FANUC Series 0*i*-MODEL F Plus

Common to Lathe System / Machining Center System

OPERATOR'S MANUAL

FRYER MACHINE S
Rob Zahensky

B-64694EN/01

- No part of this manual may be reproduced in any form.
- The appearance and specifications of this product are subject to change without notice.

The products in this manual are controlled based on Japan's “Foreign Exchange and Foreign Trade Law". The export from Japan may be subject to an export license by the government of Japan.
Further, re-export to another country may be subject to the license of the government of the country from where the product is re-exported. Furthermore, the product may also be controlled by re-export regulations of the United States government.
Should you wish to export or re-export these products, please contact FANUC for advice.

The products in this manual are manufactured under strict quality control. However, when a serious accident or loss is predicted due to a failure of the product, pay careful attention to safety.

In this manual, we endeavor to include all pertinent matters.
There are, however, a very large number of operations that must not or cannot be performed, and if the manual contained them all, it would be enormous in volume.
It is, therefore, requested to assume that any operations that are not explicitly described as being possible are "not possible".

This manual contains the program names or device names of other companies, some of which are registered trademarks of respective owners. However, these names are not followed by ® or ™ in the main body.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

SAFETY PRECAUTIONS

This section describes the safety precautions related to the use of CNC units.
It is essential that these precautions be observed by users to ensure the safe operation of machines equipped with a CNC unit (all descriptions in this section assume this configuration). Note that some precautions are related only to specific functions, and thus may not be applicable to certain CNC units.
Users must also observe the safety precautions related to the machine, as described in the relevant manual supplied by the machine tool builder. Before attempting to operate the machine or create a program to control the operation of the machine, the operator must become fully familiar with the contents of this manual and relevant manual supplied by the machine tool builder.

DEFINITION OF WARNING, CAUTION, AND NOTE

This manual includes safety precautions for protecting the user and preventing damage to the machine. Precautions are classified into **Warning** and **Caution** according to their bearing on safety. Also, supplementary information is described as a **Note**. Read the **Warning**, **Caution**, and **Note** thoroughly before attempting to use the machine.

⚠ **WARNING**
Used if a danger resulting in the death or serious injury of the user is expected to occur if he or she fails to observe the approved procedure.

⚠ **CAUTION**
Used if a danger resulting in the minor or moderate injury of the user or equipment damage is expected to occur if he or she fails to observe the approved procedure.

NOTE
Used if a supplementary explanation not related to any of DANGER, WARNING, and CAUTION is to be indicated.

- Read this manual carefully, and store it in a safe place.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

GENERAL WARNINGS AND CAUTIONS

⚠ WARNING

1 Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

2 Before operating the machine, thoroughly check the entered data.
Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

3 Ensure that the specified feedrate is appropriate for the intended operation.
Generally, for each machine, there is a maximum allowable feedrate.
The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate.
If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

4 When using a tool compensation function, thoroughly check the direction and amount of compensation.
Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

5 The parameters for the CNC and PMC are factory-set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change.
Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

FRYER MACHINE SYSTEMS INC
Rob Zehansky

⚠ CAUTION

1 Immediately after switching on the power, do not touch any of the keys on the MDI unit until the position display or alarm screen appears on the CNC unit. Some of the keys on the MDI unit are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.

2 The OPERATOR'S MANUAL and programming manual supplied with a CNC unit provide an overall description of the machine's functions. Note that the-functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.

3 Some functions may have been implemented at the request of the machine-tool builder. When using such functions, refer to the manual supplied by the machine-tool builder for details of their use and any related cautions.

4 The liquid-crystal display is manufactured with very precise fabrication technology. Some pixels may not be turned on or may remain on. This phenomenon is a common attribute of LCDs and is not a defect.

NOTE

Programs, parameters, and macro variables are stored in non-volatile memory in the CNC unit. Usually, they are retained even if the power is turned off.
Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from non-volatile memory as part of error recovery.
To guard against the occurrence of the above, and assure quick restoration of deleted data, backup all vital data, and keep the backup copy in a safe place.
The number of times to write machining programs to the non-volatile memory is limited.
You must use "High-speed program management" when registration and the deletion of the machining programs are frequently repeated in such case that the machining programs are automatically downloaded from a personal computer at each machining.
In "High-speed program management", the program is not saved to the non-volatile memory at registration, modification, or deletion of programs.

FRYER MACHINE SYSTEMS INC
Bob Zaremsky

12.3 RELATION WITH CONVENTIONAL FUNCTIONS 232
12.3.1 Relation with Folders 232
12.3.2 Relation with Program Names 233
12.3.3 Related Parameters 234
12.3.4 Part Program Storage Size / Number of Registerable Programs 235

13 PROGRAM CONFIGURATION 236
13.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS 237
13.2 PROGRAM SECTION CONFIGURATION 239
13.3 SUBPROGRAM (M98, M99) 245
13.4 COMMAND PATH OF MACRO CALL OR SUBPROGRAM CALL 249

14 FUNCTIONS TO SIMPLIFY PROGRAMMING 251
14.1 FIGURE COPYING (G72.1, G72.2) 251
14.2 3-DIMENSIONAL COORDINATE SYSTEM CONVERSION 257
14.3 TILTED WORKING PLANE INDEXING 270
14.3.1 Tilted Working Plane Indexing 270
14.3.1.1 Tilted working plane indexing based on Eulerian angle 273
14.3.1.2 General specifications of the tilted working plane indexing 274
14.3.1.3 Tilted working plane indexing based on roll-pitch-yaw 279
14.3.1.4 Tilted working plane indexing based on three points 281
14.3.1.5 Tilted working plane indexing based on two vectors 285
14.3.1.6 Tilted working plane indexing based on projection angles 288
14.3.1.7 Tilted working plane indexing by tool axis direction 291
14.3.2 Multiple Command of Tilted Working Plane Indexing 301
14.3.2.1 Absolute multiple command 301
14.3.2.2 Incremental multiple command 303
14.3.3 Tool Axis Direction Control 305
14.3.3.1 Tool axis direction control 305
14.3.3.2 Tool center point retention type tool axis direction control 322
14.3.4 Tilted Working Plane Indexing in Tool Length Compensation 328
14.3.5 Restrictions of Tilted Working Plane Indexing 332

15 COMPENSATION FUNCTION 336
15.1 TOOL LENGTH COMPENSATION (G43, G44, G49) 336
15.1.1 Overview 336
15.1.2 G53, G28, and G30 Commands in Tool Length Compensation Mode 341
15.2 SCALING (G50, G51) 342
15.3 PROGRAMMABLE MIRROR IMAGE (G50.1, G51.1) 349
15.4 NORMAL DIRECTION CONTROL (G40.1,G41.1,G42.1) 351
15.5 TOOL OFFSET FOR MILLING AND TURNING FUNCTION (G43.7) 356

16 CUSTOM MACRO 358
16.1 VARIABLES 358
16.2 SYSTEM VARIABLES 364
16.3 DISPLAY OF SYSTEM VARIABLES 413
16.4 READING AND WRITING VARIABLES FOR ANOTHER PATH 414
16.5 ARITHMETIC AND LOGIC OPERATION 418
16.6 INDIRECT AXIS ADDRESS SPECIFICATION 425
16.7 READING PARAMETERS 426
16.8 MACRO STATEMENTS AND NC STATEMENTS 427
16.9 BRANCH AND REPETITION 428

FRYER MACHINE SYSTEMS INC
R.b Zahensky

⚠ **WARNING**

7 **Tool post interference check**
A tool post interference check is performed based on the tool data specified during automatic operation. If the tool specification does not match the tool actually being used, the interference check cannot be made correctly, possibly damaging the tool or the machine itself, or causing injury to the user. After switching on the power, or after selecting a tool post manually, always start automatic operation and specify the tool number of the tool to be used.

8 **Same address command in same block**
The G code or M code including the same address cannot be commanded on the same block. If you use the same address, it may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user. Command on separate block.（About address P, refer to the appendix “List of functions include address P in the program command”）

⚠ **CAUTION**

1 **Absolute/incremental mode**
If a program created with absolute values is run in incremental mode, or vice versa, the machine may behave unexpectedly.

2 **Plane selection**
If an incorrect plane is specified for circular interpolation, helical interpolation, or a canned cycle, the machine may behave unexpectedly. Refer to the descriptions of the respective functions for details.

3 **Torque limit skip**
Before attempting a torque limit skip, apply the torque limit. If a torque limit skip is specified without the torque limit actually being applied, a move command will be executed without performing a skip.

4 **Programmable mirror image**
Note that programmed operations vary considerably when a programmable mirror image is enabled.

5 **Compensation function**
If a command based on the machine coordinate system or a reference position return command is issued in compensation function mode, compensation is temporarily canceled, resulting in the unexpected behavior of the machine. Before issuing any of the above commands, therefore, always cancel compensation function mode.

FRYER MACHINE SYSTEMS INC
Rob Zaherusky

WARNINGS AND CAUTIONS RELATED TO HANDLING

This section presents safety precautions related to the handling of machine tools. Before attempting to operate your machine, read the supplied OPERATOR'S MANUAL carefully, such that you are fully familiar with their contents.

⚠ **WARNING**

1 **Manual operation**
When operating the machine manually, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and feedrate have been specified correctly. Incorrect operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

2 **Manual reference position return**
After switching on the power, perform manual reference position return as required.
If the machine is operated without first performing manual reference position return, it may behave unexpectedly. Stroke check is not possible before manual reference position return is performed.
An unexpected operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3 **Manual handle feed**
In manual handle feed, rotating the handle with a large scale factor, such as 100, applied causes the tool and table to move rapidly. Careless handling may damage the tool and/or machine, or cause injury to the user.

4 **Disabled override**
If override is disabled (according to the specification in a macro variable) during threading, rigid tapping, or other tapping, the speed cannot be predicted, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

5 **Origin/preset operation**
Basically, never attempt an origin/preset operation when the machine is operating under the control of a program. Otherwise, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the tool, or causing injury to the user.

6 **Workpiece coordinate system shift**
Manual intervention, machine lock, or mirror imaging may shift the workpiece coordinate system. Before attempting to operate the machine under the control of a program, confirm the coordinate system carefully.
If the machine is operated under the control of a program without making allowances for any shift in the workpiece coordinate system, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

FRYER MACHINE SYSTEMS INC
Rob Zakrensky

⚠ **WARNING**

7 **Software operator's panel and menu switches**

Using the software operator's panel and menu switches, in combination with the MDI unit, it is possible to specify operations not supported by the machine operator's panel, such as mode change, override value change, and jog feed commands.

Note, however, that if the MDI unit keys are operated inadvertently, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

8 **RESET key**

Pressing the RESET key stops the currently running program. As a result, the servo axes are stopped. However, the RESET key may fail to function for reasons such as an MDI unit problem. So, when the motors must be stopped, use the emergency stop button instead of the RESET key to ensure security.

⚠ **CAUTION**

1 **Manual intervention**

If manual intervention is performed during programmed operation of the machine, the tool path may vary when the machine is restarted. Before restarting the machine after manual intervention, therefore, confirm the settings of the manual absolute switches, parameters, and absolute/incremental command mode.

2 **Feed hold, override, and single block**

The feed hold, feedrate override, and single block functions can be disabled using custom macro system variable #3004. Be careful when operating the machine in this case.

3 **Dry run**

Usually, a dry run is used to confirm the operation of the machine. During a dry run, the machine operates at dry run speed, which differs from the corresponding programmed feedrate. Note that the dry run speed may sometimes be higher than the programmed feed rate.

4 **Program editing**

If the machine is stopped, after which the machining program is edited (modification, insertion, or deletion), the machine may behave unexpectedly if machining is resumed under the control of that program. Basically, do not modify, insert, or delete commands from a machining program while it is in use.

5 **PS alarm**

When a PS alarm is occurred during executing a blolck, the axis movement of the block is continued to the end of block. After finishing the executing the block, the axis movement stops.

FRYER MACHINE SYSTEMS INC
Rob Zahonsky

WARNINGS RELATED TO DAILY MAINTENANCE

⚠ **WARNING**

1 **Memory backup battery replacement**

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.

When replacing the batteries, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).

Touching the uncovered high-voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The CNC uses batteries to preserve the contents of its memory, because it must retain data such as programs, offsets, and parameters even while external power is not applied.

If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.

When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the contents of the CNC's memory will be lost.

Refer to the Section “Method of replacing battery” in the OPERATOR’S MANUAL (Common to Lathe System/Machining Center System) for details of the battery replacement procedure.

⚠ **WARNING**

2 **Absolute pulse coder battery replacement**

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.

When replacing the batteries, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).

Touching the uncovered high-voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The absolute pulse coder uses batteries to preserve its absolute position.

If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.

When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the absolute position data held by the pulse coder will be lost.

Refer to the FANUC SERVO MOTOR αi series Maintenance Manual for details of the battery replacement procedure.

FRYER MACHINE SYSTEMS INC
Ron Zahersky

⚠ **WARNING**

3 Fuse replacement

Before replacing a blown fuse, however, it is necessary to locate and remove the cause of the blown fuse.
For this reason, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing a fuse with the cabinet open, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching an uncovered high-voltage circuit presents an extremely dangerous electric shock hazard.

⚠ **WARNING**

4 When using the controller unit, display unit, MDI unit, or machine operator's panel, prevent these units from directly exposing to chips or coolants. Even if direct exposure to coolants is prevented, coolants containing sulfur or chlorine at a high activation level, oil-free synthetic-type coolants, or water-soluble coolants at a high alkali level particularly have large effects on the control unit and peripheral units, possibly causing the following failures.

- Coolants containing sulfur or chlorine at a high activation level
  Some coolants containing sulfur or chlorine are at an extremely high activity level. If such a coolant adheres to the CNC or peripheral units, it reacts chemically with a material, such as resin, of equipment, possibly leading to corrosion or deterioration. If it gets in the CNC or peripheral units, it corrodes metals, such as copper and silver, used as component materials, possibly leading to a defective component.
- Synthetic-type coolants having a high permeability
  Some synthetic-type coolants whose lubricating component is, for example, PAG (polyalkylene glycol) have an extremely high permeability. If such a coolant is used even in equipment having a high closeness, it can readily flow into the CNC or peripheral units through, for example, gaskets. It is likely that, if the coolant gets in the CNC or a peripheral unit, it may deteriorate the insulation and damage the components.
- Water-soluble coolants at a high alkali level
  Some coolants whose pH is increased using alkanolamine are so strong alkali that its standard dilution will lead to pH10 or higher. If such a coolant spatters over the surface of the CNC or peripheral unit, it reacts chemically with a material, such as resin, possibly leading to corrosion or deterioration.

FRYER MACHINE SYSTEMS INC
Rob Zanensky

FRYER MACHINE SYSTEMS INC
Rob Zahensky

TABLE OF CONTENTS

SAFETY PRECAUTIONS s-1

I. GENERAL

1 GENERAL 3
1.1 NOTES ON READING THIS MANUAL 6
1.2 NOTES ON VARIOUS KINDS OF DATA 6

II. PROGRAMMING

1 GENERAL 9
1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE-INTERPOLATION 9
1.2 FEED-FEED FUNCTION 11
1.3 PART DRAWING AND TOOL MOVEMENT 12
1.3.1 Reference Position (Machine-specific Position) 12
1.3.2 Coordinate System on Part Drawing and Coordinate System Specified by CNC - Coordinate System 13
1.3.3 How to Indicate Command Dimensions for Moving the Tool (Absolute and Incremental Programming) 18
1.4 CUTTING SPEED - SPINDLE FUNCTION 21
1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING - TOOL FUNCTION 22
1.6 COMMAND FOR MACHINE OPERATIONS - AUXILIARY FUNCTION 23
1.7 PROGRAM CONFIGURATION 24
1.8 TOOL MOVEMENT RANGE - STROKE 26

2 CONTROLLED AXES 27
2.1 NUMBER OF CONTROLLED AXES 27
2.2 NAMES OF AXES 27
2.3 INCREMENT SYSTEM 28
2.4 MAXIMUM STROKE 29
2.5 MAXIMUM COMMAND VALUES 30

3 PREPARATORY FUNCTION (G FUNCTION) 32
3.1 G CODE LIST IN THE MACHINING CENTER SYSTEM 33
3.2 G CODE LIST IN THE LATHE SYSTEM 36

4 INTERPOLATION FUNCTIONS 40
4.1 POSITIONING (G00) 40
4.2 SINGLE DIRECTION POSITIONING (G60) 41
4.3 LINEAR INTERPOLATION (G01) 43
4.4 CIRCULAR INTERPOLATION (G02, G03) 45
4.5 HELICAL INTERPOLATION (G02, G03) 49
4.6 POLAR COORDINATE INTERPOLATION (G12.1, G13.1) 51

FRYER MACHINE SYSTEMS INC
Ron Zalensky

4.7 CYLINDRICAL INTERPOLATION (G07.1) 59
4.7.1 Cylindrical Interpolation 59
4.7.2 Cylindrical Interpolation by Plane Distance Command 62
4.7.3 Cutting Point Interpolation for Cylindrical Interpolation 64
4.8 SMOOTH TOLERANCE+ CONTROL 74
4.8.1 Change Tolerance in Smooth Tolerance+ Control Mode 76
4.9 VARIABLE LEAD THREADING (G34) 78
4.10 CIRCULAR THREADING (G35, G36) 79
4.11 SKIP FUNCTION (G31) 83
4.12 MULTI-STEP SKIP (G31) 85
4.13 HIGH-SPEED SKIP (G31) 86
4.14 SKIP POSITION MACRO VARIABLE IMPROVEMENT 86
4.15 CONTINUOUS HIGH-SPEED SKIP FUNCTION 87
4.16 TORQUE LIMIT SKIP 88

5 FEED FUNCTIONS 91
5.1 OVERVIEW 91
5.2 RAPID TRAVERSE 92
5.3 CUTTING FEED 93
5.4 CUTTING FEEDRATE CONTROL 100
5.4.1 Exact Stop (G09, G61), Cutting Mode (G64), Tapping Mode (G63) 101
5.4.2 Automatic Corner Override 102
5.4.2.1 Automatic override for inner corners (G62) 102
5.4.2.2 Internal circular cutting feedrate change 104
5.5 FEEDRATE INSTRUCTION ON IMAGINARY CIRCLE FOR A ROTARY AXIS 105
5.6 DWELL 108

6 REFERENCE POSITION 110
6.1 REFERENCE POSITION RETURN 110

7 COORDINATE SYSTEM 118
7.1 MACHINE COORDINATE SYSTEM 119
7.2 WORKPIECE COORDINATE SYSTEM 122
7.2.1 Setting a Workpiece Coordinate System 122
7.2.2 Selecting a Workpiece Coordinate System 124
7.2.3 Changing Workpiece Coordinate System 125
7.2.4 Workpiece Coordinate System Preset (G92.1) 128
7.2.5 Addition of Workpiece Coordinate System Pair (G54.1 or G54) 130
7.2.6 Automatic Coordinate System Setting 132
7.2.7 Workpiece Coordinate System Shift 132
7.3 LOCAL COORDINATE SYSTEM 134
7.4 PLANE SELECTION 136

8 COORDINATE VALUE AND DIMENSION 137
8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING 137
8.2 INCH/METRIC CONVERSION (G20, G21) 139
8.3 DECIMAL POINT PROGRAMMING 142
8.4 DIAMETER AND RADIUS PROGRAMMING 144

9 SPINDLE SPEED FUNCTION (S FUNCTION) 145

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

9.1 SPECIFYING THE SPINDLE SPEED WITH A CODE 145
9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5-DIGIT COMMAND) 145
9.3 CONSTANT SURFACE SPEED CONTROL (G96, G97) 145
9.4 SPINDLE POSITIONING FUNCTION 152
9.4.1 Spindle Orientation 152
9.4.2 Spindle Positioning 153
9.4.3 Canceling Spindle Positioning 154
9.5 SPINDLE SPEED FLUCTUATION DETECTION 156
9.6 SPINDLE CONTROL WITH SERVO MOTOR 162
9.6.1 Spindle Control with Servo Motor 162
9.6.2 Spindle Indexing Function 163

10 TOOL FUNCTION (T FUNCTION) 165
10.1 TOOL SELECTION FUNCTION 165
10.2 TOOL MANAGEMENT FUNCTION 166
10.3 TOOL MANAGEMENT EXTENSION FUNCTION 181
10.3.1 Customization of Tool Management Data Display 181
10.3.2 Setting of Spindle Position / Standby Position Display 185
10.3.3 Input of Customize Data with the Decimal Point 187
10.3.4 Protection of Tool Information with the KEY Signal 189
10.3.5 Selection of a Tool Life Count Period 190
10.3.6 Each tool Data Screen 190
10.3.7 Total Life Time Display for Tools of The Same Type 190
10.4 TOOL MANAGEMENT FUNCTION FOR OVERSIZE TOOLS 191
10.5 TOOL LIFE MANAGEMENT 192
10.5.1 Tool Life Management Data 194
10.5.2 Registering, Changing, and Deleting Tool Life Management Data 196
10.5.3 Tool Life Management Commands in Machining Program 201
10.5.4 Tool Life Counting and Tool Selection 207
10.5.5 Tool Life Count Restart M Code 209
10.5.6 Disabling Life Count 211
10.5.7 Remaining Tool Number Check Function 211

11 AUXILIARY FUNCTION 213
11.1 AUXILIARY FUNCTION (M FUNCTION) 213
11.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK 214
11.3 M CODE GROUPING FUNCTION 215
11.3.1 Setting an M Code Group Number Using the Setting Screen 215
11.3.2 Setting an M Code Group Number Using a Program 217
11.3.3 M Code Group Check Function 217
11.4 SECOND AUXILIARY FUNCTIONS (B CODES) 218
11.5 AUXILIARY FUNCTION OUTPUT IN MOVING AXIS 220

12 PROGRAM MANAGEMENT 226
12.1 FOLDERS 226
12.1.1 Folder Configuration 226
12.1.2 Folder Attributes 228
12.1.3 Default Folders 229
12.2 PROGRAMS 229
12.2.1 Program Name 229
12.2.2 Program Attributes 231

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3 RELATION WITH CONVENTIONAL FUNCTIONS 232
12.3.1 Relation with Folders 232
12.3.2 Relation with Program Names 233
12.3.3 Related Parameters 234
12.3.4 Part Program Storage Size / Number of Registerable Programs 235

13 PROGRAM CONFIGURATION 236
13.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS 237
13.2 PROGRAM SECTION CONFIGURATION 239
13.3 SUBPROGRAM (M98, M99) 245
13.4 COMMAND PATH OF MACRO CALL OR SUBPROGRAM CALL 249

14 FUNCTIONS TO SIMPLIFY PROGRAMMING 251
14.1 FIGURE COPYING (G72.1, G72.2) 251
14.2 3-DIMENSIONAL COORDINATE SYSTEM CONVERSION 257
14.3 TILTED WORKING PLANE INDEXING 270
14.3.1 Tilted Working Plane Indexing 270
14.3.1.1 Tilted working plane indexing based on Eulerian angle 273
14.3.1.2 General specifications of the tilted working plane indexing 274
14.3.1.3 Tilted working plane indexing based on roll-pitch-yaw 279
14.3.1.4 Tilted working plane indexing based on three points 281
14.3.1.5 Tilted working plane indexing based on two vectors 285
14.3.1.6 Tilted working plane indexing based on projection angles 288
14.3.1.7 Tilted working plane indexing by tool axis direction 291
14.3.2 Multiple Command of Tilted Working Plane Indexing 298
14.3.2.1 Absolute multiple command 298
14.3.2.2 Incremental multiple command 298
14.3.3 Tool Axis Direction Control 298
14.3.3.1 Tool axis direction control 298
14.3.3.2 Tool center point retention type tool axis direction control 298
14.3.4 Tilted Working Plane Indexing in Tool Length Compensation 298
14.3.5 Restrictions of Tilted Working Plane Indexing 298

15 COMPENSATION FUNCTION 336
15.1 TOOL LENGTH COMPENSATION (G43, G44, G49) 336
15.1.1 Overview 336
15.1.2 G53, G28, and G30 Commands in Tool Length Compensation Mode 341
15.2 SCALING (G50, G51) 342
15.3 PROGRAMMABLE MIRROR IMAGE (G50.1, G51.1) 349
15.4 NORMAL DIRECTION CONTROL (G40.1,G41.1,G42.1) 351
15.5 TOOL OFFSET FOR MILLING AND TURNING FUNCTION (G43.7) 356

16 CUSTOM MACRO 358
16.1 VARIABLES 358
16.2 SYSTEM VARIABLES 364
16.3 DISPLAY OF SYSTEM VARIABLES 413
16.4 READING AND WRITING VARIABLES FOR ANOTHER PATH 414
16.5 ARITHMETIC AND LOGIC OPERATION 418
16.6 INDIRECT AXIS ADDRESS SPECIFICATION 425
16.7 READING PARAMETERS 426
16.8 MACRO STATEMENTS AND NC STATEMENTS 427
16.9 BRANCH AND REPETITION 428

FRYER MACHINE SYSTEMS INC
Rob Zahensky

16.9.1 Unconditional Branch (GOTO Statement) / Unconditional Branch with label (LGOTO Statement) ... 428
16.9.2 GOTO Statement Using Stored Sequence Numbers / LGOTO Statement Using Stored Label ... 429
16.9.3 Conditional Branch (IF Statement) ... 431
16.9.4 Repetition (WHILE Statement) ... 436
16.9.5 Precision Specification for Macro Relational Operators ... 438
16.9.6 Multidirectional branch (SWITCH-CASE Statement) ... 440
16.10 MACRO CALL ... 445
16.10.1 Simple Call (G65) ... 446
16.10.2 Modal Call: Call After the Move Command (G66) ... 453
16.10.3 Modal Call: Each Block Call (G66.1) ... 456
16.10.4 Macro Call Using a G Code ... 458
16.10.5 Macro Call Using a G Code (Specification of Multiple Definitions) ... 459
16.10.6 Macro Call Using a G Code with a Decimal Point (Specification of Multiple Definitions) ... 460
16.10.7 Macro Call Using an M Code ... 461
16.10.8 Macro Call Using an M Code (Specification of Multiple Definitions) ... 462
16.10.9 Special Macro Call using M code ... 463
16.10.10 Subprogram Call Using an M Code ... 465
16.10.11 Subprogram Call Using an M Code (Specification of Multiple Definitions) ... 466
16.10.12 Subprogram Calls Using a T Code ... 466
16.10.13 Subprogram Calls Using an S Code ... 467
16.10.14 Subprogram Calls Using a Secondary Auxiliary Function ... 467
16.10.15 Subprogram Call Using a Specific Address ... 468
16.11 MACRO CALL ARGUMENT FOR AXIS NAME EXPANSION ... 471
16.12 PROCESSING MACRO STATEMENTS ... 472
16.13 REGISTERING CUSTOM MACRO PROGRAMS ... 475
16.14 CODES AND RESERVED WORDS USED IN CUSTOM MACROS ... 475
16.15 EXTERNAL OUTPUT COMMANDS ... 476
16.16 RESTRICTIONS ... 481
16.17 INTERRUPTION TYPE CUSTOM MACRO ... 483
16.17.1 Specification Method ... 484
16.17.2 Details of Functions ... 485
16.18 DISPLAYING VARIABLE IN MACRO ALARM OR MACRO MESSAGE ... 492

17 REAL-TIME CUSTOM MACRO ... 497
17.1 TYPES OF REAL TIME MACRO COMMANDS ... 499
17.1.1 Modal Real Time Macro Command / One-shot Real Time Macro Command ... 499
17.1.2 Cancel Command for Real Time Custom Macro ... 505
17.2 VARIABLES ... 507
17.2.1 Variables Dedicated to Real Time Custom Macros ... 508
17.2.1.1 System variables ... 508
17.2.1.2 Real time macro variables (RTM variables) ... 510
17.2.2 Custom Macro Variables ... 511
17.2.2.1 System variables ... 512
17.2.2.2 Local variables ... 513
17.2.3 Reading and writing variables for another path ... 514
17.3 ARITHMETIC AND LOGICAL OPERATION ... 515
17.4 CONTROL ON REAL TIME MACRO COMMANDS ... 517
17.4.1 Conditional Branch (ZONCE Statement) ... 518
17.4.2 Condition Transition (ZEDGE Statement) ... 519
17.4.3 Repetition (ZWHILE Statement) ... 520

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

17.4.4 Multi-statement (ZDO...ZEND Statement) 521
17.5 MACRO CALL 523
17.6 OTHERS 524
17.7 AXIS CONTROL COMMAND 525
17.8 AUXILIARY FUNCTION 535
17.9 NOTES 538
17.10 LIMITATION 540

18 PROGRAMMABLE DATA INPUT (G10) 542
18.1 PROGRAMMABLE DATA INPUT (G10) 542
18.2 PROGRAMMABLE PARAMETER INPUT (G10) 545

19 PATTERN DATA INPUT 550
19.1 OVERVIEW 550
19.2 EXPLANATION 550
19.3 EXPLANATION OF OPERATION 553
19.4 DEFINITION OF THE SCREEN 556
19.4.1 Definition of the Pattern Menu Screen 557
19.4.2 Definition of the Custom Macro Screen 559
19.4.3 Setting the Character-codes 562

20 HIGH-SPEED CUTTING FUNCTIONS 567
20.1 AI CONTOUR CONTROL FUNCTION I AND AI CONTOUR CONTROL FUNCTION II (G05.1) 567
20.2 MACHINING CONDITION SELECTING FUNCTION 583
20.3 OPTIMUM ACCELERATION/DECELERATION FOR RIGID TAPPING 584
20.4 G CODE PREVENTING BUFFERING 585
20.5 Speeding up of rigid tapping 587

21 AXIS CONTROL FUNCTIONS 590
21.1 AXIS SYNCHRONOUS CONTROL 590
21.2 POLYGON TURNING (G50.2, G51.2, G250, G251) 592
21.3 SYNCHRONOUS, COMPOSITE AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND (G50.4, G51.4, G50.5, G51.5, G50.6, AND G51.6) 597
21.4 ROTARY AXIS ROLL-OVER 601
21.4.1 Rotary Axis Roll-over 601
21.5 TOOL RETRACT AND RECOVER 602
21.5.1 Improvement of Tool compensation for Tool Retract and Recover 605
21.6 ELECTRONIC GEAR BOX 607
21.6.1 Electronic Gear Box 607
21.6.2 Spindle Electronic Gear Box 613
21.6.3 Electronic Gear Box Automatic Phase Synchronization 620
21.6.4 Skip Function for EGB Axis 624
21.6.5 Electronic Gear Box 2 Pair 626
21.6.5.1 Specification method (G80.5, G81.5) 626
21.6.5.2 Description of commands compatible with those for a hobbing machine (G80, G81) 628
21.6.5.3 Controlled axis configuration example 632
21.6.5.4 Sample programs 632
21.6.5.5 Synchronization ratio specification range 635

FRYER MACHINE SYSTEMS INC
Rob Zanensky

21.6.5.6 Retract function 638
21.6.6 U-axis Control 639
21.7 TANDEM CONTROL 640
21.8 SKIP FUNCTION FOR FLEXIBLE SYNCHRONIZATION CONTROL 640
21.9 HIGH PRECISION OSCILLATION FUNCTION 641

22 MUITI-PATH CONTROL FUNCTION 650
22.1 OVERVIEW 650
22.2 WAITING FUNCTION FOR PATHS 650
22.3 WAITING M CODES OF HIGH-SPEED TYPE 654
22.4 COMMON MEMORY BETWEEN EACH PATH 654
22.5 SPINDLE CONTROL BETWEEN EACH PATH 656
22.6 SYNCHRONOUS/COMPOSITE/SUPERIMPOSED CONTROL 657
22.7 WAITING FUNCTION BY SPECIFYING START POINT 659
22.8 FUNCTION FOR LOADER CONTROL 665

III. OPERATION

1 GENERAL 669
1.1 MANUAL OPERATION 669
1.2 TOOL MOVEMENT BY PROGRAMING - AUTOMATIC OPERATION 671
1.3 AUTOMATIC OPERATION 672
1.4 TESTING A PROGRAM 673
1.4.1 Check by Running the Machine 673
1.4.2 How to View the Position Display Change without Running the Machine 674
1.5 EDITING A PROGRAM 674
1.6 DISPLAYING AND SETTING DATA 675
1.7 DISPLAY 678
1.7.1 Program Display 678
1.7.2 Current Position Display 679
1.7.3 Alarm Display 680
1.7.4 Parts Count Display, Run Time Display 680

2 OPERATIONAL DEVICES 681
2.1 POWER ON/OFF 681
2.1.1 Turning on the Power 681
2.1.2 Power Disconnection 682
2.2 SETTING AND DISPLAY UNITS 682
2.2.1 10.4" LCD CNC Display Panel (15"/19" LCD CNC Display Panel) 682
2.2.2 Standard MDI Unit (ONG Key) 683
2.2.3 Standard MDI Unit (QWERTY Key) 684
2.2.4 Small MDI Unit (ONG Key) 685
2.3 EXPLANATION OF THE MDI UNIT 686
2.4 FUNCTION KEYS AND SOFT KEYS 688
2.4.1 General Screen Operations 688
2.4.2 Function Keys 690
2.4.3 Soft Keys 691
2.5 EXTERNAL I/O DEVICES 698

3 MANUAL OPERATION 700
3.1 MANUAL REFERENCE POSITION RETURN 700

FRYER MACHINE SYSTEMS INC
Rob Zahensky

3.2 JOG FEED (JOG) 701
3.3 INCREMENTAL FEED 703
3.4 MANUAL HANDLE FEED 704
3.5 MANUAL ABSOLUTE ON AND OFF 707
3.6 MANUAL LINEAR/CIRCULAR INTERPOLATION 711
3.7 RIGID TAPPING BY MANUAL HANDLE 715
3.8 3-DIMENSIONAL MANUAL FEED 718
3.8.1 Tool Axis Direction Handle Feed / Tool Axis Direction JOG Feed / Tool Axis Direction Incremental Feed 719
3.8.2 Tool Axis Right-Angle Direction Handle Feed / Tool Axis Right-Angle Direction JOG Feed / Tool Axis Right-Angle Direction Incremental Feed 721
3.8.3 Tool Tip Center Rotation Handle Feed / Tool Tip Center Rotation JOG Feed / Tool Tip Center Rotation Incremental Feed 725
3.8.4 Table Vertical Direction Handle Feed / Table Vertical Direction JOG Feed / Table Vertical Direction Incremental Feed 727
3.8.5 Table Horizontal Direction Handle Feed / Table Horizontal Direction JOG Feed / Table Horizontal Direction Incremental Feed 728
3.9 DISTANCE CODED LINEAR SCALE INTERFACE 732
3.9.1 Procedure for Reference Position Establishment 732
3.9.2 Reference Position Return 733
3.9.3 Distance Coded Rotary Encoder 733
3.9.4 Axis Synchronization Control 734
3.9.5 PMC Axis Control 735
3.9.6 Angular Axis Control 735
3.9.7 Note 736
3.10 LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL) 737

4 AUTOMATIC OPERATION 742
4.1 MEMORY OPERATION 742
4.2 MDI OPERATION 744
4.3 DNC OPERATION 747
4.4 SCHEDULE OPERATION 750
4.5 EXTERNAL SUBPROGRAM CALL (M198) 752
4.6 EXTERNAL SUBPROGRAM CALLS USING THE DATA SERVER AVAILABLE IN MULTI-PATH SYSTEMS 757
4.7 MANUAL HANDLE INTERRUPTION 759
4.7.1 Manual Interruption of 3-dimensional Coordinate System Conversion 764
4.8 MIRROR IMAGE 765
4.9 PROGRAM RESTART 768
4.9.1 Auxiliary Function Output in Program Restart Function 784
4.10 QUICK PROGRAM RESTART 788
4.10.1 Suppress Motion of Quick Program Restart 805
4.10.2 Quick Program Restart for a Machining Cycle 809
4.10.3 Quick Program Restart for Multi Path System 817
4.10.4 Improvement of Quick program restart for non-machining program 824
4.11 TOOL RETRACT AND RECOVER 825
4.11.1 Retract 828
4.11.2 Withdrawal 828
4.11.3 Return 829
4.11.4 Repositioning 829

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

4.11.5 Tool Retract and Recover for Threading 829
4.11.6 Operation Procedure for a Canned Cycle for Drilling 833
4.12 MANUAL INTERVENTION AND RETURN 835
4.13 RETRACE 838
4.14 RETRACTION FOR RIGID TAPPING / RETRACTION FOR 3-DIMENSIONAL RIGID TAPPING 847

5 TEST OPERATION 850
5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK 850
5.1.1 Machine Lock 850
5.1.2 Auxiliary Function Lock 851
5.2 FEEDRATE OVERRIDE 852
5.3 RAPID TRAVERSE OVERRIDE 852
5.4 DRY RUN 853
5.5 SINGLE BLOCK 854
5.6 HIGH SPEED PROGRAM CHECK FUNCTION 855
5.7 MANUAL HANDLE RETRACE 857
5.7.1 Displaying Program Being Executed in Backward Movement 870
5.8 AUXILIARY FUNCTION OUTPUT BLOCK REVERSE MOVEMENT FOR MANUAL HANDLE RETRACE 871
5.9 MANUAL HANDLE RETRACE FUNCTION FOR MULTI-PATH 871
5.10 EXTENSION OF THE MANUAL HANDLE RETRACE FUNCTION 874

6 SAFETY FUNCTIONS 878
6.1 EMERGENCY STOP 878
6.2 OVERTRAVEL 878
6.3 STORED STROKE CHECK 879
6.4 STROKE LIMIT CHECK BEFORE MOVE 883
6.4.1 Check of the Tool Path between Blocks by Stroke Limit Check before Move 886
6.5 STROKE LIMIT AREA CHANGING FUNCTION 892
6.6 STORED STROKE LIMIT RANGE SWITCHING FUNCTION BY SIGNAL 893
6.7 WRONG OPERATION PREVENTION FUNCTIONS 893
6.7.1 Functions that are Used When Data is Set 894
6.7.1.1 Input data range check 894
6.7.1.2 Confirmation of incremental input 895
6.7.1.3 Prohibition of the absolute input by the soft key 896
6.7.1.4 Confirmation of the deletion of the program 896
6.7.1.5 Confirmation of the deletion of all data 897
6.7.1.6 Confirmation of a data update during the data setting process 897
6.7.2 Functions that are Used when the Program is Executed 898
6.7.2.1 Display of updated modal information 898
6.7.2.2 Start check signal 899
6.7.2.3 Axis status display 899
6.7.2.4 Confirmation of the start from a middle block 900
6.7.2.5 Data range check 901
6.7.2.6 Maximum incremental value check 902
6.7.2.7 Warning display during a reset in program operation 902
6.7.3 Setting Screen 903
6.7.3.1 Operation confirmation function setting screen 903
6.7.3.2 Tool offset range setting screen 905
6.7.3.3 Workpiece origin offset range setting screen 908
6.7.3.4 Y-axis tool offset range setting screen 910

FRYER MACHINE SYSTEMS INC
ROD Zahensky

6.7.3.5 Workpiece shift range setting screen 912

7 ALARM AND SELF-DIAGNOSIS FUNCTIONS 914

7.1 ALARM DISPLAY 914
7.2 ALARM HISTORY DISPLAY 916
7.3 CHECKING BY DIAGNOSTIC DISPLAY 918
7.4 RETURN FROM THE ALARM SCREEN 919
7.4.1 Return from the Alarm Screen 919
7.4.2 Relationship with Other Functions 920

8 DATA INPUT/OUTPUT 922

8.1 OVERWRITING FILES ON A MEMORY CARD/USB MEMORY 923
8.2 INPUT/OUTPUT ON EACH SCREEN 925
8.2.1 Inputting and Outputting a Program 925
8.2.1.1 Inputting a program 925
8.2.1.2 Outputting a program 927
8.2.1.3 Inputting and outputting of O8-digit 929
8.2.2 Inputting and Outputting Parameters 929
8.2.2.1 Inputting parameters 929
8.2.2.2 Outputting parameters 930
8.2.3 Inputting and Outputting Offset Data 931
8.2.3.1 Inputting offset data 931
8.2.3.2 Outputting offset data 932
8.2.4 Inputting and Outputting Pitch Error Compensation Data 935
8.2.4.1 Inputting pitch error compensation data 935
8.2.4.2 Outputting pitch error compensation data 937
8.2.4.3 Input/output format of pitch error compensation data 937
8.2.5 Inputting and Outputting Custom Macro Common Variables 939
8.2.5.1 Inputting custom macro common variables 939
8.2.5.2 Outputting custom macro common variables 940
8.2.6 Inputting and Outputting Workpiece Coordinates System Data 941
8.2.6.1 Inputting workpiece coordinate system data 941
8.2.6.2 Outputting workpiece coordinate system data 942
8.2.7 Inputting and Outputting Operation History Data 943
8.2.7.1 Outputting operation history data 943
8.2.7.2 Inputting operation history signal selection data 944
8.2.7.3 Outputting operation history signal section data 945
8.2.7.4 Input/output format of operation history signal selection data 945
8.2.8 Inputting and Outputting Tool Management Data 946
8.2.8.1 Inputting tool management data 946
8.2.8.2 Outputting tool management data 948
8.2.8.3 Inputting magazine data 949
8.2.8.4 Outputting magazine data 950
8.2.8.5 Inputting tool life status name data 951
8.2.8.6 Outputting tool life status name data 952
8.2.8.7 Inputting name data of customize data 952
8.2.8.8 Outputting name data of customize data 953
8.2.8.9 Inputting customize data displayed as tool management data 954
8.2.8.10 Outputting customize data displayed as tool management data 955
8.2.8.11 Inputting spindle waiting position name data 956
8.2.8.12 Outputting spindle waiting position name data 957
8.2.8.13 Inputting decimal point position data of customize data 957
8.2.8.14 Outputting decimal point position data of customize data 958
8.2.8.15 Inputting tool geometry data 959
8.2.8.16 Outputting tool geometry data 960
8.2.9 Inputting and Outputting Tool Life Management Data 961

FRYER MACHINE SYSTEMS INC
Rob Zaranisky

8.2.9.1 Inputting tool life management data 961
8.2.9.2 Outputting tool life management data 962
8.3 INPUT/OUTPUT ON THE ALL IO SCREEN 963
8.3.1 Inputting/Outputting a Program 966
8.3.2 Inputting/Outputting all Programs and Folders 968
8.3.3 Inputting and Outputting Parameters 970
8.3.4 Inputting and Outputting Offset Data 971
8.3.5 Inputting/Outputting Pitch Error Compensation Data 973
8.3.6 Inputting/Outputting Custom Macro Common Variables 974
8.3.7 Inputting and Outputting Workpiece Coordinates System Data 976
8.3.8 Inputting and Outputting Operation History Data 977
8.3.9 Inputting and Outputting Tool Management Data 978
8.3.10 Inputting and Outputting All Tool Management Data at a Time 982
8.3.11 Inputting and Outputting Workpiece Setting Error Compensation Value 986
8.3.12 File Format and Error Messages 987
8.4 EMBEDDED ETHERNET OPERATIONS 988
8.4.1 Host File List Display 988
8.4.2 FTP File Transfer Function 993
8.4.3 DNC Operation File Set 996
8.4.4 M198 Folder Set 996
8.5 SCREEN HARD COPY FUNCTION 996
8.6 USB FUNCTION 998
8.7 NC DATA OUTPUT FUNCTION 999
8.7.1 All Data Output Screen 999
8.7.1.1 Output operation 1000
8.7.1.2 Format 1001
8.7.2 Output Directory 1001
8.7.2.1 LCD mounted type CNC 1001
8.7.2.2 Stand-alone type CNC (without personal computer function) 1001
8.7.2.3 Stand-alone type CNC (with personal computer function / HSSB connection) 1002
8.7.2.4 Stand-alone type CNC (with personal computer function / Ethernet connection) 1003
8.7.3 List of Output Data 1004
8.7.3.1 Text data 1004
8.7.3.2 SRAM data and User files 1005
8.7.4 List of Message 1005
8.7.4.1 Warning message 1005
8.7.4.2 Error message 1006
8.8 INPUT/OUTPUT FILE NAME FOR MULTI-PATH SYSTEM 1006

9 CREATING PROGRAMS 1008
9.1 CREATING PROGRAMS USING THE MDI UNIT 1008
9.2 AUTOMATIC INSERTION OF SEQUENCE NUMBERS 1009
9.3 CREATING PROGRAMS IN TEACH IN MODE (PLAYBACK) 1010
9.4 CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION 1013

10 EDITING PROGRAMS 1017
10.1 EDIT DISABLE ATTRIBUTE AND DISPLAY DISABLE ATTRIBUTE 1018
10.2 INSERTING, ALTERING AND DELETING A WORD 1019
10.2.1 Word Search 1020
10.2.2 Heading a Program 1021
10.2.3 Inserting a Word 1022
10.2.4 Altering a Word 1022

FRYER MACHINE SYSTEMS INC
Rob Zahensky

10.2.5 Deleting a Word 1023
10.3 REPLACING A WORD OR ADDRESS 1024
10.4 DELETING BLOCKS 1025
10.4.1 Deleting a Block 1025
10.4.2 Deleting Multiple Blocks 1025
10.5 PROGRAM SEARCH 1026
10.6 SEQUENCE NUMBER SEARCH 1027
10.7 DELETING PROGRAMS 1029
10.7.1 Deleting One Program 1029
10.7.2 Deleting All Programs 1029
10.8 EDITING OF CUSTOM MACROS 1030
10.9 CURSOR MOVEMENT LIMITATIONS ON PROGRAM EDITING 1031
10.10 PASSWORD FUNCTION 1032
10.11 EDITING PROGRAM CHARACTERS 1034
10.11.1 Available Keys 1036
10.11.2 Input Mode 1037
10.11.3 Line Number Display 1037
10.11.4 Search 1037
10.11.5 Replacement 1037
10.11.6 Reversing Edit Operations (Undo Function) 1038
10.11.7 Copy 1038
10.11.8 Cut 1039
10.11.9 Paste 1039
10.11.10 Saving 1039
10.11.11 Creation 1039
10.11.12 Line Search 1040
10.12 KEYS AND PROGRAM ENCRYPTION 1040
10.13 SIMULTANEOUS EDITING OF MULTIPATH PROGRAMS 1043
10.14 MULTI-PATH EDITING FUNCTION 1046
10.14.1 Overview 1046
10.14.2 Details 1046

11 PROGRAM MANAGEMENT 1052
11.1 SELECTING A DEVICE 1053
11.1.1 Selecting a Memory Card Program as a Device 1053
11.1.2 Selecting a From Cassette as a Device 1057
11.1.3 Selecting a USB Memory as a Device 1058
11.2 CREATING A FOLDER 1059
11.3 RENAMING A FOLDER 1060
11.4 CHANGING CURRENT FOLDER 1060
11.5 CHANGING FOLDER ATTRIBUTES 1062
11.6 DELETING A FOLDER 1063
11.7 SELECTING A DEFAULT FOLDER 1063
11.8 RENAMING A FILE 1064
11.9 DELETING A FILE 1064
11.10 CHANGING FILE ATTRIBUTES 1065
11.11 SELECTING A MAIN PROGRAM 1066
11.12 MAKING A PROGRAM COMPACT 1067
11.13 PROGRAM AND FOLDER COPY/MOVE FUNCTION 1068
11.13.1 Copy and movement between different devices 1071

FRYER MACHINE SYSTEMS INC
Rob Zahensky

11.13.2 Copy of program file with memory card device....1072
11.13.3 Deleting files in memory card device....1074
11.14 FOLDER MANAGEMENT 1075
11.14.1 Program Management under the Path Folder....1076
11.14.2 Program Management only in the Path Folder....1076
11.14.3 Folder for Subprogram/Macro Calls 1078
11.15 PROGRAM VERIFICATION 1079
11.16 HIGH-SPEED PROGRAM MANAGEMENT 1080

12 SETTING AND DISPLAYING DATA....1081

12.1 SCREENS DISPLAYED BY FUNCTION KEY POS 1099
12.1.1 Position Display in the Workpiece Coordinate System 1100
12.1.2 Position Display in the Relative Coordinate System....1101
12.1.3 Overall Position Display 1104
12.1.4 Workpiece Coordinate System Preset 1106
12.1.5 Actual Feedrate Display 1107
12.1.6 Display of Run Time and Parts Count....1109
12.1.7 Operating Monitor Display 1111
12.1.8 Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount)....1113
12.1.9 Overall Position Display (15/19-inch Display Unit)....1116
12.1.10 Workpiece Coordinate System Preset (15/19-inch Display Unit)....1119
12.1.11 Actual Feedrate Display (15/19-inch Display Unit)....1120
12.1.12 Display of Run Time and Parts Count (15/19-inch Display Unit) 1122
12.1.13 Operating Monitor Display (15/19-inch Display Unit)....1124
12.1.14 Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount) (15/19-inch Display Unit) 1126

12.2 SCREENS DISPLAYED BY FUNCTION KEY PROG 1128
12.2.1 Program Contents Display....1129
12.2.1.1 Displaying the executed block (BC) 1129
12.2.1.2 Text display.... 1131
12.2.1.3 Call stack display.... 1132
12.2.1.4 Display interpreted and original programs on the same screen.... 1134
12.2.2 Editing a Program....1137
12.2.3 Program Screen for MDI Operation 1139
12.2.4 Program Folder Screen....1140
12.2.4.1 Split display on the program folder screen 1141
12.2.4.2 Folder tree display.... 1145
12.2.5 Next Block Display Screen 1147
12.2.6 Program Check Screen 1148
12.2.6.1 Modal G codes display in 3 columns on program check screen 1149
12.2.6.2 Current block screen 1150
12.2.7 Background Editing....1158
12.2.8 Stamping the Machining Time 1164
12.2.9 Screen for Assistance in Entering Tilted Working Plane Indexing....1172
12.2.9.1 Command type selection screen.... 1178
12.2.9.2 Tilted working plane data setting screen.... 1179
12.2.9.3 Details of the tilted working plane data setting screen.... 1182
12.2.9.4 Tilted Working Plane Indexing during manual operation.... 1189
12.2.9.5 Limitation.... 1190
12.2.10 Program Contents Display (15/19-inch Display Unit) 1190
12.2.10.1 Displaying the executed block (BC) 1191
12.2.10.2 Call stack display (15/19-inch display unit).... 1191

FRYER MACHINE SYSTEMS INC
Rob Zaranosky

12.2.10.3 Display interpreted and original programs on the same screen (15/19-inch display unit) 1194
12.2.11 Editing a Program (15/19-inch Display Unit) 1196
12.2.12 Program Screen for MDI Operation (15/19-inch Display Unit) 1198
12.2.13 Program Folder Screen (15/19-inch Display Unit) 1199
12.2.13.1 Split display on the program folder screen 1200
12.2.13.2 Folder tree display (15/19-inch display unit) 1204
12.2.14 Next Block Display Screen (15/19-inch Display Unit) 1204
12.2.15 Program Check Screen (15/19-inch Display Unit) 1205
12.2.16 Background Editing (15/19-inch Display Unit) 1206
12.2.17 Stamping the Machining Time (15/19-inch Display Unit) 1212
12.2.18 Screen for Assistance in Entering Tilted Working Plane Indexing (15/19-inch Display Unit) 1220
12.2.18.1 Command type selection screen 1226
12.2.18.2 Tilted working plane data setting screen 1227
12.2.18.3 Details of the tilted working plane data setting screen 1230
12.2.18.4 Limitation 1236
12.3 SCREENS DISPLAYED BY FUNCTION KEY 1237
12.3.1 Displaying and Entering Setting Data 1237
12.3.2 Sequence Number Comparison and Stop 1240
12.3.3 Displaying and Setting Run Time, Parts Count, and Time 1241
12.3.4 Displaying and Setting the Workpiece Origin Offset Value 1243
12.3.5 Direct Input of Workpiece Origin Offset Value Measured 1244
12.3.6 Displaying and Setting Custom Macro Common Variables 1246
12.3.7 Displaying and Setting Real Time Custom Macro Data 1247
12.3.8 Displaying and Setting the Software Operator's Panel 1249
12.3.9 Setting and Displaying Tool Management Data 1251
12.3.9.1 Displaying and setting magazine screen 1251
12.3.9.2 Displaying and setting tool management screen 1253
12.3.9.3 Each tool data screen 1258
12.3.9.4 Displaying the total life of tools of the same type 1261
12.3.9.5 Tool geometry data screen 1265
12.3.10 Displaying and Switching the Display Language 1269
12.3.11 Protection of Data at Eight Levels 1271
12.3.11.1 Operation level setting 1271
12.3.11.2 Password modification 1272
12.3.11.3 Protection level setting 1273
12.3.11.4 Setting the change protection level and output protection level of a program 1276
12.3.12 Precision Level Selection 1277
12.3.13 Machining Level Selection 1279
12.3.13.1 Smoothing level selection 1279
12.3.13.2 Precision level selection 1280
12.3.14 Displaying and Setting Tool Life Management Data 1280
12.3.14.1 Tool life management (list screen) 1281
12.3.14.2 Tool life management (group editing screen) 1285
12.3.15 Displaying and Setting Pattern Data Inputs 1291
12.3.16 Setting and Displaying Data When the Tool Offset for Milling and Turning Function Is Enabled 1293
12.3.17 Eco Setting Screen 1295
12.3.18 Eco Monitoring Screen 1299
12.3.19 Displaying and Entering Setting Data (15/19-inch Display Unit) 1303
12.3.20 Sequence Number Comparison and Stop (15/19-inch Display Unit) 1306
12.3.21 Displaying and Setting Run Time, Parts Count, and Time (15/19-inch Display Unit) 1308

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.22 Displaying and Setting the Workpiece Origin Offset Value (15/19-inch Display Unit) 1310
12.3.23 Direct Input of Workpiece Origin Offset Value Measured (15/19-inch Display Unit) 1311
12.3.24 Displaying and Setting Custom Macro Common Variables (15/19-inch Display Unit) 1313
12.3.25 Displaying and Setting Real Time Custom Macro Data (15/19-inch Display Unit) 1314
12.3.26 Displaying and Setting the Software Operator's Panel (15/19-inch Display Unit) 1317
12.3.27 Setting and Displaying Tool Management Data (15/19-inch Display Unit) 1319
12.3.27.1 Displaying and setting magazine screen (15/19-inch display unit) 1319
12.3.27.2 Displaying and setting tool management screen (15/19-inch display unit) 1321
12.3.27.3 Each tool data screen (15/19-inch display unit) 1326
12.3.27.4 Displaying the total life of tools of the same type (15/19-inch display unit)1329
12.3.27.5 Tool geometry data screen (15/19-inch display unit) 1332
12.3.28 Displaying and Switching the Display Language (15/19-inch Display Unit) 1336
12.3.29 Protection of Data at Eight Levels (15/19-inch Display Unit) 1338
12.3.29.1 Operation level setting (15-inch display unit) 1338
12.3.29.2 Password modification (15/19-inch display unit) 1339
12.3.29.3 Protection level setting (15/19-inch display unit) 1340
12.3.29.4 Setting the change protection level and output protection level of a program (15/19-inch display unit) 1343
12.3.30 Precision Level Selection (15/19-inch Display Unit) 1344
12.3.31 Machining Level Selection (15/19-inch Display Unit) 1345
12.3.31.1 Smoothing level selection 1345
12.3.31.2 Precision level selection 1346
12.3.32 Displaying and Setting Tool Life Management Data (15/19-inch Display Unit)1347
12.3.32.1 Tool life management (list screen) (15/19-inch display unit) 1348
12.3.32.2 Tool life management (group editing screen) (15/19-inch display unit) 1352
12.3.33 Displaying and Setting Pattern Data Inputs (15/19-inch Display Unit) 1358
12.3.34 Setting and Displaying Data When the Tool Offset for Milling and Turning Function Is Enabled (15/19-inch Display Unit) 1360
12.3.35 Eco Setting Screen (15/19-inch Display Unit) 1361
12.3.36 Eco Monitoring Screen (15/19-inch Display Unit) 1366

12.4 SCREENS DISPLAYED BY FUNCTION KEY SYSTEM 1370

12.4.1 Displaying and Setting Parameters 1370
12.4.2 Servo Parameters 1373
12.4.3 Servo Tuning 1374
12.4.4 Spindle Setting 1375
12.4.5 Spindle Tuning 1376
12.4.6 Spindle Monitor 1377
12.4.7 Color Setting Screen 1378
12.4.8 Machining Parameter Tuning 1380
12.4.8.1 Machining parameter tuning (AI contour) 1380
12.4.9 Displaying Memory Data 1386
12.4.10 Parameter Tuning Screen 1388
12.4.10.1 Displaying the menu screen and selecting a menu item 1388
12.4.10.2 Parameter tuning screen (system setting) 1391
12.4.10.3 Parameter tuning screen (axis setting) 1392
12.4.10.4 Displaying and setting the FSSB servo amplifier setting screen 1393
12.4.10.5 Displaying and setting the FSSB spindle amplifier setting screen 1393
12.4.10.6 Displaying and setting the FSSB axis setting screen 1394
12.4.10.7 Displaying and setting the servo setting screen 1394

FRYER MACHINE SYSTEMS INC
Ron Zalansky

12.4.10.8 Parameter tuning screen (spindle setting) 1395
12.4.10.9 Parameter tuning screen (miscellaneous settings)..... 1395
12.4.10.10 Displaying and setting the servo tuning screen..... 1396
12.4.10.11 Displaying and setting the spindle tuning screen..... 1396
12.4.10.12 Displaying and setting the machining parameter tuning screen..... 1397
12.4.11 Periodic Maintenance Screen 1401
12.4.12 System Configuration Screen 1408
12.4.13 Power Consumption Monitoring Screen 1411
12.4.14 Displaying and Setting Parameters (15/19-inch Display Unit) 1413
12.4.15 Servo Parameters (15/19-inch Display Unit) 1415
12.4.16 Servo Tuning (15/19-inch Display Unit).....1416
12.4.17 Spindle Setting (15/19-inch Display Unit).....1417
12.4.18 Spindle Tuning (15/19-inch Display Unit).....1418
12.4.19 Spindle Monitor (15/19-inch Display Unit) 1419
12.4.20 Color Setting Screen (15/19-inch Display Unit) 1420
12.4.21 Machining Parameter Tuning (15/19-inch Display Unit).....1422
12.4.21.1 Machining parameter tuning (AI contour) 1422
12.4.22 Displaying Memory Data (15/19-inch Display Unit).....1428
12.4.23 Parameter Tuning Screen (15/19-inch Display Unit) 1429
12.4.23.1 Displaying the menu screen and selecting a menu item (15/19-inch display unit)..... 1429
12.4.23.2 Parameter tuning screen (system setting) (15/19-inch display unit) 1432
12.4.23.3 Parameter tuning screen (axis setting) (15/19-inch display unit)..... 1434
12.4.23.4 Displaying and setting the FSSB servo amplifier setting screen (15/19-inch display unit)..... 1435
12.4.23.5 Displaying and setting the FSSB spindle amplifier setting screen (15/19-inch display unit)..... 1435
12.4.23.6 Displaying and setting the FSSB axis setting screen (15/19-inch display unit)..... 1436
12.4.23.7 Displaying and setting the servo setting screen (15/19-inch display unit).. 1436
12.4.23.8 Parameter tuning screen (spindle setting) (15/19-inch display unit)..... 1437
12.4.23.9 Parameter tuning screen (miscellaneous settings) (15/19-inch display unit)1437
12.4.23.10 Displaying and setting the servo tuning screen (15/19-inch display unit) .. 1438
12.4.23.11 Displaying and setting the spindle tuning screen (15/19-inch display unit)1438
12.4.23.12 Displaying and setting the machining parameter tuning screen (15/19-inch display unit)..... 1439
12.4.24 Periodic Maintenance Screen (15/19-inch Display Unit).....1443
12.4.25 System Configuration Screen (15/19-inch Display Unit) 1450
12.4.26 Power Consumption Monitoring Screen (15/19-inch Display Unit).....1453
12.5 SCREENS DISPLAYED BY FUNCTION KEY MESSAGE 1455
12.5.1 External Operator Message History 1455
12.6 SWITCHING BETWEEN MULTI-PATH DISPLAY AND SINGLE-PATH DISPLAY FUNCTION 1458
12.7 PATH NAME EXPANSION DISPLAY FUNCTION 1462
12.8 SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION 1465
12.9 LOAD METER SCREEN 1466
12.9.1 Single-path Display 1466
12.9.2 Two-path Display and Three-path Display 1468
12.10 DISPLAYING THE PROGRAM NUMBER/NAME, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION 1470
12.10.1 Displaying the Program Number, Program Name, and Sequence Number.....1471
12.10.2 Displaying the Status and Warning for Data Setting or Input/Output Operation1472

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.10.3 Displaying the Program Number, Program Name, and Sequence Number (15/19-inch Display Unit) 1475
12.10.4 Displaying the Program Name 1476
12.10.5 Displaying the Status and Warning for Data Setting or Input/Output Operation (15/19-inch Display Unit) 1477
12.11 CALCULATION KEY FUNCTION 1480
12.11.1 Overview 1480
12.11.2 Operation 1480
12.11.3 Details 1480

13 GRAPHIC FUNCTION 1483
13.1 GRAPHIC DISPLAY 1483
13.2 DYNAMIC GRAPHIC DISPLAY 1494
13.2.1 Path Drawing 1495
13.2.1.1 GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen 1495
13.2.1.2 PATH GRAPHIC screen 1500
13.2.1.3 PATH GRAPHIC (TOOL POSITION) screen 1506
13.2.2 Animation 1508
13.2.2.1 GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen 1508
13.2.2.2 ANIMATION GRAPHIC screen 1511
13.2.3 Programmable Data Input (G10) for Blank Figure Drawing Parameters 1517
13.2.4 Warning Messages 1519
13.2.5 Note 1519
13.2.6 Restrictions 1520

14 VIRTUAL MDI KEY FUNCTION 1526
14.1 VIRTUAL MDI KEY 1526
14.1.1 Limitations 1529

IV. MAINTENANCE

1 ROUTINE MAINTENANCE 1533
1.1 ACTION TO BE TAKEN WHEN A PROBLEM OCCURRED 1533
1.2 BACKING UP VARIOUS DATA ITEMS 1534
1.3 METHOD OF REPLACING BATTERY 1535
1.3.1 Replacing Battery for Control Unit 1536
1.3.2 Battery in the PANEL *i* (3 VDC) 1538
1.3.3 Replacing Battery for Absolute Pulsecoders 1539
1.3.3.1 Overview 1539
1.3.3.2 Replacing batteries 1540
1.3.3.3 Replacing the batteries in a separate battery case 1540
1.3.3.4 Replacing the battery built into the servo amplifier 1541

APPENDIX

A PROGRAM CODE LIST 1545

B LIST OF FUNCTIONS AND PROGRAM FORMAT 1548

C RANGE OF COMMAND VALUE 1557

D NOMOGRAPHS 1559
D.1 INCORRECT THREADED LENGTH 1559
D.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH 1560

FRYER MACHINE SYSTEMS INC
Rob Zahensky

D.3 TOOL PATH AT CORNER 1562
D.4 RADIUS DIRECTION ERROR AT CIRCLE CUTTING 1565

E SETTINGS AT POWER-ON, IN THE CLEAR STATE, OR IN THE RESET STATE 1566

F CHARACTER-TO-CODES CORRESPONDENCE TABLE 1569
F.1 CHARACTER-TO-CODES CORRESPONDENCE TABLE 1569
F.2 FANUC DOUBLE-BYTE CHARACTER CODE TABLE 1570

G ALARM LIST 1576

H WARNING LIST 1664

I PC TOOL FOR MEMORY CARD PROGRAM OPERATION/ EDITING 1679
I.1 PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING ... 1679
I.1.1 Usage Notes 1679
I.1.2 List of Functions of PC Tool 1679
I.1.3 Explanation of Operations 1680
I.2 NAMING RULES 1688
I.2.1 Naming Rules of Program File 1688
I.2.2 Naming Rules of Folder 1688
I.3 RULES OF CHARACTERS IN PROGRAM FILE 1689
I.3.1 Usable Characters in Program File 1689
I.4 ERROR MESSAGE AND NOTE 1690
I.4.1 List of Error Message 1690
I.4.2 Note 1691

J ISO/ASCII CODE CONVERSION TOOL 1692

K LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE PROGRAM COMMAND 1695
K.1 LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF G CODE 1695
K.2 LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF M AND S CODE 1697

FRYER MACHINE SYSTEMS INC
Rob Zahersky

I. GENERAL

FRYER MACHINE SYSTEMS INC
Rob Zahensky

FRYER MACHINE SYSTEMS INC
Rob Zahensky

1 GENERAL

This manual consists of the following parts:

About this manual

I. GENERAL
Describes chapter organization, applicable models, related manuals, and notes for reading this manual.

II. PROGRAMMING
Describes each function: Format used to program functions in the NC language, explanations, and limitations.

III. OPERATION
Describes the manual operation and automatic operation of a machine, procedures for inputting and outputting data, and procedures for editing a program.

IV. MAINTENANCE
Describes procedures for daily maintenance and replacing batteries.

APPENDIX
Describes supplementary materials.

NOTE

1 This manual describes the functions common to the lathe system and machining center system. For the functions specific to the lathe system or machining center system, refer to the OPERATOR'S MANUAL (Lathe System) (B-64694EN-1) or the OPERATOR'S MANUAL (Machining Center System) (B-64694EN-2).

2 Some functions described in this manual may not be applied to some products. For detail, refer to the Descriptions manual (B-64692EN).

3 This manual does not detail the parameters not mentioned in the text. For details of those parameters, refer to the Parameter Manual (B-64700EN).
Parameters are used to set functions and operating conditions of a CNC machine tool, and frequently-used values in advance. Usually, the machine tool builder factory-sets parameters so that the user can use the machine tool easily.

4 This manual describes not only basic functions but also optional functions. Look up the options incorporated into your system in the manual written by the machine tool builder.

Applicable models

This manual describes the models indicated in the table below.
In the text, the abbreviations indicated below may be used.

Model name	Abbreviation		
FANUC Series 0*i*-TF Plus	0*i*-TF Plus	Series 0*i*-F Plus	Series 0*i*
FANUC Series 0*i*-MF Plus	0*i*-MF Plus		

NOTE

1 For an explanatory purpose, the following descriptions may be used according to the CNC model :
- 0i-TF Plus : Lathe system (T series)
- 0i-MF Plus : Machining center system (M series)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE
2 Some functions described in this manual may not be applied to some products. For details, refer to the Chapter, "LIST OF SPECIFICATION"(B-64692EN).

Special symbols

This manual uses the following symbols:

- M

Indicates a description or function that is valid only for the machine center system (M series) set as system control type.
The term "M series" used in the text means "machining center system type".

- T

Indicates a description or function that is valid only for the lathe system (T series) set as system control type.
The term "T series" in the text means "lathe system type".

-

Indicates the end of a description of a system control type.
When a system control type mark mentioned above is not followed by this mark, the description of the system control type is assumed to continue until the next item or paragraph begins. In this case, the next item or paragraph provides a description common to the control types.

- **IP**

Indicates a combination of axes such as X_ Y_ Z_
In the underlined position following each address, a numeric value such as a coordinate value is placed (used in PROGRAMMING.).

- **;**

Indicates the end of a block. It actually corresponds to the ISO code LF or EIA code CR.

Related manuals of Series 0*i*-F Plus

The following table lists the manuals related to Series 0*i*-F Plus. This manual is indicated by an asterisk (*).

Table 1 Related manuals

Manual name	Specification number	
DESCRIPTIONS	B-64692EN	
CONNECTION MANUAL (HARDWARE)	B-64693EN	
CONNECTION MANUAL (FUNCTION)	B-64693EN-1	
OPERATOR'S MANUAL (Common to Lathe System/Machining Center System)	B-64694EN	*
OPERATOR'S MANUAL (For Lathe System)	B-64694EN-1	
OPERATOR'S MANUAL (For Machining Center System)	B-64694EN-2	
MAINTENANCE MANUAL	B-64695EN	
PARAMETER MANUAL	B-64700EN	
Programming		
Macro Executor PROGRAMMING MANUAL	B-63943EN-2	
Macro Compiler PROGRAMMING MANUAL	B-66263EN	
C Language Executor PROGRAMMING MANUAL	B-63943EN-3	
PMC		
PMC PROGRAMMING MANUAL	B-64513EN	
Network		
PROFIBUS-DP Board CONNECTION MANUAL	B-63993EN	

FOXER MACHINE SYSTEMS INC
Rob Zahensky

Manual name	Specification number	
Industrial Ethernet CONNECTION MANUAL	B-64013EN	
Fast Ethernet / Fast Data Server OPERATOR'S MANUAL	B-64014EN	
DeviceNet Board CONNECTION MANUAL	B-64043EN	
FL-net Board CONNECTION MANUAL	B-64163EN	
CC-Link Board CONNECTION MANUAL	B-64463EN	
Operation guidance function		
MANUAL GUIDE *i* (Common to Lathe System/Machining Center System) OPERATOR'S MANUAL	B-63874EN	
MANUAL GUIDE *i* (For Machining Center System) OPERATOR'S MANUAL	B-63874EN-2	
MANUAL GUIDE *i* (Set-up Guidance Functions) OPERATOR'S MANUAL	B-63874EN-1	
MANUAL GUIDE 0*i* OPERATOR'S MANUAL	B-64434EN	
Dual Check Safety		
Dual Check Safety CONNECTION MANUAL	B-64483EN-2	

Related manuals of SERVO MOTOR αi/βi series

The following table lists the manuals related to SERVO MOTOR αi/βi series

Table 2 Related manuals

Manual name	Specification number
FANUC AC SERVO MOTOR αi-B series FANUC AC SERVO MOTOR αi series DESCRIPTIONS	B-65262EN
FANUC AC SPINDLE MOTOR αi-B / βi-B series DESCRIPTIONS	B-65452EN
FANUC AC SERVO MOTOR βi-B series FANUC AC SERVO MOTOR βi series DESCRIPTIONS	B-65302EN
FANUC SERVO AMPLIFIER αi-B series DESCRIPTIONS	B-65412EN
FANUC SERVO AMPLIFIER βi-B series DESCRIPTIONS	B-65422EN
FANUC AC SERVO MOTOR αi series FANUC AC SPINDLE MOTOR αi series FANUC SERVO AMPLIFIER αi series MAINTENANCE MANUAL	B-65285EN
FANUC AC SERVO MOTOR βi series FANUC AC SPINDLE MOTOR βi series FANUC SERVO AMPLIFIER βi series MAINTENANCE MANUAL	B-65325EN
FANUC AC SERVO MOTOR αi series FANUC AC SERVO MOTOR βi series FANUC LINEAR MOTOR LiS series FANUC SYNCHRONOUS BUILT-IN SERVO MOTOR DiS series PARAMETER MANUAL	B-65270EN
FANUC AC SPINDLE MOTOR αi/βi series, BUILT-IN SPINDLE MOTOR Bi series PARAMETER MANUAL	B-65280EN

The above servo motors and the corresponding spindles can be connected to the CNC covered in this manual.

This manual mainly assumes that the FANUC SERVO MOTOR αi series of servo motor is used. For servo motor and spindle information, refer to the manuals for the servo motor and spindle that are actually connected.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

1.1 NOTES ON READING THIS MANUAL

⚠ **CAUTION**

1 The function of an CNC machine tool system depends not only on the CNC, but on the combination of the machine tool, its magnetic cabinet, the servo system, the CNC, the operator's panels, etc. It is too difficult to describe the function, programming, and operation relating to all combinations. This manual generally describes these from the stand-point of the CNC. So, for details on a particular CNC machine tool, refer to the manual issued by the machine tool builder, which should take precedence over this manual.

2 In the header field of each page of this manual, a chapter title is indicated so that the reader can reference necessary information easily.
By finding a desired title first, the reader can reference necessary parts only.

3 This manual describes as many reasonable variations in equipment usage as possible. It cannot address every combination of features, options and commands that should not be attempted.
If a particular combination of operations is not described, it should not be attempted.

1.2 NOTES ON VARIOUS KINDS OF DATA

⚠ **CAUTION**

Machining programs, parameters, offset data, etc. are stored in the CNC unit internal non-volatile memory. In general, these contents are not lost by the switching ON/OFF of the power. However, it is possible that a state can occur where precious data stored in the non-volatile memory has to be deleted, because of deletions from a maloperation, or by a failure restoration. In order to restore rapidly when this kind of mishap occurs, it is recommended that you create a copy of the various kinds of data beforehand.
The number of times to write machining programs to the non-volatile memory is limited.
You must use "High-speed program management" when registration and the deletion of the machining programs are frequently repeated in such case that the machining programs are automatically downloaded from a personal computer at each machining.
In "High-speed program management", the program is not saved to the non-volatile memory at registration, modification, or deletion of programs.

FRYER MACHINE SYSTEMS INC
Bob Zamorsky

II. PROGRAMMING

FRYER MACHINE SYSTEMS INC
Rob Zahensky

FRYER MACHINE SYSTEMS INC
Rob Zahensky

1 GENERAL

1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE-INTERPOLATION

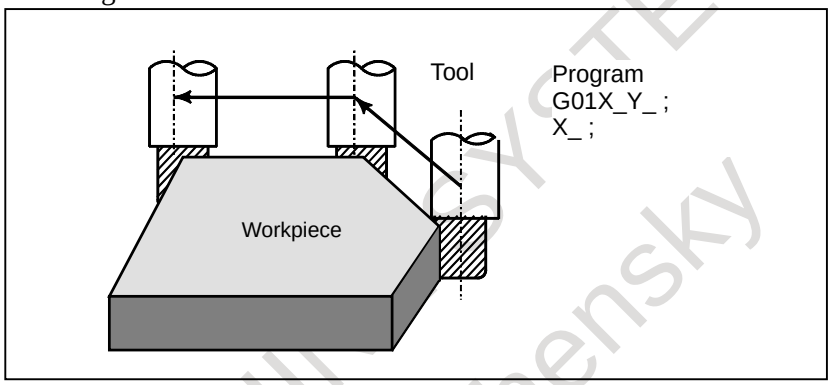
The tool moves along straight lines and arcs constituting the workpiece parts figure (See Chapter, “INTERPOLATION FUNCTIONS”).

Explanation

The function of moving the tool along straight lines and arcs is called the interpolation.

- Tool movement along a straight line

- For milling machining



- For lathe cutting

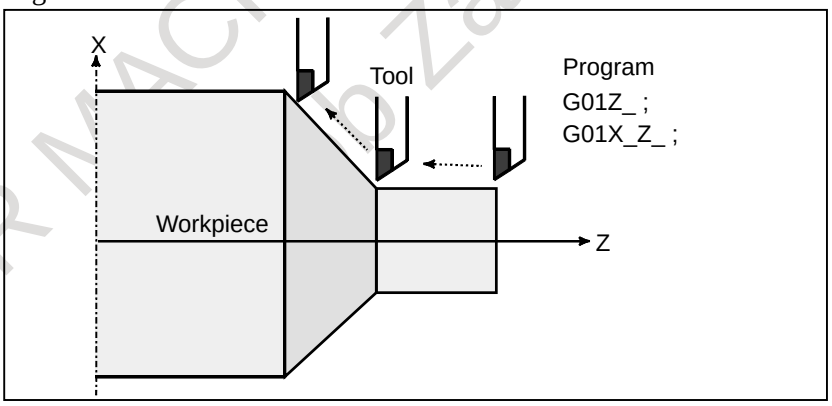
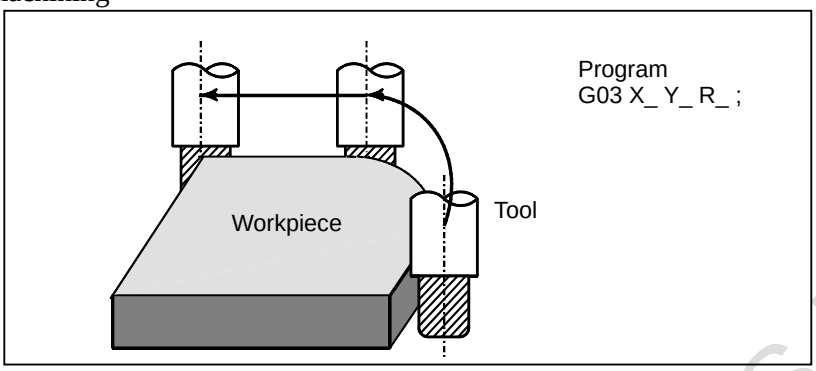


Fig. 1.1 (a) Tool movement along a straight line

FRYER MACHINE SYSTEMS INC
by Zahensky

- Tool movement along an arc

- For milling machining



- For lathe cutting

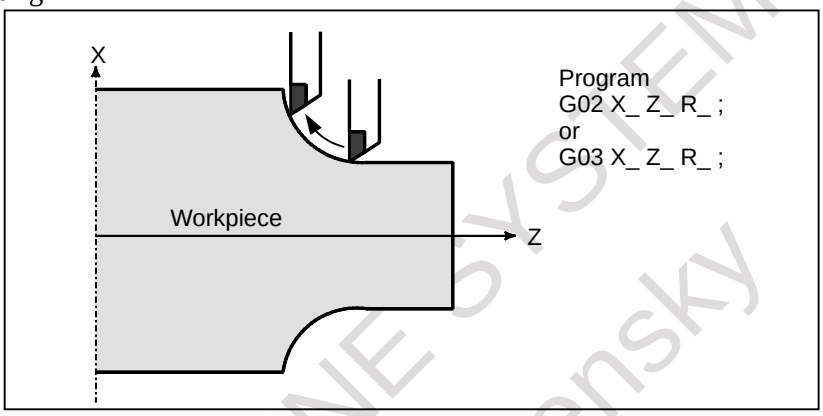


Fig. 1.1 (b) Tool movement along an arc

The term interpolation refers to an operation in which the tool moves along a straight line or arc in the way described above.
Symbols of the programmed commands G01, G02, G03,... are called the preparatory function and specify the type of interpolation conducted in the control unit.

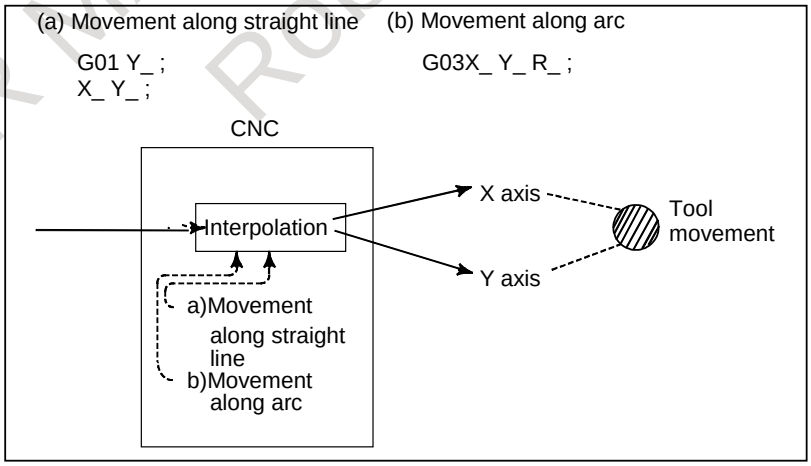


Fig. 1.1 (c) Interpolation function

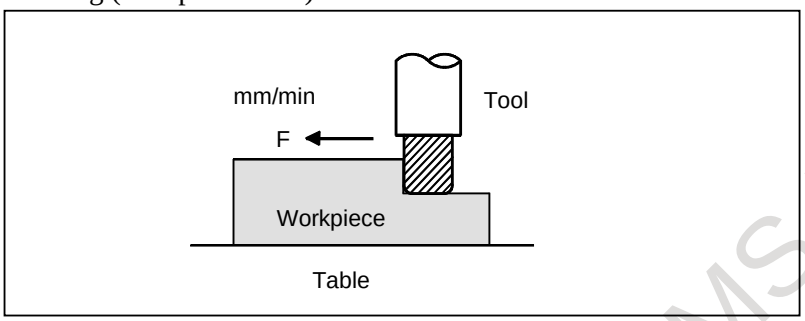
NOTE
Some machines move tables instead of tools but this manual assumes that tools are moved against workpieces.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

1.2 FEED-FEED FUNCTION

Movement of the tool at a specified speed for cutting a workpiece is called the feed.

- For milling machining (feed per minute)



- For lathe cutting (feed per revolution)

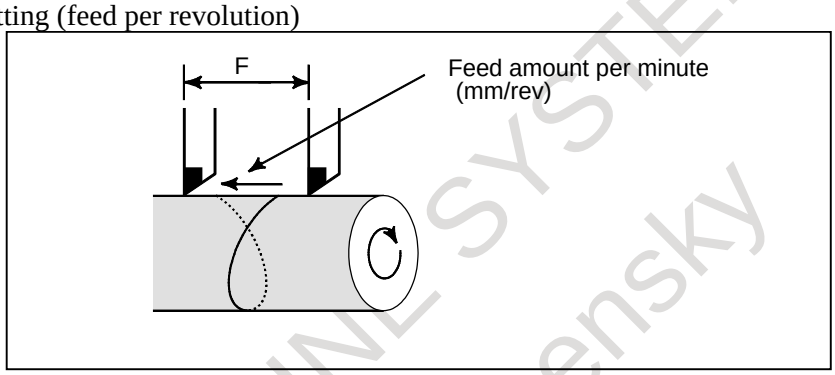


Fig.1.2 Feed function

For example, to feed the tool at a rate of 150 mm/min (feed per minute) or 150 mm/rev (feed per revolution), specify the following in the program:
F150.0
The function of deciding the feed rate is called the feed function (See Chapter, “FEED FUNCTIONS”).

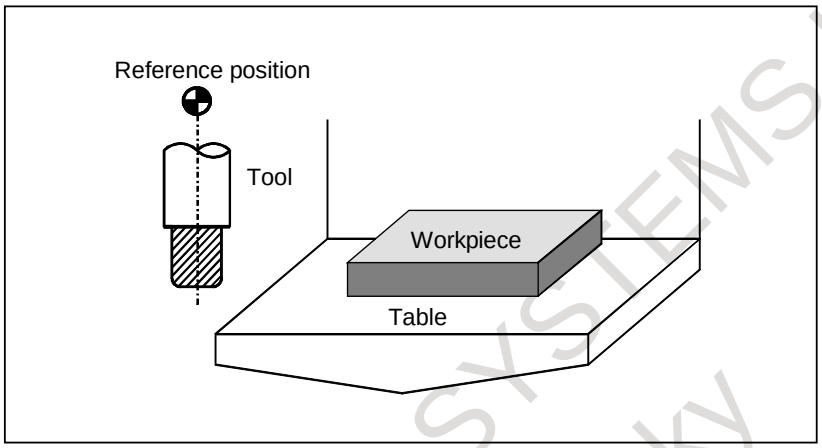
FRYER MACHINE SYSTEMS INC
Rob Zarnensky

1.3 PART DRAWING AND TOOL MOVEMENT

1.3.1 Reference Position (Machine-specific Position)

A CNC machine tool is provided with a fixed position. Normally, tool change and programming of absolute zero point as described later are performed at this position. This position is called the reference position.

- For milling machining



- For lathe cutting

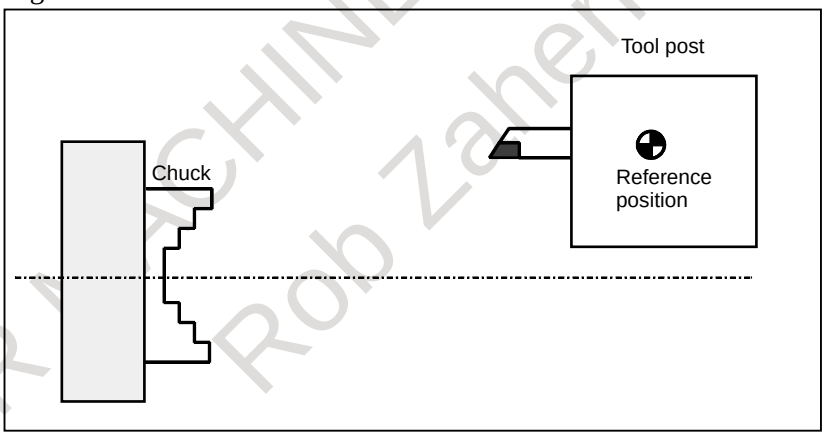


Fig.1.3.1 Reference position

Explanation

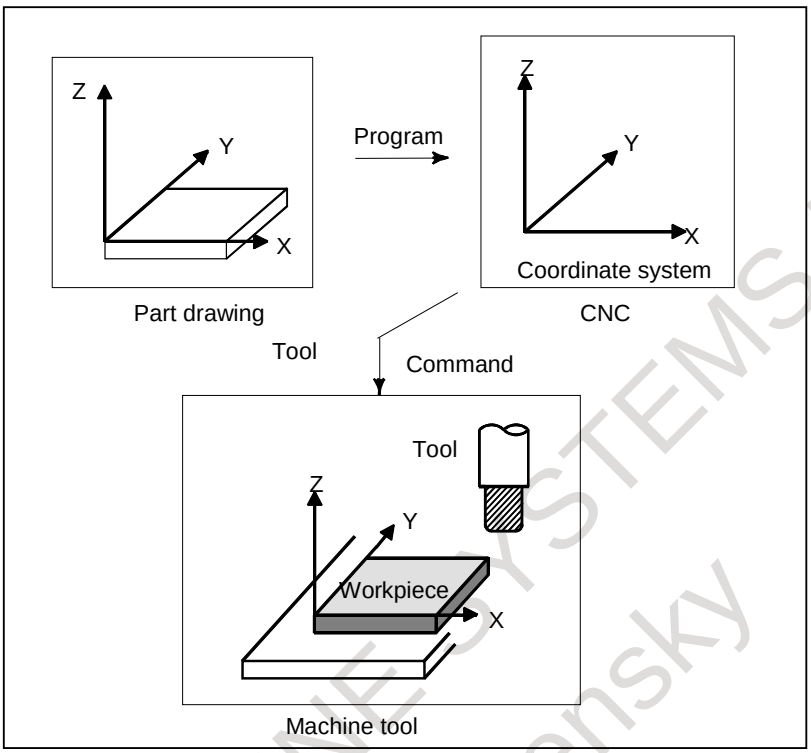
The tool can be moved to the reference position in two ways:

1. Manual reference position return (See Section, “MANUAL REFERENCE POSITION RETURN”)
   Reference position return is performed by manual button operation.
2. Automatic reference position return (See Section, “REFERENCE POSITION RETURN”)
   In general, manual reference position return is performed first after the power is turned on. In order to move the tool to the reference position for tool change thereafter, the function of automatic reference position return is used.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

1.3.2 Coordinate System on Part Drawing and Coordinate System Specified by CNC - Coordinate System

- For milling machining



- For lathe cutting

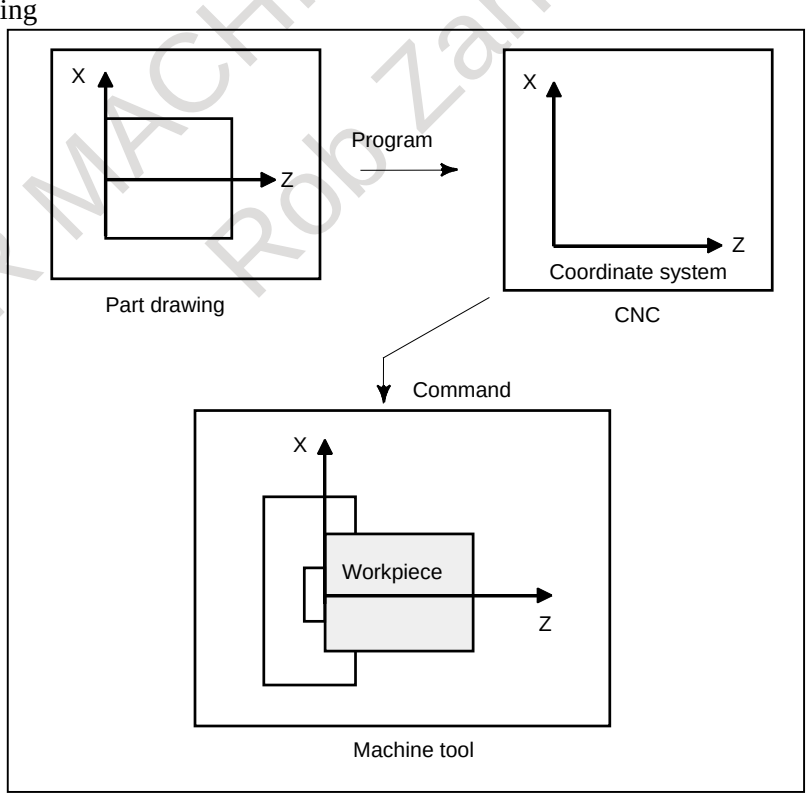


Fig. 1.3.2 (a) Coordinate system

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

- Coordinate system

The following two coordinate systems are specified at different locations:
(See Chapter, “COORDINATE SYSTEM”)

1 Coordinate system on part drawing
The coordinate system is written on the part drawing. As the program data, the coordinate values on this coordinate system are used.

2. Coordinate system specified by the CNC
The coordinate system is prepared on the actual machine tool table. This can be achieved by programming the distance from the current position of the tool to the zero point of the coordinate system to be set.

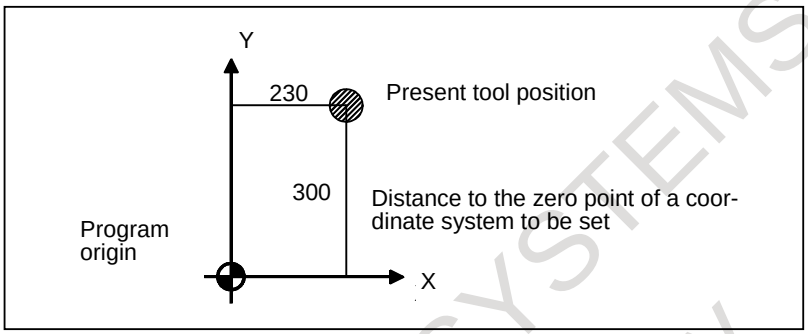
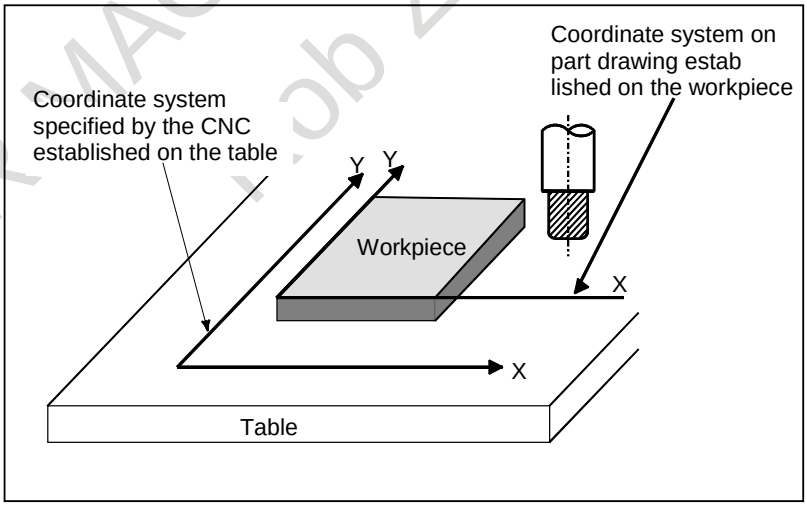


Fig. 1.3.2 (b) Coordinate system specified by the CNC

Concrete programming methods for setting coordinate systems specified by the CNC are explained in Chapter, “COORDINATE SYSTEM”

The positional relation between these two coordinate systems is determined when a workpiece is set on the table.

- For milling machining



FRYER MACHINE SYSTEMS INC
Rob Zaharensky

- For lathe cutting

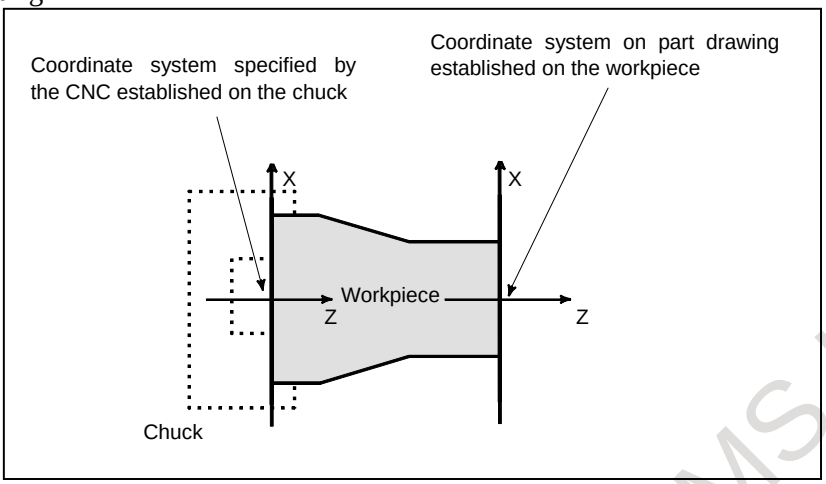


Fig. 1.3.2 (c) Coordinate system specified by CNC and coordinate system on part drawing

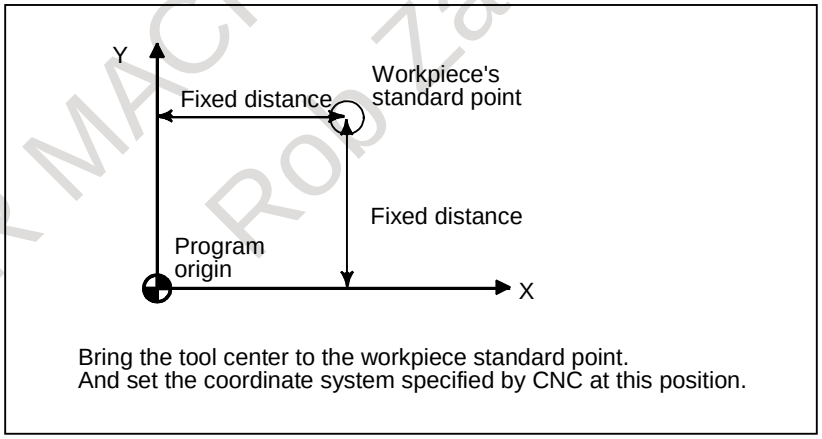
The tool moves on the coordinate system specified by the CNC in accordance with the command program generated with respect to the coordinate system on the part drawing, and cuts a workpiece into a shape on the drawing.
Therefore, in order to correctly cut the workpiece as specified on the drawing, the two coordinate systems must be set at the same position.

- Methods of setting the two coordinate systems in the same position

M

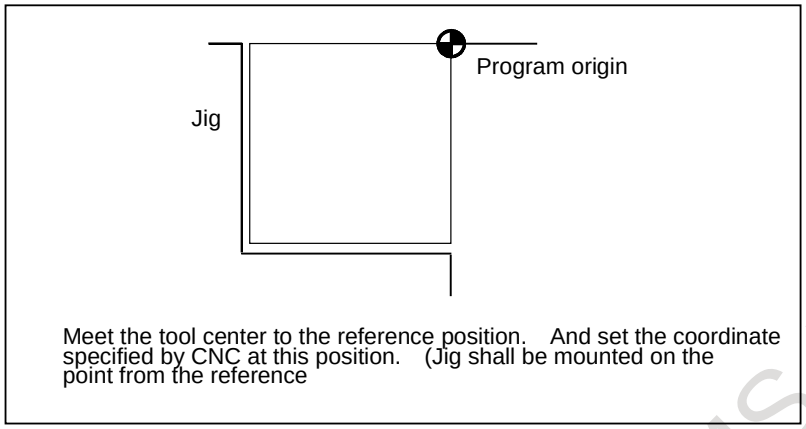
To set the two coordinate systems at the same position, simple methods shall be used according to workpiece shape, the number of machinings.

1. Using a standard plane and point of the workpiece.

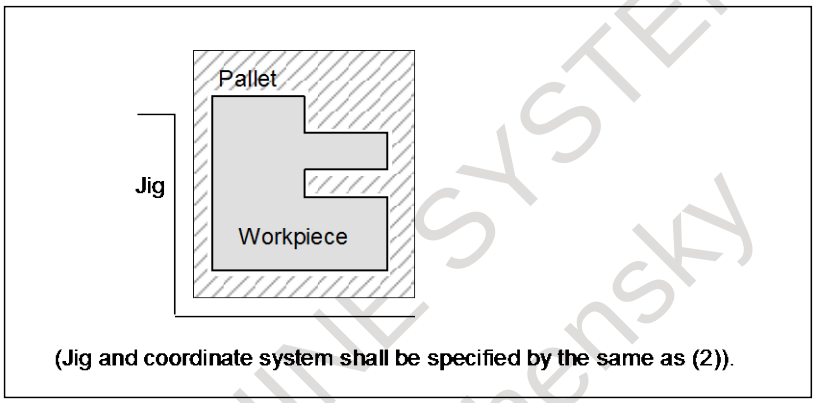


FRYER MACHINE SYSTEMS INC
Rob Zahensky

2. Mounting a workpiece directly against the jig



3. Mounting a workpiece on a pallet, then mounting the workpiece and pallet on the jig

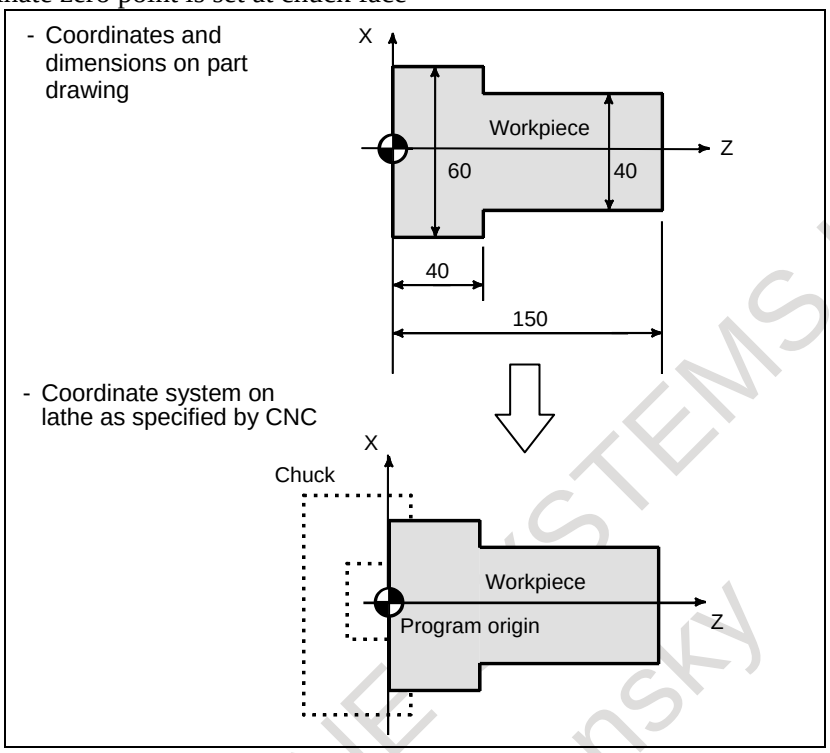


FRYER MACHINE SYSTEMS INC
Rob Zahensky

T

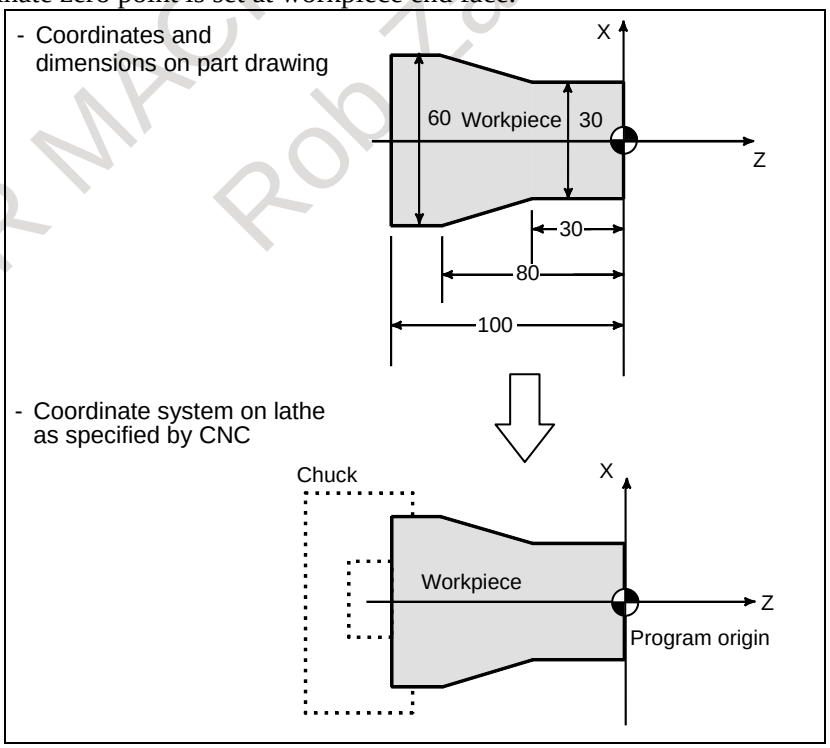
The following method is usually used to define two coordinate systems at the same location.

1 When coordinate zero point is set at chuck face



When the coordinate system on the part drawing and the coordinate system specified by the CNC are set at the same position, the program origin can be set on the chuck face.

2. When coordinate zero point is set at workpiece end face.



When the coordinate system on the part drawing and the coordinate system specified by the CNC are set at the same position, the program origin can be set on the end face of the workpiece.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

1.3.3 How to Indicate Command Dimensions for Moving the Tool (Absolute and Incremental Programming)

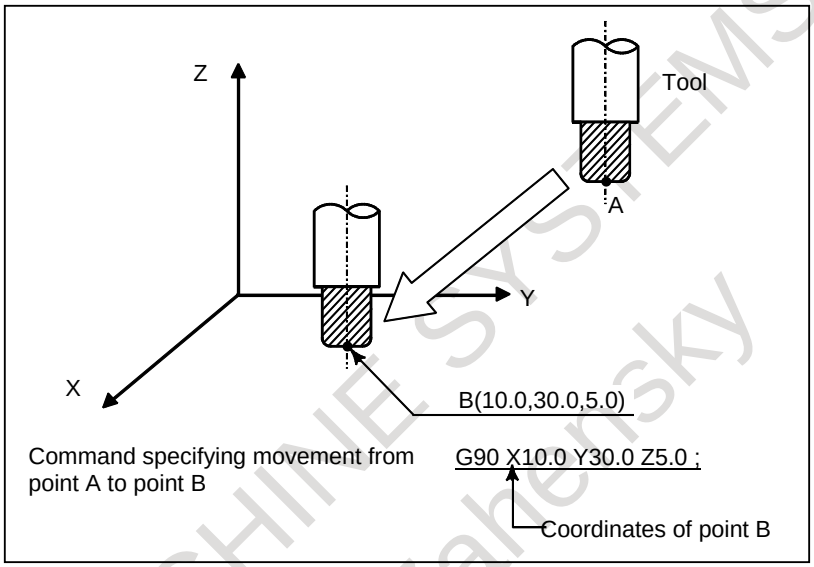
Explanation

Command for moving the tool can be indicated by absolute command or incremental command (See Section, “ABSOLUTE AND INCREMENTAL PROGRAMMING”).

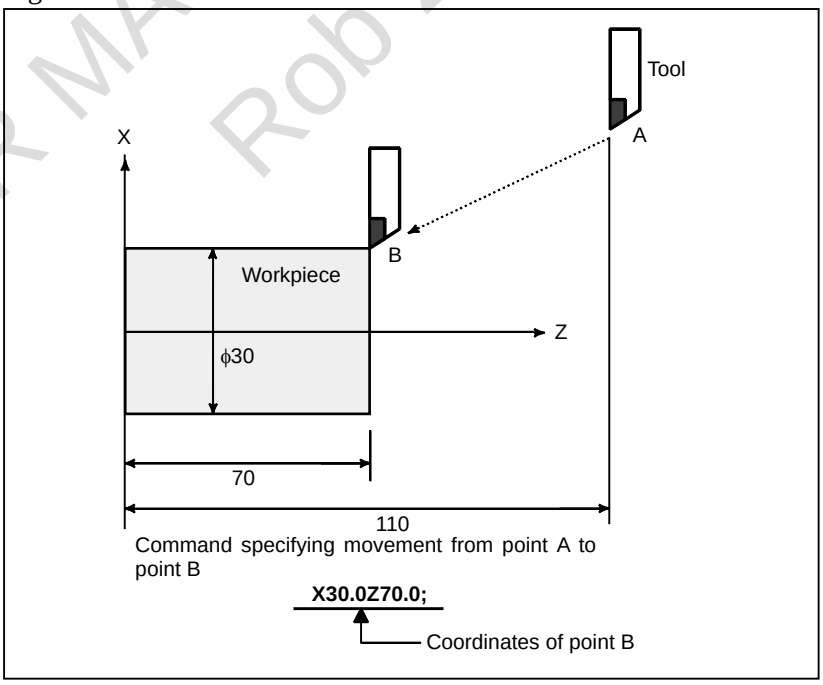
- Absolute command

The tool moves to a point at "the distance from zero point of the coordinate system" that is to the position of the coordinate values.

- For milling machining



- For lathe cutting

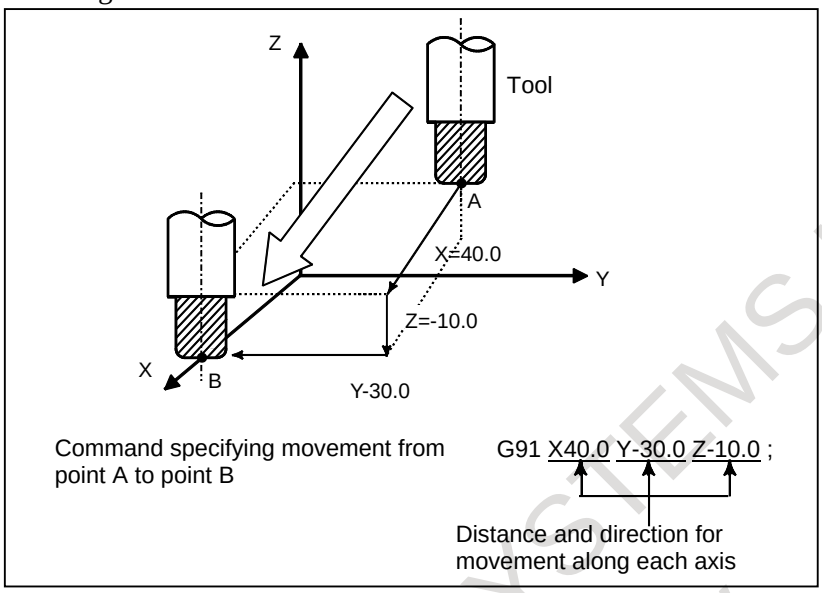


FRYER MACHINE SYSTEMS INC
Rob Zahensky

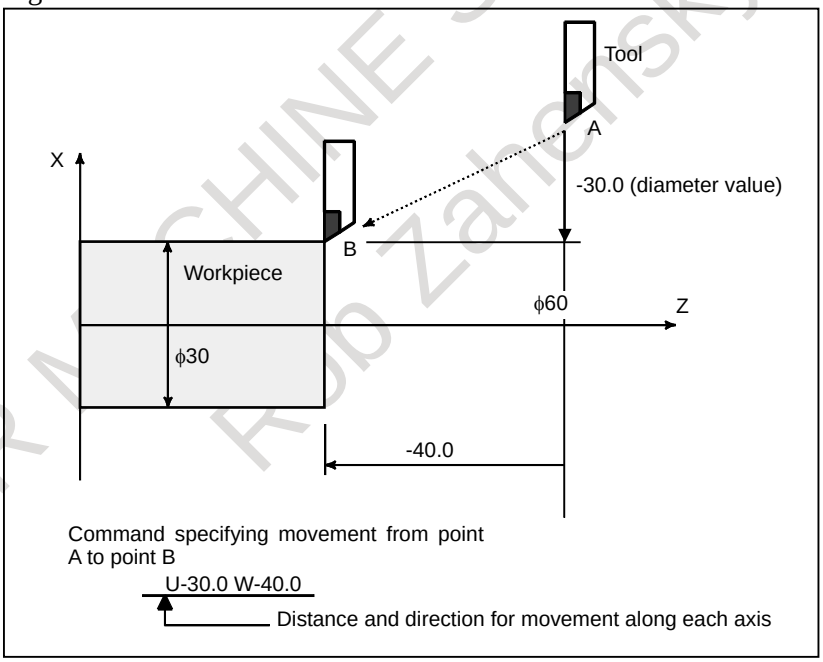
- Incremental command

Specify the distance from the previous tool position to the next tool position.

- For milling machining



- For lathe cutting

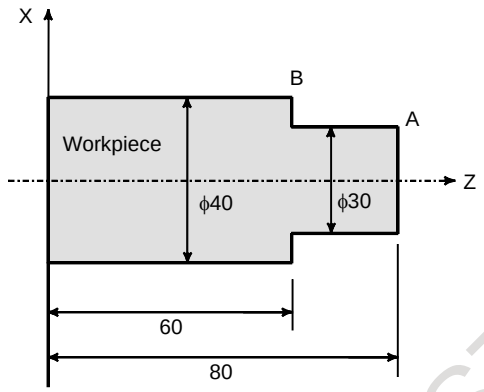


FRYER MACHINE SYSTEMS INC
Rob Zahorsky

- Diameter programming / radius programming

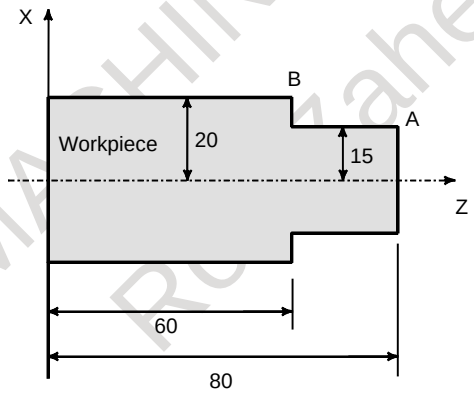
Dimensions of the X-axis can be set in diameter or in radius. Which programming is used is determined according to the setting of bit 3 (DIA) of parameter No. 1006.

1. Diameter programming
   In diameter programming, specify the diameter value indicated on the drawing as the value of the X-axis.



Coordinate values of points A and B **A(30.0, 80.0), B(40.0, 60.0)**

2. Radius programming
   In radius programming, specify the distance from the center of the workpiece, i.e. the radius value as the value of the X-axis.



Coordinate values of points A and B **A(15.0, 80.0), B(20.0, 60.0)**

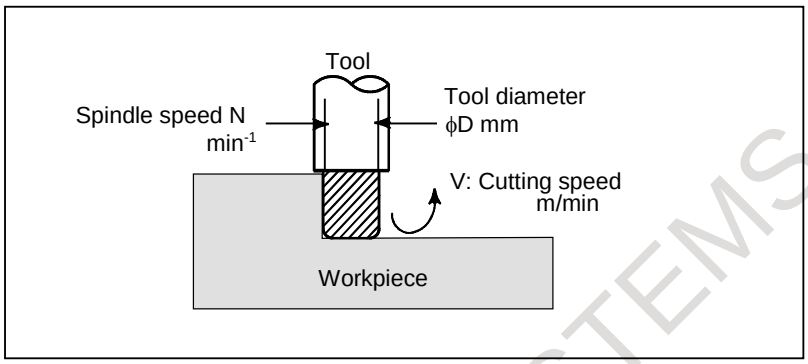
FRYER MACHINE SYSTEMS INC
Roman Zahensky

1.4 CUTTING SPEED - SPINDLE FUNCTION

Explanation

The speed of the tool with respect to the workpiece when the workpiece is cut is called the cutting speed. As for the CNC, the cutting speed can be specified by the spindle speed in min^{-1} unit.

- For milling machining

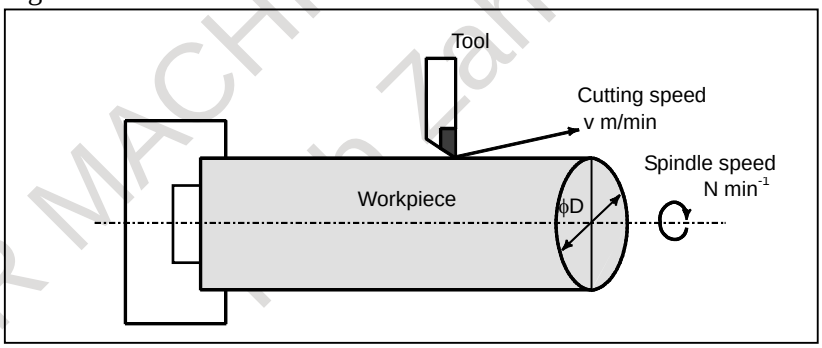


<When a workpiece should be machined with a tool 100 mm in diameter at a cutting speed of 80 m/min.>
The spindle speed is approximately 250 min^{-1}, which is obtained from N=1000v/πD. Hence the following command is required:

S250;

Commands related to the spindle speed are called the spindle speed function (See Chapter, "SPINDLE SPEED FUNCTION (S FUNCTION)").

- For lathe cutting



<When a workpiece 200 mm in diameter should be machined at a cutting speed of 300 m/min.>
The spindle speed is approximately 478 min^{-1}, which is obtained from N=1000v/πD. Hence the following command is required:

S478;

Where v is the cutting speed and D is the diameter of the workpiece.
Commands related to the spindle speed are called the spindle speed function (See Chapter, "SPINDLE SPEED FUNCTION (S FUNCTION)").
The cutting speed v (m/min) can also be specified directly by the speed value. Even when the workpiece diameter is changed, the CNC changes the spindle speed so that the cutting speed remains constant.
This function is called the constant surface speed control function (See Section, "CONSTANT SURFACE SPEED CONTROL (G96, G97)").

FOYER MACHINE SYSTEMS INC
Yuri Zahensky

1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING - TOOL FUNCTION

For each of various types of machining (such as drilling, tapping, boring, and milling for milling machining, or rough machining, semifinish machining, finish machining, threading, and grooving for lathe cutting), a necessary tool is to be selected. When a number is assigned to each tool and the number is specified in the program, the corresponding tool is selected.

Examples

M

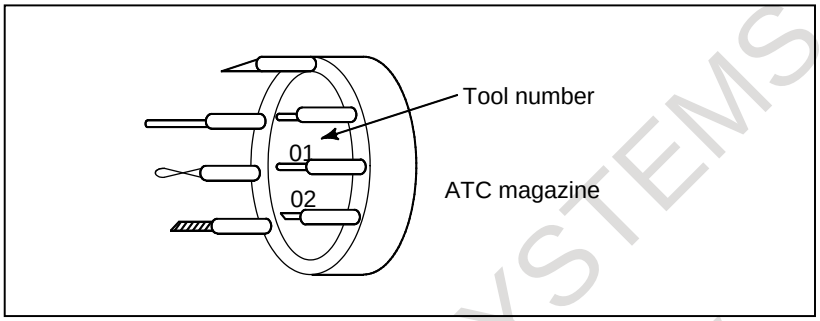


Fig. 1.5 (a) Tool used for various machining

<When No.01 is assigned to a drilling tool>
When the tool is stored at location 01 in the ATC magazine, the tool can be selected by specifying T01. This is called the tool function (See Chapter, “TOOL FUNCTION (T FUNCTION)”).

T

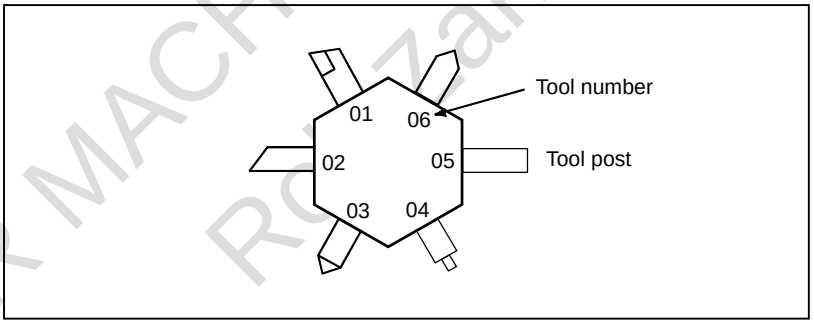


Fig. 1.5 (b) Tool used for various machining

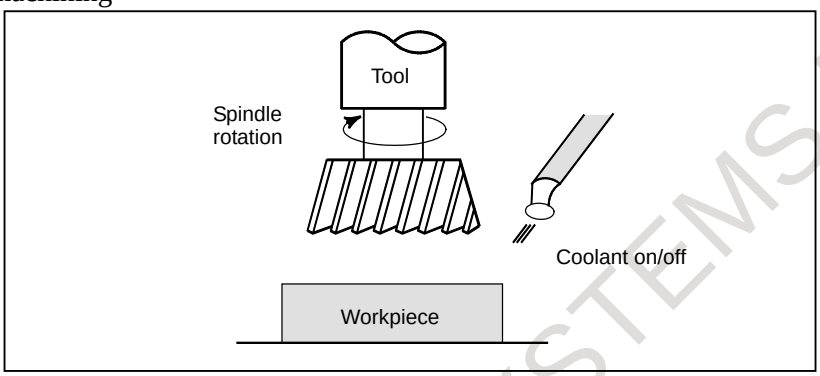
<When No.01 is assigned to a roughing tool>
When the tool is stored at location 01 of the tool post, the tool can be selected by specifying T0101. This is called the tool function (See Chapter, “TOOL FUNCTION (T FUNCTION)”).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

1.6 COMMAND FOR MACHINE OPERATIONS - AUXILIARY FUNCTION

When a workpiece is actually machined with a tool, the spindle is rotated, coolant is supplied, and the chuck is opened/closed. So, control needs to be exercised on the spindle motor of the machine, coolant valve on/off operation, and chuck open/close operation.

- For milling machining



- For lathe cutting

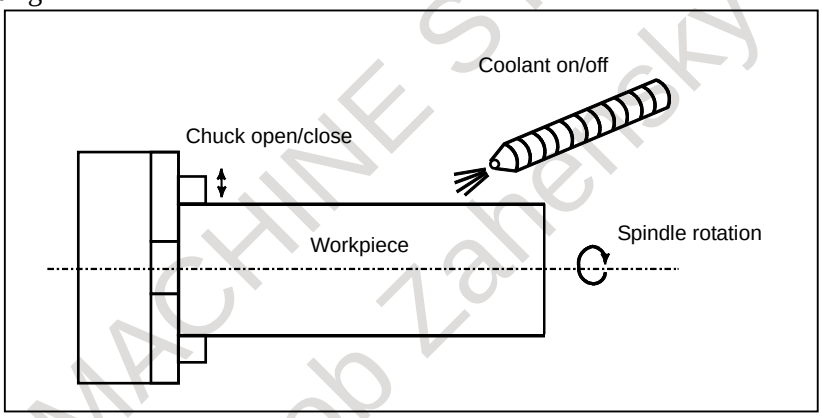


Fig. 1.6 Auxiliary function

The function of specifying the on-off operations of the components of the machine is called the auxiliary function. In general, the function is specified by an M code (See Chapter, “AUXILIARY FUNCTION”). For example, when M03 is specified, the spindle is rotated clockwise at the specified spindle speed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

1.7 PROGRAM CONFIGURATION

A group of commands given to the CNC for operating the machine is called the program. By specifying the commands, the tool is moved along a straight line or an arc, or the spindle motor is turned on and off. In the program, specify the commands in the sequence of actual tool movements.

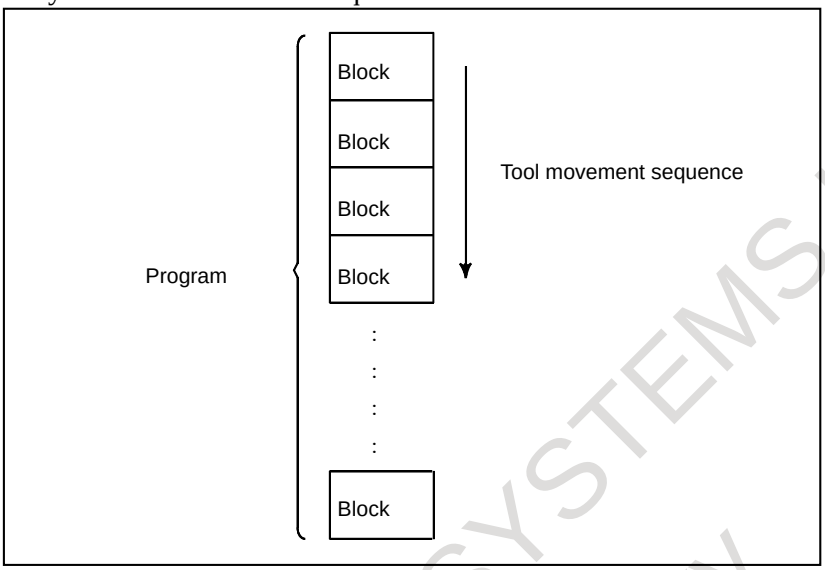


Fig. 1.7 (a) Program configuration

A group of commands at each step of the sequence is called the block. The program consists of a group of blocks for a series of machining. The number for discriminating each block is called the sequence number, and the number for discriminating each program is called the program number (See Chapter, “PROGRAM CONFIGURATION”).

Explanation

The block and the program have the following configurations.

- Block

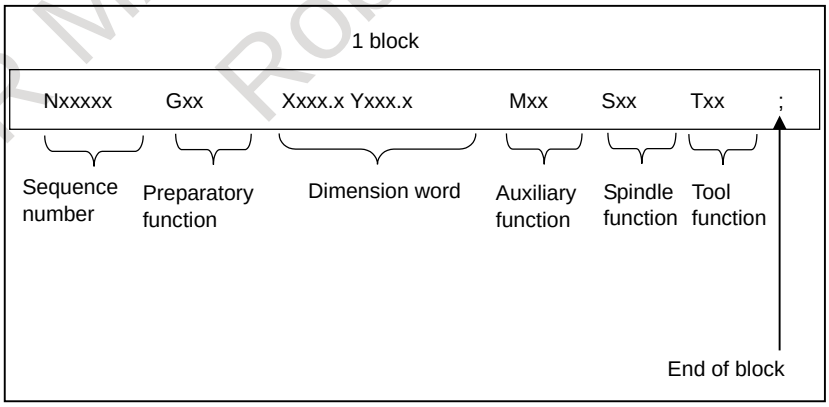


Fig. 1.7 (b) Block configuration

A block starts with a sequence number to identify the block and ends with an end-of-block code.
This manual indicates the end-of-block code by; (LF in the ISO code and CR in the EIA code).
The contents of the dimension word depend on the preparatory function. In this manual, the portion of the dimension word may be represent as IP_.

FRYER MACHINE SYSTEMS INC
Rob Zaherski

- Program

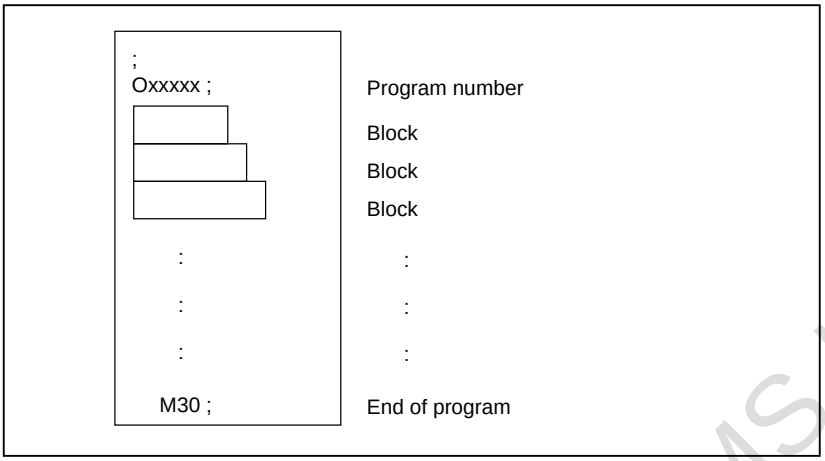


Fig. 1.7 (c) Program configuration

Normally, a program number is specified after the end-of-block (;) code at the beginning of the program, and a program end code (M02 or M30) is specified at the end of the program.

- Main program and subprogram

When machining of the same pattern appears at many portions of a program, a program for the pattern is created. This is called the subprogram. On the other hand, the original program is called the main program. When a subprogram execution command appears during execution of the main program, commands of the subprogram are executed. When execution of the subprogram is finished, the sequence returns to the main program.

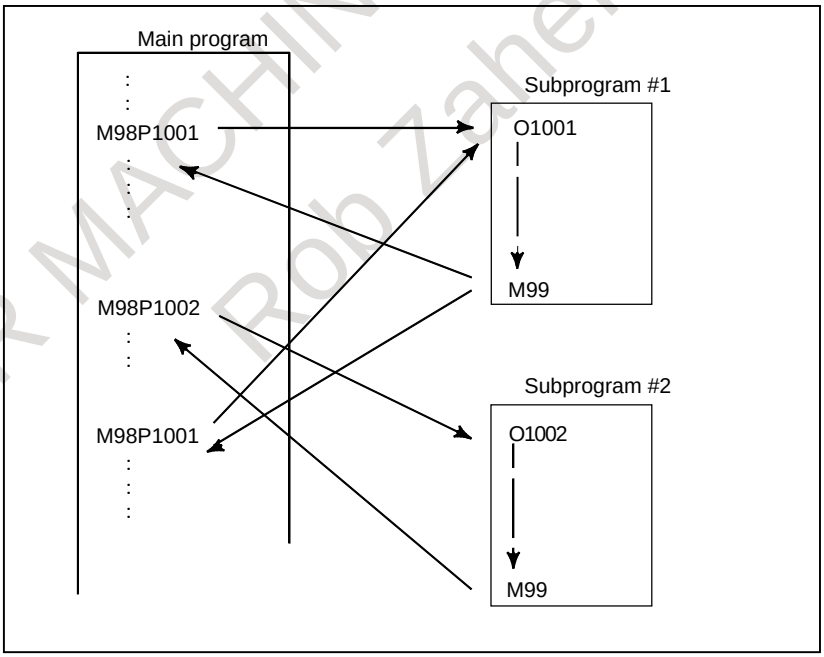
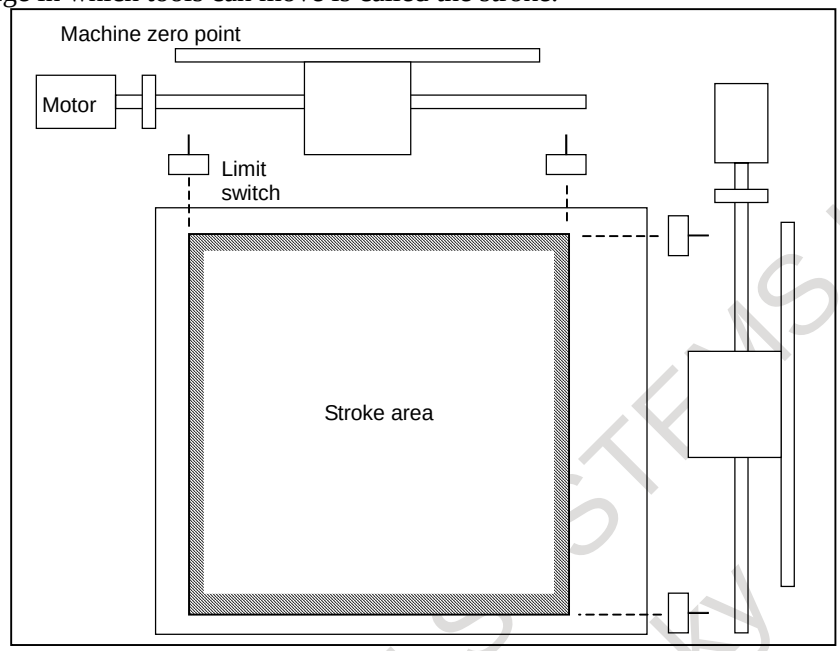


Fig. 1.7 (d) Subprogram execution

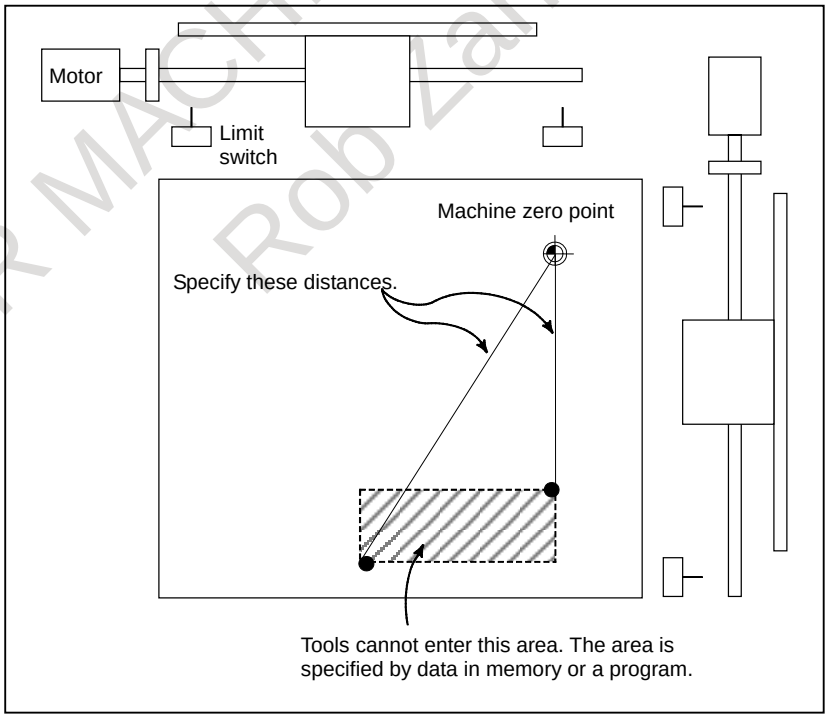
FRYER MACHINE SYSTEMS INC
Rob Zahensky

1.8 TOOL MOVEMENT RANGE - STROKE

Limit switches are installed at the ends of each axis on the machine to prevent tools from moving beyond the ends. The range in which tools can move is called the stroke.



Besides strokes defined with limit switches, the operator can define an area which the tool cannot enter using a program or data in memory. This function is called stroke check (See Section, “STORED STROKE CHECK”).



FRYER MACHINE SYSTEMS INC
Rob Zaherisky

2 CONTROLLED AXES

2.1 NUMBER OF CONTROLLED AXES

Explanation

The number of controlled axes used with this NC system depends on the model and system control type as indicated below.

Item	M			T		
	Type 0, Type 1		Type 3 Type 5	Type 0, Type 1		Type 3 Type 5
	1-path system	2-path system		1-path system	2-path system	
Maximum total number of control axes (total/each path)	9 axes	11 axes/ 9 axes	6 axes	9 axes	12 axes/ 9 axes	6 axes
Maximum number of feed axes (total/each path)	7 axes	9 axes/ 7 axes	5 axes	7 axes	10 axes/ 7 axes	5 axes
Maximum number of spindle axes (total/each path)	2 axes	4 axes/ 3 axes	2 axes	3 axes	6 axes/ 4 axes	2 axes
Simultaneously controlled axes (each path)	4 axes in the time	4 axes in the time	4 axes in the time	4 axes in the time	4 axes in the time	4 axes in the time
PMC axis control (total)	4 axes in the time	8 axes in the time	4 axes in the time	4 axes in the time	8 axes in the time	4 axes in the time

NOTE

1 The maximum number of controlled axes that can be used is limited depending on the option configuration. Refer to the manual provided by the machine tool builder for details.

2 The number of simultaneously controllable axes for manual operation (jog feed, manual reference position return, or manual rapid traverse) is 1 or 3 (1 when bit 0 (JAX) of parameter No. 1002 is set to 0 and 3 when it is set to 1).

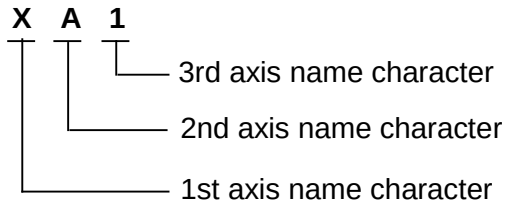
2.2 NAMES OF AXES

Explanation

The move axes of machine tools are assigned names. These names are referred to as addresses or axis names. Axis names are determined according to the machine tool. The naming rules comply with standards such as the ISO standards.

With complex machines, one character would become insufficient for representing axis names. So, up to three characters can be used for axis names. A move axis may be named "X", "X1", or "XA1". The first character of the three characters is called the first axis name character, the second character is called the second axis name character, and third character is called the third axis name character.

Example)



FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE
1 Axis names are predetermined according to the machine used. Refer to the manual supplied by the machine tool builder.
2 Since many ordinary machines use one character to represent each address, one-character addresses are used in the description in this manual.

2.3 INCREMENT SYSTEM

Explanation

The increment system consists of the least input increment (for input) and least command increment (for output). The least input increment is the least increment for programming the travel distance. The least command increment is the least increment for moving the tool on the machine. Both increments are represented in mm, inch, or deg.
Three types of increment systems are available as indicated in Table 2.3. For each axis, an increment system can be set using a bit from bit 0 and bit 1 (ISA, ISC) of parameter No. 1013.

Table 2.3 Increment system

Name of increment system	Least input increment		Least command increment	
IS-A	0.01	mm	0.01	mm
	0.001	inch	0.001	inch
	0.01	deg	0.01	deg
IS-B	0.001	mm	0.001	mm
	0.0001	inch	0.0001	inch
	0.001	deg	0.001	deg
IS-C	0.0001	mm	0.0001	mm
	0.00001	inch	0.00001	inch
	0.0001	deg	0.0001	deg

The least command increment is either metric or inch depending on the machine tool. Set metric or inch to the bit 0 (INM) of parameter No.1001.
For selection between metric and inch for the least input increment, G code (G20 or G21) or a setting parameter selects it.
Combined use of the inch system and the metric system is not allowed. There are functions that cannot be used between axes with different unit systems (circular interpolation, cutter compensation, etc.). For the increment system, see the machine tool builder's manual.

NOTE
1 The unit (mm or inch) in the table is used for indicating a diameter value for diameter programming (when bit 3 (DIA) of parameter No. 1006 is set to 1) or a radius value for radius programming.
2 Some increment systems are unavailable depending on the model. For details, refer to “Descriptions” (B-64692EN).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

2.4 MAXIMUM STROKE

Explanation

The maximum stroke controlled by this CNC is shown in the Table 2.4(a):
Maximum stroke = Least command increment × 999999999 (99999999 for IS-A)
Commands that exceed the maximum stroke are not permitted.

Table 2.4 (a) Maximum strokes

Name of increment system	Least input increment		Maximum stroke	
IS-A	0.01	mm	±999999.99	mm
	0.001	inch	±99999.999	inch
	0.01	deg	±999999.99	deg
IS-B	0.001	mm	±999999.999	mm
	0.0001	inch	±99999.9999	inch
	0.001	deg	±999999.999	deg
IS-C	0.0001	mm	±99999.9999	mm
	0.00001	inch	±9999.99999	inch
	0.0001	deg	±99999.9999	deg

NOTE

1 The actual stroke depends on the machine tool.
2 The unit (mm or inch) in the table is used for indicating a diameter value for diameter programming (when bit 3 (DIA) of parameter No. 1006 is set to 1) or a radius value for radius programming.
3 Some increment systems are unavailable depending on the model. For details, refer to "Descriptions" (B-64692EN).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

2.5 MAXIMUM COMMAND VALUES

The basic addresses and specifiable value ranges are indicated below (Table 2.5 (a)). Note, however, that the information below represents restrictions imposed by the CNC, and is totally irrelevant to the restrictions imposed by each machine.

Table 2.5 (a) Major addresses and ranges of command values

Function		Address	Input in mm	Input in inch
Program number		O (*1)	1 to 99999999	1 to 99999999
Sequence number		N	1 to 99999999	1 to 99999999
Preparatory function		G	0 to 99.9	0 to 99.9
Dimension word	Increment system IS-A	X,Y,Z,U,V,W,A,B,C,I, J,K,R (*2)	±999999.99 mm ±999999.99 deg.	±99999.999 inch (*3) ±999999.99 deg.
	Increment system IS-B		±999999.999 mm ±999999.999 deg.	±99999.9999 inch (*3) ±999999.999 deg.
	Increment system IS-C		±99999.9999 mm ±99999.9999 deg.	±9999.99999 inch (*3) ±99999.9999 deg.
Feed per minute	Increment system IS-A	F	0.01 to 999000.00mm/min	0.001 to 96000.000inch/min
	Increment system IS-B		0.001 to 999000.000mm/min	0.0001 to 9600.0000inch/min
	Increment system IS-C		0.0001 to 99999.9999mm/min	0.00001 to 4000.00000inch/min
Feed per revolution		F	0.001 to 50000mm/rev	0.0001 to 50.0000inch/rev
Spindle speed function		S (*4)	0 to 99999999	0 to 99999999
Tool function		T (*4)	0 to 99999999	0 to 99999999
Auxiliary function		M (*4)	0 to 99999999	0 to 99999999
		B (*4)	0 to 99999999	0 to 99999999
Offset number (M series only)		H, D	0 to 999	0 to 999
Dwell	Increment system IS-A	X, U (T series only)	0 to 999999.99 sec	0 to 999999.99 sec
	Increment system IS-B		0 to 99999.999 sec	0 to 99999.999 sec
	Increment system IS-C		0 to 9999.9999 sec	0 to 9999.9999 sec
Dwell		P	1 to 99999999	1 to 99999999
Designation of a program number		P	1 to 99999999	1 to 99999999
Number of sub program repetitions		L	1 to 99999999	1 to 99999999
		P	0 to 9999	0 to 9999

*1 In ISO code, the colon (:) can also be used as the address of a program number.

*2 When address I, J, K, or R is used to specify the radius for circular interpolation, the specifiable range is as follows (Table 2.5 (b)):

Table 2.5 (b) Specifiable range when address I, J, K, or R is used to specify the radius for circular interpolation

Increment system	Input in mm	Input in inch
IS-A	±999999999.99 mm	±99999999.999 inch
IS-B	±999999999.999 mm	±99999999.9999 inch
IS-C	±99999999.9999 mm	±9999999.99999 inch

*3 For inch input/millimeter machines, the maximum specifiable range of dimension words is as follows (Table 2.5 (c)):

FRYER MACHINE SYSTEMS INC Rob Zaremsky

Table 2.5 (c) Maximum specifiable range of dimension words

Increment system	Maximum specifiable range
IS-A	±39370.078 inch
IS-B	±39370.0787 inch
IS-C	±3937.00787 inch

*4 A maximum specifiable value for addresses M, S, T, and B is 99999999. Note that, however, values longer than the permissible number of digits set in parameters cannot be specified. The values and uses for some codes are limited by parameter setting. (For example, some M codes are not buffered.) For details, refer to “FANUC Series 0*i*- MODEL F Plus PARAMETER MANUCAL” (B-64700EN).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

3 PREPARATORY FUNCTION (G FUNCTION)

A number following address G determines the meaning of the command for the concerned block.
G codes are divided into the following two types.

Type	Meaning
One-shot G code	The G code is effective only in the block in which it is specified.
Modal G code	The G code is effective until another G code of the same group is specified.

(Example)

G01 and G00 are modal G codes.

```
G01 X_ ;
    Z_ ;  } G01 is effective in this range.
    X_ ;
G00 Z_ ;  } G00 is effective in this range.
    X_ ;
G01 X_ ;
  :
```

T

There are three G code systems in the lathe system: A, B, and C (Table 3.1(a)). Select a G code system using bits 6 (GSB) and 7 (GSC) of parameter No. 3401. Generally, OPERATOR'S MANUAL describes the use of G code system A, except when the described item can use only G code system B or C. In such cases, the use of G code system B or C is described.

Explanation

1. When the clear state (bit 6 (CLR) of parameter No. 3402) is set at power-up or reset, the modal G codes are placed in the states described below.
   (1) The modal G codes are placed in the states marked with ◩ as indicated in Table.
   (2) G20 and G21 remain unchanged when the clear state is set at power-up or reset.
   (3) Which status G22 or G23 at power on is set by bit 7 (G23) of parameter No. 3402. However, G22 and G23 remain unchanged when the clear state is set at reset.
   (4) The user can select G00 or G01 by setting bit 0 (G01) of parameter No. 3402.
   (5) The user can select G90 or G91 by setting bit 3 (G91) of parameter No. 3402.
   When G code system B or C is used in the lathe system, setting bit 3 (G91) of parameter No. 3402 determines which code, either G90 or G91, is effective.
   (6) In the machining center system, the user can select G17, G18, or G19 by setting bits 1 (G18) and 2 (G19) of parameter No. 3401.
2. G codes other than G10 and G11 are one-shot G codes.
3. When a G code not listed in the G code list is specified, or a G code that has no corresponding option is specified, alarm PS0010, "IMPROPER G-CODE" occurs.
4. Multiple G codes can be specified in the same block if each G code belongs to a different group. If multiple G codes that belong to the same group are specified in the same block, only the last G code specified is valid.
5. If a G code belonging to group 01 is specified in a canned cycle for drilling, the canned cycle for drilling is cancelled. This means that the same state set by specifying G80 is set. Note that the G codes in group 01 are not affected by a G code specifying a canned cycle for drilling.
6. G codes are indicated by group.
7. The group of G60 is switched according to the setting of bit 0 (MDL) of parameter No. 5431. (When the MDL bit is set to 0, the 00 group is selected. When the MDL bit is set to 1, the 01 group is selected.)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

T

8. When G code system A is used, absolute or incremental programming is specified not by a G code (G90/G91) but by an address word (X/U, Z/W, C/H, Y/V). Only the initial level is provided at the return point of the canned cycle for drilling..

3.1 G CODE LIST IN THE MACHINING CENTER SYSTEM

M

Table 3.1(a) G code list

G code	Group	Function	
G00	01	Positioning (rapid traverse)	
G01		Linear interpolation (cutting feed)	
G02		Circular interpolation CW or helical interpolation CW	
G03		Circular interpolation CCW or helical interpolation CCW	
G04	00	Dwell	
G04.1		G code preventing buffering	
G05		AI contour control (high-precision contour control compatible command)	
G05.1		AI contour control	
G05.4		HRV3 on/off	
G07.1	00	Cylindrical interpolation	
G08		AI contour control (advanced preview control compatible command)	
G09		Exact stop	
G10		Programmable data input	
G10.6		Tool retract and recover	
G11		Programmable data input mode cancel	
G15	17	Polar coordinates command cancel	
G16		Polar coordinates command	
G17	02	XpYp plane selection	Xp: X axis or its parallel axis
G18		ZpXp plane selection	Yp: Y axis or its parallel axis
G19		YpZp plane selection	Zp: Z axis or its parallel axis
G20 (G70)	06	Input in inch	
G21 (G71)		Input in mm	
G22	04	Stored stroke check function on	
G23		Stored stroke check function off	
G25	19	Spindle speed fluctuation detection off	
G26		Spindle speed fluctuation detection on	
G27	00	Reference position return check	
G28		Automatic return to reference position	
G28.2		In-position check disable reference position return	
G29		Movement from reference position	
G30		2nd, 3rd and 4th reference position return	
G30.2		In-position check disable 2nd, 3rd, or 4th reference position return	
G31		Skip function	
G31.8		EGB-axis skip	
G33	01	Threading	
G37	00	Automatic tool length measurement	
G38		Tool radius/tool nose radius compensation : preserve vector	
G39		Tool radius/tool nose radius compensation : corner circular interpolation	
G40	07	Tool radius/tool nose radius compensation : cancel	
G41		Tool radius/tool nose radius compensation : left	
G42		Tool radius/tool nose radius compensation : right	

FLYER MACHINE SYSTEMS INC
Rob Zahensky

Table 3.1(a) G code list

G code	Group	Function
G40.1	18	Normal direction control cancel mode
G41.1		Normal direction control on : left
G42.1		Normal direction control on : right
G43	08	Tool length compensation +
G44		Tool length compensation -
G43.7		Tool offset
G45	00	Tool offset : increase
G46		Tool offset : decrease
G47		Tool offset : double increase
G48		Tool offset : double decrease
G49 (G49.1)	08	Tool length compensation cancel
G50	11	Scaling cancel
G51		Scaling
G50.1	22	Programmable mirror image cancel
G51.1		Programmable mirror image
G50.4	00	Cancel synchronous control
G50.5		Cancel composite control
G50.6		Cancel superimposed control
G51.4		Start synchronous control
G51.5		Start composite control
G51.6		Start superimposed control
G52		Local coordinate system setting
G53		Machine coordinate system setting
G53.1		Tool axis direction control
G53.2		Selecting a machine coordinate system with feedrate
G53.6		Tool center point retention type tool axis direction control
G54 (G54.1)	14	Workpiece coordinate system 1 selection
G55		Workpiece coordinate system 2 selection
G56		Workpiece coordinate system 3 selection
G57		Workpiece coordinate system 4 selection
G58		Workpiece coordinate system 5 selection
G59		Workpiece coordinate system 6 selection
G60	00	Single direction positioning
G61	15	Exact stop mode
G62		Automatic corner override
G63		Tapping mode
G64		Cutting mode
G65	00	Macro call
G66	12	Macro modal call A
G66.1		Macro modal call B
G67		Macro modal call A/B cancel
G68	16	Coordinate system rotation start or 3-dimensional coordinate conversion mode on
G69		Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off
G68.2		Tilted working plane indexing
G68.3		Tilted working plane indexing by tool axis direction
G68.4		Tilted working plane indexing (incremental multi-command)
G72.1	00	Figure copying (rotary copy)
G72.2		Figure copying (linear copy)
G73	09	Peck drilling cycle
G74		Left-handed tapping cycle
G75	01	Plunge grinding cycle
G76	09	Fine boring cycle

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Table 3.1(a) G code list

G code	Group	Function
G77	01	Plunge direct sizing/grinding cycle
G78		Continuous-feed surface grinding cycle
G79		Intermittent-feed surface grinding cycle
G80	09	Canned cycle cancel Electronic gear box : synchronization cancellation
G80.4	34	Electronic gear box: synchronization cancellation
G81.4		Electronic gear box: synchronization start
G80.5	24	Electronic gear box 2 pair: synchronization cancellation
G81.5		Electronic gear box 2 pair: synchronization start
G81	09	Drilling cycle or spot boring cycle Electronic gear box : synchronization start
G81.1	00	High precision oscillation
G82	09	Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G84.2		Rigid tapping cycle (FS15 format)
G84.3		Left-handed rigid tapping cycle (FS15 format)
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle
G90	03	Absolute programming
G91		Incremental programming
G91.1	00	Checking the maximum incremental amount specified
G92		Setting for workpiece coordinate system or clamp at maximum spindle speed
G92.1		Workpiece coordinate system preset
G93	05	Inverse time feed
G94		Feed per minute
G95		Feed per revolution
G96	13	Constant surface speed control
G97		Constant surface speed control cancel
G96.1	00	Spindle indexing execution (waiting for completion)
G96.2		Spindle indexing execution (not waiting for completion)
G96.3		Spindle indexing completion check
G96.4		SV speed control mode ON
G98	10	Canned cycle : return to initial level
G99		Canned cycle : return to R point level
G107	00	Cylindrical interpolation
G160	20	In-feed control cancel
G161		In-feed control

FRYER MACHINE SYSTEMS INC
Rob Zahensky

3.2 G CODE LIST IN THE LATHE SYSTEM

T

Table 3.2 (a) G code list

G code system			Group	Function
A	B	C		
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation CW or helical interpolation CW
G03	G03	G03		Circular interpolation CCW or helical interpolation CCW
G04	G04	G04	00	Dwell
G04.1	G04.1	G04.1		G code preventing buffering
G05	G05	G05		AI contour control (command compatible with high precision contour control)
G05.1	G05.1	G05.1		AI contour control
G05.4	G05.4	G05.4		HRV3 on/off
G07.1 (G107)	G07.1 (G107)	G07.1 (G107)	00	Cylindrical interpolation
G08	G08	G08		AI contour control (advanced preview control compatible command)
G09	G09	G09		Exact stop
G10	G10	G10		Programmable data input
G10.6	G10.6	G10.6		Tool retract and recover
G10.9	G10.9	G10.9		Programmable switching of diameter/radius specification
G11	G11	G11		Programmable data input mode cancel
G12.1 (G112)	G12.1 (G112)	G12.1 (G112)	21	Polar coordinate interpolation mode
G13.1 (G113)	G13.1 (G113)	G13.1 (G113)		Polar coordinate interpolation cancel mode
G17	G17	G17	16	XpYp plane selection
G18	G18	G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G22	G22	G22	09	Stored stroke check function on
G23	G23	G23		Stored stroke check function off
G25	G25	G25	08	Spindle speed fluctuation detection off
G26	G26	G26		Spindle speed fluctuation detection on
G27	G27	G27	00	Reference position return check
G28	G28	G28		Return to reference position
G28.2	G28.2	G28.2		In-position check disable reference position return
G29	G29	G29		Movement from reference position
G30	G30	G30		2nd, 3rd and 4th reference position return
G30.2	G30.2	G30.2		In-position check disable 2nd, 3rd, or 4th reference position return
G31	G31	G31		Skip function
G31.8	G31.8	G31.8		EGB-axis skip
G32	G33	G33	01	Threading
G34	G34	G34		Variable lead threading
G35	G35	G35		Circular threading CW
G36	G36	G36		Circular threading CCW (When bit 3 (G36) of parameter No. 3405 is set to 1) or Automatic tool offset (X axis) (When bit 3 (G36) of parameter No. 3405 is set to 0)

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

Table 3.2 (a) G code list

G code system A	G code system B	G code system C	Group	Function
G37	G37	G37		Automatic tool offset (Z axis) (When bit 3 (G36) of parameter No. 3405 is set to 0)
G37.1	G37.1	G37.1	01	Automatic tool offset (X axis) (When bit 3 (G36) of parameter No. 3405 is set to 1)
G37.2	G37.2	G37.2		Automatic tool offset (Z axis) (When bit 3 (G36) of parameter No. 3405 is set to 1)
G38	G38	G38		Tool radius/tool nose radius compensation: with vector held
G39	G39	G39		Tool radius/tool nose radius compensation: corner rounding interpolation
G40	G40	G40	07	Tool radius/tool nose radius compensation : cancel
G41	G41	G41		Tool radius/tool nose radius compensation : left
G42	G42	G42		Tool radius/tool nose radius compensation : right
G40.1	G40.1	G40.1	19	Normal direction control cancel mode
G41.1	G41.1	G41.1		Normal direction control left on
G42 .1	G42 .1	G42 .1		Normal direction control right on
G43	G43	G43	23	Tool length compensation + (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G44	G44	G44		Tool length compensation - (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G43.7 (G44.7)	G43.7 (G44.7)	G43.7 (G44.7)		Tool offset (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G49 (G49.1)	G49 (G49.1)	G49 (G49.1)		Tool length compensation cancel (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G50	G92	G92	00	Coordinate system setting or max spindle speed clamp
G50.3	G92.1	G92.1		Workpiece coordinate system preset
G50.1	G50.1	G50.1	22	Programmable mirror image cancel
G51.1	G51.1	G51.1		Programmable mirror image
G50.2 (G250)	G50.2 (G250)	G50.2 (G250)	20	Polygon turning cancel
G51.2 (G251)	G51.2 (G251)	G51.2 (G251)		Polygon turning
G50.4	G50.4	G50.4	00	Cancel synchronous control
G50.5	G50.5	G50.5		Cancel composite control
G50.6	G50.6	G50.6		Cancel superimposed control
G51.4	G51.4	G51.4		Start synchronous control
G51.5	G51.5	G51.5		Start composite control
G51.6	G51.6	G51.6		Start superimposed control
G52	G52	G52		Local coordinate system setting
G53	G53	G53		Machine coordinate system setting
G53.1	G53.1	G53.1		Tool axis direction control
G53.2	G53.2	G53.2		Selecting a machine coordinate system with feedrate
G53.6	G53.6	G53.6		Tool center point retention type tool axis direction control
G54 (G54.1)	G54 (G54.1)	G54 (G54.1)	14	Workpiece coordinate system 1 selection
G55	G55	G55		Workpiece coordinate system 2 selection
G56	G56	G56		Workpiece coordinate system 3 selection
G57	G57	G57		Workpiece coordinate system 4 selection
G58	G58	G58		Workpiece coordinate system 5 selection
G59	G59	G59		Workpiece coordinate system 6 selection
G61	G61	G61	15	Exact stop mode
G63	G63	G63		Tapping mode
G64	G64	G64		Cutting mode
G65	G65	G65	00	Macro call

FEYER MACHINE SYSTEMS INC
Rob Zaremsky

Table 3.2 (a) G code list

G code system			Group	Function
A	B	C		
G66	G66	G66	12	Macro modal call A
G66.1	G66.1	G66.1		Macro modal call B
G67	G67	G67		Macro modal call A/B cancel
G68	G68	G68	04	Mirror image on for double turret or balance cutting mode
G68.1	G68.1	G68.1	17	Coordinate system rotation start or 3-dimensional coordinate system conversion mode on
G68.2	G68.2	G68.2		Tilted working plane indexing
G68.3	G68.3	G68.3		Tilted working plane indexing by tool axis direction
G68.4	G68.4	G68.4		Tilted working plane indexing (incremental multi-command)
G69	G69	G69	04	Mirror image off for double turret or balance cutting mode cancel
G69.1	G69.1	G69.1	17	Coordinate system rotation cancel or 3-dimensional coordinate system conversion mode off
G70	G70	G72	00	Finishing cycle
G71	G71	G73		Stock removal in turning
G72	G72	G74		Stock removal in facing
G73	G73	G75		Pattern repeating cycle
G74	G74	G76		End face peck drilling cycle
G75	G75	G77		Outer diameter/internal diameter drilling cycle
G76	G76	G78		Multiple-thread cutting cycle
G71	G71	G72	01	Traverse grinding cycle
G72	G72	G73		Traverse direct sizing/grinding cycle
G73	G73	G74		Oscillation grinding cycle
G74	G74	G75		Oscillation direct sizing/grinding cycle
G80	G80	G80	10	Canned cycle cancel for drilling Electronic gear box : synchronization cancellation
G81.1	G81.1	G81.1	00	High precision oscillation function
G80.4	G80.4	G80.4	28	Electronic gear box: synchronization cancellation
G81.4	G81.4	G81.4		Electronic gear box: synchronization start
G80.5	G80.5	G80.5	27	Electronic gear box 2 pair: synchronization cancellation
G81.5	G81.5	G81.5		Electronic gear box 2 pair: synchronization start
G81	G81	G81	10	Spot drilling (FS15-T format) Electronic gear box : synchronization start
G82	G82	G82		Counter boring (FS15-T format)
G83	G83	G83		Cycle for face drilling
G83.1	G83.1	G83.1		High-speed peck drilling cycle (FS15-T format)
G83.5	G83.5	G83.5		High-speed peck drilling cycle
G83.6	G83.6	G83.6		Peck drilling cycle
G84	G84	G84		Cycle for face tapping
G84.2	G84.2	G84.2		Rigid tapping cycle (FS15-T format)
G85	G85	G85		Cycle for face boring
G87	G87	G87		Cycle for side drilling
G87.5	G87.5	G87.5		High-speed peck drilling cycle
G87.6	G87.6	G87.6		Peck drilling cycle
G88	G88	G88		Cycle for side tapping
G89	G89	G89		Cycle for side boring
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Threading cycle
G94	G79	G24		End face turning cycle
G91.1	G91.1	G91.1	00	Maximum specified incremental amount check
G96	G96	G96	02	Constant surface speed control
G97	G97	G97		Constant surface speed control cancel

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Table 3.2 (a) G code list

G code system			Group	Function
A	B	C		
G96.1	G96.1	G96.1	00	Spindle indexing execution (waiting for completion)
G96.2	G96.2	G96.2		Spindle indexing execution (not waiting for completion)
G96.3	G96.3	G96.3		Spindle indexing completion check
G96.4	G96.4	G96.4		SV speed control mode ON
G98	G94	G94	05	Feed per minute
◤ G99	◤ G95	◤ G95		Feed per revolution
-	◤ G90	◤ G90	03	Absolute programming
-	G91	G91		Incremental programming
-	G98	G98	11	Canned cycle : return to initial level
-	G99	G99		Canned cycle : return to R point level

FRYER MACHINE SYSTEMS INC
Rob Zahensky

4 INTERPOLATION FUNCTIONS

Interpolation functions specify the way to make an axis movement (in other words, a movement of the tool with respect to the workpiece or table).

4.1 POSITIONING (G00)

The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental programming at a rapid traverse rate.
In the absolute programming, coordinate value of the end point is programmed.
In the incremental programming the distance the tool moves is programmed.

Format

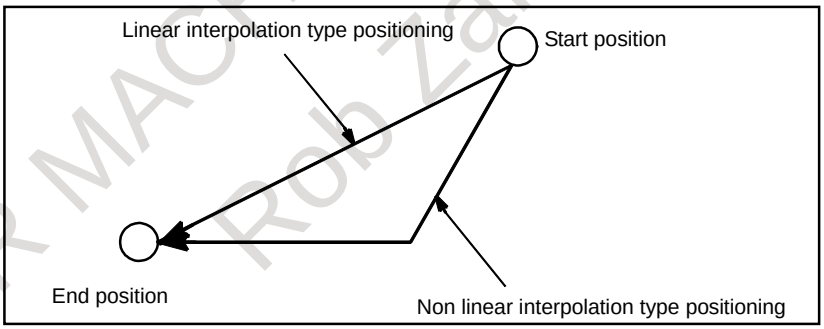
G00 IP_ ;

IP_ : For an absolute programming, the coordinates of an end point, and for an incremental programming, the distance the tool moves.

Explanation

Either of the following tool paths can be selected according to bit 1 (LRP) of parameter No. 1401.

- Nonlinear interpolation type positioning
  The tool is positioned with the rapid traverse rate for each axis separately. The tool path is normally straight.
- Linear interpolation type positioning
  The tool is positioned within the shortest possible time at a speed that is not more than the rapid traverse rate for each axis.



The rapid traverse rate in G00 command is set to the parameter No. 1420 for each axis independently by the machine tool builder. In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution proceeds to the next block after confirming the in-position.
"In-position " means that the feed motor is within the specified range.
This range is determined by the machine tool builder by setting to parameter No. 1826.

Limitation

The rapid traverse rate cannot be specified in the address F.
Even if linear interpolation type positioning is specified, nonlinear type interpolation positioning is used in the following cases. Therefore, be careful to ensure that the tool does not foul the workpiece.

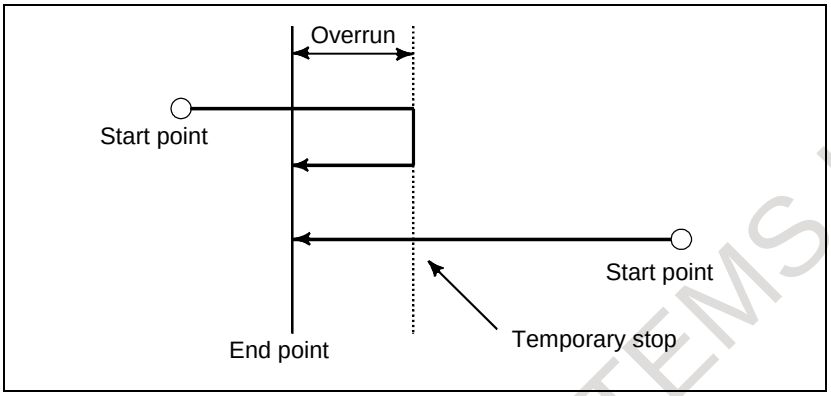
- G28 specifying positioning between the reference and intermediate positions.
- G53

FRYER MACHINE SYSTEMS INC
Rob Zarensky

4.2 SINGLE DIRECTION POSITIONING (G60)

M

For accurate positioning without play of the machine (backlash), final positioning from one direction is available.



Format

G60 IP_ ;

IP_ : For an absolute programming, the coordinates of an end point, and for an incremental programming, the distance the tool moves.

Explanation

An overrun and a positioning direction are set by the parameter No. 5440. Even when a commanded positioning direction coincides with that set by the parameter, the tool stops once before the end point.
G60, which is a one-shot G-code, can be used as a modal G-code in group 01 by setting 1 to the bit 0 (MDL) of parameter No. 5431.
This setting can eliminate specifying a G60 command for every block. Other specifications are the same as those for a one-shot G60 command. When a one-shot G code is specified in the single direction positioning mode, the one-shot G command is effective like G codes in group 01.

(Example)

```
When one-shot G60 commands are used.
G90;
G60 X0Y0;  }
G60 X100;  }   Single direction positioning
G60 Y100;  }
G04 X10;
G00 X0Y0;
When modal G60 command is used.
G90G60;        Single direction positioning mode start
X0Y0;      }
X100;      }   Single direction positioning
Y100;      }
G04X10;
G00X0 Y0;      Single direction positioning mode cancel
```

FANUC MACHINE SYSTEMS INC
Rob Zakrensky

- Overview of operation

- **In the case of positioning of non-linear interpolation type (bit 1 (LRP) of parameter No. 1401 = 0)**
  As shown below (Fig. 4.2 (a)), single direction positioning is performed independently along each axis.

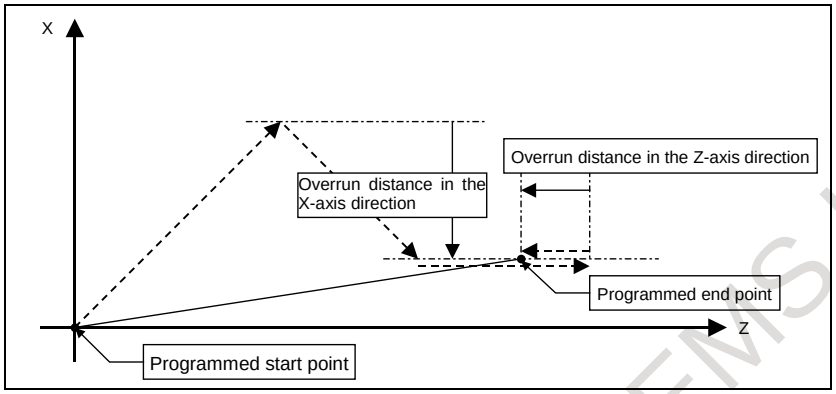


Fig. 4.2 (a)

- **In the case of positioning of linear interpolation type (bit 1 (LRP) of parameter No. 1401 = 1)**
  Positioning of interpolation type is performed until the tool once stops before or after a specified end point. Then, the tool is positioned independently along each axis until the end point is reached.

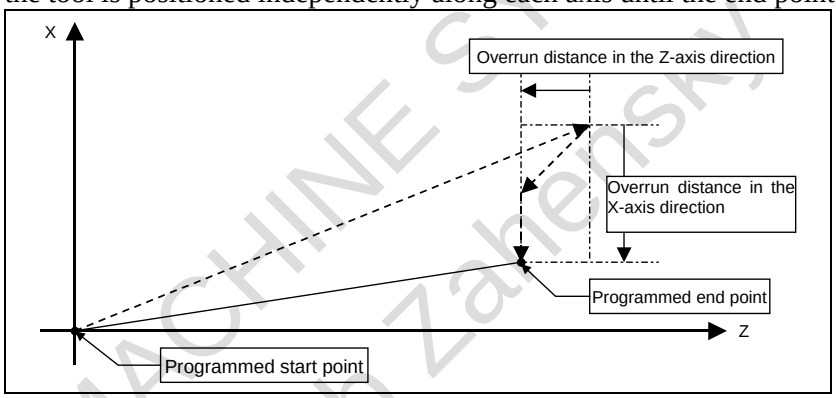


Fig. 4.2 (b)

Limitation

- Single direction positioning is not performed along an axis for which no overrun distance is set in parameter No. 5440.
- Single direction positioning is not performed along an axis for which travel distance 0 is specified.
- The mirror image function is not applied in a parameter-set direction. Even in the mirror image mode, the direction of single direction positioning remains unchanged. If positioning of linear interpolation type is used, and the state of mirror image when a single direction positioning block is looked ahead differs from the state of mirror image when the execution of the block is started, an alarm (DS0025)"G60 CANNOT BE EXECUTED" is issued. When switching mirror image in the middle of a program, disable looking ahead by specifying a non-buffering M code. Then, switch mirror image when there is no look-ahead block.
- In the cylindrical interpolation mode (G07.1), single direction positioning cannot be used.
- When specifying single direction positioning on a machine that uses arbitrary angular axis control, first position the angular axis then specify the positioning of the Cartesian axis. If the reverse specification order is used, or the angular axis and Cartesian axis are specified in the same block, an incorrect positioning direction can result.
- In positioning at a restart position by program restart function, single direction positioning is not performed.
- During canned cycle for drilling, no single direction positioning is effected in drilling axis.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- The single direction positioning does not apply to the shift motion in the canned cycles of G76 and G87.

4.3 LINEAR INTERPOLATION (G01)

Tools can move along a line.

Format

G01 IP_ **F_ ;**

IP_ : For an absolute programming, the coordinates of an end point, and for an incremental programming, the distance the tool moves.

F_ : Speed of tool feed (Feedrate)

Explanation

A tools move along a line to the specified position at the feedrate specified in F.
The feedrate specified in F is effective until a new value is specified. It need not be specified for each block.
The feedrate commanded by the F code is measured along the tool path. If the F code is not commanded, the feedrate is regarded as zero.
The feedrate of each axis direction is as follows.

G01 αα ββ γγ ζζ Ff ;

Feed rate of α axis direction : $F\alpha = \frac{\alpha}{L} \times f$

Feed rate of β axis direction : $F\beta = \frac{\beta}{L} \times f$

Feed rate of γ axis direction : $F\gamma = \frac{\gamma}{L} \times f$

Feed rate of ζ axis direction : $F\zeta = \frac{\zeta}{L} \times f$

$L = \sqrt{\alpha^2 + \beta^2 + \gamma^2 + \zeta^2}$

The feedrate of the rotary axis is commanded in the unit of deg/min (the unit is decimal point position).
When the straight line axis α (such as X, Y, or Z) and the rotating axis β (such as A, B, or C) are linearly interpolated, the feedrate is that in which the tangential feedrate in the α and β cartesian coordinate system is commanded by F (mm/min).
β-axis feedrate is obtained ; at first, the time required for distribution is calculated by using the above formula, then the β-axis feedrate unit is changed to deg/min.

A calculation example is as follows.
G91 G01 X20.0C40.0 F300.0 ;
This changes the unit of the C axis from 40.0 deg to 40mm with metric input.
The time required for distribution is calculated as follows:

$$\frac{\sqrt{20^2 + 40^2}}{300} \doteqdot 0.14907(mm)$$

The feedrate for the C axis is

$$\frac{40}{0.14907} \doteqdot 268.3 \text{deg/min}$$

In simultaneous 3 axes control, the feedrate is calculated the same way as in 2 axes control.

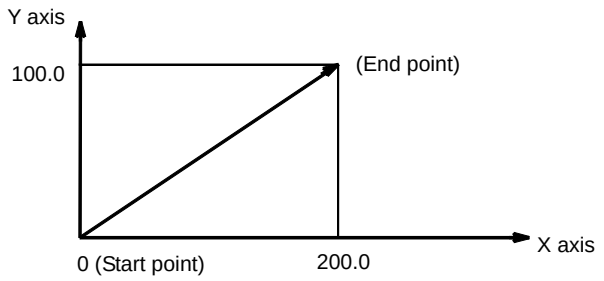
FRYER MACHINE SYSTEMS INC
Bob Zahensky

Example

- Linear interpolation

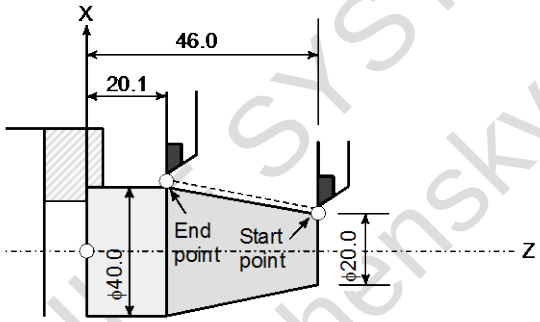
- For milling machining

(G91) G01X200.0Y100.0F200.0;



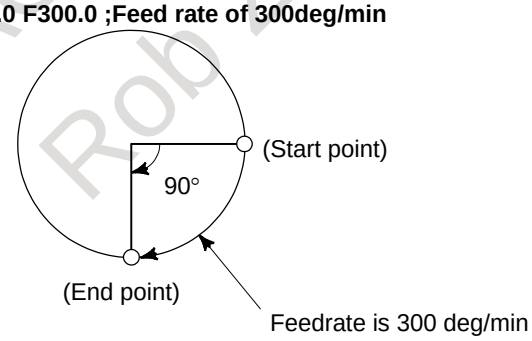
- For lathe cutting

(Diameter programming)
G01X40.0Z20.1F20; (Absolute programming)
or
G01U20.0W-25.9F20; (Incremental programming)



- Feedrate for the rotary axis

G91G01C-90.0 F300.0 ;Feed rate of 300deg/min



FRYER MACHINE SYSTEMS INC
Rob Zahensky

4.4 CIRCULAR INTERPOLATION (G02, G03)

The command below will move a tool along a circular arc.

Format

Arc in the XpYp plane

$$\mathbf{G17}\left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\ \mathbf{Xp\_\ Yp\_}\left\{\begin{matrix}\mathbf{I\_\ J\_}\\ \mathbf{R\_}\end{matrix}\right\}\ \mathbf{F\_\ ;}$$

Arc in the ZpXp plane

$$\mathbf{G18}\left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\ \mathbf{Zp\_\ Xp\_}\left\{\begin{matrix}\mathbf{I\_\ K\_}\\ \mathbf{R\_}\end{matrix}\right\}\ \mathbf{F\_\ ;}$$

Arc in the YpZp plane

$$\mathbf{G19}\left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\ \mathbf{Yp\_\ Zp\_}\left\{\begin{matrix}\mathbf{J\_\ K\_}\\ \mathbf{R\_}\end{matrix}\right\}\ \mathbf{F\_\ ;}$$

Command	Description
G17	Specification of arc on XpYp plane
G18	Specification of arc on ZpXp plane
G19	Specification of arc on YpZp plane
G02	Circular Interpolation : Clockwise direction (CW)
G03	Circular Interpolation : Counterclockwise direction (CCW)
Xp	Command values of X axis or its parallel axis (set by parameter No. 1022)
Yp	Command values of Y axis or its parallel axis (set by parameter No. 1022)
Zp	Command values of Z axis or its parallel axis (set by parameter No. 1022)
I_	Xp axis distance from the start point to the center of an arc with sign
J_	Yp axis distance from the start point to the center of an arc with sign
K_	Zp axis distance from the start point to the center of an arc with sign
R_	Arc radius (with sign, radius value for lathe cutting)
F_	Feedrate along the arc

T

NOTE

1 The U-, V-, and W-axes can be used with G-codes B and C.

2 When a circular interpolation with I,J,K command and a T-code are specified in the same block, tool path might become a spiral shape instead of an arc shape. Don’t specify a circular interpolation command and a T-code in the same block.

FRYER MACHINE SYSTEMS INC
Rob Zamansky

Explanation

- Direction of the circular interpolation

"Clockwise"(G02) and "counterclockwise"(G03) on the XpYp plane (ZpXp plane or YpZp plane) are defined when the XpYp plane is viewed in the positive-to-negative direction of the Zp axis (Yp axis or Xp axis, respectively) in the Cartesian coordinate system. See the figure below (Fig. 4.4 (a)).

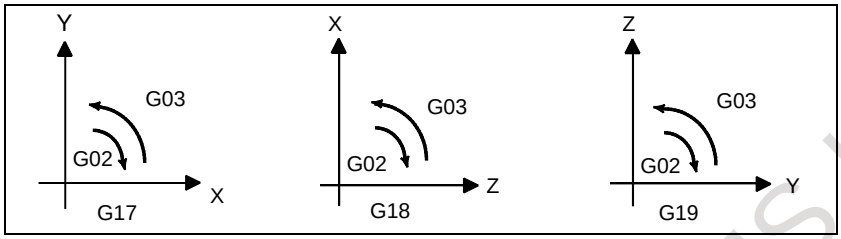


Fig. 4.4 (a)

- Distance moved on an arc

The end point of an arc is specified by address Xp, Yp or Zp, and is expressed as an absolute or incremental value according to G90 or G91. For the incremental value, the distance of the end point which is viewed from the start point of the arc is specified with sign.

- Distance from the start point to the center of arc

The arc center is specified by addresses I, J, and K for the Xp, Yp, and Zp axes, respectively. The numerical value following I, J, or K, however, is a vector component in which the arc center is seen from the start point, and is always specified as an incremental value irrespective of G90 and G91, as shown below (Fig. 4.4 (b)).

I, J, and K must be signed according to the direction.

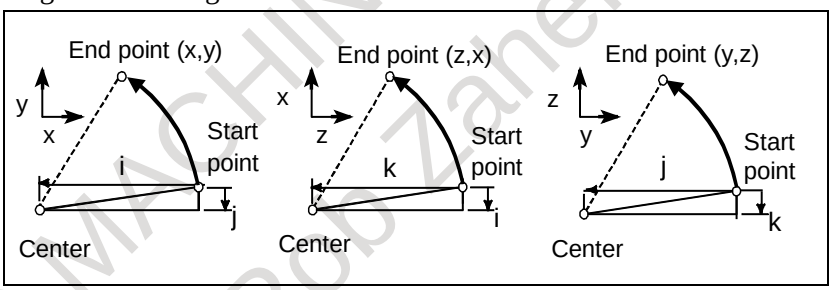


Fig. 4.4 (b)

I0, J0, and K0 can be omitted.

If the difference between the radius at the start point and that at the end point exceeds the permitted value in a parameter No.3410, an alarm PS0020, "OVER TOLERANCE OF RADIUS" occurs.

- Command for a circle

When Xp, Yp, and Zp are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360° arc (circle) is specified.

G02 I_ ; Command for a circle

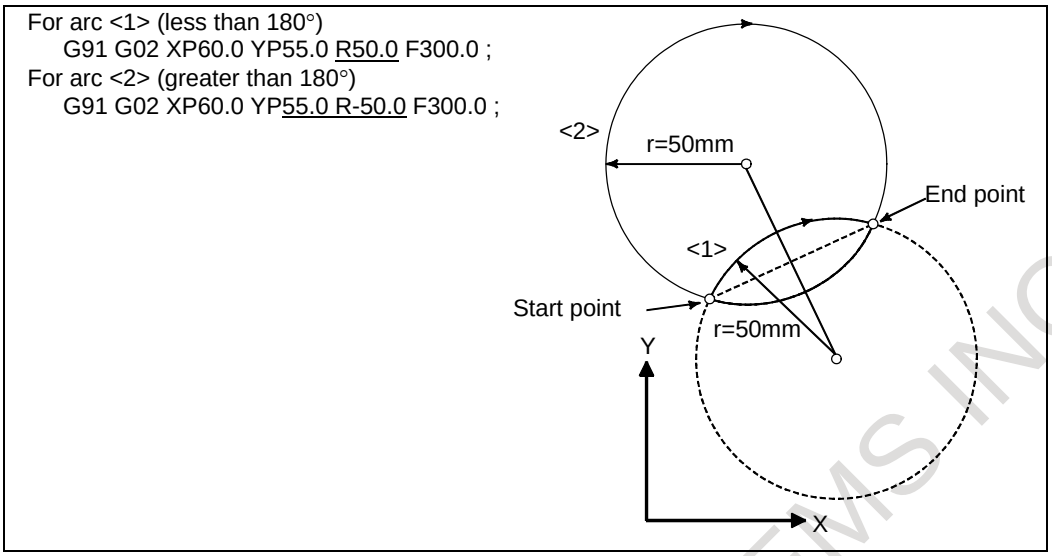
- Arc radius

The distance between an arc and the center of a circle that contains the arc can be specified using the radius, R, of the circle instead of I, J, and K.

In this case, one arc is less than 180°, and the other is more than 180° are considered. When an arc exceeding 180° is commanded, the radius must be specified with a negative value. If Xp, Yp, and Zp are all omitted, if the end point is located at the same position as the start point and when R is used, an arc of 0° is programmed

G02R_ ; (The cutter does not move.)

FRYER MACHINE SYSTEMS INC
Rob Zanensky



- Feedrate

The feedrate in circular interpolation is equal to the feedrate specified by the F code, and the feedrate along the arc (the tangential feedrate of the arc) is controlled to be the specified feedrate.
The error between the specified feedrate and the actual tool feedrate is ±2% or less. However, this feedrate is measured along the arc after the tool radius compensation is applied

Limitation

- Simultaneously specifying R with I, J, and K

If I, J, K, and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.

- Specifying an axis that is not contained in the specified plane

If an axis not comprising the specified plane is commanded, an alarm PS0021, "ILLEGAL PLANE SELECT" occurs.
For example,
For milling machining:
If the X-axis and a U-axis parallel to the X-axis are specified when the XY plane is specified
For lathe cutting:
If the X-axis and a U-axis parallel to the X-axis are specified when the ZX plane is specified with G code system B or C

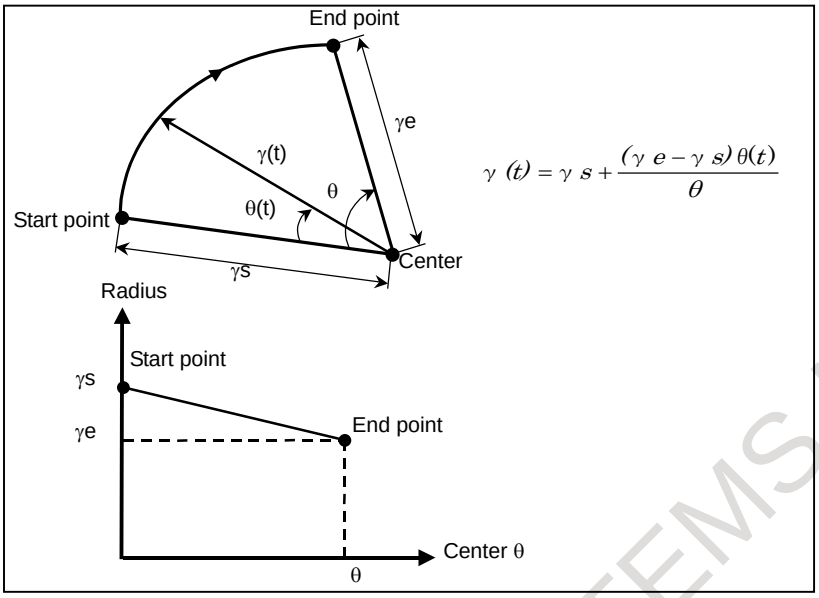
- Specifying a semicircle with R

When an arc having a center angle approaching 180° is specified, the calculated center coordinates may contain an error. In such a case, specify the center of the arc with I, J, and K.

- Difference in the radius between the start and end points

If the difference in the radius between the start and end points of the arc exceeds the value specified in parameter No. 3410, alarm PS0020, "OVER TOLERANCE OF RADIUS" is generated.
When an end point does not lie on the arc, a spiral results, as shown below.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

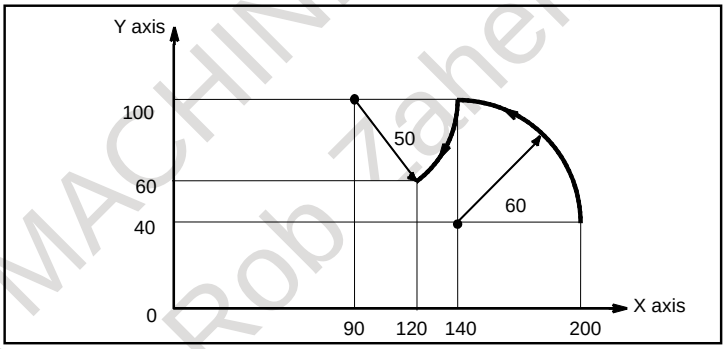


$$\gamma(t) = \gamma s + \frac{(\gamma e - \gamma s)\,\theta(t)}{\theta}$$

The arc radius changes linearly with the center angle θ(t). Spiral interpolation is performed using a circular command that specifies one arc radius for the start point and another arc radius for the end point. To use spiral interpolation, set a large value in parameter No. 3410, used to specify the limit on the arc radius error.

Example

M



The above tool path can be programmed as follows;

(1) In absolute programming

```
G92X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0 R60.0 F300. ;
G02 X120.0 Y60.0 R50.0 ;
```

or

```
G92X200.0 Y40.0Z0 ;
G90 G03 X140.0 Y100.0I-60.0 F300. ;
G02 X120.0 Y60.0I-50.0 ;
```

(2) In incremental programming

```
G91 G03 X-60.0 Y60.0 R60.0 F300. ;
G02 X-20.0 Y-40.0 R50.0 ;
```

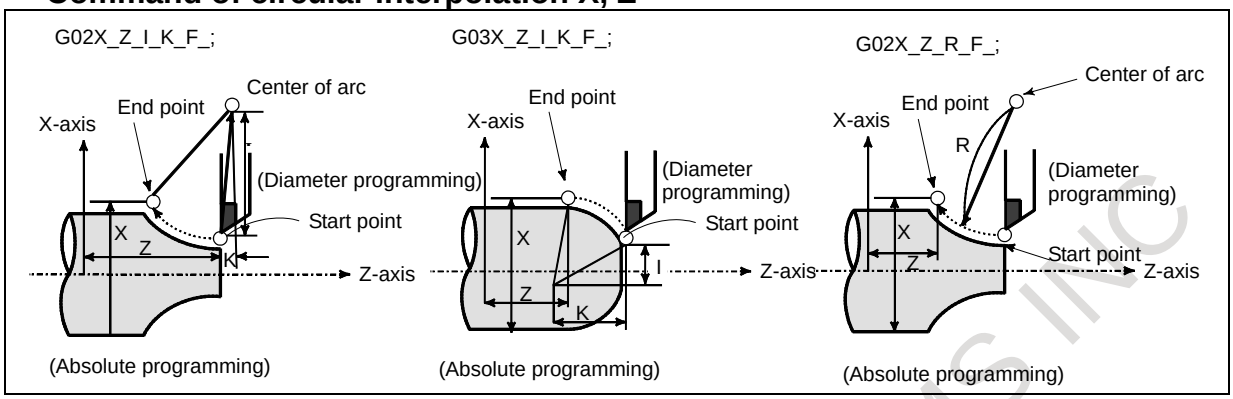
or

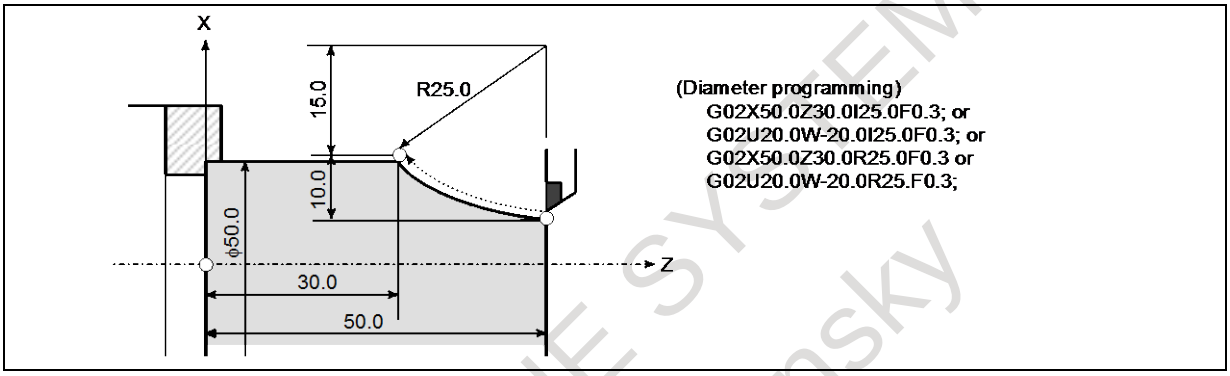
```
G91 G03 X-60.0 Y60.0 I-60.0 F300. ;
G02 X-20.0 Y-40.0 I-50.0 ;
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

T

- **Command of circular interpolation X, Z**





4.5 HELICAL INTERPOLATION (G02, G03)

Helical interpolation which moved helically is enabled by specifying up to two other axes which move synchronously with the circular interpolation by circular commands.

Format

Arc in the XpYp plane

$$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Xp\_\ Yp\_ \begin{Bmatrix} I\_\ J\_ \\ R\_ \end{Bmatrix} \alpha\_\ (\beta\_)\ F\_\ ;$$

Arc in the ZpXp plane

$$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Zp\_\ Xp\_ \begin{Bmatrix} K\_\ I\_ \\ R\_ \end{Bmatrix} \alpha\_\ (\beta\_)\ F\_\ ;$$

Arc in the YpZp plane

$$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Yp\_\ Zp\_ \begin{Bmatrix} J\_\ K\_ \\ R\_ \end{Bmatrix} \alpha\_\ (\beta\_)\ F\_\ ;$$

α, β : Any one axis where circular interpolation is not applied.
Up to two other axes can be specified.

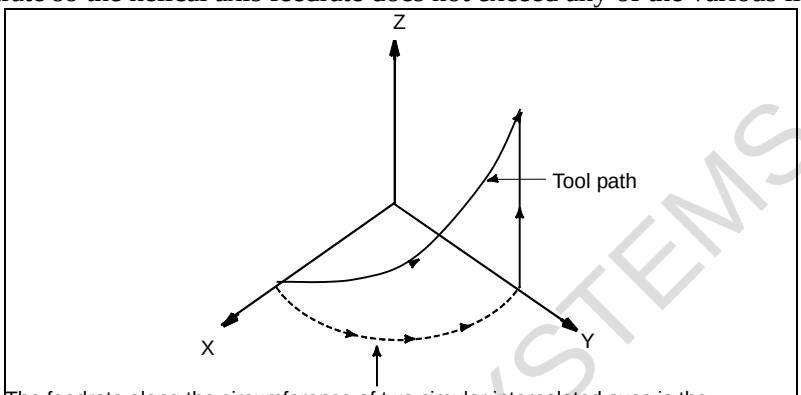
FRYER MACHINE SYSTEMS INC
Rob Zarensky

Explanation

A tangential velocity of an arc in a specified plane or a tangential velocity about the helical axis can be specified as the feedrate, depending on the setting of bit 5 (HTG) of parameter No.1403.
An F command specifies a feedrate along a circular arc, when HTG is specified to 0. Therefore, the feedrate of the helical axis is as follows:

$$F \times \frac{\text{Length of helical axis}}{\text{Length of circular arc}}$$

Determine the feedrate so the helical axis feedrate does not exceed any of the various limit values.



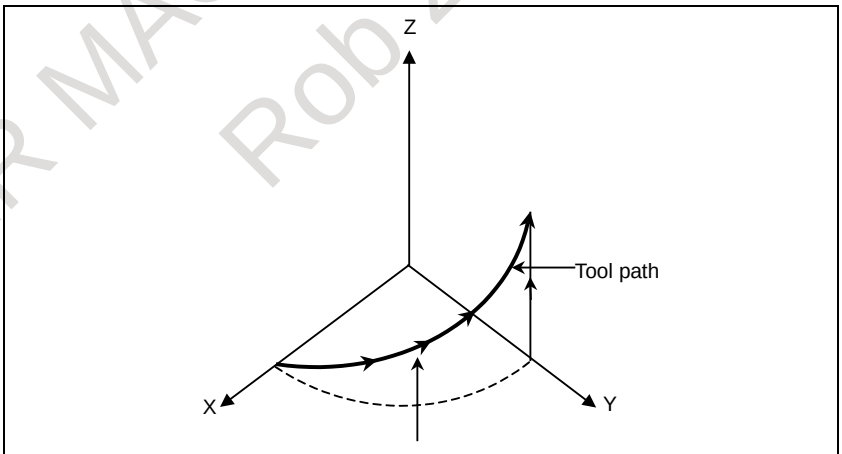
The feedrate along the circumference of two circular interpolated axes is the specified feedrate.

If HTG is set to 1, specify a feedrate along the tool path about the helical axis. Therefore, the tangential velocity of the arc is expressed as follows:

$$F \times \frac{\text{Length of arc}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of helical axis})^2}}$$

The velocity along the helical axis is expressed as follows:

$$F \times \frac{\text{Length of helical axis}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of helical axis})^2}}$$



The feedrate along the tool path is specified.

If three or more axes were specified as helical axes in the helical interpolation mode, alarm (PS0232)"TOO MANY HELICAL AXIS COMMAND" is issued.

Limitation

- Tool radius/tool nose radius compensation is applied only for a circular arc.
- Tool offset and tool length compensation cannot be used in a block in which a helical interpolation is commanded.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

4.6 POLAR COORDINATE INTERPOLATION (G12.1, G13.1)

T

Overview

Polar coordinate interpolation is a function that exercises contour control in converting a command programmed in a Cartesian coordinate system to the movement of a linear axis (movement of a tool) and the movement of a rotary axis (rotation of a workpiece). This function is useful for grinding a cam shaft.

Format

G12.1; Starts polar coordinate interpolation mode (enables polar coordinate interpolation).

Specify linear or circular interpolation using coordinates in a Cartesian coordinate system consisting of a linear axis and rotary axis (hypothetical axis).

G13.1; Polar coordinate interpolation mode is cancelled (for not performing polar coordinate interpolation).

Specify G12.1 and G13.1 in Separate Blocks.

Explanation

- Polar coordinate interpolation mode (G12.1)

The axes of polar coordinate interpolation (linear axis and rotary axis) should be specified in advance, with corresponding parameters. Specifying G12.1 places the system in the polar coordinate interpolation mode, and selects a plane (called the polar coordinate interpolation plane) formed by one linear axis and a hypothetical axis intersecting the linear axis at right angles. The linear axis is called the first axis of the plane, and the hypothetical axis is called the second axis of the plane. Polar coordinate interpolation is performed in this plane.

In the polar coordinate interpolation mode, both linear interpolation and circular interpolation can be specified by absolute or incremental programming.

Tool radius compensation can also be performed. The polar coordinate interpolation is performed for a path obtained after tool radius compensation.

The tangential velocity in the polar coordinate interpolation plane (Cartesian coordinate system) is specified as the feedrate, using F.

- Polar coordinate interpolation cancel mode (G13.1)

Specifying G13.1 cancels the polar coordinate interpolation mode.

- Polar coordinate interpolation plane

G12.1 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane (Fig. 4.6). Polar coordinate interpolation is performed on this plane.

FRYER MACHINE SYSTEMS INC
Rob Zakensky

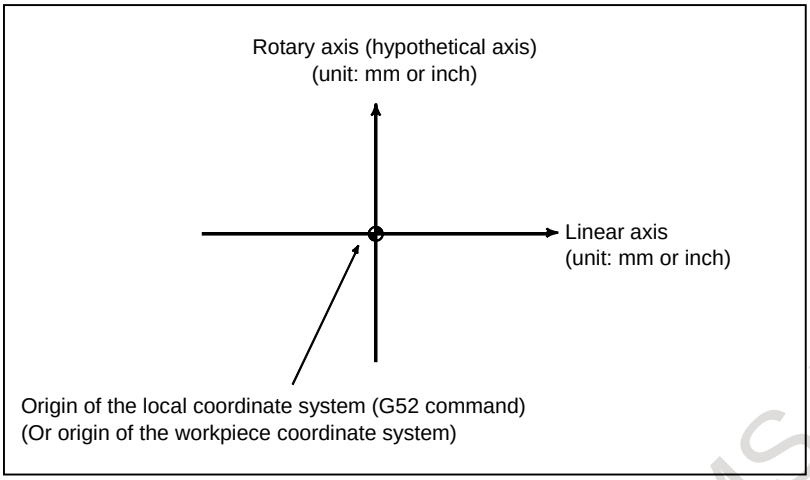


Fig. 4.6 Polar coordinate interpolation plane

When the power is turned on or the system is reset, polar coordinate interpolation is canceled (G13.1). The linear and rotation axes for polar coordinate interpolation must be set in parameters Nos. 5460 and 5461 beforehand.

> ⚠ **CAUTION**
> The plane used before G12.1 is specified (plane selected by G17, G18, or G19) is canceled. It is restored when G13.1 (canceling polar coordinate interpolation) is specified.
> When the system is reset, polar coordinate interpolation is canceled and the plane specified by G17, G18, or G19 is used.

- Distance moved and feedrate for polar coordinate interpolation

- The unit for coordinates on the hypothetical axis is the same as the unit for the linear axis (mm/inch). In the polar coordinate interpolation mode, program commands are specified with Cartesian coordinates on the polar coordinate interpolation plane. The axis address for the rotary axis is used as the axis address for the second axis (hypothetical axis) in the plane. Whether a diameter or radius is specified for the first axis in the plane is the same as for the rotary axis regardless of the specification for the first axis in the plane.
  The hypothetical axis is at coordinate 0 immediately after G12.1 is specified. Polar interpolation is started assuming the rotation angle of 0 for the position of the tool when G12.1 is specified.
  Example)

```
Input in millimeters
G12.1;
G01 X10.0 F1000. ; ......A 10.0-mm movement is made on the Cartesian coordinate system.
C20.0 ;.........................A 20.0-mm movement is made on the Cartesian coordinate system.
G13.1;

Input in inches
G12.1;
G01 X10.0 F1000. ;  ....A 10.0-inch movement is made on the Cartesian coordinate system.
C20.0 ;.........................A 20.0-inch movement is made on the Cartesian coordinate system.
G13.1;
```

- The unit for the feedrate is mm/min or inch/min.
  Specify the feedrate as a speed (relative speed between the workpiece and tool) tangential to the polar coordinate interpolation plane (Cartesian coordinate system) using F.

FRYER MACHINE SYSTEMS INC
Rob Zaherinsky

- G codes which can be specified in the polar coordinate interpolation mode

G01.....Linear interpolation
G02, G03.....Circular interpolation
G04.....Dwell, Exact stop
G40, G41, G42.....Tool radius compensation (Polar coordinate interpolation is applied to the path after tool radius compensation.)
G65, G66, G67.....Custom macro command
G90, G91.....Absolute programming, incremental programming
G94, G95.....Feed per minute, feed per revolution

- Circular interpolation in the polar coordinate plane

The addresses for specifying the radius of an arc for circular interpolation (G02 or G03) in the polar coordinate interpolation plane depend on the first axis in the plane (linear axis).

- I and J in the Xp-Yp plane when the linear axis is the X-axis or an axis parallel to the X-axis.
- J and K in the Yp-Zp plane when the linear axis is the Y-axis or an axis parallel to the Y-axis.
- K and I in the Zp-Xp plane when the linear axis is the Z-axis or an axis parallel to the Z-axis.

The radius of an arc can be specified also with an R command.

NOTE
In a lathe system, the parallel axes U, V, and W can be used in the G code system B or C.

- Movement along axes not in the polar coordinate interpolation plane in the polar coordinate interpolation mode

The tool moves along such axes normally, independent of polar coordinate interpolation.

- Current position display in the polar coordinate interpolation mode

Actual coordinates are displayed. However, the remaining distance to move in a block is displayed based on the coordinates in the polar coordinate interpolation plane (Cartesian coordinates).

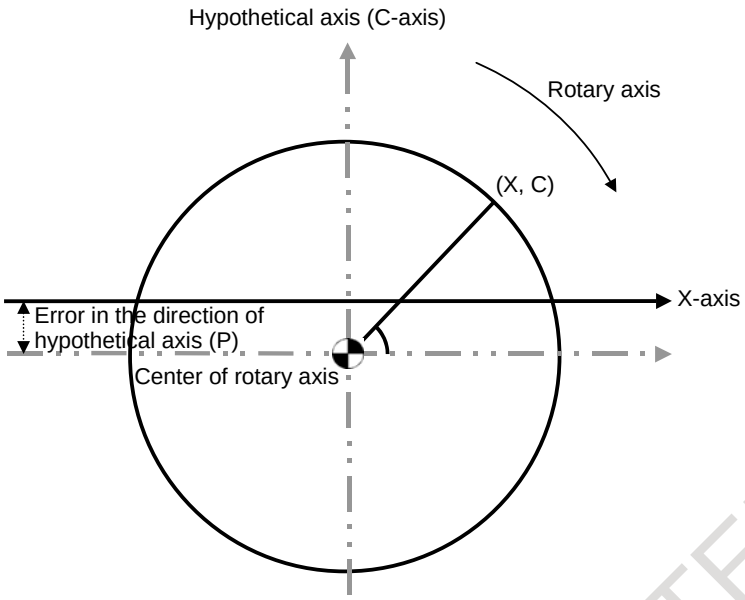
- Coordinate system for the polar coordinate interpolation

Basically, before G12.1 is specified, a local coordinate system (or workpiece coordinate system) where the center of the rotary axis is the origin of the coordinate system must be set.

- Compensation in the direction of the hypothetical axis in polar coordinate interpolation

If the first axis of the plane has an error from the center of the rotary axis in the hypothetical axis direction, in other words, if the rotary axis center is not on the X-axis, the hypothetical axis direction compensation function in the polar coordinate interpolation mode is used. With the function, the error is considered in polar coordinate interpolation. The amount of error is specified in parameter No. 5464.

FFYER MACHINE SYSTEMS INC
Rob Zakiensky



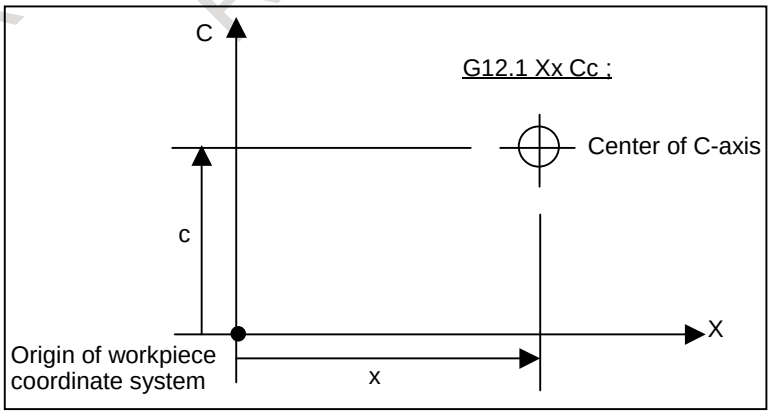
(X, C) : Point in the X-C plane (The center of the rotary axis is considered to be the origin of the X-C plane.)
X : X coordinate in the X-C plane
C : Hypothetical axis coordinate in the X-C plane
P : Error in the direction of the hypothetical axis (specified in parameter No. 5464)

- Shifting the coordinate system in polar coordinate interpolation

In the polar coordinate interpolation mode, the workpiece coordinate system can be shifted. The current position display function shows the position viewed from the workpiece coordinate system before the shift. The function to shift the coordinate system is enabled when bit 2 (PLS) of parameter No. 5450 is specified accordingly.

The shift can be specified in the polar coordinate interpolation mode, by specifying the position of the center of the rotary axis C (A, B) in the X-C (Y-A, Z-B) interpolation plane with reference to the origin of the workpiece coordinate system, in the following format.

```
G12.1 X_ C_ ;   (Polar coordinate interpolation for the X-axis and C-axis)
G12.1 Y_ A_ ;   (Polar coordinate interpolation for the Y-axis and A-axis)
G12.1 Z_ B_ ;   (Polar coordinate interpolation for the Z-axis and B-axis)
```



FRYER MACHINE SYSTEMS INC
Rob Zahensky

Limitation

- Changing the coordinate system during polar coordinate interpolation

In the G12.1 mode, the coordinate system must not be changed (G92, G52, G53, relative coordinate reset, G54 through G59, etc.).

- Tool radius/tool nose radius compensation

The polar coordinate interpolation mode (G12.1 or G13.1) cannot be started or terminated in the tool radius/tool nose radius compensation mode (G41 or G42). G12.1 or G13.1 must be specified in the tool radius/tool nose radius compensation canceled mode (G40).
For the tool radius/tool nose radius compensation canceled mode (G40) command, be sure to specify the polar coordinate axis to cancel the offset vector.

- Mirror image for double turret

The polar coordinate interpolation mode (G12.1 or G13.1) cannot be started or terminated in the mirror image for double turret mode (G68). G12.1 or G13.1 must be specified in the mirror image for double turret off mode (G69).

- Tool length offset command

Tool length offset must be specified in the polar coordinate interpolation cancel mode before G12.1 is specified. It cannot be specified in the polar coordinate interpolation mode. Furthermore, no offset values can be changed in the polar coordinate interpolation mode.

- Tool offset command

A tool offset must be specified before the G12.1 mode is set. No offset can be changed in the G12.1 mode.

- Tool geometry/wear compensation

Complete the output of the amount of compensation by tool geometry/wear compensation before the polar coordinate interpolation mode (G12.1) is set. When the setting of bit 6 (LWM) of parameter No.5002 is 1, compensation is output in the polar coordinate interpolation plane if the polar coordinate interpolation mode is set without moving command after tool changed.

- Program restart

For a block in the G12.1 mode, the program and the block cannot be restarted.

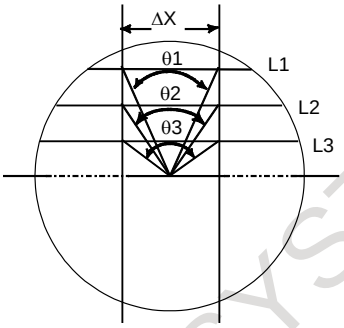
- Cutting feedrate for the rotary axis

Polar coordinate interpolation converts the tool movement for a figure programmed in a Cartesian coordinate system to the tool movement in the rotary axis (C-axis) and the linear axis (X-axis). When the tool comes close to the center of the workpiece, the C-axis velocity component increases. If the maximum cutting feedrate for the C-axis (parameter No. 1430) is exceeded, the automatic feedrate override function and automatic speed clamp function are enabled.
As well as the case of C axis, if the maximum cutting feedrate for the X-axis is exceeded, the automatic feedrate override function and automatic speed clamp function are enabled.

FRYER MACHINE SYSTEMS INC
Rob Zehensky

⚠ **WARNING**

1 Consider lines L1, L2, and L3. ΔX is the distance the tool moves per time unit at the feedrate specified with address F in the Cartesian coordinate system. As the tool moves from L1 to L2 to L3, the angle at which the tool moves per time unit corresponding to ΔX in the Cartesian coordinate system increases from θ1 to θ2 to θ3. It means that the amount of the movement of the C-axis per unit time increases, in other words, the C-axis component of the feedrate becomes larger as the tool moves closer to the center of the workpiece. The C component of the feedrate may exceed the maximum cutting feedrate for the C-axis because the tool movement in the Cartesian coordinate system has been converted to the tool movement for the C-axis and the X-axis.



L: Distance (in mm) between the tool center and workpiece center when the tool center is the nearest to the workpiece center

R: Maximum cutting feedrate (deg/min) of the C axis

Then, a speed specifiable with address F in polar coordinate interpolation can be given by the formula below. If the maximum cutting feedrate for the C-axis is exceeded, the automatic speed control function for polar coordinate interpolation automatically controls the feedrate.

$$F < L \times R \times \frac{\pi}{180} \text{ (mm/min)}$$

2 The following function cannot be used for the rotary axis of polar coordinate interpolation.

- Index table indexing function

- Automatic speed control for polar coordinate interpolation

If the velocity component of the rotary axis exceeds the maximum cutting feedrate in the polar coordinate interpolation mode, the speed is automatically controlled.

- Automatic override

If the velocity component of the rotary axis exceeds the permissible velocity (maximum cutting feedrate multiplied by the permission factor specified in parameter No. 5463), the feedrate is automatically overridden as indicated below.

Override = (Permissible velocity) ÷ (Velocity component of rotary axis) × 100(%)

- Automatic speed clamp

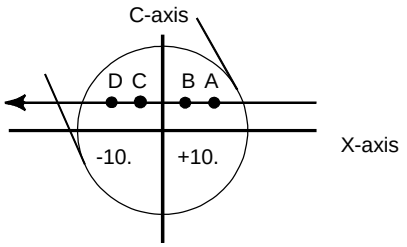
If the velocity component of the rotary axis after automatic override still exceeds the maximum cutting feedrate, the speed of the rotary axis is automatically clamped. As a result, the velocity component of the rotary axis will not exceed the maximum cutting feedrate.

The automatic speed clamp function works only when the center of the tool is very close to the center of the rotary axis.

FRYER MACHINE SYSTEMS INC
Rob Zaborsky

[Example]

```
G90 G00 X10.0 C0. ;
G12.1 ;
G01 C0.1 F1000 ;
X-10.0 :
G13.1 ;
```



Automatic speed control for polar coordinate interpolation

Suppose that the maximum cutting feedrate of the rotary axis is 360 (3600 deg/min) and that the permission factor of automatic override for polar coordinate interpolation (parameter No. 5463) is 0 (90%). If the program indicated above is executed, the automatic override function starts working when the X coordinate becomes 2.273 (point A). The automatic speed clamp function starts working when the X coordinate becomes 0.524 (point B).
The minimum value of automatic override for this example is 3%. The automatic speed clamp function continues working until the X coordinate becomes -0.524 (point C). Then, the automatic override function works until the X coordinate becomes -2.273 (point D).
(The coordinates indicated above are the values in the Cartesian coordinate system.)

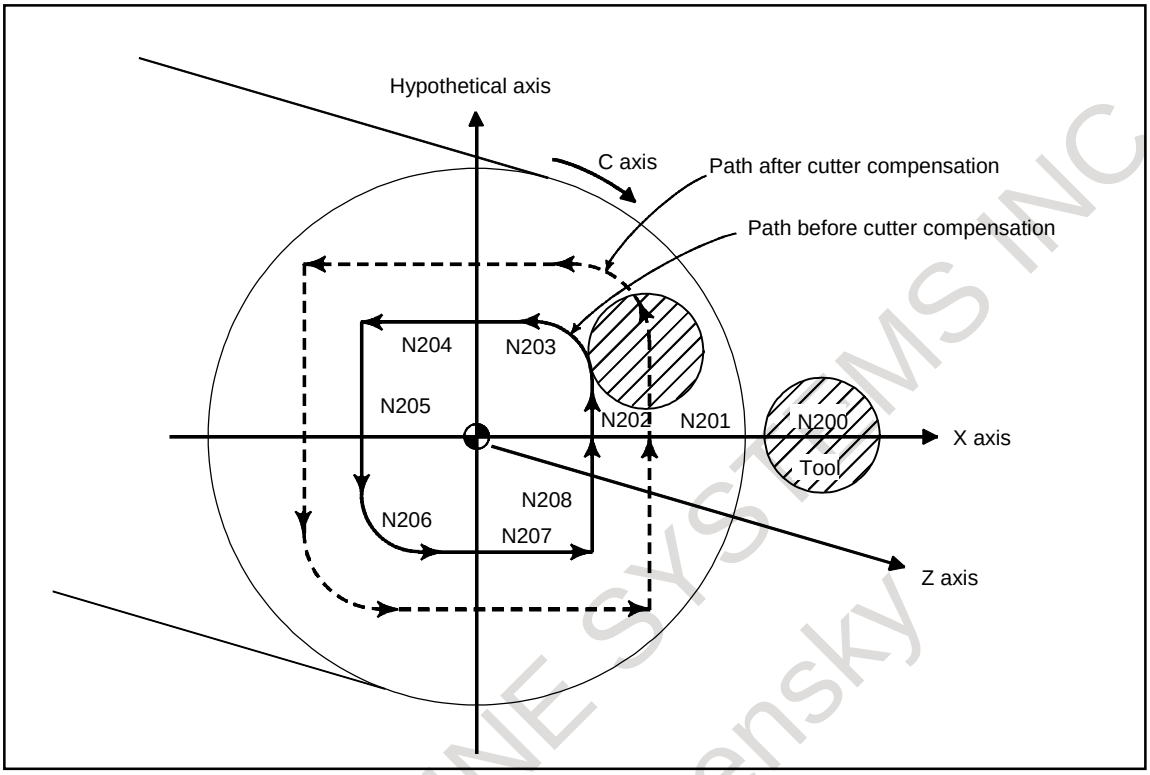
NOTE

1 While the automatic speed clamp function is working, the machine lock or interlock function may not be enabled immediately.
2 If a feed hold stop is made while the automatic speed clamp function is working, the automatic operation halt signal *SP is output. However, the operation may not stop immediately.
3 The clamped speed may exceed the clamp value by a few percent.

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

Example

Sample program for polar coordinate interpolation in a Cartesian coordinate system consisting of the X-axis (a linear axis) and a hypothetical axis




```
O0001;
  .
N010 T0101
  .
N0100 G90 G00 X60.0 C0 Z__;          Positioning to start point
N0200 G12.1;                         Start of polar coordinate interpolation
N0201 G42 G01 X20.0F__;
N0202 C10.0;
N0203 G03 X10.0 C20.0 R10.0;
N0204 G01 X-20.0;                    Geometry program
N0205 C-10.0;                        (program based on cartesian coordinates on
N0206 G03 X-10.0 C-20.0 I10.0 J0;    X axis-hypothetical axis plane)
N0207 G01 X20.0;
N0208 C0;
N0209 G40 X60.0;
N0210 G13.1;                         Cancellation of polar coordinate interpolation
N0300 Z__;
N0400 X__C__;
  .
N0900M30;
```

FIXER MACHINE SYSTEMS INC
Rob Zahensky

4.7 CYLINDRICAL INTERPOLATION (G07.1)

4.7.1 Cylindrical Interpolation

In cylindrical interpolation function, the amount of movement of a rotary axis specified by angle is converted to the amount of movement on the circumference to allow linear interpolation and circular interpolation with another axis.
Since programming is enabled with the cylinder side face expanded, programs such as a program for grooving cylindrical cams can be created very easily.

Format

G07.1 IP r; Starts the cylindrical interpolation mode (enables cylindrical interpolation).
:
:
:
G07.1 IP 0; The cylindrical interpolation mode is cancelled.
IP : One of address for the rotary axis
r : The radius of the workpiece
Specify G07.1 IPr; and G07.1 IP0; in separate blocks.
G107 can be used instead of G07.1.

NOTE
Only a positive value is effective as the radius of the workpiece. If a negative value is specified, alarm PS0175, "ILLEGAL G07.1 AXIS", is issued.

Explanation

- Plane selection (G17, G18, G19)

To specify a G code for plane selection, set the rotary axis in parameter No. 1022 as a linear axis that is one of the basic three axes of the basic coordinate system or an axis parallel to one of the basic axes. For example, when rotary axis C-axis is assumed to be parallel to the X-axis, specifying G17, axis address C, and Y at the same time can select a plane formed by the C-axis and Y-axis (the Xp-Yp plane).
Only one rotation axis of Cylindrical interpolation can be set.

T

NOTE
The U-, V-, and W-axes can be used with G-codes B and C.

- Feedrate

A feedrate specified in the cylindrical interpolation mode is the feedrate on the circumference.

- Circular interpolation (G02, G03)

Circular interpolation can be performed between the rotary axis set for cylindrical interpolation and another linear axis. Radius R is used in commands in the same way as described.
The unit for a radius is not degrees but millimeters (for metric input) or inches (for inch input).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

<Example Circular interpolation between the Z axis and C axis>

For the C axis of parameter No.1022, 5 (axis parallel with the X axis) is to be set. In this case, the command for circular interpolation is

```
G18 Z_C_;
G02 (G03) Z_C_R_;
```

For the C axis of parameter No.1022, 6 (axis parallel with the Y axis) may be specified instead. In this case, however, the command for circular interpolation is

```
G19 C_Z_;
G02 (G03) Z_C_R_;
```

- Tool radius/tool nose radius compensation

To perform tool radius/tool nose radius compensation in the cylindrical interpolation mode, cancel any ongoing tool radius/tool nose radius compensation mode before entering the cylindrical interpolation mode. Then, start and terminate tool radius/tool nose radius compensation within the cylindrical interpolation mode.

- Cylindrical interpolation accuracy

In the cylindrical interpolation mode, the amount of travel of a rotary axis specified by an angle is once internally converted to a distance of a linear axis on the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back to an angle. For this conversion, the amount of travel is rounded to a least input increment.
So when the radius of a cylinder is small, the actual amount of travel can differ from a specified amount of travel. Note, however, that such an error is not accumulative.
If manual operation is performed in the cylindrical interpolation mode with manual absolute on, an error can occur for the reason described above.

$$\text{The actual amount of travel} = \left[\frac{\text{MOTION REV}}{2\times 2\pi\text{R}}\times\left[\text{Specified value}\times\frac{2\times 2\pi\text{R}}{\text{MOTION REV}}\right]\right]$$

MOTION REV : The amount of travel per rotation of the rotary axis (360°)
R : Workpiece radius
[ ] : Rounded to the least input increment

Limitation

- Arc radius specification in the circular interpolation

In the cylindrical interpolation mode, an arc radius cannot be specified with word address I, J, or K.

- Positioning

In the cylindrical interpolation mode, positioning operations (including those that produce rapid traverse cycles such as G28, G53, G73, G74, G76, G80 to G89) cannot be specified. Before positioning can be specified, the cylindrical interpolation mode must be cancelled. Cylindrical interpolation (G07.1) cannot be performed in the positioning mode (G00).

- Cylindrical interpolation mode setting

In the cylindrical interpolation mode, the cylindrical interpolation mode cannot be reset. The cylindrical interpolation mode must be cancelled before the cylindrical interpolation mode can be reset.

- Rotary axis

Only one rotary axis can be set for cylindrical interpolation. Therefore, it is impossible to specify more than one rotary axis in the G07.1 command.

- Rotary axis roll-over

If a rotary axis using the roll-over function is specified at the start of the cylindrical interpolation mode, the roll-over function is automatically disabled in the cylindrical interpolation mode. After the cylindrical interpolation mode is canceled, the roll-over function is enabled automatically.

FRYER MACHINE SYSTEMS INC
Rob Zahansky

- Tool radius/tool nose radius compensation

If the cylindrical interpolation mode is specified when tool radius/tool nose radius compensation is already being applied, correct compensation is not performed. Specify compensation in the cylindrical interpolation mode.

- Canned cycle for drilling

Canned cycles (G73, G74, and G81 to G89 for M series / G80 to G89 for T series) for drilling, cannot be specified during cylindrical interpolation mode.

M

- Coordinate system setting

In the cylindrical interpolation mode, a workpiece coordinate system (G92, G54 to G59) or local coordinate system (G52) cannot be specified.

- Tool offset

A tool offset must be specified before the cylindrical interpolation mode is set. No offset can be changed in the cylindrical interpolation mode.

- Index table indexing function

Cylindrical interpolation cannot be specified when the index table indexing function is being used.

T

- Coordinate system setting

In the cylindrical interpolation mode, a workpiece coordinate system G50 cannot be specified.

- Mirror image for double turret

Mirror image for double turret, G68 and G69, cannot be specified during cylindrical interpolation mode.

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

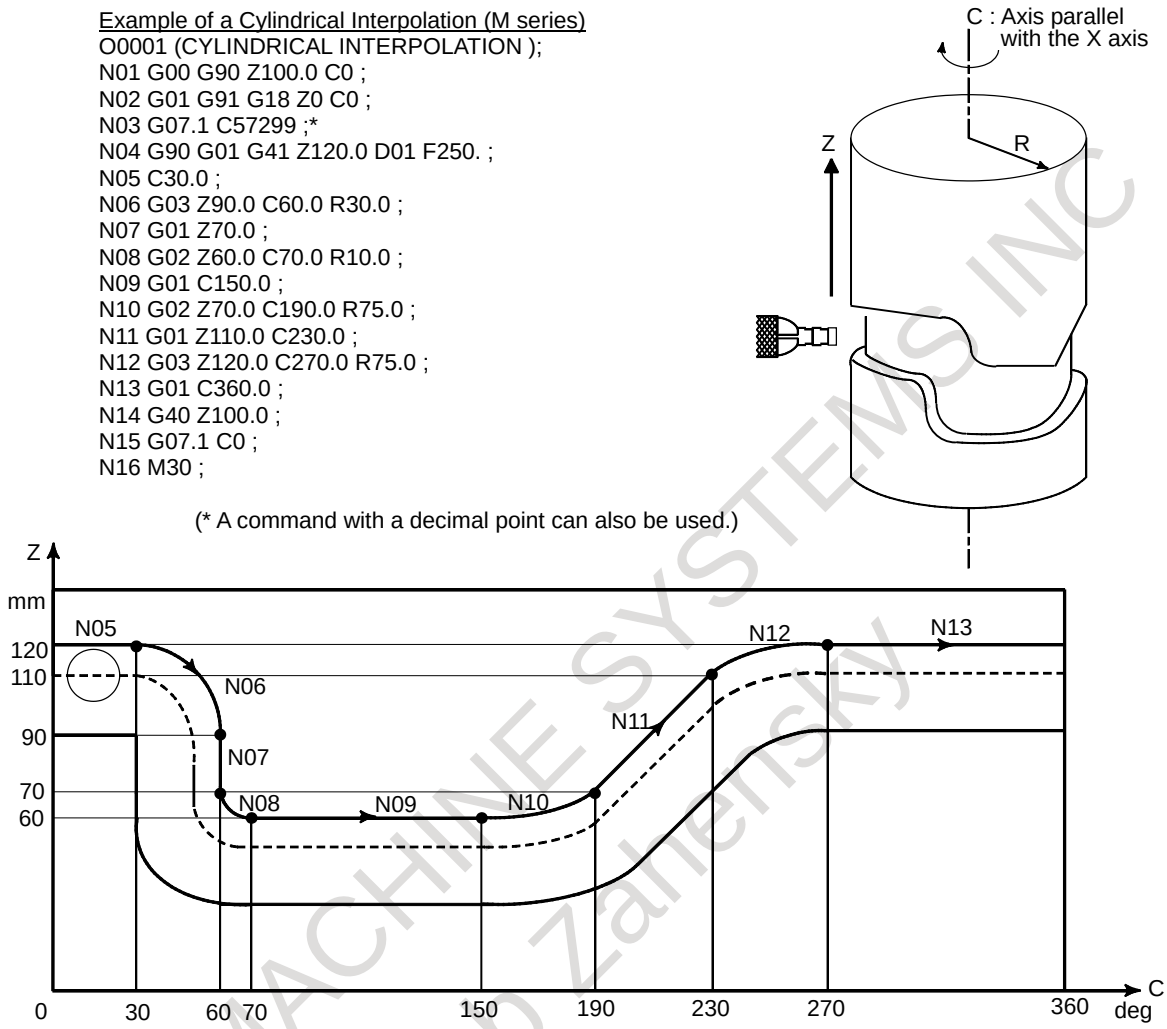
Example

M

Example of a Cylindrical Interpolation (M series)

```
O0001 (CYLINDRICAL INTERPOLATION );
N01 G00 G90 Z100.0 C0 ;
N02 G01 G91 G18 Z0 C0 ;
N03 G07.1 C57299 ;*
N04 G90 G01 G41 Z120.0 D01 F250. ;
N05 C30.0 ;
N06 G03 Z90.0 C60.0 R30.0 ;
N07 G01 Z70.0 ;
N08 G02 Z60.0 C70.0 R10.0 ;
N09 G01 C150.0 ;
N10 G02 Z70.0 C190.0 R75.0 ;
N11 G01 Z110.0 C230.0 ;
N12 G03 Z120.0 C270.0 R75.0 ;
N13 G01 C360.0 ;
N14 G40 Z100.0 ;
N15 G07.1 C0 ;
N16 M30 ;
```

(* A command with a decimal point can also be used.)



4.7.2 Cylindrical Interpolation by Plane Distance Command

Overview

In the conventional rotary axis command in cylindrical interpolation, the angle of the rotary axis is specified.

This function enables the rotary axis command in cylindrical interpolation to be specified by distance on the developed plane by setting parameters.

Format

G07.1 IP r; Starts the cylindrical interpolation mode (enables cylindrical interpolation).

:

G07.1 IP 0; The cylindrical interpolation mode is cancelled.

IP : One of address for the rotary axis

r : The radius of the workpiece

Specify G07.1 IPr; and G07.1 IP0; in separate blocks.

G107 can be used instead of G07.1.

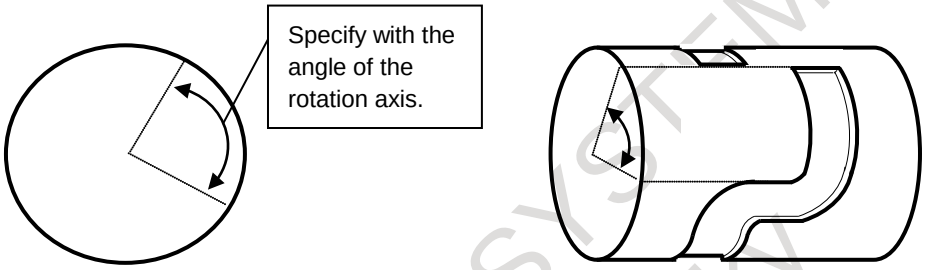
FRYER MACHINE SYSTEMS INC
Rod Zaharensky

NOTE
Only a positive value is effective as the radius of the workpiece. If a negative value is specified, alarm PS0175, "ILLEGAL G07.1 AXIS", is issued.

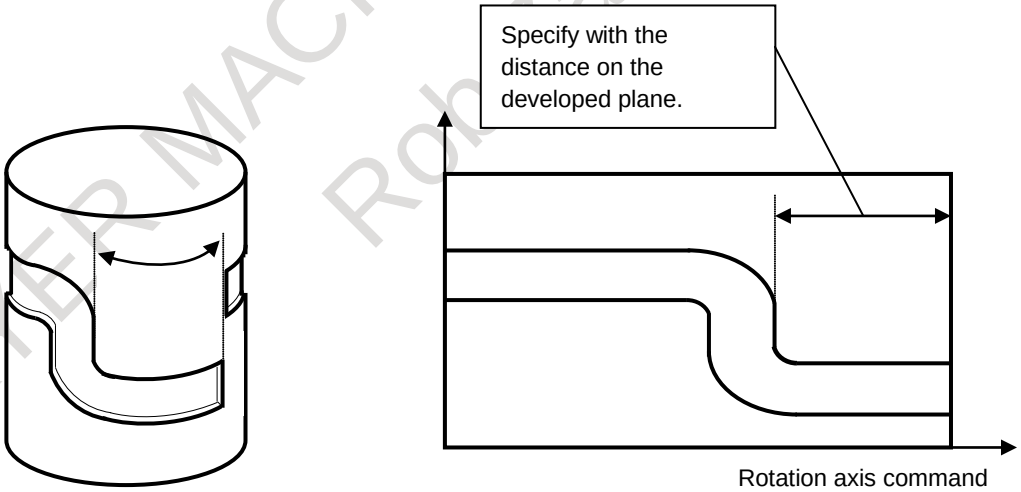
Explanation

By using bit 2 (DTO) of parameter No. 3454, it is possible to switch the rotation axis command during cylindrical interpolation between the angle of the rotation axis and the distance on the developed plane.

- In the case of the angle of the rotation axis (When bit 2 (DTO) of parameter No. 3454 is set to 0)
  The rotation axis command in cylindrical interpolation mode is executed with the angle of the rotation axis. From the program, specify the angle of the rotation axis that corresponds to the specified point on the developed plane.
  The rotation axis command uses the angle of the rotation axis [deg].



- In the case of the distance on the developed plane (When bit 2 (DTO) of parameter No. 3454 is set to 1)
  The rotation axis command in cylindrical interpolation is executed with the distance on the developed plane. The rotation axis command uses the distance on the developed plane and, therefore, the command unit varies depending on which of inch or metric input to use.



Note

NOTE
For details of the operation of cylindrical interpolation, as well as limitations, see Subsection, "Cylindrical Interpolation" in OPERATOR'S MANUAL (Common to Lathe System/Machining Center System) (B-64694EN).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

4.7.3 Cutting Point Interpolation for Cylindrical Interpolation

The conventional cylindrical interpolation function controls the tool center so that the tool axis always moves along a specified path on the cylindrical surface, towards the rotary axis (cylindrical axis) of the workpiece. The cutting point interpolation for cylindrical interpolation function controls the tool so that the tangent of the tool and the cutting surface of a contour always passes the rotation center of the workpiece. This means that the cutting surface of the contour is always perpendicular to the cylinder. With this function, the figure on the cutting surface can always be kept constant regardless of the tool radius compensation value of the tool used.

Format

This command is specified in the same way as for the conventional cylindrical interpolation function.

G07.1 IPr ; Circular interpolation mode on start (enabling cylindrical interpolation)

:

G07.1 IP0 ; Circular interpolation mode cancel

IP : One rotary axis address
r : Cylinder radius of rotary axis
Specify each of G07.1 IPr; and G07.1 IP0; singly in a block.
G107 can also be specified instead of G07.1.

NOTE

Only a positive value is effective as the radius of the workpiece. If a negative value is specified, alarm PS0175, "ILLEGAL G07.1 AXIS", is issued.

Explanation

- Comparison with conventional circular interpolation

As shown in Fig. 4.7.3 (a), the tool is controlled in the offset axis (Y-axis) direction that is perpendicular to the tool center and workpiece rotation center.

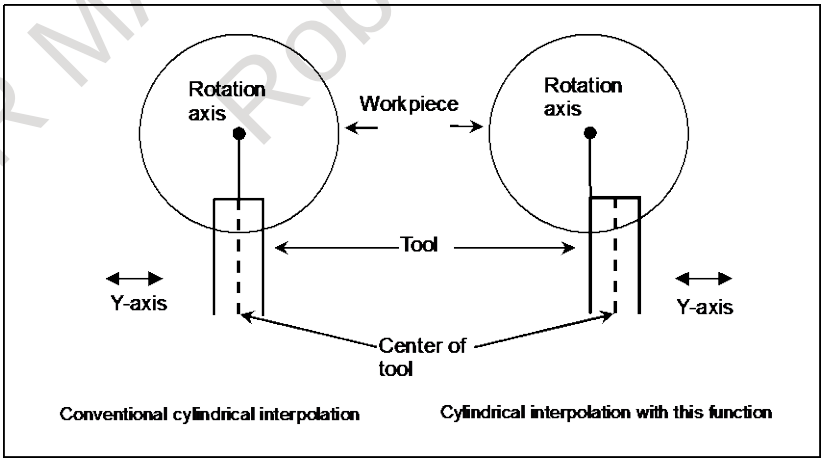


Fig. 4.7.3 (a) Comparison with conventional circular interpolation

FRYER MACHINE SYSTEMS INC
Rob Zaherosky

- Cutting point compensation

(1) Cutting point compensation between blocks

As shown in Fig. 4.7.3 (b), cutting point compensation is achieved by moving between blocks N1 and N2.

(a) Let C1 and C2 be the heads of the vectors normal to N1 and N2 from S1, which is the intersection of the tool center paths of blocks N1 and N2

(b) After the tool moves to S1 according to the command of N1, the tool moves through V on the C-axis as a result of cutting point compensation, then through $-V \times \frac{\pi}{180} \times r$ along the Y-axis.

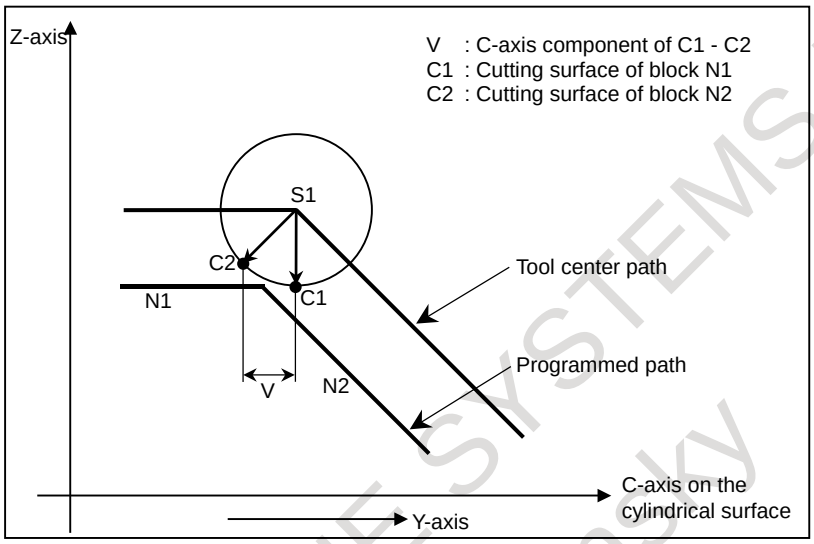


Fig. 4.7.3 (b) Cutting point compensation between blocks

(2) Cutting point compensation in a circular command block

As shown in Fig.4.7.3 (c), the movement required for cutting point compensation is made simultaneously with circular interpolation in block N1.

(a) Let C0 be the head of the vector normal to N1 from S0, which is the tool center position at the start point of circular block N1. Let C1 be the head of the similar vector at the end point.

(b) As the tool moves from S0 to S1, a superimposed movement is made by the C-axis component of (C1 - C0) (V in the figure) on the C-axis, and a superimposed movement is made by $-V \times \frac{\pi}{180} \times r$ along the Y-axis. along the Y-axis.

That is, the following expressions are valid. As movement is made through L as shown in Fig.4.7.3 (c), the superimposed movements are made on the C-axis and Y-axis as follows:

$$\Delta C = \Delta V$$

$$\Delta Y = -\frac{\pi}{180}(\Delta V)r$$

ΔV : Cutting point compensation value (ΔV2 - ΔV1) for movement of ΔL

ΔV1 : C-axis component of the vector normal to N1 from the tool center of the start point of ΔL

ΔV2 : C-axis component of the vector normal to N1 from the tool center of the end point of ΔL

R : Arc radius

FRYER MACHINE SYSTEMS INC
Rob Zalensky

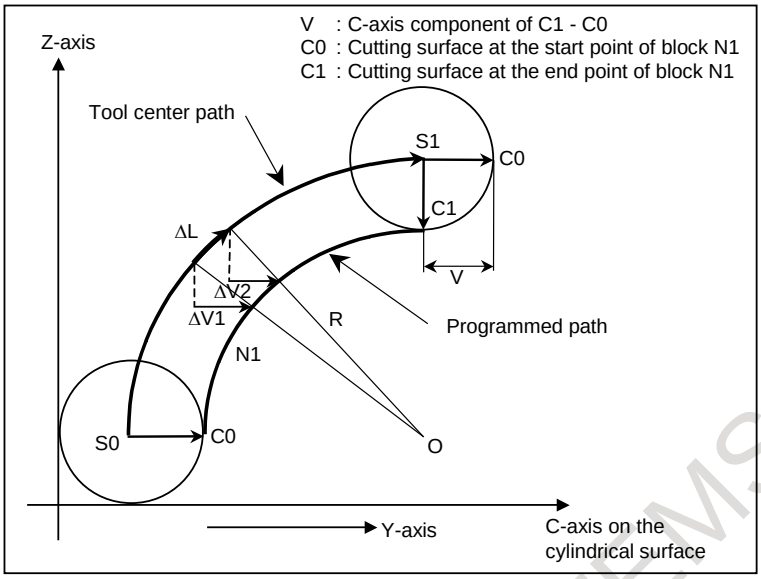


Fig.4.7.3 (c) Cutting point compensation in a circular command block

(3) When cutting point compensation is not applied between blocks

When, as shown in Fig.4.7.3 (d) and Fig.4.7.3 (e), the cutting point compensation value between blocks N1 and N2 (V in the Fig.4.7.3 (d) and Fig.4.7.3 (e)) is less than the value set in parameter No. 19534, one of the operations below is performed. (The operation that is performed depends on the setting of bit 6 (CYS) of parameter No. 19530.

(a) When bit 6 (CYS) of parameter No.19530 is set to 1

Cutting point compensation is not applied between blocks N1 and N2, but is applied when block N2 is executed.

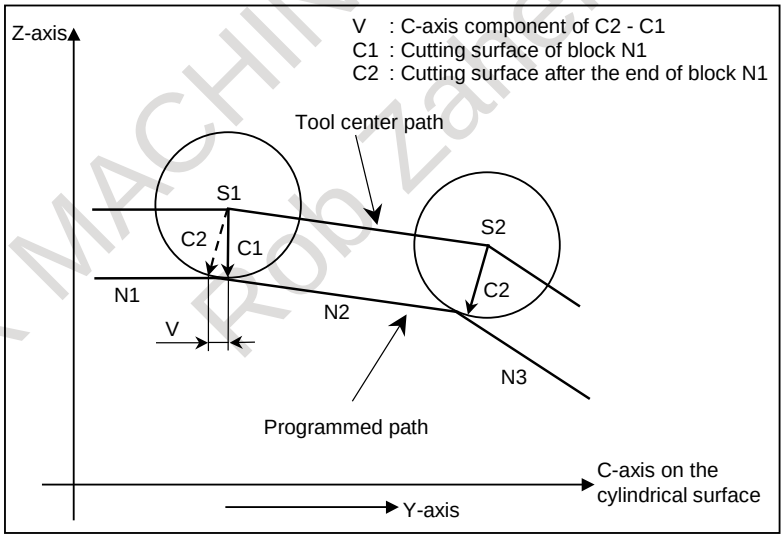


Fig.4.7.3 (d) When bit 6 (CYS) of parameter No. 19530 is set to 1

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

(b) When bit 6 (CYS) of parameter No. 19530 is set to 0
Cutting point compensation is not performed between blocks N1 and N2. Whether to apply cutting point compensation between block N2 and N3 is determined by taking the cutting point compensation value between blocks N2 and N3 (V in the Fig.4.7.3 (e)) into consideration.

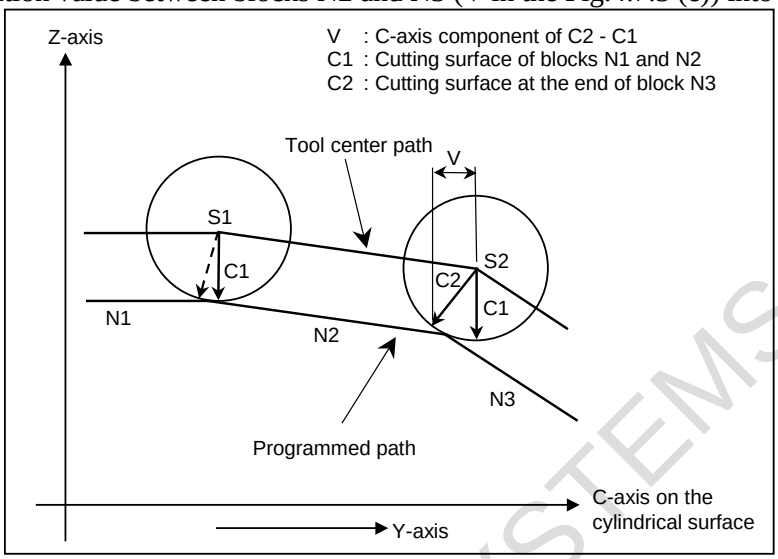


Fig.4.7.3 (e) When bit 6 (CYS) of parameter No. 19530 is 0

(c) When the amount of travel (L1) of block N2 is less than the value set in parameter No. 19535, as shown in Fig.4.7.3 (f), cutting point compensation is not applied between blocks N1 and N2. Instead, block N2 is executed with the cutting point compensation of the previous block. When the amount of travel (L2) of block N3 is greater than the value set in parameter No. 19535, cutting point compensation is applied between blocks N2 and N3.

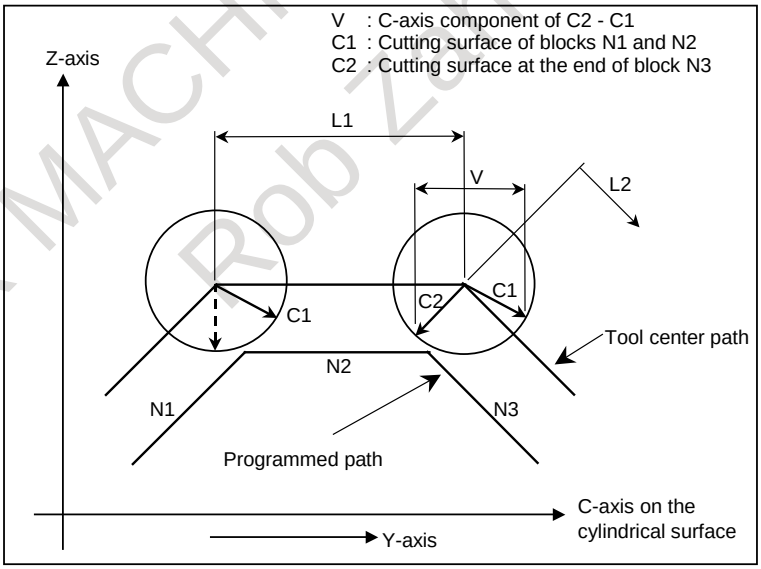


Fig.4.7.3 (f) When the amount of travel (L1) of block N2 is less than the parameter value

FRYER MACHINE SYSTEMS INC
Rob Zaherosky

(d) When, as shown in Fig.4.7.3 (g), the diameter of an arc (R in the figure) is less than the value set in parameter No. 19535, cutting point compensation is not applied simultaneously with circular interpolation

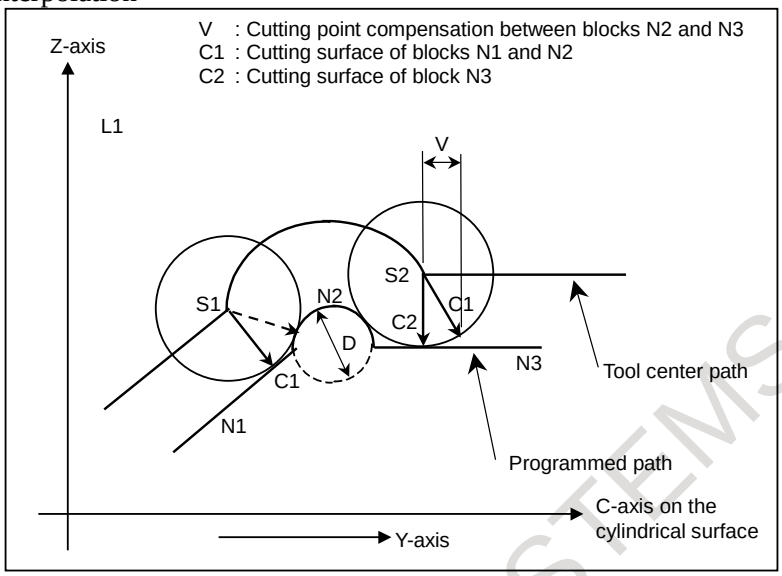


Fig.4.7.3 (g) When the diameter of an arc is less than the parameter value

- When cutting point compensation is used with normal direction control

When cutting point compensation is used with normal direction control, cutting point compensation between specified blocks is performed regardless of the method described in "**Cutting point compensation**" above, but is performed simultaneously with the movement of the normal direction control axis (C-axis).

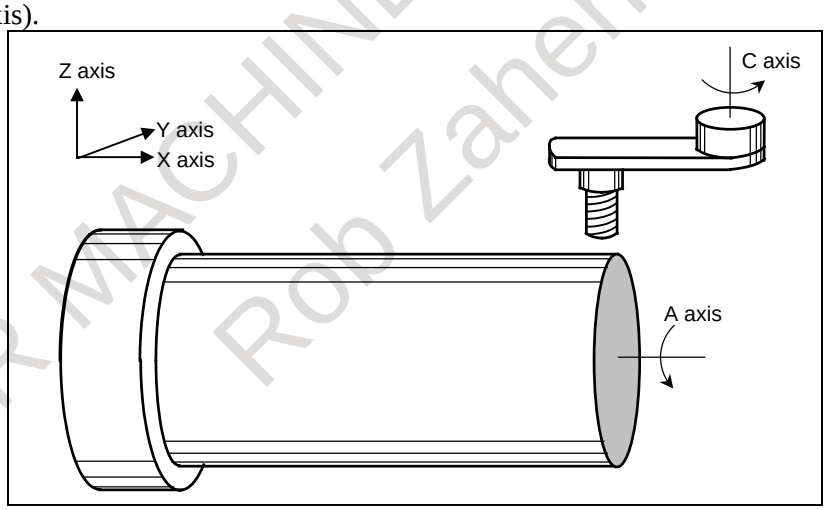


Fig. 4.7.3 (h) When used with normal direction control

FRYER MACHINE SYSTEMS INC
Rob Zahensky

(1) When the normal direction changes between blocks N1 and N2, cutting point compensation is also performed between blocks N1 and N2.
As shown in Fig. 4.7.3 (i), cutting point compensation described in (1) in "**Cutting point compensation**" is performed simultaneously with the movement made by normal direction control between blocks N1 and N2.

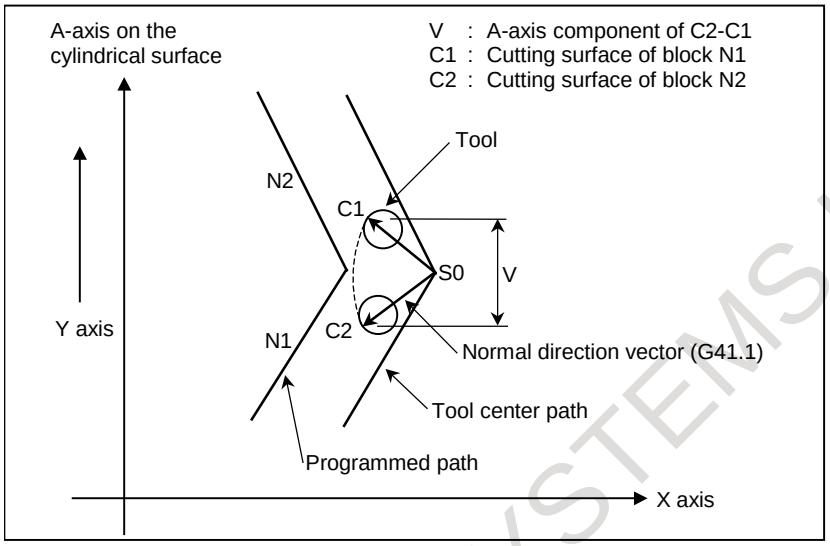


Fig. 4.7.3 (i) When the normal direction changes between blocks N1 and N2

(2) When a specified block is executed while the normal direction control axis is held in the normal direction set at the end point of the previous block, cutting point compensation is not performed, and cutting point compensation applied in the previous block is maintained. As shown in Fig. 4.7.3 (j), when the movement amount in N2 (L1 in the figure) is smaller than the setting of parameter No. 5483, so the normal direction control axis does not rotate at S1 while the movement amount in N3 (L2 in the figure) is greater than the setting of parameter No. 5483, so the normal direction axis rotates at S2, cutting point compensation is not performed at S1 but is performed by the movement for vector V2 at S2.

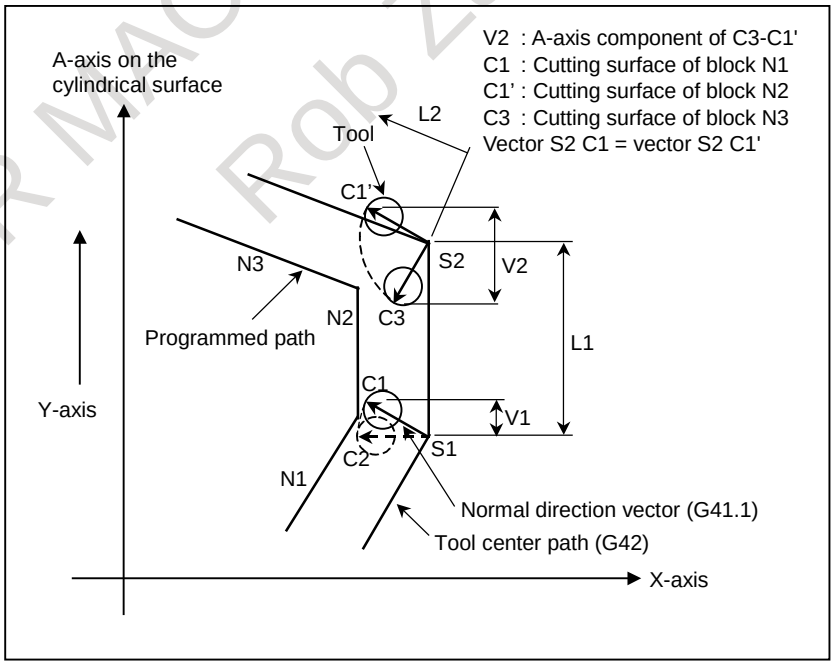


Fig. 4.7.3 (j) When the normal direction is the same as in the previous block

FRYER MACHINE SYSTEMS INC
Rob Zaretsky

- Feedrate during cutting point compensation

(1) The tool moves at a specified feedrate while cutting point compensation is being applied between blocks.

(2) The actual speed indication and feedrate during circular interpolation are as described below.
Actual speed indication
The speed component of each axis after cutting point compensation at a point in time during circular interpolation is as follows:

$Fz' = Fz$: Speed component of linear axis

$Fc' = Fc + (Vce - Vcs)$: Speed component of rotary axis

$Fy' = -(Vce - Vcs)\frac{\pi r}{180}$: Speed component of offset axis

Fz : Speed component of a cylindrical interpolation linear axis before cutting point compensation
Fc : Speed component of cylindrical interpolation rotation axis before cutting point compensation
Vcs: Rotation axis component of a tool contact point vector (Vs in the figure) at the start point at a point in time
Vce: Rotation axis component of tool contact point vector (Ve in the figure) at the end point at a point in time
r : Radius of the cylinder of a rotary axis

Accordingly, the actual speed indication during circular interpolation is greater than the specified value when |Fc'| > |Fc|(inner offset of the arc). Conversely, the actual speed indication during circular interpolation is less than the specified value when |Fc'| < |Fc|(outer offset of the arc).

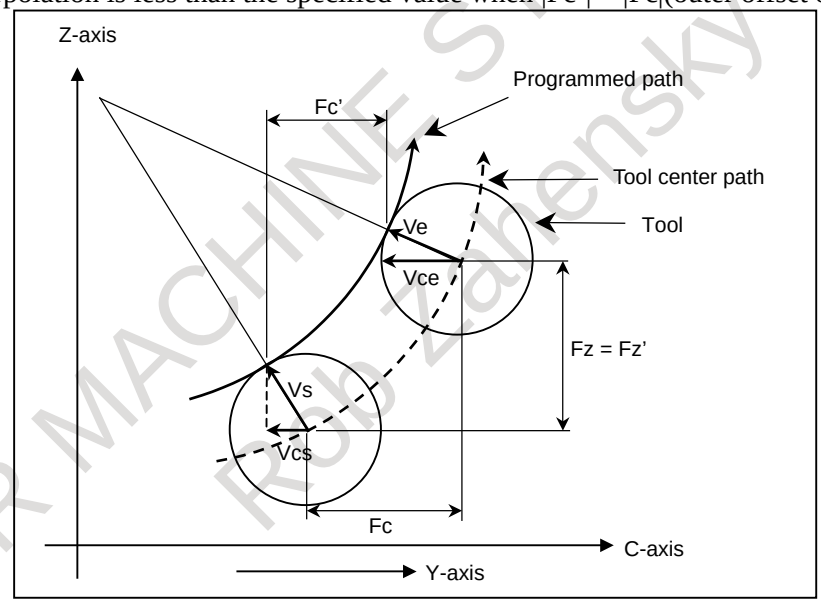


Fig. 4.7.3 (k) Actual speed indication during circular interpolation

- Usable G codes

(1) In any of the following G code modes, cutting point interpolation for cylindrical interpolation can be specified:

G01, G02, G03	: Linear interpolation, circular interpolation
G17, G18, G19	: Plane selection
G22	: Stored stroke check function on
G64	: Cutting mode
G90, G91	: Absolute programming, incremental programming
G94	: Feed per minute

(2) Any of the following G codes can be specified in cutting point interpolation for cylindrical interpolation mode:

G01, G02, G03	: Linear interpolation, circular interpolation
G04	: Dwell

FRYER MACHINE SYSTEMS INC
Rob Zahensky

G40, G41, G42	: Tool radius compensation
G40.1 to G42.1	: Normal direction control
G64	: Cutting mode
G65 to G67	: Macro call
G90, G91	: Absolute programming, incremental programming

- Parameter

To enable this function, set bit 5 (CYA) of parameter No. 19530 to 1.

Limitation

- Overcutting during inner corner cutting

Theoretically, when the inner area of a corner is cut using linear interpolation as shown in Fig.4.7.3 (l), this function slightly overcuts the inner walls of the corner. This overcutting can be avoided by specifying a value of R that is slightly greater than the radius of the tool at the corner.

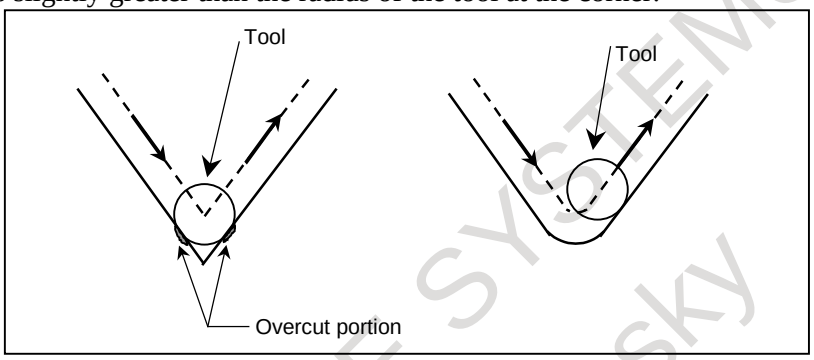


Fig. 4.7.3 (l) Overcutting

- Setting the minimum input increment for an offset axis (Y-axis)

Set the same minimum input increment for an offset axis and linear axis when cylindrical interpolation is performed.

- Workpiece radius specification

When specifying the radius of a workpiece, use the minimum input increment (with no decimal point) for the linear axis used in cylindrical interpolation.

- Reference axis setting (parameter No. 1031)

When different increment systems are used for a linear axis and a rotary axis in cylindrical interpolation, specify the number of the linear axis in cylindrical interpolation to set the axis as the reference axis.

- Rotary axis roll-over function

When a rotary axis for which the rollover function is used is specified as a rotary axis to be used for cylindrical interpolation, the roll-over function is disabled in the cylindrical interpolation mode.
When the cylindrical interpolation is canceled, the rollover function is automatically enabled.

- Program restart

In a program restart operation, G07.1 for the cylindrical interpolation mode must not be specified.
Otherwise, alarm PS0175, "ILLEGAL G07.1 AXIS", is issued.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Example

- Example of cutting point interpolation for cylindrical interpolation

The sample program below indicates the positional relationships between a workpiece and tool.
Tool radius compensation No.01 is 30 mm.

```
O0001 (CYLINDRICAL INTERPOLATION1) ;
N01 G00 G90 Z100.0 C0 ;
N02 G01 G91 G19 Z0 C0 ;
N03 G07.1 C57299 ;
N04 G01 G42 G90 Z120.0 D01 F250. ;.................(1)
N05 C20.0 ;     ....................................(2)
N06 G02 Z110.0 C60.0 R10.0 ; .......................(3)
N07 G01 Z100.0 ;....................................(4)
N08 G03 Z60.0 C70.0 R40.0 ; ........................(5)
N09 G01 C100.0 ;
   :
M30 ;
```

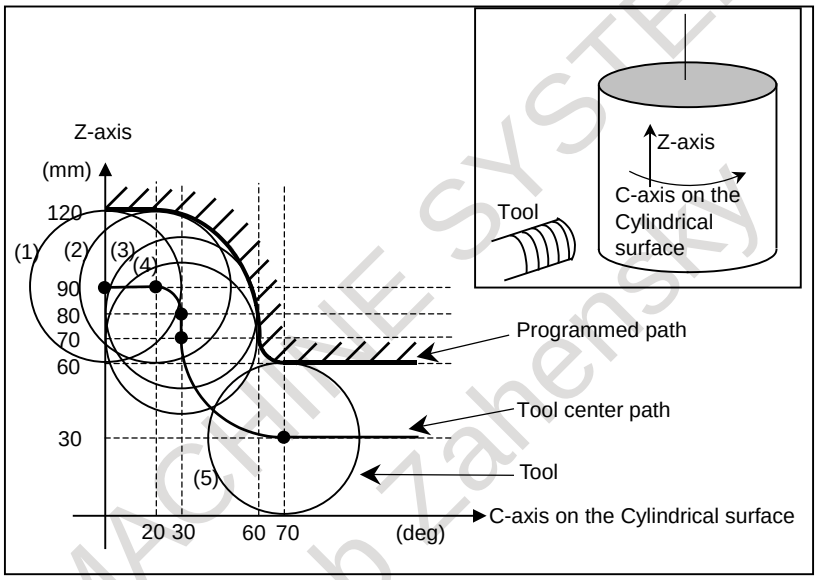


Fig. 4.7.3 (m) Path of sample program for cutting point interpolation for cylindrical interpolation

FRYER MACHINE SYSTEMS INC
Rob Zahensky

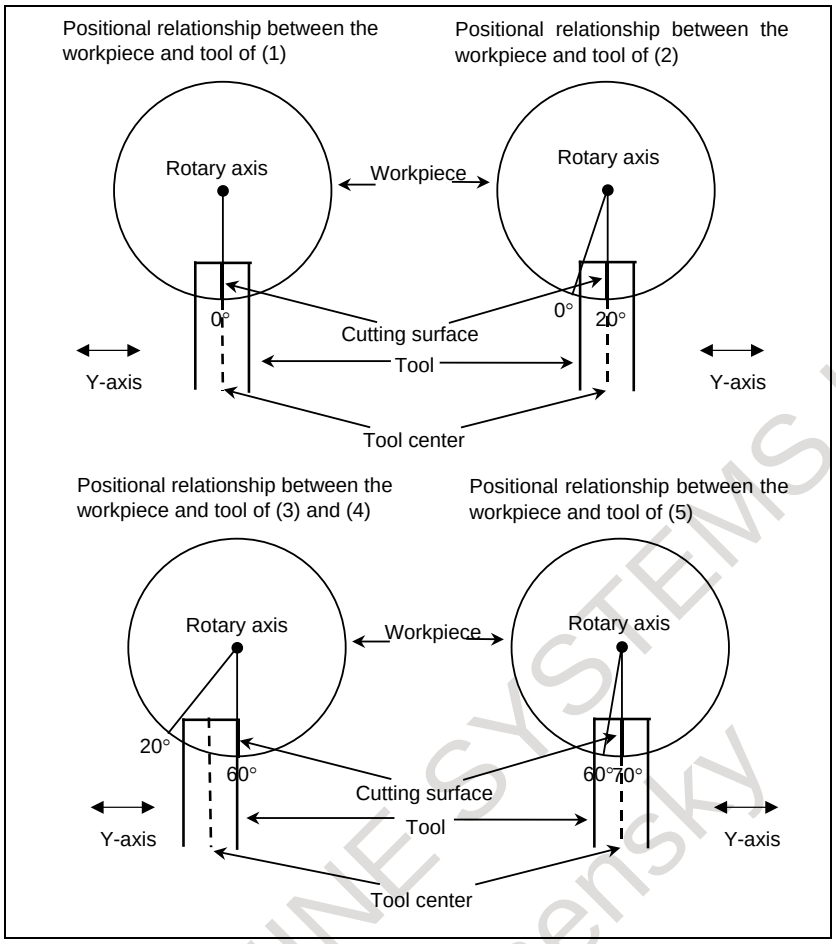


Fig. 4.7.3 (n) Positional relationships between workpiece and tool of sample program

The cutting surface in the rotary axis direction in (3) and (4) are uniform even if the tool radius compensation amount is modified.

- Example of specifying cutting point interpolation for cylindrical interpolation and normal direction control at the same time

Tool radius compensation No.01 is 30 mm.

```
O0002 (CYLINDRICAL INTERPOLATION2) ;
N01 G00 G90 X100.0 A0 ;
N02 G01 G91 G17 X0 A0 ;
N03 G07.1 C57299 ;
N04 G01 G41 G42.1 G90 X120.0 D01 F250. ;
N05 A20.0 ;
N06 G03 X80.0 A60.0 R40.0 ;
N07 G01 X70.0 ;
N08 G02 X70.0 A70.0 R10.0 ;
N09 G01 A150.0 ;
N10 G02 X70.0 A190.0 R85.0 ;
N11 G01 X110.0 A265.0 ;
N12 G03 X120.0 A305.0 R85.0 ;
N13 G01 A360.0 ;
N14 G40 G40.1 X100.0 ;
N15 G07.1 A0 ;
N16 M30 ;
```

FRYER MACHINE SYSTEMS INC
Rob Zamensky

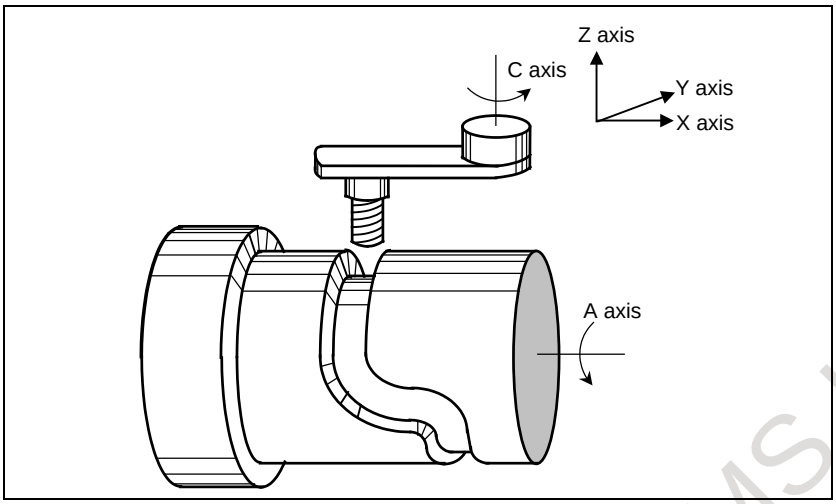


Fig. 4.7.3 (o) Example of specifying normal direction control at the same time

4.8 SMOOTH TOLERANCE+ CONTROL

M

Smooth tolerance+ control is a function that generates smooth machining path within specified tolerance. In the case of die mold machining program that consists of small line segments, this function acts as smoothing function and makes joints of blocks smooth. As the result, machining surface gets high quality.

Smoothing small line segments

This function smooths small line segments. If the length of linear blocks is less than the value set at parameter No. 19595, those blocks are treated as small line segment blocks, and smooth curves not depending on commanded points are generated, which are within tolerance from commanded points.
Smoothing tolerance is set as same value as CAM tolerance which generating small line segments. Tolerance is specified by parameters (No.11786, 11787) or NC program (G10.8L4Q_R_).

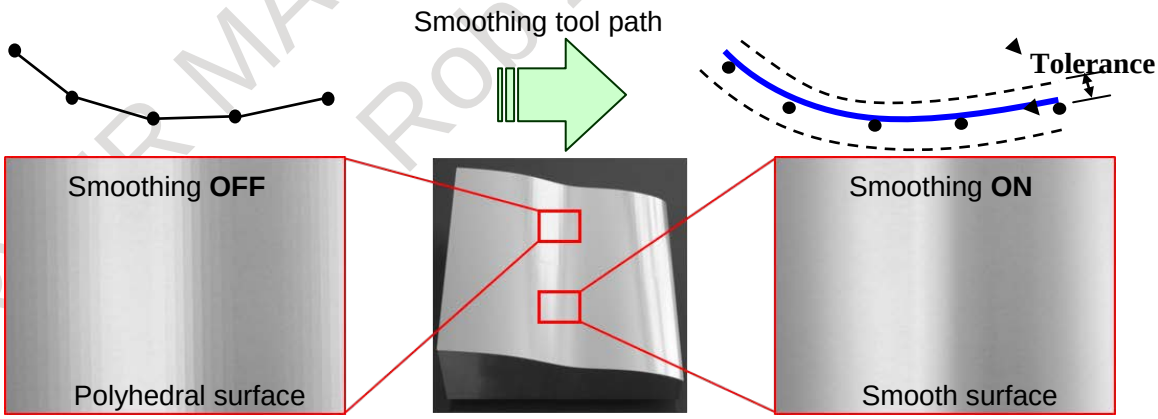


Fig.4.8 (a) Smoothing small line segments

FRYER MACHINE SYSTEMS INC
Rob Zahrensky

Command format (In Case of 3-axis Machining)

G05.1 Q3 Xp0 Yp0 Zp0;	**Smooth tolerance$^+$ control mode on**
G10.8 L4 Q_;	**Specify tolerance**
:	
G05.1 Q0;	**Smooth tolerance$^+$ control mode off**

Xp: X-axis or an axis parallel to the X-axis
Yp: Y-axis or an axis parallel to the Y-axis
Zp: Z-axis or an axis parallel to the Z-axis
Q: Tolerance for linear axis on curves

NOTE

1. Specify G05.1 alone in a block.
   (To avoid specifying any other G code in the same block)
2. Unit of “Q” in G10.8 command depends on the increment system of the basic axis.
3. The value of parameter No. 11786 is effective as tolerance between G05.1 Q3 command and G10.8 L4 command.
4. Smooth tolerance$^+$ control is disabled when the tolerance for linear axis is set to 0.
5. Smooth tolerance$^+$ control mode is activated at the start of automatic operation by setting bit 0 (CAT) of parameter No. 11785.

Description of the Function

- Automatically turning on AI contour control with Smooth tolerance$^+$ control

Specifying G5.1 Q3 also enables Smooth tolerance$^+$ control and AI contour control to be turned on at the same time.
The automatic velocity control by AI contour control reduces impacts on the mechanical system.

- Conditions for enabling Smooth tolerance$^+$ control

Smooth tolerance$^+$ control is enabled when the following conditions are satisfied.
In a block that does not satisfy the conditions for enabling it, Smooth tolerance$^+$ control is canceled, and it is judged
in the next block whether to perform Smooth tolerance$^+$ control again.

- The mode is all of the following:
  - Cutting mode (G64)
  - Linear interpolation (G01)
  - Feed per minute (G94, however G98 in G code system A of T series)
    or Feed per revolution(G95, however G99 in G code system A of T series)
  - Constant surface speed control cancel (G97)
  - Normal direction control cancel (G40.1)
  - Polar coordinate command cancel (G15)
- The block does not contain a one shot G code command.
- The block does not contain MST command.
- The block does not suppress look ahead (buffering).
- The block contains a move for at least a linear axis.
- The block contains a move command for only axes subject to Smooth tolerance$^+$ control.
- The difference between a radius at the beginning point and a radius at the end point is less than 20μm.
- Smooth tolerance$^+$ control disable signal STCD <Gn599.2> is “0”.
- Block length is less than the value of parameter No. 19595.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Conditions for canceling making corner path into curve

In Smooth tolerance+ control mode, making corner path into curve is canceled at the connection of blocks on the following conditions.

1. In case moving direction is inverted between linear interpolation blocks.
2. In case moving direction is inverted between circular interpolations blocks whose central coordinate values are exact the same.
3. In case moving direction is same between linear interpolation blocks.
4. In case between curves represented by small segment, or between a curve represented by small segment and linear / circular block.

On the conditions 1 or 2, movements of axes temporary stop at the connection of blocks.

- Interlock

When an axis for Smooth tolerance+ control is interlocked, all axes are interlocked in the block which Smooth tolerance+ control is enabled, even if the interlocked axis is not commanded in the block.

- Axis moving signals

In the block which Smooth tolerance+ control is enabled, axis moving signals MV1 to MV8 <Fn102> for axes for Smooth tolerance+ control are set to one regardless of the movement.

- Use with other functions

In the case Smooth tolerance+ control is used with the following functions, it controls paths which each function is applied.

- Cutter compensation and tool nose radius compensation
- Tool length compensation
- Programmable mirror image
- Scaling
- Coordinate system rotation

- Status display in the Smooth Tolerance+ control mode

Character string “TOLCON” blinks in the status bar when Smooth tolerance+ control mode is enabled.

Fig.4.8 (b) Status display in the Smooth Tolerance+ control mode

4.8.1 Change Tolerance in Smooth Tolerance+ Control Mode

M

The tolerance in smooth tolerance+ control mode can be changed any time by specifying G10.8 L4. The appropriate tolerance can be used which depends on situations in a program.

Format

Change Tolerance in Smooth Tolerance+ Control Mode

G10.8 L4 Q_ R_;	Directly specify tolerance
G10.8 L4;	Use paramter (No.11786, No.11787) as default tolerance
Q:	Tolerance for linear axis on curves
R:	Tolerance for rotary axis on curves

NOTE

1 Specify G10.8 alone in a block.
(Avoid specifying any other G code in the same block)

2 G10.8 L4 is a one-shot G code.

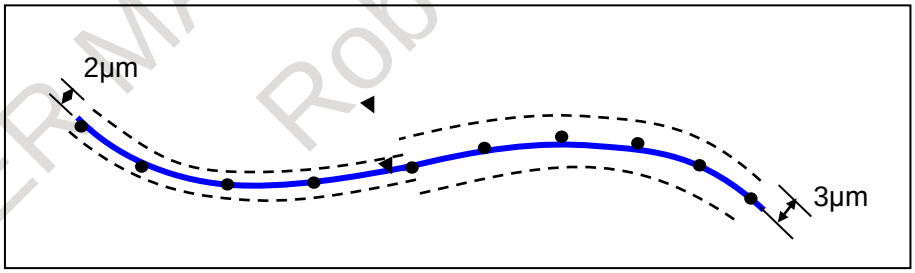
BRYER MACHINE SYSTEMS INC
Roman Zahensky

3 Unit of “Q and “R” in G10.8 command depends on the increment system of the basic axis.
4 Specifying a negative value to “Q” or “R” causes alarm PS2010, “ILL. COMMAND IN TOLERANCE CON.”.
5 When G10.8 L4 specified not in smooth tolerance+ control mode (G05.1 Q3), alarm PS0412, “ILLEGAL G CODE” is issued.
6 The value of parameter No. 11786 and No. 11787 do not change by specifying G10.8 L4.

Example

Here is an example for changing tolerance in smoothing small line segments.

```
O0012;
G28 G91 X0 Y0;
G05.1 Q3 X0 Y0 Z0;          (Smooth tolerance+ control mode on)
G10.8 L4 Q0.002;            (Tolerance is set.)
G00 G90 X100.0 Y100.0 Z100.0;
G01 X100.121 Y100.321 F1000;
X100.121 Y100.321;
X100.321 Y100.881;
X100.721 Y101.365;          Smoothing (Tolerance = 2μm)
  :
  :
G10.8 L4 Q0.003;            (Tolerance is changed.)
X110.121 Y120.321;
X110.321 Y120.881;
X110.721 Y121.365;          Smoothing (Tolerance = 3μm)
  :
  :
G05.1 Q0;                   (Smooth tolerance+ control mode off)
M30;
```



NOTE

- Functions Which Smooth tolerance+ control is Temporary Disabled

This function is temporary disabled when the following functions are used together.

Functions
Any kind of interpolations except liner interpolation.
Inverse time feed
Constant surface speed control cancel
Normal direction control
Polar coordinate command
Polar coordinate interpolation
Polygon turning
Tool length compensation command block
Canned cycle

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Background graphic display

The background graphic display function draws the path in Smooth tolerance+ control mode by linear interpolation.

- Functions Which Does Not Work with Smooth Tolerance+ Control

This function doesn't work with the following functions.

Functions	Alarm (1)
Interruption type custom macro	PS2012

(1) Functions which issue alarms in case using with Smooth tolerance+ control.

4.9 VARIABLE LEAD THREADING (G34)

T

Specifying an increment or a decrement value for a lead per screw revolution enables variable lead threading to be performed.

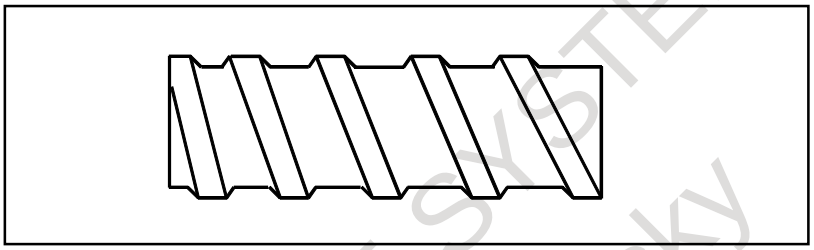


Fig. 4.9 (a) Variable lead screw

Format

G34 IP_ F_ K_ Q_ ;

IP_ : End point
F_ : Lead in longitudinal axis direction at the start point
K_ : Increment and decrement of lead per spindle revolution
Q_ : Shift amount of starting angle of thread cutting

Explanation

Address other than K are the same as in straight/taper thread cutting with G32.
The K value depends on the increment system of the reference axis, as indicated in Table 4.9 (a).
If the specified K value exceeds the range indicated in Table 4.9 (a), if the maximum lead is exceeded after a change due to the K value, or if the lead value is negative, an alarm PS0313, "ILLEGAL LEAD COMMAND", will be issued.

Table 4.9 (a) Range of valid K values

Increment system of reference axis	Metric input (mm/rev)	Inch input (inch/rev)
IS-A	±0.001 to ±500.000	±0.00001 to ±50.00000
IS-B	±0.0001 to ±500.0000	±0.000001 to ±50.000000
IS-C	±0.00001 to ±50.00000	±0.0000001 to ±5.0000000

Table4.9 (b) Range of valid lead values

Metric input (mm)	Inch input (inch)
0.0001 to 500.0000	0.000001 to 50.000000

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Continuous threading

The "continuous threading" is effective for G34.

> ⚠ **CAUTION**
> The "thread cutting cycle retract" is not effective for G34.

- Multiple threading

Multiple threading can be performed by specifying the Q value.
As for specifying the Q value, refer to " MULTIPLE THREADING ".

Example

If, in the increment system IS-B,
G34 Z-72.0 F8.0 K0.3 ;
is specified,

Lead at the start point: 8.0 mm
Lead increment: 0.3 mm/rev

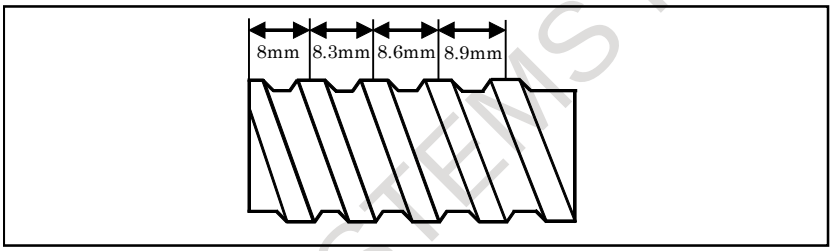


Fig.4.9 (b) Example of variable lead threading

4.10 CIRCULAR THREADING (G35, G36)

T

Using the G35 and G36 commands, a circular thread, having the specified lead in the direction of the major axis, can be machined.

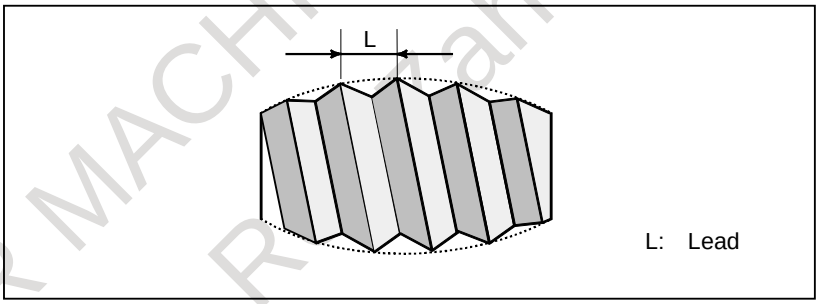


Fig. 4.10 (a) Circular threading

FRYER MACHINE SYSTEMS INC
R. Zahensky

Format

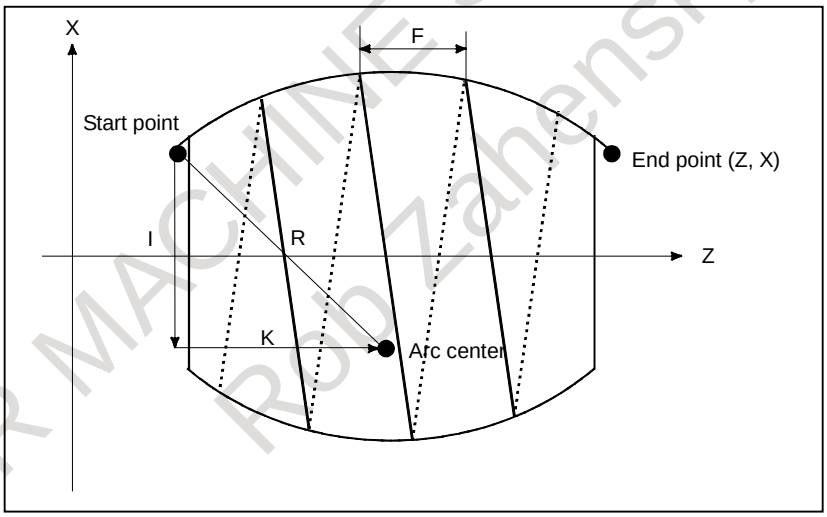
A sample format for the G18 plane (Z-X plane) is indicated below. In the G17 plane (X-Y plane) and the G19 plane (Y-Z plane), a sample format can be specified by following format. When using the format for the G17 plane (X-Y plane), change the addresses Z, X, K, and I to X, Y, I, and J respectively. When using the format for the G19 plane (Y-Z plane), change the addresses Z, X, K, and I to Y, Z, J, and K respectively.

$$\begin{Bmatrix} \mathbf{G35} \\ \mathbf{G36} \end{Bmatrix} \ \mathbf{X(U)\_\ Z(W)\_} \ \begin{Bmatrix} \mathbf{I\_\ K\_} \\ \mathbf{R\_} \end{Bmatrix} \ \mathbf{F\_\ Q\_\ ;}$$

G35 : Clockwise circular threading command
G36 : Counterclockwise circular threading command

X(U), Z(W) : Specify the arc end point (in the same way as for G02, G03).

I, K : Specify the arc center relative to the start point, using relative coordinates (in the same way as for G02, G03).
R : Specify the arc radius.
F : Specify the lead in the direction of the major axis.
Q : Specify the shift of the threading start angle
(0° to 360°, with least input increment of 0.001)
(The value cannot be programmed with a decimal point.)



Explanation

- Specifying the arc radius

If R is specified with I and K, only R is effective.

- Shift angle

If an angle greater than 360° is programmed, it is set to 360°.

- Continuous threading

The "continuous threading" is effective for G35, G36.

- Thread cutting cycle retract

The "thread cutting cycle retract" is not effective for G35, G36.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Automatic tool compensation

The G36 command is used to specify the following two functions: Automatic tool compensation X and counterclockwise circular threading. The function for which G36 is to be used depends on bit 3 (G36) of parameter No. 3405.

- When parameter G36 is set to 0, the G36 command is used for automatic tool compensation X.
- When parameter G36 is set to 1, the G36 command is used for counterclockwise circular threading.

G37.1 can be used to specify automatic tool compensation X and G37.2 can be used to specify automatic tool compensation Z.

(Specification method)
G37.1 X_
G37.2 Z_

- G code when bit 3 (G36) of parameter No. 3405 is set to 1

G code	G code group	Function
G35	01	Clockwise circular threading
G36		Counterclockwise circular threading
G37	00	Automatic tool compensation Z
G37.1		Automatic tool compensation X
G37.2		Automatic tool compensation Z

Limitation

- Range of specifiable arc

An arc must be specified such that it falls within a range in which the major axis of the arc is always the Z-axis or always the X-axis, as shown in Fig. 4.10 (b) and Fig. 4.10 (c). If the arc includes a point at which the major axis changes from the X-axis to Z-axis, or vice versa, as shown in Fig. 4.10 (d), an alarm PS5058, "G35/G36 FORMAT ERROR", is issued.

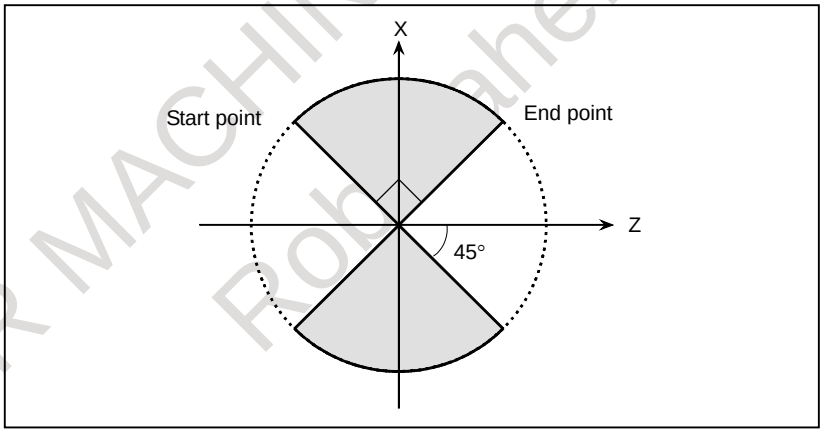


Fig. 4.10 (b) Range in which the Z-axis is the major axis

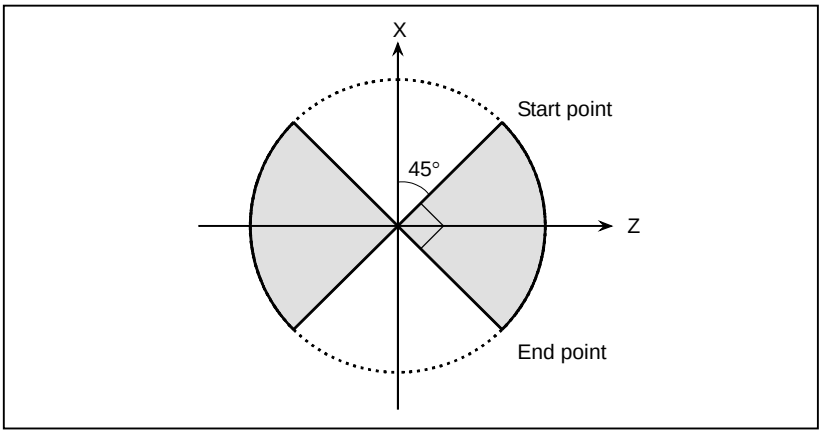


Fig. 4.10 (c) Range in which the X-axis is the major axis

FRYER MACHINE SYSTEMS INC
Robert Lahensky

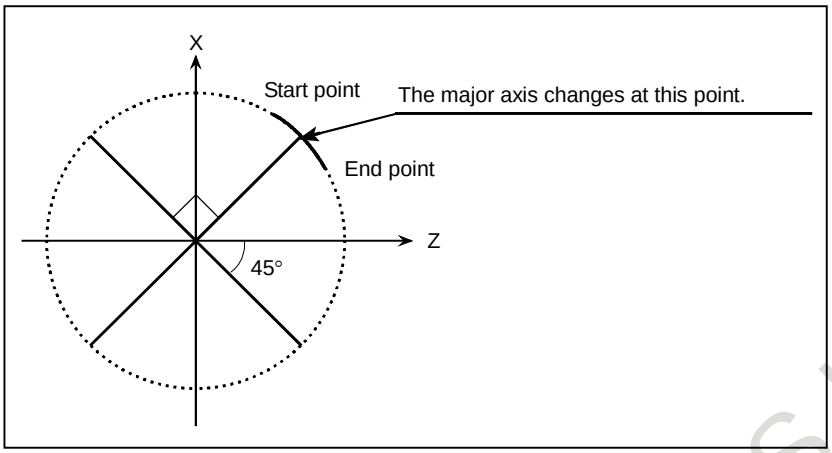


Fig. 4.10 (d) Example of arc specification which causes an alarm

- End point not on an arc

If the end point is not on an arc, a movement on an axis is made to a position of which coordinate matches the corresponding coordinate of the end point. Then, a movement is made on another axis to reach the end point.

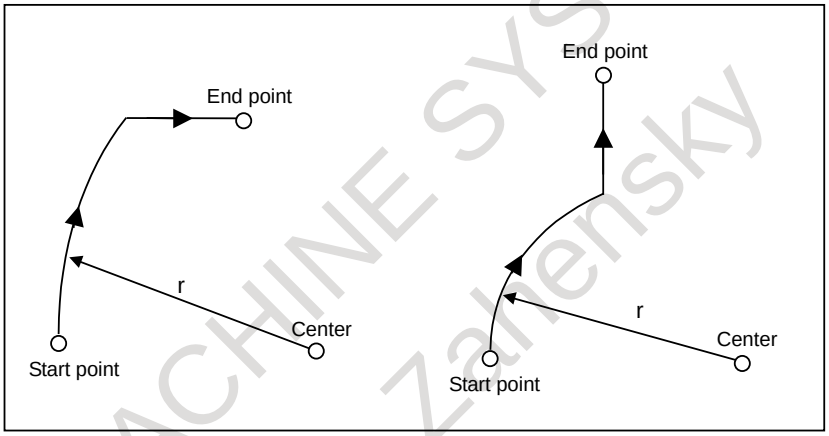


Fig. 4.10 (e) Movement when the end point is not on an arc

FRYER MACHINE SYSTEMS INC
Rob Zakensky

4.11 SKIP FUNCTION (G31)

Linear interpolation can be commanded by specifying axial move following the G31 command, like G01. If an external skip signal SKIP is input during the execution of this G31 command, execution of the G31command is interrupted and the next block is executed.
The skip function is used when the end of machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimensions of a workpiece.

Format

G31 IP ;

G31 : One-shot G code (If is effective only in the block in which it is specified)

Explanation

The coordinate values when the skip signal SKIP is turned on can be used in a custom macro because they are stored in the custom macro system variable #100151 to #100182, as follows. For systems with up to 20 axes, FS16-compatible system variables (#5061 to #5080) may also be used.

#100151	(#5061) :	First axis coordinate value
#100152	(#5062) :	Second axis coordinate value
:		
#100170	(#5080) :	20th axis coordinate value
#100171 :		21st axis coordinate value
#100172 :		22nd axis coordinate value
:		
#100182 :		32nd axis coordinate value

- Feedrate

Two types of feedrate can be selected for the G31 block, using bit 1 (SFP) of parameter No. 6207.

(1) Parameter SFP (No. 6207#1) = 0
Feedrate specified with address F (may be specified before or in the G31 block).

(2) Parameter SFP (No. 6207#1) = 1
Feedrate set for parameter No.6281.
Even if bit 7 (SKF) of parameter No. 6200 is set to 1, dry run and automatic acceleration/deceleration are disabled when a feedrate with the G31 command is the feedrate which is set for parameter No. 6281.

⚠ CAUTION

To increase the precision of the tool position when the skip signal SKIP is input, feedrate override, dry run, and automatic acceleration/deceleration are disabled in moving by the skip function (G31) regardless of the feedrate which is specified as a feed per minute or a feed per rotation.
To enable these functions, set bit 7 (SKF) of parameter No. 6200 to 1.
However, even if bit 7 (SKF) of parameter No. 6200 is set to 1, dry run and automatic acceleration/deceleration are disabled when bit 1 (SFP) of parameter No. 6207 is set to 1.

NOTE

If G31 command is issued while tool radius/tool nose radius compensation is applied, an alarm PS0035, "CAN NOT COMMANDED G31", is displayed.
Cancel the tool radius compensation with the G40 command before the G31 command is specified.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Example

- **The next block to G31 is an incremental programming**

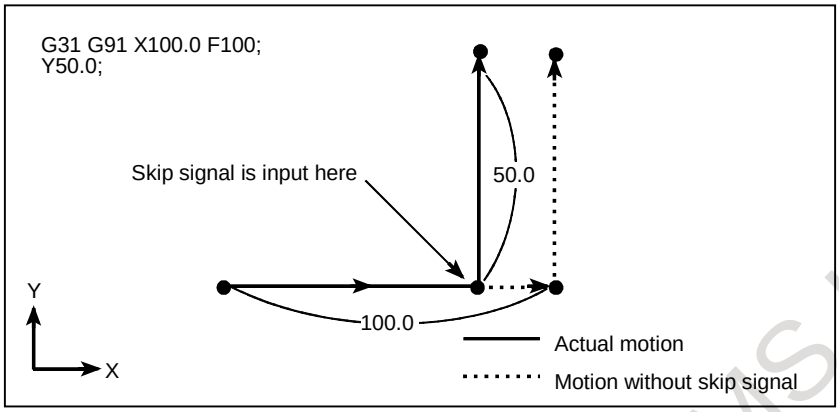


Fig. 4.11 (a) The next block is an incremental programming

- **The next block to G31 is an absolute programming for 1 axis**

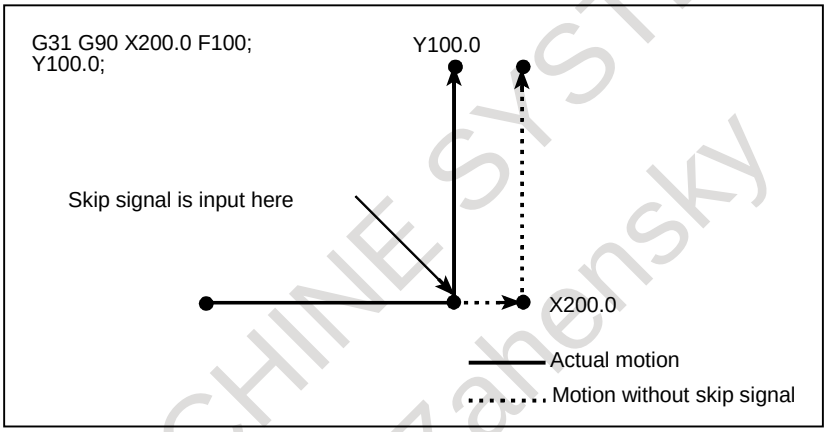


Fig. 4.11 (b) The next block is an absolute programming for 1 axis

- **The next block to G31 is an absolute programming for 2 axes**

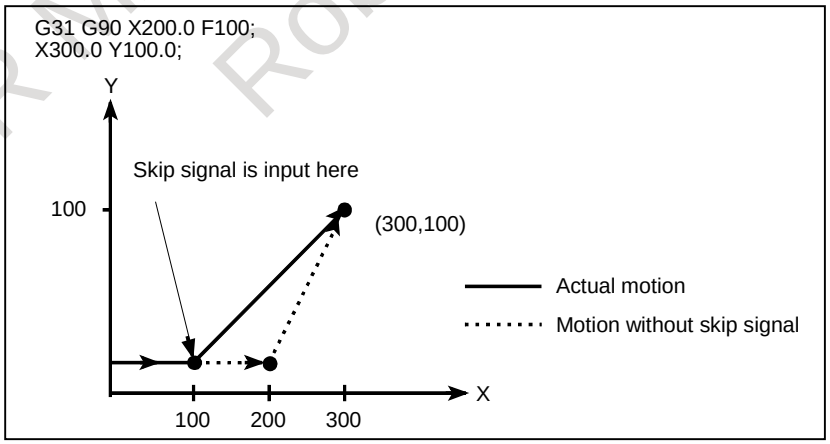


Fig. 4.11 (c) The next block is an absolute programming for 2 axes

FRYER MACHINE SYSTEMS INC
Rob Zahensky

4.12 MULTI-STEP SKIP (G31)

In a block specifying P1 to P4 after G31, the multi-step skip function stores coordinates in a custom macro variable when a skip signal (4-point or 8-point ; 8-point when a high-speed skip signal is used) is turned on. In the block where Q1 to Q4 are specified after G04, dwell can be skipped when skip signals (four or eight signals, or eight signals when high-speed skip signals are used) are input.
A skip signal from equipment such as a fixed-dimension size measuring instrument can be used to skip programs being executed.
In plunge grinding, for example, a series of operations from rough machining to spark-out can be performed automatically by applying a skip signal each time rough machining, semi-fine machining, fine-machining, or spark-out operation is completed.

Format

Move command

G31 IP_ F_ P_ ;

IP_ : End point
F_ : Feedrate
P_ : P1 to P4

Dwell

G04 X(U,P)_ (Q_);

X(U,P)_ : Dwell time
Q_ : Q1 to Q4

Explanation

Multi-step skip is caused by specifying P1, P2, P3, or P4 in a G31 block. For an explanation of selecting (P1, P2, P3, or P4), refer to the manual supplied by the machine tool builder.
Specifying Q1, Q2, Q3, or Q4 in G04 (dwell command) enables dwell skip in a similar way to specifying G31. A skip may occur even if Q is not specified. For an explanation of selecting (Q1, Q2, Q3, or Q4), refer to the manual supplied by the machine tool builder.

- Feedrate

Two types of feedrate can be selected for the multi-step skip block, using bit 2 (SFN) of parameter No.6207.

(1) Parameter SFN (No.6207#2) = 0
Feedrate specified with address F (may be specified before or in the multi-step skip block).

(2) Parameter SFN (No.6207#2) = 1
Feedrate set for parameters Nos.6282 to 6285.
Even if bit 7 (SKF) of parameter No.6200 is set to 1, dry run and automatic acceleration/deceleration are disabled when a feedrate with the G31 command is the feedrate which is set for parameters Nos.6282 to 6285.

⚠ **CAUTION**
To increase the precision of the tool position when the skip signal is input, feedrate override, dry run, and automatic acceleration/deceleration are disabled in moving by the skip function (G31) regardless of the feedrate which is specified as a feed per minute or a feed per rotation.
To enable these functions, set bit 7 (SKF) of parameter No.6200 to 1.
However, even if bit 7 (SKF) of parameter No.6200 is set to 1, dry run and automatic acceleration/deceleration are disabled when bit 2 (SFN) of parameter No.6207 is set to 1.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Correspondence to skip signals

Parameters Nos. 6202 to 6205 can be used to specify whether the 4-point or 8-point skip signal is used (when a high-speed skip signal is used). Specification is not limited to one-to-one correspondence. It is possible to specify that one skip signal correspond to two or more Pn's or Qn's (n=1, 2, 3, 4). Also, bits 0 (DS1) and 7 (DS8) parameter No.6206 can be used to specify dwell.

> ⚠ **CAUTION**
> Dwell is not skipped when Qn is not specified and bits 0 (DS1) and 7 (DS8) parameter No.6206 are not set.

4.13 HIGH-SPEED SKIP (G31)

The skip function operates based on a high-speed skip signal (connected directly to the CNC; not via the PMC) instead of an ordinary skip signal. In this case, up to eight signals can be input.
Delay and error of skip signal input is 0 - 2 msec at the CNC side (not considering those at the PMC side).
This high-speed skip signal input function keeps this value to 0.1 msec or less, thus allowing high precision measurement.
For details, refer to the appropriate manual supplied from the machine tool builder.

Format

G31 IP ;

G31; One-shot G code (If is effective only in the block in which it is specified)

4.14 SKIP POSITION MACRO VARIABLE IMPROVEMENT

Overview

In macro variables #100151 to #100200 (#5061 to #5080) for reading the skip position when executing the skip command (G31), by setting bit 7 (SKM) of parameter No. 6007 to 1, it is now possible to read the skip position in the workpiece coordinate system at the time a skip is made even if the workpiece coordinate system setting/selection command is executed after the skip command is executed.
By setting bit 7 (SKM) of parameter No. 6007 to 0, it is possible to read the skip position that reflects the workpiece coordinate system at the time it is read, as usual, if the workpiece coordinate system setting/selection command is executed after the skip command is executed.

Macro variable

The following are the macro variables that can read the position at the time a skip is made even if the workpiece coordinate system setting/selection command is executed after the skip command is executed, by setting bit 7 (SKM) of parameter No. 6007 to 1.

System variable number	System variable name	Attribute	Description
#5061 to #5080	[#_ABSKP[n]]	R	Skip position (workpiece coordinate system) Note) Subscript n represents an axis number (1 to 20).
#100151 to #100200			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#5421 to #5440	[#_SKPDTC[n]]	R	Skip position (workpiece coordinate system, detection unit) Note) Subscript n represents an axis number (1 to 20).
#100701 to #100750			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

4.15 CONTINUOUS HIGH-SPEED SKIP FUNCTION

Overview

The continuous high-speed skip function is used to read absolute coordinates using high-speed skip signals HDI0 to HDI7. Inputting a high-speed skip signal in a G31P90 block causes absolute coordinates to be stored in custom macro variables #5061 to #5080. For a system with more than 20 axes, they are stored in variables #100151 to #100182. An axis movement does not stop even if a signal is input, so that the coordinates of multiple points can be read.
Using bit 5 (CSE) of parameter No. 6201, it is possible to enable both the rising and falling edges of a high-speed skip signal.
For details, refer to the manual supplied by the machine tool builder.

Format

G31 P90 IP ;

G31; One-shot G code (If is effective only in the block in which it is specified)

Explanation

- Custom macro variables

If a high-speed skip signal is input when G31P90 is issued, absolute coordinates are stored in custom macro variables #5061 to #5080. For a system exceeding 20 axes, they are stored in variables #100151 to #100182.
These variables are updated as soon as the axis reaches a new skip position. It is, therefore, necessary to specify a speed so that the axis reaches the next skip position in a period of time longer than it takes for the application to read the variable values.

System with 20 or less axes		System with more than 20 axes	
#5061	Coordinates on the first axis	#100151	Coordinates on the first axis
#5062	Coordinates on the second axis	#100152	Coordinates on the second axis
#5063	Coordinates on the third axis	#100153	Coordinates on the third axis
	:		:
#5079	Coordinates on the 19th axis	#100181	Coordinates on the 31st axis
#5080	Coordinates on the 20th axis	#100182	Coordinates on the 32nd axis

- High-speed skip

This function works with a high-speed skip signal only.
Select a high-speed skip signal with bits 0 to 7 (9S1 to 9S8) of parameter No. 6208.

- Ending a block

A G31P90 block is ended when the axis reaches the end point.

Limitation

The continuous high-speed skip function (G31P90) block must be a command for a single axis only. If an attempt is made to specify two or more axes, alarm PS5068, “FORMAT ERROR IN G31 P90” is issued.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

4.16 TORQUE LIMIT SKIP

Overview

Executing the move command following G31P99 (or G31P98) while overriding the torque limit[*1] on the servo motor enables cutting feed in to be performed in the same way as in linear interpolation (G01). If, during the movement with this command, the torque of the servo motor reaches the torque limit value (torque limit on the servo motor multiplied by the override) due to pressing or other causes or a skip signal (including a high-speed skip signal) is input, any remaining move commands will be canceled, and the next block is executed. (The operation of canceling any remaining move commands and executing the next block is called a skip operation in the remainder of this document.)

It is possible to override the torque limit on the servo motor with the following command methods:

(1) Execute the torque limit override command in the PMC window.

(2) Execute an address Q command in the same block as that containing a G31P99 (or G31P98) command.

*1 : The torque limit on the servo motor is automatically set to a value conforming to the motor type setting. The override is applied to the maximum torque of the servo motor.

In order to override the torque limit on the servo motor, set bit 0 (TQO) of parameter No. 6286 to 1.

Format

G31 P98 Q_ α_ F_

G31 P99 Q_ α_ F_

G31 : Skip command (one-shot G code)

P98 : Performs a skip operation if the torque of the servo motor reaches the limit value.

P99 : Performs a skip operation if the torque of the servo motor reaches the limit value or if a skip signal is input.

Q : Override value for the torque limit
Range of valid settings: 1 to 254 (%)
0 to 255 correspond to 0% to 100%.
The Q command is optional. If omitting it, specify the torque limit command in the PMC window beforehand. If it is omitted, and no torque limit override is set beforehand, alarm PS0035, “CAN NOT COMMANDED G31” is issued.
If an attempt is made to specify a value out of range, alarm PS0369, “G31 FORMAT ERROR” is issued.
The specified override value is effective only to the block in which it is specified.
At the end of the skip operation, the override value returns to the one assumed immediately before the G31 command.

α : Axis address on any one axis

F : Feedrate

- Conditions for performing a skip operation -

Table 4.16

Condition	Command	
	G31P98	G31P99
The torque limit value is reached.	A skip operation is performed.	A skip operation is performed.
A skip signal is input.	No skip operation is performed.	A skip operation is performed.

FRYER MACHINE SYSTEMS INC
Ron Zaremsky

- Operation during a torque limit skip

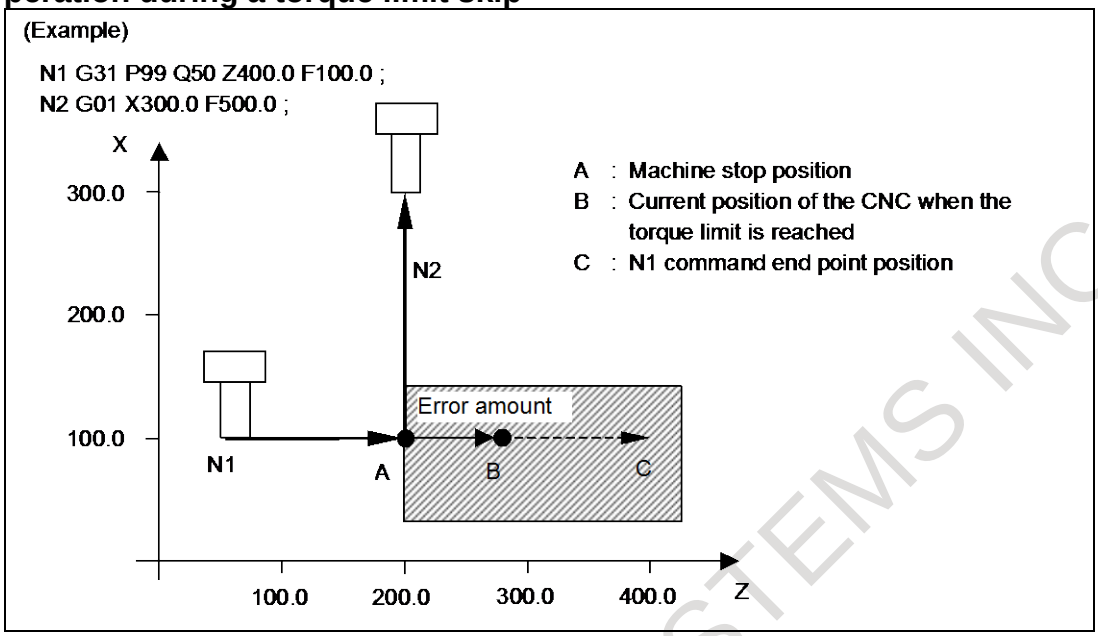


Fig. 4.16 (a)

A torque limit skip presses a specified axis against a previously prepared part or other item while the torque limit command is being executed on the servo motor, and then performs a skip operation when the servo motor reaches the torque limit value. A skip operation is performed with the fact that the torque limit value as detected in the servo motor is reached. It is, therefore, not necessary to input a skip signal using a separate sensor or other device unlike with normal skip functions.

(1) At point A, the machine comes in contact with the object under measurement and stops. At this time, because the torque limit value is not reached, no skip operation is performed, move commands are continuously output, and the current position of the CNC is updated.

(2) Because move commands are output but the machine remains stopped, there occurs a difference (error amount) between the current position of the CNC and the machine position, and torque is applied to the servo motor.

(3) When the torque limit value is reached, a skip operation is performed at machine stop position, point A, and the N2 command is executed. Assuming that the current position of the CNC when the torque limit is reached is point B, the error amount during the torque limit skip is (A - B).

- Torque limit command

If, with the torque limit skip command, no torque limit override value is specified with address Q and no torque limit command is issued from the PMC window, etc., alarm PS0035 is issued.

When no toque limit command is issued, the torque limit override value is either 0% or 100%.

If, in a torque limit skip command block, no torque limit override value is specified with address Q, a torque limit is to be specified as in the program example below.

(Program example)
O0012
:
Mxx (Specify a torque limit from the PMC via the window)
:
G31 P99 X200.0 F100.(Torque limit skip command)
:
G01 X100.0 F500.(Move command with the torque limit being still effective)
:
Myy (Cancel the torque limit from the PMC)
:
M30

FRYER MACHINE SYSTEMS INC
Rob Zaprisky

- Positional deviation limit during the torque limit command

While the torque limit skip command is being executed, the positional deviation limit check with the settings of parameters Nos. 1828 and 1829 is not performed. Instead, the positional deviation limit check with the settings of parameter No. 6287 is performed. If the positional deviation exceeds the limit, alarm SV0004, “EXCESS ERROR (G31)” is issued and an instantaneous stop occurs.

- Custom macro variables

When the torque limit skip command is executed, the custom macro system variables #5061 to #5080 (skip signal position; for a system with more than 20 axes, variables #100151 to #100182 are used) store the coordinate position assumed at the end of the skip. In reality, there is a deviation due to the delay of the servo system between the machine position and the current position of the CNC when a skip operation is executed. This deviation can be determined from the positional deviation of the servo. By setting bit 2 (TSE) of parameter No. 6201, it is possible to select whether or not the skip signal position to be stored in system variables should be compensated for with the error (positional deviation) of the servo system.

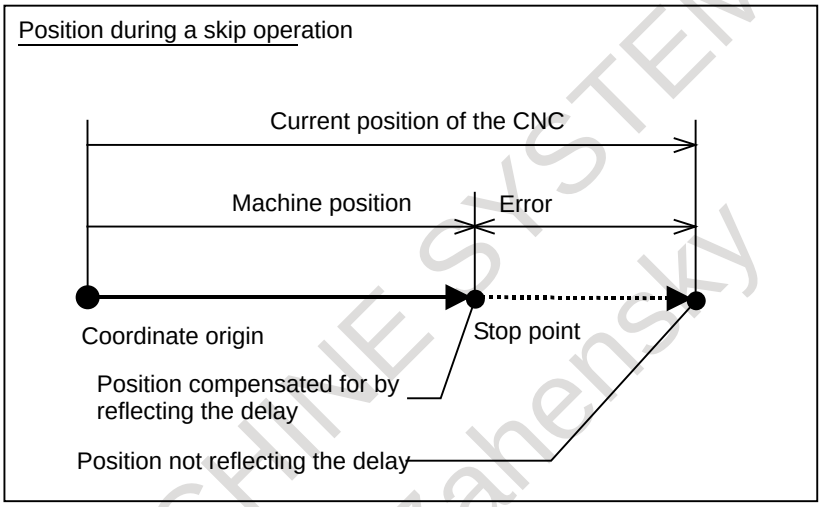


Fig. 4.16 (b)

NOTE

1 Specify only a single axis with the torque limit skip command. If no axis is specified or an attempt is made to specify more than one, alarm PS0369 is issued.
2 Do not issue the torque limit skip command in 3-dimensional coordinate conversion mode. Otherwise, alarm PS0035 is issued.
3 Do not issue the torque limit skip command in G41 or G42 mode. Otherwise, alarm PS0035 is issued.
4 A torque limit arrival signal TRQL1 to TRQL8 is output regardless of the torque limit skip command.
5 Do not issue the torque limit skip command for an axis being synchronized with synchronization control (such as synchronization control, or electronic gear box).
6 Do not specify the torque limit skip command in a continuous block.
7 The higher the movement speed, the larger the error between the position at which the machine stops and the position at which a skip is actually detected. Also, the error increases as the speed is varied during movement. Do not vary the speed with override and so on.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

5 FEED FUNCTIONS

5.1 OVERVIEW

The feed functions control the feedrate of the tool. The following two feed functions are available:

- Feed functions

1. Rapid traverse
   When the positioning command (G00) is specified, the tool moves at a rapid traverse feedrate set in the CNC (parameter No. 1420).
2. Cutting feed
   The tool moves at a programmed cutting feedrate.

- Override

Override can be applied to a rapid traverse rate or cutting feedrate using the switch on the machine operator's panel.

- Automatic acceleration/deceleration

To prevent a mechanical shock, acceleration/deceleration is automatically applied when the tool starts and ends its movement (Fig. 5.1 (a)).

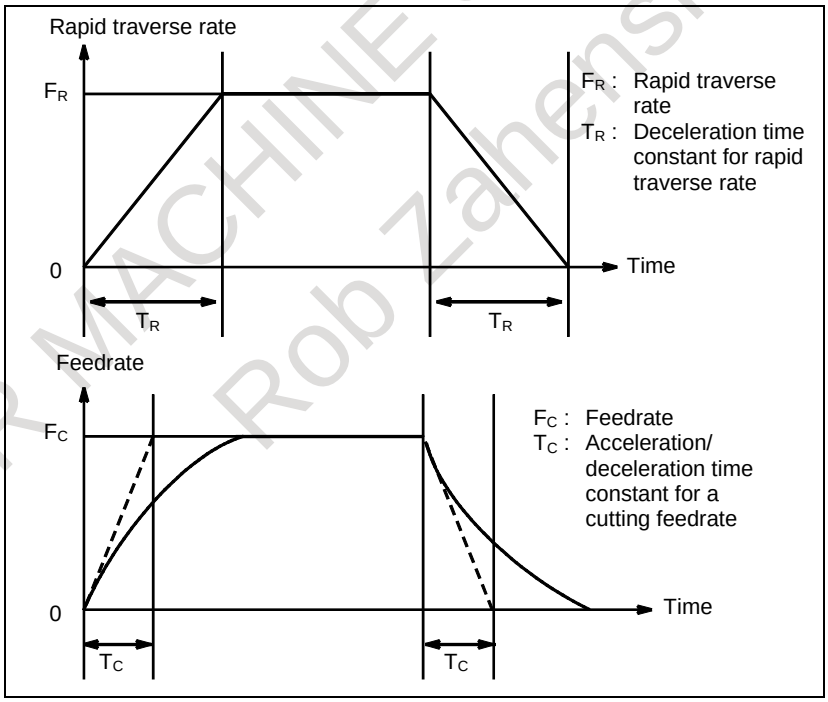


Fig. 5.1 (a) Automatic acceleration/deceleration (example)

- High-speed and high-precious machining function

The machining precision can be improved by using AI contour control function I or II. Refer to the Section of "AI CONTOUR CONTROL FUNCTION I AND AI CONTOUR CONTROL FUNCTION II" in "HIGH-SPEED CUTTING FUNCTION" for detail.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Tool path in a cutting feed

When the movement direction changes between a specified block and the next block during cutting feed, the tool path may be rounded because of the relationship between the time constant and feedrate (Fig. 5.1 (b)).

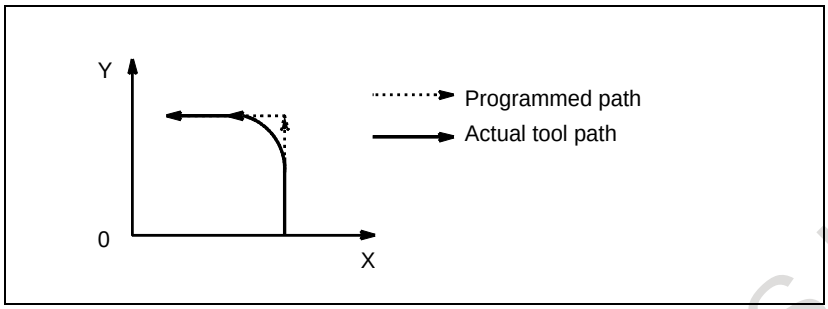


Fig. 5.1 (b) Example of tool path between two blocks

In circular interpolation, a radial error occurs (Fig. 5.1 (c)).

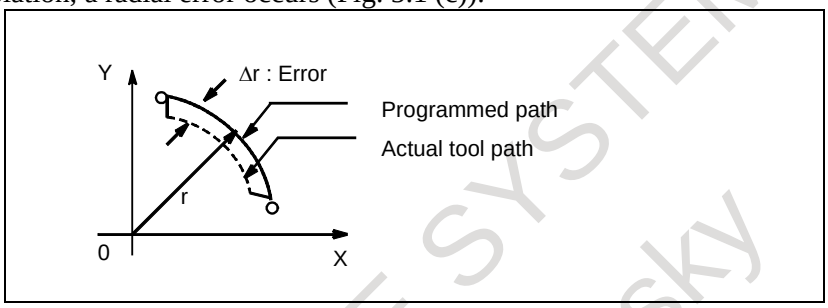


Fig. 5.1 (c) Example of radial error in circular interpolation

The rounded-corner path shown in Fig. 5.1 (b) and the error shown in Fig. 5.1 (c) depend on the feedrate. So, the feedrate needs to be controlled for the tool to move as programmed.

5.2 RAPID TRAVERSE

Format

G00 IP_ ;

G00 : G code (group 01) for positioning (rapid traverse)
IP_ : Dimension word for the end point

Explanation

The positioning command (G00) positions the tool by rapid traverse. In rapid traverse, the next block is executed after each axis stops at the end point and the servo motor reaches a certain range set by the machine tool builder (in-position check).
A rapid traverse rate is set for each axis by parameter No. 1420, so no rapid traverse feedrate need be programmed.
The following overrides can be applied to a rapid traverse rate with the switch on the machine operator's panel:F0, 25%, 50%, 100%
F0: Allows a fixed feedrate to be set for each axis by parameter No. 1421.
For detailed information, refer to the appropriate manual of the machine tool builder.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

5.3 CUTTING FEED

Overview

Feedrate of linear interpolation (G01), circular interpolation (G02, G03), etc. are commanded with numbers after the F code.
In cutting feed, the next block is executed so that the feedrate change from the previous block is minimized.

M

Four modes of specification are available:

1. Feed per minute (G94)
   After F, specify the amount of feed of the tool per minute.
2. Feed per revolution (G95)
   After F, specify the amount of feed of the tool per spindle revolution.
3. Inverse time feed (G93)
   Specify the inverse time (FRN) after F.
4. One-digit F code feed
   Specify a desired one-digit number after F. Then, the feedrate set with the CNC for that number is set.

And, the number of effective digits after the decimal point of feedrate to feedrate command F is as follows.
Example) In the case of increment system IS-B (0.001mm/ 0.0001inch)

Command	Decimal point of F command	Parameter	The number of effective digits after the decimal point	
			mm	inch
Feed per Minute (G94)	Not specified	Nothing	0 digit	2 digits
			F1234=1234(mm/min)	F1234=12.34(inch/min)
	Specified	Nothing	3 digits	5 digits
			F1.234=1.234(mm/min)	F1.23456=1.23456(inch/min)
Feed per revolution (G95) Threading (G33)	Not specified	No.1405#1 =0 (NOTE)	2 digits	4 digits
			F1234=12.34(mm/rev)	F1234=0.1234(inch/rev)
		No.1405#1 =1 (NOTE)	3 digits	5 digits
			F1234=1.234(mm/rev)	F1234=0.01234(inch/rev)
	Specified	Nothing	3 digits	5 digits
			F1.234=1.234(mm/rev)	F1.23456=1.23456(inch/rev)

NOTE
Setting of bit 1 (FCV) of parameter No.0001 doesn't influence.

T

Two modes of specification are available:

1. Feed per minute (G98 or G94)
   After F, specify the amount of feed of the tool per minute.
2. Feed per revolution (G99 or G95)
   After F, specify the amount of feed of the tool per spindle revolution.

And, the number of effective digits after the decimal point of feedrate to feedrate command F is as follows.
Example) In the case of increment system IS-B (0.001mm/ 0.0001inch)

FRYER MACHINE SYSTEMS INC Rob Zahensky

Command	Decimal point of F command	Parameter	The number of effective digits after the decimal point	
			mm	inch
Feed per Minute (G98 or G94)	Not specified	No.1404#2 =0	0 digit	2 digits
			F1234=1234(mm/min)	F1234=12.34(inch/min)
		No.1404#2 =1	3 digits	5 digits
			F1234=1.234(mm/min)	F123456=1.23456(inch/min)
	Specified	Nothing	3 digits	5 digits
			F1.234=1.234(mm/min)	F1.23456=1.23456(inch/min)
Feed per revolution (G99 or G95) Threading (G32)	Not specified	No.0001#1 =0	4 digits	6 digits
			F12345=1.2345(mm/rev)	F1234567=1.234567(inch/rev)
		No.0001#1 =1	2 digits	4 digits
			F1234=12.34(mm/rev)	F12345=1.2345(inch/rev)
	Specified	Nothing	4 digits	6 digits
			F1.2345=1.2345(mm/rev)	F1.234567=1.234567(inch/rev)

Format

M

Feed per minute

G94 ; G code (group 05) for feed per minute

F_ ; Feedrate command (mm/min or inch/min)

Feed per revolution

G95 ; G code (group 05) for feed per revolution

F_ ; Feedrate command (mm/rev or inch/rev)

Inverse time feed (G93)

G93 ; Inverse time feed command G code (05 group)

F_ ; Feedrate command (1/min)

One-digit F code feed

Fn ;

n : Number from 1 to 9

T

G code system A

Feed per minute

G98 ; G code (group 05) for feed per minute

F_ ; Feedrate command (mm/min or inch/min)

Feed per revolution

G99 ; G code (group 05) for feed per revolution

F_ ; Feedrate command (mm/rev or inch/rev)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

G code system B/C

Feed per minute

G94 ; G code (group 05) for feed per minute

F_ ; Feedrate command (mm/min or inch/min)

Feed per revolution

G95 ; G code (group 05) for feed per revolution

F_ ; Feedrate command (mm/rev or inch/rev)

Explanation

- Direction of the cutting feedrate

Cutting feed is controlled so that the tangential feedrate is always set at a specified feedrate.

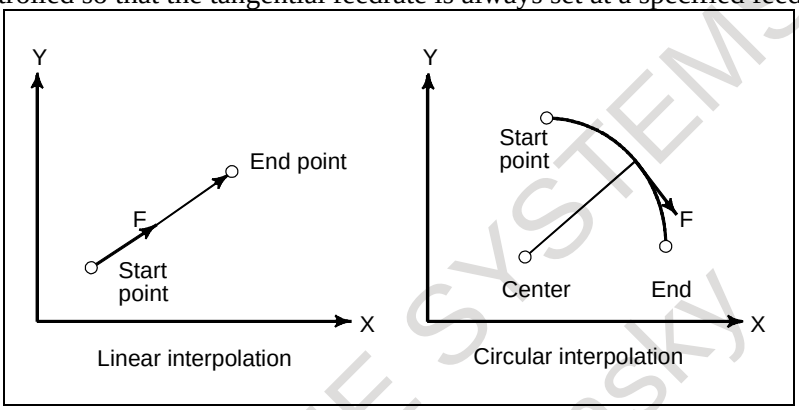


Fig. 5.3 (a) Tangential feedrate (F)

- Feed per minute

After specifying G code for feed per minute (in the feed per minute mode), the amount of feed of the tool per minute is to be directly specified by setting a number after F. G code for feed per minute is a modal code. Once a G code for feed per minute is specified, it is valid until another G code belonging to the group 05 (feed per revolution, inverse time feed) is specified.

M

At power-on, the feed per minute mode is set.

T

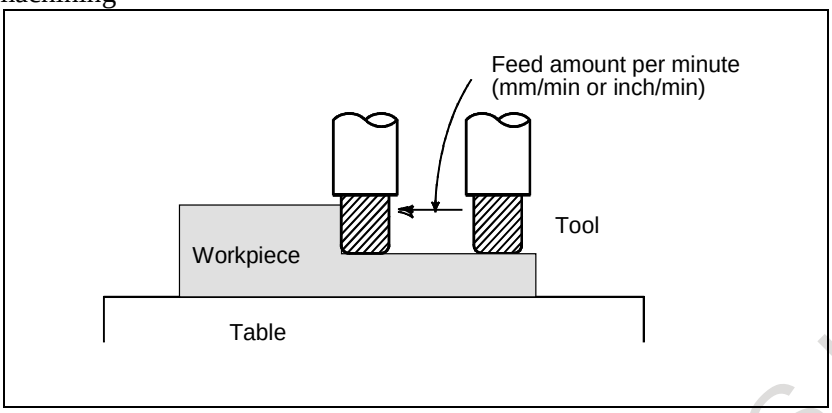
Bit 4 (FPM) of parameter No.3402 is set to 0:
At power-on, the feed per revolution mode is set.

Bit 4 (FPM) of parameter No.3402 is set to 1:
At power-on, the feed per minute mode is set.

An override from 0% to 254% (in 1% steps) can be applied to feed per minute with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- For milling machining



- For lathe cutting

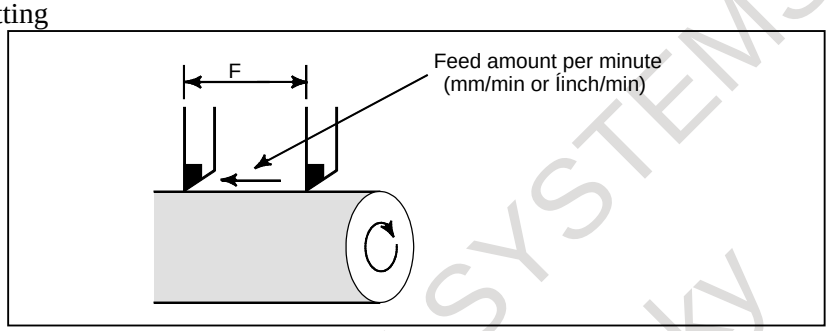


Fig. 5.3 (b) Feed per minute

> ⚠ **CAUTION**
> No override can be used for some commands such as for threading.

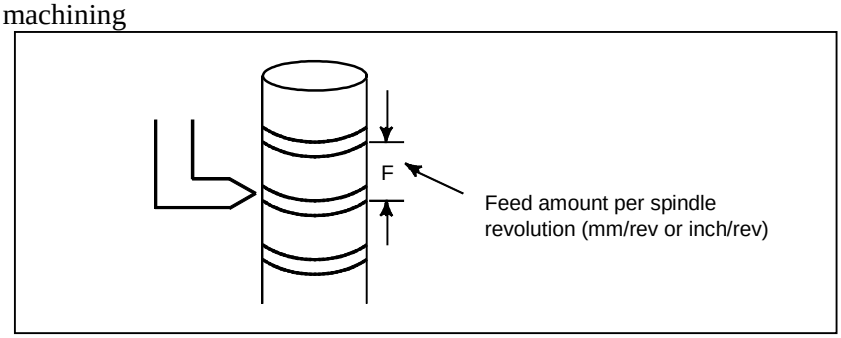
- Feed per revolution

After specifying G code for feed per revolution (in the feed per revolution mode), the amount of feed of the tool per spindle revolution is to be directly specified by setting a number after F. G code for feed per revolution is a modal code. Once a G code for feed per revolution is specified, it is valid until another G code belonging to the group 05 (feed per minute, inverse time feed) is specified.
An override from 0% to 254% (in 1% steps) can be applied to feed per revolution with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

T

If bit 0 (NPC) of parameter No. 1402 has been set to 1, feed per revolution commands can be specified even when a position coder is not being used. (The CNC converts feed per revolution commands to feed per minute commands.)

- For milling machining



FRYER MACHINE SYSTEMS INC
Rob Zahensky

- For lathe cutting

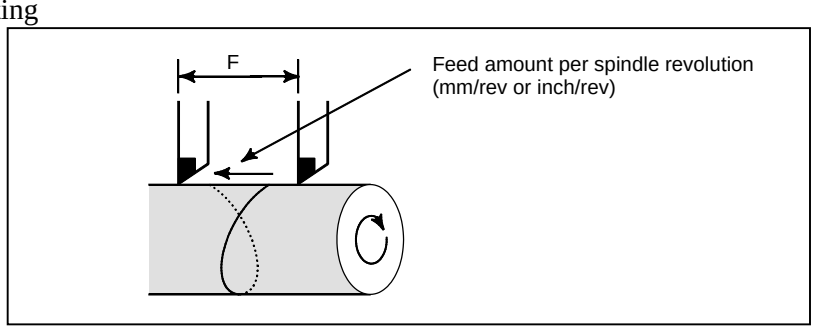


Fig. 5.3 (c) Feed per revolution

> ⚠ **CAUTION**
> When the speed of the spindle is low, feedrate fluctuation may occur.
> The slower the spindle rotates, the more frequently feedrate fluctuation occurs.

M

- Inverse time feed

When G code for inverse time feed is specified, the inverse time specification mode (G93 mode) is set. Specify the inverse time (FRN) with an F code.
A value from 0.001 to 9999.999 can be specified as FRN, regardless of whether the input mode is inches or metric, or the increment system is IS-B or IS-C.

F code specification value	FRN
F1	0.001
F1 (*1)	1.000
F1.0	1.000
F9999999	9999.999
F9999 (*1)	9999.000
F9999.999	9999.999

> **NOTE**
> *1 Value specified in fixed-point format with bit 0 (DPI) of parameter No. 3401 set to 1

G code for inverse time feed is a modal G code and belongs to group 05 (includes G code for feed per revolution and G code for feed per minute).
When an F value is specified in inverse time specification mode and the feedrate exceeds the maximum cutting feedrate, the feedrate is clamped to the maximum cutting feedrate.
In the case of circular interpolation, the feedrate is calculated not from the actual amount of movement in the block but from the arc radius. This means that actual machining time is longer when the arc radius is longer than the arc distance and shorter when the arc radius is shorter than the arc distance. Inverse time feed can also be used for cutting feed in a canned cycle.

> **NOTE**
> 1 In the inverse time specification mode, an F code is not handled as a modal code and therefore needs to be specified in each block. If an F code is not specified, alarm PS1202, “NO F COMMAND AT G93 ” is issued.
> 2 When F0 is specified in inverse time specification mode, alarm PS0011, “FEED ZERO (COMMAND)” is issued.
> 3 Inverse time feed cannot be used when PMC axis control is in effect.
> 4 If the calculated cutting feedrate is smaller than the allowable range, alarm PS0011, “FEED ZERO (COMMAND)” is issued.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Example

- For linear interpolation (G01)

$$FRN = \frac{1}{time(min)} = \frac{feedrate}{distance}$$

Feedrate: mm/min (for metric input)
inch/min (for inch input)
Distance: mm (for metric input)
inch (for inch input)

- To end a block in 1 (min)

$$FRN = \frac{1}{time(\text{min})} = \frac{1}{1(\text{min})} = 1$$ Specify F1.0.

- To end a block in 10 (sec)

$$FRN = \frac{1}{time(\text{sec})/60} = \frac{1}{10/60(\text{sec})} = 6$$ Specify F6.0.

- To find the movement time required when F0.5 is specified

$$TIME(\text{min}) = \frac{1}{FRN} = \frac{1}{0.5} = 2$$ 2 (min) is required.

- To find the movement time required when F10.0 is specified

$$TIME(\text{min}) = \frac{1 \times 60}{FRN} = \frac{60}{10} = 6$$ 6 (sec) is required.

- For circular interpolation (G02, G03)

$$FRN = \frac{1}{time(\text{min})} = \frac{feedrate}{arcradius}$$

Feedrate: mm/min (for metric input)
inch/min (for inch input)
Arc radius: mm (for metric input)
inch (for inch input)

NOTE
In the case of circular interpolation, the feedrate is calculated not from the actual amount of movement in the block but from the arcadias.

- Switching of a modal code in group 05

Usually, if a modal code in group 05 is switched using the feed selection command and the axis command is executed in cutting feed mode without specifying a feedrate (F), the feedrate (F) is inherited as modal data.

If bit 4 (MFC) of parameter No. 13450 is set to 1, the feedrate (F) is cleared if a modal code in group 05 is switched. This causes alarm PS0011, "FEED ZERO (COMMAND)", to be issued in the cutting block. Thus, it is possible to prevent cutting from being executed at an unintended feedrate (F) that is inherited as modal data.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Example

Execute the program below by setting bit 4 (MFC) of parameter No. 13450 to 1.
At the block for switching from G94 to G95 (N06), alarm PS0011 is issued.

```
O0001 ;
N01 G90 G00 X0 Y0 ;
N02 M03 S100 ;
N03 G91 G01 ;
N04 G94 X10.0 F100.0 ;
N05 Y10.0 ;
N06 G95 X10.0 ; ⇒ Alarm PS0011 is issued.
N07 Y10.0;
M30;
```

NOTE

1. In G93 mode, if the axis command and the feedrate (F) command are not in the same block, alarm PS1202, "NO F COMMAND AT G93" is issued regardless of the setting of this parameter.
2. If the G code of group 05 is cleared due to a reset, by setting bit 6 (CLR) of parameter No. 3402 to 1 and bit 5 (C05) of parameter No. 3406 to 0, so that the modal G code is switched, the feedrate (F) will be cleared even if bit 7 (CFH) of parameter No. 3409 is set to 1.
3. If bit 7 (FC0) of parameter No. 1404 is set to 1, alarm PS0011 is not issued and the block is executed with a feedrate of 0 even if the feed selection command is used to switch the modal code of group 05 and the axis command is executed in cutting feed mode without specifying a feedrate (F). In G93 mode, alarm PS1202 is issued regardless of the setting of the parameter bit FC0.
4. Alarm PS0011 or PS1202 is not used even if the feed selection command is used to switch the modal code of group 05 and the axis command is executed in cutting feed mode without specifying a feedrate (F), provided that the travel distance is 0.
5. Alarms PS0011 and PS1202 are issued if the feed selection command is used to switch the modal code of group 05 and the axis command is executed in cutting feed mode without specifying a feedrate (F), even if cutting feedrate (parameter No. 1411) during automatic operation is set. (M series.)

- One-digit F code feed

M

When a one-digit number from 1 to 9 is specified after F, the feedrate set for that number in a parameters Nos. 1451 to 1459 is used. When F0 is specified, the rapid traverse rate is applied.
The feedrate corresponding to the number currently selected can be increased or decreased by turning on the switch for changing one-digit F feedrate on the machine operator's panel, then by rotating the manual pulse generator.
The increment/decrement, ΔF, in feedrate per scale of the manual pulse generator is as follows:

$$\Delta F = \frac{F\max}{100X}$$

Fmax : Feedrate upper limit for F1-F4 set by parameter No.1460, or feedrate upper limit for F5-F9 set by parameter No.1461

X : Any value of 1-127 set by parameter No.1450

The feedrate set or altered is kept even while the power is off. The current feed rate is displayed on the LCD screen.

FRYER MACHINE SYSTEMS INC Rob Zahensky

- Cutting feedrate clamp

Parameter No. 1430 can be used to specify the maximum cutting feedrate for each axis. When the cutting feedrate along an axis exceeds the maximum feedrate for the axis as a result of interpolation, the cutting feedrate is clamped to the maximum feedrate.

Reference

See Appendix D for range of feedrate command value.

5.4 CUTTING FEEDRATE CONTROL

Cutting feedrate can be controlled, as indicated in Table 5.4(a).

Table 5.4 Cutting Feedrate Control

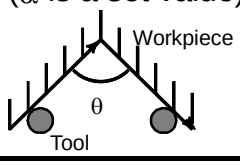
Function name		G code	Validity of G code	Description
Exact stop		G09	This function is valid for specified blocks only.	The tool is decelerated at the end point of a block, then an in-position check is made. Then the next block is executed.
Exact stop mode		G61	Once specified, this function is valid until G62, G63, or G64 is specified.	The tool is decelerated at the end point of a block, then an in-position check is made. Then the next block is executed.
Cutting mode		G64	Once specified, this function is valid until G61, G62, or G63 is specified.	The tool is not decelerated at the end point of a block, but the next block is executed.
Tapping mode		G63	Once specified, this function is valid until G61, G62, or G64 is specified.	The tool is not decelerated at the end point of a block, but the next block is executed. When G63 is specified, feedrate override and feed hold are invalid.
Automatic corner override	Automatic override for inner corners	G62	Once specified, this function is valid until G61, G63, or G64 is specified.	When the tool moves along an inner corner during tool radius compensation, override is applied to the cutting feedrate to suppress the amount of cutting per unit of time so that a good surface finish can be produced.
	Internal circular cutting feedrate change	-	This function is valid in the tool radius compensation mode, regardless of the G code.	The internal circular cutting feedrate is changed.

NOTE

1 The purpose of in-position check is to check that the servo motor has reached within a specified range (specified with a parameter by the machine tool builder). In-position check is not performed when bit 5 (NCI) of parameter No. 1601 is set to 1.

2 Spindle override is enabled during tapping mode.

3 Inner corner angle θ: $2° < \theta \leq \alpha \leq 178°$
(α is a set value)



FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Format

Exact stop	**G09 IP_ ;**
Exact stop mode	**G61 ;**
Cutting mode	**G64 ;**
Tapping mode	**G63 ;**
Automatic corner override	**G62 ;**

5.4.1 Exact Stop (G09, G61), Cutting Mode (G64), Tapping Mode (G63)

Explanation

The inter-block paths followed by the tool in the exact stop mode, cutting mode, and tapping mode are different.

At the end point of a block specifying exact stop (G09), or each block during exact stop mode (G61), cutting feed is decelerated and an in-position check is performed (Fig. 5.4.1 (a)).

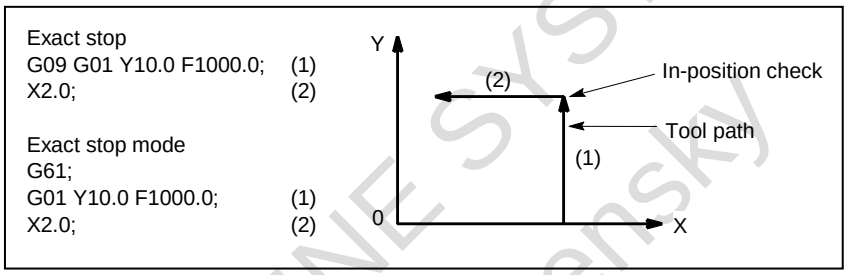


Fig. 5.4.1 (a) Example of tool path when G09 or G61 is executed

At the end of each block during tapping mode (G63) or cutting mode (G64), deceleration of cutting feed is not performed and execution proceeds to the next block (Fig. 5.4.1 (b)).

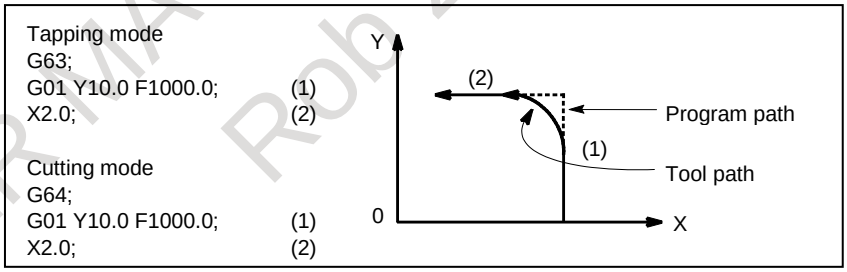


Fig. 5.4.1 (b) Example of tool path when G63 or G64 is executed

> ⚠ **CAUTION**
> The cutting mode (G64 mode) is set at power-on or system clear.

- Restriction on tapping mode (G63)

There are restrictions when the following functions are used. Please refer to each function for detail.

- Retrace
- AI contour control

FRYER MACHINE SYSTEMS INC
Rob Zahensky

5.4.2 Automatic Corner Override

M

When tool radius compensation is performed, the movement of the tool is automatically decelerated at an inner corner and internal circular area. This reduces the load on the tool and produces a smoothly machined surface.

5.4.2.1 Automatic override for inner corners (G62)

Explanation

- Override condition

When G62 is specified, and the tool path with tool radius compensation applied forms an inner corner, the feedrate is automatically overridden at both ends of the corner.

There are four types of inner corners (Fig. 5.4.2.1 (a)).

$2° \leq \theta \leq \theta p \leq 178°$ in Fig. 5.4.2.1 (a).

θp is a value set with parameter No. 1711. When θ is approximately equal to θp, the inner corner is determined with an error of 0.001° or less.

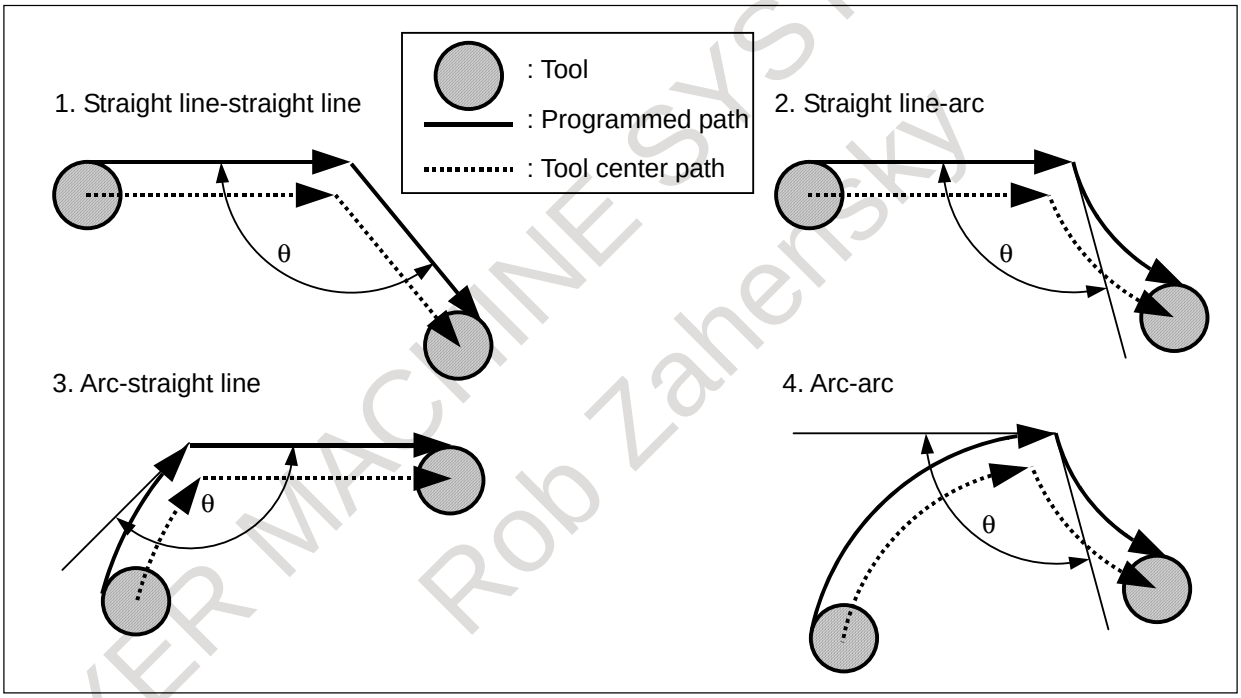


Fig. 5.4.2.1 (a) Inner corner

- Override range

When a corner is determined to be an inner corner, the feedrate is overridden before and after the inner corner. The distances Ls and Le, where the feedrate is overridden, are distances from points on the tool center path to the corner (Fig. 5.4.2.1 (b), Fig. 5.4.2.1 (c), Fig. 5.4.2.1 (d)). Ls and Le are set with parameters Nos. 1713 and 1714.

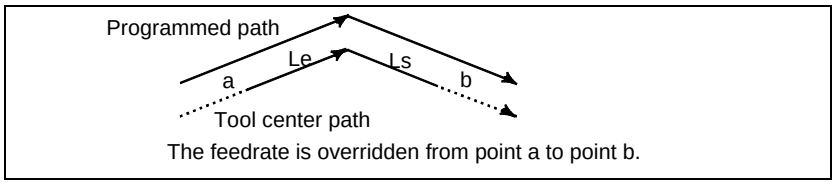


Fig. 5.4.2.1 (b) Override Range (Straight Line to Straight Line)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

When a programmed path consists of two arcs, the feedrate is overridden if the start and end points are in the same quadrant or in adjacent quadrants (Fig. 5.4.2.1 (c)).

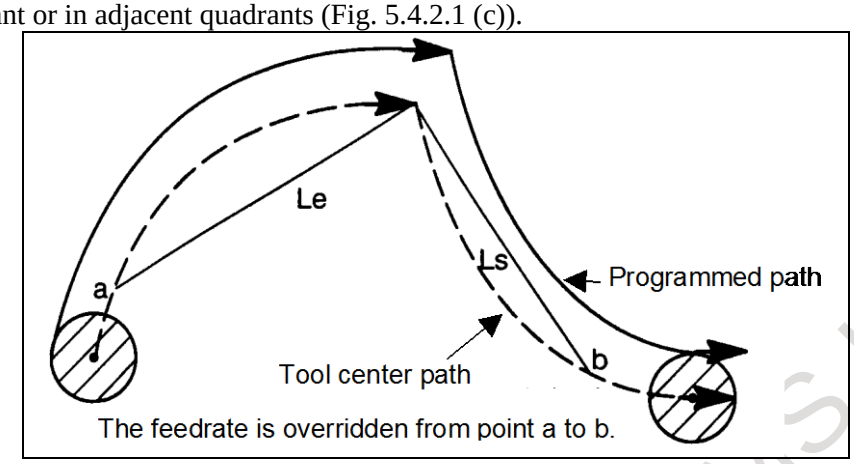


Fig. 5.4.2.1 (c) Override Range (Arc to Arc)

Regarding program (2) of an arc, the feedrate is overridden from point a to point b and from point c to point d (Fig. 5.4.2.1 (d)).

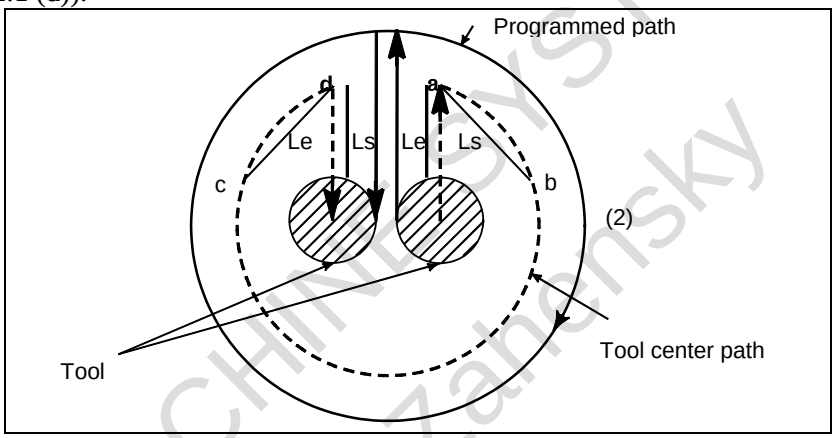


Fig. 5.4.2.1 (d) Override Range (Straight Line to Arc, Arc to Straight Line)

- Override value

An override value is set with parameter No. 1712. An override value is valid even for dry run and one-digit F code feed specification.
In the feed per minute mode, the actual feedrate is as follows:

F × (automatic override for inner corners) × (feedrate override)

Limitation

- Look-ahead acceleration/deceleration before interpolation

Override for inner corners is disabled during look-ahead acceleration/deceleration before interpolation.

- Start-up/G41, G42

Override for inner corners is disabled if the corner is preceded by a start-up block or followed by a block including G41 or G42.

- Offset

Override for inner corners is not performed if the offset is zero.

FRYER MACHINE SYSTEMS INC
Rob Zahansky

5.4.2.2 Internal circular cutting feedrate change

For internally offset circular cutting, the feedrate on a programmed path is set to a specified feedrate (F) by specifying the circular cutting feedrate with respect to F, as indicated below (Fig. 5.4.2.2 (a)). This function is valid in the tool radius compensation mode, regardless of the G62 code.

$F \times Rc / Rp$

Rc : Tool center path radius

Rp : Programmed radius

It is also valid for the dry run and the one-digit F code feed command.

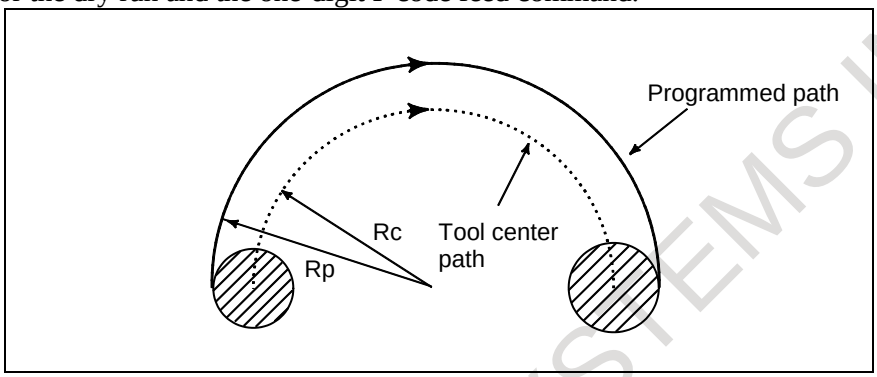


Fig. 5.4.2.2 (a) Internal circular cutting feedrate change

If Rc is much smaller than Rp, Rc/Rp ≒ 0; the tool stops. A minimum deceleration ratio (MDR) is to be specified with parameter No. 1710. When Rc/Rp≤MDR, the feedrate of the tool is (F×MDR).
If parameter No. 1710 is 0, the minimum deceleration ratio (MDR) is at 100%.

> ⚠ **CAUTION**
> When internal circular cutting must be performed together with override for inner corners, the feedrate of the tool is as follows:
> $$F \times \frac{Rc}{Rp} \times (\text{override for the inner corners}) \times (\text{feedrate override})$$

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

5.5 FEEDRATE INSTRUCTION ON IMAGINARY CIRCLE FOR A ROTARY AXIS

Overview

For the feedrate instruction on imaginary circle for a rotary axis, a circle with the radius specified with parameter No. 1465 (which is referred to as an imaginary circle) is considered, and the feedrate of the rotary axis is set to the feedrate on its circumference.

Explanation

- Cutting feedrate

Conventional method

Usually, the feedrate of a rotary axis is set to the rotation feedrate (deg/mm).

Feedrate of liner axis(X axis) $F_X = F \times \frac{\Delta X}{L} \ (mm/min)$

Feedrate of rotary axis(C axis) $F_C = F \times \frac{\Delta C}{L} \ (deg/min)$

Synthetic movement distance $L = \sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2 + \Delta C^2} \ (mm)$

Movement time $T = \frac{L}{F} \ (min)$

Feedrate instruction on imaginary circle of a rotary axis

For the feedrate instruction on imaginary circle for a rotary axis, the feedrate of the rotary axis is set to the movement feedrate on an imaginary circle with the radius specified with parameter No. 1465.
By setting the imaginary radius to 0, the rotary axis can be excluded from feedrate calculation.

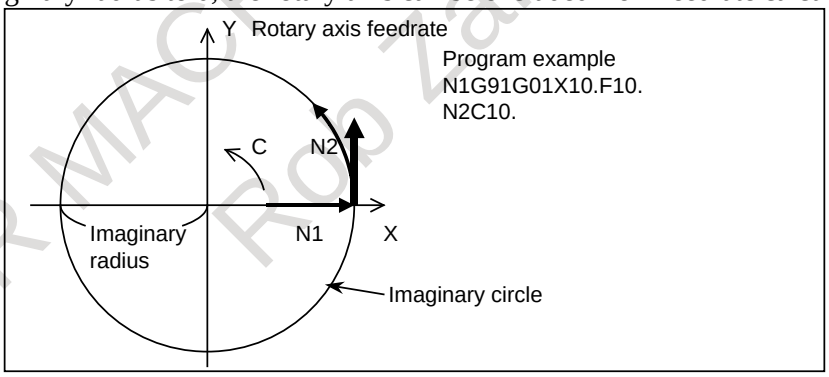


Fig. 5.5 (a)

Feedrate of liner axis(X axis) $F_X = F \times \frac{\Delta X}{L'} \ (mm/min)$

Feedrate of rotary axis(C axis) $F_C = F \times \frac{\Delta C}{L'} \ (deg/min)$

Synthetic movement distance $L' = \sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2 + \left(\frac{\pi \times l_C \times \Delta C}{180}\right)^2} \ (mm)$

Movement time $T' = \frac{L'}{F} \ (min)$

l_C : imaginary radius(parameter No.1465)

FRYER MACHINE SYSTEMS INC
Rob Zamansky

For the feedrate instruction on imaginary circle for a rotary axis, due to the difference in how to determine the movement distance, if a particularly small value is set for the imaginary radius, the movement on the axis will be fast. Pay careful attention to the input to the parameter.

A cutting feedrate clamp is performed based on the per-axis maximum cutting feedrate parameter No. 1430 and the actual speed on the axis (data before conversion). Thus, if a large value is set for the imaginary radius parameter No. 1465, a command can be issued with a feedrate exceeding the setting of the maximum cutting feedrate, and if a small value is set for the imaginary radius, the feedrate is clamped below the setting of the maximum cutting feedrate.
Note that the feedrate instruction on imaginary circle for a rotary axis is also effective to a dry run.

- AI Contour Control

AI contour control is performed on the movement speed on an imaginary circumference. The movement feedrate may, therefore, not be the feedrate for the calculation in the feedrate instruction on imaginary circle for a rotary axis if the feedrate is changed with the feedrate control of AI contour control. The feedrate during AI contour control is clamped with parameter No. 1432. If parameter No. 8465 is not 0, the feedrate is clamped with parameter No. 8465.

- If the imaginary radius is 0 mm

If the imaginary radius is 0 mm, the travel distance on the rotary axis is 0 mm. Thus, the combined travel distance is

$$L' = \sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2}$$

It is, therefore, possible to exclude the speed component on the rotary axis and regard the movement speed on the linear axis as specified speed F.
If only a rotary axis is specified with this setting, a movement occurs at the maximum cutting feedrate.

Examples

- Example 1

If, in the increment system IS-B,
G91 G01 C10. F10. ;
is specified,

(1) If 10.000 (10 mm) is set for the imaginary radius parameter No. 1465, the calculating formula is:

$$L' = \sqrt{\left(\frac{\pi \times l_C \times \Delta C}{180}\right)^2} = \sqrt{\left(\frac{\pi \times 10_{(mm)} \times 10_{(deg)}}{180}\right)^2} = 1.7453292\cdots_{(mm)}$$

$$F_C = 10_{(mm/\min)} \times \frac{10_{(deg)}}{1.7453292\cdots_{(mm)}} = 57.2957795\cdots_{(deg/min)}$$

$$T' = \frac{L'}{F} = \frac{1.7453292\cdots_{(mm)}}{10_{(mm/min)}} = 0.17453292\cdots_{(min)} = 10.4719755\cdots_{(sec)}$$

Thus, the movement time is about 10.472 sec, and the rotation speed is about 57.296 deg/min. The speed on the imaginary radius of 10.000 mm in Fig. 5.5 (b) above is regarded as the specified speed, 10.000 mm/min.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

(2) If 36.000 (36 mm) is set for the imaginary radius parameter No. 1465, the movement time is:

$$L' = \sqrt{\left(\frac{\pi \times l_C \times \Delta C}{180}\right)^2} = \sqrt{\left(\frac{\pi \times 36_{(mm)} \times 10_{(deg)}}{180}\right)^2} = 6.28318530\cdots_{(mm)}$$

$$F_C = 10_{(mm/min)} \times \frac{10_{(deg)}}{6.28318530\cdots_{(mm)}} = 15.9154943\cdots_{(deg/min)}$$

$$T' = \frac{L'}{F} = \frac{6.28318530\cdots_{(mm)}}{10_{(mm/min)}} = 0.628318530\cdots_{(min)} = 37.6991118\cdots_{(sec)}$$

Thus, the movement time is about 37.700 sec, and the rotation speed is about 15.915 deg/min. The speed on the imaginary radius of 36.000 mm in Fig. 5.5 (b) above is regarded as the specified speed, 10.000 mm/min.

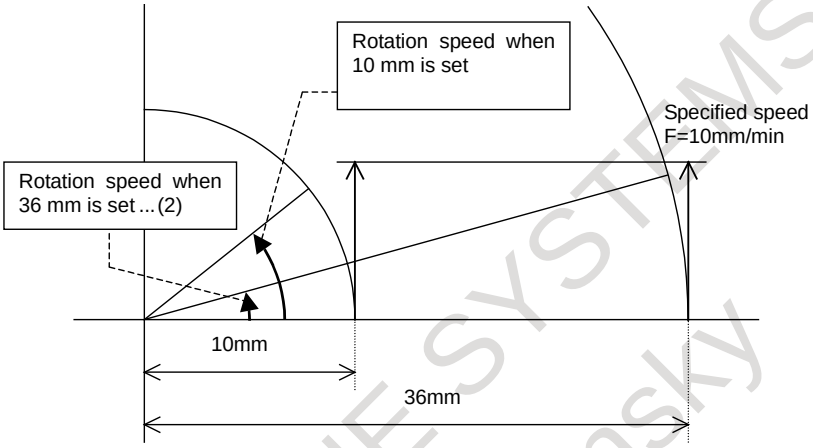


Fig. 5.5 (b)

- **Example 2**

In a machine in which the tool direction is changed with a rotary axis, such as that shown in Fig. 5.5 (c), it is possible to specify the movement speed at the reference position with F by enabling the feedrate instruction on imaginary circle for a rotary axis and setting the imaginary radius parameter No. 1465 to 0 mm.

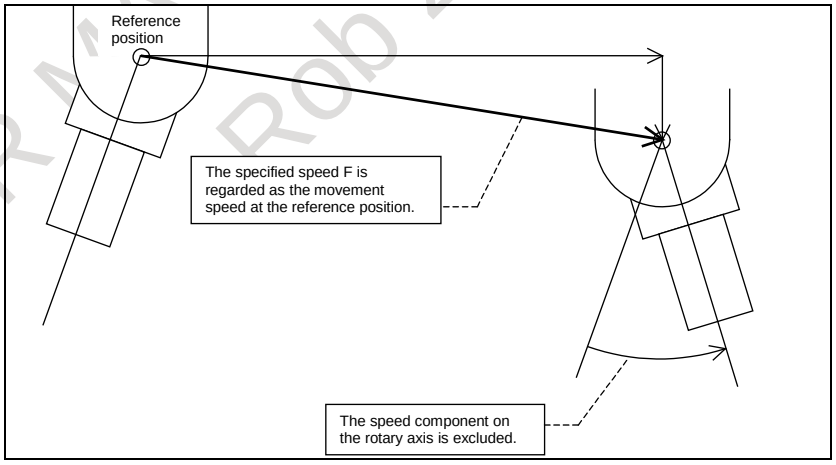


Fig. 5.5 (c)

Limitation

The feedrate instruction on imaginary circle for a rotary axis supports linear interpolation (G01) only.
It does not support the following functions:

- Feed per revolution (G95)
- Normal direction control
- Cylindrical interpolation
- PMC axis control

FRYER MACHINE SYSTEMS INC
Rob Zohensky

M

- Inverse time feed (G93)

T

- Polar coordinate interpolation

NOTE

1 The feedrate instruction on imaginary circle for a rotary axis is enabled if bit 0 (ROTx) of parameter No. 1006 and bit 0 (RFDx) of parameter No. 1408 are 1.
2 The imaginary radius parameter No. 1465 of the feedrate instruction on imaginary circle for a rotary axis can be overwritten with a programmable parameter input (G10).
3 If the feedrate instruction on imaginary circle for a rotary axis is enabled, setting 0 in the imaginary radius parameter No. 1465 and only a rotary axis is specified, a movement occurs with the maximum cutting feed.
4 Great care is required for the settings of bit 0 (RFDx) of parameter No. 1408 and the imaginary radius parameter No. 1465. Of particular note is that if a small value is set for the imaginary radius and the feedrate instruction on imaginary circle for a rotary axis is used, the movement on the axis is faster than when the command is not used.
5 If feed axis synchronization control is used in the feedrate instruction on imaginary circle for a rotary axis, the settings of parameters Nos. 1408 and 1465 for the master axis are also used for the slave axis.

5.6 DWELL

Format

M

G04 X_; or G04 P_;

X_ : Specify a time or spindle speed (decimal point permitted)
P_ : Specify a time or spindle speed (decimal point not permitted)

T

G04 X_ ; or G04 U_ ; or G04 P_ ;

X_ : Specify a time or spindle speed (decimal point permitted)
U_ : Specify a time or spindle speed (decimal point permitted)
P_ : Specify a time or spindle speed (decimal point not permitted)

Explanation

By specifying a dwell, the execution of the next block is delayed by the specified time. (Dwell per second)

By setting bit 1 (DWL) of parameter No. 3405 in the feed per revolution mode, the execution of the next block is delayed until the rotation count of the spindle reaches the specified number. (Dwell per revolution)

Table 5.6 (a) Command value range of the dwell time (Command by X or U)

Increment system	Command value range	Dwell time unit
IS-A	0.01 to 999999.99	s or rev
IS-B	0.001 to 99999.999	
IS-C	0.0001 to 9999.9999	

FRYER MACHINE SYSTEMS INC Rob Zahensky

Table 5.6 (b) Command value range of the dwell time (Command by P)

Increment system	Command value range	Dwell time unit
IS-A	1 to 99999999	0.01 s or rev
IS-B	1 to 99999999	0.001 s or rev
IS-C	1 to 99999999	0.0001 s or rev

In the case of dwell per second, the specification unit for dwell time specified with P can be fixed at 0.001 second by setting bit 7 (DWT) of parameter No. 1015 to 1.

NOTE

1 When X, U, or P is specified without a decimal point, the specification unit does not depend on inch/metric input. Depending on whether the X-axis is present, the following increment system is used:
   - When the X-axis is present
     The increment system of the X-axis is used.
   - When the X-axis is not present
     The increment system of the reference axis is used.

2 When P is specified, bit 7 (IPR) of parameter No. 1004 exercises no influence.

3 Specify dwell in a single block specifying no other codes.

While dwell is being executed, 1 is set for diagnosis data No. 0002.

M

Specify dwell also to make an exact stop in the cutting mode (G64 mode).
If the specification of P and X is omitted, an exact stop occurs.

Diagnosis data

2	Dwell execution status

While dwell is being executed, 1 is displayed, and 0 is displayed during the non-execution.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

6 REFERENCE POSITION

A CNC machine tool has a special position where, generally, the tool is exchanged or the coordinate system is set, as described later. This position is referred to as a reference position.

6.1 REFERENCE POSITION RETURN

Overview

- Reference position

The reference position is a fixed position on a machine tool to which the tool can easily be moved by the reference position return function.
For example, the reference position is used as a position at which tools are automatically changed. Up to four reference positions can be specified by setting coordinates in the machine coordinate system in parameters Nos. 1240 to 1243.

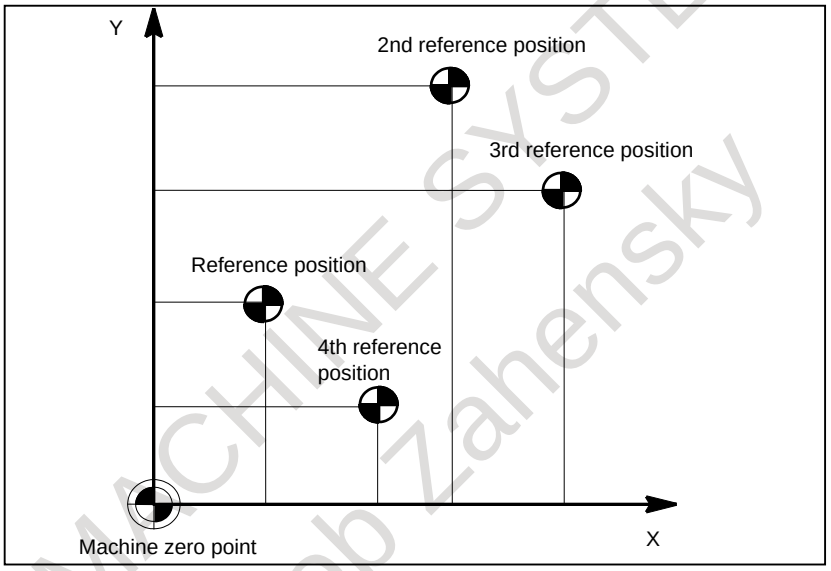


Fig. 6.1 (a) Machine zero point and reference positions

- Automatic reference position return (G28) and movement from the reference position (G29)

The automatic reference position return (G28) function returns tools automatically to the reference position via an intermediate position along a specified axis.
The intermediate position means the relay point for the axes movement in order to avoid collision when there is the interference object on the way of the reference position return.
When reference position return is completed, the lamp for indicating the completion of reference position return goes on. The return from reference position (G29) function moves tools from the reference position to a specified position via an intermediate position along a specified axis.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

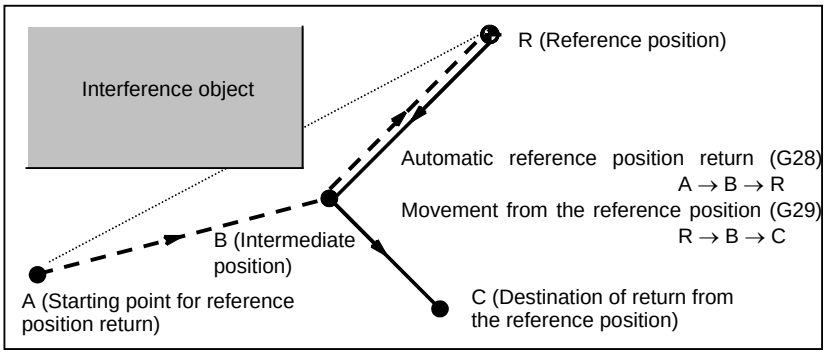


Fig. 6.1 (b) Reference position return and movement from the reference position

If you want to return directly the axes to the reference position from the current position, please specify the incremental command of travel distance zero or the absolute command to current position in the G28 block of the return axes. In this case, an intermediate position is regarded to the current position. The axes are moved to commanded position after passing the starting point of reference position return when the G29 command is specified.

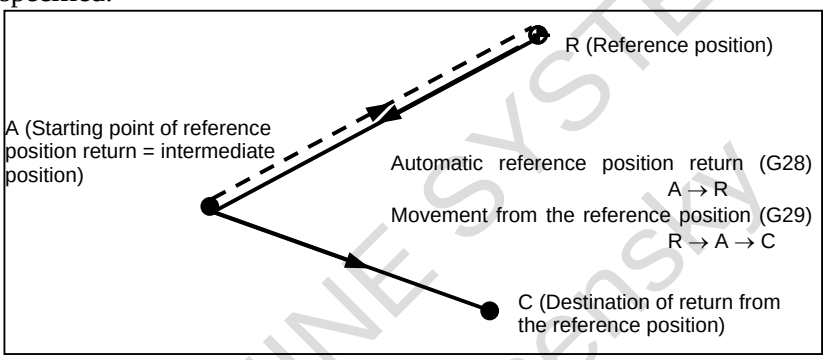


Fig. 6.1 (c) Reference position return and movement from the reference position

- Reference position return check (G27)

The reference position return check (G27) is the function which checks whether the tool has correctly returned to the reference position as specified in the program. If the tool has correctly returned to the reference position along a specified axis, the lamp for the axis for indicating the completion of reference position return goes on.

If the tool has not reached the reference position, an alarm PS0092, "ZERO RETURN CHECK (G27) ERROR" is issued.

When no movement was made along the axis, whether the current position is the reference position is checked.

- In-position check disable reference position return (G28.2, G30.2)

You can disable in-position check at a middle point and reference position by specifying G28.2 or G30.2 as a reference point return command.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Format

- Automatic reference position return and 2nd/3rd/4th reference position return In-position check disable reference position return

G28 IP_;	Reference position return
G30 P2 IP_;	2nd reference position return (P2 can be omitted.)
G30 P3 IP_;	3rd reference position return
G30 P4 IP_;	4th reference position return
G28.2 IP_;	In-position check disable reference position return
G30.2 P2 IP_;	In-position check disable 2nd reference position return (P2 can be omitted.)
G30.2 P3 IP_;	In-position check disable 3rd reference position return
G30.2 P4 IP_;	In-position check disable 4th reference position return

IP : Specify the intermediate position in the absolute coordinate system. (absolute/incremental programming)
There is no need to calculate an actual travel distance between the intermediate position and the reference position.

- Movement from reference position

G29 IP_;

IP : Specify the destination of return from the reference position in the absolute coordinate system. (absolute/incremental programming)
The intermediate position is determined by G28 or G30 specified immediately before this command.

- Reference position return check

G27 IP_;

IP : Specify positioning to the reference position in the absolute coordinate system so as to return to the reference position. (absolute/incremental programming)

Explanation

- Automatic reference position return (G28)

Positioning to the intermediate or reference positions are performed at the rapid traverse rate of each axis. Therefore, for safety, the compensation functions, such as the tool radius compensation and tool length compensation, should be cancelled before executing this command.
The coordinates for the intermediate position are stored in the CNC for the axes for which a value is specified in a G28 block. For the other axes, the previously specified coordinates are used.

(Example)

N1 G28 X40.0 ;	(The tool moves to the reference position along the X-axis and the intermediate position (X40.0) is stored.)
N2 G28 Y60.0 ;	(The tool moves to the reference position along the Y-axis and the intermediate position (Y60.0) is stored.)
N3 G29 X10.0 Y20.0 ;	(The tool moves to the position specified with G29 via the intermediate position (X40.0 Y60.0) previously specified with G28 along the X-axis and Y-axis.)

FRIEP MACHINE SYSTEMS INC
Rob Zahensky

- 2nd, 3rd, and 4th reference position return (G30)

The 2nd, 3rd, and 4th reference position return (G30) function can be used after the reference positions are established.
The G30 command is generally used when the automatic tool changer (ATC) position differs from the reference position.

- Movement from the reference position (G29)

This function is executed after the tool is returned to the reference position by G28 or G30.
For incremental programming, the command value specifies the incremental value from the intermediate point.
The tool moves to the intermediate and specified positions at the feedrate specified with a parameter.
When the workpiece coordinate system is changed after the tool reaches the reference position through the intermediate point by the G28 command, the intermediate point also shifts to a new coordinate system. If G29 is then commanded, the tool moves to the commanded position through the intermediate point which has been shifted to the new coordinate system.
The same operations are performed also for G30 command.
After the power is turned on, an alarm PS0305, “INTERMEDIATE POSITION IS NOT ASSIGNED” is issued if an attempt is made to execute G29 (movement from the reference position) before G28 (automatic reference position return) or G30 (2nd, 3rd, and 4th reference position return) is executed.

- Reference position return check (G27)

G27 command positions the tool at rapid traverse rate. If the tool reaches the reference position, the lamp for indicating the completion of reference position return lights up.
When the tool returns to the reference position along only one axis, the lamp for the axis for indicating the completion of reference position return lights up.
After positioning, if the tool has not reached the reference position along a specified axis, an alarm PS0092, "ZERO RETURN CHECK (G27) ERROR" is issued.
When no movement was made along the axis, whether the current position is the reference position is checked.

- In-position check disable reference position return (G28.2, G30.2)

You can disable in-position check at a middle point and reference position by specifying G28.2 or G30.2 as a reference point return command.
Disabling the in-position check at the middle point and reference position reduces the cycle time.
Note that, if the G28.2 command causes reference position return at low speed, the in-position check is disabled at the middle point but enabled at the reference position.
In the operations of and restrictions on reference position return, G28.2 and G30.2 are the same as G28 and G30, except for in-position check.

- Setting of the reference position return feedrate

Before a reference position is established with the first reference position return after power-on, the manual and automatic reference position return feedrates and automatic rapid traverse rate conform to the setting of parameter No. 1428 for each axis.
After a reference position is established upon the completion of reference position return, the manual reference position return feedrate conforms to the setting of the parameter No. 1428 for each axis.

FRYER MACHINE SYSTEMS INC
Rob Zalensky

NOTE

1 To this feedrate, a rapid traverse override (F0,25,50,100%) is applied, for which the setting is 100%.

2 After a reference position has been established upon the completion of reference position return, the automatic reference position return feedrate will conform to the ordinary rapid traverse rate.

3 When a value is set for parameter No. 1428, the feedrates conform to the parameter settings shown below.

1420: Rapid traverse rate 1423: Jog feedrate
1424: Manual rapid traverse rate 1428: Reference position return feedrate

	Before a reference position is established	After a reference position is established
Automatic reference position return (G28)	No. 1428	No.1420
Automatic rapid traverse (G00)	No.1428	No.1420
Manual reference position return (*1)	No.1428	No.1428 (*3)
Manual rapid traverse rate	No.1423 (*2)	No.1424

When parameter No. 1428 is set to 0, the feedrates conform to the parameter settings shown below.

	Before a reference position is established	After a reference position is established
Automatic reference position return (G28)	No. 1420	No.1420
Automatic rapid traverse (G00)	No.1420	No.1420
Manual reference position return (*1)	No.1424	No.1424 (*3)
Manual rapid traverse rate	No.1423 (*2)	No.1424

*1 By using bit 2 (JZR) of parameter No. 1401, the manual reference position return feedrate can always be set as a jog feedrate.

*2 When bit 0 (RPD) of parameter No. 1401 is 1, the setting of parameter No. 1424 (manual rapid traverse rate) is used.
When the setting of parameter No. 1424 (manual rapid traverse rate) is 0, parameter No. 1420 (rapid traverse rate) is used.

*3 When reference position return without dogs is performed in rapid traverse mode, or when manual reference position return is performed in rapid traverse mode regardless of deceleration dogs after a reference position is established, the reference position return feedrate for each of these functions (setting of bit 1 (DLF) of parameter No. 1404) is used.

Limitation

- Status the machine lock being turned on

The lamp for indicating the completion of reference position return does not go on when the machine lock is turned on, even when the tool has automatically returned to the reference position. In this case, it is not checked whether the tool has returned to the reference position even when a reference position return check command is specified.

- When automatic reference position return (G28) is executed if no reference position is established

When automatic reference position return (G28) is executed if no reference position is established, movement from the intermediate position in a reference position direction is the same as that in manual reference position return.
(This movement is referred to as a low-speed type of automatic reference position return (G28).)

FRYER MACHINE SYSTEMS INC Rob Zahensky

In this case, the tool moves in the direction for reference position return specified in bit 5 (ZMIx) of parameter No. 1006. Therefore the specified intermediate position must be a position to which reference position return is possible.

NOTE

When automatic reference position return (G28) is executed after a reference position is established, positioning is performed from the intermediate position to the reference position. This movement is referred to as a high-speed type of automatic reference position return (G28).

- Reference position return check in an offset mode

In an offset mode, the position to be reached by the reference position return check is the position obtained by adding the offset value.
Therefore, if the position with the offset value added is not the reference position, the lamp for indicating the completion of reference position return does not light up, but an alarm is displayed instead. Usually, cancel offsets before G27 is commanded.

- Lighting the lamp when the programmed position does not coincide with the reference position

When the machine tool system is an inch system with metric input, the lamp for indicating the completion of reference position return may also light up even if the programmed position is shifted from the reference position by the least setting increment. This is because the least setting increment of the machine tool system is smaller than its least command increment.

- Automatic reference position return (G28) if roll over is used on a rotation axis (type A)

If, in an absolute command for automatic reference position return (G28) on a rotation axis, the rotation axis is of type A (bit 0 (ROT) of parameter No. 1006 = 1 and bit 1 (ROS) of parameter No. 1006 = 0) and roll over is used (bit 0 (ROA) of parameter No. 1008 = 1), movement to the midpoint follows the setting of bit 1 (RAB) of parameter No. 1008, and movement from the midpoint to the reference position follows the setting of bit 5 (ZMI) of parameter No. 1006.
If, however, bit 6 (RRF) of parameter No. 1008 is set to 1, movement from the midpoint to the reference position also follows the setting of bit 1 (RAB) of parameter No. 1008.

- Restriction on automatic reference position return (G28)

There are restrictions when the following functions are used. Please refer to each function for detail.

- Mirror image
- Plane conversion
- Machine lock
- Extension of the manual handle retrace function
- Improvement of tool compensation for tool retract and recover
- Retrace
- Positioning by optimum accelerations
- In-position check disable signal
- Axis switching
- Spindle electronic gear box
- Superimposed control
- Simple spindle electronic gear box
- Electronic gear box
- Figure copying
- Programmable mirror image
- Flexible synchronization control
- Inter-path flexible synchronization control

FRYER MACHINE SYSTEMS INC
Ron Zalensky

- High precision oscillation function
- High-speed program check function
- Synchronous control
- Composite control
- Tool functions of lathe system
- Normal direction control
- Compensation function
- Coordinate system rotation
- Cutter compensation and tool nose radius compensation
- Tool length compensation
- Scaling

- Restriction on 2nd, 3rd, and 4th reference position return (G30)

There are restrictions when the following functions are used. Please refer to each function for detail.

- Plane conversion
- Machine lock
- Improvement of tool compensation for tool retract and recover
- Retrace
- In-position check disable signal
- Axis switching
- Superimposed control
- Electronic gear box
- Programmable mirror image
- Flexible synchronization control
- Tool functions of lathe system
- Normal direction control
- Compensation function
- Coordinate system rotation
- Cutter compensation and tool nose radius compensation
- Tool length compensation
- Scaling

- Restriction on reference position return check (G27)

There are restrictions when the following functions are used. Please refer to each function for detail.

- Machine lock
- Retrace
- Superimposed control
- Electronic gear box
- Programmable mirror image
- Flexible synchronization control
- Compensation function
- Coordinate system rotation
- Scaling

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Example

- When the reference position return is executed directly from the current position without passing an intermediate position.

G28G91X0Y0 ; (The axes are directly moved to reference position R from starting point A without passing an intermediate position.)

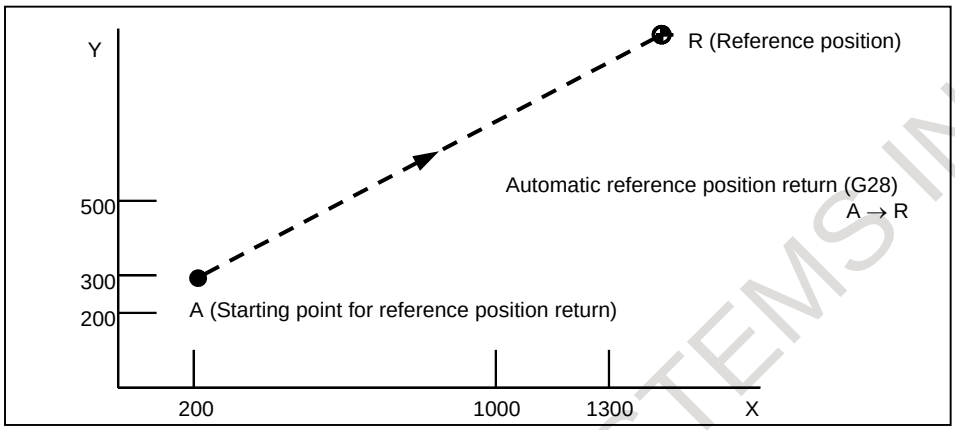


Fig. 6.1 (d) Reference position return and movement from the reference position

- When the movement passes an intermediate position.

G28G90X1000.0Y500.0 ; (Programs movement from A to B. The tool moves to reference position R via intermediate position B.)

T1111 ; (Changing the tool at the reference position)

G29X1300.0Y200.0 ; (Programs movement from B to C. The tool moves from reference position R to C specified with G29 via intermediate position B.)

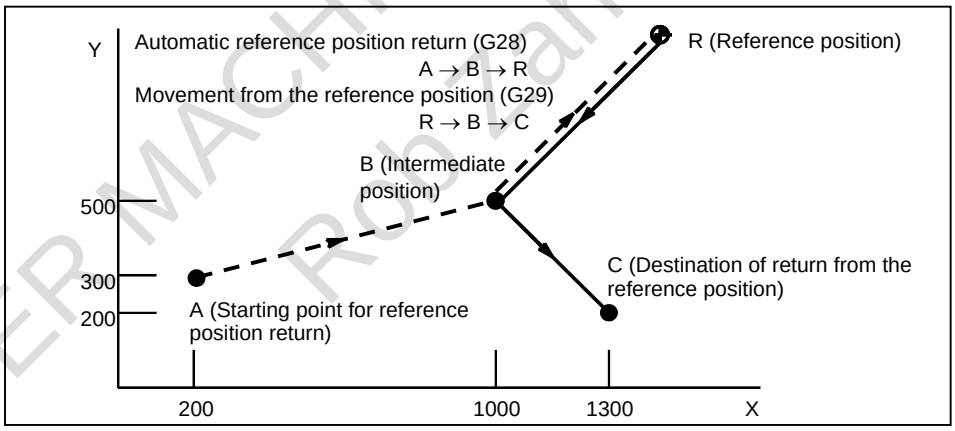


Fig. 6.1 (e) Reference position return and movement from the reference position

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

7 COORDINATE SYSTEM

By teaching the CNC a desired tool position, the tool can be moved to the position. Such a tool position is represented by coordinates in a coordinate system. Coordinates are specified using program axes.
When three program axes, the X-axis, Y-axis, and Z-axis, are used, coordinates are specified as follows:
X_Y_Z_
This command is referred to as a dimension word.

- For milling machining (Tool position specified by X40.0Y50.0Z25.0)

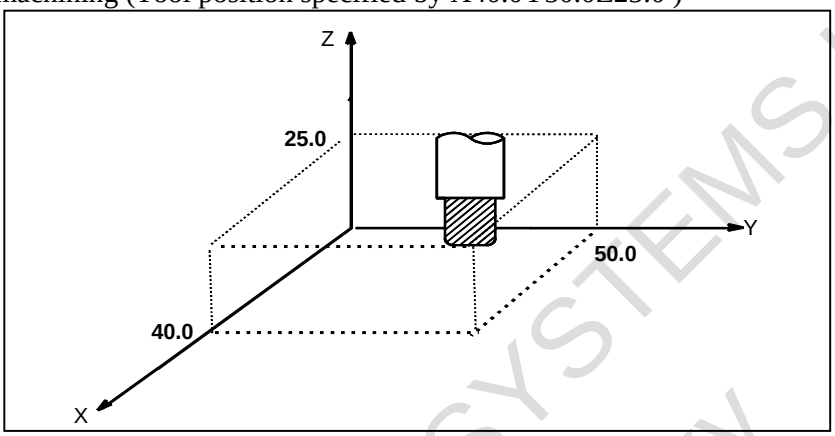


Fig. 7 (a)

- For lathe cutting (Tool position specified by X50.0 Z40.0)

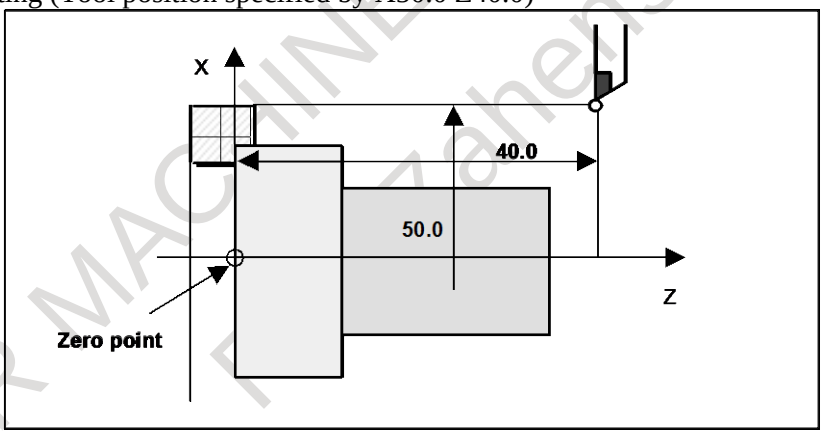


Fig. 7 (b)

Coordinates are specified in one of following three coordinate systems:
(1) Machine coordinate system
(2) Workpiece coordinate system
(3) Local coordinate system
The number of the axes of a coordinate system varies from one machine to another. So, in this manual, a dimension word is represented as IP_.

FRYER MACHINE SYSTEMS INC
R. Zahensky

7.1 MACHINE COORDINATE SYSTEM

The point that is specific to a machine and serves as the reference of the machine is referred to as the machine zero point. A machine tool builder sets a machine zero point for each machine.
A coordinate system with a machine zero point set as its origin is referred to as a machine coordinate system.
A machine coordinate system is set by performing manual reference position return after power-on (see III Section 3.1, “MANUAL REFERENCE POSITION RETURN”). A machine coordinate system, once set, remains unchanged until the power is turned off.
The reference position is not always the origin of the machine coordinate system.
(See Item, "Setting a machine coordinate system" described later.)

Format

M

(G90)G53 IP _ P1;

IP_ : Absolute command dimension word
P1 : Enables the high-speed G53 function.

(G90)G53.2 G01 IP__F__;

IP_ : Absolute command dimension word
F_ : Feedrate

T

G53 IP _ P1;

IP_ : Absolute command dimension word
P1 : Enables the high-speed G53 function.

G53.2 G01 IP__F__;

IP_ : Absolute command dimension word
F_ : Feedrate

Explanation

- Selecting a machine coordinate system (G53)

When a command is specified the position on a machine coordinate system, the tool moves to the position by rapid traverse. G53, which is used to select a machine coordinate system, is a one-shot G code; that is, it is valid only in the block in which it is specified on a machine coordinate system. Specify an absolute command for G53. When an incremental command is specified, the G53 command is ignored. When the tool is to be moved to a machine-specific position such as a tool change position, program the movement in a machine coordinate system based on G53.

- High-speed G53 function

This function enables the inter-rapid traverse block overlap function between machine coordinate selection command (G53) and positioning (rapid traverse) command (G00) blocks, thus making it possible to execute the next rapid traverse command (G00) without decelerating to a stop at the end of the machine coordinate selection command (G53). Therefore, high-speed positioning is available even when the machine coordinate selection command (G53) is used.
Specifying P1 in a G53 block enables the high-speed G53 function.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Selecting a machine coordinate system with feedrate (G53.2)

Positioning of machine coordinate system at a feed rate is available with the command of G53.2. The feed rate can be used in the modal of G01.
G53.2 is one shot G code. Moreover, the tool offset value is temporarily canceled by G53.2 command like G53.
As for the time constant, a usual feed rate are used. At the point of G53.2 command, the axis decelerates to a stop once like G53 command.
Feed per minute, feed per revolution, and inverse time feed of G codes of 05 group are available.
When group 01 G-code except G00/G01 is commanded with G53.2, the alarm PS5372, “IMPROPER MODAL G-CODE” is generated. And, when G53.2 is commanded while group 01 Modal G-code is the one except G00/G01, the alarm PS5372, “IMPROPER MODAL G-CODE (G53.2)” is generated.

Limitation

- Cancel of the compensation function

When the G53 command is specified, cancel the compensation functions such as the cutter compensation, tool length compensation, tool nose radius compensation, and tool offset beforehand.

- G53 command immediately after power supply turning on

The Reference point by the manual Reference point return or G28 command should return once after turning on the power supply because it should set the machine coordinate system before it instructs in G53.
However, the operation is unnecessary for the machine with the absolute-position detector.

- Specification in the same block

M

Commands G50/G51, G50.1/G51.1, and G68/G69 cannot be specified in the same block where the G53 command is specified.

T

Commands G50/G51 (except for G code system A), G50.1/G51.1, and G68.1/G69.1 cannot be specified in the same block where the G53 command is specified.

Note

NOTE
G53 is a G code for disabling buffering.

FRYER MACHINE SYSTEMS Inc
Rob Zahensky

Reference

- Setting a machine coordinate system

When manual reference position return is performed after power-on, a machine coordinate system is set so that the reference position is at the coordinate values of (α, β) set using parameter No.1240.

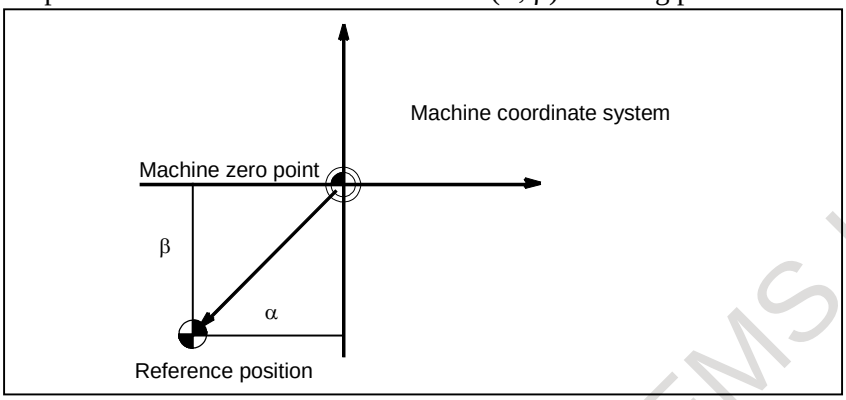


Fig. 7.1 (a)

Example (selecting a machine coordinate system with specifying a feedrate)

```
N1 G90 G01 ;
N2 G53.2 X50.0 Y100.0 F1000 ; Absolute command with feedrate F1000
N3 G53.2 X150.0 F500 ;        Absolute command with feedrate F500
```

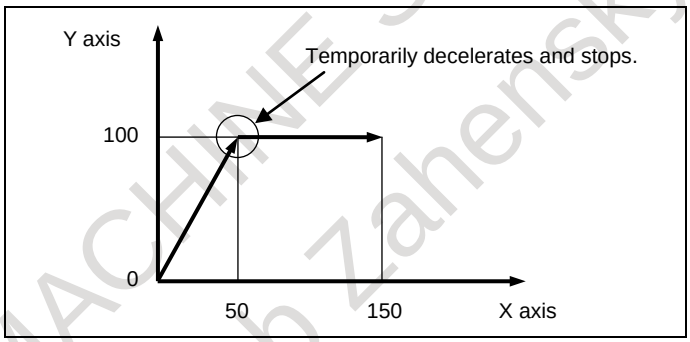


Fig. 7.1 (b)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

7.2 WORKPIECE COORDINATE SYSTEM

Overview

A coordinate system used for machining a workpiece is referred to as a workpiece coordinate system. A workpiece coordinate system is to be set with the CNC beforehand (setting a workpiece coordinate system).
A machining program sets a workpiece coordinate system (selecting a workpiece coordinate system).
A set workpiece coordinate system can be changed by shifting its origin (changing a workpiece coordinate system).

7.2.1 Setting a Workpiece Coordinate System

A workpiece coordinate system can be set using one of three methods:

(1) Method using a workpiece coordinate system setting G code
A workpiece coordinate system is set by specifying a value in the program after a workpiece coordinate system setting G code.

(2) Automatic setting
If bit 0 (ZPR) of parameter No. 1201 is set to 1, a workpiece coordinate system is automatically set when manual reference position return is performed. (See Section, “MANUAL REFERENCE POSITION RETURN”.)

(3) Method using a workpiece coordinate system selection G code
Six workpiece coordinate systems can be set beforehand using the MDI unit. Program commands G54 to G59 can be used to select the workpiece axis to be used. (See Subsection, “Displaying and Setting the Workpiece Origin Offset Value”.)

When using an absolute command, establish the workpiece coordinate system in any of the above ways.

Format

- Setting a workpiece coordinate system

M

```
(G90) G92 IP_
```

T

```
G50 IP_
```

Explanation

A workpiece coordinate system is set so that a point on the tool, such as the tool tip, is at specified coordinates.

M

If a coordinate system is set using G92 during tool length offset, a coordinate system in which the position before offset matches the position specified in G92 is set. Cutter compensation is cancelled temporarily with G92.

T

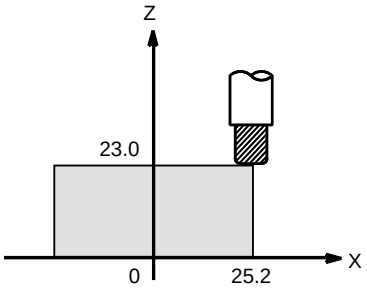
If IP_ is an incremental command value, the workpiece coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position.
When bit 0 (WAB) of parameter No. 11279 is set to 1, the setting of the workpiece coordinate system is executed with absolute value even if the incremental mode (G91) in the G-code system B/C of the lathe system is selected.
If a coordinate system is set using G50 during offset, a coordinate system in which the position before offset matches the position specified in G50 is set.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

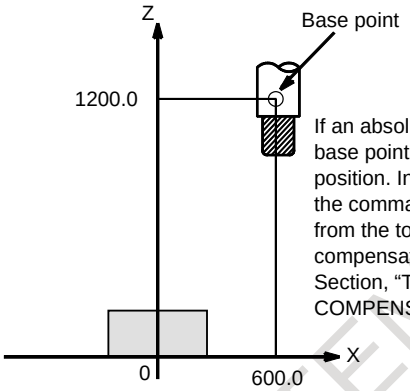
Example

M

(Example 1)
Setting the coordinate system by the G92X25.2Z23.0; command (The tool tip is the start point for the program.)



(Example 2)
Setting the coordinate system by the G92X600.0Z1200.0; command (The base point on the tool holder is the start point for the program.)

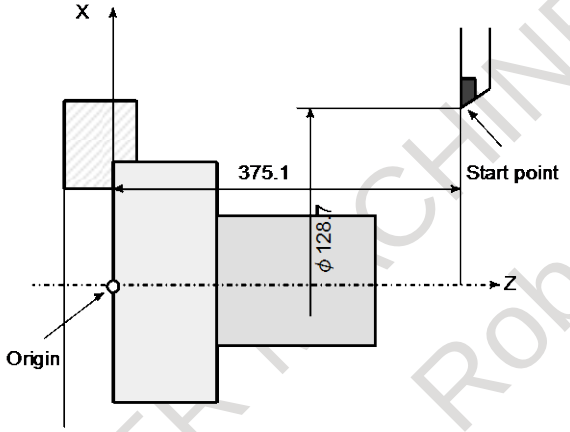


If an absolute command is issued, the base point moves to the commanded position. In order to move the tool tip to the commanded position, the difference from the tool tip to the base point is compensated by tool length offset (see Section, "TOOL LENGTH COMPENSATION (G43, G44, G49)").

Fig. 7.2.1 (a)

T

(Example 1)
Setting the coordinate system by the G50X128.7Z375.1; command (Diameter designation) (The tool nose is the start point for the program.)



(Example 2)
Setting the coordinate system by the G50X1200.0Z700.0; command (Diameter designation) (The base point on the turret is the start point for the program.)

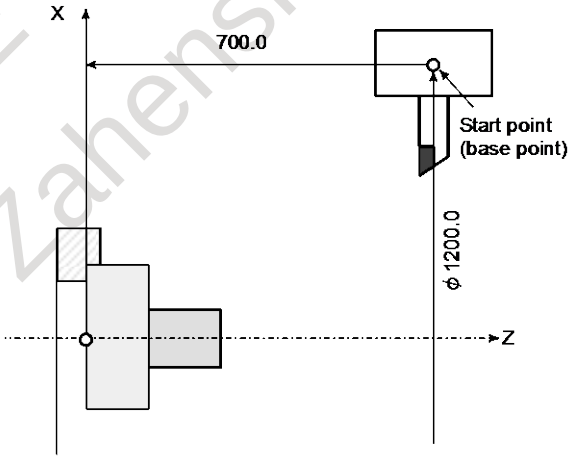


Fig. 7.2.1 (b)

> ⚠ **CAUTION**
> The set workpiece coordinate system depends on diameter programming or radius programming.

Notes

- Manual reference position return

When the manual reference position return is performed, the value of the workpiece coordinate system set by G code (G92, or G50 for the lathe system G-code system A) is cleared to 0. And when the automatic reference position return (G28) is performed, this workpiece coordinate system is not cleared to 0 but stored.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Command for setting a workpiece coordinate system / clamp at maximum spindle speed in the tool length compensation mode

Executing a workpiece coordinate system setting G code command (G92 or, for G code system A in the lathe system, G50) presets a coordinate system in such a way that the specified position will be a pre-compensation position.
However, setting for workpiece coordinate system / clamp at maximum spindle speed cannot be used together with a block where tool length compensation vectors vary. For example, it cannot be used together with the following blocks.

Example
1 Block in which G43/G44 is issued
2 Block which is in the G43 or G44 mode and in which an H code is issued
3 Block which is in the G43 or G44 mode and in which G49 is issued
4 Block in which, in the G43 or G44 mode, compensation vectors are canceled using a G code such as G28 or G53 and then resumed again

The current tool length offset can not be changed by following method:
Stop operation just before this G code, and change the correct offset by MDI or other method.

7.2.2 Selecting a Workpiece Coordinate System

The user can choose from set workpiece coordinate systems as described below. (For information about the methods of setting, see Subsection, “Setting a Workpiece Coordinate System”.)

(1) Once a workpiece coordinate system is set by a workpiece coordinate system setting G code or by automatic workpiece coordinate system setting, absolute commands indicate positions in the workpiece coordinate system.

(2) Choosing from six workpiece coordinate systems set using the MDI unit
By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.

G54 :	Workpiece coordinate system 1	G55 :	Workpiece coordinate system 2
G56 :	Workpiece coordinate system 3	G57 :	Workpiece coordinate system 4
G58 :	Workpiece coordinate system 5	G59 :	Workpiece coordinate system 6

Workpiece coordinate system 1 to 6 are established after reference position return after the power is turned on. When the power is turned on, G54 coordinate system is selected.

When bit 2 (G92) of parameter No. 1202 is set to 1, executing the workpiece coordinate system setting G92 code command results in the issue of an alarm PS0010, “IMPROPER G-CODE”. This is designed to prevent the user from confusing coordinate systems.

⚠ **CAUTION**
The set workpiece origin offset value depends on diameter programming or radius programming.

FRYER MACHINE SYSTEMS INC
Ron Zalensky

Example

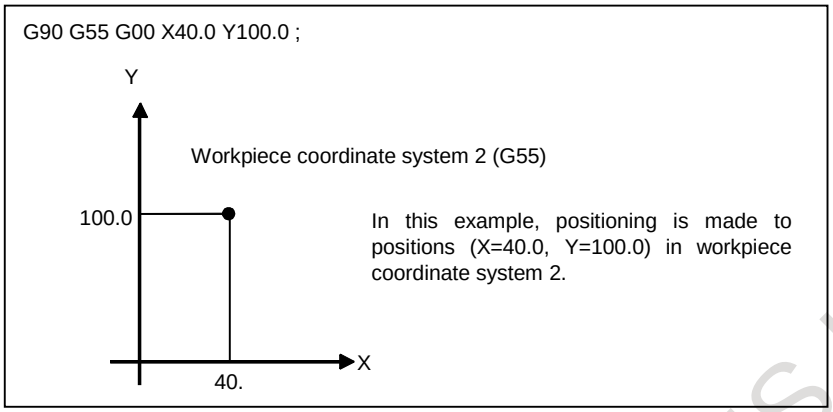


Fig. 7.2.2

7.2.3 Changing Workpiece Coordinate System

The six workpiece coordinate systems specified with G54 to G59 can be changed by changing an external workpiece origin offset value or workpiece origin offset value.
Three methods are available to change an external workpiece origin offset value or workpiece origin offset value.

(1) Inputting from the MDI unit (see Subsection, “Displaying and Setting the Workpiece Origin Offset Value”.)
(2) Programming (using a programmable data input G code or a workpiece coordinate system setting G code)
(3) Using the external data input function
An external workpiece origin offset value can be changed by input signal to CNC. Refer to machine tool builder's manual for details.

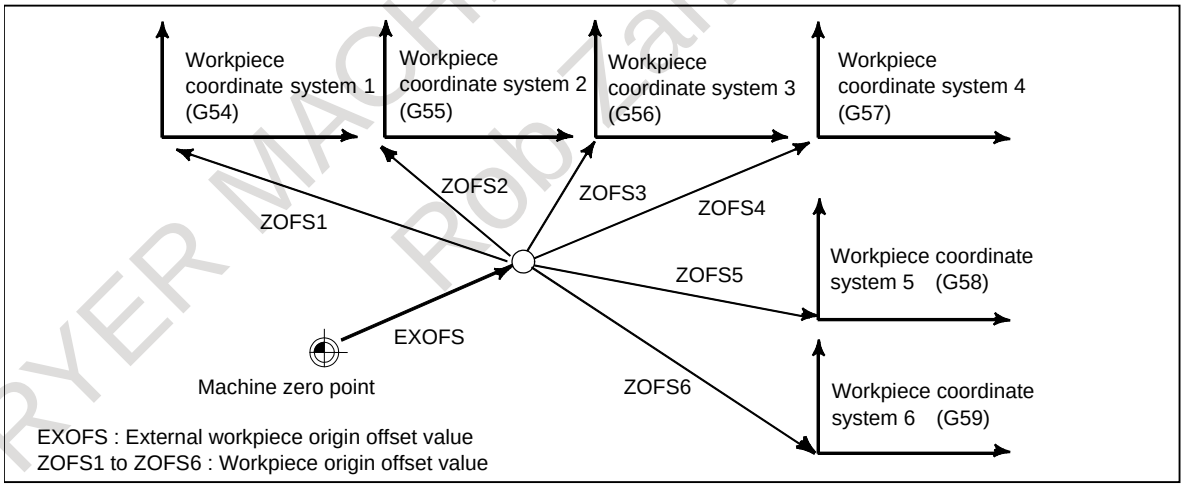


Fig. 7.2.3 (a) Changing an external workpiece origin offset value or workpiece origin offset value

Format

- Changing by inputting programmable data

G10 L2 Pp IP_;

p=0 : External workpiece origin offset value
p=1 to 6 : Workpiece origin offset value correspond to workpiece coordinate system 1 to 6
IP_ : For an absolute command, workpiece origin offset for each axis.
For an incremental command, value to be added to the set workpiece origin offset for each axis (the result of addition becomes the new workpiece origin offset).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Changing by setting a workpiece coordinate system

M

```
G92 IP_ ;
```

T

```
G50 IP_ ;
```

Explanation

- Changing by inputting programmable data

By specifying a programmable data input G code, the workpiece origin offset value can be changed for each workpiece coordinate system.

- Changing by setting a workpiece coordinate system

By specifying a workpiece coordinate system setting G code, the workpiece coordinate system (selected with a code from G54 to G59) is shifted to set a new workpiece coordinate system so that the current tool position matches the specified coordinates (IP_).
Then, the amount of coordinate system shift is added to all the workpiece origin offset values. This means that all the workpiece coordinate systems are shifted by the same amount.

> ⚠ **CAUTION**
> When a coordinate system is set with workpiece coordinate system setting G92 code command after an external workpiece origin offset value is set, the coordinate system is not affected by the external workpiece origin offset value. When G92X100.0Z80.0; is specified, for example, the coordinate system having its current tool reference position at X = 100.0 and Z = 80.0 is set.

T

If IP is an incremental command value, the workpiece coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position. (Coordinate system shift)

Example

M

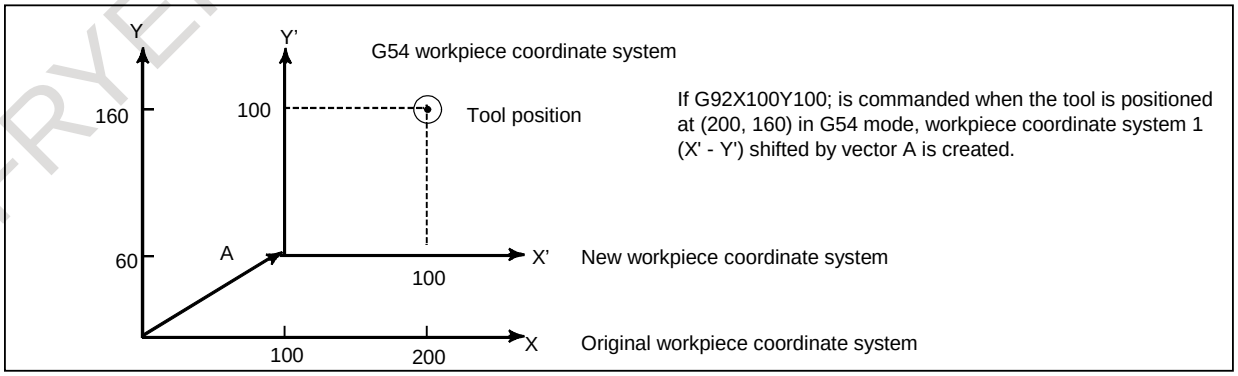
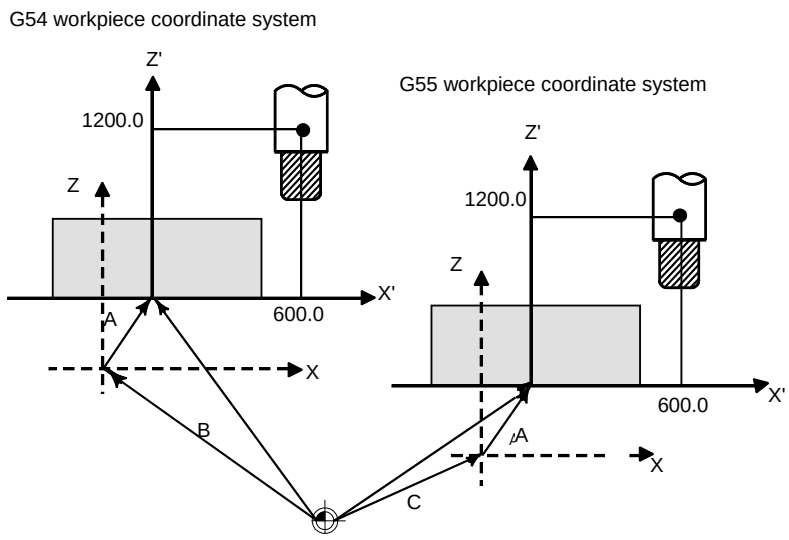


Fig. 7.2.3 (b)

FRYER MACHINE SYSTEMS INC
Rob Zaharsky



Suppose that a G54 workpiece coordinate system is specified. Then, a G55 workpiece coordinate system where the black circle on the tool (figure at the left) is at (600.0,1200.0) can be set with the following command if the relative relationship between the G54 workpiece coordinate system and G55 workpiece coordinate system is set correctly:

G92X600.0Z1200.0;

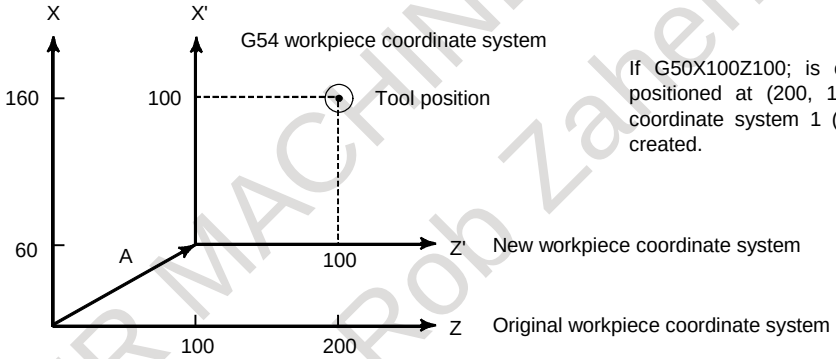
Also, suppose that pallets are loaded at two different positions. If the relative relationship of the coordinate systems of the pallets at the two positions is correctly set by handling the coordinate systems as the G54 workpiece coordinate system and G55 workpiece coordinate system, a coordinate system shift with G92 in one pallet causes the same coordinate system shift in the other pallet. This means that workpieces on two pallets can be machined with the same program just by specifying G54 or G55.

X' - Z' .....New workpiece coordinate system
X - Z .....Original workpiece coordinate system
A : Offset value created by G92
B : Workpiece origin offset value in the G54
C : Workpiece origin offset value in the G55

Fig. 7.2.3 (c)

Example

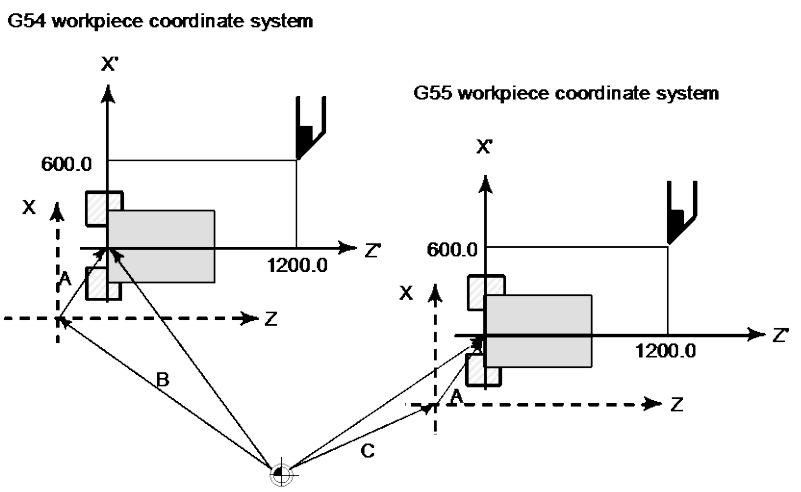
T



If G50X100Z100; is commanded when the tool is positioned at (200, 160) in G54 mode, workpiece coordinate system 1 (X' - Z') shifted by vector A is created.

Fig. 7.2.3 (d)

FRYER MACHINE SYSTEMS INC
Rob Zahensky



Suppose that a G54 workpiece coordinate system is specified. Then, a G55 workpiece coordinate system where the black tool nose point on the tool (figure at the left) is at (600.0,1200.0) can be set with the following command if the relative relationship between the G54 workpiece coordinate system and G55 workpiece coordinate system is set correctly:

G50X600.0Z1200.0;

Also, suppose that loading is performed at two different places and that the G54 and G55 workpiece coordinate systems are set for these two places. If the relative relationship between the coordinate systems in the two places is correctly set, a coordinate system shift with G50 in one loading place causes the same coordinate system shift in the other loading place. This means that workpieces on the two loading positions can be machined with the same program just by specifying G54 or G55.

X' - Z'.....New workpiece coordinate system
X - Z.....Original workpiece coordinate system
A : Offset value created by G50
B : Workpiece origin offset value in G54
C : Workpiece origin offset value in G55

Fig. 7.2.3 (e)

7.2.4 Workpiece Coordinate System Preset (G92.1)

The workpiece coordinate system preset function presets a workpiece coordinate system shifted by manual intervention to the pre-shift workpiece coordinate system. The latter system is displaced from the machine zero point by a workpiece origin offset value.

There are two methods for using the workpiece coordinate system preset function. One method uses a programmed command. The other uses MDI operations on the absolute position display screen, relative position display screen, and overall position display screen (see Subsection, “Workpiece Coordinate System Preset”).

Format

M

G92.1 IP 0 ;

IP 0 : Specifies axis addresses subject to the workpiece coordinate system preset operation. Axes that are not specified are not subject to the preset operation.

T

G50.3 IP0 ; (G92.1 IP 0; for G code system B or C)

IP 0 : Specifies axis addresses subject to the workpiece coordinate system preset operation. Axes that are not specified are not subject to the preset operation.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

When manual reference position return operation is performed in the reset state, a workpiece coordinate system is shifted by the workpiece origin offset value from the machine coordinate system zero point. Suppose that the manual reference position return operation is performed when a workpiece coordinate system is selected with G54. In this case, a workpiece coordinate system is automatically set which has its origin displaced from the machine zero point by the G54 workpiece origin offset value; the distance from the origin of the workpiece coordinate system to the reference position represents the current position in the workpiece coordinate system.

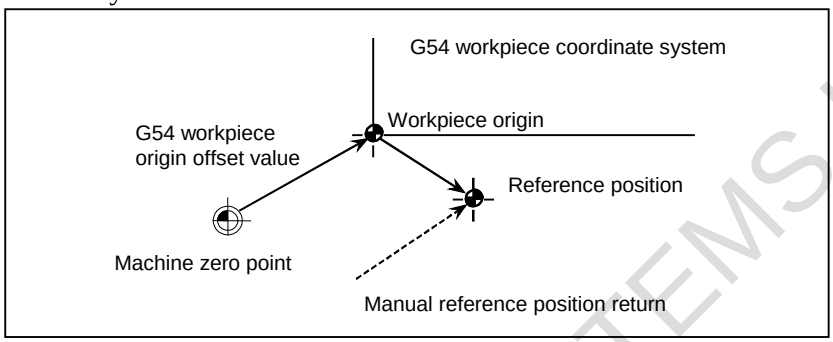


Fig. 7.2.4 (a)

If an absolute position detector is provided, the workpiece coordinate system automatically set at power-up has its origin displaced from the machine zero point by the G54 workpiece origin offset value. The machine position at the time of power-up is read from the absolute position detector and the current position in the workpiece coordinate system is set by subtracting the G54 workpiece origin offset value from this machine position. The workpiece coordinate system set by these operations is shifted from the machine coordinate system using the commands and operations listed below.

(a) Manual intervention performed when the manual absolute signal *ABSM is off
(b) Move command executed in the machine lock state
(c) Movement by handle interruption
(d) Operation using the mirror image function
(e) Shifting the workpiece coordinate system by setting the local coordinate system or workpiece coordinate system

In the case of (a) above, the workpiece coordinate system is shifted by the amount of movement during manual intervention.

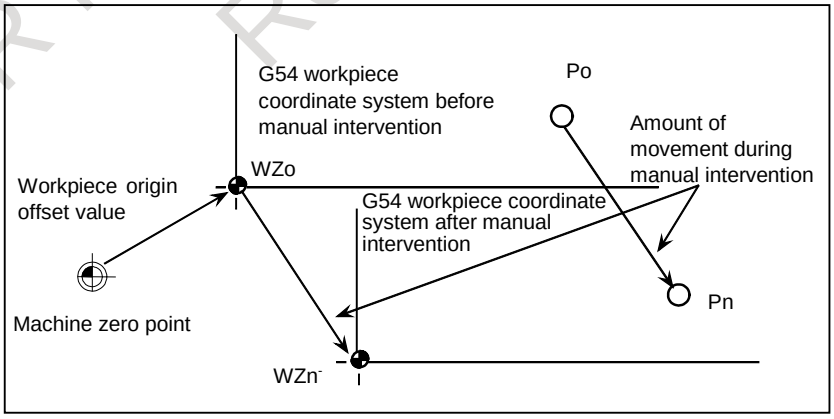


Fig. 7.2.4 (b)

In the operation above, a workpiece coordinate system once shifted can be preset using G code (G92.1) specification or MDI operation (See the section of “Workpiece Coordinate System Preset” in “SETTING AND DISPLAYING DATA”) to a workpiece coordinate system displaced by a workpiece origin offset value from the machine zero point.

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

Bit 3 (PPD) of parameter No. 3104 specifies whether to preset relative coordinates as well as absolute coordinates. Moreover, the difference below the least input increment between absolute coordinate point and relative coordinate point is corrected to the value of the preset(or origin) of relative coordinates when bit 4 (RPP) of parameter No.3129 is set to 1.

Limitation

- **Tool radius · tool nose radius compensation, tool length compensation, tool offset**

When using the workpiece coordinate system preset function, cancel compensation modes: Tool radius · tool nose radius compensation, tool length compensation, and tool offset. If the function is executed without canceling these modes, compensation vectors are cancelled.

M

- **Tool length compensation**

When using the workpiece coordinate system preset function, cancel tool length compensation. If the function is executed without canceling these modes, compensation vectors are cancelled.

- **Program restart**

The workpiece coordinate system preset function is not executed during program restart.

M

- **Prohibited modes**

Do not use the workpiece coordinate system preset function when the scaling, coordinate system rotation, programmable image, or figure copy mode is set.

7.2.5 Addition of Workpiece Coordinate System Pair (G54.1 or G54)

M

Besides the six workpiece coordinate systems (standard workpiece coordinate systems) selectable with G54 to G59, 48 or 300 additional workpiece coordinate systems (additional workpiece coordinate systems) can be used.

Format

- **Selecting the additional workpiece coordinate systems**

G54.1Pn ; or **G54Pn ;**

Pn : Codes specifying the additional workpiece coordinate systems
n : 1 to 48 or 1 to 300

- **Setting the workpiece origin offset value in the additional workpiece coordinate systems (G10)**

G10L20Pn IP_;

Pn : Codes specifying the workpiece coordinate system for setting the workpiece origin offset value
n : 1 to 48 or 1 to 300
IP_ : Axis addresses and a value set as the workpiece origin offset

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

- Selecting the additional workpiece coordinate systems

When a P code is specified together with G54.1 (G54), the corresponding coordinate system is selected from the additional workpiece coordinate systems (1 to 48 or 1 to 300).
A workpiece coordinate system, once selected, is valid until another workpiece coordinate system is selected. Standard workpiece coordinate system 1 (selectable with G54) is selected at power-on.

G54.1 P1..... Additional workpiece coordinate system 1
G54.1 P2..... Additional workpiece coordinate system 2
:
G54.1 P48..... Additional workpiece coordinate system 48
:
G54.1 P300..... Additional workpiece coordinate system 300

As with the standard workpiece coordinate systems, the following operations can be performed for a workpiece origin offset in an additional workpiece coordinate system:

(1) The workpiece origin offset value setting screen can be used to display and set a workpiece origin offset value.
(2) The G10 function enables a workpiece origin offset value to be set by programming (see Subsection, “Changing Workpiece Coordinate System”).
(3) A custom macro allows a workpiece origin offset value to be handled as a system variable.
(4) Workpiece origin offset data can be entered or output as external data.
(5) The PMC window function enables workpiece origin offset data to be read as program command modal data.

- Setting the workpiece origin offset value in the additional coordinate systems (G10)

When a workpiece origin offset value is specified using an absolute value, the specified value is the new offset value. When it is specified using an incremental value, the specified value is added to the current offset value to obtain a new offset value.

Limitation

- Specifying P codes

A P code must be specified after G54.1 (G54). If G54.1 is not followed by a P code in the same block, additional workpiece coordinate system 1 (G54.1P1) is assumed.
If a value not within the specifiable range is specified in a P code, an alarm PS0030, “ILLEGAL OFFSET NUMBER” is issued.
P codes other than workpiece offset numbers cannot be specified in a G54.1 (G54) block.

Example 1) G54.1G04P1000;
Example 2) G54.1M98P48;

FRYER MACHINE SYSTEMS INC
Rob Zaharisky

7.2.6 Automatic Coordinate System Setting

When bit 0 (ZPR) of parameter No. 1201 for automatic coordinate system setting is 1, a coordinate system is automatically determined when manual reference position return is performed.
Once α, β, and γ are set with parameter No. 1250, a workpiece coordinate system is set upon reference position return so that the base point on the tool holder or the tip of the basic tool is positioned at X = α, Y = β, and Z = γ.
This processing occurs as if the following are specified at the reference position:

M

G92XαYβZγ;

T

G50XαZγ;

T

When the setting of a workpiece coordinate system shift amount is other than 0, a workpiece coordinate system shifted by the amount is set.

7.2.7 Workpiece Coordinate System Shift

T

Explanation

When the coordinate system actually set by the G50 command or the automatic system setting deviates from the programmed workpiece system, the set coordinate system can be shifted (see III Section 3.1, “MANUAL REFERENCE POSITION RETURN”).
Set the desired shift amount in the workpiece coordinate system shift memory.

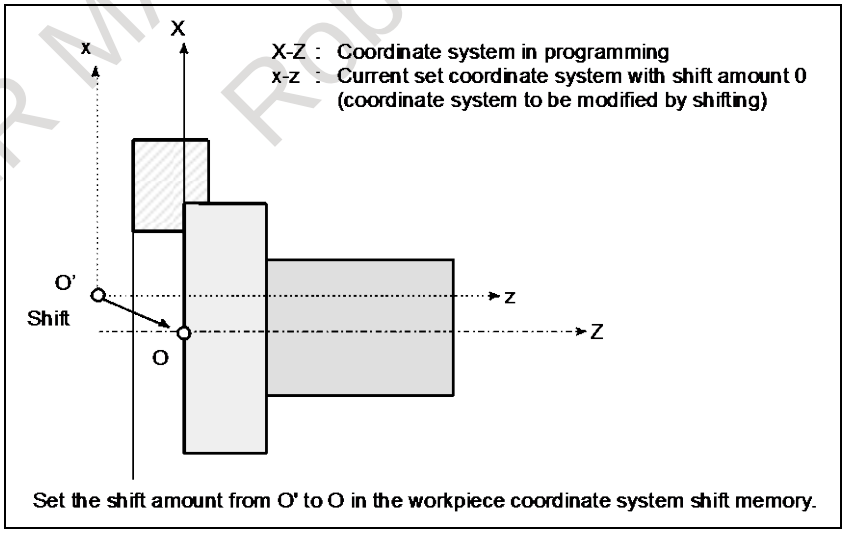


Fig. 7.2.7 (a) Workpiece coordinate system shift

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Format

- Changing the workpiece coordinate system shift amount

G10 P0 IP_;

IP：Settings of an axis address and a workpiece coordinate system shift amount

⚠ **CAUTION**

1 A single block can contain a combination of X, Y, Z, C, U, V, W, and H (in G code system A). In this case, if commands are specified for the same axis, whichever appears later becomes valid.
2 Be sure not to specify other G codes for the same block. When specified, only modal information is updated.
3 Be sure not to specify any auxiliary functions / spindle speed functions / tool functions or 2nd auxiliary functions for the same block. When specified, alarm PS1144 "G10 FORMAT ERROR" is issued.
4 When it commanded with bit 6 (NWS) of parameter No. 1201 is set to 1, alarm PS0010, "IMPROPER G-CODE" is issued.

Limitation

- Shift amount and coordinate system setting command

Specifying a coordinate system setting command (G50 or G92) invalidates the shift amount that has already been set.

Example)

When G50X100.0Z80.0; is specified, a coordinate system is set so that the current base position of the tool is at X =100.0 and Z = 80.0, regardless of which value has been set for the workpiece coordinate system shift amount.

- Shift amount and coordinate system setting

After a shift amount is set, when automatic coordinate system setting is performed upon manual reference position return, the set coordinate system is immediately shifted by the set amount.

- Diameter and radius values

The workpiece coordinate system shift amount depends on diameter programming or radius programming.

Example)

Although the base point should be positioned at X = ϕ120.0 (diameter value) and Z = 70.0 from the workpiece origin, the actual position is at X = ϕ121.0 and Z = 69.0 from the origin. Set a shift amount as shown below:

X=1.0, Z=-1.0

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

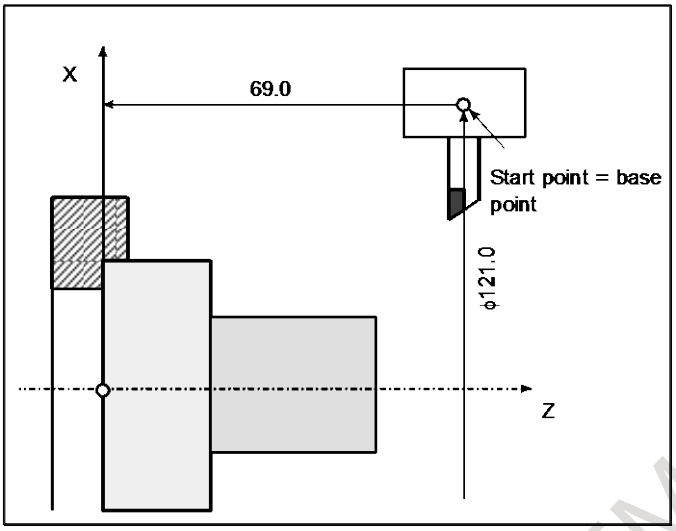


Fig. 7.2.7 (b)

7.3 LOCAL COORDINATE SYSTEM

When a program is created in a workpiece coordinate system, a child workpiece coordinate system can be set for easier programming. Such a child coordinate system is referred to as a local coordinate system.

Format

G52 IP_; Setting the local coordinate system

:

G52 IP 0 ; Canceling of the local coordinate system

IP_ : Origin of the local coordinate system

Explanation

By specifying G52 IP_;, a local coordinate system can be set in all the workpiece coordinate systems (G54 to G59). The origin of each local coordinate system is set at the position specified by IP_ in the workpiece coordinate system.

Once a local coordinate system is established, the coordinates in the local coordinate system are used in an axis shift command. The local coordinate system can be changed by specifying the G52 command with the origin of a new local coordinate system in the workpiece coordinate system.

To cancel the local coordinate system or specify the coordinate value in the workpiece coordinate system, match the origin of the local coordinate system with that of the workpiece coordinate system.

FRIEDMACHINE SYSTEMS INC
Rob Zarensky

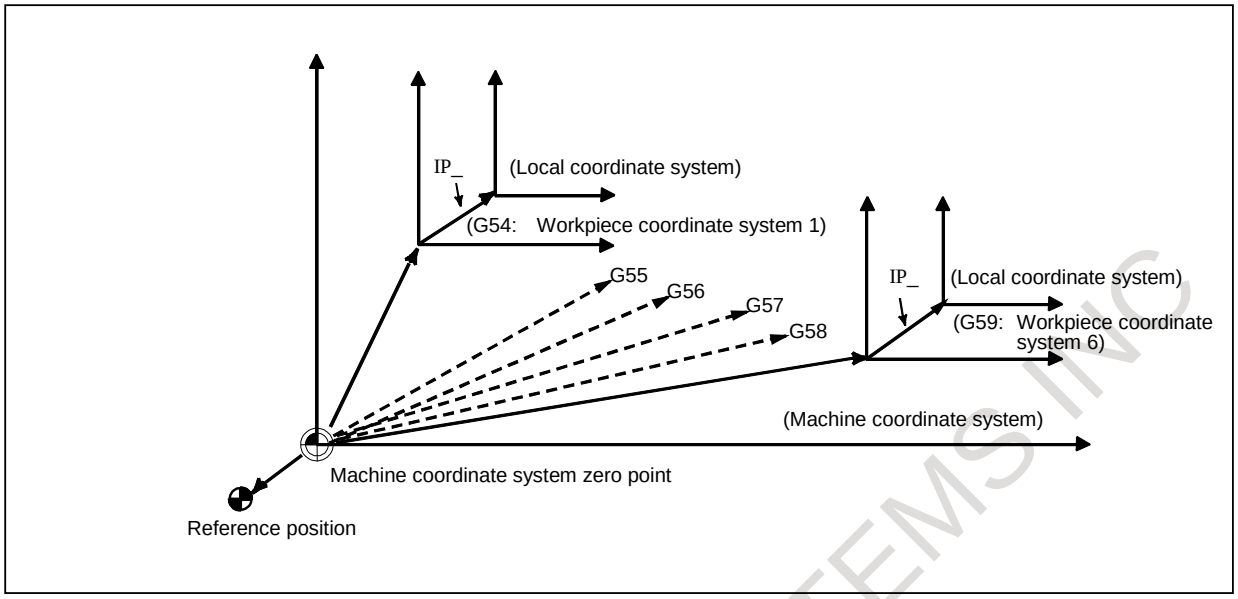


Fig. 7.3 Setting the local coordinate system

⚠ CAUTION

1 When bit 2 (ZCL) of parameter No.1201 is set to 1 and an axis returns to the reference position by the manual reference position return function, the origin of the local coordinate system of the axis matches that of the workpiece coordinate system. The same is true when the following command is issued:
   G52α0;
   α: Axis which returns to the reference position
2 The local coordinate system setting does not change the workpiece and machine coordinate systems.
3 Whether the local coordinate system is canceled at reset depends on the parameter setting. The local coordinate system is canceled when either bit 6 (CLR) of parameter No.3402 or bit 3 (RLC) of parameter No.1202 is set to 1. In 3-dimensional coordinate conversion mode, however, the local coordinate system is not canceled when bit 2 (D3R) of parameter No. 5400 is set to 1.
4 When a workpiece coordinate system is set with the G92 (G50 for G code system A in the T series) command, the local coordinate system is canceled. However, the local coordinate system of an axis for which no coordinate system is specified in a G92 (G50 for G code system A in the T series) block remains unchanged.
5 G52 cancels the offset temporarily in tool radius · tool nose radius compensation.
6 Command a move command immediately after the G52 block in the absolute mode.

FRYER MACHINE SYSTEMS INC Rob Zanensky

7.4 PLANE SELECTION

Select the planes for circular interpolation, cutter compensation, and drilling by G-code.
The following table lists G-codes and the planes selected by them.

Explanation

Table 7.4 Plane selected by G code

G code	Selected plane	Xp	Yp	Zp
G17	Xp Yp plane	X-axis or an axis parallel to it	Y-axis or an axis parallel to it	Z-axis or an axis parallel to it
G18	Zp Xp plane			
G19	Yp Zp plane			

Xp, Yp, Zp are determined by the axis address appeared in the block in which G17, G18 or G19 is commanded.
When an axis address is omitted in G17, G18 or G19 block, it is assumed that the addresses of basic three axes are omitted.
Parameter No. 1022 is used to specify that an optional axis be parallel to the each axis of the X-, Y-, and Z-axes as the basic three axes.
The plane is unchanged in the block in which G17, G18 or G19 is not commanded.
The movement instruction is irrelevant to the plane selection.

M

When the power is turned on or the CNC is reset, G17 (XY plane), G18 (ZX plane), or G19 (YZ plane) is selected by bits 1 (G18) and 2 (G19) of parameter No. 3402).

T

When the power is turned on, G18 (ZX plane) is selected.
By setting bit 0 (TPS) of parameter No. 3458 to 1, it is possible to select a modal plane by setting bits 1 (G18) and 2 (G19) of parameter No. 3402.

NOTE
U-, V-, and W-axes can be used with G-codes B and C.

Example

Plane selection when the X-axis is parallel with the U-axis.

```
G17  X_Y_ ;   XY plane,
G17  U_Y_ ;   UY plane
G18  X_Z_ ;   ZX plane
     X_Y_ ;   Plane is unchanged (ZX plane)
G17 ;         XY plane
G18 ;         ZX plane
G17 U_ ;      UY plane
G18Y_ ;       ZX plane, Y axis moves regardless without any relation to the plane.
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

8 COORDINATE VALUE AND DIMENSION

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING

There are two ways to command travels of the tool; the absolute programming, and the incremental programming. In the absolute programming, coordinate value of the end position is programmed. The incremental programming is used to program the amount of a tool movement.

M

G90 and G91 are used to programming absolute or incremental programming, respectively.

T

Absolute programming or incremental programming is used depending on the programming used. See following tables.

Table 8.1 (a)

G code system	A	B or C
Command method	Address word	G90, G91

Format

M

Absolute programming **G90 IP_ ;**
Incremental programming **G91 IP_ ;**

T

- **G code system A**

Table 8.1 (b)

	Absolute programming	Incremental programming
X axis move command	X	U
Z axis move command	Z	W
Y axis move command	Y	V
C axis move command	C	H

- **G code system B or C**

Absolute programming **G90 IP_ ;**
Incremental programming **G91 IP_ ;**

FREEDMACHINE SYSTEMS INC
Rob Zahensky

Example

M

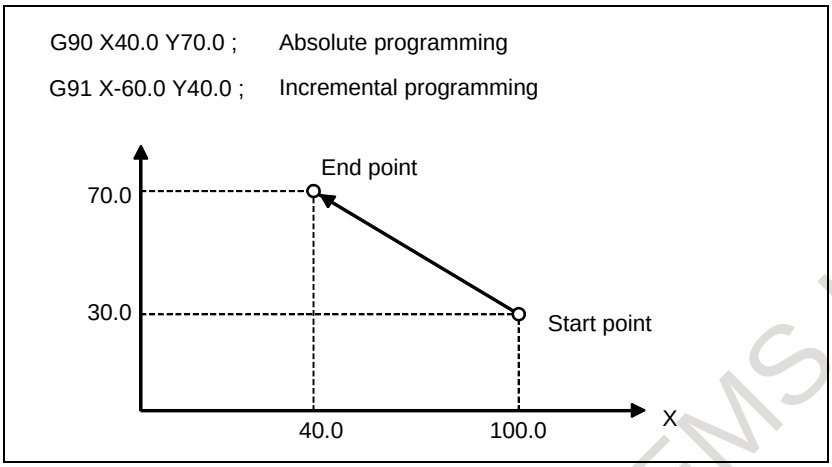


Fig. 8.1 (a)

T

Tool movement from point P to point Q (diameter programming is used for the X-axis)

Table 8.1 (c)

	G code system A	G code system B or C
Absolute programming	X400.0 Z50.0 ;	G90 X400.0 Z50.0 ;
Incremental programming	U200.0 W-400.0 ;	G91 X200.0 Z-400.0 ;

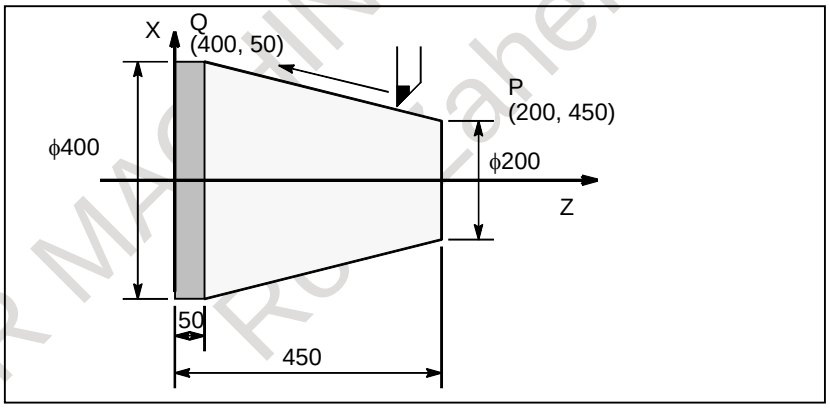


Fig. 8.1 (b)

NOTE

1 Absolute programming and incremental programming can be used together in a block.
In the above example, the following command can be specified : X400.0 W-400.0 ; (in the G code system A)

2 When both X and U or Z and W are used together in a block, the one specified later is effective.

3 Incremental programming cannot be used when names of the axes are A and B during G code system A is selected.

FRYER MACHINE SYSTEMS INC
Roman Zahensky

8.2 INCH/METRIC CONVERSION (G20, G21)

Either inch or metric input (least input increment) can be selected by G code.

Format

G20 ;	Inch input
G21 ;	Metric input

This G code must be specified in an independent block before setting the coordinate system at the beginning of the program. After the G code for inch/metric conversion is specified, the unit of input data is switched to the least inch or metric input increment of increment system (see Section, “INCREMENT SYSTEM”). The unit of data input for degrees remains unchanged. The unit systems for the following values are changed after inch/metric conversion:

- Feedrate commanded by F code
- Positional command
- Workpiece origin offset value
- Tool compensation value
- Unit of scale for manual pulse generator
- Movement distance in incremental feed
- Some parameters

When the power is turned on, the G code is the same as that held before the power was turned off.

⚠ WARNING
G20 and G21 must not be switched during a program.

NOTE

1 When the least input increment and the least command increment systems are different, the maximum error is half of the least command increment. This error is not accumulated.
(Example)
- Increment system : IS-B
- Least input increment (for input) : In case of "0.0001" inch
- Least command increment (for output) : "0.00254" -> "0.003" (Difference = 0.00046 : About 1/2 of least command increment)

Input "0.0001 inch" -> the moving increment of the machine "0.00254 mm".
However, when the increment system is IS-B, the value of the movement of the machine becomes "0.003 mm".
Therefore, the error margin of "0.00046 mm" (About 1/2 of least command increment) is generated as a difference.

2 The inch and metric input can also be switched using settings (see Subsection, “Displaying and Entering Setting Data”).

3 If a function selected using bit 2 (IRFx) of parameter No. 14000 or bit 0 (NIM) of parameter No. 11222 is not used, be sure to perform inch/metric conversion at the machine coordinate system origin.

4 G code of inch/metric conversion must be specified in a single block in principle. However, G17, G18, G19, G90, and G91 can be exceptionally specified in the same block. If especially “move command”, “tool offset” and “command that changes coordinate system” are specified in the same block, the machine may behave unexpectedly.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Performing inch/metric conversion in the reference position (parameter No. 1240 is not 0)

Conventionally, inch/metric conversion must be performed at the machine coordinate system origin. However, setting bit 2 (IRFx) of parameter No. 14000 to 1 enables inch/metric conversion to be performed in the reference position (parameter No. 1240).

If an attempt is made to perform inch/metric conversion when an axis with this function enabled is not in the reference position, alarm PS5362, “CONVERT INCH/MM AT REF-POS” is issued to cancel the attempt.
Before trying to perform inch/metric conversion, be sure to set the axis of interest to the reference position, using the G28 command, for example.

In addition, if the workpiece coordinate system has been shifted from the machine coordinate system, using the following commands or operations, bit 1 (CIM) of parameter No. 11222 can be used to select whether to issue alarm PS1298, “ILLEGAL INCH/METRIC CONVERSION” or to clear the offset.

- Manual intervention performed with the manual absolute signal *ABSM being off
- Move command issued with the machine locked
- Move command issued using a handle interrupt
- Mirror image-based operation
- Workpiece coordinate system shift caused by local coordinate system setting (G52) or workpiece coordinate system setting (G92)

Switching conditions

Performing inch/metric conversion in any position other than the reference position requires satisfying all of the following conditions. Failing to satisfy any of the conditions results in alarm PS1298, “ILLEGAL INCH/METRIC CONVERSION” being issued. For electronic gear box 2 pair synchronization or electronic gear box synchronization, alarm PS1595, “ILL-COMMAND IN EGB MODE” is issued.

- Positioning or linear interpolation
- Polar coordinate command cancel
- Spindle Speed fluctuation detection off
- Tool radius · tool nose radius compensation cancel
- Normal direction control cancel
- Tool length offset cancel
- Programmable mirror image cancel
- Macro modal call A/B cancel
- Coordinate system rotation or 3-dimensional coordinate system conversion mode off
- Mirror image for double turret or balanced cutting mode cancel (T series only)
- Canned cycle cancel
- Electronic gear box 2 pair synchronization cancel
- Electronic gear box synchronization cancel
- Constant surface speed control cancel

M

- Scaling cancel

T

- Polar coordinate interpolation cancel mode
- Polygon turning cancel

Restrictions

When performing the following operations, make sure that the axis of interest is in the machine coordinate system origin.

- Inch/metric conversion based on bit 2 (INI) of setting parameter No. 0000
- Inch/metric conversion based on programmable parameter input (G10)
- Inch/metric conversion based on custom macro variable No. 3005

TRYER MACHINE SYSTEMS INC
Rob Zahorsky

Performing inch/metric conversion in positions other than the reference position

Setting bit 0 (NIM) of parameter No. 11222 enables inch/metric conversion to be performed even in positions other than the reference position.

In addition, if the workpiece coordinate system has been shifted from the machine coordinate system, using the following commands or operations, bit 1 (CIM) of parameter No. 11222 can be used to select whether to issue alarm PS1298 or to clear the offset.

- Manual intervention performed with the manual absolute signal *ABSM being off
- Move command issued with the machine locked
- Move command issued using a handle interrupt
- Mirror image-based operation
- Workpiece coordinate system shift caused by local coordinate system setting (G52) or workpiece coordinate system setting (G92)

If an axis is under any of the following controls, however, no automatic coordinate system conversion based on this function can be carried out for the axis.

- PMC axis control
- Axis synchronous control (for slave axes when the master axis is a PMC axis)
- Flexible synchronization control (for slave axes when the master axis is a PMC axis)
- Spindle control with servo motor
- Servo/spindle synchronous control

Switching conditions

Performing inch/metric conversion in any position other than the reference position requires satisfying all of the following conditions. Failing to satisfy any of the conditions results in alarm PS1298 being issued. For electronic gear box 2 pair synchronization or electronic gear box synchronization, alarm PS1595 is issued.

- Positioning or linear interpolation
- Polar coordinate command cancel
- Spindle Speed fluctuation detection off
- Tool radius · tool nose radius compensation cancel
- Normal direction control cancel
- Tool length offset cancel
- Programmable mirror image cancel
- Macro modal call A/B cancel
- Coordinate system rotation or 3-dimensional coordinate system conversion mode off
- Mirror image for double turret or balanced cutting mode cancel (T series only)
- Canned cycle cancel
- Electronic gear box 2 pair synchronization cancel
- Electronic gear box synchronization cancel
- Constant surface speed control cancel

M

- Scaling cancel

T

- Polar coordinate interpolation cancel mode
- Polygon turning cancel

Restrictions

When performing the following operations, make sure that the axis of interest is in the machine coordinate system origin.

- Inch/metric conversion based on programmable parameter input (G10)
- Inch/metric conversion based on custom macro variable No. 3005

FRYER MACHINE SYSTEMS INC Rob Zaharsky

Limited to the reference position

Setting bit 0 (NIM) of parameter No. 11222 and bit 3 (IMRx) of parameter No. 14000 enables inch/metric conversion to limit in the reference position (parameter No. 1240). This function is used when the parameter No. 1240≠0.0 and the inch/metric conversion based on bit 2 (INI) of setting parameter No. 0000 is limited to the reference position.

An alarm (PS5362) will be generated if inch/metric conversion is performed without the reference position return. Also, inch/metric conversion based on programmable parameter input (G10) or custom macro variable No. 3005 can not be commanded. An alarm (PS1298) will be generated. The operation of bit 1 (CIM) of parameter No. 11222 and the switching condition etc., it is the same as the specification that bit 0 (NIM) of parameter No. 1122 is set to 1.

8.3 DECIMAL POINT PROGRAMMING

Numerical values can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:

M

X, Y, Z, U, V, W, A, B, C, I, J, K, Q, R, F

T

X, Y, Z, U, V, W, A, B, C, I, J, K, R, F,H(*1)
(*1) only G code system A.

Explanation

There are two types of decimal point notation: calculator-type notation and standard notation.
When calculator-type decimal notation is used, a value without decimal point is considered to be specified in millimeters inch, or deg. When standard decimal notation is used, such a value is considered to be specified in least input increments. Select either calculator-type or standard decimal notation by using the bit 0 (DPI) of parameter No.3401.Values can be specified both with and without decimal point in a single program.

Example

Table 8.3 (a)

Program command	Pocket calculator type decimal point programming	Standard type decimal point programming
X1000 Command value without decimal point	1000mm Unit :mm	1mm Unit : Least input increment (0.001mm)
X1000.0 Command value with decimal point	1000mm Unit :mm	1000mm Unit :mm

⚠ **CAUTION**
When specifying a dimension word for a command G code in a block, be sure to place the dimension word after the command G code.

FLYER MACHINE SYSTEMS INC
Rob Zarnensky

NOTE

1 A specified value less than the minimum unit is treated as described below.

Example 1)

When a value is specified directly at an address (in the case of 1/1000mm IS-B)

X1.2345 ; Treated as X1.235

X-1.2345 ; Treated as X-1.234

Example 2)

When a value is assigned to a macro variable (in the case of 1/1000mm IS-B)

#100=1.2345;

X#100 ; Treated as X1.235

#100=-1.2345;

X#100 ; Treated as X-1.234

2 When more than the digits indicated in table below are specified, an alarm occurs.

Table8.3 (b) Maximum command digit number

increment system	When address I, J, K or R is used to specify the radius for circular interpolation	The address other than the radius of circular interpolation
IS-A	11 digits	8 digits
The increment systems other than IS-A	12 digits	9 digits

If a value is entered with a decimal point, the number of digits is also checked after the value is converted to an integer according to the least input increment.

Examples:

X1.23456789;

When increment system is IS-A, alarm PS0003, “TOO MANY DIGIT” occurs because more than eight digits are specified.

X1234567.8;

If the least input increment is 0.01 mm, the value is converted to integer 123456780. When increment system is IS-A, because the integer has more than eight digits, an alarm occurs.

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

8.4 DIAMETER AND RADIUS PROGRAMMING

Since the workpiece cross section is usually circular in CNC lathe control programming, its dimensions can be specified in two ways :

Diameter and Radius

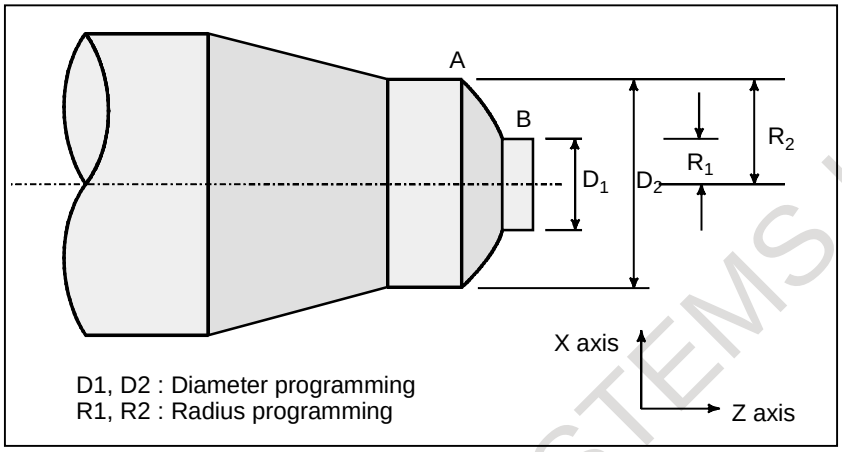


Fig. 8.4

When the diameter is specified, it is called diameter programming and when the radius is specified, it is called radius programming.

Explanation

- Notes on diameter programming/radius programming for each command

Radius programming or diameter programming can be specified by bit 3 (DIA) of parameter No.1006. When using diameter programming, note the conditions listed in the Table 8.4 (a).

Table 8.4 (a) Notes on specifying diameter value

Item	Notes
X axis command	Specified with a diameter value
Incremental command	Specified with a diameter value In the above figure, specifies D2 minus D1 for tool path B to A.
Coordinate system setting (G50)	Specifies a coordinate value with a diameter value
Component of tool offset value	Bit 1 of parameter No.5004 determines either diameter or radius value
Parameters in canned cycle, such as cutting depth along X axis. (R)	Specifies a radius value
Radius designation in circular interpolation (R, I, K, and etc.)	Specifies a radius value
Feedrate along axis	Specifies change of radius/rev. or change of radius/min.
Display of axis position	Displayed as diameter value

FRYER MACHINE SYSTEMS INC Rob Zanensky

9 SPINDLE SPEED FUNCTION (S FUNCTION)

9.1 SPECIFYING THE SPINDLE SPEED WITH A CODE

When a value is specified after address S, the code signal and strobe signal are sent to the machine to control the spindle rotation speed.
A block can contain only one S code. Refer to the appropriate manual provided by the machine tool builder for details such as the number of digits in an S code or the execution order when a move command and an S code command are in the same block.

9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5-DIGIT COMMAND)

The spindle speed can be specified directly by address S followed by a max. five-digit value (min^{-1}). The unit for specifying the spindle speed may vary depending on the machine tool builder. Refer to the appropriate manual provided by the machine tool builder for details.

9.3 CONSTANT SURFACE SPEED CONTROL (G96, G97)

Specify the surface speed (relative speed between the tool and workpiece) following S. The spindle is rotated so that the surface speed is constant regardless of the position of the tool.

Format

- **Constant surface speed control command**

G96S<u>xxxxx</u> ;

↑ Surface speed (m/min or feet/min)
This surface speed unit may change according to machine tool builder's specification.

- **Constant surface speed control cancel command**

G97S<u>xxxxx</u> ;

↑ Spindle speed (min^{-1})
This surface speed unit may change according to machine tool builder's specification.

- **Constant surface speed controlled axis command**

G96Pα ;

P0 : Axis set in the parameter No. 3770
P1 : 1st axis, P2 : 2nd axis, P3 : 3rd axis, P4 : 4th axis
P5 : 5th axis, P6 : 6th axis, P7 : 7th axis, P8 : 8th axis

NOTE
If multi-spindle control (spindle selecting based on address P or spindle name expansion) is enabled, axis specification based on address P is disabled. Use parameter No. 3770 for axis specification.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Clamp of maximum spindle speed

G92 S_ ;

The maximum spindle speed (min^{-1}) follows S.

T

G50 S_ ;

The maximum spindle speed (min^{-1}) follows S.

NOTE

G50 can be used with G code system A.

Explanation

- Constant surface speed control command (G96)

G96 (constant surface speed control command) is a modal G code. After a G96 command is specified, the program enters the constant surface speed control mode (G96 mode) and specified S values are assumed as a surface speed. A G96 command must specify the axis along which constant surface speed control is applied. A G97 command cancels the G96 mode. When constant surface speed control is applied, a spindle speed higher than the value specified in G92S_; or G50S_; (maximum spindle speed) is clamped at the maximum spindle speed. When the power is turned on, the maximum spindle speed is not yet set and the speed is not clamped. S (surface speed) commands in the G96 mode are assumed as S = 0 (the surface speed is 0) until M03 (rotating the spindle in the positive direction) or M04 (rotating the spindle in the negative direction) appears in the program.

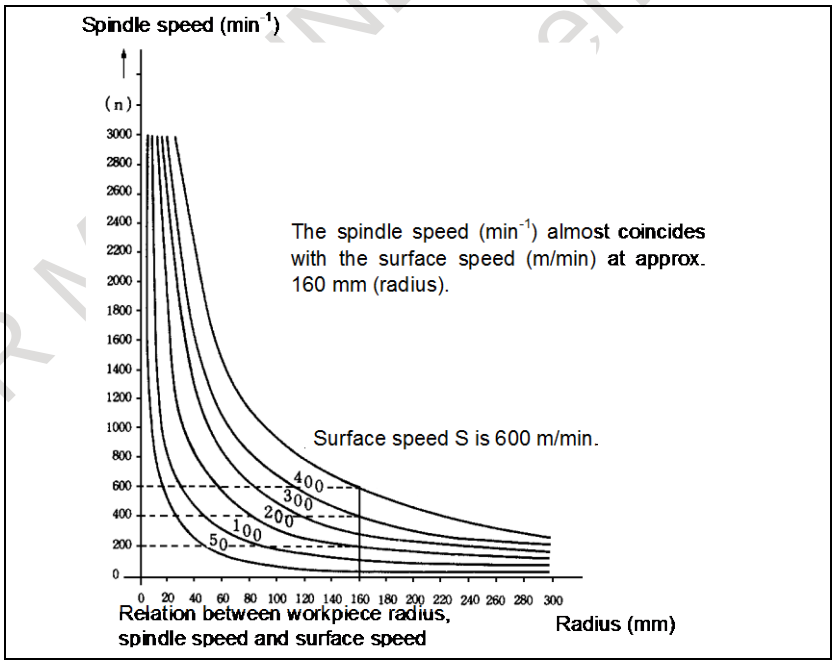


Fig. 9.3 (a) Relation between workpiece radius, spindle speed and surface speed

FRYER MACHINE SYSTEMS INC
Ensky

- Setting the workpiece coordinate system for constant surface speed control

To execute the constant surface speed control, it is necessary to set the workpiece coordinate system , and so the coordinate value at the center of the rotary axis, for example, Z axis, (axis to which the constant surface speed control applies) becomes zero.

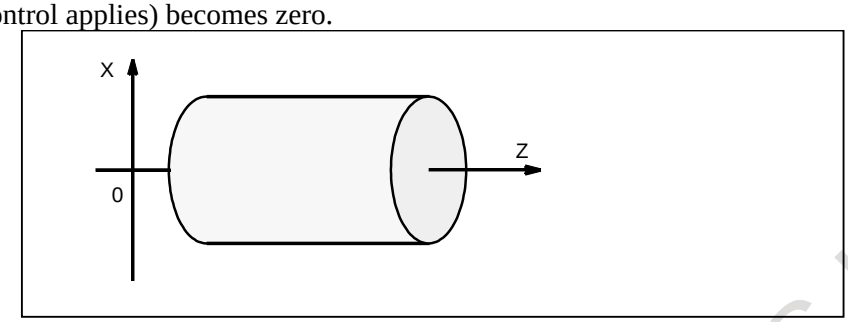
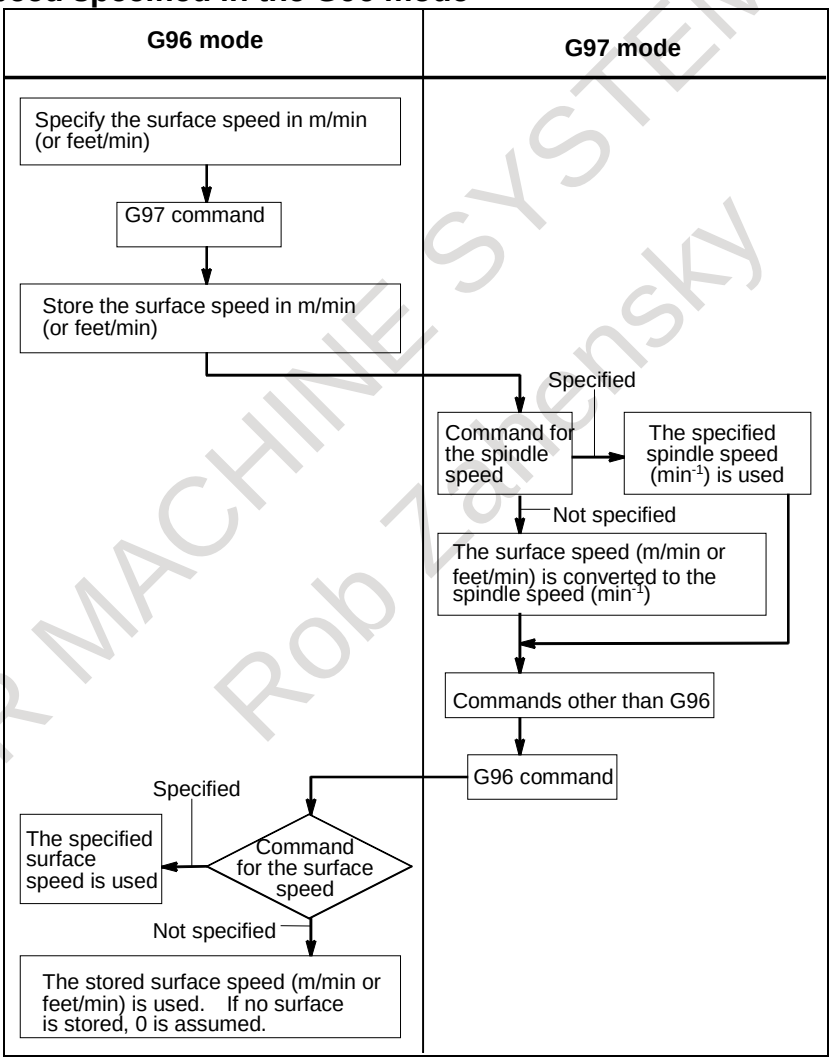


Fig. 9.3 (b) Example of the workpiece coordinate system for constant surface speed control

- Surface speed specified in the G96 mode



FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Maximum spindle speed clamp

If the spindle rotating at the maximum speed specified with the relevant parameter may cause a problem in the constant surface speed control mode, execute maximum spindle speed clamp (M series: G92S_;, T series: G50S_;) before starting constant surface speed control so that the spindle speed does not exceed the specified speed.
When maximum spindle speed clamp has not been executed after power-on, constant surface speed control may be specified due to a program error or operator error including the use of an improper procedure for executing a machining program. To avoid this problem, bit 4 (CSA) of parameter No. 3712 can be set to 1 to issue alarm PS5557, “NO MAX SP SPEED CLAMP COMMAND” in the above case to prevent the spindle from rotating at an unexpected speed.
When bit 0 (CLM) of parameter No.3785 is set to 0, and spindle selection by extended spindle name is enabled, if extended spindle name is commanded at clamp command of maximum spindle speed G92 (G50 in G code system A of T series), alarm PS0539 “MAX SP SPEED CLAMP COMMAND ERROR” is issued. “S” has to be commanded at clamp command of maximum spindle speed.

Example
Suppose that extended spindle name is set as follows.

Table 9.3 (a) Setting of extended spindle name

Spindle number	Extended spindle name
First spindle	S
Second spindle	SB

If “G92 SB30 ;” is commanded, alarm PS0539 “MAX SP SPEED CLAMP COMMAND ERROR” is issued. Clamp of maximum spindle speed has to be command “G92 S30 ;”. If “S” is set as extended spindle name, alarm PS0539 is not issued.

- Selection of axis as the calculation reference in constant surface speed control

When bit 0 (CLM) of parameter No.3785 is set to 0, and spindle selection by address P or extended spindle name of multi-spindle control is enabled, selection of axis as the calculation reference in constant surface speed control “G96 P_;” is commanded, alarm PS0190 “ILLEGAL AXIS SELECTED (G96)” is issued. Please command programmable parameter input G10 in changing axis as the calculation reference in constant surface speed control.

Example
Suppose that spindle selection by address P is enabled.
- If “G96 P_ ;” is commanded, alarm PS0190 is issued.
- If “G96 S_ P_ ;” is commanded, spindle selected by address P is controlled by constant surface speed control.

Example
Suppose that extended spindle name is set as Table 9.3 (a).
- If “G96 P_ ;” is commanded, alarm PS0190 is issued.
- If “G96 S_ P_ ;” , or “G96 SB_ P_ ” is commanded, alarm PS0190 is issued.

When extended spindle name is enabled, address P cannot be commanded at G96 block.

- G96 command without surface speed

When bit 1 (G96) of parameter No.3785 is set to 1, and spindle selection by address P or extended spindle name is enabled, if G96 command without surface speed is commanded, alarm PS5355 “S CODE IS NOT COMMANDED AT G96” is issued.

FRYER MACHINE SYSTEMS INC.
Rob Zarensky

NOTE

When spindle selection by address P or extended spindle name is enabled, if G96 is commanded without surface speed, surface speed which is commanded at previous G96 command is applied. Axis as the calculation reference in constant surface speed control which is commanded previously regardless of G96/G97 mode is applied. In order to prevent rotation of unexpected spindle by G96 command without surface speed, please command surface speed at G96 command.

- Spindle speed calculation during the 3-dimensional coordinate system conversion / the tilted working plane indexing

In the constant surface speed control in the 3-dimensional coordinate system conversion / the tilted working plane indexing mode, the coordinate system which is the standard of speed calculation can be changed by the parameter 3CS(No.11221#6).

Table 9.3 (b) The standard coordinate system of spindle speed calculation

Parameter 3CS	Coordinate system of standard
0	Workpiece coordinate
1	Program coordinate (3-dimensional coordinate system conversion) / Feature coordinate (Tilted working plane indexing)

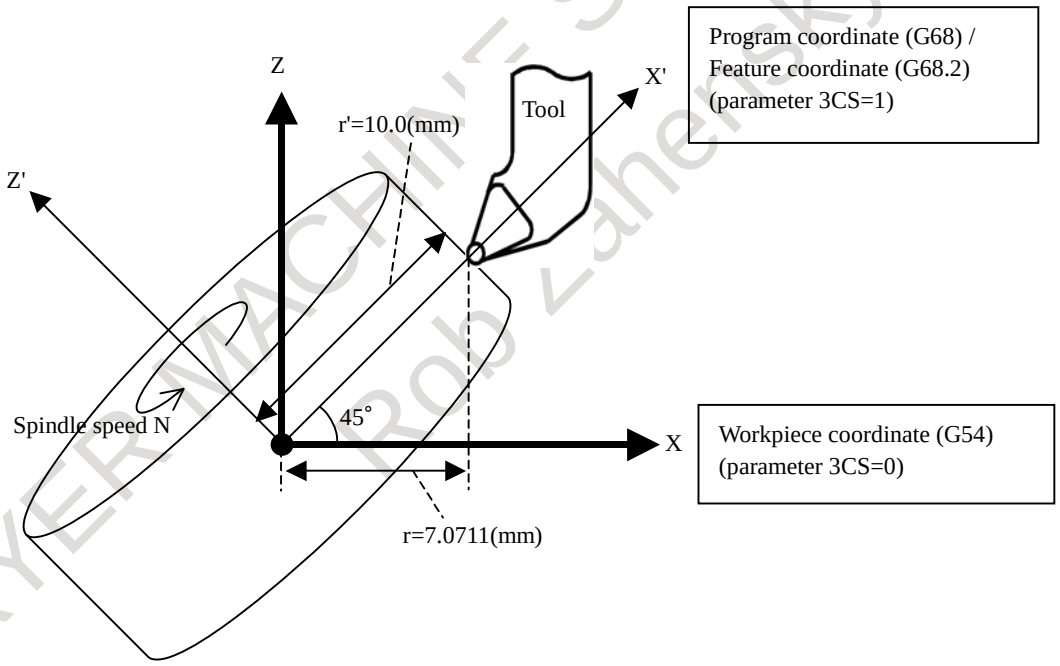


Fig.9.3 (c) Spindle speed calculation during the 3-dimensional coordinate system conversion / the tilted working plane indexing

Example)
When the followings are set by the programming.
- the surface speed S=10(m/min)
- the axis as the calculation reference is X

```
N1 G68 X0 Y0 Z0 I0 J1 K0 R-45.0
N2 G01 G90 X0 Y0 Z0 F100.0
N3 G96 S10 P1
N4 X10.0 Y0 Z0
```


FRYER MACHINE SYSTEMS INC Rob Zaherisky

N5 G97
N6 G69

Table 9.3 (c) The spindle speed in N4 (calculation by relation of N=S/2π r)

Parameter 3CS	Radius value r (mm) In the axis (X) as the calculation reference	Spindle speed N(mm^{-1})
0	7.0711	225
1	10.0	159

Limitation

- Constant surface speed control for threading

The constant surface speed control is also effective during threading. Accordingly, it is recommended that the constant surface speed control be invalidated with G97 command before starting the scroll threading and taper threading, because the response problem in the servo system may not be considered when the spindle speed changes.

- Constant surface speed control for rapid traverse (G00)

In a rapid traverse block specified by G00, the constant surface speed control is not made by calculating the surface speed to a transient change of the tool position, but is made by calculating the surface speed based on the position at the end point of the rapid traverse block, on the condition that cutting is not executed at rapid traverse.

- Rigid tapping

If rigid tapping is commanded during constant surface speed control, alarm (PS0200), ”ILLEGAL S CODE COMMAND” is issued. Command rigid tapping after canceling constant surface speed control.

Notes

- Clamp at maximum spindle speed in the tool length compensation mode

Clamp at maximum spindle speed cannot be used together with a block where tool length compensation vectors vary. For example, it cannot be used together with the following blocks.

Example

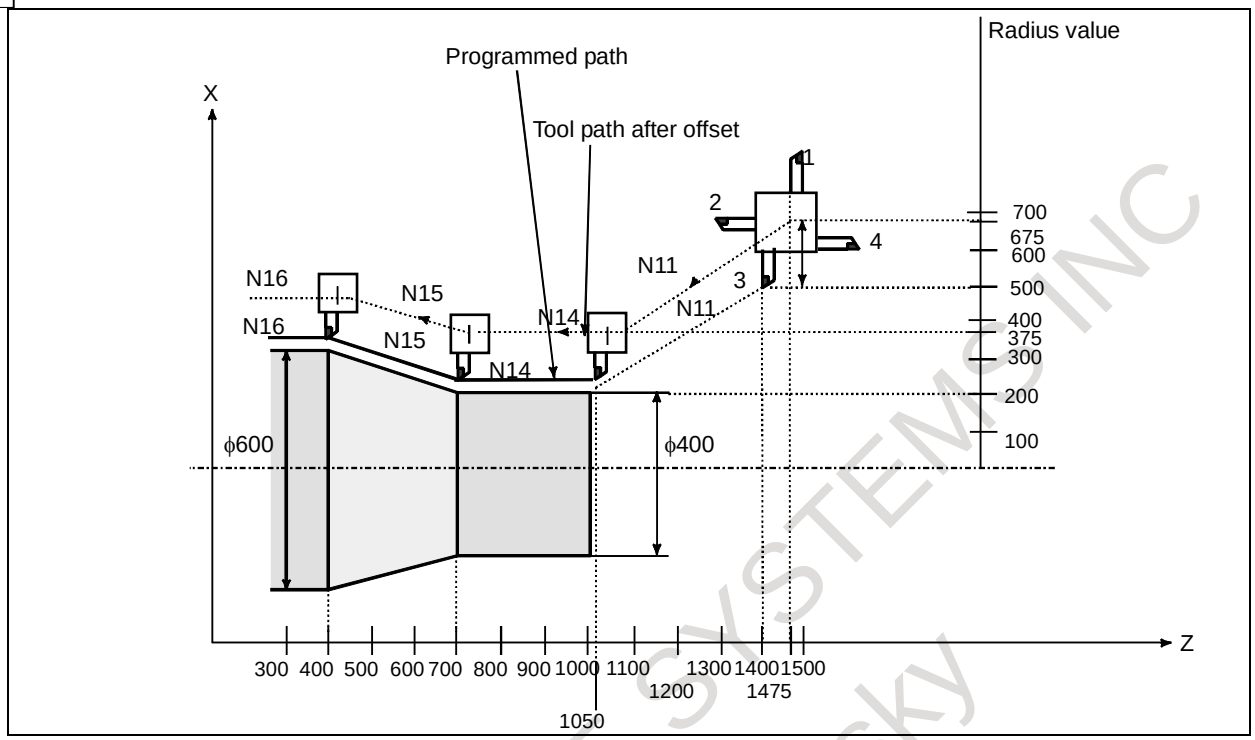
1 Block in which G43/G44 is issued
2 Block which is in the G43 or G44 mode and in which an H code is issued
3 Block which is in the G43 or G44 mode and in which G49 is issued
4 Block in which, in the G43 or G44 mode, compensation vectors are canceled using a G code such as G28 or G53 and then resumed again

The current tool length offset can not be changed by following method:
Stop operation just before this G code, and change the correct offset by MDI or other method.

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

Example

T



N8 G00 X1000.0Z1400.0 ;
N9 T33;
N11 X400.0Z1050.0;
N12 G50S3000 ; (Designation of max. spindle speed)
N13 G96S200 ; (Surface speed 200 m/min)
N14 G01 Z 700.0F1000 ;
N15 X600.0Z 400.0;
N16 Z_ ;

The CNC calculates the spindle speed which is proportional to the specified surface speed at the position of the programmed coordinate value on the X axis. This is not the value calculated according to the X axis coordinate after offset when offset is valid. At the end point N15 in the example above, the speed at 600 dia. (Which is not the turret center but the tool nose) is 200 m/min. If X axis coordinate value is negative, the CNC uses the absolute value.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

9.4 SPINDLE POSITIONING FUNCTION

Overview

In turning, the spindle connected to the spindle motor is rotated at a certain speed to rotate the workpiece mounted on the spindle. This spindle control status is referred to as spindle rotation mode.
The spindle positioning function turns the spindle connected to the spindle motor by a certain angle to position the workpiece mounted on the spindle at a certain angle. This spindle control status is referred to as spindle positioning mode.

The spindle positioning function involves the following three operations:

1. Canceling the spindle rotation mode and entering the spindle positioning mode
   Place the spindle in the spindle positioning mode and establish a reference position by specifying a given M code (set with a parameter). (Spindle orientation)
2. Positioning the spindle in the spindle positioning mode
   The spindle is positioned in either of the two methods:
   1) Positioning with an arbitrary angle by an axis address
   2) Positioning with a semi-fixed angle by a given M code (set with a parameter)
3. Canceling the spindle positioning mode, and entering the spindle rotation mode
   Place the spindle in the spindle rotation mode by specifying a given M code (set with a parameter).

The least command increment, least input increment, and maximum value for the spindle positioning axis are as follows:

- Least command increment
  $\frac{360}{4096} \fallingdotseq 0.088$ deg (when the gear ratio of the spindle to the position coder
- Least input increment
  0.001 deg (IS-B)
- Maximum value
  ±999999.999 deg

9.4.1 Spindle Orientation

When spindle positioning is first performed after the spindle motor is used for normal spindle operation, or when spindle positioning is interrupted, the spindle orientation is required.
Orientation permits the spindle to stop at a predetermined position.
Orientation is directed by the M code set in parameter No. 4960. The direction of orientation can be set with a parameter.
For the serial spindle, it is set in bit 4 (RETSV) of parameter No. 4000. With the grid shift function, the orientation position can be shifted in a range of 0 to 360 deg with parameter No. 4073 for a serial spindle.

- Feedrate during spindle orientation

An orientation feedrate for a serial spindle is determined by a spindle parameter setting.
In orientation, the serial spindle stops at the orientation position after several rotations of the spindle motor.

- Omission of orientation

By using bit 2 (ISZ) of parameter No. 4950, orientation upon switching to spindle positioning mode can be omitted if it is unnecessary (for example, when no start position is specified and incremental positioning from the current position is only required). More specifically, when an M code for switching to spindle positioning mode is specified, the spindle control mode is simply switched to spindle positioning mode and then the processing is completed without orientation.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Program reference position

The position at which orientation is completed is assumed to be a program reference position. However, the program reference position can be changed through coordinate system setting (G92 or G50) or automatic coordinate system setting (bit 0 (ZPR) of parameter No. 1201).
When a setting is made to omit orientation, a program reference position is not established, and operation by an absolute command is unpredictable during spindle positioning with an axis address.

9.4.2 Spindle Positioning

The spindle can be positioned with a semi-fixed angle or arbitrary angle.

- Positioning with a semi-fixed angle

Use an M code to specify a positioning angle. The specifiable M code value may be one of the six values from Mα to M (α+5). Value α must be set in parameter No. 4962 beforehand. The positioning angles corresponding to Mα to M (α+5) are listed below. Value β must be set in parameter No. 4963 beforehand.

M-code (Ex.) β = α + 5	Positioning angle	(Ex.) β = 30°
Mα	β	30°
M (α + 1)	2β	60°
M (α + 2)	3β	90°
M (α + 3)	4β	120°
M (α + 4)	5β	150°
M (α + 5)	6β	180°

When the number of M codes to be used, value γ, is specified in parameter No. 4964, a specifiable M code value may be in a range of values from Mα to M (α + (γ - 1)), up to 255 values from Mα to M (α + (255 - 1)).

M code (Ex.) γ = 11	Positioning angle	(Ex.) β = 30°
Mα	β	30°
M (α + 1)	2β	60°
M (α + 2)	3β	90°
M (α + 3)	4β	120°
...	...	...
M (α + 11 - 1)	11β	330°

The direction of rotation can be specified in bit 1 (IDM) of parameter No.4950.

- Positioning with an arbitrary angle

Specify the position with an arbitrary angle using the axis address followed by a signed numeric value or numeric values. The axis address must be specified in the G00 mode.
(The explanation below is given assuming that a C axis address is set.)

(Example) C-45000
C180.000

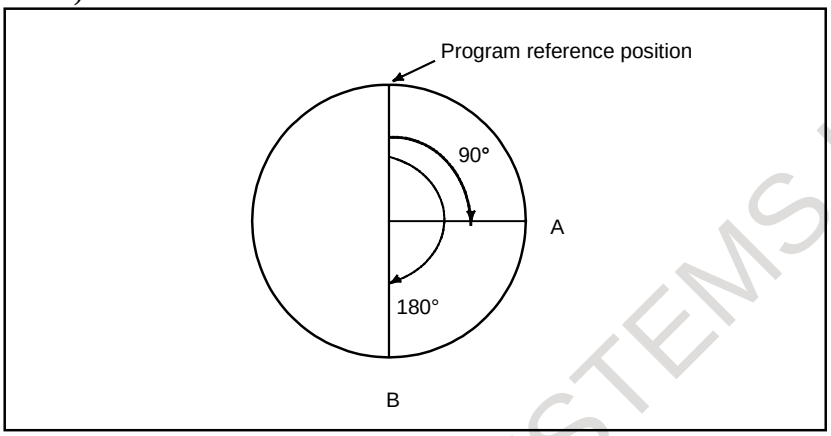
A numeric with the decimal point can be entered. The value must be specified in degrees.

(Example) C36.0=C36 deg

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Absolute commands and incremental commands

Incremental commands are always used for positioning with a semi-fixed angle (using M codes).
The direction of rotation can be specified with bit 1 (IDM) of parameter No. 4950.
Absolute and incremental commands can be used for positioning with an arbitrary angle.
With absolute commands for positioning with an arbitrary angle, when the rotation axis rollover function is used (bit 0 (ROA) of parameter No. 1008 is 1), shortcut control is also enabled (bit 1 (RAB) of parameter No. 1008 is 0).



Command format		G-code system A in lathe system		G-code system B or C in lathe system, and machining center system	
		Address used	Command A-B in the above figure	Address used and G code	Command A-B in the above figure
Absolute command	Specify the end point with a distance from the program reference position.	C	C180.0 ;	G90, C	G90 C180 .;
Incremental command	Specify a distance from the start point to the end point.	H	H90.0 ;	G91, C	G91 C90 .;

- Feedrate during positioning

The feedrate during positioning equals the rapid traverse speed specified in parameter No. 1420.
For the specified speed, an override of 100%, 50%, 25%, and F0 (parameter No. 1421) can be applied.

9.4.3 Canceling Spindle Positioning

When modes are to be switched from spindle positioning to normal spindle rotation, the M code set in parameter No. 4961 must be specified.

Also, the spindle positioning mode is canceled and the spindle rotation mode is set in the following cases:

<1> A reset operation (including an emergency stop) occurs when a servo alarm is issued.
<2> A reset operation (including an emergency stop) occurs when a spindle alarm is issued.
<3> An orientation operation in progress is stopped due to a reset or alarm, or for some other reason.
<4> A reset operation (including an emergency stop) occurs when bit 0 (IOR) of parameter No. 4950 is 1.

⚠ CAUTION

1 During execution of spindle positioning sequences (canceling the spindle rotation mode and entering the spindle positioning mode, positioning the spindle in the spindle positioning mode, and canceling the spindle positioning mode and entering the spindle rotation mode), the automatic operation stop signal *SP is invalid. This means that automatic operation does not stop until all the sequences are completed, even when the *SP signal becomes “0”.

FRYER MACHINE SYSTEMS INC
Rob Zahenski

⚠ CAUTION

2 Dry run and machine lock cannot be performed during spindle positioning.
3 Auxiliary function lock is disabled for M codes for the spindle positioning function.
4 The serial spindle Cs-axis contour control function and the spindle positioning function cannot be used at a time. If both functions are specified, the spindle positioning function has priority.
5 The spindle positioning axis is handled as a controlled axis. Therefore, controlled axis-related signals (such as the overtravel signal) must be set.
6 When using the rigid tapping function and the spindle positioning function concurrently, do not specify rigid tapping in spindle positioning mode or spindle positioning in rigid tapping mode.

NOTE

1 M code commands for positioning of a spindle must be specified in a single block. Other commands must not be contained in the same block. (Also, M code commands for positioning of another spindle must not be contained in the same block.)
Even when the single-block, multiple-M code command function is also used, related M codes must be specified in a single block.
2 Even when the single-block, multiple-M code command function is also used, related M codes must be specified in a single block.
3 Axis address commands for positioning of a spindle must be specified in a single block. Other commands must not be contained in the same block. However, the following commands can be contained in the same block where axis address commands are specified:
G00, G90, G91, G92 (G-code systems B and C in the T series, M series)
G00, G50 (G-code system A in the T series)
4 M code commands for spindle positioning specify M codes that are not buffered.
5 Spindle positioning cannot be performed by manual operation (in jog feed, manual handle feed, or other mode).
6 Spindle positioning cannot be performed by PMC axis control.
7 For spindle positioning, program restart and block restart operations cannot be performed. Use the MDI for these operations.
8 The stored stroke limit check is disabled for the spindle positioning axis.
9 The axis removal function is disabled for the spindle positioning axis.
10 The pitch error compensation function is disabled for the spindle positioning axis.
11 When a setting is made to omit spindle orientation, the reference position return completion signal does not become “1”.
12 In spindle orientation, all-axis interlock and axis-specific interlock are checked only when a block is started. A signal is ignored if input during the execution of the block.
13 A difference between a specified travel distance and an actual travel distance is maintained until spindle positioning mode is canceled.

FRYER MACHINE SYSTEMS INC Rob Zanensky

9.5 SPINDLE SPEED FLUCTUATION DETECTION

Overview

With this function, an overheat alarm OH0704, “OVERHEAT” is raised and the spindle speed fluctuation detection alarm signal SPAL is issued when the spindle speed deviates from the specified speed due to machine conditions.
This function is useful, for example, for preventing the seizure of the guide bushing.
G26 enables spindle speed fluctuation detection. G25 disables spindle speed fluctuation detection.

Format

- Spindle speed fluctuation detection on

G26 Pp Qq Rr Ii;

p: Time (in ms) from the issue of a new spindle rotation command (S command) to the start of checking whether the actual spindle speed is so fast that an overheat can occur.
When a specified speed is reached within the time period of P, a check is started at that time.

q: Tolerance (%) of a specified spindle speed

$$q = \left| 1 - \frac{actual\ spindle\ speed}{specified\ spndle\ speed} \right| \times 100$$

If a specified spindle speed lies within this range, it is regarded as having reached the specified value. Then, the checking of an actual spindle speed is started.

r: Spindle speed fluctuation (%) at which the actual spindle speed is so fast that an overheat can occur

$$r = \left| 1 - \frac{speed\ that\ can\ cause\ overheat}{speciofied\ spndle\ speed} \right| \times 100$$

If the fluctuation of the actual spindle speed to the specified spindle speed exceeds the spindle speed fluctuation of R, the actual spindle speed is regarded as being so fast that an overheat can occur.

i: Spindle speed fluctuation width at which the actual spindle speed (min^{-1}) is so fast that an overheat can occur
If the fluctuation (width) between the specified and actual spindle speeds exceeds the spindle speed fluctuation width of I, the actual spindle speed is regarded as being so fast that an overheat can occur.

NOTE
Specify G26 in a separate block.

G26 enables the spindle speed fluctuation detection function. The values specified for P, Q, R, and I are set in the following parameters: No. 4914, No. 4911, No. 4912, and No. 4913, respectively. Each command address corresponds to a parameter number as listed below.

Command address	Parameter number
Q	No.4911
R	No.4912
I	No.4913
P	No.4914

If the P, Q, R, or I command address is omitted, the function detects the fluctuation of the actual spindle speed according to the value set in the corresponding parameters Nos. 4914, 4911, 4912, or 4913.
The parameters Nos. 4914, 4911, 4912, and 4913 for the spindle on which the currently selected position coder is mounted are used for the setting and spindle speed fluctuation detection check.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Spindle speed fluctuation detection off

G25;

NOTE
Specify G25 in a separate block.

G25 disables the spindle speed fluctuation detection function.
When G25 is specified, the parameters Nos. 4914, 4911, 4912, and 4913 are unchanged.

Explanation

The function for detecting spindle speed fluctuation checks whether the actual speed varies for the specified speed or not. Si or Sr, whichever is greater, is taken as the allowable fluctuation speed (Sm). An alarm (OH0704) is activated when the actual spindle speed varies for the commanded speed (Sc) under the condition that the variation width exceeds the allowable variation width (Sm).

|Sc – Sa| > Sm

Sc : Specified spindle speed
Sa : Actual spindle speed
Si : The allowable constant variation width which is independent of the specified spindle speed (parameter No.4913)
Sr : The allowable variation width which is obtained by multiplying Sc (commanded spindle speed) by r (constant ratio). (r = parameter No.4912)

Bit 0 (FLR) of parameter No. 4900= 0	Bit 0 (FLR) of parameter No. 4900= 1
$Sr = Sc \times \frac{r}{100}$	$Sr = Sc \times \frac{r}{1000}$

Sm: Si or Sr, whichever is greater

- Conditions to start spindle speed fluctuation detection

If the specified spindle speed Sc changes, spindle speed fluctuation detection starts when one of the conditions below is met.
When bit 7 (FDTs) of parameter No. 4900 is set to 1, spindle speed fluctuation detection starts when condition <2> is met.

<1> The actual spindle speed falls in a range of (Sc - Sq) to (Sc + Sq)
Sc : Specified spindle speed
Sq : Tolerance within which the spindle is assumed to attain the programmed speed (parameter No.4911)

Parameter FLR= 0	Parameter FLR= 1
$Sq = Sc \times \frac{q}{100}$	$Sq = Sc \times \frac{q}{1000}$

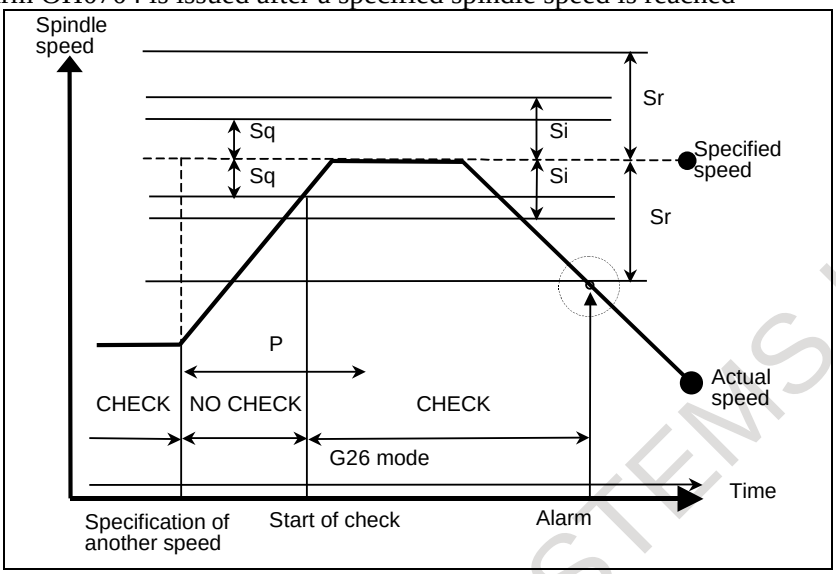
<2> When time p specified in parameter No. 4914 elapses after the specified speed Sc changes.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Examples of spindle speed fluctuation detection

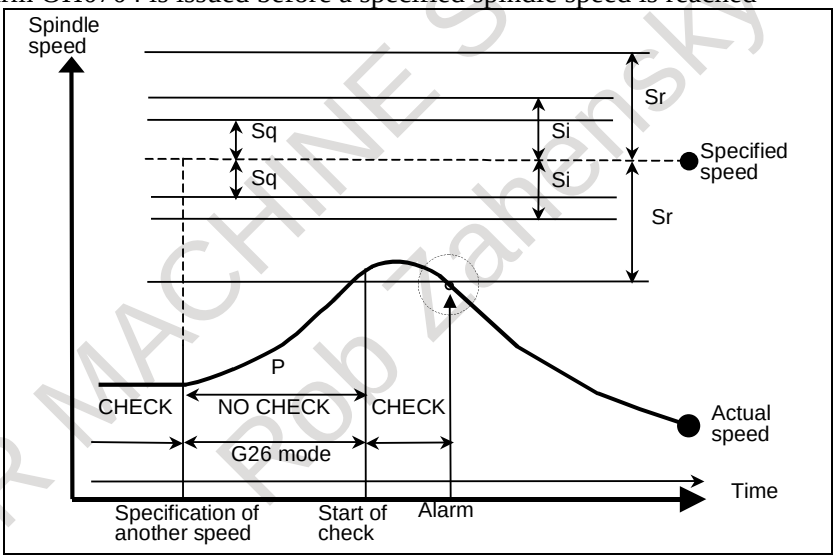
(Example 1)

When an alarm OH0704 is issued after a specified spindle speed is reached



(Example 2)

When an alarm OH0704 is issued before a specified spindle speed is reached



Specified speed : (Speed specified by address S and five-digit value) × (spindle override)
Actual speed : Speed detected with a position coder
p : Period after a change occurs in the actual spindle speed until detection starts
Parameter No.4914, address P
Sq: (Specified spindle speed) × (Detection start tolerance (q))
Parameter No.4911, address Q

Parameter FLR = 0	Parameter FLR = 1
$\frac{q}{100}$	$\frac{q}{1000}$

Sr : (Specified spindle speed) × (Allowable variation (r))
Parameter No.4912, address R

Parameter FLR = 0	Parameter FLR = 1
$\frac{r}{100}$	$\frac{r}{1000}$

FRYER MACHINE SYSTEMS INC
Rob Zaborsky

Si : Allowable variation width
Parameter No.4913, address I

If the difference between the specified speed and actual speed exceeds both Sr and Si, an alarm OH0704 is raised.

- Relationship between spindle speed control and each spindle

Function \ Spindle	Serial spindle			
	1st spindle	2nd spindle	3rd spindle	4th spindle
Spindle speed fluctuation detection	Possible	Possible(*1)	Possible(*1)	Possible(*1)

NOTE

1 A multi spindle control function is necessary.
2 The spindle speed fluctuation detection function is effective for a single spindle. The function cannot be executed for two or more spindles.
The spindle speed fluctuation detection function is effective for a spindle on which the currently selected position coder is mounted. Just a single position coder can be selected. Multiple position coders cannot be selected. For the selection of a position coder, see the section of "Multi spindle."
* Position coder selection signals (PC2SLC, PC3SLC, PC4SLC)
3 The parameters that become valid are the parameters of the spindle speed fluctuation detection function (No.4911, No.4912, No.4913, No.4914) for the spindle on which the currently selected position coder is mounted.

- Spindle for which to detect the spindle speed fluctuation

For the spindle for which to detect the spindle speed fluctuation, refer to the appropriate manual provided by the machine tool builder.

- About spindle speed fluctuation detection at power-on or cleared state

At power-on or cleared state by reset, whether spindle speed fluctuation detection function is disabled (G25 mode) or is enabled (G26 mode) depends on the setting of bit 0 (G26) of parameter No. 4902 as follows.

- When bit 0 (G26) of parameter No. 4902 set to 0, spindle speed fluctuation detection function is disabled.
- When bit 0 (G26) of parameter No. 4902 set to 1, spindle speed fluctuation detection function is enabled.

Placing the G25/G26 group in the cleared state requires the following parameter settings:

- Bit 6 (CLR) of parameter No. 3402 set to 1
- Bit 0 (C08) of parameter No. 3407 set to 1 (T series) / Bit 3 (C19) of parameter No. 3408 set to 1 (M series)

About these parameters and G25/G26 mode state at power-on or reset, refer to following.

FRYER MACHINE SYSTEMS INC
Rob Zaherlsky

G26(No.4902#0)	CLR(No.3402#6)	(T series) C08(No.3407#0) (M series) C19(No.3408#3)	At power-on	At reset in G25 mode	At reset in G26 mode
0	0	0	G25	G25	G26
		1	G25	G25	G26
	1	0	G25	G25	G25
		1	G25	G25	G26
1	0	0	G26	G25	G26
		1	G26	G25	G26
	1	0	G26	G26	G26
		1	G26	G25	G26

- Parameters of spindle speed fluctuation detection at power-on

At power-on when bit 6 (UDPs) of parameter No.4900 is set to 1, parameters Nos. 4921 to 4924 (dp, dq, dr, di) are valid and parameters Nos. 4911 to 4914 (p, q, r, i) are invalid.
Because of specified address P, Q, R, I in the machining program, value of parameters Nos. 4911 to 4914 (p, q, r, i) are changed and value of parameters Nos. 4921 to 4924 (dp, dq, dr, di) are not changed. So, parameters Nos. 4921 to 4924 (dp, dq, dr, di) are used as default parameter at power-on.

When parameters Nos. 4921 to 4924 (dp, dq, dr, di) are valid, if address P, Q, R, I are specified in the machining program even once as following,
G26 Pp Qq Rr Ii ;
the specified values are set for parameters Nos. 4911 to 4914 (p, q, r, i) and these parameters are valid as parameters of spindle speed fluctuation detection. If address P, Q, R, I are not specified in the machining program, parameters Nos. 4921 to 4924 (dp, dq, dr, di) are still valid. Examples of valid spindle speed fluctuation detection parameters are shown below.

(Example 1)

Current mode and valid parameters

G25 mode		p		q		r		i	
G26 mode	○	dp	○	dq	○	dr	○	di	○

↓ G26 Pp Qq Rr Ii ; ——— machining program

G25 mode		p	○	q	○	r	○	i	○
G26 mode	○	dp		dq		dr		di	

(Example 2)

G25 mode		p		q		r		i	
G26 mode	○	dp	○	dq	○	dr	○	di	○

↓ G26 Rr Ii ;

G25 mode		p		q		r	○	i	○
G26 mode	○	dp	○	dq	○	dr		di	

At cleared state, G25 / G26 mode changes according to parameter bit 0 (G26) of parameter No.4902, but valid parameters of spindle speed fluctuation detection don't change. And, When mode is change into G25 mode, Valid state of the parameters of spindle speed fluctuation detection is held. After that, when mode is change into G26 mode again, this valid state is applied. Examples are shown below.

FRYE MACHINE SYSTEMS INC
Bob Lahensky

(Example 3)

G25 mode		p		q		r		i	
G26 mode	○	dp	○	dq	○	dr	○	di	○

↓ G26 Rr Ii ;

G25 mode		p		q		r	○	i	○
G26 mode	○	dp	○	dq	○	dr		di	

↓ clear (bit 0 (G26) of parameter No.4902 is set to 1)

G25 mode		p		q		r	○	i	○
G26 mode	○	dp	○	dq	○	dr		di	

↓ G25 ;

G25 mode	○	p		q		r		i	
G26 mode		dp		dq		dr		di	

↓ G26 ;

G25 mode		p		q		r	○	i	○
G26 mode	○	dp	○	dq	○	dr		di	

(Example 4)

G25 mode		p		q		r	○	i	○
G26 mode	○	dp	○	dq	○	dr		di	

↓ G25 ;

G25 mode	○	p		q		r		i	
G26 mode		dp		dq		dr		di	

↓ clear (bit 0 (G26) of parameter No.4902 is set to 1)

G25 mode		p		q		r	○	i	○
G26 mode	○	dp	○	dq	○	dr		di	

NOTE

Alarm PS1305 is issued because value of specified address P, Q, R, I in the machining program is out of valid data range. At this time, about address of value out of valid data range, if type of parameters Nos. 4921 to 4924 (dp, dq, dr, di) are valid before machining program, type of parameters Nos. 4921 to 4924 (dp, dq, dr, di) are still valid.

FRYER MACHINE SYSTEMS INC Rob Zaleensky

9.6 SPINDLE CONTROL WITH SERVO MOTOR

9.6.1 Spindle Control with Servo Motor

- Command with a program

This function provides SV speed control mode in which spindle rotation commands, S commands, are effective to a rotation axis with a servo motor and position control mode in which normal positioning can be performed. To place a servo motor under speed control, you must turn SV speed control mode ON (G96.4). Once SV speed control mode is turned ON,
S commands are effective to the servo motor until it is canceled.
You cannot specify positioning commands while SV speed control mode is ON. To perform positioning, you must cancel SV speed control mode.
To cancel SV speed control mode (turn position control mode ON), you must specify a spindle indexing command, G96.1/G96.2.
In position control mode, S commands are invalid as in the normal servo axis state. Note, however, S command information is stored; when SV speed control mode is ON, rotation starts at the already specified speed.

- Command with a signal

You can also turn SV speed control mode ON/OFF with the SV speed control mode signal SRVON1 to SRVON8. When you turn SV speed control mode ON with the signal, and if you turn SV speed control mode OFF with a program command, you must specify the SV speed control mode signal or specify G96.4 to turn SV speed control mode ON again. You can check the state of SV speed control mode with the SV speed control mode in-progress signal SVREV1 to SVREV8.
When you turn SV speed control mode signal OFF while rotating, spindle indexing is executed. Then SV speed control mode is turned off. Spindle indexing is executed with R0(absolute position 0).

NOTE
To specify a M code to turn SV speed control mode ON/OFF by the SV speed control mode signal SRVON1 to SRVON8, use an independent block.

Format

G96.4 P_ ; SV speed control mode ON
M03 (M04) S_ P_ ; Rotation command
S: Spindle speed [min^{-1}] (numeric value of up to five digits)
P: Spindle selection with multi-spindle control

To specify a spindle selection command P, use parameter No. 3781 (P code for selecting a spindle in multi-spindle control).
Using G96.4 and a spindle selection command P can turn the SV speed control mode ON for each axis separately. Likewise, using a spindle indexing command (G96.1/G96.2) and a spindle selection command P can cancel the SV speed control mode, that is, turn on the position control mode ON, for each axis separately.
To specify “G96.4 P_ ;”, use an independent block.
However, it is possible to specify "G96.4 P_ S_ ;".

FRYER MACHINE SYSTEMS INC
Rob Zaherskiy

9.6.2 Spindle Indexing Function

Format

G96.1 P_ R_ ;	After spindle indexing is completed, the operation of the next block is started.
G96.2 P_ R_ ;	Before spindle indexing is completed, the operation of the next block is started.
G96.3 P_ ;	After it is confirmed that spindle indexing is completed, the operation of the next block is started.

P: Spindle selection with multi-spindle control
R: Stoppage angle [deg] (0 to (parameter No. 1260))

To specify a spindle selection command P, use parameter No. 3781 (P code for selecting a spindle in multi-spindle control).
To turn the position control mode ON without performing spindle indexing, do so after canceling the SV speed control mode by issuing a G96.1 command with no R specified when the motor is at a halt.
When the motor is rotating, issuing a G96.1 (or G96.2) command with no R specified results in the motor coming to a halt by behaving in the same manner as for R0.

NOTE
1 When using G96.2, issue G96.3 before another spindle move command, to make sure that the spindle is at a complete halt.
2 To issue G96.1, G96.2, or G96.3, use an independent block.

Spindle indexing command

- Move command

(1) Command waiting for spindle indexing to be completed
If G96.1 is issued, the next block is executed after spindle indexing is completed.
(2) Command not waiting for spindle indexing to be completed
If G96.2 is issued, the next block can be executed before spindle indexing is completed.

- Movement completion check command

G96.3 is used to check to see if spindle indexing is completed. If it has not been completed, the next block waits for spindle indexing to be completed. If it is completed, the next block is executed.

- SV speed control mode cancellation

If G96.1 is used to perform spindle indexing, the SV speed control mode is canceled when spindle indexing is completed.
If G96.2 is used to perform spindle indexing, G96.3 can be used to check to see if spindle indexing is completed and, if completed, cancel the SV speed control mode. Issuing G96.2 not followed by G96.3 cannot cancel the SV speed control mode even if spindle indexing is completed.
Start SV speed control mode cancellation at a path to which the axis of interest belongs.

- Spindle indexing command during spindle rotation

Issuing G96.1 or G96.2 with a position specified during spindle rotation causes the spindle to stop at the specified position.
Example)
M03 S1000 ;..... Rotation at S1000
G96.1 P1 R180. ;..... Stoppage of rotation at the 180deg position

FRYER MACHINE SYSTEMS INC Rob Zahensky

- Spindle indexing speed

Issuing G96.1 or G96.2 causes a move speed to be dedicated to spindle indexing. Specify the move speed for spindle indexing, using parameter No. 11012.

Spindle indexing command (absolute coordinate/machine coordinate)

Using bit 0 (SIC) of parameter No. 11005 can select which coordinate system, absolute or machine, is to be used in spindle indexing.

Example:

If the difference between the machine and absolute coordinates (machine coordinate – absolute coordinate) is 100.000:

- Spindle indexing performed by turning the SV speed control mode signals SRVON1 to SRVON8 OFF (“1” to “0”) or issuing a spindle indexing G code (with no R specified)
  1. If bit 0 (SIC) of parameter No. 11005 = 0:
     Spindle indexing is performed with a machine coordinate of 100.000 and an absolute coordinate of 0.000.
  2. If bit 0 (SIC) of parameter No. 11005 = 1:
     Spindle indexing is performed with a machine coordinate of 0.000 and an absolute coordinate of 260.000.
- G code-based spindle indexing (with R specified)
  Assuming that spindle indexing is specified with R100.000:
  1. If bit 0 (SIC) of parameter No. 11005 = 0:
     Spindle indexing is performed with a machine coordinate of 200.000 and an absolute coordinate of 100.000.
  2. If bit 0 (SIC) of parameter No. 11005 = 1:
     Spindle indexing is performed with a machine coordinate of 100.000 and an absolute coordinate of 0.000.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

10 TOOL FUNCTION (T FUNCTION)

10.1 TOOL SELECTION FUNCTION

By specifying an up to 8-digit numerical value following address T, a code signal and a strobe signal are transmitted to the machine tool. This is used to select tools on the machine.
One T code can be commanded in a block. Refer to the machine tool builder's manual for the number of digits commendable with address T and the correspondence between the T codes and machine operations.
When a move command and a T code are specified in the same block, the commands are executed in one of the following two ways:

(i) Simultaneous execution of the move command and T function commands.

(ii) Executing T function commands upon completion of move command execution.

The selection of either (i) or (ii) depends on the machine tool builder's specifications. Refer to the manual issued by the machine tool builder for details.

Explanations

T

The value after the T code indicates the desired tool. Part of the value is also used as a tool offset number that specifies the amount of tool offset or the like. The tool can be selected as follows according to the specification method and parameter settings.

Description of a T code (Note 1)		How to specify the offset number for each parameter setting (Note 2)
Bit 1 (LGN) of parameter No. 5002 = 0	**Bit 1 (LGN) of parameter No. 5002 = 1**	
TOOOOOOO O ↑ ↑ Tool selection Tool geometry tool wear offset	TOOOOOOO O ↑ ↑ Tool selection tool geometry offset Tool wear offset	The tool wear offset number is specified using the low-order one digit of a T code.
		When parameter No.5028 is set to 1
TOOOOOO OO ↑ ↑ Tool selection Tool geometry tool wear offset	TOOOOOO OO ↑ ↑ Tool selection tool geometry offset Tool wear offset	The tool wear offset number is specified using the low-order two digits of a T code.
		When parameter No.5028 is set to 2
TOOOOO OOO ↑ ↑ Tool selection Tool geometry tool wear offset	TOOOOO OOO ↑ ↑ Tool selection tool geometry offset Tool wear offset	The tool wear offset number is specified using the low-order three digits of a T code.
		When parameter No.5028 is set to 3

NOTE

1 The maximum number of digits of a T code can be specified by parameter No.3032 as 1 to 8.

2 When parameter No.5028 is set to 0, the number of digits used to specify the offset number in a T code depends on the number of tool offsets.
Example)
When the number of tool offsets is 1 to 9: Low-order one digit
When the number of tool offsets is 10 to 99: Low-order two digits
When the number of tool offsets is 100 to 999: Low-order three digits

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Refer to the machine tool builder's manual for correspondence between the T-code and the tool and the number of digit to specify tool selection.

Example (T2+2)
N1G00X1000Z1400;
N2T0313; (Select tool No. 3 and tool offset value No. 13)
N3X400Z1050;

10.2 TOOL MANAGEMENT FUNCTION

Overview

The tool management function totally manages tool information including information about tool offset and tool life.

Explanation

A tool type number is specified with a T code. The tool type number is any number the user can define freely. With tool type numbers, tools can be grouped by various conditions such as life, compensation value, and cutting conditions. When each type is assumed to have a single tool, tool type numbers are equivalent to unique tool numbers.

For each tool, an information storage area is prepared in the CNC (SRAM). This area contains information such as a tool type number, tool life, tool status (such as a breakage condition), tool compensation number (H, D, G, or W), spindle speed (S), cutting feedrate (F), and freely-definable customize data. Such data is called tool management data.
A cartridge management table that links cartridge information and tool management data is provided so that the CNC can manage the cartridges of the machine and tool change operations. In addition, areas for managing the tools in the spindle and tool standby positions are prepared.

When a tool type number is specified using a T code command, a tool that has the tool type number and the shortest life is searched for, and the cartridge number and pot number of the location where the tool is stored are output to the PMC. Then, a tool change operation using the cartridge number and pot number is enabled by the PMC ladder sequence.

Tool information in the CNC is managed by the tool management data and cartridge management table (including the spindle management table and standby position table).

- Tool management data

As tool management data, information about each tool is registered with a tool management data number.
The following number of sets of tool management data can be used:

Tool management function 64 sets	64 sets in total
Tool management function 240 sets	240 sets in total
Tool management function 1000 sets	1000 sets in total

NOTE
For the number of tool management data sets, refer to the relevant manual issued by the machine tool builder.

FRYER MACHINE SYSTEMS INC
Rob Zamonsky

- Details of data

The following details the tool management data registered for each data number:

- Tool type number (T code)

Item	Description
Data length	4byte
Valid data range	0,1 to 99,999,999

- Tool life counter

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds)

The value of an increment counter, that is, the number of use times (time) is indicated. The remaining life value is [the maximum tool life value minus tool life counter value].

- Maximum tool life value

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds)

- Notice life value

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds)

If a value other than 0 is set, the tool life expiration notice signal TLCHB or TLCHBx is output when the remaining life value of the tool (= maximum tool life value minus tool life counter value) has reached the set value.

- Tool life status

Item	Description
Data length	1byte
Detail data	0: Life management is not performed. 1: Tool not yet used 2: Life remains. 3: Life expired. 4: Tool breakage (skip)

The machine (PMC) determines tool breakage and stores corresponding information through the window. In tool management of the CNC, a broken tool is regarded as being equivalent to tools whose lives have expired.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Tool information

Item	Description
Data length	1 byte (flag data)
#0 RGS	0: Tool management data is invalid. (-) 1: Tool management data is valid. (R)
#1 TIM	0: Type for counting the number of use times (C) 1: Type for counting time (T)
#2 BDT	0: Normal tool (N) 1: Oversize tool (B)
#3 LOC	0: Data accessible (U) 1: Data not accessible (L)
#4 SEN	When the tool life status indicates that tool management is not performed: 0: This tool is not searched. (-). 1: This tool is searched. (S)
#5 to #7	Reserved

When RGS is set to 0 in tool management data, the tool management data is regarded as not being registered even when values are set for other items.

NOTE
Make sure you specify the same life count type for all tools of the same type. You can check tools of the same type for any difference in life count type, using the "check function."

M

- Tool length compensation number (H)

Item	Description
Data length	2byte
Valid data range	0 to 999

- Cutter compensation number (D)

Item	Description
Data length	2byte
Valid data range	0 to 999

T

- Tool geometry compensation number (G)

Item	Description
Data length	2byte
Valid data range	0 to 999

- Tool wear compensation number (W)

Item	Description
Data length	2byte
Valid data range	0 to 999

NOTE
When the machine control type is the combined system type, tool length compensation and cutter compensation numbers are used for paths for the machining center system, and for paths for the lathe system, tool geometry compensation and tool wear compensation numbers are used.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Spindle speed (S)

Item	Description
Data length	4byte
Unit of data	min^{-1}
Valid data range	1 to 99,999

- Feedrate (F)

Item	Description
Data length	4byte
Unit of data	mm/min, inch/min, deg/min, mm/rev, inch/rev
Valid data range	0 to 99,999,999

As additional tool management data, areas for setting customize data (5 data items including customize data 0 and customize data 1 to 4) are provided without defining specific usage. The user can use these customize data areas freely to set a warning life value, cutting resistance, override value, spindle current value, maximum and minimum S/F, and other items according to the targeted application.

- Customize data 0

Item	Description
Data length	1 byte (bit type)
Valid data range	0 or 1 on a bit-by-bit basis

- Customize data 1 to 4 (to 20) (to 40)

Item	Description
Data length	4byte
Valid data range	-99,999,999 to 99,999,999

NOTE
For the maximum number of tool management function customization data items, refer to the relevant manual issued by the machine tool builder.

- Cartridge management table

The storage status of tools in cartridges is managed with the cartridge management table.

- Multiple cartridge numbers can be defined. (Numbers from 1 to 4, up to four cartridges)
- The maximum number of pots for all cartridges is 64, 240, or 1000, which depends on whether a tool management data option is selected.
- Cartridge numbers and pot numbers can be assigned freely within cartridge management data by parameter setting (described later).
- The tool management data number adjacent to a pot number is linked with tool data defined in the tool management table. Therefore, the tool attached to the pot is indicated.
- Zero set as a data number indicates that no tool is attached.
- The cartridge management table can be read from and written to through the PMC window and FOCAS2.
- The spindle management table and standby position table are provided to indicate special cartridge positions.
- Spindle positions and standby positions, regarded as special cartridge positions, have fixed cartridge numbers 11 to 14 (the positions of the first to fourth spindles) and 21 to 24 (the first to fourth standby positions).
- With the PMC window, the spindle position table and standby position table can be read from and written to.
- Tool life counting is performed only for the tools at the spindle positions.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

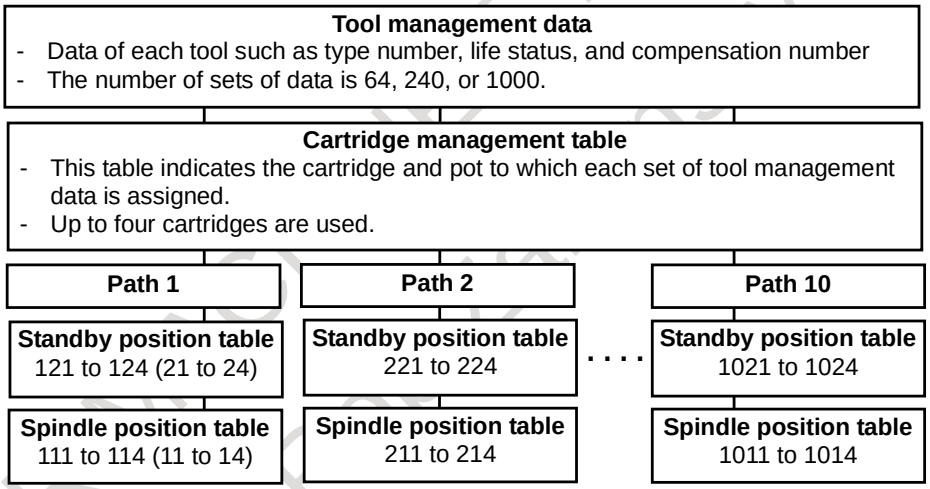
- Multi-path system

The tool management data and cartridge management table are common data among the paths. The spindle management table and standby position table, however, are treated as independent data for each path.
When the spindle table or standby position table is specified as a cartridge from the PMC window, specify the following, in which the path number is set in the hundred's place:

	Spindle position			
	First	**Second**	**Third**	**Fourth**
First path	111(11)	112(12)	113(13)	114(14)
Second path	211	212	213	214

	Standby position			
	First	**Second**	**Third**	**Fourth**
First path	121(21)	122(22)	123(23)	124(24)
Second path	221	222	223	224

NOTE
When specifying 111, 121, and so on to specify the first path, you may specify just 11, 21, and so on.



- M/T series

The tool management data and cartridge management table are data common to the M and T series.

- Tool life management and tool change

The CNC performs tool life management, regarding tools having the same tool type number as a group. When a tool type number (T code) is specified by an NC program, the tool management data registered in the CNC is searched to find a tool having the shortest life among the tools with the same tool type number.
The cartridge number and pot number corresponding to the searched tool are output as a T code signal to the PMC. Based on the output cartridge number and pot number, the PMC performs preparation for a tool change (to the next tool). A different tool can also be selected on the PMC side.
The CNC performs tool life counting for each tool that is at a spindle position in the spindle management table.

When the lives of all tools having the tool type number specified by the T code have expired, alarm PS5317, "ALL TOOL LIFE IS OVER" is issued. If there is a tool in the spindle position or standby position, that tool is selected to continue machining.

As the T code signal, a specified tool type number instead of the cartridge number and pot number can also be output directly by parameter setting.

There are two types of tool life management counting methods: counting the number of use times and counting cutting time. One of the counting methods is set in tool information of tool management data.

Other major specifications related to tool life management are as follows:

Tool type number (T code) : Up to 8 digits (1 to 99,999,999)
Maximum tool life value : 99,999,999 times when the number of use times is specified
: 999 hours 59 minutes 59 seconds when time is specified
Life count interval when time is specified : 1 second
Tool life management count restart M code : Enabled
Tool life count override : Enabled

To specify a certain tool directly without performing tool life management by tool type number, use the following format:

M (value in parameter No. 13252) T (cartridge number) (pot number) ;

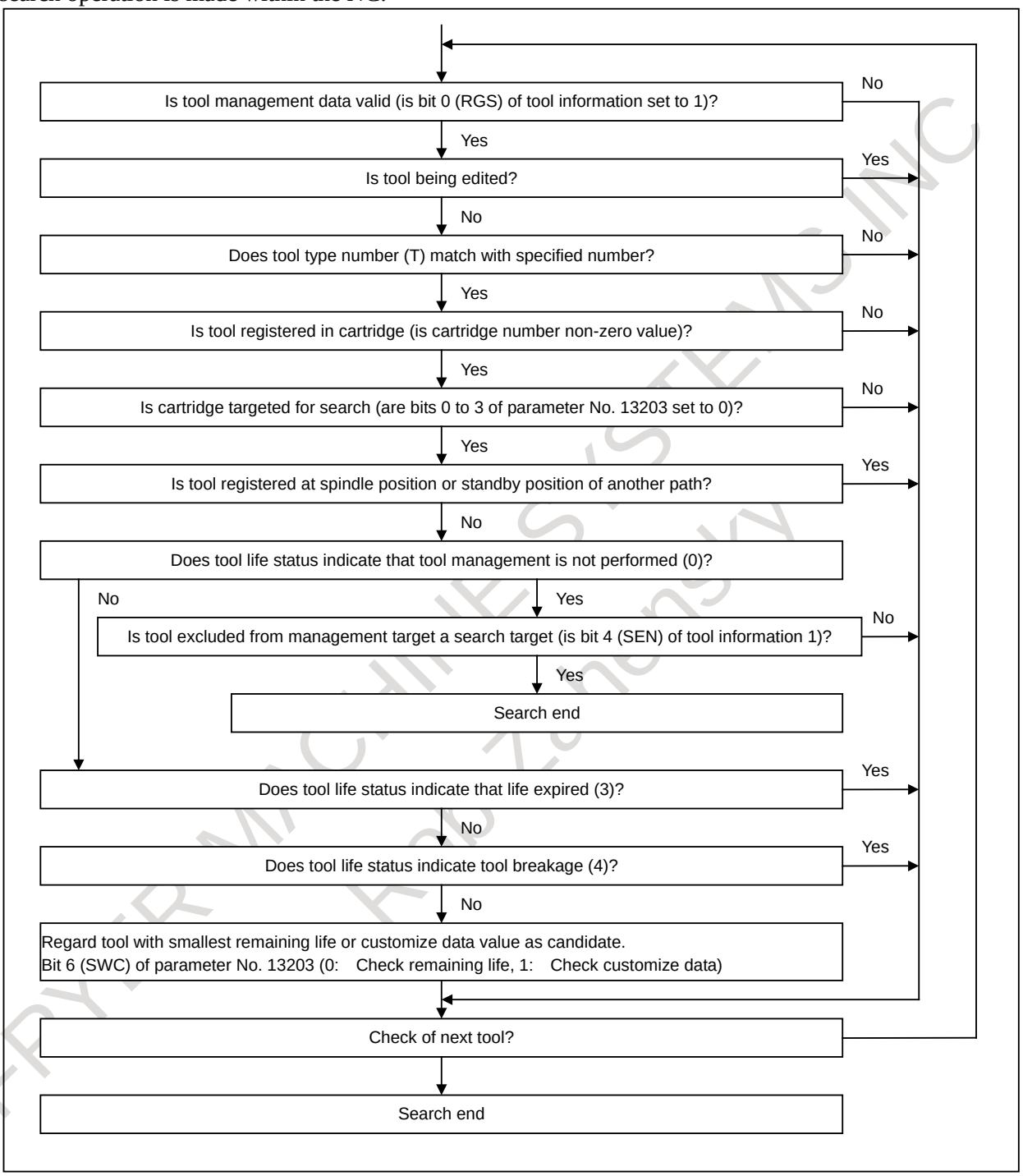
When parameter No. 13252 is set to 333, the cartridge number is 2, and the pot number is 27, for example, the following command is specified:

M333 T20027;

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Tool search order

Tools having a tool type number (T) specified by a program are searched sequentially from tool management data number 1 while registered data contents are checked. The following shows how a search operation is made within the NC:



- System variables

The following tool management data of the tool being used as a spindle after a tool change by M06 and the tool to be used next which is specified by a T code can be read through custom macro variables:

Being used	Item	
#8401	Tool management data number	
#8402	Tool type number (T)	
#8403	Tool life counter	
#8404	Maximum tool life value	
#8405	Tool notice life value	
#8406	Tool life status	
#8407	Customize data 0 (bit)	
#8408	Tool information	
#8409	Tool length compensation number (H)	For machining center systems
#8410	Cutter compensation number (D)	
#8411	Spindle speed (S)	
#8412	Cutting feedrate (F)	
#8413	Tool geometry compensation number (G)	For lathe systems
#8414	Tool wear compensation number (W)	
#8431	Customize data 1	
#8432	Customize data 2	
#8433	Customize data 3	
#8434	Customize data 4	
#8435	Customize data 5	
#8436	Customize data 6	
#8437	Customize data 7	
#8438	Customize data 8	
#8439	Customize data 9	
#8440	Customize data 10	
#8441	Customize data 11	
#8442	Customize data 12	
#8443	Customize data 13	
#8444	Customize data 14	
#8445	Customize data 15	
#8446	Customize data 16	
#8447	Customize data 17	
#8448	Customize data 18	
#8449	Customize data 19	
#8450	Customize data 20	
#8451	Customize data 21	
#8452	Customize data 22	
#8453	Customize data 23	
#8454	Customize data 24	
#8455	Customize data 25	
#8456	Customize data 26	
#8457	Customize data 27	
#8458	Customize data 28	
#8459	Customize data 29	
#8460	Customize data 30	
#8461	Customize data 31	
#8462	Customize data 32	
#8463	Customize data 33	
#8464	Customize data 34	
#8465	Customize data 35	
#8466	Customize data 36	
#8467	Customize data 37	

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Being used	Item
#8468	Customize data 38
#8469	Customize data 39
#8470	Customize data 40

When a cartridge number of a spindle position (11 to 14) or standby position (21 to 24) is specified in #8400, information about the corresponding position can be read.
If the spindle position table or standby position table has an empty pot, <empty> is read from #8402 to #8470.
Value 0 is read from #8401 (data number).

Therefore, machining conditions registered in tool management data can be specified directly by coding, for example, D#8410, H#8409, S#8411, and F#8412 with a tool change macro (such as M06). Similarly, customize data can be referenced by a custom macro, and machining programs can be customized according to the tool used.

NOTE
For the maximum number of tool management function customization data items, refer to the relevant manual issued by the machine tool builder.

- Specifying a tool compensation number

M

When parameter No. 13265 is 0, a compensation number registered as tool management data of a tool attached at a spindle position can be selected by specifying H99 or D99.
(99 is treated as a special number, so 99 cannot be specified directly as a compensation number.)
When other than 0 is set in parameter No. 13265, the number set in the parameter can be specified instead of 99. For example, if parameter No. 13265 is 3, specifying H3 specifies the tool length compensation number registered for the tool at the spindle position.

T

When the tool management function is not used, a tool compensation number is specified also with a T code; when the tool management function is used, the T code is used only to specify a tool type number, so a tool compensation number must be specified with address D.
Except the specifications for addresses, the specifications for the number of permissible digits (parameter No. 3032) and the number of digits consisting of a compensation number (parameter No. 5028), separation of geometry compensation numbers and wear compensation numbers (bit 1 (LGN) of parameter No. 5002), and so on are the same as for T.

If parameter No. 13265 is set to 0, when a compensation number registered for a tool attached at a spindle position is specified, the specification format varies according to the number of digits as follows, which is the same as for conventional T:
When the compensation number is 1 digit long : D9
When the compensation number is 2 digits long : D99
When the compensation number is 3 digits long : D999
Because 9, 99, or 999 is treated as a special number, it cannot be specified directly as a compensation number.
Unless parameter No. 13265 is set to 0, the number set in the parameter can be specified instead of 9, 99, or 999. When parameter No. 13265 is set to 3, specifying D3 specifies the tool geometry compensation number and tool wear compensation number registered for a tool attached at the spindle position.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

Multi-path system

Depending on whether the local path is a machining center system or a lathe system, tool compensation numbers are specified by using one of the above methods.

Spindle selection

When specifying compensation numbers of a tool attached to a spindle other than the first spindle, specify the spindle number with address P within the same block that contains H/D. When specifying the first spindle, you can omit P.

D99 P3; Specifying compensation numbers registered for the tool attached at the third spindle
D99; Specifying compensation numbers registered for the tool attached at the first spindle

- Read/write operations for tool management data and cartridge management table

The tool management data and cartridge management table can be read from and written to by using the CNC/PMC data window library (FOCAS2). Therefore, a specific tool management system including all available tool data not registered in the CNC can be built easily by using the personal computer function.

Similarly, the tool management data and cartridge management table can also be read from and written to using the PMC window.

Tool management data of the tool being used as a spindle after a tool change operation performed by M06 and the tool being selected by specifying a T code can be read using a custom macro.

The user can modify tool management data by MDI from the tool management function screen of the CNC. Addition, modification, and deletion of the above tool management data can be made from a part program (G10). Such data can also be input to and output from external I/O equipment by using the ALL I/O screen and tool management function screen.

- G10 format

Addition, modification, and deletion are performed for the tool management data and cartridge management table from programs. For use of this function, the programmable data input option function is required.

If a format error is found in the commands from G10 L75/L76/L77 to G11, or if a value beyond the valid data range is specified, alarm PS5312, “ILLEGAL COMMAND IN G10 L75/76/77” is issued. In such a case, correct the program. Within the range from G10 to G11, no decimal point can be specified with any address. If a decimal point is specified, alarm PS0007, “ILLEGAL USE OF DECIMAL POINT” results.

- Registering new tool management data

Tool management data can be registered. When data is output to an external device from the tool management data screen, this format is used. The specification of those items that are not registered may be omitted.

FLYER MACHINE SYSTEMS INC
Rob Zaherski

G10 L75 P1;
N_ ; Tool management data number specification
T_ C_ L_ I_ B_ Q_ H_ D_ S_ F_ J_ K_ ;
P0 R_ ; Customization data 0
P1 R_ ; Customization data 1
P2 R_ ; Customization data 2
P3 R_ ; Customization data 3
P4 R_ ; Customization data 4
N_ ; Next tool management data number specification
:
G11;

N_	Tool management data No.	1 to 64 (1 to 240, 1 to 1000)
T_	Tool type No. (T)	0 to 99,999,999
C_	Tool life counter	0 to 99,999,999
L_	Maximum tool life	0 to 99,999,999
I_	Noticed life	0 to 99,999,999
B_	Tool life state	0 to 4
Q_	Tool information	Bit format (8 bits)
H_	Tool length compensation No. (H)	0 to 999 (M series)
D_	Cutter compensation No. (D)	0 to 999 (M series)
S_	Spindle speed (S)	0 to 99,999
F_	Cutting feedrate (F)	0 to 99,999,999
J_	Tool geometry compensation No. (G)	0 to 999 (T series)
K_	Tool geometry compensation No. (W)	0 to 999 (T series)
P_	Customization data No.	0 to 4 (0 to 20, 0 to 40)
R_	Customization data value	-99,999,999 to 99,999,999

Specify customization data in the following format:

P (customization-number) R (value)

Use the bit format only when specifying the customization data 0 (P0). Specify other data in the binary format. The specification of customization data that need not be set may be omitted.

Example)

```
G10 L75 P1;
N1 ;                          Tool management data No.1
T10000001 C0 L1000 B1 Q00000001 H1 D1 S4000 F10000 ;   } Registering tool management data No.1
P0 R11101101 ;                Customization data 0
P4 R99999999 ;                Customization data 4
N2 ;                          Tool management data No.2
T10000002 C0   L2000 B1 Q00000001 H2 D2 S3000 F6000 ;  } Registering tool management data No.2
P1 R200 ;                     Customization data 1
P2 R10 ;                      Customization data 2
N5 ;                          Tool management data No.5
T10000005 C0   L5000 B1 Q00000001 H5 D5 S5000 F11000 ; } Registering tool management data No.5
P2 R1 ;                       Customization data 2
P3 R500 ;                     Customization data 3
G11 ;
```

FB YEP MACHINE SYSTEMS INC
Rob Zadersky

Modifying tool management data

Tool management data can be modified.
The specification of those items that are not modified may be omitted.

```
G10 L75 P2 ;
N_ ;
T_ C_ L_ I_ B_ Q_ H_ D_ S_ F_ J_ K_ ;
P_ R_ ;
N_ ;
 :
G11 ;
```

Deleting tool management data

The data of a specified data number can be deleted from tool management data.
The cartridge management table data corresponding to a deleted tool management data number is also deleted. (The tool management data number in the cartridge management table is cleared to 0.)

```
G10 L75 P3 ;
N_ ;
N_ ;
 :
N_ ;
G11 ;
```

Registering new cartridge management table data

A tool management data number can be registered with a free pot in the cartridge management table.

```
G10 L76 P1 ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
G11 ;
```

For a spindle position table and standby position table, only cartridge number data is specified.
Example)

G10 L76 P2 ;
N11 R1; Changes the tool management data number of the spindle position to No. 1.
N21 R29; Changes the tool management data number of the standby position to No. 29.
G11 ;

Modifying the cartridge management table

Tool management data numbers in the cartridge management table can be modified.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

```
G10 L76 P2 ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
G11 ;
```

For a spindle position table and standby position table, only cartridge number data is specified.
Example)

G10 L76 P2 ;
N11 R1; Changes the tool management data number of the spindle position to No. 1.
N21 R29; Changes the tool management data number of the standby position to No. 29.
G11 ;

Deleting cartridge management table data

Tool management data numbers can be deleted from the cartridge management table.

```
G10 L76 P3 ;
N cartridge-number P pot-number ;
N cartridge-number P pot-number ;
N cartridge-number P pot-number ;
N cartridge-number P pot-number ;
G11 ;
```

For a spindle position table and standby position table, only cartridge number data is specified.
Example)

G10 L76 P3 ;
N11 ; Deletes the tool management data number of the spindle position (clears the number to 0).
N21 ; Deletes the tool management data number of the standby position (clears the number to 0).
G11 ;

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Naming customization data

The display name of customization data (0 to 40) can be set.

```
G10 L77 P1 ;
N_ ;
P_ R_ ;
P_ R_ ;
;
N_ ;
P_ R_ ;
P_ R_ ;
G11 ;
```

N_: Customization data No. (0 to 40)
P_: Character No. (1 to 16)
R_: Character code (ASCII)
- Specify 0 to clear data.

- A set display name can be checked on the tool management data screen only.
- When no name is registered, a name such as the default "Customize 3" is displayed.
- A name consisting of up to 16 characters is displayed. For a name shorter than 16 characters, register 0 in the empty area. Those characters that are registered immediately before 0 are displayed.
- To clear data, set 0 as a character code.
- No character code data check is made.

When the name of customization data 3 is set as "MEASURE5", for example, specify the following:
Example)

```
G10 L77 P1 ;
N3 ;        Specifies customization data 3.
P1 R32 ;    character ‘space’ ASCII code 20h
P2 R77 ;    character “M” ASCII code 4Dh
P3 R69 ;    character “E” ASCII code 45h
P4 R65 ;    character “A” ASCII code 41h
P5 R83 ;    character “S” ASCII code 53h
P6 R85 ;    character “U” ASCII code 55h
P7 R82 ;    character “R” ASCII code 54h
P8 R69 ;    character “E” ASCII code 45h
P9 R53 ;    character “5” ASCII code 35h
P10 R0 ;    Clears data. (Not displayed. End)
G11 ;
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Naming tool life states

The display name of a tool life state (0 to 4) can be set.

```
G10 L77 P2 ;
N_ ;
P_ R_ ;
P_ R_ ;
N_ ;
P_ R_ ;
P_ R_ ;
G11 ;
```

N_:	Tool life state (0 to 4)
P_:	Character No. (1 to 12)
R_:	Character code (ASCII)
-	Specify 0 to clear data.

- A set display name can be checked on the tool management data screen only.
- If no name is registered, a default such as "Remaining" and "Unremaining" is displayed.
- A name consisting of up to 12 characters is displayed. For a name shorter than 12 characters, register 0 in the empty area. Those characters that are registered immediately before 0 are displayed.
- To clear data, set 0 as a character code.
- No character code data check is made.
- The defaults are as follows:
  - 0: Invalid
  - 1: Remaining
  - 2: Remaining
  - 3: Unremaining
  - 4: Broken

To set "NORMAL" as the name of tool life state 2 (Remaining), specify the following:
Example)

```
G10 L77 P2 ;
N2 ;          Specifies tool life state 2.
P1 R78 ;      character “N” ASCII code 4Eh
P2 R79 ;      character “O” ASCII code 4Fh
P3 R82 ;      character “R” ASCII code 52h
P4 R77 ;      character “M” ASCII code 4Dh
P5 R65 ;      character “A” ASCII code 41h
P6 R76 ;      character “L” ASCII code 4Ch
P7 R0 ;       Clears data. (Not displayed. End)
G11 ;
```

FRYER MACHINE SYSTEMS INC
Rob Zahenski

10.3 TOOL MANAGEMENT EXTENSION FUNCTION

Overview

The following functions have been added to the tool management function:

1. Customization of tool management data display
2. Setting of spindle position/standby position display
3. Input of customize data with the decimal point
4. Protection of various tool information items with the KEY signal
5. Selection of a tool life count period
6. Each tool data screen
7. Total life time display for tools of the same type

They will be explained in detail below.

10.3.1 Customization of Tool Management Data Display

With the tool management data screen display customization function, the display positions of screen elements (type number, tool information, life counter, and so forth) on the tool management screen can be changed and whether to display or hide such screen elements can be chosen using the G10 format. This function enables a customized tool management screen to be configured.

Format

G10L77P3;

N_ R_;

G11;

N_: Tool management data screen display position number
R_: Tool management data display item

Explanation

- **number (N_)**
- A display position number represents an ordinal number from the leftmost position on the tool management data screen.
- As N, a number from 1 to 200 can be specified.

- **Tool management data display item (R_)**
- For a value to be set (in R_), see the table below.
- For actual setting, consider a display width plus 1.

- Items common to machining center systems and lathe systems

R	Item	Display width	Remarks
-1	End of setting	-	
0	Blank column	10	
1	No.	4	For spindle/standby positions for 2 paths
2	TYPE-NO.	8	
3	MG	4	
4	POT	5	
5	T-INFORMATION	10	
6	L-COUNT	10	
7	MAX-LIFE	10	
8	NOTICE-L	10	

FRYER MACHINE SYSTEMS INC
Rob Zahensky

R	Item	Display width	Remarks
9	L-STATE	6 or 12	The display width is switched by bit 1 of parameter No. 13201.
10	S (Spindle speed)	10	
11	F (Feedrate)	10	
12	Tool figure number (A)	3	

- Offset-related items for machining center systems

R	Item	Display width	Remarks
20	H (Tool length compensation number)	4	
21	D (Cutter compensation number)	4	
22	OFFSET-M	10	Tool offset memory A
25	GEOMETRY(H)	10	Tool offset memory C
26	WEAR(H)	10	
27	GEOMETRY(D)	10	
28	WEAR(D)	10	

- Offset-related items for lathe systems

R	Item	Display width	Remarks
40	TG (Tool geometry compensation number)	4	
41	TW (Tool wear compensation number)	4	
42	OFFSET-X	10	
43	OFFSET-Z	10	
44	OFFSET-R	10	Tool nose radius compensation
45	TIP	10	
46	OFFSET-Y	10	Y-axis offset
47	OFFSET-B	10	B-axis control
48	GEOMETRY (X)	10	Tool geometry/wear compensation
49	WEAR (X)	10	
50	GEOMETRY (Z)	10	
51	WEAR (Z)	10	
52	GEOMETRY (R)	10	Tool nose radius compensation, Tool geometry/wear compensation
53	GEOMETRY TIP	10	
54	WEAR (R)	10	
55	WEAR TIP	10	
56	GEOMETRY (Y)	10	Y-axis offset, Tool geometry/ wear compensation
57	WEAR (Y)	10	
58	GEOMETRY (B)	10	B-axis control, Tool geometry/ wear compensation
59	WEAR (B)	10	
60	GEOMETRY (X2)	10	2nd geometry tool offset
61	GEOMETRY (Z2)	10	
62	GEOMETRY (Y2)	10	

Items related to customize data

R	Item	Display width	Remarks
80	CUSTOM 0	10	
81	CUSTOM 1	10	
82	CUSTOM 2	10	
83	CUSTOM 3	10	
84	CUSTOM 4	10	

FRYER MACHINE SYSTEMS INC
Rob Zahensky

R	Item	Display width	Remarks
85	CUSTOM 5	10	Tool management function customize data extension (5 to 20) or tool management function customize data extension (5 to 40)
86	CUSTOM 6	10	
87	CUSTOM 7	10	
88	CUSTOM 8	10	
89	CUSTOM 9	10	
90	CUSTOM 10	10	
91	CUSTOM 11	10	
92	CUSTOM 12	10	
93	CUSTOM 13	10	
94	CUSTOM 14	10	
95	CUSTOM 15	10	
96	CUSTOM 16	10	
97	CUSTOM 17	10	
98	CUSTOM 18	10	
99	CUSTOM 19	10	
100	CUSTOM 20	10	
101	CUSTOM 21	10	Tool management function customize data extension (5 to 40)
102	CUSTOM 22	10	
103	CUSTOM 23	10	
104	CUSTOM 24	10	
105	CUSTOM 25	10	
106	CUSTOM 26	10	
107	CUSTOM 27	10	
108	CUSTOM 28	10	
109	CUSTOM 29	10	
110	CUSTOM 30	10	
111	CUSTOM 31	10	
112	CUSTOM 32	10	
113	CUSTOM 33	10	
114	CUSTOM 34	10	
115	CUSTOM 35	10	
116	CUSTOM 36	10	
117	CUSTOM 37	10	
118	CUSTOM 38	10	
119	CUSTOM 39	10	
120	CUSTOM 40	10	

NOTE

1 If G10 L77 P3 is terminated normally, the power must be turned off before operation is continued.

2 The setting becomes effective after the power is turned off then back on.

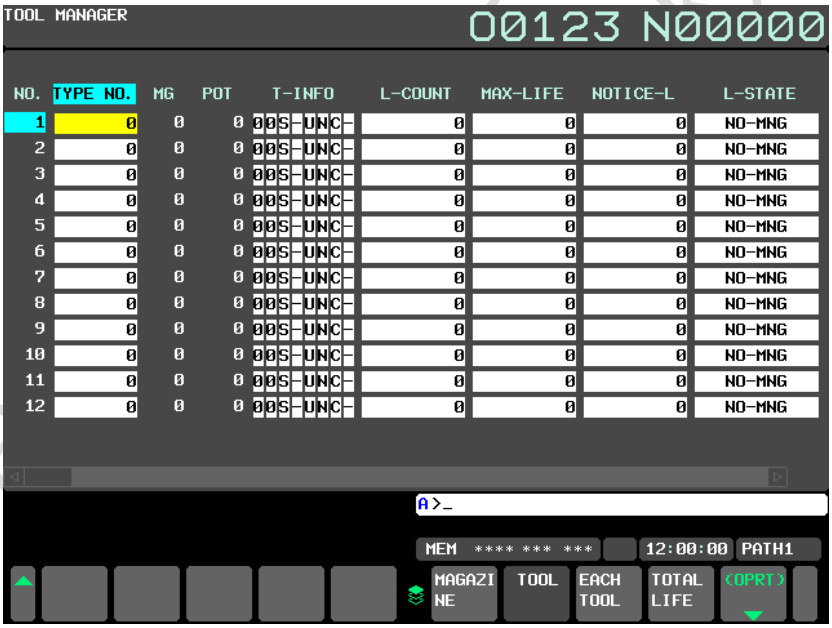
FRYER MACHINE SYSTEMS INC
Rob Zahensky

Example

Example of setting tool offset memory A

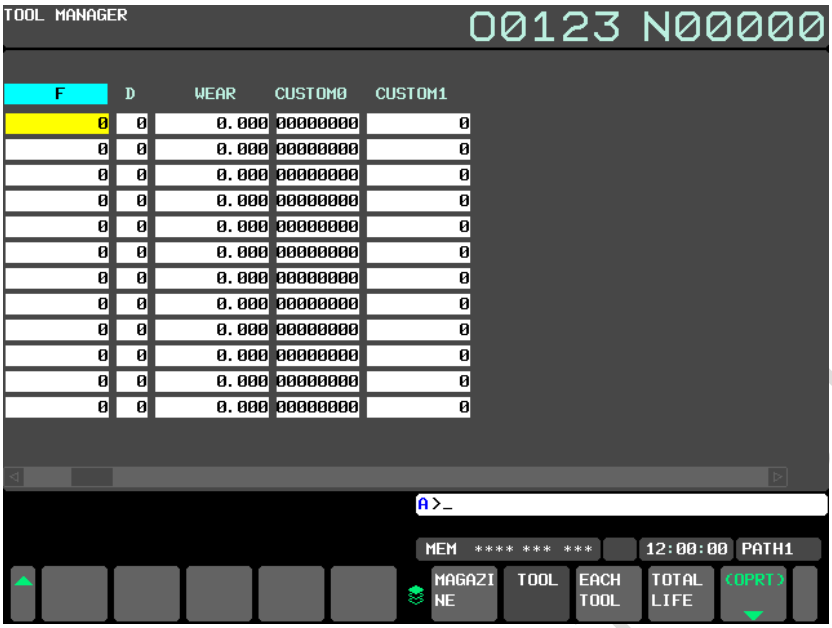
G10L77P3;	Set tool management data screen display customization
N1 R1;	Set No. as number 1
N2 R2;	Set TYPE-NO. as number 2
N3 R3;	Set MG as number 3
N4 R4;	Set POT as number 4
N5 R5;	Set T-INFORMATION as number 5
N6 R6;	Set L-COUNT as number 6
N7 R7;	Set MAX-LIFE as number 7
N8 R8;	Set NOTICE-L as number 8
N9 R9;	Set L-STATE as number 9
N10 R11;	Set F (Feedrate) as number 10
N11 R21;	Set D as number 11
N12 R22;	Set OFFSET-M as number 12
N13 R80;	Set CUSTOM 0 as number 13
N14 R81;	Set CUSTOM 1 as number 14
N15 R-1;	End
G11;	Cancel the setting mode

On the tool management data screen, the first page displays "No., Type Number, MG, Pot, Tool Information, Life Counter, Maximum Life, Noticed Life, and Life State".
The second page displays "F (Feedrate), D (Cutter Compensation Number), Tool Offset-M, Customize Data 0, and Customize Data 1".



Example 1: Page 1

FRYER SYSTEMS INC



Example 1: Page 2

NOTE

1 This setting is enabled when bit 0 (TDC) of parameter No. 13201 is set to 1.
2 Up to 20 pages can be set.
3 Be sure to specify an end.
4 If an item that requires the corresponding option is specified without specifying the option, the item is displayed as a blank field.
5 No items after an end are displayed.
(Example) In the case of N1 → R1 (No.), N2 → R2 (Type Number), N 3 → R3 (MG), N4 → R-1 (End), N5 → R4 (Pot), only the items "No., Type Number, and MG" are displayed on the screen.
6 If G10 L77 P3 is specified, the setting immediately before is initialized.
7 If an invalid value is set in R (tool management data display item), a blank field is displayed.

10.3.2 Setting of Spindle Position / Standby Position Display

In MG on the tool management data screen, a spindle position or standby position is displayed as a number such as 11, 12, and 13. With the spindle position/standby position display setting function, three arbitrary characters can be displayed using the G10 format.

Format

G10L77P4;
N_;
P_R_;
G11;

N_: Spindle position/standby position setting
P_: Character number
R_: Character code

FRIER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

- Spindle position/standby position setting (N_)

Specify a spindle position or standby position to be renamed. The table below indicates the values to be specified.

	Spindle position			
	First	**Second**	**Third**	**Fourth**
1st path	111	112	113	114
2nd path	211	212	213	214

	Standby position			
	First	**Second**	**Third**	**Fourth**
1st path	121	122	123	124
2nd path	221	222	223	224

- Character number (P_)

Specify a character number (1 to 3). Up to three characters are displayed. If a character string to be specified is shorter than three characters, specify 0 in the leading blank character position(s). A character string immediately before 0 is displayed.

- Character code (R_)

Set the name of a spindle position/standby position by using a character code (ASCII code).

- For a character code, use ASCII code.
- No character code data check is made.

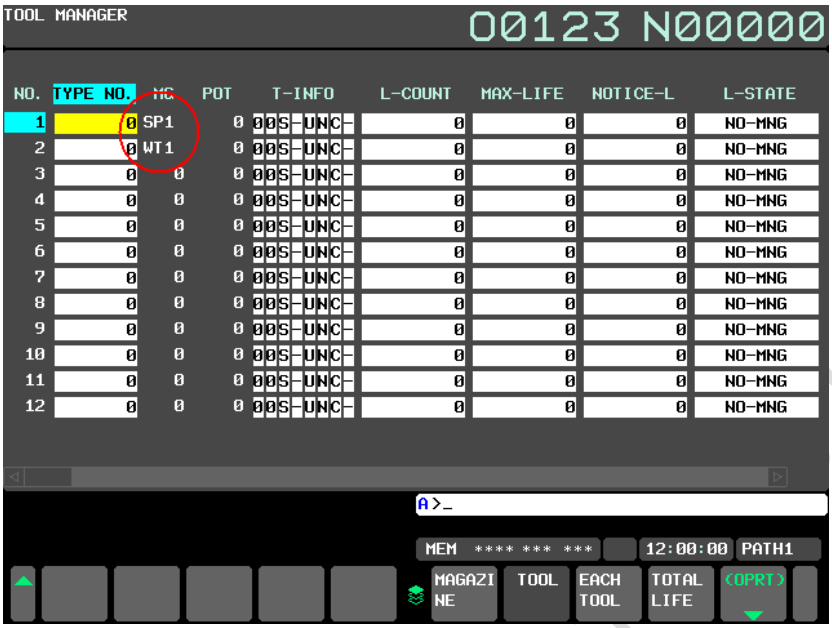
Example

When spindle 1 is named "SP1", and standby 1 is named "WT1"

```
G10L77P4;      Set spindle position/standby position
N111;          Specify spindle 1
P1 R83;        53h as ASCII code for "S"
P2 R80;        50h as ASCII code for "P"
P3 R49;        31h as ASCII code for "1"
N121;          Specify standby 1
P1 R87;        57h as ASCII code for "W"
P2 R84;        54h as ASCII code for "T"
P3 R49;        31h as ASCII code for "1"
G11;           Cancel the setting mode
```

In the MG item on the tool management data screen, spindle 1 is displayed as "SP1", and standby 1 is displayed as "WT1".

FRYER MACHINE SYSTEMS INC
Rob Zalensky



NOTE

Data registered becomes effective after the screen display is switched to the tool management screen.

10.3.3 Input of Customize Data with the Decimal Point

With the function for input of customize data with the decimal point, the number of decimal places is set using the G10 format for each customize data item (customize data 1, ..., 40) to enable data input with the decimal point.

Format

G10L77P5;

N_R_;

G11;

N_: Customize data number
R_: Decimal point position

Explanation

- **Customize data number (N_)**

- A number from 1 through 4 can be specified for N.
- When tool management function customize data extension (from 5 through 20) is available, A number from 1 through 20 can be specified.
- When tool management function customize data extension (from 5 through 40) is available, A number from 1 through 40 can be specified.

- **Decimal point position (R_)**

Specify a decimal point position (from 0 through 7). When 0 is set, the number of decimal places of customize data input cannot be performed.

FRYER MACHINE SYSTEMS INC
Rob Zahansky

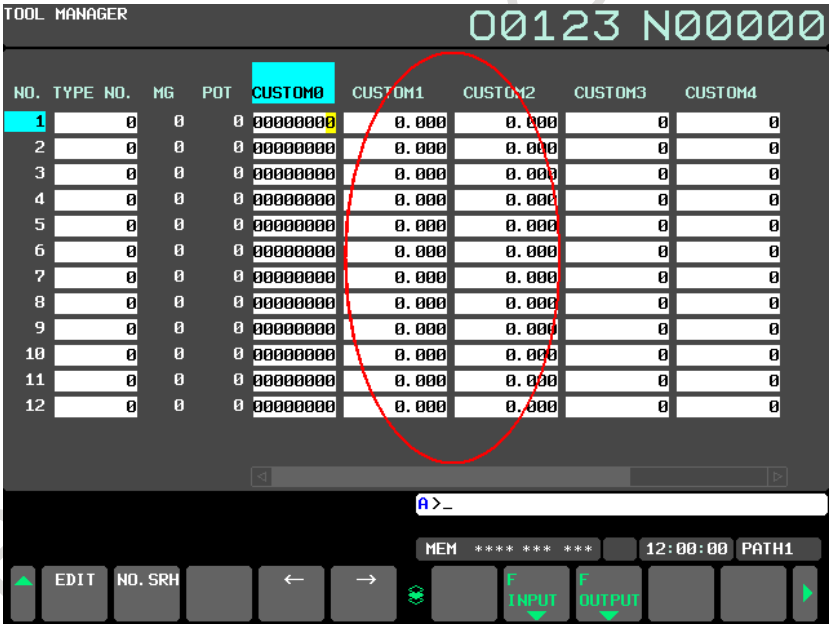
NOTE

1 If G10 L77 P5 is terminated normally, the power must be turned off before operation is continued.
2 The setting becomes effective after the power is turned off then back on.
3 When the number of decimal places is set for customize data, the number of decimal places can be input only through the MDI keys.
4 To set the number of decimal places for customize data 5 to 20, the customize data extension (5 to 20) option for the tool management function or the customize data extension (5 to 40) for the tool management function is required.
5 To set the number of decimal places for customize data 21 to 40, the customize data extension (5 to 40) for the tool management function is required.

Example 1

When customize data 1 and customize data 2 are input with three decimal places

G10L77P5;	Set the number of decimal places for customize data
N1 R3;	Set the number of decimal places to 3 for customize data 1
N2 R3;	Set the number of decimal places to 3 for customize data 2
G11;	Cancel the setting mode



- Use in the G10 format

Between G10 and G11, the decimal point cannot be specified at any address. So, if decimal point input for customize data is enabled with this function, a G10 L75/L76/L77 command using a custom macro variable (#8431 to #8450) reads and sets the value of the integer part.

Example 2

(Example 1)

Condition:

"3" is set as the decimal point position of customize data 1.
"1" is set as the decimal point position of customize data 2.

Operation:

Data is transferred from customize data 1 to customize data 2 by using a custom macro variable.

FRYER SYSTEMS INC

```
G10L77P5;          <1> Set customize data decimal point position
N1 R3;             <2> Set 3 as decimal point position of customize data 1
N2 R1;             <3> Set 1 as decimal point position of customize data 2
G11;               <4> Cancel the setting mode
;
G10 L75 P1;        <5> Register tool management data
N01;               <6> Register with No. 1
P1 R12345;         <7> Set “12.345” for customize data 1
P2 R#8431;         <8> Set “1.2” for customize data 2
G11;               <9> Cancel the setting mode
```

In Example 1, customize data 1 is directly set in customize data 2 by using a custom macro variable. Customize data 1 holds "12.345". In step <8>, only the integer part is read and processed as "P2 R12".
So, "1.2" is set in customize data 2.

(Example 2)

Condition:

"3" is set as the decimal point position of customize data 1.
"1" is set as the decimal point position of customize data 2.

Operation:

By using the custom macro variable of customize data 1, the data is transferred to customize data 2 after being multiplied by 1000.

```
G10L77P5;          <1> Set customize data decimal point position
N1 R3;             <2> Set 3 as decimal point position of customize data 1
N2 R1;             <3> Set 1 as decimal point position of customize data 2
G11;               <4> Cancel the setting mode
;
G10 L75 P1;        <5> Register tool management data
N01;               <6> Register with No. 1
P1 R12345;         <7> Set “12.345” for customize data 1
P2 R[#8431*1000];  <8> Set “1234.5” for customize data 2
G11;               <9> Cancel the setting mode
```

In Example 2, customize data 1 is directly set in customize data 2 by using a custom macro variable. Customize data 1 holds "12.345". In step <8>, the data is multiplied by 1000 to eliminate the fractional part.
So, this command is equivalent to "P2 R12345", so that "1234.5" is set in customize data 2.

10.3.4 Protection of Tool Information with the KEY Signal

When tool management data is in the edit state, information in it can be modified. By setting bit 0 of parameter No. 13204 to 1, tool management data is protected with the KEY signal so that information in it cannot be registered, modified, and deleted.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

10.3.5 Selection of a Tool Life Count Period

A tool life count period can be chosen either 1 sec or 8 msec tool-by-tool.

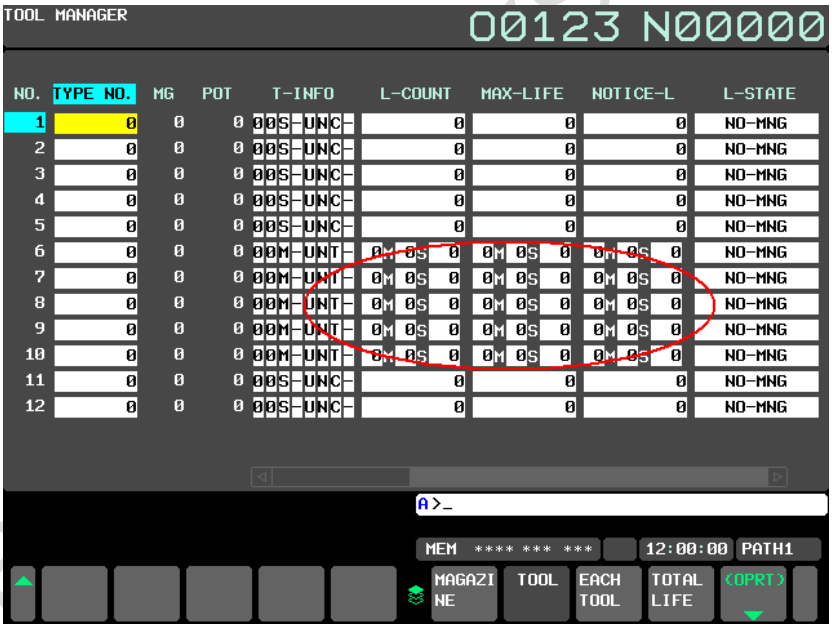
- Tool life count period selection

Bit 5 of tool information is used to select a life count period.

Item		Description
Data length		1 byte (flag data)
#5	REV	0: A life count period of 1 sec is used. (S) 1: A life count period of 8 msec is used. (M)

Range of count is as follows.
1sec: 0 to 3,599,999 seconds (999 hours 59 minutes 59 seconds)
8msec: 0 to 3,599,992 ms (59 minutes 59 seconds 992 milliseconds)

NOTE
This function is valid when the tool information TIM (#1) is set to 1.



10.3.6 Each tool Data Screen

All data for a specified tool can be extracted and displayed.

10.3.7 Total Life Time Display for Tools of The Same Type

The remaining lives of tools with the same type numbers are totaled, and totals are displayed in order by tool type number or by remaining life. Also, data of tools with the same tool type numbers are displayed in a list.

FRYER SYSTEMS INC

10.4 TOOL MANAGEMENT FUNCTION FOR OVERSIZE TOOLS

Overview

Tool management function for oversize tools is added to the tool management function.
The figure of an oversize tool can be defined freely, and the figure of each oversize tool is registered. When an oversize tool is stored in a cartridge, interference with tools stored in other pots is considered. This function is usable with cartridges of chain type and matrix type.

Format

By G10 data input, an oversize tool number can be set in the tool management data. Moreover, tool figure data can be set.

<Registration of new tool management data>

```
G10 L75 P1 ;
N_;
A_;
  G11 ;
  N_:    Tool management data number
  A_:    Specify tool figure number (0 to 20).
```

<Modification of tool management data>

```
G10 L75 P2;
N_;
A_;
G11 ;
  N_: Tool management data number
  A_:    Specify tool figure number (0 to 20).
```

<Registration of tool figure data>

```
G10 L77 P6 ;
N_P_Q_R_S_T_;
G11 ;
  N_: Tool figure number
  P_:    Number of pots to be occupied in left direction (in steps of 0.5 pot)
  Q_:    Number of pots to be occupied in right direction (in steps of 0.5 pot)
  R_:    Number of pots to be occupied in upper direction (in steps of 0.5 pot)
  S_:    Number of pots to be occupied in lower direction (in steps of 0.5 pot)
  T_:    0 for figure A, or 1 for figure B
```

NOTE

1. If a target tool is registered in a cartridge and interferes with other tools in registration or modification of tool figure data of the tool management data, alarm PS5360, “TOOL INTERFERENCE CHECK ERROR” is issued. (The data is not input.)
2. If a tool interferes with other tools in registration or modification to the cartridge management table, alarm PS5360 is issued. (The data is not input.)
3. If an attempt is made to modify tool figure data, and the tool for which the tool figure data number subject to modification is set is registered in the cartridge, alarm PS5360 is issued. (The data is not input.)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

10.5 TOOL LIFE MANAGEMENT

Tools are classified into several groups, and a tool life (use count or use duration) is specified for each group in advance. Each time a tool is used, its life is counted, and when the tool life expires, a new tool that is sequenced next within the same group is selected automatically. With this function, the tool life can be managed while machining is being performed continuously. Data for tool life management consists of tool group numbers, tool life values, tool numbers, and codes for specifying a tool offset value. These data items are registered in the CNC.

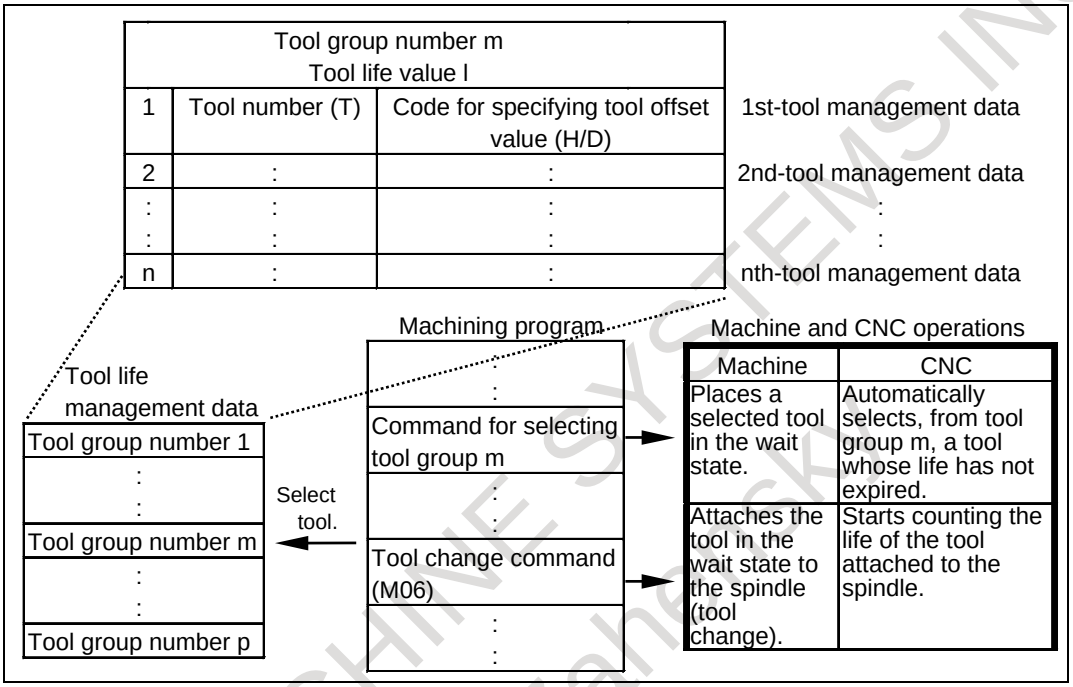


Fig. 10.5 (a) Tool selection from machining program

M

A group is selected by a T code, and tool life counting is started by the M06 command.

T

The T series has two tool change types (the turret type and the ATC type). The turret type uses only a T code to select a group, specify a tool offset value, and start tool life counting, whereas the ATC type, like the M series, uses a T code to select a group and the M06 command to start tool life counting. With the ATC type, only a D code is used for specifying a tool offset value. The tool change type is selected by bit 3 (TCT) of parameter No. 5040.

> ⚠ **CAUTION**
> This function cannot be used if bit 1 (LGN) of parameter No. 5002 is set to 1 to use the same number as the tool selection number to specify a geometric offset number.

- Life management B function

If the tool life management B function is enabled, the maximum tool life value can be extended, and the tool life expiration prior notice signal can be output to post tool life expiration in advance when the remaining life (the life value minus the life counter value) has reached the remaining life setting. The remaining life setting is registered as tool life management data in the CNC in advance.
The tool life management B function is enabled by setting bit 4 (LFB) of parameter No. 6805 to 1.

FRYER MACHINE SYSTEMS INC
Rob Zaharski

M

If the tool life management B function is enabled, the function for selecting a tool group by an arbitrary group number can be used.

T

The tool life management B function can be used. However, the function for selecting a tool group by an arbitrary group number can be used only if the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1).

- Maximum number of tool life management groups and multipath system

Up to 256 tool life management groups can be used in the entire CNC system.
For each path, set a maximum number of groups to be used in parameter No. 6813.
The maximum number of groups must be a multiple of the minimum number of groups (eight groups). If the maximum number of groups is 0, the tool life management function is disabled.
The total number of groups in the entire CNC system can be expanded to 1024 groups by using the additional tool life management groups.

Multipath system (when tool life management is performed for three paths)
Suppose that, of the total groups in the system (256 groups), 128 groups are allocated to path 1, 64 groups to path 2, and 64 groups to path 3. Then, set parameter No. 6813 as follows:

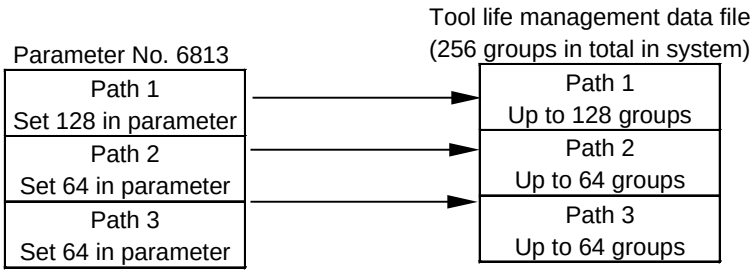


Fig. 10.5 (b) Group allocation in entire system

> ⚠ **CAUTION**
> Re-setting this parameter results in the tool life management data file being cleared at power-on. Once the parameter is re-set, register tool life management data for each path again.

- System variables

The following system variables for life data can be used in tool life management.

System variable number	System variable name	Attribute	Description
#181000	[#_TLMGN_C]	R	Tool life management data (Tool group numbers)
#181001	[#_TLMML_C]	R	Tool life management data (Tool life values)
#181002	[#_TLMLC_C]	R	Tool life management data (Life counter values)

Tool life values and life counter values are read by the following unit according to the life count type.

Life count type	Unit
Count	1-time
Duration	1-minute

NOTE

1. Zero is read from the system variables #181000 to #181002 when a group being used not exists.
2. Bit 0 (FCO) of parameter No.6805 not affect to the unit of tool life values and life counter values.
3. The information before change might be read from the system variables when the following states. The system variables should be used after passing 100ms from changing.
   - Immediate aftermath of changing a group being used by M06 (When T system with setting bit 3 (TCT) of parameter No.5040 to 1 or M system.)/T codes (When T system with setting bit 3 (TCT) of parameter No.5040 to 0).
   - Immediate aftermath of changing tool life management data by G10L3 command.
4. When the system variables #181000 to #181002 is specified, alarm PS0115 “VARIABLE NO. OUT OF RANGE” occurs if tool life management is invalid.
5. Real-time custom macro does not support the system variables #181000 to #181002. When the system variables #181000 to #181002 is specified in real-time custom macro, alarm PS0390 “ILLEGAL MACRO VAR” occurs.

10.5.1 Tool Life Management Data

Tool life management data consists of tool group numbers, tool numbers, codes for specifying a tool offset value, tool life values, arbitrary group numbers, and remaining life settings.
Whether to use arbitrary group numbers and remaining life settings can be specified by setting bit 5 (TGN) and bit 3 (GRP) of parameter No. 6802.

Explanation

- Tool group number

Up to 256 tool life management groups can be used in the entire CNC system.
In parameter No. 6813, set a maximum number of groups to be used. With the set maximum number of groups, up to four tools can be registered per group. The combination of the maximum number of registrable groups and the maximum number of tools can be changed by setting bits 0 (GS1) and bit 1 (GS2) of parameter No. 6800 as follows.

Table 10.5.1 Maximum numbers of registrable groups and tools

Bit 1 (GS2) of parameter No. 6800	Bit 0 (GS1) of parameter No. 6800	Number of groups	Number of tools
0	0	1/8 of maximum number of groups (parameter No. 6813)	32
0	1	1/4 of maximum number of groups (parameter No. 6813)	16
1	0	1/2 of maximum number of groups (parameter No. 6813)	8
1	1	Maximum number of groups (parameter No. 6813)	4

⚠ CAUTION

After changing the setting of bits 0 (GS1) and 1 (GS2) of parameter No. 6800, re-register tool life management data by issuing the G10L3 (registration after deletion of data for all groups).
Otherwise, the newly set combination does not become valid.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE
The total number of groups in the entire CNC system can be expanded to up to 1024 groups, using the additional tool life management groups.

- Tool number

A tool number is specified with a T code. A number consisting of up to eight digits (99999999) can be specified.

NOTE
The maximum number of digits usable in the T code is specified in parameter No. 3032.

- Codes for specifying a tool offset value

M

Codes for specifying a tool offset value include an H code (for tool length offset) and a D code (for cutter compensation). Numbers up to 999 (up to three digits long) can be registered as codes for specifying tool offset values. However, the number of registered codes must not exceed the number of tool offsets the CNC can use.

NOTE
If codes for specifying tool offset values are not used, the registration of these codes can be omitted.

T

If the tool change type is the turret type (bit 3 (TCT) of parameter No. 5040 = 0), neither the H code nor D code is used to specify a tool offset value. The T code includes a compensation code. If the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), only the D code is used.

- Tool life value

A tool life value can be registered in terms of use duration or use count. The maximum value is as follows:
Up to 4300 minutes can be registered if duration specification is selected, or up to 65535 times can be registered if use count specification is selected.
When the tool life management B function is enabled, up to 100000 minutes (or 60000 minutes if the life is counted every 0.1 second) can be registered if duration specification is selected, or up to 99999999 times can be registered if use count specification is selected.

- Arbitrary group number

M

If a function for allowing specification of arbitrary group numbers is used (bit 5 (TGN) of parameter No. 6802 = 1), an arbitrary group number can be specified with a T code to select a tool life management group.
An arbitrary group number ranging from 1 to 99999999 can be specified.

T

Arbitrary group numbers can be used only if the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1).

FRYER MACHINE SYSTEMS INC
Rob Zakrensky

- Remaining life setting

Bit 3 (GRP) of parameter No. 6802 is used to specify which value, a value set for each group or a parameter-set value (parameters Nos. 6844 and 6845), is to be used as the remaining life setting until a new tool is selected.
If a value set for each group is used (bit 3 (GRP) of parameter No. 6802 = 1), the value is registered as tool life management data.
A value ranging from 1 to 100000 minutes (or 60000 minutes if the life is counted every 0.1 second) can be specified if duration specification is selected, or a value up to 99999999 times can be specified if use count specification is selected.

10.5.2 Registering, Changing, and Deleting Tool Life Management Data

By programming, tool life management data can be registered in the CNC, and registered tool life management data can be changed or deleted.

Explanation

The program format varies depending on the following four types of operation:

- Registration after deletion of all groups

After all registered tool life management data is deleted, programmed tool life management data is registered.

- Change of tool life management data

Tool life management data can be set for a group for which no tool life management data is registered, and already registered tool life management data can be changed.

- Deletion of tool life management data

Tool life management data can be deleted.

- Setting of tool life count type

A count type (use duration or use count) can be set for each group separately.

Format

- Registration after deletion of all groups

M

Format	Meaning
G10L3; P-L-; T-H-D-; T-H-D-; : P-L-; T-H-D-; T-H-D-; : G11; M02(M30);	G10L3: Register data after deleting data of all groups. P-: Group number L-: Tool life value T-: Tool number H-: Code for specifying tool offset value (H code) D-: Code for specifying tool offset value (D code) G11: End of registration

FRYER MACHINE SYSTEMS INC
Rob Zahensky

T

Format	Meaning
G10L3; P-L-; T-(D-); T-(D-); : P-L-; T-(D-); T-(D-); : G11; M02(M30);	G10L3: Register data after deleting data of all groups. P-: Group number L-: Tool life value T-: For turret type (bit 3 (TCT) of parameter No. 5040 = 0), tool number and tool offset number For ATC type (bit 3 (TCT) of parameter No. 5040 = 1), tool number G11: End of registration (D-): For ATC type (bit 3 (TCT) of parameter No. 5040 = 1), code for specifying tool offset value

NOTE

No H code is used even if the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1).

If more than one offset value is to be used for the same tool within one process, specify the command as shown below (on the assumption that tool offset numbers are two digits long).

Format	Meaning
G10L3; P-L-; T0101; T0102; T0103; : G11; M02(M30);	 Tool number 01, tool offset number 01 Tool number 01, tool offset number 02 Tool number 01, tool offset number 03

- Change of tool life management data

M

Format	Meaning
G10L3P1; P-L-; T-H-D-; T-H-D-; : P-L-; T-H-D-; T-H-D-; : G11; M02(M30);	G10L3P1: Start changing group data. P-: Group number L-: Tool life value T-: Tool number H-: Code for specifying tool offset value (H code) D-: Code for specifying tool offset value (D code) G11: End of change of group

FRYER MACHINE SYSTEMS INC
Bob Zahensky

T

Format	Meaning
G10L3P1; P-L-; T-(D-); T-(D-); : P-L-; T-(D-); T-(D-); : G11; M02(M30);	G10L3P1: Start changing group data. P-: Group number L-: Tool life value T-: For turret type (bit 3 (TCT) of parameter No. 5040 = 0), tool number and tool offset number For ATC type (bit 3 (TCT) of parameter No. 5040 = 1), tool number G11: End of registration (D-): For ATC type (bit 3 (TCT) of parameter No. 5040 = 1), code for specifying tool offset value

NOTE
No H code is used even if the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1).

- Deletion of tool life management data

Format	Meaning
G10L3P2; P-; P-; P-; P-; : G11; M02(M30);	G10L3P2: Start deleting group data. P-: Group number G11: End of deletion

- Setting of tool life count type

Format	Meaning
G10L3 (or G10L3P1); P-L-Q-; T-H-D-; T-H-D-; : G11; M02(M30);	Q: Life count type (1: Use count. 2: Duration.)

⚠ **CAUTION**
If the Q command is omitted, the life count type is set according to the setting of bit 2 (LTM) of parameter No. 6800.

- Arbitrary group number

M

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1), and use of the function for specifying an arbitrary group number is set (bit 5 (TGN) of parameter No. 6802 = 1), an arbitrary group number can be registered by programming a command.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Format	Meaning
G10L3; (or G10L3P1); P-L-T-; T-H-D-; T-H-D-; : G11; M02(M30);	T-: Arbitrary group number (1 to 99999999)

NOTE
1 A group number that is the same as the group number of another group cannot be registered.
An attempt to register such a group number results in alarm PS0431, “ILLEGAL T/R DATA OF TOOL LIFE”.
2 If an arbitrary group number (T) is 0, alarm PS0431 is issued.
3 If ever an arbitrary group number is specified, the T command must be specified. If the T command is omitted, alarm PS0149, “FORMAT ERROR IN G10L3” is issued.

T

The function for specifying an arbitrary group number is available only if the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1).
If the tool change type is the ATC type, the format used for setting an arbitrary group number is the same as for the M series.

Remaining life setting

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1), and a remaining life value is to be set for each group separately (bit 3 (GRP) of parameter No. 6802 = 1), a remaining life value can be set as tool life management data by programming a command.

Format	Meaning
G10L3; (or G10L3P1); P-L-R-; T-H-D-; T-H-D-; : G11; M02(M30);	R-: Remaining life setting (remaining life until a new tool is selected)

NOTE
1 If the remaining life setting (R) is 0 or omitted, the remaining life is assumed to be 0. In this case, the tool life expiration prior notice function is disabled.
2 The remaining life setting (R) cannot exceed the life value (L). If an attempt is made to specify a value greater than the life value, alarm PS0431 is issued.
3 The remaining life setting (R) must be an unsigned value having no decimal point.
4 The unit of the remaining life setting (R) depends on the setting of bit 1 (FGL) of parameter No. 6805, which selects the unit of the tool life value.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Tool life value

- If the life management B function is disabled

If the tool life management B function is disabled (bit 4 (LFB) of parameter No. 6805 = 0), a tool life value is registered as a duration or a use count according to the setting of bit 2 (LTM) of parameter No. 6800 or the setting of a count type (Q command). The maximum values are listed below.

Table 10.5.2 (a) Life count types and maximum life values

Bit 4 (LFB) of parameter No. 6805	Bit 2 (LTM) of parameter No. 6800	Life count type	Maximum life value
0	0	Use count specification	65535 times
	1	Duration specification	4300 min

If the count type is duration specification, the unit of a life value specified with address L in a program may be one minute or 0.1 second, which is determined by the setting of bit 1 (FGL) of parameter No. 6805.

Table 10.5.2 (b) Life value unit and maximum value in L command

Bit 4 (LFB) of parameter No. 6805	Bit 1 (FGL) of parameter No. 6805	Life value unit	Maximum value in L command	Example
0	0	1 min	4300	L100: Life value is 100 min.
	1	0.1 sec	2580000	L1000: Life value is 100 sec.

- If the life management B function is enabled

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1)

Table 10.5.2 (c) Life count types and maximum life values

Bit 4 (LFB) of parameter No. 6805	Bit 2 (LTM) of parameter No. 6800	Life count type	Maximum life value
1	0	Use count specification	99999999 times
	1	Duration specification	100000 min
			60000 min(Note)

Table 10.5.2 (d) Life value unit and maximum value in L command

Bit 4 (LFB) of parameter No. 6805	Bit 1 (FGL) of parameter No. 6805	Life value unit	Maximum value in L command	Example
1	0	1 min	100000	L100: Life value is 100 min.
			60000(Note)	
	1	0.1 sec	60000000	L1000: Life value is 100 sec.
			36000000(Note)	

NOTE

If the life is counted every 0.1 second (bit 0 (FCO) of parameter No. 6805 = 1), the maximum value specified by duration is 60000 minutes.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

10.5.3 Tool Life Management Commands in Machining Program

Explanation

M

- **Commands**

The following commands are used for tool life management:

T○○○○○○○○;

Specifies a tool group number.
The tool life management function selects, from a specified group, a tool whose life has not expired, and outputs its T code signal.
In ○○○○○○○○, specify the sum of the tool life management ignore number specified in parameter No. 6810 and a desired group number.
Example:

To specify tool group number 1 when the tool life management ignore number is 100, specify "T101;".

> **NOTE**
> If ○○○○○○○○ is not greater than a tool life management ignore number, the T code is treated as an ordinary T code.

If an arbitrary group number is used (bit 4 (LFB) of parameter No. 6805 = 1, and bit 5 (TGN) of parameter No. 6802 = 1), it must be a preset arbitrary group number rather than a tool group number specified in ○○○○○○○○.
Example:

To specify tool group number 1 when arbitrary group number 1234 is set for tool group number 1, specify T1234.

> **NOTE**
> 1 If an arbitrary group number is used, the tool life management ignore number specified in parameter No. 6810 is not used.
> 2 If the group specified by ○○○○○○○○ is not present, the CNC assumes that a tool not controlled under the tool life management function is selected.

M06;

Terminates life management for the previously used tools, and begins counting the life of a new tool selected with the T code.

> **NOTE**
> 1 M06 is treated as an M code not involving buffering.
> 2 If multiple M codes are to be specified in the same block, M06 can be specified anywhere in the M codes.

H99;

Selects the H code registered in tool life management data for the currently used tool to enable tool length offset. Parameter No. 13265 can be used to enable compensation according to an H code other than H99.

H00;

Cancels tool length offset.

FRYER MACHINE SYSTEMS INC Rob Zanensky

D99;

Selects the D code registered in tool life management data for the currently used tool to perform cutter compensation.
Parameter No. 13266 can be used to enable compensation according to a D code other than D99.

D00;

Cancels cutter compensation.

> **NOTE**
> H99 and D99 must be specified after the M06 command. If a code other than H99 or D99 or a code other than the H or D code set, respectively, in parameter No. 13265 or 13266 is specified following M06, no H code or D code is selected for tool life management data.

- Types

For tool life management, the four tool change types (types A to D) listed below are used. Which type to use varies from one machine to another. For details, refer to the relevant manual of each machine tool builder.

Table 10.5.3 Differences among tool change types

Tool change type	A		B		C		D	
Parameters M6T and M6E M6T (No.6800#7) M6E (No.6801#7)	M6T	M6E	M6T	M6E	M6T	M6E	M6T	M6E
	0	0	1	0	1	0		1
Tool group number specified in the same block as the tool change command (M06)	Tool group already used		Tool group to be used next					
Time when tool life is counted	Life counting is performed for a tool in the specified tool group if M06 is specified next.						Life counting is performed if a tool in the tool group specified in the same block as M06 is specified.	
Remarks	If the T command (retract tool group) following the M06 command is not the currently used tool group, alarm PS0155, “ILLEGAL T-CODE COMMAND” is issued (if bit 6 (IGI) of parameter No. 6800 = 0).		Normally, if a tool group number command is specified alone, type B is used. However, even if a tool group number command is specified alone with type C, no alarm is issued. (This means that there is no difference in operation between types B and C.)				If M06 is specified alone, alarm PS0153, “T-CODE NOT FOUND” is raised.	

> **NOTE**
> If a tool group number is specified and a new tool is selected, the new tool selection signal is output.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

T

If the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), commands are specified in the same manner as for the M series except that neither H99 nor H00 is used for the T series. See the description for the M series. For details on the tool compensation types applied when the tool change type is the ATC type, refer to Section, "EXTENDED TOOL SELECTION" of the OPERATOR'S MANUAL (Lathe System) (B-64694EN-1).
The command specification method for the turret type (bit 3 (TCT) of parameter No. 5040 = 0) is explained below.

> ⚠ **CAUTION**
> This function cannot be used if the same number as the tool selection number is to be used to specify geometry offset number (bit 1 (LGN) of parameter No. 5002 = 1).

- Commands

TOOOOOO99;

The tool life management function ends counting the life of the tool used so far, selects, from the group specified by OOOOOO, a tool whose life has not expired, outputs the T code signal for the tool, and starts counting the life of the tool.

Example:

Suppose that a T199 command (with a tool offset specified with the lower two digits) is issued to cause the life management function to select T10001 of tool group 1. Then, T code 100 is output, and the tool offset number 1 is selected.

If the condition that a new tool be selected is not met, and the second or subsequent selection of the same group is made since the entry of the control unit into the automatic operation start state from the reset state, the next to the currently selected T code is selected if more than one offset is registered.
If the third selection is made, for example, the third offset is selected from among multiple offsets registered for the same tool.

Example:

Suppose that the tool offset is specified with the lower two digits, and that two offset numbers are set for the same tool number in group 1 with the following two T codes:

T10001 T10002

The first T199 command issued since the entry of the control unit into the automatic operation start state from the reset state selects the first T code, T10001. Then, if T199 is issued again before the control unit is reset, the second T code, T10002, is selected. Furthermore, if T199 is issued again before the control unit is reset, the second T code, T10002, is selected because no third offset is present.
Setting bit 1 (TSM) of parameter No. 6801 set to 1 enables life counting to be performed for each T code separately even if T codes specifying multiple offsets for the same number are registered.

TOOOOOO88;

The offset of the tool whose life is currently counted by tool life management is canceled. The tool offset code is set to 00, and the tool number is output as a T code signal.

Example:

Suppose that the tool number of the tool currently used by the life management function is 100. Then, issuing a T188 command (with a tool offset specified with the lower two digits) outputs T code 100 and selects offset number 0, canceling the offset.

FRYER MACHINE SYSTEMS INC
Prop Zarensky

NOTE
If life counting is not performed, or if the specified tool does not belong to the group for which life counting is being performed, alarm PS0155 is issued.

The numbers of digits in ○○○○○○ and 99/88 vary as follows:

No.5028	99	88
1	T○○○○○○○ 9 ↑ ↑ Select group Start life counting	T○○○○○○○ 8 ↑ ↑ Select group Cancel tool offset
2	T○○○○○○ 99 ↑ ↑ Select group Start life counting	T○○○○○○ 88 ↑ ↑ Select group Cancel tool offset
3	T○○○○○ 999 ↑ ↑ Select group Start life counting	T○○○○○ 888 ↑ ↑ Select group Cancel tool offset

The maximum number of digits in T codes is set in parameter No. 3032.
The number of digits used for specifying an offset number is selected by parameter No. 5028. If 0 is selected, the number of digits depends on the number of tool offsets.
Example:

If there are one to nine tool offsets:	Lowest digit
If there are 10 to 99 tool offsets:	Lower two digits
If there are 100 to 999 tool offsets:	Lower three digits

NOTE
Offset start and cancel operations involve compensation by moving a tool or by shifting the coordinate system. Using bit 6 (LWM) of parameter No. 5002 can select whether to perform a compensation operation if a T code is specified or if an axis move command is specified.
For details, refer to refer to Subsection, “Offset” of the OPERATOR’S MANUAL (Lathe System) (B-64694EN-1).

T○○○○○○ΔΔ;

If the tool offset number in ΔΔ is neither 99 nor 88, the T code is treated as an ordinary T code. If life counting is being currently performed, it is ended.

Examples

M

- Tool change type A

If a block specifying a tool change command (M06) also contains a tool group command (T code), the T code is used as a command for returning the tool to its cartridge. By specifying a tool group number with a T code, the number of the tool used so far is output as a T code signal. If the specified tool number is not a tool number of the tool group of the tool being used, alarm PS0155 is issued. The alarm, however, can be suppressed by setting bit 6 (IGI) of parameter No. 6800 to 1.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

Example: Suppose that the tool life management ignore number is 100.	
T101 ;	A tool whose life has not expired is selected from group 1.
:	(Suppose that tool number 010 is selected.)
M06 ;	Tool life counting is performed for the tool in group 1.
:	(The life of tool number 010 is counted.)
T102 ;	A tool whose life has not expired is selected from group 2.
:	(Suppose that tool number 100 is selected.)
M06 T101 ;	Tool life counting is performed for the tool in group 2.
:	(The life of tool number 100 is counted.)
:	The number of the tool currently used (in group 1) is output with a T code signal. (Tool
:	number 010 is output.)
T103 ;	A tool whose life has not expired is selected from group 3.
:	(Suppose that tool number 200 is selected.)
M06 T102 ;	Tool life counting is performed for the tool in group 3.
:	(The life of tool number 200 is counted.)
G43 H99 ;	Tool length offset for the tool selected from group 3 is used.
:	
G41 D99 ;	Cutter compensation for the tool selected from group 3 is used.
:	
D00 ;	Cutter compensation is canceled.
:	
H00 ;	Tool length offset is canceled.

- Tool change types B and C

If a block specifying a tool change command (M06) also contains a tool group command (T code), the T code is used to specify a tool group number for which life counting is to be performed by the next tool change command.

Example: Suppose that the tool life management ignore number is 100.	
T101 ;	A tool whose life has not expired is selected from group 1.
:	(Suppose that tool number 010 is selected.)
M06 T102 ;	Tool life counting is performed for the tool in group 1.
:	(The life of tool number 010 is counted.)
:	A tool whose life has not expired is selected from group 2.
:	(Suppose that tool number 100 is selected.)
M06 T103 ;	Tool life counting is performed for the tool in group 2.
:	(The life of tool number 100 is counted.)
:	A tool whose life has not expired is selected from group 3.
:	(Suppose that tool number 200 is selected.)
G43 H99 ;	Tool length offset for the tool selected from group 2 is used.
:	
G41 D99 ;	Cutter compensation for the tool selected from group 2 is used.
:	
D00 ;	Cutter compensation is canceled.
:	
H00 ;	Tool length offset is canceled.
:	
M06 T104 ;	Tool life counting is performed for the tool in group 3.
:	(The life of tool number 200 is counted.)
	A tool whose life has not expired is selected from group 4.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Tool change type D

For a tool selected by a tool group command (T code), life counting is performed by a tool change command (M06) specified in the same block as the tool group command. Specifying a T code alone does not results in an alarm; however, specifying an M06 command alone results in alarm PS0153. However, the alarm can be suppressed by setting bit 7 (TAD) of parameter No. 6805 to 1.

Example: Suppose that the tool life management ignore number is 100.	
T101 M06 ;	A tool whose life has not expired is selected from group 1.
:	(Suppose that tool number 010 is selected.)
:	Tool life counting is performed for the tool in group 1.
:	(The life of tool number 010 is counted.)
T102 M06 ;	A tool whose life has not expired is selected from group 2.
:	(Suppose that tool number 100 is selected.)
:	Tool life counting is performed for the tool in group 2.
:	(The life of tool number 100 is counted.)
G43 H99 ;	Tool length offset for the tool selected from group 2 is used.
:	
G41 D99 ;	Cutter compensation for the tool selected from group 2 is used.
:	
D00 ;	Cutter compensation is canceled.
:	
H00 ;	Tool length offset is canceled.
:	
T103 M06 ;	A tool whose life has not expired is selected from group 3.
:	Tool life counting is performed for the tool in group 3.

T

For the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), commands are specified in the same manner as for the M series except that D99 rather than H99 is used. For the ATC type, see the description for the M series.
The following shows how commands are specified if the turret type is used (bit 3 (TCT) of parameter No. 5040 = 0):

Example: Suppose that offset numbers are two digits long.	
T0199 ;	A tool whose life has not expired is selected from group 1.
:	(Suppose that T1001 is selected. The tool number is 10, and the offset number is 01.)
:	
:	Tool life counting is performed for the tool in group 1.
:	(The life of tool number 10 is counted.)
:	
T0188 ;	The offset of the tool being used in group 1 is canceled.
:	(Since the tool being used is T1001, the tool number is 10, and the offset number is 00.)
:	
T0299 ;	A tool whose life has not expired is selected from group 2.
:	(Suppose that T2002 is selected. The tool number is 20, and the offset number is 02.)
:	Tool life counting is performed for the tool in group 2.
:	(The life of tool number 20 is counted.)
:	
T0299 ;	If more than one offset number is specified for the currently used tool in group 2, the next offset number is selected.
:	(Suppose that T2002 and T2003 are registered with tool number 20. In this case, T2003 is selected. The tool number is 20, and the offset number is 03.)
:	
:	
T0301 ;	Life counting for the tool in group 2 ends, and this command is treated as an ordinary T code.
:	(The tool number is 03, and the offset number is 01.)

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

10.5.4 Tool Life Counting and Tool Selection

Either use count specification or duration specification is selected as the tool life count type according to the state of bit 2 (LTM) of parameter No. 6800. Life counting is performed for each group separately, and the life counter contents are preserved even after the power is turned off.

Table 10.5.4 (a) Tool life management count types and intervals

Tool life count type	Use count specification	Time specification
Bit 2 (LTM) of parameter No. 6800	0	1
Life count interval	Incremented by one for tools used in one program Counting can be resumed by using tool life count restart M code (parameter No. 6811).	Bit 0 (FCO) of parameter No. 6805 0: Every second 1: Every 0.1 second Can be changed by override.

Explanation

M

- Use count specification (LTM=0)

If a tool group (T code) is specified, a tool whose life has not expired is selected from the specified tool group. The life counter for the selected tool is then incremented by one by a tool change command (M06). Unless a tool life count restart M code is specified, selection of a new tool and the increment operation can be performed only if a tool group number command and a tool change command are issued for the first time since the entry of the control unit into the automatic operation start state from the reset state.

> ⚠ **CAUTION**
> No matter how many times the same tool group number is specified in a program, the use count is not incremented, and no new tool is selected.

- Duration specification (LTM=1)

After all registered tool life management data is deleted, programmed tool life management data is registered.
If a tool group command (T code) is specified, a tool whose life has not expired is selected from the specified tool group. Then, life management for the selected tool is started by a tool change command (M06). Life management (counting) is performed by measuring the time during which the tool is actually used in the cutting mode at regular intervals (every second or 0.1 second). The life count interval is specified by bit 0 (FCO) of parameter No. 6805. The time required for single block stop, feed hold, rapid traverse, dwell, machine lock, and interlock operations is not counted.
Setting bit 2 (LFV) of parameter No. 6801 enables the life count to be overridden as directed by tool life count override signals. An override from 0 times to 99.9 times can be applied. If 0 times is specified, counting is not performed.

T

If the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), the same specification method as for the M series applies. See the description for the M series.
The following explains the specification method for the turret type (bit 3 (TCT) of parameter No. 5040 = 0):

FRYER MACHINE SYSTEMS INC Rob Zaremsky

- Use count specification (LTM=0)

If a tool group command (T○○99 code) is issued, a tool whose life has not expired is selected from the specified tool group, and the life counter for the selected tool is incremented by one. Unless a tool life count restart M code is specified, selection of a new tool and the increment operation can be performed only if a tool group number command and a tool change command are issued for the first time since the entry of the control unit into the automatic operation start state from the reset state.

> ⚠ **CAUTION**
> No matter how many times the same tool group number is specified in a program, the use count is not incremented, and no new tool is selected.

- Duration specification (LTM=1)

If a tool group command (T○○99 code) is specified, a tool whose life has not expired is selected from the specified tool group, and tool management for the selected tool starts.
Life management (counting) is performed by measuring the time during which the tool is actually used in the cutting mode at regular intervals (every second or 0.1 second). The life count interval is specified by bit 0 (FCO) of parameter No. 6805. The time required for single block stop, feed hold, rapid traverse, dwell, machine lock, and interlock operations is not counted.
Setting bit 2 (LFV) of parameter No. 6801 enables the life count to be overridden according tool life count override signals. An override from 0 times to 99.9 times can be applied. If 0 times is specified, counting is not performed.

M	
T	

> **NOTE**
> 1 When a tool is selected, tools are searched starting from the current tool toward the last tool to find a tool whose life has not expired. When the last tool is reached during this search, the search restarts from the first tool. If the search does find any tool whose life has not expired, the last tool is selected.
> When the tool currently used is changed by the tool skip signal, the next new tool is selected using the method described here.
> 2 If tool life counting indicates that the life of the last tool in a group has expired, the tool change signal is output. If the life count type is duration specification, the signal is output as soon as the life of the last tool in the group has expired. If the life count type is use count specification, the signal is output when the CNC is reset by a command such as M02 or M30 or when the tool life count restart M code is specified after the life of the last tool in the group has expired.
> 3 If a T command is specified, a group and a tool in the group are selected while the T command is buffered. This means that if a block to be buffered contains a T command specifying a group while machining is being performed with that group selected, the next T command is already buffered even if the tool life expires during machining, so the next tool is not selected. To prevent this, if the life count type is duration specification and a T command is to be specified to select the same group successively, insert an M code for suppressing buffering into a place immediately before the T command.
> 4 When the tool life is counted, the remaining life of a group (the life value minus the life counter value) is compared with the remaining life setting, and the status of the tool life expiration prior notice signal is changed according to the result of the comparison.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- M99

If the life count is specified by use count and bit 0 (T99) of parameter No. 6802 is 1, the tool change signal TLCH is output and the automatic operation is stopped if the life of at least one tool group has expired when the M99 command is executed. If the life count type is duration specification, the tool change signal is output immediately when the life of at least one tool group has expired; when the M99 command is specified, the automatic operation is stopped, but no more tool change signal is output.

M

If the life count is specified by use count, a tool group command (T code) issued after the M99 command selects, from a specified group, a tool whose life has not expired, and the next tool change command (M06) increments the tool life counter by one.

T

If the life count is specified by use count, when a tool group command (T code) is specified after the M99 command is specified, a tool whose life has not expired is selected from a specified group, and the tool life counter is incremented by one.
If the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), the same specifications as for the M series apply.

10.5.5 Tool Life Count Restart M Code

Explanation

M

If the life count is specified by use count, the tool change signal is output if the life of at least one tool group has expired when a tool life count restart M code is issued. A tool group command (T code) issued after the tool life count restart M code selects, from a specified group, a tool whose life has not expired, and the next tool change command (M06) increments the tool life counter by one. This enables the tool life to be counted by a tool change command (M06) even if the command is not the first tool change command (M06) issued since the entry of the CNC into the automatic operation start state from the reset state. The tool life count restart M code is specified in parameter No. 6811.

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

```
Example:
    Suppose that M16 is a tool life count restart M code and that the tool life
    management ignore number is 100.
    Also suppose that the life count is specified by use count.
    T101 ;   A tool whose life has not expired is selected from group 1.
       :
       :
    M06 ;    Tool life management is performed for group 1.
       :     (The tool life counter is incremented by one.)
       :
    T102 ;   A tool whose life has not expired is selected from group 2.
       :
       :
    M06 ;    Tool life management is performed for group 2.
       :     (The tool life counter is incremented by one.)
       :
    M16 ;    Tool life counting is restarted.
    T101 ;   A tool whose life has not expired is selected from group 1.
       :
       :
    M06 ;    Tool life management is performed for group 1.
       :     (The tool life counter is incremented by one.)
```

T

If the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), the same specifications as for the M series apply. See the description for the M series.

The following explanation assumes that the tool change type is the turret type (bit 3 (TCT) of parameter No. 5040 = 0).

If the life count is specified by use count, the tool change signal is output if the life of at least one tool group has expired when a tool life count restart M code is specified. A tool group command (T code) issued after the tool life count restart M code selects a tool whose life has not expired from a specified group, and the tool life counter is incremented by one. This enables the tool life to be counted by a tool group command (T code) even if the command is not the first tool group command issued since the entry of the CNC into the automatic operation start state from the reset state. The tool life count restart M code is specified in parameter No. 6811.

```
Example:
    Suppose that M16 is a tool life count restart M code.
    Also suppose that the life count is specified by use count.
    T199 ;   A tool whose life has not expired is selected from group 1.
       :     Tool life management is performed for group 1.
       :     (The tool life counter is incremented by one.)
       :
    T299 ;   A tool whose life has not expired is selected from group 2.
       :     Tool life management is performed for group 2.
       :     (The tool life counter is incremented by one.)
       :
    M16 ;    Tool life counting is restarted.
    T199 ;   A tool whose life has not expired is selected from group 1.
       :     Tool life management is performed for group 1.
       :     (The tool life counter is incremented by one.)
```

FRYER MACHINE SYSTEMS INC
Rob Zahonsky

M

T

NOTE

1 The tool life count restart M code is treated as an M code not involved in buffering.
2 If the life count type is use count specification, the tool change signal is output if the life of at least one tool group has expired when the tool life count restart M code is specified. If the life count type is duration specification, specifying the tool life count restart M code causes nothing.
3 If multiple M codes are to be specified in the same block, the tool life count restart M code can be specified anywhere in the M codes.

10.5.6 Disabling Life Count

Explanation

M

T

If bit 6 (LFI) of parameter No. 6804 is 1, the tool life count disable signal LFCIV can be used to select whether to cancel the tool life count.

If the tool life count disable signal LFCIV is “1”, the tool life count disabled signal LFCIF becomes “1”, and the tool life count is disabled.
If the tool life count disable signal LFCIV is “0”, the tool life count disabled signal LFCIF becomes “0”, and the tool life count is enabled.

NOTE

No buffering should occur when the state of the tool life count disable signal LFCIV is changed. So, use, for example, M codes not involved in buffering to change the signal state. If M06 (for the M series) or a tool change T code (for the T series) is issued in a block that follows directly a block in which an auxiliary function code with buffering enabled is used to turn on or off the tool life count disable signal LFCIV, it is likely that a command for specifying whether to count may become incorrect.

10.5.7 Remaining Tool Number Check Function

Overview

M

In tool life management, this function outputs the remaining tool number notice signal if the number of remaining tools in the group selected with a T code command is equal to or less than the value set in parameter No. 6846.

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

Explanation

M

If, in tool life management, the remaining tool number in the group selected with a T code command is equal to or less than the value set in parameter No. 6846, the remaining tool number notice signal TLAL becomes "1".
If the setting of parameter No. 6846 is 0, the remaining tool number notice signal TLAL is not output.

The remaining tool number notice signal TLAL becomes "0" when:
- A value is input to parameter No. 6846.
- Using a G10 command, registration is performed that involves the erasure of all life management data groups.
- Using a T code command, a group is selected in which the remaining tool number is greater than the setting of parameter No. 6846.

The remaining tool number notice signal TLAL becomes "0" when any of the following is executed in the group that has caused the remaining tool number notice signal TLAL to be generated.
- Clearing of execution data, using the tool change reset signal.
- Change and deletion of tool life management data, using a G10 command.
- Batch deletion of tool groups from the group edit screen of tool life management, addition of tool data to the screen, and deletion from the screen.
- Clearing of tool execution data from the group edit screen of tool life management.
- Clearing of execution data from the list screen of tool life management.
- Execution of any of the FOCAS2 functions given in Table 10.5.7(a).

Table 10.5.7(a) FOCAS2 functions that cause TLAL to become "0"

Function name	Description
cnc_clrcntinfo	Clearing of tool life management data (life counter and tool information) (range specification)
cnc_deltlifegrp	Deletion of tool life management data (tool group)
cnc_deltlifedt	Deletion of tool life management data (tool data) (Effective if the group contains only one tool.)
cnc_instlifedt	Addition of tool life management data (tool data)
cnc_wr1tlifedata	Writing of tool life management data (tool data) (Effective if tool information is changed to "expired" or if tool information is changed to "registered" for a tool that is "skipped".)
cnc_wr1tlifedat2	
cnc_wr2tlifedata	

FRYER MACHINE SYSTEMS INC
Rob Zaborsky

11 AUXILIARY FUNCTION

Overview

There are two types of auxiliary functions; auxiliary function (M code) for specifying spindle start, spindle stop, program end, and so on, and secondary auxiliary function (B code) for specifying index table positioning.

When a move command and auxiliary function are specified in the same block, the commands are executed in one of the following two ways:

(1) Simultaneous execution of the move command and auxiliary function commands.

(2) Executing auxiliary function commands upon completion of move command execution.

The selection of either sequence depends on the machine tool builder's specification. Refer to the manual issued by the machine tool builder for details.

11.1 AUXILIARY FUNCTION (M FUNCTION)

When a numeral is specified following address M, code signal and a strobe signal are sent to the machine. The machine uses these signals to turn on or off its functions.

Usually, only one M code can be specified in one block. Depending on the setting of bit 7 (M3B) of parameter No. 3404, up to three M codes can be specified. And, depending on the setting of bit 5 (M5B) of parameter No. 11630, up to five M codes can be specified.

Which M code corresponds to which machine function is determined by the machine tool builder. The machine processes all operations specified by M codes except those specified by M98, M99, M198 or called subprogram (parameters Nos.6071 to 6079), or called custom macro (parameters Nos.6080 to 6089). Refer to the machine tool builder's instruction manual for details.

Explanation

The following M codes have special meanings.

- M02, M30 (End of program)

This indicates the end of the main program.

Automatic operation is stopped and the CNC unit is reset. (This differs with the machine tool builder.)

After a block specifying the end of the program is executed, control returns to the start of the program.

Bits 5 (M02) and 4 (M30) of parameter No. 3404 can be used to disable M02, M30 from returning control to the start of the program.

- M00 (Program stop)

Automatic operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged. The automatic operation can be restarted by actuating the cycle operation. (This differs with the machine tool builder.)

- M01 (Optional stop)

Similarly to M00, automatic operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel has been pressed.

- M98 (Calling of subprogram)

This code is used to call a subprogram. The code and strobe signals are not sent. Refer to the Section, “SUBPROGRAM (M98, M99)” for details.

- M99 (End of subprogram)

This code indicates the end of a subprogram.

M99 execution returns control to the main program. The code and strobe signals are not sent. Refer to the Section, “SUBPROGRAM (M98, M99)” for details.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

11.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK

Usually, only one M code can be specified in one block. By setting bit 7 (M3B) of parameter No. 3404 to 1, however, up to three M codes can be specified simultaneously in one block. And, by setting bit 5 (M5B) of parameter No. 11630, to 1, however, up to five M codes can be specified simultaneously in one block.

Up to five M codes specified in one block are output to the machine at the same time. So, when compared with a case where a single M code is specified in one block, a reduced machining cycle time can be achieved.

Explanation

CNC allows up to five M codes to be specified in one block. However, some M codes cannot be specified at the same time due to mechanical operation restrictions. For detailed information about the mechanical operation restrictions on simultaneous specification of multiple M codes in one block, refer to the manual of each machine tool builder.
M00, M01, M02, M30, M98, M99, or M198 must not be specified together with another M code.
Some M codes other than M00, M01, M02, M30, M98, M99, and M198 cannot be specified together with other M codes; each of those M codes must be specified in a single block.
Such M codes include these which direct the CNC to perform internal operations in addition to sending the M codes themselves to the machine. To be specified, such M codes are M codes for calling program numbers 9001 to 9009 and M codes for disabling advance reading (buffering) of subsequent blocks. Meanwhile, multiple of M codes that direct the CNC only to send the M codes themselves (without performing internal operations) can be specified in a single block.
However, it is possible to specify multiple M codes that are sent to the machine in the same block unless they direct the CNC to perform internal operations. (Since the processing method depends on the machine, refer to the manual of the machine tool builder.)
If bit 0 (MMB) of parameter No. 11658 is set to 1, M code preventing buffering specified by any of parameters Nos. 3411 to 3420, parameters Nos. 3421 to 3432 or parameters Nos. 11290 to 11299 can be specified together with another M code.

Example

One M command in a single block	Multiple M commands in a single block
M40; M50; M60; M70; M80; G28G91X0Y0Z0; : : :	M40M50M60M70M80; G28G91X0Y0Z0; : : : : :

FRYER MACHINE SYSTEMS INC
Rob Zahenskiy

11.3 M CODE GROUPING FUNCTION

Overview

Classifying a maximum of 500 M codes into a maximum of 127 groups allows the user:

- To receive an alarm if an M code that must be specified alone is included when multiple M codes are specified in a block.
- To receive an alarm if M codes belonging to the same group are specified in the same block when multiple M codes are specified in a block.

11.3.1 Setting an M Code Group Number Using the Setting Screen

- Procedure for displaying the M code group setting screen

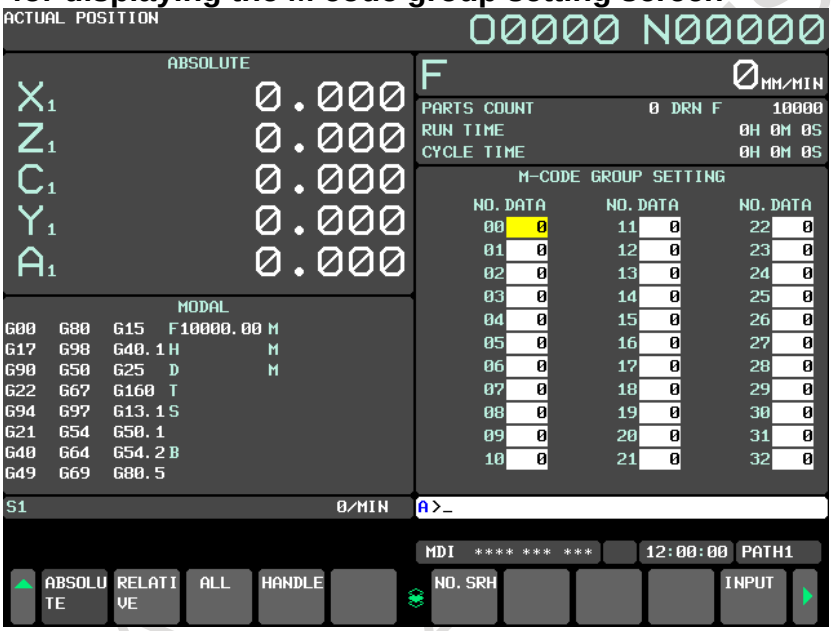


Fig. 11.3.1 M code group setting screen

You can use the “M code group setting screen (Fig. 11.3.1)” to set a group number for each M code.

Display the “M code group setting screen” using the following procedure:

(1) Press function key SYSTEM and the continuous menu key several times. Soft key [M CODE GROUP] appears.

(2) Press soft key [M CODE GROUP].

In the “NO.” field, M codes for which an M code group can be set are displayed.
An M code group can be set for the following M codes: M00 to M99, and any 400 M codes selected from M100 and subsequent M codes. For details of how to add the 100th and subsequent M codes, see the explanation of parameters Nos. 3441 to 3444.

In the “DATA” field, the M code group number corresponding to each M code is displayed.

- Setting a group number

To set an M code group number on the “M code group setting screen (Fig. 11.3.1),” use he following procedure:

1 Select the MDI mode.
2 Set “PARAMETER WRITE” on the setting screen to 1.
3 Display the “M code group setting screen”.
4 Move the cursor to the M code to be set using page keys and cursor keys. You can also enter the number of the M code to be set and press soft key [NO.SRH] to move the cursor to the M code.
5 Enter a group number and press soft key [INPUT] or the INPUT key.

The valid range of M code group numbers is from 1 to 127 (127 groups). If a value of 0 is input, it is not registered as an M code group.

- Examples of setting parameters Nos. 3441 to 3444

In the following examples, the number of digits of an M code is 4.
<1> to <4> indicate parameters Nos. 3441 to 3444.

(1) When <1> = 300, <2> = 400, <3> = 500, and <4> = 900 are set

Number
0000
:
0099 } 100 codes
0300
:
0399 } 100 codes
0400
:
0499 } 100 codes
0500
:
0599 } 100 codes
0900
:
0999 } 100 codes

M code groups can be set for M0000 to M0099, M0300 to M0599, and M0900 to M999.
M codes M0300 to M0599 and M0900 to M999 are added to the M code group setting screen.

(2) When <1> = 200, <2> = 0, <3> = 550, and <4> = 800 are set

Number
0000
:
0099
0200
:
0299
0550
:
0649
0800
:
0899

M code groups can be set for M0000 to M0099, M0200 to M0299, M0550 to M0649, and M0800 to M0899.
(The setting of parameter <2> is invalid because it is 0.)
In this case, M codes M0200 to M0299, M0550 to M0649, and M0800 to M0899 are added to the M code group setting screen.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

11.3.2 Setting an M Code Group Number Using a Program

You can execute a program to set an M code group number and M code name. The command format is shown below.

Format

G10 L40 Pn Rg ;

Pn: “n” specifies an M code.
Rg: “g” specifies an M code group number.

NOTE

1 If the format is invalid, an alarm PS1144, “G10 FORMAT ERROR” is issued.
2 If an M code group cannot be set for the M code specified for the P command or if the group number specified for the R command is not within the range between 0 and 127, an alarm PS1305, “DATA OUT OF RANGE” is issued.

(Example)

Executing the following program sets M code group “1” to M03:

```
G10 L40 P03 R1 ;
```

11.3.3 M Code Group Check Function

When multiple M commands in a single block (enabled when bit 7 (M3B) of parameter No. 3404 is set to 1) are used, you can check the following items. You can also select whether to check the items using bit 1 (MGC) of parameter No. 3400.

(1) M code to be specified in a single block containing no other M codes
If an M code which must be specified in a single block containing no other M codes is specified together with another M code, an alarm PS5016, “ILLEGAL COMBINATION OF M CODES” is issued.
(2) M codes in the same group
If multiple M codes in the same group are specified together, an alarm PS5016 is issued.

The valid range of group numbers is from 0 to 127 (128 groups).
Group numbers 0 and 1 have special meaning. Note the following points:

- Each M code with group number 1 is assumed to be an M code to be specified in a single block containing no other M codes.
- For each M code with group number 0, the “same group M code check” is ignored. That is, when multiple M codes with group number 0 are specified in a single block, the alarm is not issued.
- For each M code with group number 0, the “check for an M code to be specified in a single block containing no other M codes” is not ignored. That is, if an M code with group number 1 and an M code with group number 0 are specified in a single block, the alarm is issued.
- For M codes that are not to be output to the machine such as M98, M99, M198, and M codes for subprogram and macro calls (set in parameters Nos. 6071 to 6079 and Nos. 6080 to 6089 and with the macro executor), be sure to set 0 as the group number.
- For M00, M01, M02, M30, and M codes for which buffering is suppressed (set in parameters Nos. 3411 to 3432 and No.11290 to 11299), be sure to set 1 as the group number.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

11.4 SECOND AUXILIARY FUNCTIONS (B CODES)

Overview

If a value with a maximum of eight digits is specified after address B, the code signal and strobe signal are transferred for calculation of the rotation axis. The code signal is retained until the next B code is specified.
Only one B code can be specified for each block. When the maximum number of digits are specified by parameter No.3033, an alarm is issued if the number of digits of a command exceeds the specified number.
In addition, the address used for specifying the second auxiliary function can be changed to an address other than address B (address A, C, U, V, or W) by setting parameter No.3460.
However, the address used for the second auxiliary function cannot also be used as the address of the controlled axis. For details, refer to the manual available from the machine tool builder.

Explanation

- Range of specification

-99999999 to 99999999

- Output value

The value specified after the address of the second auxiliary function is output on the code signals B00 to B31. Note the following about a output value.

1. When a command with a decimal point or a negative command is disabled
   (When bit 0 (AUP) of parameter No.3450 is set to 0)

   When the second auxiliary function with no decimal point is specified, the specified value is output on the code signals as is, regardless of the desktop calculator decimal point setting (bit 0 (DPI) of parameter No.3401).
   Example:

Specified value	Output value
B10	10

   When the second auxiliary function with a decimal point is specified, alarm PS0007, “ILLEGAL USE OF DECIMAL POINT” is issued.
   When the second auxiliary function is specified with a negative value, alarm PS0006, “ILLEGAL USE OF MINUS SIGN” is issued.

2. When a command with a decimal point or a negative command is enabled
   (When bit 0 (AUP) of parameter No.3450 is set to 1)

   When the desktop calculator decimal point setting is not specified (when bit 0 (DPI) of parameter No.3401 is set to 0), if the second auxiliary function with no decimal point is specified, the specified value is output on the code signals as is.
   Example:

Specified value	Output value
B10	10

   When desktop calculator decimal point input is specified (when bit 0 (DPI) of parameter No.3401 is set to 1), if the second auxiliary function with no decimal point is specified, the specified value multiplied by a magnification is output on the code signals. (Magnifications are shown in Table 11.4 (a).)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Example:

Specified value	Output value
B10	10000 (When metric input is used and the reference axis is IS-B. The magnification is 1000.)

When the second auxiliary function with a decimal point is specified, the specified value multiplied by a magnification is output to the code signals. (Magnifications are shown in Table 11.4 (a).)

Example:

Specified value	Output value
B10.	10000 (When metric input is used and the reference axis is IS-B. The magnification is 1000.)
B0.123	1230 (When inch input is used, the reference axis is IS-B, and parameter AUX is set to 1. The magnification is 10000.)

The magnification is determined as shown below according to the setting unit of the reference axis (specified by parameter No.1031) and bit 0 (AUX) of parameter No.3405.

Table 11.4 (a) Magnifications for an output value when the second auxiliary function with a decimal point is specified for desktop calculator decimal point input

Setting unit		Bit 0 (AUX) of parameter No.3405 = 0	Bit 0 (AUX) of parameter No.3405 = 1
Metric input system	Reference axis: IS-A	100×	100×
	Reference axis: IS-B	1000×	1000×
	Reference axis: IS-C	10000×	10000×
Inch input system	Reference axis: IS-A	100×	1000×
	Reference axis: IS-B	1000×	10000×
	Reference axis: IS-C	10000×	100000×

⚠ **CAUTION**

If a decimal fraction remains after multiplying the specified value with a decimal point by a magnitude in Table 11.4 (a), the fraction is truncated.

Example:

Specified value	Output value
B0.12345	1234 (When inch input is used, the reference axis is IS-B, and parameter AUX is set to 1. The magnification is 10000.)

NOTE

If the number of digits of the specified value exceeds the allowable number of digits (set by parameter No. 3033), alarm PS0003, “TOO MANY DIGIT” is issued.

When the specified value is multiplied by a magnitude in Table 11.4 (a), the allowable number of digits must be set for the resultant value.

Limitation

Addresses used for the second auxiliary functions (addresses specified with B or parameter No. 3460) cannot be used as the addresses used for controlled axis names.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

11.5 AUXILIARY FUNCTION OUTPUT IN MOVING AXIS

Overview

By specifying absolute coordinate values and auxiliary functions (M, B) at G50.9 block, the auxiliary functions are output to PMC when absolute coordinate enters the specified area in the next movement block.

G50.9 can be specified up to 2 blocks consecutively. In other words, auxiliary function output points in a movement block can be specified up to two.

Code signals and strobe signals are output to the same signal address as a usual auxiliary function.

Format

G50.9 IP_ Mm11 Mm12 Mm13 Bb11 ;

G50.9 IP_ Mm21 Mm22 Mm23 Bb21 ;

(Movement command block) Mm01 Mm02 Mm03 Bb01 Ss01 Tt01 ;

IP_ : Auxiliary function output start point. (Absolute coordinate value)

M m11 : 1st M auxiliary function (1st block)
M m12 : 2nd M auxiliary function (1st block)
M m13 : 3rd M auxiliary function (1st block)
B b11 : 2nd auxiliary function (1st block)

M m21 : 1st M auxiliary function (2nd block)
M m22 : 2nd M auxiliary function (2nd block)
M m23 : 3rd M auxiliary function (2nd block)
B b21 : 2nd auxiliary function (2nd block)

M m01 : 1st M auxiliary function (movement block)
M m02 : 2nd M auxiliary function (movement block)
M m03 : 3rd M auxiliary function (movement block)
B b01 : 2nd auxiliary function (movement block)
S s01 : Spindle speed command
T t01 : T command (Tool compensation)

- At least one axis of auxiliary function output start point must be commanded at G50.9 block. The alarm (PS5330)," G50.9 FORMAT ERROR" occurs if there is not an axis address at G50.9 block.
- At least one auxiliary function must be commanded at G50.9 block. The alarm (PS5330)," G50.9 FORMAT ERROR" occurs if an auxiliary function isn't commanded.
- It is possible to command G50.9 block up to 2 blocks consecutively. The alarm (PS5330)," G50.9 FORMAT ERROR" occurs if more than 3 blocks are commanded.
- Incremental programming cannot be specified in G50.9 block. The alarm (PS0009),"IMPROPER NC ADDRESS" occurs. Also, in case of incremental mode (G91) in G code system B/C, incremental programming in G50.9 block is regarded as absolute programming.

FRYER MACHINE SYSTEMS INC
Rob Zalensky

Operation

(1) At the start point of movement block, M/S/T/B codes (m01,m02,m03,b01,s01,t01) of movement block are output to PMC, and moving is started.

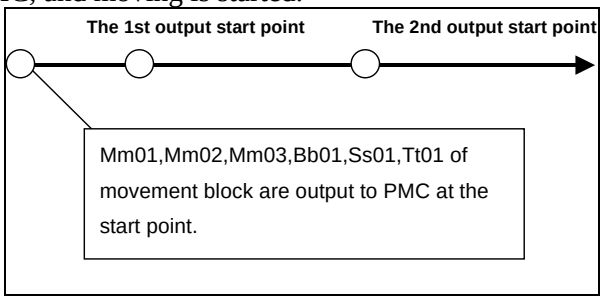


Fig.11.5 (a)

(2) When moving to the 1st output start point that is specified at the 1st G50.9 block, if FIN processing of M/S/T/B code which was output to PMC in (1) has completed, M code (m11,m12,m13) and B code (b11) of the 1st G50.9 block are output to PMC.
If FIN processing has not completed, M code (m11,m12,m13) and B code (b11) are not output to PMC at the 1st output start point. They are output immediately after FIN processing completes. Code signals and strobe signals are output to the same signal address as usual auxiliary function.

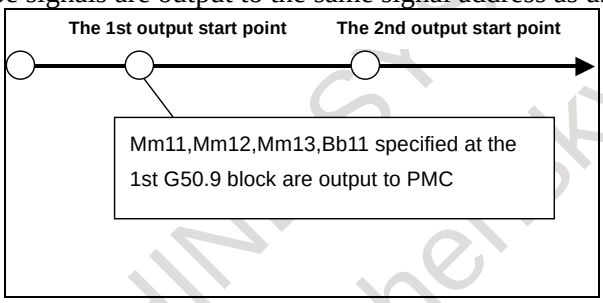


Fig.11.5 (b)

(3) When moving to the 2nd output start point that is specified at the 2nd G50.9 block, if FIN processing of M/S/T/B code which was output to PMC in (2) has completed, M code (m21,m22,m23) and B code (b21) of the 2nd G50.9 block are output to PMC.
If FIN processing has not completed, M code (m21,m22,m23) and B code (b21) are not output to PMC at the 2nd output start point. They are output immediately after FIN processing completes.

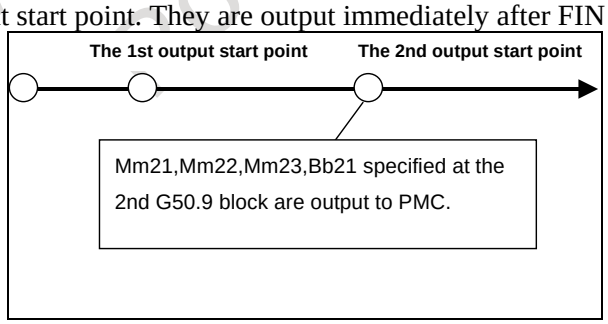


Fig.11.5 (c)

(4) When movement completes, if absolute coordinate has not reached the value that was specified at G50.9 blocks, the alarm (PS5331), “ILLEGAL COMMANDED POSITION“ occurs.

FRYER MACHINE SYSTEMS INC
Bob Zahensky

Specification of output start point

When coordinates for 2 axes are specified

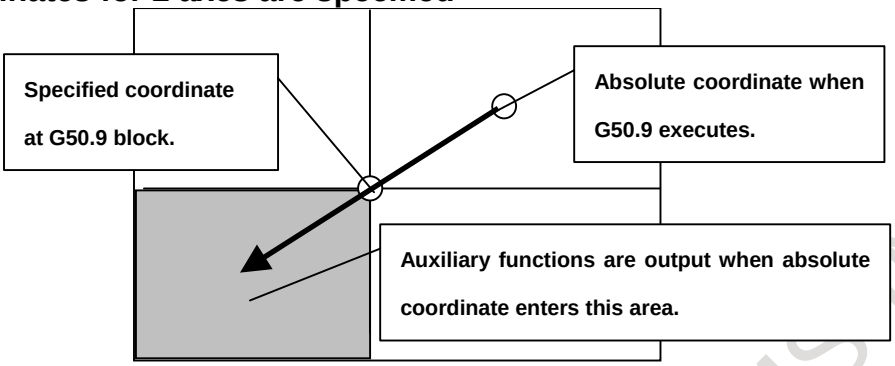


Fig.11.5 (d)

When coordinate for 1 axis is specified

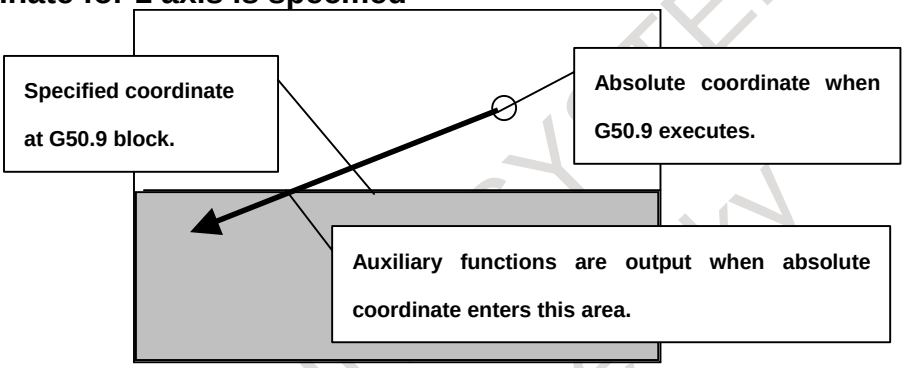


Fig.11.5 (e)

Priority of auxiliary function output

If absolute coordinate have entered the area that was specified at the 2nd G50.9 block while they have not entered the area that was specified at the 1st G50.9 block, auxiliary function of the 2nd G50.9 block are not output at that time. After absolute coordinate entered the area that was specified at the 1st G50.9 block and FIN processing completed, auxiliary function of the 2nd G50.9 block are output finally.

Alarm at movement completion

When movement completes, if absolute coordinate has not reached the value that was specified at G50.9 blocks, the alarm (PS5331), ”ILLEGAL COMMANDED POSITION” occurs.
The alarm (PS5331), ”ILLEGAL COMMANDED POSITION” occurs if next block after G50.9 block is no movement command because absolute coordinate does not reach to the value that was specified at G50.9 block.

Example 1) M code

```
G00 X100.0 ;
G50.9 X200.0 M128 ;
M01;    ← The alarm (PS5331) occurs.
G00 X3000. ;
```

Example 2) Empty block

```
G00 X100.0 ;
G50.9 X200.0 M128 ;
; (Empty block)          ← The alarm (PS5331) occurs.
G00 X300.0 ;
```

FRYER MACHINERY SYSTEMS INC
Ron Zaher

Note

Tool nose radius compensation/Cutter compensation C

Because G50.9 block is treated as a block without movement, “Tool nose radius compensation” and “Cutter compensation C” are temporarily canceled when two G50.9 blocks are commanded consecutively.

Tool compensation

When tool compensation amount is changed by T code at the movement block, absolute coordinate value changes according to tool compensation amount. Therefore, absolute coordinate value sometimes enters the area specified by G50.9 block, at the head of the movement block.

Program restart

The histories of M code displayed in the screen of program restart are not in actual output order but in commanded block order.

Multiple M commands in a single block

The bit 7(M3B) of parameter 3404 must be set to 1 in order to command multiple M codes in one block.

M30, M02, M00, M01

Decode signals DM30, DM02, DM01, DM00 are not output while axes are moving if M30, M02, M01, M00 are specified at G50.9 block. These decode signals are output at movement completion, but not output if FIN processing for M30, M02, M01, M00 have already completed.

M98, M198

Subprogram call M code (M98, M198) cannot be commanded at G50.9 block.
The alarm (PS0076), ”PROGRAM NOT FOUND” occurs.

Macro call M code

Custom macro call M code cannot be commanded at G50.9 block.

Tool life management, Tool management

Even if the M code (M06, etc.) which counts tool life is commanded at G50.9 block, tool life count isn't done.

Waiting M code

Even if a waiting M code is commanded at the G50.9 block, waiting function isn't executed. It is recognized as a usual auxiliary function.

Rigid tapping beginning M code

Even if M code (M29) for beginning rigid tap is commanded at G50.9 block, a mode doesn't switch over to rigid tapping.

M code group check

If plural M codes of one group are commanded at G50.9 block, the alarm (PS5016), ”ILLEGAL COMBINATION OF MCODES” occurs.

Manual handle retrace

G50.9 block and movement block cannot move backward in manual handle retrace mode.
Only forward movement is possible.

Canned cycle

Canned cycle cannot be specified as movement command block.

FRYER MACHINE SYSTEMS INC
Ron Zahensky

Auxiliary function lock

According to the condition of Auxiliary function lock signal AFL at beginning of movement block, auxiliary functions are output to PMC.

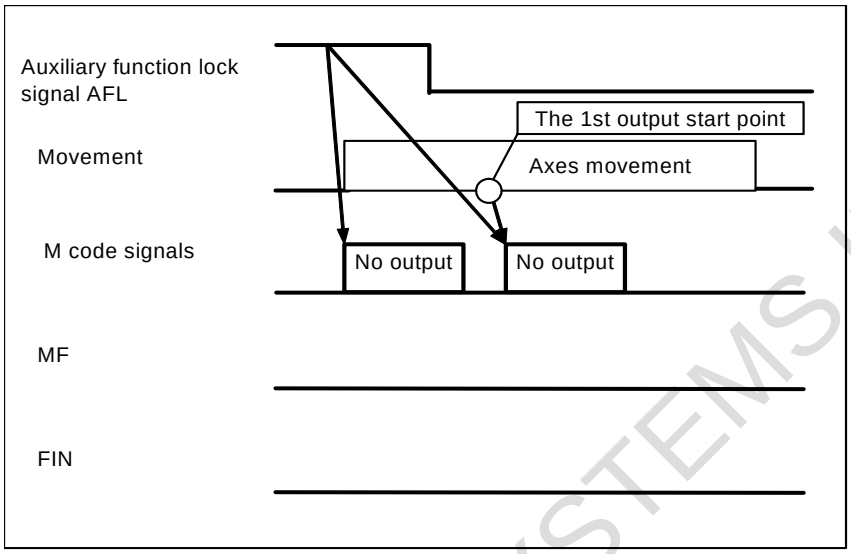


Fig.11.5 (f)

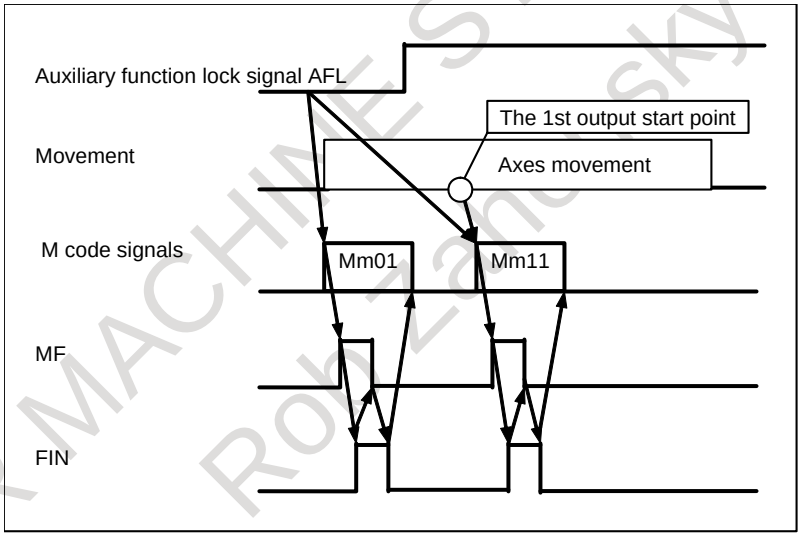


Fig.11.5 (g)

M code for spindle command (M03/M04/M05)

M code for spindle command (M03/M04/M05) is cannot be specified in G50.9 block.
These M codes are recognized as usual auxiliary functions.

M code that is set by parameter

M code in which specific operation is enabled by parameter setting is cannot be specified in G50.9 block.
This M code is recognized as a usual auxiliary function.

Retrace (M system)

G50.9 block and movement block cannot move backward. Only forward movement is possible.

Dynamic graphic display

Instruction information of M code that is specified in G50.9 block is not displayed in Path Drawing screen/Animation screen

FOYER MACHINE SYSTEMS INC
Roh Zanowsky

Manual intervention

Do not operate manual intervention in G50.9 block.

Single block

During single block operation, G50.9 block stops.

Incremental programming

Incremental programming cannot be specified in G50.9 block. The alarm (PS0009), ”IMPROPER NC ADDRESS” occurs.
Also, in case of incremental mode (G91) in G code system B/C, incremental programming in G50.9 block is treated as absolute programming.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12 PROGRAM MANAGEMENT

Operation around program management is described in "PROGRAM MANAGEMENT" in OPERATION.

12.1 FOLDERS

Overview

Folders can be created in program memory.

12.1.1 Folder Configuration

The following folders can be created:

- Folder names are up to 32 characters long.
- The following characters can be used in folder names:
  Alphabetical characters (uppercase and lowercase letters), numeric characters, and the symbols below:
  – + _ .
  Because "." and ".." are reserved folder names, they cannot be used.

- Initial folders

When program memory is initialized, folders having predefined structures and names are created. These folders are called initial folders.

(1) Root folder
The parent folder of all folders

(2) System folder（SYSTEM）
Contains subprograms and macro programs of the system.

(3) MTB dedicated folder 1 (MTB1)
Contains subprograms and macro programs created by the machine tool builder.

(4) MTB dedicated folder 2 (MTB2)
Contains subprograms and macro programs created by the machine tool builder.

(5) User folder
Contains programs created by the user.
The following folders are further created in this folder:
(a) Path folders (PATHn: As many folders as the number of paths are created.)
Each contains main programs, subprograms, and macro programs used for the corresponding path.
(b) Common program folder (LIBRARY)
Contains subprograms and macro programs used commonly.

NOTE
The initial folders can neither be deleted nor renamed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

[Initial folder configuration]

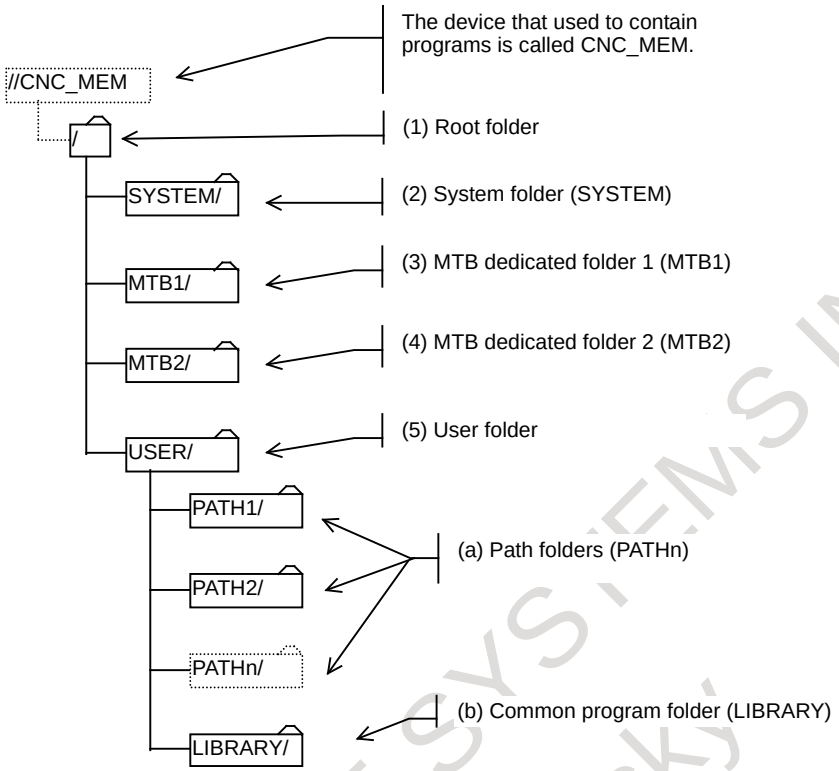


Fig.12.1.1 (a)

- User created folders

Folders other than the initial folders are called user created folders.
User created folders can be created in the following initial folders:

- User folder
- Path folders

User created folders can contain user created main programs, subprograms, and macro programs.

NOTE

1 Folder names must each be unique in the same folder.
2 Each time a user created folder is created, the number of programs that can be registered decreases by one.
3 Folder hierarchical levels of user created folders are limited.
Up to three hierarchical levels starting from the user folder (USER/) are permitted.

FRYER MACHINE SYSTEMS INC
Bob Zahensky

[Sample folder configuration]

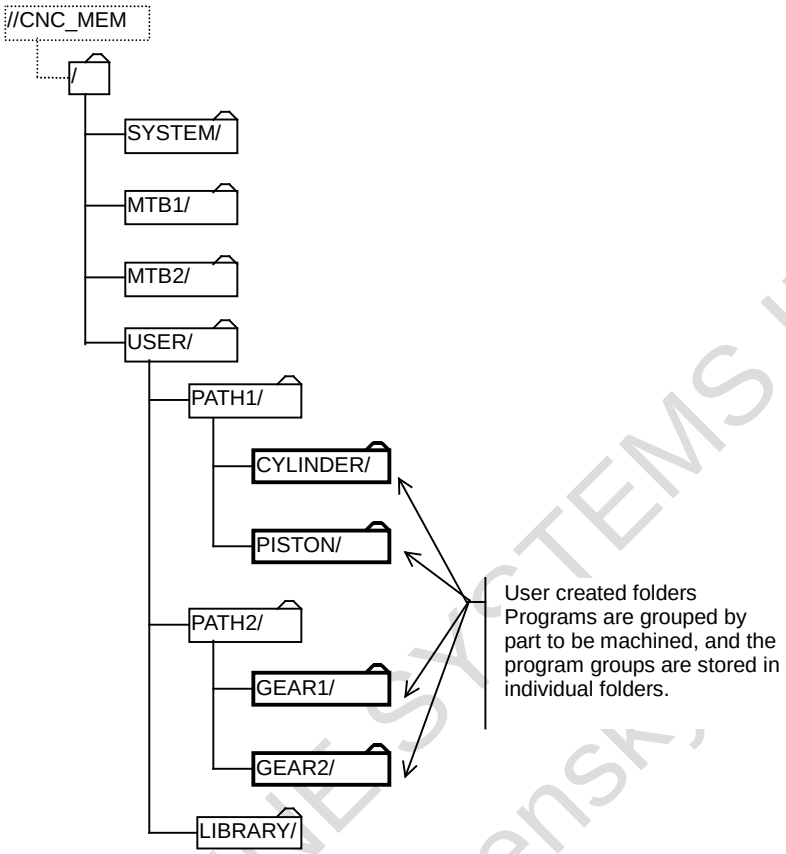


Fig.12.1.1 (b)

12.1.2 Folder Attributes

The following attributes can be set for folders except the root folder:

- Edit disable
- Edit/display disable

- Edit disable

Editing of the programs and folders in a folder can be disabled.
A program in the folder can be output to an external device.
A program cannot be input (registered) from an external device to the folder.

- Edit/display disable

Editing and display of the programs and folders in a folder can be disabled.
When this attribute is set for a folder, the programs and folders in the folder become invisible. (The folder appears to contain nothing.)
A program in the folder cannot be output to an external device and a program cannot be input (registered) from an external device to a folder.

FRYER MACHINE SYSTEMS INC
Rob Zahenski

12.1.3 Default Folders

Default folders are folders on which operations are performed when no folder is specified. There are two types of default folders as follows:

- Foreground default folder
- Background default folder

How to select a default folder is described in "PROGRAM MANAGEMENT" in OPERATION.

- Foreground default folder

A folder used for foreground operations except automatic operations and program editing is set.
The target operations include:

- Program input/output
- External data input
- External workpiece number search

- Background default folder

A folder used for background operations is set.
The target operations include:

- Program input/output

NOTE

1 When the foreground or background default folder is not set, a path folder, which is an initial folder, is assumed.
2 Foreground and background default folder settings are stored in default folder setting files.
3 When a program file, program folder, or program folder management file is cleared, the default folder setting file is cleared at the same time.

12.2 PROGRAMS

Overview

Desired program names can be given to part programs in program memory.

12.2.1 Program Name

Program names can be set as follows:

- Program names are up to 32 characters long.
- The following characters can be used in program names:
  Alphabetical characters (uppercase and lowercase letters), numeric characters, and the symbols below:
  – + _ .
  Because "." and ".." are reserved program names, they cannot be used.

- Program names and program numbers

Program names are associated with program numbers as explained below.
When the program name of a program consists of "O" plus a numeric with the following limitation, the program can be handled also by program number.

- The numeric must be a leading zero suppressed value from 1 to 9999. (Bit 3 (ON8) of parameter No.11304 = 0)
- The numeric must be a leading zero suppressed value from 1 to 99999999. (Bit 3 (ON8) of parameter No.11304 = 0)

FRYER MACHINE SYSTEMS INC
Rob Zamansky

When the program name of a program does not have the above format, the program cannot be handled by program number.
When a program name consisting of "O" plus a numeric does not satisfy the above limitation, the program cannot be created.

Example)

Program names that can be treated as program numbers

O123	Program number 123
O1	Program number 1
O3000	Program number 3000
O9999	Program number 9999
O12345678	Program number 12345678

Program names that cannot be treated as program numbers

ABC
o123
O123.4

NOTE

1 Program names must each be unique in the same folder.
2 When the program name of a program is not treated as a program number, the program is restricted as follows:
- The program cannot be specified by program number.
- Information output by program number is impossible..

- Displaying Program Names and Program Numbers

The program name of the program selected or being executed as the main program is displayed as shown in Fig. 12.2.1 (a) to Fig. 12.2.1 (c).

- For program names that can be handled as program numbers, the program number is displayed.

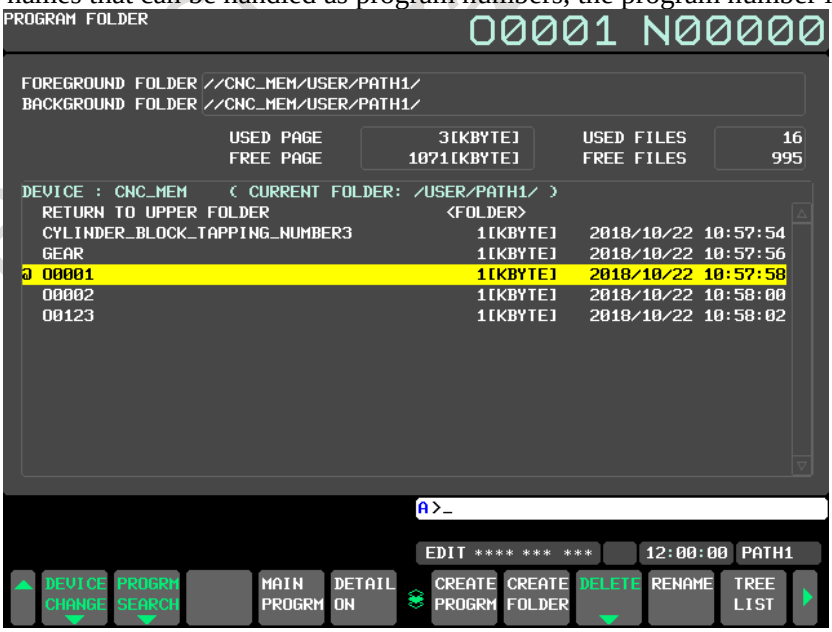


Fig. 12.2.1 (a) Program name display 1

FRYER MACHINE SYSTEMS INC Chernovtsy

- For program names that cannot be handled as program numbers, the program name is displayed. The display layout varies as shown below, depending on the length of a program name.

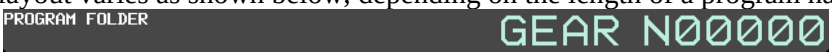


Fig. 12.2.1 (b) Program name display 2

Fig. 12.2.1 (c) Program name display 3

12.2.2 Program Attributes

The following attributes can be set for programs:
- Edit disable
- Edit/display disable
- Encoding
- Change protection level/output protection level

- Edit disable

Editing of a specified program can be disabled.
A program can be output to an external device.
A program cannot be input (registered) from an external device to the folder.

- Edit/display disable

Editing and display of a specified program can be disabled.
When this attribute is set, the program becomes invisible.
(The folder appears not to contain this program.)
An output to an external device and an input from an external device (program registration) cannot be made.

- Encoding

A specified program can be encoded.
For details of encoding, see the description of the “Program encryption” function.
An output to an external device and an input from an external device (program registration) cannot be made.

- Change protection level/output protection level

With the 8-level data protection function, change and output protection can be provided for a specified program.
For details of the 8-level data protection function, see the description of the “Protection of data at eight levels” function.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3 RELATION WITH CONVENTIONAL FUNCTIONS

This section explains relation with conventional functions when folder names and program names are used.

12.3.1 Relation with Folders

This subsection explains how folders are used for operations and editing.

- Automatic operation

Main program

As the main program running for an automatic operation, a program in a desired folder can be selected.

Subprogram (called by M98/G72.1/G72.2)
Macro program (called by G65/G66/G66.1/M96)

- Subprogram call (M98)
- Macro call (simple call G65/modal call G66, G66.1)
- Macro interrupt (M96)
- Figure copying (G72.1, G72.2)

When a call listed above is made, folders are searched in the following order, and the program found first is called:

<1> Folders containing the main program
<2> Common program folder, which is an initial folder (LIBRARY)

Bit 7 (SCF) of parameter No. 3457 can be used to add the following search folders. (The folders are searched in the order listed below.) The folder to be actually enabled is separately set by bits 1 to 3 of parameters No. 3457.

<3> MTB dedicated folder 2, which is an initial folder (MTB2)
<4> MTB dedicated folder 1, which is an initial folder (MTB1)
<5> System folder, which is an initial folder (SYSTEM)

Subprogram (called by M code/specific address/2nd auxiliary function)
Macro program (called by G code/M code/T code/one touch macro)

- Subprogram call by M code/specific address/2nd auxiliary function code
- Macro call by G code/M code/T code
- One touch macro call

For programs called as listed above, the folders to be searched are set in bits 0 to 3 of parameter No.3457 in advance. (The search order is described below.)
The folders set as the search targets are searched, and the program found first is called.

<1> Common program folder, which is an initial folder (LIBRARY)
<2> MTB dedicated folder 2, which is an initial folder (MTB2)
<3> MTB dedicated folder 1, which is an initial folder (MTB1)
<4> System folder, which is an initial folder (SYSTEM)

- Program editing

A program in any folder can be edited.

- Program I/O

The following functions are performed for default folders:

- Program input from external devices
- Program output to external devices

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- External data input

A set foreground default folder is searched for an external program.

- External workpiece number search

A set foreground default folder is searched for an external workpiece number.

- Macro executor

Programs called by execution macros, conversational macros, and auxiliary macros are programs in P-Code files regardless of the default folder.
For programs on the macro executor, the folder function and program name function are unavailable.

12.3.2 Relation with Program Names

Program names can be used with the following functions:

- Subprogram call (M98)
- Macro call (simple call G65/modal call G66, G66.1)
- Interruption type macro call (M96)
- Subprogram call in figure copying (G72.1, G72.2)
- Program I/O with external devices

- Subprogram call by program name
- Macro call by program name

- Subprogram call (M98)
- Macro call (G65/G66/G66.1)
- Interruption type macro call (M96)
- Figure copying (G72.1, G72.2)

When a program is called in the above functions, a subprogram call by program name and a macro call by program name can be used.

- Subprogram call by program name
  M98 <program-name> Lxx ;
- Macro call by program name
  G65 <program-name> Lxx argument-specification ;
  G66 <program-name> Lxx argument-specification ;
  G66.1 <program-name> Lxx argument-specification ;
- Interruption type macro call
  M96 <program-name> ;
- Subprogram call by program name in figure copying (G72.1, G72.2)
  G72.1 <program-name> Lxx Xxx Yxx Rxx ;
  G72.2 <program-name> Lxx Ixx Jxx ;

In the above subprogram and macro calls, the program having the program name indicated by <program-name> is called.

Format examples:

- Subprogram call
  M98 <R50> L1 ;
- Macro call
  G65 <R50> L1 A0 ;
  G66 <R50> L1 A1 ;
  G66.1 <R50> L1 A2 ;
- Interruption type macro call
  M96 <R50> ;

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Subprogram call in figure copy
  G72.1 <R50> L1 X0 Y0 R0 ;
  G72.2 <R50> L1 I0 J0 ;

NOTE
1 When characters in <> are read, they are treated in the same way as for characters in comments. So, note that these characters are treated differently from other significant information portions.
Refer to Appendix B “PROGRAM CODE LIST” for details.
2 The <program-name> word must be placed immediately after each word for calling (M98, G65, and so forth).

12.3.3 Related Parameters

This subsection lists the meanings of parameters related to program numbers and the folders and programs to be manipulated or executed.

Table 12.3.3

Parameter No.	Bit No.	Description	Manipulation/execution target
3202	0 (NE8)	Disables or enables editing of programs O8000 to O8999.	Corresponding programs in all folders
	4 (NE9)	Disables or enables editing of programs O9000 to O9999.	Same as above
3204	3 (P8E)	Disables or enables editing of programs O80000000 to O8999999.	Same as above
	4 (P9E)	Disables or enables editing of programs O90000000 to O99999999.	Same as above
	5 (SPR)	Assumes or does not assume a particular program number in the nine thousands to be a number obtained by adding 90000000.	Same as above
3210/3211	-	Password/keyword for protecting programs in the nine thousands	Same as above
3222/3223	-	Program protection range (minimum value)/(maximum value)	Same as above
3404	2 (SBP)	In the subprogram call function, address P in the M198 block specifies a file/program number.	(Not dependent on the folder function and file name function)
6001	5 (TCS)	Calls or does not call a custom macro by T code.	The following initial folders can be selected by parameter setting: - Common program folder (LIBRARY) - MTB dedicated folder 2 (MTB2) - MTB dedicated folder 1 (MTB1) - System folder (SYSTEM)
6050 to 6059	-	G code for calling a custom macro with program No. 9010 to 9019	Same as above
6071 to 6079	-	M code for calling a subprogram with program No. 9001 to 9009	Same as above
6080 to 6089	-	M code for calling a custom macro with program No. 9020 to 9029	Same as above
6090/6091	-	ASCII code for calling a subprogram with program No. 9004/9005	Same as above
8341	-	Target program number (for 4- and 8-digit O number) for sequence number comparison and stop	Foreground or background default folder depending on the mode
11304	3 (ON8)	Program number is 4-digit / 8-digit	Corresponding programs in all folders

FRYER MACHINE SYSTEMS INC
Rob Zalensky

12.3.4 Part Program Storage Size / Number of Registerable Programs

The following table lists the combinations of program storage sizes and the total number of registrable programs.

Table 12.3.4

Part program storage size	Number of registerable programs	Number of registerable programs expansion 1	Number of registerable programs expansion 2
32Kbyte	63	-	-
64Kbyte	63	125	-
128Kbyte	63	250	-
256Kbyte	63	500	-
512Kbyte	63	1000	-
1Mbyte	63	1000	2000
2Mbyte	63	1000	4000
4Mbyte	63	1000	4000
8Mbyte	63	1000	4000

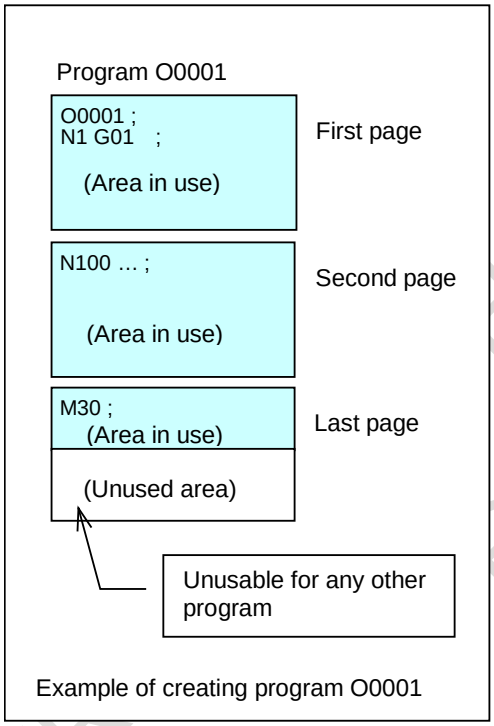


Fig. 12.3.4 (a)Unit of program storage

NOTE

1 Creating one folder results in the number of programs yet to be registerable decreasing one.

2 The program storage size means the maximum size of a program if the program is the one and only program registered.

3 If more than one program is registered, the total size of registerable programs reduces for the following reason.
The unit of program storage is managed also in page units. When a program is created, as many pages as necessary to store the program are secured, and the program is stored on these pages. Generally, the last program storage page has an unused area (Fig. 12.3.4(a)). This unused area cannot be used to store any other program. For the sake of program management, it is regarded as an area in use.

FANUC MACHINE SYSTEMS INC Robert Zaremsky

13 PROGRAM CONFIGURATION

Overview

- Main program and subprogram

There are two program types, main program and subprogram. Normally, the CNC operates according to the main program. However, when a command calling a subprogram is encountered in the main program, control is passed to the subprogram. When a command specifying a return to the main program is encountered in a subprogram, control is returned to the main program.

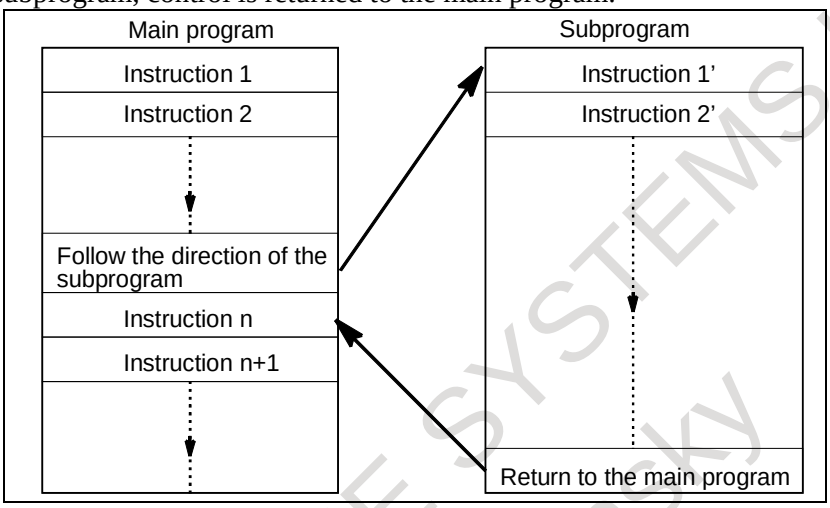


Fig. 13 (a) Main program and Subprogram

The CNC memory can hold up to 4000 main programs and subprograms (63 as standard). A main program can be selected from the stored main programs to operate the machine. Refer to the Chapter, "CREATING PROGRAMS" and Section, "PROGRAM SEARCH" for the methods of registering and selecting programs.

- Program components

A program consists of the following components:

Table 13 (a) Program components

Components	Descriptions
Program code start	Symbol indicating the start of a program file
Leader section	Used for the title of a program file, etc.
Program start	Symbol indicating the start of a program
Program section	Commands for machining
Comment section	Comments or directions for the operator
Program code end	Symbol indicating the end of a program file

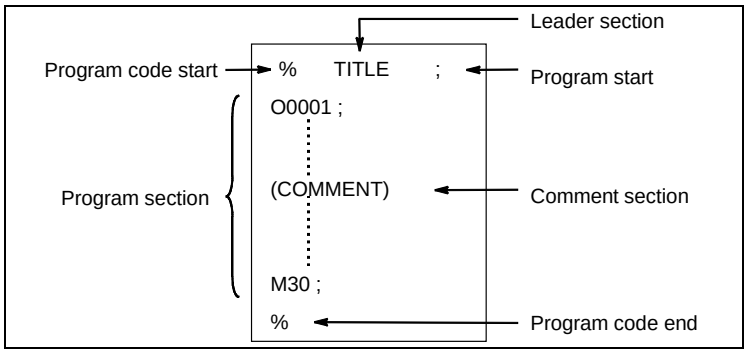


Fig. 13 (b) Program configuration

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Program section configuration

A program section consists of several blocks. A program section starts with a program number or program name and ends with a program end code.

Program section configuration	Program section
Program number	O0001 ;
Block 1	N1 G91 G00 X120.0 Y80.0 ;
Block 2	N2 G43 Z-32.0 H01 ;
:	:
Block n	Nn Z0 ;
Program end	M30 ;

A block contains information necessary for machining, such as a move command or coolant on/off command. Specifying a slash (/) at the start of a block disables the execution of some blocks (Refer to "optional block skip").

13.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS

This section describes program components other than program sections. Refer to the Section, “PROGRAM SECTION CONFIGURATION” for a program section.

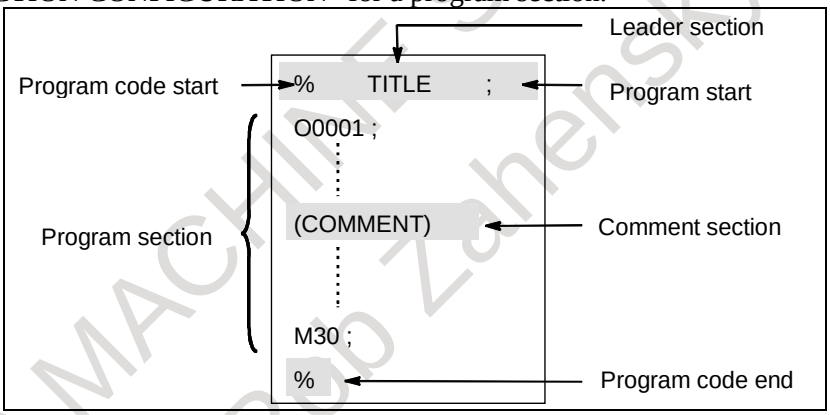


Fig. 13.1 (a) Program configuration

Explanation

- Program code start

The program code start indicates the start of a file that contains CNC programs.
The mark is not required when programs are entered using SYSTEM P or ordinary personal computers. The mark is not displayed on the screen. However, if the file is output, the mark is automatically output at the start of the file.

Table 13.1 (a) Code of a program code start

Name	ISO code	EIA code	Notation in this manual
Program code start	%	ER	%

- Leader section

Data entered before the programs in a file constitutes a leader section.
When machining is started, the label skip state is usually set by turning on the power or resetting the system. In the label skip state, all information is ignored until the first end-of-block code is read. When a file is read into the CNC unit from an I/O device, leader sections are skipped by the label skip function.
A leader section generally contains information such as a file header. When a leader section is skipped, even a TV parity check is not made. So a leader section can contain any codes except the EOB code.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Program start

The program start code is to be entered immediately after a leader section, that is, immediately before a program section.
This code indicates the start of a program, and is always required to disable the label skip function.
With SYSTEM P or ordinary personal computers, this code can be entered by pressing the return key.

Table 13.1 (b) Code of a program start

Name	ISO code	EIA code	Notation in this manual
Program start	LF	CR	;

NOTE
If one file contains multiple programs, the EOB code for label skip operation must not appear before a second or subsequent program number.

- Comment section

Any information enclosed by the control-out and control-in codes is regarded as a comment.
The user can enter a header, comments, directions to the operator, etc. in a comment section.

Table 13.1 (c) Codes of a control-in and a control-out

Name	ISO code	EIA code	Notation in this manual	Meaning
Control-out	(	2-4-5	(	Start of comment section
Control-in	)	2-4-7	)	End of comment section

When a program is read into memory for memory operation, comment sections, if any, are not ignored but are also read into memory. Note, however, that codes other than those listed in the code table in Appendix A are ignored, and thus are not read into memory.
When data in memory is output on external I/O device (Refer to the Chapter, “DATA INPUT/OUTPUT”), the comment sections are also output.
When a program is displayed on the screen, its comment sections are also displayed. However, those codes that were ignored when read into memory are not output or displayed.
During memory operation or DNC operation, all comment sections are ignored.
The TV check function can be used for a comment section by setting bit 1 (CTV) of parameter No. 0100.

⚠ CAUTION
If a long comment section appears in the middle of a program section, a move along an axis may be suspended for a long time because of such a comment section. So a comment section should be placed where movement suspension may occur or no movement is involved.

NOTE
1 If only a control-in code is read with no matching control-out code, the read control-in code is ignored.
2 The following codes cannot be used in the comment section:
- EOB
- % (ER for EIA)

- Program code end

A program code end is to be placed at the end of a file containing NC programs.
If programs are entered using the automatic programming system, the mark need not be entered.
The mark is not displayed on the screen. However, when a file is output, the mark is automatically output at the end of the file.
If an attempt is made to execute % when M02 or M30 is not placed at the end of the program, the alarm PS5010, “END OF RECORD” is occurred.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Table 13.1 (d) Code of a program code end

Name	ISO code	EIA code	Notation in this manual
Program code end	%	ER	%

13.2 PROGRAM SECTION CONFIGURATION

This section describes elements of a program section. Refer to the Section, “PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS” for program components other than program sections.

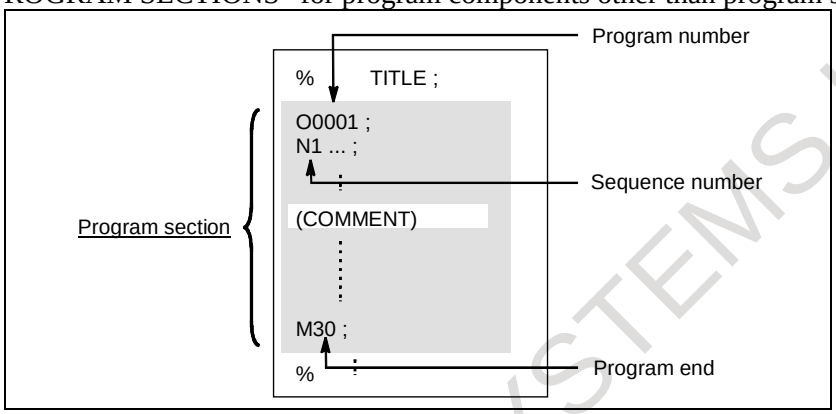


Fig. 13.2 (a) Program configuration

- Program number

A program number consisting of address O followed by a four-digit number is assigned to each program at the beginning registered in memory to identify the program. When the 8-digit number function is selected, the program number consists of eight digits.
In ISO code, the colon (:) can be used instead of O.
When no program number is specified at the start of a program, the sequence number (N....) at the start of the program is regarded as its program number. If a five-digit sequence number is used, the lower four digits are registered as a program number. If the lower four digits are all 0, the program number registered immediately before added to 1 is registered as a program number. Note, however, that N0 cannot be used for a program number.
If there is no program number or sequence number at the start of a program, a program number must be specified using the MDI unit when the program is stored in memory (Refer to the Section, “INPUT/OUTPUT ON EACH SCREEN” or “CREATING PROGRAMS USING THE MDI UNIT”)

NOTE
Program numbers 8000 to 9999 may be used by machine tool builders, and the user may not be able to use these numbers.

- Program name

A program name can be assigned instead of a program number.
When coding a program name, be sure to place the program name enclosed in "<" and ">" at the beginning of a program.

```
Example) % ;
         <PARTS_1> ;
         N1 ...
          :
         M30 ;
         %
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

A program name can be coded:
- At the beginning of a program
- Immediately after M98, G65, G66, G66.1, M96, G72.1, or G72.2

Do not code a file name in other than the above.

- Sequence number and block

A program consists of several commands. One command unit is called a block. One block is separated from another with an EOB of end of block code.

Table 13.2 (a) EOB code

Name	ISO code	EIA code	Notation in this manual
End of block (EOB)	LF	CR	;

At the head of a block, a sequence number consisting of address N followed by a number not longer than eight digits (1 to 99999999) can be placed. Sequence numbers can be specified in a random order, and any numbers can be skipped. Sequence numbers may be specified for all blocks or only for desired blocks of the program. In general, however, it is convenient to assign sequence numbers in ascending order in phase with the machining steps (for example, when a new tool is used by tool replacement, and machining proceeds to a new surface with table indexing.)

N300X200.0Z300.0 ; A sequence number is underlined.

Fig. 13.2 (b) Sequence number and block (example)

NOTE

N0 must not be used for the reason of file compatibility with other CNC systems. Program number 0 cannot be used. So 0 must not be used for a sequence number regarded as a program number.

- TV check (Vertical parity check)

A parity check is made for each block of input data. If the number of characters in one block (starting with the code immediately after an EOB and ending with the next EOB) is odd, an alarm PS0002, “TV ERROR” is output.

No TV check is made only for those parts that are skipped by the label skip function. Bit 1 (CTV) of parameter No. 0100 is used to specify whether comments enclosed in parentheses are counted as characters during TV check. The TV check function can be enabled or disabled by setting on the MDI unit (Refer to the Subsection, “Displaying and Entering Setting Data”).

- Block configuration (word and address)

A block consists of one or more words. A word consists of an address followed by a number some digits long. (The plus sign (+) or minus sign (-) may be prefixed to a number.)

For an address, one of the letters (A to Z) is used ; an address defines the meaning of a number that follows the address.

Word = Address + number (Example : X-1000)

Table 13.2 (b) indicates the usable addresses and their meanings.

The same address may have different meanings, depending on the preparatory function specification.

Table 13.2 (b) Major functions and addresses

Function	Address	Meaning
Program number	O(*)	Program number
Sequence number	N	Sequence number
Preparatory function	G	Specifies a motion mode (linear, arc, etc.)

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Function	Address	Meaning
Dimension word	X, Y, Z, U, V, W, A, B, C	Coordinate axis move command
	I, J, K	Coordinate of the arc center
	R	Arc radius
Feed function	F	Rate of feed per minute, Rate of feed per revolution
Spindle speed function	S	Spindle speed
Tool function	T	Tool number
Auxiliary function	M	On/off control on the machine tool
	B	Table indexing, etc.
Program number designation	P	Subprogram number
Number of repetitions	P, L	Number of subprogram repetitions
Parameter	P, Q	Canned cycle parameter

M

Offset number	D, H	Offset number
Dwell	P, X	Dwell time

T

Dwell	P, X, U	Dwell time

NOTE
(*) In ISO code, the colon (:) can also be used as the address of a program number.

N_	G_	X_ Y_	F_	S_	T_	M_	;
Sequence number	Preparatory function	Dimension word	Feed-function	Spindle speed function	Tool function	Auxiliary function	

Fig. 13.2 (c) 1 block (example)

- Major addresses and ranges of command values

Major addresses and the ranges of values specified for the addresses are shown below. Note that these figures represent limits on the CNC side, which are totally different from limits on the machine tool side. For example, the CNC allows a tool to traverse up to about 100 m (in millimeter input) along the X axis. However, an actual stroke along the X axis may be limited to 2 m for a specific machine tool.
Similarly, the CNC may be able to control a cutting feedrate of up to 240 m/min, but the machine tool may not allow more than 3 m/min. When developing a program, the user should carefully read the manuals of the machine tool as well as this manual to be familiar with the restrictions on programming.

FRYER MACHINE SYSTEMS INC
Pon Zalensky

Table 13.2 (c) Major addresses and ranges of command values

Function		Address	Input in mm	Input in inch
Program number		O (*1)	1 to 99999999	1 to 99999999
Sequence number		N	1 to 99999999	1 to 99999999
Preparatory function		G	0 to 99.9	0 to 99.9
Dimension word	Increment system IS-A	X,Y,Z,U,V, W,A,B,C,I, J,K,R (*2)	±999999.99 mm ±999999.99 deg.	±99999.999 inch *3 ±999999.99 deg.
	Increment system IS-B		±999999.999 mm ±999999.999 deg.	±99999.9999 inch *3 ±999999.999 deg.
	Increment system IS-C		±99999.9999 mm ±99999.9999 deg.	±9999.99999 inch *3 ±99999.9999 deg.
Feed per minute	Increment system IS-A	F	0.01 to 999000.00mm/min	0.001 to 96000.000inch/min
	Increment system IS-B		0.001 to 999000.000mm/min	0.0001 to 9600.0000inch/min
	Increment system IS-C		0.0001 to 99999.9999mm/min	0.00001 to 4000.00000inch/min
Feed per revolution		F	0.001 to 50000mm/rev	0.0001 to 50.0000inch/rev
Spindle speed function		S (*4)	0 to 99999999	0 to 99999999
Tool function		T (*4)	0 to 99999999	0 to 99999999
Auxiliary function		M (*4)	0 to 99999999	0 to 99999999
		B (*4)	0 to 99999999	0 to 99999999
Offset number (M series only)		H, D	0 to 999	0 to 999
Dwell	Increment system IS-A	X, U (T series only)	0 to 999999.99 sec	0 to 999999.99 sec
	Increment system IS-B		0 to 99999.999 sec	0 to 99999.999 sec
	Increment system IS-C		0 to 9999.9999 sec	0 to 9999.9999 sec
Dwell		P	1 to 99999999	1 to 99999999
Designation of a program number		P	1 to 99999999	1 to 99999999
Number of subprogram repetitions		L	1 to 99999999	1 to 99999999
		P	0 to 9999	0 to 9999

*1 In ISO code, the colon (:) can also be used as the address of a program number.

*2 When address I, J, K, or R is used to specify the radius for circular interpolation, the specifiable range is as follows:

Increment system	Input in mm	Input in inch
IS-A	±999999999.99 mm	±99999999.999 inch
IS-B	±999999999.999 mm	±99999999.9999 inch
IS-C	±99999999.9999 mm	±9999999.99999 inch

*3 For inch input/millimeter machines, the maximum specifiable range of dimension words is as follows:

Increment system	Maximum specifiable range
IS-A	±39370.078 inch
IS-B	±39370.0787 inch
IS-C	±3937.00787 inch

*4 The maximum value of addresses M, S, T, and B is 99999999. Note that, however, values longer than the permissible number of digits set in parameters Nos. 3030 to 3033 cannot be specified.
The values and uses for some codes are limited by parameter setting. (For example, some M codes are not buffered.) For details, refer to the parameter manual.

- Optional block skip

When a slash followed by a number (/n (n=1 to 9)) is specified at the head of a block, and optional block skip signals BDT1 to BDT9 are set to “1” during automatic operation, the information (/n to the end of the block (EOB)) contained in the block for which /n corresponding to signal BDTn is specified is ignored.

Example 1)

```
/2    N123    X100.0    Y200.0 ;
```


FRYER MACHINE SYSTEMS INC
Rob Zahenski

Example 2)

//3 N123 X100.0 Y200.0 ; → Incorrect

/1 /3 N123 X100.0 Y200.0 ; → Correct

Input signal and program code

Input signal	Start code to be ignored
BDT1	/ or /1(NOTE)
BDT2	/2
BDT3	/3
BDT4	/4
BDT5	/5
BDT6	/6
BDT7	/7
BDT8	/8
BDT9	/9

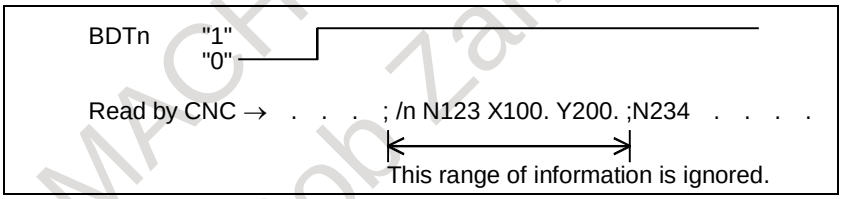
NOTE

1 Number 1 for /1 can be omitted. However, when two or more optional block skips are specified for one block, number 1 for /1 cannot be omitted.

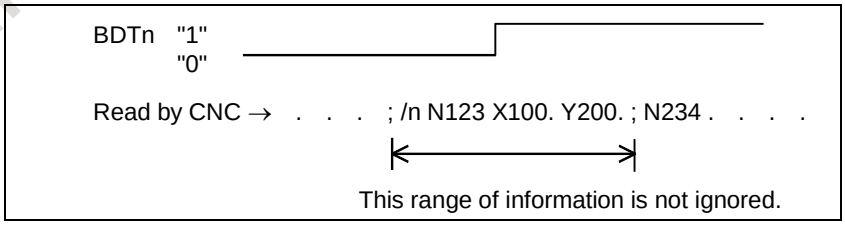
2 Depending on the machine tool, all optional block skip signals (1 to 9) may not be usable. Refer to manuals of the machine tool builder to find which switches are usable.

The following shows the relationship between the timing at which optional block skip signals BDT1 to BDT9 are set to "1" and the range of information to be ignored.

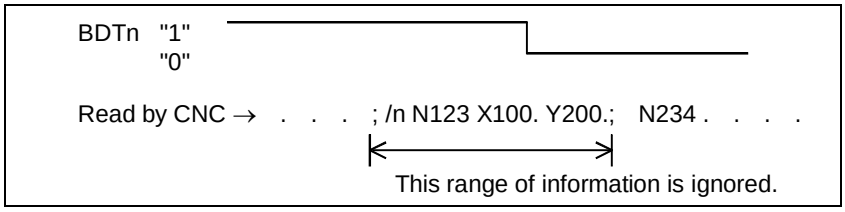
1. When the signal BDTn is set to "1" before the CNC starts reading a block that contains /n, the block is ignored.



2. When the signal BDTn is set to "1" while the CNC is reading a block that contains /n, the block is not ignored.

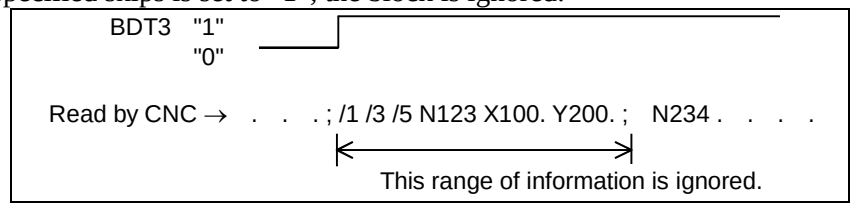


3. When the signal BDTn is set to "0" while the CNC is reading a block that contains /n, the block is ignored.



FRYER MACHINE SYSTEMS INC
Rob Zahensky

4. Two or more optional block skips can be specified in one block. When the signal corresponding to any of the specified skips is set to “1”, the block is ignored.



NOTE

1 This function is not used when a program is registered in memory. A block containing / is registered in memory regardless of the statuses of optional block skip signals. When a program in memory is also output regardless of the statuses of optional block skip signals.
In addition, the optional block skip function is enabled during a search for a sequence number.

2 Position of a slash
A block skip slash (/) must be specified at the beginning of a block. Note that if a slash is placed elsewhere, the information from the slash to immediately before the EOB code is ignored.

3 TV and TH check
When the optional block skip signal is set to “1”, TH and TV checks are made for the skipped portions in the same way as when the optional block skip signal is set to “0”.

- Program end

The end of a program is indicated by programming one of the following codes at the end of the program:

Table 13.2 (d) Code of a program end

Code	Meaning usage
M02	For main program
M30	
M99	For subprogram

If one of the program end codes is executed in program execution, the CNC terminates the execution of the program, and the reset state is set. When the subprogram end code is executed, control returns to the program that called the subprogram.

⚠ CAUTION

A block containing an optional block skip code such as /M02 ; , /M30 ; , or /M99 ; is not regarded as the end of a program. (Refer to "Optional block skip".)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

13.3 SUBPROGRAM (M98, M99)

If a program contains a fixed sequence or frequently repeated pattern, such a sequence or pattern can be stored as a subprogram in memory to simplify the program.
A subprogram can be called from the main program.
A called subprogram can also call another subprogram.

Format

- Subprogram configuration

One subprogram

Oxxxx ;	Subprogram number or subprogram name (or the colon (:) optionally in the case of ISO)
:	
M99;	Program end

M99 need not constitute a separate block as indicated below.

Example) X100.0Y100.0M99;

- Subprogram call

- When a subprogram with a 4-digit or shorter program number is called

M98 Pxxxx xxxx ;

Subprogram number
Number of times the subprogram is called repeatedly

or

M98 Pxxxx Lxxxxxxxx ;

Number of times the subprogram is called repeatedly
Subprogram number

- When a subprogram with a 5-digit or longer program number is called

M98 Pxxxxxxxx Lxxxxxxxx ;

Subprogram number
Number of times the subprogram is called repeatedly

- When a subprogram is called by program name

M98 <xxxx> Lxxxxxxxx ;

Subprogram name
Number of times the subprogram is called repeatedly

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Called program and folders to be searched

The order in which folders are searched depends on the method of calling a subprogram. Folders are searched in sequence and the program found first is called. For details, refer to the Chapter "Managing Programs".

NOTE

1 When calling a subprogram of a 4-digit or shorter subprogram number repeatedly (P8 digit), make the subprogram number 4 digits by prefixing it with 0 if the subprogram number is shorter than 4 digits.
Example)
P100100: Call subprogram No. 100 ten times.
P50001: Call subprogram No. 1 five times.

2 If the number of times the subprogram is called repeatedly is omitted when a subprogram with a 4-digit or shorter program number is called, the subprogram is called just once.
In this case, it is not necessary to adjust the subprogram number length to 4 digits as described in Item 1 above.

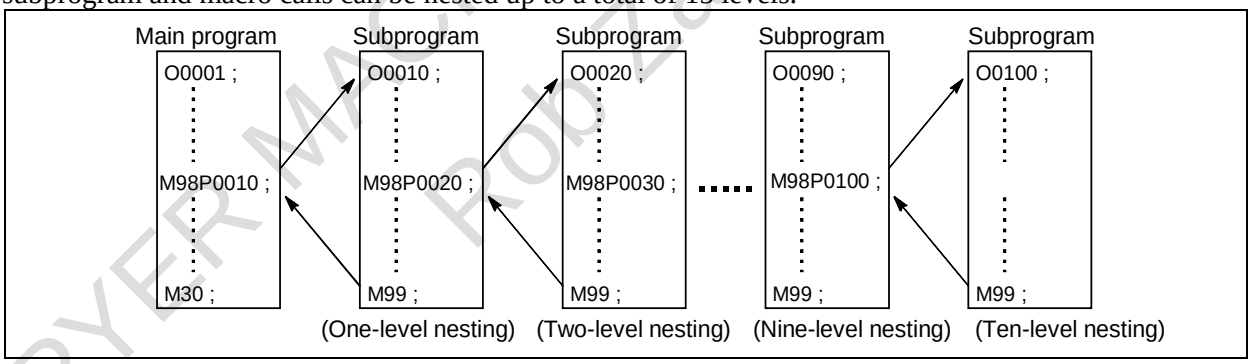
3 When calling a subprogram of a 4-digit or shorter program number repeatedly (P8 digit), do not specify address L in the same block.

4 When calling a subprogram with a 5-digit or longer program number, do not omit the specification of the number of repeats.

5 When calling a subprogram by program name, be sure to specify the program name immediately after M98.

Explanation

When the main program calls a subprogram, it is regarded as a one-level subprogram call. Thus, subprogram calls can be nested up to ten levels as shown below. In addition, when custom macro is used, subprogram and macro calls can be nested up to a total of 15 levels.



A single call command can repeatedly call a subprogram up to 99999999 times. For compatibility with automatic programming systems, in the first block, Nxxxxxxxx can be used instead of a subprogram number that follows O (or :). A sequence number after N is registered as a subprogram number.

NOTE

1 The M98 and M99 code signal and strobe signal are not output to the machine tool.

2 If the subprogram number specified by address P cannot be found, an alarm PS0078, “SEQUENCE NUMBER NOT FOUND” is output.

3 When an attempt is made to call a subprogram by program name, but the specified program cannot be found, alarm PS0310, “FILE NOT FOUND” is issued.

FRYER MACHINE SYSTEMS INC
Rob Zarensky

Example

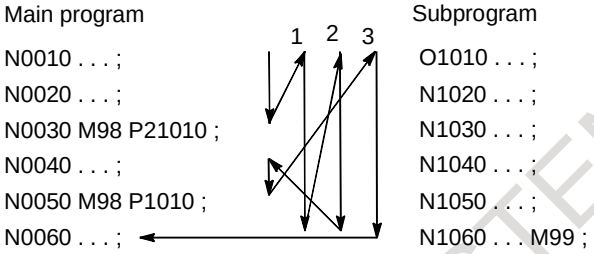
M98 P51002 ;

This command specifies "Call the subprogram (number 1002) five times in succession." A subprogram call command (M98P_) can be specified in the same block as a move command.

X1000.0 M98 P1200 ;

This example calls the subprogram (number 1200) after an X movement.

Execution sequence of subprograms called from a main program

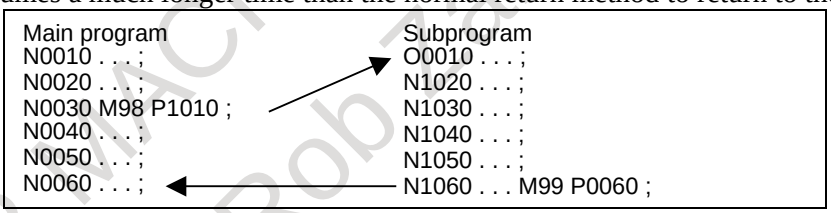


A subprogram can call another subprogram in the same way as a main program calls a subprogram.

Special usage

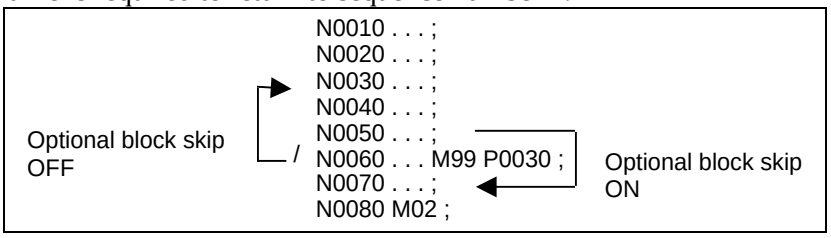
- Specifying the sequence number for the return destination in the main program

If P is used to specify a sequence number when a subprogram is terminated, control does not return to the block after the calling block, but returns to the block with the sequence number specified by P. Note, however, that P is ignored if the main program is operating in a mode other than memory operation mode. This method consumes a much longer time than the normal return method to return to the main program.



- Using M99 in the main program

If M99 is executed in a main program, control returns to the start of the main program. For example, M99 can be executed by placing /M99 ; at an appropriate location of the main program and setting the optional block skip function to off when executing the main program. When M99 is executed, control returns to the start of the main program, then execution is repeated starting at the head of the main program.
Execution is repeated while the optional block skip function is set to off.
If the optional block skip function is set to on, the /M99 ; block is skipped ; control is passed to the next block for continued execution.
If/M99Pn ; is specified, control returns not to the start of the main program, but to sequence number n. In this case, a longer time is required to return to sequence number n.



FRYER MACHINE SYSTEMS INC Rob Zanensky

- Using a subprogram only

A subprogram can be executed just like a main program by searching for the start of the subprogram with the MDI.
(Refer to the Section, “PROGRAM SEARCH” for information about search operation.)
In this case, if a block containing M99 is executed, control returns to the start of the subprogram for repeated execution. If a block containing M99Pn is executed, control returns to the block with sequence number n in the subprogram for repeated execution. To terminate this program, a block containing /M02 ; or /M30 ; must be placed at an appropriate location, and the optional block switch must be set to off ; this switch is to be set to on first.

```
N1010 . . . ;
N1020 . . . ;
N1030 . . . ;
/ N1040 . . . M02 ;
N1050 M99 P1020 ;        Optional block skip ON
```

- Subprogram call with sequence number

Setting bit 0 (SQC) of parameter No. 6005 to 1 can call a specified sequence number in the subprogram for execution.
In a subprogram call command, specify the letter Q followed by a sequence number to be called after the letter P for specifying a program number.

```
M98 Pxxxx Qxxxxx ;
          Sequence number
     Program number
```

This command causes program execution to start at the called sequence number in the subprogram. If a repetition count is specified, program execution is repeated from the specified sequence number.

Main program	Sub program
N0010···;	O1010···;
N0020···;	N1020···;
N0030 M98 P1010 Q1030 ;	N1030···;
N0040···;	N1040···;
N0050···;	N1050···;
N0060···;	N1060···M99 ;

This function enables a sequence number in the same program to be called for execution as shown below.
This method, however, requests the programmer to be aware of an allowable call nesting level. If an attempt is made to exceed the allowable nesting level, alarm PS0077, “TOO MANY SUB,MACRO NESTING” is issued.

```
O0001 ;
N0010···;
N0020 M98 (P0001) Q0050 ;
N0030···;
N0040···;
N0050···;
N0060···;
N0070···M99;
```

For a call within the same program, specification of Pxxxx in a block can be omitted when the block includes M98.

This function is usable only for subprogram calls by M98; it is unusable for non-M98 calls, such as macro calls or external subprogram calls based on M198.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

NOTE
If bit 0 (SQC) of parameter No. 6005 is 0, and an M98 Pxxxx Qxxxxx command is specified, alarm PS0009, "IMPROPER NC-ADDRESS", is issued.

13.4 COMMAND PATH OF MACRO CALL OR SUBPROGRAM CALL

The call of program by the command of path can be executed at Subprogram call and Macro call.
This function can be used by setting the parameter CPD (No.3233#4) =1.

Format

Case of subprogram call (M98)

1. Command absolute path

M98 <//device name(*1)/folder name/file name> L_ ;

L: Number of times the subprogram is called repeatedly
（*1）The device name can be omitted. If the device name is omitted, the same device as folder of main program is selected.

2. Command relative path

M98 <./../folder name/file name> L_ ;

. : It means the folder of main program.
..: It means the folder of one high hierarchy.
L: Number of times the subprogram is called repeatedly

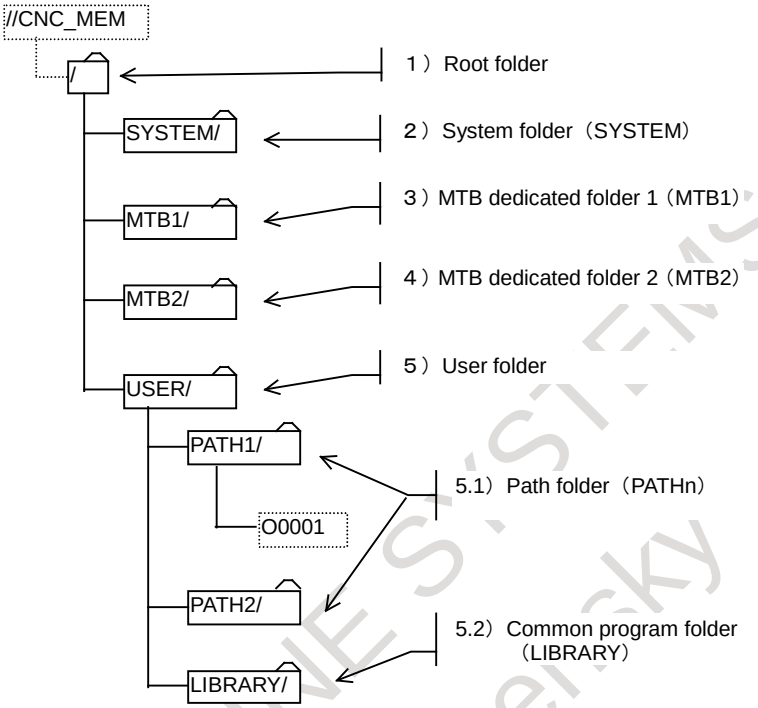
NOTE
1 If the path for file name is omitted, the program is searched in the regulations of order as usual, and called.
2 If parameter CPD is set to 1 and the path for file name is commanded, the program is not searched in the regulations of order as usual.
3 The following command can be use.
- Subprogram call (M98)
- Macro call (G65,G66,G66.1)
- Subprogram call in figure copying (G72.1,G72.2)
4 “.” can be specified once at the head of the path name.
“..” can be specified only for continuing after“.”.
5 “..” is not possible to go up to the hierarchy of the device name.
If the command of relative path is executed, It is not possible to call the program of the different device.
6 When the path command called by DNC operation with the program is not selected in MEM mode, only the absolute path command include device name can be executed. If another path command is specified, the alarm occurs.
7 In Word editing mode,
- when CPD is set to 0, ' /' cannot be used between ' <' and ' >'.
- when CPD is set ti 1, ' /' can be used between ' <' and ' >'.
8. When the specified program cannot be found, alarm PS0310, “FILE NOT FOUND” is issued.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

In case of the following folder components, program O1 is called by absolute path of subprogram call.

［Folder components］



M98<//CNC_MEM/USER/PATH1/O1> is coded.

Limitation

The following functions are not supported.

- Macro interrupt (M96)
- Macro executor

Other limitations are the same as those for subprogram call or macro call.

FRYER MACHINE SYSTEMS INC
Rob Zaherosky

14 FUNCTIONS TO SIMPLIFY PROGRAMMING

14.1 FIGURE COPYING (G72.1, G72.2)

Machining can be repeated after moving or rotating the figure using a subprogram.

Format

- Rotational copying

Xp-Yp plane (specified by G17) : **G72.1 P_ L_ Xp_Yp_R_ ;**

Zp-Xp plane (specified by G18) : **G72.1 P_ L_ Zp_Xp_R_ ;**

Yp-Zp plane (specified by G19) : **G72.1 P_ L_ Yp_Zp_R _;**

P :Subprogram number
L :Number of times the operation is repeated
Xp :Center of rotation on the Xp axis (Xp: X-axis or an axis parallel to the X-axis)
Yp :Center of rotation on the Yp axis (Yp: Y-axis or an axis parallel to the Y-axis)
Zp :Center of rotation on the Zp axis (Zp: Z-axis or an axis parallel to the Z-axis)
R :Angular displacement (A positive value indicates a counterclockwise angular displacement. Specify an incremental value.)

Specify a plane selection command (G17, G18, or G19) to select the plane on which the rotational copying is made.

- Linear copying

Xp-Yp plane (specified by G17) : **G72.2 P_ L_ I_ J_ ;**

Zp-Xp plane (specified by G18) : **G72.2 P_ L_ K_ I_ ;**

Yp-Zp plane (specified by G19) : **G72.2 P_ L_ J_ K_;**

P :Subprogram number
L :Number of times the operation is repeated
I :Shift along the Xp axis
J :Shift along the Yp axis
K :Shift along the Zp axis

Specify a plane selection command (G17, G18, or G19) to select the plane on which the linear copying is made.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Explanation

- First block of the subprogram

The first block of a subprogram that performs a rotational or linear copying must be the block including absolute move command. If the first block does not have a move command, movement may stop at the start point of the figure made by the n-th (n = 1, 2, 3, ...) copying.

(Example of an incorrect program)

```
O1234 ;
G00 G90 X100.0 Y200.0 ;
. . . . . ;
. . . . . ;
M99 ;
```

(Example of a correct program)

```
O1000 G00 G90 X100.0 Y200.0 ;
. . . . . ;
. . . . . ;
M99 ;
```

- Combination of rotational and linear copying

The linear copying command can be specified in a subprogram for a rotational copying. Also, the rotational copying command can be specified in a subprogram for a linear copying.

- Subprogram call

In a subprogram for rotational or linear copying, M98 for calling another subprogram or G65 for calling a macro can be specified.

- Specifying the center of rotation

The center of rotation specified with G72.1 is processed as an absolute position even in the incremental mode.

- Specifying address

In a block with G72.1, addresses other than P, L, Xp, Yp, Zp, or R are ignored. The subprogram number (P), coordinates of the center of rotation (Xp, Yp, Zp), and angular displacement (R) must be specified.
In a block with G72.2, addresses other than P, L, I, J, or K are ignored.
The subprogram number (P) and shift (I, J, K) must be specified.

- Address P

If the subprogram number specified with P is not found, alarm PS0310, “FILE NOT FOUND” occurs. If P is not specified, alarm PS0076, “PROGRAM NOT FOUND” occurs.

- Address L

If L is omitted, the repetition count is assumed to be 1 and the subprogram is called only once.

- Increment in angular displacement or shift

In a block with G72.1, an increment in angular displacement is specified with address R. The angular displacement of the figure made by the n-th rotation is calculated as follows : $R \times (n - 1)$.
In a block with G72.2, an increment in shift is specified with addresses I, J, and K. The shift of the figure made by the n-th movement is calculated as follows : (Programmed shift) $\times$ (n - 1).

- Nesting level of a subprogram

If a subprogram is called by G72.1 or G72.2, the nesting level is increased by one in the same manner as when M98 is specified.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Block end position

The coordinates of a figure moved rotationally or linearly (block end position) can be read from #5001 and subsequent system variables of the custom macro of rotational or linear copying.

- Disagreement between end point and start point

If the end point of the figure made by the n-th copy does not agree with the start point of the figure to be made by the next (n + 1) copy, the figure is moved from the end point to the start point, then copying is started. (Generally, this disagreement occurs if an incorrect angular displacement or shift is specified.)

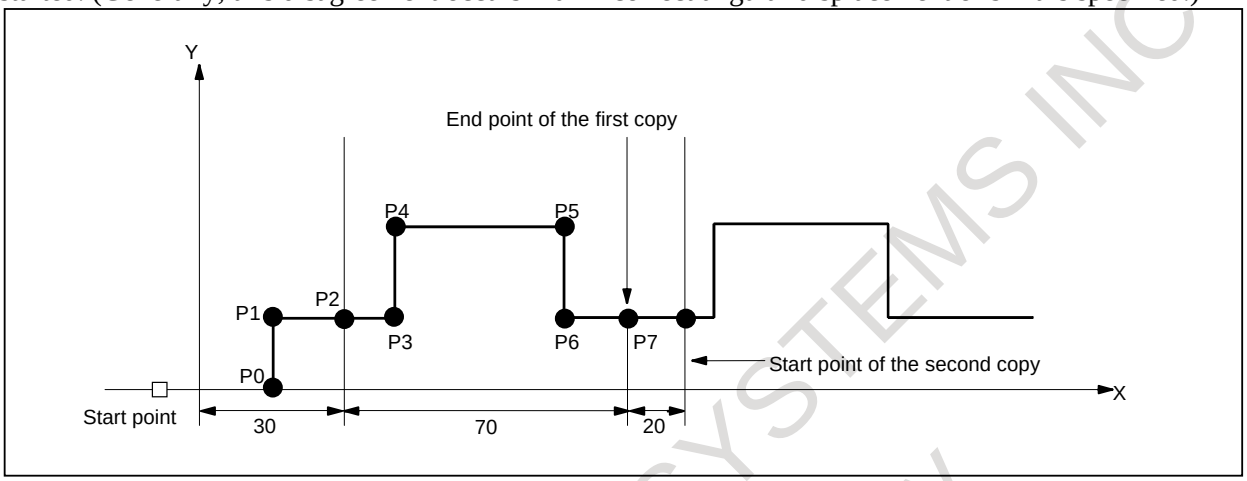


Fig. 14.1 (a)

```
Main program
O1000 ;
N10 G92 X-20.0 Y0.0 ;
N20 G00 G90 X0.0 Y0.0 ;
N30 G01 X20.0 Y0.0 F10 ;        (P0)
N40 Y20.0 ;                     (P1)
N50 X30.0 ;                     (P2)
N60 G72.2 P2000 L3 I90.0 J0.0 ;
```

Although a shift of 70 mm was required, I90.0 was specified instead of I70.0. Since an incorrect shift was specified, the end point of the figure made by the n-th copy disagrees with the start point of the figure to be made by the next (n + 1) copy.

```
Subprogram
O2000 G90 G01 X40.0 ;           (P3)
N100 Y40.0 ;                    (P4)
N200 X80.0 ;                    (P5)
N300Y20.0 ;                     (P6)
N400 X100.0 ;                   (P7)
N500 M99;
```

- Incremental command after figure copying

If the move command specified after figure copying is not absolute but incremental one, the incremental command operation depends on the setting of bit 2 (FCI) of parameter No. 11279.

(1) If bit 2 (FCI) of parameter No. 11279 is set to 0
The starting point for the incremental command is the end point of the figure made by the first copy.

(2) If bit 2 (FCI) of parameter No. 11279 is set to 1 (FS16*i* compatible)
The starting point for the incremental command is the end point of the figure made by the last copy.

FRYER MACHINE SYSTEMS INC
Rick Zahensky

Though the incremental command is specified after the following figure copying, the starting point for the incremental command is the end point of the figure made by the first copy regardless of the setting of bit 2 (FCI) of parameter No. 11279.

- Rotational copying specified in a subprogram for a linear copying
- Linear copying specified in a subprogram for a rotational copying

Limitation

- Specifying two or more commands to copy a figure

G72.1 cannot be specified more than once in a subprogram for making a rotational copying (If this is attempted, alarm PS0308, “G72.1 NESTING ERROR” will occur). G72.2 cannot be specified more than once in a subprogram for making a linear copying (If this is attempted, alarm PS0309, “G72.2 NESTING ERROR” will occur).

- Commands that must not be specified

Within a program that performs a rotational or linear copying, the following must not be specified:

- Command for changing the selected plane (G17 to G19)
- Command for specifying polar coordinates
- Reference position return command
- Coordinate system rotation, programmable mirror image
  The command for rotational or linear copying can be specified after a command for coordinate system rotation or programmable mirror image is executed.

M

- Scaling
  The command for rotational or linear copying can be specified after a command for scaling image is executed.

- Modes that must not be selected

Figure copying cannot be specified in the following modes.

- Chamfering/Corner R
- Tool offset
- Tilted working plane indexing command
- 3-dimensional coordinate system conversion

T

- Polar coordinate interpolation

- Unit system

The two axes of the plane for copying a figure must have an identical unit system.

- Single block

Single-block stops are not performed in a block with G72.1 or G72.2.

- Specifying tool radius compensation and the workpiece coordinate system

In a subprogram for copying a figure, the G code for tool radius / tool nose radius compensation or compensation amount (H or D code) cannot be changed. G92 and G54 to G59 cannot be changed either. Those codes must be specified before figure copying is started.

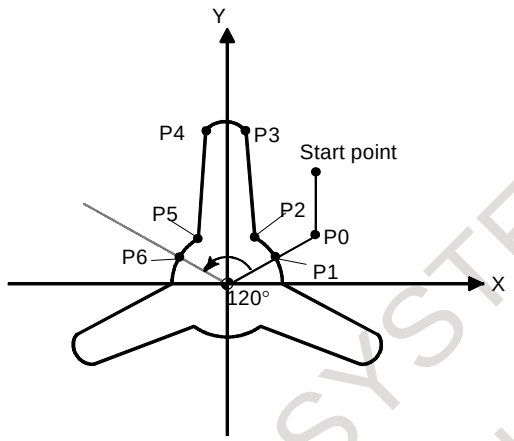
FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Copy axially excluding the axis direction of plane selection

Rotational copying and Linear copying for the copy axially excluding the axis direction of plane selection cannot be executed. At the Rotational copying, the rotation center axis command excluding the axis direction of plane selection (for example, Z command in Xp-Yp plane (specified by G17)) is ignored. At the Linear copying, the shift along the axis excluding the axis direction of plane selection (for example, K command in Xp-Yp plane (specified by G17)) is ignored.

Example

- Rotational copying



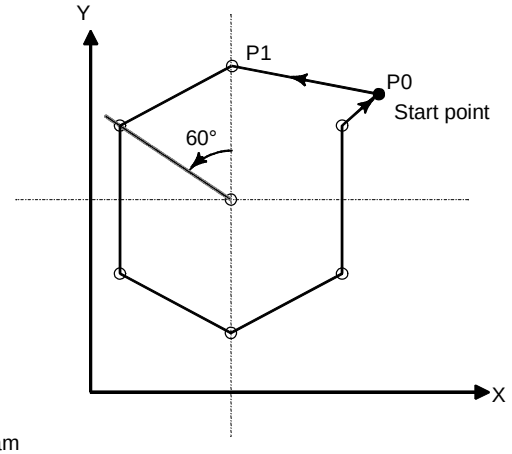

```
Main program
  O1000 ;
  N10 G92 X40.0 Y50.0 ;
  N20 G00 G90 X_ Y_ ;                    (P0)
  N30 G01 G17 G41 X_ Y_ D01 F10 ;        (P1)
  N40 G72.1 P2000 L3 X0 Y0 R120.0 ;
  N50 G40 G01 X_ Y_ I_ J_ ;              (P0)
  N60 G00 X40.0 Y50.0 ;
  N70 M30 ;

Sub program
  O2000 G03 X_ Y_ R30.0 ;                (P2)
  N100 G01 X_ Y_ ;                       (P3)
  N200 G03 X_ Y_ R10.0 ;                 (P4)
  N300 G01 X_ Y_ ;                       (P5)
  N400 G03 X_ Y_ R30.0 ;                 (P6)
  N500 M99;
```

Fig. 14.1 (b)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Rotational copying (spot boring)



Main program

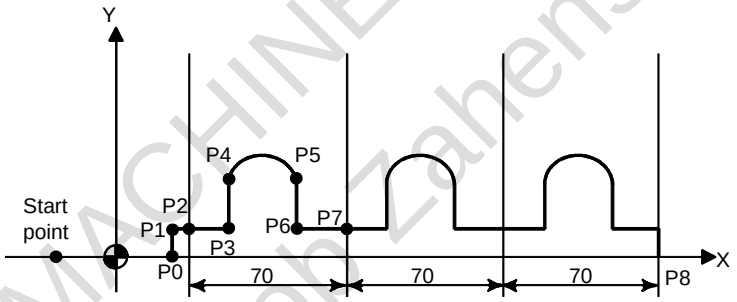
```
O3000 ;
N10 G92 G17 X80.0 Y50.0 ;                (P0)
N20 G72.1 P4000 L6 X0 Y0 R60.0 ;
N30 G80 G00 X80.0 Y50.0 ;                (P0)
N40 M30 ;
```

Subprogram

```
O4000 N100 G90 G81 X_ Y_ R_ Z_ F_ ;      (P1)
N200 M99 ;
```

Fig. 14.1 (c)

- Linear copying



Main program

```
O1000 ;
N10 G92 X-20.0 Y0 ;
N20 G00 G90 X0 Y0 ;
N30 G01 G17 G41 X20. Y0 D01 F10 ;    (P0)
N40 Y20. ;                           (P1)
N50 X30. ;                           (P2)
N60 G72.2 P2000 L3 I70.0 J0 ;
N70 X_ Y_ ;                          (P8)
N80 X0 ;
N90 G00 G40 X-20.0 Y0 ;
N100 M30 ;
```

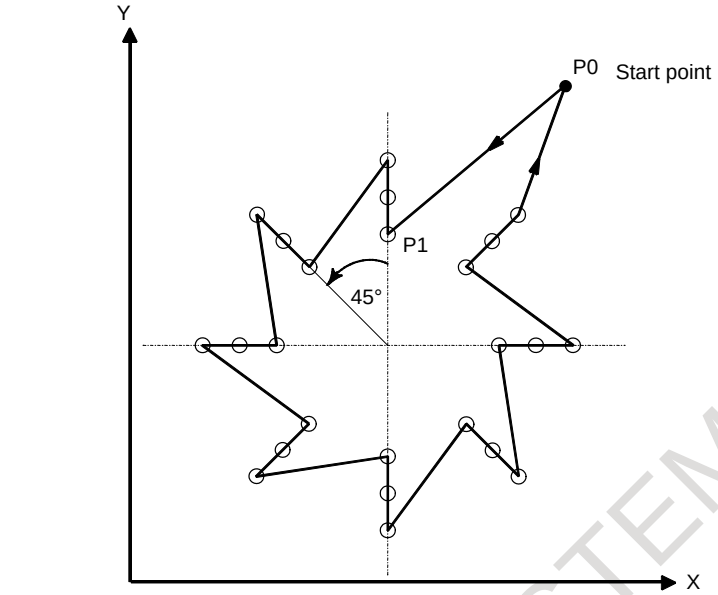
Subprogram

```
O2000 G90 G01 X_ ;                   (P3)
N100 Y_ ;                            (P4)
N200 G02 X_ I_ ;                     (P5)
N300 G01 Y_ ;                        (P6)
N400 X_ ;                            (P7)
N500 M99 ;
```

Fig. 14.1 (d)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- **Combination of rotational copying and linear copying (bolt hole circle)**



Main program

```
O1000 ;
N10 G92 G17 X100.0 Y80.0 ;                (P0)
N20 G72.1 P2000 X0 Y0 L8 R45.0 ;
N30 G80 G00 X100.0 Y80.0 ;                (P0)
N40 M30 ;
```

Subprogram (rotational copy)

```
O2000 N100 G72.2 P3000 I0 J_ L3 ;
N200 M99 ;
```

Subprogram (linear copy)

```
O3000 N110 G90 G81 X_ Y_ R_ Z_ F_ ;       (P1)
N210 M99 ;
```

Fig. 14.1 (e)

14.2 3-DIMENSIONAL COORDINATE SYSTEM CONVERSION

Coordinate system conversion about an axis can be carried out if the center of rotation, direction of the axis of rotation, and angular displacement are specified. This function is very useful in 3-dimensional machining by a die-sinking machine or similar machine. For example, if a program specifying machining on the XY plane is converted by the 3-dimensional coordinate system conversion function, the identical machining can be executed on a desired plane in 3-dimensional space.

- For milling machining

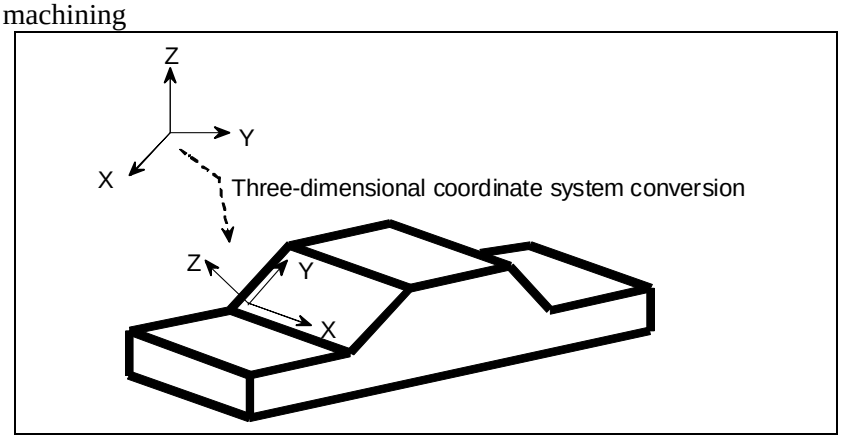


Fig. 14.2 (a)

FFYER MACHINE SYSTEMS INC
Rob Zahensky

- For lathe cutting

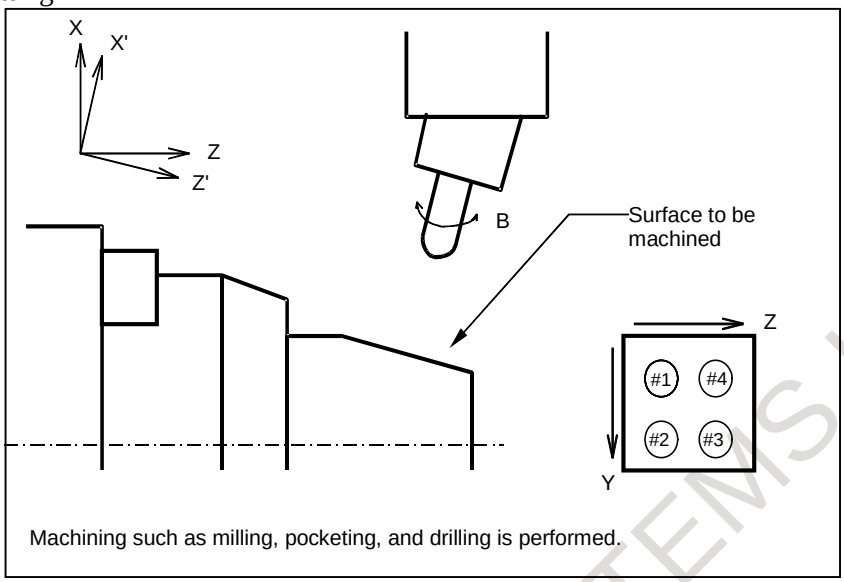


Fig. 14.2 (b)

Format

M

G68 Xp$_{x1}$ Yp$_{y1}$ Zp$_{z1}$ I$_{i1}$ J$_{j1}$ K$_{k1}$ Rα ; Starting 3-dimensional coordinate system conversion

:
: } 3-dimensional coordinate system conversion mode

G69 ; Canceling 3-dimensional coordinate system conversion

Xp, Yp, Zp : Center of rotation (absolute coordinates) on the X, Y, and Z axis or parallel axes
I, J, K : Direction of the axis of rotation
R : Angular displacement

T

G68.1 Xp$_{x1}$ Yp$_{y1}$ Zp$_{z1}$ I$_{i1}$ J$_{j1}$ K$_{k1}$ Rα ; Starting 3-dimensional coordinate system conversion

:
: } 3-dimensional coordinate system conversion mode

G69.1 ; Canceling 3-dimensional coordinate system conversion

Xp, Yp, Zp : Center of rotation (absolute coordinates) on the X, Y, and Z axis or parallel axes
I, J, K : Direction of the axis of rotation
R : Angular displacement

NOTE
The G code of this function is hereinafter described using the format (G68/G69) for the machining center system in this section.

FRYER MACHINE SYSTEMS INC
Rob Zaborsky

Explanation

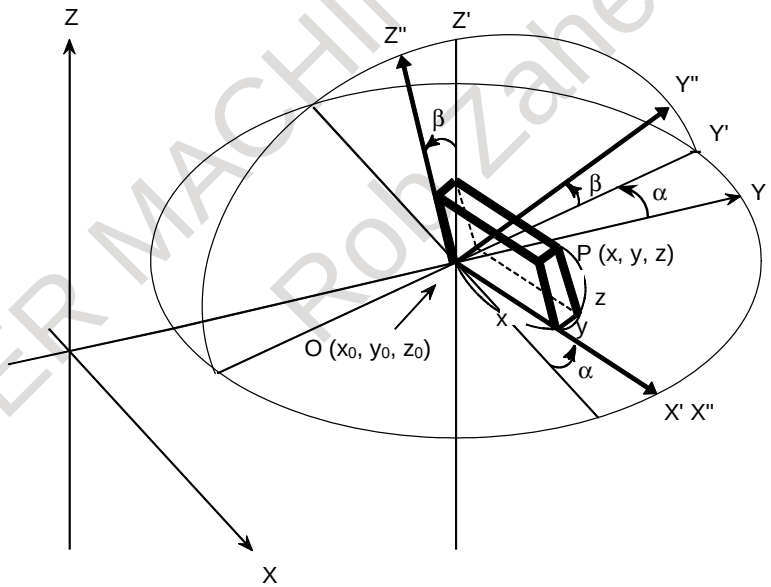
- Command for 3-dimensional coordinate system conversion (program coordinate system)

N1 G68 Xp x_1 Yp y_1 Zp z_1 I i_1 J j_1 K k_1 R α ;
N2 G68 Xp x_2 Yp y_2 Zp z_2 I i_2 J j_2 K k_2 R β ;
N3
:
Nn G69 ;

3-dimensional coordinate system conversion can be executed twice.
In the N1 block, specify the center, direction of the axis of rotation, and angular displacement of the first rotation.

When this block is executed, the center of the original coordinate system is shifted to (x_1, y_1, z_1), then rotated around the vector (i_1, j_1, k_1) by angular displacement α. The new coordinate system is called X'Y'Z'. In the N2 block, specify the center, direction of the axis of rotation, and angular displacement of the second rotation. In the N2 block, specify coordinates and the angle with the coordinate system formed after the N1 block in Xp, Yp, Zp, I, J, K, and R. When the N2 block is executed, the X'Y'Z' coordinate system is shifted to (x_2, y_2, z_2), then rotated around the vector (i_2, j_2, k_2) by angular displacement β. The newest coordinate system is called X"Y"Z". In the subsequent N3 block, coordinates in the X"Y"Z" coordinate system are specified with Xp, Yp, and Zp. The X"Y"Z" coordinate system is called the program coordinate system.
If (Xp, Yp, Zp) is not specified in the N2 block, (Xp, Yp, Zp) in the N1 block is assumed to be the center of the second rotation (the N1 and N2 blocks have a common center of rotation). If the coordinate system is to be rotated only once, the N2 block need not be specified.

Example) G68 Xx_0 Yy_0 Zz_0 I0 J0 K1 Rα ;
G68 I1 J0 K0 Rβ ;



X, Y, Z	:	Workpiece coordinate system
X', Y', Z'	:	Coordinate system formed after the first conversion
X", Y", Z"	:	Coordinate system formed after the second conversion
α	:	Angular displacement of the first rotation
β	:	Angular displacement of the second rotation
O (x_0, y_0, z_0)	:	Center of rotation
P (x, y, z)	:	Coordinates in the X"Y"Z" coordinate system (program coordinate system)

Fig. 14.2 (c)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Format error

If one of the following format errors is detected, alarm PS5044, “G68 FORMAT ERROR” occurs:

1. When I, J, or K is not specified in a block with G68
2. When I, J, and K are all set to 0 in a block with G68
3. When R is not specified in a block with G68

- Center of rotation

Specify absolute coordinates with Xp, Yp, and Zp in the G68 block.

- Equation for 3-dimensional coordinate system conversion

The following equation shows the general relationship between (x, y, z) in the program coordinate system and (X, Y, Z) in the original coordinate system (workpiece coordinate system).

$$\begin{pmatrix} X \\ Y \\ Z \end{pmatrix} = (M_1)\begin{pmatrix} x \\ y \\ z \end{pmatrix} + \begin{pmatrix} x_1 \\ y_1 \\ z_1 \end{pmatrix}$$

When conversion is carried out twice, the relationship is expressed as follows:

$$\begin{pmatrix} X \\ Y \\ Z \end{pmatrix} = (M_1)(M_2)\begin{pmatrix} x \\ y \\ z \end{pmatrix} + (M_1)\begin{pmatrix} x_2 \\ y_2 \\ z_2 \end{pmatrix} + \begin{pmatrix} x_1 \\ y_1 \\ z_1 \end{pmatrix}$$

X, Y, Z : Coordinates in the original coordinate system (workpiece coordinate system)
x, y, z : Programmed value (coordinates in the program coordinate system)
x_1, y_1, z_1 : Center of rotation of the first conversion
x_2, y_2, z_2 : Center of rotation of the second conversion
(coordinates in the coordinate system formed after the first conversion)
M_1 : First conversion matrix
M_2 : Second conversion matrix

M_1 and M_2 are conversion matrices determined by an angular displacement and rotation axis. Generally, the matrices are expressed as shown below.

$$\begin{pmatrix} n_1^2 + (1-n_1^2)\cos\theta & n_1 n_2 (1-\cos\theta) - n_3 \sin\theta & n_1 n_3 (1-\cos\theta) + n_2 \sin\theta \\ n_1 n_2 (1-\cos\theta) + n_3 \sin\theta & n_2^2 + (1-n_2^2)\cos\theta & n_2 n_3 (1-\cos\theta) - n_1 \sin\theta \\ n_1 n_3 (1-\cos\theta) - n_2 \sin\theta & n_2 n_3 (1-\cos\theta) + n_1 \sin\theta & n_3^2 + (1-n_3^2)\cos\theta \end{pmatrix}$$

n_1 : Cosine of the angle made by the rotation axis and X-axis i/p
n_2 : Cosine of the angle made by the rotation axis and Y-axis j/p
n_3 : Cosine of the angle made by the rotation axis and Z-axis k/p
θ : Angular displacement

Value p is obtained by the following:

$$p = \sqrt{i^2 + j^2 + k^2}$$

Conversion matrices for rotation on two-dimensional planes are shown below:

(1) Coordinate system conversion on the XY plane

$$M = \begin{pmatrix} \cos\theta & -\sin\theta & 0 \\ \sin\theta & \cos\theta & 0 \\ 0 & 0 & 1 \end{pmatrix}$$

FRYER MACHINE SYSTEMS INC
Rob Zahensky

(2) Coordinate system conversion on the YZ plane

$$M = \begin{pmatrix} 1 & 0 & 0 \\ 0 & \cos\theta & -\sin\theta \\ 0 & \sin\theta & \cos\theta \end{pmatrix}$$

(3) Coordinate system conversion on the ZX plane

$$M = \begin{pmatrix} \cos\theta & 0 & \sin\theta \\ 0 & 1 & 0 \\ -\sin\theta & 0 & \cos\theta \end{pmatrix}$$

In the starting block of 3-dimensional coordinate system conversion (G68 : in machining center system, G68.1 : in lathe system), the coordinates(x,y,z) in the program coordinate system are generated from the coordinates(X,Y,Z) in the original coordinate system(workpiece coordinate system) by the inverse of the above conversion equation.
The generated coordinates(x,y,z) is the starting point for following commands in the program coordinate system.

- Three basic axes and their parallel axes

3-dimensional coordinate system conversion can be applied to a desired combination of three axes selected out of the basic three axes (X, Y, Z) and their parallel axes. The 3-dimensional coordinate system subjected to 3-dimensional coordinate system conversion is determined by axis addresses specified in the G68 block. If Xp, Yp, or Zp is not specified, X, Y, or Z of the basic three axes is assumed. However, if the basic three axes are not specified in parameter No. 1022, alarm PS0048, “BASIC 3 AXIS NOT FOUND” occurs.
In a single G68 block, both a basic axis and a parallel axis cannot be specified.
If this is attempted, alarm PS0047, “ILLEGAL AXIS SELECT” occurs.
(Example)

When U-axis, V-axis, and W-axis are parallel to the X-axis, Y-axis, and Z-axis respectively
(If in lathe system, assume that G code system B or C is used.)

```
G68   X_ I_ J_ K_ R_ ;        XYZ coordinate system
G68   U_V_ Z_ I_ J_ K_ R_ ;   UVZ coordinate system
G68   W_ I_ J_ K_ R_ ;        XYW coordinate system
```

- Specifying the second conversion

3-dimensional coordinate system conversion can be executed twice. The center of rotation of the second conversion must be specified with the axis addresses specified for the first conversion. If the axis addresses of the second conversion are different from the axis addresses of the first conversion, the different axis addresses are ignored. An attempt to execute 3-dimensional coordinate system conversion three or more times causes alarm PS5043, “TOO MANY G68 NESTING”.

- Angular displacement R

A positive angular displacement R indicates a clockwise rotation along the axis of rotation. Specify angular displacement R in 0.001 degrees within the range of -360000 to 360000.
To specify angular displacement R in 0.00001 degrees (one hundred-thousandth), set bit 0 (FRD) of parameter No. 11630 to 1. In this case, angular displacement R is specified within the range of -36000000 to 36000000.

- G codes that can be specified

The following G codes can be specified in the 3-dimensional coordinate system conversion mode:

G00 Positioning
G01 Linear interpolation
G02 Circular interpolation / Helical interpolation (clockwise)

FRYER MACHINE SYSTEMS INC
Robo Zahensky

G03 Circular interpolation / Helical interpolation (counterclockwise)
G04 Dwell
G10 Data setting
G17 Plane selection (XY)
G18 Plane selection (ZX)
G19 Plane selection (YZ)
G28 Reference position return
G29 Movement from the reference position
G30 Second, third, or fourth reference position return
G31 Skip function
G53 Selecting the machine coordinate system
G65 Custom macro call
G66 Custom macro modal call
G67 Custom macro modal call cancel
G40 Canceling tool radius / tool nose radius compensation
G41 Tool radius / tool nose radius compensation to the left
G42 Tool radius / tool nose radius compensation to the right
G73, G74, G76, G80 to G89 Canned cycle for drilling

M

G43 Increasing tool length compensation
G44 Decreasing tool length compensation
G45 Increasing the tool offset
G46 Decreasing the tool offset
G47 Doubling the tool offset
G48 Halving the tool offset
G49(G49.1)Canceling tool length compensation
G50.1 Canceling programmable mirror image
G51.1 Programmable mirror image
G90 Absolute programming
G91 Incremental programming
G94 Feed per minute
G95 Feed per revolution
G98 Canned cycle (return to the initial level)
G99 Canned cycle (return to the level of point R)

T

G12.1 Polar coordinate interpolation
G90 Absolute programming (when G code system B or C is used.)
G91 Incremental programming (when G code system B or C is used.)
G94 Feed per minute (when G code system B or C is used.)
G95 Feed per revolution(when G code system B or C is used.)
G98 Canned cycle (return to the initial level) (when G code system B or C is used.)
G99 Canned cycle (return to the level of point R) (when G code system B or C is used.)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Rapid traverse in a canned cycle for drilling

By using bit 1 (D3R) of parameter No. 11221, rapid traverse in the drilling direction in a canned cycle for drilling in the tilted working plane indexing mode or 3-dimensional coordinate conversion mode can be selected and the rapid traverse mode for shift operation in fine boring (G76) or back boring (G87) for the M system can also be selected. When bit 1 (D3R) of parameter No. 11221 is set to 0, the cutting feed mode is selected; when it is set to 1, the rapid traverse mode is selected. For positioning to the initial level at the start of a canned cycle for drilling, however, the rapid traverse mode is used, regardless of whether bit 1 (D3R) of parameter No. 11221 is set to 0 or 1. Rapid traverse override can be applied in the rapid traverse mode, and cutting feed override can be applied in the cutting feed mode. Bit 1 (D3R) of parameter No. 11221 can also be used to change the mode of rapid traverse in the rigid tapping direction.

For acceleration/deceleration after interpolation

When bit 1 (D3R) of parameter No. 11221 is set to 1 (for the rapid traverse mode), rapid traverse in the drilling direction in a canned cycle for drilling in the tilted working plane indexing mode or 3-dimensional coordinate conversion mode compensation mode is accelerated/decelerated along each axis independently according to the time constants specified in parameters Nos. 1620 and 1621. So, when performing positioning of linear interpolation type with bit 1 (D3R) of parameter No. 11221 set to 1, make the following settings:

Bit 1 of (LRP) of parameter No. 1401=1	Selects positioning of linear interpolation type.
Bit 4 of (PRT) of parameter No. 1603=1	Selects acceleration/deceleration of constant time type.
Parameter No. 1620	Set the same value for all.
Parameter No. 1621	Set the same value for all.

For look-ahead acceleration/deceleration before interpolation

In a canned cycle for drilling, linear rapid traverse in the drilling direction is performed at all times.

Feedrate

When bit 1 (D3R) of parameter No. 11221 is set to 1 (for the rapid traverse mode), the rapid traverse rate in the drilling direction in a canned cycle for drilling in the tilted working plane indexing mode or 3-dimensional coordinate conversion mode is clamped to a parameter value indicated below if the rapid traverse override is 100%. In the rapid traverse override F0 mode, however, the rapid traverse rate is clamped to the value of parameter No. 1421 or a parameter value listed in Table 14.2 (a), whichever smaller.

Table 14.2 (a) Feedrate in the rapid traverse mode

Parameter No. 5412=0	Parameter No. 1420
Parameter No. 5412≠0	Value of parameter No. 1420 or value of parameter No. 5412, whichever smaller

However, when the external deceleration function is enabled, the feedrate is clamped to a parameter value listed in Table 14.2 (a) that is overridden or an external deceleration speed, whichever smaller.

NOTE

1 Set a rapid traverse rate for each axis in parameter No. 1420 and a tangential rapid traverse rate in parameter No. 5412.
2 If rapid traverse of nonlinear interpolation type is performed, the feedrate is clamped to the value of parameter No. 1420, regardless of the setting of parameter No. 5412.
3 When the cutting feed mode is selected (when bit 1 (D3R) of parameter No. 11221 is set to 0), set a tangential rapid traverse rate in parameter No. 5412.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Compensation functions

M

If tool length compensation, tool radius / tool nose radius compensation, or tool offset is specified with 3-dimensional coordinate system conversion, compensation is performed first, followed by 3-dimensional coordinate system conversion.

T

If tool radius / tool nose radius compensation is specified with 3-dimensional coordinate system conversion, compensation is performed first, followed by 3-dimensional coordinate system conversion.

- Relationship between 3-dimensional and two-dimensional coordinate system conversion

3-dimensional and two-dimensional coordinate system conversion use identical G codes (G68 and G69). A G code specified with I, J, and K is processed as the command for 3-dimensional coordinate system conversion. A G code not specified with I, J, and K is processed as the command for two-dimensional coordinate system conversion.

- Custom macro system variables

Coordinates on the workpiece coordinate system are assigned to system variables #5041 to #5048 (current position on each axis).

- Reset

If a reset occurs during 3-dimensional coordinate system conversion mode, the mode is canceled and the continuous-state G code is changed to G69.
Bit 2 (D3R) of parameter No. 5400 determines whether just the G69 code is used to cancel the 3-dimensional coordinate system conversion mode (G68). When this setting is selected, a CNC reset by a reset operation or by an input signal from the PMC will not cancel the 3-dimensional coordinate system conversion mode.

- Absolute position display

The absolute coordinates based on the program or workpiece coordinate system can be displayed in the 3-dimensional coordinate system conversion mode. Specify a desired coordinate system in bit 6 (DAK) of parameter No. 3106.

- Distance to go display

The distance to go based on the program or workpiece coordinate system can be displayed in the 3-dimensional coordinate system conversion mode. Specify a desired coordinate system in bit 5 (D3D) of parameter No.19602.

- 3-dimensional rigid tapping

By specifying the rigid tapping command in 3-dimensional coordinate system conversion mode, tapping can be executed in the direction of the angle programmed by the 3-dimensional coordinate system conversion command.
In 3-dimensional coordinate system conversion mode, "Position Error Z", displayed on the spindle adjustment screen, is taken from the longitudinal tapping axis after 3-dimensional conversion.
Positioning in 3-dimensional coordinate system conversion mode must be linear interpolation positioning (bit 1 (LRP) of parameter No.1401 is set to 1).

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- 3-dimensional coordinate system conversion during tool length compensation

By setting bit 3 (TLC) of parameter No. 11221 to 1, 3-dimensional coordinate system conversion can be used during tool length compensation. In this case, the tool length compensation vector is converted to the program coordinate system subject to 3-dimensional coordinate system conversion. This makes machining possible while keeping the tool direction to the Z-axis direction of the workpiece coordinate system.

Tool length compensation and 3-dimensional coordinate system conversion must be nested inside each other.

(Example)

```
N10 G43 H1 X_ Y_ Z_ ;
      :
N20 G68 X0 Y0 Z0 I0.0 J1.0 K0.0 R-30.0 ;
      :
G69 ;
      :
G49 X_ Y_ Z_ ;
```

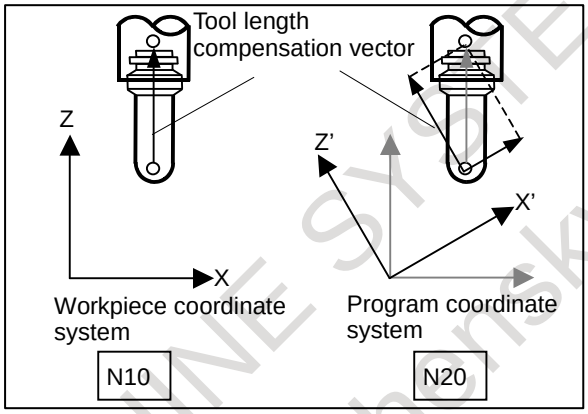


Fig. 14.2 (d)

- Cutting feedrate clamp

The cutting feedrate is clamped so that the feedrate of each real axis after the 3-dimensional coordinate system conversion does not exceed the maximum cutting feedrate（Parameter No.1432 if look-ahead acceleration/deceleration before interpolation is enabled and parameter No.1430 otherwise）.

T

- Polar coordinate interpolation in 3-dimensional coordinate system conversion

G12.1 in 3-dimensional coordinate system conversion selects a polar coordinate interpolation plane. The linear axis of polar coordinate interpolation plane is the axis of program coordinate system of 3-dimensional coordinate system conversion. Polar coordinate interpolation is performed on this plane.

FRYER MACHINE SYSTEMS INC
Rob Zaharnsky

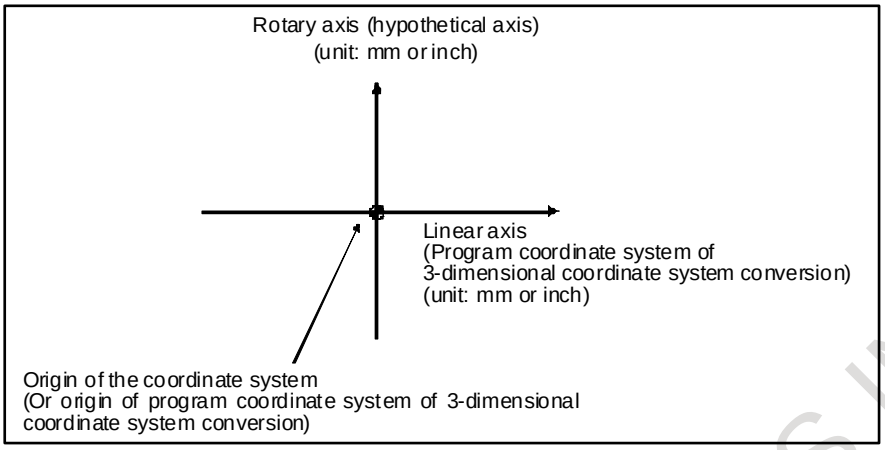


Fig. 14.2 (e) Polar coordinate interpolation plane

- Interlock signal for each axis direction, interlock signal for each axis, external deceleration signal

When bit 2 (D3IT) of parameter No.11600 is set to 1 and bit 5 (ITM) of parameter No.11223 is set to 1, the following signals are enabled for axis of machine coordinate system.

- Interlock signal for each axis direction during manual intervention / manual handle interrupt in 3-dimensional coordinate system conversion mode
- Interlock signal for each axis during manual intervention / manual handle interrupt in 3-dimensional coordinate system conversion mode
- External deceleration signal during manual intervention / manual handle interrupt in 3-dimensional coordinate system conversion mode
- External deceleration signal during 3-dimensional coordinate conversion

By this setting, all signals of controlled axis type are enabled for axis of machine coordinate system during 3-dimensional coordinate conversion.

Table14.2 (b) Relationship between coordinate system of target axis of each signal during 3-dimensional coordinate conversion (automatic operation) and parameter setting

D3IT	ITM	Interlock for each axis direction	Interlock for each axis	External deceleration
0	-	Programming coordinate system	Interlock all X, Y, Z-axis	Programming coordinate system
1	0	Machine coordinate system	Machine coordinate system	Programming coordinate system
	1	Machine coordinate system	Machine coordinate system	Machine coordinate system

Table14.2 (c) Relationship between coordinate system of target axis of each signal during manual intervention / manual handle interrupt in 3-dimensional coordinate system conversion mode and parameter setting

D3IT	ITM	Interlock for each axis direction	Interlock for each axis	External deceleration
0	-	Programming coordinate system	Programming coordinate system	Programming coordinate system
1	0	Programming coordinate system	Programming coordinate system	Programming coordinate system
	1	Machine coordinate system	Machine coordinate system	Machine coordinate system

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Limitation

- Manual intervention

3-dimensional coordinate system conversion does not affect the degree of manual intervention or manual handle interrupt.

- Positioning in the machine coordinate system

3-dimensional coordinate system conversion does not affect positioning in the machine coordinate system (e.g. specified with G28, G30, or G53).

- Specifying rapid traverse

Specify linear rapid traverse when 3-dimensional coordinate system conversion is executed. (Set bit 1 (LRP) of parameter No.1401 to 1.)

- Block with G68 or G69

In a block with G68 or G69, other G codes must not be specified. G68 must be specified with I, J, and K.

- Position display and compensation

To display the absolute position when 3-dimensional coordinate system conversion is executed, set bits 4 to 7 (DRL, DRC, DAL, and DAC) of parameters No. 3104 to 0.

- Positioning by optimum accelerations

The function for positioning by optimum accelerations is disabled during the 3-dimensional coordinate system conversion mode.

- Mirror image

M

Programmable mirror image can be specified, but external mirror image (mirror image by the mirror image signal or setting) cannot be specified. 3-dimensional coordinate system conversion is carried out after the programmable mirror image function is executed.

T

External mirror image (mirror image by the mirror image signal or setting) cannot be specified.

- 3-dimensional coordinate system conversion and other modal commands

M

Canned cycles, G41, G42, or G51.1 must be nested between G68 and G69.
(Example)

```
G68 X100.0 Y100.0 Z100.0 I0.0 J0.0 K1. R45.0 ;
    :
G41 D01 ;
    :
G40 ;
    :
G69 ;
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

T

Canned cycles, G41 or G42 must be nested between G68.1 and G69.1.
(Example)

```
G68.1 X100.0 Y100.0 Z100.0 I0.0 J0.0 K1.0R45.0 ;
    :
G41 X_ Z_ I_ K_ ;
    :
G40 ;
    :
G69.1 ;
```

T

- Relationship between 3-dimensional coordinate system conversion and tool offset

When using a tool offset command, nest the tool offset command within the 3-dimensional coordinate system conversion mode.
(Example)

```
G68.1 X100.0 Y100.0 Z100.0 I0.0 J0.0 K1.0 R45.0 ;
    :
T0101;
    :
T0100;
    :
G69.1 ;
```

When tool geometry and wear compensation with coordinate shift is enabled and it is not canceled by reset, if tool geometry and wear compensation is executed in 3-dimensional coordinate system conversion mode, 3-dimensional coordinate system conversion is not canceled by reset. (Even if it is a setting that 3-dimensional coordinate system conversion is canceled by reset (bit 2 (D3R) of parameter No. 5400 is set to 0), it is not canceled.)

- PMC axis control

In the 3-dimensional coordinate system conversion mode, PMC axis control cannot be performed for the three axes related to the conversion (alarm PS5131).

- Manual operation

When manual feeding is performed during a 3-dimensional coordinate system conversion, the tangent velocity in the coordinate system after conversion (program coordinate system) equals the lowest feedrate of those on the selected axes.

- Workpiece coordinate system

Avoid changing the workpiece coordinate system in the 3-dimensional coordinate system conversion mode.

- Manual reference position return

If manual reference position return is performed in the 3-dimensional coordinate system conversion mode, alarm PS5324 occurs. If you want to perform manual reference position return, cancel the 3-dimensional coordinate system conversion mode in advance.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Cs contour axis

When specifying the Cs contour axis and rapid traverse at the same time in the 3-dimensional coordinate system conversion mode, make a reference position return on the Cs contour axis in advance. If a reference position return is made in the first rapid traverse after the Cs contour axis is selected (bit 1 (NRF) of parameter No.3700 is set to 0), avoid specifying the reference position return command in the 3-dimensional coordinate system conversion mode.

Example

N1 G90 X0 Y0 Z0 ;	Carries out positioning to zero point H.
N2 G68 X10.0 Y0 Z0 I0 J1 K0 R30.0 ;	Forms new coordinate system X'Y'Z'.
N3 G68 X0 Y-10.0 Z0 I0 J0 K1 R-90.0 ;	Forms other coordinate system X"Y"Z". The origin agrees with (0, -10, 0) in coordinate system X'Y'Z.
N4 G90 X0 Y0 Z0 ;	Carries out positioning to zero point H" on coordinate system X"Y"Z".
N5 X10. Y10. Z0 ;	Carries out positioning to (10, 10, 0) on coordinate system X"Y"Z".

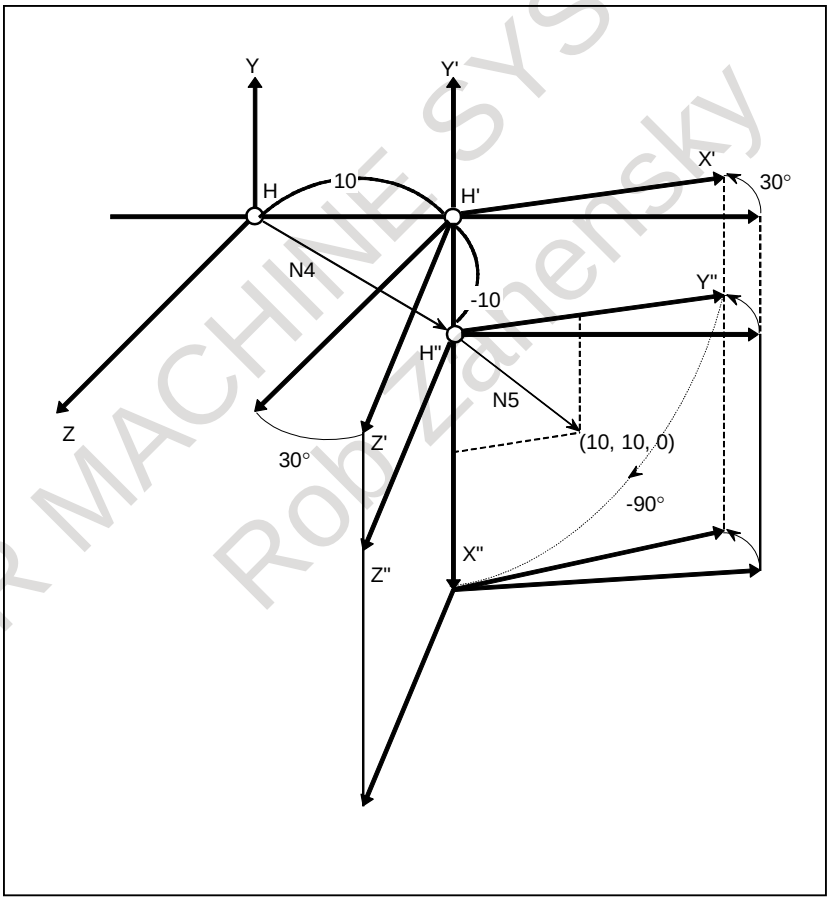


Fig. 14.2 (f)

FRYER MACHINE SYSTEMS INC
Rob Zamensky

14.3 TILTED WORKING PLANE INDEXING

14.3.1 Tilted Working Plane Indexing

Overview

Programming for creating holes, pockets, and other figures in a datum plane tilted with respect to the workpiece would be easy if commands can be specified in a coordinate system fixed to this plane (called a feature coordinate system). This function enables commands to be specified in the feature coordinate system. The feature coordinate system is defined in the workpiece coordinate system.
For explanations about the relationship between the feature coordinate system and workpiece coordinate system, see Fig. 14.3.1 (a).

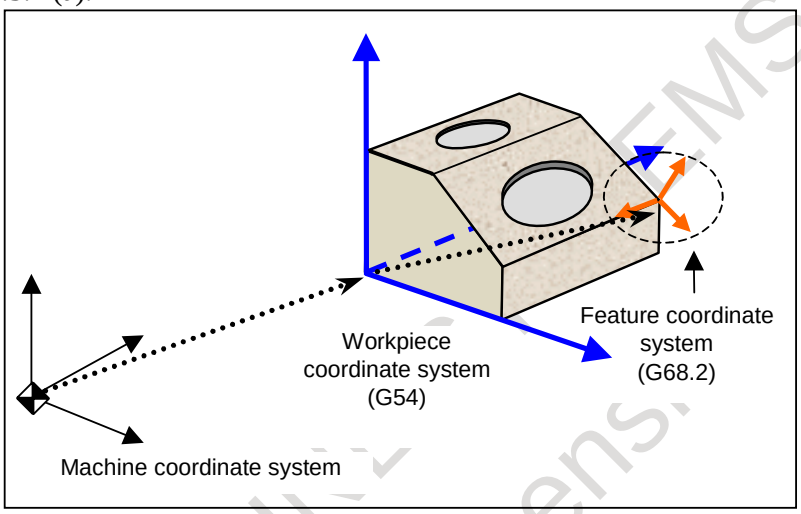


Fig. 14.3.1 (a) Feature coordinate system

The G68.2 command causes the programming coordinate system to switch to the feature coordinate system. The commands in all subsequent blocks are assumed to be specified in the feature coordinate system until G69 appears.
If G68.2 specifies the relationship between the feature coordinate system and the workpiece coordinate system, G53.1 automatically specifies the +Z direction of the feature coordinate system as the tool axis direction even if no angle is specified for the rotary axis. (See Fig. 14.3.1 (c).)
For explanations about the tool axis direction, see Fig. 14.3.1 (b).

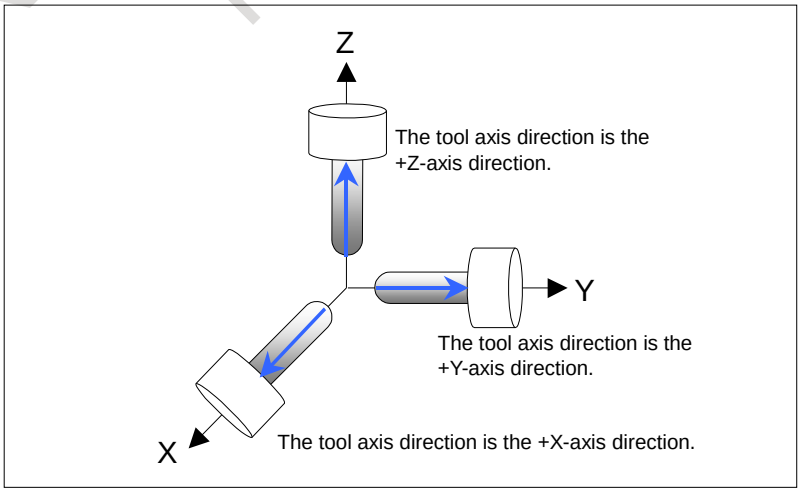


Fig. 14.3.1 (b) Tool axis direction

FRYER MACHINE SYSTEMS INC
Rob Zahens

This function regards the direction normal to the machining plane as the +Z-axis direction of the feature coordinate system. After the G53.1 command, the tool is controlled so that it remains perpendicular to the machining plane.

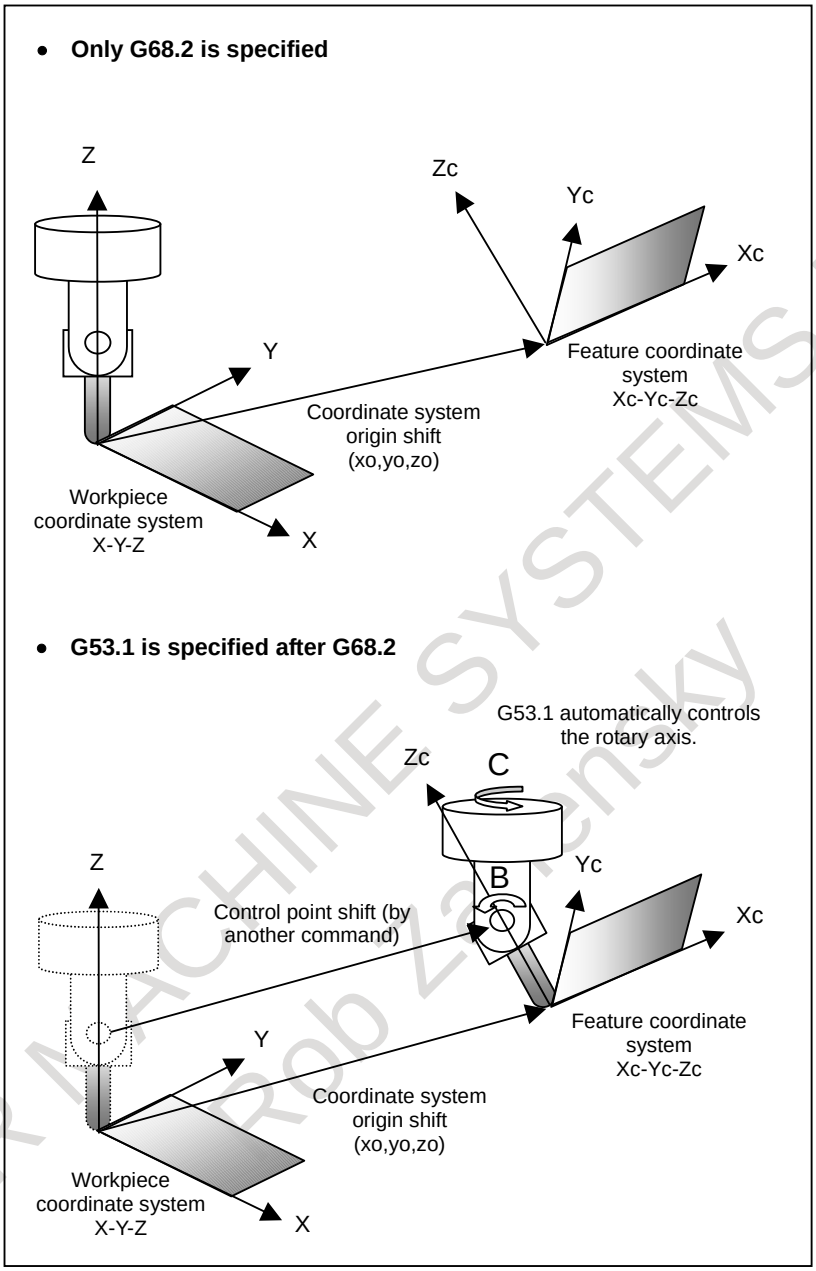


Fig. 14.3.1 (c) G68.2 and G53.1 commands

This function is applicable to the following machine configurations. (See Fig. 14.3.1 (d).)

<1> Tool rotation type machine controlled with two tool rotation axes
<2> Table rotation type machine controlled with two table rotation axes
<3> Composite type machine controlled with one tool rotation axis and one rotary axis

FRYER MACHINE SYSTEMS INC Rob Zarensky

The function can also be used for a machine configuration in which the rotary axis for controlling the tool does not intersect the rotary axis for controlling the table.

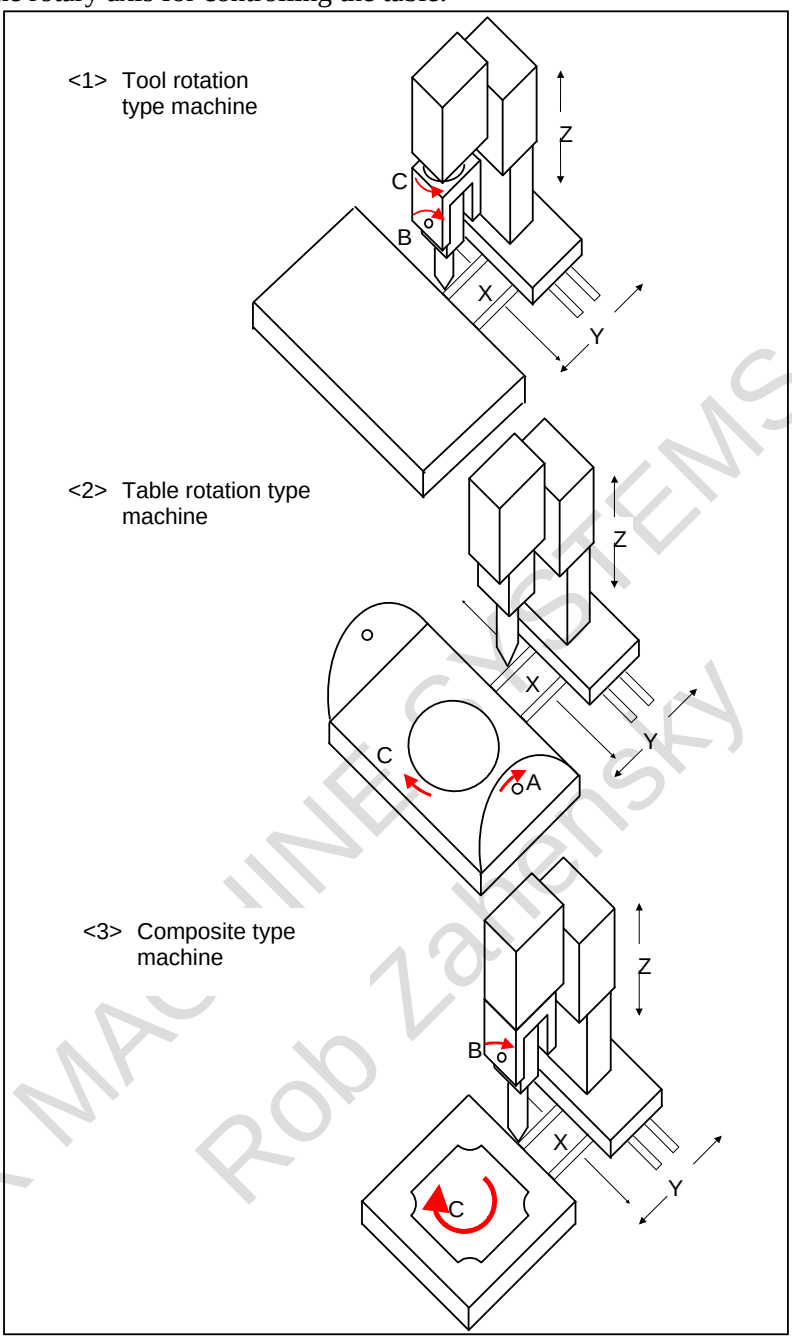


Fig. 14.3.1 (d) Three types of 5-axis machine

FRYER MACHINE SYSTEMS INC
Rob Zaltensky

14.3.1.1 Tilted working plane indexing based on Eulerian angle

Format

- **Tilted working plane indexing (G68.2)**

M

G68.2 X x_0 Y y_0 Z z_0 Iα Jβ Kγ ;	Tilted working plane indexing
G69 ;	Cancels the tilted working plane indexing.

X,Y,Z: Feature coordinate system origin
The axes specified here are the three axes of the feature coordinate system. Specify the three basic axes or parallel axes set by parameter No. 1022. When specification is omitted, the X, Y, and Z of the basic three axes are assumed to be 0.
I,J,K: Euler's angle for determining the orientation of the feature coordinate system

T

G68.2 X x_0 Y y_0 Z z_0 Iα Jβ Kγ ;	Tilted working plane indexing
G69.1 ;	Cancels the tilted working plane indexing.

X,Y,Z: Feature coordinate system origin
The axes specified here are the three axes of the feature coordinate system. Specify the three basic axes or parallel axes set by parameter No. 1022. When specification is omitted, the X, Y, and Z of the basic three axes are assumed to be 0.
I,J,K: Euler's angle for determining the orientation of the feature coordinate system

- **Tool axis direction control (G53.1)**

G53.1 ;	Controls the tool axis direction.

⚠ **CAUTION**

1 G53.1 must be specified in a block after the block that contains G68.2. An alarm occurs if G53.1 is specified without G68.2 being specified in a preceding block.
2 G53.1 must be specified in a block in which there is no other command.
3 The rotary axis moves at the maximum rapid traverse federate in the case of rapid traverse and at the specified federate in the case of cutting feed.

Explanation

- **Coordinate conversion using an Euler's angle**

Coordinate conversion by rotation is assumed to be performed around the workpiece coordinate system origin.
Let the coordinate system obtained by rotating the workpiece coordinate system around the Z-axis by an angle of α degrees be coordinate system 1. Similarly, let the coordinate system obtained by rotating coordinate system 1 around the X'-axis by an angle of β be coordinate system 2. The feature coordinate system is the coordinate system obtained by shifting the coordinate system that is obtained by rotating coordinate system 2 around the Z"-axis through an angle of γ degrees from the workpiece coordinate system origin by (Xo, Yo, Zo).
Fig. 14.3.1.1 shows the relationship between the workpiece coordinate system and the feature coordinate system.
The figure also gives examples of displacement on the X-Y plane.

FRYER MACHINE SYSTEMS INC Rob Zaharsky

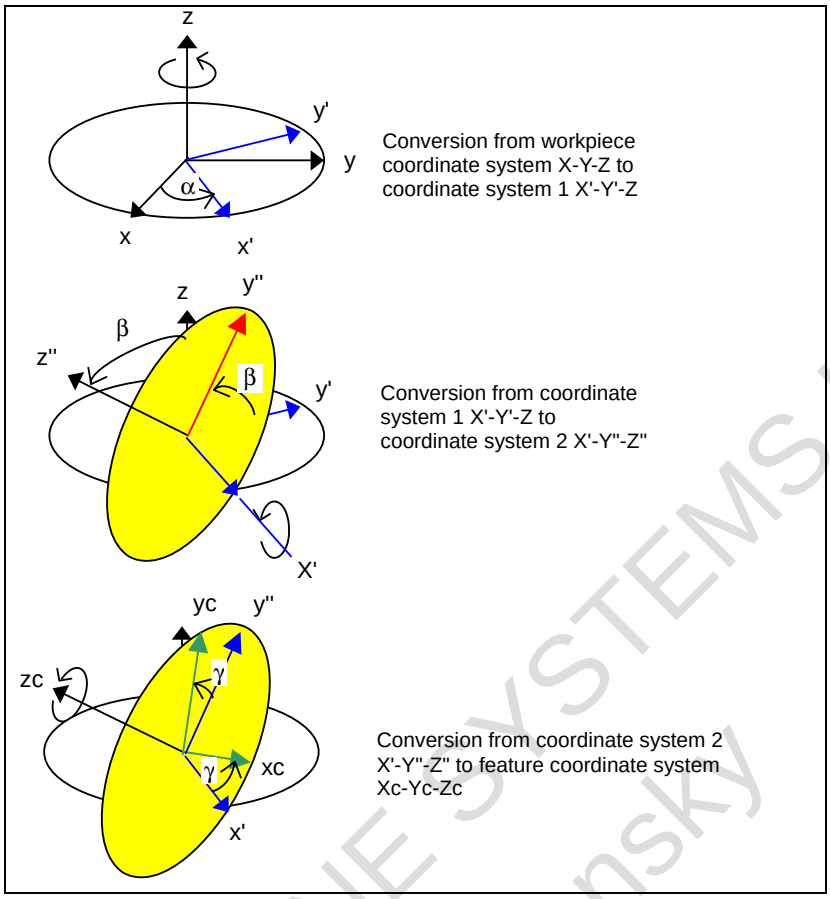


Fig. 14.3.1.1 Coordinate conversion using an Euler's angle

- I0 J0 K0 command

When I0 J0 K0 is specified as an Euler's angle, the alarm PS5457, “G68.2 FORMAT ERROR” usually occurs. When bit 1 (ATW) of parameter No. 13451 is set to 1, the feature coordinate system with a tilted angle of 0 degree is used.

14.3.1.2 General specifications of the tilted working plane indexing

- Constant surface speed control

Constant surface speed control is exercised by using, as the reference, the machine axis specified in address P in a G96 block or the machine axis (not in the feature coordinate system but in the actually operating workpiece coordinate system) set in parameter No. 3770.

- Workpiece coordinate system selection command during the tilted working plane indexing

By executing the workpiece coordinate system selection command (G54 to G59, G54.1) during the tilted working plane indexing when bit 6 (3TW) of parameter No. 1205 is 1, it is possible to change the workpiece coordinate system. In this case, the coordinate system zero point shift of the tilted working plane indexing is maintained.

If an attempt is made to execute the workpiece coordinate system selection command (G54 to G59, G54.1) during the tilted working plane indexing when bit 6 (3TW) of parameter No. 1205 is 0, alarm PS5462, ”ILLEGAL COMMAND(G68.2/G69)” is issued.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

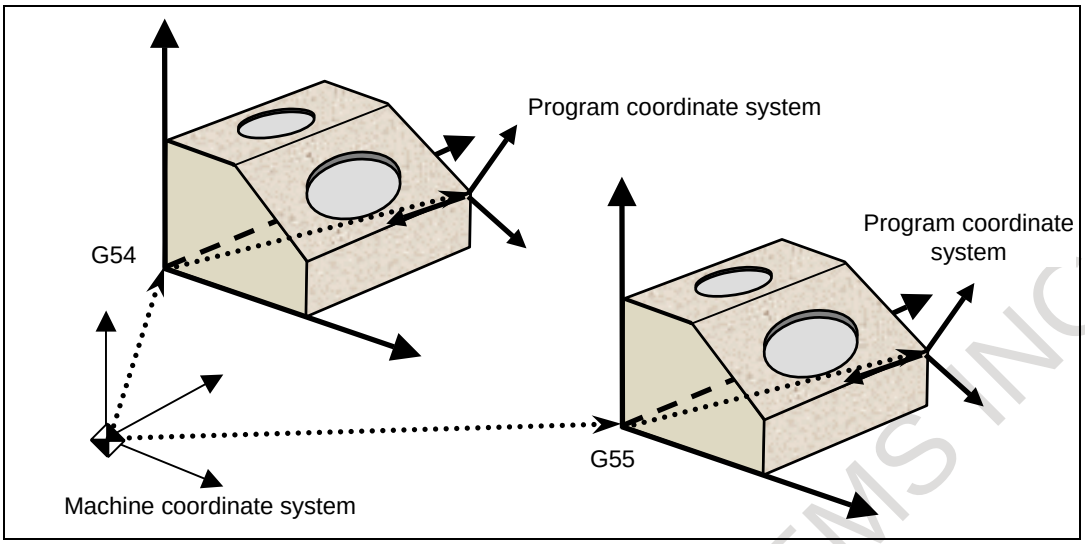


Fig. 14.3.1.2 (a)

- Minimum command unit of rotation angles

The minimum command unit of the rotation angles (I, J, K, and R) of the tilted working plane indexing is 0.001 degree regardless of the increment system. By setting bit 2 (TFR) of parameter No. 11630 to 1, the minimum command unit of the rotation angles can be set to 0.00001 degree.

- System variables of skip positions during the tilted working plane indexing

System variable number and coordinate system of skip are as Table14.3.1.2 (a). The coordinate system of #100105- and #151001- changes according to bit 5 (LV3) of parameter No.5400.

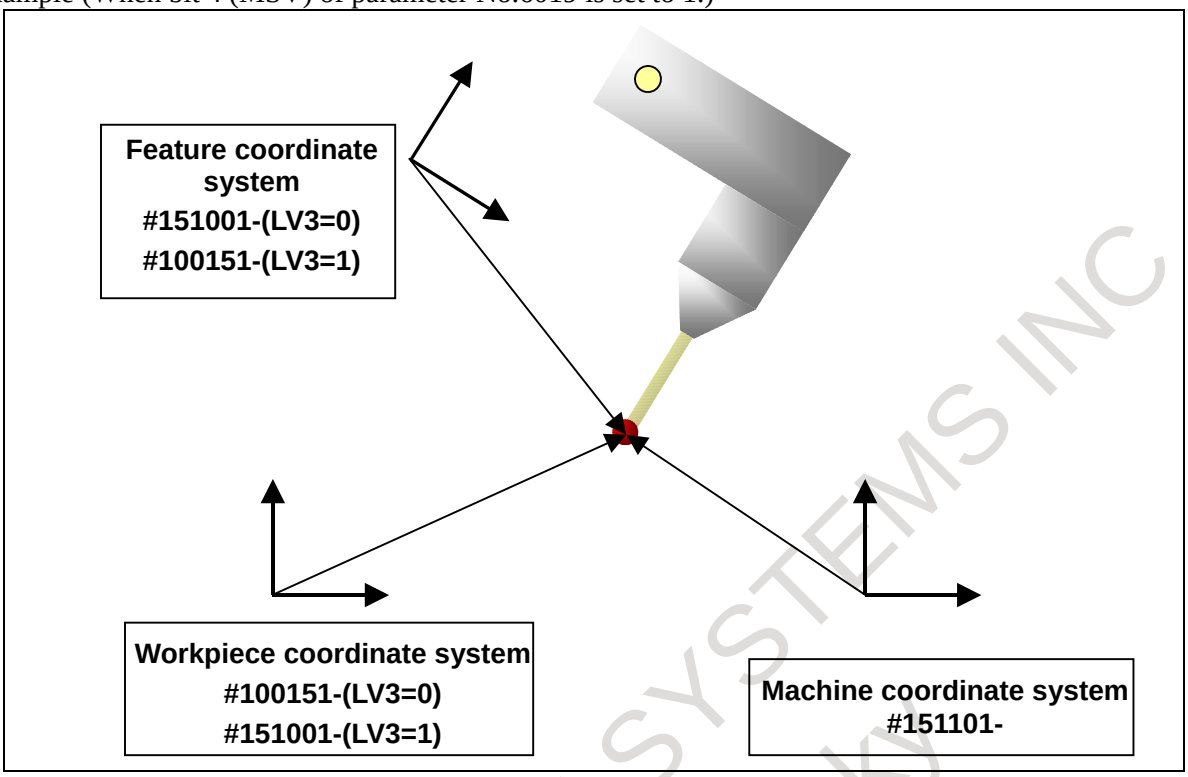
Table14.3.1.2 (a) Parameter LV3 and coordinate system of skip positions

System variable number	Coordinate system of skip positions	
	Parameter LV3=0	Parameter LV3=1
#100151 -	Workpiece coordinate system	Feature coordinate system
#151001 -	Feature coordinate system	Workpiece coordinate system
#151101 -	Machine coordinate system	

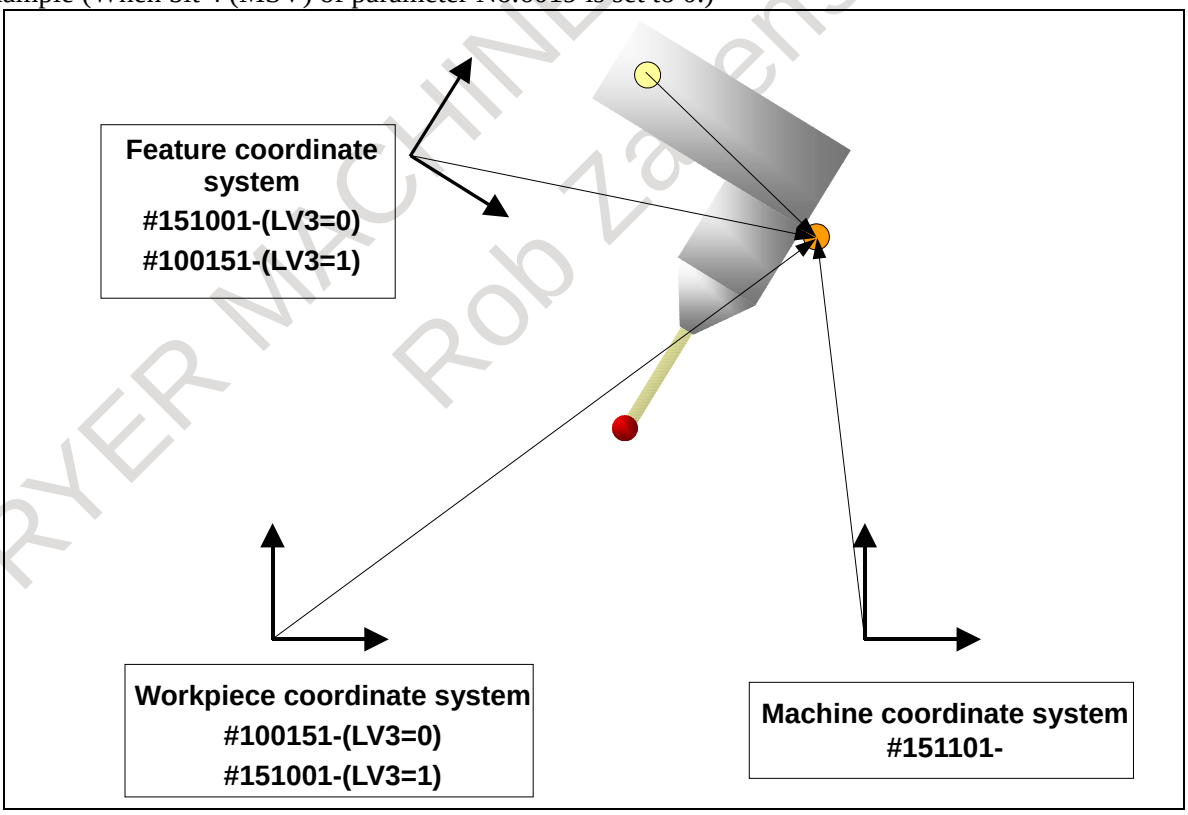
Moreover, skip positions of tool tip position can be read by setting bit 4 (MSV) of parameter No. 6019 for machining center system or bit 4 (TSV) of parameter No. 6021 for lathe system.
When parameter MSV or TSV is set to 0, system variable includes tool length compensation offset (control point position).
When parameter MSV or TSV is set to 1, system variable does not include tool length compensation offset (tool tip position).

FRIED MACHINE SYSTEMS INC
Rob Zakelsky

Example (When bit 4 (MSV) of parameter No.6019 is set to 1.)



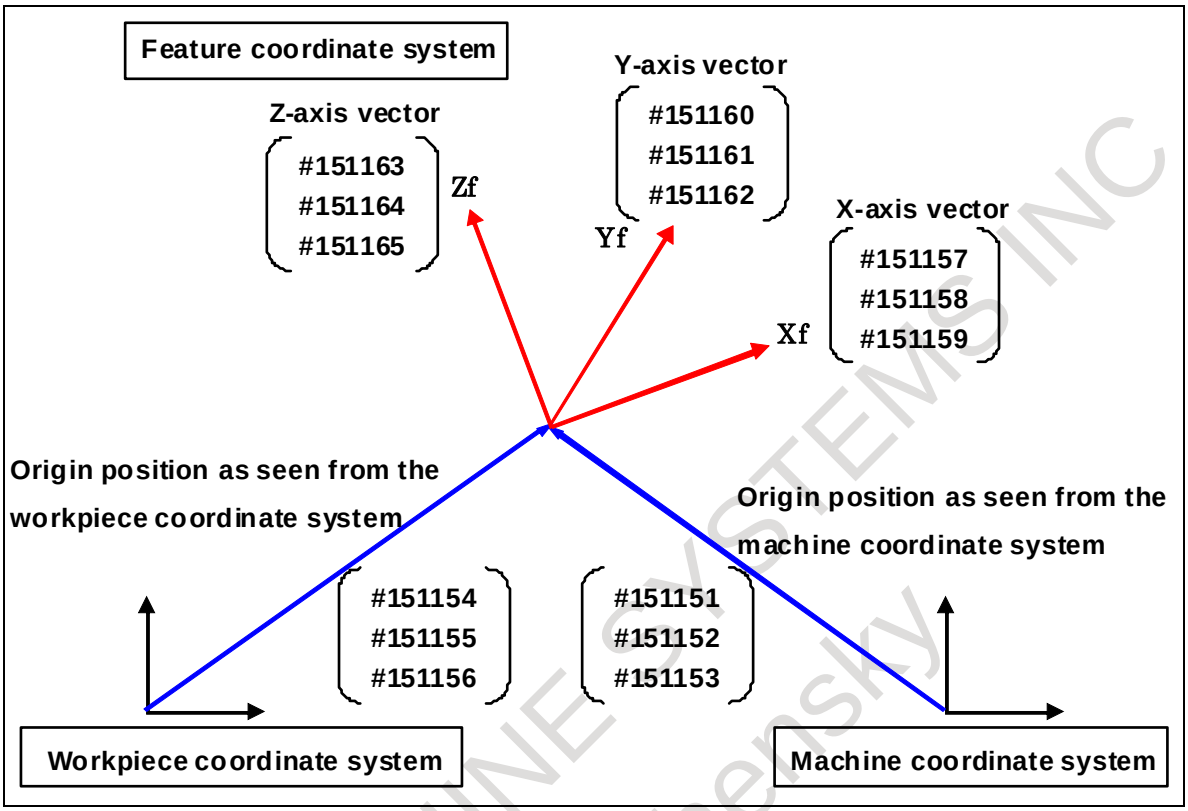
Example (When bit 4 (MSV) of parameter No.6019 is set to 0.)



FRYER MACHINE SYSTEMS INC
Rob Zalensky

- System variables of the feature coordinate system information

By using custom macro variables #151151 to #151165, the feature coordinate system information can be read.



System variable number	System variable name	Attribute	Description
#151151 to #151165	[#_FCOORD [n]]	R	The feature coordinate system information during the tilted working plane indexing. Note) Subscript n represents a compensation number (1 to 15).

R is attribute of a variable and indicates read-only.
Details of each variable are as follows.

System variable number	System variable name	Description
#151151	[#_FCOORD [1]]	Origin position X as seen from the machine coordinate system
#151152	[#_FCOORD [2]]	Origin position Y as seen from the machine coordinate system
#151153	[#_FCOORD [3]]	Origin position Z as seen from the machine coordinate system
#151154	[#_FCOORD [4]]	Origin position X as seen from the workpiece coordinate system
#151155	[#_FCOORD [5]]	Origin position Y as seen from the workpiece coordinate system
#151156	[#_FCOORD [6]]	Origin position Z as seen from the workpiece coordinate system
#151157	[#_FCOORD [7]]	X of X-axis vector as seen from the machine/workpiece coordinate system
#151158	[#_FCOORD [8]]	Y of X-axis vector as seen from the machine/workpiece coordinate system
#151159	[#_FCOORD [9]]	Z of X-axis vector as seen from the machine/workpiece coordinate system
#151160	[#_FCOORD [10]]	X of Y-axis vector as seen from the machine/workpiece coordinate system
#151161	[#_FCOORD [11]]	Y of Y-axis vector as seen from the machine/workpiece coordinate system
#151162	[#_FCOORD [12]]	Z of Y-axis vector as seen from the machine/workpiece coordinate system
#151163	[#_FCOORD [13]]	X of Z-axis vector as seen from the machine/workpiece coordinate system
#151164	[#_FCOORD [14]]	Y of Z-axis vector as seen from the machine/workpiece coordinate system
#151165	[#_FCOORD [15]]	Z of Z-axis vector as seen from the machine/workpiece coordinate system

When it is not in the tilted working plane indexing mode, all variables are set to 0.0.

FAYER MACHINE SYSTEMS INC Rob Zahensky

The length of the each axis vector on the feature coordinate system (#151157 to #151165) is 1. The vector variable is displayed by 9-digits in the decimal part

Example) Execute the block N20:O1234 of the following NC program.

```
O1234 ;
N10 G54 X0.0 Y0.0 Z0.0 ;
N20 G68.2 X5.0 Y10.0 Z15.0 I30.0 J0.0 K0.0;
```

Set the workpiece coordinate system (G54: X=100.0 Y=200.0 Z=300.0)

Specifying the tilted working plane indexing.

The feature coordinate system made in N20 is the workpiece coordinate system that shifted X5.0 Y10.0 Z15.0 in parallel and rotated around the Z-axis by angle of 30 degree. The value of each system variable at this time is as follows.

System variable number	Value	System variable number	Value	System variable number	Value	System variable number	Value	System variable number	Value
#151151	105.0	#151154	5.0	#151157	0.866025404	#151160	-0.5	#151163	0.0
#151152	210.0	#151155	10.0	#151158	0.5	#151161	0.866025404	#151164	0.0
#151153	315.0	#151156	15.0	#151159	0.0	#151162	0.0	#151165	1.0

- Minimum command unit of the rotation angles

The minimum command unit for the rotation angles (I, J, K, R) of the tilted working plane indexing is 0.001 degree regardless of the setting unit. When bit 2 (TFR) of parameter No.11630 is set to 1, the minimum command unit for the rotation angles is 0.00001 degree.

- Cutting feedrate clamp

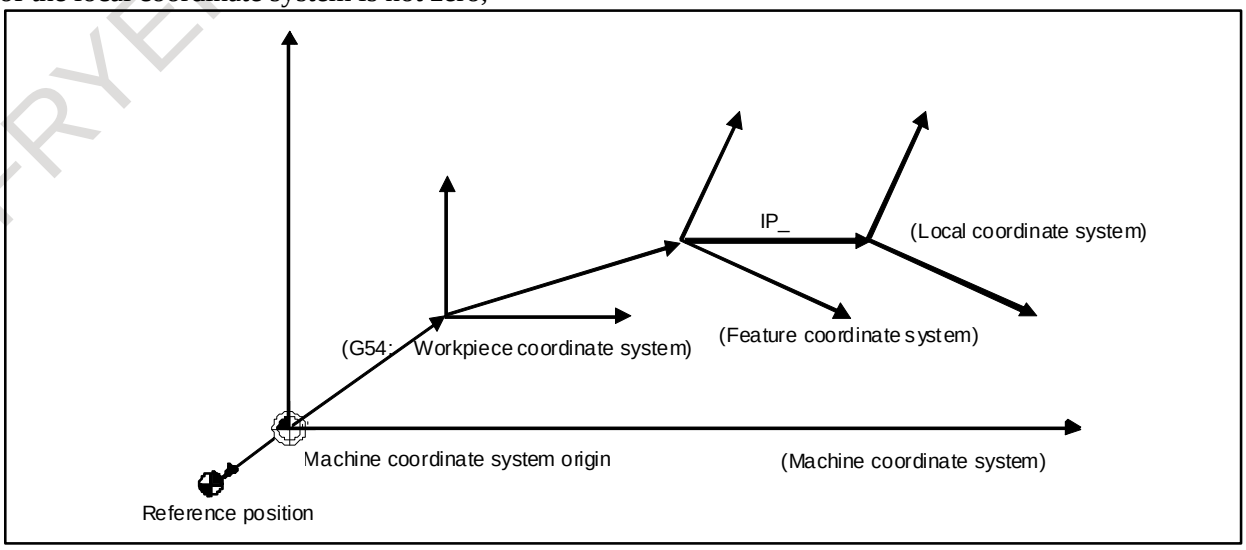
The cutting feedrate is clamped so that the feedrate of each real axis after the conversion by the tilted working plane indexing does not exceed the maximum cutting feedrate（Parameter No.1432 if look-ahead acceleration/deceleration before interpolation is enabled and parameter No.1430 otherwise).

- Local Coordinate System

The local coordinate system is available to the feature coordinate system during the tilted working plane indexing.

X,Y,Z commands of the local coordinate system defines a local coordinate system that the feature coordinate system is translated in X,Y,Z direction.

Alarm PS5462 occurs when the tilted working plane indexing is specified on the condition that the offset of the local coordinate system is not zero,



FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- **Absolute position display**

The absolute coordinates based on the program or workpiece coordinate system can be displayed during the tilted working plane indexing. Specify a desired coordinate system in bit 6 (DAK) of parameter No. 3106.

- **Distance to go display**

The distance to go based on the program or workpiece coordinate system can be displayed during the tilted working plane indexing. Specify a desired coordinate system in bit 5 (D3D) of parameter No.19602.

14.3.1.3 Tilted working plane indexing based on roll-pitch-yaw

Overview

With the tilted working plane indexing, coordinate system conversion by rotation about the X-axis, Y-axis, and Z-axis of a workpiece coordinate system in this order can be used (roll-pitch-yaw).
The order of rotary axes can be specified using address Q.

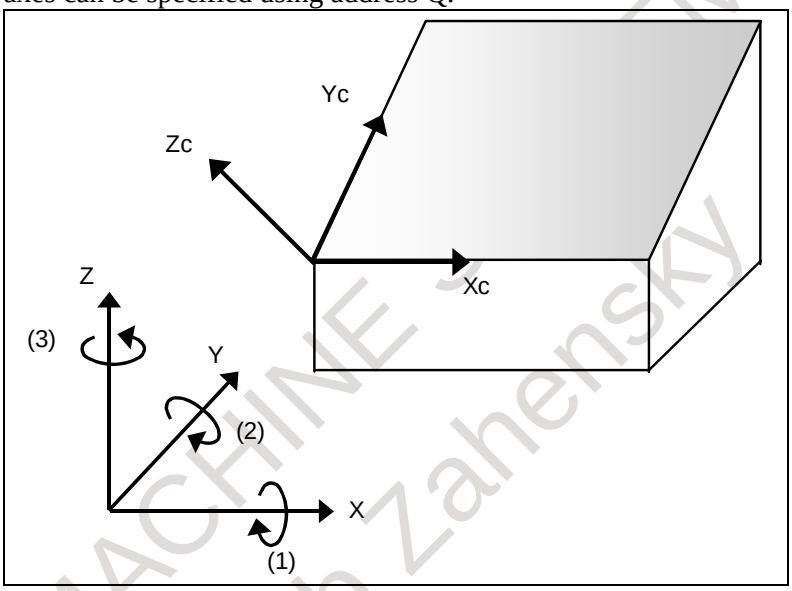


Fig. 14.3.1.3 (a)

Format

Format	
G68.2 P1 Qq X_ Y_ Z_ Iα Jβ Kγ;	Tilted working plane indexing
G69 ;	Cancel tilted working plane indexing (M series).
G69.1;	Cancel tilted working plane indexing (T series).
Explanation of symbols	
Q:	Order in which axes are rotated
X_ Y_ Z_:	Origin of a feature coordinate system
I:	Angle of rotation about the X-axis (roll angle)
J:	Angle of rotation about the Y-axis (pitch angle)
K:	Angle of rotation about the Z-axis (yaw angle)

The values of address Q and the order in which axes are rotated are shown below.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Table 14.3.1.3 (a)

	First rotation axis	Second rotation axis	Third rotation axis
Q 123	X axis	Y axis	Z axis
Q 132	X axis	Z axis	Y axis
Q 213	Y axis	X axis	Z axis
Q 231	Y axis	Z axis	X axis
Q 312	Z axis	X axis	Y axis
Q 321	Z axis	Y axis	X axis

⚠ CAUTION

1 When address Q is omitted, the X-axis, Y-axis, and Z-axis are rotated in this order. (equivalent to Q123)

2 When address Q is set to a value other than the above, alarm PS5457, “G68.2/G68.3 FORMAT ERROR” is issued.

Explanation

Suppose that the coordinate system is rotated about (1) the X-axis, (2) the Y-axis, and (3) the Z-axis in this order.

A "workpiece coordinate system" rotated by angle α about the X-axis is "coordinate system 1".

"Coordinate system 1" rotated by angle β about the Y-axis is "coordinate system 2". "Coordinate system 2" rotated by angle γ about the Z-axis then shifted by (X_0,Y_0,Z_0) from the workpiece coordinate system origin is a "feature coordinate system".

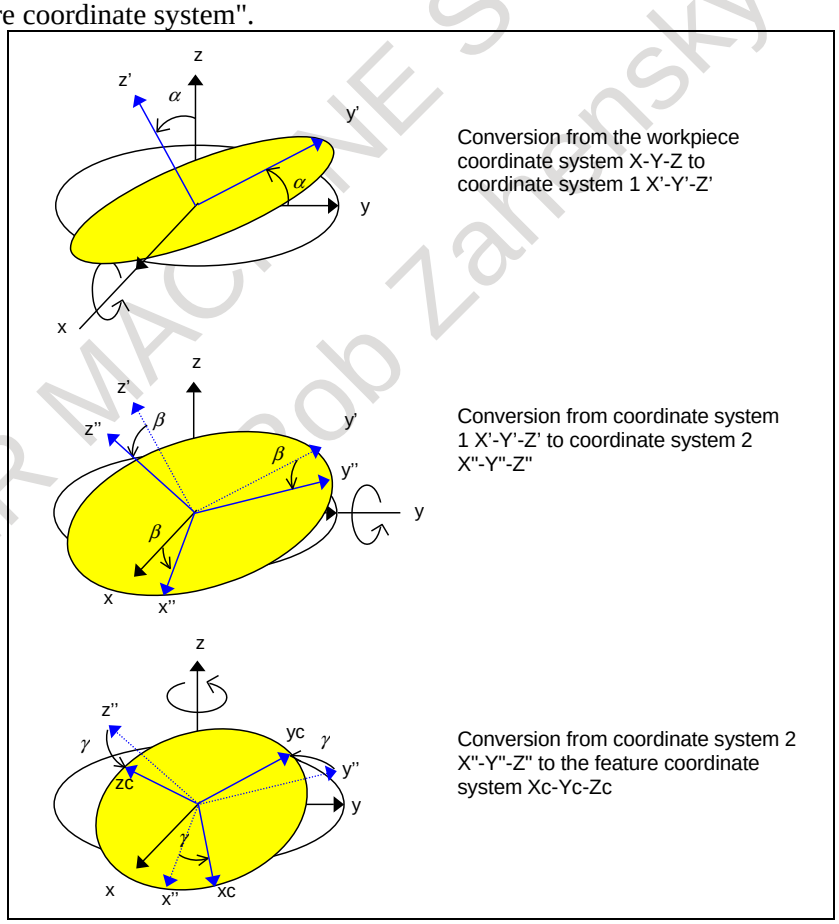


Fig. 14.3.1.3 (b)

FRYER MACHINE SYSTEMS INC
Bob Zahnsky

Example

The example of a program when feature coordinate system like the figure below is used is shown below.

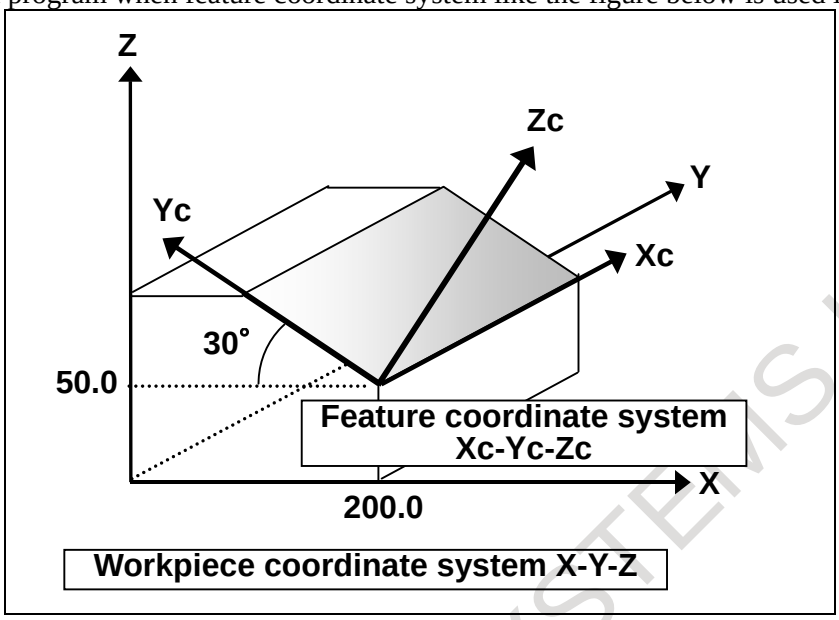


Fig. 14.3.1.3 (c)

- Feature coordinate system origin : (200.0, 0, 50.0)
- Order in which axes are rotated : X axis → Y axis → Z axis
- Angle of rotation about the X-axis : 30 degrees
- Angle of rotation about the Y-axis : 0 degree
- Angle of rotation about the Z-axis : 90 degrees

Example of a program

```
G68.2 P1 Q123 X200.0 Y0 Z50.0 I30.0 J0 K90.0 ;
G53.1 ;
  :
```

14.3.1.4 Tilted working plane indexing based on three points

Overview

With the tilted working plane indexing, a tilted working plane can be specified by specifying three points in a feature coordinate system.

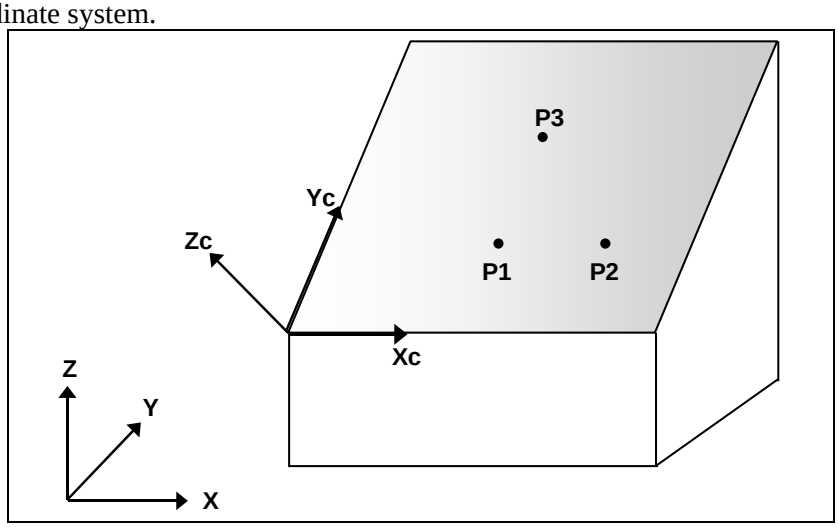


Fig. 14.3.1.4 (a)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Format

Format
G68.2 P2 Q0 X x_0 Y y_0 Z z_0 Rα ; **G68.2 P2 Q1 X x_1 Y y_1 Z z_1 ;** **G68.2 P2 Q2 X x_2 Y y_2 Z z_2 ;** **G68.2 P2 Q3 X x_3 Y y_3 Z z_3 ; Tilted working plane indexing** **G69 ; Cancel tilted working plane indexing (M series).** **G69.1; Cancel tilted working plane indexing (T series).**
Explanation of symbols
Q0 X x_0 Y y_0 Z z_0 : Amount of shift from the first point to the origin of a feature coordinate system By default, this value is 0. Q1 X x_1 Y y_1 Z z_1 : First point. (origin of a feature coordinate system) Q2 X x_2 Y y_2 Z z_2 : Second point. Q3 X x_3 Y y_3 Z z_3 : Third point. R : Angle of rotation about the Z-axis of a feature coordinate system By default, this value is 0. Any block in the G68.2 P2 command can be specified.

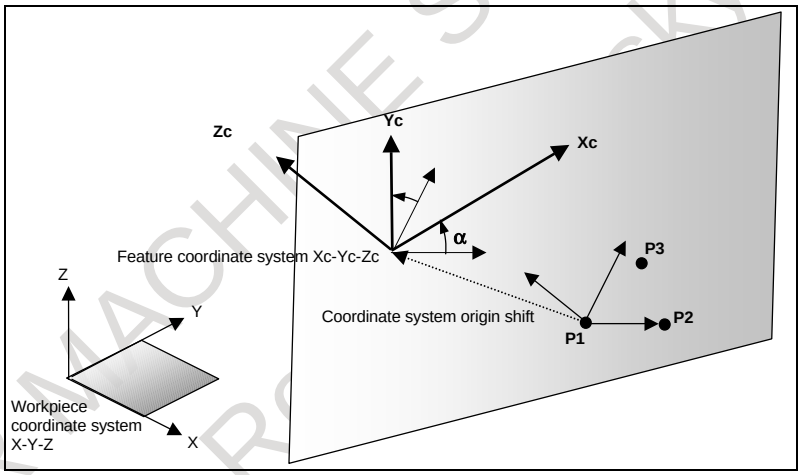


Fig. 14.3.1.4 (b)

⚠ CAUTION

1 Three G68.2P2 commands (Q1, Q2, and Q3) determine a tilted plane. If the G68.2P2 commands are interrupted, alarm PS5457, "G68.2/G68.3 FORMAT ERROR" is issued.

2 When any of the following conditions is met or address Q is set to a value other than the above, alarm PS5457 is issued.
(1) When two or more points are duplicate (the plane is not determined.)
(2) When three points are placed on a line (the plane is not determined.)
(3) When the distance between a line passing two of the three points and the remaining point is smaller than the setting of parameter No. 11220 (the plane is unstable.)

FRYER MACHINE SYSTEMS INC
Rocky

Explanation

- Determination of a feature coordinate system

Three entered points are named P1, P2, and P3 in the order of entry.
The P1-to-P2 direction is defined as the X-axis of a feature coordinate system. Among the directions that are on the plane containing the three points and are normal to the X-axis of the feature coordinate system, the direction that makes a smaller angle with the P1→P3 vector is defined as the Y-axis of the feature coordinate system. The Z-axis of the feature coordinate system is defined according to the right-handed system.

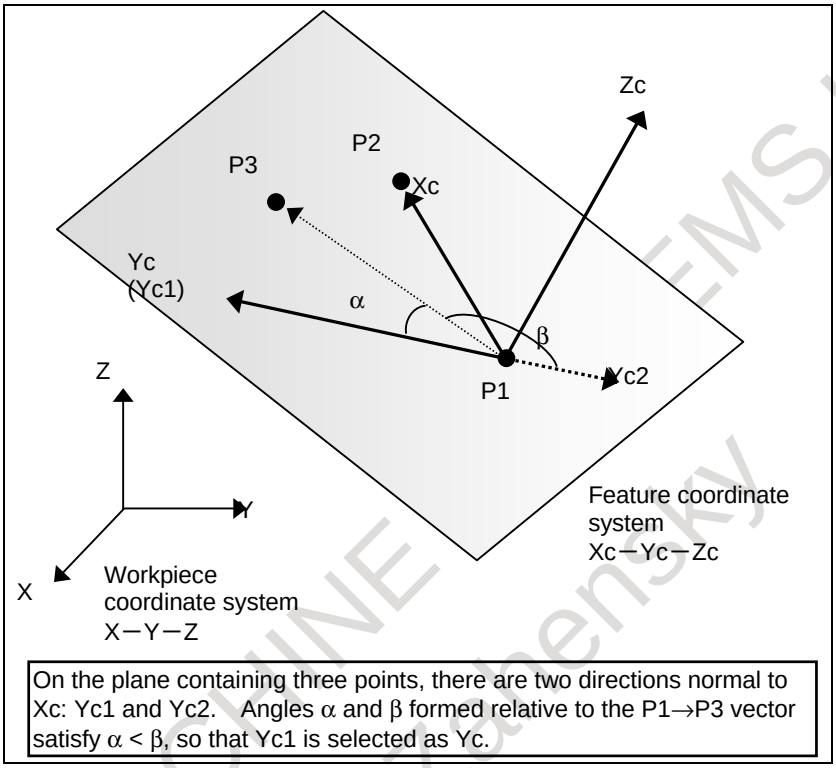


Fig. 14.3.1.4 (c)

- Origin of the feature coordinate system

The origin of the feature coordinate system is the first specified point P1.
By setting an origin shift amount (G68.2 P2 Q0 X_Y_Z_), the origin of the feature coordinate system is shifted by (X,Y,Z) from P1. Specify (X,Y,Z) in the feature coordinate system.

- Angular displacement R

The angular displacement R is positive for clockwise rotation viewed in the Z-axis direction in the feature coordinate system.

FRIEP MACHINE SYSTEMS INC
Rob Zabensky

Example

The example of a program when feature coordinate system like the figure below is used is shown below.

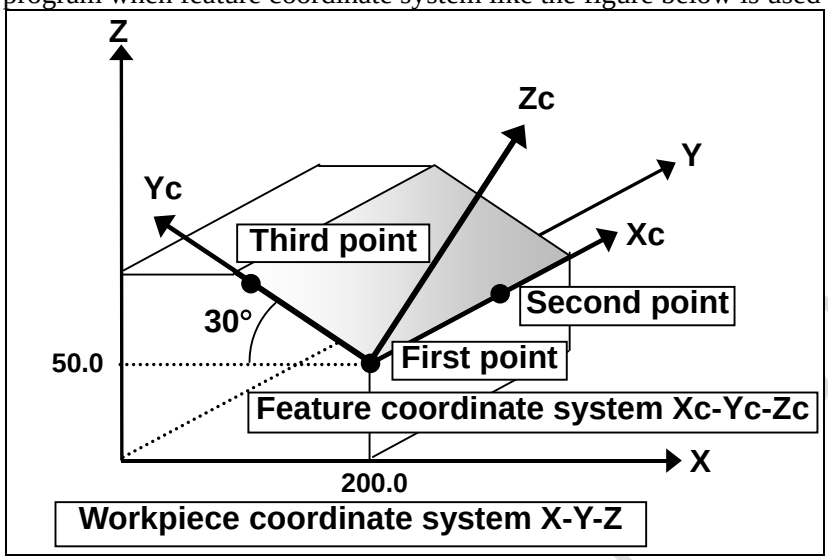


Fig. 14.3.1.4 (d)

- First point (origin of a feature coordinate system) : (200.0, 0, 50.0)
- Second point : (200.0, 100.0, 50.0)
- Third point : (26.795, 0, 150.0)

Example of a program

```
G68.2 P2 Q1 X200.0 Y0 Z50.0 ;
G68.2 P2 Q2 X200.0 Y100.0 Z50.0 ;
G68.2 P2 Q3 X26.795 Y0 Z150.0 ;
G53.1 ;
 . . .
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

14.3.1.5 Tilted working plane indexing based on two vectors

Overview

With the tilted working plane indexing, a tilted working plane can be specified by specifying an X-axis direction vector and a Z-axis direction vector in the feature coordinate system.

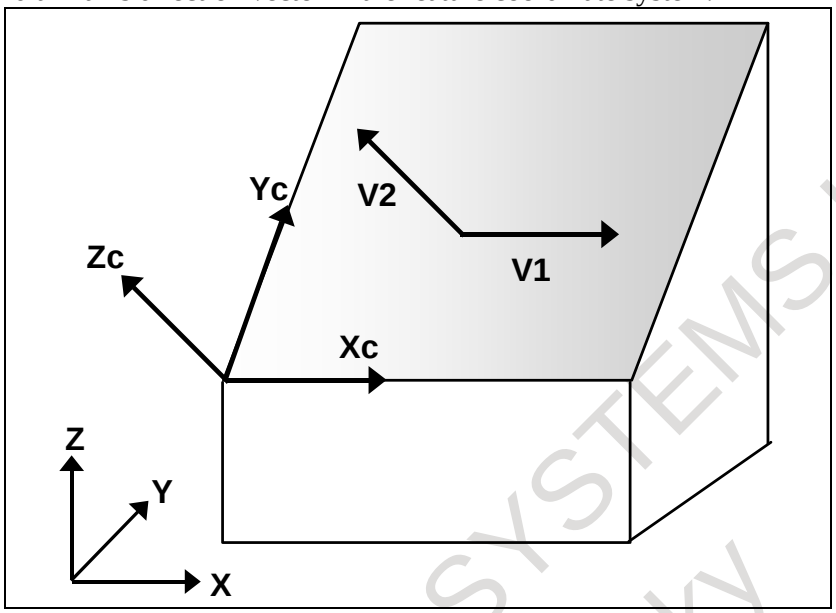


Fig. 14.3.1.5 (a)

Format

Format
G68.2 P3 Q1 X_ Y_ Z_ Iα_1 Jβ_1 Kγ_1 ; **G68.2 P3 Q2 Iα_2 Jβ_2 Kγ_2 ; Tilted working plane indexing** **G69 ; Cancel tilted working plane indexing (M series).** **G69.1; Cancel tilted working plane indexing (T series).**
Explanation of symbols
X_ Y_ Z_ : Origin of a feature coordinate system (specifiable in the Q1 block) Q1 Iα_1 Jβ_1 Kγ_1 : Feature coordinate system X-axis direction relative to the workpiece coordinate system (first vector) Q2 Iα_2 Jβ_2 Kγ_2 : Feature coordinate system Z-axis direction relative to the workpiece coordinate system (second vector)

⚠ CAUTION

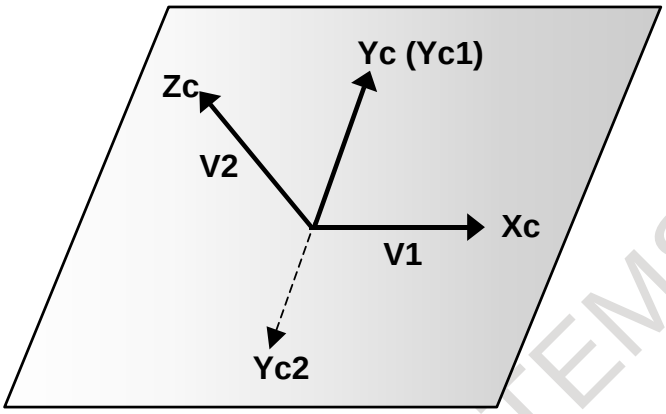
1. The two G68.2P3 commands (Q1 and Q2) determine a tilted plane. The G68.2P3 commands are interrupted, alarm PS5457, “G68.2/G68.3 FORMAT ERROR” is issued.
2. If the angle between the two vectors is 5 degrees or more off the 90 degrees, alarm PS5457 is issued.
3. If (I, J, K) is set to a 0 vector, alarm PS5457 is issued.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

- Determination of a feature coordinate system

The first vector is defined as the X-axis of the feature coordinate system, and the second vector is defined as the Z-axis of the feature coordinate system. The Y-axis of the feature coordinate system is defined according to the right-handed system.

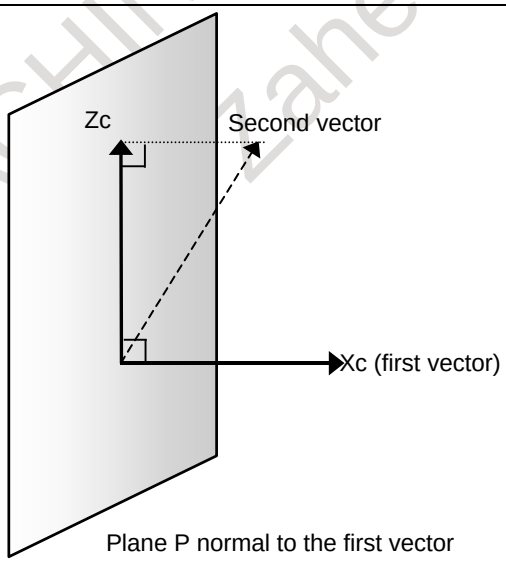


There are two vectors normal to the Xc-axis and Zc-axis. However, Yc1 is defined as the Yc-axis of the feature coordinate system according to the right-handed system.

Fig. 14.3.1.5 (b)

- When the first and second vectors are not normal

When the first vector and second vector are not normal, the orthographical vector from the second vector to plane P normal to the first vector is defined as the Z-axis of the feature coordinate system.



Plane P normal to the first vector

FRYER MACHINE SYSTEMS INC
Zahensky

Example

The example of a program when feature coordinate system like the figure below is used is shown below.

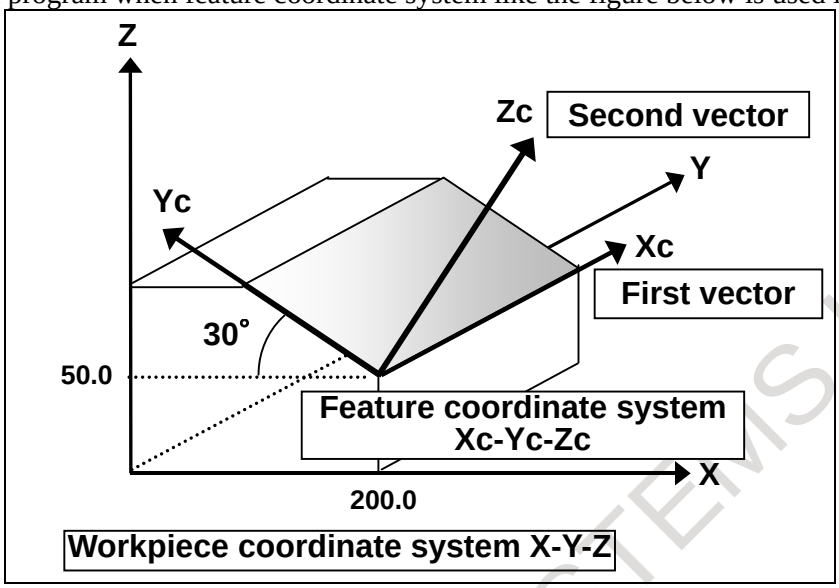


Fig. 14.3.1.5 (d)

Origin of a feature coordinate system : (200.0, 0, 50.0)
X-axis direction of a feature coordinate system (First vector) : (0, 1.0, 0)
Z-axis direction of a feature coordinate system (Second vector) : (100.0, 0, 173.205)

Example of a program

```
G68.2 P3 Q1 X200.0 Y0 Z50.0 I0 J1.0 K0 ;
G68.2 P3 Q2 I100.0 J0 K173.205 ;
G53.1 ;
 . . .
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

14.3.1.6 Tilted working plane indexing based on projection angles

Overview

With the tilted working plane indexing, a tilted working plane can be specified based on projection angles.

A plane determined by vector A and vector B produced by rotating the X-axis vector and Y-axis vector of the workpiece coordinate system is defined to be a tilted working plane.

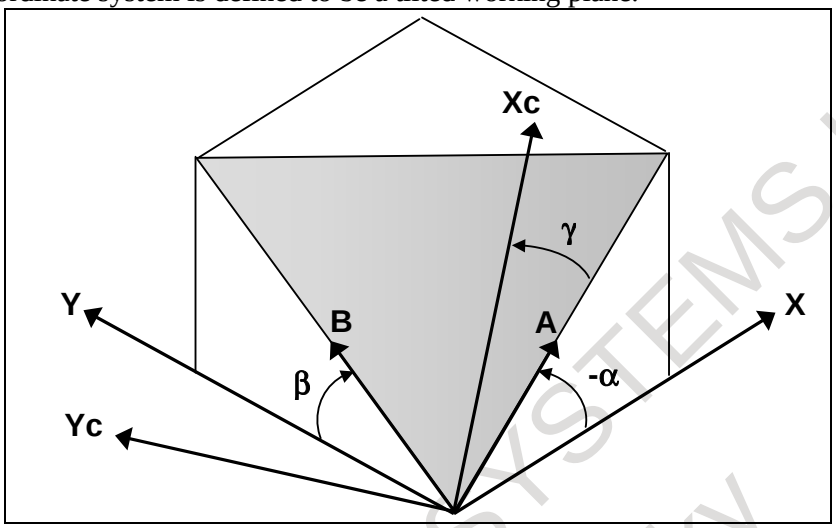


Fig. 14.3.1.6 (a)

Format

Format
G68.2 P4 X_ Y_ Z_ Iα Jβ Kγ; Tilted working plane indexing **G69 ; Cancel tilted working plane indexing (M series).** **G69.1; Cancel tilted working plane indexing (T series).**
Explanation of symbols
X_ Y_ Z_ : Origin of a feature coordinate system α : Angle by which the X-axis vector is rotated about the Y-axis of the workpiece coordinate system β : Angle by which the Y-axis vector is rotated about the X-axis of the workpiece coordinate system γ : Angle of rotation about the Z-axis of the feature coordinate system

Explanation

- Determination of a feature coordinate system

The X-axis direction vector of the workpiece coordinate system rotated by α about the Y-axis of the workpiece coordinate system is defined as vector A. The Y-axis direction vector of the workpiece coordinate system rotated by β about the X-axis of the workpiece coordinate system is defined to be vector B.

The direction normal to plane P containing vector A and vector B (direction of the outer product of A × B) is defined to be the Z-axis direction of the feature coordinate system.

Vector A rotated by γ about the Z-axis of the feature coordinate system is defined to be the X-axis direction of the feature coordinate system. The Y-axis of the feature coordinate system is defined according to the right-handed system.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

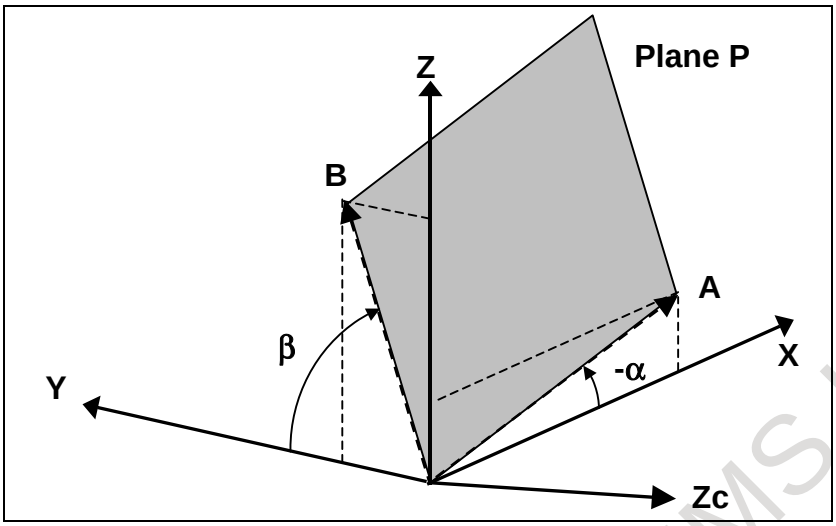


Fig. 14.3.1.6 (b)

By the third command angle α and second command angle β, the Z-axis of the feature coordinate system are determined.

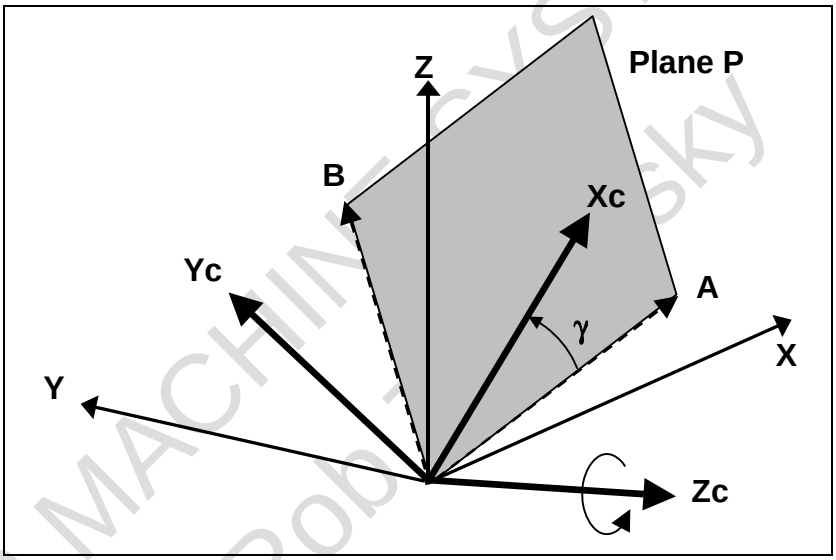


Fig. 14.3.1.6 (c)

By the third command angle γ, the X-axis and Y-axis of the feature coordinate system are determined.

> ⚠ **CAUTION**
> When vector A and vector B are considered to be parallel with each other (when the angle formed by the two vectors is smaller than 1°), alarm PS5457, "G68.2/G68.3 FORMAT ERROR" is issued.

FRYER MACHINE SYSTEMS INC Robotsky

Example

The example of a program when feature coordinate system like the figure below is used is shown below.

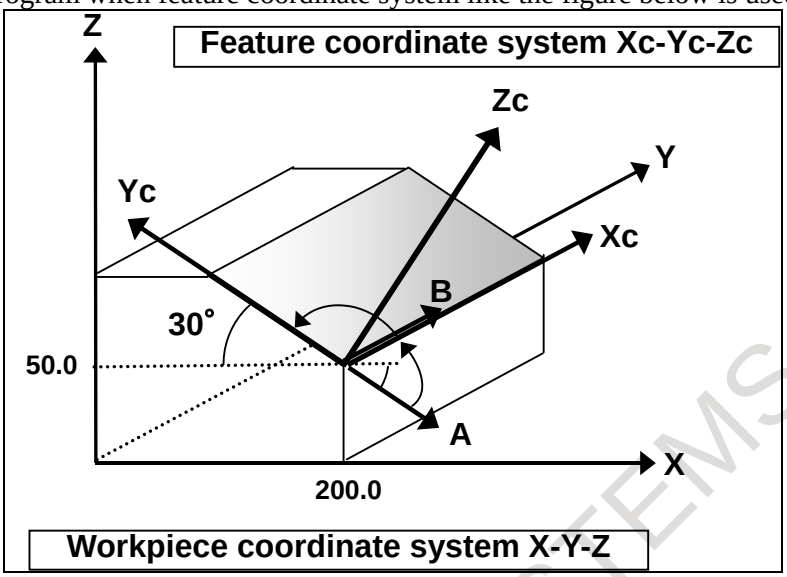


Fig. 14.3.1.6 (d)

- Origin of a feature coordinate system : (200.0, 0, 50.0)
- Angle by which the X-axis vector is rotated about the Y-axis of the workpiece coordinate system : 30 degrees
- Angle by which the Y-axis vector is rotated about the X-axis of the workpiece coordinate system : 0 degree
- Angle of rotation about the Z-axis of the feature coordinate system : 90 degrees

Example of a program

```
G68.2 P4 X200.0 Y0 Z50.0 I30.0 J0 K90.0 ;
G53.1 ;
      :
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

14.3.1.7 Tilted working plane indexing by tool axis direction

Overview

By specifying G68.3, a coordinate system (feature coordinate system) where the tool axis direction is the +Z-axis direction can be automatically specified. When a feature coordinate system is used, a program for cutting a hole or pocket in a plane tilted relative to the workpiece coordinate system can be made simpler. This function can automatically generate a feature coordinate system that is normal to the tool direction.

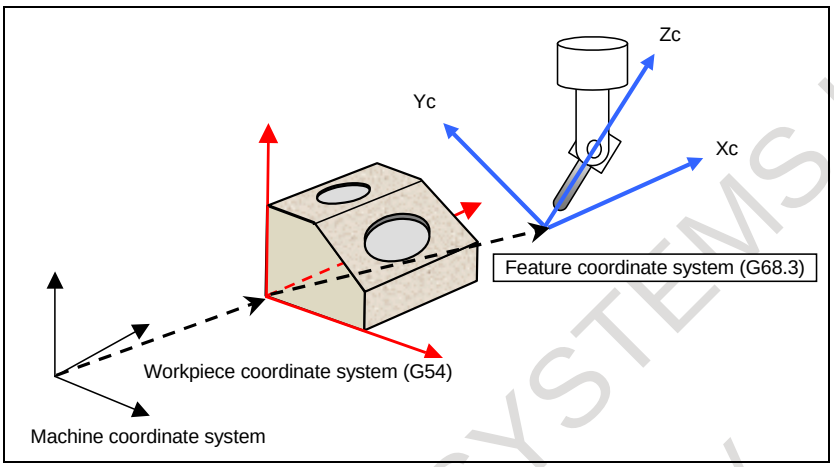


Fig. 14.3.1.7 (a) Feature coordinate system

When G68.3 is specified in a block, the coordinate system for programming is changed to a feature coordinate system. All commands after the block are regarded as commands in the feature coordinate system until G69 is specified.

Format

Format	
G68.3 X x_0 Y y_0 Z z_0 Rα ;	**Tilted working plane indexing**
G68.3 P1 X x_0 Y y_0 Z z_0 ;	**Tilted working plane indexing**
G69 ;	**Cancel tilted working plane indexing (M series).**
G69.1;	**Cancel tilted working plane indexing (T series).**

Explanation of symbols	
X,Y,Z	: Origin of a feature coordinate system (absolute) By default, the current position is used as the origin of the feature coordinate system.
R	: Angular displacement about the Z-axis in the feature coordinate system. The default is 0°.
P1	: The feature coordinate system is defined corresponding to tool rotation axis position.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

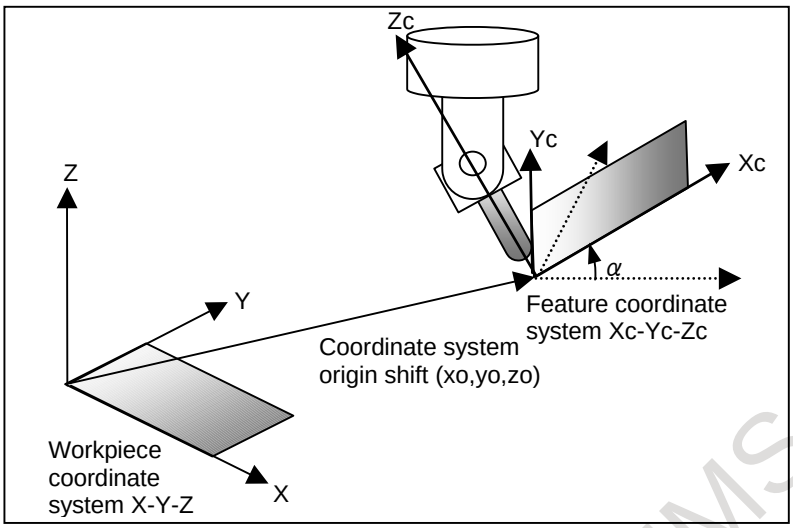


Fig.14.3.1.7 (b) G68.3 command

Explanation

- Feature coordinate system

By specifying G68.3, a feature coordinate system with the tool axis direction being the +Z-axis direction can be created. The tool axis direction means the tool axis direction based on the rotation axis position reached by automatic operation or manual operation.

- Origin of a feature coordinate system

The origin of a feature coordinate system must be specified using an absolute command.
Even in the incremental command mode, the specified origin is regarded as an absolute position.
When 1 address or 2 addresses are omitted in X, Y, Z, alarm PS5457, "G68.2/G68.3 FORMAT ERROR" is issued.
When the origin of a feature coordinate system is not specified, the position when G68.3 is specified is used as the origin.

- Determination of a feature coordinate system

Determination of a feature coordinate system depends on whether P1 command is present.

(1) When G68.3 block does not include P1 command (G68.3)

When G68.3 is specified, the tool axis direction vector ($\vec{T}$) represents the +Z direction ($\vec{Zc}$) of the feature coordinate system.

The vector normal to a plane formed by the +Z direction ($\vec{Zc}$) of the feature coordinate system and the vertical axis direction vector ($\vec{P}$) (parameter No. 12321) represents the +X direction ($\vec{Xc}$) of the feature coordinate system.

Expression: $\vec{Xc} = \vec{P} \times \vec{Zc}$

The vector normal to the +Z direction ($\vec{Zc}$) of the feature coordinate system and the +X direction ($\vec{Xc}$) of the feature coordinate system represents the +Y direction ($\vec{Yc}$) of the feature coordinate system.

Expression: $\vec{Yc} = \vec{Zc} \times \vec{Xc}$

When R is commanded, a coordinate system rotated by R around Zc from the above-mentioned coordinate system is the feature coordinate system.

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

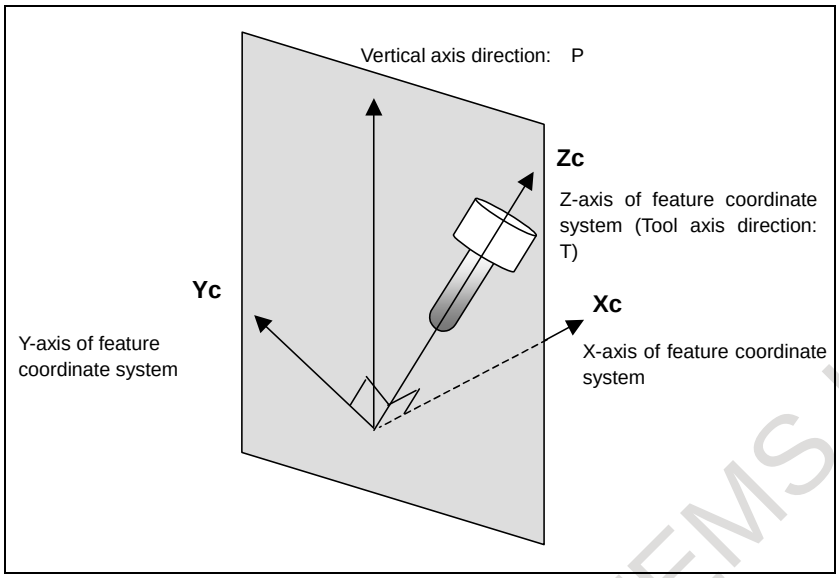


Fig.14.3.1.7 (c) Determination of a feature coordinate system

When the tool axis direction vector ($\vec{T}$) is parallel with the vertical axis direction vector ($\vec{P}$) (parameter No. 12321) (when the angle between the vector ($\vec{T}$) and the vector ($\vec{P}$) is equal to or less than the value of parameter No. 12322), the feature coordinate system Xc-Yc-Zc is as indicated below. By specifying angular displacement R, a feature coordinate system rotated about the Z-axis of this coordinate system can be specified.

Table 14.3.1 (a)

Parameter No. 12321	Z-axis of feature coordinate system Zc	X-axis of feature coordinate system Xc	Y-axis of feature coordinate system Yc
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

When 0 is set in parameter No. 12321, the vertical axis direction is the reference tool axis direction (parameter No. 19697).
If a value other than 0 through 3 is set in parameter No. 12321, alarm PS5459, “MACHINE PARAMETER INCORRECT” is issued.

> ⚠ **CAUTION**
> Tool axis direction is Z-axis direction of feature coordinate system regardless of the reference tool axis direction (parameter No. 19697).

- Angular displacement R

Angular displacement R is positive when a rotation is made clockwise viewed in the Z-axis direction of the feature coordinate system. The range of angular displacement R is: 0.0° ≤ R ≤ 360.0°.

(2) When G68.3 block includes P1 command (G68.3 P1)

"G68.3 P1" command defines the feature coordinate system corresponding to tool rotation axis position. The direction of the feature coordinate system is the direction of reference coordinate system rotated by tool rotation axes.

FRYER MACHINE SYSTEMS INC Rob Zaitensky

M

The direction of this feature coordinate system is the same as the direction of tool axis direction feed / tool axis right-angle direction feed of 3-dimensional manual feed when the parameter FLL (No.12320#1) is set to zero.

The reference coordinate system of feature coordinate system (the feature coordinate system that is defined when absolute coordinate system of tool rotation axes is zero) is as follows by the parameter (No.19697) for reference tool axis direction.
The feature coordinate system defined by "G68.3 P1" command is the coordinate system that the reference coordinate system is rotated by tool rotation axis position and parameter RA (No.19698), RB (No.19699).

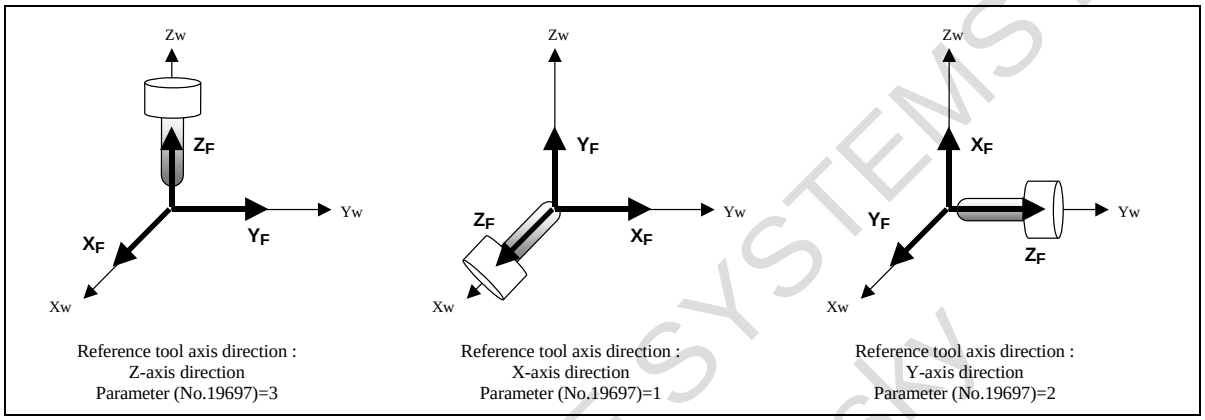
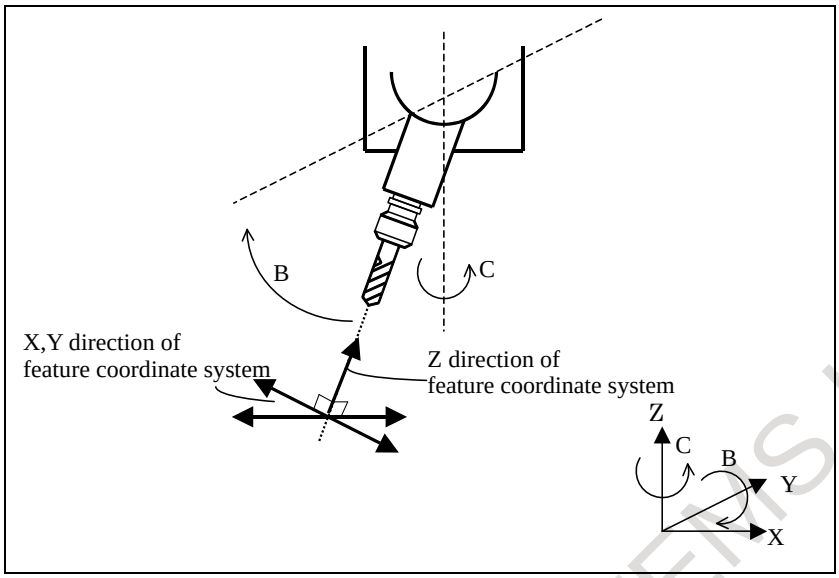


Fig.14.3.1.7 (d) The reference coordinate system of feature coordinate system (G68.3 P1)

FRYER MACHINE SYSTEMS INC
Rob Zahensky



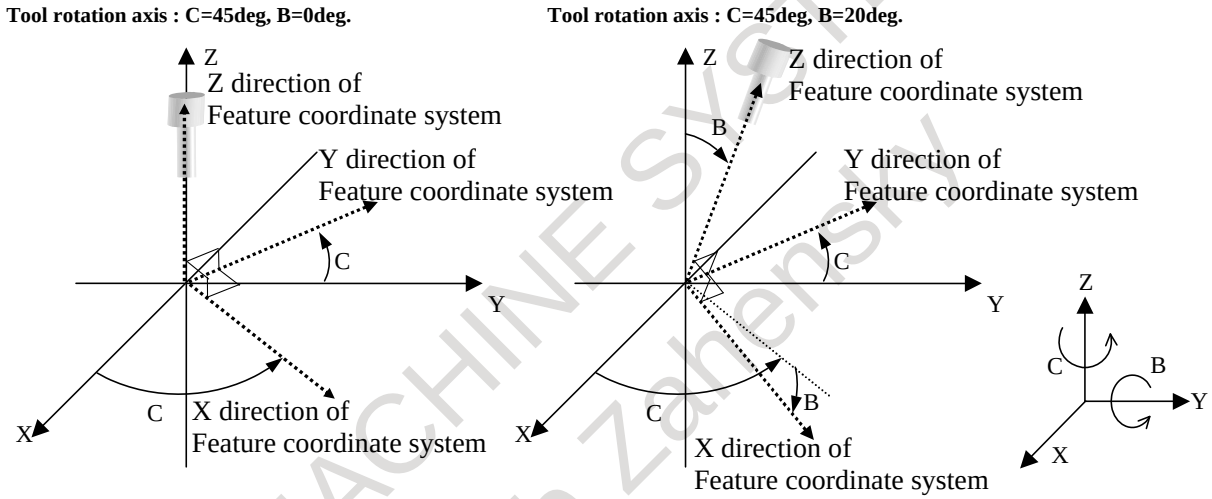


Fig.14.3.1.7 (e) Example that reference tool axis direction is Z direction

- Machine of table rotation type

On a machine of table rotation type, the tool direction remains unchanged. So, a feature coordinate system based on the reference tool axis direction (parameter No. 19697) is set. However, the origin specification of the feature coordinate system and angular displacement R about the Z-axis are valid.

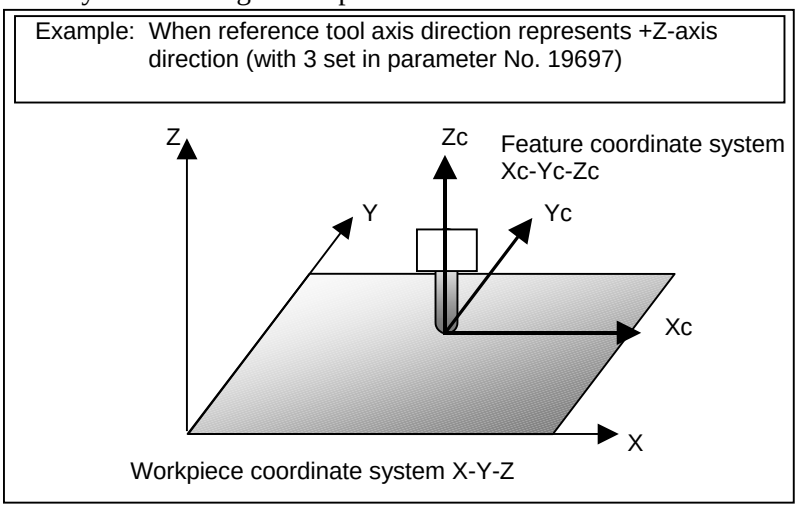


Fig.14.3.1.7 (f)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- G53.1 / G53.6 command in G68.3 mode

When G53.1 or G53.6 command is executed in G68.3 mode, alarm PS5458, ”ILLEGAL USE OF G53.1/G53.6” occurs.

- Use in combination with tool length compensation

The G68.3 command can be specified even during tool length compensation.

- Example of operation

An example of operation on a machine of tool rotation type is given below.
The machine configuration is "BC type reference tool axis Z-axis".

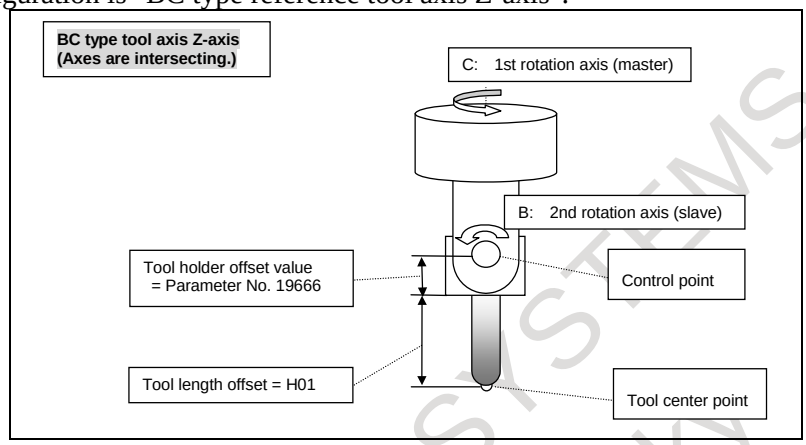


Fig.14.3.1.7 (g)

Sample program 1

```
O0100 ;
N1 G55 ;
N2 G90 G01 X0Y0Z50.0 F1000 ;
N3 G43 H01 X0 Y0 Z0 ;
N4 B-45.0 ;
N5 G68.3 ;
:
N6 G69 ;
:
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

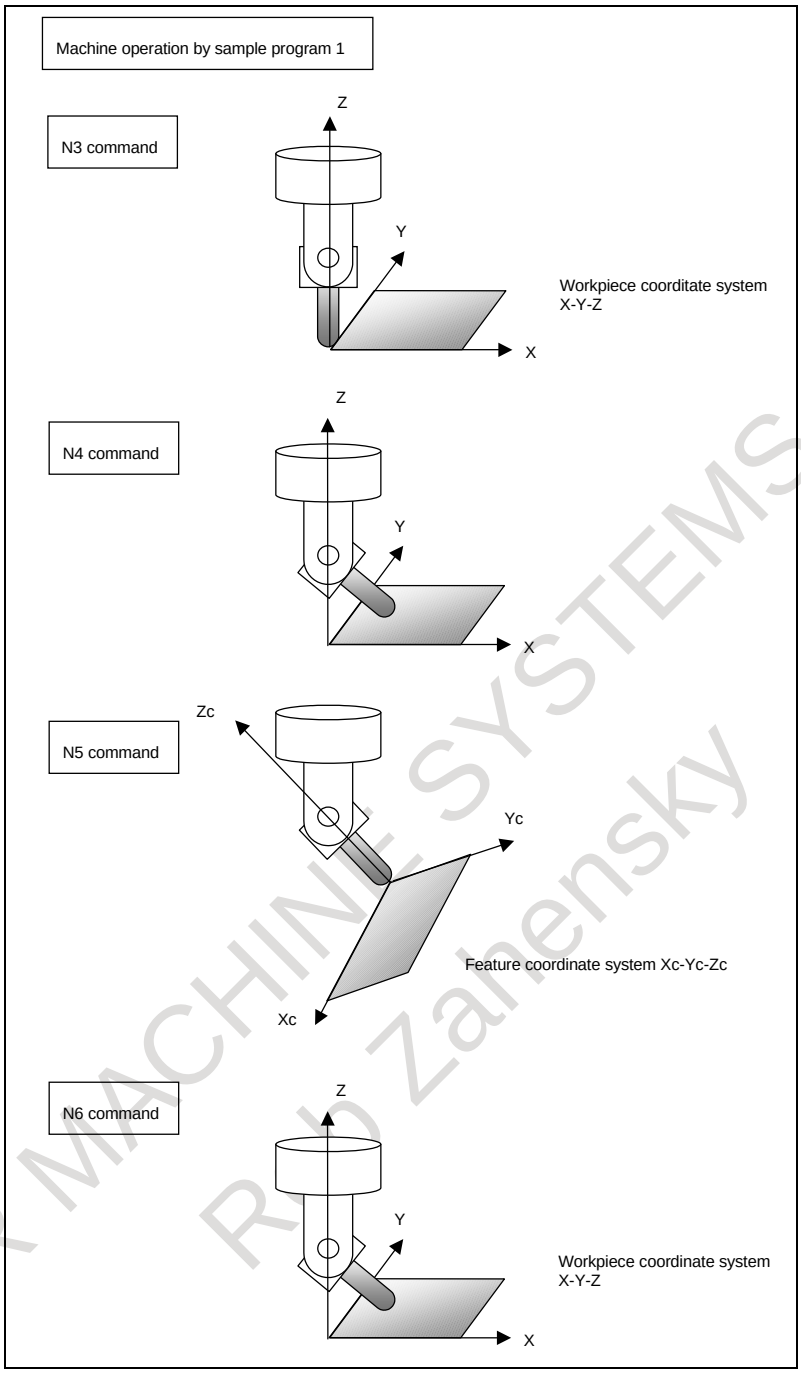


Fig.14.3.1.7 (h)

N3 block: Performs tool length compensation in the workpiece coordinate system.
The tool center point moves to the origin of the workpiece coordinate system.

N4 block: Tilts the tool.

N5 block: Sets a feature coordinate system where the tool axis direction is the Z-axis direction and the tool center point is placed at the origin.

N6 block: Cancels the feature coordinate system to return to the workpiece coordinate system.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Sample program 2
O0100 ;
N1 G54 G90 G00 B0 C0 ;
N2 B45.0 ;
N3 C60.0 ;
N4 G68.3 P1 X0 Y0 Z0 ;

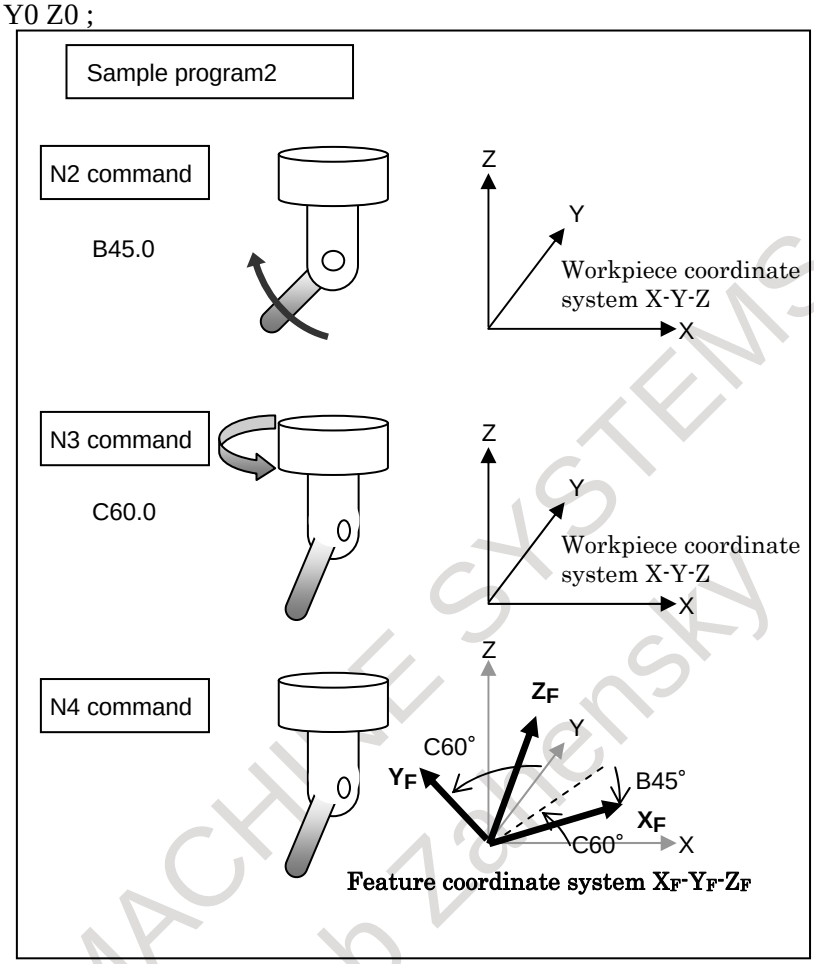


Fig.14.3.1.7 (i)

N2 block: Tilts the tool. (B45 deg)
N3 block: Tilts the tool. (C60 deg)
N4 block: The direction of the reference coordinate system of feature coordinate system is the direction of workpiece coordinate system because the reference tool axis direction is Z direction. The feature coordinate system is the coordinate system that the reference coordinate system is rotated by 45 deg. around Y direction and 60 deg. around Z direction.

X_F : The direction that X direction of workpiece coordinate system is rotated by 60 deg. around Z direction after by 45 deg. around Y direction of workpiece coordinate system.

Y_F : The direction that Y direction of workpiece coordinate system is rotated by 60 deg. round Z direction of workpiece coordinate system.

Z_F : The direction that Z direction of workpiece coordinate system is rotated by 60 deg. around Z direction after by 45 deg. around Y direction of workpiece coordinate system.

- Multiple G68.3

After the tool axis direction is changed in G68.3 mode, by specifying G68.3, a new feature coordinate system where the tool axis direction is the +Z-axis direction can be specified.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Example of operation

An example of operation on a machine of tool rotation type is given below.
The machine configuration is "AC type reference tool axis Z-axis".

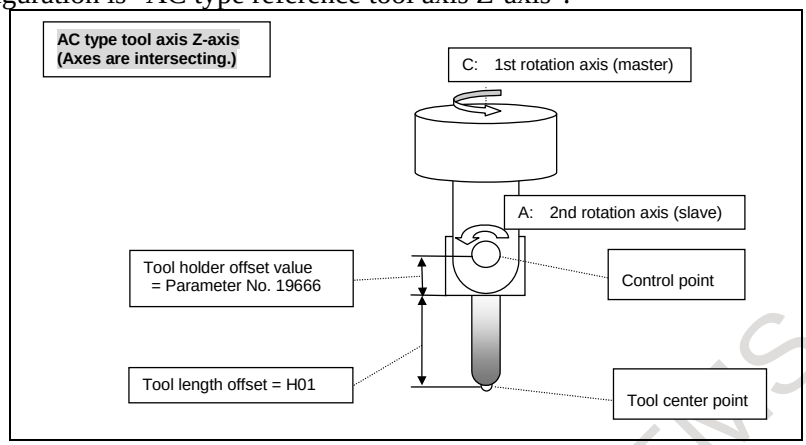


Fig.14.3.1.7 (j)

Sample program 2

```
O0200 ;
N1 G55 ;
N2 G01 A90.0 F1000 ;
N3 G68.3 X0 Y0 Z0 R0;
:
N4 X10.0 Y0 Z0 ;
N5 C90.0;
N6 G68.3 X10.0 Y0 Z0 ;
:
N7 G69 ;
:
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

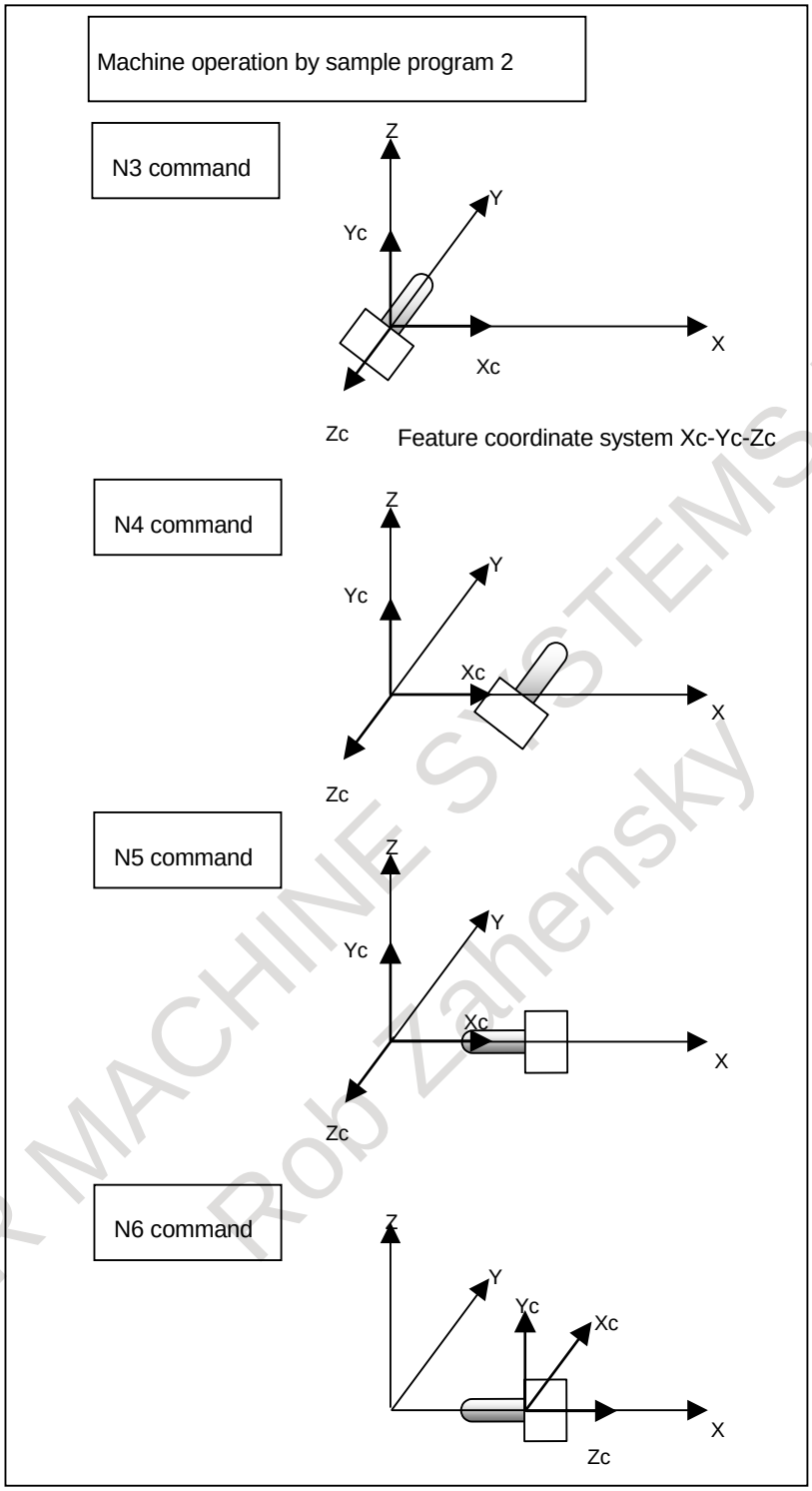


Fig.14.3.1.7 (k)

N3 block: Sets a feature coordinate system according to the tool direction.
N4 block: Specifies coordinates in the feature coordinate system.
N5 block: Changes the tool direction.
N6 block: Sets a feature coordinate system according to the tool direction.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

14.3.2 Multiple Command of Tilted Working Plane Indexing

14.3.2.1 Absolute multiple command

By additionally specifying G68.2 in the tilted working plane indexing mode, a feature coordinate system produced by additionally applying coordinate system conversion to the workpiece coordinate system can be set. The workpiece coordinate system is resumed by specifying G69.
This function is enabled by setting bit 0 (MTW) of parameter No. 11221.

Format

The format of the tilted working plane indexing (G68.2) is applicable.
Specify the origin of a feature coordinate system in the workpiece coordinate system.

NOTE
Before specifying G68.2, cancel tool length compensation and tool radius compensation. If G68.2 is specified during tool length compensation or tool radius compensation, alarm PS5462, “ILLEGAL COMMAND (G68.2/G69)” is issued.

Example of operation

An example of operation on a tool rotation type machine is explained below.
The machine configuration is "BC type with the reference tool axis being the Z-axis".

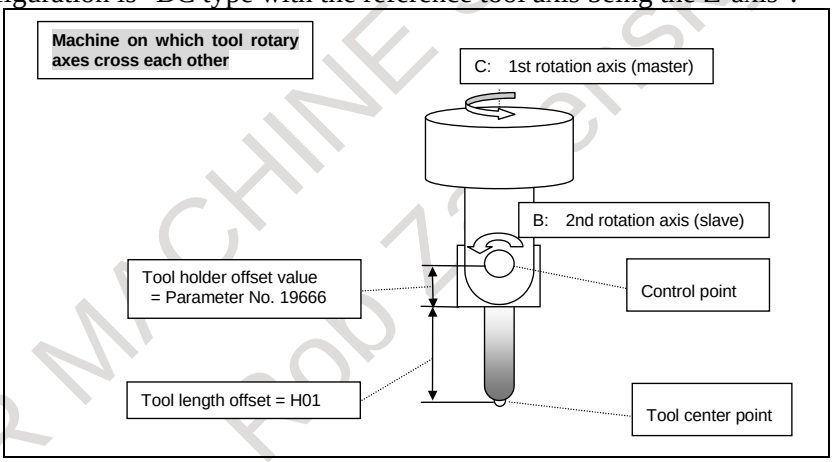


Fig. 14.3.2.1 (a)

Sample program 2

```
O0100 ;
N1 G55 ;
N2 G90 G01 X20.0 Y5.0 Z0 F1000 ;
N3 G68.2 X20.0 Y5.0 Z0 I0 J90.0 K0 ;
N4 G53.1 ;
  :
N5 X-15.0 Y0 Z-15.0 ;
N6 G68.2 X5.0 Y20.0 Z0 I90.0 J90.0 K0 ;
N7 G53.1;
  :
N8 G69 ;
  :
```

FRYER MACHINE SYSTEMS INC Rod Zaleski

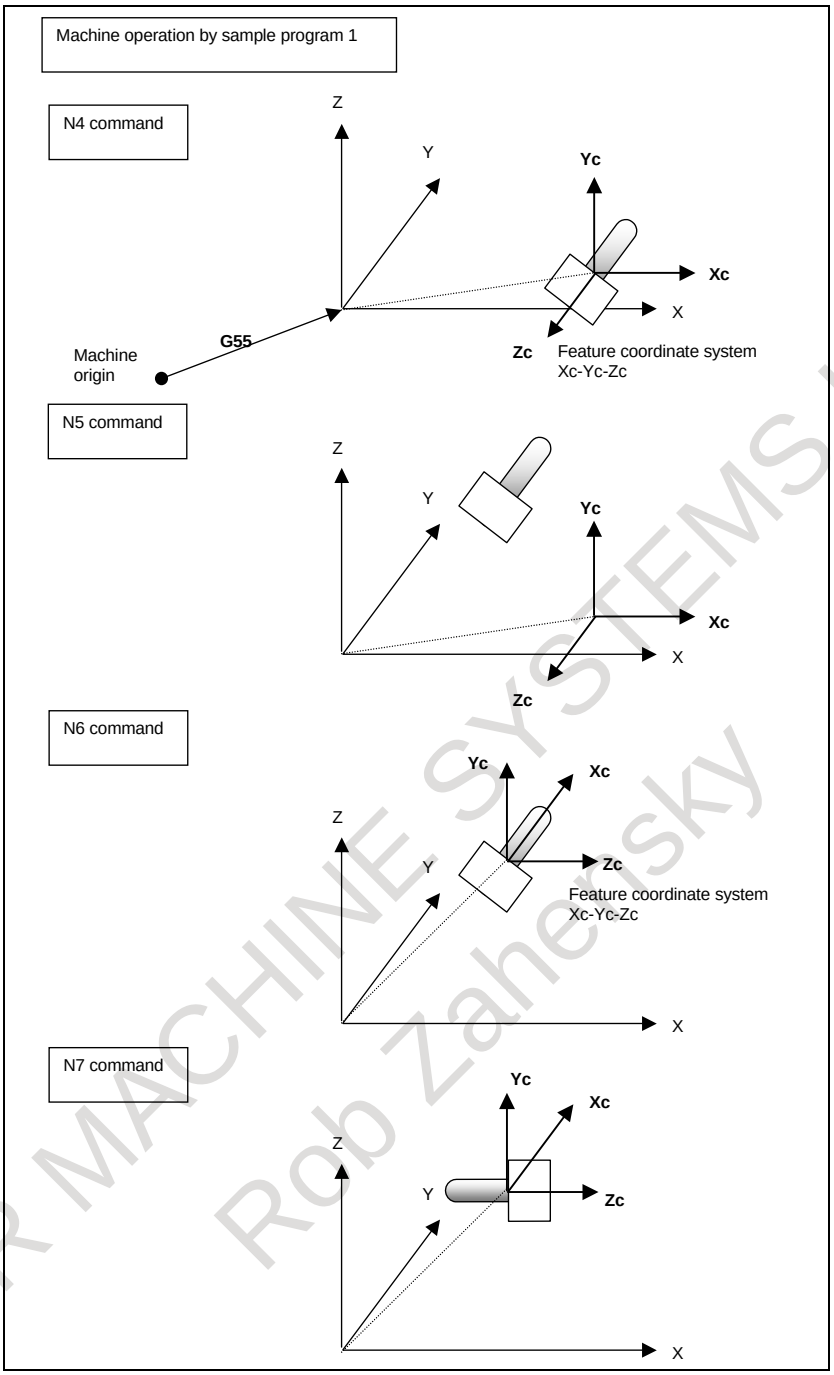


Fig. 14.3.2.1 (b)

N4 block: Rotates the tool in the Z-axis direction in the feature coordinate system.
N5 block: Specifies coordinates in the feature coordinate system.
N6 block: Sets a new feature coordinate system.
N7 block: Rotates the tool in the Z-axis direction in the new feature coordinate system.

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

14.3.2.2 Incremental multiple command

By specifying G68.4, coordinate system conversion can be applied to the currently set feature coordinate system.
This function is enabled by setting bit 0 (MTW) of parameter No. 11221.

Format

The format of the tilted working plane indexing (G68.2) is applicable.
Specify the origin of a feature coordinate system in the immediately preceding feature coordinate system.

Table 14.3.2.2 (a)

Specification method	Incremental multiple command
Eulerian angle	G68.4
Roll-pitch-yaw	G68.4 P1
Three points	G68.4 P2
Two vectors	G68.4 P3
Projection angles	G68.4 P4

NOTE
Before specifying G68.4, cancel tool radius compensation. If G68.4 is specified during tool radius compensation, alarm PS5462, “ILLEGAL COMMAND (G68.2/G69)” is issued.

Example of operation

An example of operation on a tool rotation type machine is explained below.
Rotary axis C rotates about the Z-axis (master axis).
Rotary axis B rotates about the Y-axis (slave axis).

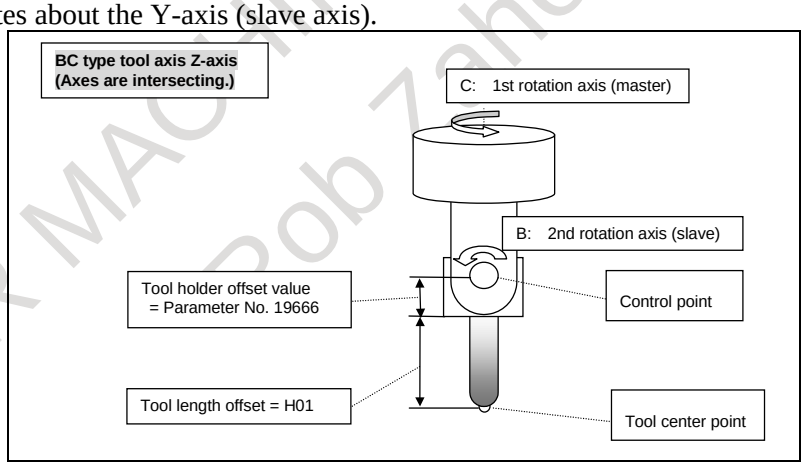


Fig. 14.3.2.2 (a)

Sample program 3

```
O0200 ;
N1 G55 ;
N2 G90 G01 X20.0 Y5.0 Z0 F1000 ;
N3 G68.2 X20.0 Y5.0 Z0 I0 J90.0 K0 ;
N4 G53.1 ;
  :
N5 X-15.0 Y0 Z-15.0 ;
N6 G68.4 X-15.0 Y0 Z-15.0 I90.0 J90.0 K-90.0 ;
N7 G53.1;
  :
N8 G69 ;
  :
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

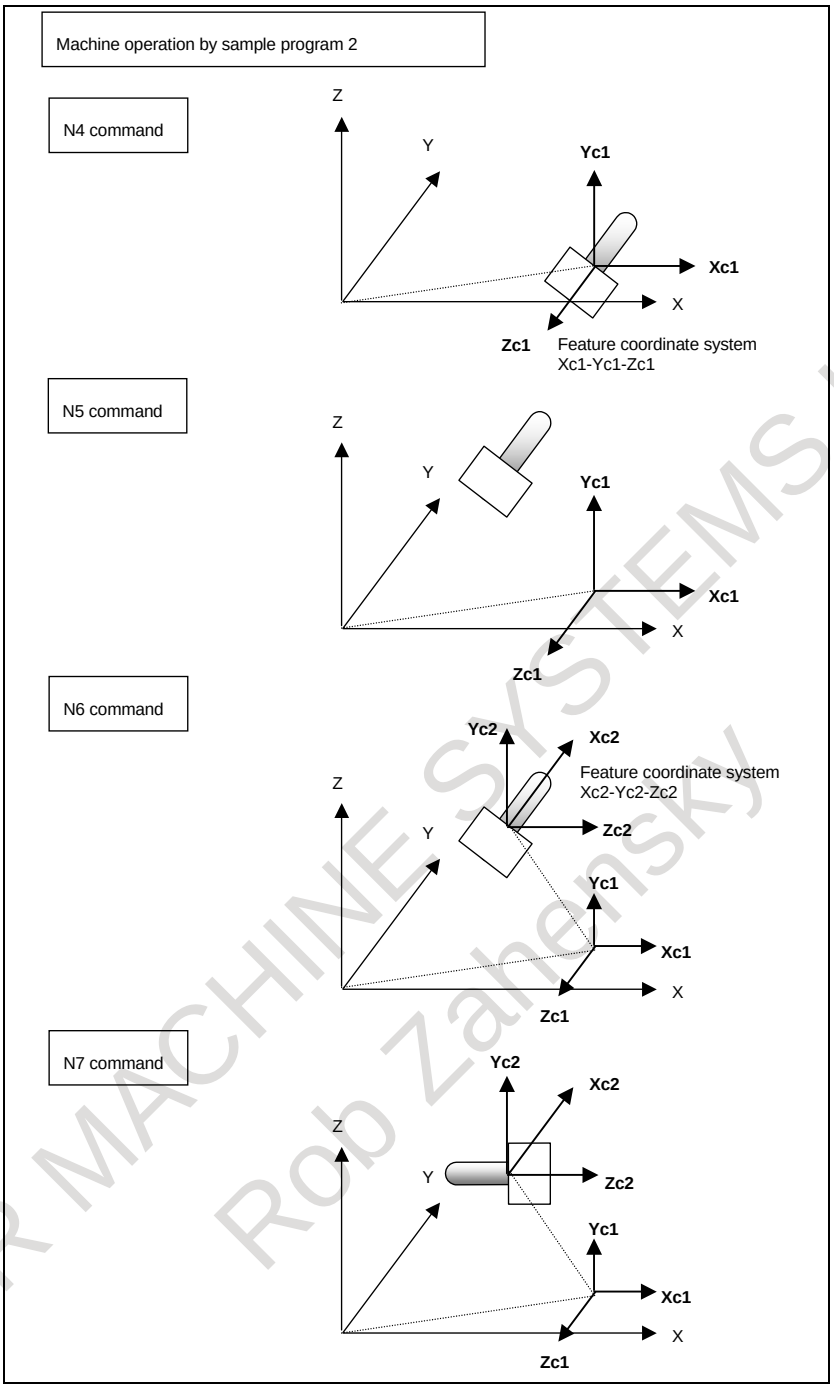


Fig. 14.3.2.2 (b)

N4 block: Rotates the tool in the Z-axis direction in the feature coordinate system.
N5 block: Specifies coordinates in the feature coordinate system.
N6 block: Applies coordinate system conversion to the feature coordinate system based on Eulerian angle to set a new feature coordinate system.
N7 block: Rotates the tool in the Z-axis direction in the new feature coordinate system.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

14.3.3 Tool Axis Direction Control

14.3.3.1 Tool axis direction control

G53.1 automatically specifies the +Z direction of the feature coordinate system as the tool axis direction.

Example of operation

The following gives an operation example in the machine configuration below.

- Table rotation type
- Master axis: About the Y-axis (B axis)
- Slave axis: About the Z-axis (C axis)
- Reference tool axis direction: Z direction
- Increment system for the rotation axis: 1/10(IS-C)

Program example 1

G68.2 I90.0 J0.0005 K-90.0 (rotation by 0.0005 degree about the Y-axis)
G53.1

The rotation axis position after the G53.1 command is as follows:

- Bit 2 (TFR) of parameter No. 11630 is set to 0 (minimum command unit of the rotation angles: 0.001 degree):
  B axis: 0.0010 degree
  C axis: 0.0000 degree
- Bit 2 (TFR) of parameter No. 11630 is set to 1 (minimum command unit of the rotation angles: 0.00001 degree):
  B axis: 0.0005 degree
  C axis: 0.0000 degree

Program example 2

G68.2 I90.0 J10 K-90.0 (rotation by (minimum command unit of the rotation angles × 10) about the Y-axis)
G53.1

The rotation axis position after the G53.1 command is as follows:

- Bit 2 (TFR) of parameter No. 11630 is set to 0 (minimum command unit of the rotation angles: 0.001 degree):
  B axis: 0.0100 degree
  C axis: 0.0000 degree
- Bit 2 (TFR) of parameter No. 11630 is set to 1 (minimum command unit of the rotation angles: 0.00001 degree):
  B axis: 0.0001 degree
  C axis: 0.0000 degree

- Tool rotation type machine

The following paragraphs describe several cases of the tool rotation type machine operation.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Operation description 1:
When G43 (tool length compensation) is specified for a machine with its axes crossing one another

The G53.1 command, when specified after the G68.2 command, automatically controls the rotary axis in such a way that the tool axis will be oriented in the +Z direction of the feature coordinate system.

Example)

```
O100 (Sample Program1) ;
N1 G55 ;
N2 G90 G01 X0 Y0 Z30.0 F1000 ;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0 ;
N4 G01 X0 Y0 Z30.0 F1000 ;
N5 G53.1 ;
N6 G43 H01 X0 Y0 Z0 ;
N7 . . .
```

In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
In addition, the tool axis, tool rotation axis B, and tool rotation axis C cross one another.

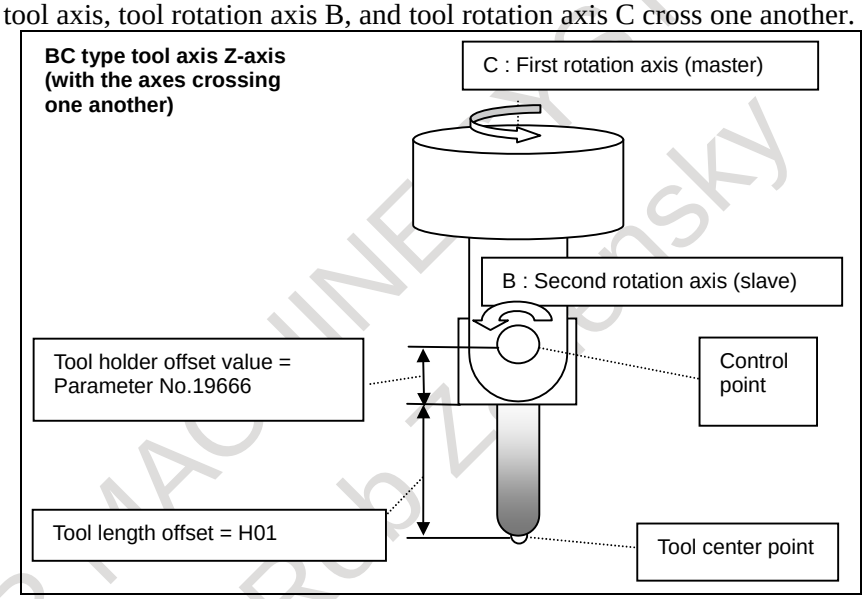


Fig. 14.3.3.1 (a)

Block N3 : Defines a feature coordinate system in the workpiece coordinate system.
Block N4 : Shifts the control point to point Z30.0 in the feature coordinate system.
Block N5 : Exerts automatic control over the rotary axes.
Block N6 : Performs tool length compensation in the feature coordinate system.
The tool center point is shifted to the origin of the feature coordinate system.

FIXER MACHINERY SYSTEMS INC
Rob Zaremsky

Fig. 14.3.3.1 (b) shows the behavior of the machine when it runs sample program 1.

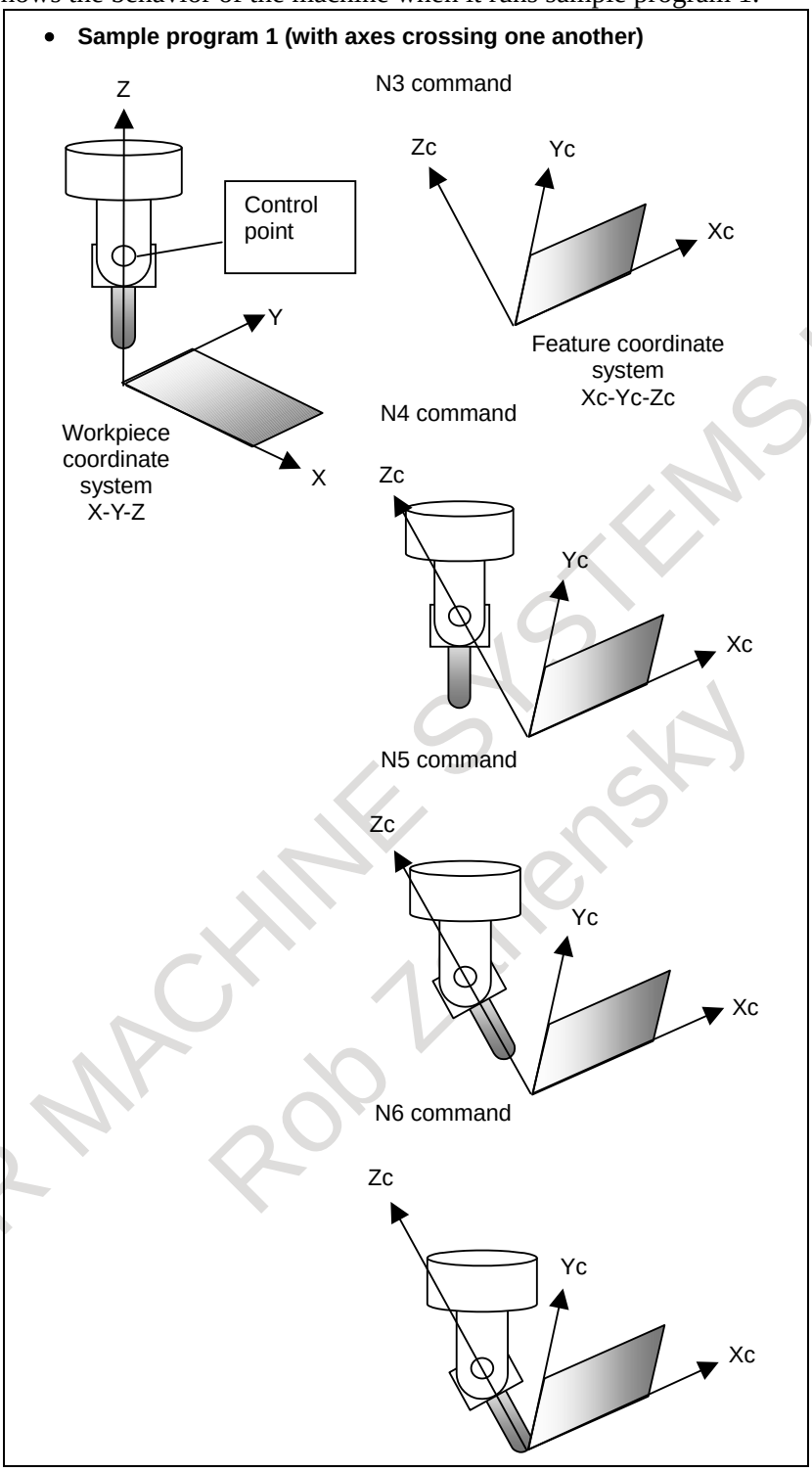


Fig. 14.3.3.1 (b) Tool axis direction control 1

FRYER MACHINE SYSTEMS INC
Rob Zamensky

Operation description 2:
When G43 (tool length compensation) is specified for a machine with no axis crossing

Here is the case where no axis of the machine crosses any other axis.
It is assumed that sample program 1 is used.

In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
It is assumed, however, that the tool axis does not cross the B-axis while the B-axis and C-axis cross each other.

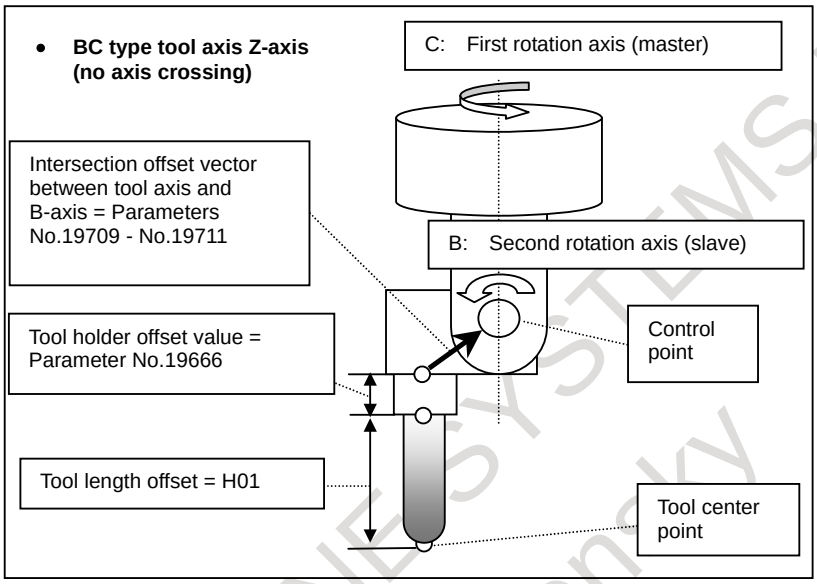


Fig. 14.3.3.1 (c)

Block N4 : Shifts the control point to point Z30.0 in the feature coordinate system.
Block N5 : Exerts automatic control over the rotary axes.
Block N6 : An intersection offset vector between the tool axis and the B-axis with automatic control for rotary axes taken into consideration is output in the feature coordinate system.
Performs tool length compensation in the feature coordinate system.
The tool center point is shifted to the origin of the feature coordinate system.

This is also true when the B-axis does not cross the C-axis.
For explanations about the offset to be applied when the B-axis does not cross the C-axis, see the descriptions about parameters No.19712, No.19713, and No.19714.

Fig. 14.3.3.1 (d) shows the behavior of the machine when it runs sample program 1.

FRYER MACHINE SYSTEMS INC
Rob Zahenky

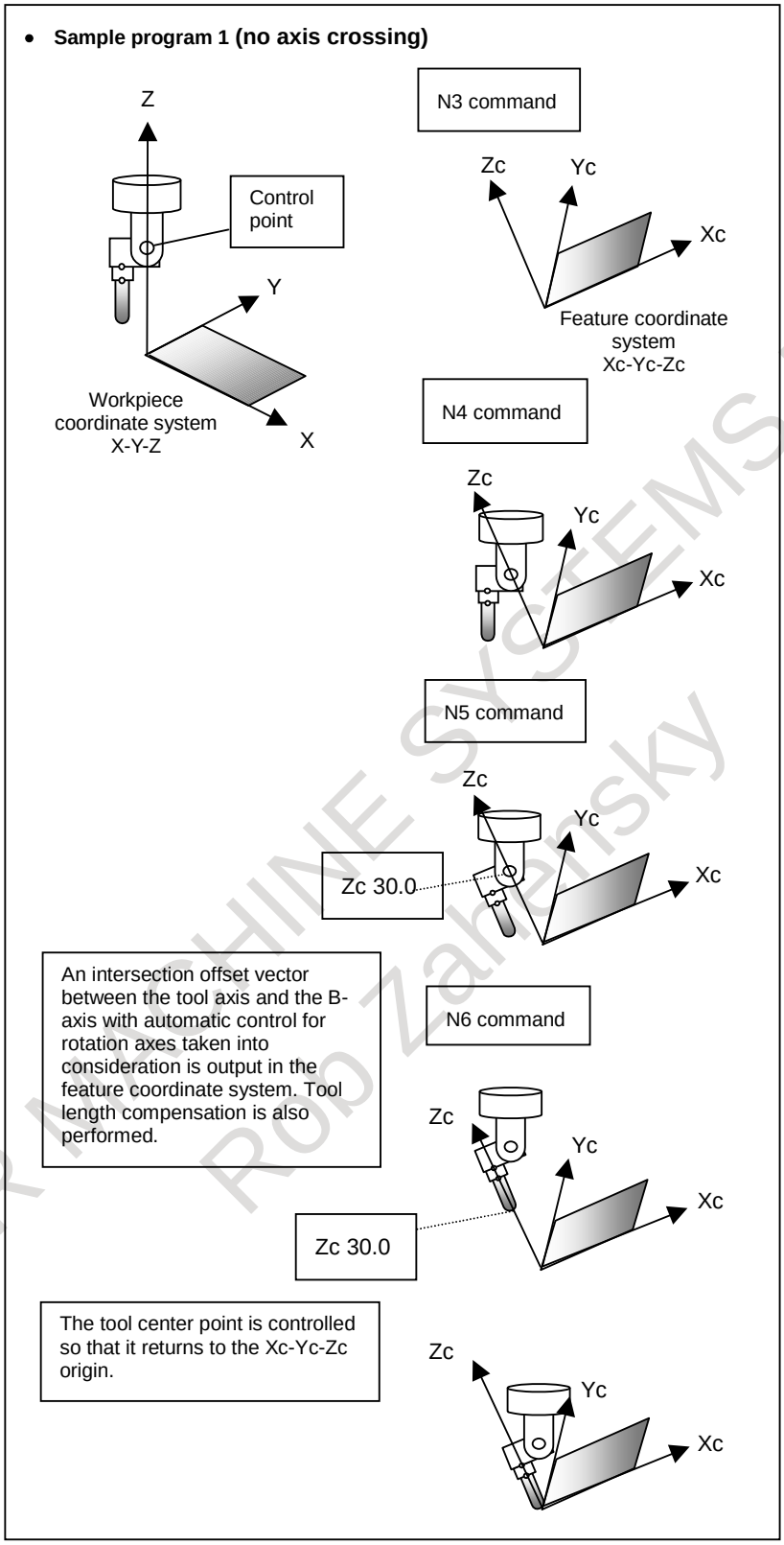


Fig. 14.3.3.1 (d) Tool axis direction control 2

FRYER MACHINE SYSTEMS INC
Rob Zahernsky

Operation description 3:
When no G43 (tool length compensation) command is specified or if no G53.1 (tool axis direction control) command is specified

Sample program 2 of O200 is equivalent to sample program 1 except that sample program 2 has no tool length compensation command (G43).
Example)

```
O200 (Sample Program2) ;
N1 G55 ;
N2 G90 G01 X0 Y0 Z30.0 F1000 ;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0 ;
N4 G01 X0 Y0 Z0 F1000 ;
N5 G53.1 ;
N6 . . . ;
```

In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
The case in which the axes cross one another and the case in which no axis crosses any other axis are described.
Fig. 14.3.3.1 (e) shows the behavior of the machine when it runs sample program 2.

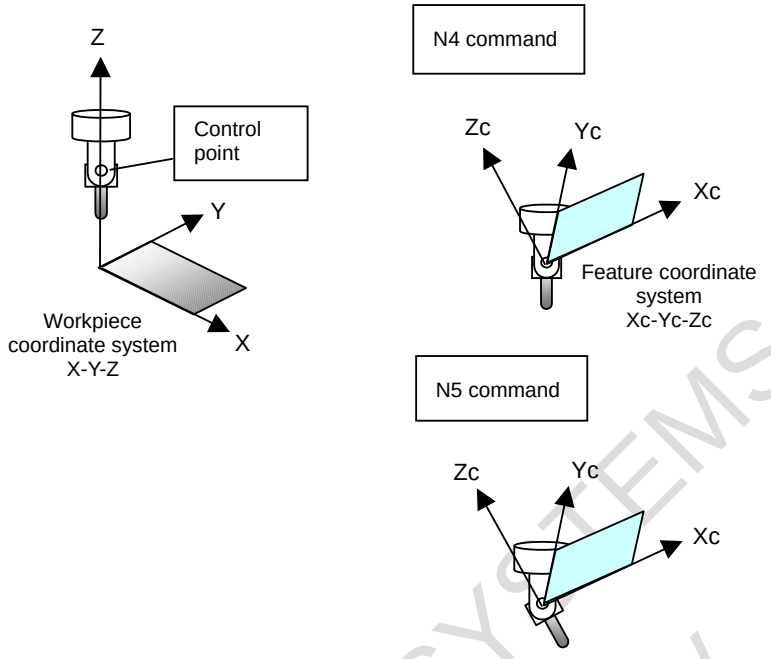
Sample program 3 of O300 is equivalent to sample program 1 except that sample program 3 has no tool axis direction control command (G53.1).
Example)

```
O300 (Sample Program3) ;
N1 G55 ;
N2 G90 G01 X0 Y0 Z30.0 F1000 ;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0 ;
N4 G01 X0 Y0 Z0 F1000 ;
N5 G43 H01 ;
N6 . . . ;
```

In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
The case in which the axes cross one another and the case in which no axis crosses any other axis are described.
Tool length compensation is applied in the +Z-axis direction of the feature coordinate system.
Fig. 14.3.3.1 (f) shows the behavior of the machine when it runs sample program 3.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- **Sample program 2 (with axes crossing one another)**



- **Sample program 2 (no axis crossing)**

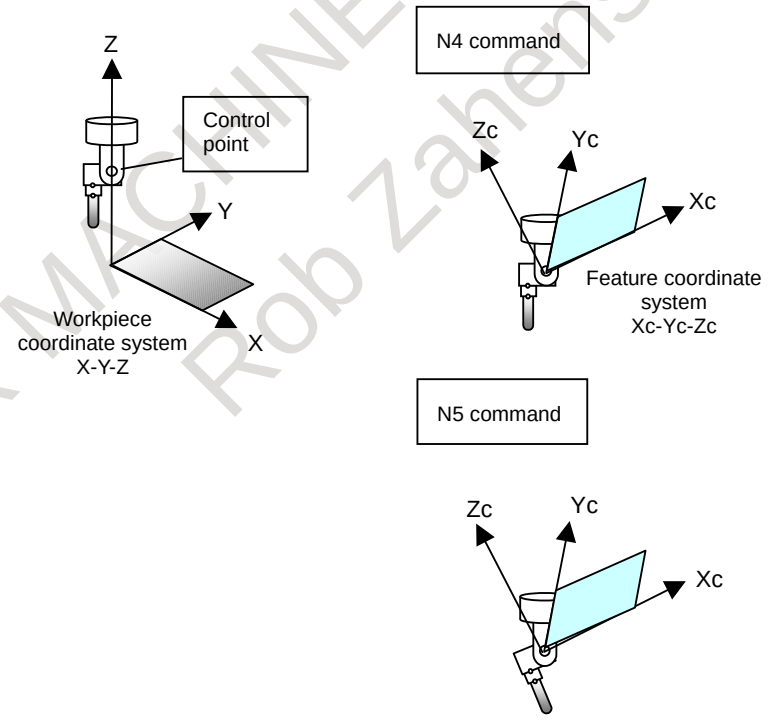
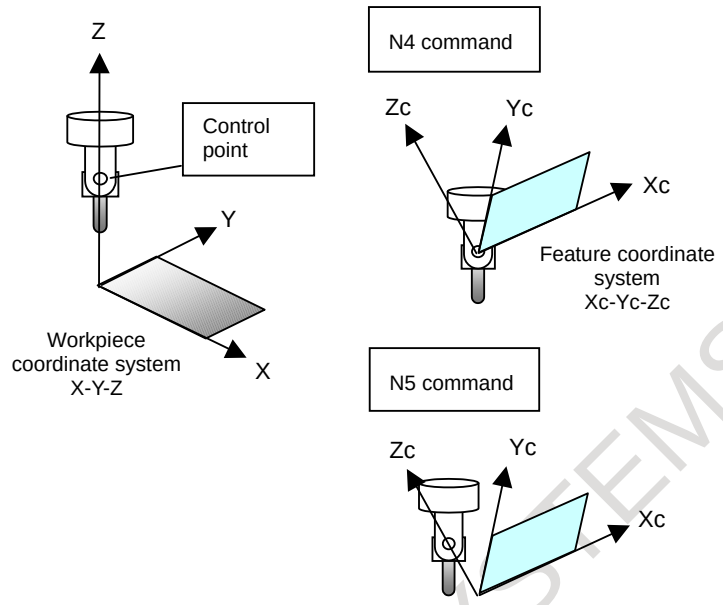


Fig. 14.3.3.1 (e) When the tool length compensation command is not specified

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- **Sample program 3 (with axes crossing one another)**



- **Sample program 3 (no axis crossing)**

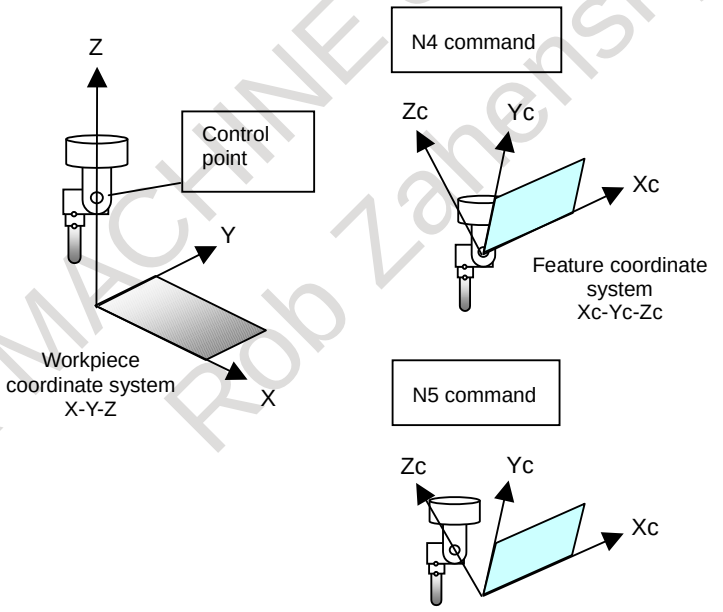


Fig. 14.3.3.1 (f) When the tool axis direction control command is not specified

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Composite type machine
Basic operation

This function is also available for a composite type machine in which the tool head rotates on the tool rotation axis and the table rotates on the table rotation axis.
The feature coordinate system Xc-Yc-Zc is set in the workpiece coordinate system based on the coordinate system origin shift (xo, yo, zo) and the Euler's angle.
Given the A-axis and B-axis shown in Fig. 14.3.3.1 (g), control is performed in such a way that the A-axis rotates until Zc comes in the X-Z plane and the B-axis is controlled so that the tool axis is oriented toward the +Z-axis direction of the feature coordinate system.

- **Tool axis direction control for a composite type machine**

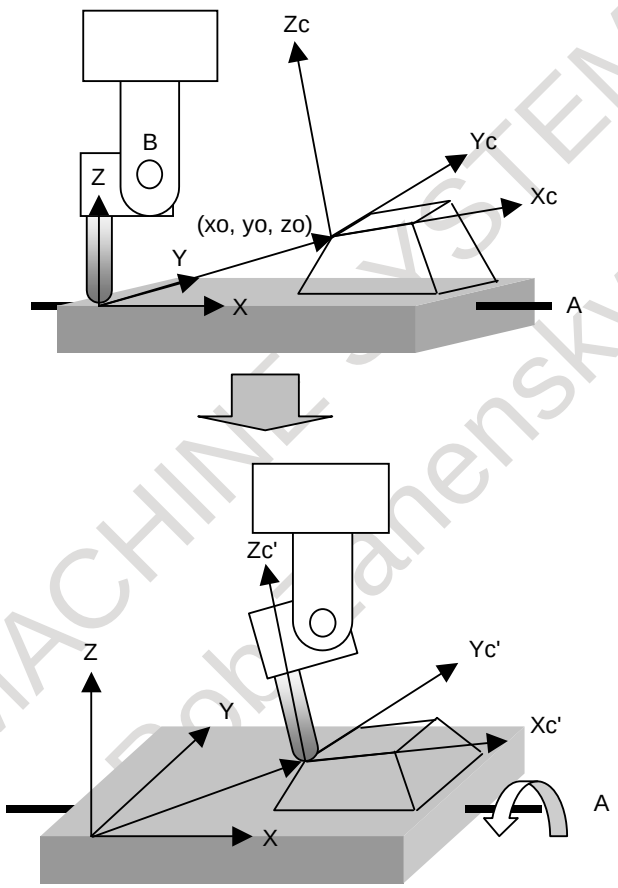


Fig. 14.3.3.1 (g) Composite type machine

FRYER MACHINE SYSTEMS INC
Bob Lanensky

- Feature coordinate system with the table rotated by G53.1 (tool axis direction control)

The composite type machine shown in Fig. 14.3.3.1 (g) is explained as an example.
If the table rotates by the tool axis direction control command (G53.1), the feature coordinate system (called the first feature coordinate system), which is set in the workpiece coordinate system by the tilted working plane indexing (G68.2), rotates as much as the table rotates.
The feature coordinate system that has rotated is called the second feature coordinate system.
Once G53.1 is specified, the subsequent machining commands are assumed to be specified in the second feature coordinate system. (See Fig. 14.3.3.1 (h).)
In the composite type machine, the specified feature coordinate system (the first feature coordinate system) may differ from the feature coordinate system to be used for machining (the second feature coordinate system).

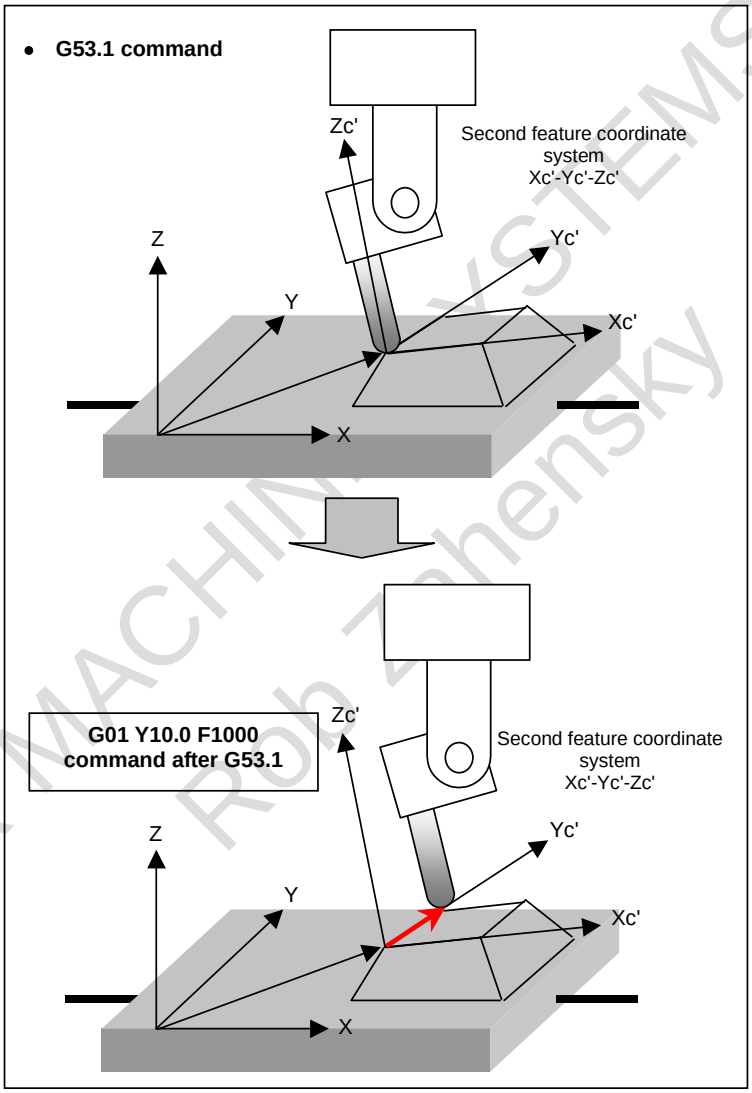


Fig. 14.3.3.1 (h) Resetting of the feature coordinate system

FRYER MACHINE SYSTEMS INC
Rob Zehensky

- Rotation direction of the table rotation axis

The composite type machine shown in Fig. 14.3.3.1 (g) is explained as an example.

Set parameter No.19684 to 1 if the rotation direction of the rotation table corresponding to the positive-direction move command is clockwise when viewed from the positive direction of the rotation center axis on which the table rotation axis rotates. If the rotation direction is counterclockwise, set parameter No.19684 to 0.

Let's take sample program 4 of O400 as an example, where the movement of the table is specified by G53.1.

If parameter No.19684 is set to 1, control is performed in such a way that the table is rotated to A-45.0.

If parameter No.19684 is set to 0, control is performed in such a way that the table is rotated to A45.0.

Example)

```
O400 (Sample Program4) ;
N1 G68.2 X100.0 Y100.0 Z0 I180.0 J45.0 K0 ;
N2 G53.1 ;
N3 . . . ;
```

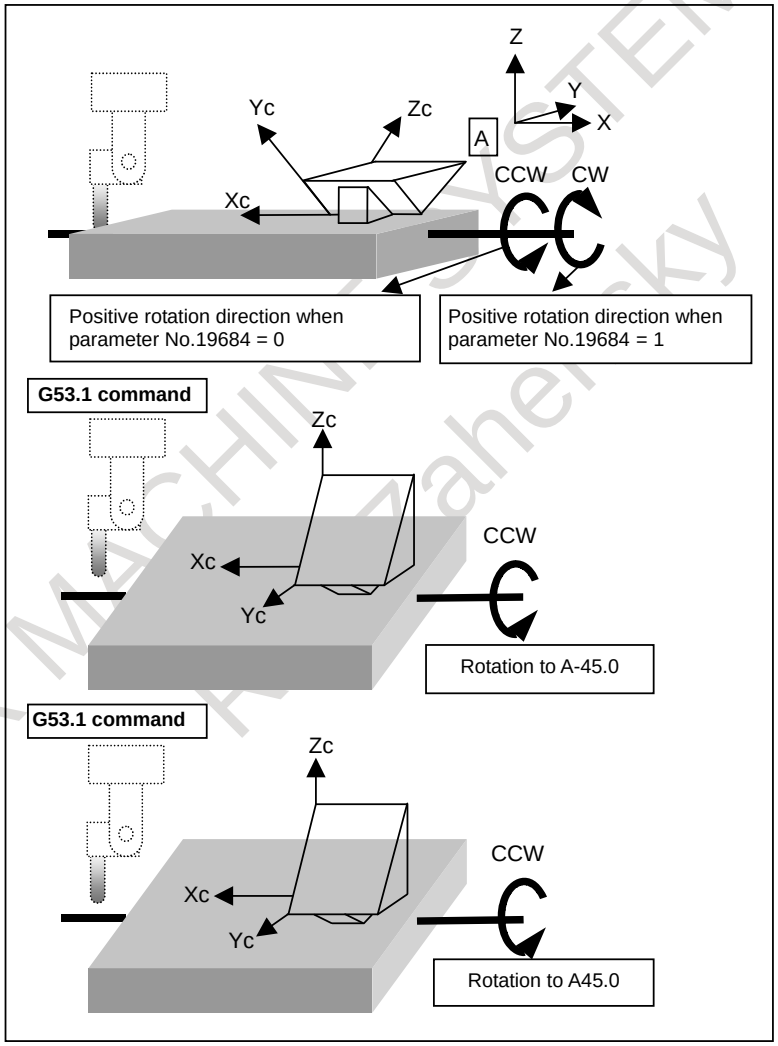


Fig. 14.3.3.1 (i) Rotation direction of the table rotation axis

FRYER MACHINE SYSTEMS INC
R. Zahensky

- Table rotation type machine Basic operation

This function is also usable for a table rotation type machine with two table rotation axes.
The feature coordinate system Xc-Yc-Zc is set in the workpiece coordinate system based on the coordinate system origin shift (xo, yo, zo) and the Euler's angle.
Given the A-axis and C-axis shown in Fig. 14.3.3.1 (j), the A-axis and C-axis rotate until Zc comes in the X-Z plane and the tool axis is directed toward the +Z-axis direction of the feature coordinate system.

- **Tool axis direction control for a table rotation type machine**

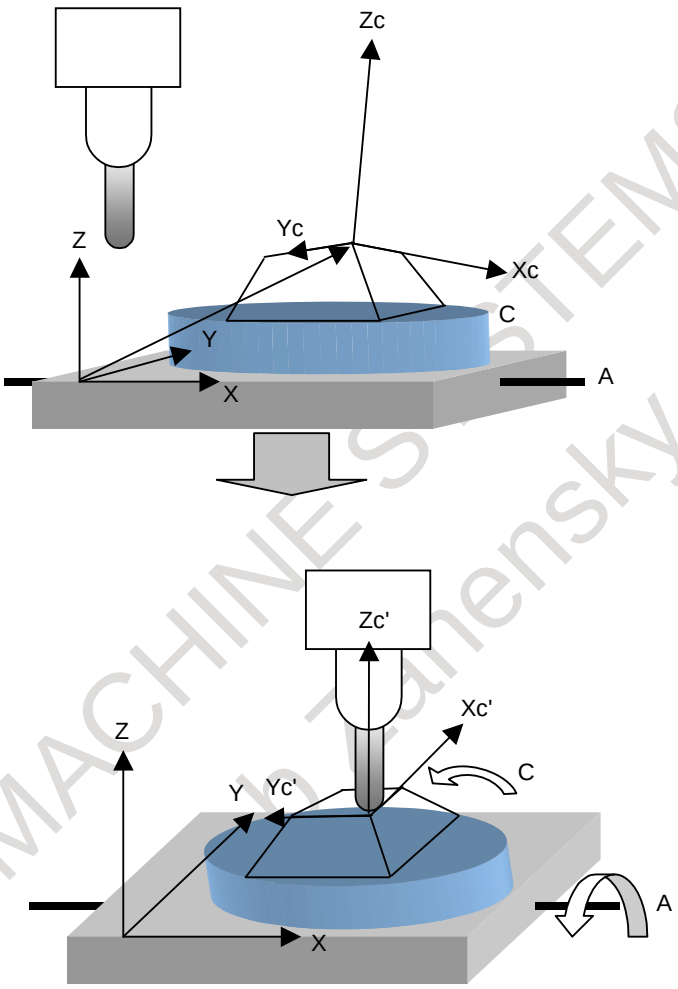


Fig. 14.3.3.1 (j) Table rotation type machine

FRYER MACHINE SYSTEMS INC
In Zahensky

- Feature coordinate system with the table rotated by G53.1 (tool axis direction control)

The table rotation type machine shown in Fig. 14.3.3.1 (j) is explained as an example.
If the table rotates by the tool axis direction control command (G53.1), the feature coordinate system (called the first feature coordinate system), which is set in the workpiece coordinate system by the tilted working plane indexing (G68.2), rotates as much as the table rotates.
The feature coordinate system that has rotated is called the second feature coordinate system.
Once G53.1 is specified, the subsequent machining commands are assumed to be specified in the second feature coordinate system. (See Fig. 14.3.3.1 (k).)
In the table rotation type machine, the specified feature coordinate system (the first feature coordinate system) may differ from the feature coordinate system to be used for machining (the second feature coordinate system).

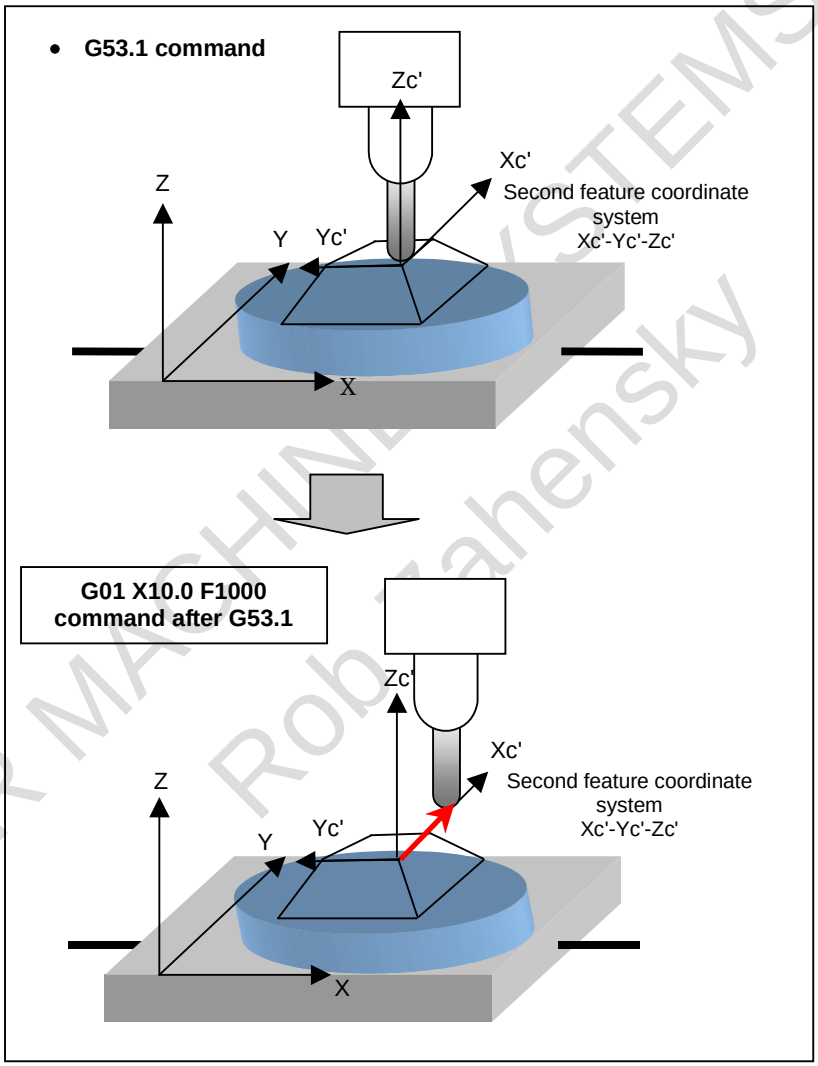


Fig. 14.3.3.1 (k) Resetting of the feature coordinate system

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Angle of the rotary axis

When tool axis direction control (G53.1) has been performed, more than two pairs of "computed angles" of the rotary axes usually exist.
The "computed angle" is the candidate angle at which the rotary axis is to be controlled in the tool axis direction specified by G53.1.
The "output angle" is determined from the "computed angle" based on the "output judgment conditions" described below.
When the upper and lower limits of the movement range of rotary axes are specified by parameters No. 19741 to No. 19744 at this time, a calculated angle that makes the two axes fall within the movement range is selected.

"Output judgment conditions"

Tool rotation type or table rotation type machine	
<1>	The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) moving angle is smaller. ↓ ↓ When the master axis moving angle is the same ↓
<2>	The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) moving angle is smaller. ↓ ↓ When the slave axis moving angle is the same ↓
<3>	The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees). ↓ ↓ When the master axis angle is equally near to 0 degree ↓
<4>	The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).
Composite type machine	
<1>	The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) moving angle is smaller. ↓ ↓ When the table moving angle is the same ↓
<2>	The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) moving angle is smaller. ↓ ↓ When the tool moving angle is the same ↓
<3>	The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees). ↓ ↓ When the master axis angle is equally near to 0 degree ↓
<4>	The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

The process of judging whether the moving angle is smaller or larger as the output judgement condition is called "movement judgement."
When bit 5 (PRI) of parameter No.19608 is 1, the movement judgements for the first rotary axis and second rotary axis are made in reverse order.

The "movement judgement" process is explained below.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

When the "computed angle" is within the range between 0 and 360 degrees, it is called the "basic computed angle."
Usually, two pairs of "basic computed angles" exist.
For example, assume that the machine has rotary axis A (master) and rotary axis B (slave) and that there are two pairs of basic computed angles as follows:
(A θ1 degree; B ϕ1 degree)
(A θ2 degrees; B ϕ2 degrees) where θ1 ≤ θ2 and ϕ1 ≤ ϕ2.
The "computed angle" is obtained from either of the following expressions: "basic computed angle" + 360 degrees × N or "basic computed angle" - 360 degrees × N.
The current position of rotary axis A (master) is PA, and that of rotary axis B (slave) is 0 degree.
Based on the PA angle, the "movement judgement" process is done as follows (when bit 5 (PRI) of parameter No.19608 is 0).

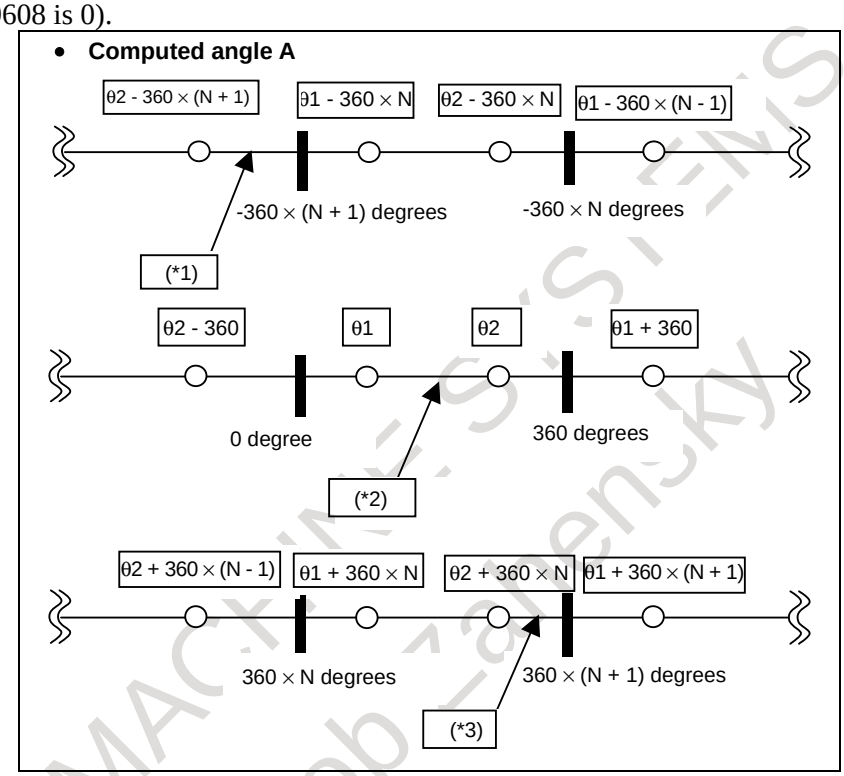


Fig. 14.3.3.1 (l) "Movement judgment"

When the PA angle is (*1):
The output angle is: (A θ2 - 360 × (N + 1) degrees; B ϕ2 degrees).
Namely, θ2 - 360 × (N + 1) degrees is adopted that is nearer to the computed angle of A, and ϕ2, which is the same group as θ2, is adopted as the computed angle of B.

When the PA angle is (*2):
The output angle is: (A θ1 degrees; B ϕ1 degrees).
Namely, θ1 degrees is adopted that is nearer to the computed angle of A, and ϕ1, which is the same group as θ1, is adopted as the computed angle of B.

When the PA angle is (*3):
The output angle is: (A θ2 + 360 × N degrees; B ϕ2 degrees).
Namely, θ2 + 360 × N degrees is adopted that is nearer to the computed angle of A, and ϕ2, which is the same group as θ2. is adopted as the computed angle of B.

When the moving angle of rotary axis A (master) is the same, a "movement judgement" is made for rotary axis B (slave) according to the "output judgment conditions."

FRYER MACHINE SYSTEMS INC Rob Zahensky

If the "output angle" of rotary axis A is determined by the "movement judgement" for rotary axis A, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis B. Similarly, if the "output angle" of rotary axis B is determined by the "movement judgement" for rotary axis B, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis A.

⚠ CAUTION

1 To use the rotary axis roll-over function, set parameter No. 1260 (amount of rotary axis movement per rotation) to 360 degrees.
2 A stroke limit before movement is applied to the rotary axis subject to tool axis direction control.
3 If the setting of the lower limit (parameters Nos. 19742 and 19744) is greater than that of the upper limit (parameters Nos. 19741 and 19743), alarm PS5459 , “MACHINE PARAMETER INCORRECT” is issued.
4 If there is no calculated angle that falls within the movement range because the movement range is too small, alarm PS5459 is issued.
5 When the parameters that specify the upper limit and lower limit of the movement range are set to 0, no movement range is assumed to be specified.

The "output angle" is explained below using a tool rotation type machine as an example. (Assume that bit 5 (PRI) of parameter No.19608 is 0.)
This example illustrates a machine having a "BC type tool axis Z."

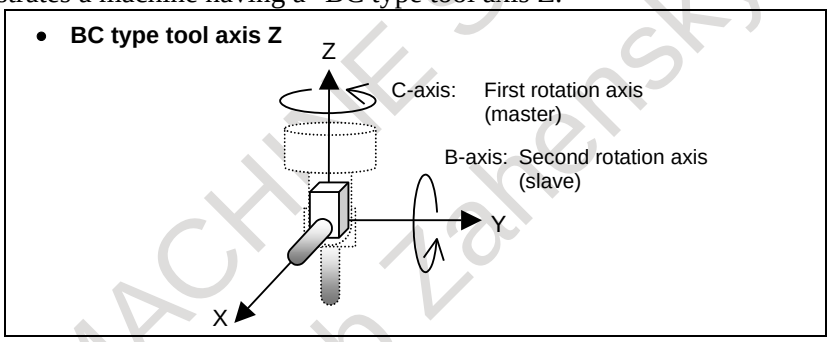


Fig. 14.3.3.1 (m) BC type tool axis Z

The following two pairs of "computed basic angles" exist that direct the tool axis toward the +X-axis direction.
(B 90 degrees; C 180 degrees)
(B 270 degrees; C 0 degree)

<1> When the current rotary axis angles are (B -70 degrees; C 30 degrees)
The "output angles" are (B -90 degrees; C 0 degree).
0 degree is adopted because it is nearer to the current position (30 degrees) of the C-axis that is the master axis. For the B-axis, 270 degrees is adopted which is the same group. However, this is changed to -90 degrees (270 degrees - 360 degrees) which is the nearest to the current position of the B-axis (-70 degrees).

<2> When the current rotary axis angles are (B 80 degrees; C 500 degrees)
The "output angles" are (B 90 degrees; C 540 degrees).
540 degrees (180 degrees + 360 degrees) is adopted because it is nearer to the current position (500 degrees) of the C-axis that is the master axis. For the B-axis, 90 degrees is adopted which is the same group.

<3> When the current rotary axis angles are (B 60 degrees; C 90 degrees)
The "output angles" are (B 90 degrees; C 180 degrees).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. 90 degrees is adopted

FRYER MACHINE SYSTEMS INC
Rob Zakensky

because it is nearer to the current position (60 degrees) of the B-axis that is the slave axis. For the C-axis, 180 degrees is adopted which is the same group.

<4> When the current rotary axis angles are (B 180 degrees; C 90 degrees)
The "output angles" are (B 270 degrees; C 0 degree).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. In this case, however, the two candidates are also equally near to the current position of the B-axis (180 degrees). Therefore, the candidate is adopted in which the C-axis (master axis) is nearer to 0 degree.
That is, the pair is adopted whose C-axis angle is 0 degree and whose B-axis angle is 270 degrees.

When the slave axis angle is 0 degree, the direction of the tool axis becomes fixed regardless of the master axis angle.
In that case, the master axis does not move from the current angle.

An explanation is shown below using a machine having a "BC type tool axis Z" as an example.

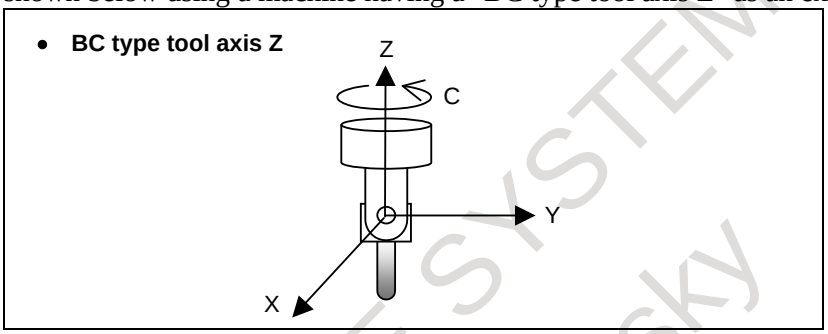


Fig. 14.3.3.1 (n) BC type tool axis Z

When the current rotary axis angles are (B 45 degrees; C 90 degrees), the "output angles" are (B 0 degree; C 90 degrees).

- Specification for rotary axes

The tilted working plane indexing assumes that positioning of a rotary axis is performed by tool axis direction control (G53.1) and then machining is performed without the rotary axis being moved.
When movement is made with the rotary axis specified directly, the movement of the rotary axis is not considered in the movement of the linear axis. Do not specify the movement of the rotary axis during execution of the tilted working plane indexing to ensure correct machining.

Second Rotation Axis Control in Tool Axis Direction Control Where the End Point is a Singular Point

When the end point of tool axis direction control (G53.1/G53.6) during execution of the tilted working plane indexing is a singular point, the second rotation axis is controlled so that the direction of the second feature coordinate system matches that of the workpiece coordinate system.
To enable this function, set bit 4 (CFW) of parameter No. 11221 to 1.

When bit 4 (CFW) of parameter No. 11221 is set to 0, if the end point of tool axis direction control (G53.1/G53.6) is a singular point, the second rotation axis does not operate and only the first rotation axis turns. As a result, the X- and Y-directions of the feature coordinate system depend on the second rotation axis immediately before tool axis direction control. (Fig. 14.3.3.1 (o), Fig. 14.3.3.1 (p))

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

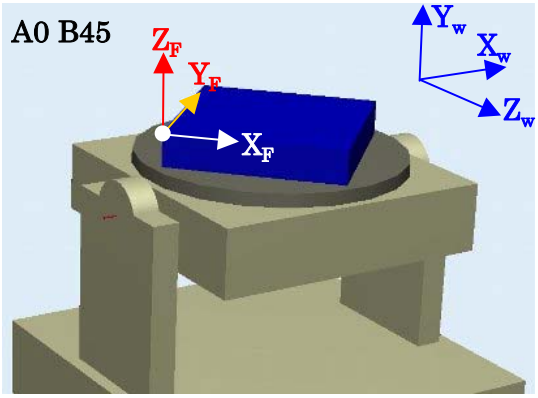


Fig. 14.3.3.1 (o) Before G53.1

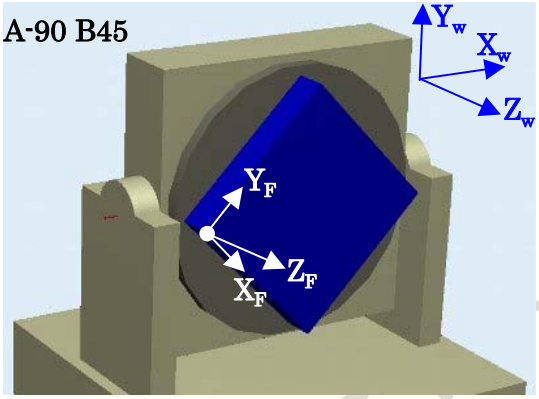


Fig. 14.3.3.1 (p) After G53.1
The second rotation axis (B-axis) does not turn.

When this function is enabled (bit 4 (CFW) of parameter No. 11221 = 1), the second rotation axis is controlled so that the direction of the second feature coordinate system matches that of the workpiece coordinate system. (Fig. 14.3.3.1 (q),Fig. 14.3.3.1 (r))

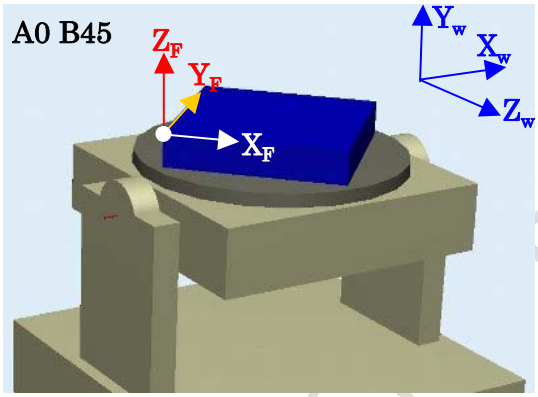


Fig. 14.3.3.1 (q) Before G53.1

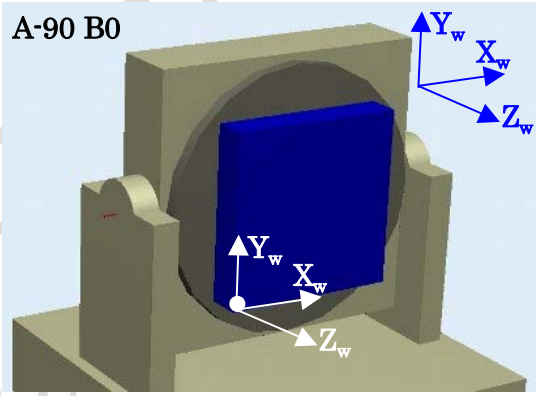


Fig. 14.3.3.1 (r) After G53.1
The second rotation axis (B-axis) moves so that the direction of the second feature coordinate system matches that of the workpiece coordinate system.

14.3.3.2 Tool center point retention type tool axis direction control

In tilted working plane indexing, tool center point retention type tool axis direction control (G53.6) can be specified so that tool can be perpendicular to the tilted plane with the tool center point maintained on the workpiece. In tool center point retention type tool axis direction control, end point of tool center point can be specified. Then tool center point moves on the feature coordinate system fixed on the workpiece. The cycle time can be shortened by moving tool axis direction and tool center point simultaneously.
By specifying the distance between the tool center point to the rotation center with R, it is possible to move the tool so that it is perpendicular to the titled plane while retaining the rotation center shifted from the tool center point.

Format

M

G53.6 (H_) (R_) X_ Y_ Z_ ; Tool center point retention type tool axis direction control

H : Tool compensation number
R : Distance from the tool center point to the rotation center
(Specify R as a radius value.)
X_ Y_ Z_ : End point of tool center point

G53.6 is a one shot G code.
To use tool center point retention type tool axis direction control, a tool length offset number is required. If no H command is specified, the H code is regarded as modal information.
To shift the rotation center from the tool center point, enter the distance from the tool center point to the rotation center, using the R command. If no R command is specified, the distance is regarded as R0, and the tool moves while retaining the tool center position on the workpiece so that it is perpendicular to the tilted plane.
Specify end point of tool center point in feature coordinate system.

T

G53.6 (D_) (R_) X_ Y_ Z_ ; Tool center point retention type tool axis direction control

D : Tool compensation number
R : Distance from the tool center point to the rotation center
(Specify R as a radius value.)
X_ Y_ Z_ : End point of tool center point

To use tool center point retention type tool axis direction control on a T series machine, enable the extended tool selection function (set bit 3 (TCT) of parameter No. 5040 = 1).
Use D as the tool length offset number.
Other restrictions are the same as those on the M series.

⚠ **CAUTION**

1 Specify tool center point retention type tool axis direction control (G53.6) in tool length compensation cancel mode (G49). If it is specified in a mode other than G49 mode, alarm PS5458, ”ILLEGAL USE OF G53.1/G53.6” is issued.
2 If G68.2 is not specified before G53.6 is specified, alarm PS5458 is issued.
3 Specify G53.6 independently. If an axis movement command is specified in the same block, alarm PS5458 is issued.
4 The tool center point retention type tool axis direction control (G53.6) block becomes a block that suppresses buffering.
5 If tool center point retention type tool axis direction control does not have an H command (D command for the T series), and the modal information for H (D) is 0, alarm PS5458 is issued.

FRYER MACHINE SYSTEMS INC Rob Zahensky

⚠ CAUTION

6 For the feedrate, the movement speed of the rotation axis is applied. During rapid traverse, it is regarded as the maximum rapid traverse rate, and as the specified speed during cutting feed.
7 Specify tool center point retention type tool axis direction control (G53.6) in cutter compensation cancel mode (G40). If it is specified in a mode other than G40 mode, alarm PS5458 is issued.
8 Specify tool center point retention type tool axis direction control (G53.6) in either G00 or G01 mode.
9 Specify tool center point retention type tool axis direction control (G53.6) with feed per minute or feed per revolution.
10 Do not perform manual intervention during tool center point retention type tool axis direction control (G53.6). Otherwise, alarm PS5458 is issued.
11 If the R command is specified, and if an exceedingly large value is input as R, alarm PS0143, ”COMMAND DATA OVERFLOW” may be issued.

Example

Tool center point retention type tool axis direction control with R not specified

Fig. 14.3.3.2 (a) and Fig. 14.3.3.2 (b) show tool center point retention type tool axis direction control with R not specified. The tool moves so that it is perpendicular to the tilted plane while retaining the tool center point on the workpiece.

This function can be used by specifying G53.6 and specifying a tool length offset number with H (D for the T series). (If H (D) has the modal information for the currently used tool, this function can be used without specifying H (D).)

```
O0002(TCP-HOLD-TYPE)
G00 B0 C0
G5.1 Q1
G68.2 X0 Y0 Z0 I90.0 J45.0 K0
G53.6 H1 (G53.6 D1 for the T series)
```

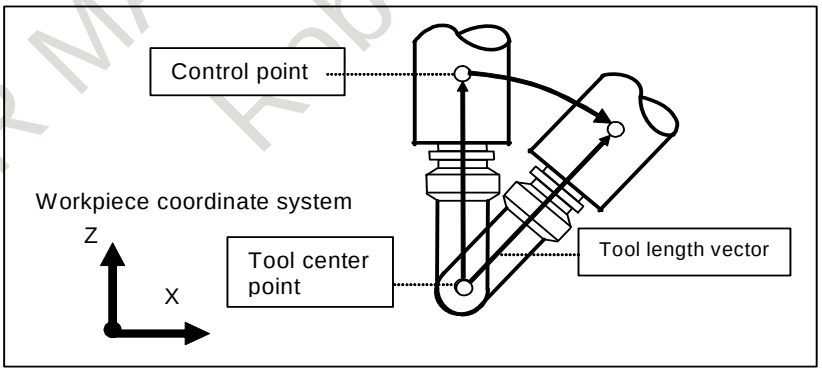


Fig. 14.3.3.2 (a) Operation of tool center point retention type tool axis direction control (tool rotation type)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

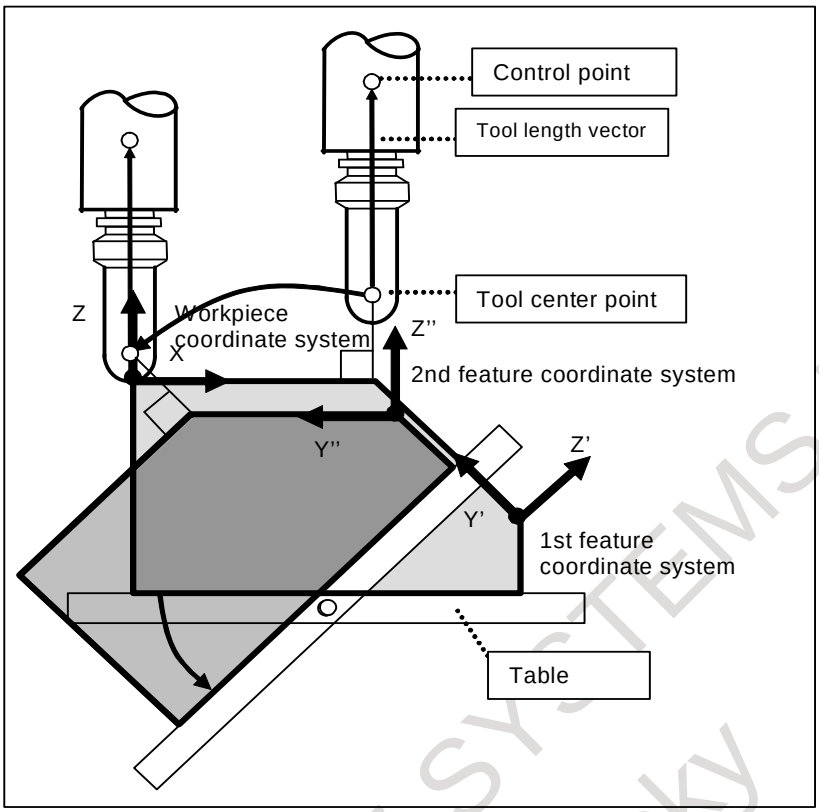


Fig. 14.3.3.2 (b) Operation of tool center point retention type tool axis direction control (table rotation type)

Fig.14.3.3.2 (c) and Fig.14.3.3.2 (d) show tool center point retention type tool axis direction control specified with end point of tool center point. Tool center point moves on the feature coordinate system fixed on the workpiece.

O0012(TCP-HOLD-TYPE-TOOL_ROT)
G00 B0 C0
G5.1 Q1
G68.2 X0 Y0 Z0 I90.0 J45.0 K-90.0
G53.6 H1 X100.0 Y0 Z0 (G53.6 D1 X100.0 Y0 Z0 for the T series)

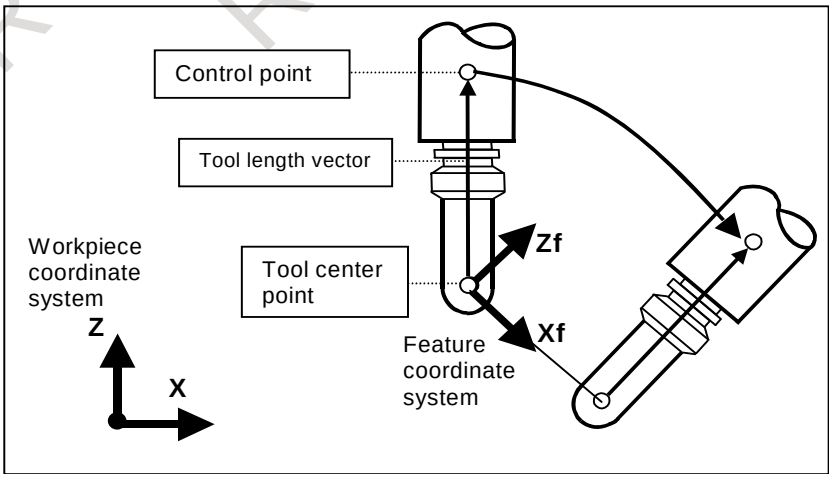


Fig.14.3.3.2 (c) Operation of tool center point retention type tool axis direction control specified with end point of tool center point (tool rotation type)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

O0022(TCP-HOLD-TYPE-TABLE_ROT)
G00 B0 C0
G5.1 Q1
G68.2 X0 Y0 Z0 I90.0 J45.0 K-90.0
G53.6 H1 X0 Y0 Z0 (G53.6 D1 X0 Y0 Z0 for the T series)

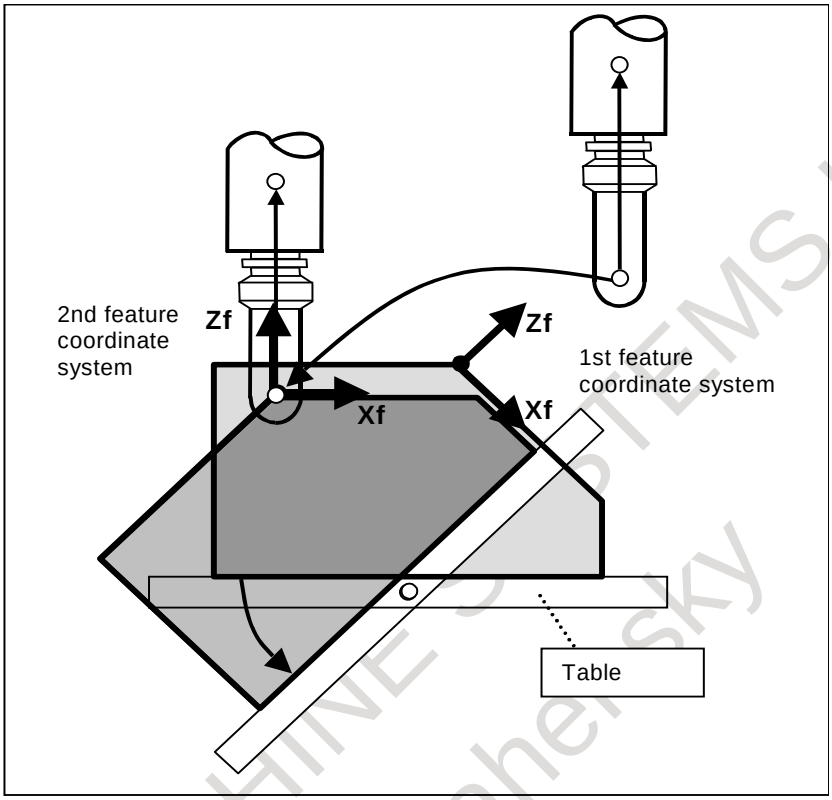


Fig.14.3.3.2 (d) Operation of tool center point retention type tool axis direction control specified with end point of tool center point (table rotation type)

Rotation center compensation with the tool center point retention type
In tool center point retention type tool axis direction control, the rotation center can be shifted from the tool center point by specifying R.
Fig. 14.3.3.2 (e) and Fig. 14.3.3.2 (f) show cases in which the rotation center is shifted by specifying the distance from the tool center point to the workpiece with R.
By specifying this, the tool moves so that it is perpendicular to the titled plane while retaining the rotation center on the workpiece.
This function can be used by specifying G53.6 and specifying a tool length offset number with H (D for the T series) and specifying the distance from the tool center point to the rotation center with R.

O0003(CENTER-OF-ROTATION-HOLD-TYPE)
G00 B0 C0
G5.1 Q1
G68.2 X0 Y0 Z0 I90.0 J45.0 K0
G53.6 H1 R200.0 (G53.6 D1 R200.0 for the T series)

FRYER MACHINE SYSTEMS INC
Rob Zaherlsky

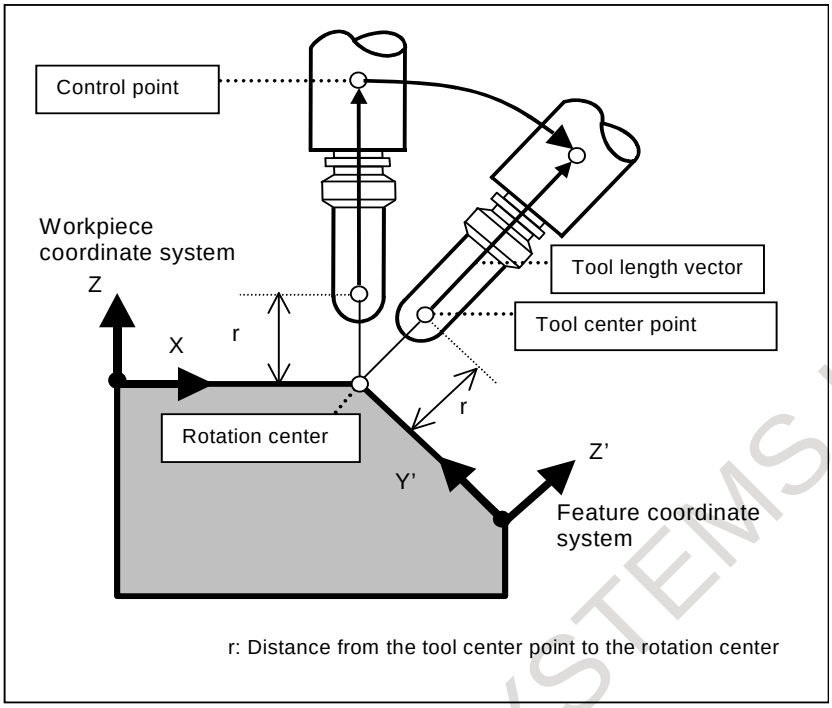


Fig. 14.3.3.2 (e) Operation of rotation center compensation with tool center point retention type tool axis direction control (tool rotation type)

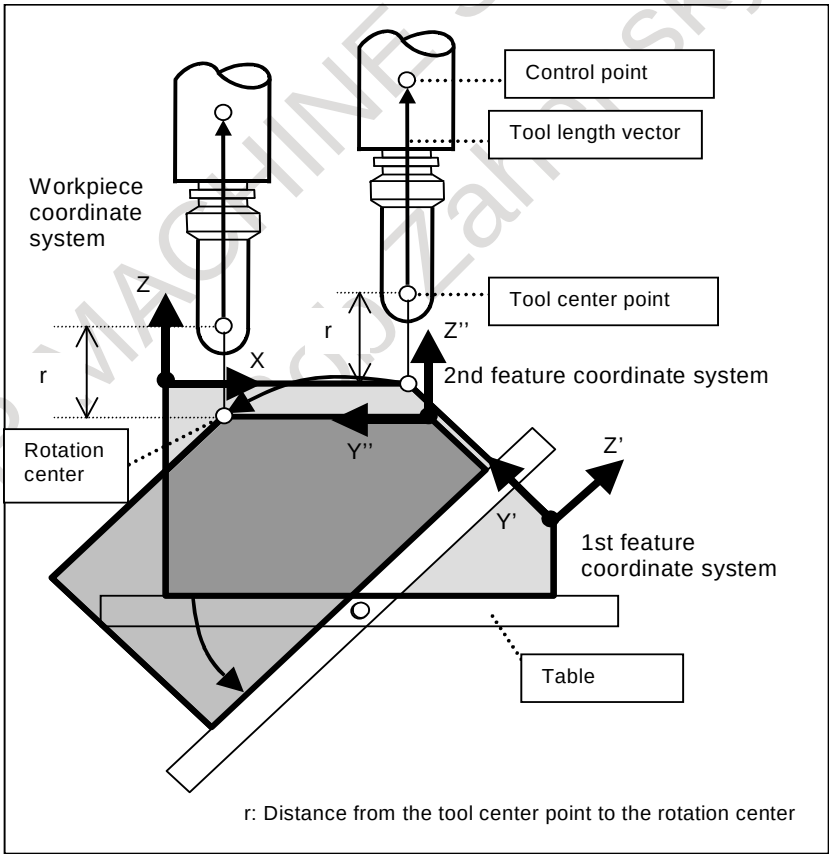


Fig. 14.3.3.2 (f) Operation of rotation center compensation with tool center point retention type tool axis direction control (table rotation type)

FRYER MACHINE SYSTEMS INC

14.3.4 Tilted Working Plane Indexing in Tool Length Compensation

Overview

In tool length compensation (G43), G68.2/G68.4 (tilted working plane indexing) and G53.1 (tool axis direction control)/G53.6 (tool center point retention type tool axis direction control) can be specified.
Accordingly, the G68.2/G68.4 and G53.1/G53.6 commands can be used without canceling tool length compensation.

Explanation

- G68.2/G68.4 command in tool length compensation

The G68.2/G68.4 command can be executed in tool length compensation.
Absolute coordinates after the G68.2/G68.4 command are based on the position of the tool center point on the feature coordinate system.
When the tilted working plane indexing is executed with the tool or table tilted on the rotation axis, absolute coordinates are based on the position of the tool center point with the position of the rotation axis considered.
Accordingly, machining is allowed even when the tool axis direction is not the Z-axis direction on the feature coordinate system.

Example of operation 1

N10 G69 ;
N20 G54 **G43** H1 X0 Y0 Z0 ;
N30 **G68.2** X_ Y_ Z_ I90. 0J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N40 X100.0 Y0 Z0 ;

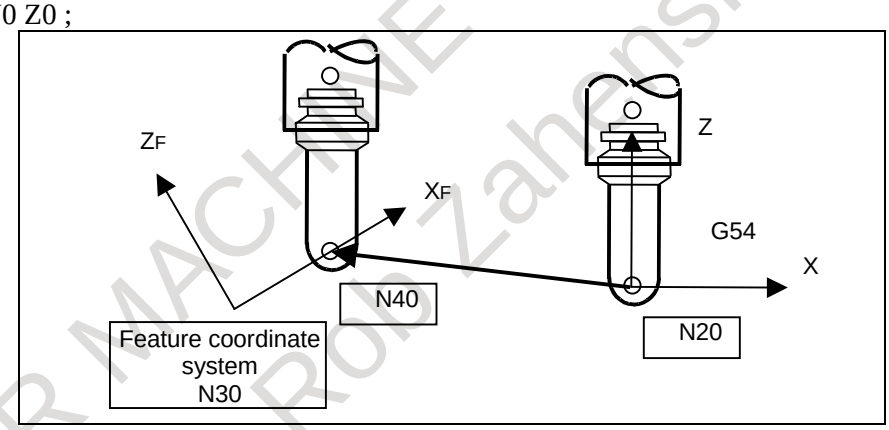


Fig. 14.3.4 (a) Example of operation 1

Example of operation 2

N10 G69 ;
N20 G54 **G43** H1 X0 Y0 Z0 B0 ;
N30 B20.0 ;
N40 **G68.2** X_ Y_ Z_ I90.0 J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N50 X100.0 Y0 Z0 ;

FRYER MACHINE SYSTEMS INC
Rob Zahensky

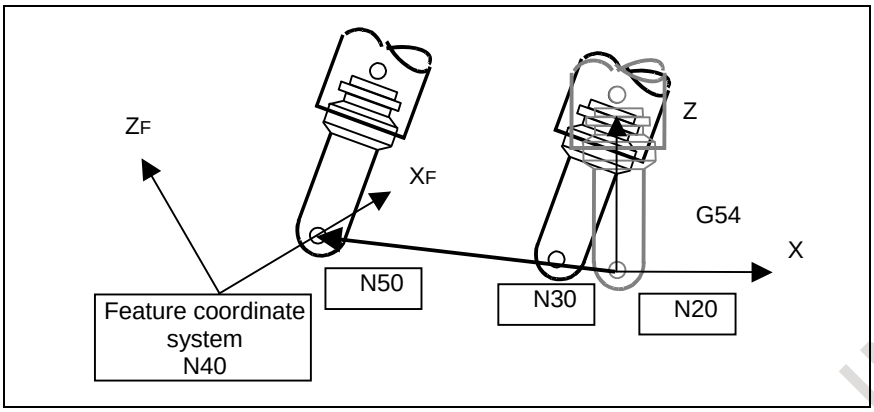


Fig. 14.3.4 (b) Example of operation 2 (tool rotation type)

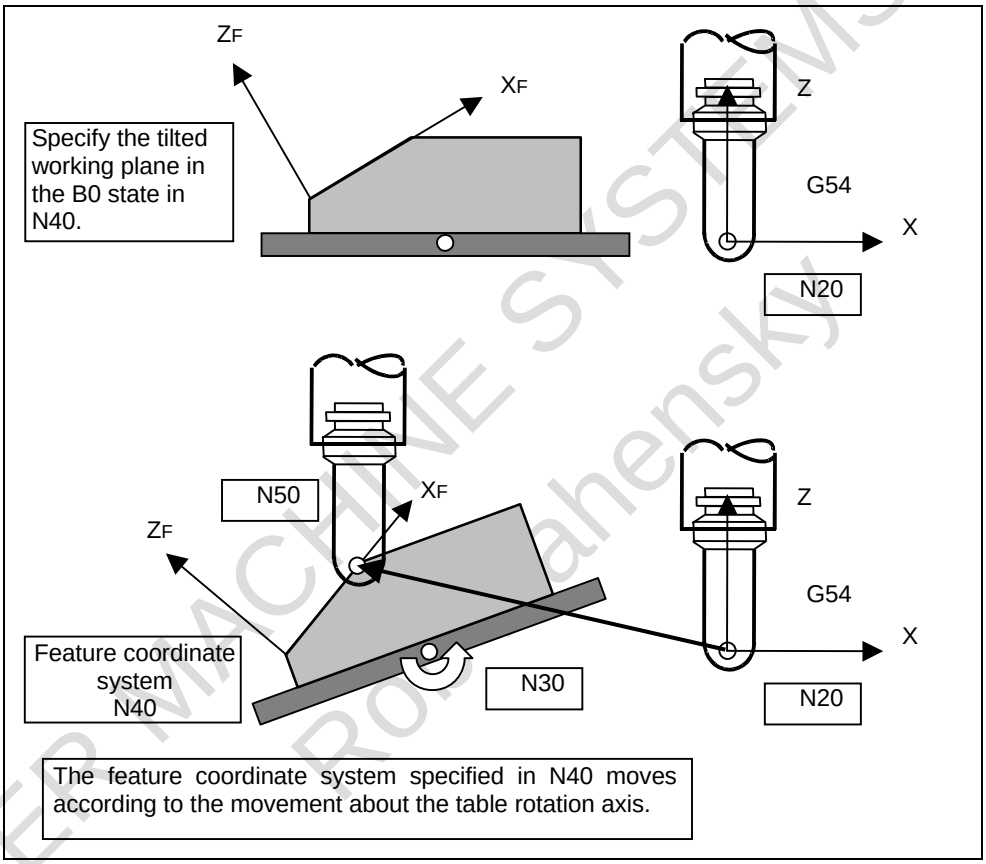


Fig. 14.3.4 (c) Example of operation 2 (table rotation type)

Example of operation 3

N30 **G43** X0 Y0 Z0 B20.0 ;
N40 **G68.2** X_ Y_ Z_ I90.0 J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N50 X100.0 Y0 Z0 ;
N60 B-20.0
N70 **G68.4** X_ Y_ Z_ I90.0 J40.0 K-90.0 ; （incremental multiple command: rotation by 40 degrees about the Y-axis）
N80 X100.0 Y0 Z0 ;

FRYER MACHINE SYSTEMS INC
Rob Lahensky

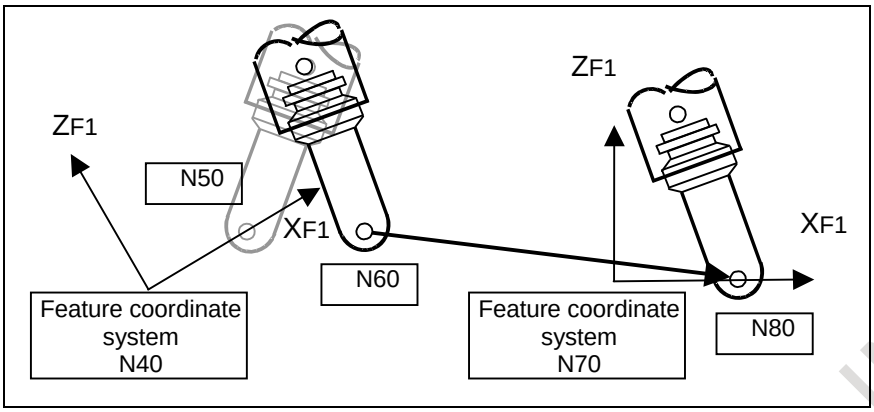


Fig. 14.3.4 (d) Example of operation 3 (tool rotation type)

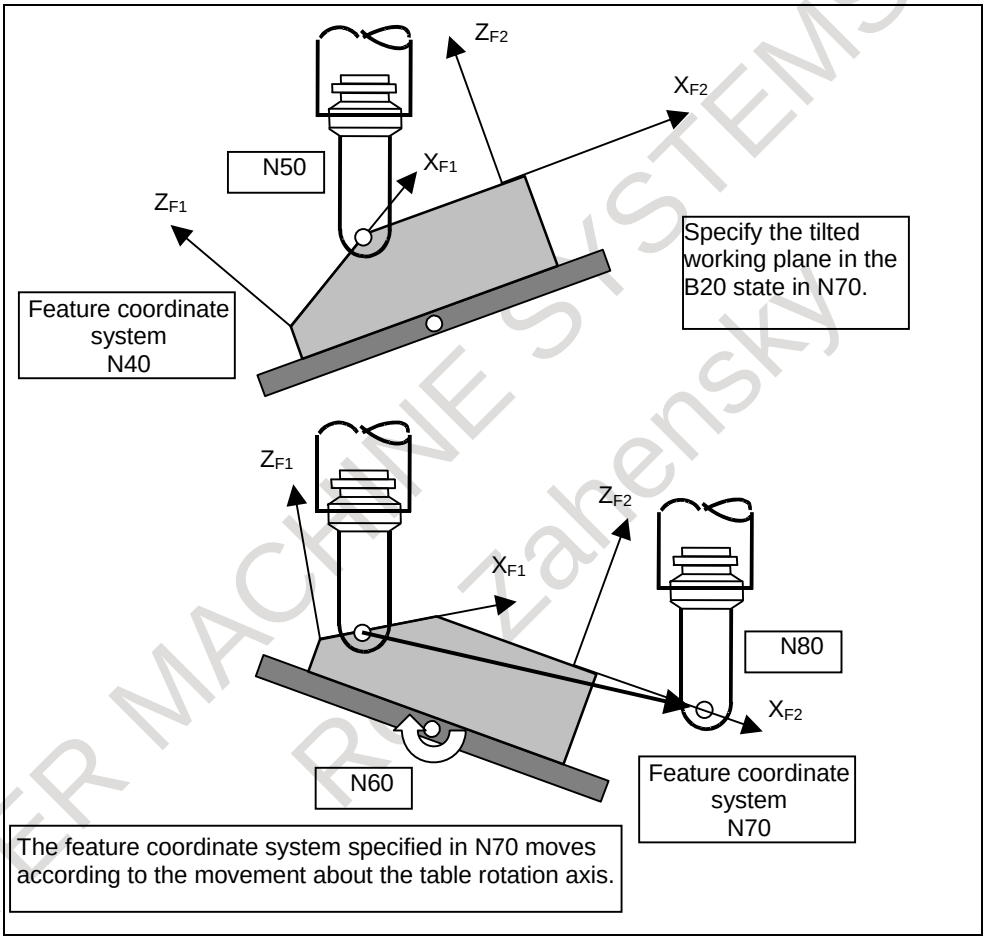


Fig. 14.3.4 (e) Example of operation 3 (table rotation type)

FRYER MACHINE SYSTEMS INC
R. Zahensky

G53.1 command in tool length compensation

The G53.1 command can be executed in tool length compensation.

G53.1 operation in tool length compensation is performed in the same way as in the tool length compensation cancel mode.

Absolute coordinates after the G53.1 command are based on the position of the tool center point on the feature coordinate system after the G53.1 command is specified.

Example of operation 4

N10 G54 **G43** H1 X_ Y_ Z_ ;
N20 **G68.2** X_ Y_ Z_ I90.0 J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N30 **G53.1** ;
N40 X100.0 Y0 Z0 ;

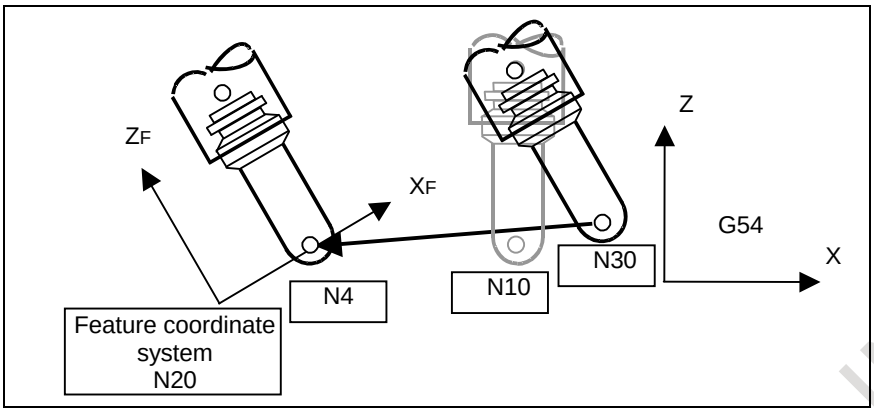


Fig. 14.3.4 (f) Example of operation 4 (tool rotation type)

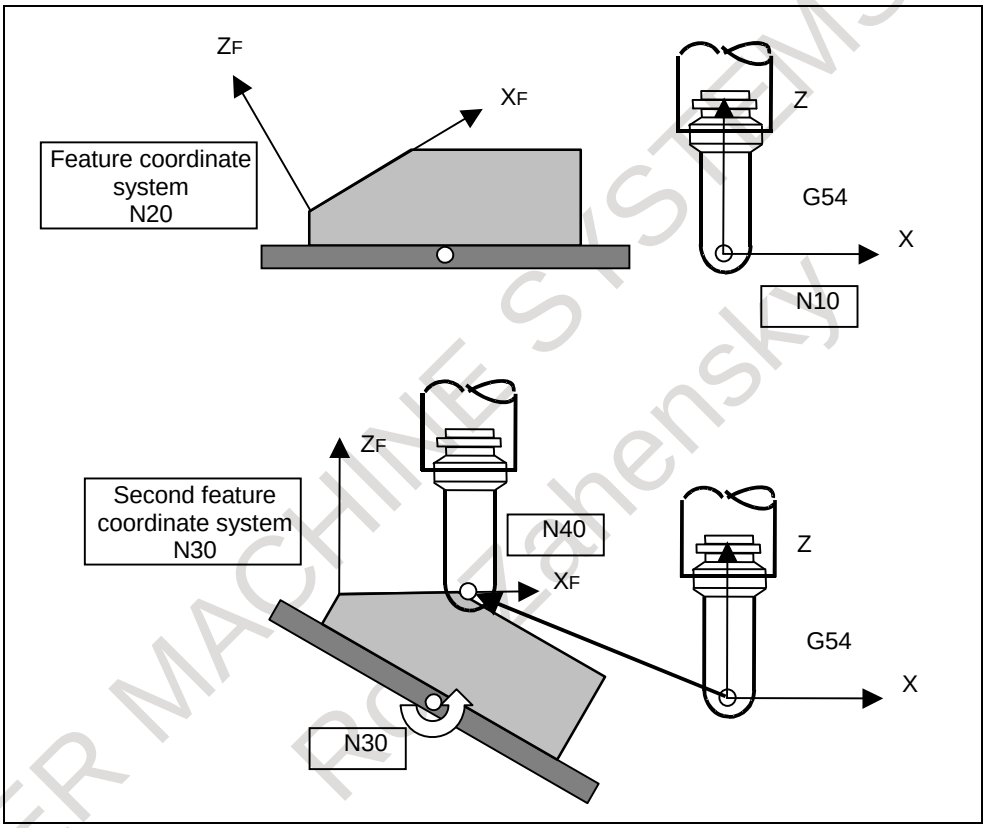


Fig. 14.3.4 (g) Example of operation 4 (table rotation type)

G53.6 command in tool length compensation

The G53.6 command can be executed in tool length compensation.

G53.6 operation in tool length compensation is performed in the same way as in the tool length compensation cancel mode.

G69 command in tool length compensation

The G69 command can be executed in tool length compensation.

After G69 operation in tool length compensation, the tool length compensation vector represents the Z direction of the workpiece coordinate system regardless of the position of the rotation axis.

FAYER MACHINE SYSTEMS INC
Roman Lahensky

14.3.5 Restrictions of Tilted Working Plane Indexing

- Basic restrictions

The restrictions imposed on 3-dimensional coordinate conversion also apply to the tilted working plane indexing.

- Increment system

The same increment system must be used for the basic three axes used by this function.

- Rapid traverse command

Linear rapid traverse (bit 1 (LRP) of parameter No. 1401 = 1) must be set for the rapid traverse command.

- 3-dimensional coordinate system conversion

If an attempt is made to set a new coordinate system by performing 3-dimensional coordinate conversion in a feature coordinate system, an alarm is also raised.

- Positioning in the machine coordinate system

Positioning commands in the machine coordinate system such as G28, G30, and G53 operate in the machine coordinate system rather than the feature coordinate system.

- External mirror image

If an attempt is made to use this function and an external mirror image function simultaneously, this function takes effect before the external mirror image function.

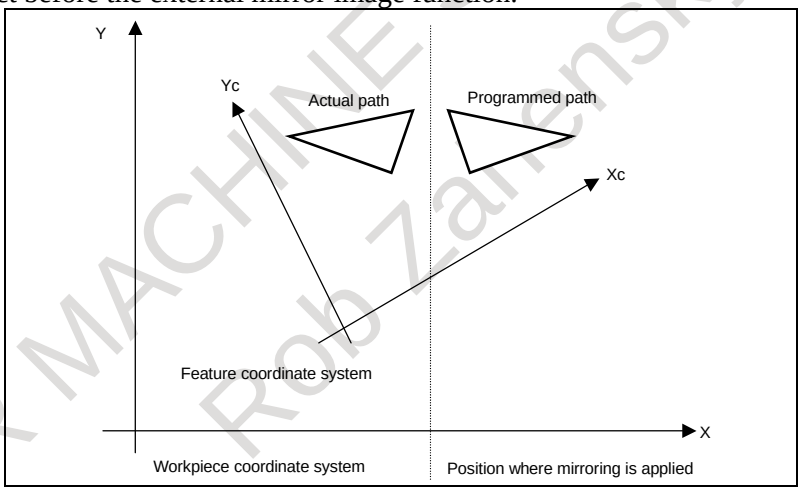


Fig. 14.3.5

- Tool center point retention type tool axis direction control

Performing a reset during tool center point retention type tool axis direction control results in the operation to be performed when the following parameters are set regardless of whether the reset is performed in the foreground or background. Thus, do not use the background during tool center point retention type tool axis direction control.

(1) Bit 2 (D3R) of parameter No. 5400 = 0: Clears the tilted working plane indexing.
(2) Bit 6 (CLR) of parameter No. 3402 = 1: Clears with a reset.
(3) Bit 0 (C08) of parameter No. 3407 = 0: Clears the G code in group 08.
(4) Bit 7 (C23) of parameter No. 3408 = 0: Clears the G code in group 23.
(5) Bit 7 (CFH) of parameter No. 3409 = 0: Clears the F, H, D, and T codes.
(6) Bit 6 (LVK) of parameter No. 5003 = 0: Clears the tool length compensation vector.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Relationships with other modal commands

G41, G42, and G40 (cutter compensation), G43, G49 (tool length compensation), G51.1 and G50.1 (programmable mirror image), and canned cycle commands must have nesting relationships with G68.2.
In other words, first issue G68.2 when the modes mentioned above are off, turn the modes on and off, then issue G69.

- Manual reference position return

Manual reference position return in the tilted working plane indexing mode results in alarm PS5324, “REFERENCE RETURN INCOMPLETE”. If you want to perform manual reference position return, cancel the tilted working plane indexing mode.

- Hypothetical axis of a table rotation axis

When a table rotation axis is set as a hypothetical axis, tilted working plane indexing is performed on the assumption that the table rotation axis is at 0 degrees.

- Specifiable G codes

The G codes that can be specified in the tilted working plane indexing mode are listed below.
Specifying a G code other than these codes results in alarm PS5462, “ILLEGAL COMMAND (G68.2/G69)”.

- Positioning (G00)
- Linear interpolation (G01)
- Circular interpolation / helical interpolation (G02/G03)
- Dwell (G04)
- AI contour control OFF/ON (G05.1 Q0/Q1)
- Exact stop(G09)
- Programmable data input (G10)
- Tool retract and recover (G10.6)
- Programmable data input mode cancel (G11)
- Plane selection (G17/G18/G19)
- Automatic return to reference position (G28)
- Movement from reference position (G29)
- 2nd, 3rd and 4th reference position return (G30)
- Skip function (G31)
- Cutter compensation : cancel (G40)
- Tool radius or tool nose radius compensation(G41/G42)
- Tool length compensation + (G43)
- Tool length compensation - (G44)
- Tool length compensation cancel (G49,G49.1)
- Programmable mirror image cancel (G50.1)
- Local coordinate system (G52)
- Machine coordinate system setting (G53)
- Tool axis direction control (G53.1)
- Workpiece coordinate system selection (G54 to G59, G54.1)
- Exact stop (G61)
- Tapping mode (G63)
- Cutting mode (G64)
- Macro call (G65)
- Macro modal call A (G66)
- Macro modal call B (G66.1)
- Macro modal call A/B cancel (G67)
- Canned cycle for drilling (G73, G74, G76, G80 to G89)
- Absolute programming (G90)
- Incremental programming (G91)
- Constant surface speed control (G96)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Constant surface speed control cancel (G97)
- Canned cycle : return to initial level (G98)
- Canned cycle : return to R point level (G99)

M

- Scaling cancel (G50)
- Scaling(G51)
- Programmable mirror image (G51.1)
- Automatic corner override (G62)
- Coordinate system rotation cancel or 3-dimensional coordinate system conversion mode off (G69)
- Inverse time feed (G93)
- Feed per minute (G94)
- Feed per revolution (G95)

T

- Coordinate system rotation cancel or 3-dimensional coordinate system conversion mode off (G69.1)
- Feed per minute (G98 (G94))
- Feed per revolution (G99 (G95))

- Modal G codes that allow specification of a tilted working plane indexing

A tilted working plane indexing can be specified in the modal G code states listed below.
In a modal state other than the following modal G codes, specifying the tilted working plane indexing results in alarm PS5462:

- Positioning (G00)
- Linear interpolation (G01)
- Programmable data input mode cancel (G11)
- Plane selection (G17/G18/G19)
- Polar coordinates command cancel (G15)
- Input in inch (G20 (G70))
- Input in mm (G21 (G71))
- Stored stroke check function (G22/G23)
- Cutter compensation : cancel (G40)
- Tool length compensation cancel (G49,G49.1)
- Programmable mirror image cancel (G50.1)
- Workpiece coordinate system selection (G54 to G59, G54.1)
- Exact stop mode (G61)
- Tapping mode (G63)
- Cutting mode (G64)
- Macro modal call A/B cancel (G67)
- Canned cycle cancel (G80)
- Absolute programming (G90)
- Incremental programming (G91)
- Constant surface speed control cancel (G97)
- Canned cycle : return to initial level (G98)
- Canned cycle : return to R point level (G99)

M

- Scaling cancel (G50)
- Automatic corner override (G62)
- Coordinate system rotation cancel or 3-dimensional coordinate system conversion mode off (G69)
- Inverse time feed (G93)
- Feed per minute (G94)
- Feed per revolution (G95)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

T

- Polar coordinate interpolation mode cancel (G13.1)
- Polygon turning cancel (G50.2)
- Mirror image for double turret off/balanced cutting mode cancel (G69)
- Coordinate system rotation cancel or 3-dimensional coordinate system conversion mode off (G69.1)
- Feed per minute (G98 (G94))
- Feed per revolution (G99 (G95))
- Polar coordinate interpolation mode cancel (G113)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

15 COMPENSATION FUNCTION

15.1 TOOL LENGTH COMPENSATION (G43, G44, G49)

This function can be used by setting the difference between the tool length assumed during programming and the actual tool length of the tool used into the offset memory. It is possible to compensate the difference without changing the program.
Specify the direction of offset with G43 or G44. Select a tool length compensation value from the offset memory by entering the corresponding address and number (H code).

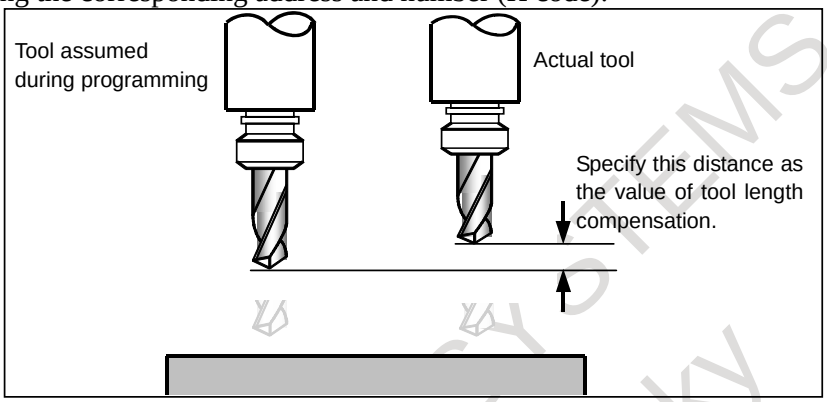


Fig. 15.1 Tool length compensation

15.1.1 Overview

The following three methods of tool length compensation can be used, depending on the axis along which tool length compensation can be made.

- Tool length compensation A
  Compensates for the difference in tool length along the basic Z-axis.
- Tool length compensation B
  Compensates for the difference in tool length in the direction normal to a selected plane.
- Tool length compensation C
  Compensates for the difference in tool length along a specified axis.

Format

Type	Format	Description
Tool length compensation A	**G43 Z_ H_ ;** **G44 Z_ H_ ;**	G43 : Positive offset G44 : Negative offset G17 : XY plane selection G18 : ZX plane selection G19 : YZ plane selection α : Address of a specified axis H : Address for specifying the tool length compensation value X, Y, Z : Offset move command
Tool length compensation B	**G17 G43 Z_ H_ ;** **G17 G44 Z_ H_ ;** **G18 G43 Y_ H_ ;** **G18 G44 Y_ H_ ;** **G19 G43 X_ H_ ;** **G19 G44 X_ H_ ;**	
Tool length compensation C	**G43 α_H_ ;** **G44 α_H_ ;**	
Tool length compensation cancel	**G49 ; or H0 ;**	

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

- Selection of tool length compensation

Select tool length compensation A, B, or C, by setting bits 0 (TLC) and 1 (TLB) of parameter No. 5001 .

Parameter No.5001		Type
Bit 1 (TLB)	Bit 0 (TLC)	
0	0	Tool length compensation A
1	0	Tool length compensation B
0/1	1	Tool length compensation C

- Direction of the offset

When G43 is specified, the tool length compensation value (stored in offset memory) specified with the H code is added to the coordinates of the end position specified by a command in the program. When G44 is specified, the same value is subtracted from the coordinates of the end position. The resulting coordinates indicate the end position after compensation, regardless of whether the absolute or incremental mode is selected.

When the specification of an axis is omitted, a movement is made by the tool length compensation value.

G43 and G44 are modal G codes. They are valid until another G code belonging to the same group is used.

- Specification of the tool length compensation value

The tool length compensation value assigned to the number (offset number) specified in the H code is selected from offset memory and added to or subtracted from the moving command in the program.

Example

```
  :
H1 ;        The offset value of offset number 1 is selected.
  :
G43 Z_ ;    Offset is applied according to the offset value of offset number 1.
  :
H2 ;        Offset is applied according to the offset value of offset number 2.
  :
H0 ;        Offset is applied according to the offset value 0.
  :
H3 ;        Offset is applied according to the offset value of offset number 3.
  :
G49 ;       Offset is canceled.
  :
H4 ;        The offset value of offset number 4 is selected.
  :
```

A tool length compensation value is to be set in the offset memory corresponding to an offset number.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

⚠ CAUTION

When another offset number is specified, the tool length compensation value just changes to a new value. The new tool length compensation value is not added to the old tool length compensation value.

H1 : Tool length compensation value 20.0
H2 : Tool length compensation value 30.0
G90 G43 Z100.0 H1 ; Z will move to 120.0
G90 G43 Z100.0 H2 ; Z will move to 130.0

NOTE

The tool length compensation value corresponding to offset No. 0, that is, H0 always means 0. It is impossible to set any other tool length compensation value to H0.

- Performing tool length compensation along two or more axes

Tool length compensation B can be executed along two or more axes when the axes are specified in two or more blocks.

By setting bit 3 (TAL) of parameter No. 5001 to 1, tool length compensation C can also can be executed along two or more axes when the axes are specified in two or more blocks. If no axis is specified in the same block, the alarm PS0027, “NO AXES COMMANDED IN G43/G44” is issued. If two or more axes are specified in the same block, the alarm PS0336, “TOOL COMPENSATION COMMANDED MORE TWO AXES” is issued.

Example 1

When tool length compensation B is executed along the X-axis and Y-axis

```
G19 G43 H_ ;      Offset in X axis
G18 G43 H_ ;      Offset in Y axis
```

Example 2

When tool length compensation C is executed along the X-axis and Y-axis

```
G43 X_ H_ ;       Offset in X axis
G43 Y_ H_ ;       Offset in Y axis
```

Example 3

When an alarm is issued with tool length compensation C

```
G43 X_ Y_ H_ ;    An alarm PS0336 occurs
```

- Tool length compensation cancel

To cancel tool length compensation, specify G49 or H0. After G49 or H0 is specified, the system immediately cancels the offset mode.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

1 If offset is executed along two or more axes, offset along all axes is canceled by specifying G49. If H0 is used to specify cancellation, offset along only the axis normal to a selected plane is canceled in the case of tool length compensation B, or offset along only the last axis specified by G43 or G44 is canceled in the case of tool length compensation C.

2 If offset is executed along three or more axes, and offset along all axes is canceled using G49, the alarm PS0015, “TOO MANY SIMULTANEOUS AXES” may be issued. By using H0 together, for example, cancel offset so that the number of simultaneously controlled axes (the number of axes along which movements are made simultaneously) does not exceed the allowable range of the system.

3 If tool length compensation and 3-dimensional coordinate system conversion are canceled by a reset with 3-dimensional coordinate system conversion performed during tool length compensation, the direction of tool length compensation vector cancellation becomes incorrect. By setting bit 6 (LVK) of parameter No. 5003 to 1 and setting bit 2 (D3R) of parameter No. 5400 to 1, ensure that the tool length compensation vector and 3-dimensional coordinate system conversion are not canceled by a reset.

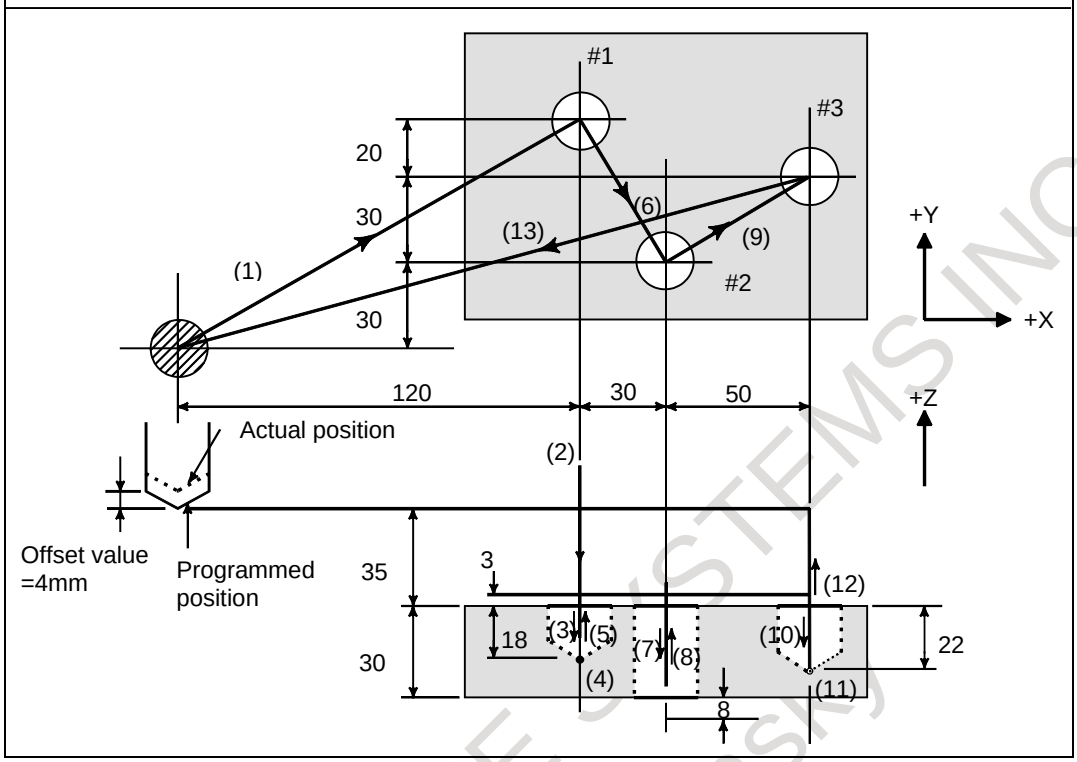
Example)

```
G43 H1 ;
G68 X_ Y_ Z_ I_ J_ K_ R_ ;
  :      } Ensure that tool length compensation and three-dimensional
  :        coordinate conversion are not canceled by a reset in this range.
  :
G69 ;
G49 ;
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Example

Tool length compensation (in boring holes #1, #2, and #3)



Program
H1=-4.0 (Tool length compensation value)
N1 G91 G00 X120.0 Y80.0 ; .....(1)
N2 G43 Z-32.0 H1 ; .....(2)
N3 G01 Z-21.0 F1000 ; .....(3)
N4 G04 P2000 ; .....(4)
N5 G00 Z21.0 ; .....(5)
N6 X30.0 Y-50.0 ; .....(6)
N7 G01 Z-41.0 ; .....(7)
N8 G00 Z41.0 ; .....(8)
N9 X50.0 Y30.0 ; .....(9)
N10 G01 Z-25.0 ; .....(10)
N11 G04 P2000 ; .....(11)
N12 G00 Z57.0 H0 ; .....(12)
N13 X-200.0 Y-60.0 ; .....(13)
N14 M2 ;

T

When this function is used in the lathe system, "EXTENDED TOOL SELECTION" function is needed. Moreover, it should be ATC type (parameter TCT (No.5040#3)=1) as for the tool exchange method.

Notes

- Command for setting a workpiece coordinate system in the tool length compensation mode

Executing a workpiece coordinate system setting G code command (G92 or, for G code system A in the lathe system, G50) presets a coordinate system in such a way that the specified position will be a pre-compensation position.
However, this G code cannot be used together with a block where tool length compensation vectors vary. For details, see “Notes” in Subsection 7.2.1, “Setting a Workpiece Coordinate System.”

FRITZ MACHINE SYSTEMS INC
Rob Zahensky

15.1.2 G53, G28, and G30 Commands in Tool Length Compensation Mode

This section describes the tool length compensation cancellation and restoration performed when G53, G28, or G30 is specified in tool length compensation mode. Also described is the timing of tool length compensation.
As mentioned in "SAFETY PRECAUTIONS WARNINGS AND CAUTIONS RELATED TO PROGRAMMING" on this manual, it is recommended to cancel tool length compensation mode before executing the G53, G28, and G30 commands.

Explanation

- Tool length compensation vector cancellation

When G53, G28, or G30 is specified in tool length compensation mode, tool length compensation vectors are canceled as described below. However, the previously specified modal G code remains displayed; modal code display is not switched to G49.

(1) When G53 is specified

Command	Specified axis	Operation
G53 IP_	Tool length compensation axis	Canceled upon movement being performed
	Other than tool length compensation axis	Not canceled
G49 G53 IP_	Tool length compensation axis	Canceled upon movement being performed
	Other than tool length compensation axis	Canceled

(IP_ : Dimension word)

> ⚠ **CAUTION**
> If tool length compensation is applied along multiple axes, the offset vector along the axis specified by G53 is canceled.

(2) When G28,or G30 is specified

Command	Specified axis	Operation
G28 IP_	Tool length compensation axis	Not canceled at an intermediate point. Canceled at the reference position.
	Other than tool length compensation axis	Not canceled at an intermediate point. Canceled at the reference position.
G49 G28 IP_	Tool length compensation axis	Canceled when a movement is made to an intermediate point.
	Other than tool length compensation axis	Canceled when a movement is made to an intermediate point.

(IP_ : Dimension word)

> ⚠ **CAUTION**
> If tool length compensation is applied along multiple axes, the offset vector along the axis on which a reference position return operation has been performed is canceled.

FRYER MACHINE SYSTEMS INC
Rob Zahonsky

- Tool length compensation vector restoration

Tool length compensation vectors, canceled by specifying G53, G28, or G30 in tool length compensation mode, are restored as described below.

Type	Bit 6 (EVO) of parameter No. 5001	Restoration condition
A/B	0	The H command or G43 (G44) is specified.
	1	Restored by the next buffered block.
C		The H command or G43 (G44)IP_ is specified.

(IP_ : Dimension word)

⚠ CAUTION

1 If a tool length compensation vector is restored only with H_, G43, or G44 when tool length compensation is applied along multiple axes, the tool length compensation vector along only the axis normal to a selected plane is restored in the case of tool length compensation B, or the tool length compensation vector along only the last axis for which tool length compensation is specified is restored in the case of tool length compensation C. The tool length compensation vector along any other axes is not restored.

2 In the block in which the tool length compensation vector is restored, do not execute commands other than positioning with G00 or G01, G04, and a single block with EOB.

15.2 SCALING (G50, G51)

M

Overview

A programmed figure can be magnified or reduced (scaling).

Two types of scaling are available, one in which the same magnification rate is applied to each axis and the other in which different magnification rates are applied to different axes.

The magnification rate can be specified in the program.

Unless specified in the program, the magnification rate specified in the parameter is applied.

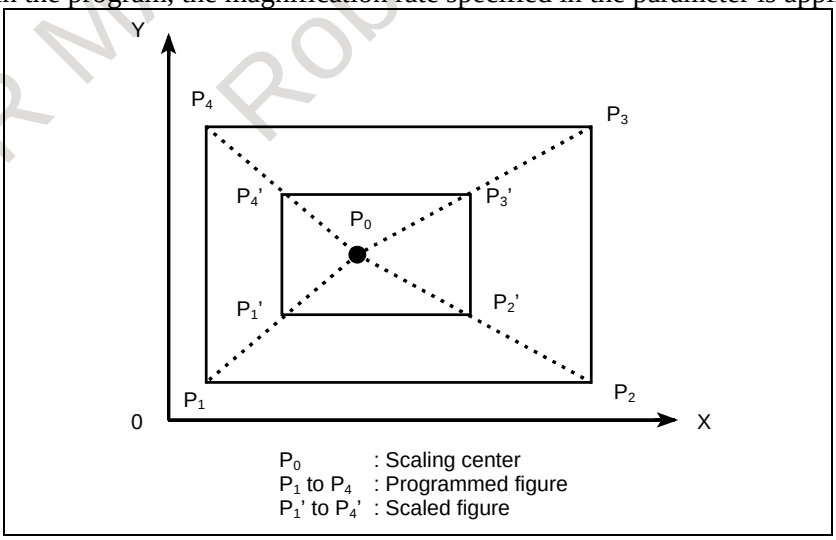


Fig. 15.2 (a) Scaling

FRYER MACHINE SYSTEMS INC
Rob Zehensky

Format

Scaling up or down along all axes at the same rate of magnification
(When bit 6 (XSC) of parameter No. 5400 = 0)

Format	Meaning of command
G51 IP_P_ ; Scaling start ¦ } Scaling is effective. (Scaling mode) ¦ **G50 ;** Scaling cancel	**IP_** : Absolute command for center coordinate value of scaling **P_** : Scaling magnification

Scaling up or down along each axes at a different rate of magnification (mirror image)
(When bit 6 (XSC) of parameter No. 5400 = 1)

Format	Meaning of command
G51 IP_I_J_K_; Scaling start ¦ } Scaling is effective. (Scaling mode) ¦ **G50 ;** Scaling cancel	**IP_** : Absolute command for center coordinate value of scaling **I_J_K_** : Scaling magnification for basic 3 axes (X, Y, and Z axes) respectively

T

NOTE
This function is available when the G-code system B or C is set.

⚠ **CAUTION**
1 Specify G51 in a separate block.
2 After the figure is enlarged or reduced, specify G50 to cancel the scaling mode.

NOTE
1 Entering electronic calculator decimal point input mode (bit 0 (DPI) of parameter No. 3401 = 1) does not cause the units of the magnification rates P, I, J, and K to change.
2 Setting the least input increment equal to 10 times the least command increment (bit 7 (IPR) of parameter No. 1004 = 1) does not cause the units of the magnification rates P, I, J, and K to change.
3 An attempt to specify 0 as a magnification rate causes alarm PS0142, “ILLEGAL SCALE RATE” to occur in a G51 block.

Explanation

- Axis for which scaling is to be enabled

For the axis for which scaling is to be enabled, set bit 0 (SCL) of parameter No. 5401 to 1.

- Minimum unit of scaling magnification

Least input increment of scaling magnification is: 0.001 or 0.00001.
It is 0.00001 (one hundred thousandth) if bit 7 (SCR) of parameter No. 5400 is 0 and 0.001 if it is 1.

- Scaling center

Even in incremental command (G91) mode, the scaling center coordinates IP_ specified in the G51 block are assumed those of an absolute position.
If the scaling center coordinates are omitted, the position assumed when G51 is specified is assumed the scaling center.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

> ⚠ **CAUTION**
> With the move command subsequent to the G51 block, execute an absolute (G90 mode) position command.
> If no absolute position command is executed after the G51 block, the position assumed when G51 is specified is assumed the scaling center; once an absolute position command is executed, the scaling center assumes the coordinates specified in the G51 block, after that block.

- Scaling along each axis at the same rate of magnification

Set bit 6 (XSC) of parameter No. 5400 to 0.
If the scaling magnification P is not specified, the magnification set in parameter No. 5411 is used.
Decimal point input is not accepted as the magnification P. If decimal point input is made, alarm PS0007, “ILLEGAL USE OF DECIMAL POINT” will occur.
A negative value cannot be specified as the magnification P. If a negative value is specified, alarm PS0006, “ILLEGAL USE OF MINUS SIGN” will occur.
The allowable magnification range is from 0.00001 to 9999.99999.

- Scaling of each axis, programmable mirror image (negative magnification)

Each axis can be scaled by different magnifications. Also when a negative magnification is specified, a mirror image is applied. The axis subject to the mirror image is the one that contains the scaling center.
Set bit 6 (XSC) of parameter No. 5400 to 1 to validate each axis scaling (mirror image).
Using I, J, and K, specify the scaling magnifications for the basic 3 axes (X to Z axes). Use parameter No. 1022 to specify which axes to use as the basic 3 axes. For those of the X to Z axes for which I, J, and K are not specified and for axes other than the basic 3 axes, the magnification set with parameter No. 5421 is used.
A value other than 0 must be set to parameter No. 5421.
Decimal point programming can not be used to specify the rate of magnification (I, J, K).
Magnification can be set within the range of ±0.00001 to ±9999.99999.

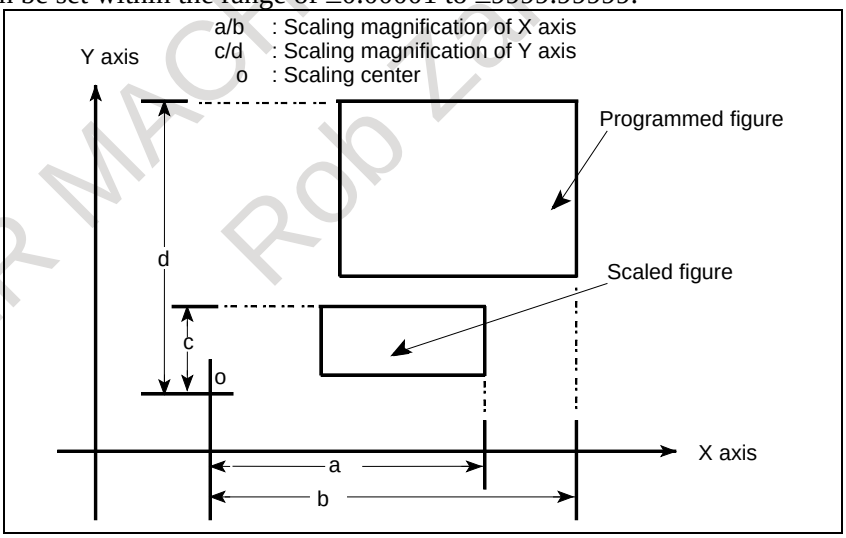


Fig. 15.2 (b) Scaling of each axis

FRYER MACHINE SYSTEMS INC
Rob Zahensky

⚠ CAUTION

Specifying the following commands at the same time causes them to be executed in the order indicated below:

<1> Programmable mirror image (G51.1)
<2> Scaling (G51) (including a mirror image with a negative magnification)
<3> Mirror image due to the external switch of the CNC or the settings of the CNC

In this case, the programmable mirror image is effective to the scaling center and magnification as well.
To specify G51.1 and G51 at the same time, specify them in this order; to cancel them, specify them in the reverse order.

- Scaling of circular interpolation

Even if different magnifications are applied to each axis in circular interpolation, the tool will not trace an ellipse.

```
G90 G00 X0.0 Y100.0 Z0.0;
G51 X0.0 Y0.0 Z0.0 I2000 J1000;
```

(A magnification of 2 is applied to the X-component and a magnification of 1 is applied to the Y-component.)

```
G02 X100.0 Y0.0 I0 J–100.0 F500;
```

Above commands are equivalent to the following command:

```
G90 G00 X0.0 Y100.0 Z0.0;
G02 X200.0 Y0.0 I0 J–100.0 F500;
```

(Because the end point is not on an arc, spiral interpolation is assumed.)

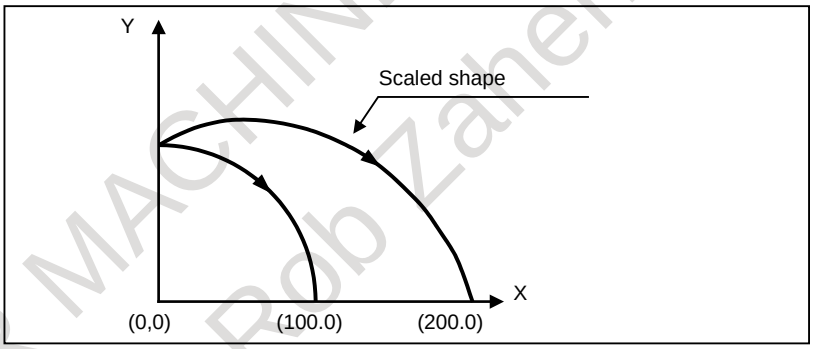


Fig. 15.2 (c) Scaling for circular interpolation

Even for an R-specified arc, scaling is applied to each of I, J, and K after the radius value (R) is converted into a vector in the center direction of each axis.
If, therefore, the above G02 block contains the following R-specified arc, the operation will be same as that in which I and J are specified.

```
G02 X100.0 Y0.0 R100.0 F500 ;
```

- Scaling and coordinate system rotation

If both scaling and coordinate system rotation are specified at the same time, scaling is performed first, followed by coordinate system rotation. In this case, scaling is effective to the rotation center as well.
To specify both of them, specify scaling first and then coordinate system rotation. To cancel them, specify them in the reverse order.

Example
Main program

```
O1
G90 G00 X20.0 Y10.0 ;
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

```
    M98 P1000 ;
    G51 X20.0 Y10.0 I3000 J2000 ;  (×3 in the X direction and ×2 in the Y direction)
    M98 P1000 ;
    G17 G68 X35.0 Y20.0 R30. ;
    M98 P1000 ;
    G69 ;
    G50 ;
    M30 ;
Subprogram
    O1000 ;
    G01 X20.0 Y10.0 F500 ;
    G01 X50.0 ;
    G01 Y30.0 ;
    G01 X20.0 ;
    G01 Y10.0 ;
    M99 ;
```

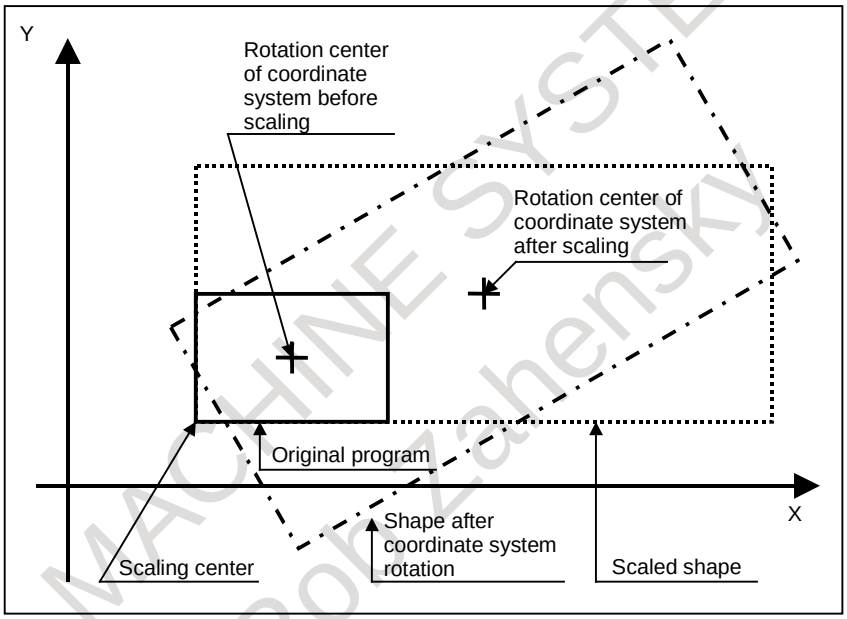


Fig. 15.2 (d) Scaling and coordinate system rotation

- Scaling and optional chamfering/corner R

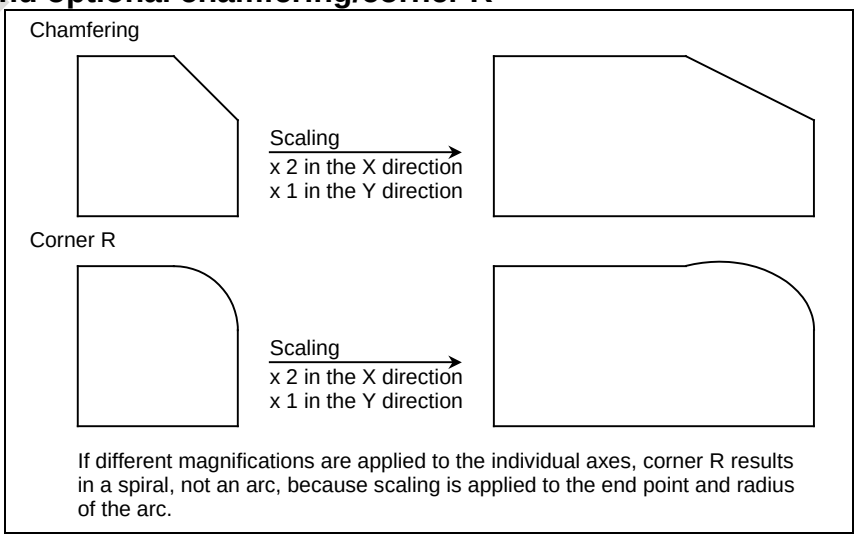


Fig. 15.2 (e) Scaling and optional chamfering/corner R

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Limitation

- Tool compensation

This scaling is not applicable to tool radius tool nose radius compensation values, tool length compensation values, and tool offset values (Fig. 15.2 (f)).

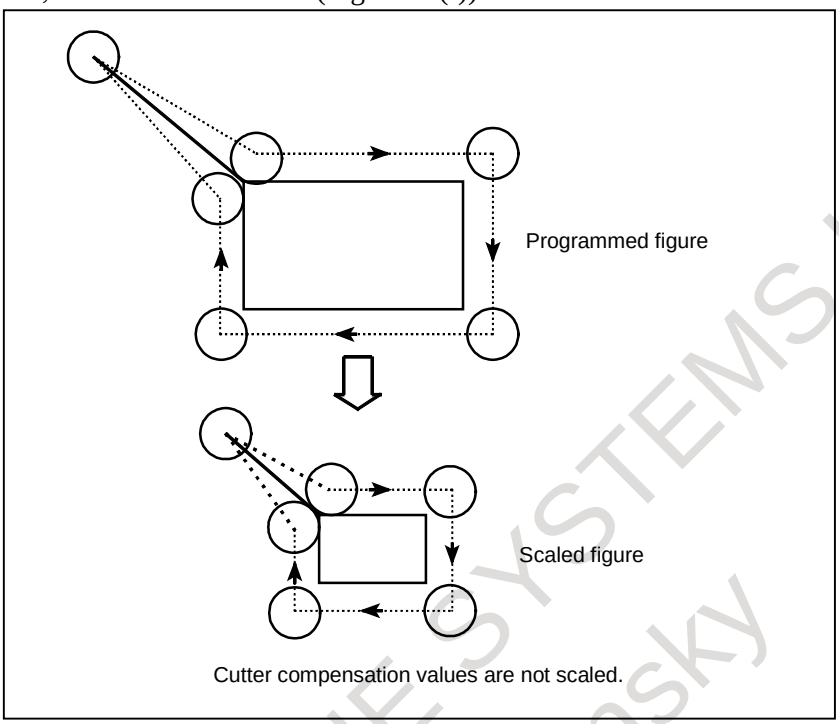


Fig. 15.2 (f) Scaling during cutter compensation

- Invalid scaling

M

Scaling is not applied to the travel distance during canned cycle shown below.

- Cut-in value Q and retraction value d of peck drilling cycle (G83, G73).
- Fine boring cycle (G76)
- Shift value Q of X and Y axes in back boring cycle (G87).

In manual operation, the travel distance cannot be increased or decreased using the scaling.

T

This function is available to G code systems B and C only; it is not available to G code system A.

During scaling, the following functions cannot be used. If any of them is specified, alarm PS0300, “ILLEGAL COMMAND IN SCALING” will occur.

- Finishing cycle (G70, G72)
- Outer surface rough cutting cycle (G71, G73)
- End rough cutting cycle (G72, G74)
- Closed loop cutting cycle (G73, G75)
- End cutting off cycle (G74, G76)
- Outer/inner surface cutting off cycle (G75, G77)
- Multiple threading cycle (G76, G78)
- Traverse grinding cycle (for grinding machines) (G71, G72)
- Traverse direct constant-size grinding cycle (for grinding machines) (G72, G73)
- Oscillation grinding cycle (for grinding machines) (G73, G74)
- Oscillation direct constant-size grinding cycle (for grinding machines) (G74, G75)
- Face drilling cycle (G83, G83)
- Face tapping cycle (G84, G84)

FIXER MACHINE SYSTEMS INC
Rob Zahensky

- Face boring cycle (G85, G85)
- Side drilling cycle (G87, G87)
- Side tapping cycle (G88, G88)
- Side boring cycle (G89, G89)
- Outer/inner surface turning cycle (G77, G20)
- Threading cycle (G78, G21)
- End turning cycle (G79, G24) (The G code in system B is given first, followed by that of system C.)

In manual operation, the travel distance cannot be increased or decreased using the scaling.
If, during scaling, tool position compensation with a coordinate system shift (bit 2 (LWT) of parameter No. 5002 = 1 or bit 4 (LGT) of parameter No. 5002 = 0) is specified, alarm PS0509, "TOOL OFFSET COMMAND IS NOT AVAILABLE", is issued. This is also true if bit 6 (EVO) of parameter No. 5001 is set to 1 and the tool position compensation value is changed.

⚠ CAUTION

1 If a parameter setting value is employed as a scaling magnification without specifying P, the setting value at G51 command time is employed as the scaling magnification, and a change of this value, if any, is not effective.
2 Before specifying the G code for reference position return (G27, G28, G29, G30, etc.) or coordinate system setting (G52 to G59, G92, etc.), cancel the scaling mode. If any of them is specified during scaling, alarm PS0412, “AN UNUSABLE G CODE WAS USED”, will occur.
3 If scaling results are rounded by counting fractions of 5 and over as a unit and disregarding the rest, the move amount may become zero. In this case, the block is regarded as a no movement block, and therefore, it may affect the tool movement by cutter compensation. See the description of cutter compensation.
4 Refrain from scaling on a rotation axis for which the rollover function is enabled. Otherwise, the tool may rotate in a short-cut manner, possibly resulting in unexpected movement.

NOTE

1 The position display represents the coordinate value after scaling.
2 When a mirror image was applied to one axis of the specified plane, the following results:
(1) Circular command ..... Direction of rotation is reversed.
(2) Tool radius · tool nose radius compensation . Offset direction is reversed.
(3) Coordinate system rotation ..... Rotation angle is reversed.

Example

Sample program of a scaling in each axis

```
O1;
G51 X20.0 Y10.0 I750 J250; (× 0.75 in the X direction, × 0.25 in the Y direction)
G00 G90 X60.0 Y50.0;
G01 X120.0 F100;
G01 Y90;
G01 X60;
G01 Y50;
G50;
M30;
```

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

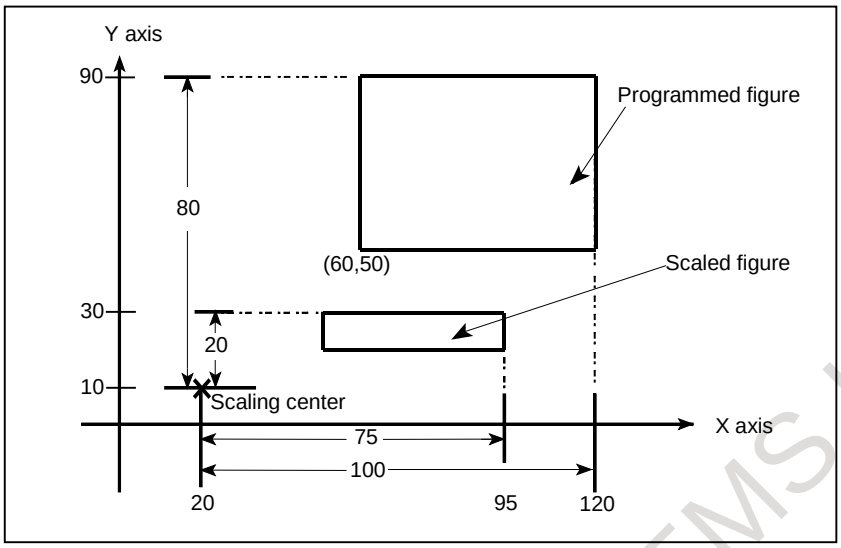
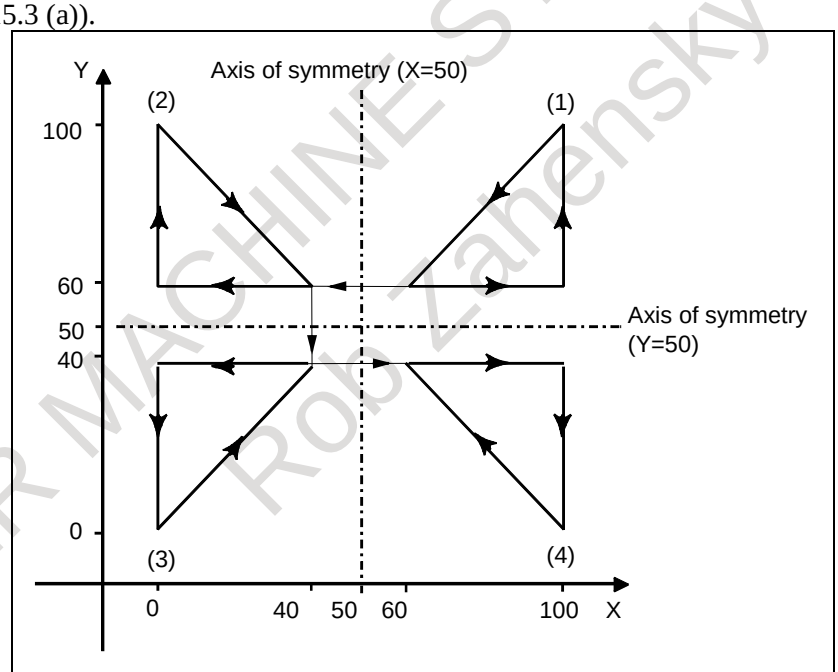


Fig. 15.2 (g) Program example of scaling in each axis

15.3 PROGRAMMABLE MIRROR IMAGE (G50.1, G51.1)

A mirror image of a programmed command can be produced with respect to a programmed axis of symmetry (Fig. 15.3 (a)).



(1) Original image of a programmed command
(2) Image symmetrical about a line parallel to the Y-axis and crossing the X-axis at 50
(3) Image symmetrical about point (50, 50)
(4) Image symmetrical about a line parallel to the X-axis and crossing the Y-axis at 50

Fig. 15.3 (a) Programmable mirror image

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Format

G51.1 IP_ ; Setting a programmable image

: } A mirror image of a command specified in these blocks is produced with respect to the axis of symmetry specified by G51.1 IP_;.

G50.1 IP_ ; Canceling a programmable mirror image

IP_ : Point (position) and axis of symmetry for producing a mirror image when specified with G51.1.
Axis of symmetry for producing a mirror image when specified with G50.1. Point of symmetry is not specified.

Explanation

- Mirror image by setting

If the programmable mirror image function is specified when the command for producing a mirror image is also selected by a CNC external switch or CNC setting (see III Section 4.8, “MIRROR IMAGE”), the programmable mirror image function is executed first.

- Mirror image on a single axis in a specified plane

Applying a mirror image to one of the axes on a specified plane changes the following commands as follows :

Command	Explanation
Circular command	G02 and G03 are interchanged.
Tool radius · tool nose radius compensation	G41 and G42 are interchanged.
Coordinate system rotation	CW and CCW (directions of rotation) are interchanged.

- Mirror image to rotary axis

The programmable mirror image function is executed to a rotary axis as well as a linear axis.

Limitation

- Scaling and coordinate system rotation

Processing proceeds from program mirror image to scaling and coordinate system rotation in the stated order. The commands should be specified in this order, and, for cancellation, in the reverse order. Do not specify G50.1 or G51.1 during scaling or coordinate system rotation mode.

- Commands related to reference position return and coordinate system

In programmable mirror image mode, G codes related to reference position return (G27, G28, G29, G30, etc.) and those for changing the coordinate system (G52 to G59, G92, etc.) must not be specified. If any of these G codes is necessary, specify it only after canceling the programmable mirror image mode.

T

If, during the programmable mirror image mode, tool position compensation with a coordinate system shift (bit 2 (LWT) of parameter No. 5002 = 1 or bit 4 (LGT) of parameter No. 5002 = 0) is specified, alarm PS0509, "TOOL OFFSET COMMAND IS NOT AVAILABLE", is issued.
This is also true if bit 6 (EVO) of parameter No. 5001 is set to 1 and the tool position compensation value is changed.

FRYER MACHINE SYSTEMS INC
Rob Zaharchuk

15.4 NORMAL DIRECTION CONTROL (G40.1,G41.1,G42.1)

Overview

When a tool with a rotation axis (C-axis) is moved in the XY plane during cutting, the normal direction control function can control the tool so that the C-axis is always perpendicular to the tool path (Fig. 15.4 (a)).

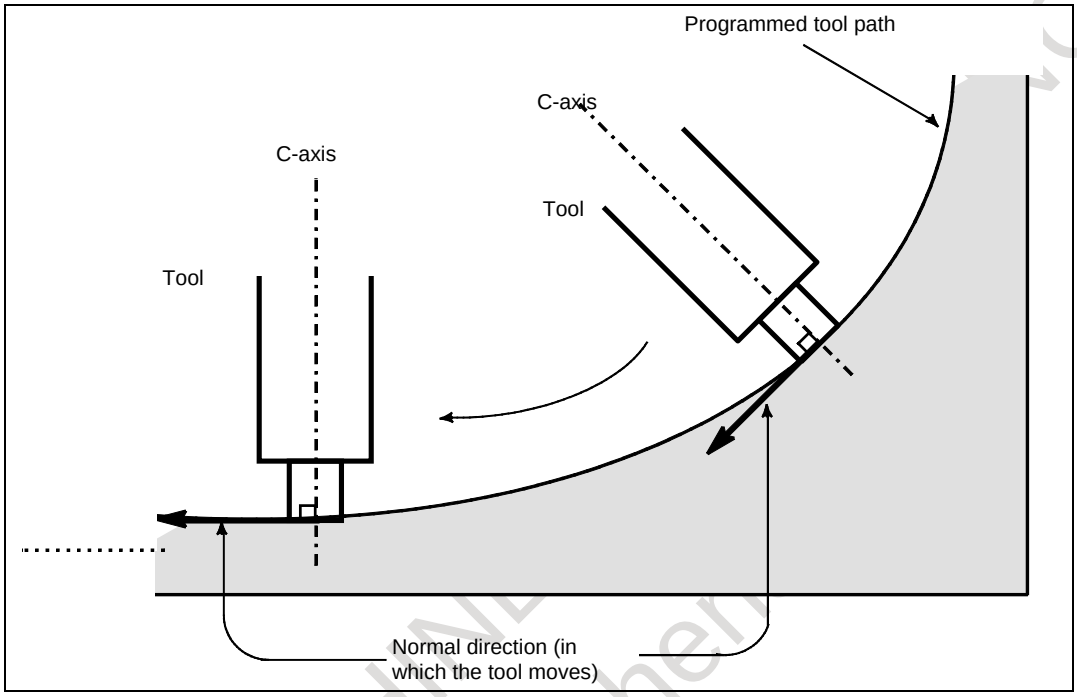


Fig. 15.4 (a) Sample Movement of the tool

Format

G41.1 ; Normal direction control, left

G42.1 ; Normal direction control, right

G40.1 ; Cancel normal direction control

The normal direction control, left (G41.1) command is used when the workpiece is on the right side of the tool as viewed while you are looking into the tool's way.
Once either G41.1 or G42.1 is issued, normal direction control is enabled (normal direction control mode).
Issuing G40.1 cancels the normal direction control mode.

FRYER MACHINE Rob Zahen

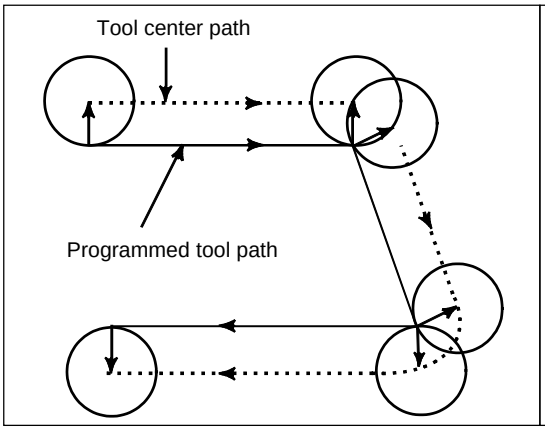


Fig. 15.4 (b) Normal direction control, left (G41.1)

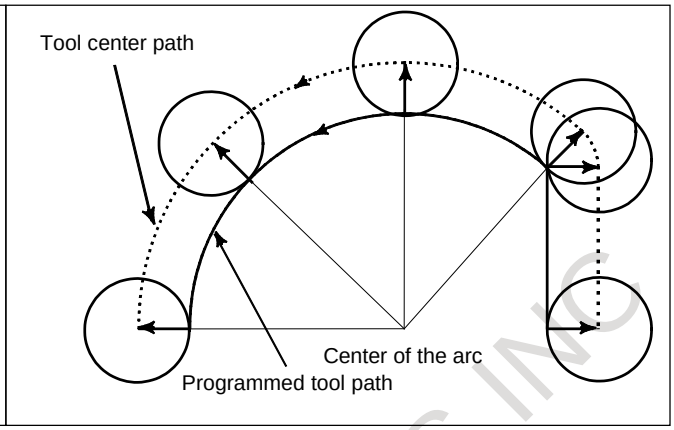


Fig. 15.4 (c) Normal direction control, right (G42.1)

Explanation

- Angle of the C axis

When viewed from the center of rotation around the C-axis, the angular displacement about the C-axis is determined as shown in Fig. 15.4 (d). The positive side of the X-axis is assumed to be 0, the positive side of the Y-axis is 90°, the negative side of the X-axis is 180°, and the negative side of the Y-axis is 270°.

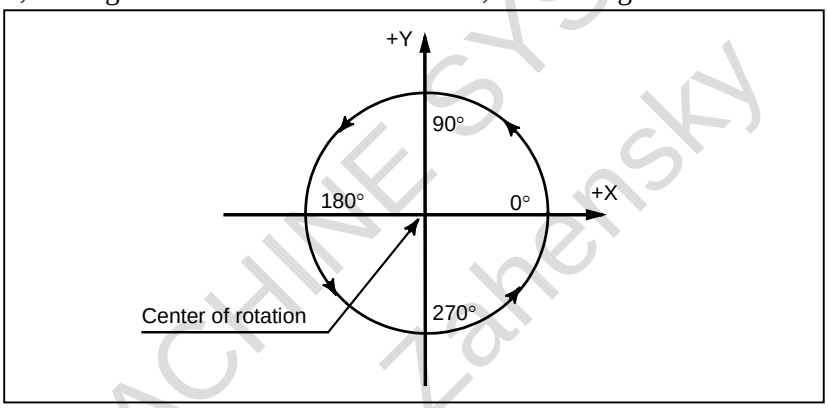


Fig. 15.4 (d) Angle of the C axis

- Normal direction control of the C axis

When the cancel mode is switched to the normal direction control mode, the C-axis becomes perpendicular to the tool path at the beginning of the block containing G41.1 or G42.1.

In the interface between blocks in the normal direction control mode, a command to move the tool is automatically inserted so that the C-axis becomes perpendicular to the tool path at the beginning of each block. The tool is first oriented so that the C-axis becomes perpendicular to the tool path specified by the move command, then it is moved along the X- and Y axes.

In the cutter compensation mode, the tool is oriented so that the C-axis becomes perpendicular to the tool path created after compensation.

In single-block operation, the tool is not stopped between a command for rotation of the tool and a command for movement along the X- and Y-axes. A single-block stop always occurs after the tool is moved along the X- and Y-axes.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

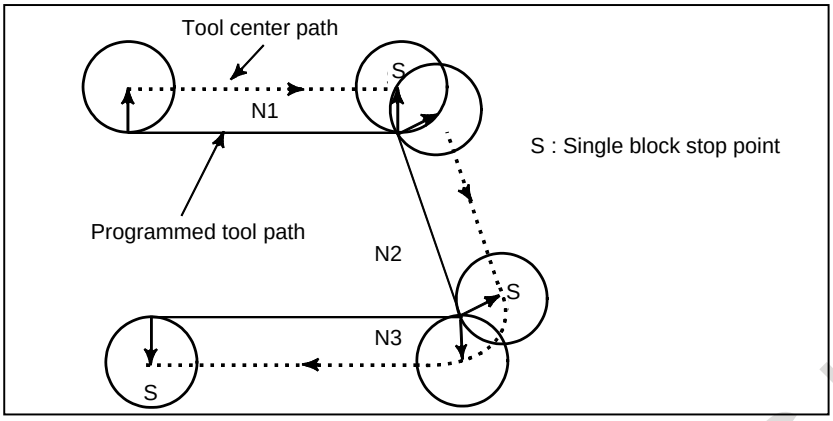


Fig. 15.4 (e) Point at which a single-block stop occurs in the normal direction control mode

Before circular interpolation is started, the C-axis is rotated so that the C-axis becomes normal to the arc at the start point. During circular interpolation, the tool is controlled so that the C-axis is always perpendicular to the tool path determined by circular interpolation.

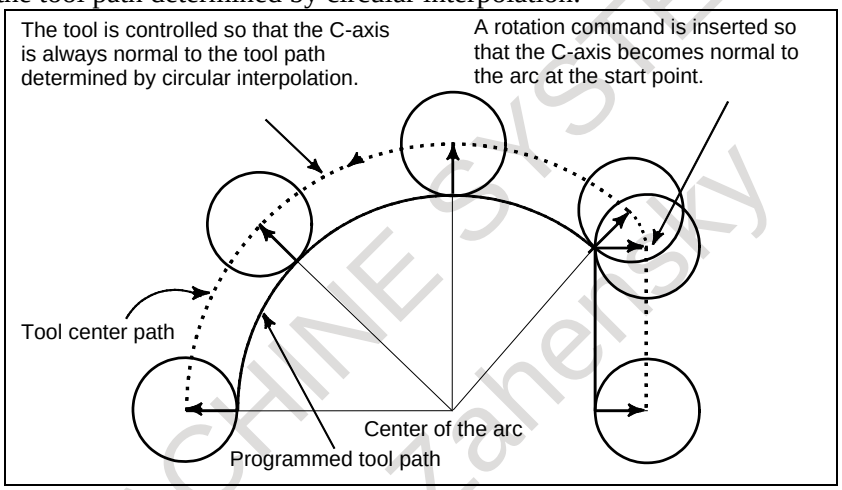


Fig. 15.4 (f) Normal direction control of the circular interpolation

NOTE

During normal direction control, the C axis always rotates through an angle less than 180 deg. I.e., it rotates in whichever direction provides the shorter route.

- C axis feedrate

Movement of the tool inserted at the beginning of each block is executed at the feedrate set in parameter 5481. If dry run mode is on at that time, the dry run feedrate is applied. If the tool is to be moved along the X-and Y-axes in rapid traverse (G00) mode, the rapid traverse feedrate is applied.
The feedrate of the C axis during circular interpolation is defined by the following formula.

$$F \times \frac{\text{Amount of movement of the C axis (deg)}}{\text{Length of arc (mm or inch)}} \quad \text{(deg/min)}$$

F : Feedrate (mm/min or inch/min) specified by the corresponding block of the arc
Amount of movement of the C axis : The difference in angles at the beginning and the end of the block.

NOTE

If the feedrate of the C axis exceeds the maximum cutting speed of the C axis specified to parameter No. 1430, the feedrate of each of the other axes is clamped to keep the feedrate of the C axis below the maximum cutting speed of the C axis.

FRYER MACHINE SYSTEMS INC
Rob Zaherlsky

- Normal direction control axis

A C-axis to which normal-direction control is applied can be assigned to any axis with parameter No. 5480.

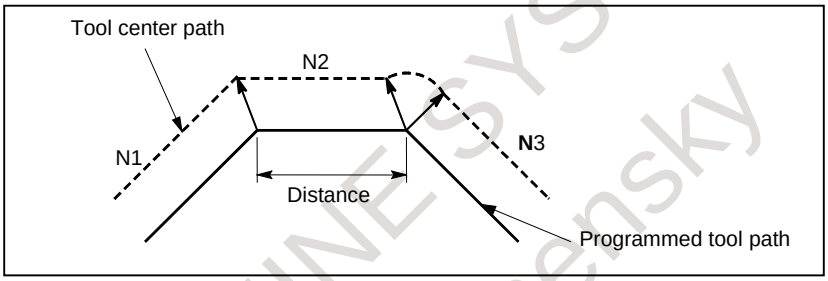
- Angle for which figure insertion is ignored

When the rotation angle to be inserted, calculated by normal-direction control, is smaller than the value set with parameter No. 5482, the corresponding rotation block is not inserted for the axis to which normal-direction control is applied. This ignored rotation angle is added to the next rotation angle to be inserted, the total angle being subject to the same check at the next block.
If an angle of 360 degrees or more is specified, the corresponding rotation block is not inserted.
If an angle of 180 degrees or more is specified in a block other than that for circular interpolation with a C-axis rotation angle of 180 degrees or more, the corresponding rotation block is not inserted.

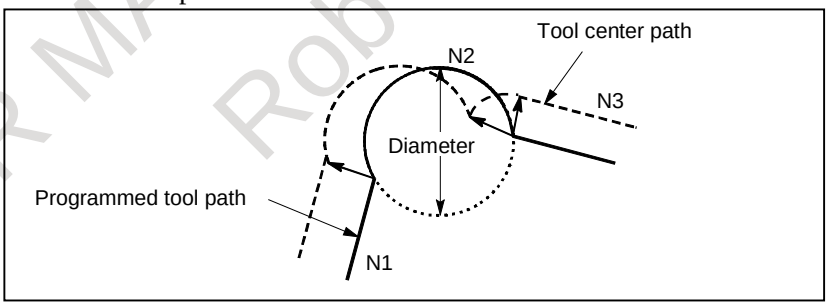
- Movement for which arc insertion is ignored

Specify the maximum distance for which machining is performed with the same normal direction as that of the preceding block.

- Linear movement
  When distance N2, shown below, is smaller than the set value, machining for block N2 is performed using the same direction as that for block N1.



- Circular movement
  When the diameter of block N2, shown below, is smaller than the set value, machining for block N2 is performed using the same normal direction as that for block N1. And control as compensation along the circular arc is not performed.



FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

1 Do not specify any command to the C axis during normal direction control. Any command specified at this time is ignored.
2 Before processing starts, it is necessary to correlate the workpiece coordinate of the C axis with the actual position of the C axis on the machine using the coordinate system setting (G92) or the like.
3 The helical cutting option is required to use this function. Helical cutting cannot be specified in the normal direction control mode.
4 Normal direction control cannot be performed by the G53 move command.
5 The C-axis must be a rotation axis.
6 The following functions must be commanded with normal direction control cancel mode (G40.1).
   - Plane selection command
   - AI contour control

T

NOTE

In the normal direction control mode, the following commands cannot be issued. An attempt to issue any of them results in alarm PS1471 being raised.
- Plane selection command (G17, G18, G19)
- Automatic reference position return(G28)
- 2nd/3rd/4th reference position return(G30)
- Move from the reference position (G29)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

15.5 TOOL OFFSET FOR MILLING AND TURNING FUNCTION (G43.7)

T

Overview

In the past, the tool offset screen for the machining center system was different from that for the lathe system. This function enables display and manipulation of offset data on a common tool offset screen.

Format

Conforms to the format for the lathe system. For details, refer to "EXTENDED TOOL SELECTION" in "OPERATOR'S MANUAL (For Lathe System) (B-64694EN-1)"

Explanation

- Tool length compensation on the Z-axis

Of the compensation amounts used with the functions below, only the value set for "Z/LENGTH" on the tool offset screen are used. Values set for "X", "Y", "NOSE R/RAD", etc. are not used.

G43 / G44: Tool length offset

- Programmable data input (G10)

The data format used to input/output offset data on the tool offset screen and to input offset data due to program operation is the G10 format, below.

- G10 data format

G10 Lbaaa ＋ Various tool offset formats

- L format: Lbaaa

```
aaa  : 200-  Tool offset, tool-nose radius compensation (cutter compensation),
             virtual tool tip direction
             Tool wear compensation
     : 201-  Tool offset, tool-nose radius compensation (cutter compensation),
             virtual tool tip direction
             Tool geometry compensation
b    : 0-    Writes the specified data over the existing data.
     : 1-    Adds the specified data to the existing data.
```

- Various tool offset formats

```
aaa  : 200,201-  Tool offset, tool-nose radius compensation (cutter
                 compensation), virtual tool tip direction
                 P_  : Tool offset number
                 X_  : Tool offset amount on the X-axis
                 Z_  : Tool offset amount on the Z-axis (tool length compensation)
                 R_  : Tool-nose radius compensation amount (cutter
                       compensation amount)
                 Q_  : Virtual tool tip direction
                 Y_  : Tool offset amount on the Y-axis
```

FRYER MACHINE SYSTEMS INC
Rob Zanensky

- Example

```
O0010;
G10L200P1X_Z_R_Q_Y_;   Overwrite tool wear compensation, such as tool offset.
G10L1200P1X_Z_R_Q_Y_;  Add tool wear compensation, such as tool offset.
G10L201P1X_Z_R_Y_;     Overwrite tool geometry compensation, such as tool offset.
G10L1201P1X_Z_R_Y_;    Add tool geometry compensation, such as tool offset.
:      :
M02;
```

- Input in the G10 format

Offset data can be input in the G10 format of the lathe system.
G10P1X1.001Z2.001R3.001Q4Y5.001

Pn : Tool offset number (1 to number of tool compensation values)
  Pn ≤ 10000 : Tool wear compensation
  Pn ≥ 10001 : Tool geometry compensation
X_ : X axis tool offset
Z_ : Z axis tool offset
R_ : Tool nose radius compensation
Q_ : Virtual tool tip direction
Y_ : Y axis tool offset

- Custom macro system variables

In this function, there is no change in the referred custom macro system variables.

Limitation

(1) Functions that cannot be used together with this function
  - MANUAL GUIDE *i*

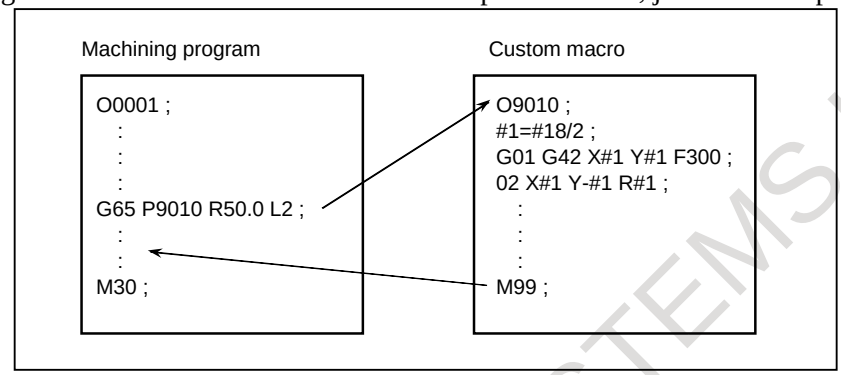
  If the above function is ordered together with this function, the tool offset screen is replaced by the conventional screen, so that G43.7 cannot be used.

(2) If this function is used, part of the specifications of the function below is limited.
  Tool offset protection cannot be performed with the 8-level data protection function.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

16 CUSTOM MACRO

Although subprograms are useful for repeating the same operation, the custom macro function also allows use of variables, arithmetic and logic operations, and conditional branches for easy development of general programs such as pocketing and user-defined canned cycles.
A machining program can call a custom macro with a simple command, just like a subprogram.



16.1 VARIABLES

An ordinary machining program specifies a G code and the travel distance directly with a numeric value; examples are G00 X100.0.
With a custom macro, numeric values can be specified directly or using a variable number. When a variable number is used, the variable value can be changed by a program or using operations on the MDI unit.

```
#1=#2+100 ;
G01 X#1 F300 ;
```

Explanation

- Variable representation

When specifying a variable, specify a number sign (#) followed by a variable number.
#i (i = 1, 2, 3, 4, ...)
[Example] #5
#109
#1005
A variable can also be represented as follows using <expression> described in the section about arithmetic and logic operation commands.
#[<expression>]
[Example] #[#100]
#[#1001-1]
#[#6/2]
Variable #i shown in the following can be replaced with a variable of #[<expression>].

- Types of variables

Variables can be classified as local variables, common variables, and system variables according to the variable number. Each of those variables has its own usage and characteristics. Read-only system constants are also provided.

FFYFR MACHINE SYSTEMS INC
Rob Zaherisky

- Range of variable values

Local and common variables can have a value in the following ranges. If the result of calculation exceeds the range, an alarm PS0111, “OVERFLOW :FLOATING” is issued.

When bit 0 (F16) of parameter No.6008 = 0

Maximum value: approx. $\pm 10^{308}$

Minimum value: approx. $\pm 10^{-308}$

Numeric data handled by a custom macro conforms to the IEEE standard and is handled as a double-precision real number. An error resulting from operation depends on the precision.

When bit 0 (F16) of parameter No.6008 = 1

Maximum value: approx. $\pm 10^{47}$

Minimum value: approx. $\pm 10^{-29}$

- Local variable (#1-#33)

A local variable is a variable that is used locally in a macro. That is, local variable #i used by a macro called at a certain time is different from that used by a macro called at another time, regardless of whether the two macros are the same. Therefore, for example, when macro A calls macro B during multiple calls or the like, it is impossible for macro B to corrupt a local variable used by macro A by erroneously using the variable.

A local variable is used to pass arguments. For information on correspondence between arguments and addresses, see the section about macro calling commands. The initial state of a local variable to which no arguments are passed is <null> and the user can freely use the variable. The attribute of a local variable is READ/WRITE enabled.

- Common variable (#100-#199, #500-#999, #98000-#98499)

A common variable is shared among the main program, subprograms called by the main program, and macros while a local variable is used locally in a macro. That is, #i used by a macro is the same as that used by another macro. Therefore, a resultant common variable obtained by using a macro can be used by another macro. The attribute of a common variable is basically READ/WRITE enabled. However, the common variable can be protected (its attribute is set to READ only) by specifying its variable number using parameters Nos.6031 and 6032. A common variable can be freely used by the user even when its usage is not defined by the system. The number of common variables can be specified by selecting one of the following.

(a) 100 common variables (Bit 6 (NCV) of parameter No. 8135 is 1)

The common variables #100 to #149 and #500 to #549 can be used. The variables #100 to #149 are cleared during power-down, but the variables #500 to #549 are retained during power-down.

(b) 600 common variables (Bit 6 (NCV) of parameter No. 8135 is 0)

The common variables #100 to #199 and #500 to #999 can be used. The variables #100 to #199 are cleared during power-down, but the variables #500 to #999 are retained during power-down.

(c) 1100 common variables (Addition of custom macro variables 1000 required.)

The common variables #100 to #199, #500 to #999 and #98000 to #98499 can be used. The variables #100 to #199 are cleared during power-down, but the variables #500 to #999 and #98000 to #98499 are retained during power-down.

NOTE

The default value of the common variables (#100 to #149(#199)) that are cleared during power-down is <null>.

The default value of the common variables (#500 to #549(#999) , (#98000 to #98499)) that are retained during power-down is 0.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Write protection of a common variable

Multiple common variables (#500 to #999) can be protected (their attributes are set to READ only) by setting variable numbers in parameters Nos.6031 and 6032. This protection is enabled for both Input/All Clear by MDI on the macro screen and write operation by a macro program. If the NC program specifies WRITE operation (used in the left side) for a common program in the set range, an alarm PS0116, “WRITE PROTECTED VARIABLE” is issued.

- Custom macro common variables between each path

Some or all of the custom macro common variables (#100 to #149 (#199, #499) and #500 to #599 (#999)) can be made common to all paths by setting parameter No. 6036 (#100 to #149 (#199, #499)) and parameter No. 6037 (#500 to #599 (#999)) appropriately. Also, whether or not to make the variables common can be selected on a path-by-path basis by setting bit 0 (NC1) of parameter No. 6020 for parameter No. 6036 and bit 1 (NC2) of parameter No. 6020 for parameter No. 6037 appropriately.

[Example]

Assume that system common parameter No. 6036 is set to 20.

If the setting of parameter No. 6036 is not to be reflected in the fourth path only, make the settings below (Table 16.1 (a)).

Table 16.1 (a)

Path number	No.6036	NC1	Area of the custom macro variables used
1	20	0	Variables #100 to #119 are common to all paths, while other variables for each path are independent.
2		0	
3		0	
4		1	Variables for path 4 are all independent.

- System variable

A variable whose usage does not vary in the system. The attribute of a system variable is READ only, WRITE only, or READ/WRITE enabled depending on the nature of a system variable.

- System constant

A system constant can be referenced as with a variable even though its value is fixed. The attribute of a system constant is READ only.

- Omission of the decimal point

When a variable value is defined in a program, the decimal point can be omitted.

[Example] When #1 = 123; is defined, the actual value of variable #1 is 123.000.

- Referencing variables

The value following an address can be replaced with a variable. When programming as <address>#i or <address>-#i, the variable value or the complement of it is used as the specified value of the address.

[Example] F#33 is the same as F1.5 when #33 = 1.5.
Z-#18 is the same as Z-20.0 when #18 = 20.0.
G#130 is the same as G3 when #130 = 3.0.
A variable cannot be referenced using address/, :, or O and N.

[Example] Programming such as O#27 or N#1 is not allowed.
n (n = 1 to 9) in the optional block skip /n cannot be a variable.
A variable number cannot be specified by a direct variable.

[Example] When replacing 5 in #5 with #30, specify #[#30] instead of ##30. No values exceeding the maximum allowable value for each address can be specified.

[Example] When #140 = 10000, G#140 exceeds the maximum allowable value.
When a variable is used as address data, the variable is automatically rounded off to the number of significant figures of each address or less.

FREEMACHINE SYSTEMS INC
Rob Zaharnsky

[Example] For a machine with an increment system of 1/1000 mm (IS-B), when #1 = 12.3456, G00 X#1; becomes G00 X12.346;.
If <expression>, described later, is used, the value following an address can be replaced with <expression>.
<address>[<expression>] or <address>-[<expression>]
The program code shown above indicates the value of <expression> or the complement of the value is used as an address value. Note that a constant with no decimal point, enclosed in brackets ([ ]), is assumed to have a decimal point at the end.

[Example] X[#24+#18*COS[#1]]
Z-[#18+#26]

- Undefined variable

When the value of a variable is not defined, such a variable is referred to as a "null" variable. Variables #0 and #3100 are always null variables. They cannot be written to, but they can be read.

(a) Quotation
When an undefined variable is quoted, the address itself is also ignored.

Original command	G90 X100 Y#1
Equivalent command when #1 = <null>	G90 X100
Equivalent command when #1 = 0	G90 X100 Y0

(b) Definition/replacement, addition, multiplication
When a local variable or common variable is directly replaced with <null>, the result is <null>. When a system variable is directly replaced with <null> or the result of calculation including <null> is replaced, a variable value of 0 is assumed.

Original expression (local variable)	#2=#1	#2=#1*5	#2=#1+#1
Replacement result (when #1 = <null>)	<null>	0	0
Replacement result (when #1 = 0)	0	0	0

Original expression (common variable)	#100=#1	#100=#1*5	#100=#1+#1
Replacement result (when #1 = <null>)	<null>	0	0
Replacement result (when #1 = 0)	0	0	0

Original expression (system variable)	#2001=#1	#2001=#1*5	#2001=#1+#1
Replacement result (when #1 = <null>)	0	0	0
Replacement result (when #1 = 0)	0	0	0

(c) Comparison
<null> differs from 0 only for EQ and NE.
<null> is equal to 0 for GE, GT, LE, and LT.
- When <null> is assigned to #1

Conditional expression	#1 EQ #0	#1 NE 0	#1 GE #0	#1 GT 0	#1 LE #0	#1 LT 0
Evaluation result	Established (true)	Established (true)	Established (true)	Not established (false)	Established (true)	Not established (false)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- When 0 is assigned to #1

Conditional expression	#1 EQ #0	#1 NE 0	#1 GE #0	#1 GT 0	#1 LE #0	#1 LT 0
Evaluation result	Not established (false)	Not established (false)	Established (true)	Not established (false)	Established (true)	Not established (false)

- Specifying a system variable (constant) by its name

A system variable (constant) is specified by its variable number, but it can also be specified by its predetermined system variable (constant) name. A system variable (constant) name begins with an underscore (_), followed by up to seven uppercase letters, numerics, or underscores. For axis-dependent variables (such as coordinates) or variables having a lot of data of similar types (such as tool compensation), subscript [n] (n: integer) can be used to specify values. In this case, n can be specified in <expression> format (calculation format).

The command format must be specified in [#system-variable-name] format, as shown below.

[#_DATE]

[Example]

[#_DATE]=20040117 ; : 2004.01.17 is assigned to #3011 (year month date).
[#_TIME]=161705 ; : 16:17:05 is assigned to #3012 (time minute second).
#101=[#_ABSMT[1]] ; : #5021 (machine coordinate value of the 1st axis) is read off and assigned to #101.
#102=[#_ABSKP[#500*2]] ; : #506x (skip position of [#500*2]th axis) is read off and assigned to #102.

If a value other than an integer is specified for subscript n, a variable value is referenced, assuming that the fractional portion is rounded off.

[Example]

[#_ABSIO[1.4999999]] : This value is assumed to be [#_ABSIO[1]], that is, #5001.
[#_ABSIO[1.5000000]] : This value is assumed to be [#_ABSIO[2]], that is, #5002.

NOTE

1 When the specified variable name is not registered, an alarm PS1098, “NO VARIABLE NAME” is issued.
2 When a negative or other invalid subscript is specified, an alarm PS1099, “ILLLEGAL SUFFIX [ ]” is issued.

- System constant #0, #3100-#3102 (Attribute: R)

Constants used as fixed values in the system can be used as system variables. Such constants are called system constants. The system constants provided are shown below.

Constant number	Constant name	Description
#0, #3100	[#_EMPTY]	Null
#3101	[#_PI]	Circular constant π = 3.14159265358979323846
#3102	[#_E]	Base of natural logarithm e = 2.71828182845904523536

- Specifying a common variable by its name

Specifying a variable name set by the SETVN command described later allows reading from or writing to a common variable.

The command must be specified in the form [#common-variable-name] such as [#VAR500].

[Example]

X[#POS1] Y[#POS2] ; : Specifying a position by the variable name
[#POS1] = #100+#101 ; : Executing a assignment statement by the variable name
#[100+[#ABS]] = 500 ; : Same as above (by a variable number)
#500 = [1000+[#POS2]*10] ; : Reading a variable by a variable name

FRIER MACHINE SYSTEMS INC Rob Zahansky

- Setting and specifying the name of a common variable (SETVN)

For the 50 common variables, #500 to #549, a name of up to eight characters can be specified by using a command as shown below. If the custom macro variable name expansion function is enabled, a name of up to 31 characters can be specified.

SETVN n [VAR500, VAR501, VAR502,.....] ;

n represents the starting number of a common variable for which the name is specified.
VAR500 is the variable name of variable n, VAR501 is the variable name of variable n+1, and VAR502 is the variable name of variable number n+2, and so on. Each string is delimited by a comma (,). All codes that can be used as meaningful information in a program except control in, control out, [,], EOB, EOR, and : (colon in a program number) can be used. However, each name must begin with an alphabetical character. Variable names are not cleared on switch-off.
Specifying a set variable name allows reading from or writing to the common variable. The command must be specified in the form [#common-variable-name] such as [#VAR500].

[Example] SETVN 510[TOOL_NO, WORK_NO, COUNTER1, COUNTER2];
The command above names the variables as follows.

Variable	Name
#510	TOOL_NO
#511	WORK_NO
#512	COUNTER1
#513	COUNTER2

The names specified by the command can be used in a program. For example, when 10 is assigned to #510, the expression [#TOOL_NO]=10; can be used instead #510=10;.
If the custom macro variable name expansion function is enabled,
the SETVN 510[TOOL_NUMBER000001_CHANGE_DETECT]; command assigns a name of 31 characters to the variable.

Variable	Name
#510	TOOL_NUMBER000001_CHANGE_DETECT

NOTE
If the same name was specified for different common variables, only the variable which has the smaller variable number can be referenced with the specified name.

FRYER MACHINE SYSTEMS INC
Rob Zaharisky

16.2 SYSTEM VARIABLES

System variables can be used to read and write internal CNC data such as tool compensation values and current position data. System variables are essential for automation and general-purpose program development.

List of system variables and constants

n represents a subscript.

R, W, and R/W are attributes of a variable and indicate read-only, write-only, and read/write enabled, respectively.

- Interface signals

System variable number	System variable name	Attribute	Description
#1000-#1031	[#_UI[n]]	R	Interface input signals (BIT), UI000-UI031 NOTE) Subscript n represents a BIT position (0-31).
#1032-#1035	[#_UIL[n]]	R	Interface input signals (LONG), UI000-UI031/ UI100-UI131/ UI200-UI231/UI300-UI331 NOTE) Subscript n (0-3): 0 = UI000-UI031, 1 = UI100-UI131, 2 = UI200-UI231, 3 = UI300-UI331
#1036-#1067	[#_UI[n]]	R	Interface input signals (BIT), UI400-UI431 NOTE) Subscript n represents a BIT position (32-63).
#1068-#1071	[#_UIL[n]]	R	Interface input signals (LONG), UI400-UI431 / UI500-UI531 / UI600-UI631 / UI700-UI731 NOTE) Subscript n (4-7): 4=UI400-UI431, 5=UI500-UI531, 6=UI600-UI631, 7=UI700-UI731
#1100-#1131	[#_UO[n]]	R/W	Interface output signals (BIT), UO000-UO031 NOTE) Subscript n represents a BIT position (0-31).
#1132-#1135	[#_UOL[n]]	R/W	Interface output signals (LONG), UO000-UO031/ UO100-UO131/UO200-UO231/UO300-UO331 NOTE) Subscript n (0-3): 0 = UO000-UO031, 1 = UO100-UO131, 2 = UO200-UO231, 3 = UO300-UO331
#1136-#1167	[#_UO[n]]	R/W	Interface output signals (BIT), UO400-UO431 NOTE) Subscript n represents a BIT position (32-63).
#1168-#1171	[#_UOL[n]]	R/W	Interface output signals (LONG), UO400-UO431/ UO500-UO531 / UO600-UO631 / UO700-UO731 NOTE) Subscript n (4-7): 4=UO400-UO431, 5=UO500-UO531, 6=UO600-UO631, 7=UO700-UO731

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Tool compensation value

M

When bit 3 (V15) of parameter No.6000 = 0

System variable number	System variable name	Attribute	Description
#2001-#2200	[#_OFSHW[n]]	R/W	Tool compensation value (H code, wear) Note) Subscript n represents a compensation number (1 to 200).
#10001-#10999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2400	[#_OFSHG[n]]	R/W	Tool compensation value (H code, geometry) Note) Subscript n represents a compensation number (1 to 200).
#11001-#11999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#12001-#12999	[#_OFSDW[n]]	R/W	Tool compensation value (D code, wear) Note) Subscript n represents a compensation number (1 to 999).
# 13001-#13999	[#_OFSDG[n]]	R/W	Tool compensation value (D code, geometry) Note) Subscript n represents a compensation number (1 to 999).

When bit 3 (V15) of parameter No.6000 = 1

System variable number	System variable name	Attribute	Description
#2001-#2200	[#_OFSHG[n]]	R/W	Tool compensation value (H code, geometry) Note) Subscript n represents a compensation number (1 to 200).
#10001-#10999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2400	[#_OFSHW[n]]	R/W	Tool compensation value (H code, wear) Note) Subscript n represents a compensation number (1 to 200).
#11001-#11999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2401-#2600	[#_OFSDG[n]]	R/W	Tool compensation value (D code, geometry) (Note 1) Subscript n represents a compensation number (1 to 200). Note 1) Enabled when bit 5 (D15) of parameter No.6004 = 1.
#12001-#12999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2601-#2800	[#_OFSDW[n]]	R/W	Tool compensation value (D code, wear) (Note 1) Subscript n represents a compensation number (1 to 200). Note 1) Enabled when bit 5 (D15) of parameter No.6004 = 1.
#13001-#13999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).

System variables not dependent on bit 3 (V15) of parameter No. 6000

System variable number	System variable name	Attribute	Description
#21001-#21999	[#_CORR_G[n]]	R/W	Corner R offset (geometry) Note) Subscript n represents a compensation number (1 to 999).
#22001-#22999	[#_CORR_W[n]]	R/W	Corner R offset (wear) Note) Subscript n represents a compensation number (1 to 999).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Tool compensation value

T

Without tool geometry/wear compensation memory

System variable number	System variable name	Attribute	Description
#2001-#2064 #10001-#10999	[#_OFSX[n]]	R/W	X-axis compensation value (*1) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2101-#2164 #11001-#11999	[#_OFSZ[n]]	R/W	Z-axis compensation value (*1) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2264 #12001-#12999	[#_OFSR[n]]	R/W	Tool nose radius compensation value Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2301-#2364 #13001-#13999	[#_OFST[n]]	R/W	Virtual tool tip T position Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2401-#2449 #14001-#14999	[#_OFSY[n]]	R/W	Y-axis compensation value (*1) Note) Subscript n represents a compensation number (1 to 49). When the number of sets is larger than 49, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

With tool geometry/wear compensation memory

System variable number	System variable name	Attribute	Description
#2001-#2064 #10001-#10999	[#_OFSXW[n]]	R/W	X-axis compensation value (wear) (*1) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2101-#2164 #11001-#11999	[#_OFSZW[n]]	R/W	Z-axis compensation value (wear) (*1) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2264 #12001-#12999	[#_OFSRW[n]]	R/W	Tool nose radius compensation value (wear) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2301-#2364 #13001-#13999	[#_OFST[n]]	R/W	Virtual tool tip T position Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

System variable number	System variable name	Attribute	Description
#2401-#2449 #14001-#14999	[#_OFSYW[n]]	R/W	Y-axis compensation value (wear) (*1) Note) Subscript n represents a compensation number (1 to 49). When the number of sets is larger than 49, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2451-#2499 #19001-#19999	[#_OFSYG[n]]	R/W	Y-axis compensation value (geometry) (*1) Note) Subscript n represents a compensation number (1 to 49). When the number of sets is larger than 49, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2701-#2749 #15001-#15999	[#_OFSXG[n]]	R/W	X-axis compensation value (geometry) (*1) Note) Subscript n represents a compensation number (1 to 49). When the number of sets is larger than 49, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 49).
#2801-#2849 #16001-#16999	[#_OFSZG[n]]	R/W	Z-axis compensation value (geometry) (*1) Note) Subscript n represents a compensation number (1 to 49). When the number of sets is larger than 49, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2901-#2964 #17001-#17999	[#_OFSRG[n]]	R/W	Tool nose radius compensation value (geometry) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- Workpiece coordinate system shift amount

T

System variable number	System variable name	Attribute	Description
#2501	[#_WKSFTX]	R/W	X-axis workpiece shift amount
#2601	[#_WKSFTZ]	R/W	Z-axis workpiece shift amount
#100751-#100800	[#_WZ_SFT [n]]	R/W	nth axis workpiece shift amount Note) Subscript n represents an axis number (1 to 50).

X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes

- Automatic operation or the like

System variable number	System variable name	Attribute	Description
#3000	[#_ALM]	W	Macro alarm
#3001	[#_CLOCK1]	R/W	Clock 1 (ms)
#3002	[#_CLOCK2]	R/W	Clock 2 (hr)
#3003	[#_CNTL1]	R/W	Enable or disable the suppression of single block stop. Enable or disable the waiting of the auxiliary function completion signal.
#3003 bit 0	[#_M_SBK]	R/W	Enable or disable the suppression of single block stop.
#3003 bit 1	[#_M_FIN]	R/W	Enable or disable waiting for the auxiliary function completion signal.
#3004	[#_CNTL2]	R/W	Enable or disable feed hold. Enable or disable feedrate override. Enable or disable exact stop check.

FRYER MACHINE SYSTEMS INC
Rob Zahenski

System variable number	System variable name	Attribute	Description
#3004 bit 0	[#_M_FHD]	R/W	Enable or disable feed hold.
#3004 bit 1	[#_M_OV]	R/W	Enable or disable feedrate override.
#3004 bit 2	[#_M_EST]	R/W	Enable or disable exact stop check.
#3005	[#_SETDT]	R/W	Read/write setting data.
#3006	[#_MSGSTP]	W	Stop with a message.
#3007	[#_MRIMG]	R	Status of a mirror image (DI and setting)
#3008	[#_PRSTR]	R	Restarting/not restarting a program

- Time

System variable number	System variable name	Attribute	Description
#3011	[#_DATE]	R	Year/Month/Date
#3012	[#_TIME]	R	Hour/Minute/Second

- Path number of the parameter to be read or written

System variable number	System variable name	Attribute	Description
#3018	—	R/W	Path number of the parameter to be read or written

- Number of the path on which a macro is being executed

System variable number	System variable name	Attribute	Description
#3019	[#_PATH_NO]	R	Number of the path on which a macro is being executed

- Number of parts

System variable number	System variable name	Attribute	Description
#3901	[#_PRTSA]	R/W	Total number of parts
#3902	[#_PRTSN]	R/W	Number of required parts

- Tool compensation memory

M

System variable number	System variable name	Attribute	Description
#3980	[#_OFSMEM]	R	Tool compensation memory information

- Main program number

System variable number	System variable name	Attribute	Description
#4000	[#_MAINO]	R	Main program number

- Modal information

M

System variable number	System variable name	Attribute	Description
#4001-#4030	[#_BUFG[n]]	R	Modal information on blocks that have been specified by last minute (G code) Note) Subscript n represents a G code group number.
#4102	[#_BUFB]	R	Modal information on blocks that have been specified by last minute (B code)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

System variable number	System variable name	Attribute	Description
#4107	[#_BUFD]	R	Modal information on blocks that have been specified by last minute (D code)
#4108	[#_BUFE]	R	Modal information on blocks that have been specified by last minute (E code)
#4109	[#_BUFF]	R	Modal information on blocks that have been specified by last minute (F code)
#4111	[#_BUFH]	R	Modal information on blocks that have been specified by last minute (H code)
#4113	[#_BUFM]	R	Modal information on blocks that have been specified by last minute (M code)
#4114	[#_BUFN]	R	Modal information on blocks that have been specified by last minute (sequence number)
#4115	[#_BUFO]	R	Modal information on blocks that have been specified by last minute (program number)
#4119	[#_BUFS]	R	Modal information on blocks that have been specified by last minute (S code)
#4120	[#_BUFT]	R	Modal information on blocks that have been specified by last minute (T code)
#4130	[#_BUFWZP]	R	Modal information on blocks that have been specified by last minute (additional workpiece coordinate system number)
#4201-#4230	[#_ACTG[n]]	R	Modal information on the block currently being executed (G code) Note) Subscript n represents a G code group number.
#4302	[#_ACTB]	R	Modal information on the block currently being executed (B code)
#4307	[#_ACTD]	R	Modal information on the block currently being executed (D code)
#4308	[#_ACTE]	R	Modal information on the block currently being executed (E code)
#4309	[#_ACTF]	R	Modal information on the block currently being executed (F code)
#4311	[#_ACTH]	R	Modal information on the block currently being executed (H code)
#4313	[#_ACTM]	R	Modal information on the block currently being executed (M code)
#4314	[#_ACTN]	R	Modal information on the block currently being executed (sequence number)
#4315	[#_ACTO]	R	Modal information on the block currently being executed (program number)
#4319	[#_ACTS]	R	Modal information on the block currently being executed (S code)
#4320	[#_ACTT]	R	Modal information on the block currently being executed (T code)
#4330	[#_ACTWZP]	R	Modal information on the block currently being executed (additional workpiece coordinate system number)
#4401-#4430	[#_INTG[n]]	R	Modal information on interrupted blocks (G code) Note) Subscript n represents a G code group number.
#4502	[#_INTB]	R	Modal information on interrupted blocks (B code)
#4507	[#_INTD]	R	Modal information on interrupted blocks (D code)
#4508	[#_INTE]	R	Modal information on interrupted blocks (E code)
#4509	[#_INTF]	R	Modal information on interrupted blocks (F code)
#4511	[#_INTH]	R	Modal information on interrupted blocks (H code)
#4513	[#_INTM]	R	Modal information on interrupted blocks (M code)
#4514	[#_INTN]	R	Modal information on interrupted blocks (sequence number)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

System variable number	System variable name	Attribute	Description
#4515	[#_INTO]	R	Modal information on interrupted blocks (program number)
#4519	[#_INTS]	R	Modal information on interrupted blocks (S code)
#4520	[#_INTT]	R	Modal information on interrupted blocks (T code)
#4530	[#_INTWZP]	R	Modal information on interrupted blocks (additional workpiece coordinate system number)

T

System variable number	System variable name	Attribute	Description
#4001-#4030	[#_BUFG[n]]	R	Modal information on blocks that have been specified by last minute (G code) Note) Subscript n represents a G code group number.
#4102	[#_BUFB]	R	Modal information on blocks that have been specified by last minute (B code)
#4108	[#_BUFE]	R	Modal information on blocks that have been specified by last minute (E code)
#4109	[#_BUFF]	R	Modal information on blocks that have been specified by last minute (F code)
#4113	[#_BUFM]	R	Modal information on blocks that have been specified by last minute (M code)
#4114	[#_BUFN]	R	Modal information on blocks that have been specified by last minute (sequence number)
#4115	[#_BUFO]	R	Modal information on blocks that have been specified by last minute (program number)
#4119	[#_BUFS]	R	Modal information on blocks that have been specified by last minute (S code)
#4120	[#_BUFT]	R	Modal information on blocks that have been specified by last minute (T code)
#4130	[#_BUFWZP]	R	Modal information on blocks that have been specified by last minute (additional workpiece coordinate system number)
#4201-#4230	[#_ACTG[n]]	R	Modal information on the block currently being executed (G code) Note) Subscript n represents a G code group number.
#4302	[#_ACTB]	R	Modal information on the block currently being executed (B code)
#4308	[#_ACTE]	R	Modal information on the block currently being executed (E code)
#4309	[#_ACTF]	R	Modal information on the block currently being executed (F code)
#4313	[#_ACTM]	R	Modal information on the block currently being executed (M code)
#4314	[#_ACTN]	R	Modal information on the block currently being executed (sequence number)
#4315	[#_ACTO]	R	Modal information on the block currently being executed (program number)
#4319	[#_ACTS]	R	Modal information on the block currently being executed (S code)
#4320	[#_ACTT]	R	Modal information on the block currently being executed (T code)
#4330	[#_ACTWZP]	R	Modal information on the block currently being executed (additional workpiece coordinate system number)
#4401-#4430	[#_INTG[n]]	R	Modal information on interrupted blocks (G code) Note) Subscript n represents a G code group number.
#4502	[#_INTB]	R	Modal information on interrupted blocks (B code)

FRYER MACHINE SYSTEMS INC
Rob Ashensky

System variable number	System variable name	Attribute	Description
#4508	[#_INTE]	R	Modal information on interrupted blocks (E code)
#4509	[#_INTF]	R	Modal information on interrupted blocks (F code)
#4513	[#_INTM]	R	Modal information on interrupted blocks (M code)
#4514	[#_INTN]	R	Modal information on interrupted blocks (sequence number)
#4515	[#_INTO]	R	Modal information on interrupted blocks (program number)
#4519	[#_INTS]	R	Modal information on interrupted blocks (S code)
#4520	[#_INTT]	R	Modal information on interrupted blocks (T code)
#4530	[#_INTWZP]	R	Modal information on interrupted blocks (additional workpiece coordinate system number)

- Position information

System variable number	System variable name	Attribute	Description
#5001-#5020	[#_ABSIO[n]]	R	End point position of the previous block (workpiece coordinate system) Note) Subscript n represents an axis number (1 to 20)
#100001-#100050			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#5021-#5040	[#_ABSMT[n]]	R	Specified current position of input unit (machine coordinate system) Note) Subscript n represents an axis number (1 to 20).
#100051-#100100			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#5041-#5060	[#_ABSOT[n]]	R	Specified current position (workpiece coordinate system) Note) Subscript n represents an axis number (1 to 20).
#100101-#100150			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#5061-#5080	[#_ABSKP[n]]	R	Skip position (workpiece coordinate system) Note) Subscript n represents an axis number (1 to 20).
#100151-#100200			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#118601-#118650	[#_MCNPOS[n]]	R	Specified current position of machine unit (machine coordinate system). Value of the specified current position without servo delay and acceleration / deceleration delay. Note) Subscript n represents an axis number (1 to 50).
#118651-#118700	[#_ACTMCN [n]]	R	Current position of machine unit (machine coordinate system). Value of the current position with servo delay and acceleration / deceleration delay. Note) Subscript n represents an axis number (1 to 50).

- Tool length compensation value

M

System variable number	System variable name	Attribute	Description
#5081-#5100	[#_TOFS[n]]	R	Tool length compensation value Note) Subscript n represents an axis number (1 to 20).
#100201-#100250			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).

FRYER MACHINE SYSTEMS INC
Rob Zamensky

- Tool offset value

T

System variable number	System variable name	Attribute	Description
#5081, #100201	[#_TOFSWX]	R	X-axis tool offset (wear)
#5082, #100202	[#_TOFSWZ]		Z-axis tool offset (wear)
#5083, #100203	[#_TOFSWY]		Y-axis tool offset (wear)
#5084 - #5100	[#_TOFS[n]]		Tool offset (wear) for an arbitrary axis Note) Subscript n represents an axis number (4 to 20).
#100204-#100250			Tool offset (wear) for an arbitrary axis Note) Subscript n represents an axis number (4 to 50).
#5121, #100901	[#_TOFSGX]	R	X-axis tool offset (geometry)
#5122, #100902	[#_TOFSGZ]		Z-axis tool offset (geometry)
#5123, #100903	[#_TOFSGY]		Y-axis tool offset (geometry)
#5124 - #5140	[#_TOFSG[n]]		Tool offset (geometry) for an arbitrary axis Note) Subscript n represents an axis number (4 to 20).
#100904-#100950			Tool offset (geometry) for an arbitrary axis Note) Subscript n represents an axis number (4 to 50).

X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- Servo position deviation

System variable number	System variable name	Attribute	Description
#5101-#5120	[#_SVERR[n]]	R	Servo positional deviation Note) Subscript n represents an axis number (1 to 20).
#100251-#100300			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).

- Manual handle interruption

System variable number	System variable name	Attribute	Description
#5121-#5140	[#_MIRTP[n]]	R	Manual handle interruption Note) Subscript n represents an axis number (1 to 20).
#100651-#100700			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).

- Distance to go

System variable number	System variable name	Attribute	Description
#5181-#5200	[#_DIST[n]]	R	Distance to go Note) Subscript n represents an axis number (1 to 20).
#100801-#100850			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).

- Workpiece origin offset value, extended workpiece origin offset value

M

System variable number	System variable name	Attribute	Description
#5201-#5220	[#_WZCMN[n]]	R/W	External workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5221-#5240	[#_WZG54[n]]	R/W	G54 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).

FRYER MACHINE SYSTEMS INC
Rob Zaranosky

System variable number	System variable name	Attribute	Description
#5241-#5260	[#_WZG55[n]]	R/W	G55 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5261-#5280	[#_WZG56[n]]	R/W	G56 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5281-#5300	[#_WZG57[n]]	R/W	G57 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5301-#5320	[#_WZG58[n]]	R/W	G58 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5321-#5340	[#_WZG59[n]]	R/W	G59 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#100301-#100350	[#_WZCMN[n]]	R/W	External workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100351-#100400	[#_WZG54[n]]	R/W	G54 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100401-#100450	[#_WZG55[n]]	R/W	G55 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100451-#100500	[#_WZG56[n]]	R/W	G56 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100501-#100550	[#_WZG57[n]]	R/W	G57 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100551-#100600	[#_WZG58[n]]	R/W	G58 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100601-#100650	[#_WZG59[n]]	R/W	G59 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
Extended workpiece origin offset value			
#7001-#7020	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#7021-#7040	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
: :	: :	: :	: :
#7941-#7960	[#_WZP48[n]]	R/W	G54.1P48 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#14001-#14020	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#14021-#14040	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
: :	: :	: :	: :
#19981-#20000	[#_WZP300[n]]	R/W	G54.1P300 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#101001-#101050	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#101051-#101100	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
: :	: :	: :	: :
#115901-#115950	[#_WZP299[n]]	R/W	G54.1P299 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#115951-#116000	[#_WZP300[n]]	R/W	G54.1P300 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).

FRYER MACHINE SYSTEMS INC
Rob Zawensky

T

System variable number	System variable name	Attribute	Description
#5201-#5220	[#_WZCMN[n]]	R/W	External workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5221-#5240	[#_WZG54[n]]	R/W	G54 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5241-#5260	[#_WZG55[n]]	R/W	G55 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5261-#5280	[#_WZG56[n]]	R/W	G56 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5281-#5300	[#_WZG57[n]]	R/W	G57 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5301-#5320	[#_WZG58[n]]	R/W	G58 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5321-#5340	[#_WZG59[n]]	R/W	G59 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#100301-#100350	[#_WZCMN[n]]	R/W	External workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100351-#100400	[#_WZG54[n]]	R/W	G54 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100401-#100450	[#_WZG55[n]]	R/W	G55 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100451-#100500	[#_WZG56[n]]	R/W	G56 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100501-#100550	[#_WZG57[n]]	R/W	G57 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100551-#100600	[#_WZG58[n]]	R/W	G58 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100601-#100650	[#_WZG59[n]]	R/W	G59 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
Extended workpiece origin offset value			
#7001-#7020	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#7021-#7040	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
: :	: :	: :	: :
#7941-#7960	[#_WZP48[n]]	R/W	G54.1P48 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#101001-#101050	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#101051-#101100	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
: :	: :	: :	: :
#115901-#115950	[#_WZP299[n]]	R/W	G54.1P299 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#115951-#116000	[#_WZP300[n]]	R/W	G54.1P300 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).

FRYER MACHINE SYSTEMS INC
Robert Fryer

- Skip position (detection unit)

System variable number	System variable name	Attribute	Description
#5421-#5440	[#_SKPDTC[n]]	R	Skip position (detection unit) Note) Subscript n represents an axis number (1 to 20).
#100701-#100750			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).

- Second geometry tool offset value

T

System variable number	System variable name	Attribute	Description
#5801-#5832 #27001-#27999	[#_OFSX2G[n]]	R/W	Second geometry tool offset X-axis compensation value Note) Subscript n represents a compensation number (1 to 32). If the number of pairs is greater than 32, the numbers on the left are also permitted. Note) Subscript n represents a compensation number (1 to 999).
#5833-#5864 #28001-#28999	[#_OFSZ2G[n]]	R/W	Second geometry tool offset Z-axis compensation value Note) Subscript n represents a compensation number (1 to 32). If the number of pairs is greater than 32, the numbers on the left are also permitted. Note) Subscript n represents a compensation number (1 to 999).
#5865-#5896 #29001-#29999	[#_OFSY2G[n]]	R/W	Second geometry tool offset Y-axis compensation value Note) Subscript n represents a compensation number (1 to 32). If the number of pairs is greater than 32, the numbers on the left are also permitted. Note) Subscript n represents a compensation number (1 to 999).

- Tool management function

M

System variable number	System variable name	Attribute	Description
#8400	-	R/W	Cartridge number
#8401	-	R	Data number
#8402	-	R	Tool type number (T)
#8403	-	R	Tool life counter
#8404	-	R	Maximum tool life value
#8405	-	R	Tool notice life value
#8406	-	R	Tool life status
#8407	-	R	Customize data 0 (bit)
#8408	-	R	Tool information
#8409	-	R	Tool length compensation number (H)
#8410	-	R	Cutter compensation number (D)
#8411	-	R	Spindle speed (S)
#8412	-	R	Cutting feedrate (F)
#8431-#8470	-	R	Customize data 1 to 40

FRYER MACHINE SYSTEMS INC
Rob Zahansky

T

System variable number	System variable name	Attribute	Description
#8400	-	R/W	Cartridge number
#8401	-	R	Data number
#8402	-	R	Tool type number (T)
#8403	-	R	Tool life counter
#8404	-	R	Maximum tool life value
#8405	-	R	Tool notice life value
#8406	-	R	Tool life status
#8407	-	R	Customize data 0 (bit)
#8408	-	R	Tool information
#8411	-	R	Spindle speed (S)
#8412	-	R	Cutting feedrate (F)
#8413	-	R	Tool geometry compensation number (G)
#8414	-	R	Tool wear compensation number (W)
#8431-#8470	-	R	Customize data 1 to 40

- Feedrate reduction ratio for rapid traverse overlap

System variable number	System variable name	Attribute	Description
#100851- #100900	[#_ROVLP [n]]	R/W	Feedrate reduction ratio for rapid traverse overlap Note) Subscript n represents an axis number (1 to 50).

- Serial spindle

System variable number	System variable name	Attribute	Description
#100951-#100954	[#_SPSTAT[n]]	R	State of each spindle Note) Subscript n represents a spindle number (1 to 4).

- Clamp value of spindle maximum speed

System variable number	System variable name	Attribute	Description
#100959	[#_CSSSMAX]	R	Spindle maximum speed commanded at clamp command of maximum spindle speed

- Dynamic standard tool compensation value

M

System variable number	System variable name	Attribute	Description
#118051-#118100	[#_DOFS1[n]]	R/W	Dynamic standard tool compensation value (first set) Note) Subscript n represents an axis number (1 to 50).
#118101-#118150	[#_DOFS2[n]]	R/W	Dynamic standard tool compensation value (second set) Note) Subscript n represents an axis number (1 to 50).
#118151-#118200	[#_DOFS3[n]]	R/W	Dynamic standard tool compensation value (third set) Note) Subscript n represents an axis number (1 to 50).
#118201-#118250	[#_DOFS4[n]]	R/W	Dynamic standard tool compensation value (fourth set) Note) Subscript n represents an axis number (1 to 50).
#118251-#118300	[#_DOFS5[n]]	R/W	Dynamic standard tool compensation value (fifth set) Note) Subscript n represents an axis number (1 to 50).
#118301-#118350	[#_DOFS6[n]]	R/W	Dynamic standard tool compensation value (sixth set) Note) Subscript n represents an axis number (1 to 50).

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

System variable number	System variable name	Attribute	Description
#118351-#118400	[#_DOFS7[n]]	R/W	Dynamic standard tool compensation value (seventh set) Note) Subscript n represents an axis number (1 to 50).
#118401-#118450	[#_DOFS8[n]]	R/W	Dynamic standard tool compensation value (eighth set) Note) Subscript n represents an axis number (1 to 50).

- Local coordinate system offset value

System variable number	System variable name	Attribute	Description
#118501-#118550	[#_LCLOFS[n]]	R	Local coordinate system offset value Note) Subscript n represents an axis number (1 to 50).

- Torque difference of Axis synchronous

System variable number	System variable name	Attribute	Description
#180801-#180850	[#_SYNTD[n]]	R	Torque difference of master axis and slave axis in Axis synchronous control Range：-32768 to 32767 Unit：Unit in which the set value corresponding to Nominal current limit of Amplifier [Ap] is 6554. (Note) Subscript n represents an axis number (1 to 50).

- Tool life management

System constant number	System constant name	Attribute	Description
#181000	[#_TLMGN_C]	R	Tool life management data (Tool group numbers)
#181001	[#_TLMML_C]	R	Tool life management data (Tool life values)
#181002	[#_TLMLC_C]	R	Tool life management data (Life counter values)

- Others

System constant number	System constant name	Attribute	Description
#8570	-	R/W	Switching of P-CODE variables/system variables (#10000 and above)

- System constant

System constant number	System constant name	Attribute	Description
#0, #3100	[#_EMPTY]	R	Null
#3101	[#_PI]	R	Circular constant π = 3.14159265358979323846
#3102	[#_E]	R	Base of natural logarithm e = 2.71828182845904523536

- Exclusion control between each path

System constant number	System constant name	Attribute	Description
#182001-#182010	[#_PRIO[n]]	R/W	System variable for exclusion control NOTE) Subscript n represents a number of system variables for exclusion control (1-10).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

R, W, and R/W are attributes of a variable and represents read-only, write-only, and read/write enabled, respectively.

- Interface signal #1000-#1031, #1032, #1033-#1035 (Attribute: R)
#1100-#1131, #1132, #1133-#1135 (Attribute: R/W)

[Input signal]

The status of interface input signals can be obtained by reading the value of system variables #1000 to #1032.

Variable number	Variable name	Point	Interface input signal
#1000	[#_UI[0]]	1	UI000 (2^0)
#1001	[#_UI[1]]	1	UI001 (2^1)
#1002	[#_UI[2]]	1	UI002 (2^2)
#1003	[#_UI[3]]	1	UI003 (2^3)
#1004	[#_UI[4]]	1	UI004 (2^4)
#1005	[#_UI[5]]	1	UI005 (2^5)
#1006	[#_UI[6]]	1	UI006 (2^6)
#1007	[#_UI[7]]	1	UI007 (2^7)
#1008	[#_UI[8]]	1	UI008 (2^8)
#1009	[#_UI[9]]	1	UI009 (2^9)
#1010	[#_UI[10]]	1	UI010 (2^{10})
#1011	[#_UI[11]]	1	UI011 (2^{11})
#1012	[#_UI[12]]	1	UI012 (2^{12})
#1013	[#_UI[13]]	1	UI013 (2^{13})
#1014	[#_UI[14]]	1	UI014 (2^{14})
#1015	[#_UI[15]]	1	UI015 (2^{15})
#1016	[#_UI[16]]	1	UI016 (2^{16})
#1017	[#_UI[17]]	1	UI017 (2^{17})
#1018	[#_UI[18]]	1	UI018 (2^{18})
#1019	[#_UI[19]]	1	UI019 (2^{19})
#1020	[#_UI[20]]	1	UI020 (2^{20})
#1021	[#_UI[21]]	1	UI021 (2^{21})
#1022	[#_UI[22]]	1	UI022 (2^{22})
#1023	[#_UI[23]]	1	UI023 (2^{23})
#1024	[#_UI[24]]	1	UI024 (2^{24})
#1025	[#_UI[25]]	1	UI025 (2^{25})
#1026	[#_UI[26]]	1	UI026 (2^{26})
#1027	[#_UI[27]]	1	UI027 (2^{27})
#1028	[#_UI[28]]	1	UI028 (2^{28})
#1029	[#_UI[29]]	1	UI029 (2^{29})
#1030	[#_UI[30]]	1	UI030 (2^{30})
#1031	[#_UI[31]]	1	UI031 (2^{31})
#1032	[#_UIL[0]]	32	UI000-UI031
#1033	[#_UIL[1]]	32	UI100-UI131
#1034	[#_UIL[2]]	32	UI200-UI231
#1035	[#_UIL[3]]	32	UI300-UI331

Variable value	Input signal
1.0	Contact closed
0.0	Contact opened

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Since the read value is 1.0 or 0.0 regardless of the unit system, the unit system must be considered when a macro is created.
The input signals at 32 points can be read at a time by reading from system variables #1032 to #1035.

$$\#1032 = \sum_{i=0}^{30} \#[1000+i] \times 2^{i} - \#1031 \times 2^{31}$$

$$\#[1032+n] = \sum_{i=0}^{30} \left\{ 2^{i} \times V_{i} \right\} - 2^{31} \times V_{31}$$

When $UIn_i = 0$, $V_i = 0$.
When $UIn_i = 1$, $V_i = 1$.
n = 0-3

[Output signal]
Interface output signals can be sent by assigning values to system variables #1100 to #1132 for sending interface signals.

Variable number	Variable name	Point	Interface input signal
#1100	[#_UO[0]]	1	UO000 (2^0)
#1101	[#_UO[1]]	1	UO001 (2^1)
#1102	[#_UO[2]]	1	UO002 (2^2)
#1103	[#_UO[3]]	1	UO003 (2^3)
#1104	[#_UO[4]]	1	UO004 (2^4)
#1105	[#_UO[5]]	1	UO005 (2^5)
#1106	[#_UO[6]]	1	UO006 (2^6)
#1107	[#_UO[7]]	1	UO007 (2^7)
#1108	[#_UO[8]]	1	UO008 (2^8)
#1109	[#_UO[9]]	1	UO009 (2^9)
#1110	[#_UO[10]]	1	UO010 (2^{10})
#1111	[#_UO[11]]	1	UO011 (2^{11})
#1112	[#_UO[12]]	1	UO012 (2^{12})
#1113	[#_UO[13]]	1	UO013 (2^{13})
#1114	[#_UO[14]]	1	UO014 (2^{14})
#1115	[#_UO[15]]	1	UO015 (2^{15})
#1116	[#_UO[16]]	1	UO016 (2^{16})
#1117	[#_UO[17]]	1	UO017 (2^{17})
#1118	[#_UO[18]]	1	UO018 (2^{18})
#1119	[#_UO[19]]	1	UO019 (2^{19})
#1120	[#_UO[20]]	1	UO020 (2^{20})
#1121	[#_UO[21]]	1	UO021 (2^{21})
#1122	[#_UO[22]]	1	UO022 (2^{22})
#1123	[#_UO[23]]	1	UO023 (2^{23})
#1124	[#_UO[24]]	1	UO024 (2^{24})
#1125	[#_UO[25]]	1	UO025 (2^{25})
#1126	[#_UO[26]]	1	UO026 (2^{26})
#1127	[#_UO[27]]	1	UO027 (2^{27})
#1128	[#_UO[28]]	1	UO028 (2^{28})
#1129	[#_UO[29]]	1	UO029 (2^{29})
#1130	[#_UO[30]]	1	UO030 (2^{30})
#1131	[#_UO[31]]	1	UO031 (2^{31})
#1132	[#_UOL[0]]	32	UO000-UO031
#1133	[#_UOL[1]]	32	UO100-UO131
#1134	[#_UOL[2]]	32	UO200-UO231
#1135	[#_UOL[3]]	32	UO300-UO331

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Variable value	Input signal
1.0	Contact closed
0.0	Contact opened

The output signals at 32 points can be written at a time by writing to system variables #1132 to #1135. The signals can also be read.

$$\#1132 = \sum_{i=0}^{30} \#[1000+i] \times 2^{i} - \#1131 \times 2^{31}$$

$$\#[1132+n] = \sum_{i=0}^{30} \left\{ 2^{i} \times V_{i} \right\} - 2^{31} \times V_{31}$$

When UIn_i = 0, V_i = 0.
When UIn_i = 1, V_i = 1.
n = 0-3

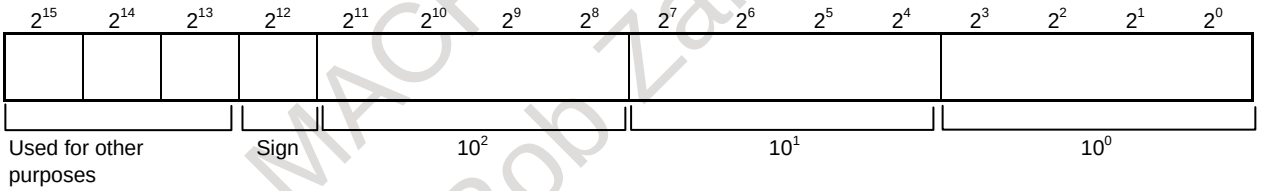
NOTE

1 When a value other than 1.0 or 0.0 is assigned to variables #1100 to #1131, it is assumed as follows.
<null> is assumed to be 0.
A value other than <null> or 0 is assumed to be 1.
Where, a value less than 0.00000001 is undefined.

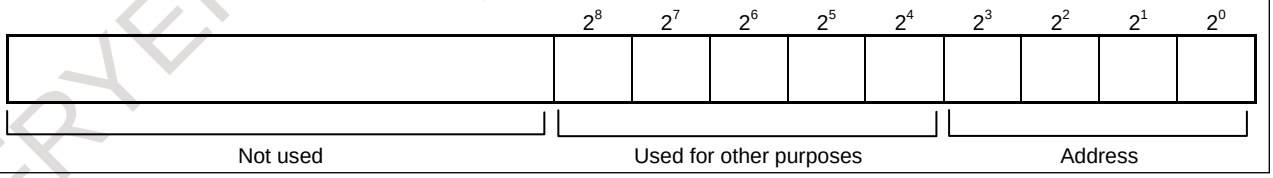
2 When any of UI016 to UI031, UI100 to UI131, UI200 to UI231, UI300 to UI331, UO016 to UO031, UO200 to UO231, and UO300 to UO331 are used, bit 0 (MIF) of parameter No. 6001 must be set to 1.

Example

Structure of DI



Structure of DO



<1> The specified address D value is output to DO and signed BCD 3 digits are read into #100.

Macro calling instruction

```
G65 P9100 D (address);
```

A custom macro body is created as follows.

```
O9100 ;
#1132 = #1132 AND 496 OR#7 ;      : Address output
G65 P9101 T60 ;                   : Timer macro. For details, refer to the example of clock (#3001).
#100 = BIN[#1032 AND 4095] ;      : BCD 3 digits are read.
IF [#1012 EQ 0] GOTO 9100 ;       : A sign is attached.
#100 = -#100
N9100 M99
```


FRYER MACHINE SYSTEMS INC
Rob Zahensky

<2> Eight types of specified address D values are output to DO and signed BCD 6 digits (3-digit integer part + 3-digit fractional part) are read into #101.

Structure on the machine side

When DO 2^0 = 0: Data with 3 decimal places
When DO 2^0 = 1: Data with 3-digit integer part
When DO 2^3 to 2^1 = 000: No1 data when #1 = 0
When DO 2^3 to 2^1 = 001: No2 data when #2 = 0
:
When DO 2^3 to 2^1 = 111: No8 data when #8 = 0

Macro calling instruction

```
G65    P9111    D (data number);
```

A custom macro body is created as follows.

```
O9111 ;
G65 P9100 D[#7*2+1] ;          : Call the macro of <1> O9100
#101 = #100    ;
G65 P9100 D[#7*2] ;
#101 = #101 + #100/1000 ;
M99 ;
```

- Interface signal R address
#1036 to #1067, #1068, #1069 to #1071 (Attribute: R)
#1136 to #1167, #1168, #1169 to #1171 (Attribute: R/W)

By setting bit 2 (IFR) of parameter No. 6020 to 1, this function is enabled. Set the start address of each R area in parameter No. 6093 or 6094. From the start address, 128 signal inputs or outputs are assigned. Set a multiple of 4 (0, 4, 8, ...) in parameters Nos. 6093 and 6094.

[Input signal]
The status of the input signal at each R address can be read with the value of system variable #1036 to #1071.

Variable number	Variable name	Point	Interface input signal
#1036	[#_UI[32]]	1	UI400 (2^0)
#1037	[#_UI[33]]	1	UI401 (2^1)
to	to	to	to
#1067	[#_UI[63]]	1	UI431 (2^{31})
#1068	[#_UIL[4]]	32	UI400 to UI431
#1069	[#_UIL[5]]	32	UI500 to UI531
#1070	[#_UIL[6]]	32	UI600 to UI631
#1071	[#_UIL[7]]	32	UI700 to UI731

Variable value	Input signal
1.0	Contact closed
0.0	Contact opened

[Output signal]
A value can be written to the output signal at each R address and the status of the signal can be read with the value of system variable #1136 to #1171.

Variable number	Variable name	Point	Interface input signal
#1136	[#_UO[32]]	1	UO400 (2^0)
#1137	[#_UO[33]]	1	UO401 (2^1)
to	to	to	to

FRYER MACHINE SYSTEMS INC
Rob Zamonsky

Variable number	Variable name	Point	Interface input signal
#1167	[#_UO[63]]	1	UO431 (2^{31})
#1168	[#_UOL[4]]	32	UO400 to UO431
#1169	[#_UOL[5]]	32	UO500 to UO531
#1170	[#_UOL[6]]	32	UO600 to UO631
#1171	[#_UOL[7]]	32	UO700 to UO731

Variable value	Input signal
1.0	Contact closed
0.0	Contact opened

NOTE

1 When signals UO400 to UO431 are handled from system variables #1136 to #1171, do not write any of signals UO400 to UO431 from a ladder or another interface. Specifying a write may cause a conflict between read from and write to the signal, resulting in an unexpected change to the signal.

2 For signals UO400 to UO731, R addresses are used, while for signals UO000 to UO331, F addresses are used. Therefore, to handle these signals with a ladder, it is necessary to consider synchronization processing of I/O signals. For details, refer to Subsection, “Synchronization Processing of I/O Signals”, in the “PMC Programming Manual (B-64513EN)”.

- Tool compensation value #2001-#2800, #10001-#13999, #21001-#22999 (Attribute: R/W)

M

The compensation values can be obtained by reading system variables #2001 to #2800, #10001 to #13999, or #21001 to #22999 for tool compensation. The compensation values can also be changed by assigning values to the system variables.

- When the number of compensations is 200 or less
  When bit 3 (V15) of parameter No.6000 = 0

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2201	[#_OFSHG[1]] or [#_OFSZG[1]]	#2001	[#_OFSHW[1]] or [#_OFSZW[1]]
2	#2202	[#_OFSHG[2]] or [#_OFSZG[2]]	#2002	[#_OFSHW[2]] or [#_OFSZW[2]]
3	#2203	[#_OFSHG[3]] or [#_OFSZG[3]]	#2003	[#_OFSHW[3]] or [#_OFSZW[3]]
:	:	:	:	:
199	#2399	[#_OFSHG[199]] or [#_OFSZG[199]]	#2199	[#_OFSHW[199]] or [#_OFSZW[199]]
200	#2400	[#_OFSHG[200]] or [#_OFSZG[200]]	#2200	[#_OFSHW[200]] or [#_OFSZW[200]]

FRYER MACHINE SYSTEMS INC
Ron Zarensky

When bit 3 (V15) of parameter No.6000 = 1

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2001	[#_OFSHG[1]] or [#_OFSZG[1]]	#2201	[#_OFSHW[1]] or [#_OFSZW[1]]
2	#2002	[#_OFSHG[2]] or [#_OFSZG[2]]	#2202	[#_OFSHW[2]] or [#_OFSZW[2]]
3	#2003	[#_OFSHG[3]] or [#_OFSZG[3]]	#2203	[#_OFSHW[3]] or [#_OFSZW[3]]
:	:	:	:	:
199	#2199	[#_OFSHG[199]] or [#_OFSZG[199]]	#2399	[#_OFSHW[199]] or [#_OFSZW[199]]
200	#2200	[#_OFSHG[200]] or [#_OFSZG[200]]	#2400	[#_OFSHW[200]] or [#_OFSZW[200]]

D code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2401	[#_OFSDG[1]] or [#_OFSRG[1]]	#2601	[#_OFSDW[1]] or [#_OFSRW[1]]
2	#2402	[#_OFSDG[2]] or [#_OFSRG[2]]	#2602	[#_OFSDW[2]] or [#_OFSRW[2]]
3	#2403	[#_OFSDG[3]] or [#_OFSRG[3]]	#2603	[#_OFSDW[3]] or [#_OFSRW[3]]
:	:	:	:	:
199	#2599	[#_OFSDG[199]] or [#_OFSRG[199]]	#2799	[#_OFSDW[199]] or [#_OFSRW[199]]
200	#2600	[#_OFSDG[200]] or [#_OFSRG[200]]	#2800	[#_OFSDW[200]] or [#_OFSRW200]]

NOTE

1 When #2401 to #2800 are used for reading or writing of D codes, bit 5 (D15) of parameter No.6004 must be set to 1.

2 When bit 5 (D15) of parameter No.6004 is set to 1, system variables #2500 to #2806 for workpiece origin offset cannot be used. Use system variables #5201 to #5324.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- When the number of compensations is more than 200 (For compensation with a compensation number of 200 or less, #2001 to #2800 can also be used.)
  When bit 3 (V15) of parameter No.6000 = 0

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#11001	[#_OFSHG[1]] or [#_OFSZG[1]]	#10001	[#_OFSHW[1]] or [#_OFSZW[1]]
2	#11002	[#_OFSHG[2]] or [#_OFSZG[2]]	#10002	[#_OFSHW[2]] or [#_OFSZW[2]]
3	#11003	[#_OFSHG[3]] or [#_OFSZG[3]]	#10003	[#_OFSHW[3]] or [#_OFSZW[3]]
:	:	:	:	:
998	#11998	[#_OFSHG[998]] or [#_OFSZG[998]]	#10998	[#_OFSHW[998]] or [#_OFSZW[998]]
999	#11999	[#_OFSHG[999]] or [#_OFSZG[999]]	#10999	[#_OFSHW[999]] or [#_OFSZW[999]]

D code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#13001	[#_OFSDG[1]] or [#_OFSRG[1]]	#12001	[#_OFSDW[1]] or [#_OFSRW[1]]
2	#13002	[#_OFSDG[2]] or [#_OFSRG[2]]	#12002	[#_OFSDW[2]] or [#_OFSRW[2]]
3	#13003	[#_OFSDG[3]] or [#_OFSRG[3]]	#12003	[#_OFSDW[3]] or [#_OFSRW[3]]
:	:	:	:	:
998	#13998	[#_OFSDG[998]] or [#_OFSRG[998]]	#12998	[#_OFSDW[998]] or [#_OFSRW[998]]
999	#13999	[#_OFSDG[999]] or [#_OFSRG[999]]	#12999	[#_OFSDW[999]] or [#_OFSRW[999]]

FRYER MACHINE SYSTEMS INC
Rob Zaherlsky

When bit 3 (V15) of parameter No.6000 = 1

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#10001	[#_OFSHG[1]] or [#_OFSZG[1]]	#11001	[#_OFSHW[1]] or [#_OFSZW[1]]
2	#10002	[#_OFSHG[2]] or [#_OFSZG[2]]	#11002	[#_OFSHW[2]] or [#_OFSZW[2]]
3	#10003	[#_OFSHG[3]] or [#_OFSZG[3]]	#11003	[#_OFSHW[3]] or [#_OFSZW[3]]
:	:	:	:	:
998	#10998	[#_OFSHG[998]] or [#_OFSZG[998]]	#11998	[#_OFSHW[998]] or [#_OFSZW[998]]
999	#10999	[#_OFSHG[999]] or [#_OFSZG[999]]	#11999	[#_OFSHW[999]] or [#_OFSZW[999]]

D code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#12001	[#_OFSDG[1]] or [#_OFSRG[1]]	#13001	[#_OFSDW[1]] or [#_OFSRW[1]]
2	#12002	[#_OFSDG[2]] or [#_OFSRG[2]]	#13002	[#_OFSDW[2]] or [#_OFSRW[2]]
3	#12003	[#_OFSDG[3]] or [#_OFSRG[3]]	#13003	[#_OFSDW[3]] or [#_OFSRW[3]]
:	:	:	:	:
998	#12998	[#_OFSDG[998]] or [#_OFSRG[998]]	#13998	[#_OFSDW[998]] or [#_OFSRW[998]]
999	#12999	[#_OFSDG[999]] or [#_OFSRG[999]]	#13999	[#_OFSDW[999]] or [#_OFSRW[999]]

System variables not dependent on bit 3 (V15) of parameter No. 6000

Corner R offset				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#21001	[#_CORR_G[1]]	#22001	[#_CORR_W[1]]
2	#21002	[#_CORR_G[2]]	#22002	[#_CORR_W[2]]
3	#21003	[#_CORR_G[3]]	#22003	[#_CORR_W[3]]
:	:	:	:	:
998	#21998	[#_CORR_G[998]]	#22998	[#_CORR_W[998]]
999	#21999	[#_CORR_G[999]]	#22999	[#_CORR_W[999]]

FRYER MACHINE SYSTEMS INC
Roh Zahensky

- Tool compensation value #2001-#2964, #10001-#19999 (Attribute: R/W)

T

The compensation values can be obtained by reading system variables #2001 to #2964 or #10001 to #19999 for tool compensation. The compensation values can also be changed by assigning values to the system variables.

<1> Without tool geometry/wear compensation memory

- When the number of compensations is 64 or less

Compensation number	Variable number	Variable name	Description
1	#2001	[#_OFSX[1]]	X-axis compensation value (*1)
2	#2002	[#_OFSX[2]]	
3	#2003	[#_OFSX[3]]	
:	:	:	
63	#2063	[#_OFSX[63]]	
64	#2064	[#_OFSX[64]]	
1	#2101	[#_OFSZ[1]]	Z-axis compensation value (*1)
2	#2102	[#_OFSZ[2]]	
3	#2103	[#_OFSZ[3]]	
:	:	:	
63	#2163	[#_OFSZ[63]]	
64	#2164	[#_OFSZ[64]]	
1	#2201	[#_OFSR[1]]	Tool nose radius compensation value
2	#2202	[#_OFSR[2]]	
3	#2203	[#_OFSR[3]]	
:	:	:	
63	#2263	[#_OFSR[63]]	
64	#2264	[#_OFSR[64]]	
1	#2301	[#_OFST[1]]	Virtual tool tipT position
2	#2302	[#_OFST[2]]	
3	#2303	[#_OFST[3]]	
:	:	:	
63	#2363	[#_OFST[63]]	
64	#2364	[#_OFST[64]]	
1	#2401	[#_OFSY[1]]	Y-axis compensation value (*1)
2	#2402	[#_OFSY[2]]	
3	#2403	[#_OFSY[3]]	
:	:	:	
48	#2448	[#_OFSY[48]]	
49	#2449	[#_OFSY[49]]	

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

FRYER MACHINE SYSTEMS INC
Rob Zumensky

- When the number of compensations is more than 64 (For compensation with a compensation number of 64 or less, #2001 to #2449 can also be used.)

Compensation number	Variable number	Variable name	Description
1	#10001	[#_OFSX[1]]	X-axis compensation value (*1)
2	#10002	[#_OFSX[2]]	
3	#10003	[#_OFSX[3]]	
:	:	:	
998	#10998	[#_OFSX[998]]	
999	#10999	[#_OFSX[999]]	
1	#11001	[#_OFSZ[1]]	Z-axis compensation value (*1)
2	#11002	[#_OFSZ[2]]	
3	#11003	[#_OFSZ[3]]	
:	:	:	
998	#11998	[#_OFSZ[998]]	
999	#11999	[#_OFSZ[999]]	
1	#12001	[#_OFSR[1]]	Tool nose radius compensation value
2	#12002	[#_OFSR[2]]	
3	#12003	[#_OFSR[3]]	
:	:	:	
998	#12998	[#_OFSR[998]]	
999	#12999	[#_OFSR[999]]	
1	#13001	[#_OFST[1]]	Virtual tool tip T position
2	#13002	[#_OFST[2]]	
3	#13003	[#_OFST[3]]	
:	:	:	
998	#13998	[#_OFST[998]]	
999	#13999	[#_OFST[999]]	
1	#14001	[#_OFSY[1]]	Y-axis compensation value (*1)
2	#14002	[#_OFSY[2]]	
3	#14003	[#_OFSY[3]]	
:	:	:	
998	#14998	[#_OFSY[998]]	
999	#14999	[#_OFSY[999]]	

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

<2> With tool geometry/wear compensation memory

- When the number of compensations is 64 or less

Compensation number	Variable number	Variable name	Description
1	#2001	[#_OFSXW[1]]	X-axis compensation value (wear) (*1)
2	#2002	[#_OFSXW[2]]	
3	#2003	[#_OFSXW[3]]	
:	:	:	
63	#2063	[#_OFSXW[63]]	
64	#2064	[#_OFSXW[64]]	
1	#2101	[#_OFSZW[1]]	Z-axis compensation value (wear) (*1)
2	#2102	[#_OFSZW[2]]	
3	#2103	[#_OFSZW[3]]	
:	:	:	
63	#2163	[#_OFSZW[63]]	
64	#2164	[#_OFSZW[64]]	

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Compensation number	Variable number	Variable name	Description
1	#2201	[#_OFSRW[1]]	Tool nose radius compensation value (wear)
2	#2202	[#_OFSRW [2]]	
3	#2203	[#_OFSRW [3]]	
:	:	:	
63	#2263	[#_OFSRW [63]]	
64	#2264	[#_OFSRW [64]]	
1	#2301	[#_OFST[1]]	Virtual tool tip T position
2	#2302	[#_OFST[2]]	
3	#2303	[#_OFST[3]]	
:	:	:	
63	#2363	[#_OFST[63]]	
64	#2364	[#_OFST[64]]	
1	#2401	[#_OFSYW[1]]	Y-axis compensation value (wear) (*1)
2	#2402	[#_OFSYW [2]]	
3	#2403	[#_OFSYW [3]]	
:	:	:	
48	#2448	[#_OFSYW [48]]	
49	#2449	[#_OFSYW [49]]	
1	#2451	[#_OFSYG[1]]	Y-axis compensation value (geometry) (*1)
2	#2452	[#_OFSYG [2]]	
3	#2453	[#_OFSYG [3]]	
:	:	:	
48	#2498	[#_OFSYG [48]]	
49	#2499	[#_OFSYG [49]]	
1	#2701	[#_OFSXG[1]]	X-axis compensation value (geometry) (*1)
2	#2702	[#_OFSXG[2]]	
3	#2703	[#_OFSXG [3]]	
:	:	:	
48	#2748	[#_OFSXG [48]]	
49	#2749	[#_OFSXG [49]]	
1	#2801	[#_OFSZG[1]]	Z-axis compensation value (geometry) (*1)
2	#2802	[#_OFSZG[2]]	
3	#2803	[#_OFSZG[3]]	
:	:	:	
48	#2848	[#_OFSZG[48]]	
49	#2849	[#_OFSZG[49]]	
1	#2901	[#_OFSRG[1]]	Tool nose radius compensation value (geometry)
2	#2902	[#_OFSRG[2]]	
3	#2903	[#_OFSRG[3]]	
:	:	:	
63	#2963	[#_OFSRG[63]]	
64	#2964	[#_OFSRG[64]]	

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- When the number of compensations is more than 64 (For compensation with a compensation number of 64 or less, #2001 to #2964 or #10001 to #19999 can also be used.)

FRYER MACHINE SYSTEMS INC
Robert Ahensky

Compensation number	Variable number	Variable name	Description
1	#10001	[#_OFSXW[1]]	X-axis compensation value (wear) (*1)
2	#10002	[#_OFSXW[2]]	
3	#10003	[#_OFSXW[3]]	
:	:	:	
998	#10998	[#_OFSXW[998]]	
999	#10999	[#_OFSXW[999]]	
1	#11001	[#_OFSZW[1]]	Z-axis compensation value (wear) (*1)
2	#11002	[#_OFSZW[2]]	
3	#11003	[#_OFSZW[3]]	
:	:	:	
998	#11998	[#_OFSZW[998]]	
999	#11999	[#_OFSZW[999]]	
1	#12001	[#_OFSRW[1]]	Tool nose radius compensation value (wear)
2	#12002	[#_OFSRW [2]]	
3	#12003	[#_OFSRW [3]]	
:	:	:	
998	#12998	[#_OFSRW [998]]	
999	#12999	[#_OFSRW [999]]	
1	#13001	[#_OFST[1]]	Virtual tool tip T position
2	#13002	[#_OFST[2]]	
3	#13003	[#_OFST[3]]	
:	:	:	
998	#13998	[#_OFST[998]]	
999	#13999	[#_OFST[999]]	
1	#14001	[#_OFSYW[1]]	Y-axis compensation value (wear) (*1)
2	#14002	[#_OFSYW [2]]	
3	#14003	[#_OFSYW [3]]	
:	:	:	
998	#14998	[#_OFSYW [998]]	
999	#14999	[#_OFSYW [999]]	
1	#15001	[#_OFSXG[1]]	X-axis compensation value (geometry) (*1)
2	#15002	[#_OFSXG[2]]	
3	#15003	[#_OFSXG [3]]	
:	:	:	
998	#15998	[#_OFSXG [998]]	
999	#15999	[#_OFSXG [999]]	
1	#16001	[#_OFSZG[1]]	Z-axis compensation value (geometry) (*1)
2	#16002	[#_OFSZG[2]]	
3	#16003	[#_OFSZG[3]]	
:	:	:	
998	#16998	[#_OFSZG[998]]	
999	#16999	[#_OFSZG[999]]	
1	#17001	[#_OFSRG[1]]	Tool nose radius compensation value (geometry)
2	#17002	[#_OFSRG[2]]	
3	#17003	[#_OFSRG[3]]	
:	:	:	
998	#17998	[#_OFSRG[998]]	
999	#17999	[#_OFSRG[999]]	
1	#19001	[#_OFSYG[1]]	Y-axis compensation value (geometry) (*1)
2	#19002	[#_OFSYG[2]]	
3	#19003	[#_OFSYG[3]]	
:	:	:	
998	#19998	[#_OFSYG[998]]	
999	#19999	[#_OFSYG[999]]	

FRYER MACHINE SYSTEMS INC
Roman Ahensky

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- Alarm #3000 (Attribute: W)

When an error is detected in a macro, an unit can enter the alarm state. In addition, an alarm message of up to 60 characters with alphabet and numerals can be specified between a control-out and a control-in after the expression. When an alarm message is not specified, a macro alarm is used instead.

Variable number	Variable name	Description
#3000	[#_ALM]	Macro alarm

When bit 1 (MCA) of parameter No.6008 = 0

#3000 = n (ALARM MESSAGE); (n: 0-200)

On the screen, the alarm number obtained by adding the value of #3000 to 3000 and alarm message appear after MC.

(Example) #3000 = 1 (ALARM MESSAGE);
=> "MC3001 ALARM MESSAGE" appears on the alarm screen.

When bit 1 (MCA) of parameter No.6008 = 1

#3000 = n (ALARM MESSAGE); (n: 0-4095)

On the screen, the alarm number of #3000 and alarm message appear after MC.

(Example) #3000=1 (ALARM MESSAGE);
=> "MC0001 ALARM MESSAGE" appears on the alarm screen.

When #3000 is commanded with the comment section, the order of alarm message and the comment section change places by setting value of bit 3 (NCM) of parameter No.6020.

When bit 3 (NCM) of parameter No.6020 is set to 0

The first section is an alarm message and followings are comments.

(Example) #3000 =1 (ALARM MESSAGE) (COMMENT 1) (COMMENT 2);

When bit 3 (NCM) of parameter No.6020) is set to 1

The last section is an alarm message and others are comments.

(Example) #3000 =1 (COMMENT 1) (COMMENT 2) (ALARM MESSAGE);

- Clock #3001, #3002 (Attribute: R/W)

The clock time can be obtained by reading system variables #3001 and #3002 for clocks. The time can be preset by entering a value in the system variables.

Type	Variable number	Variable name	Unit	At power-on	Count condition
Clock 1	#3001	[#_CLOCK1]	1 ms	Reset to 0	Anytime
Clock 2	#3002	[#_CLOCK2]	1 hour	Same as at power-down	When the STL signal is on

The clock accuracy is 16 ms. Clock 1 returns to 0 after a lapse of 2147483648 ms. Clock 2 returns to 0 after a lapse of 9544.37176 hours.

[Example]

```
Timer
Macro calling command
G65 P9101 T (wait time) ms ;
A macro is created as follows.
O9101;
#3001 = 0;   Initial setting
WHILE [#3001 LE #20] DO1:   Wait for a specified time
END1 ;
M99 ;
```

FRYER MACHINE SYSTEMS INC
Rob Zalensky

- Controlling of single block stop and waiting for the auxiliary function completion signal #3003 (Attribute: R/W)

Assigning the following values in system variable #3003 allows the specification of whether single block stop is disabled in the following blocks or whether a wait for the completion signal (FIN) of the auxiliary function (M, S, T, or B) before going to the next block is enabled. When a wait for completion signal is disabled, the distribution end signal (DEN) is not sent. Be careful not to specify the next auxiliary function without waiting for the completion signal.

Variable number and variable name	Value	Single block stop	Auxiliary function completion signal
#3003 [#_CNTL1]	0	Enabled	Waiting
	1	Disabled	Waiting
	2	Enabled	Not waiting
	3	Disabled	Not waiting

In addition, the following variable names can be used to enable or disable single block stop and a wait for the auxiliary function completion signal, individually.

Variable name	Value	Single block stop	Auxiliary function completion
[#_M_SBK]	0	Enabled	-
	1	Disabled	-
[#_M_FIN]	0	-	Waiting
	1	-	Not waiting

[Example]

Drill cycle (for incremental programming)
(G81 equivalent)
Macro calling command
G65 P9081 L Iterations R R point Z Z point;
A custom macro body is created as follows.

```
O9081 ;
#3003 = 1 ;
G00 Z#18 ;
G01 Z#26 ;
G00 Z-[ ROUND[#18] + ROUND[#26] ] ;
#3003 = 0 ;
M99 ;
```

Disable single block stop.
#18 corresponds to R and #26 to Z.

NOTE
#3003 is cleared by a reset.

- Enabling of feed hold, feedrate override, and exact stop check #3004 (Attribute: R/W)

Assigning the following values in system variable #3004 allows the specification of whether feed hold and feedrate override are enabled in the following blocks or whether exact stop in G61 mode or by G09 command is disabled.

Variable number and variable name	Value	Field hold	Feedrate override	Exact stop
#3004 [#_CNTL2]	0	Enabled	Enabled	Enabled
	1	Disabled	Enabled	Enabled
	2	Enabled	Disabled	Enabled
	3	Disabled	Disabled	Enabled
	4	Enabled	Enabled	Disabled
	5	Disabled	Enabled	Disabled
	6	Enabled	Disabled	Disabled
	7	Disabled	Disabled	Disabled

FOKER MACHINE SYSTEMS NO
Rob Zahensky

In addition, the following variable names can be used to enable or disable feed hold, feedrate override, and exact stop in G61 mode or by the G09 command, individually.

Variable number and variable name	Value	Feed hold	Feedrate override	Exact stop
[#_M_FHD]	0	Enabled	-	-
	1	Disabled	-	-
[#_M_OV]	0	-	Enabled	-
	1	-	Disabled	-
[#_M_EST]	0	-	-	Enabled
	1	-	-	Disabled

NOTE

1 These system variables are provided to maintain compatibility with conventional NC programs. It is recommended that functions provided by G63, G09, G61, and other G codes be used to enable or disable feed hold, feedrate override, and exact stop.

2 When the feed hold button is pressed during execution of a block for which feed hold is disabled:

<1> If the feed hold button is kept pressed, operation stops after execution of the block. When single block stop is disabled, however, operation does not stop.

<2> If the pressed feed hold button is released, the feed hold lamp lights, but operation does not stop until the end of the first block that was enabled.

3 #3004 is cleared by a reset.

4 If exact stop is disabled by #3004, the original exact stop position between cutting feed and positioning block is not affected. #3004 can temporarily disable exact stop in G61 mode or by the G09 command between cutting feed and cutting feed.

- Settings #3005 (Attribute: R/W)

Settings can be read and written.
Binary values are converted to decimals.

#3005

	#15	#14	#13	#12	#11	#10	#9	#8
Setting							FCV	
	#7	#6	#5	#4	#3	#2	#1	#0
Setting			SEQ			INI	ISO	TVC

#9 (FCV) : Whether to use the FANUC Series 15 program format conversion capability
#5 (SEQ) : Whether to automatically insert sequence numbers
#2 (INI) : Millimeter input or inch input
#1 (ISO) : Whether to use EIA or ISO as the output code
#0 (TVC) : Whether to make a TV check

- Stop with a message #3006 (Attribute: W)

When "#3006=1 (MESSAGE);" is commanded in the macro, the program executes blocks up to the immediately previous one and then stops. When a message of up to 60 characters with alphabet and numerals, which is enclosed by a control-in character and control-out character, is programmed in the same block, the message is displayed on the external operator message screen.

Variable number	Variable name	Description
#3006	[#_MSGSTP]	Stop with a message

FRYER MACHINE SYSTEMS INC Rob Zahorsky

When #3006 is commanded with the comment section, the order of message and the comment section changes places by setting value of bit 3 (NCM) of parameter No.6020.
When bit 3 (NCM) of parameter No.6020 is set to 0
It is assumed that it message first, and assumes since the second to be comment section.
(Example) #3000 =1 (MACRO MESSAGE) (COMMENT 1) (COMMENT 2);
When bit 3 (NCM) of parameter No.6020 is set to 1
The end is assumed to be message, and it assumes to be comment section from it ahead.
(Example) #3000 =1 (COMMENT 1) (COMMENT 2) (MACRO MESSAGE);

NOTE
If the single block stop is disabled by #3003=1 command, the program does not stop even if #3006=1 is commanded. However, the message commanded by #3006=1 is displayed on the external operator message screen.

- Status of a mirror image #3007 (Attribute: R)

The status of an mirror image (setting or DI) at that point in time can be obtained for each axis by reading #3007.

Variable number	Variable name	Description
#3007	[#_MRIMG]	Status of a mirror image

When the status is indicated in binary, each bit corresponds with an axis as follows.

Bit	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16
nth axis	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17

Bit	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
nth axis	16	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1

For the 32 bits, 0 indicates that a mirror image is disabled and 1 indicates that a mirror image is enabled.
[Example] When #3007 is 3, a mirror image is enabled for the 1st and 2nd axes.

NOTE
1 The status of a programmable mirror image is not reflected on this variable.
2 When the mirror image function is set for the same axis by the mirror image signal and setting, the signal value and setting value are ORed and then output.
3 When mirror image signals for axes other than the controlled axes are turned on, they are not read into system variable #3007.

- Status during restart of a program #3008 (Attribute: R)

Whether a program is restarting can be determined by reading #3008.

Variable number	Variable name	Description
#3008	[#_PRSTR]	0: Program is not restarting. 1: Program is restarting.

- Time #3011, #3012 (Attribute: R)

Year/month/date and hour/minute/second can be obtained by reading system variables #3011 and #3012. This variable is read-only. To change year/month/date and hour/minute/second, use the timer screen.

[Example] May 20, 2004, PM 04:17:05
#3011 = 20040520
#3012 = 161705

FRYER MACHINE SYSTEMS INC Rob Zahensky

- Path number of the parameter to be read or written #3018 (Attribute: R/W)

If a parameter for another path is to be read or written using parameter reading with the operation command PRM or using parameter writing with programmable parameter input (G10L52), this system variable is used to specify that path.
Relationships between specified values and path numbers are as follows:

#3018	Selected path
0	Local path
1	Path 1
2	Path 2
3	Path 3
4	Path 4

Variable number	Variable name	Description
#3018	–	Path number of the parameter to be read or written

- Number of the path on which a macro is being executed #3019 (Attribute: R)

This system variable can be used to read the number of the path on which a macro is being executed.
By using this variable, it is possible to execute a single program differently for different paths.
The path number can be read as follows:

Path	#3019
Path 1	1
Path 2	2
Path 3	3
Path 4	4

Variable number	Variable name	Description
#3019	[#_PATH_NO]	Path number

- Total number of parts and the number of required parts #3901 and #3902 (Attribute: R/W)

The number of required parts and the number of machined parts can be displayed on the screen by using the operation time and part number displaying function. When the (total) number of machined parts reaches the number of required parts, a signal indicating the fact is sent to the machine (PMC side).
The system variables can be used to read or write the total number of parts and the number of required parts.

Variable number	Variable name	Description
#3901	[#_PRTSA]	Total number of parts
#3902	[#_PRTSN]	Number of required parts

- Type of tool compensation memory #3980 (Attribute: R)

M

System variable #3980 can be used to read the type of compensation memory.

Variable number	Variable name	Description
#3980	[#_OFSMEM]	Types of tool compensation memory 0: Tool compensation memory A 2: Tool compensation memory C

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Main program number #4000 (Attribute: R)

System variable #4000 can be used to read the main program number regardless of the level of a subprogram.

Variable number	Variable name	Description
#4000	[#_MAINO]	Main program number

NOTE

1 The main program number indicates the number of the program that is first started.

2 When an O number is specified by MDI during execution of the main program or when the second O number is specified in DNC mode, the value of #4000 changes to the specified O number. In addition, when no programs are registered or when no O numbers are specified in DNC mode, the value of #4000 changes to 0.

- Modal information #4001-#4130, #4201-#4330, #4401-#4530 (Attribute: R)

The modal information specified before the previous block of the macro statement that reads system variables #4001 to #4130 can be obtained in the block currently being looked ahead, by reading system variables #4001 to #4130.

The modal information of the block currently being executed can be obtained by reading system variables #4201 to #4330.

The modal information specified before the block interrupted by an interruption type custom macro by reading system variables #4401 to #4530.

The unit used when it was specified is applied.

M

(Category: <1> Previous block, <2> Running block, <3> Interrupted block)

Category	Variable number	Variable name	Description
<1> <2> <3>	#4001 #4201 #4401	[#_BUFG[1]] [#_ACTG[1]] [#_INTG[1]]	Modal information (G code: group 1)
<1> <2> <3>	#4002 #4202 #4402	[#_BUFG[2]] [#_ACTG[2]] [#_INTG[2]]	Modal information (G code: group 2)
:	:	:	:
<1> <2> <3>	#4030 #4230 #4430	[#_BUFG[30]] [#_ACTG[30]] [#_INTG[30]]	Modal information (G code: group 30)
<1> <2> <3>	#4102 #4302 #4502	[#_BUFB] [#_ACTB] [#_INTB]	Modal information (B code)
<1> <2> <3>	#4107 #4307 #4507	[#_BUFD] [#_ACTD] [#_INTD]	Modal information (D code)
<1> <2> <3>	#4108 #4308 #4508	[#_BUFE] [#_ACTE] [#_INTE]	Modal information (E code)
<1> <2> <3>	#4109 #4309 #4509	[#_BUFF] [#_ACTF] [#_INTF]	Modal information (F code)
<1> <2> <3>	#4111 #4311 #4511	[#_BUFH] [#_ACTH] [#_INTH]	Modal information (H code)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Category	Variable number	Variable name	Description
<1> <2> <3>	#4113 #4313 #4513	[#_BUFM] [#_ACTM] [#_INTM]	Modal information (M code)
<1> <2> <3>	#4114 #4314 #4514	[#_BUFN] [#_ACTN] [#_INTN]	Modal information (sequence number N)
<1> <2> <3>	#4115 #4315 #4515	[#_BUFO] [#_ACTO] [#_INTO]	Modal information (sequence number O)
<1> <2> <3>	#4119 #4319 #4519	[#_BUFS] [#_ACTS] [#_INTS]	Modal information (S code)
<1> <2> <3>	#4120 #4320 #4520	[#_BUFT] [#_ACTT] [#_INTT]	Modal information (T code)
<1> <2> <3>	#4130 #4330 #4530	[#_BUFWZP] [#_ACTWZP] [#_INTWZP]	Modal information (additional workpiece coordinate system number P)

T

(Category: <1> Previous block, <2> Running block, <3> Interrupted block)

Category	Variable number	Variable name	Description
<1> <2> <3>	#4001 #4201 #4401	[#_BUFG[1]] [#_ACTG[1]] [#_INTG[1]]	Modal information (G code: group 1)
<1> <2> <3>	#4002 #4202 #4402	[#_BUFG[2]] [#_ACTG[2]] [#_INTG[2]]	Modal information (G code: group 2)
:	:	:	:
<1> <2> <3>	#4030 #4230 #4430	[#_BUFG[30]] [#_ACTG[30]] [#_INTG[30]]	Modal information (G code: group 30)
<1> <2> <3>	#4102 #4302 #4502	[#_BUFB] [#_ACTB] [#_INTB]	Modal information (B code)
<1> <2> <3>	#4108 #4308 #4508	[#_BUFE] [#_ACTE] [#_INTE]	Modal information (E code)
<1> <2> <3>	#4109 #4309 #4509	[#_BUFF] [#_ACTF] [#_INTF]	Modal information (F code)
<1> <2> <3>	#4113 #4313 #4513	[#_BUFM] [#_ACTM] [#_INTM]	Modal information (M code)
<1> <2> <3>	#4114 #4314 #4514	[#_BUFN] [#_ACTN] [#_INTN]	Modal information (sequence number N)
<1> <2> <3>	#4115 #4315 #4515	[#_BUFO] [#_ACTO] [#_INTO]	Modal information (sequence number O)
<1> <2> <3>	#4119 #4319 #4519	[#_BUFS] [#_ACTS] [#_INTS]	Modal information (S code)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Category	Variable number	Variable name	Description
<1> <2> <3>	#4120 #4320 #4520	[#_BUFT] [#_ACTT] [#_INTT]	Modal information (T code)
<1> <2> <3>	#4130 #4330 #4530	[#_BUFWZP] [#_ACTWZP] [#_INTWZP]	Modal information (additional workpiece coordinate system number P)

NOTE

1 Previous block and running block
Since the CNC reads the block that is ahead of the block currently being executed by the machining program, the block being retrieved by the CNC is normally different from that currently being executed. The previous block indicates the block that is ahead of the block being retrieved by the CNC, that is, the block that is ahead of the program block in which #4001 to #4130 are specified.

2 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.

[Example]
```
O1234 ;
N10 G00 G90 X200. Y200. ;
N20 G01 G91 X1000. Y1000. F10. ;
      :
      :
N50 G00 G90 X500. Y500. ;
N60 #1 = #4003 ;
```

Assume that the CNC is currently executing N20. If the CNC retrieved and processed the blocks up to N60 as shown above, the running block is N20 and the previous block is N50. Therefore, group 3 modal information in the running block is G91 and group 3 modal information in the previous block is G90.
When N60 #1 = #4203, #1 = 91.
When N60 #1 = #4003, #1 = 90.

- Position information #5001-#5080, #100001-#100200, #118601-#118700 (Attribute: R)

The end position of the previous block, the specified current position (for the machine coordinate system and workpiece coordinate system), and the skip signal position can be obtained by reading the values of system variables #5001 to #5080 or #100001 to -#100200.
System variable #118601 to #118650 can obtain the specified current position (machine coordinate system) in machine units, this machine coordinate is the value of the specified position without servo delay (servo position deviation) and acceleration / deceleration delay.
System variable #118651 to #118700 can obtain the current position (machine coordinate system) in machine units, this machine coordinate is the value with the servo delay(servo position deviation) and the acceleration / deceleration delay together.

FAYER MACHINE SYSTEMS INC
Rob Zohensky

Variable number	Variable name	Position information	Coordinate system	Tool position/tool length/cutter compensation	Servo delay and acceleration/ deceleration delay	Reading operation during movement
#5001 #5002 : #5020	[#_ABSIO[1]] [#_ABSIO[2]] : [#_ABSIO[20]]	1st axis block end point position 2nd axis block end point position : 20th axis block end point position	Workpiece coordinate system	Not included	Not included	Enabled
#100001 #100002 : #100050	[#_ABSIO[1]] [#_ABSIO[2]] : [#_ABSIO[50]]	1st axis block end point position 2nd axis block end point position : 50th axis block end point position				
#5021 #5022 : #5040	[#_ABSMT[1]] [#_ABSMT[2]] : [#_ABSMT[20]]	1st axis current position 2nd axis current position : 20th axis current position	Machine coordinate system (input unit)	Included	Not included	Disabled
#100051 #100052 : #100100	[#_ABSMT[1]] [#_ABSMT[2]] : [#_ABSMT[50]]	1st axis current position 2nd axis current position : 50th axis current position				
#5041 #5042 : #5060	[#_ABSOT[1]] [#_ABSOT[2]] : [#_ABSOT[20]]	1st axis current position 2nd axis current position : 20th axis current position	Workpiece coordinate system	Included	Not included	Disabled
#100101 #100102 : #100150	[#_ABSOT[1]] [#_ABSOT[2]] : [#_ABSOT[50]]	1st axis current position 2nd axis current position : 50th axis current position				
#5061 #5062 : #5080	[#_ABSKP[1]] [#_ABSKP[2]] : [#_ABSKP[20]]	1st axis skip position 2nd axis skip position : 20th axis skip position	Workpiece coordinate system	Included	Bit 1(SEB) of parameter No.6201	Enabled
#100151 #100152 : #100200	[#_ABSKP[1]] [#_ABSKP[2]] : [#_ABSKP[50]]	1st axis skip position 2nd axis skip position : 50th axis skip position				
#118601 #118602 : #118650	[#_MCNPOS[1]] [#_MCNPOS[2]] : [#_MCNPOS[50]]	1st axis current position 2nd axis current position : 50th axis current position	Machine coordinate system (mahine unit)	Included	Not included	Disabled
#118651 #118652 : #118700	[#_ACTMCN[1]] [#_ACTMCN[2]] : [#_ACTMCN[50]]	1st axis current position 2nd axis current position : 50th axis current position	Machine coordinate system (machine unit)	Included	Included	Enabled

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The position information for 20th or earlier axis can be used with #5001 to #5080.
3 The block end point position (ABSIO) of the skip (G31) is the position where the skip signal is turned on. If the skip signal is not turned on, the position is the end position of the block.
4 "Read operation during movement is disabled" means that the accurate reading of values during movement is not guaranteed.

- Tool length compensation value #5081-#5100, #100201-#100250 (Attribute: R)

M

Tool length compensation in the block currently being executed can be obtained for each axis by reading system variables #5081 to #5100 or #100201 to #100250.

Variable number	Variable name	Position information	Read operation during movement
#5081	[#_TOFS[1]]	1st axis tool length compensation value	Disabled
#5082	[#_TOFS[2]]	2nd axis tool length compensation value	
:	:	:	
#5100	[#_TOFS[20]]	20th axis tool length compensation value	
#100201	[#_TOFS[1]]	1st axis tool length compensation value	
#100202	[#_TOFS[2]]	2nd axis tool length compensation value	
:	:	:	
#100250	[#_TOFS[50]]	50th axis tool length compensation value	

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The tool length compensation for 20th or earlier axis can also be used with #5081 to #5100.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Tool offset #5081-#5100, #5121-#5140 (Attribute: R) #100201-#100250, #100901-#100950

T

Tool offset in the block currently being executed can be obtained for each axis by reading system variables #5081 to #5100 or #5121 to #5140. (X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes)

<1> Without tool geometry/wear compensation memory

Variable number	Variable name	Position information	Read operation during movement
#5081	[#_TOFSWX]	X-axis tool offset value	Disabled
#5082	[#_TOFSWZ]	Z-axis tool offset value	
#5083	[#_TOFSWY]	Y-axis tool offset value	
#5084	[#_TOFS[4]]	4th axis tool offset value	
:	:	:	
#5100	[#_TOFS[20]]	20th axis tool offset value	
#100201	[#_TOFSWX]	X-axis tool offset value	
#100202	[#_TOFSWZ]	Z-axis tool offset value	
#100203	[#_TOFSWY]	Y-axis tool offset value	
#100204	[#_TOFS[4]]	4th axis tool offset value	
:	:	:	
#100250	[#_TOFS[50]]	50th axis tool offset value	

<2> With tool geometry/wear compensation memory

Variable number	Variable name	Position information	Read operation during movement
#5081	[#_TOFSWX]	X-axis tool offset value (geometry)	Disabled
#5082	[#_TOFSWZ]	Z-axis tool offset value (geometry)	
#5083	[#_TOFSWY]	Y-axis tool offset value (geometry)	
#5084	[#_TOFS[4]]	4th axis tool offset value (geometry)	
:	:	:	
#5100	[#_TOFS[20]]	20th axis tool offset value (geometry)	
#100201	[#_TOFSWX]	X-axis tool offset value (geometry)	
#100202	[#_TOFSWZ]	Z-axis tool offset value (geometry)	
#100203	[#_TOFSWY]	Y-axis tool offset value (geometry)	
#100204	[#_TOFS[4]]	4th axis tool offset value (geometry)	
:	:	:	
#100250	[#_TOFS[50]]	50th axis tool offset value (geometry)	
#5121	[#_TOFSGX]	X-axis tool offset value (geometry)	Disabled
#5122	[#_TOFSGZ]	Z-axis tool offset value (geometry)	
#5123	[#_TOFSGY]	Y-axis tool offset value (geometry)	
#5124	[#_TOFSG[4]]	4th axis tool offset value (geometry)	
:	:	:	
#5140	[#_TOFSG[20]]	20th axis tool offset value (geometry)	
#100901	[#_TOFSGX]	X-axis tool offset value (geometry)	
#100902	[#_TOFSGZ]	Z-axis tool offset value (geometry)	
#100903	[#_TOFSGY]	Y-axis tool offset value (geometry)	
#100904	[#_TOFSG[4]]	4th axis tool offset value (geometry)	
:	:	:	
#100950	[#_TOFSG[50]]	50th axis tool offset value (geometry)	

When tool geometry/wear compensation memory exists, the system variable values vary as follows depending on bit 2 (LWT) of parameter No. 5002 and bit 4 (LGT) of parameter No. 5002.

FRYER MACHINE SYSTEMS INC
Rob Zarensky

Variable number	LWT=0 LGT=0	LWT=1 LGT=0	LWT=0 LGT=1	LWT=1 LGT=1
#5081 : #5100	Wear compensation	0	Wear compensation	Wear compensation
#5121 : #5140	Geometry compensation	Wear compensation + Geometry compensation	Geometry compensation	Geometry compensation

NOTE
1 The set value is read as the tool offset regardless of bit 1 (ORC) of parameter No.5004 and bit 0 (OWD) of parameter No.5040.
2 To read the tool offset (geometry) using #5121 to #5140, set bit 2 (VHD) of parameter No.6004 to 0.

- Servo position deviation #5101-#5120, #100251-#100300 (Attribute: R)

The servo position deviation for each axis can be obtained by reading system variables #5101 to #5120 or #100251 to #100300.

Variable number	Variable name	Position information	Read operation during movement
#5101 #5102 : #5120	[#_SVERR[1]] [#_SVERR[2]] : [#_SVERR[20]]	1st axis servo position deviation 2nd axis servo position deviation : 20th axis servo position deviation	Disabled
#100251 #100252 : #100300	[#_SVERR[1]] [#_SVERR[2]] : [#_SVERR[50]]	1st axis servo position deviation 2nd axis servo position deviation : 50th axis servo position deviation	

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The servo position deviation for 20th or earlier axis can be used with #5101 to #5120.

- Manual handle interruption #5121-#5140, #100651-#100700 (Attribute: R)

The manual handle interruption for each axis can be obtained by reading system variables #5121 to #5140 or #100651 to #100700.

Variable number	Variable name	Position information	Read operation during movement
#5121 #5122 : #5140	[#_MIRTP[1]] [#_MIRTP[2]] : [#_MIRTP[20]]	1st axis manual handle interruption 2nd axis manual handle interruption : 20th axis manual handle interruption	Disabled
#100651 #100652 : #100700	[#_MIRTP[1]] [#_MIRTP[2]] : [#_MIRTP[50]]	1st axis manual handle interruption 2nd axis manual handle interruption : 50th axis manual handle interruption	

FOXER MACHINE SYSTEMS INC
Rob Zahensky

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The manual handle interruption for 20th or earlier axis can be used with #5121 to #5140.

T

NOTE
#5121 to #5140 are enabled only when bit 2 (VHD) of parameter No.6004 is set to 1.

- Distance to go #5181-#5200, #100801-#100850 (Attribute: R)

The distance to go value for each axis can be obtained by reading system variables #5181 to #5200 or #100801 to #100850.

Variable number	Variable name	Position information	Read operation during movement
#5181 #5182 : #5200	[#_DIST[1]] [#_DIST[2]] : [#_DIST[20]]	1st axis distance to go value 2nd axis distance to go value : 20th axis distance to go value	Disabled
#100801 #100802 : #100850	[#_DIST[1]] [#_DIST[2]] : [#_DIST[50]]	1st axis distance to go value 2nd axis distance to go value : 50th axis distance to go value	

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The distance to go values for 20th or earlier axis can be used with #5181 to #5200.

- Workpiece coordinate system shift value #2501, #2601, #100751-#100800 (Attribute: R/W)

T

The X-axis workpiece coordinate system shift value can be obtained by system variable #2501 while Z-axis workpiece coordinate system shift value by system variable #2601. Each workpiece coordinate system shift value can be obtained by system variable #100751-#100800. The X-axis or Z-axis workpiece coordinate system shift value can be changed by assigning values to the system variables. (X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes)

Variable number	Variable name	Description
#2501	[#_WZ_SFTX]	X-axis workpiece shift value
#2601	[#_WZ_SFTZ]	Z-axis workpiece shift value
#100751 #100752 : #100800	[#_WZ_SFT[1]] [#_WZ_SFT [2]] : [#_WZ_SFT [50]]	1st axis workpiece shift value 2nd axis workpiece shift value : 50th axis workpiece shift value

FIXER MACHINE SYSTEMS INC
Rob Zahensky

NOTE
1 The value of a variable with a number greater than the number of controlled axes is undefined.
2 The X-axis of three basic axes can also be used in system variable #2501, and the Z-axis of three basic axes can also be used in system variable #2601.

- Workpiece origin offset value #5201-#5340, #100301-#100650 (Attribute: R/W)

The workpiece origin offset value can be obtained by reading system variables #5201 to #5340 or #100301 to #100650. The offset value can also be changed by assigning values to the system variables.

Variable number	Variable name	Controlled axis	Workpiece coordinate system
#5201 #5202 : #5220	[#_WZCMN[1]] [#_WZCMN[2]] : [#_WZCMN[20]]	1st axis external workpiece origin offset value 2nd axis external workpiece origin offset value : 20th axis external workpiece origin offset value	External workpiece origin offset value (applied to all coordinate systems)
#5221 #5222 : #5240	[#_WZG54[1]] [#_WZG54[2]] : [#_WZG54[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	G54
#5241 #5242 : #5260	[#_WZG55[1]] [#_WZG55[2]] : [#_WZG55[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	G55
#5261 #5262 : #5280	[#_WZG56[1]] [#_WZG56[2]] : [#_WZG56[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	G56
#5281 #5282 : #5300	[#_WZG57[1]] [#_WZG57[2]] : [#_WZG57[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	G57
#5301 #5302 : #5320	[#_WZG58[1]] [#_WZG58[2]] : [#_WZG58[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	G58
#5321 #5322 : #5340	[#_WZG59[1]] [#_WZG59[2]] : [#_WZG59[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	G59

Variable number	Variable name	Controlled axis	Workpiece coordinate system
#100301 #100302 : #100350	[#_WZCMN[1]] [#_WZCMN[2]] : [#_WZCMN[50]]	1st axis external workpiece origin offset value 2nd axis external workpiece origin offset value : 50th axis external workpiece origin offset value	External workpiece origin offset value (applied to all coordinate systems)
#100351 #100352 : #100400	[#_WZG54[1]] [#_WZG54[2]] : [#_WZG54[50]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 50th axis workpiece origin offset value	G54
#100401 #100402 : #100450	[#_WZG55[1]] [#_WZG55[2]] : [#_WZG55[50]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 50th axis workpiece origin offset value	G55

FRYER MACHINE SYSTEMS INC
Rod Zalensky

Variable number	Variable name	Controlled axis	Workpiece coordinate system
#100451 #100452 : #100500	[#_WZG56[1]] [#_WZG56[2]] : [#_WZG56[50]]	1st axis workpiece origin offset 2nd axis workpiece origin offset : 50th axis workpiece origin offset	G56
#100501 #100502 : #100550	[#_WZG57[1]] [#_WZG57[2]] : [#_WZG57[50]]	1st axis workpiece origin offset 2nd axis workpiece origin offset : 50th axis workpiece origin offset	G57
#100551 #100552 : #100600	[#_WZG58[1]] [#_WZG58[2]] : [#_WZG58[50]]	1st axis workpiece origin offset 2nd axis workpiece origin offset : 50th axis workpiece origin offset	G58
#100601 #100602 : #100650	[#_WZG59[1]] [#_WZG59[2]] : [#_WZG59[50]]	1st axis workpiece origin offset 2nd axis workpiece origin offset : 50th axis workpiece origin offset	G59

M

The following variables can also be used when bit 5 (D15) of parameter No. 6004 is set to 0:

Axis	Function	Variable number
1st axis	External workpiece origin offset value G54 workpiece origin offset value G55 workpiece origin offset value G56 workpiece origin offset value G57 workpiece origin offset value G58 workpiece origin offset value G59 workpiece origin offset value	#2500 #2501 #2502 #2503 #2504 #2505 #2506
2nd axis	External workpiece origin offset value G54 workpiece origin offset value G55 workpiece origin offset value G56 workpiece origin offset value G57 workpiece origin offset value G58 workpiece origin offset value G59 workpiece origin offset value	#2600 #2601 #2602 #2603 #2604 #2605 #2606
3rd axis	External workpiece origin offset value G54 workpiece origin offset value G55 workpiece origin offset value G56 workpiece origin offset value G57 workpiece origin offset value G58 workpiece origin offset value G59 workpiece origin offset value	#2700 #2701 #2702 #2703 #2704 #2705 #2706
4th axis	External workpiece origin offset value G54 workpiece origin offset value G55 workpiece origin offset value G56 workpiece origin offset value G57 workpiece origin offset value G58 workpiece origin offset value G59 workpiece origin offset value	#2800 #2801 #2802 #2803 #2804 #2805 #2806

FRYER MACHINE SYSTEMS INC
Rob Zahensky

T

The following variables can be used to maintain compatibility with conventional models.

Axis	Function	Variable number
1st axis	External workpiece origin offset value	#2550
	G54 workpiece origin offset value	#2551
	G55 workpiece origin offset value	#2552
	G56 workpiece origin offset value	#2553
	G57 workpiece origin offset value	#2554
	G58 workpiece origin offset value	#2555
	G59 workpiece origin offset value	#2556
2nd axis	External workpiece origin offset value	#2650
	G54 workpiece origin offset value	#2651
	G55 workpiece origin offset value	#2652
	G56 workpiece origin offset value	#2653
	G57 workpiece origin offset value	#2654
	G58 workpiece origin offset value	#2655
	G59 workpiece origin offset value	#2656
3rd axis	External workpiece origin offset value	#2750
	G54 workpiece origin offset value	#2751
	G55 workpiece origin offset value	#2752
	G56 workpiece origin offset value	#2753
	G57 workpiece origin offset value	#2754
	G58 workpiece origin offset value	#2755
	G59 workpiece origin offset value	#2756
4th axis	External workpiece origin offset value	#2850
	G54 workpiece origin offset value	#2851
	G55 workpiece origin offset value	#2852
	G56 workpiece origin offset value	#2853
	G57 workpiece origin offset value	#2854
	G58 workpiece origin offset value	#2855
	G59 workpiece origin offset value	#2856

NOTE

1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.

2 The workpiece origin offset value for 20th or earlier axis can be used with #5201 to #5340.

M

NOTE

To use variables #2500 to #2806, #5201 to #5340, and #100301 to #100650, setting that bit 0 (NWA) of parameter No. 8136 is 0 is necessary.

T

NOTE

To use variables #2550 to #2856, #5201 to #5340, and #100301 to #100650, setting that bit 0 (NWA) of parameter No. 8136 is 0 is necessary.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Workpiece origin offset value of the additional workpiece coordinate system #7001-#7960, #101001-#116000 (Attribute: R/W)

M

#14001-#20000 (Attribute: R/W)

The workpiece origin offset value of the additional workpiece coordinate system can be obtained by reading system variables #7001 to #7960, #14001 to #20000, #101001 to #116000. The offset value can also be changed by assigning values to the system variables.

Variable number	Variable name	Controlled axis	Additional workpiece system number
#7001 #7002 : #7020	[#_WZP1[1]] [#_WZP1[2]] : [#_WZP1[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	1 (G54.1 P1)
#7021 #7022 : #7040	[#_WZP2[1]] [#_WZP2[2]] : [#_WZP2[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	2 (G54.1 P2)
#7041 #7042 : #7060	[#_WZP3[1]] [#_WZP3[2]] : [#_WZP3[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	3 (G54.1 P3)
:	:	:	:
#7941 #7942 : #7960	[#_WZP48[1]] [#_WZP48[2]] : [#_WZP48[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	48 (G54.1 P48)

System variable number = 7000 + (Coordinate system number -1) × 20 + Axis number
Coordinate number: 1 to 48
Axis number: 1 to 20

M

Variable number	Variable name	Controlled axis	Additional workpiece system number
#14001 #14002 : #14020	[#_WZP1[1]] [#_WZP1[2]] : [#_WZP1[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	1 (G54.1 P1)
#14021 #14022 : #14040	[#_WZP2[1]] [#_WZP2[2]] : [#_WZP2[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	2 (G54.1 P2)
#14041 #14042 : #14060	[#_WZP3[1]] [#_WZP3[2]] : [#_WZP3[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	3 (G54.1 P3)
:	:	:	:
#19981 #19982 : #20000	[#_WZP300[1]] [#_WZP300[2]] : [#_WZP300[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	300 (G54.1 P300)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

System variable number = 14000 + (Coordinate system number -1) × 20 + Axis number
Coordinate number: 1 to 300
Axis number: 1 to 20

Variable number	Variable name	Controlled axis	Additional workpiece system number
#101001 #101002 : #101050	[#_WZP1[1]] [#_WZP1[2]] : [#_WZP1[50]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 50th axis workpiece origin offset value	1 (G54.1 P1)
#101051 #101052 : #101100	[#_WZP2[1]] [#_WZP2[2]] : [#_WZP2[50]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 50th axis workpiece origin offset value	2 (G54.1 P2)
#101101 #101102 : #101150	[#_WZP3[1]] [#_WZP3[2]] : [#_WZP3[50]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 50th axis workpiece origin offset value	3 (G54.1 P3)
:	:	:	:
#115951 #115952 : #116000	[#_WZP300[1]] [#_WZP300[2]] : [#_WZP300[50]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 50th axis workpiece origin offset value	300 (G54.1 P300)

System variable number = 101000 + (Coordinate system number -1) × 50 + Axis number
Coordinate number: 1 to 300
Axis number: 1 to 50

M

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The workpiece origin offset of additional workpiece coordinate system for 20th or earlier axis can be used with #7001 to #7960 or #14001 to #20000.
3 To use #7001 to #7960 (G54.1 P1 to G54.1 P48), Addition of workpiece coordinate system pair (48 pairs) (bits 0 (NWZ) and 2 (NWN) of parameter No. 8136) is required. To use #14001 to #20000, and #101001 to #116000 (G54.1 P1 to G54.1 P300), Addition of workpiece coordinate system pair (300 pairs) is required. In this case, #7001 to #7960 can also be used.

T

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 To use #7001 to #7960 (G54.1 P1 to G54.1 P48), Addition of workpiece coordinate system pair (48 pairs) (bits 0 (NWZ) and 2 (NWN) of parameter No. 8136) is required.
3 To use #101001 to #116000 (G54.1 P1 to G54.1 P300), Addition of workpiece coordinate system pair (300 pairs) is required. In this case, #7001 to #7960 can also be used.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Skip position (detection unit) #5421-#5440, #100701-#100750 (Attribute: R)

The skip position with the detection unit can be obtained by reading system variables #5421 to #5440 or #100701 to #100750. The values of these variables are integers, and values in the workpiece coordinate system.

Variable number	Variable name	Position information	Read operation during movement
#5421 #5422 : #5440	[#_SKPDTC[1]] [#_SKPDTC[2]] : [#_SKPDTC[20]]	1st axis skip position (detection unit) 2nd axis skip position (detection unit) : 20th axis skip position (detection unit)	Disabled
#100701 #100702 : #100750	[#_SKPDTC[1]] [#_SKPDTC[2]] : [#_SKPDTC[50]]	1st axis skip position (detection unit) 2nd axis skip position (detection unit) : 50th axis skip position (detection unit)	

NOTE

1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The skip position (detection unit) for 20th or earlier axis can be used with #5421 to #5440.
3 To specify these variables, set bit 2 (DSK) of parameter No. 6210 to 1. (If an attempt is made to specify them when DSK is 0, alarm PS0115 is issued.)

- Second geometry tool offset value #5801-#5896, #27001-#29999 (Attribute: R/W)

T

By reading the values of system variables #5801 to #5896 and #27001 to #29999, it is possible to determine the second geometry tool offset value, and by assigning values to the system variables, it is possible to change the second geometry tool offset value.

- For compensation numbers up to 32

Compensation number	Variable number	Variable name	Attribute	Description
1	#5801	[#_OFSX2G[1]]	R/W	Second geometry tool offset X-axis compensation value
:	:	:	:	
32	#5832	[#_OFSX2G[32]]	R/W	
1	#5833	[#_OFSZ2G[1]]	R/W	Second geometry tool offset Z-axis compensation value
:	:	:	:	
32	#5864	[#_OFSZ2G[32]]	R/W	
1	#5865	[#_OFSY2G[1]]	R/W	Second geometry tool offset Y-axis compensation value
:	:	:	:	
32	#5896	[#_OFSY2G[32]]	R/W	

- For compensation numbers greater than 32 (Compensation values with compensation numbers of up to 32 can also be used in variables #5801 to #5896.)

Compensation number	Variable number	Variable name	Attribute	Description
1	#27001	[#_OFSX2G[1]]	R/W	Second geometry tool offset X-axis compensation value
:	:	:	:	
999	#27999	[#_OFSX2G[999]]	R/W	
1	#28001	[#_OFSZ2G[1]]	R/W	Second geometry tool offset Z-axis compensation value
:	:	:	:	
999	#28999	[#_OFSZ2G[999]]	R/W	
1	#29001	[#_OFSY2G[1]]	R/W	Second geometry tool offset Y-axis compensation value
:	:	:	:	

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Compensation number	Variable number	Variable name	Attribute	Description
999	#29999	[#_OFSY2G[999]]	R/W	

- Cartridge number of the tool management function #8400 (Attribute: R/W)

By specifying the cartridge number of a spindle position (11 to 14) or standby position (21 to 24) in #8400, information about the corresponding position can be read. For details of the tool management function, refer to subsection "TOOL MANAGEMENT FUNCTION".

Variable number	Variable name	Description
#8400	-	Cartridge number of the tool management function

- Tool management data #8401-#8414, #8431-#8470 (Attribute: R)

By reading system variables #8401 to #8414 and #8431 to #8470, it is possible to determine the tool management data about the position specified in #8400. For details of the tool management function, refer to subsection "TOOL MANAGEMENT FUNCTION".

M

Variable number	Variable name	Description
#8401	-	Data number
#8402	-	Tool type number (T)
#8403	-	Tool life counter
#8404	-	Maximum tool life value
#8405	-	Tool notice life value
#8406	-	Tool life status
#8407	-	Customize data 0 (bit)
#8408	-	Tool information
#8409	-	Tool length compensation number (H)
#8410	-	Cutter compensation number (D)
#8411	-	Spindle speed (S)
#8412	-	Cutting feedrate (F)
#8431 : #8470	- : -	Customize data 1 : Customize data 40

T

Variable number	Variable name	Description
#8401	-	Data number
#8402	-	Tool type number (T)
#8403	-	Tool life counter
#8404	-	Maximum tool life value
#8405	-	Tool notice life value
#8406	-	Tool life status
#8407	-	Customize data 0 (bit)
#8408	-	Tool information
#8411	-	Spindle speed (S)
#8412	-	Cutting feedrate (F)
#8413	-	Tool geometry compensation number (G)
#8414	-	Tool wear compensation number (W)
#8431 : #8470	- : -	Customize data 1 : Customize data 40

FAYER MACHINE SYSTEMS INC
Rob Zaremsky

- Feedrate reduction ratio for rapid traverse overlap #100851-#100900 (Attribute: R/W)

The feedrate reduction ratio for rapid traverse overlap can also be changed by setting values to the system variables #100851 to #100900.

Variable number	Variable name	Controlled axis
#100851	[#_ROVLP[1]]	1st axis feedrate reduction ratio for rapid traverse overlap
#100852	[#_ROVLP[2]]	2nd axis feedrate reduction ratio for rapid traverse overlap
:	:	:
#100900	[#_ROVLP[50]]	50th axis feedrate reduction ratio for rapid traverse overlap

- States of serial spindles #100951 to #100954 (Attribute: R)

Using system variables #100951 to #100954, it is possible to determine the states of serial spindles. The relationships between the values of the variables and the states of spindles are as follows:

1 : Normal operation
2 : Orientation
3 : Synchronous control
4 : Rigid tapping
5 : Cs contour control

Variable number	Variable name	Description
#100951	[#_SPSTAT[1]]	State of the first spindle on the path
#100952	[#_SPSTAT[2]]	State of the second spindle on the path
#100953	[#_SPSTAT[3]]	State of the third spindle on the path
#100954	[#_SPSTAT[4]]	State of the fourth spindle on the path

- Clamp value of spindle maximum speed #100959 (Attribute: R)

Using system variables #100959, it is possible to obtain the spindle maximum speed commanded at clamp command of maximum spindle speed. In the following case, #100959 is -1.

- Clamp command of maximum spindle speed G92 (G50 in G code system A of T series) has not been executed after turning on the power.

Variable number	Variable name	Description
#100959	[#_CSSSMAX]	Spindle maximum speed commanded at clamp command of maximum spindle speed

- Dynamic reference tool compensation value #118051-#118450 (Attribute: R/W)

M

The dynamic reference tool compensation value in the rotary head dynamic tool compensation function can be obtained by reading system variables #118051 to #118450. The dynamic reference tool compensation value can also be obtained by assigning values to the system variables.

Variable number	Variable name	Controlled axis	Dynamic tool offset number
#118051	[#_DOFS1[1]]	1st axis dynamic reference tool compensation value	1 (G43.2H1)
#118052	[#_DOFS1[2]]	2nd axis dynamic reference tool compensation value	
:	:	:	
#118100	[#_DOFS1[50]]	50th axis dynamic reference tool compensation value	
#118101	[#_DOFS2[1]]	1st axis dynamic reference tool compensation value	2 (G43.2H2)
#118102	[#_DOFS2[2]]	2nd axis dynamic reference tool compensation value	
:	:	:	
#118150	[#_DOFS2[50]]	50th axis dynamic reference tool compensation value	
#118151	[#_DOFS3[1]]	1st axis dynamic reference tool compensation value	3 (G43.2H3)
#118152	[#_DOFS3[2]]	2nd axis dynamic reference tool compensation value	
:	:	:	
#118200	[#_DOFS3[50]]	50th axis dynamic reference tool compensation value	

FRYER MACHINE SYSTEMS INC. Rob Zahensky

Variable number	Variable name	Controlled axis	Dynamic tool offset number
#118201 #118202 : #118250	[#_DOFS4[1]] [#_DOFS4[2]] : [#_DOFS4[50]]	1st axis dynamic reference tool compensation value 2nd axis dynamic reference tool compensation value : 50th axis dynamic reference tool compensation value	4 (G43.2H4)
#118251 #118252 : #118300	[#_DOFS5[1]] [#_DOFS5[2]] : [#_DOFS5[50]]	1st axis dynamic reference tool compensation value 2nd axis dynamic reference tool compensation value : 50th axis dynamic reference tool compensation value	5 (G43.2H5)
:	:	:	:
#118401 #118402 : #118450	[#_DOFS8[1]] [#_DOFS8[2]] : [#_DOFS8[50]]	1st axis dynamic reference tool compensation value 2nd axis dynamic reference tool compensation value : 50th axis dynamic reference tool compensation value	8 (G43.2H8)

NOTE
When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.

- Local coordinate system offset value #118501-#118550 (Attribute: R)

Using system variables #118501 to #118550, it is possible to determine the local coordinate system (G52) offset value.

Variable number	Variable name	Description
#118501 #118502 : #118550	[#_LCLOFS[1]] [#_LCLOFS[2]] : [#_LCLOFS[50]]	1st axis local coordinate system offset value 2nd axis local coordinate system offset value : 50th axis local coordinate system offset value

- Torque difference of Axis synchronous #180801-#180850 (Attribute: R)

Using system variables #180801 to #180850, it is possible to determine the torque difference of master axis and slave axis in Axis synchronous control. Parameter (No.8311) determines the combination of master axis and slave axis, and torque difference of master axis and slave axis is output to system variable of slave axis during synchronous operation and normal operation. The output torque difference is the value obtained by subtracting torque value of slave axis from the torque value of master axis.
System value is read “0” in master axis. The case of axis more than the number of axis in a path is read “0” as with master axis.

Variable number	Variable name	Description
#180801 #180802 : #180850	[#_SYNTD[1]] [#_SYNTD [2]] : [#_SYNTD [50]]	1st axis torque difference of Axis synchronous 2nd axis torque difference of Axis synchronous : 50th axis torque difference of Axis synchronous

NOTE
When value of parameter (No.8311) is incorrect, data of system variable cannot be read and becomes alarm (PS5580) “CANNOT GET SYSTEM VARIABLE”.

FRYER MACHINE SYSTEMS INC Rob Zahensky

- Tool life management #181000-#181002 (Attribute: R)

Using system variables #181000 to #181002, it is possible to determine the information (tool group numbers, tool life values and life counter values of a group being used).

Variable number	Variable name	Description
#181000	[#_TLMGN_C]	Tool life management data (Tool group numbers)
#181001	[#_TLMML_C]	Tool life management data (Tool life values)
#181002	[#_TLMLC_C]	Tool life management data (Life counter values)

- Switching between P-CODE variables and system variables (#10000 or later) #8570 (Attribute: R/W)

This system variable allows read/write operations of P-CODE variables (#10000 to #89999) for the macro executor function. For details on P-CODE variables, refer to the Macro Executor Programming Manual (B-63943EN-2).

System variable #8570 can be used to make variables #10000 or later correspond to either P-CODE variables or system variables.

#8570 setting	Specified variable	Corresponding variable
#8570 = 1	#10000 : #89999	P-CODE variables (#10000) : P-CODE variables (#89999)
#8570 = 0	#10000 : #89999	System variables (#10000) : System variables (#89999)

Example

```
#8570 = 0 ;
#10001 = 123 ; → Writing to system variable #10001 (tool compensation)
#8570 = 1 ;
#10001 = 456 ; → Writing to P-CODE variable #10001 (tool compensation)
```

NOTE

1. Variable #8570 can be used only when the macro executor function is enabled.
2. System variables (#10000 or later) always correspond to system variables specified by their variable names even when #8570 is 1.
3. When an attempt is made to access a variable that cannot be used with P-CODE variables (#10000 or later), an alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.

- Exclusion control between each path #182001-#182010 (Attribute: R/W)

In the custom macro of multi-path, if it is not convenient to operate at the same time in the multi-path, a single path that can be execution on the system variable will be notified.

Exclusive control in a custom macro between each path will be available by this function.

By writing a non-zero to the system variable from #182001 to #182010, execute the acquisition of priority.

If the path was able to get the priority, 1 can be read from the system variable from #182001 to #182010.

If the path has not acquired priority, 0 can be read from the system variable.

By writing a 0 to the system variable from #182001 to #182010, execute the abandon of priority.

By writing a non-zero to the system variable, the following path will be able to get the priority.

FRYER MACHINE SYSTEMS INC
Rob Zahenskyj

NOTE
By writing a NULL to the system variable from #182001 to #182010, It is treated as 0.

The state of the priority does not change at the reset input. If bit 0 (SPR) of parameter No. 6022 is set to 1, it is possible to abandon the priority at the reset.

Variable number	Variable name	Description
#182001 : #182010	[#_PRIO[1]] : [#_PRIO[10]]	System variables for the 1st exclusive control : System variables for the 10th exclusive control

16.3 DISPLAY OF SYSTEM VARIABLES

System variables are displayed on the custom macro screen.

Explanation

System variables of the previous section (#1000-#8499, #10000-#89999, #100000-#199999) can be displayed on the custom macro screen when bit 4 (CSD) of parameter No.11369 is set to 1. The value of system variables which is not used is not displayed.
Refer to the section of "Displaying and Setting Custom Macro Common Variables" for the display of the custom macro screen.

When macro executor is effective, the P-CODE variable can be used. The number of the P-CODE variable overlaps with the number of system variables (#10000-#89999) because #10000-#89999 is used for the P-CODE variable. Therefore, the treatment of the system variable and the P-CODE variable can be changed by system variable #8570. Refer to the previous section for system variable #8570.
When bit 4 (CSD) of parameter No.11369 is 1, the display of #10000-#89999 is switched as follows by setting of #8570.

- In case of #8570=0, system variables are displayed in #10000-#89999.
- In case of #8570=1, P-CODE variables are displayed in #10000-#89999.

NOTE
When macro executor is effective, the macro screen of macro executor is displayed besides the macro screen of the custom macro. (Macro screen of execution macro, conversational macro, auxiliary macro) However, bit 4 (CSD) of parameter No.11369 is invalid on these screens.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

16.4 READING AND WRITING VARIABLES FOR ANOTHER PATH

By adding a path number to the high-order 8th and 9th digits of a variable, it is possible to read and write a common variable or a system variable for another path.
For a list of variables that can be read and written, see "List of variables that can be read and written".

Format

#ppxxxxxxx

pp: Path number
Omitted = Local path
1 = First path
: :
4 = Forth path
xxxxxxx: Variable number

Common variable (#100 to #499(Note), #500 to #999(Note)) or system variable number (1000 and above, 10000 and above, 100000 and above)

NOTE
Available common variables and system variables differ depending on the system configuration.

Example

Machine coordinates for path 1		Machine coordinates for path 2	
X1	123.456	X2	-123.456
Y1	45.670	Z2	78.900
Z1	345.789	C2	45.000

In the case above, if #100=#5023 is executed on path 1, the machine coordinate 345.789 on the third axis on path 1 is read into variable #100.
If #100=#20005023 is executed, the machine coordinate 45.0 on the third axis on path 2 is read into variable #100.

NOTE
1 Variable names cannot be specified.
2 If a path number out of the range is specified, alarm PS0115, “VARIABLE NO. OUT OF RANGE” is issued.
3 Local variables for another path cannot be read or written.
4 If reading or writing variables for another path is executed using a real time custom macro, alarm PS0390, “ILLEGAL MACRO VAR” is issued.

Caution

⚠ **CAUTION**
System variables include those that affect automatic operation (for example, variables #3000 to #3999), and they affect the operation on another path. Use great caution when writing them.

FRYER MACHINE SYSTEMS INC Rob Zahensky

List of variables that can be read and written

This function can read and write the following variables for another path.
For details of system variables, see the description of variables in the previous section.
R, W, and R/W are variable attributes, and respectively represent READ only, WRITE only, and READ/WRITE.

- Common variable

System variable number	Attribute	Description
#100 to #149(#199) #200 to #499 #500 to #549(#999)	R/W	Available common variables differ depending on the system configuration.

- Interface signals

System variable number	Attribute	Description
#1000 to #1035	R	Interface input signals
#1100 to #1135	R/W	Interface output signals

- Tool compensation value

System variable number	Attribute	Description
#2001 to #2964 #10001 to #19999	R/W	Tool compensation value

- Workpiece coordinate system shift amount

T

System variable number	Attribute	Description
#100751 to #100800	R/W	Workpiece coordinate system shift amount

- Automatic operation or the like

System variable number	Attribute	Description
#3001, #3002	R/W	Clock
#3003	R/W	Control for waiting for a single block stop and the auxiliary function completion signal
#3004	R/W	Feed hold, feedrate override, exact stop check
#3005	R/W	Setting data.
#3007	R	Status of a mirror image (DI and setting)
#3008	R	State during a program restart

- Time

System variable number	Attribute	Description
#3011 to #3012	R	Year/Month/Date, Hour/Minute/Second

- Number of parts

System variable number	Attribute	Description
#3901 to #3902	R/W	Total number of parts, Number of required parts

- Tool compensation memory

M

System variable number	Attribute	Description
#3980	R	Tool compensation memory information

FOYER MACHINE SYSTEMS INC
Rob Zahorsky

- Main program number

System variable number	Attribute	Description
#4000	R	Main program number

- Modal information

System variable number	Attribute	Description
#4001 to #4530	R	Modal information

- Position information

System variable number	Attribute	Description
#5001 to #5020 #100001 to #100050	R	End point position of the block (workpiece coordinate system)
#5021 to #5040 #100051 to #100100	R	Specified current position of input unit (machine coordinate system)
#5041 to #5060 #100101 to #100150	R	Specified current position (workpiece coordinate system)
#5061 to #5080 #100151 to #100200	R	Skip position (workpiece coordinate system)
#118601 to #118650	R	Specified current position of machine unit (machine coordinate system). Value of the specified current position without servo delay and acceleration / deceleration delay.
#118651 to #118700	R	Current position of machine unit (machine coordinate system). Value of the current position with servo delay and acceleration / deceleration delay.

- Tool length compensation value

M

System variable number	Attribute	Description
#5081 to #5100 #100201 to #100250	R	Tool length compensation value

- Tool offset value

T

System variable number	Attribute	Description
#5081 to #5083 #100201～#100250	R	Tool offset value (wear)
#5121 to #5123 #100901～#100950	R	Tool offset value (geometry)

- Servo position deviation

System variable number	Attribute	Description
#5101 to #5120 #100251 to #100300	R	Servo positional deviation

- Manual handle interruption

System variable number	Attribute	Description
#5121 to #5140 #100651 to #100700	R	Manual handle interruption

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Distance to go

System variable number	Attribute	Description
#5181 to #5200 #100801 to #100850	R	Distance to go

- Workpiece origin offset value, extended workpiece origin offset value

M

System variable number	Attribute	Description
#5201 to #5340 #100301 to #100650	R/W	Workpiece origin offset value
Extended workpiece origin offset value		
#7001 to #7960 #14001 to #20000 #101001 to #116000	R/W	Workpiece origin offset value

T

System variable number	Attribute	Description
#5201 to #5340 #100301 to #100650	R/W	Workpiece origin offset value
Extended workpiece origin offset value		
#7001 to #7960 #101001 to #116000	R/W	Workpiece origin offset value

- Skip position (detection unit)

System variable number	Attribute	Description
#5421 to #5440 #100701 to #100750	R	Skip position (detection unit)

- Second geometry tool offset value

T

System variable number	Attribute	Description
#27001 to #29999	R/W	Second geometry tool offset value

- Tool life management

System variable number	Attribute	Description
#181000	R	Tool life management data (Tool group numbers)
#181001	R	Tool life management data (Tool life values)
#181002	R	Tool life management data (Life counter values)

- Others (#8570)

System variable number	Attribute	Description
#8570	R/W	Switching of P-CODE variables/system variables (#10000 and above)

FINKER MACHINE SYSTEMS INC
Rob Zarensky

16.5 ARITHMETIC AND LOGIC OPERATION

Various operations can be performed on variables. Program an arithmetic and logic operation in the same way as for a general arithmetic expression.

#i=<expression>

<Expression>

The expression to the right of the arithmetic and logic operation contains constants and/or variables combined by a function or operator. Variables #j and #k below can be replaced with a constant. If a constant used in an expression has no decimal point, it is assumed to end with a decimal point.

Table 16.5 (a) Arithmetic and logic operation

Type of operation	Operation	Description
<1> Definition or replacement	#i=#j	Definition or replacement of a variable
<2> Addition-type operations	#i=#j+#k #i=#j-#k #i=#j OR #k #i=#j XOR #k	Addition Subtraction Logical OR (bit by bit of 32 bits) Exclusive OR (bit by bit of 32 bits)
<3> Multiplication-type operations	#i=#j*#k #i=#j/#k #i=#j AND #k #i=#j MOD #k	Multiplication Division Logical AND (bit by bit of 32 bits) Remainder (A remainder is obtained after #j and #k are rounded to their nearest whole numbers. When #j is a negative value, #i is assumed to be a negative value.)
<4> Functions	#i=SIN[#j] #i=COS[#j] #i=TAN[#j] #i=ASIN[#j] #i=ACOS[#j] #i=ATAN[#j] #i=ATAN[#j]/[#k] #i=ATAN[#j,#k] #i=SQRT[#j] #i=ABS[#j] #i=BIN[#j] #i=BCD[#j] #i=ROUND[#j] #i=FIX[#j] #i=FUP[#j] #i=LN[#j] #i=EXP[#j] #i=POW[#j,#k] #i=ADP[#j] #i=PRM[#j] #i=PRM[#j,#k] #i=PRM[#j]/[#l] #i=PRM[#j,#k]/[#l]	Sine (in degrees) Cosine (in degrees) Tangent (in degrees) Arc sine Arc cosine Arc tangent (one argument), ATN can also be used. Arc tangent (two arguments), ATN can also be used. Arc tangent (two arguments), ATN can also be used. Square root, SQR can also be used. Absolute value Conversion from BCD to binary Conversion from binary to BCD Rounding off, RND can also be used. Rounding down to an integer Rounding up to an integer Natural logarithm Exponent using base e (2.718...) Power (#j to the #kth power) Addition of a decimal point Parameter reading (system common, path, or machine group parameter) Parameter reading (system common, path, or machine group parameter bit number specification) Parameter reading (axis or spindle parameter) Parameter reading (axis or spindle parameter bit number specification)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

- Angle units

The units of angles used with the SIN, COS, ASIN, ACOS, TAN, and ATAN functions are degrees. For example, 90 degrees and 30 minutes is represented as 90.5 degrees.

- ARCSIN #i = ASIN[#j];

- The solution ranges are as indicated below:
  When the bit 0 (NAT) of parameter No. 6004 is set to 0: 270° to 90°
  When the bit 0 (NAT) of parameter No. 6004 is set to 1: -90° to 90°
- When #j is beyond the range of -1 to 1, an alarm PS0119, “ARGUMENT VALUE OUT OF RANGE” is issued.
- A constant can be used instead of the #j variable.

- ARCCOS #i = ACOS[#j];

- The solution ranges from 180° to 0°.
- When #j is beyond the range of -1 to 1, an alarm PS0119, “ARGUMENT VALUE OUT OF RANGE” is issued.
- A constant can be used instead of the #j variable.

- ARCTAN #i = ATAN[#j]/[#k]; (two arguments)

- ATAN[#j,#k] is equivalent to ATAN[#j]/[#k].
- When point (#k,#j) on plane X-Y is given, this function returns the value of the arc tangent for the angle made by the point.
- A constant can be used instead of the #j variable.
- If the #j=0 and #k=0, an alarm PS0119, “ARGUMENT VALUE OUT OF RANGE” is issued.
- The solution ranges are as follows:
  When the bit 0 (NAT) of parameter No. 6004 is set to 0: 0° to 360°
  Example: When #1 = ATAN[-1]/[-1]; is specified, #1 is 225.0.
  When the bit 0 (NAT) of parameter No. 6004 is set to 1: -180° to 180°
  Example: When #1 = ATAN[-1]/[-1]; is specified, #1 is -135.0.

- ARCTAN #i = ATAN[#j]; (one argument)

- When ATAN is specified with one argument, this function returns the main value of arc tangent (-90° ≤ ATAN[#j] ≤ 90°). In other word, this function returns the same value as ATAN in calculator specifications.
- To use this function as the dividend of a division, be sure to enclose it with brackets ([]). If this function is not enclosed, ATAN[#j]/[#k] is assumed.
  Example:
  #100 = [ATAN[1]]/10 ; : Divides ATAN with one argument by 10.
  #100 = ATAN[1]/[10] ; : Executes ATAN with two arguments.
  #100 = ATAN[1]/10 ; : Assumes ATAN with two arguments, but issues an alarm PS1131, “MISSING OPEN BRACKET” because the X coordinate specification is not enclosed with brackets ([]).

- Natural logarithm #i = LN[#j];

- When the antilogarithm (#j) is zero or smaller, an alarm PS0119, “ARGUMENT VALUE OUT OF RANGE” is issued.
- A constant can be used instead of the #j variable.

- Exponential function #i = EXP[#j];

- When the result of the operation overflows, an alarm PS0111, “OVERFLOW :FLOATING” is issued.
- A constant can be used instead of the #j variable.

FRYER MACHINE SYSTEMS INC
Rob Zalenski

- ROUND function

- When the ROUND function is included in an arithmetic or logic operation command, IF statement, or WHILE statement, the ROUND function rounds off at the first decimal place.
  Example:
  When #1=ROUND[#2]; is executed where #2 holds 1.2345, the value of variable #1 is 1.0.
- When the ROUND function is used in NC statement addresses, the ROUND function rounds off the specified value according to the least input increment of the address.
  Example:
  Creation of a drilling program that cuts according to the values of variables #1 and #2, then returns to the original position
  Suppose that the increment system is 1/1000 mm, variable #1 holds 1.2345, and variable #2 holds 2.3456. Then,
  G00 G91 X-#1; Moves 1.235 mm in negative direction.
  G01 X-#2 F300; Moves 2.346 mm in negative direction.
  G00 X[#1+#2]; Since 1.2345 + 2.3456 = 3.5801 in positive direction, the travel distance is 3.580 mm, which does not return the tool to the original position.
  This difference comes from whether addition is performed before or after rounding off. G00X-[ROUND[#1]+ROUND[#2]]; must be specified to return the tool to the original position.

- Add decimal point (ADP) function

- ADP[#n] (n = 1 to 33) can be executed to add a decimal point to an argument passed with no decimal point, in the subprogram.
  Example:
  In the subprogram called with G65 P_X10;, the value of ADP[#24] is a value to which a decimal point is added at its end (that is, 10.). Use this function when you do not want to consider the increment system in the subprogram. When bit 4 (CVA) of parameter No. 6007 is set to 1, however, the ADP function cannot be used because any argument is converted to 0.01 the moment it is passed.

NOTE
For compatibility among programs, it is recommended that the ADP function be not used, and decimal points be added in the argument specification for a macro call.

- Rounding up and down to an integer (FUP and FIX)

With CNC, when the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. Conversely, when the absolute value of the integer produced by an operation on a number is less than the absolute value of the original number, such an operation is referred to as rounding down to an integer. Be particularly careful when handling negative numbers.
Example:
Suppose that #1=1.2 and #2=-1.2.
When #3=FUP[#1] is executed, 2.0 is assigned to #3.
When #3=FIX[#1] is executed, 1.0 is assigned to #3.
When #3=FUP[#2] is executed, -2.0 is assigned to #3.
When #3=FIX[#2] is executed, -1.0 is assigned to #3.

- Parameter reading (PRM)

・ The following formats are available depending on the parameter type and the data to be read.

FLYER MACHINE SYSTEMS INC
Rob Zaborsky

Format	Remarks
#i = PRM[#j];	Format for a system common, path, or machine group parameter.
#i = PRM[#j, #k];	Format for specifying the number of a bit of a system common, path, or machine group parameter.
#i = PRM[#j] /[#l];	Format for an axis or spindle parameter
#i =PRM[#j, #k] /[#l];	Format for specifying the number of a bit of an axis or spindle parameter

- For #j, set a parameter number.
- For #k, set a bit number.
- For #l, set an axis number or a spindle number.

- By specifying a path number, it is possible to read a parameter for that path. There are two ways to specify a path number.
  a) Adding a path number to a parameter number
  b) Specifying a path number using a system variable

  a) Adding a path number to a parameter number
  By adding a path number to the 8th and 9th high-order digits of the parameter number specified with the operation command PRM, it is possible to read the parameter for the specified path.
  #i = PRM[#j];
  #j = ppxxxxxxx ; For pp, set a path number and for xxxxxxx, set a parameter number.

NOTE
1 If a path number is omitted or if 0 is specified, a parameter for the local path will be read.
2 If a path number out of the range is specified, alarm PS0115, “VARIABLE NO. OUT OF RANGE” is issued.

  b) Specifying a path number using a system variable
  By specifying a path number using system variable #3018, it is possible to read a parameter for the specified path.

Example
Reading the fourth axis of parameter No. 01322 for the second path
#3018 = 2 ; ..... Set the second path.
#1 =PRM[1322] /[4] ; .. Read the fourth axis of parameter No. 01322 for the second path and write it to variable #1 for the local path.
:
#3018= 0 ; ..... Return the path number to that of the local path.

- Abbreviations of arithmetic and logic operation commands

When a function is specified in a program, the first two characters of the function name can be used to specify the function.

Example: ROUND → RO
FIX → FI

NOTE
POW cannot be abbreviated.

- Priority of operations

<1> Functions
<2> Operations such as multiplication and division (*, /, AND)
<3> Operations such as addition and subtraction (+, -, OR, XOR)

FLYER MACHINE SYSTEMS INC
Rob Zahensky

```
Example) #1=#2+#3*SIN[#4];
                  <1>
              <2>
          <3>
```

<1>, <2> and <3> indicate the order of operations.

- Bracket nesting

Brackets are used to change the order of operations. Brackets can be used to a depth of five levels including the brackets used to enclose a function. When a depth of five levels is exceeded, an alarm PS0118, “TOO MANY BRACKET NESTING” occurs.

```
Example) #1=SIN [ [ [#2+#3] *#4 +#5] *#6];
                  <1>
                <2>
              <3>
            <4>
          <5>
```

<1> to <5> indicate the order of operations.

Limitation

- Caution concerning decreased precision When bit 0 (F16) of parameter No. 6008 is set to 0

- Addition and subtraction

Note that when an absolute value is subtracted from another absolute value in addition or subtraction, the relative error may become 10^{-15} or greater.

For example, assume that #1 and #2 have the following true values in the process of operation.

(The following values are examples in the process of operation and cannot actually be specified from any program.)

#1=9876543210.987654321
#2=9876543210.987657777

You cannot obtain the following result with operation #2-#1:

#2-#1=0.000003456

This is because the precision of custom macro variables is 15 decimal digits. With this precision, the values of #1 and #2 become:

#1=9876543210.987650000
#2=9876543210.987660000

(Precisely, the actual values are slightly different from the above values because they are internally processed in binary.) Therefore, the result is:

#2-1=0.000010000

A large error occurs.

- Logical expressions

Be aware of errors that can result from conditional expressions using EQ, NE, GT, LT, GE, and LE because they are processed basically in the same way as addition and subtraction. For example, if the following statement is used to decide whether #1 is equal to #2 in the above example, a correct decision may not be resulted because errors may occur:

IF [#1 EQ #2]

Evaluate the difference between #1 and #2 with:

IF [ABS [#1-#2]LT 0.1]

Then, assume that the values are equal when the difference does not exceed the allowable error range.

FRYER MACHINE SYSTEMS INC Rob Zahensky

- Trigonometric functions
  The absolute error is guaranteed for trigonometric functions. However, the relative error is 10^{-15} or greater. Carefully perform multiplication or division after executing a trigonometric function.
- FIX function
  When using the FIX function for the result of an operation, be careful with the precision. For example, when the following operations are performed, the value of #3 may not always be 2.

```
N10 #1=0.002;
N20 #2=#1*1000;
N30 #3=FIX[#2];
```

  This is because an error may occur in operation N20 and the result may not be

```
#2=2.0000000000000000
```

  but a value a little smaller than 2 such as the following:

```
#2=1.9999999999999997
```

  To prevent this, specify N30 as follows:

```
N30 #3=FIX[#2+0.001];
```

  Generally, specify the FIX function as follows:

  FIX[expression] → FIX[expression ±ε]

  (Specify +ε when the value of the expression is positive or -ε when it is negative, and 0.1, 0.01, 0.001, ... for ε as required.)

NOTE
The operation result of exponential function #i=EXP[#j]; overflows when #j exceeds about 790.

When bit 0 (F16) of parameter No. 6008 is set to 1

Errors may occur when operations are performed.

Table 16.5 (b) Errors involved in operations

Operation	Average error	Maximum error	Type of error
a = b*c	1.55×10^{-10}	4.66×10^{-10}	Relative error(*1) $\left\lvert \frac{\varepsilon}{a} \right\rvert$
a = b / c	4.66×10^{-10}	1.88×10^{-9}	
a = $\sqrt{b}$	1.24×10^{-9}	3.73×10^{-9}	
a = b + c a = b – c	2.33×10^{-10}	5.32×10^{-10}	(*2) MIN $\left\lvert \frac{\varepsilon}{b} \right\rvert$ $\left\lvert \frac{\varepsilon}{c} \right\rvert$
a = SIN [b] a = COS [b]	5.0×10^{-9}	1.0×10^{-8}	Absolute error(*3) $\lvert \varepsilon \rvert$ deg
a = ATAN [b] / [c]	1.8×10^{-6}	3.6×10^{-6}	

NOTE
1 The relative error depends on the result of the operation.
2 Smaller of the two types of errors is used.
3 The absolute error is constant, regardless of the result of the operation.
4 Function TAN performs SIN/COS.
5 Note that, in the case of natural logarithm #i=LN[#j]; and exponential function #i=EXP[#j];, the relative error may become 10^{-8} or greater.
6 The operation result of exponential function #i=EXP[#j]; overflows when #j exceeds about 110.

- The precision of variable values is about 8 decimal digits. When very large numbers are handled in an addition or subtraction, the expected results may not be obtained.

FRYER MACHINE SYSTEMS INC Rob Zaremsky

Example:

When an attempt is made to assign the following values to variables #1 and #2:

#1=9876543210123.456
#2=9876543277777.777

the values of the variables become:

#1=9876543200000.000
#2=9876543300000.000

In this case, when #3=#2-#1; is calculated, #3=100000.000 results. (The actual result of this calculation is slightly different because it is performed in binary.)

- Also be aware of errors that can result from conditional expressions using EQ, NE, GE, GT, LE, and LT.

  Example:

  IF[#1 EQ #2] is effected by errors in both #1 and #2, possibly resulting in an incorrect decision.

  Therefore, instead find the difference between the two variables with IF[ABS[#1-#2] LT 0.001].

  Then, assume that the values of the two variables are equal when the difference does not exceed an allowable limit (0.001 in this case).

- Also, be careful when rounding down a value.

  Example:

  When #2=#1*1000; is calculated where #1=0.002;, the resulting value of variable #2 is not exactly 2 but 1.99999997.

  Here, when #3=FIX[#2]; is specified, the resulting value of variable #3 is not 2.0 but 1.0.

  In this case, round down the value after correcting the error so that the result is greater than the expected number, or round it off as follows:

  #3=FIX[#2+0.001]
  #3=ROUND[#2]

- Brackets

Brackets ([ ]) are used to enclose an expression.
Note that parentheses ( ) are used for comments.

- Divisor

When a divisor of zero is specified in a division, an alarm PS0112, “ZERO DIVIDE” occurs.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

16.6 INDIRECT AXIS ADDRESS SPECIFICATION

Overview

When the custom macro function is enabled, you can use AX[(axis-number)] in an axis address specification to indirectly specify an axis with its axis number and not to directly specify it with its axis name.

You can also use AXNUM[(axis-name)] to obtain the axis number for an axis name.

Explanation

- Indirect axis address

You can use indirect axis address AX[ ] to specify an axis with its axis number. (An equal sign (=) is always required following AX[ ].)

AX[(axis-number)] = (numeric-value) ;

(axis-number): 1 to the number of controlled axes (number of controlled axes for each path in a multi-path system)
(numeric-value): Value specified for the axis specified with its axis number

If an invalid axis number is specified, an alarm PS0331, “ILLEGAL AXIS NUMBER IN AX[]” occurs. If a value is put in a decimal place, the number is rounded off to an integer and the result is treated as the axis number.
You can also specify a variable (local, common, or system variable) for (axis-number). To specify an operation using a variable name in (axis-number), however, enclose the variable name with brackets ([ ]).

1. **AX[1]=100.0;**
   Specifies a value of 100.000 for the first axis.
2. **AX[#500]=200.0;**
   Specifies a value of 200.000 for the axis having the axis number indicated by the value stored in #500.
3. **AX[#500+1]=300.0;**
   Specifies a value of 300.000 for the axis having the axis number obtained by adding 1 to the value stored in #500.
4. **SETVN 500 [ABC];**
   AX[#ABC]=400.0;
   Specifies a value of 400.000 for the axis having the axis number indicated by the value stored in #ABC (#500).
5. **SETVN 500 [ABC];**
   AX[[#ABC]+1]=500.0;
   Specifies a value of 400.000 for the axis having the axis number obtained by adding 1 to the value stored in #ABC (#500).
6. **SETVN 500 [ABC];**
   AX[#ABC+1]=500.0;
   Alarm PS0331 occurs.

- AXNUM function

You can use AXNUM[ ] to obtain an axis number.

AXNUM[(axis-name)];

If an invalid axis name is specified, an alarm PS0332, “ILLEGAL AXIS ADDRESS IN AXNUM[]” occurs.

When bit 3 (AXN) of parameter No.6021 is set to 1, in AXNUM function of custom macro, <null> can be obtained when the axis name that does not exist in the CNC is commanded, and the alarm does not occur. Therefore, it is possible to distinguish an axis is exist or not by custom macro program. Thus, the

FRYER MACHINE SYSTEMS INC
Rob Zahensky

common custom macro program can be created among different axis name and different machine configurations.

When the number of controlled axes is 3, the name of the first axis is X, that of the second axis is Y, and that of the third axis is Z

1. **#500=AXNUM[X];**
   A value of 1 is stored in #500.
2. **#501=AXNUM[Y];**
   A value of 2 is stored in #501.
3. **#502=AXNUM[Z];**
   A value of 3 is stored in #502.
4. **#503=AXNUM[A];**
   Alarm PS0332 occurs.

Example

Examples when the name of the first axis is X, that of the second axis is Y, and that of the third axis is Z1

```
N10 SETVN 500[AXIS1,AXIS2,AXIS3] ;
N20 [#AXIS1]=AXNUM[X] ;
N30 [#AXIS2]=AXNUM[Y] ;
N40 [#AXIS3]=AXNUM[Z1] ;
N50 G92 AX[#AXIS1]=0 AX[#AXIS2]=0 AX[#AXIS3]=0 ;
N60 G90G01F1000. ;
N70 AX[#AXIS1]=100.0 AX[#AXIS2]=100.0 AX[#AXIS3]=100.0 ;
N80 G02 AX[#AXIS1]=100. 0 AX[#AXIS2]= 0.0 R50.0 ;
N90 M02;
```

Limitation

When the custom macro function is enabled, AX and AXN cannot be used as an extended axis name. AX is assumed to be AX[ ] and AXN is assumed to be AXNUM[ ].

16.7 READING PARAMETERS

Overview

By using the PRM function, it is possible to read parameters.

Format	Remarks
#i = PRM[#j, #k] ;	In the case of parameters other than axis type parameters and spindle type parameters
#i = PRM[#j, #k] / [#l] ;	In the case of axis type parameters or spindle type parameters

Explanation

- **Reading a parameter**
  #i=PRM[#j,#k]
  #i=PRM[#j,#k]/[#l]

- For #j, enter a parameter number. If the number of a parameter that cannot be read, alarm PS0119, "ARGUMENT VALUE OUT OF RANGE", is issued.
- To read a bit type parameter, specify, for #k, the bit number of the bit type parameter in the range of 0 to 7. If a bit number is specified, data with the specified bit is read. If none is specified, data with all bits is read. For parameters other than bit type parameters, the bit number is ignored.
- For #l, set the axis number of an axis type parameter or an spindle type parameter in the range of 1 to the number of controlled axes or the number of controlled spindles. If an axis type parameter or an spindle type parameter is to be read but #l is not specified, alarm PS0119 is issued. For parameters other than axis type parameters and spindle type parameters, #1 may be omitted together with '/'.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Example

1. Reading the value of the third axis of bit 0 (MIR) of bit axis type parameter No. 0012
   If parameter No. 0012 (third axis) = 10010001
   #2=12 ; Parameter number setting
   #3=0 ; Bit number setting
   #4=3 ; Axis number setting
   If reading data with all bits
   #1=PRM[#2]/[#4] ; #1=10010001
   If reading data with a specified bit
   #1=PRM[#2, #3]/[#4] ; #1=1

2. Reading the value of the fourth axis of axis type parameter No. 1322
   #2=1322 ; Parameter number setting
   #4=4 ; Axis number setting
   #1=PRM[#2]/[#4] ;

3. Reading bit 2 (SBP) of bit type parameter No. 3404
   If parameter No.3404 = 01010000
   #2=3404 ; Parameter number setting
   #3=2 ; Bit number setting
   If reading data with all bits
   #1=PRM[#2] ; #1=1010000
   If reading data with a specified bit
   #1=PRM[#2,#3] ; #1=0

16.8 MACRO STATEMENTS AND NC STATEMENTS

The following blocks are referred to as macro statements:
- Blocks containing an arithmetic or logic operation (=)
- Blocks containing a control statement (such as GOTO, DO, END)
- Blocks containing a macro call command (such as macro calls by G65, G66, G66.1, G67, or other G codes, or by M codes)

Any block other than a macro statement is referred to as an NC statement.

Explanation

- Differences from NC statements

- Even when single block mode is on, the machine does not stop. Note, however, that the machine stops in the single block mode when bit 5 (SBM) of parameter No. 6000 is 1.
- Macro blocks are not regarded as blocks that involve no movement in the cutter compensation mode.

- NC statements that have the same property as macro statements

- NC statements that include a subprogram call command (such as subprogram calls by M98 or other M codes, or by T codes) and not include other command addresses except an O, file name, N, P, or L address have the same property as macro statements.
- NC statements that include M99 and those do not include other command addresses except an O, file name, N, P, or L address have the same property as macro statements.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

16.9 BRANCH AND REPETITION

In a program, the flow of control can be changed using the GOTO statement and IF statement. Four types of branch and repetition operations are used:

Branch and repetition	GOTO / LGOTO (unconditional branch / unconditional branch with label)
	IF (conditional branch: if ..., then...)
	WHILE (repetition while ...)
	SWITCH-CASE (multidirectional branch)

16.9.1 Unconditional Branch (GOTO Statement) / Unconditional Branch with label (LGOTO Statement)

Unconditional Branch (GOTO Statement)

A branch to sequence number n occurs. When a sequence number outside of the range 1 to 99999999 is specified, an alarm PS1128, “SEQUENCE NUMBER OUT OF RANGE” occurs. A sequence number can also be specified using an expression.

GOTOn ; n: Sequence number (1 to 99999999)

Example: GOTO 1;
GOTO #10;

> ⚠ **WARNING**
> Do not specify multiple blocks with the same sequence number in a single program. It is very dangerous to specify such blocks because the destination of a branch from the GOTO statement is undefined.

> **NOTE**
> 1 A backward branch takes more time as compared with a forward branch.
> 2 In the destination of GOTOn, that is, the block with sequence number n, the sequence number must appear at the beginning of the block. If the sequence number is not at the beginning of the block, a branch cannot be made.

Unconditional Branch with label (LGOTO Statement)

It is possible to branch to the label (arbitrary character string) unconditionally.

```
LGOTO[ label ] ;
…
LBL[ label ] ;
```

- The character that can be used for the label is capital letter alphabet (A-Z), number (0-9), and underscore (_). However, the number cannot be used for the first character.
- The number of characters that can be used for the label is 31 characters or less.
- LBL[label] can be used together with the NC statement, the macro statement, the sequence number, and the comment.
- Bit 0 (NOB) of parameter No.3404 and bit 4 (NBN) of parameter No.3451 are valid to LBL[label] as well as the sequence number. When the block only of LBL[label] is executed by setting NOB=1 and NBN=0, it is ignored.

FRYER MACHINE SYSTEMS INC
Rob Zakrensky

Example

```
N1 LGOTO[BBB];
…
N11 LBL[AAA] G90 G00 X100.0 ;
…
N22 LBL[BBB] G90 G00 Y50.0 (LABEL BBB);
…
N33 LBL[CCC] #100 = 10.0 ;
…
```

It jumps to N22 block by LGOTO[BBB] of N1.

⚠ **WARNING**

Do not specify multiple blocks with the same label in a single program. It is very dangerous to specify such blocks because the destination of a branch from the LGOTO statement is undefined.

NOTE

1 LG statement cannot be used as omitted LGOTO statement. When LG statement is commanded, the alarm PS0125 "MACRO STATEMENT FORMAT ERROR" occurs.
2 When any mistake (the character that cannot be used is used , etc.) is found in the LGOTO[label] block, the alarm PS0114 "ILLEGAL EXPRESSION FORMAT" occurs.
3 When the label specified by LGOTO[label] is not found in the program, the alarm PS1135 " SPECIFIED LABEL IS NOT FOUND " occurs.
4 When LBL[label] block that the mistake (the character that cannot be used is used , etc.) is found in the format is executed, the alarm PS1151 "LABEL FORMAT ERROR" occurs.
5 A backward branch takes more time as compared with a forward branch.

16.9.2 GOTO Statement Using Stored Sequence Numbers / LGOTO Statement Using Stored Label

When the GOTO/LGOTO statement is executed in a custom macro control command, a sequence number search/a label search is made for sequence numbers/LBL[label] stored at previous execution of the corresponding blocks at a high speed.

As a "sequence number/LBL[label] stored at previous execution of the corresponding block", a unique sequence number/LBL[label] within the same program or the sequence number/LBL[label] of a subprogram call with which the block was executed is stored.

The storage type differs depending on the values of the following parameters.

(1) When bit 1 (MGO) of parameter No. 6000 is set to 1
- Fixed type: Up to 20 sequence numbers/LBL[label] stored at execution of the corresponding blocks from the start of operation

(2) When bit 4 (HGO) of parameter No. 6000 is set to 1
- Variable type: Up to 30 sequence numbers/LBL[label] stored at execution of the corresponding blocks before execution of the GOTO/LGOTO statement
- History type: Up to 10 sequence numbers/LBL[label] stored by a sequence number search/a label search previously made using the GOTO/LGOTO statement

The stored sequence numbers/LBL[label] are canceled in the following cases:

- Immediately after power-on

FRYER MACHINE SYSTEMS INC Rob Zalensky

- After a reset
- Operation after program registration or editing (including background editing and MDI program editing)

⚠ WARNING

Do not specify multiple blocks with the same sequence number/LBL[label] in a single program.

It is very dangerous to specify the sequence number/LBL[label] of the branch destination before and after the GOTO/LGOTO statement and execute the GOTO/LGOTO statement because the branch destination changes according to the values of the parameters as shown below:

When bit 1 (MGO) or 4 (HGO) of parameter No. 6000 is set to 1	**When both bits 1 (MGO) and 4 (HGO) of parameter No. 6000 are set to 0**
: N10; : GOTO10; : N10; A branch to N10 before the GOTO statement occurs.	: N10; : GOTO10; : N10; A branch to N10 after the GOTO statement occurs.
: LBL[AAA] ; : LGOTO[AAA] ; : LBL[AAA] ; A branch to LBL[AAA] before the LGOTO statement	: LBL[AAA] ; : LGOTO[AAA] ; : LBL[AAA] ; A branch to LBL[AAA] after the LGOTO statement

When bit 1 (MGO) or 4 (HGO) of parameter No. 6000 is set to 1 and the GOTO/LGOTO statement is executed, the sequence number/LBL[label] of the branch destination may not be contained in the sequence numbers/LBL[label] stored at previous execution of the corresponding blocks. In this case, a branch to the sequence number/LBL[label] in a block following the GOTO/LGOTO statement occurs (the destination is the same as when both bits are set to 0).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

1 When an external program is read and executed by DNC operation, the executed sequence numbers/LBL[label] are not stored.
When a program registered in memory is executed by a subprogram call, the sequence numbers/LBL[label] are stored.

2 Set bit 1 (SMI) of parameter No.11651 to 1 when the memory operation on the data server or the memory card is executing.
Parameter SMI
When GOTO statement using stored sequence numbers/LGOTO statement using stored label is enabled and
memory operation of a program on Data Server/Memory card is executed, sequence numbers/LBL[label] in the program are:
0:Stored.
1:Not stored.

3 The number of destinations that can be stored by LGOTO Statement using stored label is shared by GOTO statement using stored sequence numbers.
Example :
The number that can be stored by fixed type is 20 or less.
In this case, the number that can be stored is as follows.
(the number of the sequence number + the number of LBL[label]) ≤ 20

⚠ **CAUTION**

According to the restrictions on the GOTO/LGOTO statement, no branch to a sequence number within a DO-END loop cannot be made. If a program in which a branch to a sequence number within a loop occurs is executed, operation may differ depending on whether the GOTO statement using stored sequence numbers/LGOTO statement using stored label is used.

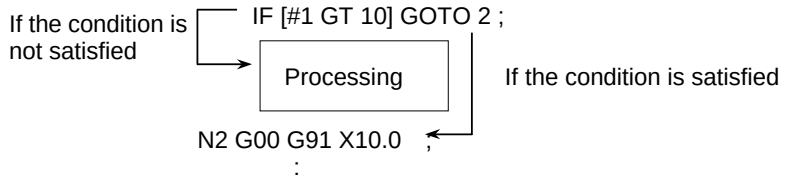
16.9.3 Conditional Branch (IF Statement)

Specify a <conditional expression> after IF.
There are 5 kinds of instruction formats as follows. The executable instruction formats are decided by setting bit 2 (ESE) of parameter No.11648.

(a) IF[<conditional expression>]GOTOn;

If the specified <conditional expression> is satisfied (true), a branch to sequence number n occurs. If the specified condition is not satisfied, the next block is executed.

If the value of variable #1 is greater than 10, a branch to sequence number N2 occurs.



(b) IF[<conditional expression>]THEN;

If the specified <conditional expression> is satisfied (true), a macro statement specified after THEN is executed.
Only a single macro statement is executed.

FRYER MACHINE SYSTEMS INC Rob Zamansky

If the values of #1 and #2 are the same, 0 is assigned to #3.
IF[#1 EQ #2] THEN#3=0 ;

(c) IF[<conditional expression>] THEN; Multiple statements; ENDIF;

If the specified <conditional expression> is satisfied (true), the multiple statements specified in between THEN and ENDIF are executed. If the specified condition is not satisfied, the multiple statements are skipped.
Two or more NC statements and macro statements can be specified between THEN and ENDIF.

If the values of #1 and #2 are the same, the multiple blocks specified between THEN and ENDIF are executed.
IF[#1 EQ #2] THEN; #101 = #4201; #102 = #5041; G91 G28 X0; ENDIF;

(d) IF[<conditional expression>] THEN Macro statement 1; ELSE Macro statement 2;

If the specified <conditional expression> is satisfied (true), Macro statement 1 specified after THEN is executed. If the specified condition is not satisfied, Macro statement 2 specified after ELSE is executed.
Only a single macro statement can be specified to Macro statement 1 or Macro statement 2.

If the values of #1 and #2 are the same, 0 is assigned to #3. If the both values are the different, 0 is assigned to #4.
IF[#1 EQ #2] THEN #3=0; ELSE #4=0;

(e) IF[<conditional expression>] THEN; Multiple statements 1; ELSE; Multiple statements 2; ENDIF;

If the specified <conditional expression> is satisfied (true), Multiple statements 1 specified between THEN and ELSE are executed. If the specified condition is not satisfied, Multiple statements 2 specified between ELSE and ENDIF are executed.
Two or more NC statements and macro statements can be specified to Multiple statements 1 or Multiple statements 2.

If the values of #1 and #2 are the same, the multiple blocks specified between THEN and ELSE are executed. If the both values are the different, the multiple blocks specified between ELSE and ENDIF are executed.

IF[#1 EQ #2] THEN;
#101 = #4201;
#111 = #5041;
G91 G28 X0;
ELSE;
#102 = #4202;
#112 = #5042;
G91 G28 Y0;
ENDIF;

Explanation

- Relation between bit 2 (ESE) of parameter No.11648 and executable function

The executable functions are decided by setting bit 2 (ESE) of parameter No.11648.
When bit 2 (ESE) of parameter No.11648 is set to 0.

- ELSE statement is invalid. 2 kinds of instruction formats a),b) are effective.
- Nesting of IF statement is impossible.
- "ELSE", and "ENDIF" are not reserved words.

FRYER MACHINE SYSTEMS INC
Rob Zaborsky

When bit 2 (ESE) of parameter No.11648 is set to 1.
- ELSE statement is effective. 5 kinds of instruction formats a)-e) are effective.
- Nesting of IF statement is enabled.
- "ELSE", and "ENDIF" are added to reserved words.

NOTE
When bit 2 (ESE) of parameter No.11648 is set to 1, the program display of Manual Guide *i* cannot treat the program including "ELSE" or "ENDIF". When the program is displayed, the delimitation of the word is not correct.

- <Conditional expression>

<Conditional expressions> are divided into <simple conditional expressions> and <complex conditional expressions>. In a <simple conditional expression>, a relational operator described in Table 16.9.3 (a) is specified between two variables or between a variable and constant to be compared. An <expression> can be used instead of a variable. With a <complex conditional expression>, an AND (logical AND), OR (logical OR), or XOR (exclusive OR) operation is performed for the results (true or false) of multiple <simple conditional expressions>.

If the values of #1 and #2 are the same and those of #3 and #4 are also the same, 0 is assigned to #5.
IF[[#1 EQ #2] AND [#3 EQ #4]] THEN#5=0 ;

If the values of #1 and #2 are the same or those of #3 and #4 are the same, 0 is assigned to #5.
IF[[#1 EQ #2] OR [#3 EQ #4]] THEN#5=0 ;

- Relational operators

Relational operators each consist of two letters and are used to compare two values to determine whether they are equal or one value is smaller or greater than the other value. Note that the equal sign (=) and inequality sign (>, <) cannot be used as a relational operator.

Table 16.9.3 (a)Relational operators

Operator	Meaning
EQ	Equal to(=)
NE	Not equal to(≠)
GT	Greater than(>)
GE	Greater than or equal to(≥)
LT	Less than(<)
LE	Less than or equal to(≤)

- About how to combine the instruction format

Instruct in the commands “IF” , “THEN”, “ELSE”, “ENDIF” so as not to contradict the combination. When the combinations are contradicted, alarm (PS1129) “MISSING IF STATEMENT” is occur.
Above-mentioned instruction format (a)-(e) are basic type.
It is possible to instruct by combining single macro statement and multiple statements as follows.

If the values of #1 and #2 are the same, 0 is assigned to #3. If the both values are the different, the multiple blocks specified between ELSE and ENDIF are executed.

IF[#1 EQ #2] THEN #3=0;
ELSE;
#102 = #4202;
#112 = #5042;
G91 G28 Y0;
ENDIF;

FRKER MACHINE SYSTEMS INC Rob Zahensky

If the values of #1 and #2 are the same, the multiple blocks specified between THEN and ELSE are executed. If the both values are the different, 0 is assigned to #4.

```
IF[#1 EQ #2] THEN;
                #101 = #4201;
                #111 = #5041;
                G91 G28 X0;
            ELSE #4=0;
```

However, it is not possible to instruct by combining GOTO statement and ELSE statement as follows.

If the values of #1 and #2 are the same, a branch to sequence number N20 occurs. If the both values are the different, a macro statement is executed.

The following instructions cannot be executed (ELSE statement is continued to GOTO statement).

```
IF[#1 EQ #2] GOTO 20;
ELSE         macro statement;
```

In such a case, divide the instructions into IF-GOTO block and the block of macro statement as follows.

```
IF[#1 EQ #2] GOTO 20;   If the values of #1 and #2 are the same, a branch to sequence number N20 occurs.
macro statement;        If the both values are the different, this block is executed as a macro statement.
```

- Nesting of IF statement

IF statement can do nesting like the following examples. Up to 10 levels of nesting of IF statement are allowed. When the maximum value of the nesting of IF statement is exceeded, alarm (PS1130) “TOO MANY IF STATEMENT NESTING” is occur.

EXAMPLE

The following is an example of 3 levels of nesting, but up to 10 levels of nesting are allowed.

```
IF[...] THEN ;
    IF [...] THEN ;
            Statement ;
                :
    ELSE ;
        IF[...] THEN ;
                Statement ;
        ENDIF ;
    ENDIF ;
Statement ;
    :
ELSE ;
    Statement ;
        :
ENDIF ;
```

3rd level

2nd level

1st level

FRYER MACHINE SYSTEMS INC
Rob Zahonsky

⚠ CAUTION

When only a single macro statement is to be executed, the macro statement can be specified immediately after THEN/ELSE. In this case, no ENDIF statement is usually required. However, an ENDIF statement is required when IF [...] THEN macro statement; is specified just before ELSE or ENDIF of the previous nesting as shown below:

```
N10 IF[...] THEN ;
        N21 IF [...] THEN macro statement;
        N22 IF [...] THEN macro statement;
                ENDIF ; ⇐ To specify the thing that ELSE statement has paired
                          with N10 IF statement, ENDIF statement that pairs
                          with N22 IF statement is needed.
    ELSE ;
        N23 IF [...] THEN macro statement;
        N24 IF [...] THEN macro statement;
                ENDIF ; ⇐ To specify the thing that next ENDIF statement has
                          paired with N10 IF statement, ENDIF statement that
                          pairs with N24 IF statement is needed.
    ENDIF;
```

NOTE

The nesting of IF statement is checked only by the custom macro.
Because macro compiler checks nesting in the execution macro, it doesn't do in automatic operation.

- MDI intervention during automatic operation

When MDI intervention during automatic operations is done, the nesting of IF statement in automatic operation is succeeded to MDI operation.

Up to 5 levels of nesting of IF statement are allowed in MDI operation after nesting becomes 5 levels by automatic operation.

```
IF[#1 EQ #1] THEN ;
  IF[#1 EQ #2] THEN ;
    IF[#1 EQ #3] THEN ;
      IF[#1 EQ #4] THEN ;
        IF[#1 EQ #5] THEN ;
          #101 = #4201;
          #102 = #5041; <= Up to 5 levels of nesting of IF statement are allowed in MDI intervention during
          G91 G28 X0 ;      automatic operations.
        ENDIF ;
      ENDIF ;
    ENDIF ;
  ENDIF ;
ENDIF ;
```

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

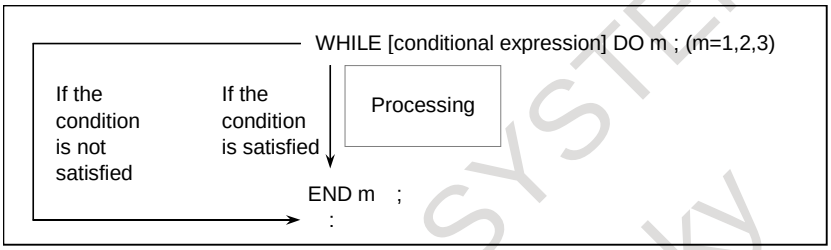
Sample program

The sample program below finds the total of numbers 1 to 10.

```
O9500;
   #1=0;................................ Initial value of the variable to hold the sum
   #2=1;................................ Initial value of the variable as an addend
N1 IF[#2 GT 10] GOTO 2;... Branch to N2 when the addend is greater than 10
   #1=#1+#2;......................... Calculation to find the sum
   #2=#2+1;........................... Next addend
   GOTO 1; ........................... Branch to N1
N2 M30;............................... End of program
```

16.9.4 Repetition (WHILE Statement)

Specify a conditional expression after WHILE. While the specified condition is satisfied, the program from DO to END is executed. If the specified condition is not satisfied, program execution proceeds to the block after END.



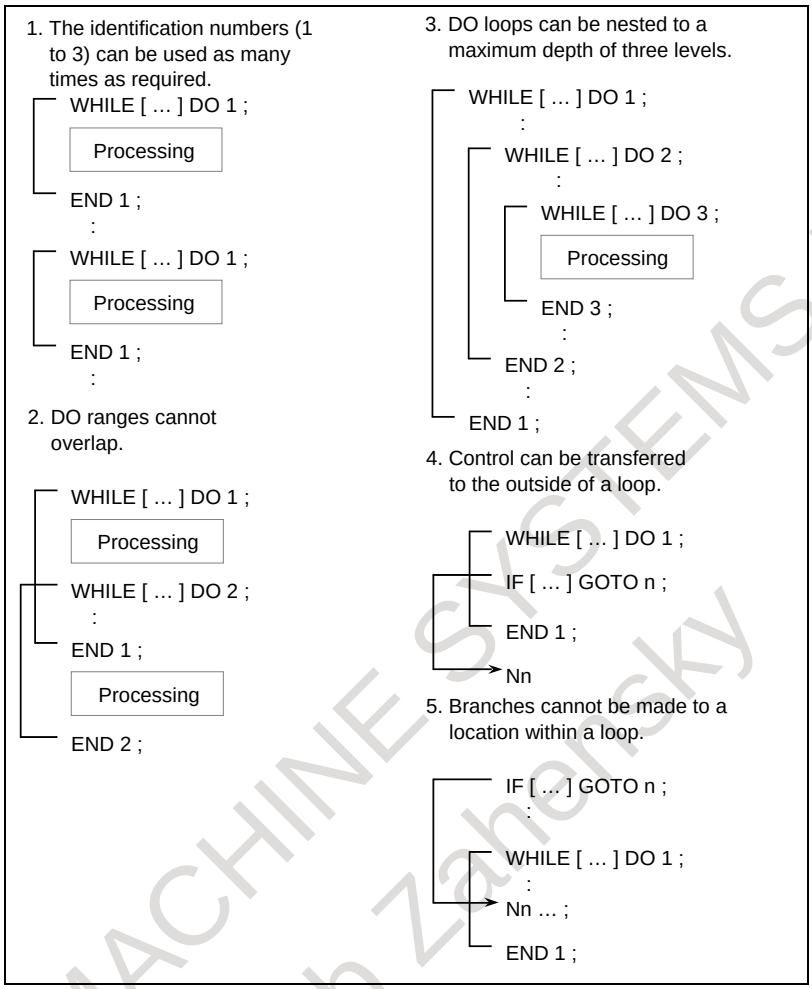
Explanation

While the specified condition is satisfied, the program from DO to END after WHILE is executed. If the specified condition is not satisfied, program execution proceeds to the block after END. The same format as for the IF statement applies. A number after DO and a number after END are identification numbers for specifying the range of execution. The numbers 1, 2, and 3 can be used. When a number other than 1, 2, and 3 is used, an alarm PS0126, “ILLEGAL LOOP NUMBER” occurs.

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

- Nesting

The identification numbers (1 to 3) in a DO-END loop can be used as many times as desired. Note, however, when a program includes overlapped DO ranges, or a program includes branches into the loop, an alarm PS1124, “MISSING DO STATEMENT” occurs when the END statement is executed.



Limitation

- Infinite loops

When DO m is specified without specifying the WHILE statement, an infinite loop ranging from DO to END is produced.

- Processing time

When a branch to the sequence number specified in a GOTO statement occurs, the sequence number is searched for. For this reason, processing in the reverse direction takes a longer time than processing in the forward direction. Therefore, in the case of processing in the reverse direction, use the WHILE statement for repetition to reduce processing time.

- Undefined variable

In a conditional expression that uses EQ or NE, a <null> and zero have different effects. In other types of conditional expressions, a <null> is regarded as zero.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

Sample program

The sample program below finds the total of numbers 1 to 10.

```
O0001;
#1=0;
#2=1;
WHILE[#2 LE 10]DO 1;
#1=#1+#2;
#2=#2+1;
END 1;
M30;
```

16.9.5 Precision Specification for Macro Relational Operators

When making a comparison using a relational operator in a custom macro, you can specify the number of decimal places in the two values to be compared. The values are compared after rounded off to the specified number of decimal places.

Setting bit 5 (EDP) of parameter No. 6019 to 1 enables this function. Specify the number of significant decimal places you want to set in parameter No. 6100.

Relational operators

The precision specification is valid for the following relational operators.

Operator	Meaning
EQ	Equal to(=)
NE	Not equal to(≠)
GT	Greater than(>)
GE	Greater than or equal to(≥)
LT	Less than(<)
LE	Less than or equal to(≤)

Limitation

The precision specification for macro relational operators is valid for the following functions:

- Custom macro
- Macro executor

NOTE
The precision specification is invalid for real-time custom macros.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Example

Macro program

```
O1000 ;
#100=1.2344567 ;
#101=1.2345678 ;
N10 IF[#100 EQ #101] GOTO 20 ;
...
N20 ;
N30 IF[#100 NE #101] GOTO 40 ;
...
N40 ;
...
```

When a comparison is executed, the target two values are compared after rounded off to the specified number of decimal places.
The actual values are not rounded.

#100=1. 2344567
ˆˆˆ The number of significant decimal places is 3.
ˆRounded off.

1. When the precision specification for macro relational operators is disabled
   Bit 5 (EDP) of parameter No. 6019 is set to 0.
   The values of #100 and #101 are compared as is. In the N10 block, a branch to N20 does not occur. In the N30 block, a branch to N40 occurs.

2. When the precision specification for macro relational operators is enabled - 1
   Bit 5 (EDP) of parameter No. 6019 is set to 1 and parameter No. 6100 is set to 2.
   The value obtained by rounding off the value of #100, 1.23, is compared with the value obtained by rounding off the value of #101, 1.23. The values of #100 and #101 are not changed.
   In the N10 block, a branch to N20 occurs. In the N30 block, a branch to N40 does not occur.

3. When the precision specification for macro relational operators is enabled - 2
   Bit 5 (EDP) of parameter No. 6019 is set to 1 and parameter No. 6100 is set to 3.
   The value obtained by rounding off the value of #100, 1.234, is compared with the value obtained by rounding off the value of #101, 1.235. The values of #100 and #101 are not changed.
   In the N10 block, a branch to N20 does not occur. In the N30 block, a branch to N40 occurs.

When variable values are negative, processing is performed as follows:

Macro program

```
O1000 ;
#100=-1.2344 ;
#101=-1.2345 ;
N10 IF[#100 EQ #101] GOTO 20 ;
...
N20 ;
...
```

#100=- 1. 2345
ˆˆˆ The number of significant decimal places is 3.
ˆRounded off.

1. When the precision specification for macro relational operators is enabled
   Bit 5 (EDP) of parameter No. 6019 is set to 1 and parameter No. 6100 is set to 3.
   The value obtained by rounding off the value of #100, -1.234, is compared with the value obtained by rounding off the value of #101, -1.235. The values of #100 and #101 are not changed.
   In the N10 block, a branch to N20 does not occur.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

16.9.6 Multidirectional branch (SWITCH-CASE Statement)

The flow of control can be changed in multidirectional.

Explanation

This function is composed of SWITCH statement, CASE statement, and DEFAULT statement. The flow of the program can be changed in multidirectional according to the expression specified by SWITCH statement. There are one block command and multiple block command in the format.

One block command format
SWITCH[<expression>] CASE[<expression 1>] GOTOn1 CASE[<expression 2>] GOTOn2 ... DEFAULT[ ] GOTOnx ;

Multiple block command format
SWITCH[<expression>] ; CASE[<expression 1>] GOTOn1 ; CASE[<expression 2>] GOTOn2 ; ... DEFAULT[ ] GOTOnx ;

- SWITCH statement : SWITCH[<expression>]
  The calculation result of <expression> is saved. The saved value is compared to the value specified in the CASE statement. The value saved by SWITCH statement is maintained until GOTO of CASE statement is executed or DEFAULT statement is executed.
- CASE statement : CASE[<expression>] GOTOn
  When the value saved by SWITCH statement is corresponding to <expression>, GOTO statement is executed. Moreover, the value saved by SWITCH statement is cleared. That is, it makes the program to the state that doesn't execute SWITCH statement. When the value saved by SWITCH statement is not corresponding to <expression>, CASE statement does not do anything.
- DEFAULT statement : DEFAULT[ ] or DEFAULT[ ] GOTOn
  It is the command for the case which the value saved by SWITCH statement is not corresponding to any <expression>. GOTO statement is unconditionally executed. Moreover, the value saved by SWITCH statement as well as CASE statement is cleared. When not command GOTO statement, DEFAULT statement only clears the value saved by SWITCH statement. Do not put anything in brackets “[ ]”. There is no problem even if not command DEFAULT statement.

One block command
N1 #101=30; N2 SWITCH[#100] CASE[10] GOTO22 CASE[20] GOTO33 CASE[#101] GOTO44 DEFAULT[ ] GOTO100; N22 #500=1; … N33 #500=2; … N44 #500=3; … N100 #500=4; … In SWITCH statement of N2 block, It jumps to N22 block when #100 is 10. It jumps to N44 block when #100 is 30. It jumps to N100 block when #100 is not 10, 20, or 30.

FRYER MACHINE SYSTEMS INC Rob Zahensky

Multiple block command

```
N1 #101=30;
N2 SWITCH[#100];
N3 CASE[10] GOTO22;
N4 CASE[20] GOTO33;
N5 CASE[#101] GOTO44;
N6 DEFAULT[ ] GOTO100;
N22 #500=1;
…
N33 #500=2;
…
N44 #500=3;
…
N100 #500=4;
…
```

In SWITCH statement of N2 block,
It jumps to N22 block in N3 block when #100 is 10.
It jumps to N44 block in N5 block when #100 is 30.
It jumps to N100 block in N6 block when #100 is not 10, 20, or 30.

NOTE

1 SWITCH statement, CASE statement, DEFAULT statement can not be omitted command (command of the first two characters only), such as in the following example. When the omitted command as SW, CA, and DE is commanded, the alarm PS0125 "MACRO STATEMENT FORMAT ERROR" occurs.
example)
ROUND -> RO
FIX -> FI

2 When the power is turned off or the reset is executed, the value saved by SWITCH statement is cleared.

3 The value of <expression> of SWITCH and CASE is treated as an integral value in which it rounds off below the decimal point. (half-adjust)

Example 1)
In the following cases, SWITCH statement saves 1.
#100=1.4999;
SWITCH[#100];

Example 2)
In the following cases, SWITCH statement saves 2.
#100=1.5000;
SWITCH[#100];

4 SWITCH statement, CASE statement, and DEFAULT statement is executed in the order in which they are commanded. Therefore, even if there is CASE statement that is corresponding to the value saved by SWITCH statement, when there is DEFAULT statement earlier, DEFAULT statement is executed.
Example 1)
In the following cases, GOTO100 of DEFAULT[ ] is executed. GOTO44 of CASE [30] is not executed.
N1 #100=30;
N2 SWITCH[#100] CASE[10] GOTO22 DEFAULT[ ] GOTO100 CASE[20]

FRYER MACHINE SYSTEMS INC
Rob Zahersky

NOTE

```
GOTO33 CASE[30] GOTO44;
…
N100 #500=1;
```

Example 2)
In the following cases, GOTO11 of DEFAULT[ ] is executed. GOTO2 of CASE [10] is not executed.

```
N1 #100=10;
N2 SWITCH[#100]
N3 DEFAULT[ ] GOTO11
N4 CASE[10] GOTO2
N5 CASE[20] GOTO4
…
N11 #500=1;
```

5 When SWITCH statement is executed , the value of <expression> is saved. Even if the value of <expression> is changed, the saved value is not changed.
Example)
In the following cases, 10 is saved in N2 block. The saved value in N3 block is not changed. Therefore, GOTO5 of N5 block is executed.

```
N1 #100=10;
N2 SWITCH[#100];
N3 #100=20;
N4 CASE[20] GOTO2;
N5 CASE[10] GOTO5;
```

6 In the case of multiple block command, the command between the blocks of SWITCH statement, CASE statement, and DEFAULT statement is executed.
Example)
In the following cases, N3 block, N6 block, and N7 block which is commanded between SWITCH statement and CASE statement are also executed. Therefore, GOTO44 of CASE [30] of N8 block is not executed.

```
N1 #100=30;
N2 SWITCH[#100];
N3 #500=1;
N4 CASE[10] GOTO22;
N5 CASE[20] GOTO33;
N6 #500=2;
N7 GOTO77;
N8 CASE[30] GOTO44;
…
N77#500=3;
```

7 When SWITCH statement is executed in succession, the saved value is the value of the SWITCH statement that last is executed.
Example)
The following case, 20 is saved by SWITCH statement of N3 block. However, because it is overwritten to 10 by SWITCH statement of N4 block, GOTO3 of N5 block is executed.

```
N1 #100=20;
```

FRYER MACHINE SYSTEMS INC
Rob Zalensky

NOTE

N2 #101=10;
N3 SWITCH[#100];
N4 SWITCH[#101];
N5 CASE[10] GOTO3;
N6 CASE[20] GOTO5;

8 When SWITCH statement contains an error, the alarm PS1148 "SWITCH STATEMENT ERROR" occurs.
The cause may be one of the following:
1. Format error
   - Not “SWITCH[<expression>]”
   - The next of “SWITCH[<expression>]” is not “EOB”, “CASE” or “DEFAULT”.
2. The value of <expression> is out-of-range (-2147483648 to 2147483647).
   <Empty> is regarded as out-of-range value.

9 When CASE statement contains an error, the alarm PS1149 "CASE STATEMENT ERROR" occurs.
The cause may be one of the following:
1. CASE statement is commanded with SWITCH statement not executed.
2. Format error
   - Not “CASE[<expression>]”
   - The next of “CASE[<expression>]” is not “GOTO”. However, when the value saved by SWITCH statement is not corresponding to <expression>, the alarm does not occur.
3. The value of <expression> is out-of-range (-2147483648 to 2147483647).
   <Empty> is regarded as out-of-range value.

10 When DEFAULT statement contains an error, the alarm PS1150 "DEFAULT STATEMENT ERROR" occurs.
The cause may be one of the following:
1. DEFAULT statement is commanded with SWITCH statement not executed.
2. Format error
   - Not “DEFAULT[ ]”
   - The next of “DEFAULT[ ]” in not “EOB” or “GOTO”.

11 When SWITCH statement, CASE statement, or DEFAULT statement contain an error, the alarm PS1148, PS1149, or PS1150 occurs. However, the alarm does not occur when GOTO statement is executed in front of a place where contains an error in one block command.
Example)
In the following cases, “#100 = 30” does not follow the format. However, the alarm does not occur because GOTO55 of CASE[10] is executed.
N1 #100=10;
N2 SWITCH[#100] CASE[10] GOTO55 CASE #100=30 [20] GOTO66 DEFAULT[ ] GOTO100;
…
N55 #500=1;

12 Place where it is possible to put a comment is place of the following example.
Example)
Comment : (AA)
SWITCH[#100] (AA) CASE[10] (AA) GOTO55 (AA) CASE[20] (AA) GOTO66 (AA) DEFAULT[ ] (AA) GOTO100 (AA) ;

13 The value saved by SWITCH statement of program that is called at some point

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

NOTE

is different from the value saved by SWITCH statement of program that is called at other times. The value saved by SWITCH statement is not succeeded.

Example 1)

O1000 from O0001 is called, and O2000 from O1000 is called. And SWITCH[10] in O0001, SWITCH[20] in O1000, and SWITCH[30] in O2000 is executed. At this time, the saved value in O0001 is 10. Therefore, in O0001, GOTO30 of N5 block is executed.

```
O0001;
N1 SWITCH[10];
N2 M98 P1000;
N3 CASE[30] GOTO10;
N4 CASE[20] GOTO20;
N5 CASE[10] GOTO30;
N10 #100=10;
N20 #101=20;
N30 #102=30;
N6 M30

O1000;
N1 SWITCH[20];
N2 M98 P2000;
N3 M99;

O2000;
N1 SWITCH[30];
N2 M99;
```

Example 2)

SWITCH statement is executed in O0002. However, the alarm PS1149 "CASE STATEMENT ERROR" occurs in CASE[30] because SWITCH statement is not executed in O3000.

```
O0002;
N1 SWITCH[50];
N2 M98 P3000;
N3 M30

O3000;
N1 CASE[30] GOTO10;
N10 #100=10;
N2 M99;
```

Example 3)

Even Macro modal call, the value saved by SWITCH statement is not succeeded. When O4000 is called from A1.0 of N2 block in O0003, SWITCH statement is executed. However, the alarm PS1149 "CASE STATEMENT ERROR" occurs in CASE[30] because CASE statement is commanded with SWITCH statement not executed when O4000 is called from A2.0 of N3 block in

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

```
O0003.

O0003;
N1 G66.1 P4000 A0;
N2 A1.0;
N3 A2.0;
N4 M30;

O4000;
N1 SWITCH[#1] CASE[1] GOTO10 CASE[2] GOTO20 DEFAULT[ ] GOTO100
N10
N2 SWITCH[10];
N3 M99;
N20
N4 CASE[10] GOTO30;
N30 #100=1;
N5 M99
N100
N6 M99
```

16.10 MACRO CALL

A macro program can be called using the following methods. The calling methods can roughly be divided into two types: macro calls and subprogram calls.
A macro program can also be called during MDI operation in the same way.

- Macro call
  - Simple call (G65)
  - Modal call (G66, G66.1, G67)
  - Macro call using a G code
  - Macro call using an M code
- Subprogram call
  - Subprogram call using an M code
  - Subprogram call using a T code
  - Subprogram call using an S code
  - Subprogram call using a second auxiliary function code
  - Subprogram call using a specific code

Limitation

- Call nesting

Macro calls can be nested to a depth of up to five levels. Subprogram calls can be nested to a depth of up to 15 levels including macro calls.

- Differences between macro calls and subprogram calls

Macro call (G66, G66.1, Ggg, or Mmm) differs from subprogram call (for example M98, Mmm, or Ttt) as described below.

- With a macro call, an argument (data passed to a macro) can be specified. A subprogram call does not have this capability.
- If a macro call block contains another NC command (such as G01 X100.0 G65 Pp), an alarm PS0127, “DUPLICATE NC,MACRO STATEMENT” occurs.
- If a subprogram call block contains another NC command (such as G01 X100.0 M98 Pp), the subprogram is called after the command is executed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

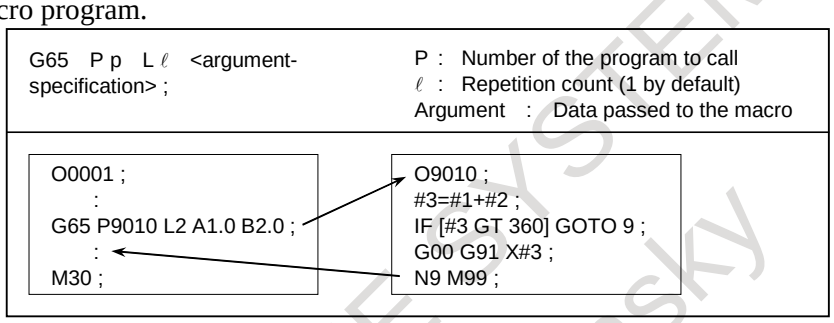
- In any macro call block, the machine does not stop in the single block mode.
  If a subprogram call block contains another NC command (such as G01 X100.0 M98 Pp), the machine stops in the single block mode.
- With a macro call, the level of local variables changes. With a subprogram call, the level of local variables does not change. (See "Local variable levels" in Limitation of Subsection, “Simple Call (G65)”.)

- Called program and folders to be searched

The order in which folders are called depends on the method of calling a macro or subprogram. Folders are searched in sequence and the program found first is called. For details, see the "Managing Programs" chapter.

16.10.1 Simple Call (G65)

When G65 is specified, the custom macro specified at address P is called. Data (argument) can be passed to the custom macro program.



Explanation

- Call

- After G65, specify at address P the program number of the custom macro to call.
- When the number of repetitions is required, specify a number from 1 to 99999999 after address L. When L is omitted, 1 is assumed.
- By using argument specification, values are assigned to corresponding local variables.

- Argument specification

Two types of argument specification are available. Argument specification I uses letters other than G, L, O, N, and P once each. Argument specification II uses A, B, and C once each and also uses I, J, and K up to ten times. The type of argument specification is determined automatically according to the letters used.

- Argument specification I

Address	Variable number
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11

Address	Variable number
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19

Address	Variable number
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

- Addresses G, L, N, O, and P cannot be used in arguments.
- Addresses that need not be specified can be omitted. Local variables corresponding to an omitted address are set to null.
- Addresses do not need to be specified alphabetically. They conform to word address format.
  I, J, and K need to be specified alphabetically, however.
  Argument specification I is always used for I, J, and K by setting bit 7 (IJK) of parameter No. 6008 to 1

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Example

- When bit 7 (IJK) of parameter No. 6008 is 0, I_J_K_ means that I = #4, J = #5, and K = #6 while K_J_I_ means K = #6, J = #8, and I= #10 because argument specification II is used.
- When bit 7 (IJK) of parameter No. 6008 is 1, K_J_I_ means that I = #4, J = #5, and K = #6, which is the same as with I_J_K_, because argument specification I is used.

- Argument specification II
  Argument specification II uses A, B, and C once each and uses I, J, and K up to ten times. Argument specification II is used to pass values such as 3-dimensional coordinates as arguments.

Address	Variable number
A	#1
B	#2
C	#3
I_1	#4
J_1	#5
K_1	#6
I_2	#7
J_2	#8
K_2	#9
I_3	#10
J_3	#11

Address	Variable number
IK_3	#12
I_4	#13
J_4	#14
K_4	#15
I_5	#16
J_5	#17
K_5	#18
I_6	#19
J_6	#20
K_6	#21
I_7	#22

Address	Variable number
J_7	#23
K_7	#24
I_8	#25
J_8	#26
K_8	#27
I_9	#28
J_9	#29
K_9	#30
I_{10}	#31
J_{10}	#32
K_{10}	#33

- Subscripts of I, J, and K for indicating the order of argument specification are not written in the actual program.

NOTE

When bit 7 (IJK) of parameter No. 6008 is 1, argument II cannot be used.

Limitation

- Format

G65 must be specified before any argument.

- Mixture of argument specifications I and II

The CNC internally identifies argument specification I and argument specification II. If a mixture of argument specification I and argument specification II is specified, the type of argument specification specified later takes precedence.

[Example]

```
G65  A1.0  B2.0  I-3.0  I4.0  D5.0  P1000 ;

(Variables)
#1:1.0
#2:2.0
#3:
#4:-3.0
#5:
#6:
#7:4.0          5.0
```

When both the I4.0 and D5.0 arguments are commanded for variable #7 in this example, the latter, D5.0, is valid.

FRYER MACHINE SYSTEMS INC
Rob Zalensky

- Position of the decimal point

The units used for argument data passed without a decimal point correspond to the least input increment of each address.

> ⚠ **CAUTION**
> The value of an argument passed without a decimal point may vary according to the system configuration of the machine. It is good practice to use decimal points in macro call arguments to maintain program compatibility.

- Extended axis name

The axis address of an extended axis name cannot be specified as an argument. If an attempt is made to specify it, alarm PS0129, “'G' AS ARGUMENT” is issued.

M

When a value is specified with no decimal point, the number of decimal places is determined as follows.

Address	For a non-axis address	For an axis address
D, E, H, M, S, or T	0	
Q or R	α (NOTE 2)	
A, C, I, J, K, X, Y, or Z	α (NOTE 2)	β (NOTE 3)
B, U, V(NOTE 1), or W	0	β (NOTE 3)
Second auxiliary function	γ (NOTE 4)	

Address	Metric input	Inch input
F (G93 mode)	3	
F (G94 mode)	0	2
F (G95 mode)	2 (NOTE 5)	4 (NOTE 5)

NOTE

1 When V is used in a call using a specific code, the number of decimal places is determined according to the setting for the reference axis.

2 α is determined according to the increment system for the reference axis (axis specified with parameter No. 1031) as listed in the table in NOTE 3.

3 β is determined according to the increment system for the corresponding axis address as listed in the following table.

Increment system	Linear axis (metric input)	Linear axis (inch input)	Rotation axis
IS-A	2	3	2
IS-B	3	4	3
IS-C	4	5	4

When bit 7 (IPR) of parameter No. 1004 is set to 1, the above values from which 1 is subtracted are used. When the increment system for an axis is IS-A, however, the setting of bit 7 (IPR) of parameter No. 1004 is not effective. When calculator-type decimal notation for each axis is used (bit 0 (ADX) of parameter No. 3455 is set to 1), the number of decimal places is 0. When bit 7 (EAP) of parameter No. 3452 is set to 1, however, calculator-type decimal notation is not effective and the number of decimal places is determined as listed in the above table.

4 γ is determined according to the increment system for the reference axis (axis specified with parameter No. 1031) as listed in the following table. (When bit 7 (BDX) of parameter No. 3450 is set to 1, γ is also determined in the same way.)

FRYER MACHINE SYSTEMS INC Rob Zahensky

NOTE

Increment system for the reference axis	AUP (No.3450#0) = 0	AUP(No.3450#0) = 1			
		AUX (No.3405#0) = 0		AUX (No.3405#0) = 1	
		Metric	Inch	Metric	Inch
IS-A	0	2		2	3
IS-B		3		3	4
IS-C		4		4	5

5 When bit 1 (FR3) of parameter No. 1405 is 1, the values in the table need to be incremented by 1.
6 When calculator-type decimal notation is used (bit 0 (DPI) of parameter No. 3401 is set to 1), the number of decimal places is 0.

T

When a value is specified with no decimal point, the number of decimal places is determined as follows.

Address	For a non-axis address	For an axis address
H, M, Q, S, or T	0	
D	α	
R	0 or α (NOTE 6)	
A, B, C, I, J, K, U, V, W, X, Y, or Z	α (NOTE 1)	β (NOTE 2)
Second auxiliary function	γ (NOTE 3)	

Address	Metric input	Inch input
E, F (G98 mode)	0 (NOTE 4)	2 (NOTE 4)
E, F (G99 mode)	4	6

NOTE

1 α is determined according to the increment system for the reference axis (axis specified with parameter No. 1031) as listed in the table in NOTE 2.
2 β is determined according to the increment system for the corresponding axis address as listed in the following table.

Increment system	Linear axis (metric input)	Linear axis (inch input)	Rotation axis
IS-A	2	3	2
IS-B	3	4	3
IS-C	4	5	4

When bit 7 (IPR) of parameter No. 1004 is set to 1, the above values from which 1 is subtracted are used. When the increment system for an axis is IS-A, however, the setting of bit 7 (IPR) of parameter No. 1004 is not effective.
When calculator-type decimal notation for each axis is used (bit 0 (ADX) of parameter No. 3455 is set to 1), the number of decimal places is 0. When bit 7 (EAP) of parameter No. 3452 is set to 1, however, calculator-type decimal notation is not effective and the number of decimal places is determined as listed in the above table.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

3 γ is determined according to the increment system for the reference axis (axis specified with parameter No. 1031) as listed in the following table. (When bit 7 (BDX) of parameter No. 3450 is set to 1, γ is also determined in the same way.)

Increment system for the reference axis	AUP (No.3450#0) = 0	AUP(No.3450#0) = 1			
		AUX (No.3405#0) = 0		AUX (No.3405#0) = 1	
		Metric	Inch	Metric	Inch
IS-A	0	2		2	3
IS-B		3		3	4
IS-C		4		4	5

4 When bit 2 (FM3) of parameter No. 1404 is 1, the values in the table need to be incremented by 3.

5 When calculator-type decimal notation is used (bit 0 (DPI) of parameter No. 3401 is set to 1), the number of decimal places is 0.

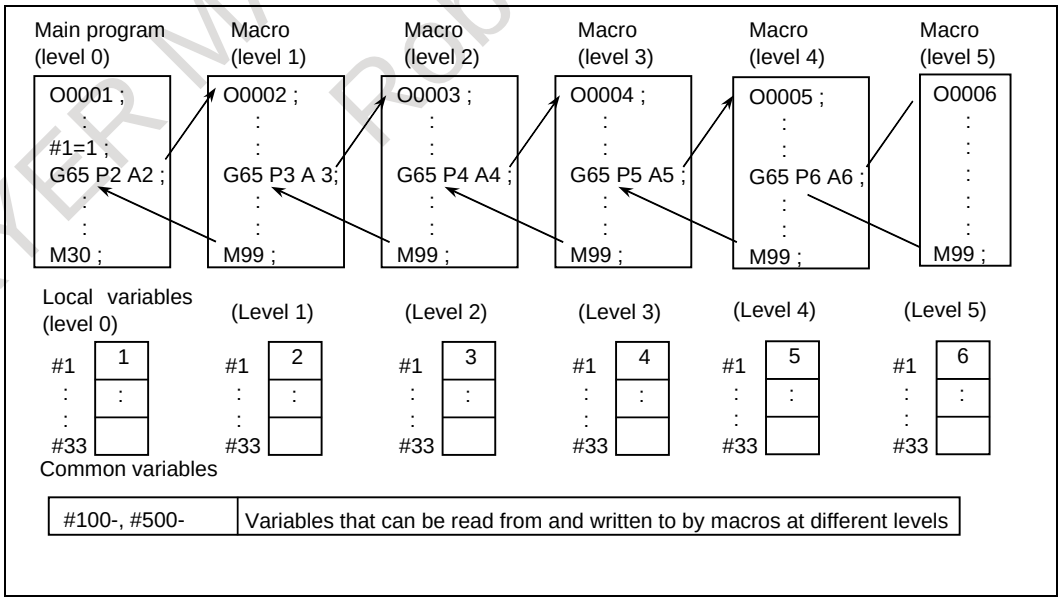
6 If bit 2 (DPD) of parameter No. 6019 is 0, the number of decimal places is 0, and if DPD is 1, it is α.

- Call nesting

Macro calls can be nested to a depth of up to five levels including simple calls (G65) and modal calls (G66/G66.1). Subprogram calls can be nested to a depth of up to 15 levels including macro calls.
A macro program can also be called during MDI operation in the same way.

- Local variable levels

- Local variables from level 0 to 5 are provided for nesting.
- The level of the main program is 0.
- Each time a macro is called (with G66, G66.1, Ggg, or Mmm), the local variable level is incremented by one. The values of the local variables at the previous level are saved in the CNC.
- When M99 is executed in a macro program, control returns to the calling program. At that time, the local variable level is decremented by one; the values of the local variables saved when the macro was called are restored.

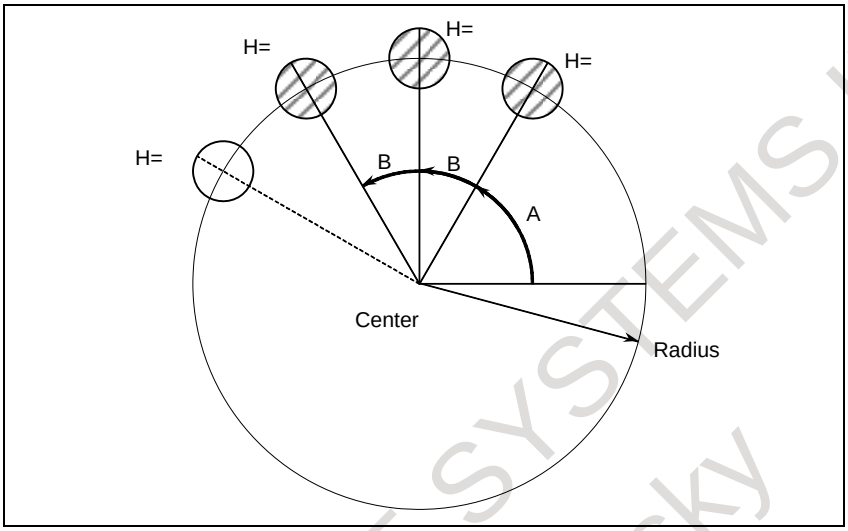


FRYER MACHINE SYSTEMS INC Rob Zabensky

Sample program (bolt hole circle)

M

A macro is created which drills H holes at intervals of B degrees after a start angle of A degrees along the periphery of a circle with radius I.
The center of the circle is (X,Y). Commands can be specified in either the absolute or incremental programming.
To drill in the clockwise direction, specify a negative value for B.



- Calling format

```
G65 P9100 Xx Yy Zz Rr Ff Ii Aa Bb Hh ;
```

X : X coordinate of the center of the circle (absolute or incremental programming) ..... (#24)
Y : Y coordinate of the center of the circle (absolute or incremental programming) ..... (#25)
Z : Hole depth ..... (#26)
R : Coordinates of an approach point ..... (#18)
F : Cutting feedrate ..... (#9)
I : Radius of the circle ..... (#4)
A : Drilling start angle ..... (#1)
B : Incremental angle (clockwise when a negative value is specified) ..... (#2)
H : Number of holes ..... (#11)

- Program calling a macro program

```
O0002;
G90 G92 X0 Y0 Z100.0;
G65 P9100 X100.0 Y50.0 R30.0 Z-50.0 F500 I100.0 A0 B45.0 H5;
M30;
```

- Macro program (called program)

```
O9100;
#3=#4003; ..............................  Stores G code of group 3.
G81 Z#26 R#18 F#9 K0; (Note) ..  Drilling cycle.
                                 Note: L0 can also be used.
IF[#3 EQ 90]GOTO 1; ............  Branches to N1 in the G90 mode.
#24=#5001+#24; .................  Calculates the X coordinate of the center.
#25=#5002+#25;  ................  Calculates the Y coordinate of the center.
N1 WHILE[#11 GT 0]DO 1; .....  Until the number of remaining holes reaches 0
#5=#24+#4*COS[#1]; ............  Calculates a drilling position on the X-axis.
```

FAYER MACHINE SYSTEMS INC
Rob Zahensky

#6=#25+#4*SIN[#1]; ..... Calculates a drilling position on the Y-axis.
G90 X#5 Y#6;..... Performs drilling after moving to the target position.
#1=#1+#2;..... Updates the angle.
#11=#11-1;..... Decrements the number of holes.
END 1;
G#3 G80;..... Returns the G code to the original state.
M99;

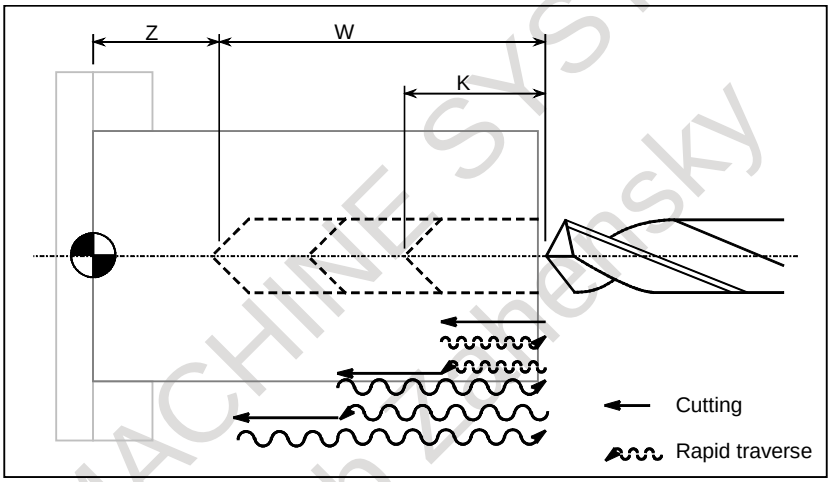
Meaning of variables:

#3: Stores the G code of group 3.
#5: X coordinate of the next hole to drill
#6: Y coordinate of the next hole to drill

Sample program (Drill cycle)

T

Move the tool beforehand along the X- and Z-axes to the position where a drilling cycle starts. Specify Z or W for the depth of a hole, K for the depth of a cut, and F for the cutting feedrate to drill the hole.



- Calling format

G65 P9100 { Zz / Ww } Kk Ff ;

Z : Hole depth (absolute programming)
W : Hole depth (incremental programming)
K : Cutting amount per cycle
F : Cutting feedrate

- Program calling a macro program

```
O0002;
G50 X100.0 Z200.0 ;
G00 X0 Z102.0 S1000 M03 ;
G65 P9100 Z50.0 K20.0 F0.3 ;
G00 X100.0 Z200.0 M05 ;
M30
```

- Macro program (called program)

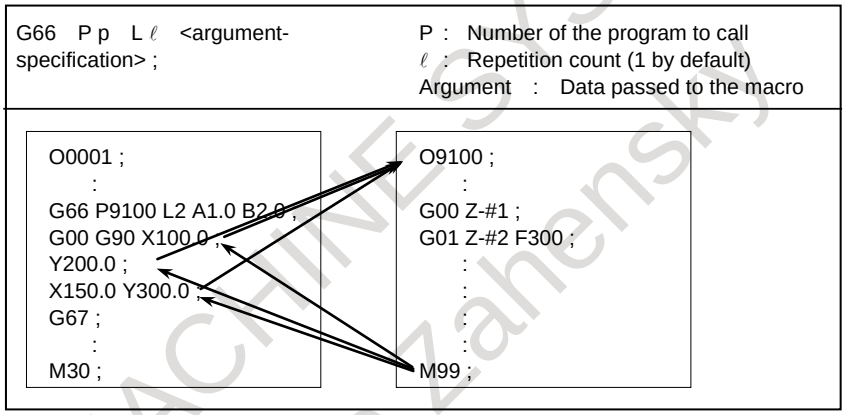
O9100;
#1=0 ;..... Clear the data for the depth of the current hole.
#2=0 ;..... Clear the data for the depth of the preceding hole.

FRYE MACHINE SYSTEMS INC Rob Zuhlensky

```
     IF [#23 NE #0] GOTO 1 ;..................... If incremental programming, specifies the jump to
                                                  N1.
     IF [#26 EQ #0] GOTO 8 ; .................... If neither Z nor W is specified, an error occurs.
     #23=#5002-#26 ; ............................ Calculates the depth of a hole.
N1   #1=#1+#6 ;..................................... Calculates the depth of the current hole.
     IF [#1 LE #23] GOTO 2 ; .................... Determines whether the hole to be cut is too deep?
     #1=#23 ;.................................... Clamps at the depth of the current hole.
N2   G00 W-#2 ; ......................................... Moves the tool to the depth of the preceding hole at
                                                  the cutting feedrate.
     G01 W- [#1-#2] F#9 ;........................... Drills the hole.
     G00 W#1 ;................................... Moves the tool to the drilling start point.
     IF [#1 GE #23] GOTO 9 ; .................... Checks whether drilling is completed.
     #2=#1 ;..................................... Stores the depth of the current hole.
     GOTO 1 ;
N9   M99
N8   #3000=1 (NOT Z OR W COMMAND).. Issues an alarm.
```

16.10.2 Modal Call: Call After the Move Command (G66)

Once G66 is issued to specify a modal call a macro is called after a block specifying movement along axes is executed. This continues until G66 is issued to cancel a modal call.



Explanation

- **Call**
- After G66, specify at address P a program number subject to a modal call.
- When the number of repetitions is required, a number from 1 to 99999999 can be specified at address L.
- As with a simple call (G65), data passed to a macro program is specified in arguments.
- In the G66 mode, a macro or subprogram can be called.

- **Cancellation**

When a G67 code is specified, modal macro calls are no longer performed in subsequent blocks.

- **Call nesting**

Macro calls can be nested to a depth of up to five levels including simple calls (G65) and modal calls (G66/G66.1). Subprogram calls can be nested to a depth of up to 15 levels including macro calls.

- **Modal call nesting**

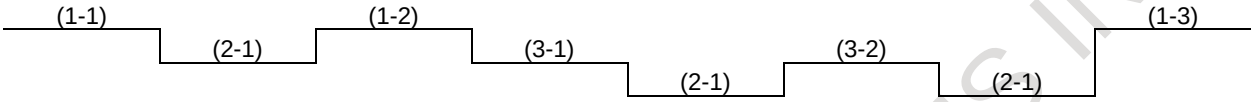
For a single modal call (when G66 is specified only once), each time the move command is executed, the specified macro is called. When nested modal macro calls are specified, the macro at the next higher level is called each time the move command for a macro call is executed.
Macros are called in reverse order in which they are specified. Each time G67 is issued, the macros are canceled one by one in reverse order in which they are specified.

FRYER MACHINE SYSTEMS INC
Rob Zahenskij

[Example]

```
G66 P9100 ;          O9100 ;          O9200 ;
X10.0 ;    (1-1)     Z50.0 ;   (2-1)  X60.0 ;   (3-1)
G66 P9200 ;          M99 ;            Y70.0 ;   (3-2)
X15.0 ;    (1-2)                      M99;
G67 ;      Cancels P9200.
G67 ;      Cancels P9100.
X-25.0 ;   (1-3)
```

Execution order of the above program (blocks not containing the move command omitted)



* No modal call is performed after (1-3) because the mode is not the macro call mode.

Limitation

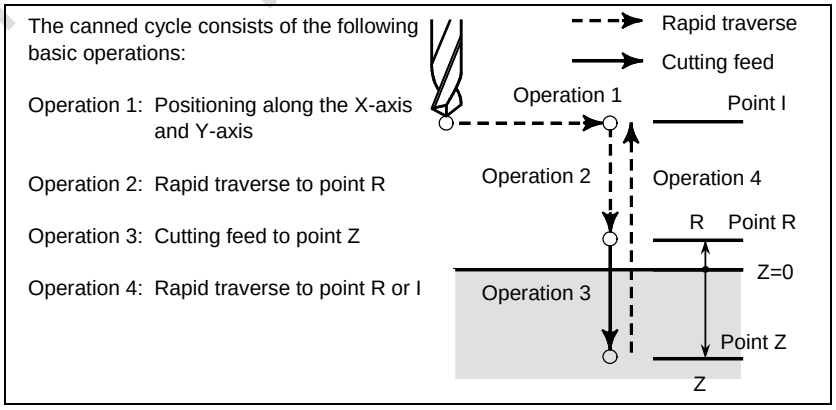
- G66 and G67 blocks are specified in pairs in the same program. If a G67 code is specified not in the G66 mode, an alarm PS1100, “CANCEL WITHOUT MODAL CALL” occurs. Bit 0 (G67) of parameter No. 6000 can be set to 1 to specify that the alarm does not occur in this case.
- In a G66 block, no macros can be called. Local variables (arguments) are set, however.
- G66 needs to be specified before any arguments.
- No macros can be called in a block which contains a code such as a auxiliary function that does not involve movement along an axis.
- Local variables (arguments) can only be set in G66 blocks. Note that local variables are not set each time a modal call is performed.

NOTE
If M99 is specified in a block in which a call is performed, it is executed after the call is performed.

Sample program

M

The same operation as the drilling canned cycle G81 is created using a custom macro and the machining program makes a modal macro call. For program simplicity, all drilling data is specified using absolute values.



- Calling format

```
G66   P9110   Zz   Rr   Ff   Ll ;
```

Z : Coordinates of position Z (absolute programming only)..... (#26)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

R : Coordinates of position R (absolute programming only)..... (#18)
F : Cutting feedrate..... (#9)
L : Repetition count

- Program that calls a macro program

```
O0001;
G28 G91 X0 Y0 Z0;
G92 X0 Y0 Z50.0;
G00 G90 X100.0 Y50.0;
G66 P9110 Z-20.0 R5.0 F500;
G90 X20.0 Y20.0;
X50.0;
Y50.0;
X70.0 Y80.0;
G67;
M30;
```

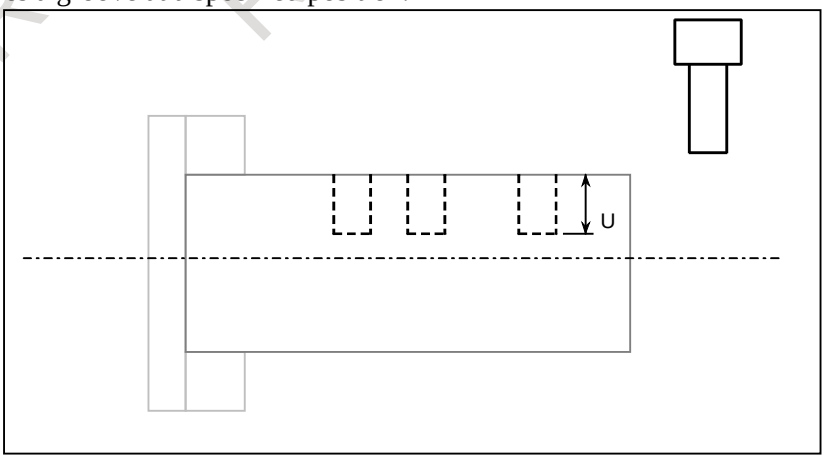
- Macro program (program called)

```
O9110;
    #1=#4001;  ..........Stores G00/G01.
    #3=#4003;..........Stores G90/G91.
    #4=#4109;..........Stores the cutting feedrate.
    #5=#5003;..........Stores the Z coordinate at the start of drilling.
    G00 G90 Z#18; .....Positioning at position R
    G01 Z#26 F#9;.....Cutting feed to position Z
    IF[#4010 EQ 98]GOTO 1; ...Return to position I
    G00 Z#18;..........Positioning at position R
    GOTO 2;
N1 G00 Z#5;.............Positioning at position I
N2 G#1 G#3 F#4;.........Restores modal information.
    M99;
```

Sample program

T

This program makes a groove at a specified position.



- Calling format

G66 P9110 Uu Ff

U : Groove depth (incremental programming)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

F : Cutting feed of grooving

- Program that calls a macro program

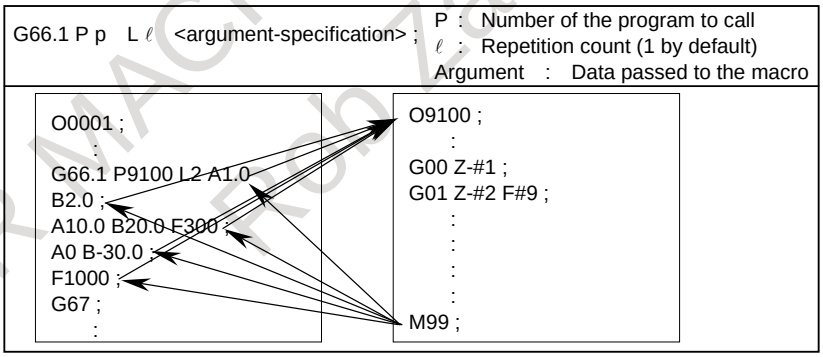
O0003 ;
G50 X100.0 Z200.0 ;
S1000 M03 ;
G66 P9110 U5.0 F0.5 ;
G00 X60.0 Z80.0 ;
Z50.0 ;
Z30.0 ;
G67 ;
G00 X00.0 Z200.0 M05 ;
M30;

- Macro program (program called)

O9110 ;
G01 U - #21 F#9 ; ..... Cuts the workpiece.
G00 U#21 ;..... Retracts the tool.
M99 ;

16.10.3 Modal Call: Each Block Call (G66.1)

In this macro call mode, the specified macro is unconditionally called for each NC command block. All data other than O, file name, N, and G codes that is specified in each block is not executed and is used as arguments. (The G code in the block in which G66.1 is specified is not used as an argument. Only the last G code specified in subsequent blocks is used as an argument.)
The NC command blocks having O or file name N have the same effect as when G65P is specified in the next position, the other NC command blocks have the same effect as when G65P is specified at the beginning.



[Example]
In the G66.1 P100 ; mode,
N001 G01 G91 X100 Y200 D1 R1000 ;
is the same as
N001 G65 P100 G01 G91 X100 Y200 D1 R1000 ;

Explanation

- Call

- After G66.1, specify at address P a program number subject to a modal call.
- When a number of repetitions is required, a number from 1 to 99999999 can be specified at address L.
- As with a simple call (G65), data passed to a macro program is specified in arguments.
- In the G66.1 mode, a macro or subprogram can be called.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Cancellation

When a G67 code is specified, modal macro calls are no longer performed in subsequent blocks.

- Call nesting

Macro calls can be nested to a depth of up to five levels including simple calls (G65) and modal calls (G66/G66.1). Subprogram calls can be nested to a depth of up to 15 levels including macro calls.

- Modal call nesting

For a single modal call (when G66.1 is specified only once), the specified macro is called for each NC command block. When nested modal macro calls are specified, the macro at the next higher level is also called in a block within a called macro in which an NC command is specified.
Macros are called in reverse order in which they are specified. Each time G67 is issued, the macros are canceled one by one in reverse order in which they are specified.

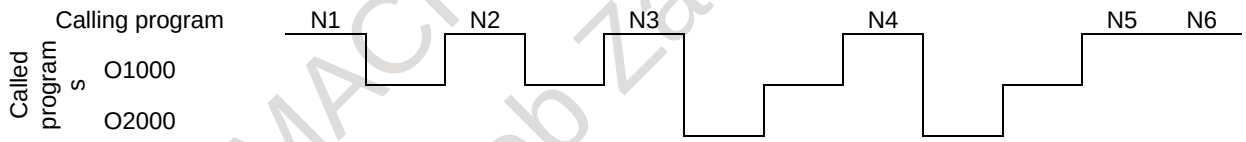
[Example] Axis specification switching

```
N1 G66.1 P1000 X10.0 ;          →  Calls O1000 and executes Y10.0.
N2 X20.0 ;                      →  Calls O1000 and executes Y20.0.
N3 G66.1 P2000 Y10.0 Z20.0 ;    →  Calls O2000 and executes Y20.0 Z10.0. Then, calls O1000 and
                                   executes X20.0 Z10.0.
N4 X10.0 Y20.0 Z30.0 ;          →  Calls O2000 and executes X10.0 Y30.0 Z20.0. Then, calls O1000
                                   and executes X30.0 Y10.0 Z20.0.
N5 G67 ;                        →  Cancels P2000.
N6 G67 ;                        →  Cancels P1000.

O1000 X#25 Y#24 Z#26 ;          (X-Y switching)
M99 ;

O2000 X#24 Y#26 Z#25 ;          (Y-Z switching)
M99 ;
```

Execution order of the above program (blocks not containing the move command omitted)



In N1 and N2 blocks, O1000 is called and the X and Y specifications are made to change places.
In N3 and N4 blocks, O2000 is called first and the Y and Z specifications are made to change places. For the switched specification, O1000 is called and the X and Y specifications are made to change places. Therefore, the X, Y, and Z specifications are switched to the Z, X, and Y specifications.

Limitation

- G66.1 and G67 blocks are specified in pairs in the same program. If a G67 code is specified not in the G66.1 mode, an alarm PS1100, “CANCEL WITHOUT MODAL CALL” occurs. Bit 0 (G67) of parameter No. 6000 can be set to 1 to specify that the alarm does not occur in this case.
- G66.1 block
  - (a) In a G66.1 block, a macro is called.
  - (b) The correspondence between addresses specified as arguments and variables is the same as for simple calls.
- Block following a G66.1 block in which a call is performed (not including G66.1 blocks)
  - (a) Addresses G, P, and L can also be used as arguments. Address G corresponds to #10; address L to #12; address P to #16. However, the restrictions on the input format of normal NC commands are put on the data. For example, ;G1000. P0.12 L-4 cannot be specified.
  - (b) When multiple G codes are specified, only the last G code is used as an argument. O, file name, and N codes and G codes not in group 00 are passed to the next and subsequent blocks.
- In the G66.1 mode, G10 cannot be specified.

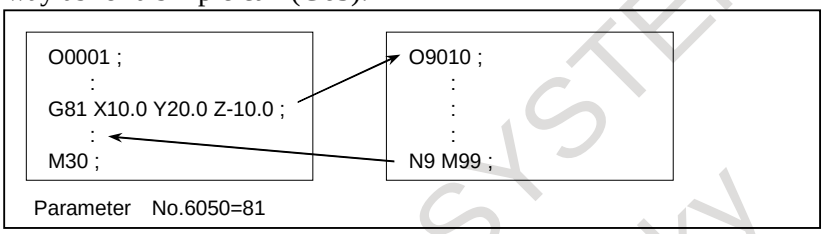
FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

1 In a block in which only an O number, file name, sequence number, EOB, macro statement, or M99 command is specified, a macro is not called for each block.
2 In each block, when an address other than O, file name, or N is specified, it is assumed to be an NC command and a macro is called for each block. When N is specified following an address other than O, file name, or N, it is used as an argument. In this case, N corresponds to variable #14 and the number of decimal places is 0.
3 If M99 is specified in a block in which a macro is called, it is executed after the call is performed.

16.10.4 Macro Call Using a G Code

By setting a G code number used to call a macro program in a parameter, the macro program can be called in the same way as for a simple call (G65).



Explanation

By setting a G code number from -9999 to 9999 used to call a custom macro program (O9010 to O9019) in the corresponding parameters Nos.6050 to 6059, the macro program can be called in the same way as with G65. To call custom macro program O9040 to O9049 using a G code with a decimal point, set bit 0 (DPG) of parameter No. 6007 to 1 and set the G code number in the corresponding parameters Nos. 6060 to 6069.

The number of decimal places of a G code is 1. Set the value obtained by multiplying a desired G code number by 10 in the corresponding parameter.

[Example] When parameter No. 6060 is set to 234, custom macro program O9040 is called using G23.4.

When a negative G code is set, a modal call is performed. In this case, bit 3 (MGE) of parameter No. 6007 can be set to select the G66 or G66.1 mode.

For example, when a parameter is set so that macro program O9010 can be called with G81, a user-specific cycle created using a custom macro can be called without modifying the machining program.

- Correspondence between parameter numbers and program numbers

G code with no decimal point		G code with a decimal point	
Parameter number	Program number	Parameter number	Program number
6050	O9010	6060	O9040
6051	O9011	6061	O9041
6052	O9012	6062	O9042
6053	O9013	6063	O9043
6054	O9014	6064	O9044
6055	O9015	6065	O9045
6056	O9016	6066	O9046
6057	O9017	6067	O9047
6058	O9018	6068	O9048
6059	O9019	6069	O9049

FRYER MACHINE SYSTEMS INC
Rob Zaranosky

- Repetition

As with a simple call, the number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used.

Limitation

- Nesting of calls using G codes

- To call another program in a program called using a G code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using an M, T, or S code, second auxiliary function, or specific code can be performed in a program called using a G code.

- G10 command

If bit 3 (MGE) of parameter No. 6007 is set to 0, G10 cannot be specified in the modal call mode.

16.10.5 Macro Call Using a G Code (Specification of Multiple Definitions)

By setting the starting G code number used to call a macro program, the number of the starting program to be called, and the number of definitions, macro calls using multiple G codes can be defined.

Explanation

As many custom macros as the number specified in parameter No. 6040 can be called using as many G codes as the number specified in parameter No. 6040. The numeric value set in parameter No. 6038 indicates the starting G code number and the program number set in parameter No. 6039 indicates the starting program number. To disable this type of call, set 0 in parameter No. 6040.
When a negative G code is set in parameter No. 6038, modal calls are performed. In this case, bit 3 (MGE) of parameter No. 6007 can be set to select the G66 or G66.1 mode.
The number of repetitions and argument specification are set in the same way as with a macro call using a G code.
[Example]

Set parameter No. 6038 to 900, parameter No. 6039 to 1000, and parameter No. 6040 to 100.
G900 → O1000
G901 → O1001
G902 → O1002
:
G999 → O1099

Custom macro calls (simple calls) for 100 combinations are defined as shown above. When parameter No. 6038 is changed to -900, custom macro calls (modal calls) for the same combinations are defined.

NOTE

1 The calls defined by this setting become all invalid in the following cases:
  <1> A value outside the valid data range is set in one of the above parameters.
  <2> (No. 6039 + No. 6040 - 1) > 99999999
2 Simple and modal calls cannot be mixed in the specification.
3 If the G code set in parameters Nos. 6050 to 6059 to call the corresponding macro program is within the G code range for calling programs using multiple G codes, the macro program corresponding to the G code set in parameters Nos. 6050 to 6059 is called.

FRYER MACHINE SYSTEMS INC
Rob Zabensky

NOTE
4 If bit 3 (MGE) of parameter No. 6007 is set to 0, G10 cannot be specified in the modal call mode.

16.10.6 Macro Call Using a G Code with a Decimal Point (Specification of Multiple Definitions)

When bit 0 (DPG) of parameter No. 6007, by setting the starting G code number with a decimal point used to call a macro program, the number of the starting program to be called, and the number of definitions, multiple macro calls using multiple G codes with a decimal point can be defined.

Explanation

As many custom macros as the number specified in parameter No. 6043 can be called using as many G codes with a decimal point as the number specified in parameter No. 6043. The numeric value set in parameter No. 6041 indicates the starting G code number with a decimal point and the program number set in parameter No. 6042 indicates the starting program number. To disable this type of call, set 0 in parameter No. 6043.
When a negative G code is set in parameter No. 6041, modal calls are performed. In this case, bit 3 (MGE) of parameter No. 6007 can be set to select the G66 or G66.1 mode.
The number of repetitions and argument specification are set in the same way as with a macro call using a G code.
[Example]

Set parameter No. 6041 to 900, parameter No. 6042 to 2000, and parameter No. 6043 to 100.
G90.0 → O2000
G90.1 → O2001
G90.2 → O2002
:
G99.9 → O2099

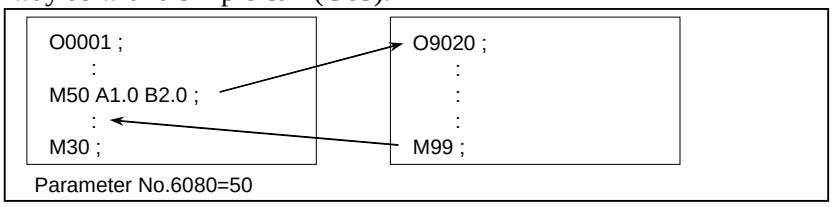
Custom macro calls (simple calls) for 100 combinations are defined as shown above. When parameter No. 6041 is changed to -900, custom macro calls (modal calls) for the same combinations are defined.

NOTE
1 The calls defined by this setting become all invalid in the following cases:
(1) A value outside the valid data range is set in one of the above parameters.
(2) (No. 6042 + No. 6043 - 1) > 99999999
(3) Bit 0 (DPG) of parameter No. 6007 is set to 0 (this setting invalidates a macro call using a G code with a decimal point).
2 Simple and modal calls cannot be mixed in the specification.
3 If the G code set in parameters Nos. 6060 to 6069 to call the corresponding macro program is within the G code range for calling programs using multiple G codes, the macro program corresponding to the G code set in parameters Nos. 6060 to 6069 is called.
4 If bit 3 (MGE) of parameter No. 6007 is set to 0, G10 cannot be specified in the modal call mode.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

16.10.7 Macro Call Using an M Code

By setting an M code number used to call a macro program in a parameter, the macro program can be called in the same way as with a simple call (G65).



Explanation

By setting an M code number from 3 to 99999999 used to call custom macro program O9020 to O9029 in the corresponding parameters Nos. 6080 to 6089, the macro program can be called in the same way as with G65.

- Correspondence between parameter numbers and program numbers

Parameter number	Corresponding program number
6080	O9020
6081	O9021
6082	O9022
6083	O9023
6084	O9024
6085	O9025
6086	O9026
6087	O9027
6088	O9028
6089	O9029

Example) When parameter No. 6080 is set to 990, O9020 is called using M990.

- Repetition

As with a simple call, the number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used. When bit 2 (MAA) of parameter No.6009 is set to 1, address G becomes an argument, too.

1. When bit 2 (MAA) of parameter No.6009 is set to 0
   The address commanded after call code become arguments. Argument specification I or argument specification II are used.
   Argument specification I

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	J	#5	S	#19
B	#2	K	#6	T	#20
C	#3	L	*2	U	#21
D	#7	M	#13 *3	V	#22
E	#8	M(Call code)	*4	W	#23
F	#9	N	#14 *5	X	#24
G	*1	P	#16	Y	#25
H	#11	Q	#17	Z	#26
I	#4	R	#18		

FRYER MACHINE SYSTEMS INC
Rob Zahensky

*1: Address G can not command. If address G is commanded, alarm PS0129 occurs.
*2: Address L is a number of times in which the macro call is repeated.
*3: Address M other than call code is passed to variable #13.
*4: Address M for call code does not become an argument.
*5: Address N is passed to variable #14.

2. When bit 2 (MAA) of parameter No.6009 is set to 1
The address commanded after call code become arguments. Address G becomes an argument, too. Argument specification I or argument specification II are used.
Argument specification I

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	J	#5	S	#19
B	#2	K	#6	T	#20
C	#3	L	*2	U	#21
D	#7	M	#13 *3	V	#22
E	#8	M(Call code)	*4	W	#23
F	#9	N	#14 *5	X	#24
G	#28 to #32 *1	P	#16	Y	#25
H	#11	Q	#17	Z	#26
I	#4	R	#18		

*1: The first five addresses G in ascending order of G code groups are used as arguments and passed to variables #28 to #32.
*2: Address L is a number of times in which the macro call is repeated.
*3: Address M other than call code is passed to variable #13.
*4: Address M for call code does not become an argument.
*5: Address N is passed to variable #14.

Limitation

- An M code used to call a macro program must be specified at the start of a block.
- To call another program in a program called using an M code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using an M code.

16.10.8 Macro Call Using an M Code (Specification of Multiple Definitions)

By setting the starting M code number used to call a macro program, the number of the starting program to be called, and the number of definitions, macro calls using multiple M codes can be defined.

Explanation

As many custom macros as the number specified in parameter No. 6049 can be called using as many M codes as the number specified in parameter No. 6049. The numeric value set in parameter No. 6047 indicates the starting M code number and the program number set in parameter No. 6048 indicates the starting program number. To disable this type of call, set 0 in parameter No. 6049.
The number of repetitions and argument specification are set in the same way as with a macro call using a M code.
[Example]

Set parameter No. 6047 to 90000000, parameter No. 6048 to 4000, and parameter No. 6049 to 100.
M90000000 → O4000
M90000001 → O4001

FRYER MACHINE SYSTEMS INC
Rob Zahensky

M90000002 → O4002
:
M90000099 → O4099

Custom macro calls (simple calls) for 100 combinations are defined as shown above.

NOTE

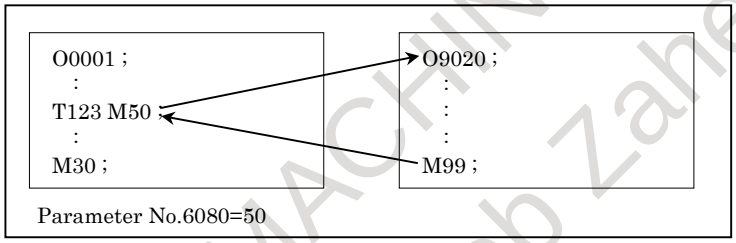
1 The calls defined by this setting become all invalid in the following cases:
  <1> A value outside the valid data range is set in one of the above parameters.
  <2> (No. 6048 + No. 6049 - 1) > 99999999
2 If the M code set in parameters Nos. 6080 to 6089 to call the corresponding macro program is within the M code range for calling programs using multiple M codes, the macro program corresponding to the M code set in parameters Nos. 6080 to 6089 is called.

16.10.9 Special Macro Call using M code

Usually, it is necessary to command M code for Macro Call using M code or Macro Call using M code (Specification of 3 sets) at the head of the block. Alarm PS0127 occurs if M code is commanded for macro call at the position of the block that is not the head.
Special Macro Call using M code is a function to execute macro call when call code is commanded at the position of the block that is not the head. By setting bit 0 (MSM) of parameter No. 6009 is set to 1, it becomes possible.

Example

When M code is commanded for a macro call at the position of the block that is not the head



When bit 0 (MSM) of parameter No. 6009 is set to 0, PS0127 alarm occurs.
When bit 0 (MSM) of parameter No. 6009 is set to 1, Macro Call using M code is executed. And, Address T becomes an argument and #20 is set to 123.0.

Explanation

When Special Macro Call using M code becomes possible because bit 0 (MSM) of parameter No. 6009 is set to 1, the execution of macro call changes by the position in which M code for macro call is commanded.

- Command position of call code

1. When M code for macro call is commanded at the head of the block
   It operates as usual Macro Call.
   For details, see Subsection "Macro Call Using an M Code".
2. When M code for macro call is commanded at the position of the block that is not the head
   It operates as Special Macro Call using M code.

- Repetition

Address L is also used as an argument, so that no repetition count can be specified.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Argument specification

All addresses are treated as arguments. However, the specifiable addresses and specification range conform to the specification address range of the CNC. Only argument specification I can be used.
When bit 2 (MAA) of parameter No. 6009 is set to 1, address G becomes an argument, too.

1. When bit 2 (MAA) of parameter No. 6009 is set to 0
   All addresses other than M for call code, G and L are treated as arguments. However, the specifiable addresses and specification range conform to the specification address range of the CNC. Only argument specification I can be used.
   Argument specification I

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	J	#5	S	#19
B	#2	K	#6	T	#20
C	#3	L	*2	U	#21
D	#7	M	#13 *3	V	#22
E	#8	M(Call code)	*4	W	#23
F	#9	N	#14 *5	X	#24
G	*1	P	#16	Y	#25
H	#11	Q	#17	Z	#26
I	#4	R	#18		

   *1: Even if address G is commanded, PS alarm does not occur. And address G does not become an argument.
   *2: Address L does not become a number of times in which the macro call is repeated, and does not become an argument.
   *3: Address M other than call code is passed to variable #13.
   *4: Address M for call code does not become an argument.
   *5: Address N is passed to variable #14, and becomes a sequence number.

2. When bit 2 (MAA) of parameter No. 6009 is set to 1
   All addresses are treated as arguments. However, the specifiable addresses and specification range conform to the specification address range of the CNC. Only argument specification I can be used. Address G, L and M for call code become arguments, too.
   Argument specification I

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	J	#5	S	#19
B	#2	K	#6	T	#20
C	#3	L	#12 *2	U	#21
D	#7	M	#13 *3	V	#22
E	#8	M(Call code)	#27 *4	W	#23
F	#9	N	#14 *5	X	#24
G	#28 to #32 *1	P	#16	Y	#25
H	#11	Q	#17	Z	#26
I	#4	R	#18		

   *1: The first five addresses G in ascending order of G code groups are used as arguments and passed to variables #28 to #32.
   *2: Address L is passed to variable #12, and does not become a number of times in which the macro call is repeated.
   *3: Address M other than call code is passed to variable #13.
   *4: Address M for call code is passed to variable #27.
   *5: Address N is passed to variable #14, and becomes a sequence number.

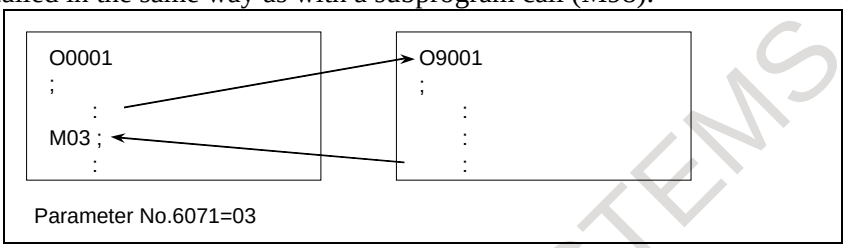
FRYER MACHINE SYSTEMS INC
Rob Zehensky

Limitation

- To call another program in a program called using an M code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using an M code.
- When multiple call commands are specified in the same block, the first call code is selected.

16.10.10 Subprogram Call Using an M Code

By setting an M code number used to call a subprogram (macro program) in a parameter, the macro program can be called in the same way as with a subprogram call (M98).



Explanation

By setting an M code number from 3 to 99999999 used to call subprogram O9001 to O9009 in the corresponding parameters Nos. 6071 to 6079, the subprogram can be called in the same way as with M98.

- Correspondence between parameter numbers and program numbers

Parameter number	Program number
6071	O9001
6072	O9002
6073	O9003
6074	O9004
6075	O9005
6076	O9006
6077	O9007
6078	O9008
6079	O9009

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

Argument specification is not allowed.

- M code

An M code in a macro program that has been called is treated as an ordinary M code.

Limitation

- To call another program in a program called using an M code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using an M code.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

16.10.11 Subprogram Call Using an M Code (Specification of Multiple Definitions)

By setting the starting M code number used to call a subprogram, the number of the starting subprogram to be called, and the number of definitions, subprogram calls using multiple M codes can be defined.

Explanation

As many subprograms as the number specified in parameter No. 6046 can be called using as many M codes as the number specified in parameter No. 6046. The numeric value set in parameter No. 6044 indicates the starting M code number and the numeric value set in parameter No. 6045 indicates the starting subprogram number. To disable this type of call, set 0 in parameter No. 6046.

[Example]

Set parameter No. 6044 to 80000000, parameter No. 6045 to 3000, and parameter No. 6046 to 100.

M80000000 → O3000
M80000001 → O3001
M80000002 → O3002
:
M80000099 → O3099

Subprogram calls for 100 combinations are defined as shown above.

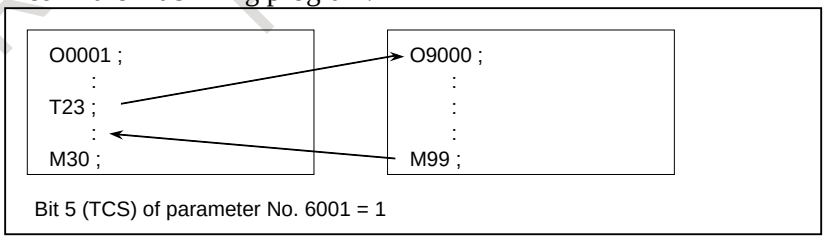
NOTE

1 The calls defined by this setting become all invalid in the following cases:
<1> A value outside the valid data range is set in one of the above parameters.
<2> (No. 6045 + No. 6046 - 1) > 99999999

2 If the M code set in parameters Nos. 6071 to 6079 to call the corresponding subprogram is within the M code range for calling subprograms using multiple M codes, the subprogram corresponding to the M code set in parameters Nos. 6071 to 6079 is called.

16.10.12 Subprogram Calls Using a T Code

By enabling subprograms to be called with a T code in a parameter, a subprogram can be called each time the T code is specified in the machining program.



Explanation

- Call

By setting bit 5 (TCS) of parameter No. 6001 to 1, subprogram O9000 can be called each time a T code is specified in a machining program. A T code specified in a machining program is assigned to common variable #149.

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

FRYER MACHINE SYSTEMS INC
Rob Zaborsky

- Argument specification

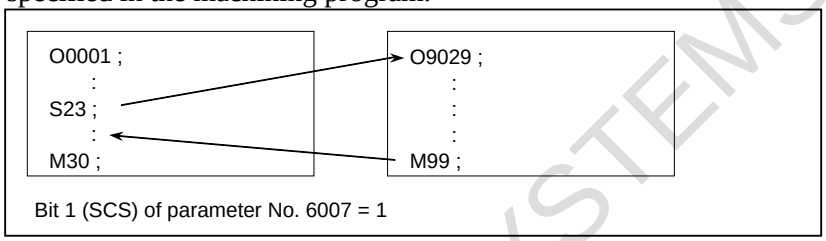
Argument specification is not allowed.

Limitation

- To call another program in a program called using a T code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using a T code.

16.10.13 Subprogram Calls Using an S Code

By enabling subprograms to be called with an S code in a parameter, a subprogram can be called each time the S code is specified in the machining program.



Explanation

- Call

By setting bit 1 (SCS) of parameter No. 6007 to 1, subprogram O9029 can be called each time a S code is specified in a machining program. An S code specified in a machining program is assigned to common variable #147.

- Repetition

As with a simple call, the number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

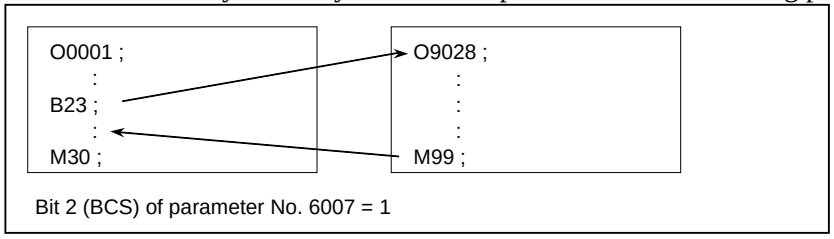
Argument specification is not allowed.

Limitation

- To call another program in a program called using an S code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using an S code.

16.10.14 Subprogram Calls Using a Secondary Auxiliary Function

By enabling subprograms to be called with a secondary auxiliary function in a parameter, a subprogram can be called each time the secondary auxiliary function is specified in the machining program.



FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

- Call

By setting bit 2 (BCS) of parameter No. 6007 to 1, subprogram O9028 can be called each time a secondary auxiliary function code is specified in a machining program. A secondary auxiliary function specified in a machining program is assigned to common variable #146.

- Repetition

As with a simple call, the number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

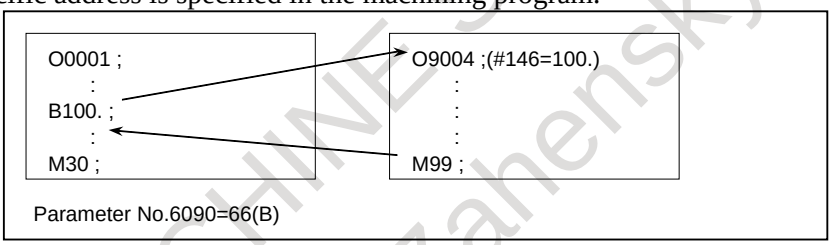
Argument specification is not allowed.

Limitation

- To call another program in a program called using a second auxiliary function, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using a second auxiliary function.

16.10.15 Subprogram Call Using a Specific Address

By enabling subprograms to be called with a specific address in a parameter, a subprogram can be called each time the specific address is specified in the machining program.



Explanation

- Call

By setting the code (ASCII code converted to decimal) corresponding to a specific address in parameter No. 6090 or No. 6091, the custom macro program, O9004 or O9005, corresponding to each parameter can be called when the specific address is specified in a machining program. The code value corresponding to a specific address specified in a machining program is assigned to the common variables (#146, #147). The table below indicates the addresses that can be set.

M

Address	Parameter setting	Address	Parameter setting
A	65	P	80
B	66	Q	81
D	68	R	82
F	70	S	83
H	72	T	84
I	73	V	86
J	74	X	88
K	75	Y	89
L	76	Z	90
M	77		

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE
When address L is set, the number of repetitions cannot be set.

T

Address	Parameter setting	Address	Parameter setting
A	65	L	76
B	66	M	77
F	70	P	80
H	72	Q	81
I	73	R	82
J	74	S	83
K	75	T	84

NOTE
When address L is set, the number of repetitions cannot be set.

- Correspondence between parameter numbers and program numbers and between the parameter numbers and common variables

Parameter number	Program number	Common variable
6090	O9004	#146
6091	O9005	#147

- Repetition

As with a simple call, the number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

Argument specification is not allowed.

Limitation

- To call another program in a program called using a specific code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using a specific code.

Sample program

By using the subprogram call function that uses M codes, the cumulative usage time of each tool is measured.

Conditions

- The cumulative usage time of each of tools T01 to T05 is measured.
  No measurement is made for tools with numbers greater than T05.
- The following variables are used to store the tool numbers and measured times:

#501	Cumulative usage time of tool number 1
#502	Cumulative usage time of tool number 2
#503	Cumulative usage time of tool number 3
#504	Cumulative usage time of tool number 4
#505	Cumulative usage time of tool number 5

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Usage time starts being counted when the M03 command is specified and stops when M05 is specified. System variable #3002 is used to measure the time during which the cycle start lamp is on. The time during which the machine is stopped by feed hold and single block stop operation is not counted, but the time used to change tools and pallets is included.

Operation check

- Parameter setting

Set 3 in parameter No. 6071, and set 5 in parameter No. 6072.

- Variable value setting

Set 0 in variables #501 to #505.

- Program that calls a macro program

```
O0001;
T01 M06;
M03;
      :
M05;..................... Changes #501.
T02 M06;
M03;
      :
M05;..................... Changes #502.
T03 M06;
M03;
      :
M05;..................... Changes #503.
T04 M06;
M03;
      :
M05;..................... Changes #504.
T05 M06;
M03;
      :
M05;..................... Changes #505.
M30;
```

- Macro program (program called)

```
O9001(M03);        ..................................... Macro to start counting
M01;
IF[#4120 EQ 0]GOTO 9; ............................ No tool specified
IF[#4120 GT 5]GOTO 9;............................ Out-of-range tool number
#3002=0;           ..................................... Clears the timer.
N9 M03;            ..................................... Rotates the spindle in the forward direction.
M99;

O9002(M05);        ..................................... Macro to end counting
M01;
IF[#4120 EQ 0]GOTO 9; ............................ No tool specified
IF[#4120 GT 5]GOTO 9;............................ Out-of-range tool number
#[500+#4120]=#3002+#[500+#4120]; ....... Calculates cumulative time.

N9 M05; ..................................................... Stops the spindle.
M99;
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

16.11 MACRO CALL ARGUMENT FOR AXIS NAME EXPANSION

Macro argument can be specified to the address of axis name expansion. By setting the parameter No. 11647, the address of axis name expansion is allocated to local variable number (#1 - #33).
This function is effective to not only an axis name expansion but also a usual axis address of one character. A usual axis address of one character can allocate to the local variable number (#1-#33).

Example

In case of the following axis configuration, if all arguments are specified, the relationship between the parameter No. 11647 and local variable number is as follows.

Axis name	Parameter (No.11647)	Local variable
XA1 (*1)	1	Argument is set to #1
XA2 (*1)	2	Argument is set to #2
Y	0	Argument is set to #25
Z	21	Argument is set to #21
C	0	Argument is set to #3
C2 (*1)	22	Argument is set to #22

(*1) Axis name expansion

```
G65   XA1=10.   XA2=20.   Y30.   Z40.   C50.   C2=60.   P1000 ;

(Variable)
#1: 10.0
#2: 20.0
#25: 30.0
#21: 40.0
#3: 50.0
#22: 60.0
```

Allocating the same local variable

Please do not allocate the same local variable to 2 or more arguments. If the same local variable is allocated, the argument specified later becomes effective.

Example

When 2 arguments of XA1=10. and D20.0 are commanded to local variable #7, the later argument D20.0 becomes effective.

Axis name	Parameter (No.11647)	Local variable
XA1 (*1)	7	Argument is set to #7

(*1) Axis name expansion

```
G65   XA1=10.   D20.0   P1000 ;

(Variable)
#7: 10.0   20.0
```

FRASER MACHINE SYSTEMS INC
Bob Zahensky

Using the same axis name

When using the same axis name, the parameter No. 11647 setting of the smallest axis number becomes effective. The setting of the other axis becomes invalid.

Example

Since the same axis name exists, setting of 2, 4, 6 axes is invalid.

Axis name	Parameter (No.11647)	Local variable
XA1 (*1)	0	The argument for XA1 is invalid
XA1 (*1)	2	Invalid setting
YA2 (*1)	3	Argument is set to #3
YA2 (*1)	4	Invalid setting
C	5	Argument is set to #5
C	6	Invalid setting

(*1) Axis name expansion

G65 XA1=10. YA2=20. C30. P1000 ;
Because the argument for XA1 is invalid, the alarm PS0129, “USE 'G' AS ARGUMENT” occurs.

G65 YA2=20. C30. P1000 ;

(Variable)
#3: 20.0
#5: 30.0

The setting range of the parameter

The setting range of the parameter No. 11647 is 0,1-33. This corresponds to local variable number (#1-#33). When other values are set, the setting of the axis is invalid. Therefore, when using the axis name expansion, the alarm PS0129, “USE 'G' AS ARGUMENT” occurs by commanding the axis. If the axis does not use the axis name expansion, the argument is assigned to original local variable (#1-#33).

16.12 PROCESSING MACRO STATEMENTS

For smooth machining, the CNC looks ahead to the NC statement to be performed next. This operation is referred to as buffering. For example, in looking ahead due to AI contour control, up to 1000 blocks of NC statements are buffered.
In the cutter compensation mode (G41 or G42), the CNC looks ahead to the NC statements in at least three blocks ahead to calculate intersections.
Macro statements for arithmetic expressions and conditional branches are processed as soon as they are read into the buffer. Therefore, the timing of the macro statement execution is not always the specified order.
At the blocks containing M00, M01, M02 or M30, blocks containing M-codes for which buffering is suppressed by setting parameters Nos. 3411 to 3420, 3421 to 3432, and 11290 to 11299, and blocks containing prevention buffering G codes such as G31 or G53, the CNC stops looking ahead to the NC statement after that. Then, the stop of the macro statement execution is guaranteed until such M codes or G codes complete its execution.

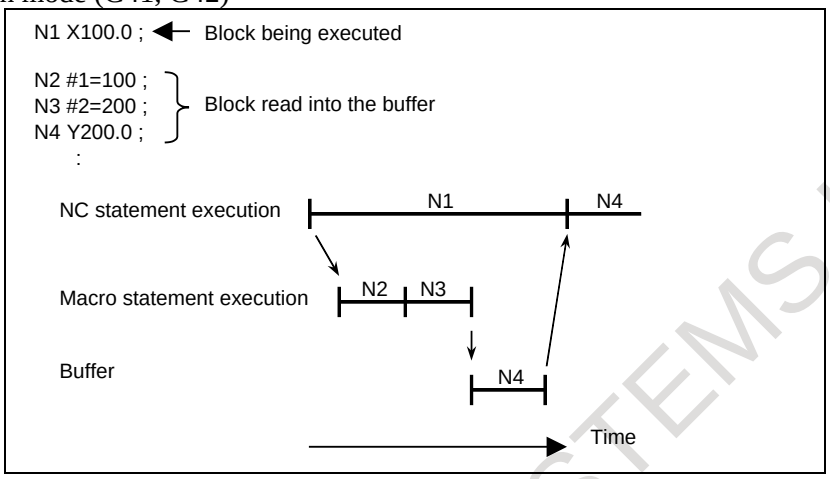
FRYER MACHINE SYSTEMS INC
Rob Zehensky

Explanation

- When the next block is buffered

Example 1:

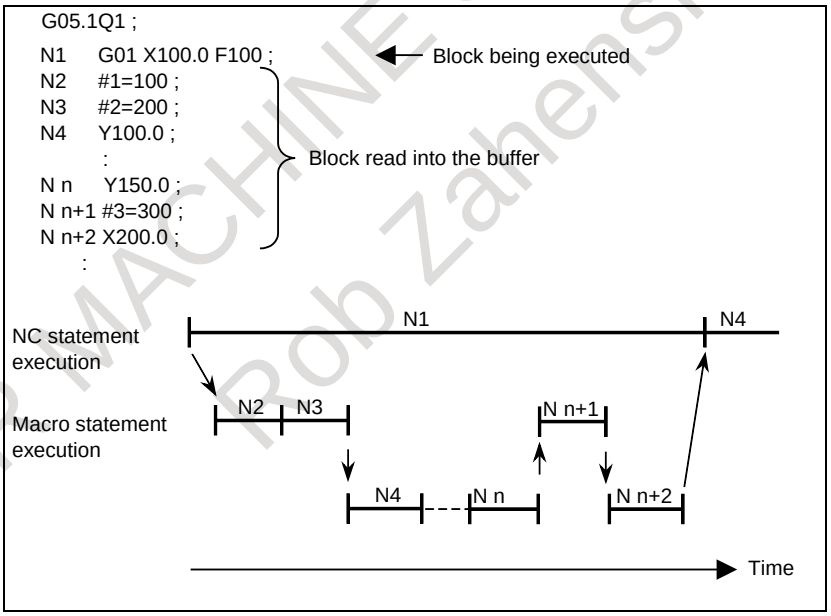
Looking ahead to the next block when the system is not in AI contour control mode or cutter compensation mode (G41, G42)



While N1 is being executed, the next NC statement in N4 is read into the buffer.
The macro statements in N2 and N3 between N1 and N4 are processed while N1 is being executed.

Example 2:

In AI contour control mode

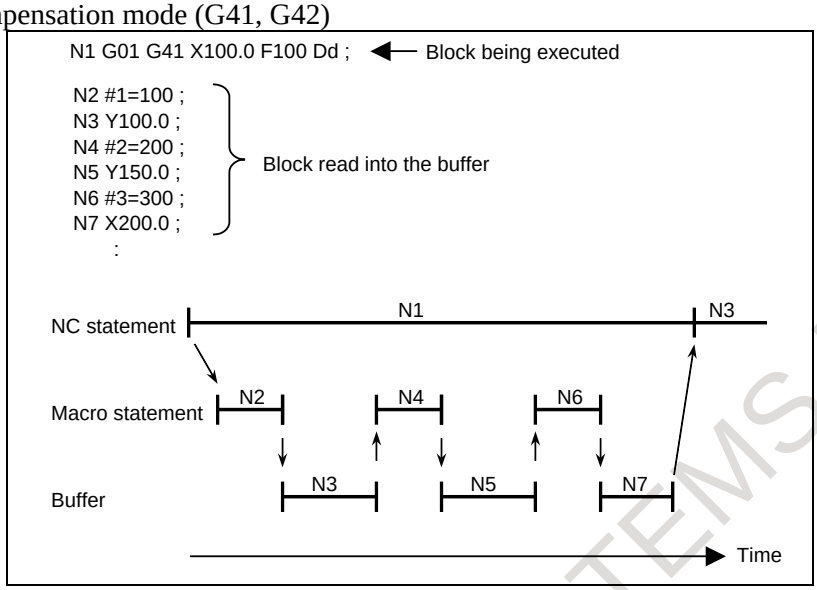


While N1 is being executed, the NC statements in up to the subsequent 1000 blocks (Nn+2) are read into the buffer. The macro statements in N2, N3, and Nn+1 between the blocks are processed while N1 is being executed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Example 3:

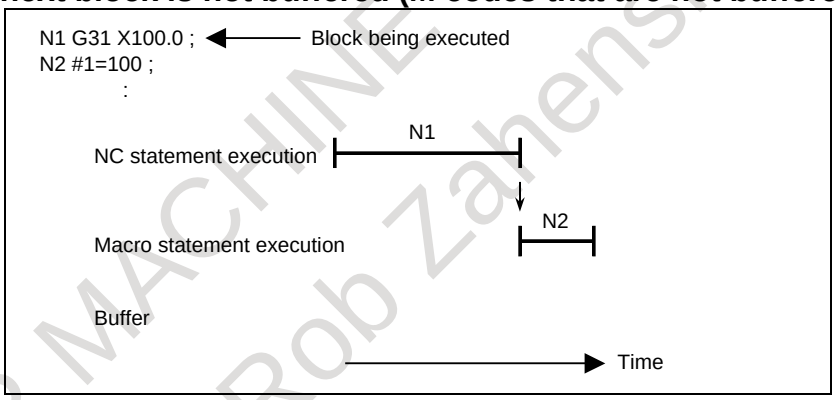
In cutter compensation mode (G41, G42)



While N1 is being executed, the NC statements in the subsequent three blocks (blocks up to N7) are read into the buffer.

The macro statements in N2, N4, and N6 between the blocks are processed while N1 is being executed.

- When the next block is not buffered (M codes that are not buffered, G31, etc.)



⚠CAUTION

In case that you need to execute the macro statement after completing the block just before the macro statement, specify M code or G code that is not buffered just before the macro statement. Specially, in case of reading/writing the system variables to control signals, coordinates, offset value, etc., it may different system variable data by the timing of the NC statement execution. To avoid this phenomenon, specify such M codes or G codes before the macro statement, if necessary.

Diagnosis data

Diagnosis	1493	Number of macro statement blocks executed by custom and execution macros

[Data type] 2-word

[Unit of data] Block

The number of macro statement blocks executed by custom and execution macros per 1024 ms is displayed.

This data provides an indication of the actual macro statement processing speed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

16.13 REGISTERING CUSTOM MACRO PROGRAMS

Custom macro programs are similar to subprograms. They can be registered and edited in the same way as subprograms. The storage capacity is determined by the total length of tape used to store both custom macros and subprograms.

16.14 CODES AND RESERVED WORDS USED IN CUSTOM MACROS

In addition to the codes used in ordinary programs, the following codes are used in custom macro programs.

Explanation

- Codes

(1) When the ISO code is used or when bit 4 (ISO) of parameter No. 6008 is set to 0
(The codes are represented in hexadecimal.)

Meaning	Code	Meaning	Code
*	0AAh	?	03Fh
=	0BDh	@	0C0h
#	0A3h	&	0A6h
[	0DBh	_	05Fh
]	0DDh	O	0CFh

(2) When the EIA code is used or when the ISO code is used with bit 4 (ISO) of parameter No. 6008 set to 1

Meaning	Code	Meaning	Code
*	Code set in parameter No. 6010	?	Code set in parameter No. 6015
=	Code set in parameter No. 6011	@	Code set in parameter No. 6016
#	Code set in parameter No. 6012	&	Code set in parameter No. 6017
[	Code set in parameter No. 6013	_	Code set in parameter No. 6018
]	Code set in parameter No. 6014		

For O, the same code as for O indicating a program number is used. Set a hole pattern for each of *, =, #, [,], ?, @, &, and _ in the ISO or EIA code in the corresponding parameters Nos. 6010 to 6018.
The code 00h cannot be used. The code indicating an alphabetic character can be used for the code indicating a character listed above, but the code can be no longer used to indicate the original character.

- Reserved words

The following reserved words are used in custom macros:

AND, OR, XOR, MOD, EQ, NE, GT, LT, GE, LE,
SIN, COS, TAN, ASIN, ACOS, ATAN, ATN, SQRT, SQR, ABS, BIN, BCD, ROUND, RND, FIX, FUP, LN, EXP, POW, ADP, IF, GOTO, WHILE, DO, END, BPRNT, DPRNT, POPEN, PCLOS, SETVN, AX, AXNUM, PRM

System variable (constant) names and registered common variable names are also used as reserved words.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

16.15 EXTERNAL OUTPUT COMMANDS

In addition to the standard custom macro commands, the following macro commands are available. They are referred to as external output commands.

- BPRNT
- DPRNT
- POPEN
- PCLOS

These commands are provided to output variable values and characters through the input/output interface. In the external output commands, RS232C, Memory card, USB memory, Data Server, and Embedded Ethernet can be specified for an external input/output device by the parameter No. 0020.

Explanation

Specify these commands in the following order:

Open command: POPEN

Before specifying a sequence of data output commands, specify this command to establish a connection to an external input/output device.

Data output command: BPRNT or DPRNT

Specify necessary data output.

Close command: PCLOS

When all data output commands have completed, specify PCLOS to release a connection to an external input/output device.

- Open command POPEN

The POPEN command establishes a connection to an external input/output device. It must be specified before a sequence of data output commands.

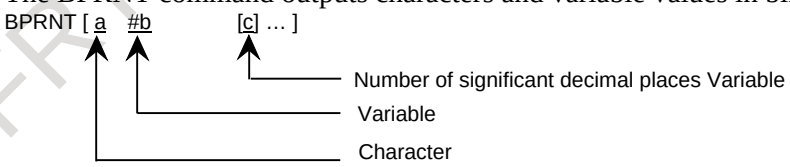
NOTE

1 When RS232C is used as an external input/output device and the specification number of the parameter No. 102, 112 or 122 is set to 0, the CNC outputs a DC2 control code by POPEN command.

2 When Memory card, USB memory, Data Server, or Embedded Ethernet is used as an external input/output device, the file is opened by POPEN command. Please refer to the later explanation about the file name.

- Data output command BPRNT

The BPRNT command outputs characters and variable values in binary.



(i) Specified characters are output.
Specifiable characters are as follows:
- Letters (A to Z)
- Numbers
- Special characters (*, /, +, -, ?, @, &, _)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE
1 An asterisk (*) is output by a space code.
2 When specifying RS232C as an external input/output device, specified characters are converted to the codes according to the setting data (bit 1 (ISO) of parameter No. 0000) that is output at that time. When using ?, @, &, and/or _, use the ISO code as the output code (setting data (ISO) = 1).

(ii) All variables are stored with a decimal point. Specify a variable followed by the number of significant decimal places enclosed in brackets. A variable value is treated as 2-word (32-bit) data, including the decimal digits. It is output as binary data starting from the highest byte.
(iii) When specified data has been output, an EOB code is output.

NOTE
When specifying RS232C as an external input/output device, specified EOB code is converted to the codes according to the setting data (bit 1 (ISO) of parameter No. 0000) that is output at that time.

(iv) <Null> variables are regarded as 0.

Example

```
BPRNT [ C** X#100 [3] Y#101 [3] M#10 [0] ]
    Variable value
      #100=0.40956
      #101=-1638.4
      #10=12.34
    are output as follows:
  C3 A0 A0 D8 00 00 01 9A 59 FF E7 00 00 4D 00 00 00 0C 0A
  ↓                                                      ↓
  C   sp sp   X0000019A   YFFE70000   M0000000C   LF
       ( * * )   (410)     (-1638400)    (12)     ( ; )
```

- Data output command DPRNT

```
DPRNT [ a    #b    [c    d] ... ]
```

- d: Number of significant decimal places
- c: Number of significant digits in the integer part
- b: Variable
- a: Character

The DPRNT command outputs characters and each digit in the value of a variable.
(i) For an explanation of the DPRNT command, see Items (i), (iii), and (iv) for the BPRNT command.
(ii) When outputting a variable, specify # followed by the variable number, then specify the number of digits in the integer part and the number of decimal places enclosed in brackets.
For the value of a variable, as many codes as the specified number of digits are output according to the settings one by one, starting with the highest digit. The decimal point is also output using the set code.
Each variable must be a numeric value consisting of up to nine digits. When high-order digits are zeros, these zeros are not output if bit 1 (PRT) of parameter No. 6001 is 1. If parameter PRT is 0, a space code is output each time a zero is encountered.
When the number of decimal places is not zero, digits in the decimal part are always output. If the number of decimal places is zero, no decimal point is output.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

When bit 1 (PRT) of parameter No. 6001 is 0, a space code is output to indicate a positive number instead of +; if parameter PRT is 1, no code is output.

Example

DPRNT [X#2 [53] Y#5 [53] T#30 [20]]

Variable value

#2=128.47398

#5=-91.2

#30=123.456

are output as follows:

(1) Bit 1 (PRT) of parameter No. 6001 = 0

D8 A0 A0 A0 B1 B2 B8 2E B4 B7 B4 59 2D A0 A0 A0 39 B1 2E B2 30 30 D4 A0 B2 33 0A

↓ ↓ ↓ ↓ ↓ ↓ ↓

X sp sp sp 128.474 Y- sp sp sp 91.200 T sp 023 LF

(2) Bit 1 (PRT) of parameter No. 6001 = 1

D8 B1 B2 B8 2E B4 B7 B4 59 2D 39 B1 2E B2 30 30 D4 A0 B2 33 0A

↓

X128.474 Y-91.200 T023 LF

- Close command PCLOS

The PCLOS command releases a connection to an external input/output device. Specify this command when all data output commands have terminated.

NOTE

1 When RS-232C is used as an external input/output device and the specification number of the parameter No.102, 112 or 122 is set to 0, the CNC outputs a DC4 control code by PCLOS command.

2 When Memory card, USB memory, Data Server, or Embedded Ethernet is used as an external input/output device, the file is closed by PCLOS command.

File name

When Memory card, USB memory, Data Server, or Embedded Ethernet is used as an external input/output device, the created file name is from “PRNT0000.DAT” to “PRNT9999.DAT”. The numerical part of the file name is sequentially counted from 0000 to 9999. When the CNC is turned off, the number is counted from 0 again. But the number can be memorized and continuously counted by the bit 7 (SFN) of parameter No. 6019.

Moreover, the created file name can be fixed as “MCR_PRNT.TXT” by the bit 3 (OFN) of parameter No. 6019.

FRYER MACHINE SYSTEMS INC
Bob Zamansky

NOTE

1 If the file which has the same file name already exists when the file is created into an external input/output device, the following operation is done by an external input/output device.

In case of memory card	: Alarm SR1973 is issued.
In case of USB memory	: The file is overwritten.
In case of Data Server (Storage mode)	: Alarm PS2032 is issued.
In case of Data Server (FTP mode)	: The file is overwritten. (NOTE2)
In case of Embedded Ethernet	: The file is overwritten. (NOTE2)

If a file with the same name already exists, the external output command cannot use bit 1 (COW) of parameter No. 11308, for whether to overwrite the file or cancel overwriting, cannot be used.

2 This operation depends on the specification of FTP server software on the Host computer. Generally, the file is overwritten. Please refer to the specification of the FTP server software for details.

Variable name, Array variable, <expression>, <conditional expression>

In addition to macro variables, variable names, array variables, <expression>, <conditional expression> are also specified for specifying numerical values in BPRNT / DPRNT. In addition, when using array variables, enable the number of custom macro variable name 1000 or enable the number of custom macro variable name 4000 is required.

Format

DPRNT [a [# variable name] [c d] …]
In the case of 1-dimensional array :
DPRNT [a [# array variable name [i]] [c d] …]
In the case of 2-dimensional array :
DPRNT [a [# array variable name [i, j]] [c d] …]
In the case of 3-dimensional array
DPRNT [a [# array variable name [i, j, k]] [c d] …]
DPRNT [a [<expression>] [c d] …]
DPRNT [a [<conditional expression>] [c d] …]

a : String
c : Number of significant digits above the decimal point
d : Number of significant digits below the decimal point

BPRNT [a [# variable name] [c] …]
In the case of 1-dimensional array :
BPRNT [a [# array variable name [i]] [c] …]
In the case of 2-dimensional array :
BPRNT [a [# array variable name [i, j]] [c] …]
In the case of 3-dimensional array :
BPRNT [a [# array variable name [i, j, k]] [c] …]
BPRNT [a [<expression>] [c] …]
BPRNT [a [<conditional expression>] [c] …]

a : String
c : Number of significant digits below the decimal point

Example 1 : Variable name (common variable name)
SETVN 500 [VAR500]

FANUC MACHINE SYSTEMS INC Rob Zahensky

```
[ #VAR500 ] = 111.0
DPRNT [ AAA [ #VAR500 ] [ 5 3 ] ]
```

Example 2 : Variable name (system variable name)

```
DPRNT [ AAA [ #_ABSMT [ 1 ] ] [ 5 3 ] ]
```

Example 3 : Array variable name

```
DEFREAL 500 [ ARRAY [ 3, 3 ] ]
[ #ARRAY [ 2, 1 ] ] = 111.0
DPRNT [ AAA [ #ARRAY [ 2, 1 ] ] [ 5 3 ] ]
```

Example 4 : <expression>

```
#100=111.0
#101=222.0
DPRNT [ AAA [ #100 + #101 ] [ 5 3 ] ]
```

Example 5 : <conditional expression>

```
#100=111.0
#101=222.0
DPRNT [ AAA [ #100 EQ #101 ] [ 5 3 ] ]
```

It is assumed to be 1.0 when the judgment result of the conditional expression is true and 0.0 when it is false.

- Notice

NOTE

1. When an external input/output device is RS-232C, it is necessary to set the settings (baud-rate etc.) for RS-232C interface. Moreover, set 0 or 4 to the specification number of the input/output device of the parameters Nos. 102, 112 and 122.
2. It is not necessary to always specify the open command (POPEN), data output command (BPRNT, DPRNT), and close command (PCLOS) together. Once an open command is specified at the beginning of a program, it does not need to be specified again except after a close command was specified.
3. Be sure to specify open commands and close commands in pairs. Specify the close command at the end of the program. However, do not specify a close command if no open command has been specified.
4. When a reset operation is performed while commands are being output by a data output command, output is stopped and subsequent data is erased. Therefore, when a reset operation is performed by a code such as M30 at the end of a program that performs data output, specify a close command at the end of the program so that processing such as M30 is not performed until all data is output.
5. When an external input/output device is Data Server (FTP mode) or Embedded Ethernet, the FTP is used. Therefore, it is necessary to execute the FTP server on the personal computer.
   Moreover, please note that there is a possibility that the FTP communication is aborted by the security function of the personal computer side when a time that the data doesn't flow is long.

FRYER MACHINE SYSTEMS INC
Ron Zamansky

16.16 RESTRICTIONS

- Sequence number search

A custom macro program cannot be searched for a sequence number.

- Single block

Even while a macro program is being executed, blocks can be stopped in the single block mode.
A block containing a macro call command (G66, G66.1, Ggg, Mmm, or G67) does not stop even when the single block mode is on.
Whether blocks containing arithmetic and logic operation commands and control commands are stopped depends on the settings of bits 5 (SBM) and 7 (SBV) of parameter No. 6000 as shown in the following table.

		Bit 5 (SBM) of parameter No. 6000	
		0	1
Bit 7 (SBV) of parameter No. 6000	0	Not stopped when the single block mode is on.	Can be stopped in the single block mode. (Variable #3003 cannot be used to disable single block stop. Single block stop is always enabled.)
	1	Can be stopped in the single bock mode. (Variable #3003 can be used to enable or disable single block stop.)	

Note that when a single block stop occurs at a macro statement in cutter compensation mode, the statement is assumed to be a block that does not involve movement, and proper compensation cannot be performed in some cases. (Strictly speaking, the block is regarded as specifying a movement with a travel distance 0.)

- Optional block skip

A / appearing in the middle of an <expression> (enclosed in brackets [ ] on the right-hand side of an arithmetic expression) is regarded as a division operator; it is not regarded as the specifier for an optional block skip code.

- Operation in EDIT mode

By setting bit 0 (NE8) of parameter No. 3202 and bit 4 (NE9) of parameter No. 3202 to 1, deletion and editing are disabled for custom macro programs and subprograms with program numbers 8000 to 8999 and 9000 to 9999. This prevents registered custom macro programs and subprograms from being destroyed by accident. When the entire memory is cleared, the contents of memory such as custom macro programs are deleted.

- Reset

With a reset operation, local variables and common variables #100 to #199 are cleared to null values. They can be prevented from clearing by setting bit 6 (CCV) of parameter No. 6001. System variables #100 to #199 are not cleared.
A reset operation clears any called states of custom macro programs and subprograms, and any DO states, and returns control to the main program.

- Display of the PROGRAM RESTART

As with M98, the M and T codes used for subprogram calls are not displayed.

- Feed hold

When feed hold is enabled during execution of macro statement, the machine stops after the execution of the macro statement. The machine also stops when a reset or an alarm occurs.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- DNC operation

The control commands (such as GOTO and WHILE-DO) cannot be executed during DNC operation. However, this restriction is removed when a program registered in program memory is called during DNC operation.

- Constant values that can be used in <expression>

+0.00000000001 to +999999999999
-999999999999 to -0.00000000001
The number of significant digits is 12 (decimal).
If this range is exceeded, an alarm PS0004, “INVALID BREAK POINT OF WORDS” occurs.

- Comment section

Usually, the position in which comment section can be inserted in macro statements is as follows.

(ABC) #100 = 1;	Head of the block
#100 = 1(ABC);	End of the block
N01 (ABC) #100 = 1;	Just behind of the sequence number

If bit 3 (NCM) of parameter No. 6020 is set to 1, the position in which comment section can be inserted in macro statements as follows, too.

#100(ABC) = 1;	Just behind of variable number
#100 = #101 +1.(ABC) *#102;	Just behind of numeric values
#100 = [#_UIL[1]](ABC) *100.;	Just behind of the name of variable

In word editing, it might become difficult to do the program edit since comment section is inserted. Such a case uses character editing.
Comment section cannot be inserted in the character string such as variable number, numeric values, name of variable, functions and control commands.
Example that cannot be inserted

```
#(ABC)100 = 1;
#100 = [#_UIL(ABC)[1]] *100.;
IF[ #100 EQ 1 ] GO(ABC)TO99;
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

16.17 INTERRUPTION TYPE CUSTOM MACRO

When a program is being executed, another program can be called by inputting an interrupt signal UINT from the machine.
This function is referred to as an interruption type custom macro function. Program an interrupt command in the following format:

Format

M96Pxxxxxxxx ;	Enables custom macro interrupt
M97 ;	Disables custom macro interrupt

Explanation

Use of the interruption type custom macro function allows the user to call a program during execution of an arbitrary block of another program. This allows programs to be operated to match situations which vary from time to time.

(1) When a tool abnormality is detected, processing to handle the abnormality is started by an external signal.
(2) A sequence of machining operations is interrupted by another machining operation without the cancellation of the current operation.
(3) At regular intervals, information on current machining is read.
:

Listed above are examples like adaptive control applications of the interruption type custom macro function.

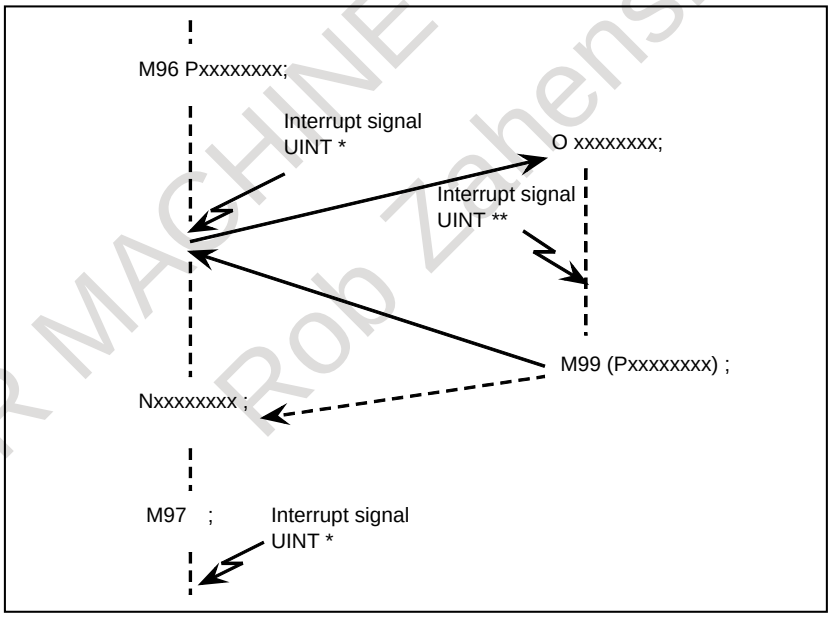


Fig 16.17 Interruption type custom macro function

When M96Pxxxx is specified in a program, subsequent program operation can be interrupted by an interrupt signal UINT input to execute the program specified by Pxxxx. When the interrupt signal UINT (marked with an asterisk (*) in Fig 16.17) is input during execution of the interrupt program or after M97, it is ignored.

FRYER MACHINE SYSTEMS INC
Rob Zehensky

16.17.1 Specification Method

Explanation

- Interrupt conditions

A custom macro interrupt is available only during program execution. It is enabled under the following conditions

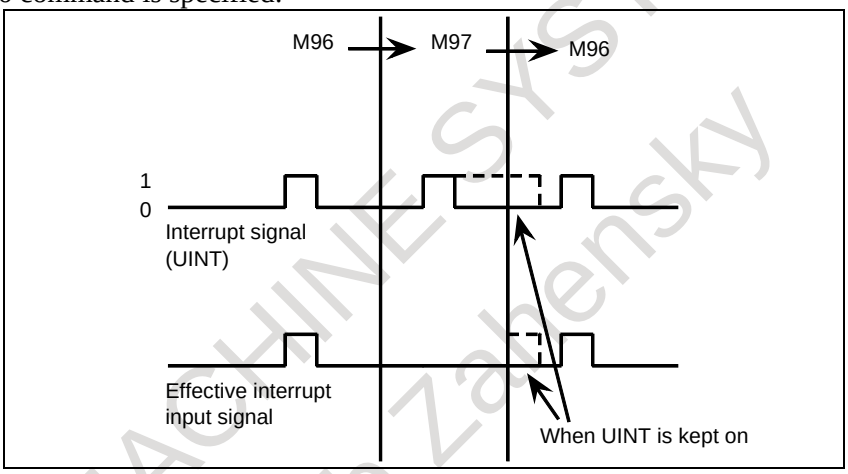
- When memory operation, DNC operation, or MDI operation is selected
- When STL (start lamp) is on
- When a custom macro interrupt is not currently being processed

A macro interrupt cannot be performed during manual operation.

- Specification

Generally, the custom macro interrupt function is used by specifying M96 to enable the interrupt signal UINT and M97 to disable the signal.

Once M96 is specified, a custom macro interrupt can be initiated by the input of the interrupt signal UINT until M97 is specified or the CNC is reset. After M97 is specified or the CNC is reset, no custom macro interrupts are initiated even when the interrupt signal UINT is input. The interrupt signal UINT is ignored until another M96 command is specified.



The interrupt signal UINT becomes valid after M96 is specified. Even when the signal is input in M97 mode, it is ignored. When the signal input in M97 mode is kept on until M96 is specified, a custom macro interrupt is initiated as soon as M96 is specified (only when the status-triggered scheme is employed); when the edge-triggered scheme is employed, the custom macro interrupt is not initiated even when M96 is specified.

NOTE
For the status-triggered and edge-triggered schemes, see the Subsection, “Details of Functions”.

FRYER MACHINE SYSTEMS INC
Bob Zabensky

16.17.2 Details of Functions

Explanation

- Subprogram-type interrupt and macro-type interrupt

There are two types of custom macro interrupts: Subprogram-type interrupts and macro-type interrupts. The interrupt type used is selected by bit 5 (MSB) of parameter No. 6003.

(a) Subprogram-type interrupt: When bit 5 (MSB) of parameter No. 6003 is set to 1
An interrupt program is called as a subprogram.
This means that the levels of local variables remain unchanged before and after the interrupt.
This interrupt is not included in the nesting level of subprogram calls.

(b) Macro-type interrupt: When bit 5 (MSB) of parameter No. 6003 is set to 0
An interrupt program is called as a custom macro.
This means that the levels of local variables change before and after the interrupt.
The interrupt is not included in the nesting level of custom macro calls.
When a subprogram call or a custom macro call is performed within the interrupt program, this call is included in the nesting level of subprogram calls or custom macro calls.
Arguments cannot be passed from the current program even when the custom macro interrupt is a macro-type interrupt. The local variables immediately after interruption are all cleared to null.

- M codes for custom macro interrupt control

In general, custom macro interrupts are controlled by M96 and M97. However, these M codes, may already being used for other purposes (such as an M function or macro M code call) by some machine tool builders.
For this reason, bit 4 (MPR) of parameter No. 6003 is provided to set M codes for custom macro interrupt control.
When specifying this parameter to use the custom macro interrupt control M codes set by parameters, set parameters Nos. 6033 and 6034 as follows:
Set the M code to enable custom macro interrupts in parameter No. 6033, and set the M code to disable custom macro interrupts in parameter No. 6034.
When specifying that parameter-set M codes are not used, M96 and M97 are used as the custom macro control M codes regardless of the settings of parameters Nos. 6033 and 6034.
The M codes used for custom macro interrupt control are processed internally (they are not output to external units). However, in terms of program compatibility, it is undesirable to use M codes other than M96 and M97 to control custom macro interrupts.

- Custom macro interrupts and NC statements

When performing a custom macro interrupt, the user may want to interrupt the NC statement being executed, or the user may not want to perform the interrupt until the execution of the current block is completed. Bit 2 (MIN) of parameter No.6003 is used to select whether to perform interrupts even in the middle of a block or to wait until the end of the block. The type of interrupt performed even in the middle of a block is called type I and the type of interrupt performed at the end of the block is called type II.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

⚠ CAUTION

For interrupt type I, operation after control is returned differs depending on whether the interrupt program contains an NC statement.
When the program number block contains EOB (;), it is assumed to contain an NC statement.

(Program containing an NC statement)

```
O0013;
#101=#5041;
#102=#5042;
#103=#5043;
M99;
```

(Program containing no NC statement)

```
O0013#101=#5041;
#102=#5042;
#103=#5043;
M99;
```

Type I (when an interrupt is performed even in the middle of a block)

(i) When the interrupt signal UINT is input, any movement or dwell being performed is stopped immediately and the interrupt program is executed.

(ii) If there are NC statements in the interrupt program, the command in the interrupted block is lost and the NC statement in the interrupt program is executed. When control is returned to the interrupted program, the program is restarted from the next block after the interrupted block.

(iii) If there are no NC statements in the interrupt program, control is returned to the interrupted program by M99, then the program is restarted from the command in the interrupted block.

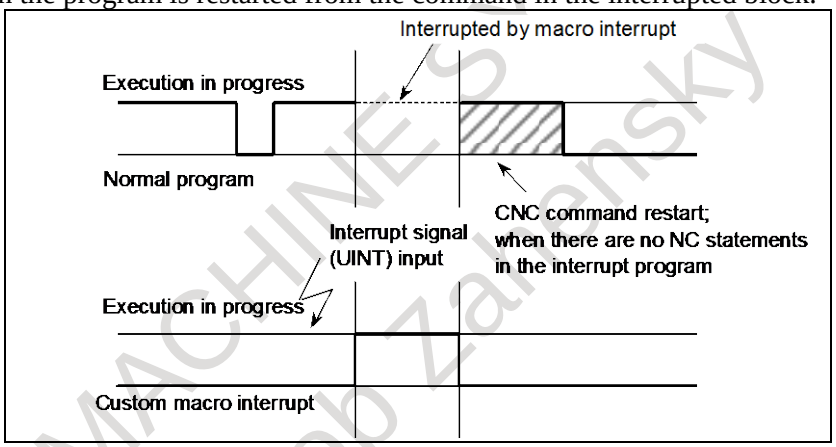


Fig. 16.17.2 (a) Custom macro interrupt and NC command (type I)

Type II (when an interrupt is performed at the end of the block)

(i) If the block being executed is not a block that consists of several cycle operations such as a drilling canned cycle and automatic reference position return (G28), an interrupt is performed as follows:
When an interrupt signal UINT is input, macro statements in the interrupt program are executed immediately unless an NC statement is encountered in the interrupt program. NC statements are not executed until the current block is completed.

(ii) If the block being executed consists of several cycle operations, an interrupt is performed as follows:
When the last movement in the cycle operations is started, macro statements in the interrupt program are executed unless an NC statement is encountered. NC statements are executed after all cycle operations are completed.

FRYER MACHINE SYSTEMS INC
Rob Zalensky

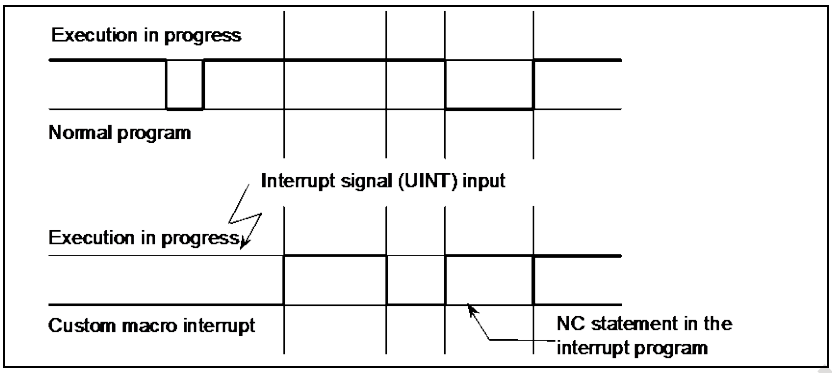


Fig. 16.17.2 (b) Custom macro interrupt and NC command (type II)

M

NOTE

During execution of a program for cycle operations, interrupt type II is performed regardless of whether bit 2 (MIN) of parameter No. 6003 is set to 0 or 1. Cycle operations are available for the following functions:

<1> Automatic reference position return
<2> Tool radius · tool nose radius compensation (generating multiple blocks using the specified block such as when the tool moves around the outside of an acute angle)
<3> Canned cycle
<4> Automatic tool length measurement
<5> Optional chamfering/corner R
<6> Normal direction control
<7> Cutting point interpolation for cylindrical interpolation

T

NOTE

During execution of a program for cycle operations, interrupt type II is performed regardless of whether bit 2 (MIN) of parameter No. 6003 is set to 0 or 1. Cycle operations are available for the following functions:

<1> Automatic reference position return
<2> Tool radius · tool nose radius compensation (generating multiple blocks using the specified block such as when the tool moves around the outside of an acute angle)
<3> Canned cycle (No interruption type custom macro can be used during execution of a multiple repetitive canned turning cycle, however.)
<4> Automatic tool offset
<5> Chamfering/corner R
<6> Cutting point interpolation for cylindrical interpolation

FRYER MACHINE SYSTEMS INC
Ron Zalensky

- Conditions for enabling and disabling the custom macro interrupt signal

The interrupt signal becomes valid after execution starts of a block that contains M96 for enabling custom macro interrupts. The signal becomes invalid when execution starts of a block that contains M97.
While an interrupt program is being executed, the interrupt signal becomes invalid. The signal becomes valid when the execution of the block that immediately follows the interrupted block in the main program is started after control returns from the interrupt program. In type I, if the interrupt program consists of only macro statements, the interrupt signal becomes valid when execution of the interrupted block is started after control returns from the interrupt program.

- Custom macro interrupt signal UINT

There are two schemes for custom macro interrupt signal UINT input: The status-triggered scheme and edge-triggered scheme. When the status-triggered scheme is used, the signal is valid when it is on. When the edge triggered scheme is used, the signal becomes valid on the rising edge when it switches from off to on status.
One of the two schemes is selected with bit 3 (TSE) of parameter No. 6003.

When the status-triggered scheme is selected by this parameter, a custom macro interrupt is generated if the interrupt signal UINT is on at the time the signal becomes valid. By keeping the interrupt signal UINT on, the interrupt program can be executed repeatedly.

When the edge-triggered scheme is selected, the interrupt signal UINT becomes valid only on its rising edge. Therefore, the interrupt program is executed only momentarily (in cases when the program consists of only macro statements). When the status-triggered scheme is inappropriate, or when a custom macro interrupt is to be performed just once for the entire program (in this case, the interrupt signal may be kept on), the edge-triggered scheme is useful.

Except for the specific applications mentioned above, use of either scheme results in the same effects. The time from signal input until a custom macro interrupt is executed does not vary between the two schemes.

In the example shown in Fig. 16.17.2 (c), an interrupt is executed four times when the status-triggered scheme is used; when the edge-triggered scheme is used, the interrupt is executed just once.

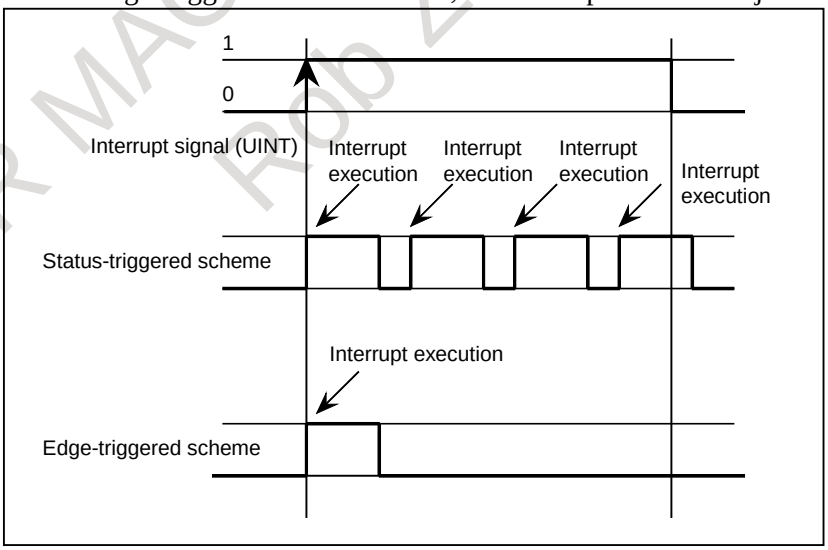


Fig. 16.17.2 (c) Custom macro interrupt signal

- Return from a custom macro interrupt

To return control from a custom macro interrupt to the interrupted program, specify M99. A sequence number in the interrupted program can also be specified using address P. If this is specified, the program is searched from the beginning for the specified sequence number. Control is returned to the first sequence number found.

FRYER MACHINE SYSTEMS INC
Rob Zahrensky

NOTE

When an M99 block consists only of address O, N, P, L, or M, this block is regarded as belonging to the previous block in the program. Therefore, a single-block stop does not occur for this block. In terms of programming, the following <1> and <2> are basically the same. (The difference is whether Gxx is executed before M99 is recognized.)

```
<1>  Gxx Xxxx ;
     M99 ;
<2>  Gxx Xxxx M99 ;
```

- Custom macro interrupt and modal information

A custom macro interrupt is different from a normal program call. It is initiated by an interrupt signal UINT during program execution. In general, any modifications of modal information made by the interrupt program should not affect the interrupted program.
For this reason, even when modal information is modified by the interrupt program, the modal information before the interrupt is restored when control is returned to the interrupted program by M99.

When control is returned from the interrupt program to the interrupted program by M99 Pxxxxxxxx, however, modal information can again be controlled by the program. In this case, the new continuous information modified by the interrupt program is passed to the interrupted program.

In this case, take the following action as required:

<1> The interrupt program provides modal information to be used after control is returned to the interrupted program.
<2> After control is returned to the interrupted program, modal information is specified again as necessary.

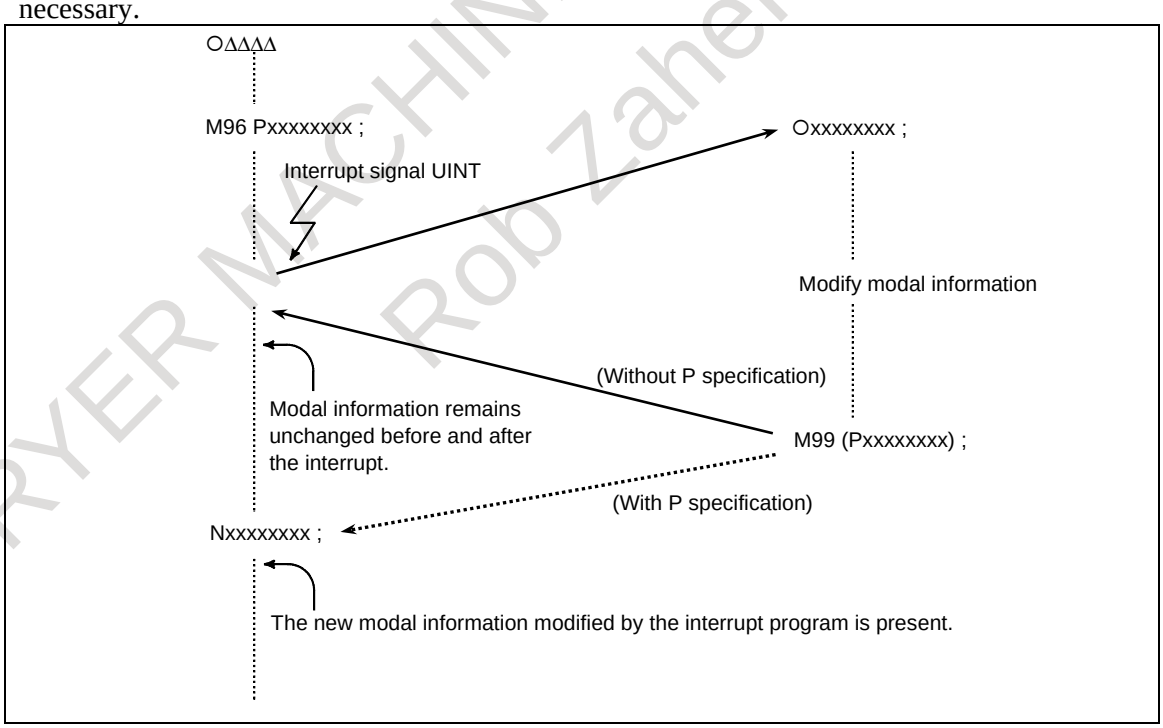


Fig. 16.17.2 (d) Custom macro interrupt and modal information

Modal information when control is returned by M99

The modal information present before the interrupt becomes valid. The new modal information modified by the interrupt program is made invalid.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Modal information when control is returned by M99 Pxxxxxxxx

The new modal information modified by the interrupt program remains valid even after control is returned.

Modal information which was valid in the interrupted block

The old modal information which was valid in the interrupted block can be read using custom macro system variables #4401 to #4530.

M

System variable	Modal information which was valid when a custom macro interrupt was generated
#4401	G code (group 01)
:	:
#4421	G code (group 21)
#4502	B code
#4507	D code
#4508	E code
#4509	F code
#4511	H code
#4513	M code
#4514	Sequence number
#4515	Program number
#4519	S code
#4520	T code
#4530	Additional workpiece coordinate system number

T

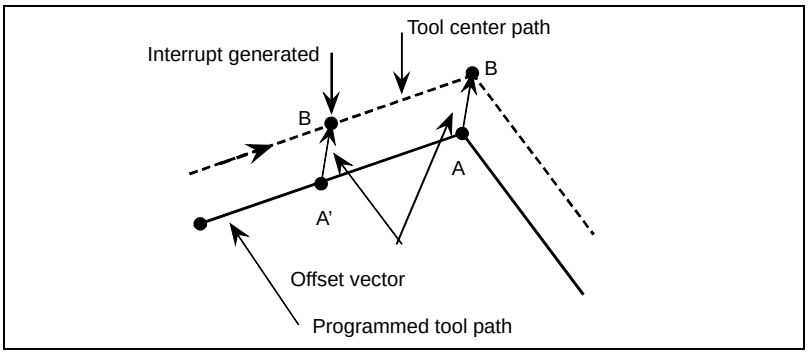
System variable	Modal information which was valid when a custom macro interrupt was generated
#4401	G code (group 01)
:	:
#4421	G code (group 21)
#4508	E code
#4509	F code
#4513	M code
#4514	Sequence number
#4515	Program number
#4519	S code
#4520	T code
#4530	Additional workpiece coordinate system number

- System variables (position information values) for the interrupt program

Position information can be read as follows.

Macro variable	Condition	Position information value
#5001 or above	Until the first NC statement appears	Coordinates of point A
	After an NC statement with no move command appears	Coordinates of point A'
	After an NC statement with a move command appears	Coordinates of the end point of the move command
#5021 or above		Machine coordinates of point B'
#5041 or above		Workpiece coordinates of point B'

FRYER MACHINE SYSTEMS INC
Rob Zahensky



- Custom macro interrupt and custom macro modal call

When the interrupt signal UINT is input and an interrupt program is called, the custom macro modal call is canceled (G67). However, when G66 is specified in the interrupt program, the custom macro modal call becomes valid. When control is returned from the interrupt program by M99, the modal call is restored to the state it was in before the interrupt was generated. When control is returned by M99 Pxxxxxxxx;, the modal call in the interrupt program remains valid.

- Custom macro interrupt and program restart

In program restart, when the interrupt signal UINT is input during dry run recovery after a search, the interrupt program is called after restart of all axes is completed.
That is, interrupt type II is assumed regardless of the parameter setting.

M

NOTE

1 Alarm PS1101, “ILLEGAL CNC STATEMENT IRT.” occurs in the following cases:
<1>An interrupt is generated in the programmable mirror image (G51.1) mode and another G51.1 is specified in the interrupt program.
<2>An interrupt is generated in the coordinate system rotation (G68) mode and another G68 is specified in the interrupt program.
<3>An interrupt is generated in the scaling (G51) mode and another G51 is specified in the interrupt program.

2 In program restart, do not input the interrupt signal UINT during dry run recovery after a search.

T

NOTE

1 Alarm PS1101, “ILLEGAL CNC STATEMENT IRT.” occurs in the following cases:
<1>An interrupt is generated in the programmable mirror image (G51.1) mode and another G51.1 is specified in the interrupt program.
<2>An interrupt is generated in the coordinate system rotation (G68.1) mode and another G68.1 is specified in the interrupt program.

2 No interruption type custom macro can be used during execution of a multiple repetitive canned turning cycle.

3 In program restart, do not input the interrupt signal UINT during dry run recovery after a search.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

16.18 DISPLAYING VARIABLE IN MACRO ALARM OR MACRO MESSAGE

This function converts the macro variable in the message of system variable #3000 (macro alarm) or #3006 (stop with a message) into the value.

Format

#3000= - n(&#var Ff.e Zz&) ;

#3006= - 1(&#var Ff.e Zz&) ;

n	: 0-200 (bit 1 (MCA) of parameter No. 6008 = 0), 0-4095 (MCA = 1)
var	: The macro variable number
Ff.e Zz	: The display format

Explanation

When a negative value is assigned in system variable #3000 or #3006, "&?" in the message becomes a control code. At this case, "&#" is the control code to convert the macro variable into the value. Then, "&#-&" is converted into the value. Also, the display format can be specified by address F and Z. The other specification follows the specification of traditional #3000 or #3006. For the details, refer to the section of the system variable of custom macro.

var

"var" is the macro variable number. The macro variable number can be specified regardless of the variable of type (local variables, common variables, system variables) if an available number. In addition, the number to read and write the other path variables (#ppxxxxxxx (pp: path number)) can also be specified.

Ff.e

The display digit and the decimal position can be specified. ‘f’ is the display digit and ‘e’ is the decimal position.
The sign (‘-’) and the decimal point (‘.’) are not included in the display digit. Therefore, the number of characters that are actually displayed is the sum of the display digit, the sign, and the decimal point. When ‘e’ is 0, the decimal point is not displayed.
f : The display digit (1-40)
e : The decimal position (0-9)

Zz

Whether or not to do the zero suppression can be specified. Only zero of left side is suppressed. When "Zz" is omitted, it is equivalent to "Z1".
z=0 : Not zero suppression.
z=1 : Zero suppression. (There is left side space.)
z=2 : Zero suppression. (There is not left side space.)

When both “Ff.e” and “Zz” are omitted

The converted string is displayed with a minimum number of characters. It is displayed without left side space and extra zero of decimal fraction part. In addition, when there is not the value of decimal fraction part, the decimal point also is not displayed.

FRYER MACHINE SYSTEMS INC
Rob Zahelnsky

Display of '&'

Command "&&" in the message.

Example
When the following message is displayed.
MC3001 ERROR : #500&#501&#502 ARE 100 (the value of #503)

The program is as follows.
#503 = 100.0 ;
#3000 = - 1 (ERROR : #500&&#501&&#502 ARE &#503&) ;

The program example

In case of #500=1234567.89, #501=-98.7654321 and #502=-12345.0, when each program is executed, the following alarm messages are displayed.

1. When there is both “Ff.e” and “Zz”. (When “Ff.e” is changed.)
The program #3000 = -1(ERR NO.&#500F11.4Z0& && NO.&#501F11.4Z0& && NO.&#502F11.4Z0&) ;
Alarm message MC3001 ERR NO.1234567.8900 & NO.-0000098.7654 & NO.-0012345.0000

The program #3000 = -1(ERR NO.&#500F11.0Z0& && NO.&#501F11.0Z0& && NO.&#502F11.0Z0&) ;
Alarm message MC3001 ERR NO.00001234568 & NO.-00000000099 & NO.-00000012345

The program #3000 = -1(ERR NO.&#500F9.2Z0& && NO.&#501F9.2Z0& && NO.&#502F9.2Z0&) ;
Alarm message MC3001 ERR NO.1234567.89 & NO.-0000098.77 & NO.-0012345.00

2. When there is both “Ff.e” and “Zz”. (When “Zz” is changed.)
The program #3000 = -1(ERR NO.&#500F11.4Z0& && NO.&#501F11.4Z0& && NO.&#502F11.4Z0&) ;
Alarm message MC3001 ERR NO.1234567.8900 & NO.-0000098.7654 & NO.-0012345.0000

The program #3000 = -1(ERR NO.&#500F11.4Z1& && NO.&#501F11.4Z1& && NO.&#502F11.4Z1&) ;
Alarm message MC3001 ERR NO.1234567.8900 & NO. -98.7654 & NO. -12345.0000

The program #3000 = -1(ERR NO.&#500F11.4Z2& && NO.&#501F11.4Z2& && NO.&#502F11.4Z2&) ;
Alarm message MC3001 ERR NO.1234567.8900 & NO.-98.7654 & NO.-12345.0000

3. When “Zz” is omitted.
The program #3000 = -1(ERR NO.&#500F11.4& && NO.&#501F11.4& && NO.&#502F11.4&) ;
Alarm message MC3001 ERR NO.1234567.8900 & NO. -98.7654 & NO. -12345.0000

4. When both “Ff.e” and “Zz” are omitted.
The program #3000 = -1(ERR NO.&#500& && NO.&#501& && NO.&#502&) ;
Alarm message MC3001 ERR NO.1234567.89 & NO.-98.7654321 & NO.-12345

FRYER MACHINE SYSTEMS INC
Rob Zuhensky

NOTE

1 When the control code “&?” in which the corresponding function is not provided is used, “&?-&” is converted into ‘!’. When there is not ‘&’ of the end, “&?-” is converted into ‘!’.

Example 1
”&1” is used.
The program #3000= -1(AA&1100F12.3Z2&BB)
Alarm message MC3001 AA!BB

Example 2
”&A” is used and there is not ‘&’ of the end.
The program #3000= -1(AA&A100F12.3Z2BB)
Alarm message MC3001 AA!

2 The following cases are format error. “&#-&” is converted into ‘?’. When there is not ‘&’ of the end, “&#-” is converted into ‘?’.
- When the format is incorrect.
- When the macro variable number that cannot be used is specified in “var”.
- When a value outside the range is used to ‘f’, ‘e’, or ‘z’.
- When ‘e’ is more than ‘f’

Example 1
'E' is an error.
The program #3000= -1(AA&#500F12.3E0&BB)
Alarm message MC3001 AA?BB

Example 2
There is not “Ff.e”.
The program #3000= -1(AA&#500Z2&BB)
Alarm message MC3001 AA?BB

Example 3
When there is not enable the embedded macro, #200 cannot be used.
The program #3000= -1(AA&#200&BB)
Alarm message MC3001 AA?BB

Example 4
‘z’ is the value outside the range.
The program #3000= -1(AA&#500F12.3Z4&BB)
Alarm message MC3001 AA?BB

Example 5
‘e’ is more than ‘f’.
The program #3000= -1(AA&#500F2.6Z1&BB)
Alarm message MC3001 AA?BB

Example 6
There is not ‘&’ of the end.
The program #3000= -1(AA&#500F12.3Z0BB)
Alarm message MC3001 AA?

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

3 When the number to exceed 15 digits (Precisely, the actual values are slightly different from the above values because they are internally processed in binary.) are displayed, “&#-&” is converted into ‘*’.

Example
#100 = 100000000.0 ;
#101 = 100000000.0 ;
#500 = #100 * #101 ;

When 10000000000000000 (17 digits) is displayed.
The program #3000= -1(AA&#500F20.0&BB)
Alarm message MC3001 AA*BB

When 100000000.000000000 (18 digits) is displayed.
The program #3000= -1(AA&#100F20.9&BB)
Alarm message MC3001 AA*BB

4 In the case of omitting both “Ff.e” and “Zz”, when the number to exceed 15 digits is displayed, integer part is preferentially displayed. When the integer part exceeds 15 digits, “&#-&” is converted into ‘*’ as well as note 3.

Example
#100 = 987654321.0 ;
#101 = 0.123456789 ;
#500 = #100 + #101 ;
The program #3000= -1(AA&#500&BB)
Alarm message MC3001 AA987654321.123457BB

5 When the value of the macro variable is <empty>, “&#-&” is converted into NULL (without a string).
Example
#100 = #0 ;
The program #3000= -1(AA&#100F12.3Z2&BB)
Alarm message MC3001 AABB

6 When the digit of displayed value is larger than the display digit that specified in ‘f’, “&#-&” is converted into ‘-’.

Example
#100 = -1234567890.0 ;
The program #3000= -1(AA&#100F5.2Z1&BB)
Alarm message MC3001 AA-BB

7 A value that is smaller than the decimal position specified in ‘e’ is rounded off.

Example
#100 = -1111.111500 ;
#101 = -1111.111499 ;

The program #3000= -1(AA&#100F15.3Z2&BB)
Alarm message MC3001 AA-1111.112BB

The program #3000= -1(AA&#101F15.3Z2&BB)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

Alarm message MC3001 AA-1111.111BB

8 The valid characters are 62 characters for the message. Therefore the conversion characters are 62 characters. The character that exceeds 62 characters is not used for the message. (The displayed message is 61 characters or less.)

Example

#3000 = - 1(&#100& && &#101& && … && &#106& && &#107& && &#108&) ;

<- 62 characters or less are converted. -><- This is not used.

MC3001 11111 & 22222 & 33333 & …… & 44444 & 55555 & 66666

<- The displayed message is 61 characters or less. ->

9 When the message exceeds 61 characters by conversion, the 61st character is converted into ‘/’.

Example

#100 = -1234.56789 ;

The program

#3000= -1(&#100& && &#100& && &#100& && &#100& && &#100&) ;

Alarm message

MC3001 -1234.56789 & -1234.56789 & -1234.56789 & -1234.56789 & -123/

10 The variable name, the array variable, and the calculation result cannot be specified directly for “var”. Substitute for another macro variable once and display it when you display these values. (It is recommended to use local variables that the number of characters is little.)

Example

#1 = [#XYZ]+ [#_ABSMT[1]] -[#ARRAY[0,1,2]]*100+#500;

#3000 = -1(AA&#1&BB) ;

FRYER MACHINE SYSTEMS INC
Rob Zaherek

17 REAL-TIME CUSTOM MACRO

Overview

Used with an NC program, the real time custom macro function controls peripheral axes and signals.

If a macro statement is used together with an NC statement, a program using the conventional custom macro function executes the macro statement immediately when the macro statement is read. So, the macro statement cannot be executed independently of the NC statement.
On the other hand, the real time custom macro function enables the following control operations when a real time macro command (RTM command) is coded in an NC program.

- A real time macro command starts operation in synchronism with the NC statement and is executed independently. During NC program execution, a real time macro command can be executed at the same time.
- PMC interface signals can be read and written (with a restriction). In an NC program, a motion using a signal as a trigger can be coded.
- Variables dedicated to a real time macro command can be read and written.
- A real time macro command can exercise axis control. (The axis control by PMC.)
- Multiple real time macro commands can be executed at the same time. Multiple real time custom macro statements can be coded in an NC program and can be controlled independently of each other.

By using a real time custom macro, control on signals and peripheral axes can be programmed.
The example below turns on and off a signal and performs operation on a peripheral axis when machining points are passed.

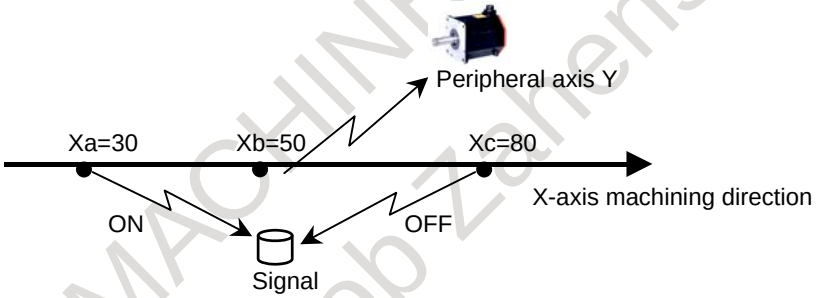


Fig. 17 (a) Signal and operation on a peripheral axis during machining (example)

During movement for machining along the X-axis in the Fig. 17 :

(1) When point Xa is passed, the G99.5 signal is set to “1”.
→ Real time macro command 1

(2) When point Xb is passed, positioning is started on peripheral axis Y. → Real time macro command 2

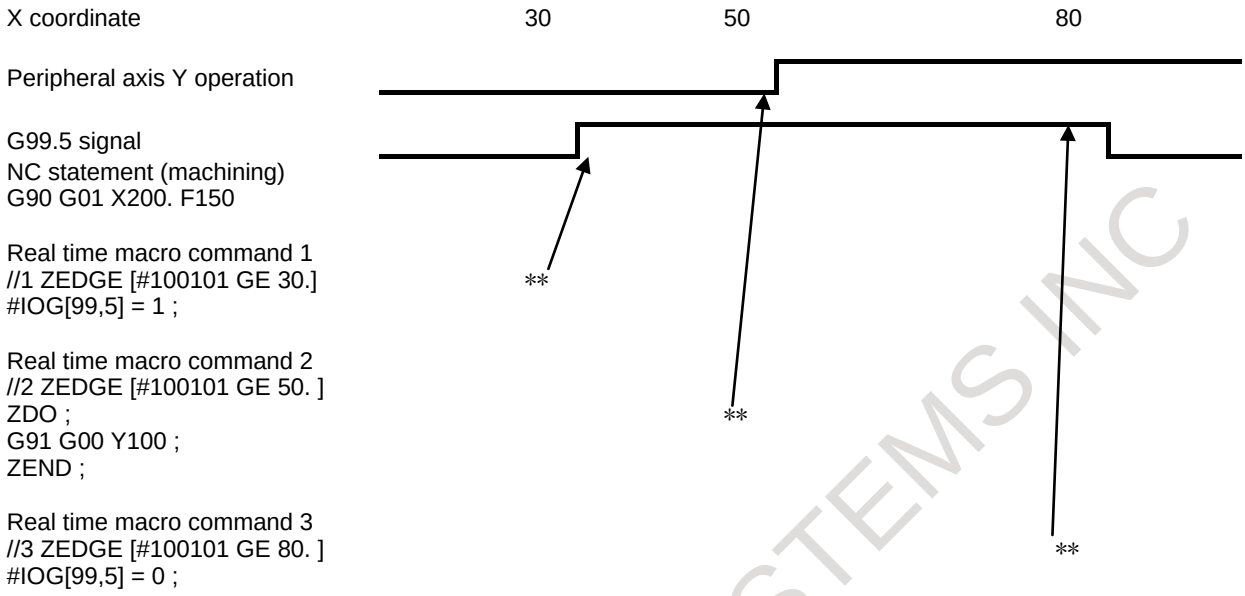
(3) When point Xc is passed, the G99.5 signal is set to “0”.
→ Real time macro command 3

The operation above is programmed using real time macro commands.

```
Program
O0001 ;
G92 X0 ;
//1 ZEDGE [#100101 GE 30. ] #IOG[99,5] = 1 ;
//2 ZEDGE [#100101 GE 50.] ZDO ;
G91 G00 Y100 ;
ZEND ;
//3 ZEDGE [#100101GE 80. ] #IOG[99,5] = 0 ;
G90 G01 X200. F150 ;
M30 ;
```

FRYER MACHINE SYSTEMS, INC
Rob Zahensky

The timing is as follows. (‘**’ represents the satisfaction of the condition.)



Explanation

To use the real time custom macro function, a real time macro command (RTM command) is coded in an NC program.

- Real time macro command (RTM command)

The real time macro command (RTM command) is a macro command that starts execution in phase with an NC statement in the program. After the execution of a real time macro command starts, the RTM command operates independently of the NC statement.
An RTM command is a statement dedicated to the real time custom macro function.
An RTM command consists of a set of one or more real time macro statements (RTM statements).

- Real time macro statement (RTM statement)

The real time macro statement (RTM statement) is a statement included in an RTM command.
One or more RTM statements make up an RTM command.
An RTM statement consists of a macro command and axis control command dedicated to the real time custom macro function.

The axis control command of an RTM statement is an RTM statement including an address. This command is used to exercise axis control.
Example

```
// ZDO ;
G91 G00 X100 ;
ZEND ;
```

(ZDO and ZEND are reserved words required for the axis control command of an RTM statement, and are detailed later.)

The macro command of an RTM statement is a macro statement used with an RTM statement for arithmetic/logical operation and signal control. (In the example below, #RV[1] and #RV[2] are variables dedicated to the real time macro command.)
Example

```
// #IOG[124, 5] = 1 ;
// #RV[1] = #RV[2] * 10 ;
```

FRYER MACHINE SYSTEMS INC
Rob Zahansky

Format

The real time macro command format is shown below.
The RTM command is a command with two slashes (//) prefixed at the start of a block.

//n <real-time-macro-statement> **or** **//n ZDO ;** **<real-time-macro-statement>** **:** **ZEND ;**
n: Modal ID (1 to 10) (Omissible) When a proper number is coded in n, a modal real time macro command is specified. When n is omitted, a one-shot real time macro command is specified.

ZDO to ZEND are detailed later.

17.1 TYPES OF REAL TIME MACRO COMMANDS

17.1.1 Modal Real Time Macro Command / One-shot Real Time Macro Command

Explanation

A command with '//' followed by an RTM statement is referred to as a one-shot real time macro command (one-shot RTM command).
Example:

```
// #RV[1] = 30 ;
```

On the other hand, a command with '//' followed by number n (1 to 10) then an RTM statement is referred to as a modal real time macro command (modal RTM command).
Example:

```
//3 #RV[1] = 30 ;
```

A one-shot RTM command starts when the execution of the first following NC command starts. A started one-shot RTM command is valid until the end of the NC command.
A modal RTM command starts when the execution of the first following NC command starts, as in the case of a one-shot RTM command.
Unlike a one-shot RTM command, however, a started modal RTM command is valid until the automatic operation is completed.

- Start of a real time macro command

An RTM command starts when the execution of the first following NC command starts.

Example:

When NC command (1) starts execution in the program below, macro commands (2) and (4) are executed in succession without waiting for the end of (1).
On the other hand, RTM command (3) starts execution when NC command (5) starts execution after the end of NC command (1).

```
O0001 ;
G90 G00 X30. ;          (1) NC command
#100=0 ;                (2) Macro command
// #RV[0]=1 ;           (3) RTM command
#102=2;                 (4) Macro command
G90 G00 X100. ;         (5) NC command
M30 ;
```


FRYER MACHINE SYSTEMS INC
Rob Zahensky

- End of a real time macro command

When one of the following conditions is satisfied, the RTM command is terminated.

Termination conditions common to one-shot RTM and modal RTM commands
- When RTM command processing is completed
- When a reset occurs

Termination condition specific to a one-shot RTM command
- When the execution of the NC command that started at the same time has ended

However, if the RTM statement being executed is an axis control command, the command is terminated when the execution of the block is completed.
If the execution of the Y10. block is completed before the X100. block in the command below, and the execution of the Y20. block starts, for example, the X100. command of the RTM statement is executed until the end.

```
// ZDO ;
X100. ;
ZEND ;
Y10. ;
Y20. ;
```

Termination condition specific to a modal RTM command
- When ZCANCEL is commanded.

NOTE

1 No one-shot RTM command can be specified using any of the commands indicated below as a trigger. When using any of these commands as a trigger, use a modal RTM command.
- Command related to reference position return
- Command related to AI contour control
- Command related to canned cycles (rigid tapping, drilling cycle, etc.)
- Command related to cutter compensation
- Command related to tool length compensation
- Command related to automatic tool length measurement
- Command related to coordinate system rotation
  Command related to scaling
- Command related to programmable mirror image

2 If an RTM command is specified using, as a trigger, a block such as a block specifying smooth tolerance+ control or a multiple repetitive canned cycle that does not necessarily pass the start point or end point of the command, operation can start or end at a point other than the start point and end point. Do not use such a block as a trigger.

3 Do not restart a program that includes an RTM command.

FRYER MACHINE SYSTEMS INC
Bob Zaharnsky

NOTE

4 When an NC statement used as a trigger for an RTM command represents an auxiliary function, execution continues even if the FIN signal is awaited.
If the following program is executed, for example, the count-up operation of #RV[0] continues until the FIN signal of M55 is returned:

```
O0001 ;
// ZWHILE [1] #RV[0] = #RV[0]+1 ;
M55 ;
G91 X200. ;
   :
```

5 If M02 follows an RTM command, execution continues until a reset occurs, even when the program itself is terminated.
If the following program is executed, for example, the count-up operation of #RV[0] continues until a reset occurs:

```
O0001 ;
// ZWHILE [1] #RV[0] = #RV[0]+1 ;
M02 ;
```

- Priority of commands

If a modal RTM command and one-shot RTM command are specified at the same time, the modal RTM command is executed first.
If multiple modal RTM commands are specified at the same time, the commands are executed in ascending order of ID values.
No priority is applicable to one-shot RTM commands.
The order of execution is as follows:

Modal RTM command with ID value being 1 — High
Modal RTM command with ID value being 2
:
:
Modal RTM command with ID value being 10
One-shot RTM command
:
One-shot RTM command — Low

Example 1)
Priority of modal RTM commands

```
O0001 ;
//1 #RV[0]=1 ;
//3 #RV[0]=3 ;
//2 #RV[0]=2 ;
M02 ;
```

When the program above is executed, the RTM commands are executed in the following order:
#RV[0]=1
#RV[0]=2
#RV[0]=3
So, the value of #RV[0] is 3.

Example 2)
Priority of modal RTM commands and a one-shot RTM command

```
O0001 ;
//3 #RV[0]=3 ;
//1 #RV[0]=1 ;
// #RV[0]=10 ;
//5 #RV[0]=5 ;
```

FRIER MACHINE SYSTEMS INC Rob Zahensky

```
M02 ;
```

When the program above is executed, the RTM commands are executed in the following order:

```
//1 #RV[0]=1
//3 #RV[0]=3
//5 #RV[0]=5
// #RV[0]=10
```

So, the value of #RV[0] is 10.

Example 3)

Priority of one-shot RTM commands

When the following program is executed, the value of #RV[0] is undefined, that is, 1, 2, or 3.

```
O0001 ;
//#RV[0]=1 ;
//#RV[0]=2 ;
//#RV[0]=3 ;
G04P10 ;
M30 ;
```

Even if a priority is applicable, the order of execution or the order of termination can change when an RTM statement includes a control code, ZWHILE or ZEDGE, or axis control command.

Example 4)

The RTM command priority of #RV[0]=1 in a modal command with its ID value being 1 is higher than the priority of #RV[1]=1 in a modal command with its ID value being 2. However, #RV[0]=1 is executed after the end of the execution of the block specifying the axis control command G91 G00 X10., so that #RV[1]=1 is actually executed earlier than #RV[0]=1.

```
O0001 ;
//1 ZDO ;
G91 G00 X10. ;
#RV[0]=1 ;
ZEND ;
//2 #RV[1]=1 ;
G04 P10 ;
M30 ;
```

Example 5)

In the RTM command priority, ZEDGE in a modal command with its ID value being 1 is always a false control code (detailed later). The RTM command priority of #RV[0]=1 in a modal command with its ID value being 1 is higher than the priority of #RV[1]=1 and #RV[2]=1 in a modal command with their ID values being 2. However, #RV[0]=1 is executed after the condition of ZEDGE becomes true (that is, at the second time or later), so that #RV[1]=1 and #RV[2]=1 are executed earlier than #RV[0]=1.

```
O0001 ;
//1 ZEDGE [ #IOG[234.0] EQ 1 ] #RV[0]=1 ;
//2 ZDO ;
#RV[1]=1 ;
#RV[2]=1 ;
ZEND ;
G04 P10 ;
M30 ;
```

- Number of real time macro commands

A program can have multiple RTM commands coded.
Up to six one-shot RTM commands can be specified.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

If one-shot RTM commands more than the maximum allowable number are specified, an alarm PS0397 "RTM BUFFER OVER" or PS0398 "ID OVER IN BUFFER" is issued.

Up to ten modal RTM commands can be specified. When specifying modal RTM commands, ensure that there is no duplicate ID.

If there is a duplicate ID, an alarm PS0397 "RTM BUFFER OVER" or PS0398 "ID OVER IN BUFFER" is issued. If an incorrect ID is specified, an alarm PS0404 "RTM ERROR" is issued.

In all paths, up to 16 RTM commands can be executed simultaneously.
When an axis control command is included, up to four commands can be executed simultaneously.

NOTE

1 In a block specifying an RTM statement, no NC command can be coded.

2 If the maximum specifiable number of commands, an alarm PS0397 "RTM BUFFER OVER" or PS0398 "ID OVER IN BUFFER" is issued. If the maximum number of simultaneously executable commands is exceeded, an alarm PS0401 "EXEC CMD NUM OVER IN SAME TIME" is issued.

3 If an NC statement to be used to trigger an RTM command is specified in a block (e.g., small block) that ends in a very short time, an RTM statement programmed to start at a different timing may be executed simultaneously. If the following is specified, for example, #RV[0]=1 and #RV[1]=2 may be executed simultaneously:

```
// #RV[0]=1 ;
G91 G01 X0.002 ; F5000
// #RV[1]=2 ;
X0.001 ;
```

4 When a function for reading multiple blocks in advance is used, the same ID must not be coded in the scope of blocks read in advance. In the following program, for example, while //1 #RV[0]=#100101 is being executed during AI contour control, //1 #RV[1]=#100101 and //1 #RV[2]=#100101 are also read in advance. So, an alarm PS0398 "ID OVER IN BUFFER" may be issued.

```
//1 #RV[0]=#100101 ;
X#100 ;
//1 #RV[1]=#100101 ;
X#100 ;
//1 #RV[2]=#100101 ;
X#100 ;
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

5 When a function for reading multiple blocks in advance is used, up to three blocks among the blocks read in advance can trigger an RTM command.
For example, if the blocks up to the block of (2) are read in advance during execution of (1) in the program below, up to three blocks can trigger an RTM command. In the program below, the number of NC blocks that trigger an RTM command exceeds 3, so that the RTM command of (a) must not be coded.

```
X30. Y50. ; ................... (1)
// Z-30. ;
// #RV[0]=#RV[0]+1 ;
X3. Y16. ;                ← NC block 1 triggering an RTM command
X-23. Y4. ;
// #RV[1]=#RV[1]+1 ;
//2 Z30. ;
X-2. Y9. ;                ← NC block 2 triggering an RTM command
X17. Y5. ;
// #RV[2]=#RV[2]+1 ;
X-2. Y9. ;                ← NC block 3 triggering an RTM command
// #RV[3]=#RV[3]+1 ; ... (a)
X-12. Y-3. ;
X-100. Y200. ; ............. (2)
```

- Check of the execution state of real time custom macro

Execution state of real time custom macro can be checked by diagnosis data.
Modal RTM command of ID1 to ID10 are displayed in the diagnosis data R01 to R10 (No.4800#0 to No. 4801#1). One-shot RTM command is displayed in the diagnosis data RSH (No.4801#2). When the RTM command to start the executed in cycle start, diagnosis data "1" is set. When the RTM command is completed, "0" is set. When the automatic operation is in single block stop state or feed hold stop state will be "0". When the reset has been input, or when the RTM command has been canceled in the ZCANCEL statement, "0" is set.

NOTE

RTM command is terminated by reset input. When the axis control of the RTM command, axis movement will continue, but the diagnosis data has become "0".

- Diagnosis information of code consumption rate in real time custom macro

The code consumption rate displayed to diagnosis data (No.7080-No.7090). The rate of code size becomes the guide for correcting/editing RTM command.

- Reserved words

The following reserved words are used with real time custom macros:

- Reserved words dedicated to real time custom macros
  ZDO, ZEND, ZONCE, ZWHILE, ZEDGE
- Reserved words shared with custom macros
  AND, OR, XOR, MOD, EQ, NE, GT, LT, GE, LE, SIN, COS, TAN, ASIN, ACOS, ATAN, ATN, SQRT, SQR, ABS, BIN, BCD, ROUND, RND, FIX, FUP, LN, EXP, POW

Be sure to fully spell out the reserved words for real time custom macros. For example, ‘ZONCE’ must not be coded as ‘ZON’ or ‘ZONC’.

FRIEP MACHINE SYSTEMS INC
Rob Zaranosky

Diagnosis data

	#7	#6	#5	#4	#3	#2	#1	#0
4800	R08	R07	R06	R05	R04	R03	R02	R01
	#7	#6	#5	#4	#3	#2	#1	#0
4801						RSH	R10	R09

[Data type] Bit path

R01 to R10 Execution state of modal RTM command ID1 to ID10.

#2 RSH Execution state of one-shot RTM command.
Execution state of RTM command will be displayed.
When the RTM command to start the executed in cycle start, diagnosis data "1" is set. When the RTM command is completed, "0" is set. When the automatic operation is in single block stop state or feed hold stop state will be "0". When the reset has been input, or when the RTM command has been canceled in the ZCANCEL statement, "0" is set.

7080	Code consumption rate for Modal RTM command with ID-1
7081	Code consumption rate for Modal RTM command with ID-2
to	to
7089	Code consumption rate for Modal RTM command with ID-10

[Data type] Word
[Unit of data] %
[Valid data range] 0 to 100

7090	Code consumption rate for One-shot RTM command

[Data type] Word
[Unit of data] %
[Valid data range] 0 to 100

17.1.2 Cancel Command for Real Time Custom Macro

Overview

Modal RTM command in execution can be canceled by the program command. RTM command in execution is canceled, and new RTM command is available in the same modal ID.

Format

```
//n ZCANCEL ;
    n : Modal ID(1 to 10)
```

Explanation

Cancel command for real time custom macro uses the ZCANCEL statement.
ZCANCEL statement is the same as RTM command; the start of the cancel command is simultaneously with the start of the execution of the first following NC command.
If ZCANCEL statement was started during the axis control of RTM statement, the axis movement does not stop immediately, axis is stopped when the block in the execution has been completed. If you immediately stop the axis control of RTM statement, it controls the PMC axis reset signal ECLRg of the relevant group.
The following shows a program example is shown in Fig. 17.1.2 (a).

FRYER MACHINE SYSTEMS INC
Rob Zahansky

Modal RTM command of ZEDGE is executed between N1 and N2 blocks, and the RTM command is canceled in the N2 block. Modal RTM command of ZWHILE is executed from the N3 block.

```
:
//1 ZEDGE[ #RV[0] NE #RV[1] ] #RVS[0] = 1.0 ;
N1 G04 ;
:
//1 ZCANCEL ;
N2 G04 ;
:
//1 ZWHILE[ #RV[2] EQ #RV[3] ] #RVS[1] = 1.0 ;
N3 G04 ;
:
M30 ;
```

Fig.17.1.2 (a) Program example of ZCANCEL statement

When the ZCANCEL statement and next RTM command by look-ahead is buffered, alarm PS0398, “ID OVER IN BUFFER” occurs. Commands the G code preventing buffering as N1 block of Fig. 17.1.2 (b). Execution of ZCANCEL statement is completed, the buffering of next RTM command is executed.

```
:
//1 ZCANCEL ;
N1 G04.1 ;
//1 ZWHILE[ #RV[2] EQ #RV[3] ] #RVS[1] = 1.0 ;
N2 G04 ;
:
M30 ;
```

Fig.17.1.2 (b) Description example of ZCANCEL statement and next RTM command.

Limitation

ZCANCEL statement can not be commanded along with other RTM statement. When the program is described as shown in Fig. 17.1.2 (c), alarm PS0394, “ILLEGAL SENTENCE CONTROL” occurs.

```
:
//1 ZDO ;
ZCANCEL ;
ZEND ;
N1 G04 ;
:
:
M30 ;
```

```
:
//1 ZDO ;
ZCANCEL ;
#RV[0] = 1.0 ;
ZEND ;
N1 G04 ;
:
M30 ;
```

Fig.17.1.2 (c) Example of alarm PS0394

RTM command can not be described after the ZCANCEL statement. Moreover, ZCANCEL statement can not be buffered with the next RTM command. When the program is described as shown in Fig.17.1.2 (d), alarm PS0398, “ID OVER IN BUFFER” occurs.

```
:
//1 ZCANCEL ;
//1 #RV[0] = 1.0 ;
N1 G04 ;
:
:
M30 ;
```

```
:
//1 ZCANCEL ;
N1 G04 ;
//1 #RV[0] = 1.0 ;
N2 G04 ;
:
M30 ;
```

Fig.17.1.2 (d) Example of alarm PS0398

ZCANCEL statement can be described in the RTM command only. When the ZCANCEL statement is described in the NC command, alarm PS0396, “ILLEGAL RTM SENTENCE CONTROL” occurs.

17.2 VARIABLES

Overview

With real time custom macros, the following variables can be handled:

- System variables dedicated to real time custom macros
- Variables (RTM variables) dedicated to real time custom macros
- System variables for some custom macros

The RTM variables mean the variables dedicated to real time custom macros.

Table 17.2 List of usable variables

		Variable number	Real time custom macro	Custom macro	Macro executor (Execution macro)
Real time custom macro variables		#RV[0] to #RV[99], #RVS[0] to #RVS[31]	Enabled	Not enabled	Not enabled
Custom macro variables	Local variables	#1 to #33	Not enabled	Enabled	Enabled
	Common variables	#100 to #199, #500 to #999, #98000 to #98499	Enabled	Enabled	Enabled
Macro executor variables	Local variables	#1 to #33	Not enabled	Not enabled	Enabled
	Common variables	#100 to #199, #500 to #999,	Not enabled	Not enabled	Enabled
	P-CODE variables	#10000 to #89999	Enabled (READ only)	Enabled	Enabled

The variables (system variables and RTM variables) dedicated to real time custom macros are the variables specific to the real time custom macro function. Those variables cannot be used with the custom macro function.

When a real time custom macro to use the P-CODE variable, system variable #8570=1 must be commanded before the real time macro command. P-CODE variable is ready for use. In real time macro command #8570=1 can not be commanded. Alarm (PS0390) "ILLEGAL MACRO VAR" is generated.
P-CODE variable can be only READ. If WRITE is commanded, alarm (PS0116) "WRITE PROTECTED VARIABLE" is generated. WRITE will use the custom macro common variable.

NOTE

1. Real time custom macros do not support <null>. Do not use it.
2. To use the custom macro common variable #150 to #199 and #550 to #999, enable the addition of custom macro common variables (bit 6 (NCV) of parameter No.8135 is 0).
3. To use the custom macro common variable #98000 to #98499, enable the addition of custom macro common variables 1000.
4. P-CODE variable can be used only when the macro executor function is enabled. Alarm (PS0390) "ILLEGAL MACRO VAR" is generated when the setting is insufficient.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

> ⚠ **CAUTION**
> Calculation processing of custom macros and execution macro is performed at the stage of buffering that program analysis. On the other hand, calculation processing of real time custom macro is performed at the stage of NC statements execution. The calculation is performed at different timings. It must be a program that is considered.

17.2.1 Variables Dedicated to Real Time Custom Macros

These variables are dedicated to real time custom macros. The variables are classified as system variables and RTM variables.

17.2.1.1 System variables

System variables dedicated to real time custom macros

Format

#IOp [m, n] Bit-by-bit read/write
#IOpB [m] Byte-by-byte read/write
#IOpW [m] Word-by-word read/write
#IOpD [m] Double word-by-double word read/write

p : Signal type (X, G, F, Y, D, R)
m : Signal byte address
n : (Used for bit-by-bit read/write only) Signal address bit number (0 to 7)

PMC interface signals are read and written. Bit-by-bit and byte-by-byte read/write operations are possible.
The following signals can be used:

Variable name	Signal type	Read	Write
#IOX #IOXB #IOXW #IOXD	X	Possible	Impossible
#IOG #IOGB #IOGW #IOGD	G	Impossible	Possible
#IOF #IOFB #IOFW #IOFD	F	Possible	Impossible
#IOY #IOYB #IOYW #IOYD	Y	Impossible	Possible
#IOD #IODB #IODW #IODD	D	Possible	Possible
#IOR #IORB #IORW #IORD	R	Possible	Possible

FRYER MACHINE SYSTEMS INC
Rob Zahensky

The valid signal address range is usable range of selected PMC memory type.
For details of PMC memory type, refer to “PMC Programming Manual (B-64513EN)”.

When writing to a signal, make the variable unprotected on the PMC signal protection screen (described later) beforehand.

Specify an address by using m and n.
Example:

#IOF[1, 3]	F1.3 bit type
#IOG[1, 5]	G1.5 bit type
#IOFB[32]	F32 byte type
#IOGB[12]	G12 byte type

Read/write operations are performed in the same as for an ordinary macro statement.
Example:

#RV[0]=#IOFB[32]	Assigns F32 to #RV[0].
#IOG[99.3] = 1	Sets G99.3 to “1”.

If a signal at a nonexistent address is specified, alarm PS0387 "ILLEGAL RTM DI/DO VAR" or PS0389 "ILLEGAL RTM SIGNAL BIT" is issued.

⚠ CAUTION

1. Controls handling other signals, such as a ladder or macro executor, must not write to a signal address being written to by an RTM statement. Ensure that a single control writes to the same byte signal address.
   For example, when the G000.0 signal is written to by an RTM statement, do not write to the G000.7 signal from a ladder.
2. Ensure that the same F signal is not read from by an RTM statement and written to by the NC simultaneously.
3. Ensure that the same G signal is not written to by an RTM statement and read from by the NC simultaneously.
4. For word-by-word read/write, specify an even-numbered address and for double word-by-double word read/write, specify an address that is a multiple of four.

NOTE

1. The system variables described above are not supported for a multi-path PMC.
2. System relay area cannot be written. In the case of PMC memory type-C or D, system relay address is Z signal address, but Z signal address area cannot be read nor written.

- PMC signal protection

Whether to enable a write to a signal handled by an RTM statement can be set. This function protects against a malfunction due to incorrect coding.

On the PMC signal protection screen, set whether to enable a write to a signal.
If an RTM statement makes an attempt to write to a signal write-protected on the PMC signal protection screen, an alarm PS0403 "ACCESS TO RTM PROTECT VAR" is issued at the time of execution.

Set whether to enable a write to each address of Y and G on a byte-by-byte basis.
For each of addresses D and R, set a write-enabled range.

For the unwritable signals (X, F), the screen is not displayed.

FRYER MACHINE SYSTEMS INC
Roy Zaherisky

Before changing an address on the PMC signal protection range specification screen, perform a clear operation first and then input a new address.

- Input/output

A value set for PMC signal protection can be input/output.

- Input/output format

After outputting PMC signal protection, one file (DIDOENBL.TXT) is created.
Please execute input/output operation in EDIT mode.
The output format is as follows:

L Specification method
  0: Byte specification
  1: Range specification
Q Alphabetic signal address
  0: G, 2: Y, 5: R, 9: D
K (for use in range specification only) Index number (0 to 9)
R Address number for byte specification
  Start address number for range specification
P Protection value for byte specification
  0: Not writable
  1: Writable
  End address number
  for range specification

Example of output

```
%
L0Q2R0000P0
L0Q2R0001P1          Y0 to Y127 protect
:                    information
L0Q2R0127P1
:
L0Q0R0000P1
L0Q0R0001P1          G0 to G767 protect
:                    information                Byte
L0Q0R0767P0                                     specification
L0Q0R1000P1
:                    G1000 to G1767 protect
L0Q0R1767P0          information
:
L1Q5K0R00002P00014 — R2 to R14 are writable
:                    (Index 0)                  Range
L1Q9K9R00032P00127 — D32 to D127 are writable   specification
M02                  (Index 9)
%
```

17.2.1.2 Real time macro variables (RTM variables)

The real time macro variables (RTM variables) are variables dedicated to real time custom macros.
The RTM variables are classified as volatile real time macro variables (volatile RTM variables) and nonvolatile real time macro variables (nonvolatile RTM variables).

The data of a nonvolatile RTM variable is preserved even when the power is turned off.
The data of a volatile RTM variable is cleared to 0 when the power is turned off.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Format

#RV [m] Volatile RTM variable

m: Volatile RTM variable number (0 to 99)

#RVS [n] Nonvolatile RTM variable

n: Nonvolatile RTM variable number (0 to 31)

NOTE

1 RTM variables can be used with an RTM statement only. RTM variables cannot be used with an NC statement and macro statement.
2 No RTM variable assumes a "null" value.
3 Volatile RTM variables are cleared to 0 by a reset. On the other hand, nonvolatile RTM variables are not cleared to 0 by a reset.

Explanation

- Input/output

RTM variables can be input/output in a specified format.
Both nonvolatile RTM variables and volatile RTM variables can be input/output.

- Input/output format

After outputting RTM variables, one file (RTMMACRO.TXT) is created.
The output format is described below.
The value of an RTM variable is output in hexadecimal as a double-precision floating-point data bit image.
When a nonvolatile RTM variable is output, G10L87 is followed by an RTM variable number and a variable value.
When a volatile RTM variable is output, G10L88 is followed by an RTM variable number and a variable value.

Example of output

```
%
G10L87P0(3FE0000000000000)
G10L87P1(4000000000000000)
:                                  Nonvolatile RTM variable
G10L87P30(4010000000000000)
G10L87P31(4014000000000000)

G10L88P0(4008000000000000)
G10L88P1(3FD9999999800000)
:                                  Volatile RTM variable
G10L88P98(3FF0000000000000)
G10L88P99(4010000000000000)
M02
%
```

Execute input/output operation in EDIT mode.

17.2.2 Custom Macro Variables

With real time custom macros, a part of the custom macro variables (part of the system variables) can be handled.

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

17.2.2.1 System variables

With real time custom macros, position-related information among the system variables of the custom macros can be handled.

- **Position information #100001 to #100200, #118601 to #118700 (Attribute: Read only)**

Block end position #100001 to #100050
Specified current position of input unit (machine coordinate system) #100051 to #100100
Specified current position (workpiece coordinate system) #100101 to #100150
Skip position #100151 to #100200
Specified current position of machine unit (machine coordinate system) #118601 to #118650
Current position of machine unit (machine coordinate system) #118651 to #118700

By readying the values of system variables #100001 to #100200, the end position of the previous block, the current positions (machine coordinate system and workpiece coordinate system), and skip signal position can be found.
System variable #118601 to #118650 can obtain the specified current position (machine coordinate system) in machine units, this machine coordinate is the value of the specified position without servo delay (servo position deviation) and acceleration / deceleration delay.
System variable #118651 to #118700 can obtain the current position (machine coordinate system) in machine units, this machine coordinate is the value with the servo delay(servo position deviation) and the acceleration / deceleration delay together.

Variable No.	Position information	Coordinate system
#100001 #100002 : #100050	Block end position on 1st axis Block end position on 2nd axis : Block end position on 50th axis	Workpiece coordinate system
#100051 #100052 : #100100	Current position on 1st axis Current position on 2nd axis : Current position on 50th axis	Machine coordinate system (input unit)
#100101 #100102 : #100150	Current position on 1st axis Current position on 2nd axis : Current position on 50th nd axis	Workpiece coordinate system
#100151 #100152 : #100200	Skip position on 1st axis Skip position on 2nd axis : Skip position on 50th axis	Workpiece coordinate system
#118601 #118602 : #118650	1st axis current position 2nd axis current position : 50th axis current position	Machine coordinate system (machine unit). Value of the specified current position without servo delay and acceleration / deceleration delay.
#118651 #118652 : #118700	1st axis current position 2nd axis current position : 50th axis current position	Machine coordinate system (machine unit). Value of the current position with servo delay and acceleration / deceleration delay.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

NOTE
1 The value of a variable with a number greater than the number of controlled axes is undefined.
2 The end position (ABSIO) of a skip (G31) block is the skip signal ON position if the skip signal is turned on. If the skip signal is not turned on, the block end point is undefined.
3 As block end position information #100001 to #100050, the end positions read in advance are obtained instead of the end positions of the block currently being executed.

- **Servo positional deviation #100251 to #100300 (Attribute: Read only)**

By reading the values of system variables #100251 to #100300, the servo positional deviation on each axis can be found.

Variable No.	Position information
#100251	Servo positional deviation on 1st axis
#100252	Servo positional deviation on 2nd axis
:	:
#100300	Servo positional deviation on 50th axis

NOTE
The value of a variable with a number greater than the number of controlled axes is undefined.

- **Remaining travel distance #100801 to#100850 (Attribute: Read only)**

By reading the values of system variables #100801 to #100850, the remaining travel distance on each axis can be read.

Variable No.	Position information
#100801	Travel distance on 1st axis
#100802	Travel distance on 2nd axis
:	:
#100850	Travel distance on 50th axis

NOTE
1 The value of a variable with a number greater than the number of controlled axes is undefined.
2 System variables not described here are unusable.
3 The name of a system variable cannot be specified.
Do not specify a command as indicated below.
//1 #RV[0]=[#_ABSOT[1]] ;

17.2.2.2 Local variables

The local variables (#1 to #33) cannot be used.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

17.2.3 Reading and writing variables for another path

Overview

By adding a path number to the high-order 8th and 9th digits of a variable, it is possible to read and write a common variable or a system variable for another path.
For a list of variables that can be read and written, see "List of variables that can be read and written".

Format

#ppxxxxxxx

```
pp : Path number
     Omitted = Local path
        1    = First path
        :       :
        4    = Fourth path
xxxxxxx: Variable number
Common variable (#100 to #499, #500 to #999, #98000 to #98499) or
P-CODE variable (#10000 to #19999, #20000 to #89999) or
system variable number (#100001 and above).
```

NOTE
Available common variables and system variables differ depending on the system configuration.

Example

In the case of Fig.17.2.3 (a), if #100=#100023 is executed on path 1, the machine coordinate 345.789 on the third axis on path 1 is read into variable #100. If #100=#20100023 is executed, the machine coordinate 45.0 on the third axis on path 2 is read into variable #100.

Machine coordinates for path 1		Machine coordinates for path 2	
X1	123.456	X2	-123.456
Y1	45.670	Z2	78.900
Z1	345.789	C2	45.000

Fig.17.2.3 (a) Example of reading variables for another path in 2 path system

NOTE
1 Variable names cannot be specified.
2 If a path number out of the range is specified, alarm PS0390, “ILLEGAL MACRO VAR” is issued.
3 Local variables for another path cannot be read or written. Alarm PS0390, “ILLEGAL MACRO VAR” is issued.

List of variables that can be read and written

This function can read and write the following variables for another path.
For details of system variables, see the description of variables in the previous section.
R and R/W is variable attributes, and respectively represents READ only, and READ/WRITE.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Common variable

System variable number	Attribute	Description
#100 to #149(#199) #200 to #499 #500 to #549(#999) #98000 to #98499	R/W	Available common variables differ depending on the system configuration.

- P-CODE variable

System variable number	Attribute	Description
#10000 to #19999 #20000 to #89999	R	

- Position information

System variable number	Attribute	Description
#100001 to #100050	R	End point position of the block (workpiece coordinate system)
#100051 to #100100	R	Specified current position of input unit (machine coordinate system)
#100101 to #100150	R	Specified current position (workpiece coordinate system)
#100151 to #100200	R	Skip position (workpiece coordinate system)
#118601 to #118650	R	Specified current position of machine unit (machine coordinate system)
#118651 to #118700	R	Current position of machine unit (machine coordinate system)

- Servo position deviation

System variable number	Attribute	Description
#100251 to #100300	R	Servo positional deviation

- Distance to go

System variable number	Attribute	Description
#100801 to #100850	R	Distance to go

17.3 ARITHMETIC AND LOGICAL OPERATION

With the real time custom macros, the following arithmetic and logical operations can be specified:

Table 17.3 (a) Arithmetic and logical operation

Type of operation	Operation	Description
(1) Definition, substitution	#i=#j	Definition or substitution of a variable
(2) Additive operation	#i=#j+#k #i=#j-#k #i=#j OR #k #i=#j XOR #k	Addition Subtraction Logical OR (Bit by bit for 32 bits) Exclusive OR (Bit by bit for 32 bits)
(3) Multiplicative operation	#i=#j*#k #i=#j/#k #i=#j AND #k #i=#j MOD #k	Multiplication Division Logical AND (Bit by bit for 32 bits) Remainder (#j and #k are rounded to an integer to find a remainder. When #j is negative, #i is also negative.)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Type of operation	Operation	Description
(4) Function	#i=SIN[#j]	Sine (in deg)
	#i=COS[#j]	Cosine (in deg)
	#i=TAN[#j]	Tangent (in deg)
	#i=ASIN[#j]	Arcsine
	#i=ACOS[#j]	Arccosine
	#i=ATAN[#j]	Arctangent (1 argument). ATN is acceptable.
	#i=ATAN[#j]/[#k]	Arctangent (2 arguments). ATN is acceptable.
	#i=ATAN[#j,#k]	Ditto
	#i=SQRT[#j]	Square root. SQR is acceptable.
	#i=ABS[#j]	Absolute value
	#i=BIN[#j]	Binary conversion from BCD
	#i=BCD[#j]	BCD conversion from binary
	#i=ROUND[#j]	Rounding. RND is acceptable.
	#i=FIX[#j]	Discarding fractional digits
	#i=FUP[#j]	Rounding fractional digits upward to an integer
	#i=LN[#j]	Natural logarithm
	#i=EXP[#j]	Exponent with e (2.718……) used as the base
	#i=POW[#j,#k]	Power (#j to #k power)

- Constant specifiable in <expression>

+0.00000000001 to +999999999999
-999999999999 to -0.00000000001
Up to 12 decimal digits can be specified.
If the maximum allowable number of digits is exceeded, alarm PS0004, “INVALID BREAK POINT OF WORDS” is issued.

NOTE

1 The ADP function is not available.
2 With an RTM statement, the external output commands (BPRNT, DPRNT, POPEN, and PCLOS) are unavailable.
3 The FS16i compatibility specifications are not applicable. Bit 0 (F16) of parameter No. 6008 = 1 (with the operation result precision based on the FS16i compatibility specifications) is invalid.
4 For information on decreased precision, see "When bit 0 (F16) of parameter No. 6008 is set to 0" in "Caution concerning decreased precision" in the CUSTOM MACRO chapter.
5 Bit 0 (NAT) of parameter No. 6004 is valid in an RTM command.
6 The setting of bit 5 (SBM) of parameter No. 6000 and bit 7 (SBV) of parameter No. 6000 is invalid. For the single block specifications, see the separate item.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

17.4 CONTROL ON REAL TIME MACRO COMMANDS

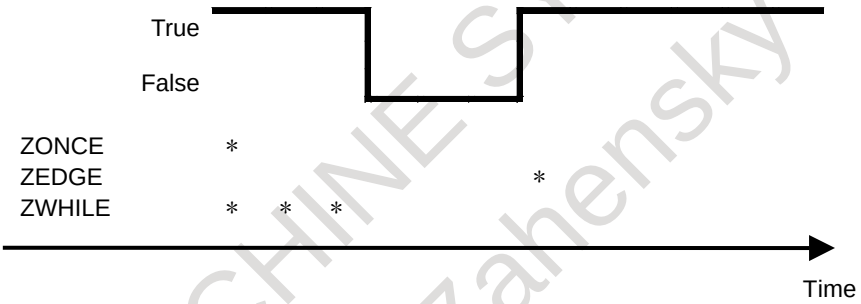
Explanation

By using a reserved word for controlling statements in an RTM command, the flow of the RTM command can be changed or multiple statements can be controlled as a set of statements. Four reserved words are used to control an RTM command as indicated below.

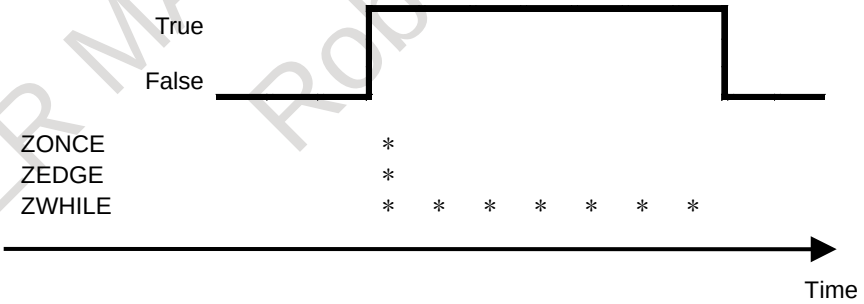
Reserved word name	Syntax	Meaning
ZONCE	// ZONCE A B	If A is true, B is executed.
ZEDGE	// ZEDGE A B	If A becomes true, B is executed.
ZWHILE	// ZWHILE A B	While A is true, B is executed repeatedly.
ZDO...ZEND	// ZDO B1 B2 B3 ZEND	(Multiple statements) B1, B2, and B3 are sequentially executed.

The timing chart of an RTM command using these reserved words is indicated below. (Multi-statement control ZDO...ZEND is excluded.)
When the condition of a each reserved word is True, it shows '*'.

When condition A makes transitions from True to False to True



When condition A makes transitions from False to True to False



FRYER MACHINE SYSTEMS INC
Rob Zahensky

17.4.1 Conditional Branch (ZONCE Statement)

After ZONCE, <conditional-expression> and <real-time-macro-statement> are coded.

- //(n) ZONCE [<conditional-expression>] <real-time-macro-statement>

If <conditional-expression> is true, <real-time-macro-statement> is executed. If <conditional-expression> is false, <real-time-macro-statement> is not executed, but control exits from the ZONCE command to terminate the execution. <conditional-expression> conforms to the custom macro specifications.

// ZONCE A B (If A is true, B is executed.)
If A is true, the command is terminated after executing B.
If A is false, the command is terminated without executing B.

If the workpiece coordinate on the first axis is greater than 30, the Y1.0 signal is output.
// ZONCE [#100101 GT 30.] #IOY[1,0] = 1 ;

If the workpiece coordinate on the first axis is greater than 30 and #RV[0] matches #RV[3], the Y1.0 signal is output.
// ZONCE[[#100101 GT 30.] AND [#RV[0] EQ #RV[3]]] #IOY[1,0]=1 ;

In <real-time-macro-statement>, multiple RTM statements can be coded.
In this case, code the following by using ZDO...ZEND of multi-statement structure:
// ZONCE [<conditional-expression>] ZDO ;
<real-time-macro-statement-1> ;
<real-time-macro-statement-2> ;
:
ZEND ;

If the workpiece coordinate on the second axis is equal to or less than 10, the rapid traverse override value is changed.
// ZONCE [#100102 LE 10.] ZDO ; **#IOG[14,0]=0 ;** **#IOG[14,1]=1 ;** **ZEND ;**

However, if <conditional-expression-1> specifies an axis control command, be sure to use ZDO...ZEND even when a single statement is used.

If the workpiece coordinate on the first axis is greater than 30 and #RV[0] matches #RV[3], a movement on the A-axis starts.
//1 ZONCE [[#100101 GT 30.] AND [#RV[0] EQ #RV[3]]] ZDO ; **G91 G00 A20. ;** **ZEND ;**

Similarly, use ZDO...ZEND for a multi-statement including an axis control command.

If the workpiece coordinate on the second axis is equal to or less than 10, a movement on the V-axis starts and the Y1.0 signal is set to “1”.
//1 ZONCE [#100102 LE 10.] ZDO ; **G91 G00 V10. ;** **#IOY[1,0] = 1 ;** **ZEND ;**

FENVER MACHINE SYSTEMS INC
Rob Zahensky

17.4.2 Condition Transition (ZEDGE Statement)

After ZEDGE, <conditional-expression> and <real-time-macro-statement> are coded.
// ZEDGE A B (If A becomes true, B is executed.)

- //(n) ZEDGE [<conditional-expression>] <real-time-macro-statement>

When <conditional-expression> makes a transition from false to true, an RTM statement specified after <conditional-expression> is executed. <conditional-expression> conforms to the custom macro specifications.
With the ZEDGE command, the result of the first evaluation of <conditional-expression> is always false.

Explanation

The difference between the ZONCE command and ZEDGE command is that if <conditional-expression> is false, the RTM statement itself is terminated in the case of ZONCE, while the evaluation of <conditional-expression> continues until <conditional-expression> becomes true in the case of ZEDGE. (With a one-shot real time macro command, the evaluation of <conditional-expression> continues until the execution of the NC statement that started operation simultaneously with the command ends.)

On the rising edge of the G address signal 4.3, the workpiece coordinate on the third axis is read.
// ZEDGE [#IOG[4,3] EQ 1] #RV[0]=#100103 ;

In the example above, even if the [#IOG[4,3] EQ 1] is true from the beginning, #RV[0]=#100103 of the RTM statement is not executed. #RV[0]=#100103 is executed when the result of evaluation of [#IOG[4,3] EQ 1] changes from false to true.

As in the case of ZONCE, multiple RTM statements can be coded in <real-time-macro-statement>.
If <conditional-expression-1> specifies an axis control command, be sure to use ZDO...ZEND even when a single statement is used.
Code the following by using ZDO...ZEND of multi-statement structure:

```
// ZEDGE [<conditional-expression>] ZDO ;
<real-time-macro-statement-1> ;
<real-time-macro-statement-2> ;
   :
  ZEND ;
```

On the falling edge of the X address signal, a movement on the B-axis is started and the Y1.0 signal is set to “1”.
// ZEDGE [#IOX[1,3] EQ 0] ZDO ; **G91 G00 B10. ;** **#IOY[1,0] = 1 ;** **ZEND ;**

On the rising edge of the G address signal, a movement on the U-axis is started.
// ZEDGE [#IOG[4,3] EQ 1] ZDO ; **G91 G00 U25. ;** **ZEND ;**

If the remaining travel distance on the second axis is equal to or less than 10, the rapid traverse override value is changed.
// ZEDGE [#100802 LE 10.] ZDO ; **#IOG[14,0]=0 ;** **#IOG[14,1]=1 ;** **ZEND ;**

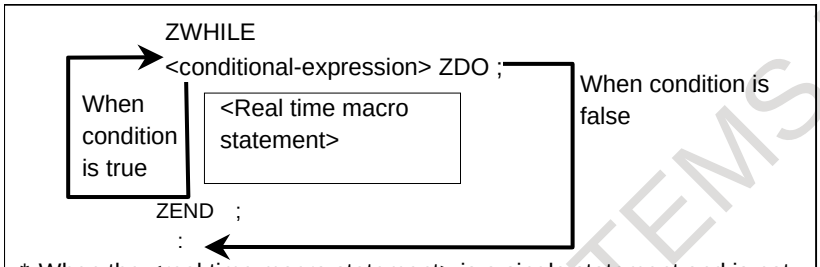
FRYER MACHINE SYSTEMS INC Rob Zahanski

17.4.3 Repetition (ZWHILE Statement)

After ZWHILE, a <conditional statement> is coded.
// ZWHILE A B (While A is true, B is executed repeatedly.)

- //(n) ZWHILE [<conditional-expression>] <real-time-macro-statement>

While <conditional-expression> is true, the RTM statement specified after <conditional-expression> is executed.
If <conditional-expression> is not satisfied, the ZWHILE statement is terminated, and the next block is processed.



* When the <real time macro statement> is a single statement and is not an axis control statement, ZDO and ZEND are not required.
* When “conditional-expression” is fulfilled and “Real time macro statement” is complete, interpolation cycle time has passed before the next evaluation of "ZWHILE conditional-expression".

Explanation

While <conditional-expression> is true, the command or commands between ZDO and ZEND after ZWHILE are executed.
If <conditional-expression> is not satisfied, the command after ZEND is processed.
The same <conditional-expression> and operators as for the ZONCE statement are used.

While the F address 234.1 signal is “1”, an incremental movement on the U-axis is repeatedly performed, and #RV[0] is incremented by 1 each time.
//1 ZWHILE [#IOF[234,1] EQ 1] ZDO ; **G91 G00 U10. ;** ** #RV[0] = #RV[0]+1 ;** **ZEND ;**

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

17.4.4 Multi-statement (ZDO...ZEND Statement)

- **//(n) ZDO ;**
  <real-time-macro-statement-1> <real-time-macro-statement-2>
  <real-time-macro-statement-3> …
  ZEND ;

One or multiple statements enclosed in ZDO...ZEND are regarded as a one RTM statement (multiple-statement structure).

Explanation

The maximum number of RTM statements in one multi-statement (between ZDO...ZEND) is roughly five, but it is subject to change (Because it depends on the contents of RTM statements).

ZDO...ZEND is used in the following cases:

- When multiple <real time macro statements> are handled as a series of commands

If the workpiece coordinate on the first axis exceeds 30, the rapid traverse override value is changed.
// ZEDGE [#100101 GT 30.] ZDO ; **#IOG[14,0]=1 ;** **#IOG[14,1]=0 ;** **ZEND ;**

- When an axis control command is specified

On the rising edge of the X signal 7.0, a movement on the V-axis is started.
// ZEDGE [#IOX[7,0] EQ 1] ZDO ; **G91 G00 V35. ;** **ZEND ;**

- Nesting

ZONCE, ZEDGE, ZWHILE, and ZDO...ZEND cannot be nested and overlapped.
For details, see the following:

1. ZONCE, ZEDGE, ZWHILE, and ZDO...ZEND may be used any number times.

```
// ZWHILE [···] ZDO ;
    Processing
ZEND ;
  :
// ZONCE[···] ZDO ;
    Processing
ZEND ;
  :
```

2. One ZDO...ZEND range must not overlap another ZDO...ZEND range.

```
// ZONCE [···] ZDO ;
    Processing
// ZWHILE [···] ZDO ;
  :
ZEND ;
    Processing
ZEND ;
```

3. ZONCE, ZEDGE, ZWHILE, and ZDO...ZEND must not be nested.

```
// ZWHILE [···] ZDO ;
  :
  ZONCE [···] ZDO ;
    :
    ZEDGE [···] ZDO ;
      Processing
    ZEND ;
    :
  ZEND ;
  :
ZEND ;
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Endless loop

An endless loop is formed if the conditional expression enclosed in brackets after the ZWHILE statement is always satisfied.

Example)

In the example below, #RV[0] is counted up unconditionally.

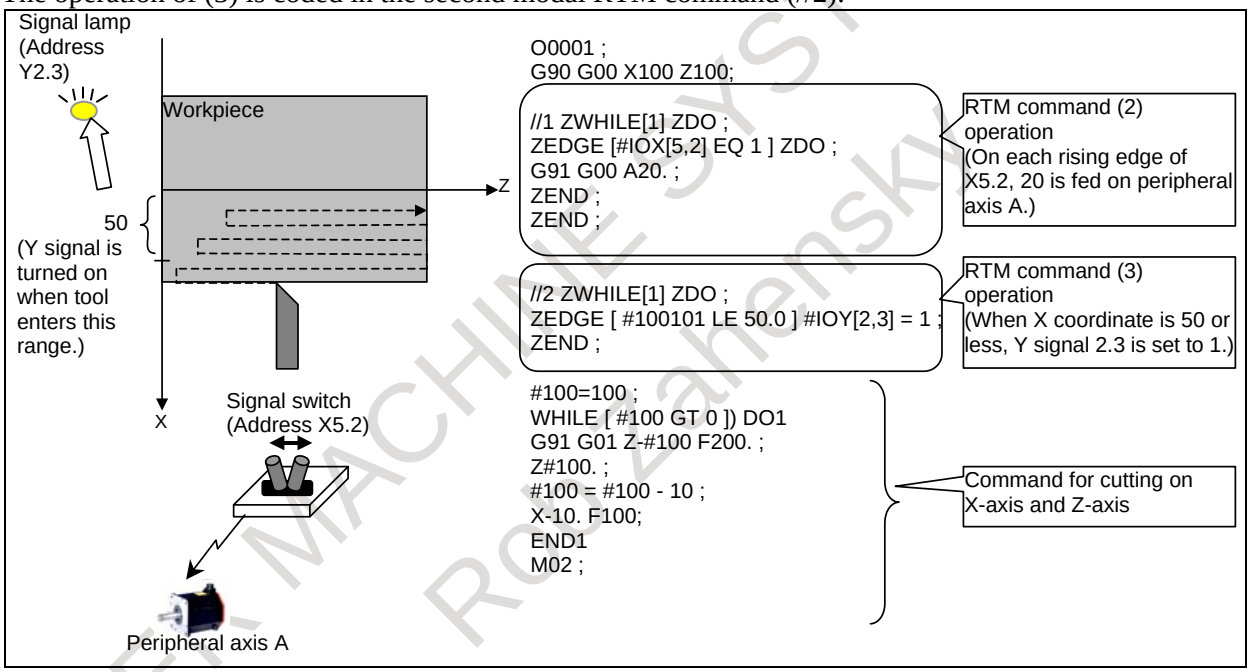
```
// ZWHILE [1] #RV[1]=#RV[1]+1 ;
```

Sample program

The sample program below exercises the following three control operations at the same time.

(1) A cutting operation is performed on the X-axis and Z-axis.
(2) On each rising edge of the X signal 5.2, 20 is fed on the peripheral axis A.
(3) When the workpiece coordinate on the X-axis (#100101) is equal to or less than 50, the Y signal 2.3 is set to “1”.

The operation of (1) is coded in the main NC statement.
The operation of (2) is coded in the first modal RTM command (//1).
The operation of (3) is coded in the second modal RTM command (//2).

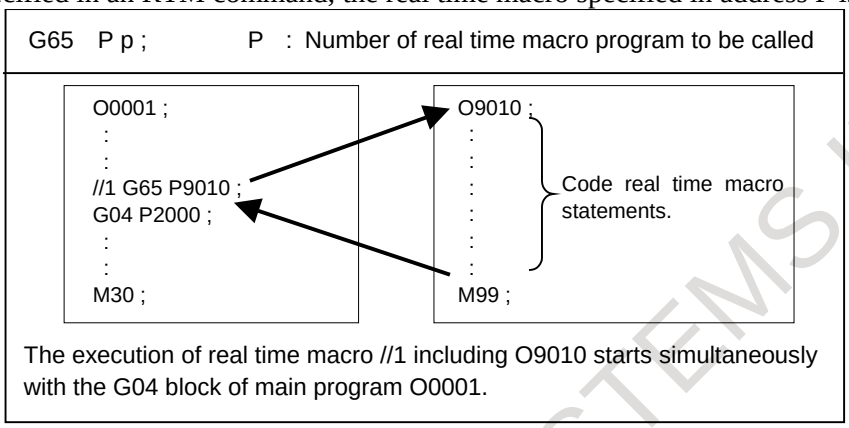


FRYER MACHINE SYSTEMS INC
Roz Zahensky

17.5 MACRO CALL

A series of RTM statements can be formed into a subprogram, which can be called from the main program.

When G65 is specified in an RTM command, the real time macro specified in address P is called.



Explanation

- Calling

In address P after G65, specify the program number of a real time custom macro to be called.

NOTE

1. In argument P of G65, only a constant value can be used. No value can be specified through a variable.
   Example:
   // G65 P9010 ; Correct
   // G65 P#RV[0] ; Incorrect
2. The inclusion of another NC command (such as G01 X100.0 G65 Pp) is not allowed. If another NC command is included, alarm PS0127, “DUPLICATE NC,MACRO STATEMENT” is issued.
3. The G65 block for calling a real time macro does not make a single block stop.
4. On the other hand, a real time macro program called by real time macro calling makes a single block stop.

- Call destination real time program

In a called real time macro program, only an RTM statement can be coded.

In a called real time macro program, no additional RTM command may be executed. (The RTM command symbol ‘//’ may not be coded.) For example, do not execute the following program:

```
O0001 ;                 O9010 ;
// G65 P9010 ;          // #RV[0]=1 ;
M02 ;                   M99 ;
```

Code M99 in the last block only.
For example, do not execute the following program:

```
O0001 ;                 O9010 ;
// G65 P9010 ;          ZEDGE [#RV[0] EQ 1] M99 ;
M02 ;                   G91 G00 X50. ;
                        M99 ;
```

FRYER MACHINE SYSTEMS INC
Rob Zahenskey

When executing an RTM statement in a subprogram, code an NC statement before the M99 block for returning to the calling program.

```
O0001 ;                 O9010 ;
G65 P9010 ;             // #RV[0] = 20 ; Real time macro statement
M02 ;                   G04 ;            NC statement
                        M99 ;
```

- Format

Specify G65 at the start of a block.

- Call nesting level

Macro call nesting is not allowed.

- Differences from macro call using a custom macro

Macro call using a real time custom macro and macro call using a custom macro differs as described below.

- In macro call using a custom macro, an argument (data to be passed to a macro) and the number of repetitions can be specified. In macro call using a real time custom macro, such information cannot be specified.
- With a real time custom macro, other types of macro call (macro call using G66, G66.1, G, and M code) and subprogram call are not allowed.

17.6 OTHERS

If an axis control command is followed by a macro command in an RTM command, the execution of the macro command starts when the axis control command is completed or deceleration starts.
If deceleration on the X-axis starts upon completion of distribution according to the axis control command of (1), for example, the macro command of (2) is executed. If acceleration/deceleration is not applied on the X-axis, (2) is executed upon completion of distribution on the X-axis.

```
// ZDO ;
G91 G00 X30 ;        (1) Axis control command of the RTM statement
#RV[0] = 1 ;         (2) Macro command of the RTM statement
ZEND ;
```

If an RTM command is executed in a custom macro WHILE DO to END loop, and the same RTM is looked ahead while the RTM command is being executed, an alarm PS0398 "'ID OVER IN BUFFER" may be issued.
For example, if the command below is executed, an alarm PS0398 is issued.

```
WHILE [1] DO1 ;
// #RV[0] = #RV[0] + 1 ;
#100=#100+1 ;
END1 ;
```

In this example, insert G04 ; block between a #100=#100+1; and END1;.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

17.7 AXIS CONTROL COMMAND

In an RTM statement, a G code for specifying a movement can be specified. For axis control, the PMC axis control interface is used. The specifications differ from the specifications for the G code used with an NC statement.

Format

```
// ZDO ;
G_ IP _ F_ ;
  :
ZEND ;
```

For an axis control command, ZDO...ZEND of multi-statement structure is used for coding even if a single statement (single RTM statement) is used.

> ⚠ **CAUTION**
> Do not specify an NC statement for an axis to be controlled with an RTM statement. Moreover, do not exercise axis control with an RTM statement on an axis to be controlled with an NC statement.

> **NOTE**
> 1 In one block, only one axis may be specified. Do not specify two or more axes in the same block.
> 2 In the 3-dimensional coordinate conversion mode, the polygon mode, etc., an RTM statement concerning that axis cannot be specified.

Explanation

- Setting

For an axis control command in an RTM statement, the interface of a PMC axis control group set by a parameter beforehand is used.
For an axis to be controlled with an RTM statement, set a group to be used with parameter No. 8010. Then, set bit 0 (XRT) of parameter No. 8011 for the axis to 1.

> **NOTE**
> An axis for which bit 0 (XRT) of parameter No. 8011 is set to 1 is dedicated to real time custom macros, so that such an axis cannot be used with PMC axis control.

- Relationship with PMC axis control

Axis control based on an RTM statement uses the PMC axis control interface. So, the specifications for a move command in each block within an RTM statement are generally equivalent to the specifications for the PMC axis control command.
For the detailed specifications and restrictions related to axis control, refer to the specifications of PMC axis control as well.

> **NOTE**
> 1 An axis used with an RTM statement must not be specified from PMC axis control.
> 2 A PMC axis control group used with an RTM statement must not be specified from PMC axis control.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Operation command code

The table below indicates the G codes that can be specified in an RTM statement.
The ◤ mark in the table indicates modal G codes when an RTM command starts.

Code	Group	Meaning
◤ G00	01	Positioning
G01		Feed at specified feedrate
G04	00	Dwell
G28		Reference position return
G53		Machine coordinate system selection
G90	03	Absolute command NOTE 1
◤ G91		Incremental command NOTE 2
◤ G94	05	Feed per minute
G95		Feed per revolution

NOTE
1 The absolute command (G90) is valid for machine coordinate system selection (G53) only.
2 The incremental command (G91) is valid for positioning (G00), linear interpolation (G01), and reference position return (G28) only.
3 In T series G code system A as well, not G98/G99 but G90/G91 are valid with an RTM command.

- Modal information

Basically, modal information such as F and modal G codes in an RTM statement is independent in an NC statement and each RTM statement.
Modal information in an RTM statement is initialized when the execution of the RTM statement is started (when a block specified with // is executed).
The initial value of modal information in an RTM statement is as follows:

State of the G codes marked with ◤ in the operation command code list
F code : F0

⚠ **CAUTION**
With the G codes (inch input/metric input) of group 06, the same information as the modal information of an NC statement is used in an RTM statement. Do not change the modal information of group 06 with an NC statement in a block after the first RTM statement coded in a program.

NOTE
With an RTM command, bit 0 (G01), bit 3 (G91), and bit 4 (FPM) of parameter No. 3402 are invalid when the power is turned on or in the cleared state.

Example 1)
Modal information can be controlled independently in an NC statement and each RTM statement.

```
O0001 ;
G90 G01 X100. Y100. F500. ;        (1)
//1 ZDO ;
Z50. ;                             (2)
G01 Z100. F100. ;                  (3)
ZEND ;
//2 ZDO ;
G01 A30. F200. ;                   (4)
A50. ;                             (5)
ZEND ;
X200. ;                            (6)
:
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

The modal information of command (2) is G00, G91, and G94, regardless of the modal information (G90 G01 set in (1)) of the NC statement.
The modal information of command (5) is G01, G91, G94, and F200.. This modal information is not affected by the modal information (command (1)) of the NC statement and the modal information (commands (2) and (3)) in the RTM statement with modal ID=1.
The modal information of command (6) is not affected by the modal information (commands (2) to (5)) in the RTM statement, so that the modal information is G01, G90, and F500..

Example 2)

Modal information is initialized when the execution of each RTM command is started. Even if the same program includes RTM commands with the same ID, the modal information of the RTM command executed first is not inherited by the RTM command executed next.

```
O0001 ;
G90 G01 X100. Y100. F500. ;
//1 ZDO ;
G01 Z100. F3000. ;                    (1)
ZEND ;
X200. ;
//1 ZDO ;
Z200. ;                               (2)
ZEND ;
X300. ;
:
```

The modal information of command (2) is G91 and G00, regardless of command (1).

- Single block stop

If an NC statement is placed in the single block stop state, for example, by the single block stop signal SBK, the axis control command of an RTM statement also comes to a single block stop.
However, the automatic operation start signal STL is turned off when all conditions on the NC statement side are satisfied, even if the axis control command of an RTM statement is not terminated.
If an RTM command consists of multiple statements and an axis control command is coded in multiple blocks, only the block of the RTM statement that is currently executing an axis command can be brought to a single block stop by setting the group-by-group single block stop signal (ESBKg) for PMC axis control corresponding to the axis to “1”.

- Feed hold

Even when an NC statement enters the automatic operation halt state, the axis control command of an RTM statement does not stop immediately but stops at the time of termination of the block currently being executed. To stop also the axis control command of an RTM statement immediately when the NC statement enters the automatic operation halt state, control the temporary stop signal (ESTPg) for PMC axis control of the related group by monitoring the automatic operation halt in-progress signal (SPL).

- Reset

Even when the CNC is reset by an MDI reset, the external reset signal (ERS), or the reset and rewind signal (RRW), the axis control command of an RTM statement does not stop immediately but stops at the time of termination of the block currently being executed. When the emergency stop state is set, however, the RTM statement also stops immediately.
During execution of the axis control command of an RTM statement, the real time macro being executed can be stopped independently of the NC statement by setting the group-by-group reset signal (ECLRg) for PMC axis control corresponding to the axis to “1”. To stop also the axis control command of an RTM statement when the CNC is reset, control the reset signal (ECLRg) for PMC axis control of the related group by monitoring the reset signal (RST).

FRYER MACHINE SYSTEMS INC
Rob Zahnsky

- Alarm stop

Even when a PS alarm is issued with an NC statement, the axis control command of the RTM statement being executed does not stop immediately but stops at the time of termination of the block currently being executed. Moreover, even when an overtravel alarm is issued for an axis other than the axis controlled by the RTM statement being executed, the RTM statement being executed does not stop immediately but stops at the time of termination of the block currently being executed.

- Alarm reset

When the group-by-group alarm signal (EIALg) for PMC axis control corresponding to the axis specified by an RTM statement is set to “1”, correct the cause of the alarm, then reset the CNC, and reset PMC axis control with the reset signal (ECLRg) for PMC axis control of the related group.
(Tip) In the following cases, the alarm signal (EIALg) is set to “1”:

(1) When a servo alarm is issued
(2) When an overtravel alarm is issued
(3) When alarm PS0130, “NC AND PMC AXIS ARE CONFLICTED” is issued
(4) When alarm PS0139, “CANNOT CHANGE PMC CONTROL AXIS” is issued

- Interlock

In an RTM statement, the interlock signal (*IT) for an NC statement, interlock signal for each axis (*ITn), interlock signal for each axis direction (+ITn/-ITn) are invalid. Instead, the axis control temporary stop signal (ESTPg) for PMC axis control is valid in an RTM statement. When temporarily stopping the axis controlled by an RTM statement, control the corresponding axis control temporary stop signal (ESTPg) for PMC axis control.

The cutting block start interlock signal *CSL (G8.1) and the block start interlock signal *BSL (G8.3) are invalid.

- Machine lock

The same machine lock signals (all axes/each axis) as used with an NC statement are used. However, by disabling machine lock for PMC axis control with the following parameters, machine lock can be disabled for the axis being controlled by an RTM statement:
Bit 0 (MLE) of parameter No. 8001
Bit 1 (MLS) of parameter No. 8006

- Dry run

With bit 2 (OVE) of parameter No. 8001, whether to use the dry run signal (DRN) for an NC statement or the dry run signal (EDRN) for a PMC axis can be chosen. When enabling a dry run feedrate for rapid traverse in an RTM statement, set bit 3 (RDE) of parameter No. 8001 to 1. If a feed command specifying a feedrate is specified when the manual rapid traverse selection signal (RT or ERT) is set to “1”, the cutting feedrate is used as the feedrate during dry run. ("Dry run feedrate × maximum manual feedrate override value" is not applicable.)

- In-position check

In the in-position state, the in-position signal (EINPg) is set to “1”.
When bit 6 (NCI) of parameter No. 8004 is set to 1, no in-position check is made during axis control based on an RTM statement. The setting of bit 5 (NCI) of parameter No. 1601 for an NC statement is invalid. (However, the in-position signal (INPx) for an NC statement is affected by the value of NCI.)

- Manual absolute

The manual absolute signal (*ABSM) is invalid.

FRYER MACHINE SYSTEMS INC
Rob Zohensky

- External deceleration

By setting bit 0 (EDC) of parameter No. 8005 to 1, the external deceleration function can be used also for the axis being controlled by an RTM statement. Note, however, that the external deceleration function is unusable for feed with a specified feedrate (feed per revolution). For an external deceleration rate, the external deceleration rate setting for rapid traverse (parameter No. 1427, No. 1441, and No. 1444) is valid, regardless of the type of feed.
The external deceleration function for the axis being controlled by an RTM statement is also valid for feed with a specified feedrate (feed per minute), regardless of the settings of bit 4 (EDP) of parameter No. 1005 and bit 5 (EDM) of parameter No. 1005. (The enabling of rapid traverse only is impossible.)

- Mirror image

At enabling mirror image for the axis being controlled by an RTM statement, set bit 0 (EMR) of parameter No. 8008 to 1, and set either bit 0 (MIRx) of parameter No.0012 or mirror image signal MIx to “1”.
Programmable mirror image is not available.

NOTE

1 The same group as the PMC axis control group used for axis control cannot be executed with an RTM statement.
2 No NC statement can specify an axis being controlled by an RTM statement.
3 An alarm PS0399 "'ID EXECUTION IN SAME TIME" is issued if, during execution of an RTM statement, an attempt is made to execute another RTM statement with the same ID. In the program below, for example, the RTM statement of (1) operates using the NC statement of (2) as a trigger, and the RTM statement of (3) operates using the NC statement of (4) as a trigger. An alarm PS0399 is issued if (1) is still operating when (4) is started after completion of the interpolation of (2).

```
//1 ZDO ;
G91 G00 A100. ; (1)  RTM statement
ZEND ;
G01 X30. Y20. ;  (2)  NC statement
//1 ZDO ;
G91 G00 B100. ; (3)  RTM statement
ZEND ;
X40. Y50. ;    (4)  NC statement
```

4 When T series G code system A is used, address words (such as U, W, H, and V) for incremental commands must not be coded. Otherwise, alarm PS0413, “ILLEGAL ADDRESS” is issued.
5 An RTM statement for performing axis control proceeds to the next command at the end of distribution. For example, in the program below, #RV[0] counts up at the time X50. distribution ends and deceleration starts.

```
// ZDO ;
G00 X50. ;
#RV[0]=#RV[0]+1 ;
ZEND ;
```

FRYER MACHINE SYSTEMS INC
Rob Zahenny

Command details

- Rapid traverse

A movement is made at a rapid traverse rate on an axis from the current position to the point separated by a specified value.

Format

```
// ZDO ;
G91 G00 IP _ ;
ZEND ;
  G91  : G code for incremental command
  IP _ : Travel distance
```

NOTE
1 Only one axis can be specified in one block.
2 The absolute command (G90) cannot be specified.
3 The rapid traverse overlap function cannot be used.
4 When using this function, be sure to set bit 0 (RPD) of parameter No. 8002 to 0.

- Rapid traverse override
  With bit 2 (OVE) of parameter No. 8001, whether to use the rapid traverse override signal (ROV) for an NC statement or the rapid traverse override signal (EROV) dedicated to PMC axis control can be chosen.

NOTE
Even if bit 4 (RF0) of parameter No. 1401 is set to 1, rapid traverse does not stop with a cutting feed override of 0%.

- Feed with a specified feedrate (feed per minute)

A movement is made at a feedrate specified in F on an axis from the current position to the point separated by a specified value.

Format

```
// ZDO ;
G94 G91 G01 IP _ F_ ;
ZEND ;
  G94 :  G code for feed per minute
  G91 :  G code for incremental command
  IP __: Travel distance
  F__ :  Feedrate command (mm/min or inch/min)
```

NOTE
1 Only one axis can be specified in one block.
2 The absolute command (G90) cannot be specified.
3 The block overlap function cannot be used.
4 When IS-A is used, a feedrate below 10 mm/min is discarded.
5 The feedrate cannot be clamped to a parameter-set maximum cutting feedrate.
6 No time is added to the cumulative cutting time.
7 Even if this command is executed, the cutting in-progress signal is not output.
8 The operation between blocks for feed with a specified feedrate is performed in the cutting mode.
9 For this command, the feedrate arrival signal SAR cannot be used.

FRYER MACHINE SYSTEMS INC
Rob Zabensky

- Feedrate specification range
  The table below indicates the feedrate specification range.

	Linear axis		Rotation axis (deg/min)
	Metric input (mm/min)	Inch input (inch/min)	
IS-A	10. to 240000.	0.1 to 6553.5	10. to 240000.
IS-B	1. to 65535.	0.01 to 655.35	1. to 65535.
IS-C	0.1 to 6553.5	0.001 to 65.535	0.1 to 6553.5

NOTE
1 Be sure to set the following parameters to 0:
  Bit 3 (F10) of parameter No. 8002
  Bit 4 (EFD) of parameter No. 8006
  Bit 4 (PF1) of parameter No. 8002
  Bit 5 (PF2) of parameter No. 8002
  When a value other than 0 is set, the feedrate specification range changes, depending on the set parameter.
2 A value less than the lowest feedrates indicated above is discarded.

- Feedrate override
  With bit 2 (OVE) of parameter No. 8001, whether to use the feedrate override signal (*FV) for an NC statement or the feedrate override signal (*EFOV) dedicated to PMC axis control can be chosen.

NOTE
1 The second feedrate override function cannot be used.
2 The feedrate override function cannot be disabled using #3004.

- Override cancel
  With bit 2 (OVE) of parameter No. 8001, whether to use the feedrate override cancel signal (OVC) for an NC statement or the feedrate override cancel signal (EOVC) dedicated to PMC axis control can be chosen.
- Operation between blocks
  If feed with a specified feedrate is performed in succession in RTM statements, no stop occurs between blocks, but the next block is executed. With the accumulated pulse zero check signal (ELCKZg) for PMC axis control, an accumulated pulse zero check can be made between blocks specifying feed with a specified feedrate.

NOTE
1 The error detect signal (SMZ) cannot be used.
2 With the accumulated pulse zero check signal (ELCKZg) for PMC axis control, an accumulated pulse zero check can be made between blocks specifying feed with a specified feedrate for feed on the same axis only.

- Acceleration/deceleration time constant
  For an acceleration/deceleration time constant to be used for feed with a specified feedrate in an RTM statement when exponential acceleration/deceleration is used, whether to use the time constant for an NC statement or the time constant dedicated to PMC axis control can be chosen using parameter No. 8030.

NOTE
Look-ahead acceleration/deceleration before interpolation cannot be used.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Feed with a specified feedrate (feed per revolution)

A movement is made at a feedrate specified in F on an axis from the current position to the point separated by a specified value.

Format

```
// ZDO ;
G95 G91 G01 IP _ F_ ;
ZEND ;
```

G95 : G code for feed per revolution
G91 : G code for incremental command
IP _ : Travel distance
F_ : Feedrate command (mm/rev or inch/rev)

NOTE

1 Only one axis can be specified in one block.
2 The absolute command (G90) cannot be specified.
3 The block overlap function cannot be used.
4 Be sure to set the parameters below to 0. If a value other than 0 is set, the feedrate specification range changes, depending on the set parameter.
  Bit 6 (FR1) of parameter No. 8002
  Bit 7 (FR2) of parameter No. 8002
5 The feedrate is clamped to the maximum cutting feedrate set in parameter No. 8022. (Parameter No. 1430 for an NC statement is not valid for this command.)
6 To use feed per revolution, a position coder is required at all times. (Bit 0 (NPC) of parameter No. 1402 is not applicable to this command.)
7 No time is added to the cumulative cutting time.
8 Even if this command is executed, the cutting in-progress signal is not output.
9 The operation between blocks for feed with a specified feedrate is performed in the cutting mode at all times.
10 For this command, the feedrate arrival signal SAR cannot be used.

- Feedrate specification range
  The table below indicates the feedrate specification range.

	Linear axis		Rotation axis (deg/rev)
	Metric input (mm/rev)	Inch input (inch/rev)	
T series	0.001 to 65.535	0.000001 to 0.65535	0.001 to 65.535
M series	0.01 to 500.00	0.0001 to 6.5535	0.01 to 500.00

- Feedrate override
  With bit 2 (OVE) of parameter No. 8001, whether to use the feedrate override signal (*FV) for an NC statement or the feedrate override signal (*EFV) dedicated to PMC axis control can be chosen.

NOTE

1 The second feedrate override function cannot be used.
2 The feedrate override function cannot be disabled using #3004.

- Override cancel
  With bit 2 (OVE) of parameter No. 8001, whether to use the feedrate override cancel signal (OVC) for an NC statement or the feedrate override cancel signal (EOVC) dedicated to PMC axis control can be chosen.

FREE MACHINE SYSTEMS INC Rob Zelensky

- Operation between blocks
  If feed with a specified feedrate is performed in succession in RTM statements, no stop occurs between blocks, but the next block is executed. With the accumulated pulse zero check signal (ELCKZg) for PMC axis control, an accumulated pulse zero check can be made between blocks specifying feed with a specified feedrate.

NOTE
1 The error detect signal (SMZ) cannot be used.
2 With the accumulated pulse zero check signal (ELCKZg) for PMC axis control, an accumulated pulse zero check can be made between blocks specifying feed with a specified feedrate for feed on the same axis only.

- Acceleration/deceleration time constant
  For an acceleration/deceleration time constant to be used for feed with a specified feedrate in an RTM statement when exponential acceleration/deceleration is used, whether to use the time constant for an NC statement or the time constant dedicated to PMC axis control can be chosen using parameter No. 8030.

NOTE
Look-ahead acceleration/deceleration before interpolation cannot be used.

- Reference position return

A movement is made at the rapid traverse rate to the first reference position on a specified axis. Upon completion of reference position return, the return completion lamp is turned on.

Format

```
// ZDO ;
G91 G28 IP 0 ;
ZEND ;
```

G91 : G code for incremental command
IP 0 : Specify an axis on which reference position return is to be performed.
Be sure to specify 0 as the travel distance.

NOTE
1 Only one axis can be specified in one block.
2 No intermediate point may be specified. (Be sure to specify 0 as the travel distance. Otherwise, alarm PS0410, “MIDDLE POINT IS NOT ZERO” is issued.) If a movement needs to be once made to an intermediate point, specify positioning with rapid traverse (G00)/machine coordinate system selection (G53) then specify this command.
3 The absolute command (G90) cannot be specified.
4 When using this function, be sure to set bit 0 (RPD) of parameter No. 8002 to 0.
5 Before this command can be specified, a reference position must be established. (With this command, do not specify an axis on which a reference position is not established.)
6 Before specifying this command with the T series, cancel tool offset and tool-nose radius compensation. Before specifying this command with the M series, cancel cutter compensation, tool length compensation, and tool offset.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Rapid traverse override
  With bit 2 (OVE) of parameter No. 8001, whether to use the rapid traverse override signal for an NC statement or the rapid traverse override signal dedicated to PMC axis control can be chosen.

NOTE
Even if bit 4 (RF0) of parameter No. 1401 is set to 1, rapid traverse does not stop with a cutting feed override of 0%.

- Machine coordinate system selection

When a position in the machine coordinate system is specified, a movement is made to the position on the axis by rapid traverse. The G53 code for machine coordinate system selection is a one-shot G code, so that a command in the machine coordinate system is valid in the block that specifies G53. When moving the tool to a machine-specific position such as a tool change position, perform programming in the machine coordinate system based on G53.

Format

```
// ZDO ;
G90 G53 IP _ ;
ZEND ;
```

G90 : G code for absolute command
IP _ : Position in machine coordinate system

NOTE
1 Only one axis can be specified in one block.
2 The incremental command (G91) cannot be specified.
3 When using this function, be sure to set bit 0 (RPD) of parameter No. 8002 to 0.
4 Before this command can be specified, a reference position must be established. (With this command, do not specify an axis on which a reference position is not established.)
5 Before specifying this command with the T series, cancel tool offset and tool-nose radius compensation. Before specifying this command with the M series, cancel cutter compensation, tool length compensation, and tool offset.

- Rapid traverse override
  With bit 2 (OVE) of parameter No. 8001, whether to use the rapid traverse override signal for an NC statement or the rapid traverse override signal dedicated to PMC axis control can be chosen.

NOTE
Even if bit 4 (RF0) of parameter No. 1401 is set to 1, rapid traverse does not stop with a cutting feed override of 0%.

- Dwell

By specifying a dwell, the execution of the next block is delayed by the specified time. (Dwell per second)
Dwell per second is executed irrespective of bit 1 (DWL) of parameter No.3405.

FRYER MACHINE SYSTEMS INC
Rob Zakowsky

Format

```
// ZDO ;
G04 IP _ ;
ZEND ;
```

IP _ : Axis address where bit 0 (XRT) of parameter No. 8011 was set

- Dwell specification range
  The table below indicates the dwell specification range.

Increment system	Command value range	Dwell time unit
IS-A	0.01 to 999999.99	s
IS-B	0.001 to 99999.999	
IS-C	0.0001 to 9999.9999	

When the diameter is specified, whether to specify a radius value or diameter value for dwell operation can be specified using bit 1 (CDI) of parameter No.8005.

NOTE
1. Address P cannot be specified. Only the axis address where bit 0 (XRT) of parameter No.8011 was set to 1 can be specified.
2. When specified without the decimal point, the specification unit depends on inch/metric.
3. When the increment system is IS-C and bit 1 (DWE) of parameter No.8002=1, the command value becomes 1/100.

17.8 AUXILIARY FUNCTION

Overview

M code of auxiliary function will be available in real-time custom macro.
This function works by using the signal region of the auxiliary function 1 (12h) of the PMC axis control.
In addition, it is possible to commanding a M code in the same block as the axis control command.

Explanation

Set the group number (parameter No. 8010) of the PMC axis control auxiliary function used by the modal RTM command to the parameter (No. 12740 to No. 12749) for each ID. To activate multiple auxiliary functions at the same time, set a different group number for each ID.
Auxiliary function completion signal EFINg, auxiliary function strobe signal EMFg, auxiliary function code signals EM11g to EM48g and auxiliary function executing signal EDENg of the group set by the parameter is controlled to operate the auxiliary function. Fig. 17.8 (a) and Fig17.8 (b) show time charts. TMF and TFIN in the figure are set with parameters (No. 3010, No. 3011).

2 bytes of output is performed for the auxiliary function code signal EM11g to EM48g. It does not depend on the bit 6 (AUX) of parameter No.8001 setting. Auxiliary function lock is enabled. When the auxiliary function lock signal AFL<Gn005.6> is "1", the auxiliary function is not executed. Multiple M codes in a single block can not be executed. If you want to activate multiple auxiliary functions at the same time, IDs of different modal RTM commands are activated at the same time.

FRITER MACHINE SYSTEMS INC
Bob Zahensky

NOTE

1 The following functions can not be used.
- PMC axis control auxiliary function 1 (12 h), auxiliary function 2 (14 h), auxiliary function 3 (15 h)
- Multiple M commands in a single block. An alarm (PS0427)"ILLEGAL M CODE" is generated.
- M code grouping function.
- Auxiliary function output in moving axis. The same operation can be executed by the RTM command.
- Check the allowable number of digits for the M code(parameter No.3030).
- Each function of M code to set parameters. Regardless of parameter setting, auxiliary function of RTM command operated.

2 The following M code can not be commanded. An alarm (PS0427)"ILLEGAL M CODE" is generated.
- M00, M01, M02, M30, M98, M99, M198, M65536 or more

3 A one-shot RTM command can not execute the auxiliary function. An alarm (PS0426)" M FUNCTION IS UNABLE TO USE " is generated.

4 Group numbers that can be set for parameters (No. 12740 to No. 12749) are up to " NC path number × 4". Set the group number set for parameter No. 8010.

5 Block stop signal ESBKg is invalid for this function. When stopping the block, control with the auxiliary function completion signal EFINg.

6 Even if the RTM command is interrupted by reset or ZCANCEL statement while executing the auxiliary function of the RTM command, the auxiliary function will not be terminated. Wait for input of auxiliary function completion signal EFINg. To terminate the auxiliary function, input the PMC axis control reset signal ECLRg of the group used by the auxiliary function.

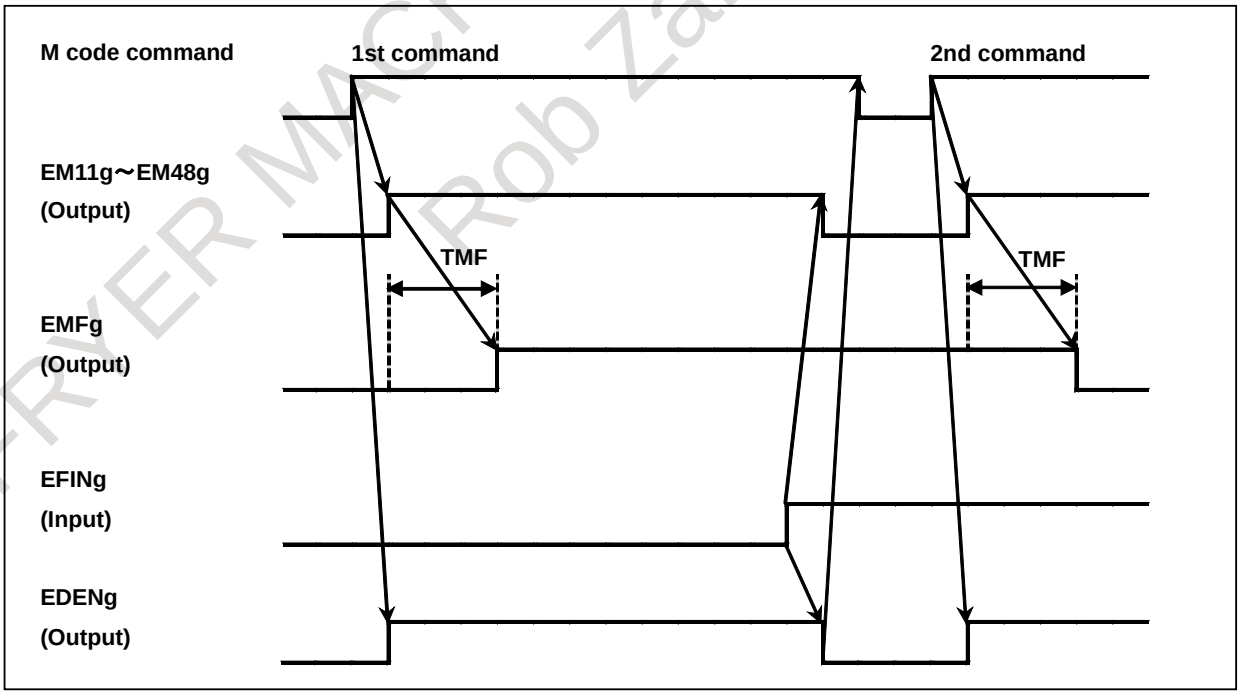


Fig. 17.8 (a) Time chart of the high-speed system for signal on the auxiliary function. (Bit 7 (MHI) of parameter No.3001 is 1)

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

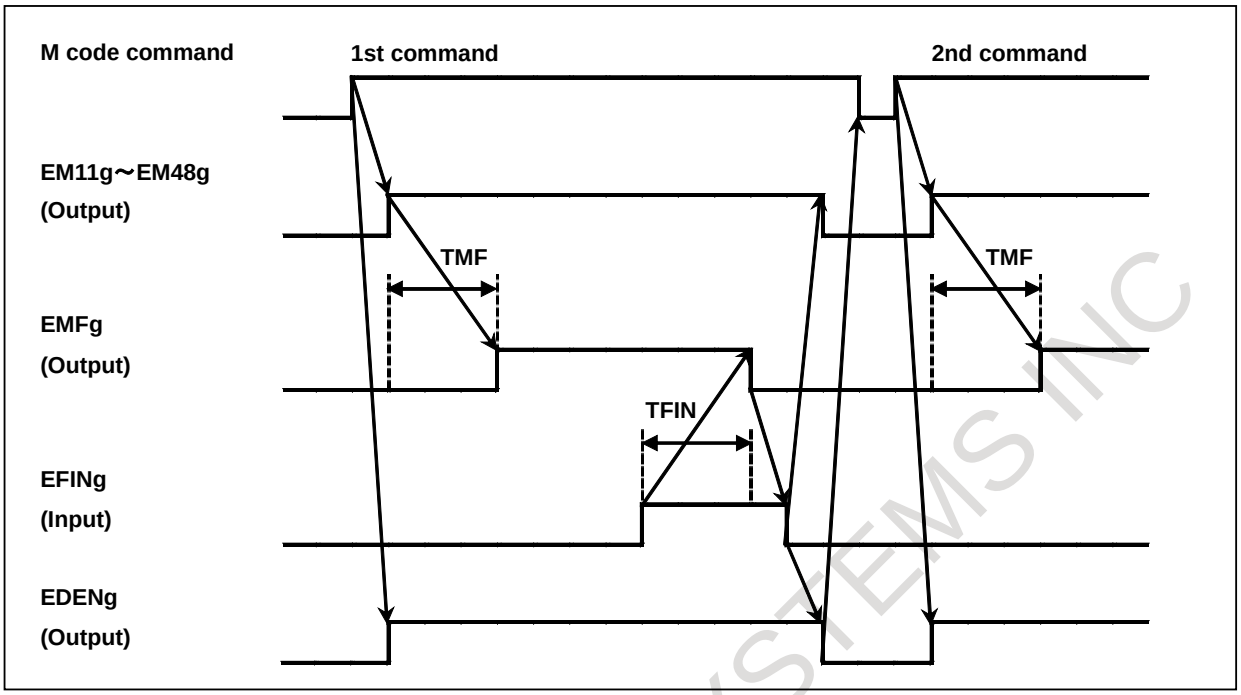


Fig. 17.8 (b) Time chart of the usual system for signal on the auxiliary function.
(Bit 7 (MHI) of parameter No.3001 is 0)

Example

Executing a single auxiliary function

If it is not necessary to execute the auxiliary function at the same time, set the same group number to the parameter (No. 12740 to No. 12749). Fig. 17.8 (c) shows an example using group 1. In this example, PMC controls the auxiliary function strobe signal EMFA#1<F131.0>, auxiliary function code signals EM11A#1 to EM48A#1<F132,F142>, auxiliary function executing signal EDENA#1<F130.3> and auxiliary function completion signal EFINA#1<G142.0>. Also, if the next auxiliary function is executed while executing the auxiliary function, an alarm (PS0428) "AUX FUNCTION IS IN USE " occurs.

Parameter	Setting value	Parameter	Setting value
No.12740	1	No.12745	1
No.12741	1	No.12746	1
No.12742	1	No.12747	1
No.12743	1	No.12748	1
No.12744	1	No.12749	1

```
O0001 ;
:
//1 M100 ;
G04.1
:
M30 ;
```

Fig. 17.8 (c) Setting example of auxiliary function of real time custom macro using group 1

Simultaneously execute multiple auxiliary functions

To execute multiple auxiliary functions at the same time, set different group numbers to parameters (No. 12740 to No. 12749) for IDs of modal RTM commands to be executed at the same time. Fig. 17.8 (d) is an example using group 1 and group 2. For IDs with different group numbers, auxiliary functions can be commanded at the same time. As shown in Fig. 17.8 (d) O0002, the auxiliary function can be commanded with ID1 and ID5 at the same time. In this example, PMC controls the auxiliary function strobe signal EMFA#1, EMFB#1<F131.0, F134.0>, auxiliary function code signals EM11A#1 to EM48A#1, EM11B#1 to EM48B#1<F132,F142, F135,F145>, auxiliary function executing signal EDENA#1, EDENB#1<F130.3, F133.3> and auxiliary function completion signal EFINA#1, EFINB#1<G142.0, G154.0>. Also, if the auxiliary function is commanded at the same time with the ID set with the same group number, an alarm (PS0428) "AUX FUNCTION IS IN USE " occurs.

FRYER MACHINE SYSTEMS INC
Ros Zarensky

Parameter	Setting value	Parameter	Setting value
No.12740	1	No.12745	2
No.12741	1	No.12746	2
No.12742	1	No.12747	2
No.12743	1	No.12748	2
No.12744	1	No.12749	2

```
O0002 ;
:
//1 M100 ;
G04.1 ;
//5 M101 ;
G04.1 ;
 :
M30 ;
```

Fig. 17.8 (d) Setting example of auxiliary function of real time custom macro using group 1 and group 2

17.9 NOTES

- Auxiliary function

Command Auxiliary function cannot be specified in a RTM statement.
An alarm PS0426 “M FUNCTION IS UNABLE TO USE” is occurs.
However, the input/output of the signal is possible.
Please achieve the specification corresponding to auxiliary function by the signal operation.

Example of executing near M code by using R signal)

```
//1ZDO
#IOR[100,0]=1.0 ;          (1)
ZEDGE [#IOR[100,1] NE 0.0] ZDO ;   (2)
#IOR[100,0]=0.0 ;          (3)
#RV[0]=1.0 ;               (4)
ZEND ;
ZEND ;
```

(1) Set R0100.0 corresponding to a code signal of auxiliary function: M00 to M31 or a strobe signal MF to 1. R0100.0 is used as a trigger and Process of M code is executed by PMC ladder.
(2) Wait until R0100.1 corresponding to the completion signal (FIN) changes into 1. After the operation of the auxiliary function is finished by PMC ladder, set R0100.1 to 1.
(3) Set R0100.0 to 0. Set R0100.1 to 0 by PMC ladder.
(4) RTM statements is coded after the auxiliary function.

NOTE
Automatic operation of NC program and the real-time custom macro are asynchronously executed. When the processing of automatic operation and the real-time custom macro is common in PMC ladder, be careful enough that you must prevent the competition of command. Exclusive control is needed.

- Address without the decimal point

In general, an NC address without the decimal point is subject to calculator-type decimal point input when bit 0 (DPI) of parameter No. 3401 or bit 0 (AXDx) of parameter No. 3455 is set to 1. In other cases, an NC address without the decimal point is regarded as a least input increment.

- Indirect axis address specification

When indirect axis address specification is used, use a direct reference argument. No indirect reference may be used.
Example)

```
AX[1]        Correct
AX[#RV[0]]   Incorrect
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Diameter specification/radius specification

For an axis to be controlled with an RTM statement, select radius specification. (Set bit 3 (DIA) of parameter No. 1006 to 0.)

- Alarm PS0224

If bit 0 (ZRN) of parameter No. 1005 is set to 0, and an axis control command is specified in an NC statement before a reference position return operation is not performed even once after power-up, alarm PS0224, “ZERO RETURN NOT FINISHED.” is issued. In axis control using an RTM statement, the alarm is not issued even when bit 0 (ZRN) of parameter No. 1005 is set to 0, unless bit 6 (EZR) of parameter No. 8006 is set to 1.

- Axis removal

Even when axes are removed, interlock is not applied to the axis being controlled by an RTM statement.

- Stroke limit check before movement

No stroke limit check before movement is made with a block in an RTM statement.

- Others

For the axis being controlled by an RTM statement, the functions disabled for PMC axis control cannot be used.

- Other disabled functions

For the axis being controlled by an RTM statement, the functions listed below as examples are disabled. Do not specify these functions for the axis being controlled by an RTM statement.

- Feed-forward
- Advanced preview feed-forward
- Fine acceleration/deceleration
- Automatic corner deceleration
- One-digit F code feed
- Scaling
- Coordinate system rotation
- Polar coordinate interpolation
- Balance cutting
- Feed stop
- Constant surface speed control
- Positioning function based on optimal acceleration, etc.

⚠ CAUTION

In an RTM statement, do not specify an axis used with any of the following functions:

1. Axis synchronous control
   Be sure to set the axis synchronous control selection signal (SYNCn) and axis synchronous control manual feed selection signal (SYNCJn) of the axis to “0”.
2. Synchronous/composite control, superimposed control
3. Index table indexing
4. 3-dimensional coordinate system conversion
5. Rigid tapping
6. Compound machining
7. Polygon turning
8. Cs contour control
9. AI contour control I/II

FRYER MACHINE SYSTEMS INC
Rob Zahensky

17.10 LIMITATION

Major general notes on RTM commands are provided below.

- Background drawing

The RTM command has no effect in background drawing. Do not specify an RTM command during background drawing.

- Interrupt-type custom macro

In an interrupt-type custom macro, no RTM command can be coded.

- Macro executor

In a macro executor, no RTM command can be coded.
Moreover, no macro executor can be coded from an RTM command.
In a series of programs, do not code a macro executor (execution macro) and RTM command at the same time.
In addition, observe the following:

- Do not code an RTM command in a program that uses an execution macro.
- Do not call an execution macro from a program with an RTM command coded.
- Do not code an RTM command in an execution macro.
- Do not use an execution macro as an RTM command trigger.

- Sequence number

No sequence number can be coded in an RTM command. No RTM command can be searched for a sequence number.

- Optional block skip

Optional block skip is unusable.
A slash (/) that appears in the middle of <expression> (enclosed in [ ] on the right-hand side of an arithmetic/logical expression) is regarded as a division operator; it is not regarded as an optional block skip code.

- Single block

When an RTM command is used, a single block stop generally occurs according to an NC statement, regardless of the values of bit 5 (SBM) of parameter No. 6000 and bit 7 (SBV) of parameter No. 6000. However, no single block stop occurs according to G65 for real time macro call.

- Processing

If there are many RTM statements between NC statements, the command flow may break.

- DNC operation

DNC operation is disabled.

- Operation in each event

If an event such as an emergency stop or alarm occurs during execution of an RTM command, the NC command and RTM command generally operate as indicated below.

Event	NC command	RTM command consisting of a macro command	RTM command including an axis control command
PS alarm issued in NC command execution	Operation stops when the block involving the PS alarm cause starts.	The RTM command stops when the NC command stops.	If the RTM statement being executed when the NC command stops is an axis control command, operation on the axis stops when the block stops.

FRYER MACHINE SYSTEMS INC
Bob Zahensky

Event	NC command	RTM command consisting of a macro command	RTM command including an axis control command
Alarm issued in RTM command execution	Operation stops with the alarm.	The RTM statement stops at that point. Other RTM commands stop when the NC command stops.	The RTM statement stops at that point. Other RTM commands stop operation on the axis when axis control by the RTM statement being executed when the NC statement stops ends.
Single block (SBK signal)	Operation stops when the command being executed ends.	The RTM command is suspended when the NC command stops. When the NC command restarts, the RTM command restarts.	If the RTM statement being executed when the NC command stops is an axis control statement, operation on the axis stops when the block ends.
Feed hold (*SP signal)	A gradual stop occurs.	The RTM command stops when the NC command stops. When the NC command restarts, the RTM command restarts.	If the RTM statement being executed when the NC command stops is an axis control statement, operation on the axis stops when the block ends.
NC command stopped by a reset (ERS signal)	A gradual stop occurs, and the command also ends.	The RTM command stops when the NC command stops.	If the RTM statement being executed when the NC command stops is an axis control command, operation on the axis stops when the block ends.
Emergency stop	An immediate stop occurs, and the command also ends.	The command immediately ends.	Operation on the axis immediately stops, and the command also immediately ends.
End of the NC command	-	If the RTM command being executed is a one-shot RTM command, the command ends. If the RTM command being executed is a modal RTM command, the command continues execution.	If the one-shot RTM command being executed when the NC command ends is an axis control command, operation on the axis stops when the block ends.
Machine lock (MLK signal)	The NC command is valid.	The RTM command operates.	The RTM command can be enabled. For details, see the description of the axis control command.
Dry run (DRN signal)	The NC command is valid.	The RTM command operates.	The RTM command can be enabled. For details, see the description of the axis control command.
Interlock (*IT signal)	The NC command is valid.	The RTM command operates.	The RTM command can be enabled. For details, see the description of the axis control command.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

18 PROGRAMMABLE DATA INPUT (G10)

18.1 PROGRAMMABLE DATA INPUT (G10)

Overview

The values of workpiece origin offset, tool data, etc. can be entered in a program.

Format

- **Programmable data entry**

```
G10 L_ (data) ;        Data entry
  L_:      Data number
```

- **Programmable data entry mode**

```
G10 L_ (data) ;        Data entry mode
(data) ;
  :
  :
G11 ;                  Data entry mode cancel
  L_:      Data number
```

Explanation

- **Data number (L)**

For the correspondence between the settable data and the data number (L) of programmable data input, refer to tables below.
For the detailed specifications such as the input format of each data, refer to the reference item.

Table 18.1 (a) Programmable data input (Common)

Data number	Data	Reference item
Without L	Workpiece coordinate system shift amount	Workpiece Coordinate System Shift / “COORDINATE SYSTEM” in PROGRAMMING
L2	Workpiece origin offset value	Changing Workpiece Coordinate System / “COORDINATE SYSTEM” in PROGRAMMING
L3 (NOTE)	Tool life management data entry mode	Registering, Changing, and Deleting Tool Life Management Data / “TOOL FUNCTION (T FUNCTION)” in PROGRAMMING
L25 (NOTE)	Machine configuration data and machine configuration name entry mode	Setting Machine Configuration Data / “5-AXIS MACHINING FUNCTION” in PROGRAMMING
L40	M code group number	Setting an M Code Group Number Using a Program / “AUXILIARY FUNCTION” in PROGRAMMING

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Data number	Data	Reference item
L60	Periodic maintenance data	Periodic Maintenance Screen / “SETTING AND DISPLAYING DATA” in OPERATION Periodic Maintenance Screen (15/19-inch) Display Unit) / “SETTING AND DISPLAYING DATA” in OPERATION
L61	Periodic maintenance item name	
L75 (NOTE)	Tool management data entry mode	TOOL MANAGEMENT FUNCTION / “TOOL FUNCTION (T FUNCTION)” in PROGRAMMING TOOL MANAGEMENT EXTENSION FUNCTION / “TOOL FUNCTION (T FUNCTION)” in PROGRAMMING TOOL MANAGEMENT FUNCTION FOR OVERSIZE TOOLS / “TOOL FUNCTION (T FUNCTION)” in PROGRAMMING
L76 (NOTE)	Cartridge management table entry mode	
L77 (NOTE)	Customization data name entry mode	
L90	Blank figure drawing parameter	Programmable Data Input (G10) for Blank Figure Drawing Parameters / “GRAPHIC FUNCTION” in OPERATION

NOTE

In the G10L_ command of data entry mode, only the data entry command according to each data format is available during the data entry mode. The data entry mode is canceled by the G11 command.

M

Table 18.1 (b) Programmable data input (Machining center system)

Data number	Data	Reference item
Without L	Tool compensation value (Tool compensation memory A)	Outputting offset data / “DATA INPUT/OUTPUT” in OPERATION OPERATOR’S MANUAL (For Machining Center System) (B-64694EN-2) TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10) / “COMPENSATION FUNCTION” in PROGRAMMING
L10	Geometry compensation value (H code) (Tool compensation memory C)	
L11	Tool compensation value (Tool compensation memory A)	
	Wear compensation value (H code) (Tool compensation memory C)	
L12	Geometry compensation value (D code) (Tool compensation memory C)	
L13	Wear compensation value (D code) (Tool compensation memory C)	
L20	Workpiece origin offset value (Additional workpiece coordinate systems)	Addition of Workpiece Coordinate System Pair (G54.1 or G54) / “COORDINATE SYSTEM” in PROGRAMMING

FRYER MACHINE SYSTEMS, INC
Rob Zahensky

T

Table 18.1 (c) Programmable data input (Lathe system)

Data number	Data	Reference item
Without L	Tool compensation amount	Outputting offset data / “DATA INPUT/OUTPUT” in OPERATION
	4th/5th axis offset	OPERATOR’S MANUAL (For Lathe System) (B-64694EN-1) 4th/5th Axis Offset / “COMPENSATION FUNCTION” in PROGRAMMING Outputting 4th/5th Axis Offset Data / “DATA INPUT/OUTPUT” in OPERATION
L200 to L203	Tool offset for milling and turning function	TOOL OFFSET FOR MILLING AND TURNING FUNCTION (G43.7) / “COMPENSATION FUNCTION” in PROGRAMMING
L1200 to L1203		

⚠ CAUTION
The canned cycle mode must be cancelled before entering of data. When not cancelled, the drilling motion may be activated.

NOTE
1 G10 and G11 must be specified in a block in which there is no other command.
2 Other NC statements cannot be specified during the data entry mode.
3 G10 and G11 are G codes for disabling buffering.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

18.2 PROGRAMMABLE PARAMETER INPUT (G10)

Overview

The values of parameters and pitch error compensation data can be entered in a program. This function is used for setting pitch error compensation data when attachments are changed or the maximum cutting feedrate or cutting time constants are changed to meet changing machining conditions.
The input format of pitch error compensation data is changed depending on bit 0 (APE) of parameter No. 3602.

Format

- Parameter entry mode

G10L52 ;	Parameter entry mode setting
N_(Q_)R_ ;	For parameters other than the axis type or spindle type
N_P_(Q_)R_ ;	For axis type or spindle type parameters
:	
:	
G11 ;	Parameter entry mode cancel

N_ : Parameter number
R_ : Parameter setting value (Leading zeros can be omitted.)
(Q_) : Bit number 0 to 7 (to be set when a bit type parameter is input)
(Enabled when bit 4 (G1B) of parameter No.3454 = 1.)
P_ : Axis number 1 to maximum controlled axis number (to be specified when an axis type parameter or spindle type parameter is specified)

NOTE
G10L52 cannot be used to enter pitch error compensation data.

- Pitch error compensation data entry mode

G10L50 ;	Pitch error compensation data entry mode setting
N_(L1)R_ ;	Pitch error compensation data entry
:	
G11 ;	Pitch error compensation data entry mode cancel

N_ : Compensation position number for pitch errors compensation +10,000
L1 : Input format of pitch error compensation data (total value type)
R_ : Pitch error compensation data

NOTE
G10L50 cannot be used to enter parameter.

FRYER MACHINE SYSTEMS INC
Rob Zahersky

Explanation

- Parameter number(N_)

Programmable parameter input (G10L52) enables writing to a parameter for a specific path by adding that path number to the parameter number to be specified (N_). The format for the parameter number is as follows:

Nppxxxxxxx : Add a path number to the high-order 8th and 9th digits of a parameter number.
For pp, set a path number, and for xxxxxxx, set a parameter number. If a path number is omitted or if 0 is set, writing to a parameter for the local path is assumed, as usual.

NOTE
1 If a path number out of the range is specified, alarm PS0115 is issued.
2 Pitch error compensation data entry do not enable selection of a path.

- Setting value (R_)

Do not use a decimal point in the setting (R_) of a parameter and pitch error compensation data. To change a bit type parameter if bit 4 (G1B) of parameter No. 3454 is 1, specify 0 or 1. If a value other than 0 and 1 is specified, alarm PS1144, "G10 FORMAT ERROR", is issued.
As the value of R, a custom macro variable can be used.
When a parameter of real type is used, set an integer value in (R_) according to the increment system of the parameter.

- Bit number (Q__)

Bit number (Q_) is effective if bit 4 (G1B) of parameter No. 3454 is 1. To set a bit type parameter, set a number in the range of 0 to 7.
A custom macro variable can be used as the value of Q.

- Axis number (P_)

As the axis number (P_) of parameter entry mode, specify the order of a controlled axis to be displayed on the CNC display screen, by using an axis type parameter.
For example, specify P2 for the control axis which is displayed second.
For a spindle type also, specify the order of an axis to be display on the CNC display screen.
A custom macro variable can be used as the value of P.

- Input format of pitch error compensation data (L1)

When the input format of pitch error compensation data is incremental value type (bit 0 (APE) of parameter No. 3602 is set to 0), L1 is not to be specified.
When the input format of pitch error compensation data is total value type (bit 0 (APE) of parameter No. 3602 is set to 1), L1 needs to be specified.
If the data with L1 is input when bit 0 (APE) of parameter No. 3602 is set to 0, or the data without L1 is input when bit 0 (APE) of parameter No. 3602 is set to 1, alarm PS1300 “ILLEGAL ADDRESS” is caused.

⚠ WARNING
1 Do not fail to perform reference position return manually after changing the pitch error compensation data or backlash compensation data. Without this, the machine position can deviate from the correct position.
2 If the compensation data is changed when the input format is total value type(bit 0 (APE) of parameter No. 3602 is set to 1), power must be turned off before operation is continued.
3 The canned cycle mode must be cancelled before entering of parameters. When not cancelled, the drilling motion may be activated.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

⚠ CAUTION

Compatibility with the Series 16*i*/18*i*/21*i*:
This model has parameters that are not compatible with the Series 16*i*/18*i*/21*i*.
So, before using this function, make a check according to the Parameter Manual (B-64700EN) of this model.

NOTE

1 G10 and G11 must be specified in a block in which there is no other command.
2 Other NC statements cannot be specified while in parameter input mode and pitch error compensation data entry mode.
3 G10 and G11 are G codes for disabling buffering.
4 A decimal point cannot be used in the address N, L, P, Q and R.

- Writing to a parameter for another path

Programmable parameter input (G10L52) enables writing to a parameter for another path by specifying that path number in either of the following ways:

- Adding a path number to a parameter number (example 4)
- Specifying a path number using system variable #3018 (example 5)

⚠ CAUTION

If changing a parameter for another path by using this function, be sure to take appropriate measures such as combining the function with an inter-path wait, so that the function is executed while the axis for which to change the parameter is stopped.

[Example]

The following shows an example of changing a parameter for path 2 from path 1, using M100 and M101 as wait M codes.

```
Path 1 program
O0100;
      :
      :
M100P12 ..............(1)
G10L52;
N200xxxxxP_R_      } (A)
G11;               }
M101P12 ..............(2)
G10L52;
NxxxxxP_R_         } (B)
G11;               }
    :
M30;
```

```
Path 2 program
O0200;
    :
M100P12.... (1)
M101P12.... (2)
      :
      :
M30;
```

(1): Waits for the start of writing to a parameter from 1st path
(2): Waits for the end of writing to a parameter from the 1st path (Subsequently, operation is performed with the parameter changed in (A).)

(A):Change to a parameter for another path
→ Must be made after it is checked that the axis on the path for which to change the parameter is stopped.
(B):Change to a parameter for the local path

FRY... ...INE SYSTEMS INC ...hensky

- Writing of a power-off parameter

It is impossible to write a power-off parameter by using programmable parameter input (G10L52) because an alarm PS1302, "ILLEGAL DATA NUMBER" is caused.
However, when a bit 2 (WPP) of parameter No.11502 is set to 1, it is possible to write a power-off parameter. In this case, an alarm PW0000, "POWER MUST BE OFF" is caused when the parameter entry mode cancel (G11) is specified.

NOTE
In case that a bit 2 (WPP) of parameter No.11502 is set to 1, it is possible to write a power-off parameter by using programmable parameter input (G10L52) even when "PARAMETER WRITE" is disabled.

- Writing of a machine unit parameter

In the writing of a machine unit parameter by using programmable parameter input (G10L52), a parameter setting value (R_) is specified in the input unit. In case that the inch/metric conversion is enabled, the parameter setting value (R_) is converted to the machine unit and set to the parameter.
Example)
In case that the increment system is IS-B and the input unit is inch on the metric machine:
When R100000 (=10.0000inch) is specified, 254.000mm is set to the parameter.

Example

1. Set bit 2 (SBP) of bit type parameter No. 3404 (when the bit 4 (G1B) of parameter No. 3454 is set to 0)

```
G10L52 ;              Parameter entry mode
N3404 R 00000100 ;    SBP setting
G11 ;                 Cancel parameter entry mode
```

2. Set bit 2 (SBP) of bit type parameter No. 3404 (when the bit 4 (G1B) of parameter No. 3454 is set to 1)

```
G10L52 ;              Parameter entry mode
N3404 Q2 R 1 ;        SBP setting
G11 ;                 Cancel parameter entry mode
```

3. Change the values for the Z-axis (3rd axis) and A-axis (4th axis) in axis type parameter No. 1322 (the coordinates of stored stroke limit 2 in the positive direction for each axis).
(When the increment systems for the 3rd and 4th axes are IS-B and millimeter machine, respectively)

```
G10L52 ;              Parameter entry mode
N1322P3R4500 ;        Change the value for the Z-axis to 4.500
N1322PP4R12000 ;      Change the value for the A-axis to 12.000
G11 ;                 Cancel parameter entry mode
```

4. Of axis type parameter No. 01322 for the 2nd path, change the values for the 3rd and 4th axes.
(When the increment systems for the 3rd and 4th axes are IS-B and metric input, respectively)

```
G10L52;               Parameter entry mode
N20001322P3R4500;     Change the value for the 3rd axis in 2nd path to 4.500
N20001322P4R12000;    Change the value for the 4th axis in 2nd path to 12.000
G11;                  Cancel parameter entry mode
```

FRASER MACHINE SYSTEMS INC
Rob Zahensky

5. Write to the 4th axis of parameter No. 01322 for the 2nd path

#3018=2;	Set the path subject to parameter writing to the 2nd path
G10L52;	Parameter entry mode
N1322P4R12000;	Change the value for the 4th axis in 2nd path to 12.000
G11;	Cancel parameter entry mode
#3018= 0 ;	Return to the local path

6. Change compensation point numbers 10 and 20 of pitch error compensation. (bit 0 (APE) of parameter No. 3602 is set to 0)

G10L50 ;	Pitch error compensation data entry mode
N10010R1 ;	Change the compensation point number from 10 to 1
N10020R5 ;	Change the compensation point number from 20 to 5
G11 ;	Pitch error compensation data entry mode

7. Change compensation point numbers 10 and 20 of pitch error compensation. (bit 0 (APE) of parameter No. 3602 is set to 1)

G10L50 ;	Pitch error compensation data entry mode
N10010L1R1 ;	Change the compensation point number from 10 to 1
N10020L1R5 ;	Change the compensation point number from 20 to 5
G11 ;	Pitch error compensation data entry mode

FRYER MACHINE SYSTEMS INC
Rob Zahensky

19 PATTERN DATA INPUT

19.1 OVERVIEW

In the program of the fixed form processing with the custom macro, the operator select the processing pattern on the menu screen and specified the size, number and so on to the variable on the custom macro screen. As above mentioned, this function enables users to perform programming simply without programming using an existing NC language. With the aid of this function, a machine tool builder can prepare the program of a hole machining cycle (such as a boring cycle or tapping cycle) using the custom macro function, and can store it into the program memory. This cycle is assigned pattern names, such as BOR1, TAP3, and DRL2. An operator can select a pattern from the menu of pattern names displayed on the screen. Data (pattern data) which is to be specified by the operator should be created in advance with variables in a drilling cycle.
The operator can identify these variables using names such as DEPTH, RETURN RELIEF, FEED, MATERIAL or other pattern data names. The operator assigns values (pattern data) to these names.
The operator selects the pattern on the menu screen, and the selected pattern number is assigned to the system variable. The custom macro of the selected pattern can be started by starting a program then referring to the system variable in the program.

19.2 EXPLANATION

This function is consist of Pattern menu screen and Custom macro screen.
The process pattern is selected on the pattern menu screen.
Then the process pattern is selected, the custom macro screen is displayed.
On this custom macro screen, the variable with the name and comment is displayed according to the selected process pattern.
The process data can be input by referring to the variable name with the numerical value on the drawing.

The Fig. 19.2 (a) to Fig. 19.2 (e) are the example for the pattern menu and the custom macro.

(1) Pattern menu screen

Fig. 19.2 (a) Pattern data menu screen (10.4-inch display unit)

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

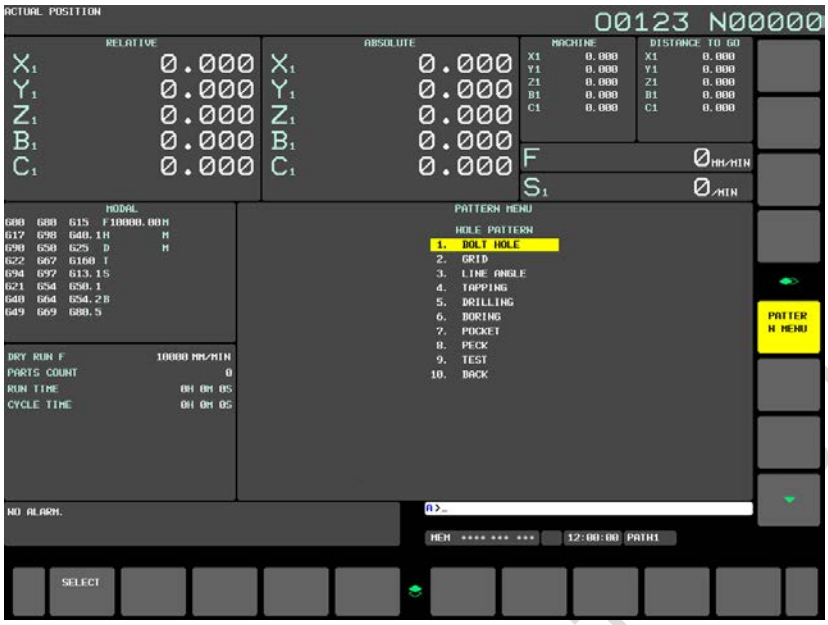


Fig. 19.2 (b) Pattern data menu screen (15-inch display unit)

(2) Custom macro screen

The name of variable and comment can be displayed on the usual custom macro screen.

The menu title and pattern name on the pattern menu screen and the variable name on the custom macro screen can be defined

The position of the comment displayed on the custom macro screen can be selected by setting bit 0 (POC) of parameter No. 11318.

On 15-/19-inch display unit, the comment is displayed at only bottom of the custom macro screen.

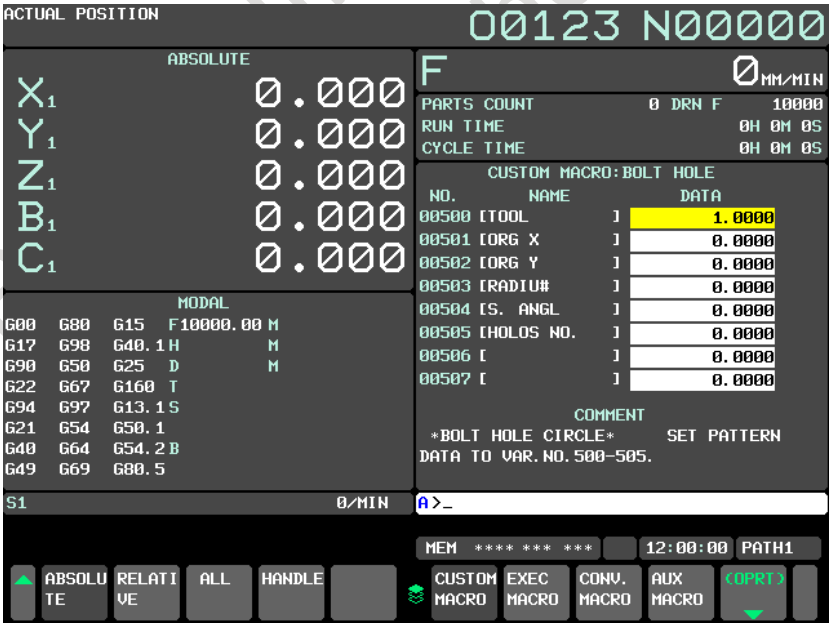


Fig. 19.2 (c) Custom macro screen（bit 0 (POC) of parameter No. 11318=0）(10.4-inch display unit)

FRYER MACHINE SYSTEMS INC

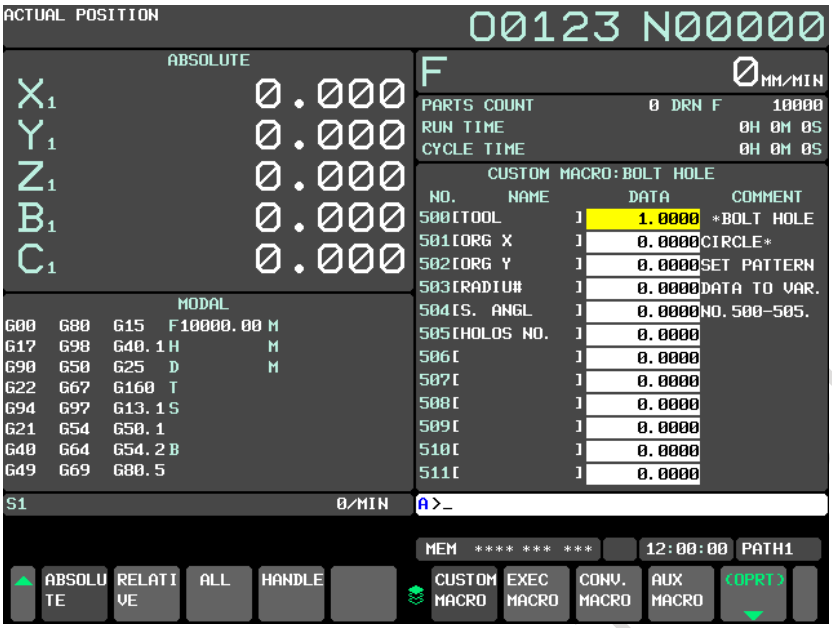


Fig. 19.2 (d) Custom macro screen (bit 0 (POC) of parameter No. 11318=1) (10.4-inch display unit)

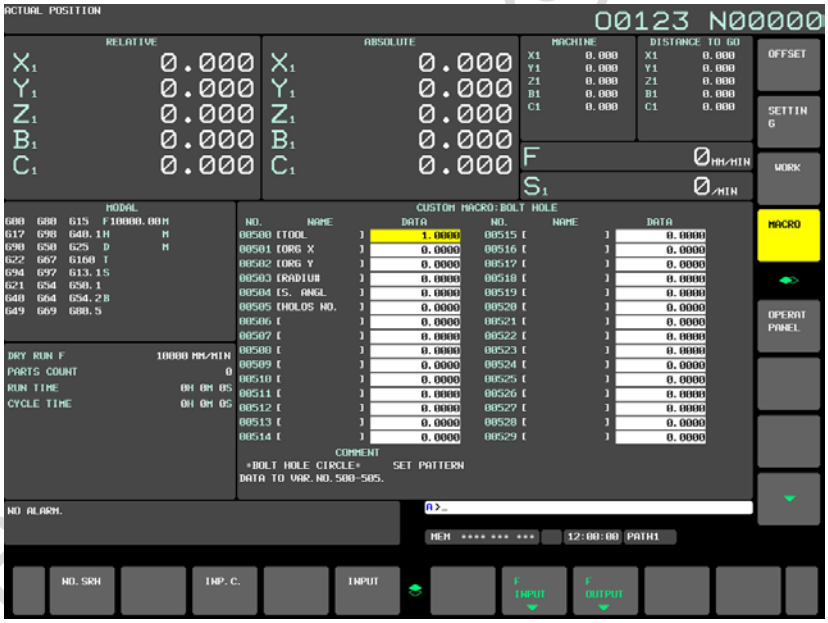


Fig. 19.2 (e) Custom macro screen (15-inch display unit)

FRYER SYSTEMS INC

19.3 EXPLANATION OF OPERATION

The following explains how to display the pattern menu screen.

For 8.4-/10.4-inch display unit

1 Press function key .
2 Press continuous menu key .
3 Press soft key [PATTERN MENU].

For a 15-/19-inch display unit

1 Press function key .
2 Press vertical soft key [NEXT PAGE] several times.
3 Press vertical soft key [PATTERN MENU].

Pattern menu screen

The pattern menu screen as Fig. 19.3 (a) or Fig. 19.3 (b) is displayed.

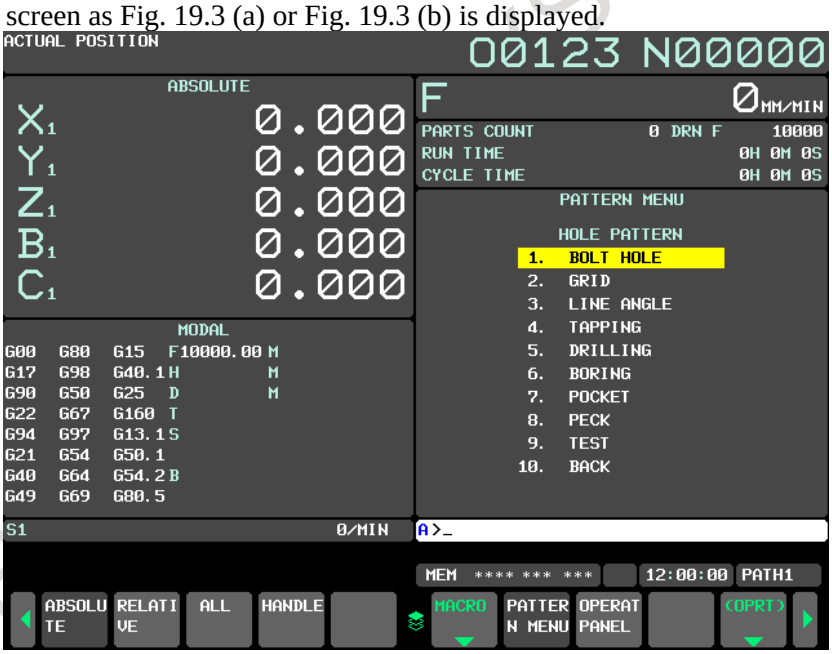


Fig. 19.3 (a) Pattern menu screen (10.4-inch display unit)

FRYER SYSTEMS INC

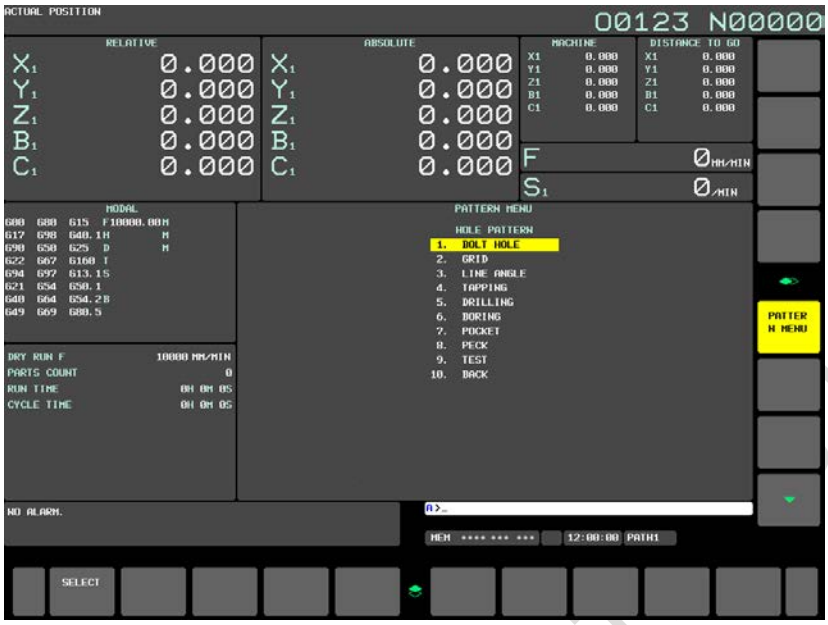


Fig. 19.3 (b) Pattern menu screen (15-inch display unit)

Select the pattern on this screen
The following two methods are effective.

- Selection by cursor

  Move the cursor to the pattern name with the cursor move keys [↑], [↓] and press the soft key [SELECT] or [INPUT].

- Selection by setting of pattern number
  The number that is displayed in the left side of the pattern name is input and press the soft key [select] or [INPUT].

The selected pattern number is registered to system variable #5900. The custom macro of the selected pattern can be started by starting a fixed program (external program No. search) with an external signal. This program refers to the system variable #5900 in the program. This system variable #5900 is kept after power-off.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Custom macro variable screen

The custom macro screen as Fig. 19.3 (c) or Fig. 19.3 (d) is displayed.

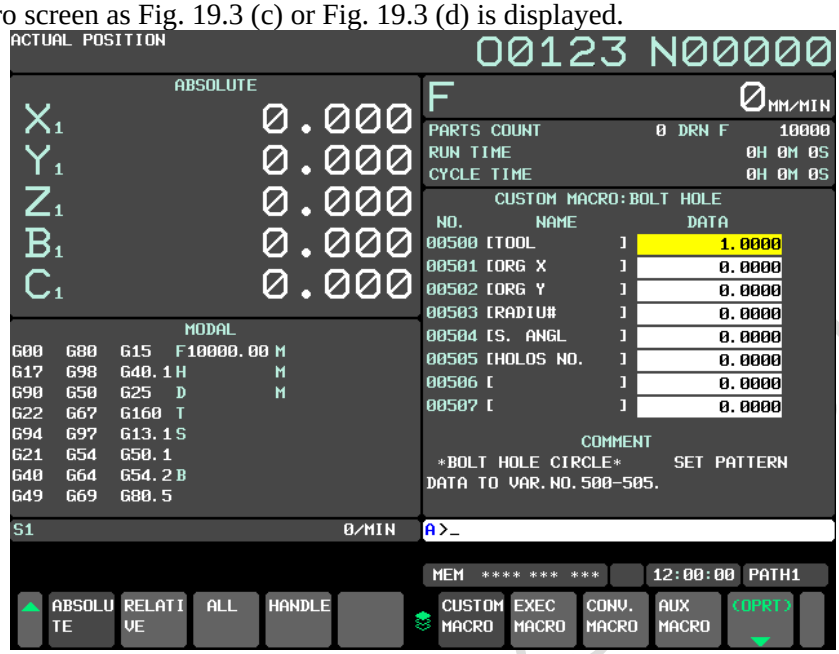


Fig. 19.3 (c) Custom macro screen when the pattern data is input (10.4-inch display unit)

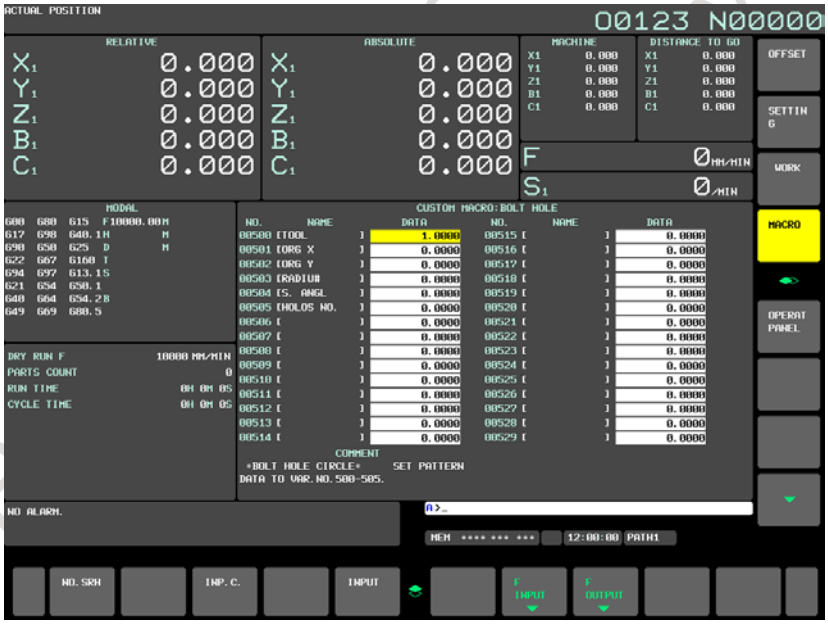


Fig. 19.3 (d) Custom macro screen when the pattern data is input (15-inch display unit)

When the screen is changed to the custom macro screen, the macro variable number that is selected first is specified with the parameters Nos.6101 to 6110. The macro variables that variable name is not defined can be input, too.

NOTE

1 The variable name that is displayed cannot be used as the common variable name of the NC program.

2 When the common variable name is defined by SETVN command, the variable name defined by pattern data input function is given priority.

NOTE

3 When bit 0 (POC) of parameter No.11318 is set to “1”, The variable number is three digit display.
And the value of 12 digits or more is input, 11 digits from head of value are displayed.
Example)
Input: -123456789.123 → Display: -123456789.1

19.4 DEFINITION OF THE SCREEN

The definition of the screen is performed by NC program.

Program configuration

This function is consist of one program for the definition of pattern menu screen and maximum ten programs for the definition of custom macro screen.
Register the macro program for the pattern data input function to the following folder.
//CNC_MEM/USER/LIBRARY/PATH1 to PATH[n]
* [n] is the maximum path number of the system
* The PATH1 to PATH[n] should be made by the machine tool builder.

The program number is as Table 19.4 (a).

Table 19.4 (a) Numbers of subprograms employed in the pattern data input function

Sub program No.	Screen
O9500	Specifies character strings displayed on the pattern data menu.
O9501	Specifies a character string of the pattern data corresponding to pattern No.1
O9502	Specifies a character string of the pattern data corresponding to pattern No.2
O9503	Specifies a character string of the pattern data corresponding to pattern No.3
O9504	Specifies a character string of the pattern data corresponding to pattern No.4
O9505	Specifies a character string of the pattern data corresponding to pattern No.5
O9506	Specifies a character string of the pattern data corresponding to pattern No.6
O9507	Specifies a character string of the pattern data corresponding to pattern No.7
O9508	Specifies a character string of the pattern data corresponding to pattern No.8
O9509	Specifies a character string of the pattern data corresponding to pattern No.9
O9510	Specifies a character string of the pattern data corresponding to pattern No.10

Table 19.4 (b) Macro commands used in the pattern data input function

G code	H code	Function
G65	H90	Specifies the menu title.
G65	H91	Specifies the pattern name.
G65	H92	Specifies the pattern data title.
G65	H93	Specifies the variable name.
G65	H94	Specifies the comment.

Table 19.4 (c) System variables employed in the pattern data input function

System variable	Function
#5900	Pattern No. selected by user.

NOTE

The pattern menu screen and the custom macro screen are defined only by preserving programs O9500 to O9510 in the above-mentioned folder, and these programs need not be executed. If these programs are executed, alarm (PS0074), “ILLEGAL PROGRAM NUMBER” is issued.

FRYER MACHINE SYSTEMS INC FOR SALE ONLY

19.4.1 Definition of the Pattern Menu Screen

Menu title and pattern name are defined as follows.

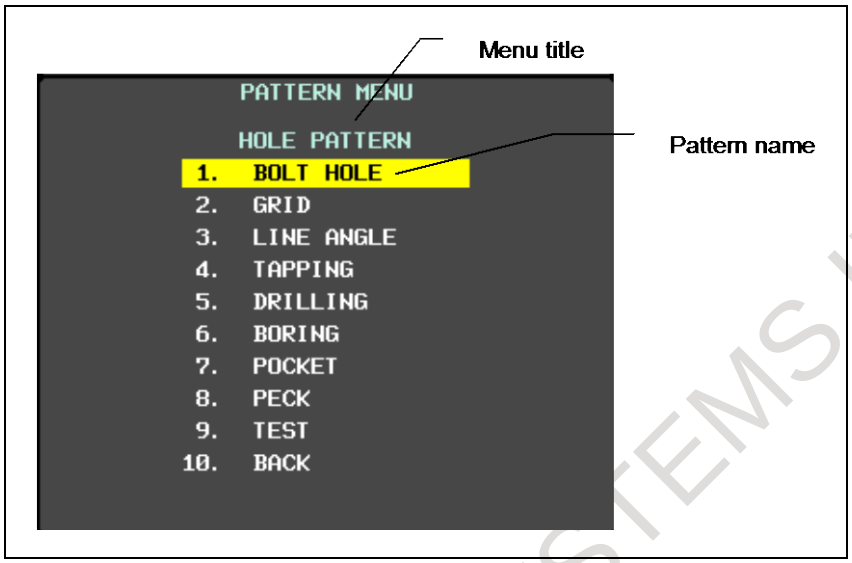


Fig. 19.4.1 (a) Pattern menu screen

Definition of menu title

The character string displayed in the menu title of the pattern menu screen is defined.
The menu title is specified up to 12 characters in a half size letter and up to 6 characters in a full size letter such as kanji character.

- Format

G65 H90 P_ Q_ R_ I_ J_ K_ ;

H90 : Specifies the menu title
P_ : The code of 1st and 2nd characters of title
Q_ : The code of 3rd and 4th characters of title
R_ : The code of 5th and 6th characters of title
I_ : The code of 7th and 8th characters of title
J_ : The code of 9th and 10th characters of title
K_ : The code of 11th and 12th characters of title
As for the way of setting the character-code, refer to the Subsection 19.4.3, "Setting the Character-codes".

Definition of pattern name

The character string displayed in the pattern name which becomes a menu item is defined.
The pattern name is specified up to 10 characters in a half size letter and up to 5 characters in a full size letter.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- **Format**

G65 H91 P_ Q_ R_ I_ J_ K_ ;

H91 : Specifies the pattern name
P_ : Specifies the menu number of the pattern name
The menu number = 1 to 10
Q_ : The code of 1st and 2nd characters of pattern name
R_ : The code of 3rd and 4th characters of pattern name
I_ : The code of 5th and 6th characters of pattern name
J_ : The code of 7th and 8th characters of pattern name
K_ : The code of 9thd and 10th characters of pattern name
As for the way of setting the character-code, refer to the Subsection 19.4.3, "Setting the Character-codes".

Example

The following is example for pattern menu screen.

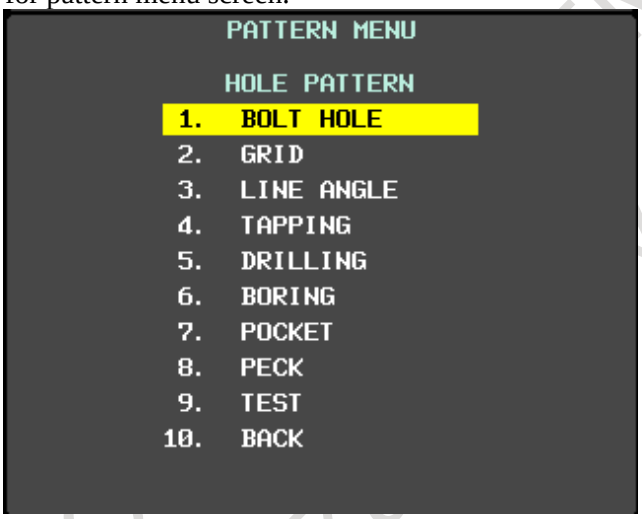


Fig. 19.4.1 (b) Pattern menu screen

```
O9500
N1 G65 H90 P072079 Q076069 R032080 I065084 J084069 K082078;... "HOLE PATTERN"
N2 G65 H91 P1 Q066079 R076084 I032072 J079076 K069032; ..... "BOLT HOLE"
N3 G65 H91 P2 Q071082 R073068; ..................................... "GRID"
N4 G65 H91 P3 Q076073 R078069 I032065 J078071 K076069; ..... "LINE ANGLE"
N5 G65 H91 P4 Q084065 R080080 I073078 J071032;  ..................... "TAPPING"
N6 G65 H91 P5 Q068082 R073076 I076073 J078071;..................... "DORILLING"
N7 G65 H91 P6 Q066079 R082073 I078071; ............................ "BORING"
N8 G65 H91 P7 Q080079 R067075 I069084; ............................ "POCKET"
N9 G65 H91 P8 Q080069 R067075; ..................................... "PECK"
N10 G65 H91 P9 Q084069 R083084; .................................... "TEST"
N11 G65 H91 P10 Q066065 R067075; ................................... "BACK"
N12 M99;
```

19.4.2 Definition of the Custom Macro Screen

The title, variable name and comment are defined as follows.

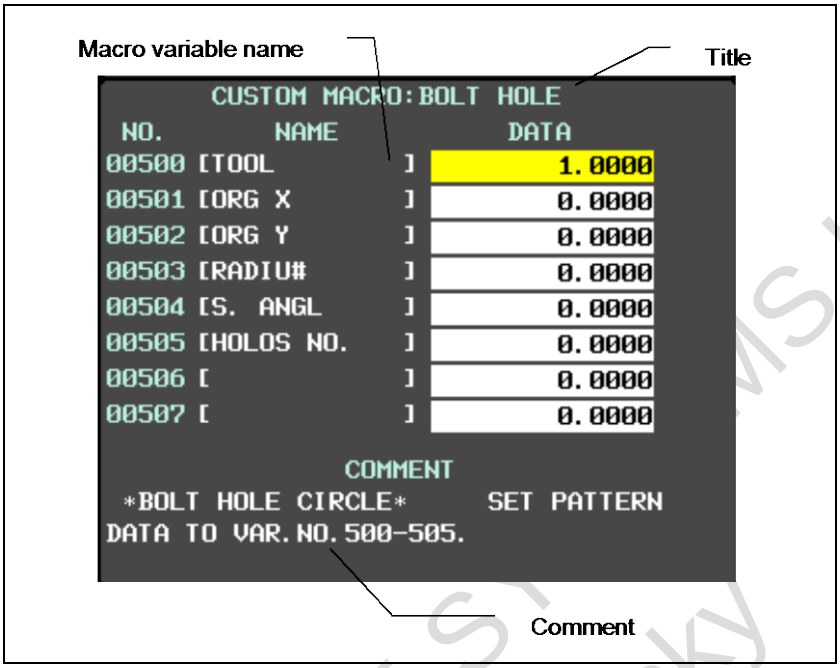


Fig. 19.4.2 (a) Custom macro screen

Definition of title

The character string displayed in the title of the custom macro screen is defined.
The title is specified up to 12 characters in a half size letter and up to 6 characters in a full size letter.

- Format

G65 H92 P_ Q_ R_ I_ J_ K_ ;

H92: Specifies the menu title
P_ : The code of 1st and 2nd characters of the menu title
Q_ : The code of 3rd and 4th characters of the menu title
R_ : The code of 5th and 6th characters of the menu title
I_ : The code of 7th and 8th characters of the menu title
J_ : The code of 9th and 10th characters of the menu title
K_ : The code of 11th and 12th characters of the menu title

As for the way of setting the character-code, refer to the Subsection 19.4.3, "Setting the Character-codes".

Definition of macro variable

The character string displayed in the macro variable name is defined.
The macro variable is specified up to 10 characters in a half size letter and up to 5 characters in a full size letter.

The variable which can be used is as follows
#100 to 199 (100 variables)
#500 to 999 (500 variables), 600 variables in total

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- **Format**

G65 H93 P_ Q_ R_ I_ J_ K_ ;

H93: Specifies the variable name
P_ : Specifies the variable number
Specifies 100 to 199 or 500 to 999
Q_ : The code of 1st and 2nd characters of the variable name
R_ : The code of 3rd and 4th characters of the variable name
I_ : The code of 5th and 6th characters of the variable name
J_ : The code of 7th and 8th characters of the variable name
K_ : The code of 9th and 10th characters of the variable name
As for the way of setting the character-code, refer to the Subsection 19.4.3, "Setting the Character-codes".

Definition of a comment

The character string of the comment displayed on the custom macro screen is defined.
The comment is specified by up to 12 characters in a half size letter and up to 6 characters in a full size letter per one block.

- When the comment is displayed at the bottom（bit 0 (POC) of parameter No.11318=0）
  The number of blocks is defined up to maximum 9 blocks.
  One line is composition of 3 blocks and maximum 3 lines can be displayed.

  The comment is displayed in the order specified by program as the Fig. 19.4.2 (b).
  The block is arranged in defined order.

Block 1	Block 2	Block 3
Block 4	Block 5	Block 6
Block 7	Block 8	Block 9

Fig. 19.4.2 (b) Block layout of comment

- When the comment is displayed at the right（bit 0 (POC) of parameter No.11318=1）
  1 line is composed by 1 blocks, the maximum number of lines is 8 on the 8.4-inch display unit or 12 on the 10.4-inch display unit.

- **Format**

G65 H94 P_ Q_ R_ I_ J_ K_ ;

H94: Specifies the comment
P_ : The code of 1st and 2nd characters of comment
Q_ : The code of 3rd and 4th characters of comment
R_ : The code of 5th and 6th characters of comment
I_ : The code of 7th and 8th characters of comment
J_ : The code of 9th and 10th characters of comment
K_ : The code of 11th and 12th characters of comment
As for the way of setting the character-code, refer to the Subsection 19.4.3, "Setting the Character-codes".

FRYER MACHINE SYSTEMS INC
Rob Zabensky

Example

The following is example of the custom macro screen.

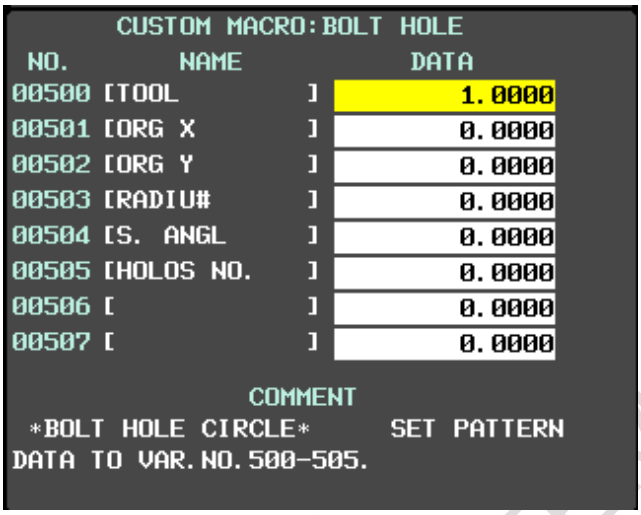


Fig. 19.4.2 (c) Custom macro screen（bit 0 (POC) of parameter No. 11318=0）

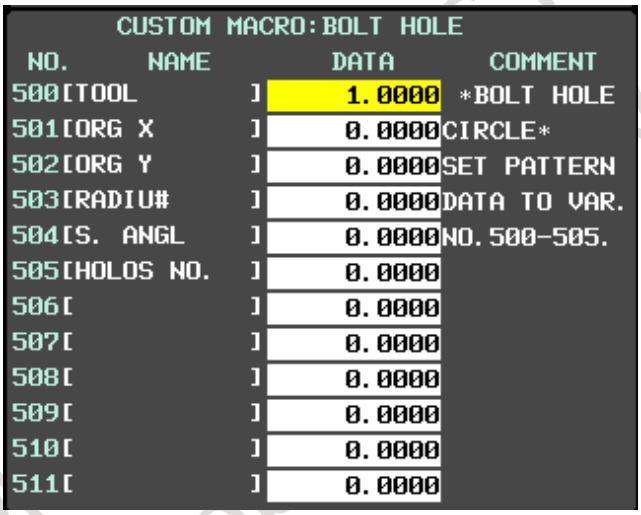


Fig. 19.4.2 (d) Custom macro screen（bit 0 (POC) of parameter No. 11318=1）

```
O9501;
N1 G65 H92 P066079 Q076084 R032072 I079076 J069032;....................."BOLT HOLE"
N2 G65 H93 P500 Q084079 R079076; ....................................................."TOOL"
N3 G65 H93 P501 Q079082 R071032 I08832; ......................................."ORG X"
N4 G65 H93 P502 Q079082 R071032 I08932; ......................................."ORG Y"
N5 G65 H93 P503 Q082065 R068073 I085803; ....................................."RADIUS"
N6 G65 H93 P504 Q083046 R032065 I078071 J076032........................."S. ANGL"
N7 G65 H93 P505 Q072079 R076079 I083032 J078079 K046032............."HOLES NO."
N8 G65 H94 P032042 Q066079 R076084 I032072 J079076 K069032;....." *BOLT HOLE"
N9 G65 H94 P067073 Q082067 R076069 I042032; ................................."CIRCLE*"
N10 G65 H94 P083069 Q084032 R080065 I084084 J069082 K078032;..."SET PATTERN"
N11 G65 H94 P068065 Q084065 R032084 I079032 J086065 K082046;..."DATA NO VAR."
N12 G65 H94 P078079 Q046053 R048048 I045053 J048053 K046032;..."NO500-505"
N13 M99;
```

19.4.3 Setting the Character-codes

The character cannot be used to specify the NC program.
Therefore, the code corresponding to the character is specified.
One character is consist of three figures in a half size letter and six figures in a full size letter.
The character code is specified for each address of the G65 instruction by six digits.

Refer to the table for the character code.
Example)
When "ABCDEFGH" is specified, the description of the code is as follows.
Encoded character string : 065 066 067 068 069 070 071 072

P065066 Q067068 R069070 I071072;
AB CD EF GH

NOTE

1 Space (032) is added ahead of the character-code, when the character-code of three digits or less is specified.
Example)
P065066 Q067; → " AB C "
032(space) is put at the end, when "ABC" is displayed.
P065066 Q067032; → " ABC "

2 It is assumed in that the space of two characters was defined in the address when there is an address not defined.
Example)
P065066 I067068; → "AB CD"

3 The character string corresponding to the block where G code other than G65 is instructed or where undefined character code is specified is not displayed.

Characters and codes to be used for the pattern data input function

Character	Code	Comment	Character	Code	Comment
A	065		6	054	
B	066		7	055	
C	067		8	056	
D	068		9	057	
E	069			032	Space
F	070		!	033	Exclamation mark
G	071		“	034	Quotation mark
H	072		#	035	Hash sign
I	073		$	036	Dollar sign
J	074		%	037	Percent
K	075		&	038	Ampersand
L	076		‘	039	Apostrophe
M	077		*	042	Asterisk
N	078		+	043	Plus sign
O	079		,	044	Comma
P	080		-	045	Minus sign
Q	081		.	046	Period
R	082		/	047	Slash
S	083		:	058	Colon
T	084		;	059	Semicolon
U	085		<	060	Left angle bracket
V	086		=	061	Equal sign
W	087		>	062	Right angle bracket

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Character	Code	Comment	Character	Code	Comment
X	088		?	063	Question mark
Y	089		@	064	At mark
Z	090		[	091	Left square bracket
0	048		¥	092	Yen sign
1	049		]	093	Right square bracket
2	050		^	094	
3	051		_	095	Underscore
4	052				
5	053				

The characters and the codes of the katakana is as follows.

Character	Code	Comment	Character	Code	Comment
ア	177		ム	209	
イ	178		メ	210	
ウ	179		モ	211	
エ	180		ヤ	212	
オ	181		ユ	213	
カ	182		ヨ	214	
キ	183		ラ	215	
ク	184		リ	216	
ケ	185		ル	217	
コ	186		レ	218	
サ	187		ロ	219	
シ	188		ワ	220	
ス	189		ヲ	166	
セ	190		ン	221	
ソ	191		ァ	167	
タ	192		ィ	168	
チ	193		ゥ	169	
ツ	194		ェ	170	
テ	195		ォ	171	
ト	196		ャ	172	
ナ	197		ュ	173	
ニ	198		ョ	174	
ヌ	199		ッ	175	
ネ	200		″	222	Diacritical mark
ノ	201		°	223	Diacritical mark
ハ	202		。	161	Punctuation
ヒ	203		「	162	Left quotation mark
フ	204		」	163	Right quotation mark
ヘ	205		、	164	Comma
ホ	206		・	165	Point
マ	207			000	Space
ミ	208				

NOTE
Diacritical mark is one character.

The characters and the codes of the hiragana and the kanji are as follows. The following hiraganas and kanjis use two characters of the alphanumeric character.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

ぁ	あ	ぃ	い	う	う	ぇ	え	ぉ	お
002 000	002 002	002 004	002 006	002 008	002 010	002 012	002 014	002 016	002 018
か	が	き	ぎ	く	ぐ	け	げ	こ	ご
002 020	002 022	002 024	002 026	002 028	002 030	002 032	002 034	002 036	002 038
さ	ざ	し	じ	す	ず	せ	ぜ	そ	ぞ
002 040	002 042	002 044	002 046	002 048	002 050	002 052	002 054	002 056	002 058
た	だ	ち	ぢ	っ	つ	づ	て	で	と
002 060	002 062	002 064	002 066	002 068	002 070	002 072	002 074	002 076	002 078
ど	な	に	ぬ	ね	の	は	ば	ぱ	ひ
002 080	002 082	002 084	002 086	002 088	002 090	002 092	002 094	002 096	002 098
び	ぴ	ふ	ぶ	ぷ	へ	べ	ぺ	ほ	ぼ
002 100	002 102	002 104	002 106	002 108	002 110	002 112	002 114	002 116	002 118
ぽ	ま	み	む	め	も	ゃ	や	ゅ	ゆ
002 120	002 122	002 124	002 126	002 128	002 130	002 132	002 134	002 136	002 138
ょ	よ	ら	り	る	れ	ろ	わ	わ	素
002 140	002 142	002 144	002 146	002 148	002 150	002 152	002 154	002 156	002 158
材	を	ん	種	類	棒	穴	成	形	質
002 160	002 162	002 164	002 166	002 168	002 170	002 172	002 174	002 176	002 178
寸	法	外	径	長	端	面	最	小	内
002 180	002 182	002 184	002 186	002 188	002 190	002 192	002 194	002 196	002 198
大	加	工	切	削	倣	正	途	中	荒
002 200	002 202	002 204	002 206	002 208	002 210	002 212	002 214	002 216	002 218
具	番	号	仕	上	込	点	方	向	速
002 220	002 222	002 224	002 226	002 228	002 230	002 232	002 234	002 236	002 238
度	送	量	開	始	深	主	軸		
002 240	002 242	002 244	002 246	002 248	002 250	002 252	002 254		
回	転	数	位	置	決	直	線	時	円
003 000	003 002	003 004	003 006	003 008	003 010	003 012	003 014	003 016	003 018
反	現	在	指	令	値	領	域	診	断
003 020	003 022	003 024	003 026	003 028	003 030	003 032	003 034	003 036	003 038
操	作	手	引	機	械	残	移	動	次
003 040	003 042	003 044	003 046	003 048	003 050	003 052	003 054	003 056	003 058
早	電	源	投	入	間	分	秒	自	運
003 060	003 062	003 064	003 066	003 068	003 070	003 072	003 074	003 076	003 078
負	荷	実	使	用	寿	命	新	規	除
003 080	003 082	003 084	003 086	003 088	003 090	003 092	003 094	003 096	003 098
隅	取	単	補	能	独	終	了	記	角
003 100	003 102	003 104	003 106	003 108	003 110	003 112	003 114	003 116	003 118
溝	刃	幅	広	設	定	ー	覧	表	部
003 120	003 122	003 124	003 126	003 128	003 130	003 132	003 134	003 136	003 138
炭	合	金	鋼	超	硬	先	付	摩	耗
003 140	003 142	003 144	003 146	003 148	003 150	003 152	003 154	003 156	003 158
仮	想	副	行	挿	消	去	山	高	準
003 160	003 162	003 164	003 166	003 168	003 170	003 172	003 174	003 176	003 178
備	完	後	弧	助	択	無	視	器	原
003 180	003 182	003 184	003 186	003 188	003 190	003 192	003 194	003 196	003 198
登	録	再	処	理	描	画	過	容	編
003 200	003 202	003 204	003 206	003 208	003 210	003 212	003 214	003 216	003 218
集	未	対	相	座	標	示	名	歯	変
003 220	003 222	003 224	003 226	003 228	003 230	003 232	003 234	003 236	003 238
呼	推	馬	力	系	選	達	閉		
003 240	003 242	003 244	003 246	003 248	003 250	003 252	003 254		

FRYER MACHINE SYSTEMS INC Rob Zaleski

禁	復	帰	書	個	桁	稼	由	両	半
004 000	004 002	004 004	004 006	004 008	004 010	004 012	004 014	004 016	004 018
逃	底	逆	下	空	四	触	平	代	辺
004 020	004 022	004 024	004 026	004 028	004 030	004 032	004 034	004 036	004 038
格	子	周	心	本	群	停	止	巾	微
004 040	004 042	004 044	004 046	004 048	004 050	004 052	004 054	004 056	004 058
状	路	範	囲	倍	率	注	側	特	殊
004 060	004 062	004 064	004 066	004 068	004 070	004 072	004 074	004 076	004 078
距	離	連	続	増	隔	件	初	期	条
004 080	004 082	004 084	004 086	004 088	004 090	004 092	004 094	004 096	004 098
経	握	圧	扱	陰	隠	右	押	横	黄
004 100	004 102	004 104	004 106	004 108	004 110	004 112	004 114	004 116	004 118
億	屋	化	何	絵	階	概	該	巻	換
004 120	004 122	004 124	004 126	004 128	004 130	004 132	004 134	004 136	004 138
気	起	軌	技	疑	供	共	境	強	教
004 140	004 142	004 144	004 146	004 148	004 150	004 152	004 154	004 156	004 158
掘	繰	係	傾	型	検	権	研	肩	見
004 160	004 162	004 164	004 166	004 168	004 170	004 172	004 174	004 176	004 178
験	元	弦	減	孔	巧	控	更	校	構
004 180	004 182	004 184	004 186	004 188	004 190	004 192	004 194	004 196	004 198
根	左	差	雑	参	散	産	算	治	耳
004 200	004 202	004 204	004 206	004 208	004 210	004 212	004 214	004 216	004 218
式	失	修	十	従	勝	商	少	尚	昇
004 220	004 222	004 224	004 226	004 228	004 230	004 232	004 234	004 236	004 238
植	色	食	伸	信	侵	振	浸		
004 240	004 242	004 244	004 246	004 248	004 250	004 252	004 254		
真	暗	以	意	異	影	鋭	越	価	可
005 000	005 002	005 004	005 006	005 008	005 010	005 012	005 014	005 016	005 018
科	果	箇	課	各	拡	核	学	掛	漢
005 020	005 022	005 024	005 026	005 028	005 030	005 032	005 034	005 036	005 038
簡	観	関	含	却	客	休	急	業	曲
005 040	005 042	005 044	005 046	005 048	005 050	005 052	005 054	005 056	005 058
均	筋	継	計	軽	言	限	互	降	採
005 060	005 062	005 064	005 066	005 068	005 070	005 072	005 074	005 076	005 078
済	細	姿	思	写	射	斜	者	車	借
005 080	005 082	005 084	005 086	005 088	005 090	005 092	005 094	005 096	005 098
縦	重	出	述	術	渉	照	省	章	証
005 100	005 102	005 104	005 106	005 108	005 110	005 112	005 114	005 116	005 118
象	身	進	人	図	違	印	沿	遠	央
005 120	005 122	005 124	005 126	005 128	005 130	005 132	005 134	005 136	005 138
奥	往	応	会	解	改	割	活	願	基
005 140	005 142	005 144	005 146	005 148	005 150	005 152	005 154	005 156	005 158
奇	寄	岐	既	近	区	矩	駆	偶	旧
005 160	005 162	005 164	005 166	005 168	005 170	005 172	005 174	005 176	005 178
求	球	究	級	欠	結	口	語	誤	交
005 180	005 182	005 184	005 186	005 188	005 190	005 192	005 194	005 196	005 198
厚	項	刻	告	黒	財	策	糸	試	資
005 200	005 202	005 204	005 206	005 208	005 210	005 212	005 214	005 216	005 218
事	持	似	釈	弱	受	収	純	順	所
005 220	005 222	005 224	005 226	005 228	005 230	005 232	005 234	005 236	005 238
序	剰	場	常	飾	水	錐	据		
005 240	005 242	005 244	005 246	005 248	005 250	005 252	005 254		

FRYER MACHINE SYSTEMS INC
Rob Zaleski

制	整	製	前	全	然	則	属	即	他
006 000	006 002	006 004	006 006	006 008	006 010	006 012	006 014	006 016	006 018
多	存	谷	探	短	徴	鎮	調	頂	鉄
006 020	006 022	006 024	006 026	006 028	006 030	006 032	006 034	006 036	006 038
添	頭	同	導	道	熱	年	濃	箱	発
006 040	006 042	006 044	006 046	006 048	006 050	006 052	006 054	006 056	006 058
抜	伴	必	百	複	物	文	聞	併	忘
006 060	006 062	006 064	006 066	006 068	006 070	006 072	006 074	006 076	006 078
末	密	有	余	与	裏	立	略	青	席
006 080	006 082	006 084	006 086	006 088	006 090	006 092	006 094	006 096	006 098
石	積	赤	接	折	粗	創	双	捜	太
006 100	006 102	006 104	006 106	006 108	006 110	006 112	006 114	006 116	006 118
打	体	待	態	替	段	知	地	致	遅
006 120	006 122	006 124	006 126	006 128	006 130	006 132	006 134	006 136	006 138
追	通	伝	得	読	凸	凹	突	鈍	敗
006 140	006 142	006 144	006 146	006 148	006 150	006 152	006 154	006 156	006 158
杯	背	配	品	不	布	並	頁	別	片
006 160	006 162	006 164	006 166	006 168	006 170	006 172	006 174	006 176	006 178
返	勉	弁	保	明	滅	木	目	歪	揺
006 180	006 182	006 184	006 186	006 188	006 190	006 192	006 194	006 196	006 198
様	溶	要	抑	良	輪	和	話	枠	節
006 200	006 202	006 204	006 206	006 208	006 210	006 212	006 214	006 216	006 218
説	絶	千	専	浅	旋	総	走	退	台
006 220	006 222	006 224	006 226	006 228	006 230	006 232	006 234	006 236	006 238
第	題	卓	室	着	柱	鋳	丁		
006 240	006 242	006 244	006 246	006 248	006 250	006 252	006 254		
低	訂	肉	日	白	薄	比	皮	被	非
007 000	007 002	007 004	007 006	007 008	007 010	007 012	007 014	007 016	007 018
美	普	伏	歩	包	門	問	絡	列	万
007 020	007 022	007 024	007 026	007 028	007 030	007 032	007 034	007 036	007 038
利	訳	礼	乱	放	枚	約	練	油	劣
007 040	007 042	007 044	007 046	007 048	007 050	007 052	007 054	007 056	007 058
例	郭	戻	冷	垂	緑	紫	許	測	精
007 060	007 062	007 064	007 066	007 068	007 070	007 072	007 074	007 076	007 078
効	→	↗	↑	↖	←	↙	↓	↘	
007 080	007 082	007 084	007 086	007 088	007 090	007 092	007 094	007 096	007 098
				板	予	〃	家	装	管
007 100	007 102	007 104	007 106	007 108	007 110	007 112	007 114	007 116	007 118
粉	等					貫	安	α	β
007 120	007 122	007 124	007 126	007 128	007 130	007 132	007 134	007 136	007 138
程	抗	張	任	破	損	御	足	守	般
007 140	007 142	007 144	007 146	007 148	007 150	007 152	007 154	007 156	007 158
納	義	丸	汎	固	毎	当	的	詳	鳥
007 160	007 162	007 164	007 166	007 168	007 170	007 172	007 174	007 176	007 178
適	論	額	縁	温	給	界	混	監	締
007 180	007 182	007 184	007 186	007 188	007 190	007 192	007 194	007 196	007 198
護	己	称	樹	脂	料	落	確	認	報
007 200	007 202	007 204	007 206	007 208	007 210	007 212	007 214	007 216	007 218
排	性	生	績	判	搬	砥	θ	島	壁
007 220	007 222	007 224	007 226	007 228	007 230	007 232	007 234	007 236	007 238
				]	[	▗	■		
007 240	007 242	007 244	007 246	007 248	007 250	007 252	007 254		

FRYER MACHINE SYSTEMS INC
Rob Zaleski

20 HIGH-SPEED CUTTING FUNCTIONS

20.1 AI CONTOUR CONTROL FUNCTION I AND AI CONTOUR CONTROL FUNCTION II (G05.1)

Overview

The AI contour control I and AI contour control II functions are provided for high-speed, high-precision machining. This function enables suppression of acceleration/deceleration delays and servo delays that become larger with increases in the feedrate and reduction of machining profile errors.
In the descriptions below that are common to AI contour control I and AI contour control II, the term "AI contour control" is used.

The functions listed below are valid in the AI contour control mode.
Valid functions are limited depending on the command format and whether AI contour control I or II is used.
Refer to the manual supplied by the machine tool builder or contact the machine tool builder for the function etc. that can be actually used.

Table20.1 (a) Valid functions

Function	AI contour control I	AI contour control II
Number of blocks read ahead	40 (When G8 is specified: 1)	200 (When G8 is specified: 1)
Speed control with feedrate difference on each axis	Enabled	Enabled
Speed control with acceleration in circular interpolation	Enabled	Enabled
Speed control with acceleration on each axis	Enabled (When G8 is specified: Not enabled)	Enabled (When G8 is specified: Not enabled)
Smooth speed control	Not enabled	Enabled (When G8 is specified: Not enabled)
Speed control with cutting load	Not enabled	Enabled
Disregard of feedrate command	Not enabled	Enabled

Format

```
G05.1 Q1 [R_] ;        AI contour control mode on
  :
G05.1 Q0 ;             AI contour control mode off
  R   :   Machining condition selecting level (1 to 10)
```

The AI contour control mode can be controlled also with the formats that have been used for the conventional advanced preview control, high-precision contour control, and AI high-precision contour control functions.

```
G08 P1 [R_] ;          AI contour control mode on
  :
G08 P0 ;               AI contour control mode off
  R   :   Machining condition selecting level (1 to 10)
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

G05 P10000 [R_] ; AI contour control mode on

:

G05 P0 ; AI contour control mode off

R : Machining condition selecting level (1 to 10)

NOTE

1 Always specify G05.1, G08, and G05 in an independent block.
（Do not specify other G codes at the same time.）

2 G05 can be specified only for AI contour control II.

3 The AI contour control mode can also be turned off at a reset.

4 The AI contour control mode can be turned on at the start of automatic operation by setting bit 0 (SHP) of parameter No. 1604.

5 G05.1, G08, and the G05 commands are separate commands respectively. Please turn off the AI contour control mode by the command that turns on the AI contour control mode. AI contour control cannot be turned off by a different command.

6 There are the following conditions when each AI contour control mode on / off is commanded in the multiple.

(1) The AI contour control mode commands are G05, G05.1, and G08 in the order of a high-ranking command (G05 > G05.1 > G08). It is necessary to turn on the AI contour control mode from the subordinate position command sequentially when the multiple commands are executed.

(2) When the AI contour control mode is turned on by the G05P10000 command, even if the AI contour control mode ON/OFF command is executed by G05.1Q1/Q0 command in the mode, the command is ignored.

Example)

```
G05P10000      AI contour control mode on by G05
G05.1Q1        This command is ignored.
G05.1Q0        This command is ignored.
G05P0          AI contour control mode off by G05
```

(3) After the AI contour control mode is turned on by the G05P10000 or the G05.1Q1, if the G08 command is executed in the mode, alarm PS5110 “IMPROPER G-CODE (AICC MODE)” occurs.

Example)

```
G05P10000(G05.1Q1)     AI contour control mode on by G05(G05.1)
G08P1(P0)              Alarm PS5110 occurs
```

(4) When the multiple commands are executed, each command that turns on/off the AI contour control mode must be nested.

Example）

```
G08P1          AI contour control mode on by G08
:
G05.1Q1        AI contour control mode on by G05.1
:
G05P10000      AI contour control mode on by G05
:
G05P0          AI contour control mode off by G05
:
G05.1Q0        AI contour control mode off by G05.1
:
G08P0          AI contour control mode off by G08
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

> ⚠ **CAUTION**
> Set bit 6 (AP5) of parameter No.11549 to 1 if AI contour control is executed simultaneously by more than 5 paths. If AI contour control is executed simultaneously by more than 5 paths without setting this parameter to 1, sufficient performance might not be obtained.

Explanation

- Method of determining the tangent acceleration

Acceleration/deceleration is performed with the largest tangent acceleration/deceleration that does not exceed the acceleration set for each axis.

(Example)

X-axis permissible acceleration: 1000 mm/sec^2
Y-axis permissible acceleration: 1200 mm/sec^2
Acceleration change time: 20 msec

Program:

```
N1 G01 G91 X20.0 F6000 ; (Move on the X-axis.)
G04 X0.01 ;
N2 Y20.0 ;       (Move on the Y-axis.)
G04 X0.01 ;
N3 X20.0 Y20.0 ; (Move in the XY direction (at 45 deg).)
```

Since N3 performs interpolation for the X and Y axes in the 45-degree direction, the acceleration of the Y axis is controlled according to the X axis to become 1000 mm/s^2. Therefore, the combined acceleration is 1414 mm/s^2.

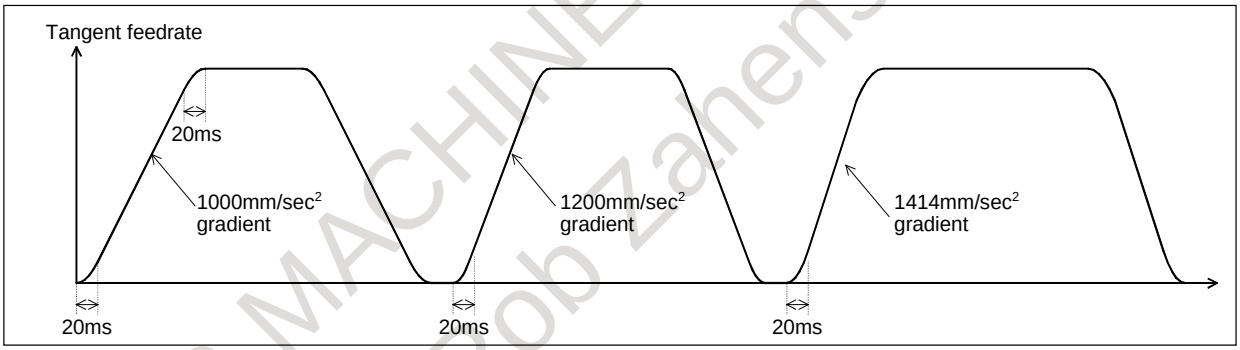


Fig. 20.1 (a)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Acceleration

Acceleration is performed so that the feedrate programmed for a block is attained at the beginning of the block.
When look-ahead acceleration/deceleration before interpolation is valid for multiple blocks, acceleration can be performed across more than one block.

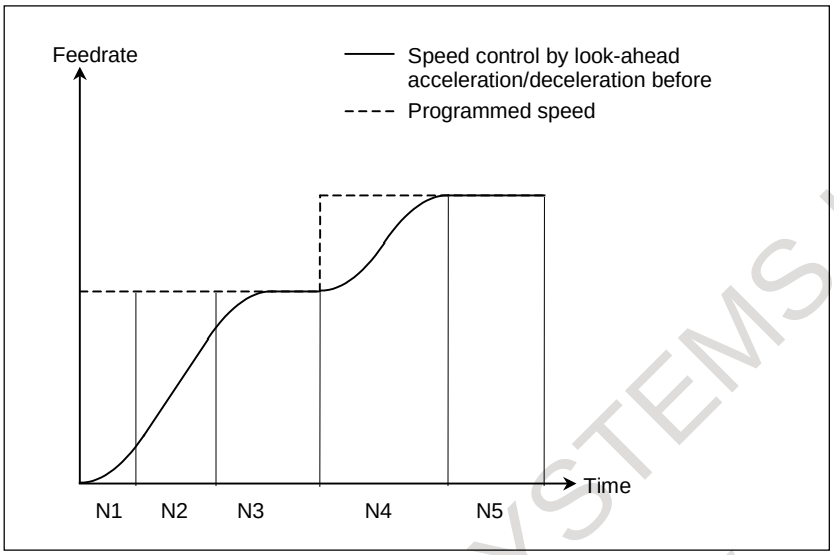


Fig. 20.1 (b)

- Deceleration

Deceleration starts in advance so that the feedrate programmed for a block is attained at the beginning of the block.
When look-ahead acceleration/deceleration before interpolation is valid for multiple blocks, deceleration can be performed across more than one block.

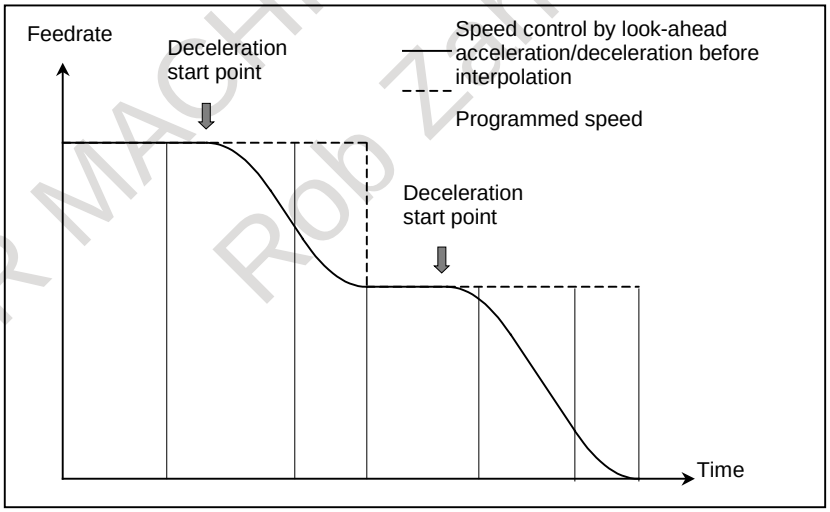


Fig. 20.1 (c)

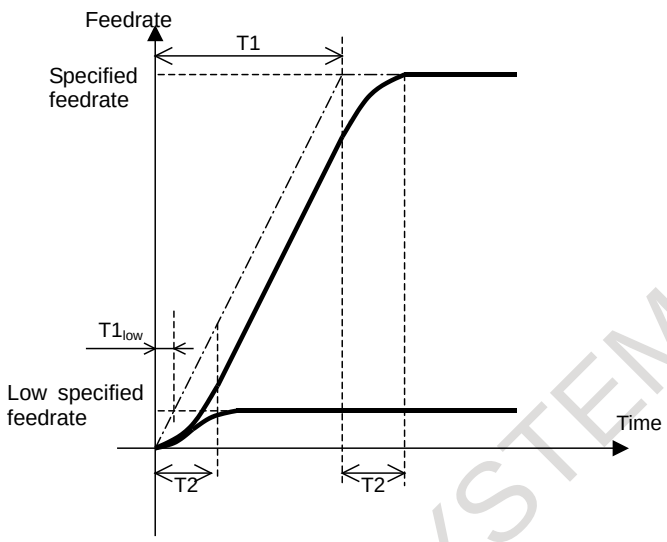
- Deceleration based on a distance

If the total distance of the blocks read ahead becomes shorter than or equal to the deceleration distance obtained from the current feedrate, deceleration starts.
If the total distance of the blocks read ahead during deceleration increases, acceleration is performed.
If the blocks of a small amount of travel are successively specified, deceleration and acceleration may be performed alternately, making the feedrate inconsistent.
To avoid this, decrease the programmed feedrate.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Function for changing time constant of look-ahead bell-shaped acceleration/deceleration

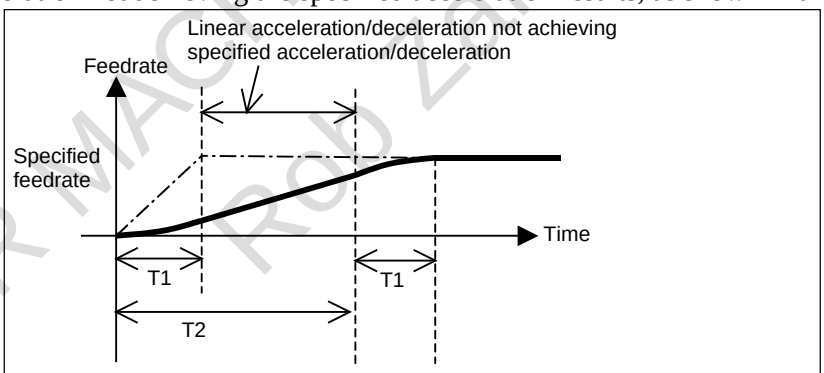
Look-ahead bell-shaped acceleration/deceleration before interpolation is performed according to the acceleration and acceleration change time set by the parameters, as shown in the figure below.



T1 : Time obtained from specified feedrate and specified acceleration (specified feedrate/acceleration (parameter No. 1660))
T2 : Acceleration change time (parameter No. 1772)

Fig. 20.1 (d)

Here, the acceleration change time (T2) remains constant regardless of the specified feedrate, while the acceleration time for the linear section (T1), which is determined by acceleration, varies with the specified feedrate. If T1 becomes shorter than T2 when the specified feedrate is low, linear acceleration/deceleration not achieving the specified acceleration results, as shown in the figure below.



T1 : Time obtained from specified feedrate and specified acceleration (specified feedrate/acceleration (parameter No. 1660))
T2 : Acceleration change time (parameter No. 1772)

Fig. 20.1 (e)

In such a case, set bit 3 (BCG) of parameter No. 7055 to 1. Then, the internal acceleration and vector time constant of look-ahead acceleration/deceleration before interpolation are changed to make the acceleration/deceleration pattern as close as possible to the optimum look-ahead bell-shaped acceleration/deceleration before interpolation based on a specified acceleration/deceleration reference speed, and so acceleration/deceleration time is reduced.

FRYER MACHINE SYSTEMS INC
Rob Zamansky

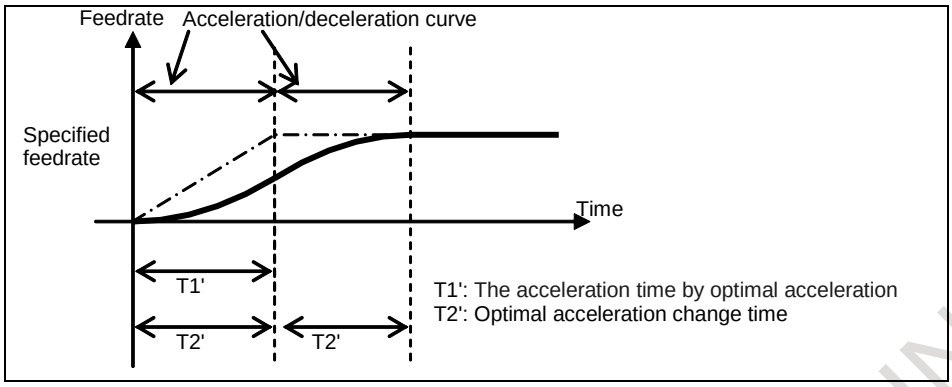


Fig. 20.1 (f)

There are three methods for specifying the acceleration/deceleration reference speed.
(1) Specifying the speed using an F in a G05.1 Q1 block
(2) Setting the speed on parameter No.7066
(3) Setting the speed specified with the F command issued at the start of cutting as the reference speed

When F is specified in a G05.1Q1 block, the specified feedrate is assumed to be the acceleration/deceleration reference speed. This command can be used only in the feed per minute mode.
If no F command is specified in a G05.1Q1 block, the feedrate specified in parameter No. 7066 is assumed to be the acceleration/deceleration reference speed. If 0 is set in parameter No. 7066, the F command specified in the cutting start block is assumed to be the acceleration/deceleration reference speed.

NOTE
When function for changing time constant of look-ahead bell-shaped acceleration/ deceleration before interpolation is enabled (bit 3 (BCG) of parameter No. 7055 is set to 1), be sure to specify acceleration rate of the reference axis (parameter No. 1660).
If acceleration rate of the reference axis is not specified, optimum acceleration rate is not obtained.

- Automatic feedrate control function

In AI contour control mode, the feedrate is automatically controlled by the reading-ahead of blocks.
The feedrate is determined using the following conditions. If the specified feedrate exceeds the determined feedrate, look-ahead acceleration/ deceleration before interpolation is performed to achieve the determined feedrate.
(1) Feedrate changes on each axis at a corner and the permissible feedrate change that has been set
(2) Expected acceleration on each axis and the permissible acceleration that has been set
(3) Cutting load that is expected from the travel direction on the Z-axis

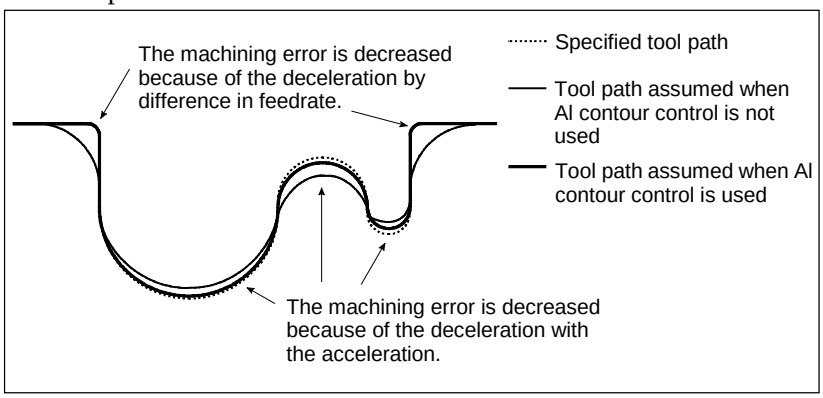


Fig. 20.1 (g)

FRIED MACHINE SYSTEMS INC Rob Zaremsky

For details, see the explanation of each function.

- Speed control based on the feedrate difference on each axis at a corner

By using the speed control based on the feedrate difference on each axis at a corner, if a feedrate change occurs on an axis on each axis at a corner, the feedrate is determined so that any feedrate difference exceeding the permissible feedrate difference on that axis that has been set for parameter No. 1783 does not occur, and deceleration is automatically performed.

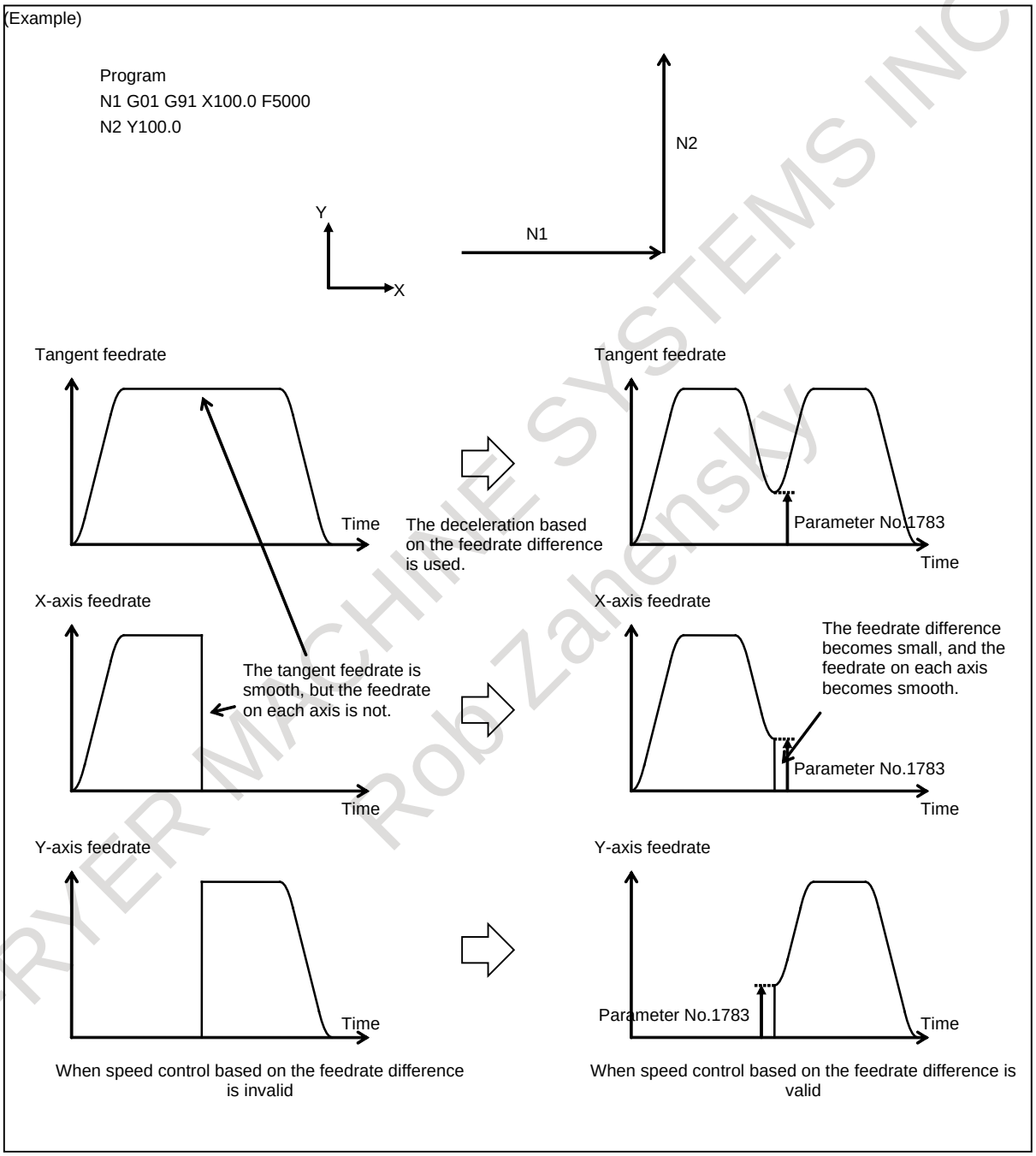


Fig.20.1 (h)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

The method of deceleration based on the feedrate difference differs depending on the setting made for bit 6 (FNW) of parameter No. 19500.
If parameter FNW is set to 0, the largest feedrate that does not exceed the permissible feedrate difference set for parameter No. 1783 is assumed to be the deceleration feedrate.
In this case, the deceleration feedrate differs if the travel direction differs, even if the shape is the same.

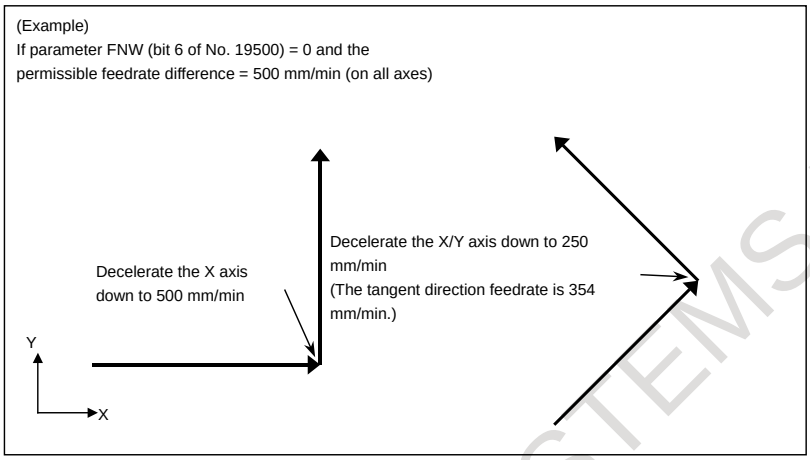


Fig.20.1 (i)

In the left-side example in the figure above, the X axis is inverted at the corner from the position direction to the negative direction, and deceleration is performed so that the feedrate difference becomes 500 mm/min. In other words, the feedrate is 250 mm/min both when the axis moves in the position direction and when it moves in the negative direction. As a result, the tangent direction feedrate becomes 354 mm/min.

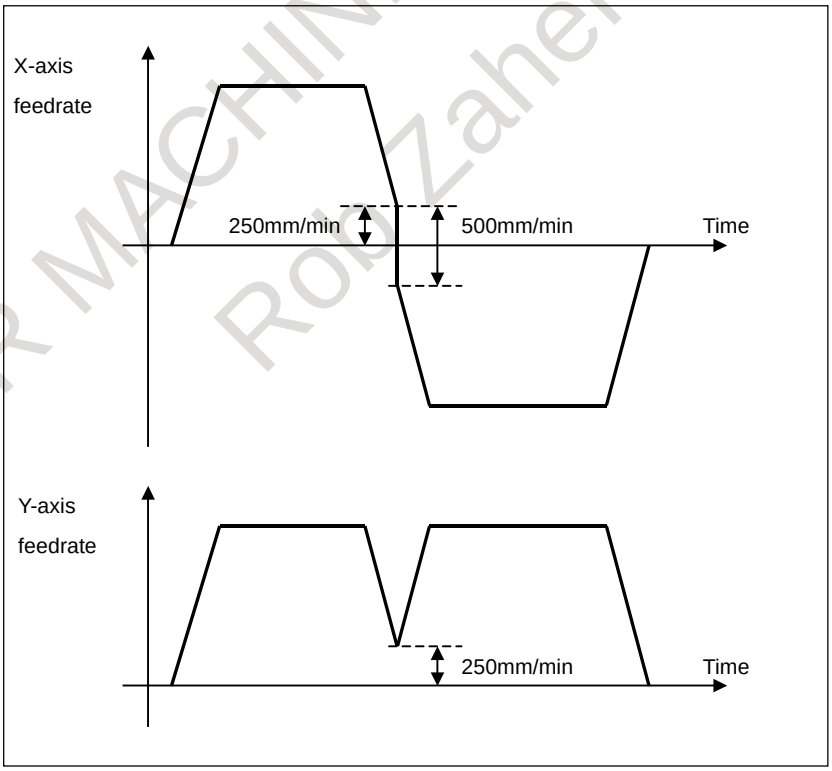


Fig.20.1 (j)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

If parameter FNW is set to 1, the feedrate is determined not only with the condition that the permissible feedrate difference and permissible acceleration on each axis are not exceeded, but also that the deceleration feedrate is constant regardless of the travel direction if the shape is the same.
If this parameter is set to 1, the deceleration feedrate determined with the feedrate difference may be up to 30% lower than that determined if this parameter is set to 0.

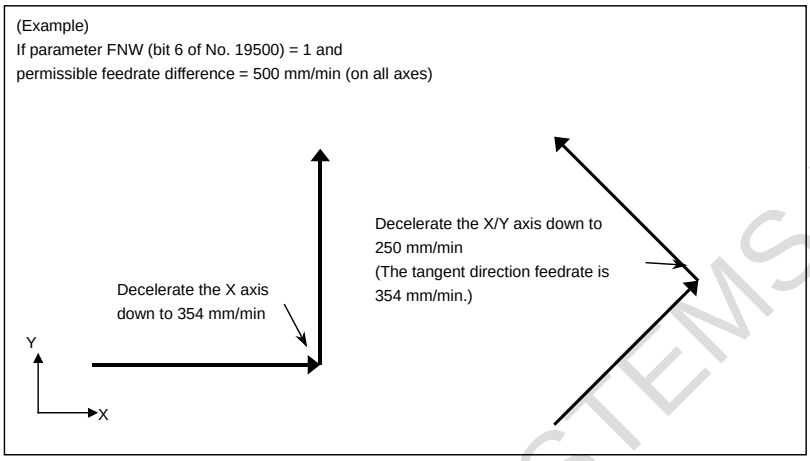


Fig.20.1 (k)

- Speed control with acceleration in circular interpolation

When high-speed cutting is performed in circular interpolation, helical interpolation, or spiral interpolation, the actual tool path has an error with respect to the programmed path. In circular interpolation, this error can be approximated from the equation given below.

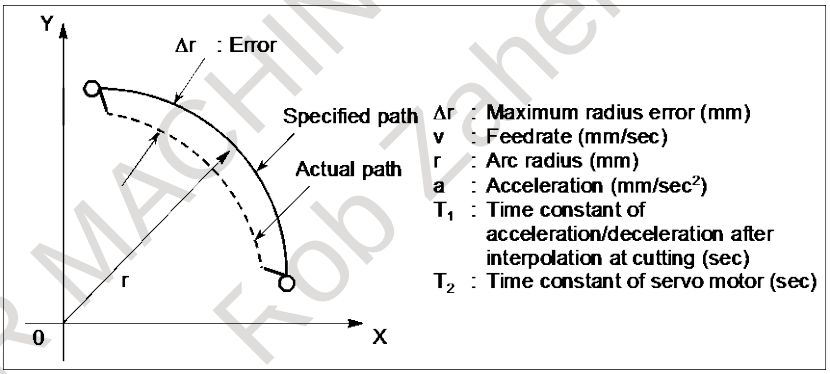


Fig.20.1 (l)

$$\Delta r = \frac{1}{2}(T_1^2 + T_2^2)\frac{v^2}{r} = \frac{1}{2}(T_1^2 + T_2^2)\cdot a \quad \text{.....(Equation 1)}$$

In actual machining, permissible error Δr is given, so the maximum permissible acceleration a (mm/sec^2) in equation 1 is determined.
Expression 1 is an expression when type of acceleration/deceleration after interpolation is exponential acceleration/deceleration. The radius error in the case of linear acceleration/deceleration after interpolation is smaller than in case of exponential acceleration/deceleration by a factor of 12, excluding any error caused by a servo motor time constant (T_2). The radius error in the case of bell-shaped acceleration/deceleration after interpolation is a half of linear acceleration/deceleration after interpolation.
When a specified feedrate causes the radial error from an arc having a programmed radius to exceed the permissible error, speed control with acceleration in circular interpolation automatically clamps the arc-cutting feedrate by using parameter settings.
Let the permissible acceleration calculated from the permissible acceleration set for each axis be A. Then, maximum permissible feedrate v with programmed radius r is expressed as follows:

FRYER MACHINE SYSTEMS INC
Rob Zahensky

$v = \sqrt{A \cdot r}$ .....(Equation 2)

If a specified feedrate exceeds feedrate v obtained from equation 2, the feedrate is clamped at feedrate v automatically.
The permissible acceleration is specified in parameter No. 1735. If there is a difference in permissible acceleration between two axes for circular interpolation, the lower acceleration is regarded as the permissible acceleration.
If the radius of an arc is small, too small value can be calculated as deceleration v. In such a case, the lower feedrate limit can be set in parameter No. 1732 to prevent the feedrate from being decreased too much.

- Speed control with the acceleration on each axis

When consecutive small lines are used to form a curve, as in the example shown in the figure below, the feedrate differences on each axis at the individual corners are not very large. Thus, deceleration with the feedrate differences is not effective. Consecutive small feedrate differences, however, cause a large acceleration on each axis, as a whole.
In such a case, deceleration can be performed to reduce the impact on the machine and the machining error caused by too large an acceleration. The deceleration feedrate is determined to be the feedrate that does not cause the acceleration on each axis to exceed the permissible acceleration set for parameter No. 1737.
The deceleration feedrate is determined for each corner. The actual feedrate is the smaller of the deceleration feedrate determined at the start point of the block and that determined at the end point.
Depending on the specified figure, a very low deceleration feedrate may be calculated. In such a case, the lower feedrate limit can be set in parameter No. 1738 to prevent the feedrate from being decreased too much.
In the following example, the acceleration (gradient of the broken line in the feedrate graph) at too large at corners N2 to N4 and N6 to N8 and, therefore, deceleration is performed.

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

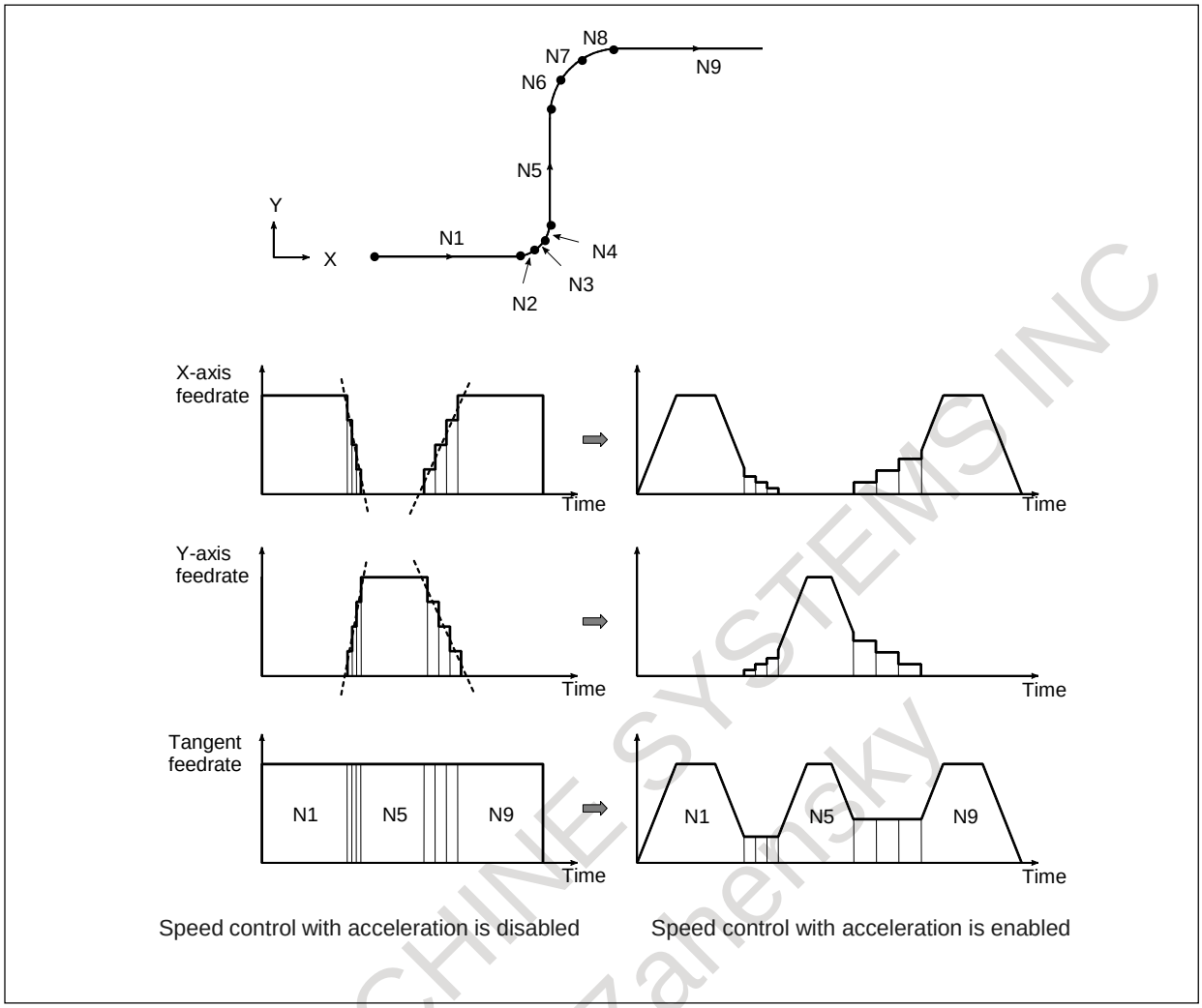


Fig.20.1 (m)

The method of determining the feedrate with the acceleration differs depending on the setting of bit 6 (FNW) of parameter No. 19500.
If parameter FNW is set to 0, the highest feedrate that does not cause the permissible acceleration set for parameter No. 1737 to be exceeded is assumed to be the deceleration feedrate. In this case, the deceleration feedrate differs depending on the travel direction even if the shape is the same, as shown in the figure below.

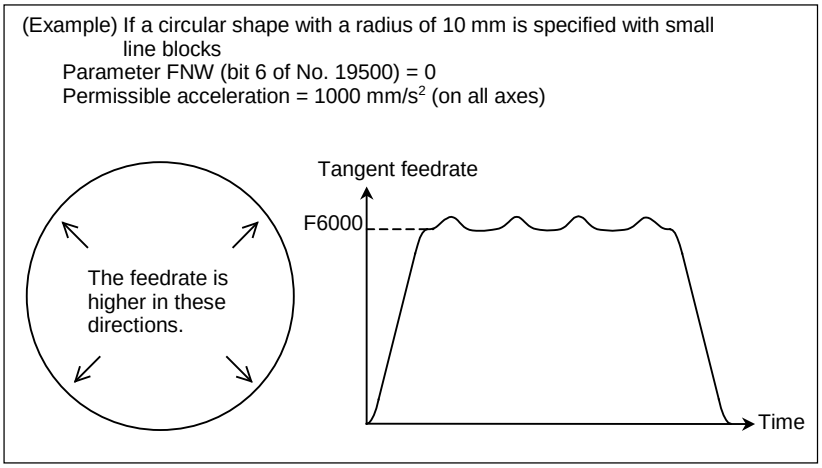


Fig.20.1 (n)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

If parameter FNW is set to 1, the feedrate is determined with not only the condition that the permissible acceleration on each axis is not exceeded but also the condition that the deceleration feedrate is constant regardless of the travel direction if the shape is the same.
If this parameter is set to 1, the deceleration feedrate determined with the feedrate difference or acceleration may be up to 30% lower than that determined if this parameter is set to 0.

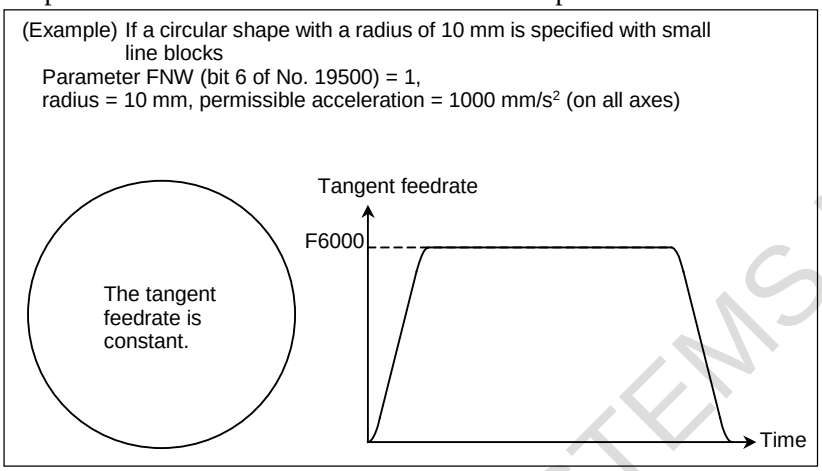


Fig.20.1 (o)

> **NOTE**
> In circular interpolation, the tangent feedrate is constant regardless of the setting of the parameter.

When this function is effective, the feedrate might be clamped as well as the travel distance in the block is small even if the travel distance in the block is long. Speed control with the acceleration is invalidated in the block longer than the block length set to parameter No. 19518 if bit 1 (HNG) of parameter No. 19517 is set to 1.

- Smooth speed control

In speed control with acceleration, the smooth speed control function recognizes the entire figure from preceding and following blocks including blocks read ahead to make a smooth feedrate determination.
When a curve is specified with successive minute straight lines, programmed values are rounded to the least input increment before issued, so the machining profile is approximated with a broken line.
When the feedrate is determined with acceleration in an ordinary manner, an optimum feedrate is automatically calculated exactly for a programmed figure, so a large acceleration may result depending on the command, which can lead to deceleration.
In such a case, the use of smooth speed control enables speed control by recognizing the entire figure, which provides smooth speed control while suppressing local deceleration, therefore increasing the feedrate.

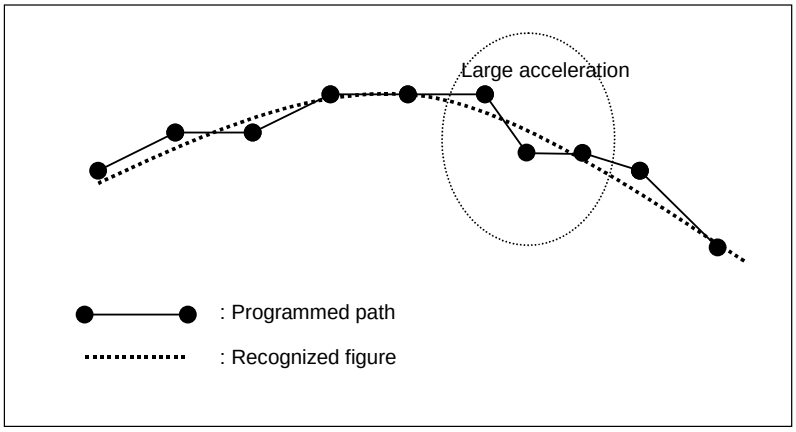


Fig.20.1 (p)

FRYER MACHINE SYSTEMS INC
Rob Zahansky

Also for a part of a programmed figure in which a large acceleration would be required, the acceleration is obtained based on the figure recognized from multiple blocks, and the feedrate is determined so that the acceleration is within the permissible acceleration set in parameter No. 1737.

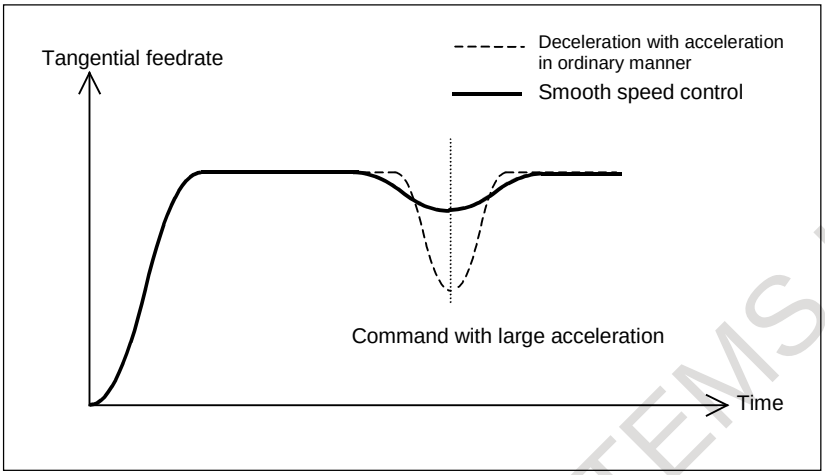


Fig.20.1 (q)

Smooth speed control obtains the acceleration by using the figure recognized from the preceding and following blocks including blocks read ahead, so smooth speed control is enabled even in parts in which the acceleration increases.
Smooth speed control is enabled under the following conditions:
(1) Speed control with acceleration is enabled in the AI contour control II mode.
(2) Successive linear interpolation commands are specified.
(3) Bit 0 (HPF) of parameter No. 19503 is set to 1.

> ⚠ **CAUTION**
> When smooth speed control is used, the feedrate in a certain figure such as a corner may become larger than the feedrate obtained by ordinary speed control with acceleration. For corners, set parameter No. 1783, which is the permissible feedrate difference parameter for speed control with the feedrate difference at corners, to perform appropriate deceleration by speed control with the corner feedrate difference.

When this function is effective, the feedrate might be clamped as well as the travel distance in the block is small even if the travel distance in the block is long. Smooth speed control is invalidated in the block longer than the travel distance set to parameter No. 19518 if bit 0 (SNG) of parameter No. 19517 is set to 1.

- Speed control with the cutting load

Usually, the cutting resistance produced when machining is performed with the bottom of the cutter as the tool lowers along the Z-axis is greater than the cutting resistance produced when machining is performed with the side of the cutter as the tool rises along the Z-axis. Therefore, deceleration is required.
In AI contour control II, the tool travel direction on the Z-axis is used as a condition for calculating the machining feedrate.
This function is enabled when bit 4 (ZAG) of parameter No. 8451 is set to 1.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

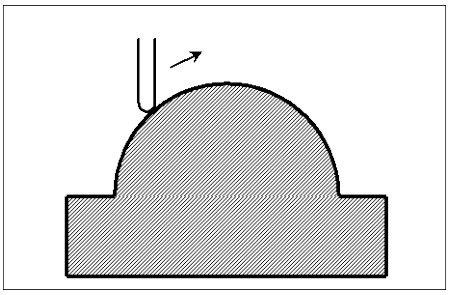


Fig.20.1 (r) During ascent on the Z-axis

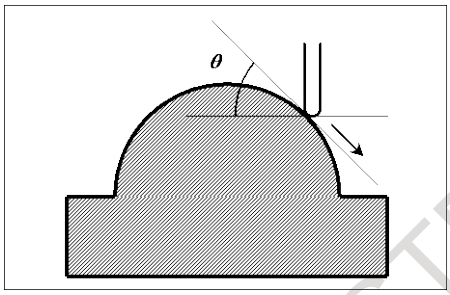


Fig.20.1 (s) During descent on the Z-axis

The descent angle θ during descent on the Z-axis (angle formed by the XY plane and the tool center path) is as shown in the figure. The descent angle is divided into four areas, and the override values for the individual areas are set for the following parameters:

Parameter No. 8456 for area 2
Parameter No. 8457 for area 3
Parameter No. 8458 for area 4

For area 1, however, no parameter is available, and an override of 100% is used at all times. The feedrate obtained according to other feedrate control is multiplied by the override value of the area to which descent angle θ belongs.

Area1 $0^\circ \leq \theta < 30^\circ$
Area2 $30^\circ \leq \theta < 45^\circ$
Area3 $45^\circ \leq \theta < 60^\circ$
Area4 $60^\circ \leq \theta < 90^\circ$

The feedrate can be overridden with an inclination by setting bit 1 (ZG2) of parameter No. 19515 to 1. In this case, specify the override value for area 1 in parameter No. 19516.

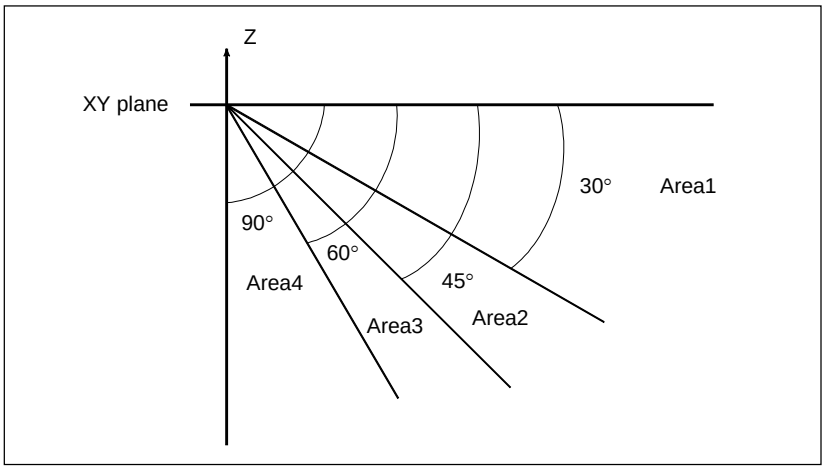


Fig.20.1 (t)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

⚠ CAUTION

1 The speed control with the cutting feed is effective only when the tool is parallel with the Z-axis. Thus, it may not be possible to apply this function, depending on the structure of the machine used.
2 In the speed control with the cutting feed, the travel direction on the Z-axis is determined with the appropriate NC command. If, therefore, manual intervention is performed on the Z-axis with manual absolute on, or if a mirror image is applied on the Z-axis, the direction on the Z-axis cannot be determined. When using the speed control with the cutting load, do not use these functions.
3 When performing 3-dimensional coordinate conversion, determine the descent angle on the Z-axis using the converted coordinate system.
4 Speed control with the cutting load is enabled for all interpolations in the AI contour control II mode. This function, however, can be made valid only for linear interpolations by setting bit 4 (ZOL) of parameter No. 19503 to 1.

- Ignoring feedrate commands

In a block in which AI contour control II is enabled, all feedrate commands (F commands) can be ignored by setting bit 7 (NOF) of parameter No. 8451.
The term feedrate commands, as used here, refer to the following commands:

(1) Modal F commands before the block in which AI contour control II is enabled
(2) F commands and modal F commands in the block in which AI contour control II is enabled

When the feedrate commands are ignored, it is assumed that the upper feedrate limit specified for parameter No. 8465 is specified.
Note, however, that any issued F commands and modal F commands are stored within the CNC.
Thus, in a block in which AI contour control II changes from the enabled state to the disabled state, the modal values of the F commands described in (1) and (2) described above are used as modal F commands, instead of the modal values of the F commands calculated by AI contour control II.

- Another example of determining the feedrate

If a specified feedrate exceeds the upper feedrate limit of AI contour control (in parameter No. 8465), the feedrate is clamped at the upper feedrate. The upper feedrate limit is clamped at the maximum cutting feedrate (parameter No. 1432) for acceleration/deceleration before interpolation..

Limitation

- Conditions for temporarily canceling the AI contour control mode

If one of the commands listed below is issued in the AI contour control mode, the AI contour control mode is canceled temporarily.
If the system becomes ready for AI contour control after it is canceled, the AI contour control mode is restored automatically.

- Positioning (rapid traverse)
- Single direction positioning
- Spindle positioning
- Rigid tapping
- Threading (single type, combined type)
- Electronic gear box
- Superimposed control
- When no move command is specified
- One-shot G code other than the following:
  - Tool offset
  - Cutter compensation vector retention
  - Cutter compensation corner rounding
  - Exact stop

FRYER MACHINE SYSTEMS INC Ron Zaremsky

Example

```
O0010
…
G5.1 Q1;
G01                 AI contour control mode
…
X1.Y2.Z3.;
M220;   Start superimposed control
…                                   AI contour control mode is
M221;   Cancel superimposed control temporarily canceled.
X2.Y2.Z4.;  Automatically return the system to AI contour control mode
…           AI contour control mode
X4.Y1.Z2.;
G5.1 Q0;
…
M30;
```

(**Note** The way to specify synchronous, composite, and superimposed controls differ from one machine tool builder to another. For details, refer to the manual issued by the machine tool builder.)

- Functions that cannot be specified in the AI contour control mode

In the AI contour control mode, the functions listed below cannot be specified. Alarm PS5110 “IMPROPER G-CODE (AICC MODE)” occurs. Before specifying these functions, turn off the AI contour control mode; after the command ends, turn on the mode again.

- Threading
- Circular threading
- Variable lead threading

Threading, circular threading, and variable lead threading can be specified in the AI contour control mode by setting bit 1 (THA) of parameter No. 1611. However, the AI contour control mode is automatically canceled.

- Modal G codes usable when AI contour control is specified

In a modal G code state listed below, AI contour control can be specified.
Do not specify AI contour control in modal states other than these.

G13.1	:	Polar coordinate interpolation mode cancel
G15	:	Polar coordinate command cancel
G40	:	Tool radius compensation cancel
G40.1	:	Normal direction control cancel
G50	:	Scaling cancel
G50.1	:	Programmable mirror image cancel
G50.2	:	Polygon turning cancel
G64	:	Cutting mode cancel
G67	:	Macro modal call cancel
G69	:	Coordinate system rotation/3-dimensional coordinate system conversion cancel
G80	:	Canned cycle cancel
G97	:	Constant surface speed control cancel

FRYER MACHINE SYSTEMS INC Rob Zahensky

- Retrace

When the reverse execution is started in AI contour control mode, a reverse execution ends immediately depending on a program then the backward movement is not possible.
During reverse execution and forward reexecution, the feedrate clamp function by acceleration under AI contour control is disabled.

- Manual handle interrupt

Manual handle interrupt is available in AI contour control mode.

- DNC operation

AI contour control is available during DNC operation.

Notes

- About processing macro statements

In AI contour control mode, the NC statements of multiple blocks are looked ahead. Macro statements such as arithmetic expressions and conditional branches are processed as soon as they are read into the buffer. Therefore, the timing of the macro statement execution is not always the specified order.
In case that you need to execute the macro statement after completing the NC block just before the macro statement, specify M code or G code that is not buffered just before the macro statement. Specially, in case of reading/writing the system variables to control signals, coordinates, offset value, etc., it may be different system variable data by the timing of the NC statement execution. To avoid this phenomenon, specify such M codes or G codes before the macro statement, if necessary.

- Setting of acceleration/deceleration, and feedrate

In the mode of look-ahead acceleration/deceleration before interpolation, when a time required for acceleration or deceleration is over one second, the feedrate could not reach the specified data.
When bit 7 (FCC) of parameter No.19500 is set to 1, the feedrate can reach the specified data.
In this case, however, the interpolation precision may be lower in curve interpolation such as circular interpolation.

20.2 MACHINING CONDITION SELECTING FUNCTION

Overview

By setting a speed- or precision-focused parameter set in an AI contour control function and specifying a precision level in accordance with the machining conditions during machining, parameters suitable to the conditions can be automatically calculated so that machining can be performed.

Format

- Changing the precision level using a program

In addition to being switched on the precision level selection screen, the precision level can be changed using a program in the format below.

```
G05.1 Q1 Rx ;
  x.......Level (1 to 10)
```

The precision level can also be changed in the format used with conventional "advanced preview control", "high precision contour control", and "AI high precision contour control".

```
G05 P10000 Rx ;
  x.......Level (1 to 10)
G08 P1 Rx ;
  x.......Level (1 to 10)
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

> ⚠ **CAUTION**
> Once specified, a level remains effective even if the AI contour control mode is canceled.

20.3 OPTIMUM ACCELERATION/DECELERATION FOR RIGID TAPPING

Overview

This function can be used to flexibly set the acceleration/deceleration during cutting in rigid tapping according to the torque characteristics of a spindle motor and the mechanical characteristics such as machine friction. Depending on the torque characteristics of a spindle motor and the mechanical characteristics, the acceleration/deceleration performance (referred to below as the maximum acceleration curve) that can be output is not symmetrical in the low-speed and high-speed parts. The conventional acceleration/deceleration (linear-shaped/bell-shaped) is symmetrical, so it was impossible to make the most of the motor performance.

This function can be used to perform acceleration/deceleration so that the actual speed curve can follow the maximum acceleration curve as close as possible. This can make the most of the motor performance and reduce the cutting time. When rigid tapping is used in an area where the acceleration of the spindle motor is constant, however, reduction in the cutting time cannot be expected.

If the acceleration pattern is set for the parameter for each gear, rigid tapping by acceleration following the maximum acceleration curve.

In acceleration/deceleration during withdrawal, the acceleration/deceleration pattern during cutting is used.

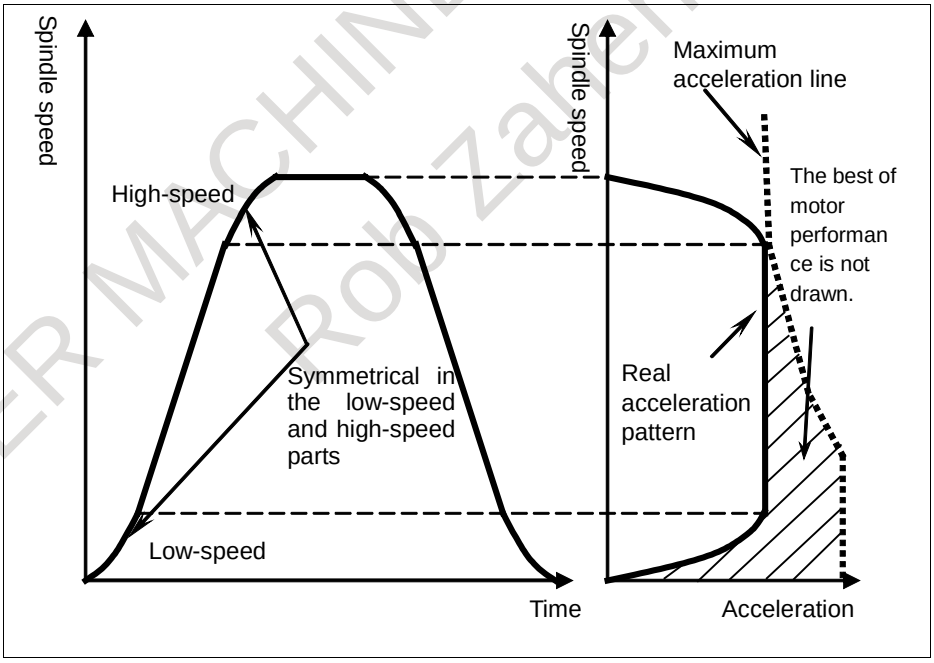


Fig. 20.3 (a) Conventional acceleration/deceleration (bell-shaped)

FRYER MACHINE SYSTEMS INC
Rob Zabensky

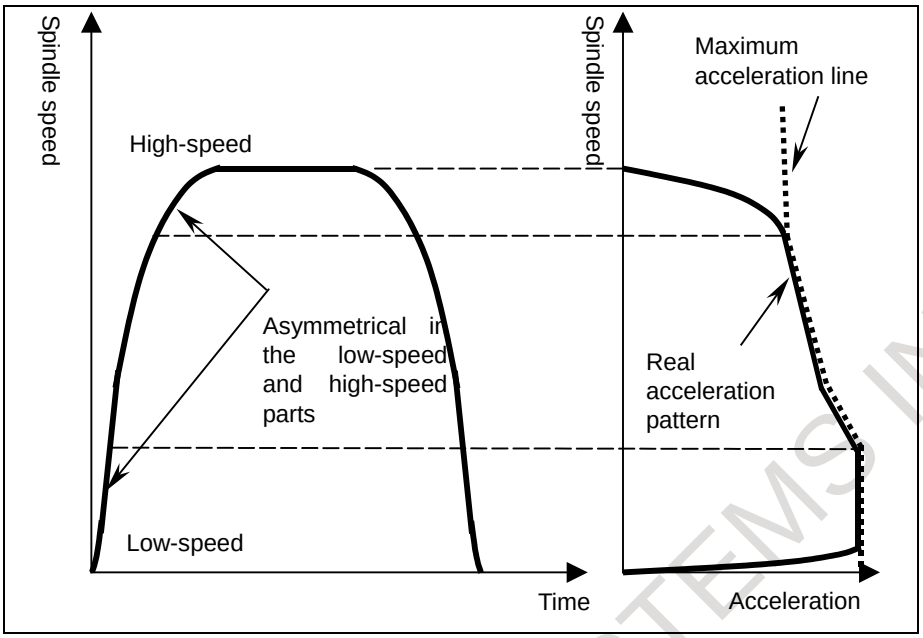


Fig. 20.3 (b) Acceleration/deceleration in which the maximum acceleration curve is followed by the actual acceleration curve by this function

20.4 G CODE PREVENTING BUFFERING

Overview

By specifying G04.1, buffering of the following block from the block of G04.1 is prevented until finishing the block of G04.1. (In this chapter, preventing buffering by G04.1 is called non-buffering command by G code. On the other hand, preventing buffering by M codes, command of only G31, command of only G53 and etc. are called non-buffering commands of normal.)

The processing time of non-buffering command by G code reduce compared with non-buffering commands of normal. Also, the position of releasing preventing buffering can be specified by the distance to go in the previous movement block. Therefore, the cycle time of automatic operation can be reduced by using non-buffering command by G code instead of non-buffering commands of normal.

Format

G04.1 (P_) ;

P_ : The operation mode of G04.1.
Compatible operation to preventing buffering by command of only G31 or G53 is possible when P1 or P2 is specified with G04.1 in the same block.
P1: Compatible operation to preventing buffering by command of only G31.
P2: Compatible operation to preventing buffering by command of only G53.

However, alarm PS2085 occurs when P code except P1 and P2 is specified.

I_ : Distance to go for releasing preventing buffering (Synthetic distance to go in all axis)

However, alarm (PS2085) occurs when the address I is specified in the state of setting the bit 5 (MRR) of parameter No.11279 to 0.

FRYER MACHINE SYSTEMS INC
Rob Zahonsky

NOTE

1 When P1 is specified, the system variables (#100151 to #100182) of skip position are updated by executed position of the block that includes G04.1. (When the system up to 20 axes, the system variables (#5061 to #5080) compatible with FS16 is uploaded too.)
2 About preventing buffering by M codes or command of only G31 or command of only G53, those behaviors are different respectively. When each preventing buffering is exchanged for G04.1, it is necessary to use the compatible operation mode.
3 Unit system as for the address I depends on the reference axis.
4 The address I is commanded by radius value regardless of diameter/radius specification of the reference axis.
5 Positive value should be specified as for the address I. If negative value is specified, the command of the address I is invalid.
6 The command of address I is invalid when the previous block of G04.1 is not the movement command.

Suppressing single block stop

Single block stop is not performed in G04.1 block when bit 6 (MSB) of parameter No.11279 is set to 1. In this case, the function of preventing buffering in G04.1 block is not invalid.

NOTE

Single block stop is not performed in G04.1 block regardless of the state of #3003 when bit 6 (MSB) of parameter No.11279 is set to 1.

Suppressing single block stop in subprogram

It is necessary that non-buffering command is specified just before system variable #3003 when system variable #3003 is used for suppressing single block stop.
When non-buffering M codes that are set by parameter No.3411 are used as non-buffering command, single block stop is performed in this block. (Example of program 1)
However, when G code preventing buffering is used as non-buffering command and bit 6 (MSB) of parameter No.11279 is set to 1, single block stop is not performed in this block. (Example of program 2)
Therefore, subprogram can be executed as well as the command of a block by specifying cycle start command once.

Example of program 1)

Main program	Subprogram	Single block stop
O0001		(Note 1)
:		Performing
:		Performing
M98P1000	O1000	(Note 1)
	M100 -----Preventing buffering of system variable #3003	Performing
	#3003=1 -----Suppressing single block stop is valid	
	:	
	:	
	M100 -----Preventing buffering of system variable #3003	
	#3003=0 -----Suppressing single block stop is invalid	
	M99	(Note 2)
:		Performing
:		Performing
M30		Performing

FDYER MACHINE SYSTEMS INC
Rob Zahensky

Example of program 2)

Main program	Subprogram	Single block stop
O0001		(Note 1)
:		Performing
:		Performing
M98P1000	O1000	(Note 1)
	G04.1 -----Preventing buffering of system variable #3003	
	#3003=1 -----Suppressing single block stop is valid	
	:	
	:	
	G04.1 -----Preventing buffering of system variable #3003	
	#3003=0 -----Suppressing single block stop is invalid	
	M99	(Note 2)
:		Performing
:		Performing
M30		Performing

NOTE

1 When bit 0 (NOB) of parameter No.3404 is set to 0, single block stop is performed in “O0001” and “O1000” block. Set bit 0 (NOB) of parameter No.3404 to 1 if it is necessary to execute subprogram by specifying cycle start command once during program operation.

2 When bit 0 (M99) of parameter No.11648 is set to 0, next block of “M98P1000” block is executed as soon as finishing subprogram because single block stop is not performed in “M99” block of subprogram. Set bit 0 (M99) of parameter No.11648 to 1 if it is necessary to perform single block stop in “M99” block during single block operation.

20.5 Speeding up of rigid tapping

Overview

Cycle time of rigid tapping can be reduced with setting the following parameter.

- Parameter HRC (No.5208#5) = 1

The start-up time for cycle operation after point R is reduced.

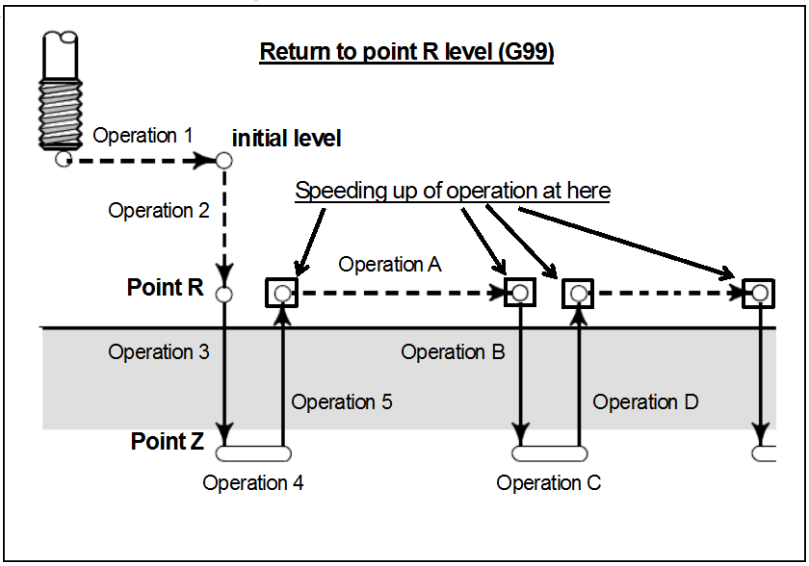


Fig 20.5 (a) Overview of operation

FRYER MACHINE SYSTEMS INC Rob Zahensky

Explanation

When parameter HRC (No.5208#5) is 1, rigid tapping cycle is speeded up because the starting time after point R is reduced.

NOTE

1 When a tapping axis is moved at both before and after point R, the tapping cycle is not speeded up.
2 This speeding up is enabled in both machining center system and lathe system. However, a rigid tapping with G code system A in lathe system is operated on mode of returning to initial level at any time. The tapping axis is moved at both before and after point R. Accordingly, this speeding up is disabled regardless of setting of parameter HRC (No.5208#5).
3 The operation of first tapping is not speeded up regardless of setting of parameter HRC (No.5208#5).

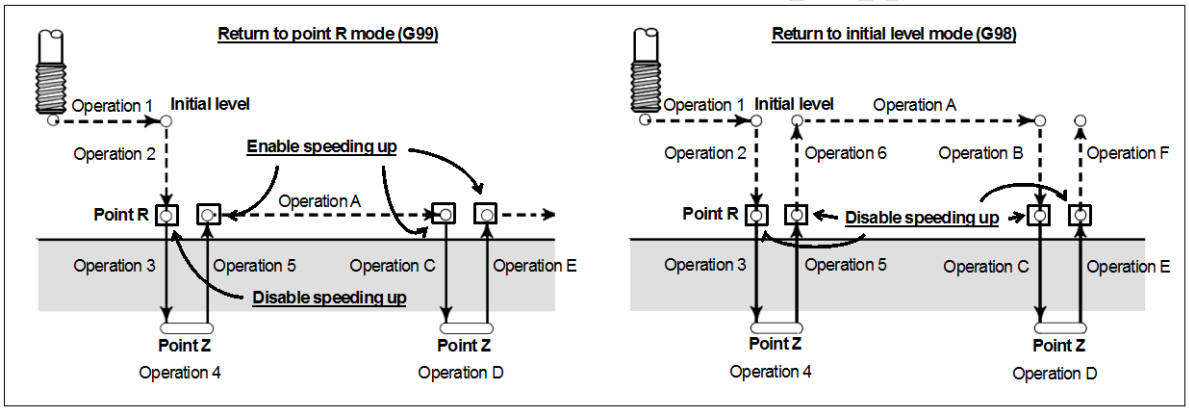


Fig 20.5 (b) Return to point R level (G99) **Fig 20.5 (c) Return to initial level (G98)**

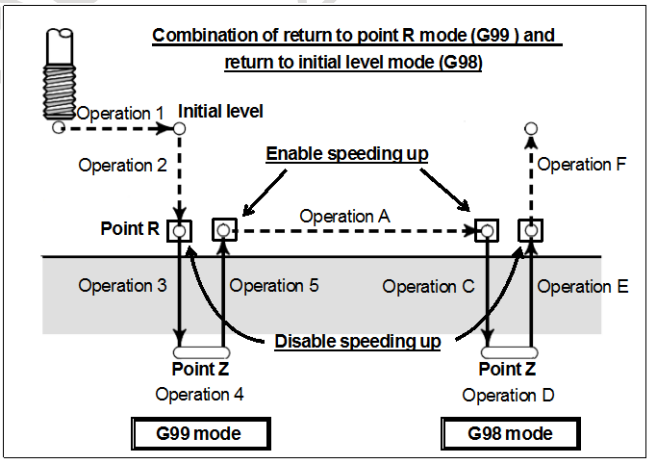


Fig 20.5 (d) combination of return to point R level (G99) and initial level (G98)

Peck rigid tapping

In peck tapping cycle (parameter PCP(No.5200#5)=1) and high-speed peck tapping cycle (parameter PCP (No.5200#5)=0), this speeding up is enabled. The operation of the first tapping from point R is speeded up. Since then, the operation during reaching to point Z is not speeded up.

Restriction

1. This speeding up is enabled when the advance preview feed forward is enabled. Please set 1 a parameter REF (No.5203#2) that feed forward is enabled in rigid tapping.
2. When a tapping axis is moved at both before and after point of speeding up, this speeding up is disabled.
   Example)
   - Return to initial level mode (G98)
3. In G code system A for lathe system, the return point level for rigid tapping is the initial level (G98) always. Therefore this speeding up is disabled regardless of the setting of parameter HRC (No.5208#5).
4. When a tapping axis is perpendicular axis for arbitrary angular axis control, this speeding up is disabled.
5. When there is an axis which is under superimposed control in a path of machining with rigid tapping, this speeding up is disabled.
6. When the multiple axes or the axis which is not specified by programming are moved as a tapping axis, this speeding up is disabled.
   Example)
   - Rigid tapping in 3-dimensional coordinate system conversion.
   - Rigid tapping in tilted working plane indexing.

   But the following functions are an exception. (this speeding up is enabled.)
   - Synchronous control
   - Composite control
   - Axis synchronous control
   - Tandem control

FRYER MACHINE SYSTEMS INC
Rob Zahensky

21 AXIS CONTROL FUNCTIONS

21.1 AXIS SYNCHRONOUS CONTROL

When a movement is made along one axis by using multiple servo motors as in the case of a large gantry machine, a command for one axis can drive the multiple motors by synchronizing one motor with the other. An axis used as the reference for axis synchronous control is called a master axis (M-axis), and an axis along which a movement is made in synchronism with the master axis is called a slave axis (S-axis).

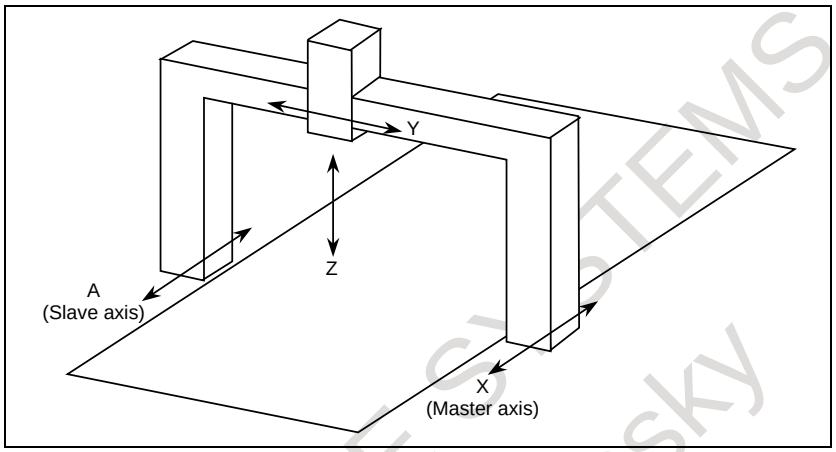


Fig. 21.1 Example of machine with X and A being synchronous axes

Operation where axis synchronous control is turned on (enabled) to make a movement along the slave axis in synchronism with the master axis is called a synchronous operation. Operation where axis synchronous control is turned off (disabled) to move a master axis and a slave axis independently is called a normal operation.

(Example)

Automatic operation when the master axis is the X-axis and the slave axis is the A-axis:

In synchronous operation, both the X-axis and the A-axis are moved according to the programmed command Xxxxx for the master axis.

In normal operation, the master axis and the slave axis are moved independently as in the case of normal CNC control. The programmed command Xxxxx makes a movement along the X-axis. The programmed command Aaaaa makes a movement along the A-axis. The programmed command Xxxxx Aaaaa makes movements along the X-axis and A-axis at the same time.

For details, refer to the relevant manual published by the machine tool builder.

FRIEDMACHINE SYSTEMS INC
Rob Zaremsky

Note

NOTE

1 During axis synchronous control, a movement based on the reference position return check (G27), automatic reference position return (G28), 2nd/3rd/4th reference position return (G30), or machine coordinate system selection (G53) command is made as described below according to the setting of bit 2 (SRF) of parameter No. 8305.
   <1> When SRF = 0, the same movement as made along the master axis is made along the slave axis.
   <2> When SRF = 1, a movement is made along the slave axis to the specified position independently of a movement made along the master axis to the specified position.

2 A command not involving a movement along an axis such as the workpiece coordinate system setting command (G92) and local coordinate system setting command (G52) is set with the master axis according to the master axis programming.

3 When switching the synchronization state in a program, be sure to specify M codes (parameters Nos. 8337 and 8338) for turning synchronization on and off. By switching between the input signals SYNC <Gn138> and SYNCJ <Gn140> from the PMC with the M codes, the synchronization state can be switched in the program.

4 If the following errors occurred during axis synchronous control, an alarm PS0213 is issued.
   <1> If the program issued the move command to the slave axis.
   <2> If the manual operation (jog feed or incremental feed) is performed to the slave axis.
   <3> If the program issued the automatic reference position return command without specifying the manual reference position return after the power was turned on.

5 A programmed command can be specified for the slave axis when switching between synchronous operation and normal operation is enabled (with bit 5 (SCAx) of parameter No. 8304 set to 0) and the signal SYNC/SYNCJ select normal operation.

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

21.2 POLYGON TURNING (G50.2, G51.2, G250, G251)

T

Polygon turning means machining a workpiece to a polygonal figure by rotating the workpiece and tool at a certain ratio.

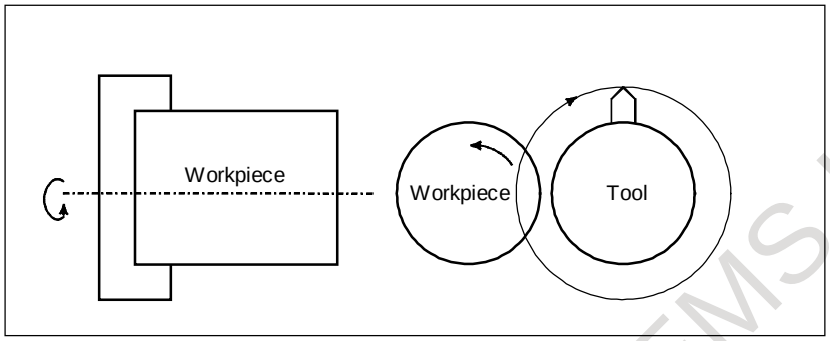


Fig. 21.2 (a) Polygon turning

By changing conditions which are rotation ratio of workpiece and tool and number of cutters, the workpiece can be machined to a square or hexagon. The machining time can be reduced as compared with polygonal figure machining using the polar coordinate interpolation. The machined figure, however, is not exactly polygonal. Generally, polygon turning is used for the heads of square and/or hexagon bolts or hexagon nuts.

As the tool rotary axis, one of the following can be used:

- CNC controlled axis (servo axis)
- Second spindle (Two serial spindles are connected.)

Polygonal machining performed using a servo axis as the tool rotary axis is referred to as polygon turning. Polygonal machining performed using the second spindle as the tool rotary axis is referred to as polygon turning with two spindles.

Function name	Workpiece axis	Tool rotary axis
Polygon turning	Spindle (Either an analog spindle or serial spindle is usable. However, a detector equivalent to a position coder is required.)	Servo axis
Polygon turning with two spindles	Spindle (Serial spindle)	Spindle (Serial spindle)

Explanation

A CNC controlled axis (servo axis) is assigned to the tool rotary axis.

This rotary axis of tool is called Y-axis in the following description. As the workpiece axis (spindle), either a serial spindle or analog spindle can be used.

The Y-axis is controlled by the G51.2 command, so that the ratio of the rotation speeds of the spindle (previously specified by S-command) and the tool becomes the specified ratio.

When simultaneous start is specified by G51.2, the one-rotation signal sent from the position codes set on the spindle is detected. After one-rotation signal detection, the Y-axis is controlled using the rotation ratio of the spindle and Y-axis specified by P and Q. So, a position coder needs to be attached to the spindle. This control will be maintained until the polygon turning cancel command is executed (G50.2). Polygon turning is cancelled by any of the following in addition to the G50.2 command:

(1) Power off
(2) Emergency stop
(3) Servo alarm

FRYER MACHINE SYSTEMS INC
Rob Zahensky

(4) Reset (external reset signal ERS, reset/rewind signal RRW, and RESET key on the MDI unit)
In polygon turning with two spindles, setting bit 0 (RPL) of parameter No.7603 to 1 prevents polygon synchronization mode from being released by reset.

(5) Occurrence of alarms PS0217 to PS0221, PS0314, and PS5018

NOTE

1 Before polygon turning, reference position return operation on the Y-axis needs to be specified to determine the rotation start position of the tool. This reference position return operation is performed by detecting a deceleration limit as in the case of manual reference position return operation. (By setting bit 7 (PLZ) of parameter No. 7600, reference position return operation can be performed without detecting a deceleration limit.)

2 The rotation direction on the Y-axis is determined by the sign of Q, and is not affected by the rotation direction of the position coder.

3 Among Y-axis position indications, the indication of a machine coordinate (MACHINE) changes within the amount of movement for 0 to 1 revolution as a movement is made on the Y-axis. The absolute coordinate and relative coordinate are not updated. So, when specifying an absolute-position command for the Y-axis after polygon turning mode cancellation, set a workpiece coordinate system after reference position return operation.

4 For the Y-axis engaged in polygon turning, jog feed and handle feed are disabled.

5 For the Y-axis not engaged in polygon turning, a move command can be specified as in the case of other controlled axes.

6 The Y-axis engaged in polygon turning is not counted in the number of simultaneously controlled axes.

7 One workpiece must be machined using a fixed spindle speed until the workpiece is finished.

8 G50.2 is the G code for suppressing buffering.

9 The following functions must be commanded with polygon turning cancel mode.
- Inch/metric conversion
- AI contour control
- Tilted working plane indexing
- Flexible path axis assignment

⚠ CAUTION

1 During polygon turning, threading cannot be performed.

2 For the Y-axis engaged in polygon turning, the signals below are valid or invalid:
Signals valid for the Y-axis
- Machine lock
- Servo-off

Signals invalid for the Y-axis
- Feed hold
- Interlock
- Override
- Dry run
(In the dry run, workpiece might not be processed to the expected shape because the one-rotation signal is not awaited though the speed of a dry run is invalid.)

FREEP MACHINE SYSTEMS INC
Rob Zahensky

Format

G50.2 (G250)	Polygon turning cancel

G51.2 (G251) P_ Q_ [R_] ;

P,Q: Rotation ratio of spindle and Y-axis
R : Relative phase difference between spindle and Y-axis (*1)
Specify range:
P: Integer from 1 to 999
Q: Integer from -999 to -1 or from 1 to 999
When Q is a positive value, Y-axis makes positive rotation.
When Q is a negative value, Y-axis makes negative rotation.
R: 0 to 360 deg
The unit of data is depend on the increment system of the reference axis.
The decimal point can be used.
If a value out of range is specified, it is rounded in 0 to 360 degrees.

(*1) R can be specified in polygon turning with two spindles.
The least command increment of R is 360/4096 degrees, and the value less than the least command increment is rounded off.
R may be omitted. If R is not specified after starting polygon turning with two spindles, the phase difference is assumed to be 0. Even if R is not specified, phase adjustment is performed.
If bit 5 (COF) of parameter No. 7602 is 1 to disable phase control, R is ignored.

NOTE

1. Specify G50.2 and G51.2 in a single block. If G51.2 and 50.2 are specified in the same block for other commands, alarm PS0219 "COMMAND G51.2/G INDEPENDENTLY" is issued.
2. If P or Q is not commanded in the G51.2 block, or the command value is out of the range, alarm PS0218 "NOT FOUND P/Q COMMAND" is issued. Please rewrite the parameter related to polygon turning during G50.2 (polygon turning mode). Then, diagnosis data No. 0471#6 is 1.
3. Please rewrite the parameter related to polygon turning during G50.2 (polygon turning mode cancel).

Example

G00 X100. 0 Z20.0 S1000 M03 ;	(Workpiece rotation speed 1000 (min^{-1}))
G51.2 P1 Q2 ;	(Tool rotation start (tool rotation speed 2000 (min^{-1}))
G01 X80.0 F10.0 ;	(X-axis infeed)
G04 X2.0 ;	(Waiting 2 seconds)
G00 X100.0 ;	(X-axis escape)
G50.2 ;	(Tool rotation stop)
M05 S0 ;	(Spindle stop)

- Principle of polygon turning

The principle of polygon turning is explained below. In the figure (Fig. 21.2 (b)) below the radius of tool and workpiece are A and B, and the angular speeds of tool and workpiece are α and β. The origin of XY Cartesian coordinates is assumed to be the center of the workpiece.
Simplifying the explanation, consider that the tool center exists at the position
P_o (A, 0) on the workpiece periphery, and the tool nose starts from position P_{to} (A-B, 0).

FRYER MACHINE SYSTEMS INC Rob Zahensky

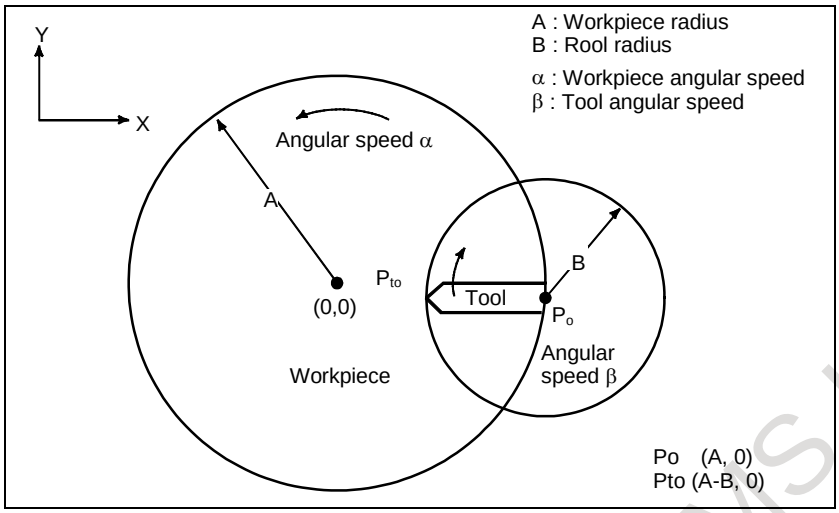


Fig. 21.2 (b) Principle of polygon turning

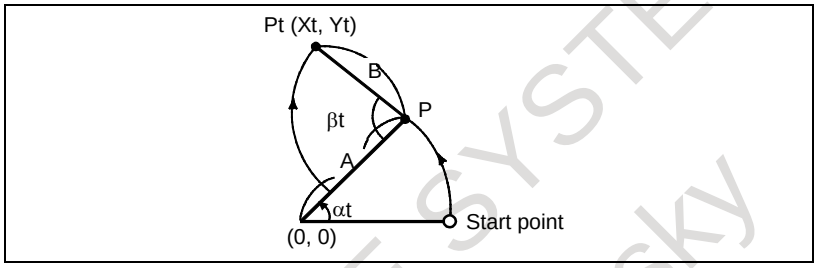


Fig. 21.2 (c) Tool nose position

In this case, the tool nose position Pt (Xt, Yt) after time t is expressed by equations 1 and 2:

$Xt=A\cos\alpha t-B\cos(\beta-\alpha)t$ (Equation 1)

$Yt=A\sin\alpha t+B\sin(\beta-\alpha)t$ (Equation 2)

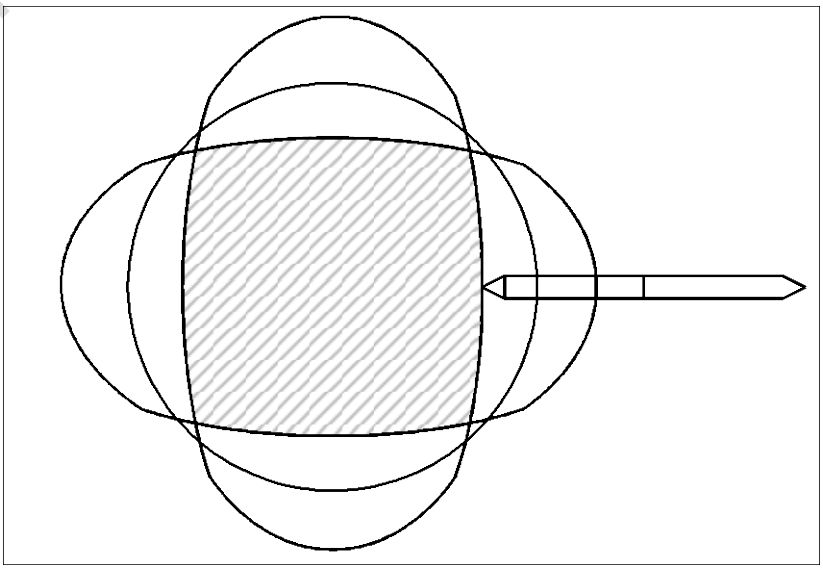
Assuming that the rotation ratio of workpiece to tool is 1:2, namely, β=2α, equations 1 and 2 are modified as follows:

$Xt=A\cos\alpha t-B\cos\alpha t=(A-B)\cos\alpha t$ (Equation 1)'

$Xt=A\sin\alpha t+B\sin\alpha t=(A+B)\sin\alpha t$ (Equation 2)'

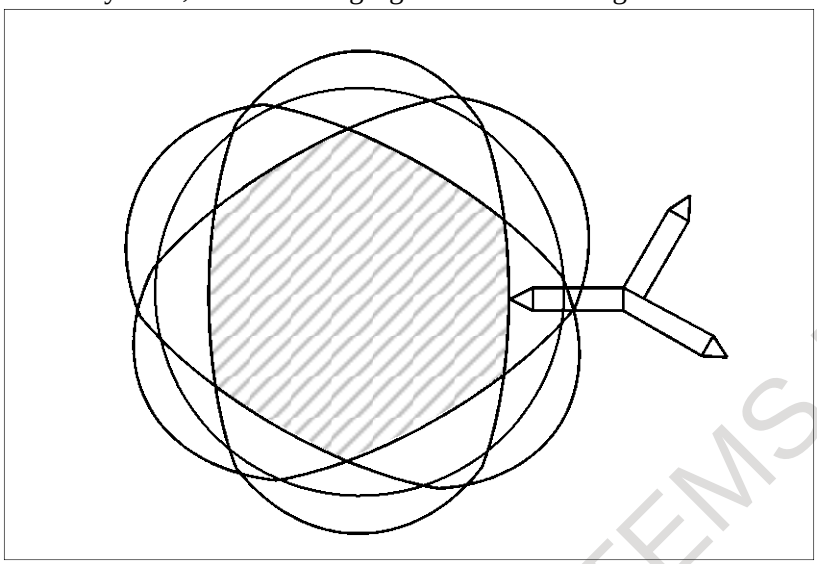
These equations indicate that the tool nose path draws an ellipse with longer diameter A+B and shorter diameter A-B.

Then consider the case when one tool is set at 180° symmetrical positions, for a total of two. A square can be machined with these tools as shown below.



FRYER MACHINE SYSTEMS INC
Rob Zahensky

If three tools are set at every 120°, the machining figure will be a hexagon as shown below.



⚠ **WARNING**
For the maximum rotation speed of the tool, see the instruction manual supplied with the machine. Do not specify a spindle speed higher than the maximum tool speed or a ratio to the spindle speed that results in a speed higher than the maximum tool speed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

21.3 SYNCHRONOUS, COMPOSITE AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND (G50.4, G51.4, G50.5, G51.5, G50.6, AND G51.6)

Synchronous control, composite control, and superimposed control can be started or canceled using a program command instead of a DI signal. Synchronous control, composite control, and superimposed control based on a DI signal is also possible.

For the basic operations of synchronous control, composite control, and superimposed control, see Sections, "SYNCHRONOUS CONTROL AND COMPOSITE CONTROL" and "SUPERIMPOSED CONTROL" in the CONNECTION MANUAL (FUNCTION).

Format

G51.4 P_ Q_ (L_) ; Synchronous control start (L is omissible.)
G50.4 Q_ ; Synchronous control cancel

P: Synchronous master axis ID number
Q: Synchronous slave axis ID number

L: Parking command
1: Master parking (slave parking cancel)
2: Slave parking (master parking cancel)
0: No parking (parking cancel)
(When L is omitted, the specification of L0 is assumed.)

G51.5 P_ Q_ ; Composite control start
G50.5 P_ Q_ ; Composite control cancel

P: Composite axis 1 ID number
Q: Composite axis 2 ID number

G51.6 P_ Q_ ; Superimposed control start
G50.6 Q_ ; Superimposed control cancel

P: Superimposed master axis ID number
Q: Superimposed slave axis ID number

As an ID number, set a unique value for identifying each axis in parameter No. 12600 for both of P and Q.

NOTE
1 These commands are one-shot G code.
2 These commands must be specified in a single block.
3 When P,Q or L command is out of range, the alarm PS5339"ILLEGAL FORMAT COMMAND IS EXECUTED IN SYNC/MIX/OVL CONTROL." occurs.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

Synchronous control

Synchronous control is performed with the G51.4/G50.4 commands, instead of simultaneously controlled axis selection signals.

Parameter setting examples for a 3-path system

- Parameter No.12600

	Path 1	Path 2	Path 3
X	101	201	301
Z	102	202	302

- Parameter No.8180

	Path 1	Path 2	Path 3
X	0	0	0
Z	0	102	102

- Program example (M100 to M103 are synchronization M codes.)

Path 1	Path 2	Path 3	Operation
N10 M100 P123 ;	M100 P123 ;	M100 P123 ;	Synchronization between paths 1, 2, and 3
N20 G51.4 P102 Q202 ; N30 G51.4 P102 Q302 ;			Start of Z1-Z2 synchronous control Start of Z1-Z3 synchronous control
N40 M101 P123 ;	M101 P123 ;	M101 P123 ;	Synchronization between paths 1, 2, and 3
N50 G00 Z100.;	Synchronous slave movement	Synchronous slave movement	Z1-Z2 and Z1-Z3 synchronous control
N60 M102 P123 ;	M102 P123 ;	M102 P123 ;	Synchronization between paths 1, 2, and 3
N70 G50.4 Q202 ; N80 G50.4 Q302 ;			Cancellation of Z1-Z2 synchronous control Cancellation of Z1-Z3 synchronous control
N90 M103 P123 ;	M103 P123 ;	M103 P123 ;	Synchronization between paths 1, 2, and 3

- Start of synchronous control

N20 G51.4 P102 Q202 :

Start of synchronous control with the Z1-axis being a master axis and the Z2-axis being a slave axis

N30 G51.4 P102 Q302 :

Start of synchronous control with the Z1-axis being a master axis and the Z3-axis being a slave axis

- Cancellation of synchronous control

N70 G50.4 Q202 :

Cancellation of synchronous control with the Z1-axis being a master axis and the Z2-axis being a slave axis

N80 G50.4 Q302 :

Cancellation of synchronous control with the Z1-axis being a master axis and the Z3-axis being a slave axis

- Parking

Use the L command to specify the start and cancellation of the parking of synchronous axes.
If the L command is omitted or if the L0 command is issued, the parking of both synchronous master axis and synchronous slave axis is canceled.

- Parameter check

If the axis number corresponding to the P specified with G51.4 is not set in slave axis parameter No. 8180, alarm PS5339, “ILLEGAL FORMAT COMMAND IS EXECUTED IN SYNC/MIX/OVL CONTROL.” is issued.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Composite control

Composite control is performed with the G51.5/G50.5 commands, instead of composite control axis selection signals.

Parameter setting examples for a 3-path system

- Parameter No.12600

	Path 1	Path 2	Path 3
X	101	201	301
Z	102	202	302

- Parameter No.8183

	Path 1	Path 2	Path 3
X	0	101	0
Z	0	102	0

- Program example (M100 to M103 are synchronization M codes.)

Path 1	Path 2	Path 3	Operation
N10 M100 P12 ;	M100 P12 ;	Independent operation	Synchronization between paths 1 and 2
N20 G51.5 P101 Q201 ; N30 G51.5 P102 Q202 ;			Start of X1-X2 composite control Start of Z1-Z2 composite control
N40 M101 P12 ;	M101 P12 ;		Synchronization between paths 1 and 2
N50 G00 X 100. Z100.;	Composite movement		X1-X2 and Z1-Z2 composite control
N60 M102 P12 ;	M102 P12 ;		Synchronization between paths 1 and 2
N70 G50.5 P101 Q201 ; N80 G50.5 P102 Q202 ;			Cancellation of X1-X2 composite control Cancellation of Z1-Z2 composite control
N90 M103 P12 ;	M103 P12 ;		Synchronization between paths 1 and 2

- Start of composite control

N20 G51.5 P101 Q201 :

Start of composite control on the X1- and X2-axes

N30 G51.5 P102 Q202 :

Start of composite control on the Z1- and Z2-axes

- Cancellation of composite control

N70 G50.5 P101 Q201 :

Cancellation of composite control on the X1- and X2-axes

N80 G50.5 P102 Q202 :

Cancellation of composite control on the Z1- and Z2-axes

- Parameter check

If the composite control axis corresponding to the P or Q specified with G51.5/G50.5 is not set in parameter No. 8183, alarm PS5339, “ILLEGAL FORMAT COMMAND IS EXECUTED IN SYNC/MIX/OVL CONTROL.” is issued.

Superimposed control

Superimposed control is performed with the G51.6/G50.6 commands, instead of superimposed control axis selection signals.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

Parameter setting examples for a 3-path system

- Parameter No.12600

	Path 1	Path 2	Path 3
X	101	201	301
Z	102	202	302

- Parameter No.8186

	Path 1	Path 2	Path 3
X	0	0	0
Z	0	0	102

- Program example (M100 to M103 are synchronization M codes.)

Path 1	Path 2	Path 3	Operation
N10 M100 P13 ;	Independent operation	M100 P13 ;	Synchronization between paths 1 and 3
N20 G51.6 P102 Q302 ;			Start of Z1-Z3 superimposed control
N30 M101 P13 ;		M101 P13 ;	Synchronization between paths 1 and 3
N40 G00 Z100.;		G00 Z-200.;	Z1-Z3 superimposed control (Z+100 superimposed on Z3)
N50 M102 P13 ;		M102 P13 ;	Synchronization between paths 1 and 3
N60 G50.6 Q302 ;			Cancellation of Z1-Z3 superimposed control
N70 M103 P13 ;		M103 P13 ;	Synchronization between paths 1 and 3

- Start of superimposed control

N20 G51.6 P102 Q302 :

Start of superimposed control with the Z1-axis being a master axis and the Z3-axis being a slave axis

- Cancellation of superimposed control

N60 G50.6 Q302 :

Cancellation of superimposed control with the Z1-axis being a master axis and the Z3-axis being a slave axis

- Parameter check

If the axis number corresponding to the P specified with G51.6 is not set in superimposed slave axis parameter No. 8186, alarm PS5339, “ILLEGAL FORMAT COMMAND IS EXECUTED IN SYNC/MIX/OVL CONTROL.” is issued.

NOTE

1 If G codes (G50.4/G50.5/G50.6) for canceling synchronous, composite, and superimposed control with program commands are issued for axes under synchronous, composite, and superimposed control with DI signals, synchronous, composite, and superimposed control is canceled.

2 If the synchronous control axis selection signal, composite control axis selection signal, and superimposed control axis selection signal are changed from '1' to '0' for axes under synchronous, composite, and superimposed control with program commands, synchronous, composite, and superimposed control is canceled.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

21.4 ROTARY AXIS ROLL-OVER

The roll-over function of the rotary axis prevents coordinates for the rotary axis from overflowing. The roll-over function of the rotary axis is enabled by setting bit 0 (ROAx) of parameter No. 1008 to 1.

21.4.1 Rotary Axis Roll-over

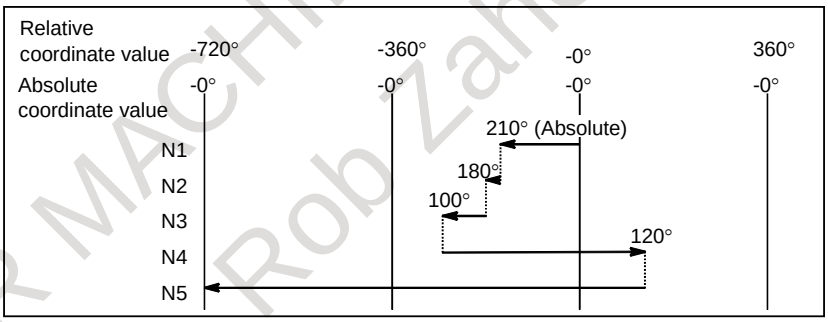
Explanation

For an incremental programming, the tool moves the angle specified in the command. For an absolute value programming, the tool moves to the angle rounded off by the amount of movement per rotation (Parameter No.1260). The tool moves in the direction in which the final coordinates are closest when bit 1 (RABx) of parameter No. 1008 is set to 0. Displayed values for relative coordinates are also rounded by the angle corresponding to one rotation when bit 2 (RRLx) of parameter No. 1008 is set to 1.

Example

Assume that axis A is the rotary axis and that the amount of movement per rotation is 360.000 (parameter No. 1260 = 360.000). When the following program is executed using the roll-over function of the rotary axis, the axis moves as shown below.

G90 A0 ;	Sequence number	Actual movement value	Absolute coordinate value after movement end
N1 G90 A-150.0 ;	N1	-150	210
N2 G90 A540.0 ;	N2	-30	180
N3 G90 A-620.0 ;	N3	-80	100
N4 G91 A380.0 ;	N4	+380	120
N5 G91 A-840.0 ;	N5	-840	0



M

NOTE

This function cannot be used together with the index table indexing function.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

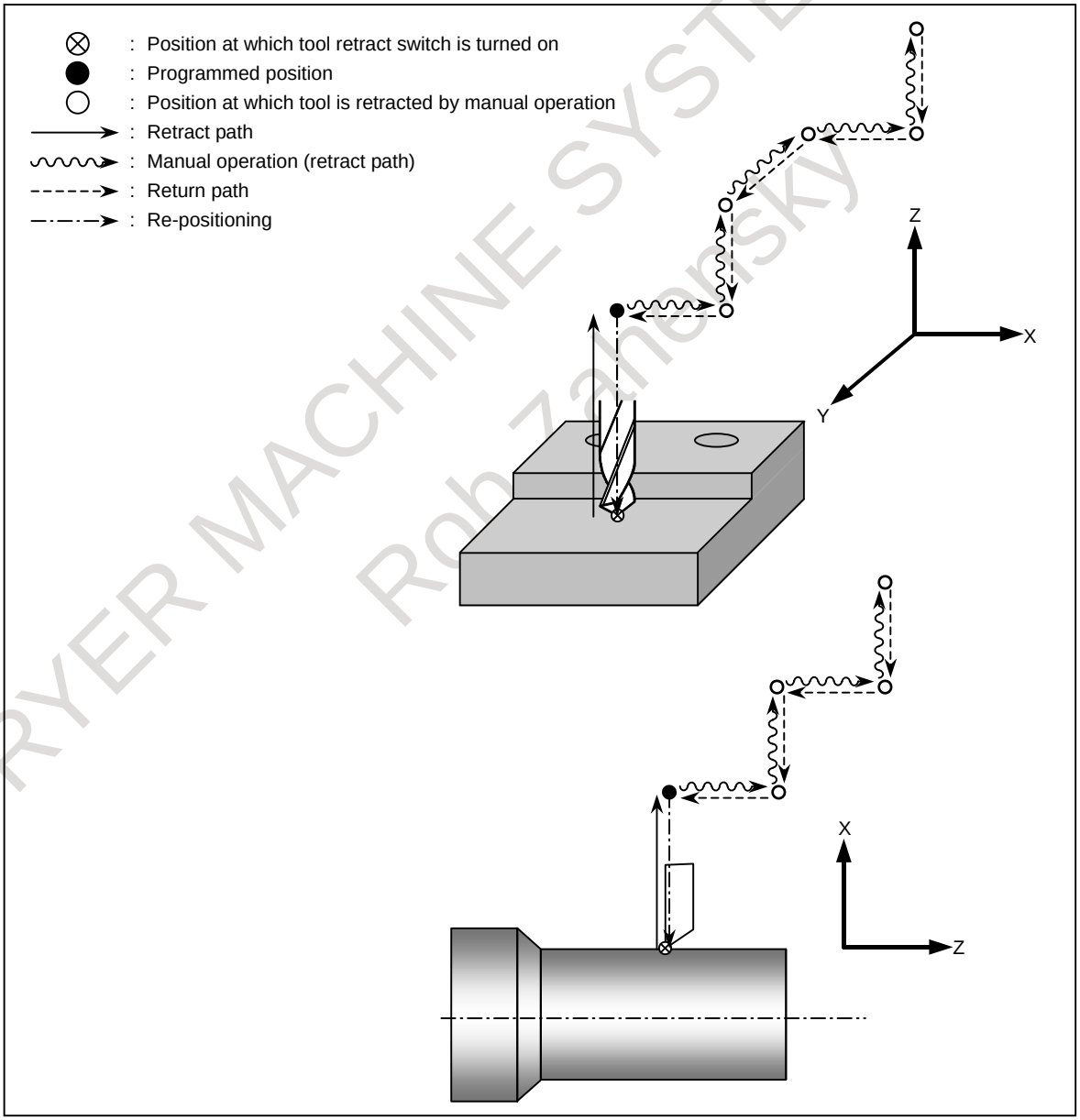
21.5 TOOL RETRACT AND RECOVER

Overview

To replace the tool damaged during machining or to check the status of machining, the tool can be withdrawn from a workpiece. The tool can then be advanced again to restart machining efficiently.
The tool retract and recover operation consists of the following four steps:

- Retract
  The tool is retracted to a predefined position using the TOOL WITHDRAW switch.
- Withdrawal
  The tool is moved to the tool-change position manually.
- Return
  The tool returns to the retract position.
- Repositioning
  The tool returns to the interrupted position.

For the tool retract and recover operations, see “Tool retract and recover“ in the Part III.



FRYER MACHINE SYSTEMS INC
Rob Zahorsky

Format

Specify a retraction axis and distance in the following format:

Specify the amount of retraction, using G10.6.

G10.6 IP_ ;

IP: In incremental mode, retraction distance from the position where the retract signal is turned on
In the absolute mode, retraction distance to an absolute position
The specified amount of retraction is effective until G10.6 is next executed. To cancel the retraction, specify the following:
G10.6 ; (as a single block containing no other commands)

Explanation

- Retraction

When the TOOL WITHDRAW switch on the machine operator's panel is turned on during automatic operation or in the automatic operation stop or hold state, the tool is retracted the length of the programmed retraction distance. This operation is called retraction. The position at which retraction is completed is called the retraction position. The tool is retracted to the retraction position in linear interpolation at the minimum feedrate among the parameter (No.7042) values of moving axes. When retraction is completed, the RETRACTIONS POSITION LED on the machine operator’s panel goes on.
When the TOOL WITHDRAW switch is turned on during execution of a block in automatic operation, execution of the block is interrupted immediately and the tool is retracted. After retraction is completed, the system enters the automatic operation hold state.
If the retraction distance and direction are not programmed, retraction is not performed. In this state, the tool can be withdrawn and returned.
When the TOOL WITHDRAW switch is turned on in the automatic operation stop or hold state, the tool is retracted, then the automatic operation stop or hold state is entered again.
When the TOOL WITHDRAW switch is turned on, the tool withdraw mode is set. When the tool withdraw mode is set, the TOOL BEING WITHDRAWN LED on the machine operator's panel goes on.

- Withdrawal

When the manual mode is set, the tool can be moved manually (jog feed, incremental feed, or handle feed) to replace the tool or measure a machined workpiece. This operation is called a withdrawal. The tool withdrawal path is automatically memorized by the CNC.

- Return

When the mode is returned to automatic operation mode and the TOOL RETURN switch on the machine operator's panel is turned off, the CNC automatically moves the tool to the retraction position by tracing the manually-moved tool path backwards. This operation is called a return. Upon completion of a return to the retraction position, the RETRACTIONS POSITION LED comes on. The return is performed at the feedrate specified by parameter (No.7042) for each axis.

- Repositioning

When the cycle start button is pressed while the tool is in the retraction position, the tool moves to the position where the TOOL WITHDRAW switch was turned on. This operation is called repositioning. The re-positioning movement is made by linear interpolation at the minimum feedrate among the parameter (No.7042) values of moving axes. Upon completion of repositioning, the TOOL BEING WITHDRAWN LED is turned off, indicating that the tool withdrawal mode has terminated. Operation after completion of repositioning depends on the automatic operation state when the tool withdrawal mode is set.

(1) When the tool withdrawal mode is set during automatic operation, operation is resumed after completion of repositioning.

FRYER MACHINE SYSTEMS INC
Rob Zakensky

(2) When the tool withdrawal mode is set when automatic operation is held or stopped, the original automatic operation hold or stop state is set after completion of repositioning. When the cycle start button is pressed again, automatic operation is resumed.

NOTE
1 If negative value or zero is set in parameter (No.7042) of moving axes, retract, recovery and re-positioning are performed at the dry run feedrate.
2 Manual feedrate override signal is effective for the feedrate of parameter (No.7042).

Limitation

- Offset

If the origin, presetting, or workpiece origin offset value (or External workpiece origin offset value) is changed after retraction is specified with G10.6 in absolute mode, the change is not reflected in the retraction position. After such changes are made, the retraction position must be respecified with G10.6.
When the tool is damaged, automatic operation can be interrupted with a tool withdrawal and return operation in order to replace the tool. Note that if the tool offset value is changed after tool replacement, the change is ignored when automatic operation is resumed from the start point or other point in the interrupted block.

- Machine lock, mirror image, and scaling

When withdrawing the tool manually in the tool withdrawal mode, never use the machine lock, mirror-image, or scaling function.

- Reset

Upon reset, the retraction data specified in G10.6 is cleared. Retraction data needs to be specified again.

- Retraction command

The tool withdrawal and return function is enabled even when the retraction command is not specified. In this case, retraction and repositioning are not performed.

⚠ **WARNING**
The retraction axis and retraction distance specified in G10.6 need to be changed in an appropriate block according to the figure being machined. Be very careful when specifying the retraction distance; an incorrect retraction distance may damage the workpiece, machine, or tool.

FRYER MACHINE SYSTEMS INC
Rob Zarensky

21.5.1 Improvement of Tool compensation for Tool Retract and Recover

Outline

In this function, when the recovery operation or re-positioning operation is started, the updated compensation value is used. Therefore the restart operation is performed with the updated compensation value.

Explanation

If this function is effective (the bit 7 (TRO) of parameter No.7002 is set to 1), when the recovery operation or re-positioning operation is started, the updated compensation value is used and the restart operation is performed with the updated compensation value. Thus, if the compensation value is updated after exchanging the tool, the tip point of tool becomes the same position as it of before exchanging tool in the restart operation thereafter.

The operation that updated compensation value is effective can be selected by the bit 6 (TNR) of parameter No.7002.
Example-1 (When the bit 6 (TNR) of parameter No.7002 is set to 0):
Assuming that the compensation value is updated from OFS_1 to OFS_2 at the manual retract position. When the recovery operation is started, the updated compensation value OFS_2 is used, the tip point of new tool and old tool is the same position in the retract position. (Refer to the Fig.21.5.1 (a))

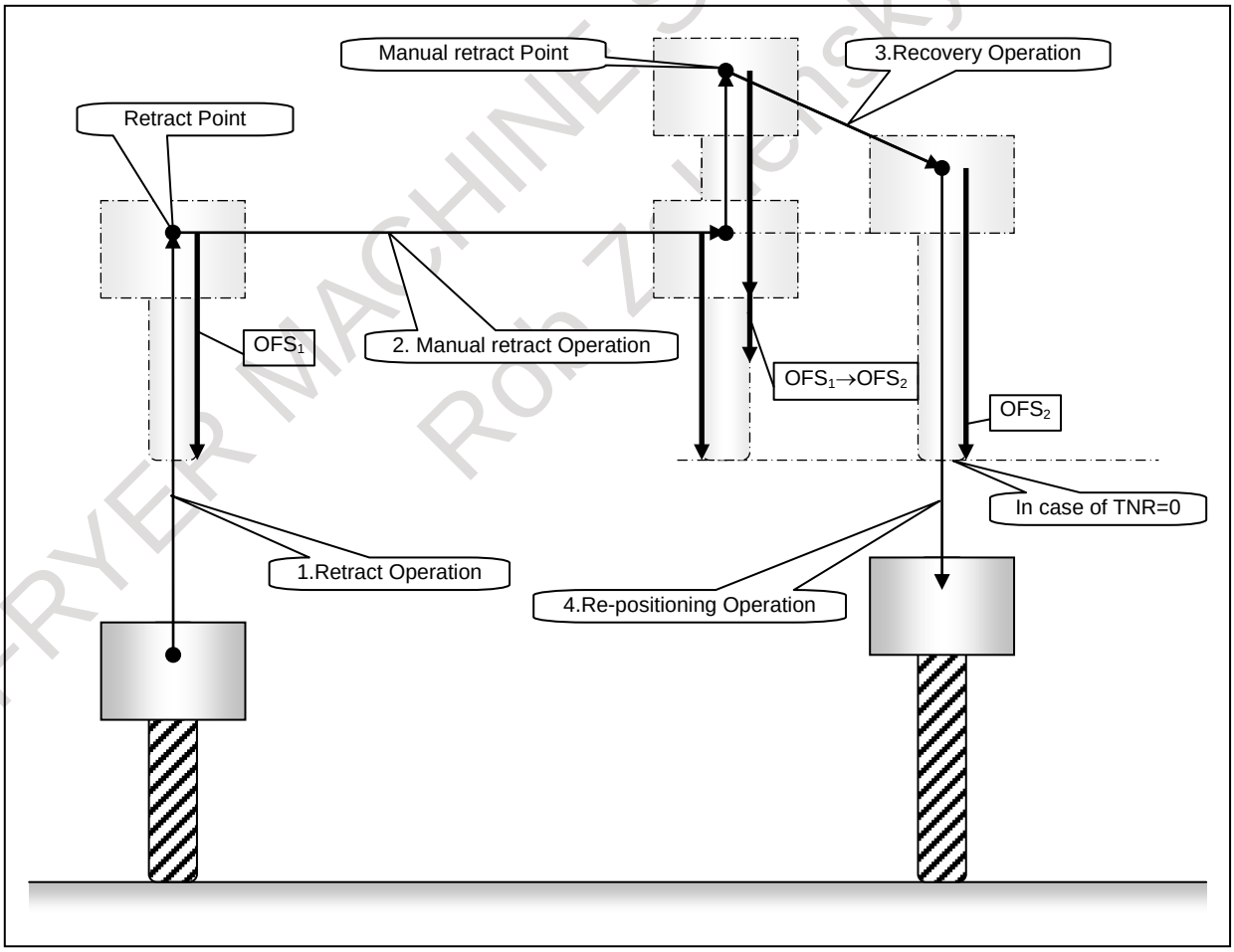


Fig.21.5.1 (a)

FRYER MACHINE SYSTEMS INC
Rob Zalensky

Example-2 (When the bit 6 (TNR) of parameter No.7002 is set to 1):
Assuming that the compensation value is updated from OFS_1 to OFS_2 in the manual retract position. When the recovery operation is started, the updated compensation value OFS_2 is not used. After re-positioning operation, the updated compensation value OFS_2 is used (Refer to the Fig.21.5.1 (b))

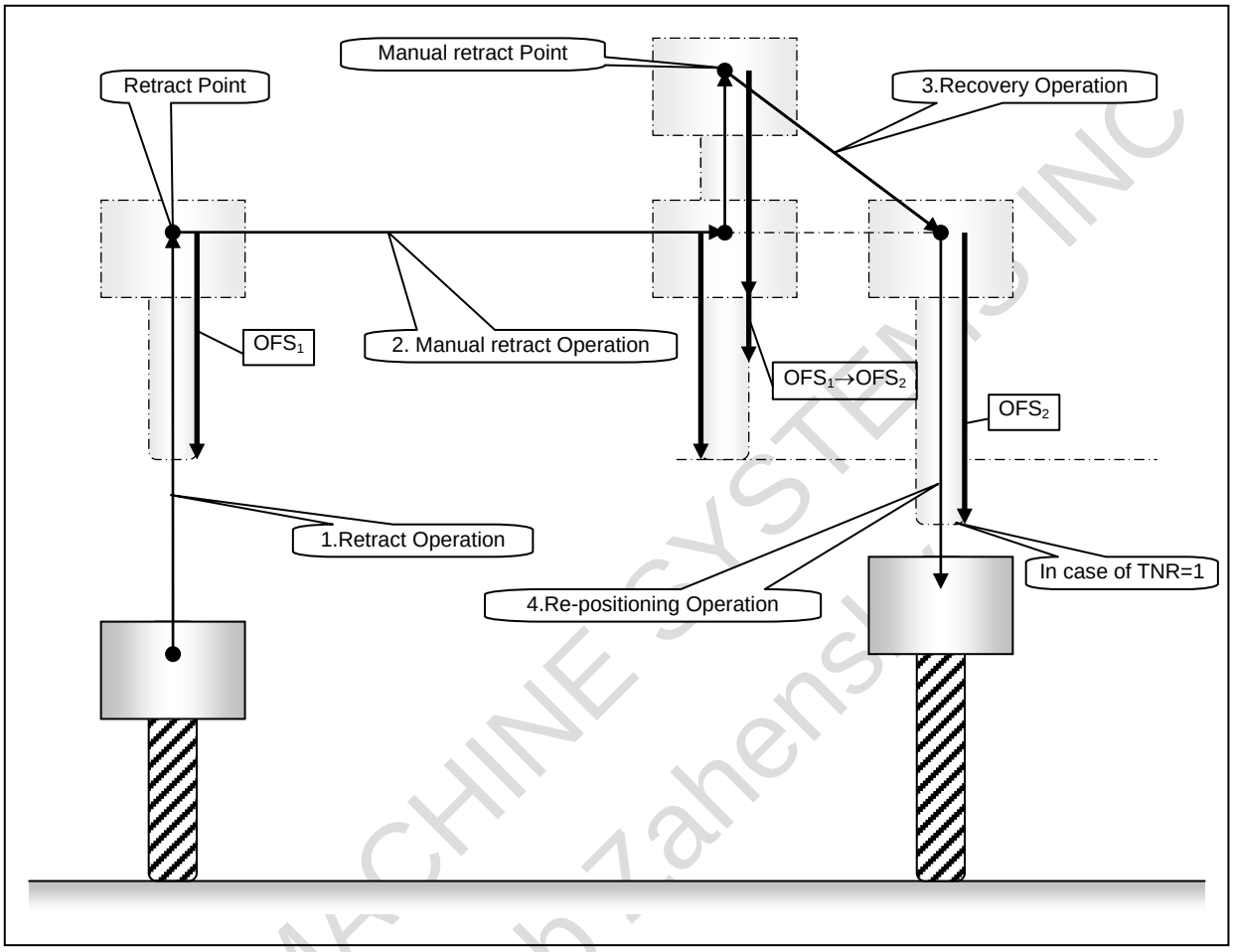


Fig.21.5.1 (b)

Notes

NOTE

1 The compensation considered by this function is as follows. Neither the cutter compensation nor the tool nose radius compensation is considered.
Machining system : Tool length compensation G43/G44 (wear/geometry),
Tool Offset G43.7/G44.7 (wear /geometry)
Lathe system : When the Extended Tool Selection is not used (the bit 3 (TCT) of parameter No.5040 is set to 0)
Tool compensation T code (wear /geometry)
When the Extended Tool Selection is not used (the bit 3 (TCT) of parameter No.5040 is set to 1)
Tool length compensation G43/G44 (wear /geometry),
Tool Offset G43.7/G44.7 (wear /geometry)

2 The compensation value can be changed only in the retract position or the manual retract position.

FRYER MACHINE SYSTEMS INC
Rob Zahenski

NOTE

3 When the tool length compensation B is used, the vertical axis against the plane from which the tool retract and recover is executed is compensated. This plane is not a same plane when G43/G44 is instructed (In case machining system)

4 When the tool length compensation C is used and G43 or G44 is commanded, the compensated axis is indefinite. Therefore this function cannot be used in this case.

5 This function cannot be used during the movement to the reference position by the G28/G30 command or the G53 command is executing.

21.6 ELECTRONIC GEAR BOX

21.6.1 Electronic Gear Box

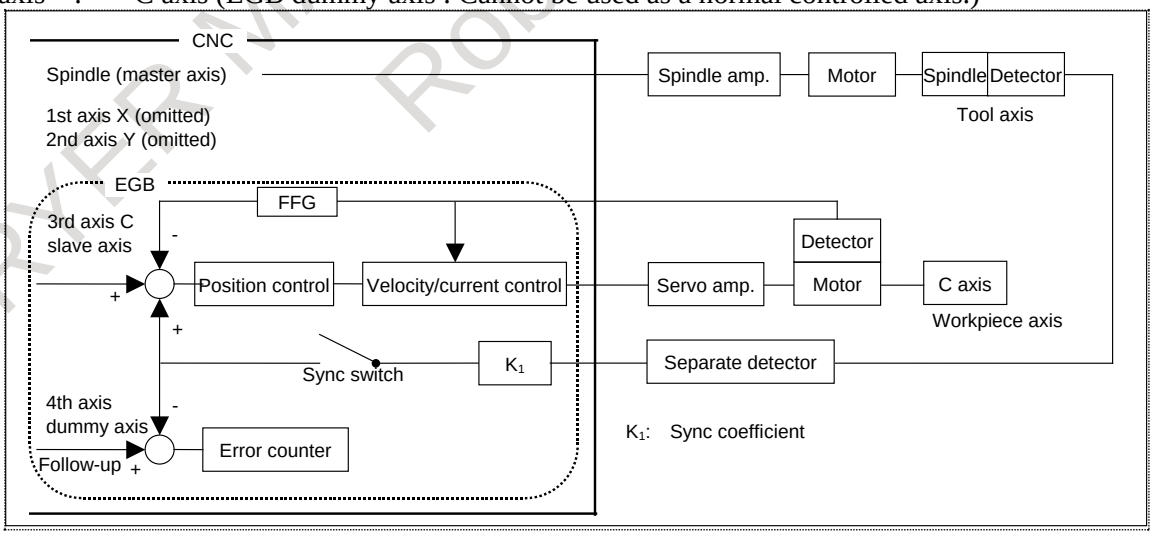
Overview

This function enables fabrication of high-precision gears, screws, and other components by rotating the workpiece in synchronization with a rotating tool or by moving the tool in synchronization with a rotating workpiece. The rate of synchronization can be specified with a program. The synchronization of tool and workpiece axes with this function adopts a system in which the synchronization is directly controlled by digital servo, so that the workpiece axis can follow up the speed fluctuations on the tool axis with no error, thereby allowing fabrication of high-precision cogwheels. In the subsequent explanation, the Electronic Gear Box is called the EGB.

There are several conditions in the setting and so on of the workpiece axis and the tool axis, and refer to the appropriate manual provided by the machine tool builder.

- Example of controlled axis configuration

Spindle : EGB master axis : Tool axis
1st axis : X axis
2nd axis : Y axis
3rd axis : C axis (EGB slave axis : Workpiece axis)
4th axis : C axis (EGB dummy axis : Cannot be used as a normal controlled axis.)



FRYER MACHINE SYSTEMS INC
Rob Zahensky

Format

	Bit 0(EFX) of parameter No.7731=0	Bit 0(EFX) of parameter No.7731=1	
		Bit 5(HBR) of parameter No.7731=1	Bit 5(HBR) of parameter No.7731=0
Start of synchronization	**G81 T_ (L_) (Q_ P_) ;**	**G81.4 R_ (L_) (Q_ P_) ;**	**G81.4 T_ (L_) (Q_ P_) ;**
Cancellation of synchronization	**G80 ;**	**G80.4 ;**	**G80.4 ;**
	(*1) (*4)	**(*2) (*4)**	**(*3) (*4)**

T(or R) : Number of teeth (Specifiable range: 1 to 5000)
L : Number of hob threads (Specifiable range: -250 to +250)
The sign of L determines the direction of rotation for the workpiece axis.
When L is positive, the direction of rotation for the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation for the workpiece axis is negative (- direction).
When L is 0, it follows the setting of bit 3 (LZR) of parameter No. 7701.
If L is not specified, the number of hob threads is assumed 1.
Q : Module or diametral pitch
Specify a module in case of metric input.
(Unit: 0.00001mm, Specifiable range: 0.01 to 25.0mm)
Specify a diametral pitch in case of inch input.
(Unit: 0.00001inch^{-1}, Specifiable range: 0.01 to 254.0 inch^{-1})
P : Gear helix angle
(Unit: 0.0001deg, Specifiable range: -90.0 to +90.0deg)
*1 Use it for machining centers.
*2 Use it for lathes.
*3 Use it for machining centers.
This format enables specification of the same G codes as those for lathes.
*4 When specifying Q and P, the user can use a decimal point.

NOTE
Specify G81, G80, G81.4, and G80.4 in a single block.

Explanation

- Master axis, slave axis, and dummy axis

The synchronization reference axis is called the master axis, while the axis along which movement is performed in synchronization with the master axis is called the slave axis. For example, if the workpiece moves in synchronization with the rotating tool as in a hobbing machine, the tool axis is the master axis and the workpiece axis is the slave axis.
Which axes to become the master and slave axes depends on the configuration of the machine. For details, refer to the manual issued by the machine tool builder.
A single servo axis is used exclusively so that digital servo can directly read the rotation position of the master axis. (This axis is called the EGB dummy axis.)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Synchronous control

(1) Start of synchronization

If G81 is issued so that the machine enters synchronization mode, the synchronization switch of the EGB function is closed, and the synchronization of the tool and workpiece axes is started. During synchronization, the rotation about the tool and workpiece axes is controlled so that the relationship between T (number of teeth) and L (number of hob threads) is maintained. During synchronization, the synchronization relationship is maintained regardless of whether the operation is automatic or manual.

Specify P and Q to use helical gear compensation.

If only either P or Q is issued, alarm PS1594, ”EGB FORMAT ERROR” is generated.

If, during synchronization, G81 is issued again without synchronization cancelation, alarm PS1595, ”ILL-COMMAND IN EGB MODE” is generated if bit 3(ECN) of parameter No. 7731 is 0. If bit 3(ECN) of parameter No. 7731 is 1, helical gear compensation is conducted with the synchronization coefficient being changed to the one newly specified with T and L commands if T and L commands are issued, and if T and L commands are not issued and only P and Q commands are issued, helical gear compensation is conducted with the synchronization coefficient kept intact. This allows consecutive fabrication of helical gears and super gears.

(2) Start of tool axis rotation

When the rotation of the tool axis starts, the rotation of the workpiece axis starts so that the synchronous relationship specified in the G81 block can be maintained.

The rotation direction of the workpiece axis depends on the rotation direction of the tool axis. That is, when the rotation direction of the tool axis is positive, the rotation direction of the workpiece axis is also positive; when the rotation direction of the tool axis is negative, the rotation direction of the workpiece axis is also negative. However, by specifying a negative value for L, the rotation direction of the workpiece axis can be made opposite to the rotation direction of the tool axis.

During synchronization, the machine coordinates of the workpiece axis and EGB axis are updated as synchronous motion proceeds. On the other hand, a synchronous move command has no effect on the absolute and relative coordinates.

(3) Termination of tool axis rotation

Synchronizing with gradual stop of the tool axis, the workpiece axis is decelerated and stopped. By specifying the G80 command after the spindle stops, synchronization is canceled, and the EGB synchronization switch is opened.

(4) Cancellation of synchronization

When cancellation of synchronization is issued, the absolute coordinate on the workpiece axis is updated in accordance with the amount of travel during synchronization. Subsequently, absolute commands for the workpiece axis will be enabled.

For a rotation axis, the amount of travel during synchronization, as rounded to 360-degree units is added to the absolute coordinate.

In the G80 block, only O and N addresses can be specified.

By setting bit 0 (HBR) of parameter No. 7700 to 0, it is possible to cancel synchronization with a reset.

Synchronization is automatically canceled under the following conditions:

(a) An emergency stop is applied.
(b) A servo alarm is generated.
(c) Alarm PW0000, “POWER MUST BE OFF” is generated.
(d) An IO alarm is generated.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

⚠ CAUTION

1 Feed hold, interlock, and machine lock are invalid to a slave axis in EGB synchronization.
2 Even if an OT alarm is issued for a slave axis in EGB synchronization, synchronization will not be canceled.
3 During synchronization, it is possible to execute a move command for a slave axis and other axes, using a program. The move command for a slave command must be an incremental one.

NOTE

1 If bit 0 (HBR) of parameter No. 7700 is set to 1, EGB synchronization will not be canceled due to a reset. Usually, set this parameter bit to 1.
2 In synchronous mode, it is not possible to specify G27, G28, G29, G30, and G53 for a slave axis.
3 It is not possible to use controlled axis detach for a slave axis.
4 During synchronization, manual handle interruption can be performed on the slave and other axes.
5 In synchronization mode, no inch/metric conversion commands (G20 and G21) cannot be issued.
6 In synchronous mode, only the machine coordinates on a slave axis are updated.
7 If bit 0 (EFX) of parameter No. 7731 is 0, no canned cycle for drilling can be used. To use a canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.
8 If bit 0 (TDP) of parameter No. 7702, is 1, the permissible range of T is 0.1 to 500 (1/10 of the specified value).
9 If, at the start of EGB synchronization (G81), L is specified as 0, synchronization starts with L assumed to be 1 if bit 3 (LZR) of parameter No. 7701 is 0; if bit 3 (LZR) of parameter No. 7701 is 1, synchronization is not started with L assumed to be 0. At this time, helical gear compensation is performed.
10 Feed per revolution is performed on the feedback pulses on the spindle. By setting of bit 0 (ERV) of parameter No. 7703, to 1, feed per revolution can be performed based on the speed on the synchronous slave axis.
11 Actual cutting feedrate display does not take synchronization pulses into consideration.
12 For an EGB slave axis, synchronous and composite control cannot be executed.
13 In EGB synchronization mode, AI contour control mode is temporarily canceled.
14 Not advanced preview feed-forward but conventional feed-forward is enabled in the path where EGB synchronization mode is effective.

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

- Helical gear compensation

For a helical gear, the workpiece axis is compensated for the movement along the Z-axis (axial feed axis) based on the torsion angle of the gear.
Helical gear compensation is performed with the following formulas:

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (for metric input)}$$

$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (for inch input)}$$

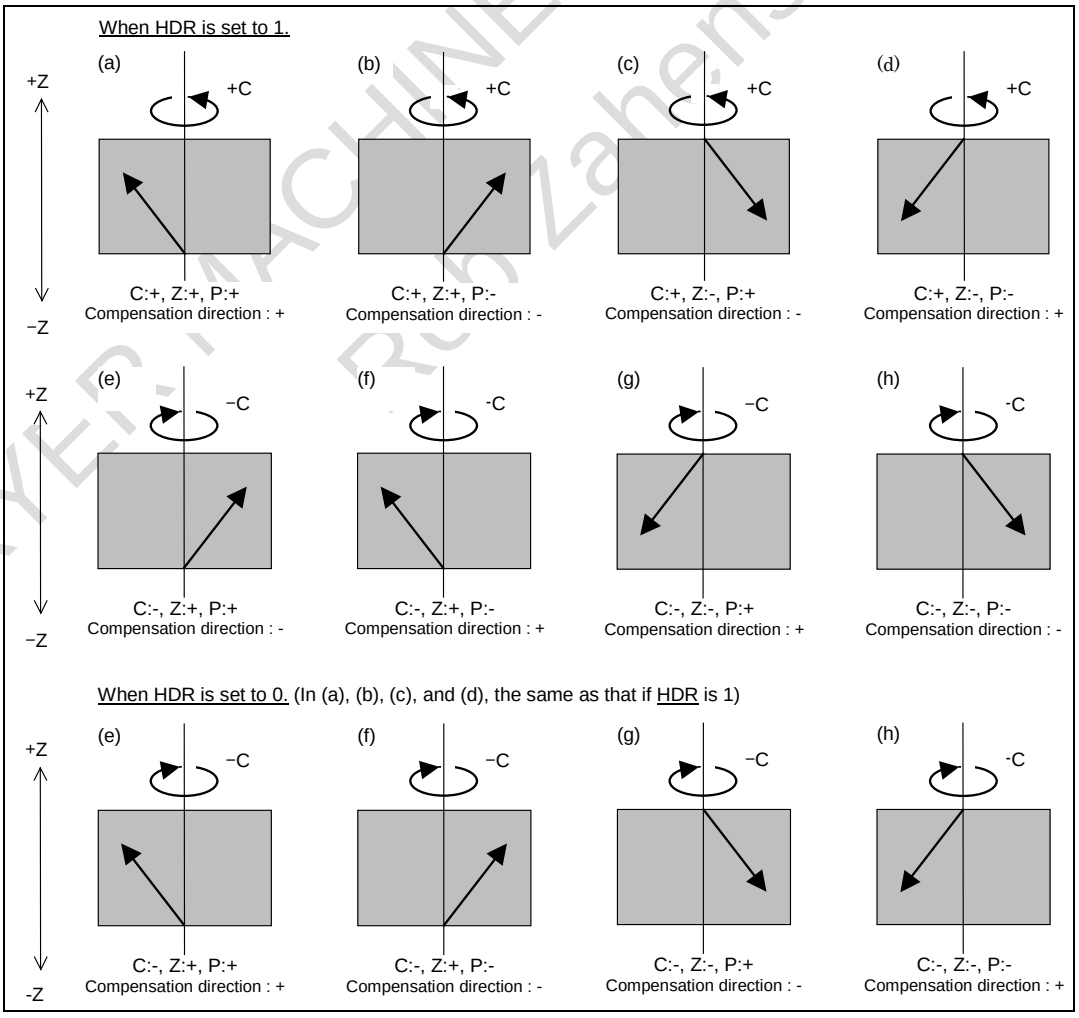
where

Compensation angle: Signed absolute value (deg)
Z : Amount of travel on the Z-axis after the specification of G81 (mm or inch)
P : Signed gear helix angle (deg)
π : Circular constant
T : Number of teeth
Q : Module (mm) or diametral pitch (inch^{-1})
Use P, T, and Q specified in the G81 block.

In helical gear compensation, the machine coordinates on the workpiece axis and the absolute coordinates are updated with helical gear compensation.

- Direction of helical gear compensation

The direction depends on bit 2 (HDR) of parameter No. 7700.



FRYER MACHINE SYSTEMS INC
Ron Zahensky

- Synchronization coefficient

A synchronization coefficient is internally represented using a fraction (Kn/Kd) to eliminate an error. The formula below is used for calculation.

$$\text{Synchronization coefficient} = \frac{Kn}{Kd} = \frac{L}{T} \times \frac{\beta}{\alpha}$$

where

L : Number of hob threads

T : Number of teeth

α : Number of pulses of the position detector per rotation about the master axis (parameter No. 7772)

β : Number of pulses of the position detector per rotation about the slave axis (parameter No. 7773)

Kn / Kd is a value resulting from reducing the right side of the above formula, but the result of reduction is subject to the following restrictions:

$-2147483648 \leq Kn \leq 2147483647$

$1 \leq Kd \leq 2147483647$

When this restriction is not satisfied, the alarm PS1596, ”EGB OVERFLOW” is issued when G81 is specified.

Example

```
O1000   ;
N0010   M19   ;              Tool axis orientation
N0020   G28   G91   C0   ;   Reference position return on the workpiece axis
N0030   G81   T20   L1   ;   Synchronous start on tool and workpiece axes
                             (Rotation about the workpiece axis by 18° per rotation about the tool axis)
N0040   S300   M03   ;       Rotation about the tool axis at 300min-1
N0050   G01 X__F__;          Movement along the X-axis (cut)
N0060   G01 Z__F__;          Movement along the Z-axis (machining)
--------------------------- ;  If necessary, axis commands such as C, X, and Z commands are allowed.
N0100   G01 X__F__;          Movement along the X-axis (escape)
N0110   M05   ;              Stop on the tool axis
N0120   G80   ;              Synchronous cancellation on tool and workpiece axes
N0130   M30   ;
```

- Retract function

(1) Retract function with an external signal

When the retract switch on the machine operator’s panel is turned on, retraction is performed with the retract amount set in parameter No. 7741 and the feedrate set in parameter No. 7740. No movement is performed along an axis for which 0 is set as the retract amount.

For the retract switch, refer to the relevant manual provided by the machine tool builder.

(2) Retract function with an alarm

If, during EGB synchronization or automatic operation, a CNC alarm is issued, retraction is performed with the retract amount set in parameter No. 7741 and the speed set in parameter No. 7740.

This can prevent the tool and the object being machined from damage if a servo alarm is generated.

No movement is performed along an axis for which 0 is set as the retract amount.

For the retract switch, refer to the relevant manual provided by the machine tool builder.

<u>Conditions under the retract function with an alarm</u>

The conditions under which the retract function is performed by an alarm can be changed using the settings of bit 1(ARE) of parameter No. 7703, and bit 2 (ARO) of parameter No. 7703.

The table below lists parameter settings and corresponding conditions.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

ARE	ARO	Condition
1	0	EGB synchronization is in progress.
1	1	Both EGB synchronization and automatic operation are in progress.
0	0	Either EGB synchronization or automatic operation is in progress.
0	1	

⚠ CAUTION

1 Retraction is performed at the speed specified in parameter No. 7740.
2 Feed hold is not effective to movement during retraction.
3 Feedrate override is not effective to movement during retraction.

NOTE

1 During a retract operation, an interlock is effective to the retract axis.
2 During a retract operation, a machine lock is effective to the retract axis.
3 The retraction direction depends on the movement direction of the machine, regardless of whether an mirror image (signal and setting) is enabled or disabled. (Mirror image is applied to the updating of absolute coordinates.)
4 If retraction is performed during automatic operation, automatic operation is halted simultaneously with a retract operation, but it is at the end of the retract operation that the operation state switches to the automatic operation halt state.
5 It is not possible to perform automatic operation during retraction.
6 The acceleration/deceleration of a retract operation is in the acceleration/deceleration state at the start of retraction.
7 Retract movement is performed with non-linear type positioning.
8 If, during a retract operation, a reset or an emergency stop is made, the operation is interrupted.
9 To enable the retract function with an alarm, bit 3 (ART) of parameter No. 7702, must be set to 1.
10 The retract function with an alarm does not perform a retract operation on the retract axis if an overtravel alarm or a servo alarm is generated on the retract axis.
11 If a new alarm is issued during retraction with the retract function with an alarm, a retract operation is not performed.

21.6.2 Spindle Electronic Gear Box

Overview

In the system with two spindles for a tool axis and a workpiece axis, a gear can be shaped (grind/cut) by synchronizing the workpiece axis rotation with the tool axis (grinding axis /hob) rotation.

To synchronize these two spindles, the spindle electronic gear box is used (Hereafter, the electronic gear box is called the EGB.). In the spindle EGB, the synchronous pulse is produced from the feedback pulse of the position detector attached to the tool axis (master axis), and the workpiece axis (slave axis) rotates with the synchronous pulse. Feedback pulse from master side to slave side is sent by the communication between spindle amplifiers.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Specification

The specifications of the spindle EGB control are as follows:

(1) The Spindle EGB synchronization is started by specifying T command (number of teeth) and L command (number of hob threads), which determine the synchronous ratio, in G81 block. The Spindle EGB synchronization is canceled by specifying G80.

(2) The synchronous ratio is calculated from T and L command in G81 block and the number of position detector pulses per rotation about the tool and the workpiece axis (parameter setting).

(3) This function has the retract function.

(4) The cutting helical gear is performed by specifying Q command (module or diametral pitch) and P command (gear helix angle) in G81 block.

(5) The Spindle EGB synchronization is maintained regardless of whether the operation is automatic or manual.

(6) In order to turn on to spindle EGB synchronization mode, the slave spindle has to be Cs contouring control mode, though the master spindle may be any mode.

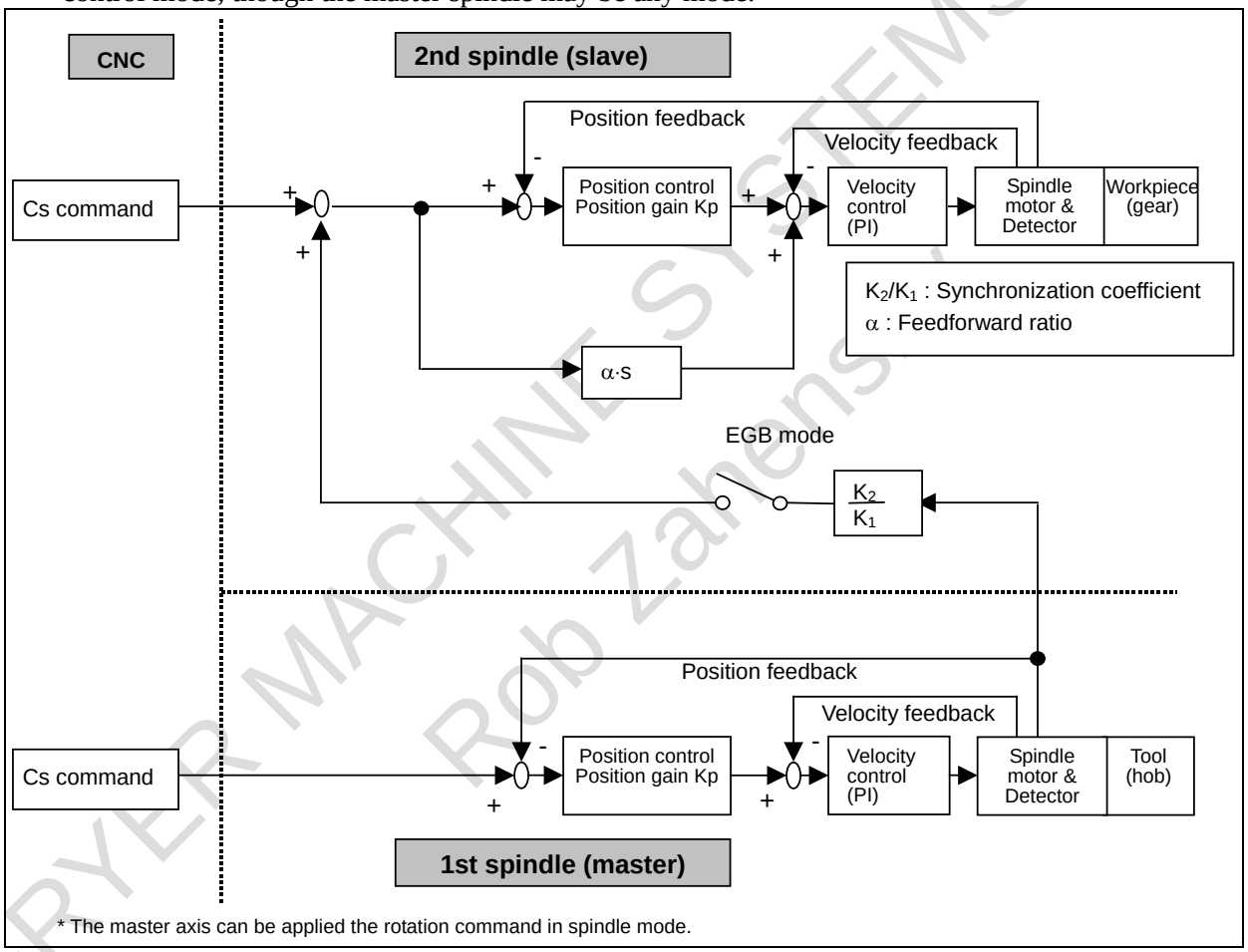


Fig. 21.6.2 (a) Block diagram of spindle EGB control

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Format

	Bit 0(EFX) of parameter No. 7731=0	Bit 0(EFX) of parameter No. 7731=1	
		Bit 5(HBR) of parameter No. 7731=1	Bit 5(HBR) of parameter No. 7731=0
Start of synchronization	**G81 T_ (L_) (Q_ P_) ;**	**G81.4 R_ (L_) (Q_ P_) ;**	**G81.4 T_ (L_) (Q_ P_) ;**
Cancellation of synchronization	**G80 ;**	**G80.4 ;**	**G80.4 ;**
	(*1) (*4)	**(*2) (*4)**	**(*3) (*4)**

T(or R) : Number of teeth (Specifiable range: 1 to 5000)
L : Number of hob threads (Specifiable range: -250 to +250)
The sign of L determines the direction of rotation for the workpiece axis.
When L is positive, the direction of rotation about the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation about the workpiece axis is negative (- direction).
When L is 0, it depends on the setting of bit 3 (LZR) of parameter No. 7701.
If L is not specified, the number of hob threads is assumed 1.
Q : Module or diametral pitch
Specify a module in the case of metric input.
(Unit: 0.00001mm, Specifiable range: 0.01 to 25.0mm)
Specify a diametral pitch in the case of inch input.
(Unit: 0.00001inch-1, Specifiable range: 0.01 to 254.0 inch-1)
P : Gear helix angle
(Unit: 0.0001deg, Specifiable range: -90.0 to 90.0deg)
*1 Use it for machining centers.
*2 Use it for lathes.
*3 Use it for machining centers.
This format enables specification of the same G codes as those for lathes.
*4 When specifying Q and P, the user can use a decimal point.

NOTE
Specify G81, G80, G81.4, and G80.4 singly in a block.

Explanation

- Parameter setting

The following parameters should be set for the Spindle EGB control.
(1) Slave axis number (Parameter No. 7710)
(2) Number of position detector pulses per rotation about the tool axis (Parameter No. 7772)
(3) Number of position detector pulses per rotation about the workpiece axis (Parameter No. 7773)
(4) Spindle EGB master axis enable (Parameter No. 4352#7)
(5) Spindle EGB slave axis enable (Parameter No. 4352#6)
(6) Number of sinusoidal waves from master spindle position detector(Parameter No. 4386)

FOYER MACHINE SYSTEMS INC
Rob Zahensky

- Start/Cancellation of synchronization

When the rotation of the tool axis (master axis) starts after G81 is specified, the EGB synchronization starts with the synchronous relationship specified in G81 block, and the rotation of the workpiece axis (slave axis) starts. When the EGB synchronization starts, the synchronization switch of the EGB function is closed.

When the rotation of the tool axis is stopped, the workpiece axis stopped with keeping synchronization. Then the EGB synchronization is canceled by specifying G80. When the EGB synchronization is canceled, the EGB synchronization switch is opened.

Specify P and Q to use helical gear compensation.

If only either P or Q is issued, alarm PS1594, “EGB FORMAT ERROR” is generated.

G81 cannot be specified again during the EGB synchronization. Moreover, the specification of T, L, Q, and P cannot be modified during the synchronization. Specify the starting and canceling of synchronization during the tool axis (master axis) stopping.

⚠ CAUTION

1 Feed hold, interlock, and machine lock are invalid to a slave axis in EGB synchronization.
2 Even if an OT alarm is issued for a slave axis in EGB synchronization, synchronization will not be canceled.
3 During synchronization, it is possible to execute a move command for a slave axis and other axes, using a program. The move command for a slave command must be an incremental one.

NOTE

1 If bit 0 (HBR) of parameter No. 7700 is set to 1, EGB synchronization will not be canceled due to a reset. Usually, set this parameter bit to 1.
2 The EGB synchronization should be started and canceled during the stop of both the master and the slave axis.
3 Reference position return of Cs contouring control axis for master and slave axis should be performed before G81 is specified. In the synchronization mode, reference position return cannot be performed. Do not change to reference position return mode in the synchronization mode.
4 In case that the parameters for axis setting (No. 7710, 4352) are not set correctly, alarm PS1593, “EGB PARAMETER SETTING ERROR” occurs when G81 specified.
5 In synchronous mode, it is not possible to specify G27, G28, G29, G30, and G53 for a slave axis.
6 In EGB synchronization mode, AI contour control mode is temporarily canceled.
7 The slave axis position display is updated by synchronous pulse as follows :
   - In synchronous mode, only the machine coordinates on a slave axis are updated. Absolute coordinates and relative coordinates are not updated.
   - When the synchronization is canceled, the pulses of the synchronization rounded to 360-degree units are added to the absolute coordinates.
8 The rotation direction of the slave axis depends on the rotation direction of the master axis. That is, when the rotation direction of the master axis is positive, the rotation direction of the slave axis is also positive; when the rotation direction of the master axis is negative, the rotation direction of the slave axis is also negative. However, by specifying a negative value for L, the rotation direction of the slave axis can be made opposite to the rotation direction of the master axis.
9 The synchronization mode is canceled by servo alarm, spindle alarm, PW alarm, or emergency stop.

FRYER MACHINE SYSTEMS INC Rob Zanensky

NOTE

10 The synchronization is not maintained in servo off status of the slave axis.
11 During synchronization, manual handle interruption can be performed on the slave and other axes.
12 In synchronization mode, no inch/metric conversion commands (G20 and G21) can be issued.
13 If bit 0 (EFX) of parameter No. 7731 is 0, no canned cycle for drilling can be used. To use a canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.
14 If bit 0 (TDP) of parameter No. 7702, is 1, the permissible range of T is 0.1 to 500 (1/10 of the specified value).
15 If, at the start of EGB synchronization (G81), L is specified as 0, synchronization starts with L assumed to be 1 if bit 3 (LZR) of parameter No. 7701, is 0; if bit 3 (LZR) of parameter No. 7701, is 1, synchronization is not started with L assumed to be 0. At this time, helical gear compensation is performed.
16 Feed per revolution is performed on the feedback pulses on the spindle. By setting bit 0 (ERV) of parameter No. 7703, to 1, feed per revolution can be performed based on the speed on the synchronous slave axis.
17 Actual cutting feedrate display does not take synchronization pulses into consideration.
18 For an EGB slave axis, synchronous and composite control cannot be executed.
19 Spindle EGB cannot be used by the G81 command together with servo EGB. Use the G81 command for spindle EGB and G81.5 command for servo EGB when using it together.
20 When simple spindle EGB function is used, this function cannot be used. When G81 is commanded for slave axis of simple spindle EGB, PS1593 alarm is generated.
21 The master axis cannot be shared by simple spindle EGB function and this function.
22 Not advanced preview feed-forward but conventional feed-forward is enabled in the path where EGB synchronization mode is effective.

- Program example

Axis configuration: X,Y,Z, B(Cs contouring control axis : tool axis/master axis), C(Cs contouring control axis : workpiece axis/slave axis)

```
O1000 ;
N0010 G80 ;
N0020 G28 G91 B0 C0 ;   Reference position return operation on the tool and workpiece axis.
N0030 G81 T20 L1 ;      Start of synchronization.
N0040 Mxx ;             Rotate the tool axis by velocity command of PMC axis control.
N0050 G04 X1000 ;       Wait until rotation of the tool axis is constant.
N0060 G01 X_ F_ ;       Movement along the X-axis(cutting).
N0070 G01 Z_ F_ ;       Movement along the Z-axis(machining).
-------------------------
-------------------------
N0100 G01 X_ F_ ;       Movement along the X-axis(escape).
N0110 M05 ;             Stop the tool axis.
N0120 G80 ;             Cancel synchronization.
N0130 M30 ;
```

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Helical gear compensation

For a helical gear, the workpiece axis is compensated for the movement along the Z-axis (axial feed axis) based on the torsion angle of the gear.
Helical gear compensation is performed with the following formulas:

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (for metric input)}$$

$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (for inch input)}$$

where

Compensation angle: Signed absolute value (deg)
Z : Amount of travel on the Z-axis after the specification of G81 (mm or inch)
P : Signed gear helix angle (deg)
π : Circular constant
T : Number of teeth
Q : Module (mm) or diametral pitch (inch^{-1})
Use P, T, and Q specified in the G81 block.

In helical compensation, the machine coordinates and the absolute coordinates on the workpiece axis are updated with helical compensation.

- Direction of helical gear compensation

The direction depends on bit 2 (HDR) of parameter No. 7700.

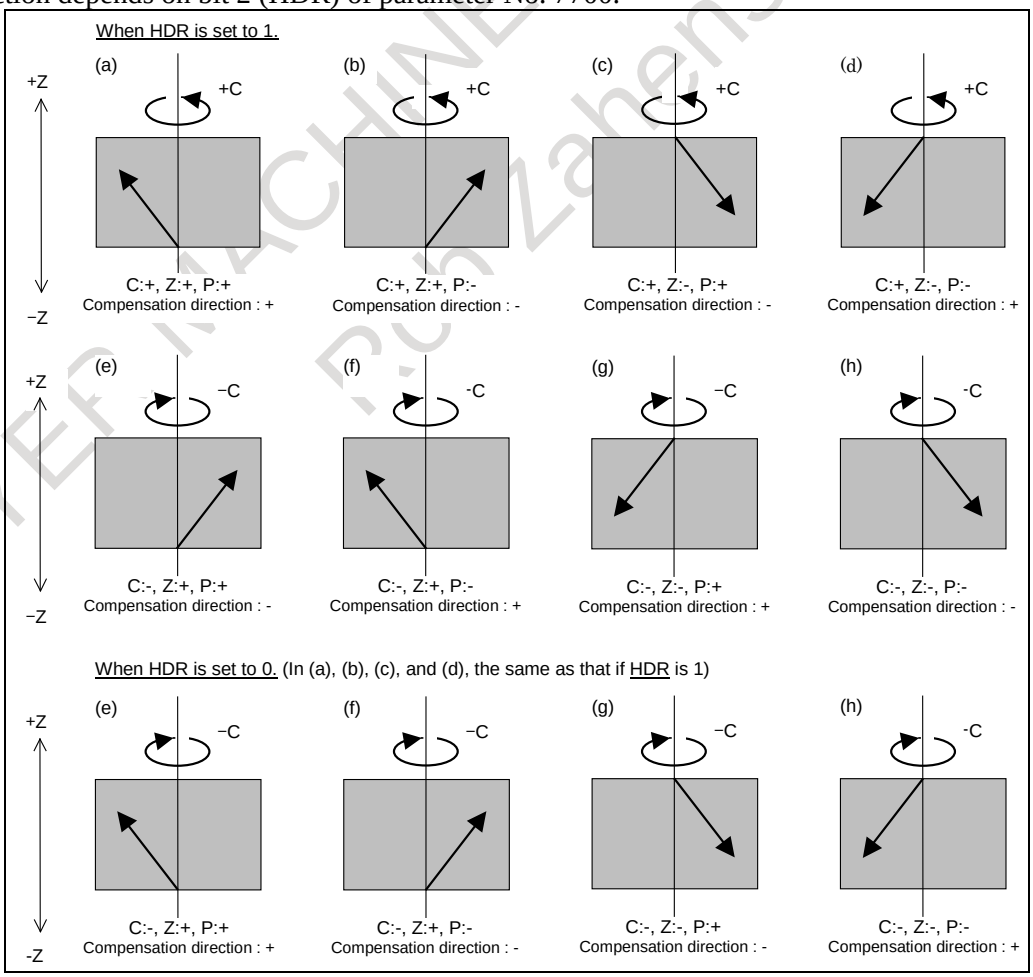


Fig. 21.6.2 (b) Direction of helical gear compensation

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Synchronization coefficient

A synchronization coefficient is internally represented using a fraction (K_2/K_1) to eliminate an error. The formula below is used for calculation.

$$\text{Synchronization coefficient} = \frac{K_2}{K_1} = \frac{L}{T} \times \frac{\beta}{\alpha}$$

where

L : Number of hob threads
T : Number of teeth
α : Number of pulses of the position detector per rotation about the master axis (parameter No. 7772)
β : Number of pulses of the position detector per rotation about the slave axis (parameter No. 7773)

K_2/K_1 is a value resulting from reducing the right side of the above formula, but the result of reduction is subject to the following restrictions:

$-32767 \le K_2 \le 32767$

$1 \le K_1 \le 32767$

When this restriction is not satisfied, the alarm PS1596, "EGB OVERFLOW" is issued when G81 is specified.

The values of K_2 and K_1 are set to the parameter No. 4387 and No. 4388 automatically when G81 is specified. If T is not specified in G81 block, the alarm PS1594, "EGB FORMAT ERROR" is issued. If L is not specified in G81 block, the synchronous ratio is calculated as L = 1.

Example)

When the pulses for one rotation (360000) are specified to the tool axis (master axis) on the following conditions, the pulses for position control are distributed as Fig. 21.6.2 (c).

Hob threads L : 10
Number of teeth of the work T : 100
Number of pulse per one revolution of position detector on the tool axis : 360000
Number of pulse per one revolution of position detector on the work axis : 360000

$$\text{Synchronization coefficient} = \frac{K_2}{K_1} = \frac{L}{T} \times \frac{\beta}{\alpha} = \frac{10}{100} \times \frac{360000}{360000} = \frac{1}{10}$$

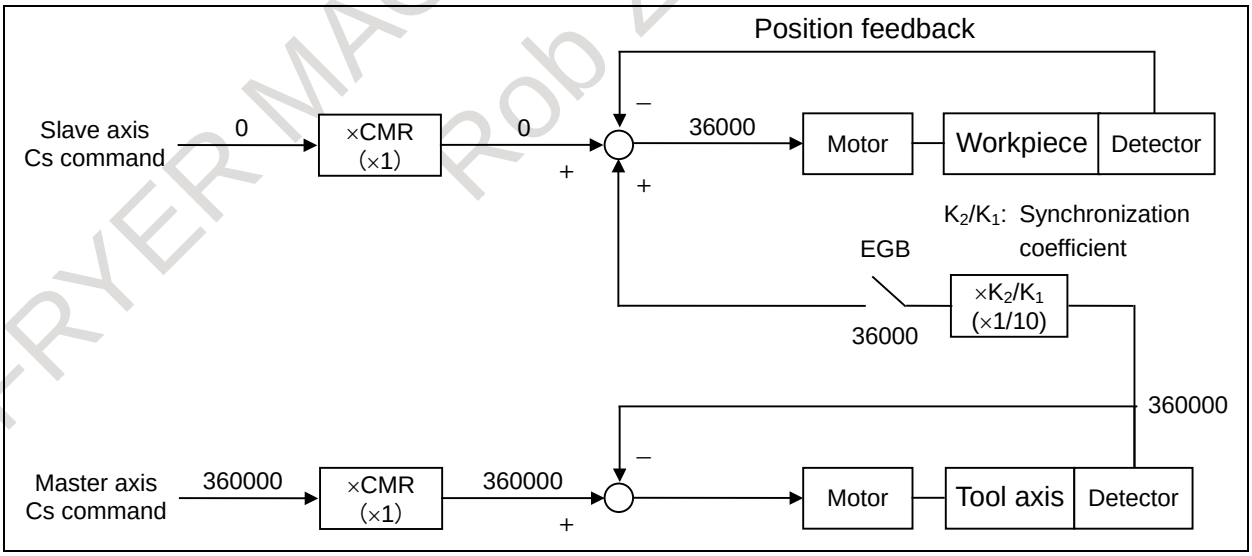


Fig. 21.6.2 (c) Command pulse distribution

As Fig. 21.6.2 (c), when 360000 pulses (Number of pulse for one rotation of the master axis) are specified, the pulses for slave axis by EGB are equal to the value which is multiplied to the number of pulse for one rotation of the slave axis by the ratio of Hob thread and number of teeth (rotation ratio between master axis and slave axis).

360000 × 1/10 = 36000

FRYER MACHINE SYSTEMS INC
Rob Zarensky

- Retract function

See Item, “Retract function” in the Subsection 21.6.1, “Electronic Gear Box”.

21.6.3 Electronic Gear Box Automatic Phase Synchronization

Overview

In the electronic gear box (EGB), when synchronization start or cancellation is specified, the synchronizing state is changed to another state gradually by applying acceleration/deceleration. This is because if synchronization is started or canceled immediately, this sudden speed change shocks a machine. Therefore, synchronization can be started or canceled while the spindle is rotating. Also, synchronization ratio can be changed while the spindle is rotating.

At the start of synchronization, automatic phase synchronization is performed such that the position where the machine coordinate is 0 about the workpiece axis matches the position corresponding to the spindle one-rotation signal. With this synchronization, the same operation is performed as synchronization start caused by a one-rotation signal in hobbing synchronization when using the functions of a hobbing machine.

The spindle corresponds to the EGB master axis and the workpiece axis corresponds to an EGB slave axis (servo axis or Cs contouring axis).

Format

M

- Acceleration/deceleration type

```
G81  T _  L _  R1  ;   Synchronization start
G80  R1  ;             Synchronization cancellation
```

T : Number of teeth (range of valid settings: 1-5000)

L : Number of hob threads (range of valid settings: -250 to +250, excluding 0)
When L is positive, the direction of rotation about the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation about the workpiece axis is negative (- direction).

- Acceleration/deceleration plus automatic phase synchronization type

```
G81  T _  L _  R2  ;   Synchronization start
G80  R2  ;             Synchronization cancellation
```

T : Number of teeth (range of valid settings: 1- to 5000)

L : Number of hob threads (range of valid settings: -250 to +250, excluding 0)
When L is positive, the direction of rotation about the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation about the workpiece axis is negative (- direction).

T

Set the following parameter when using in lathes.

Automatic phase synchronization is enabled in a command compatible with that for a hobbing machine used in lathes.

Bit 0 (EFX) of parameter No. 7731=1

Bit 5 (HBR) of parameter No. 7731=1

Bit 6 (PHS) of parameter No. 7702=1

FRYER MACHINE SYSTEMS INC
Rob Zalensky

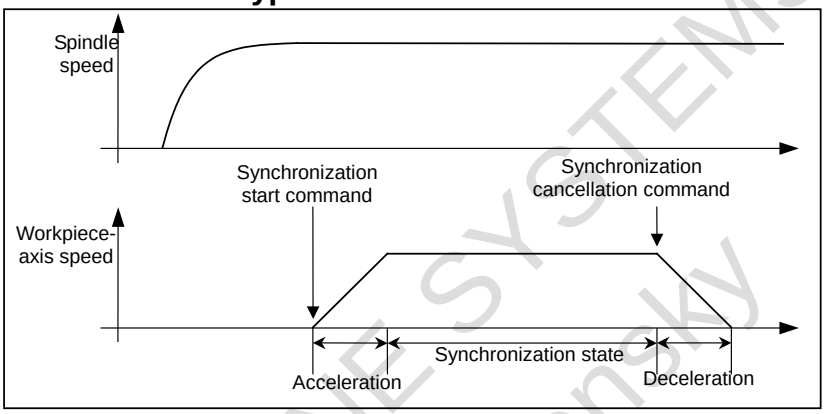
- **Acceleration/deceleration plus automatic phase synchronization type**

G81.4 R _ L _ ; Synchronization start
G80.4 ; Synchronization cancellation

R : Number of teeth (range of valid settings: 1- to 5000)
L : Number of hob threads (range of valid settings: -250 to +250, excluding 0)
When L is positive, the direction of rotation about the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation about the workpiece axis is negative (- direction).

Explanation

- **Acceleration/deceleration type**

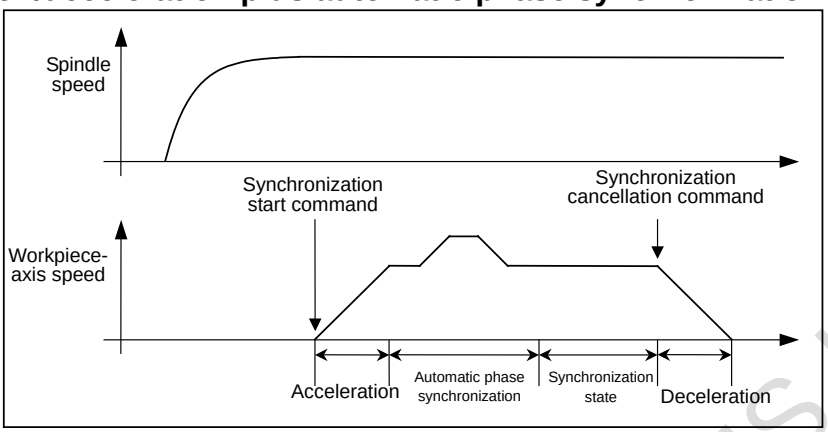


1. Specify G81R1 to start synchronization.
   When G81R1 is specified, the workpiece axis (slave axis) is accelerated with the acceleration according to the acceleration rate set in the parameter No. 7778. When the speed reaches the synchronization speed, the G81R1 block is terminated.
2. For cancellation, specify G80R1 while the tool is moved away from the workpiece.
3. When G80R1 is specified, deceleration is started immediately at the acceleration rate set in parameter No. 7778.
   When the speed is reduced to 0, the G80R1 block is terminated.

NOTE
1 During synchronization start/cancellation, acceleration/deceleration is linear type.
2 In the automatic cancellation of synchronization due to one of the following causes, deceleration is performed and synchronization is canceled:
<1> Reset
<2> Alarm PW0000, “POWER MUST BE OFF”
<3> IO alarm
3 If bit 0 (EFX) of parameter No. 7731 is 0, the canned cycle for drilling cannot be used. To use the canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Acceleration/deceleration plus automatic phase synchronization type



1. Move the workpiece axis to the position that corresponds to that of the one-rotation signal of the spindle.
2. Specify G81R2 to start synchronization.
   When G81R2 is specified, the workpiece axis is accelerated with the acceleration according to the acceleration rate set in the parameter No. 7778. Upon completion of phase synchronization, the G81R2 block terminates.
3. For cancellation, specify G80R2 while the tool is moved away from the workpiece.
4. When G80R2 is specified, deceleration is started immediately at the acceleration rate set in parameter No. 7778. When the speed is reduced to 0, the G80R2 block is terminated.

⚠ CAUTION

1 In automatic phase synchronization, specify the speed in parameter No. 7776 and the movement direction in parameter PHD, bit 7 of No. 7702.
2 In phase synchronization, rapid–traverse linear acceleration/deceleration (with the time constant specified in parameter No. 1620) is performed.
3 The workpiece–axis speed is obtained by superimposing the speed in automatic phase synchronization onto the speed corresponding to spindle rotation. In consideration of this superimposition, specify a position deviation limit in parameter No. 1828.

NOTE

1 The one-rotation signal used for automatic phase synchronization is issued not by the spindle position coder but by the separate pulse coder attached to the spindle and used to collect EGB feedback information. This means that the orientation position based on the one-rotation signal issued by the spindle position coder does not match the position used as the reference for the workpiece axis when phase synchronization is performed with automatic phase synchronization based on G81R2.
  Moreover, the one-rotation signal of the separate pulse coder must be turned on for each rotation of the spindle.
2 With the use of parameter No. 7777, the position at which the phase of the workpiece axis is matched can be shifted from the position corresponding to the one-rotation signal in automatic phase matching.
3 By setting bit 6 (EPA) of parameter No. 7731 to 1, automatic phase synchronization can be performed so that the workpiece axis position at the start of synchronization matches the position corresponding to the spindle one-rotation signal.

FRYER MACHINE SYSTEMS INC Rob Zahensky

NOTE

4 By setting bit 6 (EPA) of parameter No. 7731 to 1, in automatic phase synchronization, when a synchronization command is issued again in the synchronization state, movement about the workpiece axis is made such that the position corresponding to the one-rotation signal of the spindle matches the position about the workpiece axis specified in the G81R2 synchronization start command executed first.

5 In automatic phase synchronization, movement is performed about the workpiece axis from the current position to the nearest phase position in the phase synchronization movement direction specified by the parameter.

6 Linear acceleration/deceleration applies to synchronization start/cancellation.

7 The acceleration/deceleration plus automatic phase synchronization type can be executed by the bit 6 (PHS) of parameter No. 7702, without specifying an R2 command in a G81 or G80 block.

8 In the automatic cancellation of synchronization due to one of the following causes, deceleration is performed and synchronization canceled:
<1> Reset
<2> Alarm PW0000, “POWER MUST BE OFF”
<3> IO alarm

9 When EGB (Spindle EGB is excluded) is used, it is necessary to move the separate Pulsecoder attached to the spindle by one rotation or more before executing automatic phase synchronization.

10 When the spindle EGB is used and the control mode of the master axis is the speed control, it is necessary to execute the position coder orientation before that the automatic phase synchronization is started. In this case, set bit 7 (RFCHK3) of parameter No. 4016 for the master axis must be set ”0” to keep memorizing the one–rotation signal position of the spindle in the speed control mode.

11 When the spindle EGB is used and the control mode of the master axis is in the Cs contouring, it is necessary to execute the reference point return before the automatic phase synchronization is started.

12 The acceleration rate parameter No. 7778 must not be changed in the synchronization mode.

13 If parameter No. 7778 is 0, alarm PS1598, “EGB AUTO PHASE PARAMETER SETTING ERROR” is issued when G81 is issued.

14 For Series 16*i*, the speed and the time constant for automatic phase synchronization are set separately in parameters Nos. 2135 and 2136(Nos. 4384 and 4385 when spindle EGB is used); in this CNC, acceleration is directly set in parameter No. 7778.

15 If bit 0 (EFX) of parameter No. 7731 is 0, the canned cycle for drilling cannot be used. To use the canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Program example

- Acceleration/deceleration type

```
M03 ;                 Clockwise spindle rotation command
G81  T_  L_  R1 ;     Start of synchronization
G00  X_ ;             Positions the workpiece at the machining position.

Machining in the synchronous state

G00  X_ ;             Retract the workpiece from the tool.
G81  T_  L_  R1 ;     Synchronization ratio change.
G00  X_ ;             Positions the workpiece at the machining position.

Machining in the synchronous state

G00  X_ ;             Retract the workpiece from the tool.
G80  R1  ;            Cancel synchronization
```

- Acceleration/deceleration plus automatic phase synchronization type

```
M03  ;  Clockwise spindle rotation command
G00  G90  C_ ;        C-axis positioning
G81  T_  L_  R2 ;     Start of synchronization.
G00  X_ ;             Positions the workpiece at the machining position.

Machining in the synchronous state

G00  X_ ;             Retract the workpiece from the tool.
G81  T_  L_  R2 ;     Synchronization ratio change.
G00  X_ ;             Positions the workpiece at the machining position.

Machining in the synchronous state

G00  X_ ;             Retract the workpiece from the tool.
G80  R2  ;            Cancel synchronization
```

21.6.4 Skip Function for EGB Axis

Overview

This function enables the skip or high-speed skip signal (these signals are collectively called skip signals in the remainder of this manual) for the EGB slave axis in synchronization mode with the EGB (electronic gear box).
This function has features such as the following:

1. If a skip signal is input while an EGB axis skip command block is being executed, this block does not terminate until the specified number of skip signals have been input.
2. If a skip signal is input while an EGB axis skip command block is being executed, the tool remains in synchronous mode and the EGB slave axis is not stopped and keeps moving.
3. The machine coordinates recorded when skip signals are input and the number of input skip signals are stored in specified custom macro variables.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Format

G81 T_ L_ ; **EGB mode ON**

G31.8 G91 α0 P_ Q_ (R_) ; **EGB skip command**

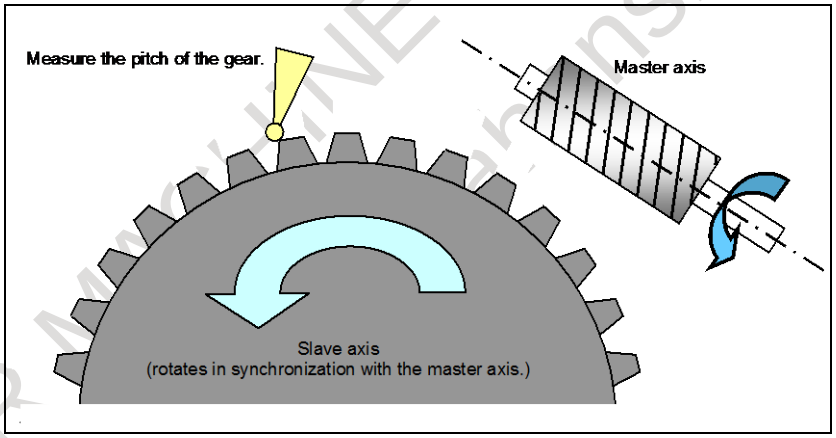
α: Specify an EGB slave axis. The specified value must always be "0".
P: Number of the first one of the custom macro variables used to store the machine coordinates recorded when skip signals are input.
Q: Number of skip signals that can be input during the execution of G31.8 (permissible range: 1 to 512).
R: Number of the custom macro variable used to store the number of input skip signals.
Specify it to check the number of input signals.

Explanation

G31.8 is a one-shot G code.
After the execution of G31.8, values of machine coordinate which is gotten at each time of skip signal input are set in custom macro variables. The numbers of variables are used from the top number commanded by P to the number added with the amount of times commanded by Q.
And the total times of skip signal input is set in the custom macro variable whose number is commanded by R.

Example

The pitch of a gear can be measured.




```
G81 T200 L2 ; ..................................EGB mode ON
X_ ;
Z_ ;
G31.8 G91 C0 P500 Q200 R1 ;...........EGB skip command
```

After 200 skip signals have been input, the 200 skip positions on the C-axis that correspond to the respective skip signals are stored in custom macro variables #500 to #699.
Also, the number of input skip signals is stored in custom macro variable #1.

NOTE

1 When specifying this function, specify only a single EGB slave axis. If no axis is specified for two or more axes are specified, alarm PS1152, "G31.9/G31.8 FORMAT ERROR" is generated.
2 If P is not specified, alarm PS1152 is generated.
3 If R is not specified, the number of input skip signals is not written to a custom macro variable.

FRYER MACHINE SYSTEMS INC www.fryerinsky.com

NOTE

4 The custom macro variable numbers specified in P and R must be existing ones. If a non-existent variable number is specified, alarm PS0115, "VARIABLE NO. OUT OF RANGE" is generated.
If a variable shortage occurs, alarm PS0115 is generated.

5 Whether to use conventional skip signals or high-speed skip signals with this function can be specified with bit 4(HSS) of parameter No. 6200. If high-speed skip signals are used, specify which high-speed signals to enable with bits 0 to 7(9S1 to 9S8) of parameter No. 6208.

6 Skip positions are calculated from feedback pulses from the machine. Thus, errors due to delay in acceleration/deceleration and the servo system are small.

21.6.5 Electronic Gear Box 2 Pair

Overview

This function enables machining of high-precision gears, screws, and other components by rotating the workpiece in synchronization with a rotating tool or by moving the tool in synchronization with a rotating workpiece. The rate of synchronization can be specified with a program.

Up to two sets of axes can be synchronized. A gear grinding machine can be controlled, for instance, by using one axis for rotating the workpiece in synchronization with the tool and another axis for performing dressing in synchronization with the tool.

The specification method differs depending on the configuration of the machine. For details, refer to the relevant manual provided by the machine tool builder.

In the subsequent explanation, the Electronic Gear Box is called the EGB.

21.6.5.1 Specification method (G80.5, G81.5)

Format

G81.5 $\left\{\begin{matrix}\mathbf{T}\,t\\ \mathbf{P}\,p\end{matrix}\right\}$ $\left\{\begin{matrix}\beta\,j\\ \beta\,0\ \ \mathbf{L}\,l\end{matrix}\right\}$ **;** Start of synchronization

↑ Amount of travel along the master axis

↑ Amount of travel along the slave axis

G80.5 β0 ; Cancellation of synchronization

Explanation

- Master axis, slave axis, and dummy axis

The synchronization reference axis is called the master axis, while the axis along which movement is performed in synchronization with the master axis is called the slave axis. For example, if the workpiece moves in synchronization with the rotating tool as in a hobbing machine, the tool axis is the master axis and the workpiece axis is the slave axis.

Which axes to become the master and slave axes depends on the configuration of the machine. For details, refer to the manual issued by the machine tool builder.

A single servo axis is used exclusively so that digital servo can directly read the rotation position of the master axis. (This axis is called the EGB dummy axis.)

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Start of synchronization

When the ratio of the master-axis travel to the slave-axis travel is specified, synchronization starts.

Specify the master-axis travel in either of the following ways.

1 Master-axis speed
T t: Master-axis speed (1≤ t ≤5000)

2 Master-axis pulse count
P p : Master-axis pulse count (1≤ p ≤999999999)
For an A/B phase detector, specify the count as four pulses corresponding with one A/B phase cycle.

Specify the slave-axis travel in either of the following ways.

1 Slave-axis travel
βj : Slave-axis address
j : Slave-axis travel indicated in units of the least command increment (the maximum specifiable range is equal to usual axis)
When j = 0, the specified command is regarded as being a command for the slave-axis speed, described below. In this case, if L is not specified, an alarm is output.

2 Slave-axis speed
β0 L±*l*
β : Slave-axis address
l : Slave axis speed(-250≤ *l* ≤250, but *l* = 0 is excluded.)

- Cancellation of synchronization

1 Canceling synchronization for each axis by issuing a command
With a G80.5 β0 command, synchronization is canceled.
β is the address of the slave axis. Synchronization of the slave axis specified by β is canceled. A cancellation command can be issued only for one axis in one block.
When β0 is not specified, the synchronization of all currently synchronized axes is canceled.
When cancellation of synchronization is issued, the absolute coordinate on the workpiece axis is updated in accordance with the amount of travel during synchronization. For a rotation axis, the amount of travel during synchronization is rounded within 360-degree and added to the absolute coordinate.

2 Canceling synchronization by a reset
By setting bit 0(HBR) of parameter No. 7700 to 0, synchronization is canceled by a reset.

3 Others
Synchronization is automatically canceled under the following conditions.
<1> An emergency stop is applied.
<2> A servo alarm is generated.
<3> Alarm (PW0000) “POWER MUST BE OFF” is generated.
<4> An IO alarm is generated.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

⚠ CAUTION

1 Feed hold, interlock, and machine lock are invalid to a slave axis in EGB synchronization.
2 Even if an OT alarm is issued for a slave axis in EGB synchronization, synchronization will not be canceled.
3 During synchronization, it is possible to execute a move command for a slave axis and other axes, using a program. The move command for a slave axis must be an incremental command.

NOTE

1 If bit 0 (HBR) of parameter No. 7700 is set to 1, EGB synchronization will not be canceled due to a reset. Usually, set this parameter bit to 1.
2 In synchronous mode, it is not possible to specify G27, G28, G29, G30, and G53 for a slave axis.
3 It is not possible to use controlled axis detach for a slave axis.
4 During synchronization, manual handle interruption can be performed on the slave and other axes.
5 In synchronization mode, no inch/metric conversion commands (G20 and G21) can be issued.
6 In synchronous mode, only the machine coordinates on a slave axis are updated.
7 If bit 0 (EFX) of parameter No. 7731 is 0, no canned cycle for drilling can be used. To use a canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1.
8 If, during synchronization, G81.5 is issued again, alarm PS1595, ”ILL-COMMAND IN EGB MODE” is issued if bit 3 (ECN) of parameter No. 7731 is 0. If bit 3 (ECN) of parameter No. 7731 is 1, the synchronization coefficient can be changed to a newly specified one.
9 Actual cutting feedrate display does not take synchronization pulses into consideration.
10 For an EGB slave axis, synchronous and composite control cannot be executed.
11 In EGB synchronization mode, AI contour control mode is temporarily canceled.
12 Not advanced preview feed-forward but conventional feed-forward is enabled in the path where EGB synchronization mode is effective.

21.6.5.2 Description of commands compatible with those for a hobbing machine (G80, G81)

A command compatible with that for a hobbing machine can be used as a synchronization command.
Usually, a hobbing machine performs machining by synchronizing the workpiece axis (usually, the C-axis) to the hobbing axis (spindle).
If there are two synchronization sets with the EGB, which synchronization set to start with this specification method can be specified with parameter No. 7710.

FRYER MACHINE SYSTEMS INC
Rob Zdrensky

Format

	Bit 0(EFX) of parameter No.7731=0	Bit 0(EFX) of parameter No.7731=1	
		Bit 5(HBR) of parameter No.7731=1	Bit 5(HBR) of parameter No.7731=0
Start of synchronization	**G81 T_ (L_) (Q_ P_) ;**	**G81.4 R_ (L_) (Q_ P_) ;**	**G81.4 T_ (L_) (Q_ P_) ;**
Cancellation of synchronization	**G80 ;**	**G80.4 ;**	**G80.4 ;**
	(*1) (*4)	**(*2) (*4)**	**(*3) (*4)**

T(or R) : Number of teeth (Specifiable range: 1 to 5000)
L : Number of hob threads (Specifiable range: -250 to +250)
The sign of L determines the direction of rotation for the workpiece axis.
When L is positive, the direction of rotation for the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation for the workpiece axis is negative (- direction).
When L is 0, it depends on the setting of bit 3 (LZR) of parameter No. 7701.
If L is not specified, the number of hob threads is assumed to be1.
Q : Module or diametral pitch
Specify a module in the case of metric input.
(Unit: 0.00001mm, Specifiable range: 0.01 to 25.0mm)
Specify a diametral pitch in the case of inch input.
(Unit: 0.00001inch^{-1}, Specifiable range: 0.01 to 254.0 inch^{-1})
P : Gear helix angle
(Unit: 0.0001deg, Specifiable range: -90.0 to +90.0deg)
*1 Use it for machining centers.
*2 Use it for lathes.
*3 Use it for machining centers.
This format enables specification of the same G codes as those for lathes.
*4 When specifying Q and P, the user can use a decimal point.

NOTE
Specify G81, G80, G81.4, and G80.4 in a single block.

Explanation

- Start of synchronization

Specify P and Q to use helical gear compensation. In this case, if only one of P and Q is specified, alarm PS1594, ”EGB FORMAT ERROR” is generated.
When G81 is issued so that the machine enters synchronization mode, the synchronization of the workpiece axis to the spindle is started.
During synchronization, the rotation about the tool and workpiece axes is controlled so that the relationship between T (number of teeth) and L (number of hob threads) is maintained.
If, during synchronization, G81 is issued again without synchronization cancellation, alarm PS1595, ”ILL-COMMAND IN EGB MODE” is generated if bit 3(ECN) of parameter No. 7731 is 0. If bit 3(ECN) of parameter No. 7731 is 1, helical gear compensation is performed with the synchronization coefficient being changed to the one newly specified with T and L commands if T and L commands are issued. If T and L commands are not issued and only P and Q commands are issued, helical gear compensation is performed with the synchronization coefficient kept intact. This allows consecutive machining of helical gears and spur gears.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Cancellation of synchronization

Synchronization of all synchronized axes is canceled.
When cancellation of synchronization is issued, the absolute coordinate on the workpiece axis is updated in accordance with the amount of travel during synchronization.
For a rotation axis, the amount of travel during synchronization is rounded within 360-degree and added to the absolute coordinate.
In the G80 block, only O and N addresses can be specified.

⚠ CAUTION

1 Feed hold, interlock, and machine lock are invalid to a slave axis in EGB synchronization.
2 Even if an OT alarm is issued for a slave axis in EGB synchronization, synchronization will not be canceled.
3 During synchronization, it is possible to execute a move command for a slave axis and other axes, using a program. The move command for a slave axis must be an incremental command.

NOTE

1 If bit 0 (HBR) of parameter No. 7700 is set to 1, EGB synchronization will not be canceled due to a reset. Usually, set this parameter bit to 1.
2 In synchronous mode, it is not possible to specify G27, G28, G29, G30, and G53 for a slave axis.
3 It is not possible to use controlled axis detach for a slave axis
4 During synchronization, manual handle interruption can be performed on the slave and other axes.
5 In synchronization mode, no inch/metric conversion commands (G20 and G21) cannot be issued.
6 In synchronous mode, only the machine coordinates on a slave axis are updated.
7 If bit 0 (EFX) of parameter No. 7731 is 0, no canned cycle for drilling can be used. To use a canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.
8 If bit 0 (TDP) of parameter No. 7702 is 1, the permissible range of T is 0.1 to 500 (1/10 of the specified value).
9 If, at the start of EGB synchronization (G81), L is specified as 0, synchronization starts with L assumed to be 1 if bit 3 (LZR) of parameter No. 7701 is 0; if bit 3 (LZR) of parameter No. 7701 is 1, synchronization is not started with L assumed to be 0. At this time, helical gear compensation is performed.
10 Feed per revolution is performed on the feedback pulses on the spindle. By setting bit 0 (ERV) of parameter No. 7703 to 1, feed per revolution can be performed based on the speed on the synchronous slave axis.
11 Actual cutting feedrate display does not take synchronization pulses into consideration.
12 For an EGB slave axis, synchronous and composite control cannot be executed.
13 In EGB synchronization mode, AI contour control mode is temporarily canceled.
14 Not advanced preview feed-forward but conventional feed-forward is enabled in the path where EGB synchronization mode is effective.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Helical gear compensation

For a helical gear, the workpiece axis is compensated for the movement along the Z axis (axial feed axis) based on the torsion angle of the gear.
Helical gear compensation is performed with the following formulas:

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (for metric input)}$$

$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (for inch input)}$$

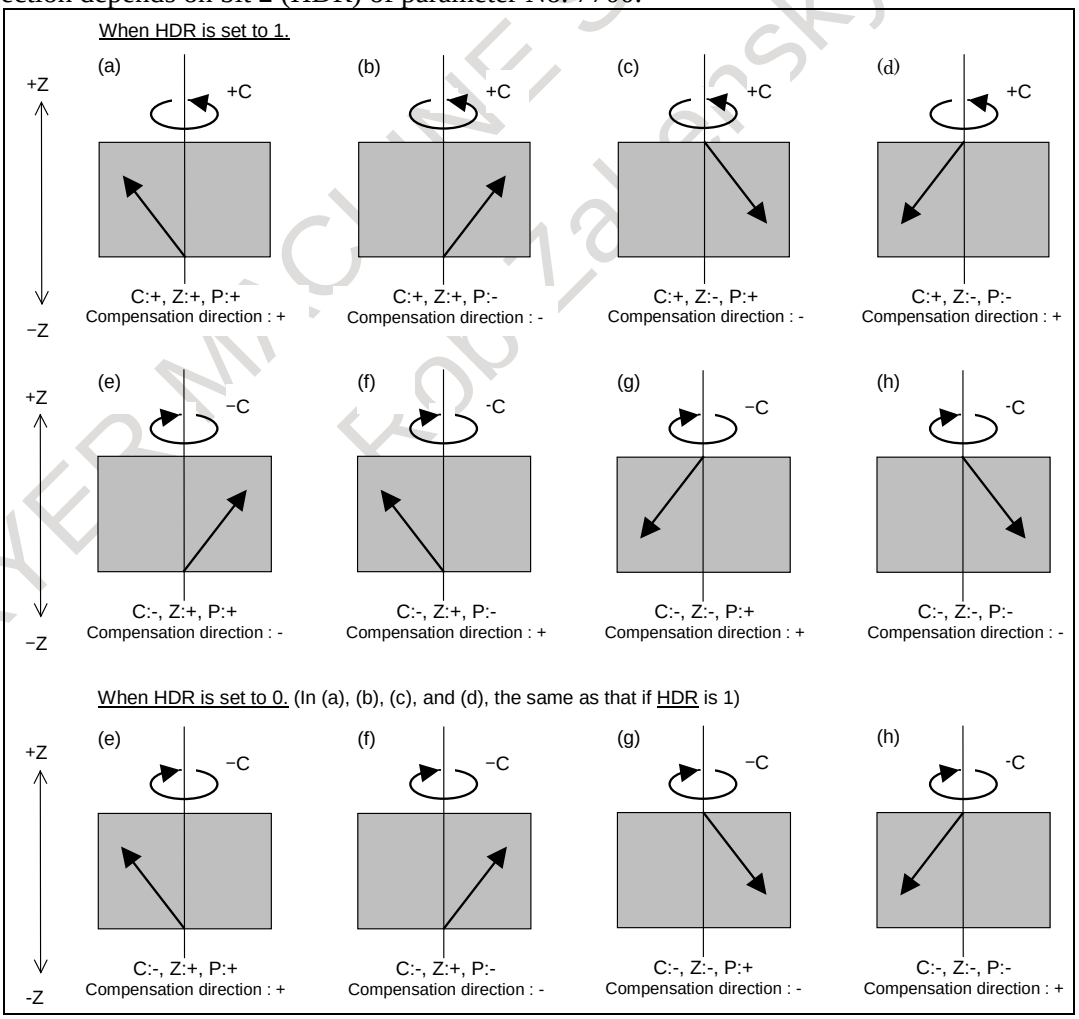
where

Compensation angle: Signed absolute value (deg)
Z : Amount of travel on the Z-axis after the specification of G81 (mm or inch)
P : Signed gear helix angle (deg)
π : Circular constant
T : Number of teeth
Q : Module (mm) or diametral pitch ($inch^{-1}$)
Use P, T, and Q specified in the G81 block.

In helical gear compensation, the machine coordinates on the workpiece axis and the absolute coordinates are updated with helical gear compensation.

- Direction of helical gear compensation

The direction depends on bit 2 (HDR) of parameter No. 7700.



FRYER MACHINE SYSTEMS INC
Fedor Zaharinsky

21.6.5.3 Controlled axis configuration example

- For gear grinders

Spindle : EGB master axis : Tool axis
1st axis : X axis
2nd axis : Y axis
3rd axis : C axis (EGB slave axis : Workpiece axis)
4th axis : C axis (EGB dummy axis : Cannot be used as a normal controlled axis)
5th axis : V axis (EGB slave axis : Dressing axis)
6th axis : V axis (EGB dummy axis : Cannot be used as a normal controlled axis)

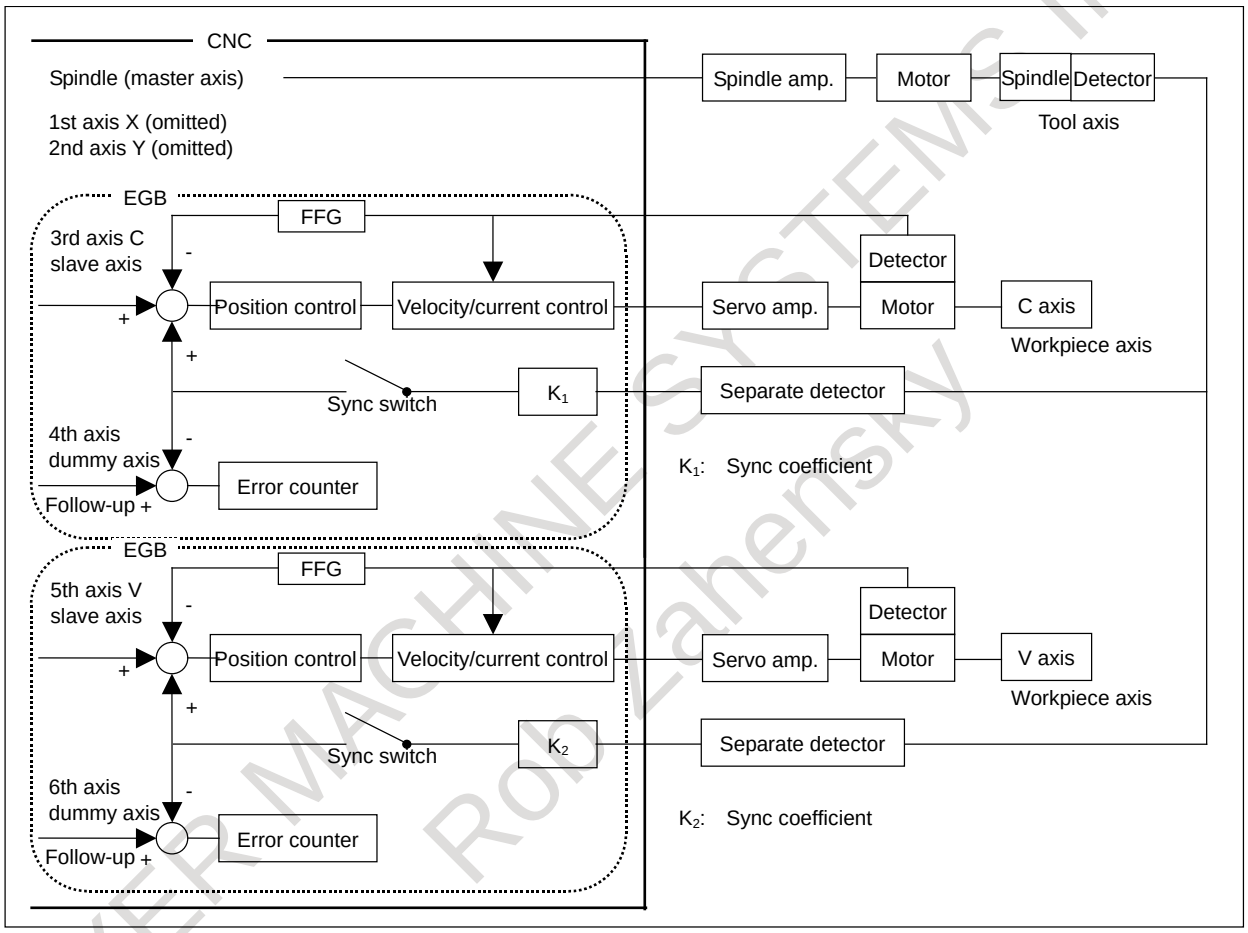


Fig. 21.6.5.3 Controlled axis configuration with a set of two EGBs

21.6.5.4 Sample programs

- When the master axis is the spindle, and the slave axis is the C-axis

(1) G81.5 T10 C0 L1 ;
Synchronization between the master axis and C-axis is started at the ratio of one rotation about the C-axis to ten rotations about the master axis.

(2) G81.5 T10 C0 L-1 ;
Synchronization between the master axis and C-axis is started at the ratio of one rotation about the C-axis to ten rotations about the master axis.
In this case, however, the direction of rotation is opposite to that of (1).

(3) G81.5 T1 C3.26 ;
Synchronization between the master axis and C-axis is started at the ratio of a 3.26-degree rotation about the C-axis per one rotation about the master axis.

FRYER MACHINE SYSTEMS INC
Rob Lahensky

(4) G81.5 P10000 C-0.214 ;
Synchronization between the master axis and C-axis is started at the ratio of a -0.214 degree rotation about the C-axis to 10,000 feedback pulses from the pulse coder of the master axis.

- When the master axis is the spindle, the slave axis is the V-axis (linear axis), and inch/metric conversion is performed

(1) For a millimeter machine and metric input
G81.5 T1 V1.0 ;
Synchronization between the master axis and V-axis is started at the ratio of a 1.00 mm movement along the V-axis per rotation about the master axis.

(2) For a millimeter machine and inch input
G81.5 T1 V1.0 ;
Synchronization between the master axis and V-axis is started at the ratio of a 1.0 inch movement (25.4 mm) along the V-axis per rotation about the master axis.

- When two groups of axes are synchronized simultaneously

Based on the controlled axis configuration described in Fig. 21.6.5.3 , the sample program below synchronizes the spindle with the V-axis while the spindle is synchronized with the C-axis.

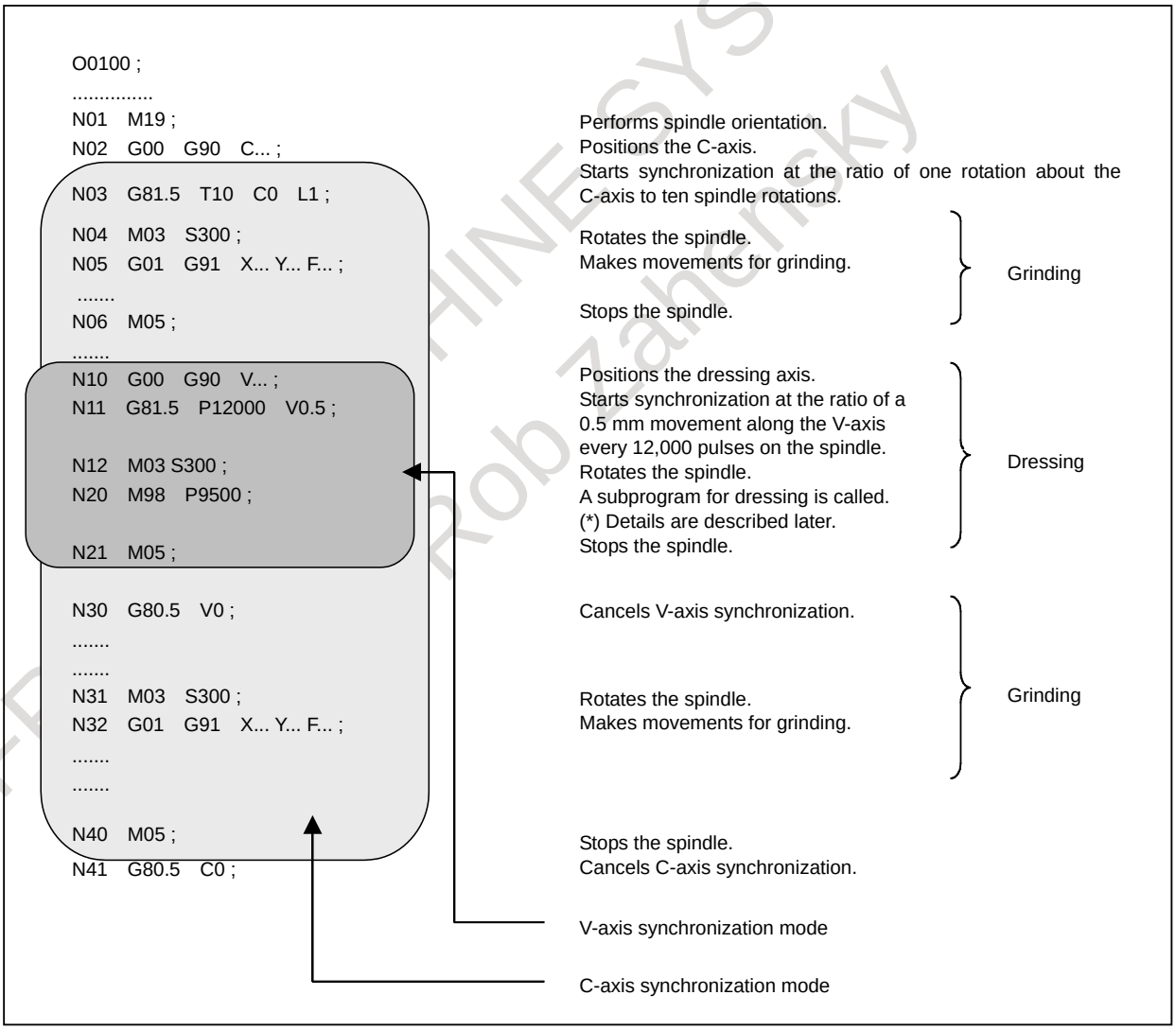
```
O0100 ;
...............
N01   M19 ;                          Performs spindle orientation.
N02   G00   G90   C... ;             Positions the C-axis.
N03   G81.5   T10   C0   L1 ;        Starts synchronization at the ratio of one rotation about the
                                     C-axis to ten spindle rotations.
N04   M03   S300 ;                   Rotates the spindle.
N05   G01   G91   X... Y... F... ;   Makes movements for grinding.        Grinding
 .......
N06   M05 ;                          Stops the spindle.
.......
N10   G00   G90   V... ;             Positions the dressing axis.
N11   G81.5   P12000   V0.5 ;        Starts synchronization at the ratio of a
                                     0.5 mm movement along the V-axis
                                     every 12,000 pulses on the spindle.
N12   M03 S300 ;                     Rotates the spindle.                   Dressing
N20   M98   P9500 ;                  A subprogram for dressing is called.
                                     (*) Details are described later.
N21   M05 ;                          Stops the spindle.

N30   G80.5   V0 ;                   Cancels V-axis synchronization.
.......
.......
N31   M03   S300 ;                   Rotates the spindle.                   Grinding
N32   G01   G91   X... Y... F... ;   Makes movements for grinding.
.......
.......

N40   M05 ;                          Stops the spindle.
N41   G80.5   C0 ;                   Cancels C-axis synchronization.

                                     V-axis synchronization mode

                                     C-axis synchronization mode
```

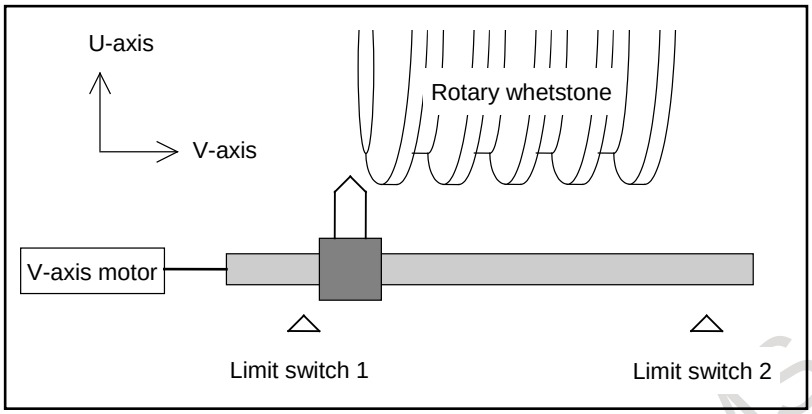


Thus, the synchronizations of each group can be started and canceled independently.

FR... HINE SYSTEMS INC
Rob Zahensky

- Example of use of dressing

Gear grinder in the following machine configuration




```
O9500 ;
N01   G01   G91   U_   F100 ;   Dressing axis approach
N02   M03   S100 ;              The M03 command causes the PMC to rotate the whetstone in the
                                positive direction.
                                In accordance with this, the tool moves along the V-axis in the
                                positive direction. When the tool reaches the position of limit switch
                                2 on the V-axis, the PMC stops the whetstone and returns FIN.
N02   U_   V_ ;                 Movement to the next dressing position
N03   M04   S100 ;              The M04 command causes the PMC to rotate the whetstone in the
                                negative direction.
                                In accordance with this, the tool moves along the V-axis in the
                                negative direction. When the tool reaches the position of limit switch
                                1 on the V-axis, the PMC stops the whetstone and returns FIN.
N04   U_ V_ ;                   Movement to the next dressing position
                                If required, N02 to N04 are repeated to conduct dressing.
..........
..........

M99 ;
```

NOTE

If the V-axis (linear axis) is synchronized with the spindle as in dressing, the V-axis travel range is determined by the rotation of the spindle. To perform dressing with the tool moving back and forth along the V-axis in a certain range, therefore, the PMC must perform an operation in which the tool is stopped temporarily and is reversed when it reaches a certain position on the V-axis.
In the above example, limit switches are provided to determine the range of travel along the V-axis and the PMC performs control so that the whetstone rotates until the tool reaches the position of each limit switch on the V-axis.
By using the position switch function instead of limit switches, dressing can be performed as in the following example, without the need to mount limit switches to the machine. By rewriting the operating ranges of the position switches (parameters Nos. 6930 to 6945 and Nos. 6950 to 6965) using the G10 programmable parameter input, the range of travel along the V-axis can be specified using a program.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Command specification for hobbing machines

Based on the controlled axis configuration described in Fig. 21.6.5.3 , the sample program below sets the C-axis (in parameter 7710) for starting synchronization with the spindle according to the command specification method for hobbing machines.

```
O1234 ;
N0010   M19   ;              Tool axis orientation
N0020   G28   G91   C0   ;   Reference position return on the work axis
N0030   G81   T20   L1 ;     Start of synchronization of the tool axis and the C-axis
                             (Rotation about the C-axis by 18° per rotation about the tool axis)
N0040   S300   M03 ;         Rotation about the tool axis at 300 min-1.
N0050   G01   X_   F_ ;      Movement along the X-axis (cutting).
N0060   G01   Y_   F_ ;      Movement along the Y-axis (grinding).
      ;                      If required, C, X, Y, and other axis command are possible.
      ;
N0100   G01   X_   F_ ;      Movement along the X-axis (escape)
N0110   M05 ;                Stop on the tool axis.
N0120   G80 ;                Cancellation of synchronization of the tool and C-axes.
N0130   M30 ;
```

21.6.5.5 Synchronization ratio specification range

The programmed ratio (synchronization ratio) of a movement along the slave axis to a movement along the master axis is converted to a detection unit ratio inside the NC automatically. If such converted data (detection unit ratio) exceeds a certain allowable data range in the NC, synchronization cannot be maintained correctly, and an alarm PS1596, ”EGB OVERFLOW” is issued.
Even when a programmed movement of master axis and a slave axis is within specifiable ranges, a detection unit ratio obtained by conversion can exceed the allowable range. This causes an alarm.
Let K be a synchronization ratio. The internal data corresponding K is the amount of slave axis movement (Kn) represented in the detection unit divided by the amount of master axis movement (Kd) represented in the detection unit;
this fraction is represented as Kn/Kd (reduced to its lowest terms) as indicated below.

$$K = \frac{Kn}{Kd} = \frac{\text{Amount of slave axis movement represented in the detection unit}}{\text{Amount of master axis movement represented in the detection unit}}$$

Kn and Kd must lie within the following ranges:

$-2147483648 \le Kn \le 2147483647$
$1 \le Kd \le 2147483647$

When Kn or Kd exceeds its allowable range above, an alarm PS1596 is issued.
In conversion to the detection unit, when the CMR (command multiplication: parameter 1820) is a fraction or when inch/millimeter conversion is used, the fraction is directly converted without modification so that no error can occur in the conversion of specified amounts of movement.
During conversion, the amount of movement is multiplied by 254/100 for inch input on a millimeter machine, and 100/254 for metric input on an inch machine. Thus, Kn and Kd can become large numbers. If a synchronization ratio cannot be reduced to its lowest terms, an alarm condition is likely to occur.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Example 1)

Based on the controlled axis configuration described in Fig. 21.6.5.3 , suppose that the spindle and V-axis are as follows:

Spindle pulse coder : 72000pulse/rev (4 pulses for one A/B phase cycle)
C-axis least command increment : 0.001deg
C-axis CMR : 5
V-axis least command increment : 0.001mm
V-axis CMR : 5

Then, the C-axis detection unit is 0.0002 degree. The V-axis detection unit is 0.0002 mm. In this case, the synchronization ratio (Kn, Kd) is related with a command as indicated below. Here, let Pm and Ps be the amounts of movements represented in the detection unit on the master axis and slave axis specified in a synchronization start command, respectively.

(1) When the master axis is the spindle, and the slave axis is the C-axis

(a) Command : G81.5 T10 C0 L1 ;
Operation : Synchronization between the spindle and C-axis is started at the ratio of one rotation about the C-axis to ten spindle rotations.
Pm : (Number of pulses per spindle rotation) × (10 rotations) → 72000 × 10
Ps : (Amount of movement per rotation about the C-axis) × CMR × (one rotation) → 360000 × 5 × 1

$$\frac{Kn}{Kd} = \frac{360000 \times 5 \times 1}{72000 \times 10} = \frac{5}{2}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(b) Command : G81.5 T10 C0 L-1 ;
Operation : Synchronization between the spindle and C-axis is started at the ratio of one rotation about the C-axis to ten spindle rotations.
In this case, however, the direction of rotation is opposite to that of (a) above.
Pm : (Number of pulses per spindle rotation) × (10 rotations) → 72000 × 10
Ps : (Amount of movement per rotation about the C-axis) × CMR × (one rotation) → -360000 × 5 × 1

$$\frac{Kn}{Kd} = \frac{-360000 \times 5 \times 1}{72000 \times 10} = \frac{-5}{2}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(c) Command : G81.5 T1 C3.263 ;
Operation : Synchronization between the spindle and C-axis is started at the ratio of a 3.263-degree rotation about the C-axis to one spindle rotation.
Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1
Ps : (Amount of C-axis movement) × CMR → 3263 × 5

$$\frac{Kn}{Kd} = \frac{3263 \times 5}{72000 \times 1} = \frac{3263}{14400}$$

Both Kn and Kd are within the allowable range. No alarm is output.

In this sample program, when T1 is specified for the master axis, the synchronization ratio (fraction) of the CMR of the C-axis to the denominator Kd can always be reduced to lowest terms, thus Kd falls in the allowable range. So, the specifiable range of C is as follows:

$-99999999 \leq C \leq 99999999$

FRYER MACHINE SYSTEMS INC
Rob Zahensky

(d) Command : G81.5 T10 C3.263 ;
Operation : Synchronization between the spindle and C-axis is started at the ratio of a 3.263-degree rotation about the C-axis to ten spindle rotations.
Pm : (Number of pulses per spindle rotation) × (10 rotations) → 72000 × 10
Ps : (Amount of C-axis movement) × CMR → 3263 × 5

$$\frac{Kn}{Kd} = \frac{3263 \times 5}{72000 \times 10} = \frac{3263}{144000}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(e) Command : G81.5 P10000 C-0.214 ;
Operation : Synchronization between the spindle and C-axis is started at the ratio of a -0.214 degree rotation of the C-axis to 10,000 feedback pulses from the pulse coder of the spindle.
Pm : (Specified number of feedback pulses from the pulse coder of the spindle) → 10000
Ps : (Amount of C-axis movement) × CMR → -214 × 5

$$\frac{Kn}{Kd} = \frac{-214 \times 5}{10000} = \frac{-107}{1000}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(2) When the master axis is the spindle, the slave axis is the V-axis (linear axis), and inch/metric conversion is performed

(a) For a millimeter machine and metric input
Command : G81.5 T1 V1.0 ;
Operation : Synchronization between the spindle and V-axis is started at the ratio of a 1.0 mm movement along the V-axis per spindle rotation.
Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1
Ps : (Amount of V-axis movement) × CMR → 1000 × 5

$$\frac{Kn}{Kd} = \frac{1000 \times 5}{72000} = \frac{5}{72}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(b) For a millimeter machine and inch input
Command : G81.5 T1 V1.0 ;
Operation : Synchronization between the spindle and V-axis is started at the ratio of a 1.0inch (25.4mm) movement along the V-axis per spindle rotation.
Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1
Ps : (Amount of V-axis movement) × CMR × 254 ÷ 100 → 10000 × 5 × 254 ÷ 100

$$\frac{Kn}{Kd} = \frac{10000 \times 5 \times 254}{72000 \times 100} = \frac{127}{72}$$

Both Kn and Kd are within the allowable range. No alarm is output.

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

(c) For a millimeter machine and inch input

Command : G81.5 T1 V0.0013 ;

Operation : Synchronization between the spindle and V-axis is started at the ratio of a 0.0013 inch (0.03302 mm) movement along the V-axis per spindle rotation.

Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1

Ps : (Amount of V-axis movement) × CMR × 254 ÷ 100 → 13 × 5 × 254 ÷ 100

$$\frac{Kn}{Kd} = \frac{13 \times 5 \times 254}{72000 \times 100} = \frac{1651}{720000}$$

Both Kn and Kd are within the allowable range. No alarm is output.

- Example 2)

Based on the controlled axis configuration described in Fig. 21.6.5.3 , suppose that the spindle and V-axis are as follows:

Spindle pulse coder : 72000pulse/rev (4 pulses for one A/B phase cycle)
C-axis least command increment : 0.001deg
C-axis CMR : 1/2
V-axis least command increment : 0.001mm
V-axis CMR : 1/2

Then, the C-axis detection unit is 0.002 degree. The V-axis detection unit is 0.002 mm.

In this case, the synchronization ratio (Kn, Kd) is related with a command as indicated below. Here, let Pm and Ps be the amounts of movements represented in the detection unit for the master axis and slave axis specified in a synchronization start command, respectively.

(1) When the master axis is the spindle, and the slave axis is the C-axis

(a) Command : G81.5 T1 C3.263 ;

Operation : Synchronization between the spindle and C-axis is started at the ratio of a 3.263-degree rotation about the C-axis per spindle rotation.

Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1

Ps : (Amount of C-axis movement) × CMR → 3263 × 1 ÷ 2

$$\frac{Kn}{Kd} = \frac{3263 \times 1}{72000 \times 2} = \frac{3263}{144000}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(b) Command : G81.5 T1 C3.26 ;

The value specified for C slightly differs from that specified for C in (a).

Operation : Synchronization between the spindle and C-axis is started at the ratio of a 3.26-degree rotation about the C-axis per spindle rotation.

Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1

Ps : (Amount of C-axis movement) × CMR → 3260 × 1 ÷ 2

$$\frac{Kn}{Kd} = \frac{3260 \times 1}{72000 \times 2} = \frac{163}{7200}$$

Both Kn and Kd are within the allowable range. No alarm is output.

21.6.5.6 Retract function

See "Retract function" in the Subsection 21.6.1, "Electronic Gear Box".

FRYER MACHINE SYSTEMS INC
Rob Zahensky

21.6.6 U-axis Control

Overview

Conventionally, the control of an axis on a spindle, such as the U-axis of a vertical lathe, from a motor mounted in a location other than the spindle has required a mechanism, consisting of a planetary gear box and differential gears, to prevent the axis from moving as the spindle is rotating.
The U-axis control function enables the U-axis to remain in a fixed position or to move at a programmed speed without using a mechanism such as a planetary gear box. This is done by causing the U-axis motor to rotate in such a way that U-axis movement, which would be caused by the rotation of the spindle, is canceled out. For details, refer to the manual supplied by the machine tool builder.

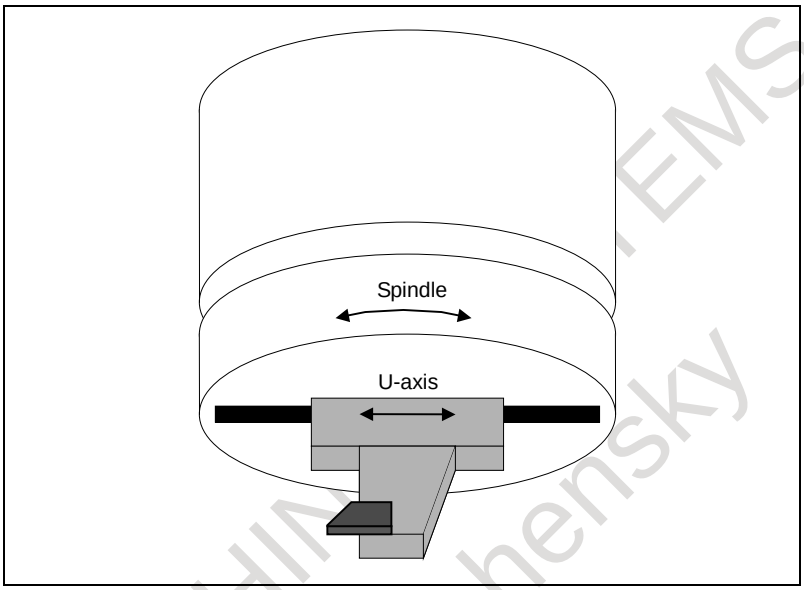


Fig. 21.6.6 (a) Example of a machine having the U-axis

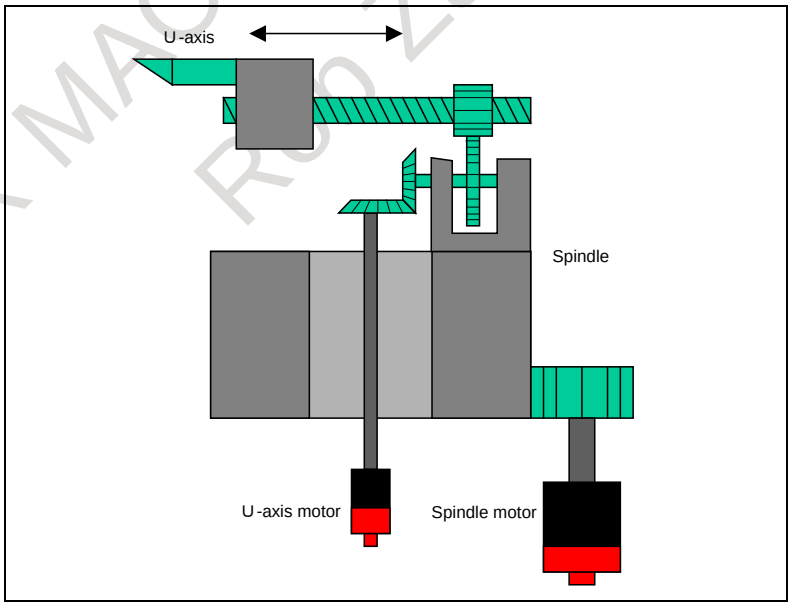


Fig. 21.6.6 (b) Example of the structure of a machine having the U-axis

In the example of the above structure, the tool moves along the U-axis when the spindle rotates. This movement is canceled out by rotating the U-axis motor.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

21.7 TANDEM CONTROL

When enough torque for driving a large table cannot be produced by only one motor, two motors can be used for movement along a single axis. Positioning is performed by the main motor only. The sub motor is used only to produce torque. With this tandem control function, the torque produced can be doubled.

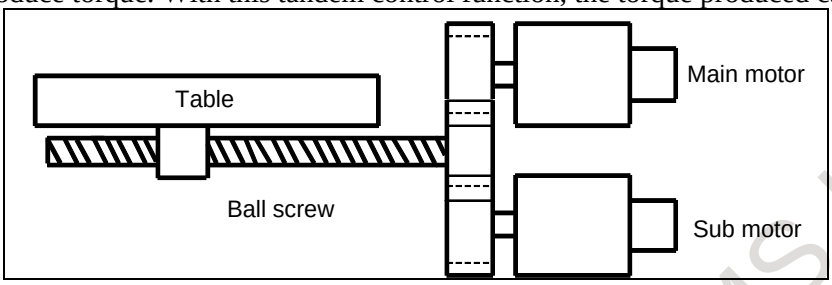


Fig. 21.7 Example of operation

In general, the NC regards tandem control as being performed for one axis. However, for servo parameter management and servo alarm monitoring, tandem control is regarded as being performed for two axes.
For details, refer to the relevant manual published by the machine tool builder.

> ⚠ **WARNING**
> In case the names of the master axis and slave axis are different, be sure not to specify a programmed command for the slave axis.

21.8 SKIP FUNCTION FOR FLEXIBLE SYNCHRONIZATION CONTROL

Outline

This function enables the skip or high-speed skip signal (in the following explanation, these signals are collectively called skip signal) for the slave axis that is moved by command of the master axis in the flexible synchronization control mode.
This function has features such as the following:

- If a skip signal is input while a skip command for flexible synchronization control block is being executed, this block does not terminate until the specified number of skip signals have been input.
- The machine coordinates assumed when skip signals are input and the number of input skip signals are stored in specified custom macro variables.
- The total number of the skip signal inputs is stored in another specified custom macro variable.

Format

Mxx ; **Flexible synchronization control mode on**

G31.8 G91 α 0 P_ Q_ R_ ; **Skip command for flexible synchronization control**

α : Specify the slave axis. The instruction value must be 0.
P_: The top number of the consecutive custom macro variables in which the machine coordinate positions of the slave axis at the skip signal inputs are stored.
Q_: The maximum allowable number of the skip signal inputs while G31.8 is running. (Range of command value: 1 to 512)
R_: The number of the custom macro variables in which the total number of the inputs is stored.
This data is usually the same as the value specified by Q.
Therefore this is not necessarily specified. Specify it to check the number of skip signal inputs.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

G31.8 is a one-shot G code.
During the execution of the G31.8 block, the machine coordinate positions of the slave axis at the skip signal inputs are stored in the consecutive custom macro variables where the top number of the variables is specified by P and the maximum allowable number of the skip signal inputs is specified by Q.
Also, this total number of the skip signal inputs is stored in the variable specified by R.

Example)

Mxx	Flexible synchronization control mode on
X--	
Y--	
G31.8 G91 A0 P100 Q30 R1	Skip command for flexible synchronization control

After 30 times of skip signal inputs, 30 machine coordinate positions of the A axis are stored respectively in the consecutive custom macro variables #100 to #129.
The total number of skip signal inputs is stored in the custom macro variable #1.

NOTE

1 In the G31.8 block, only one slave axis should be commanded. When more than two slave axes are specified, the alarm PS1152, "G31.9/G31.8 FORMAT ERROR" occurs.
2 If G31.8 is commanded out of the flexible synchronization control mode (Flexible synchronization control mode select signal switching accepted signal (MFSYNA, MFSYNB, MFSYNC or MFSYND) is “0”), the alarm PS1152 occurs.
3 If P is not specified, the alarm PS1152 occurs.
4 If R is not specified, the number of skip signal inputs is not stored in the custom macro variables
5 The number of custom macro variables specified in P and R must be the existing ones. If any nonexistent variable is specified, the alarm PS0115, "VARIABLE NO. OUT OF RANGE" occurs. If a variable shortage occurs, the alarm PS0115 occurs, too.
6 Whether to use conventional skip signals or high-speed skip signals with this function can be specified with bit 4 (HSS) of parameter No. 6200. When high speed skip is selected, specify which high-speed signals to enable with setting bits 0 to 7 (9S1 to 9S8) of parameter No. 6208.
7 The accumulated pulses and positional deviation due to acceleration/deceleration is considered and compensated when storing the machine coordinate positions to the custom macro variables.
8 The custom macro is necessary to use this function (bit 5 (NMC) of parameter No.8135 is 0).

FRYER MACHINE SYSTEMS INC Rob Zanensky

21.9 HIGH PRECISION OSCILLATION FUNCTION

Overview

In this function, the feedrate of oscillation axis (equivalent to a chopping axis that is moved vertically and repeatedly for grinding) changes along sine curve. This function is effective to improve, the accuracy of movement between upper dead point and lower dead point.
Moreover, look-ahead feed forward function can be applied to oscillation motion, then higher accuracy can be achieved even if oscillation feedrate or upper or lower dead point is changed.

Format

G81.1 Z_ Q_ R_ F_ ;

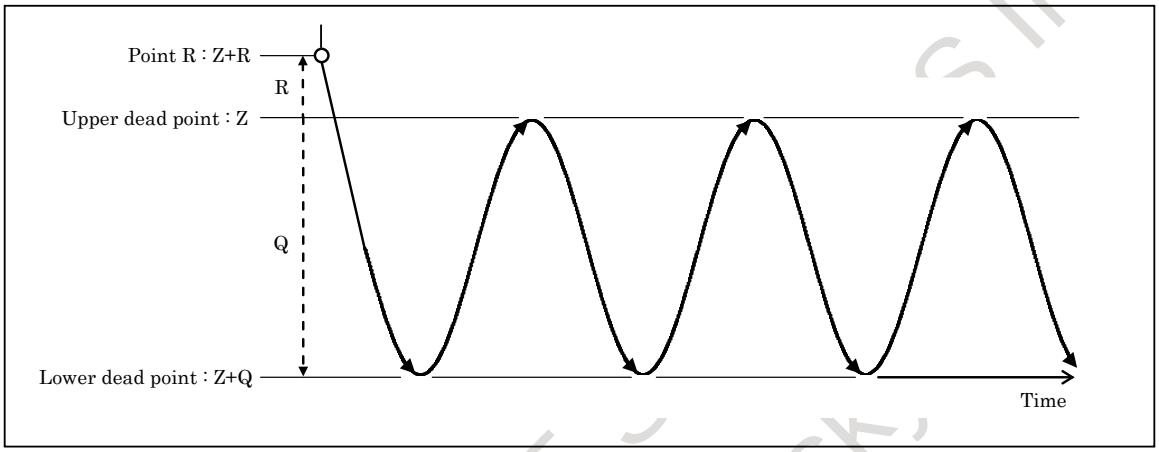
Z : Upper dead point (In case that an axis is other than the Z-axis, specify the axis address. Specify the distance as an absolute value.)

Q : Distance between the upper dead point and lower dead point
(Specify the distance as an incremental value from the upper dead point.)

R : Distance from the upper dead point to point R
(Specify the distance as an incremental value from the upper dead point.)

F : Oscillation base feedrate

G80 ; Cancels oscillation



If the addresses of Z, Q, R, or F are omitted, the oscillation motion is performed by the value of parameters. On the other hand, the value of parameters is replaced by the value commanded for each address.

Z (Axis address)	Parameter (No.8370)	: Oscillation axis number = Axis number.
	Parameter (No.8372)	: Upper dead point = Value of address Z.
Q	Parameter (No.8373)	: Lower dead point = Value of address Q + value of address Z (If address Z is omitted, the value is parameter (No.8372)).
R	Parameter (No.8371)	: Point R = Value of address R + value of address Z (If address Z is omitted, the value is parameter (No.8372)).
F	Parameter (No.8374)	: Oscillation base feedrate = Value of address F.

NOTE
1 G81.1/G80 is the G code for suppressing buffering.
2 Specify G81.1/G80 in a single block.

Explanation

- Oscillation activated by signal input

Before oscillation motion is started, oscillation axis, point R, upper dead point, lower dead point, and oscillation base feedrate must be set on the parameter screen or the oscillation screen. Oscillation motion is started once oscillation start signal CHPST has been set to "1" from "0". This signal is ignored, however, while oscillation axis is moving.

When oscillation hold signal *CHLD is set to "0" during oscillation motion, the tool immediately moves to point R and stops. Setting the oscillation hold signal *CHLD to "1" restarts oscillation motion. Oscillation motion can also be stopped by setting oscillation start signal CHPST to "0" from "1", but only when oscillation motion was started by using that signal.

Method of starting oscillation	Method of cancelling oscillation	State
Signal CHPST ="1"	Signal *CHLD = "0"	It stops. (Suspending state)
	Signal CHPST = "0"	It stops. (Cancel state)
	G80 command	It stops. (Cancel state)
G81.1 command	Signal *CHLD = "0"	It stops. (Suspending state)
	Signal CHPST = "0"	It doesn't stop.
	G80 command	It stops. (Cancel state)

NOTE

1 Switching to manual mode or suspending automatic operation, by means of feed hold, does not stop oscillation motion.
2 If a move command is specified for the oscillation axis while oscillation is being performed, an alarm PS5050, "ILL-COMMAND IN G81.1 MODE" is issued.
3 If a G81.1 command is specified during oscillation started by the signal, oscillation motion is not stopped. If point R, upper dead point, lower dead point, or oscillation feedrate has been modified by using the G81.1 command, oscillation is continued but using the modified data.
4 The use of oscillation start signal CHPST to start oscillation motion is not enabled immediately after power-on; it is not enabled until the completion of manual reference position return.
5 The M codes which are used to set the oscillation start signal CHPST to "1" or "0" should be specified as buffering mask type M code (parameter Nos. 3411 to 3432).

⚠CAUTION

After the cancel of oscillation motion during NC automatic operation, to execute a move command (in the absolute or incremental mode) for the oscillation axis in an NC block during the same operation, use G-code (G80) for the cancel of oscillation.
If it is necessary to cancel by signal input, execute the following (1) or (2) after cancelling the oscillation.
(1) M code in which Each axis workpiece coordinate system preset signal WPRST is turned ON
(2) Workpiece coordinate system preset (G92.1)

For example, in the following program (O0001), to execute the move command (N60) after the cancel of oscillation motion (Z-axis) (N40), “M code in which each axis workpiece coordinate system preset signal WPRST is turned ON” or “Workpiece coordinate system preset (G92.1)” must be specified in N50.

```
O0001
N10 G94 G90 G00 X100.0
N20 M55                    ← Oscillation start signal is turned ON
N30 G01 X30.0 F30.0
N40 M56                    ← Oscillation start signal is turned OFF
                           ← “M code in which each axis workpiece
                             coordinate system preset signal WPRST for
N50 ____                     Z-axis is turned ON” or “Workpiece
                             coordinate system preset (G92.1Z0.0)” is
                             commanded.
N60 G00 Z-50.0             ← Move command of Z-axis
N70 M02
```

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Oscillation feedrate (feedrate of movement to point R)

From the start of oscillation motion to reaching point R, the tool moves at the rapid traverse rate (specified by parameter No.1420).

The override function can be used for either the normal rapid traverse rate or oscillation feedrate, one of which can be selected by setting bit 0 (ROV) of parameter No. 8360. When the oscillation feedrate is overridden, settings excess of 100% are clamped to 100%.

- Oscillation feedrate (feedrate of movement from point R)

From point R, reached after the start of oscillation motion, to the center point between upper and lower dead points, the tool moves at the oscillation base feedrate (F). Afterwards, between the center point and the point where the oscillation motion is canceled, the tool moves with sine curve feedrate by Exp. 1.

$$f(t)_{[mm/\min]} = k \times \mathrm{F}_{[mm/\min]} \times \sin\left(\frac{2}{|\mathrm{Q}|_{[1/mm]}} \times \frac{k \times \mathrm{F}}{60}_{[mm/s]} \times \frac{180}{\pi}_{[\deg/rad]} \times t_{[s]}\right) \quad \ldots\text{Exp. 1}$$

f(t) : Sine curve feedrate [mm/min]
F : Oscillation base feedrate [mm/min]
Q : Distance between the upper dead point and lower dead point [mm]
k : Oscillation feedrate override (0.0 (0%) to 1.5 (150%))
Set by Oscillation feedrate override signal *CHP1 to *CHP8
t : Time [s]

The value (*k*F) of the product of oscillation base feedrate and oscillation override is clamped to the maximum oscillation feedrate (parameter No.8375) if the value (*k*F) is greater than the maximum oscillation feedrate. However, even if the value (*k*F) is not greater than maximum oscillation feedrate, it is clamped by the feedrate never exceeds the maximum allowable acceleration rate for acceleration/deceleration (parameter No.25652) during oscillation motion.

- Setting oscillation data

Set the following oscillation data:

Oscillation axis…… Parameter (No.8370)
Reference point (point R)…… Parameter (No.8371)
Upper dead point……. Parameter (No.8372)
Lower dead point……. Parameter (No.8373)
Oscillation base feedrate…….. Parameter (No.8374)
Maximum oscillation feedrate… Parameter (No.8375)

All data items other than the oscillation axis and maximum oscillation feedrate can be set on the oscillation screen. Moreover, the value of parameters is replaced by the value of G81.1 command.

- Stopping oscillation

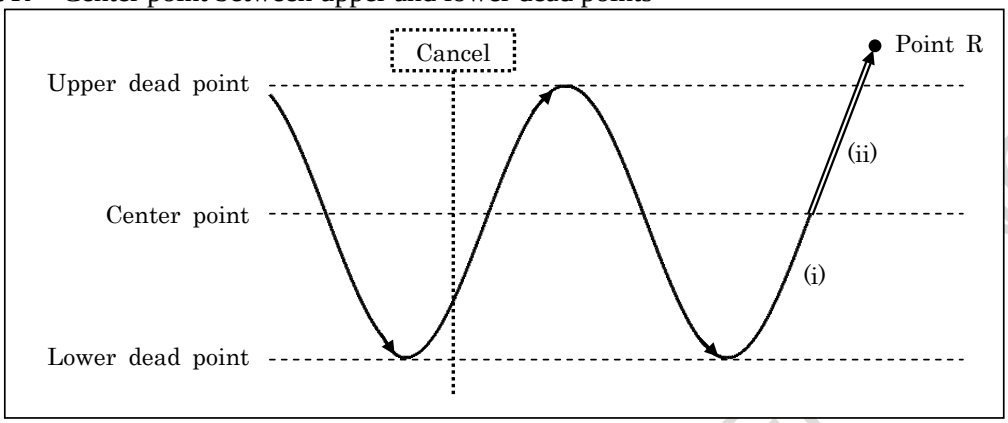
The following table lists the operations and commands that can be used to stop oscillation motion, the positions at which oscillation stops, and the operation performed after oscillation stops:

Operation/ command	Stop position	Operation after stops
G80 command	It follows parameter OST (No.25651#0). OST=0 : Point R OST=1 : Decelerated and stopped	Canceled
Reset		
PS, OT alarm		
CHPST: "0"	It follows parameters OST, SGS (No.25651#0,#2). OST=0 or SGS=0 : Point R OST=1 and SGS=1 : Decelerated and stopped	Canceled
*CHLD: "0"	It follows parameters OST, SGS, HST (No.25651#0,#2,#3). OST=0, SGS=0, or HST=0 : Point R OST=1, SGS=1, and HST=1 : Decelerated and stopped	Restart after *CHLD goes "1"
Emergency stop	The tool stops immediately.	Canceled
Servo alarm		

FRAER MACHINE SYSTEMS INC Rob Zahensky

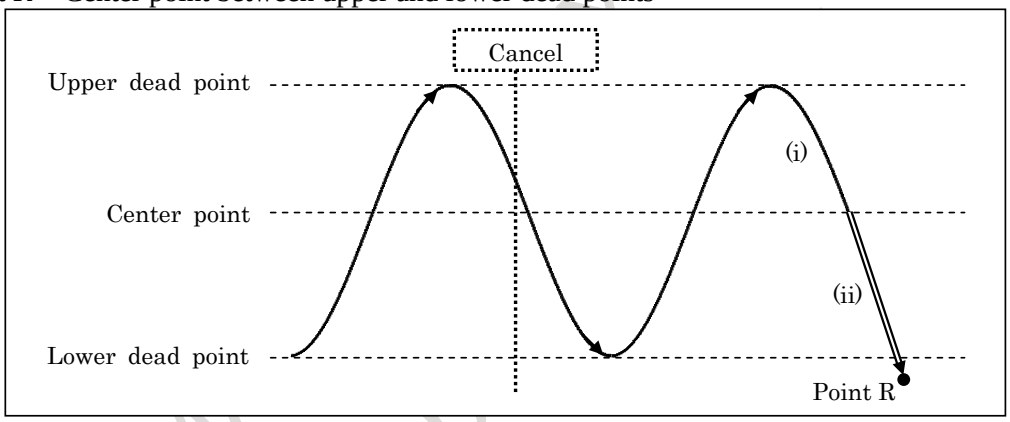
Example in the setting that stop position is point R. (In case of other than suspending by signal *CHLD = "0" and OT alarm.)

(1) Point R > Center point between upper and lower dead points



After cancel command, oscillation motion is continued until oscillation axis passes center point between upper and lower dead points via next lower dead point (i).
Afterwards, the axis moves to point R by linear interpolation (ii).

(2) Point R < Center point between upper and lower dead points



After cancel command, oscillation motion is continued until oscillation axis passes center point between upper and lower dead points via next upper dead point (i).
Afterwards, the axis moves to point R by linear interpolation (ii).

In the case of suspending by signal *CHLD = "0" or cancel by OT alarm, oscillation axis moves to point R at once.

NOTE
The timing of cancellation motion might change according to the position of point R or the setting of acceleration rate of cancellation motion.

- Movement from point R to the first dead point

The first dead point is decided by the position of point R.

Point R	The first dead point
Point R is on upper dead point side from center point or is on the center point.	Lower dead point
Point R is on lower dead point side from center point.	Upper dead point

- Omission of movements to point R and to the center point

By setting 1 to bit 0 (OST) of parameter No. 25651 and bit 1 (FFS) of parameter No. 25651, if oscillation motion is started by G81.1 command, movements to point R and to the center point can be omitted, and oscillation motion can be immediately started. In this case, the first dead point is as follows.

State before G81.1 command	The first dead point
In case of the first command of oscillation motion after the power is turned on or in the state of reset.	Lower dead point
In the state that command of cancel was specified on the way to lower dead point at last oscillation motion.	
In the state that command of cancel was specified on the way to upper dead point at last oscillation motion.	Upper dead point

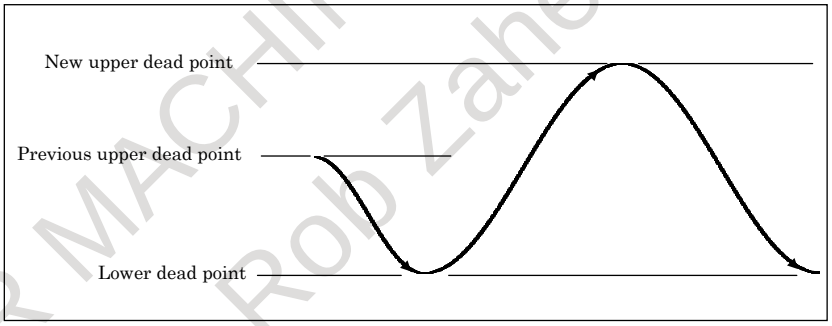
- Acceleration/deceleration

Linear acceleration/deceleration by parameter No.25653 is applied to the movement from point R to center point or the motion till stop.
Moreover, when override is changed during oscillation motion, the acceleration/deceleration set by parameter No.25653 is effective.
Acceleration/deceleration of sine curve feedrate control is enabled on oscillation motion.

- Oscillation after the upper dead point or lower dead point has been changed

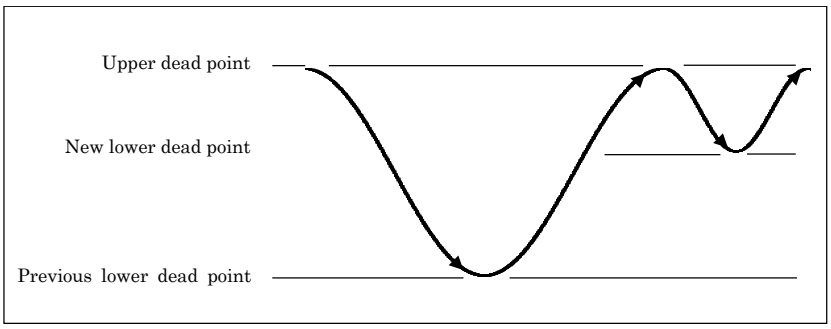
When the upper dead point or lower dead point is changed while oscillation motion is being performed, the tool moves to the dead points specified by the old data. Then, oscillation motion is changed to the new one using the new data.
The following describes the operations performed after the data has been changed.

(1) When the upper dead point is changed during movement from the upper dead point to the lower dead point



The tool first moves to the lower dead point, then to the new upper dead point.

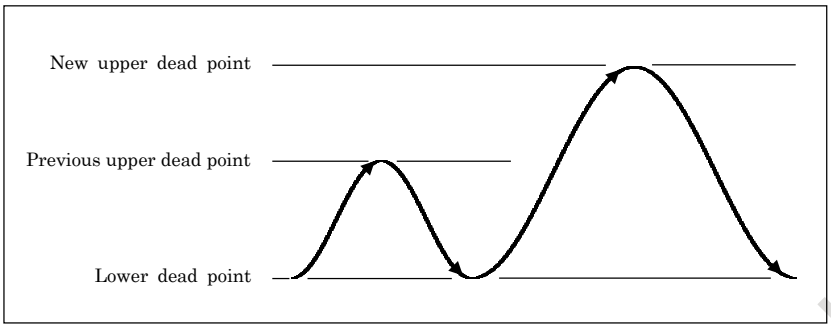
(2) When the lower dead point is changed during movement from the upper dead point to the lower dead point



The tool first moves to the previous lower dead point, then to the upper dead point, and finally to the new lower dead point.

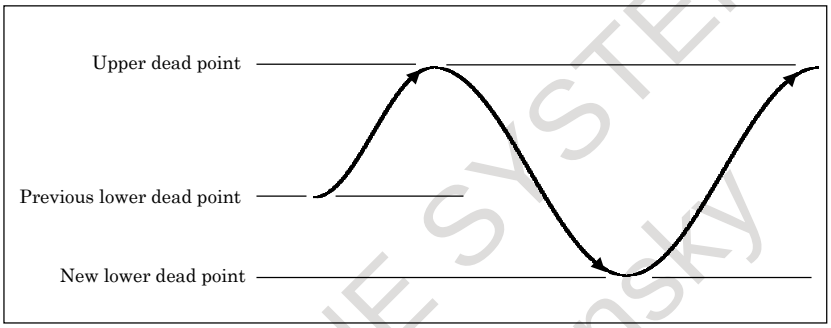
FRYER MACHINE SYSTEMS INC
Rob Zahensky

(3) When the upper dead point is changed during movement from the lower dead point to the upper dead point



The tool first moves to the previous upper dead point, then to the lower dead point, and finally to the new upper dead point.

(4) When the lower dead point is changed during movement from the lower dead point to the upper dead point



The tool first moves to the upper dead point, then to the new lower dead point.

- Look-ahead feed forward function

Look-ahead feed forward function can be effective for oscillation motion.

- Mode switching during oscillation motion

If the mode is changed during oscillation motion, oscillation motion does not stop. In manual mode, the oscillation axis cannot be moved manually. It cannot also be moved manually by means of the handle interrupt.

- Single block signal

Even when single block signal SBK is input during oscillation motion, oscillation continues.

- Interlock signal

Interlock signal for all axes *IT or Interlock signals for each axes *ITx is valid to the oscillation axis in oscillation mode. However, Interlock signal for each axis direction +MITx/-MITx is invalid to the oscillation axis in oscillation mode.
If interlock is effective to the oscillation axis during oscillation motion, the oscillation axis decelerates and stops immediately.

- Parameter setting

This oscillation function becomes effective by setting bit 0 (SSO) of parameter No. 25650 to 1.

- Flexible synchronization control

Flexible synchronization control whose master axis is the oscillation axis can be used.
If Look-ahead feed forward function is used, it is necessary to set the prameter also to the slave axis as well as the master axis.
However, Inter-path flexible synchronization control with oscillation axis cannot be used.

FIXER MACHINE SYSTEMS INC
Rob Zahensky

- Axis synchronous control

Axis synchronous control whose master axis is the oscillation axis can be used.
If Look-ahead feed forward function is used, it is necessary to set the parameters also to the slave axis as well as the master axis.

Limitation

- Workpiece coordinate system

While oscillation is being performed, do not change the workpiece coordinate system for the oscillation axis.

- PMC axis

When the oscillation axis is operating as the PMC axis, oscillation is not started.

- Mirror image

Never attempt to apply the mirror image function about the oscillation axis.

- Move command during oscillation motion

If a move command is specified for the oscillation axis while oscillation motion is being performed, an alarm (PS5050) "ILL-COMMAND IN G81.1 MODE" is issued.

- Canned Cycle

While oscillation is being performed, canned cycle can not be used.

- Program restart / Quick program restart

When a program contains G codes for starting oscillation (G81.1) and stopping oscillation (G80), an attempt to restart that program results in an alarm (PS5050) being output.
When a program that does not include the oscillation axis is restarted during oscillation motion, the coordinates and amount of travel set for the oscillation axis are not affected after the restart of the program.

- Rotation axis

Oscillation operation cannot be used with rotation axis (type-A).

- Inch/Metric conversion commands

While oscillation is being performed, do not be specified inch/metric conversion commands.

- General purpose retract

While oscillation is being performed, do not be performed retraction. The oscillation motion does not stop by retraction.

- Stored stroke check

Stored stroke check 1-I (parameter Nos. 1320 and 1321) is only effective during oscillation motion.

- Arbitrary angular axis control

Do not set angular axis and perpendicular axis of arbitrary angular axis control to oscillation axis.

- Three-dimensional coordinate conversion / Tilted working plane indexing

While oscillation is being performed, do not be specified three-dimensional coordinate conversion / tilted working plane indexing.

- Cs contour control axis

Do not set Cs contour control axis to oscillation axis. If Oscillation is specified for Cs contour control axis, an alarm PS5050, "ILL-COMMAND IN G81.1 MODE" is issued.

FRYER MACHINE SYSTEMS INC
Rob Zakensky

- Synchronous/Composite control

Do not set synchronous/composite control axis to oscillation axis.

- Polygon turning

T

Do not set control axis for polygon turning to oscillation axis.

- Manual handle interrupt

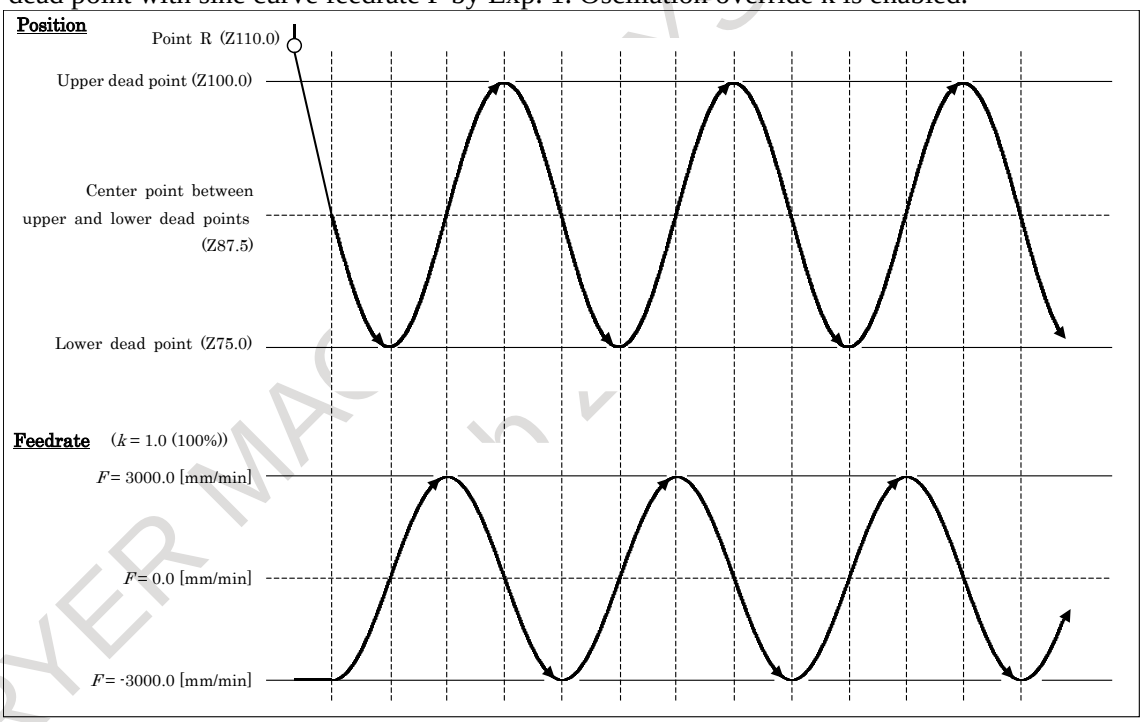
Manual handle interrupt is invalid to the axis in the oscillation mode.
(The slave axis of Flexible synchronization control or Axis synchronous control is contained)

Example

To start oscillation, specify the following command:

G90 G81.1 Z100.0 Q-25.0 R10.0 F3000.0 ;

- The tool moves to point R (Z+R = Z110.0) with rapid traverse feedrate.
- Next, move to center point (Z87.5) between upper dead point (Z100.0) and lower dead point (Z+Q = Z75.0) at oscillation base feedrate (F3000.0[mm/min]).
- Then, perform repeated movement along the Z-axis between the upper dead point and the lower dead point with sine curve feedrate F by Exp. 1. Oscillation override k is enabled.



To cancel oscillation motion, specify the following command:

G80 ;

Motion of cancel is as follows.

Bit 0 (OST) of parameter No. 25651 is

0: The tool moves to point R and stops.

1: The tool decelerates and stops.

FRYER MACHINE SYSTEMS INC

22 MUITI-PATH CONTROL FUNCTION

22.1 OVERVIEW

The multi-path control function is designed to control the independent simultaneous machining with up to 2 paths, and the peripheral device such as a loader for performing a non-machining operation. (machining path up to 2, loader path up to 2, up to 4 paths in total). This function is applicable to lathes and automatic lathes which perform cutting simultaneously with multiple tool posts, and machines which require additional control paths such as a loader control path.

For multi-path simultaneous machining, each machining program is stored in a folder in program memory for each path. When automatic operation is to be performed, each path is activated after selecting a program for machining with path 1 and programs for machining with paths 2 to 4 from the programs stored in the respective folders in program memory. Then the programs selected for the tool posts are executed independently at the same time. When tool post 1 and tool post 2 need to wait for each other during machining, the waiting function is available.
Other available functions specific to multi-path control include interference check for each path, balance cut, synchronous control, composite control, spindle control between each path, and common memory between each path.

Just one LCD/MDI is provided for the all paths. Before operation and display on the LCD/MDI, the path selection signal is used to switch between the paths.

Example) For a system with 2 paths

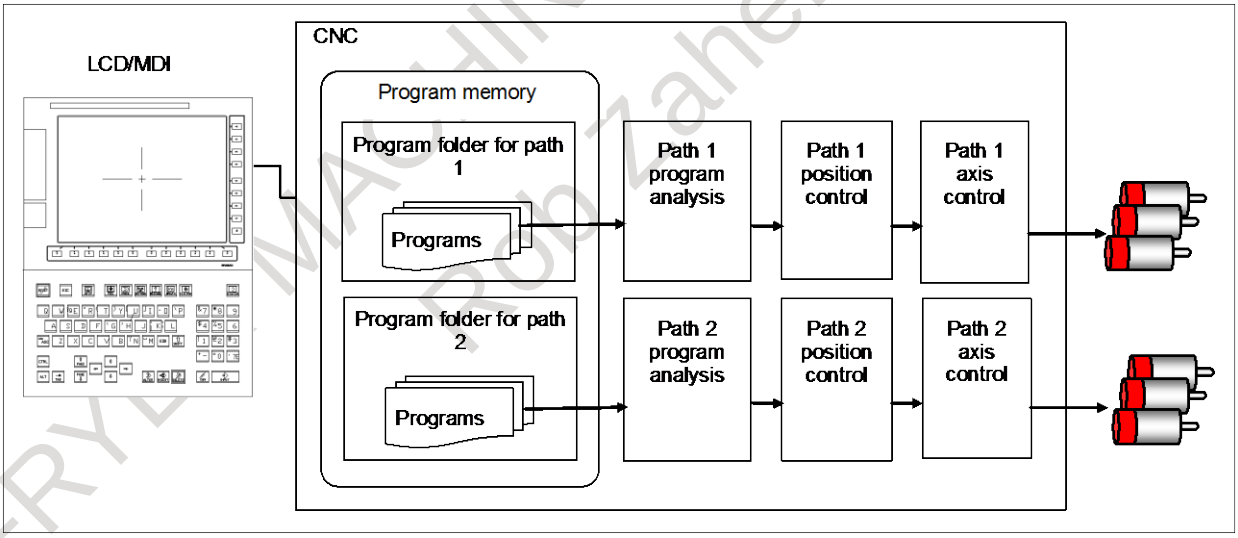


Fig. 22.1

22.2 WAITING FUNCTION FOR PATHS

Overview

Control based on M codes is used to cause one path to wait for the other during machining. When an M code for waiting is specified in a block for one path during automatic operation, the other path waits for the same M code to be specified before staring the execution of the next block.
A range of M codes used as M codes for waiting is to be set in the parameters Nos. 8110 and 8111 beforehand. Waiting can be ignored using a signal.

FRYE MACHINE SYSTEMS INC
Rob Zahensky

Format

M m (Pp) ;

m: Number of an M code for waiting

p: (1) In the binary value specification mode, specify the sum of the binary values corresponding to the numbers of paths which are to wait for one another.

(2) In the path number specification mode, specify the numbers of all paths that are to wait for one another in combination.

If P command is not specified, paths 1 and 2 wait for each other.
In case of 1 path system with loader control function, path 1 and loader path 1 wait for each other.

Explanation

The following two methods for specifying paths which are to wait for one another at address P in the same block in which an M code for waiting is specified are available and can be selected using bit 1 (MWP) of parameter No. 8103. One method is to specify the paths with the sum of their corresponding binary values (waiting set for three or more paths specified with binary values). The other is to specify them with their path numbers in combination (waiting specified with path numbers).
When address P is not specified, paths 1 and 2 wait for each other (waiting set for two paths). Always specify address P in a single block.

- Waiting specified with binary values

When bit 1 (MWP) of parameter No. 8103 is set to 0, the value specified at address P is assumed to be obtained using binary values. The following table lists the path numbers and corresponding binary values.

Table 22.2 (a)

Path number	Binary value (decimal number)
1	1
2 or Loader path 1(1 control path)	2
Loader path 1 (2 control paths) or Loader path 2(1 control path)	4
Loader path 2 (2 control paths)	8

The bit position of each path in binary representation is shown below.

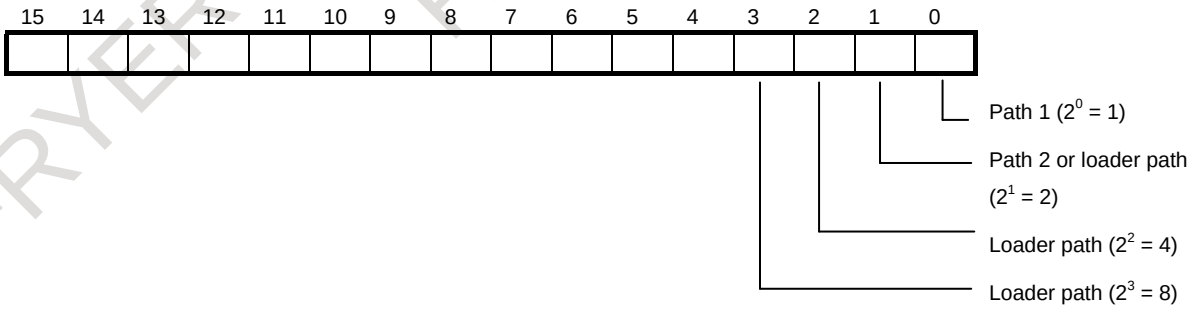


Fig. 22.2 (a)

To make all of paths 1, 2, and 3 wait for one another, the P value is obtained as follows:

Binary value of path 1	1 (0000 0000 0000 0001)
Binary value of path 2	2 (0000 0000 0000 0010)
Binary value of loader path 1	4 (0000 0000 0000 0100)
Sum	7 (0000 0000 0000 0111)

FRYER MACHINE SYSTEMS INC
Rob Zahansky

All of the three paths can be made to wait for one another by specifying P7 together with an M code for waiting.

- Waiting specified with a combination of path numbers

When bit 1 (MWP) of parameter No. 8103 is set to 1, the value specified at address P is assumed to be a combination of path numbers. The following table lists the path numbers and corresponding values.

Table 22.2 (b)

Path number	Value (decimal number)
1	1
2 or Loader path 1(1 control path)	2
Loader path 1 (2 control paths) or Loader path 2(1 control path)	3
Loader path 2 (2 control paths)	4

To make all of paths 1, 2, and loader path 1 wait for one another, the P value is a number consisting of 1, 2, and loader path 1.

Example) P123

There are no restrictions on the order in which the numeric characters are specified, and the following six possible values can be specified:

P123, P132, P213, P231, P312, P321

Path numbers specified in combination in different orders for different paths are effective as long as the numbers of the relevant paths are specified.

Example)

The following are treated as the same P value and these paths can be made to wait for one another:
M200P123 for path 1, M200P231 for path 2, and M200P321 for path 3

Example

- When the value specified at P is obtained using binary values

Assume that the waiting ignore signal for path 2 is set to 1 and M101 to M103 (parameter No. 8110 = 101 and parameter No. 8111 = 103) are set as M codes for waiting. In this case, programs O100, O200, and O300 for individual paths are executed as follows:

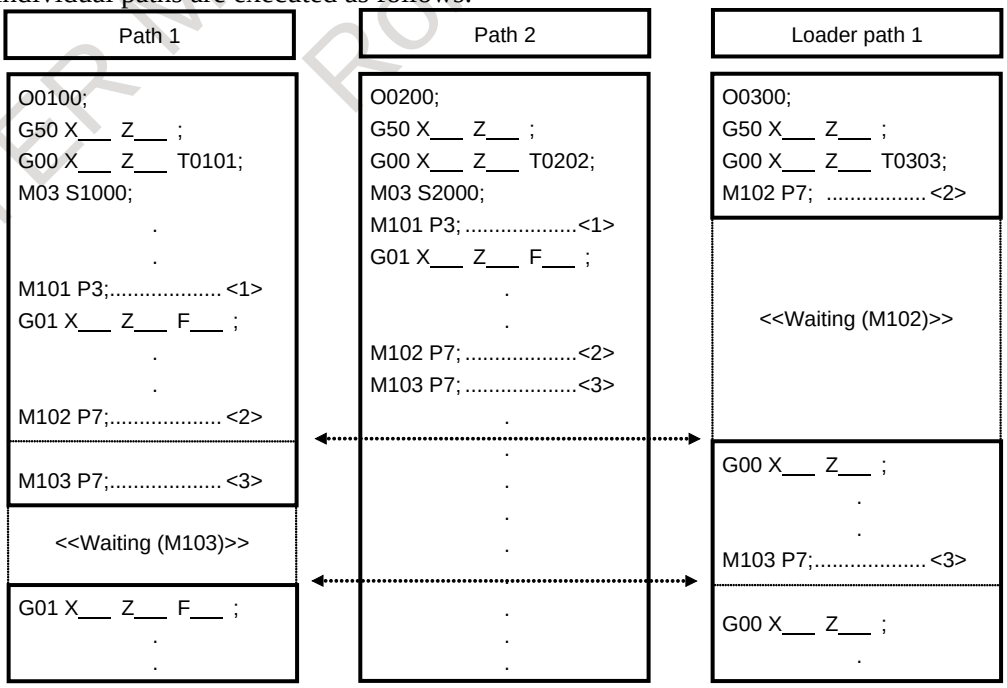


Fig. 22.2 (b)

FRYER MACHINE SYSTEMS INC
Rob Zahansky

<1> M101 P3; (making paths 1 and 2 wait for each other)
When the waiting ignore signal for path 2 is set to 0, paths 1 and 2 wait for each other. Because the signal is set to 1, however, paths 1 and 2 ignore the M code for waiting and immediately execute the next block.

<2> M102 P7; (making paths 1, 2, and loader path 1 wait for one another)
In this example, loader path 1 waits for processing on paths 1 and 2 to terminate. Because the waiting ignore signal for path 2 is set to 1, however, loader path 1 assumes that waiting is completed when processing on path 1 terminates, and executes the next block.

<3> M103 P7; (making paths 1, 2, and loader path 1 wait for one another)
In this example, paths 1 and 2 wait for processing on loader path 1 to terminate. Because the waiting ignore signal for path 2 is set to 1, however, path 2 does not wait for processing on loader path 1 to terminate and executes the next block, but path 1 waits for loader path 1.

- When the value specified at P is obtained using path numbers in combination

Assume that the waiting ignore signal for path 2 is set to 1 and M101 to M103 (parameter No. 8110 = 101 and parameter No. 8111 = 103) are set as M codes for waiting. In this case, programs O100, O200, and O300 for individual paths are executed as follows:

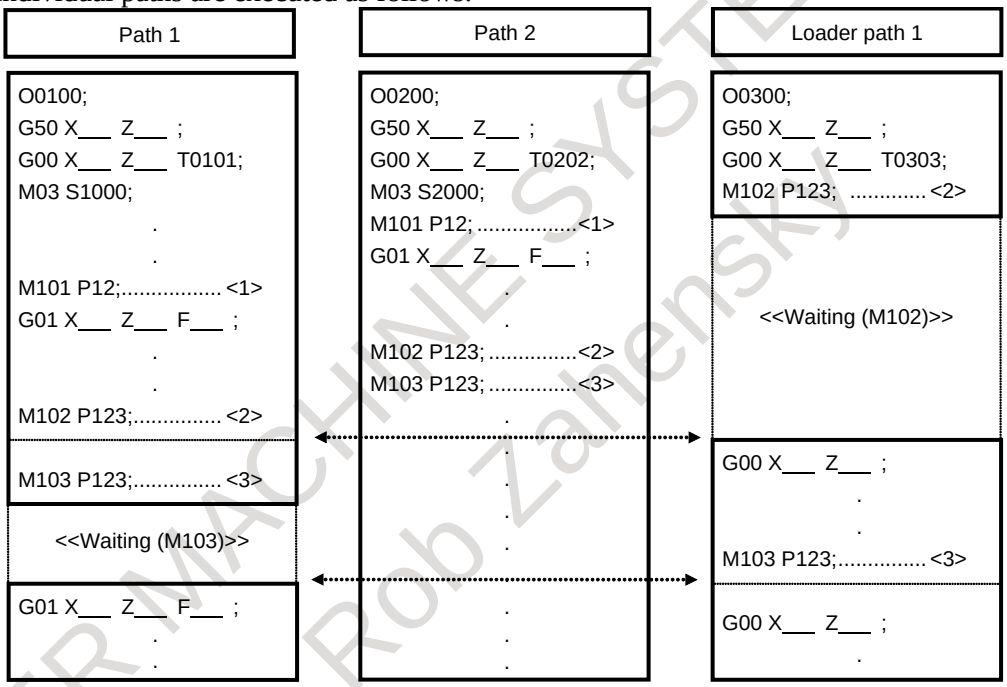


Fig. 22.2 (c)

<1> M101 P12; (making paths 1 and 2 wait for each other)
When the waiting ignore signal for path 2 is set to 0, paths 1 and 2 wait for each other. Because the signal is set to 1, however, paths 1 and 2 ignore the M code for waiting and immediately execute the next block.

<2> M102 P123; (making paths 1, 2, and loader path 1 wait for one another)
In this example, loader path 1 waits for processing on paths 1 and 2 to terminate. Because the waiting ignore signal for path 2 is set to 1, however, loader path 1 assumes that waiting is completed when processing on path 1 terminates, and executes the next block.

<3> M103 P123; (making paths 1, 2, and loader path 1 wait for one another)
In this example, paths 1 and 2 wait for processing on loader path 1 to terminate. Because the waiting ignore signal for path 2 is set to 1, however, path 2 does not wait for processing on loader path 1 to terminate and executes the next block, but path 1 waits for loader path 1.

FRYER MACHINE SYSTEMS INC
Rob Zahenskyj

⚠ CAUTION

1 An M code for waiting must always be specified in a single block.
2 Unlike other M codes, the M code for waiting is not output to the PMC.
3 If the operation of a single path is required, the M code for waiting need not be deleted. By using the signal to specify that waiting be ignored (NOWT for the system with two paths (including loader paths), NMWT for the system with three or more paths (including loader paths)), the M code for waiting in a machining program can be ignored. For details, refer to the manual supplied by the machine tool builder.
4 If using a waiting M code in block multiple-M code command mode, be sure to specify it as the first M code.

22.3 WAITING M CODES OF HIGH-SPEED TYPE

Overview

Waiting M codes of high-speed type is useful function to shorten the Cycle time of a program operation.

Format

M m (P p) ;

m : Number of an M code for waiting
p : (1) In the binary value specification mode, specify the sum of the binary values corresponding to the numbers of paths which are to wait for one another.
(2) In the path number specification mode, specify the numbers of all paths that are to wait for one another in combination.

NOTE

Specify a waiting M code only in a block.
The irrelevant command, which is specified with a waiting M code in a block, is ignored.

Explanation

In order to use Waiting M codes of high-speed type, it is necessary to set the range of the M codes to parameters Nos. 8114 and 8115.
The usage of these type M codes is similar to normal waiting M codes. For instance, the way to set a path by address P and to use a no-wait signal or so on are the same.

22.4 COMMON MEMORY BETWEEN EACH PATH

Overview

In a multi-path system, this function enables data within the specified range to be accessed as data common to all paths. The data includes tool compensation memory and custom macro common variables.

Explanation

The path common memory function enables the following operations.

- Tool compensation memory

Part or all of tool compensation memory for individual paths can be used as common data by setting parameter No. 5029.

FRYER MACHINE SYSTEMS INC Rob Zahensky

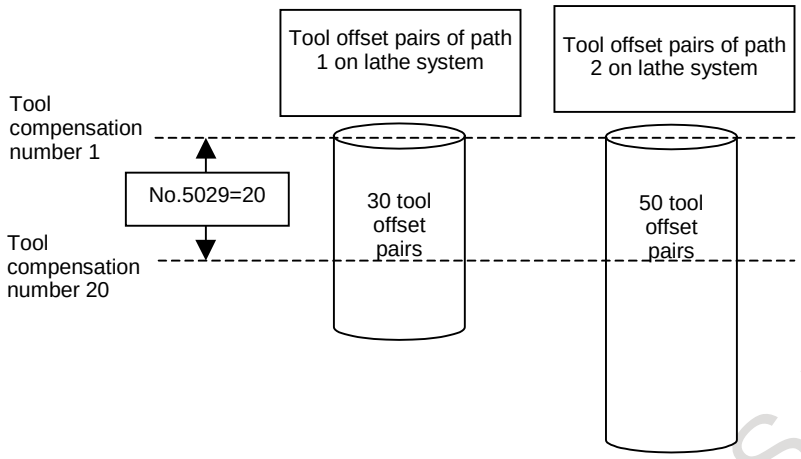


Fig. 22.4 (a)

NOTE

1 When a combined system including a machining center system and lathe system is used, data is made common according to the same path control type.
2 The same unit for tool compensation (bits 0 to 3 of parameter No. 5042) must be set for each of the machining center and lathe systems.
3 Set a value less than the number of tool compensation values for each path for parameter No. 5029.
4 If the value set for parameter No. 5029 is greater than the number of tool compensation values for each path, the minimum number of tool compensation values for individual paths is assumed.
5 For details, refer to the relevant manual of the machine tool builder.

- Custom macro common variables

All or part of custom macro common variables #100 to #149 (, #199, or #499) and #500 to #599 (or #999) can be used as common data by setting parameters No. 6036 (#100 to #149 (, #199, or #499)) and 6037 (#500 to #599 (or #999)).

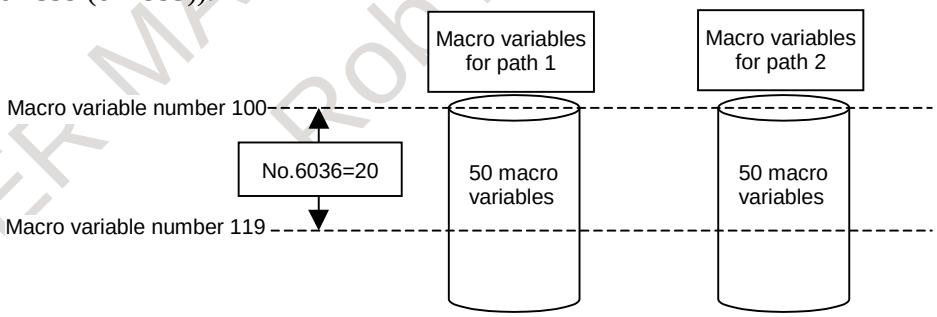


Fig. 22.4 (b)

NOTE

If the value of parameter No. 6036 or 6037 exceeds the maximum number of macro common variables, the maximum number of macro common variables is assumed.

FRYER MACHINE SYSTEMS INC Rob Zahansky

22.5 SPINDLE CONTROL BETWEEN EACH PATH

Overview

This function allows a workpiece attached to one spindle to be machined simultaneously with two tool posts and each of two workpieces attached to each of two spindles to be machined simultaneously with each of two tool posts.

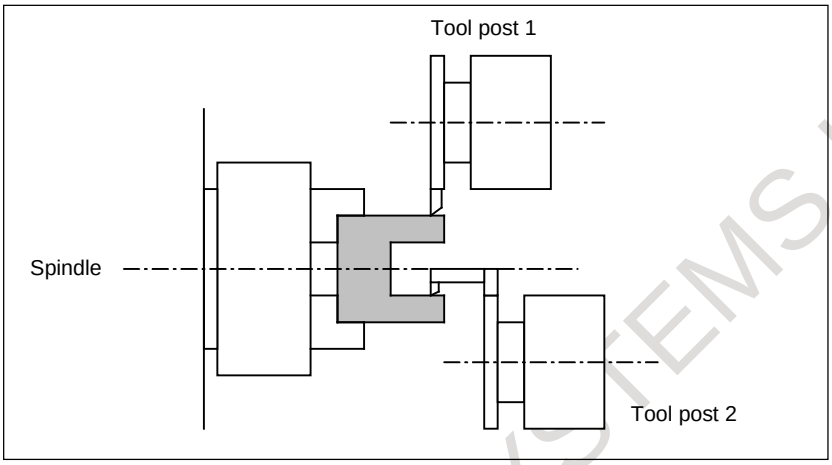


Fig. 22.5 (a) Application to a lathe with one spindle and two tool posts

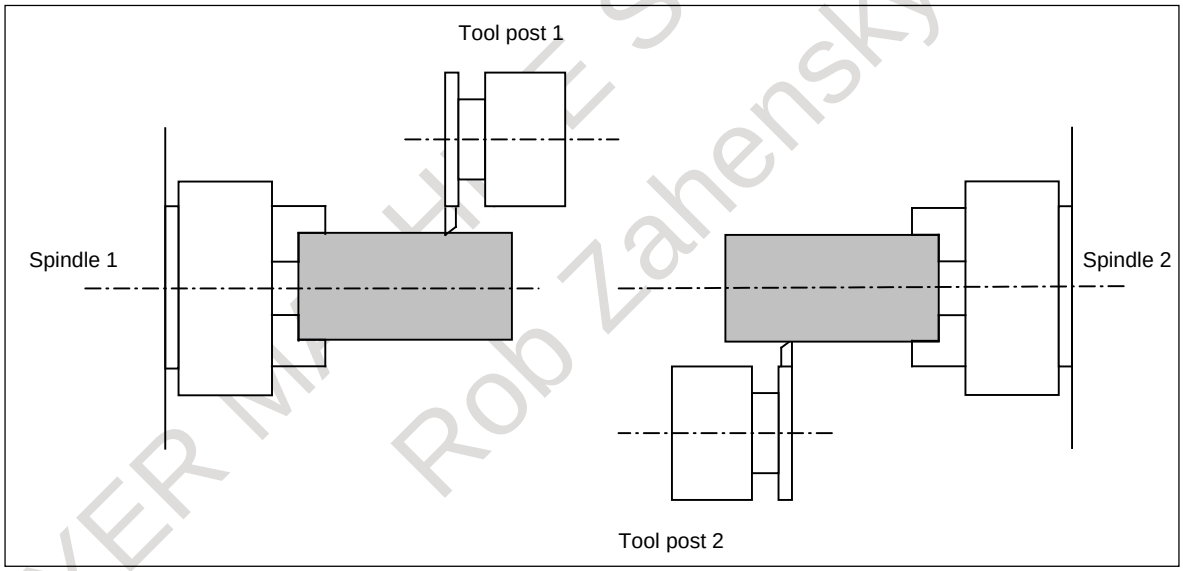


Fig. 22.5 (b) Application to a lathe with two spindles and two tool posts

The spindle belonging to each path can generally be controlled by programmed commands for the path. With path spindle command selection signals, programmed commands for any path can control the spindle belonging to any path.

NOTE

For the method of spindle command selection, refer to the relevant manual of the machine tool builder.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

22.6 SYNCHRONOUS/COMPOSITE/SUPERIMPOSED CONTROL

Overview

In multi-path control, the synchronous control function, composite control function, and superimposed control function enable synchronous control, composite control, and superimposed control in a single path system or between multi-path systems.

Explanation

- Synchronous control

- Synchronizes movement along an axis of one system with that along an axis of the other path.
  Example)
  Synchronizing movement along the Z1 (master) and Z2 (slave) axes (in the case of turning)

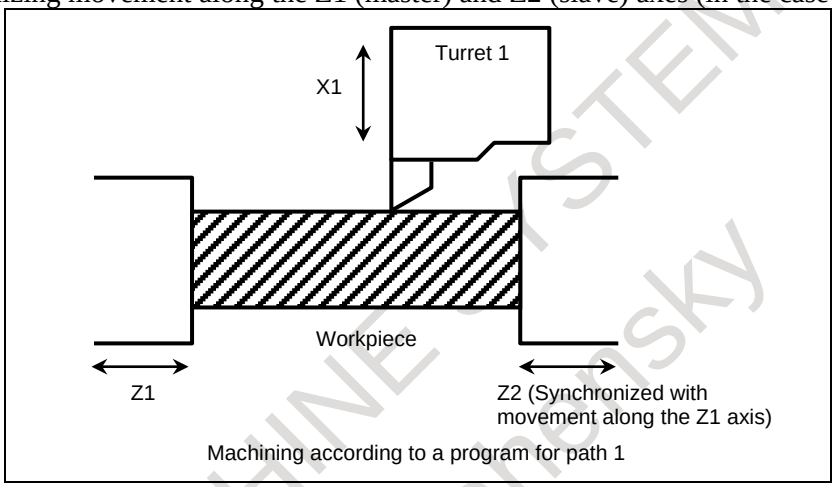


Fig. 22.6 (a)

- Synchronizes movement along an axis of one path with that along another axis of the same path.
  Example)
  Synchronizing movement along the Z1 (master) and B1 (slave) axes (in the case of turning)

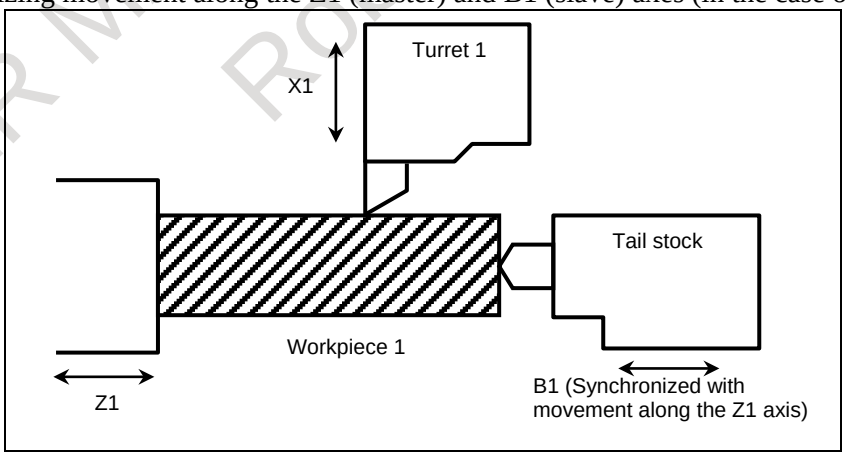


Fig. 22.6 (b)

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

- Composite control

- Exchanges the move commands for different axes of different path.
  Example)
  Exchanging the commands for the X1 and X2 axes (in the case of turning)
  → Upon the execution of a command programmed for path 1, movement is performed along the X2 and Z1 axes.
  Upon the execution of a command programmed for path 2, movement is performed along the X1 and Z2 axes.

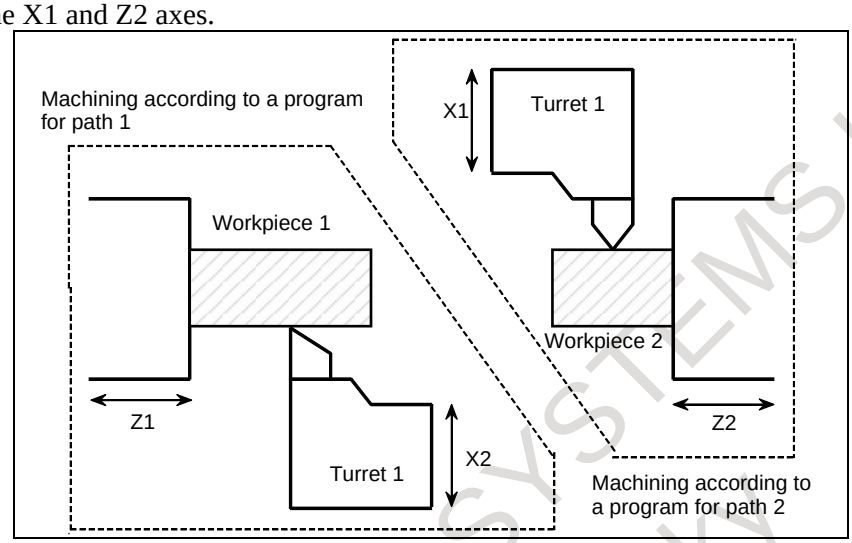


Fig. 22.6 (c)

- Superimposed control

- Provides a move command of an axis for a different axis in another path.
  Example)
  Providing the Z2 (slave) axis with a move command specified for the Z1 (master) axis (in the case of turning)

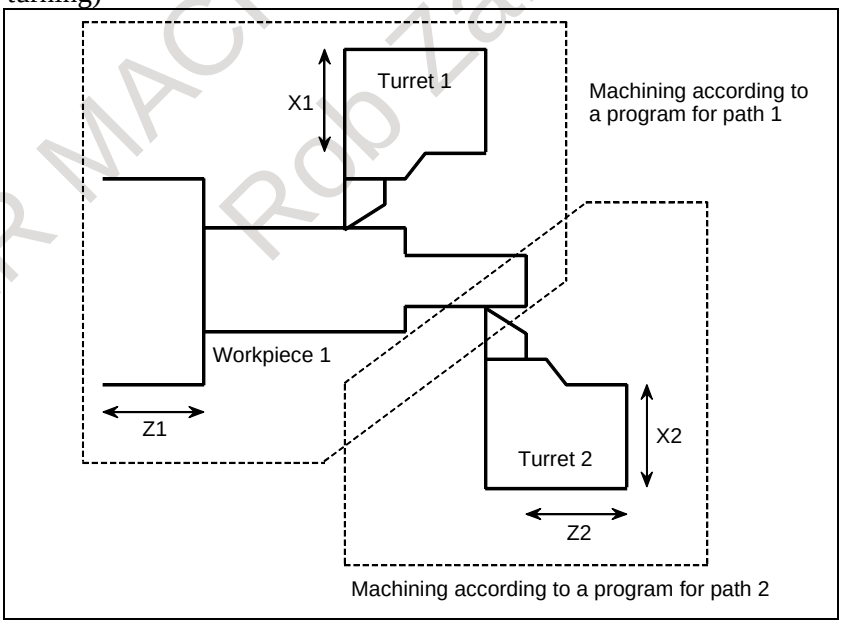


Fig. 22.6 (d)

NOTE
The method used to specify synchronous, composite, or superimposed control varies with the machine tool builder. For details, refer to the manual supplied by the machine tool builder.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

22.7 WAITING FUNCTION BY SPECIFYING START POINT

Overview

Control based on M codes is used to make one path to wait for the other during machining. When an M code for waiting is specified in a block of a path during automatic operation, the other path waits for the same M code to be specified before staring the execution of the next block.
In this function, by specifying a start point with a waiting M code, absolute coordinate value of that path or the other path can be used as the condition for waiting.

Format

M_ P_ L_ IP ;

Format to specify coordinates of a start point.

M : Waiting M code.
IP : Absolute coordinate value of a start point.
L : Type of waiting (0,1).
- L0 : This path waits until absolute coordinate on the other path arrives at start point. (When L0 is specified with P7, the alarm PS0161 occurs).
- L1 : The other path waits until absolute coordinate on this path arrives at start point.

P : Pattern of waiting (3,5,6,7).
- P3 : path 1 and path 2 wait for each other.
- P5 : path 1 and path 3 wait for each other.
- P6 : path 2 and path 3 wait for each other.
- P7 : path 1, path 2 and path 3 wait for each other.

∗The meaning of P command can be switched to “pattern of waiting with path number” by bit 1 (MWP) of parameter No.8103.
For more detail information, refer to “FANUC Series 0*i*-MODEL F CONNECTION MANUAL (FUNCTION) WAITING M CODES”.

Note.
P3, P5, P7 can be specified in path 1.
P3, P6, P7 can be specified in path 2.
P5, P6, P7 can be specified in path 3.
Specify absolute coordinate value of start point after M_ P_ L_. If start point is specified before M_ P_ L_, this block is operated as a normal waiting M code.
Specify L in an only one path.
If L is not specified in a block, this block is operated as a normal waiting M code, not the waiting function by specifying start point.

M_ P_ ;

Format not to specify coordinates of start point.

FRYER MACHINE SYSTEMS INC
Rob Zaminsky

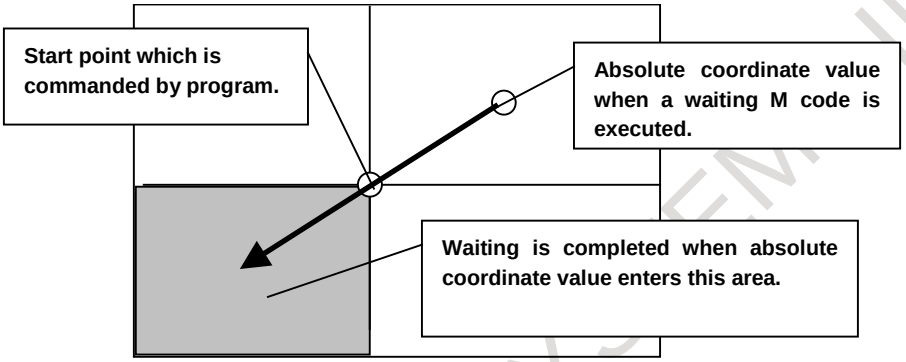
Operation

(1) When a waiting M code is specified on one path during automatic operation, that path waits until other paths execute the same waiting M code.

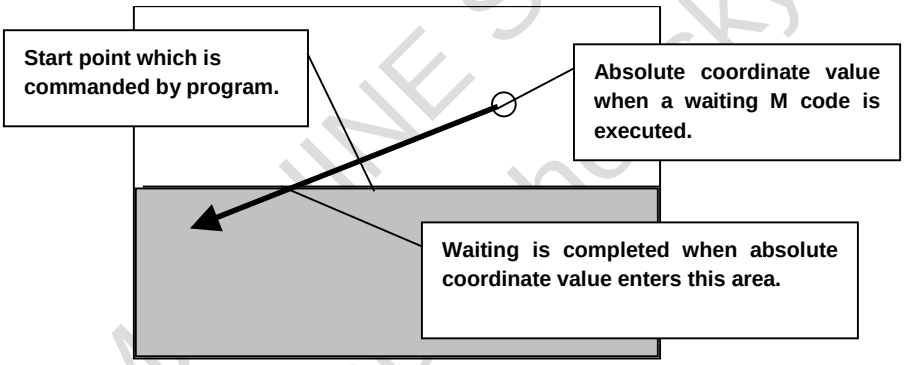
(2) After all related paths execute the same waiting M, CNC compares the programmed coordinate values with current ones, and CNC decides a waiting area.

Waiting is completed when absolute coordinate value enters the gray area in the following chart.

When command in coordinates for two axes



When command in coordinates for one axis



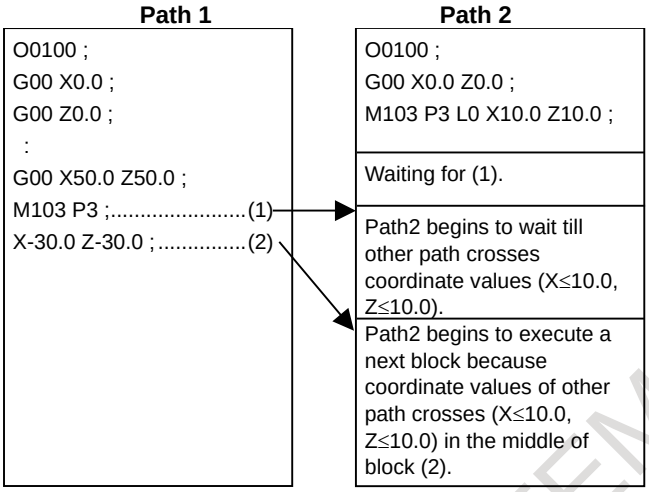
(3) If L0 is commanded on a path, a next block of a waiting M code is executed on the other path, and the path with L0 waits until absolute coordinate value on the other path enters a gray area.
If L1 is commanded, a next block of a waiting M code is executed, and the path with L1 makes the other path wait until absolute coordinate value of the path with L1 enters a gray area.

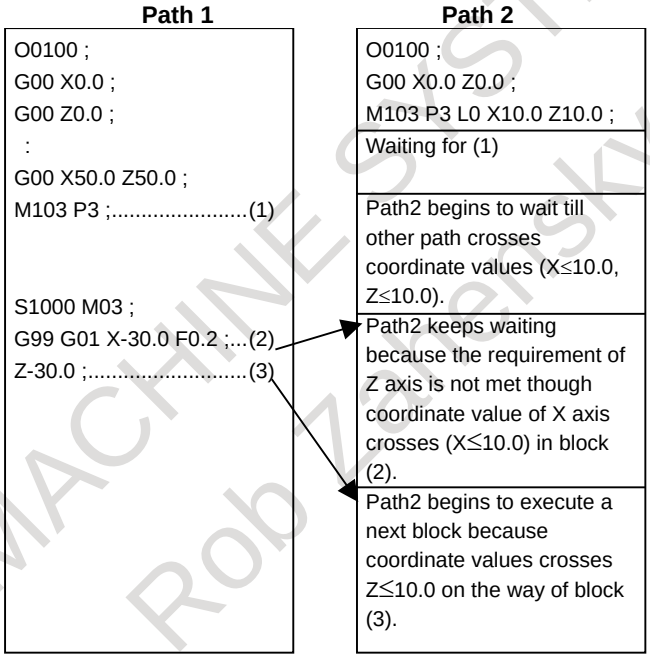
(4) After completing the waiting, the waited path begins to execute a following block of a waiting M code.

FRYER MACHINE SYSTEMS INC Robert Hocky

Example of program

L0 command





FRYER MACHINE SYSTEMS INC
Rob Zaherski

L1 command

Path 1

```
O0100 ;
G00 X0.0 ;
G00 Z0.0 ;
 :
G00 X50.0 Z50.0 ;
M103 P3 ;.......................(1)

Waiting till other path
crosses coordinate values
(X≥10.0, Z≥10.0)

Path1 begins to execute a
next block because
coordinate values crosses
(X≥10.0, Z≥10.0) in the
middle of block (2).
```

Path 2

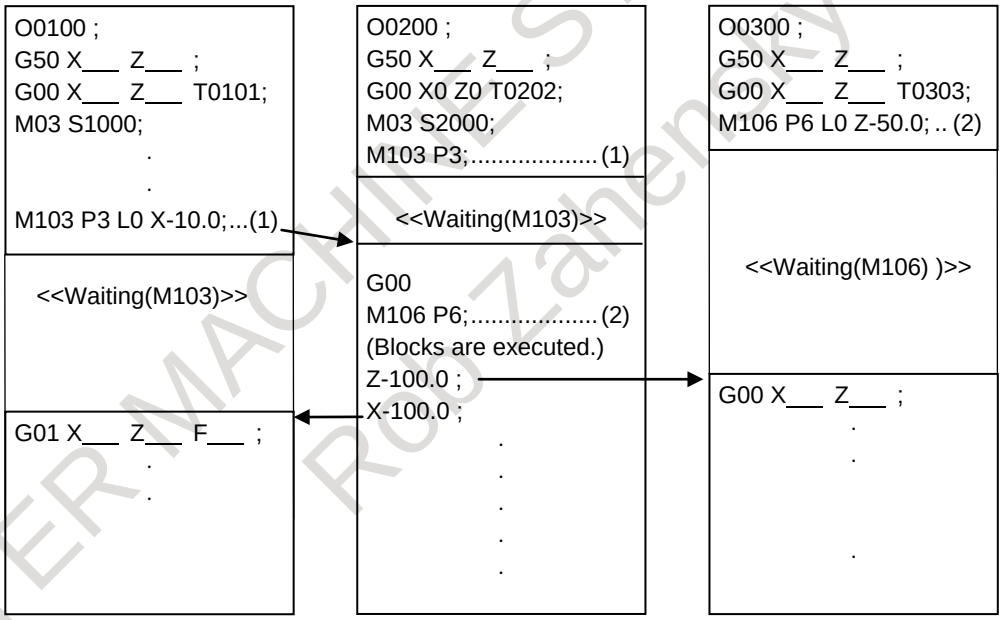
```
O0100 ;
G00 X0.0 Z0.0 ;
M103 P3 L1 X10.0 Z10.0 ;
Waiting for (1)

Path2 begins to execute a
next block

G00 X50.0 Z50.0 ; ......... (2)
```

Waiting in 3 path system

Assume M101 to M199 (parameter No. 8110 = 101 and parameter No. 8111 = 199) are set as M codes for waiting. In this case, programs O100, O200, and O300 for individual paths are executed as follows:



Waiting by specifying a start point between path 1 and path 2, and another waiting by specifying a start point between path 2 and path 3 can be specified at the same time.
In this example, both path 1 and path 3 wait for the movement of path 2

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Note

Inch / metric switching

Specifying coordinates of the waiting function by specifying start point might be failed when the input unit system is changed by G20/G21 during executing program.

```
O0100 ;
G20   ; (METRIC)
G00 X0 Z0 ;
 :
G21 ; (INCH)
G00 X50.0 Z50.0 ;
M103 P3 ;.......................(1)
X-30.0 Z-30.0 ;...............(2)
```

```
O0100 ;
G00 X0.0 Z0.0 ;
M103 P3 L0 X10.0 Z10.0 ;
(When M103 is executed,
the other path is a unit of
metric.)
Coordinates of a different
unit are compared because
the input unit system has
been changed when waiting
M code ends.
```

Current block display

The start position coordinates are not displayed in the current block display and the next block display of the program screen.

Execution macro

The waiting function by specifying start point cannot be used in execution macro (P-code program).
Also, macro cannot be called by M code of the waiting function by specifying start point.

Manual operation during waiting

Even if mode of operation is changed to the manual mode during waiting by specifying start position, waiting is completed when absolute coordinate value enters the specified area.
At this time, change to the JOG mode in only a path which axis moves, and maintain memory mode in a path that is waiting.

Repetition of waiting command

When both of paths command L0, waiting is not completed and halt because each path waits for the movement of the other path.

Path 1)	Path2)
M_ P_ L0 X_ Z_ ; Halt	M_ P_ L0 X_ Z_ ; Halt

When a path command L1 and another path command L0, waiting of a path with L1 is completed and executes a next block, however waiting of a path with L0 is not completed and halt.

Path 1)	Path2)
M_ P_ L1 X_ Z_ ; Execute a next block	M_ P_ L0 X_ Z_ ; Halt

Amount of tool compensation and current position display

When the absolute coordinate value of the current position display reaches the start position, waiting is completed. Therefore, the start position can be shifted from the program coordinates when the tool compensation (wear-out compensation and tool-nose radius compensation etc.) is effective.

Example)

```
T0101 ; (When the amount wear-out compensation is X = -1.000)
G00 X0.0 ;
M103 P3 L1 X10.0 ;
G00 X20.0 ;
```

- When bit 6 (DAL) of parameter No. 3104#=0, waiting is completed when absolute coordinate value is X = 10.000 and the amount of distance to go is X = 9.000.

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

- When bit 6 (DAL) of parameter No. 3104#=0, waiting is completed when absolute coordinate value is X = 10.000 and the amount of distance to go is X = 10.000.

Manual handle retrace

Forward movement

There is not any restriction in forward movement.

Backward movement

When this function is effective, manual handle retrace function prohibits changing direction at a waiting M code block in backward movement. Continue backward movement until waiting M code is appeared in all paths.

NOTE

When this function or waiting M code is used with manual handle retrace, set bit 4 (HMP) of parameter No.6400 to 1 in the both of paths, and when inversion or backward movement is inhibited in other path, inversion or backward movement is inhibited also for the current executed path.

Flexible path axis assignment

Do not execute flexible path axis assignment when paths are waiting.

Retrace (M system)

When the waiting function by specifying start point is available, a waiting M code cannot move backward. Only forward movement is possible.

Incremental command

Even if incremental command mode is selected by G91 command in G code system B or C at the path where M code for the waiting function by specifying start point is specified, it is assumed that absolute coordinate value is specified as a start point.

If address U, V or W, that are incremental commands, are specified as start point, the alarm (PS0009) occurs.

Rotary axis roll- over

If a rotary axis which has roll-over function is specified as an axis of start point, this coordinate value, to which roll-over function is not applied, is compared with the current absolute coordinate value which is rounded by the angle corresponding to one rotation to judge whether a condition for waiting is met.

Example)

The axis A is the rotary axis and the amount of movement per rotation is 360.000

(Parameter No.1260 =360.000)

```
N10 G00 A0.0 ;
N20 M100 P3 L1 A180.0 ;
N30 G00 A-10.0 ;
```

After the end of N30 block, the absolute coordinate value of A axis is 350.0. Therefore, waiting sequence is completed because A is larger than 180.0 in the middle of the block.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

22.8 FUNCTION FOR LOADER CONTROL

Overview

The function for loader control is used to control the devices for performing a non-machining operation (peripheral device such as a loader). When this function is valid, the path for performing a loader control is added besides a path for machining.
(In the subsequent explanation, the path which applies the function for loader control is called the loader path, while the other path is called the machining path.)

Explanation

- Path selection

A selected path can be switched between a machining path and a loader path by MDI key operation. Whenever SHIFT key and HELP key are pressed at the same time, a selected path is switched between a machining path and a loader path alternately. If the loader path 2 exists, when the loader path 1 is selected, the loader path 2 is switched by the MDI key operation.

Example)
4-path system (machining path 1, machining path 2, loader path 1, and loader path 2)

- In case of selecting the machining path 1

  Whenever MDI keys SHIFT and HELP are pressed at the same time, a selected path is switched in the following order.

  Machining path 1 → Loader path 1 → Loader path 2

Path selection by MDI key operation can be disabled by setting bit 3 (SCD) of parameter No.8106.

- Part program storage size / Number of registerable programs

A part program storage size and a number of registerable programs for the loader path are included in them of an entire system. Even when the loader path is added, part program storage size / number of registerable programs of an entire system do not change.

- Waiting M codes

Waiting M codes are available between the loader path and the machining path. For details, refer to “WAITING M CODES” in this manual.

- Custom macro common variables between each path

All or part of custom macro common variables can be used as data common to all paths including machining paths and loader paths. For details, refer to “MRMORY COMMON TO PATHS” in this manual.

- Spindle speed control

The loader path does not have spindle speed control.

- Tool functions

The loader path does not have tool functions. The loader path does not have a tool offset number and the tool compensation screen.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Usable G codes

The following table shows the G codes which are available for a loader path. When the G codes which are not listed in this table are specified in a loader path, alarm PS0010, “IMPRPOER G-CODE” is issued.

T

G code system for a loader path can be set to a different system for a machining path. Which G code system is selected is set using bits 6 (GSB) and 7 (GSC) of parameter No. 3401.

Table 22.8 G code list

Machining center system	Lathe system G code system A	B	C	Function
G00	G00	G00	G00	Positioning (Rapid traverse)
G01	G01	G01	G01	Linear interpolation (Cutting feed)
G04	G04	G04	G04	Dwell
G04.1	G04.1	G04.1	G04.1	G code preventing buffering
G09	G09	G09	G09	Exact stop
G10	G10	G10	G10	Programmable data input
G11	G11	G11	G11	Programmable data input mode cancel
G20	G20	G20	G70	Input in inch
G21	G21	G21	G71	Input in mm
G22	G22	G22	G22	Stored stroke check function on
G23	G23	G23	G23	Stored stroke check function off
G27	G27	G27	G27	Reference position return check
G28	G28	G28	G28	Return to reference position
G30	G30	G30	G30	2nd, 3rd and 4th reference position return
G31	G31	G31	G31	Skip function
G92	G50	G92	G92	Coordinate system setting
G92.1	G50.3	G92.1	G92.1	Workpiece coordinate system preset
G52	G52	G52	G52	Local coordinate system setting
G53	G53	G53	G53	Machine coordinate system setting
G54	G54	G54	G54	Workpiece coordinate system 1 selection
G54.1	G54.1	G54.1	G54.1	Additional workpiece coordinate systems
G55	G55	G55	G55	Workpiece coordinate system 2 selection
G56	G56	G56	G56	Workpiece coordinate system 3 selection
G57	G57	G57	G57	Workpiece coordinate system 4 selection
G58	G58	G58	G58	Workpiece coordinate system 5 selection
G59	G59	G59	G59	Workpiece coordinate system 6 selection
G61	G61	G61	G61	Exact stop mode
G63	G63	G63	G63	Tapping mode
G64	G64	G64	G64	Cutting mode
G65	G65	G65	G65	Macro call
G66	G66	G66	G66	Macro modal call
G67	G67	G67	G67	Macro modal call cancel
G94	G98	G94	G94	Feed per minute
G90	-	G90	G90	Absolute programming
G91	-	G91	G91	Incremental programming

FRYER MACHINE SYSTEMS INC Rob Zahensky

III. OPERATION

FRYER MACHINE SYSTEMS INC
Rob Zahensky

FRYER MACHINE SYSTEMS INC
Rob Zahensky

1 GENERAL

1.1 MANUAL OPERATION

Explanation

- Manual reference position return

The CNC machine tool has a position used to determine the machine position.
This position is called the reference position, where the tool is replaced or the coordinate are set. Ordinarily, after the power is turned on, the tool is moved to the reference position.
Manual reference position return is to move the tool to the reference position using switches and pushbuttons located on the operator's panel. (See Section, “MANUAL REFERENCE POSITION RETURN”.)

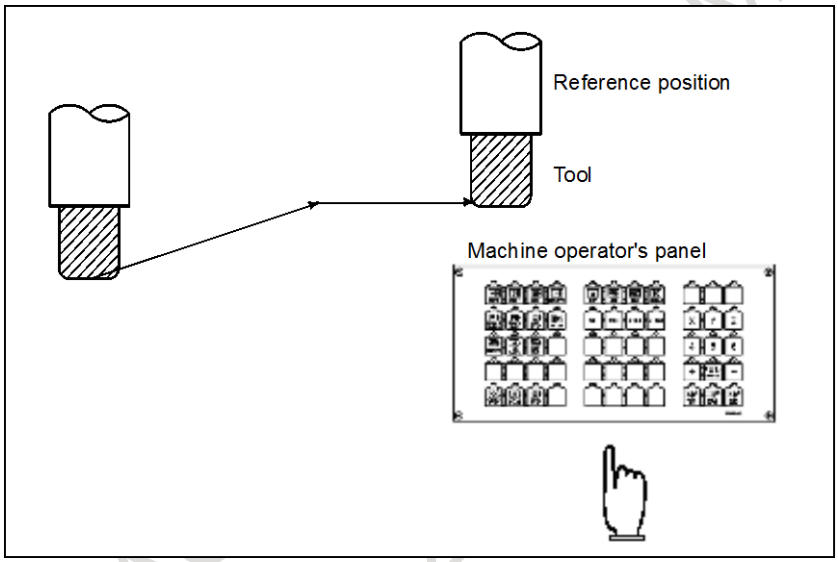


Fig. 1.1 (a) Manual reference position return

The tool can be moved to the reference position also with program commands.
This operation is called automatic reference position return. (See Chapter, “REFERENCE POSITION”.)

FRYER MA Robo MS INC

- The tool movement by manual operation

Using machine operator's panel switches, pushbuttons, or the manual handle, the tool can be moved along each axis.

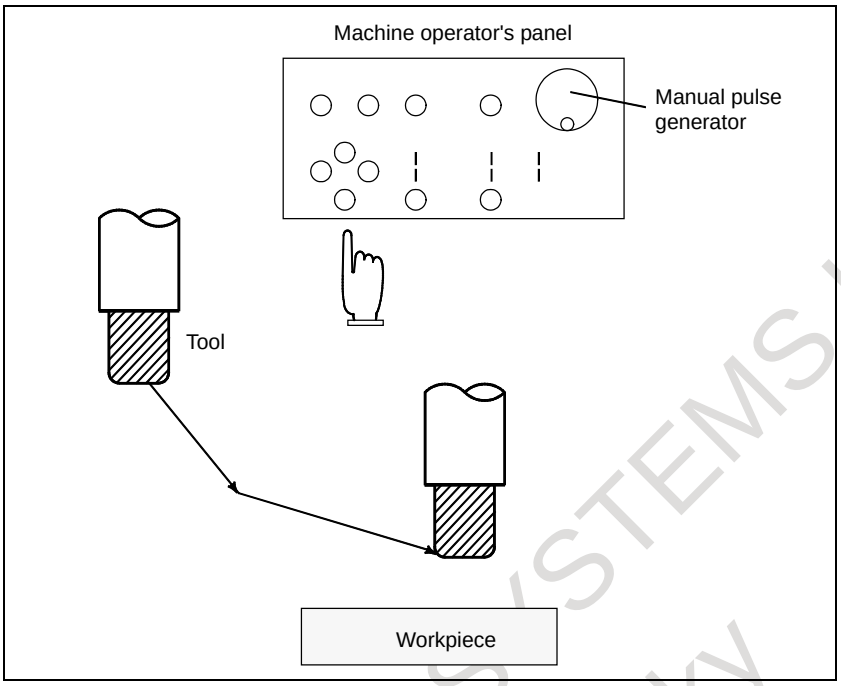


Fig. 1.1 (b) The tool movement by manual operation

The tool can be moved in the following ways:

(i) Jog feed (See Section, “JOG FEED (JOG)”.)
The tool moves continuously while a pushbutton remains pressed.

(ii) Incremental feed (See Section, “INCREMENTAL FEED”.)
The tool moves by the predetermined distance each time a button is pressed.

(iii) Manual handle feed (See Section, “MANUAL HANDLE FEED”.)
By rotating the manual handle, the tool moves by the distance corresponding to the degree of handle rotation.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

1.2 TOOL MOVEMENT BY PROGRAMING - AUTOMATIC OPERATION

Automatic operation is to operate the machine according to the created program. It includes memory, MDI and DNC operations. (See Chapter, “AUTOMATIC OPERATION”.)

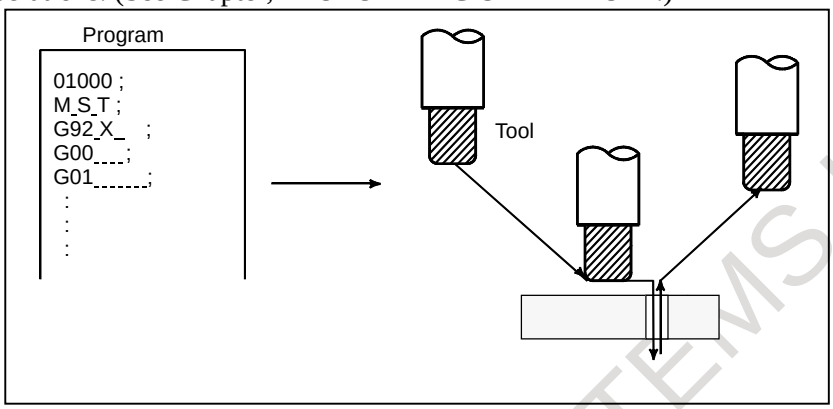


Fig. 1.2 (a) Tool Movement by programming

Explanation

- Memory operation

After the program is once registered in memory of CNC, the machine can be run according to the program instructions. This operation is called memory operation.

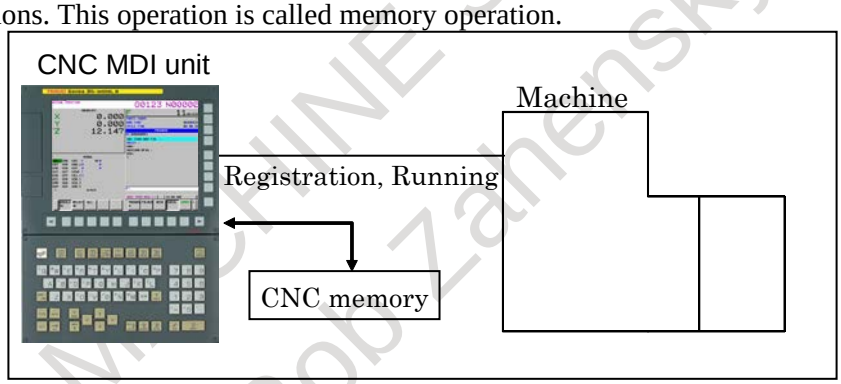


Fig. 1.2 (b) Memory operation

- MDI operation

After the program is entered, as a command group, from the MDI unit, the machine can be run according to the program. This operation is called MDI operation.

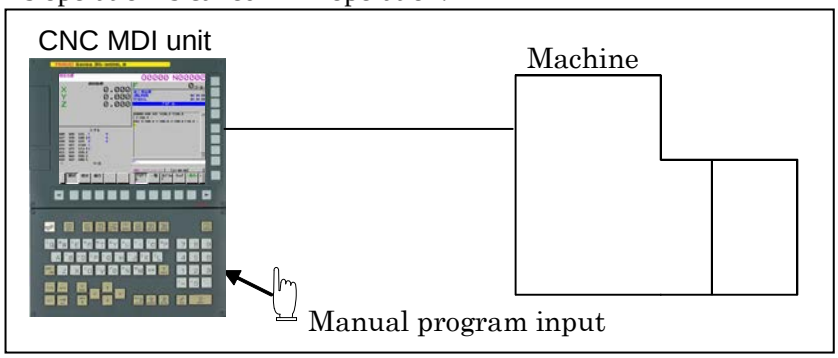


Fig. 1.2 (c) MDI operation

- DNC operation

In this mode of operation, the program is not registered in the CNC memory. It is read from the external input/output devices instead. This is called DNC operation.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

1.3 AUTOMATIC OPERATION

Explanation

- Program selection

Select the program used for the workpiece. Ordinarily, one program is prepared for one workpiece. If two or more programs are in memory, select the program to be used, by searching the program number. (See Section, “PROGRAM SEARCH”.)

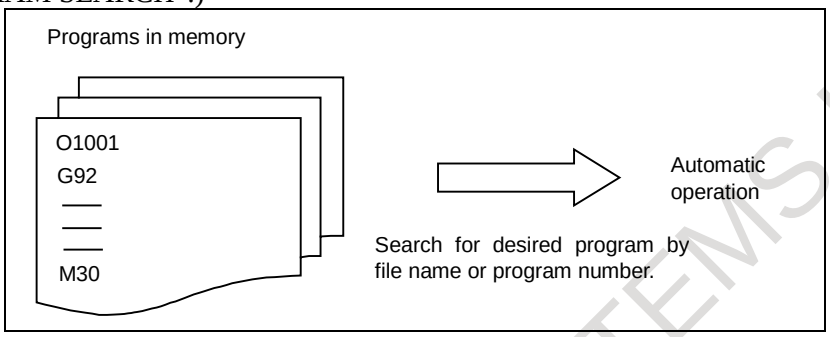


Fig. 1.3 (a) Program selection for automatic operation

- Start and stop

Pressing the cycle start pushbutton causes automatic operation to start. By pressing the feed hold or reset pushbutton, automatic operation pauses or stops. By specifying the program stop or program termination command in the program, the running will stop during automatic operation. When one process machining is completed, automatic operation stops. (See Chapter, “AUTOMATIC OPERATION”.)

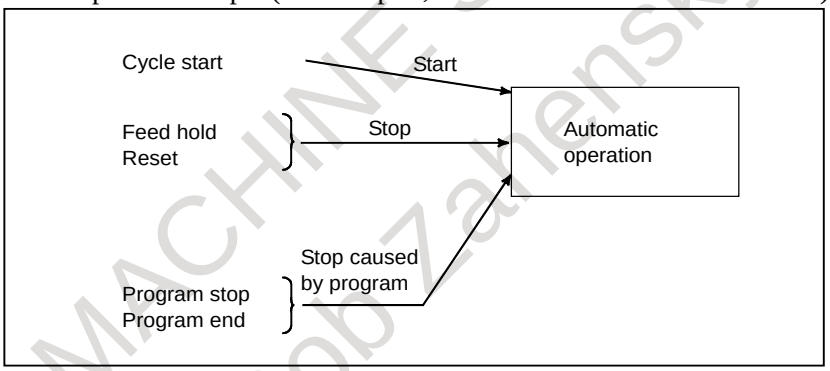


Fig. 1.3 (b) Start and stop for automatic operation

- Handle interruption

While automatic operation is being executed, tool movement can overlap automatic operation by rotating the manual handle. (See Section, “MANUAL HANDLE INTERRUPTION”.)

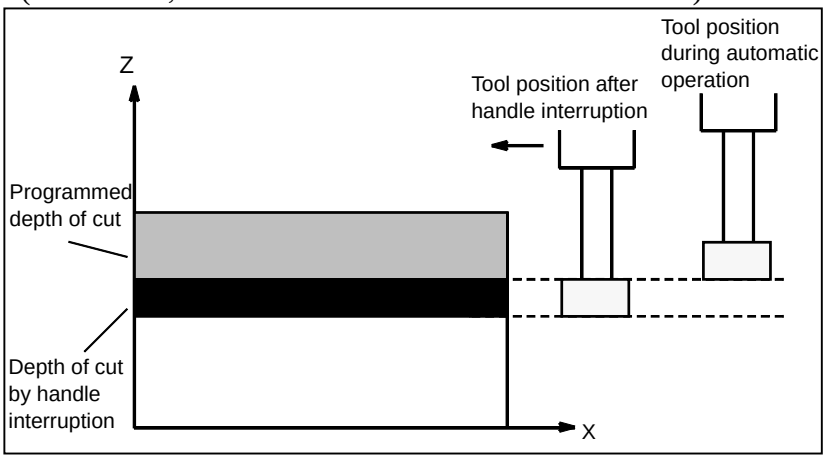


Fig. 1.3 (c) Handle interruption for automatic operation

FRYER MACHINE SYSTEMS INC
Rob Zahensky

1.4 TESTING A PROGRAM

Before machining is started, the automatic running check can be executed.
It checks whether the created program can operate the machine as desired.
This check can be accomplished by running the machine actually or viewing the position display change (without running the machine) (See Chapter, “TEST OPERATION”.).

1.4.1 Check by Running the Machine

Explanation

- Dry run

Remove the workpiece, check only movement of the tool. Select the tool movement rate using the dial on the operator's panel. (See Section , “DRY RUN”.)

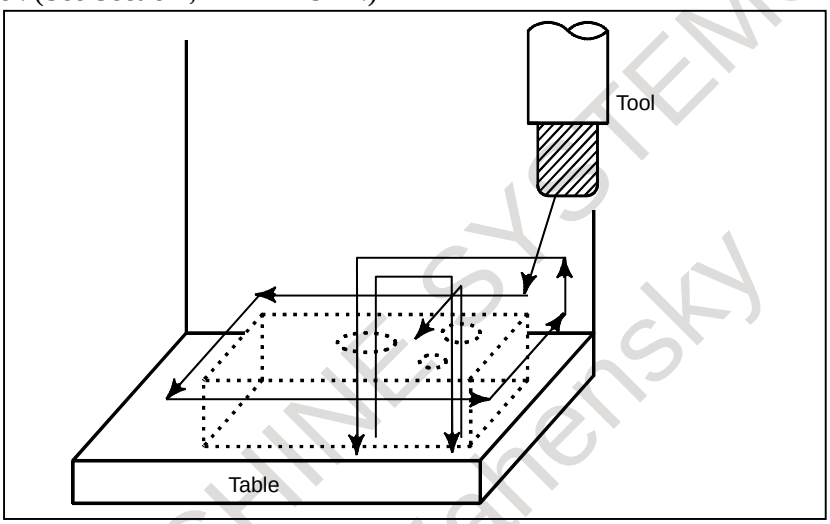


Fig. 1.4.1 (a) Dry run

- Feedrate override

Check the program by changing the feedrate specified in the program. (See Section, “FEEDRATE OVERRIDE”.)

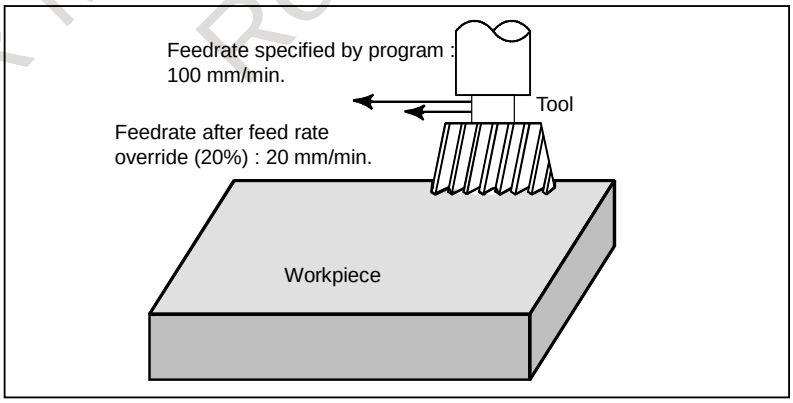


Fig. 1.4.1 (b) Feedrate override

FRYER MACHINE SYSTEMS INC
Rob Zalinsky

- **Single block**

When the cycle start pushbutton is pressed, the tool executes one operation then stops. By pressing the cycle start again, the tool executes the next operation then stops. The program is checked in this manner. (See Section, “SINGLE BLOCK”.)

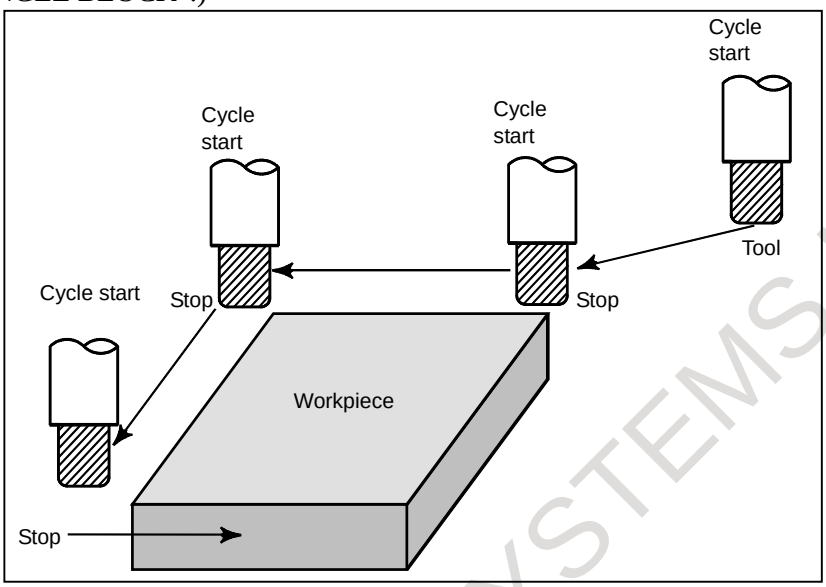


Fig. 1.4.1 (c) Single Block

1.4.2 How to View the Position Display Change without Running the Machine

Explanation

- **Machine Lock**

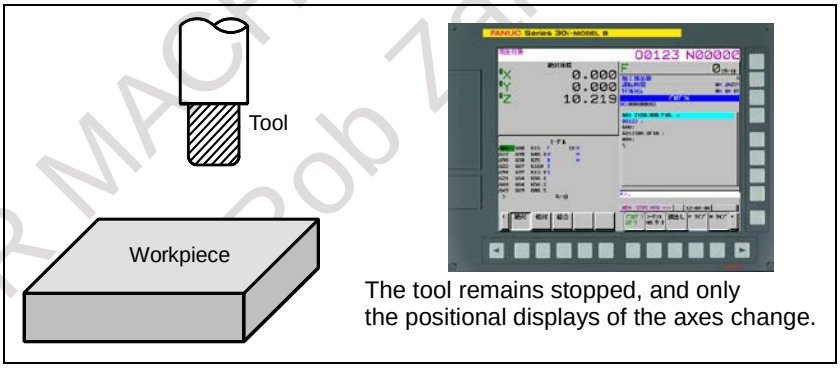


Fig. 1.4.2 (a) Machine Lock

- **Auxiliary function lock**

When automatic running is placed into the auxiliary function lock mode during the machine lock mode, all auxiliary functions (spindle rotation, tool replacement, coolant on/off, etc.) are disabled. (See Section, “MACHINE LOCK AND AUXILIARY FUNCTION LOCK”.)

1.5 EDITING A PROGRAM

After a created program is once registered in memory, it can be corrected or modified from the MDI unit. (See Chapter, “EDITING PROGRAMS”.)
This operation can be executed using the program edit function.

FRYER MACHINE SYSTEMS INC
Bob Zahensky

1.6 DISPLAYING AND SETTING DATA

The operator can display or change a value stored in CNC internal memory by key operation on the MDI screen. (See Chapter, “SETTING AND DISPLAYING DATA”.)

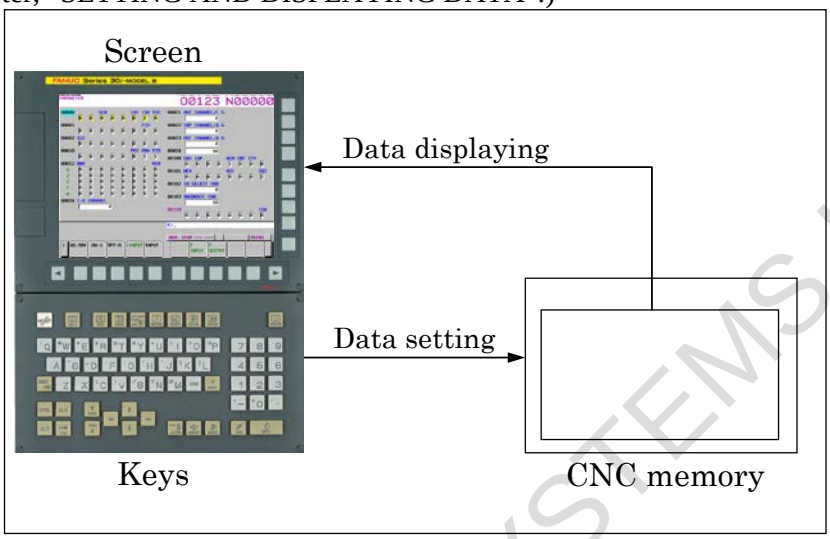


Fig. 1.6 (a) Displaying and setting data

Explanation

- Offset value

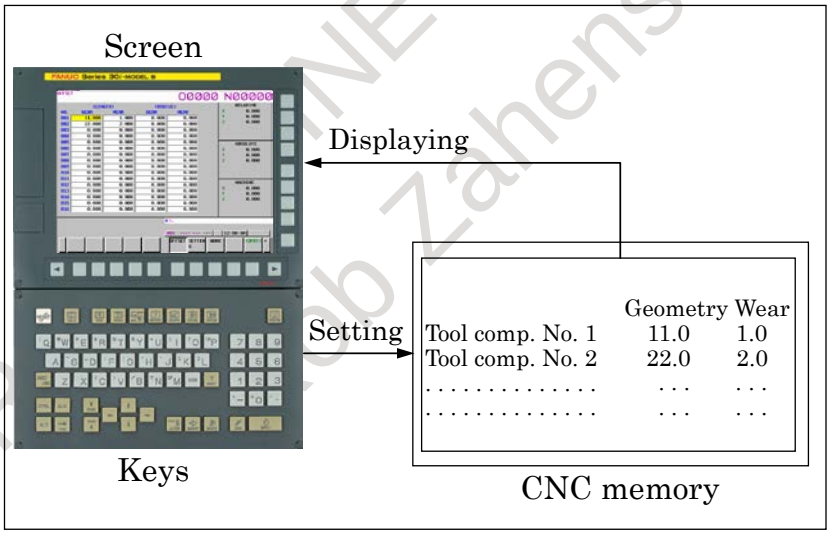


Fig. 1.6 (b) Displaying and Setting Offset Values

The tool has the tool dimension (length, diameter). When a workpiece is machined, the tool movement value depends on the tool dimensions.
By setting tool dimension data in CNC memory beforehand, CNC automatically generates tool routes that permit any tool to cut the workpiece specified by the program. Tool dimension data is called the offset value.

FRYER MACHINE SYSTEMS INC
Jacob Zahensky

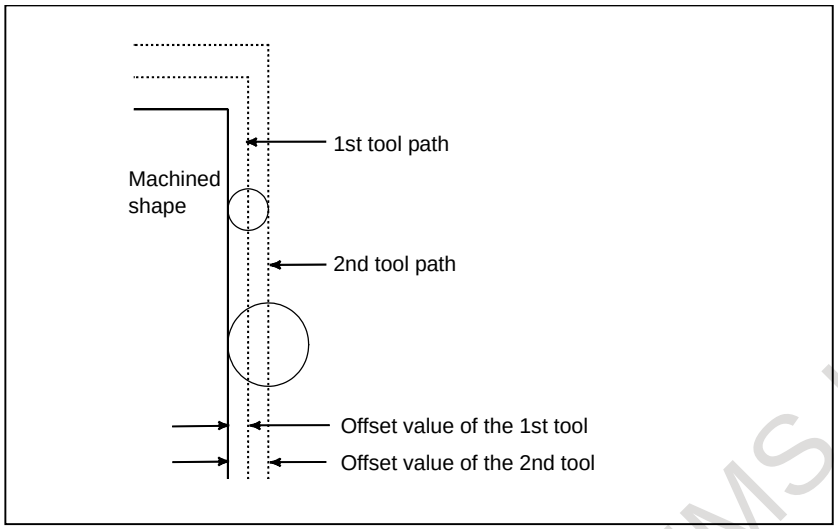


Fig. 1.6 (c) Offset value

- Displaying and setting operator's setting data

Apart from parameters, there is data that is set by the operator in operation. This data causes machine characteristics to change.

For example, the following data can be set:

- Inch/Metric switching
- Selection of I/O devices
- Mirror image cutting on/off

The above data is called setting data. (See Subsection, “Displaying and Entering Setting Data”.)

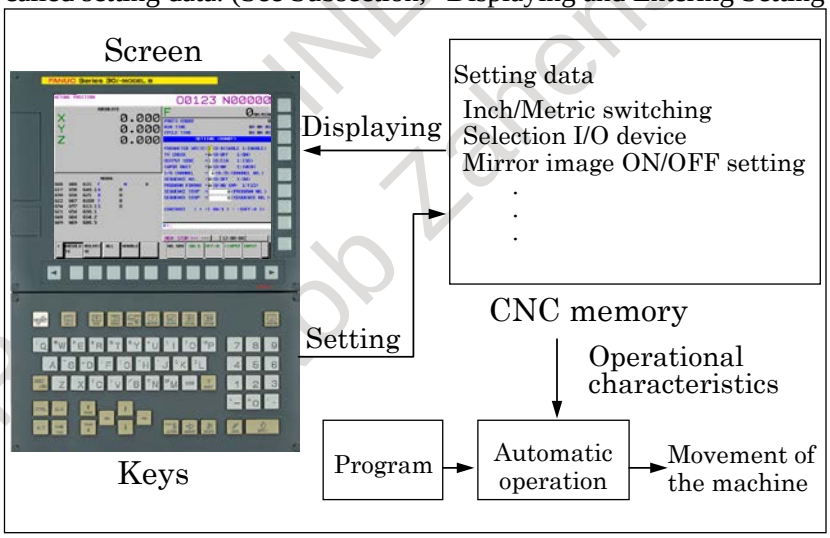


Fig. 1.6 (d) Displaying and setting operator's setting data

FRYER LINE SYSTEMS INC
Bob Zalensky

- Displaying and setting parameters

The CNC functions have versatility in order to take action in characteristics of various machines.
For example, CNC can specify the following:

- Rapid traverse rate of each axis
- Whether increment system is based on metric system or inch system.
- How to set command multiply/detect multiply (CMR/DMR)

Data to make the above specification is called parameters. (See Subsection, “Displaying and Setting Parameters”.)
Parameters differ depending on machine tool.

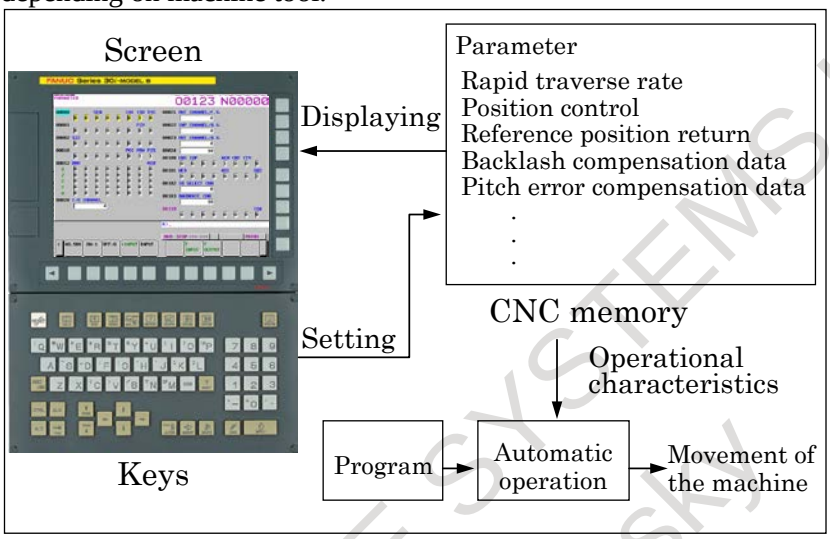


Fig. 1.6 (e) Displaying and setting parameters

- Data protection key

A key called the data protection key can be defined. It is used to prevent part programs, offset values, parameters, and setting data from being registered, modified, or deleted erroneously. (See Chapter, “SETTING AND DISPLAYING DATA”.)

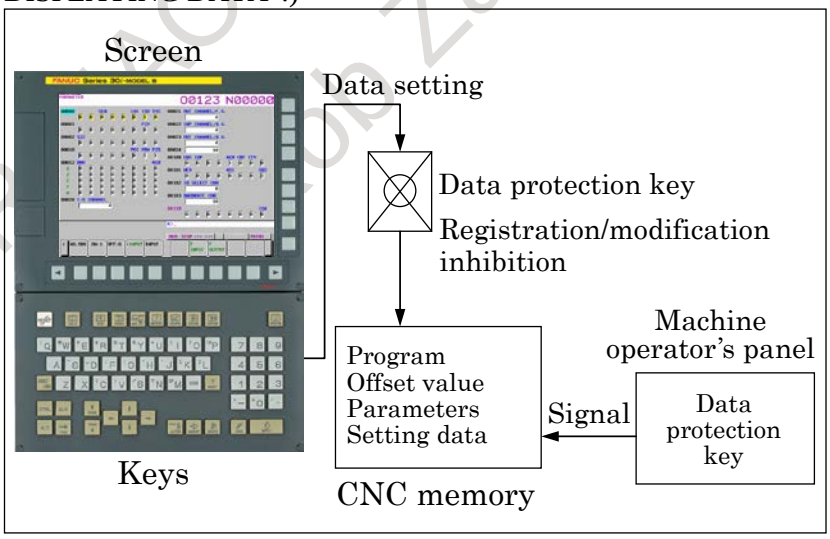


Fig. 1.6 (f) Data protection key

FRYER MACHINE SYSTEMS INC
Bob Zahensky

1.7 DISPLAY

1.7.1 Program Display

The contents of the currently active program are displayed.
(See Subsection, “Program Contents Display”.)

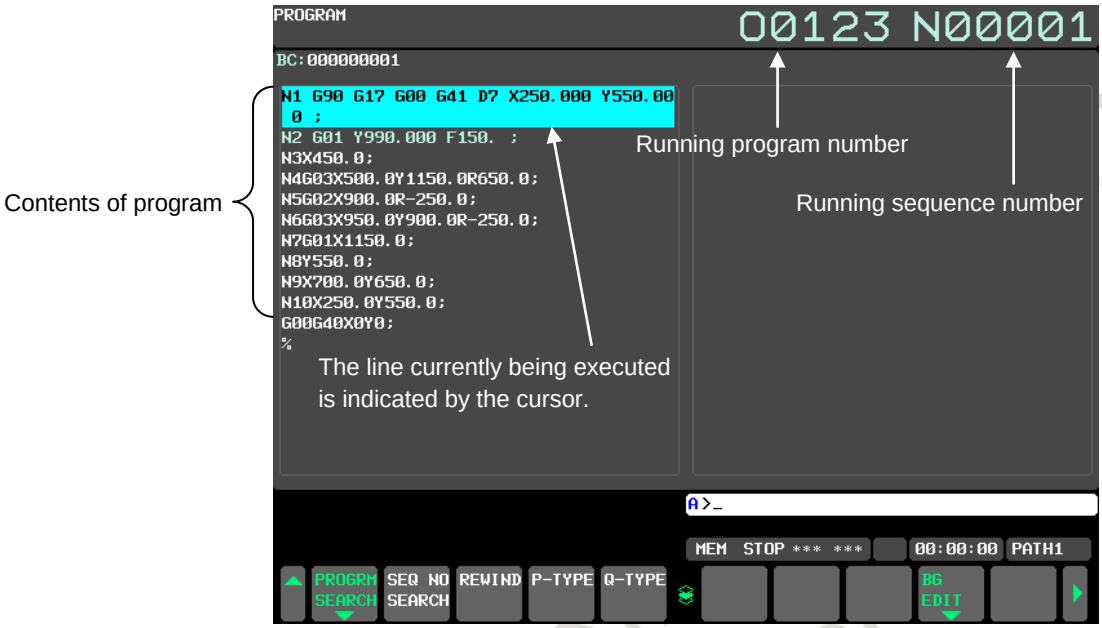


Fig. 1.7.1 (a)

A list of the programs held in the currently selected folder is displayed.

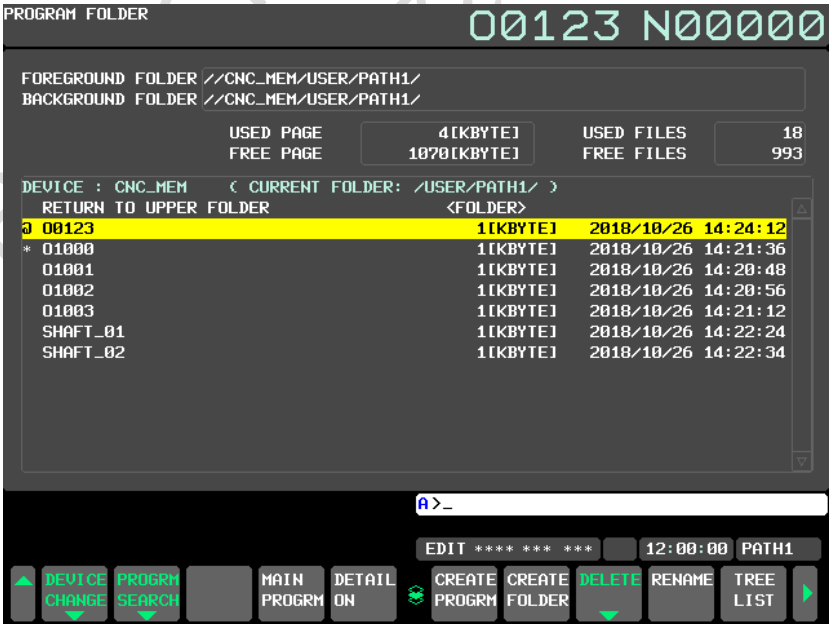


Fig. 1.7.1 (b)

1.7.2 Current Position Display

The current position of the tool is displayed with the coordinate values.
Moreover, the distance from the current position to a target point can be displayed as a remaining travel distance.
(See Subsections, “Position Display in the Workpiece Coordinate System”, “Position Display in the Relative Coordinate System”, and “Overall Position Display”.)

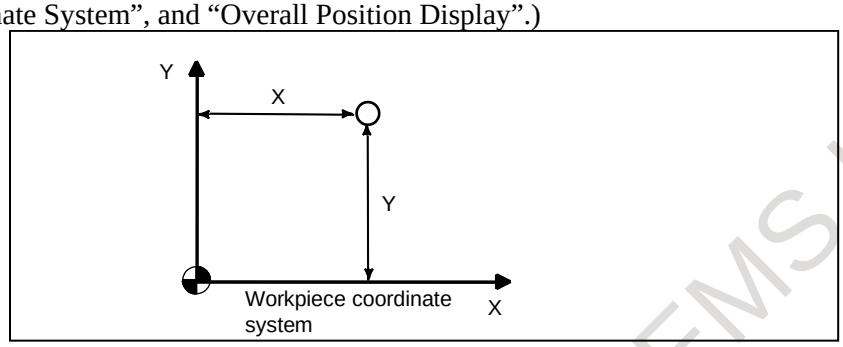


Fig. 1.7.2 (a)

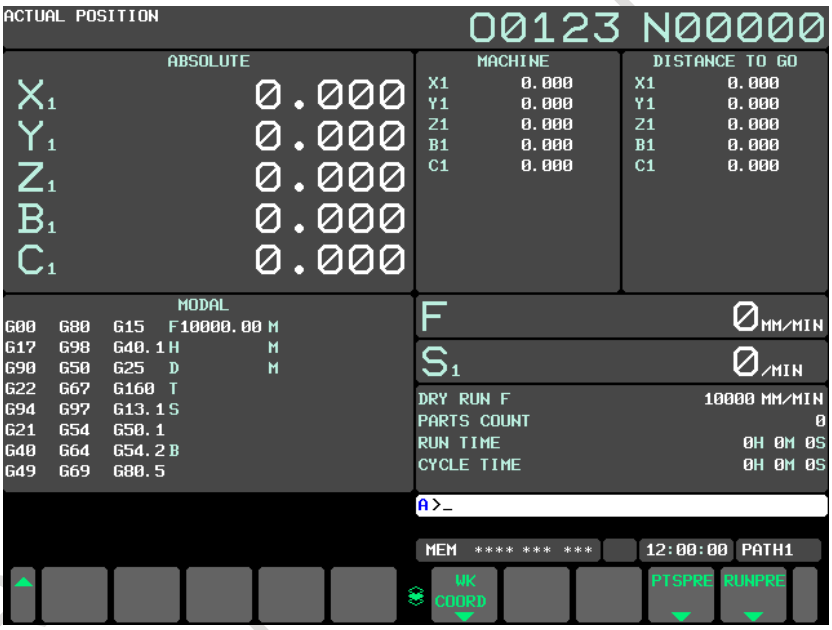


Fig. 1.7.2 (b)

Axis status display of WRONG OPERATION PREVENTION FUNCTIONS

This function displays the axis status to the left of the axis name in the display of the machine coordinates, absolute coordinates, relative coordinates, and distance to go.
For example, when the mirror image function is enabled for the X1 axis, the absolute coordinates are displayed as follows.

```
ABSOLUTE
 M  X1    10.000
    Y1    10.000
    Z1     0.000
```

The axis status is indicated as follows. These indications are listed in order of priority.

AXIS DETACH :	D
FLEXIBLE PATH AXIS ASSIGNMENT:	R
INTERLOCK :	I
MACHINE LOCK :	L
SERVO OFF :	S

TORQUE CONTROL : T
AXIS MOVING : *
MIRROR IMAGE : M
REMOVED AXIS OF

1.7.3 Alarm Display

When a trouble occurs during operation, error code and alarm message are displayed on the screen. (See Section, “ALARM DISPLA”.)
See APPENDIX G for the list of error codes and their meanings.

Fig. 1.7.3

1.7.4 Parts Count Display, Run Time Display

The position display screen displays a run time, cycle time, and parts count. (See Subsection, “Displaying and Setting Run Time, Parts Count, and Time”.)

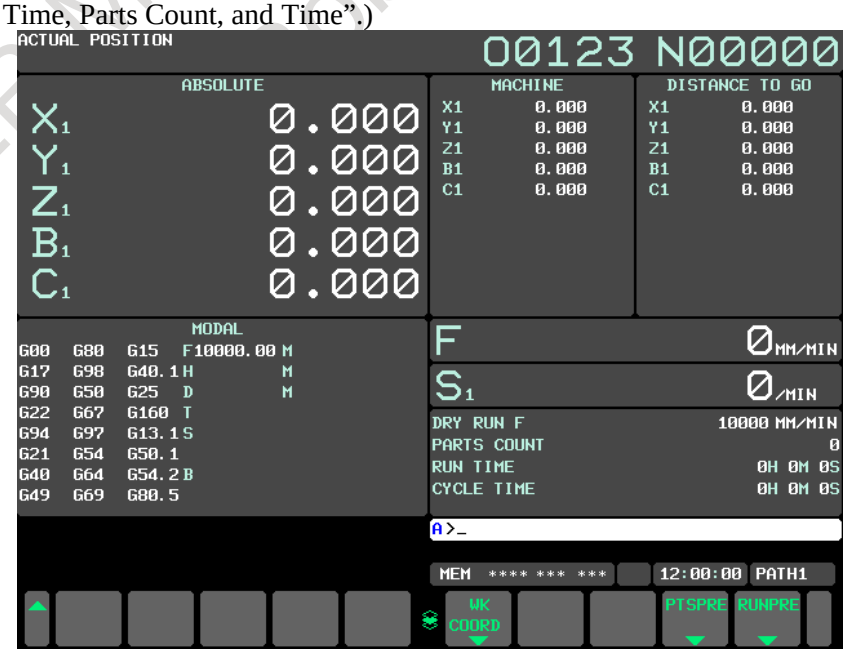


Fig. 1.7.4

2 OPERATIONAL DEVICES

As operational devices, setting and display devices attached to the CNC, and machine operator's panels are available.
For machine operator's panels, refer to the relevant manual of the machine tool builder.

2.1 POWER ON/OFF

2.1.1 Turning on the Power

Procedure of turning on the power

Procedure

1 Check that the appearance of the CNC machine tool is normal.
(For example, check that front door and rear door are closed.)
2 Turn on the power according to the manual issued by the machine tool builder.
3 After the power is turned on, check that the position screen is displayed. An alarm screen is displayed if an alarm occurs upon power-on.

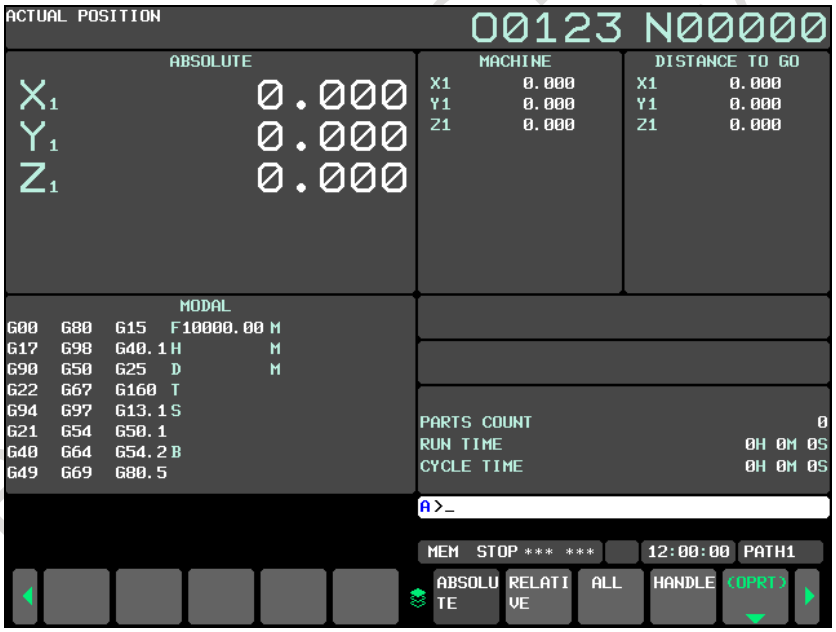


Fig. 2.1.1 (a) Position screen (for machining center system)

4 Check that the fan motor is rotating.

> ⚠ **CAUTION**
> Until the positional or alarm screen is displayed at the power on, do not touch a key on the MDI unit. Some keys are used for the maintenance or special operation purpose. When they are pressed, unexpected operation may be caused.

FRYER SYSTEMS INC

2.1.2 Power Disconnection

Procedure of power disconnection

Procedure

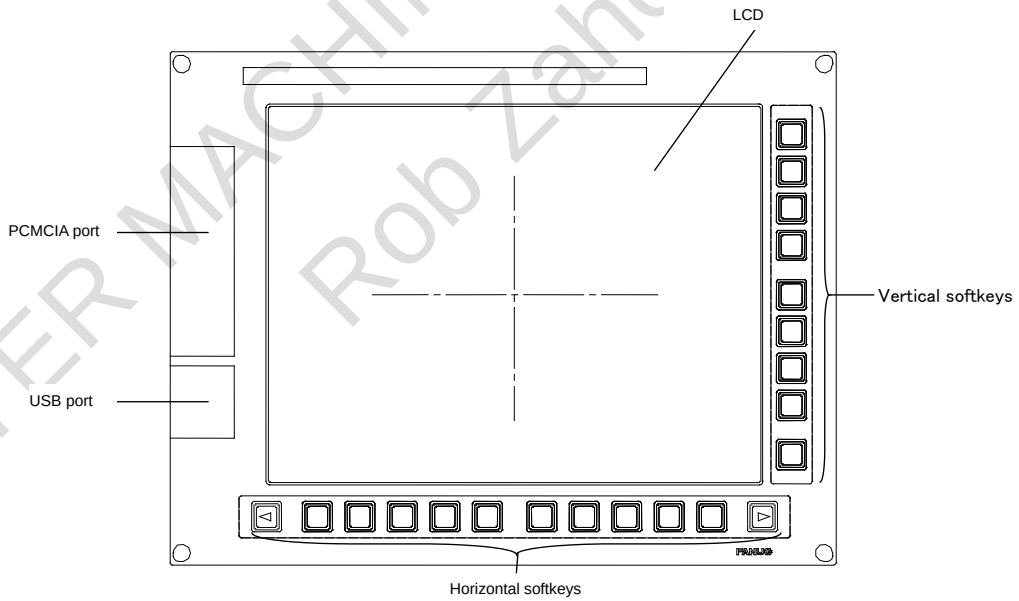
1 Check that the LED indicating the cycle start is off on the operator's panel.
2 Check that all movable parts of the CNC machine tool is stopping.
3 If an external input/output device such as the Handy File is connected to the CNC, turn off the external input/output device.
4 Continue to press the <POWER OFF> button for about 5 seconds.
5 Refer to the machine tool builder's manual for turning off the power to the machine.

2.2 SETTING AND DISPLAY UNITS

The setting and display units are shown in Subsections 2.1.1 to 2.1.4 of Part III.

10.4" LCD CNC Display Panel (15"/19" LCD CNC Display Panel) III-2.2.1
Standard MDI Unit (ONG Key) III-2.2.2
Standard MDI Unit (QWERTY Key) III-2.2.3
Small MDI Unit (ONG Key) III-2.2.4

2.2.1 10.4" LCD CNC Display Panel (15"/19" LCD CNC Display Panel)



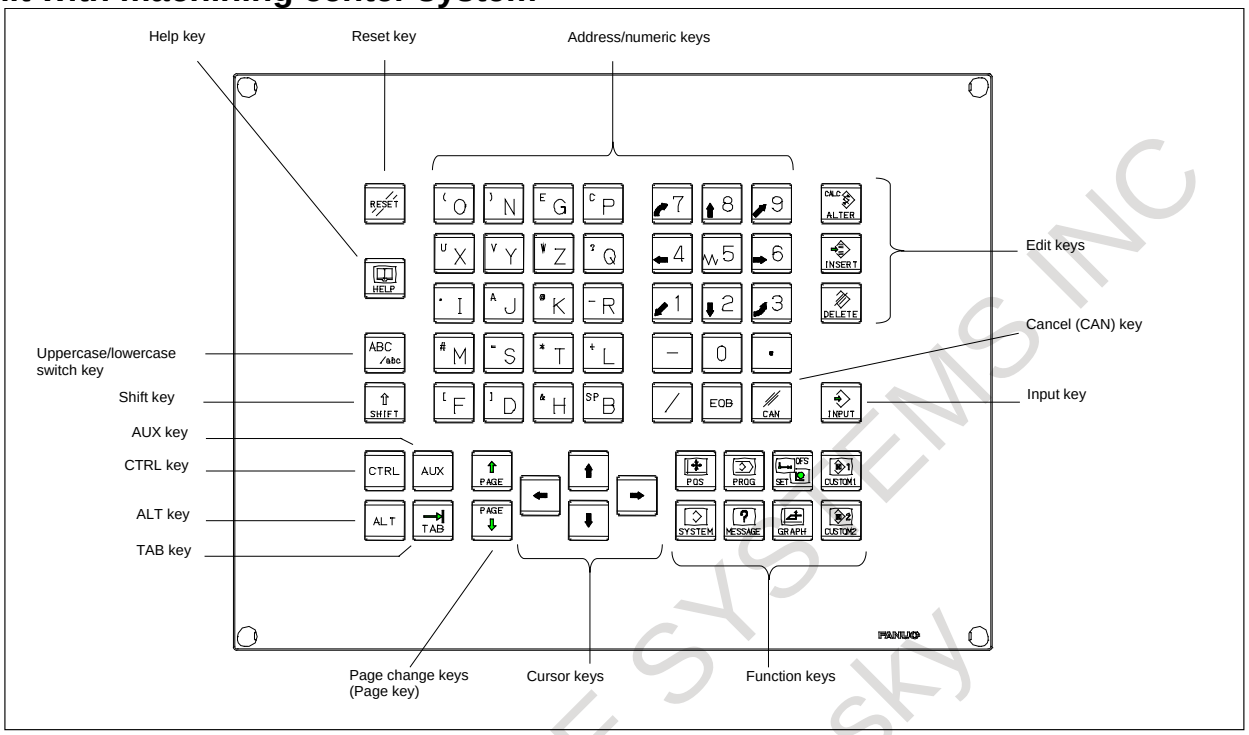
10.4"LCD CNC Display Panel

NOTE: The configuration of 15"/19" LCD CNC Display Panel is equivalent to that of 10.4" LCD CNC Display Panel.

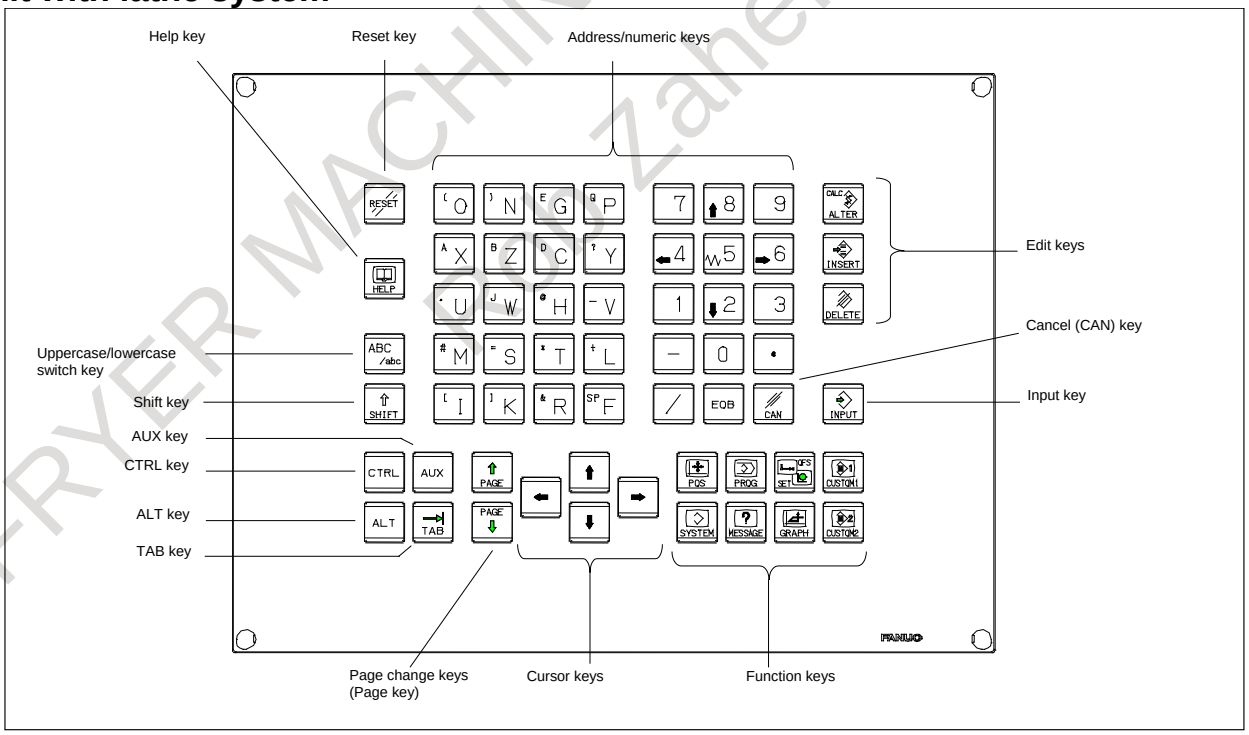
FRYER MACHINE SYSTEMS INC
Rob Zahensky

2.2.2 Standard MDI Unit (ONG Key)

Unit with machining center system

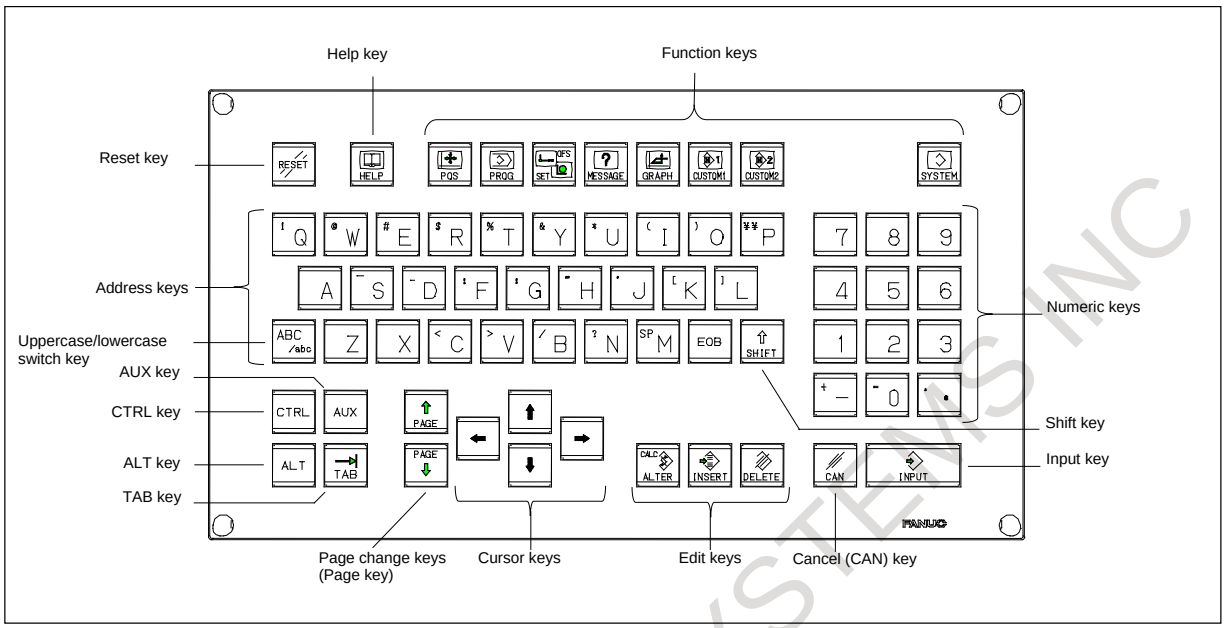


Unit with lathe system



FRYER MACHINE SYSTEMS INC
Joe Zahensky

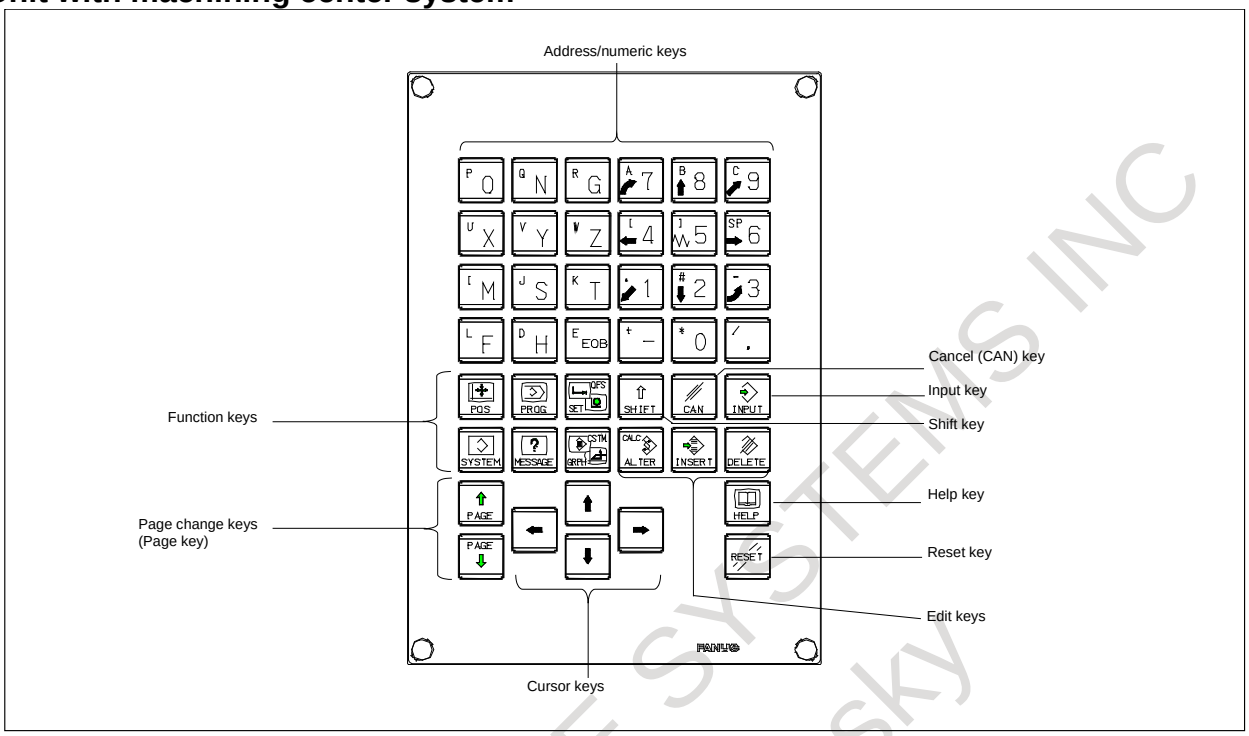
2.2.3 Standard MDI Unit (QWERTY Key)



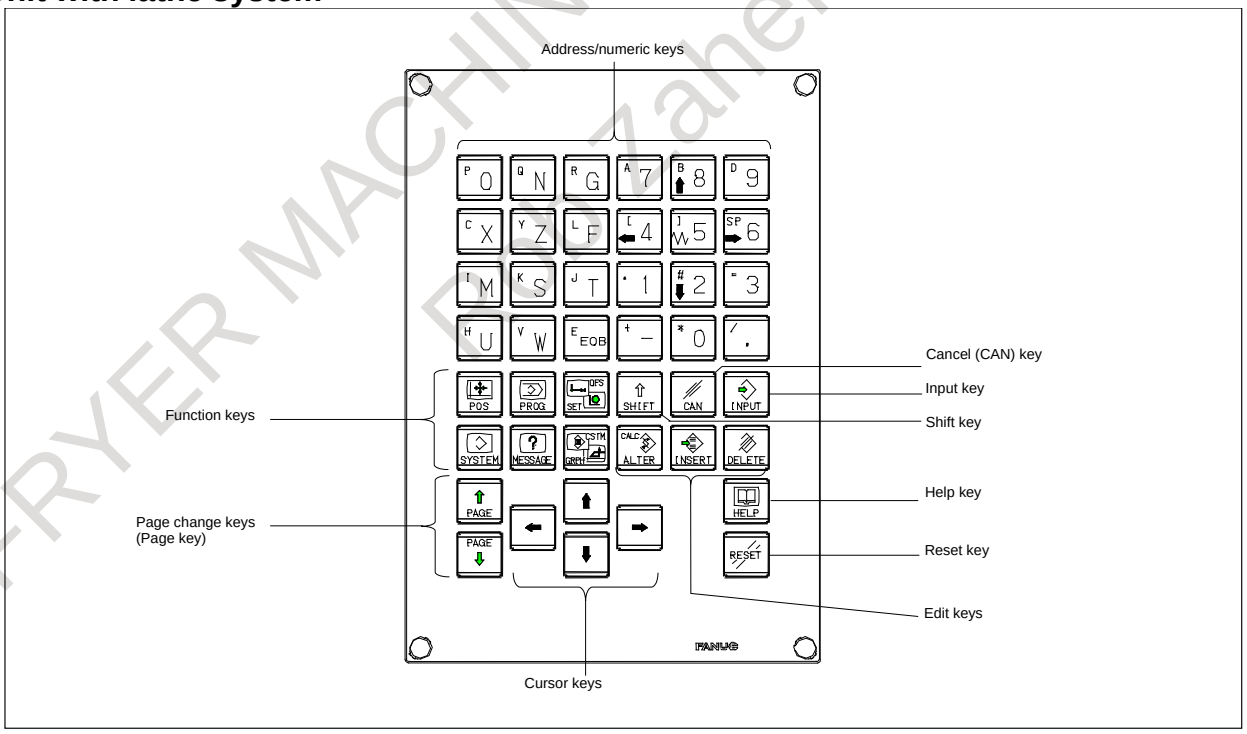
FRYER MACHINE SYSTEMS INC
Rob Zahensky

2.2.4 Small MDI Unit (ONG Key)

Unit with machining center system



Unit with lathe system



FRYER MACHINE SYSTEMS INC
Eric Zahensky

2.3 EXPLANATION OF THE MDI UNIT

Table 2.3 Explanation of the MDI unit

No.	Name	Explanation
1	RESET key	Press this key to reset the CNC, to cancel an alarm, etc.
2	HELP key	Press this key to use the help function when uncertain about the operation of an MDI key (help function).
3	Soft keys	The soft keys have various functions, according to the Applications. The soft key functions are displayed on the display unit.
4	Address and numeric keys	Press these keys to input alphabetic, numeric, and other characters.
5	SHIFT key	Some keys have two characters on their key top. Pressing the <SHIFT> key switches the characters. Special character ^ is displayed on the screen when a character indicated at the bottom right corner on the key top can be entered.
6	INPUT key	When an address or a numerical key is pressed, the data is input to the buffer, and it is displayed on the screen. To copy the data in the key input buffer to the offset register, etc., press the INPUT key. This key is equivalent to the [INPUT] key of the soft keys, and either can be pressed to produce the same result. This key is also used to move to a folder on the program folder screen.
7	CANCEL (CAN) key	Press this key to delete the last character or symbol input to the key input buffer. Example) When the key input buffer displays >N001X100Z_ and the cancel key CAN is pressed, Z is canceled and >N001X100_ is displayed.
8	Edit keys	Press these keys when editing the program. ALTER : ALTER INSERT : INSERT DELETE : DELETE
9	Function keys	Press these keys to switch display screens for each function. See Section, “Function Keys” for details of the function keys.
10	Cursor keys	There are four different cursor move keys. : This key is used to move the cursor to the right or in the forward direction. The cursor is moved in short units in the forward direction. : This key is used to move the cursor to the left or in the reverse direction. The cursor is moved in short units in the reverse direction. : This key is used to move the cursor in a downward or forward direction. The cursor is moved in large units in the forward direction. : This key is used to move the cursor in an upward or reverse direction. The cursor is moved in large units in the reverse direction.

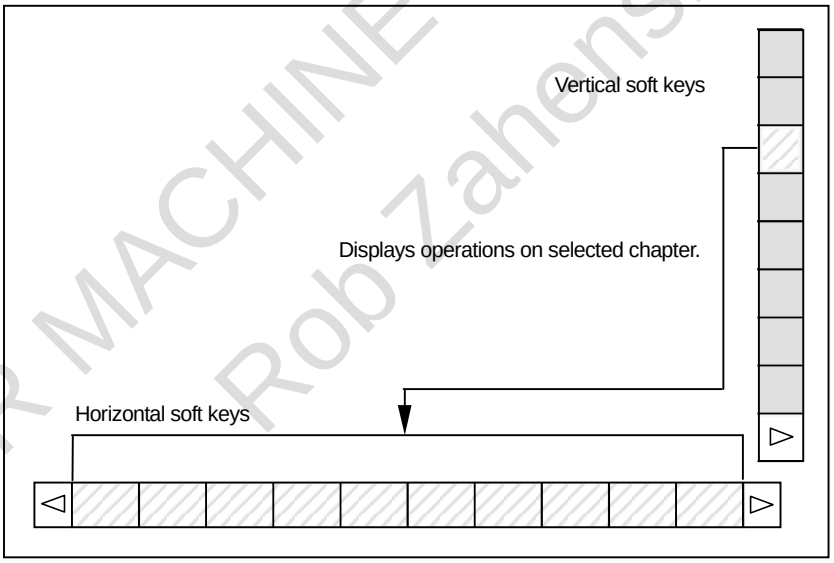
FRYER MACHINE SYSTEMS INC
Rob Zaharsky

No.	Name	Explanation
11	Page change keys (Page keys)	Two kinds of page change keys are described below. PAGE ⇩ : This key is used to changeover the page on the screen in the forward direction. PAGE ⇧ : This key is used to changeover the page on the screen in the reverse direction.
12	Uppercase/lowercase switch key	Press this key to switch between uppercase and lowercase when entering alphabetic characters.
13	PC function key	These keys are used with the personal computer functions.

Explanation

- 15” LCD/MDI soft key configuration

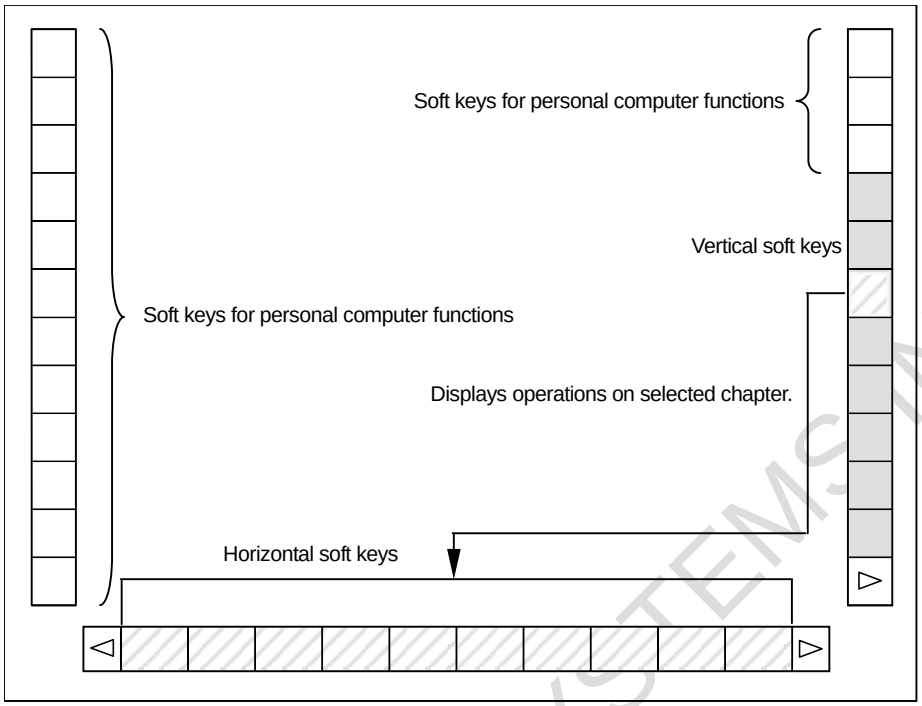
The 15” LCD panel has 12 soft keys horizontally and 9 soft keys vertically.
As shown below, the 8 vertical soft keys and the lowermost key are used as chapter selection soft keys. By pressing each of these keys, the screen (chapter) belonging to each function can be selected. The horizontal 12 soft keys are used to perform operations on the screen selected with a vertical soft key.
For an LCD display with a touch panel, touch a soft key on the screen to select it.



- 19” LCD/MDI soft key configuration

The 19” LCD panel has 12 soft keys horizontally and 12 soft keys vertically.
As shown below, the 9 lower vertical soft keys on the right side and the 12 horizontal soft keys are used for CNC screen operations and the other soft keys are available for personal computer functions.
Use the 9 lower vertical soft keys on the right side and 12 horizontal soft keys for CNC screen operations in the same way as for the 15” LCD panel.

FRYER MACHINE SYSTEMS INC
Rob Zahensky



In this manual, the descriptions below assume a 10.4" LCD display panel with 12 soft keys.

- MDI unit operation with multi-path control

In the multi-path control, be sure to select the tool post for which data is specified, using the path selection switch on the machine operator's panel. Then, perform MDI unit operation, such as displaying or specifying various data items, and editing a program.

2.4 FUNCTION KEYS AND SOFT KEYS

The function keys are used to select the type of screen (function) to be displayed. When a soft key (section select soft key) is pressed immediately after a function key, the screen (section) corresponding to the selected function can be selected.

2.4.1 General Screen Operations

- Procedure

1 By pressing a function key on the MDI unit, the chapter selection soft keys that belong to the function are displayed.

Example 1)

2 When one of the chapter selection soft keys is pressed, the screen of the chapter is displayed. If the soft key of a desired chapter is not displayed, press the continuous menu key.
In a chapter, a further choice may be made from multiple chapters.

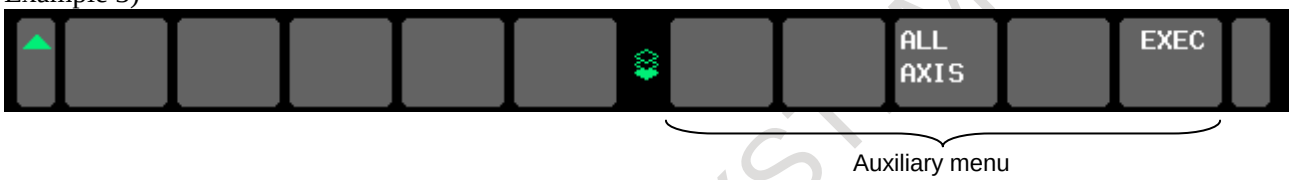
FRYER MACHINE SYSTEMS INC
Rob Zohensky

3 When the screen of a desired chapter is displayed, press the operation selection key to display operations to be performed.

4 Select a desired operation with the operation selection soft key.
Depending on the operation to be executed, an auxiliary menu of soft keys is displayed. Perform an operation according to the indications on the auxiliary menu.

Example 2)

Example 3)



5 To return to the display of chapter selection soft keys, press the return menu key.

A general screen display procedure is provided above.
The actual display procedure varies from one screen to another.
For details, see each description of operation.

- Button design change depending on soft key state

The soft keys assume one of the following states, depending on the selection target:
- Chapter selection soft keys
- Operation selection soft keys
- Auxiliary menu of operation selection soft keys

Depending on the state, the button images of the soft keys change.
From the button images, which state the soft keys are assuming can be known.

Example)
- Chapter selection soft keys

- Operation selection soft keys

- Auxiliary menu of operation selection soft keys

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

2.4.2 Function Keys

Function keys are provided to select the type of screen to be displayed. The following function keys are provided on the MDI unit:

Press this key to display the position screen.

Press this key to display the program screen.

Press this key to display the offset/setting screen.

Press this key to display the system screen.

Press this key to display the message screen.

Press this key to display the graphics screen.

Press this key to display the custom screen 1 (conversational macro screen or C Language Executor screen).

Press this key to display the custom screen 2 (C Language Executor screen).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

2.4.3 Soft Keys

By pressing a soft key after a function key, the corresponding screen of the function can be displayed.

The chapter selection soft keys of each function are described below.
The horizontal four keys on the right-hand side are assigned to chapter selection soft keys. When multiple pages are used for chapter selection soft keys, [+] is displayed on the continuous menu key (rightmost soft key). Press the continuous menu key to switch between chapter selection soft keys.

NOTE
1 Press function keys to switch between screens that are used frequently.
2 Some soft keys are not displayed depending on the option configuration.

If position indications are provided on the left half of the screen when a key other than the function key

is pressed, the left half of the soft keys is displayed as follows at all times:

Position display screen

The chapter selection soft keys that belong to the function key

and the function of each screen are described below.

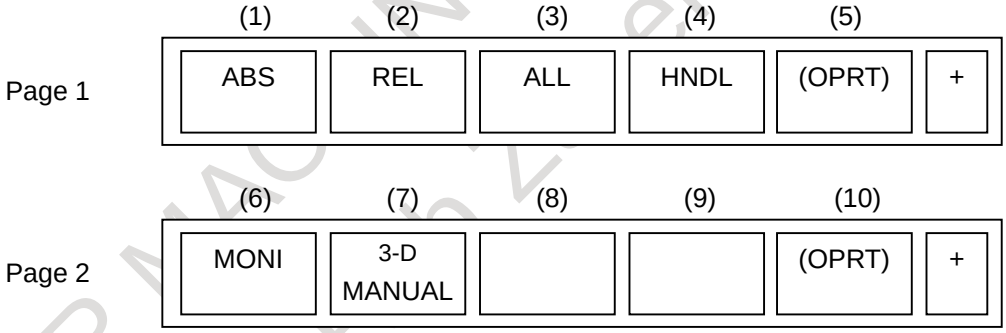


Table 2.4.3 (a) Position display screen

No.	Chapter menu	Description
(1)	ABS	Selects the absolute coordinate display screen.
(2)	REL	Selects the relative coordinate display screen.
(3)	ALL	Selects the overall coordinate display screen.
(4)	HNDL	Selects the operation screen for manual handle operation.
(6)	MONI	Selects the screen for displaying the servo axis load meter, serial spindle load meter, and speedometer.
(7)	3-D MANUAL	Displays a handle pulse interrupt amount in 3-dimensional manual feed.

FRYER MACHINE SYSTEMS INC
Pub Zaremsky

Program screen

The chapter selection soft keys that belong to the function key

and the function of each screen are described below.

Page 1

(1)	(2)	(3)	(4)	(5)	
PROGRAM	FOLDER	NEXT	CHECK	(OPRT)	+

Page 2

(6)	(7)	(8)	(9)	(10)	
TIME	JOG	RSTR	ROBOT SELECT	(OPRT)	+

Table 2.4.3 (b) Program

No.	Chapter menu	Description
(1)	PROGRAM	Selects the screen for editing or displaying a program.
(2)	FOLDER	Selects the screen for displaying a list of part programs currently registered.
(3)	NEXT	Selects the screen for displaying the command values of the block currently executed and the next block to be executed among the command values.
(4)	CHECK	Selects the screen for displaying programs, position data, modal information, and so forth simultaneously.
(6)	TIME	Selects the screen for displaying executed program operation time.
(7)	JOG	Selects the screen for executing, in the JOG mode, data specified in the program format from the MDI.
(8)	RSTR	Selects the operation screen for restarting an interrupted program operation.
(9)	ROBOT SELECT	Selects the screen for registering robot programs and part programs for the Robot connection function.

Offset/setting screen

The chapter selection soft keys that belong to the function key

and the function of each screen are described below.

Page 1

(1)	(2)	(3)	(4)	(5)	
OFFSET	SETTING	WORK		(OPRT)	+

Page 2

(6)	(7)	(8)	(9)	(10)	
MACRO		OPR	TOOL MANAGER	(OPRT)	+

Page 3

(11)	(12)	(13)	(14)	(15)	
OFST.2	W.SHFT	GEOM.2		(OPRT)	+

FRYER MACHINE SYSTEMS INC
Bob Zahensky

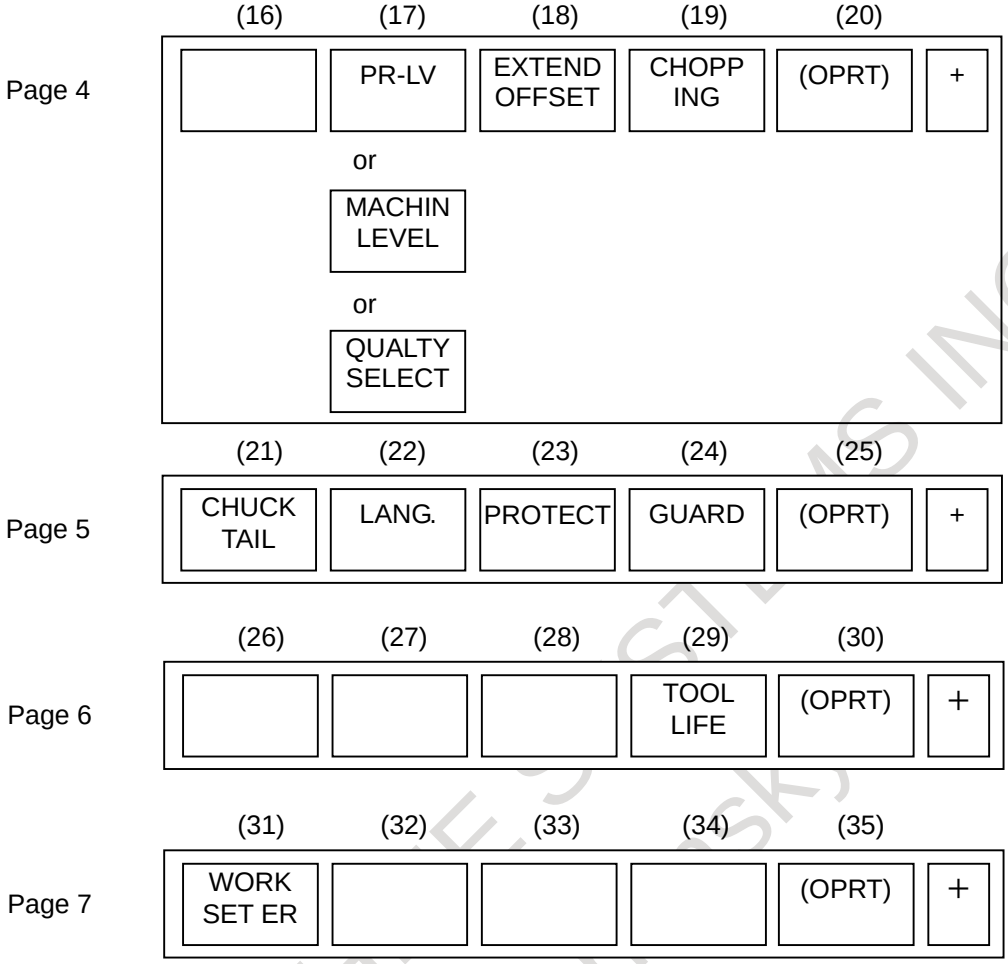


Table 2.4.3 (c) Offset

No.	Chapter menu	Description
(1)	OFFSET	Selects the screen for setting tool offset values.
(2)	SETTING	Selects the screen for setting the setting parameters.
(3)	WORK	Selects the screen for setting a workpiece coordinate system offset.
(6)	MACRO	Selects the screen for setting macro variables.
(8)	OPR	Selects the screen for operating some operation switches on the machine operator's panel as soft switches.
(9)	TOOL MANAGER	Selects the screen for setting data related to tool management.
(11)	OFST.2	Selects the screen for setting a Y-axis offset.
(12)	W.SHFT	Selects the screen for setting a workpiece coordinate system shift value.
(13)	GEOM.2	Selects the screen for setting a second geometry offset.
(17)	PR-LV	Selects the screen for setting a precision level.
(18)	EXTEND OFFSET	Selects the screen for setting the offsets of the fourth and fifth axes.
(19)	CHOPPING	Selects the screen for setting about chopping.
(21)	CHUCK TAIL	Selects the chuck tail stock barrier screen.
(22)	LANG.	Selects the screen for setting a display language.
(23)	PROTECT	Selects the screen for setting data protection.
(24)	GUARD	Selects the screen for setting wrong operation prevention.
(29)	TOOL LIFE	Selects the screen for operations and setting related to tool life management.
(31)	WORK SET ER	Selects the screen for setting errors related to workpiece mounting position.

FRYER MACHINE SYSTEMS INC Rob Zaremsky

System screen

The chapter selection soft keys that belong to the function key

and the function of each screen are described below.

	(1)	(2)	(3)	(4)	(5)	
Page 1	PARAM	DGNOS	SERVO GUIDEM	SYSTEM	(OPRT)	+

	(6)	(7)	(8)	(9)	(10)	
Page 2	MEMORY	PITCH	SERVO PARAM	SP.SET	(OPRT)	+

	(11)	(12)	(13)	(14)	(15)	
Page 3	PMC MAINTE	PMC LADDER	PMC CONFIG			+

	(16)	(17)	(18)	(19)	(20)	
Page 4	MCNG TUNING	ALL IO	ALL IO	OPEHIS	(OPRT)	+

	(21)	(22)	(23)	(24)	(25)	
Page 5	COLOR	MAINTE	M-INFO	W. DGNS	(OPRT)	+

	(26)	(27)	(28)	(29)	(30)	
Page 6		FSSB	PRMTUN	P.MATE MGR.	(OPRT)	+

	(31)	(32)	(33)	(34)	(35)	
Page 7	EMBED PORT	PCMCIA LAN	ETHER NET	PROFI MASTER	(OPRT)	+

	(36)	(37)	(38)	(39)	(40)	
Page 8	REMOTE DIAG	M CODE		3D ERR COMP	(OPRT)	+

	(41)	(42)	(43)	(44)	(45)	
Page 9	PROFI SLAVE	DEVNET MASTER	FL-net 1CH	DEVNET SLAVE	(OPRT)	+

FRYER MACHINE SYSTEMS INC

	(46)	(47)	(48)	(49)	(50)	
Page 10	DUAL CHECK	R.TIME MACRO			(OPRT)	+
	(51)	(52)	(53)	(54)	(55)	
Page 11	ID-INF				(OPRT)	+
	(56)	(57)	(58)	(59)	(60)	
Page 12	CCLink REMOTE	ROBOT STATUS	ROBOT CON.		(OPRT)	+
	(61)	(62)	(63)	(64)	(65)	
Page 13	USB	FL-net 2CH			(OPRT)	+

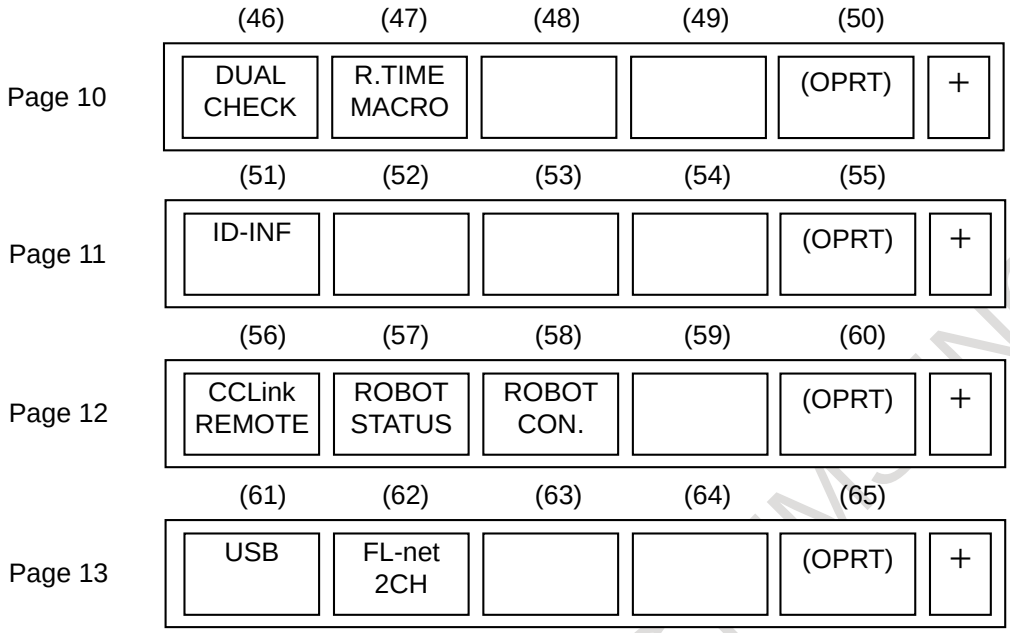


Table 2.4.3 (d) System

No.	Chapter menu	Description
(1)	PARAM	Selects the screen for setting parameters.
(2)	DGNOS	Selects the screen for displaying CNC state.
(3)	SERVO GUIDEM	Selects the screen for displaying the servo guide mate.
(4)	SYSTEM	Selects the screen for displaying the current system status.
(6)	MEMORY	Selects the screen for displaying the contents of memory.
(7)	PITCH	Selects the screen for setting pith error compensation.
(8)	SERVO PARAM	Selects the screen for setting the servo-related parameters.
(9)	SP.SET	Selects the screen for spindle-related setting.
(11)	PMC MAINTE	Selects the screen related to PMC maintenance such as PMC signal state monitoring and tracing, and PMC parameter display/editing.
(12)	PMC LADDER	Selects the screen related to ladder display/editing.
(13)	PMC CONFIG	Displays the screen for displaying/editing data other than ladders that makes up a sequence program and for setting the PMC function.
(16)	MCNG TUNING	Displays the screen for setting the parameter set for emphasis on speed (LV1) or emphasis on precision (LV10).
(17)	ALL IO (RS-232C interface)	Selects the screen for data input and output.
(18)	ALL IO (Memory card)	Selects the screen for data input to and output from the memory card.
(19)	OPEHIS	Selects the screen for displaying the history of operations performed by the operator and issued alarms.
(21)	COLOR	Selects the screen for setting colors to be used on the screen.
(22)	MAINTE	Selects the screen for setting maintenance items to be managed periodically.
(23)	M-INFO	Selects the screen for displaying information about maintenance performed.
(24)	W.DGNS	Selects the screen for displaying data such as servo positional deviation values, torque values, machine signals, and so forth as graphs.
(27)	FSSB	Selects the screen for making settings related to the high-speed serial servo bus (FSSB: Fanuc Serial Servo Bus).
(28)	PRMTUN	Selects the screen for setting parameters necessary for start-up and tuning.
(29)	P.MATE MGR.	Selects the screen for making settings related to Power mate manager.
(31)	EMBED PORT	Selects the screen for making settings related to the embedded Ethernet (embedded port).

FRYER MACHINE SYSTEMS INC Rob Zanensky

No.	Chapter menu	Description
(32)	PCMCIA LAN	Selects the screen for making settings related to the embedded Ethernet (PCMCIA Ethernet card).
(33)	ETHERNET	Selects the screen for making settings related to the fast Ethernet/fast data server.
(34)	PROFI MASTER	Selects the screen for making settings related to the PROFIBUS master function.
(36)	REMOTE DIAG	Selects the screen for making settings related to the Machine Remote Diagnosis.
(37)	M CODE	Selects the screen for setting an M code group.
(39)	3D ERR COMP	Selects the screen for making settings related to the 3-dimensional Error Compensation.
(41)	PROFI SLAVE	Selects the screen for making settings related to the PROFIBUS slave function.
(42)	DEVNET MASTER	Selects the screen for making settings related to the DeviceNet master function.
(43)	FL-net 1CH	Selects the screen for making settings related to the FL-net (port 1) function.
(44)	DEVNET SLAVE	Selects the screen for making settings related to the DeviceNet slave function.
(46)	DUAL CHECK	Select sthe screen for making settings related to the Dual check safety function.
(47)	R.TIME MACRO	Select the screen for making settings related to real-time custom macro.
(51)	ID-INF	Selects the screen for displaying CNC ID information.
(56)	CCLink REMOTE	Selects the screen for making settings related to the CC-Link remote device function.
(57)	ROBOT STATUS	Selects the robot status screen for the Robot connection function.
(58)	ROBOT CON.	Selects the screen for making settings related to the Robot connection function.
(61)	USB	Selects the USB maintenance screen.
(62)	FL-net 2CH	Selects the screen for making settings related to the FL-net (port 2) function.

Message screen

The chapter selection soft keys that belong to the function key and the function of each screen are described below.

	(1)	(2)	(3)	(4)	(5)	
Page 1	ALARM	MSG	HISTRY	MSGHIS		+

	(6)	(7)	(8)	(9)	(10)	
Page 2	EMBED LOG	PCMCIA LOG	ETHER LOG	FL-net 1CH		+

	(11)	(12)	(13)	(14)	(15)	
Page 3		USB LOG	FL-net 2CH			+

	(16)	(17)	(18)	(19)	(20)	
Page 4	DNET M HISTRY	DNET S HISTRY				+

FRYER MACHINE SYSTEMS INC
Bob Zaremsky

Table 2.4.3 (e) Message

No.	Chapter menu	Description
(1)	ALARM	Selects the alarm message screen.
(2)	MSG	Selects the operator message screen.
(3)	HISTRY	Selects the screen for displaying the details of alarms issued so far.
(4)	MSGHIS	Selects the external operator message screen.
(6)	EMBED LOG	Selects the screen for displaying error messages related to the embedded Ethernet (embedded port).
(7)	PCMCIA LOG	Selects the screen for displaying error messages related to the embedded Ethernet (PCMCIA Ethernet card).
(8)	ETHER LOG	Selects the screen for displaying error messages related to the fast Ethernet/fast data server.
(9)	FL-net 1CH	Selects the screen for displaying error messages related to the FL-net (port 1) function.
(12)	USB LOG	Selects the screen for displaying error messages related to the USB function.
(13)	FL-net 2CH	Selects the screen for displaying error messages related to the FL-net (port 2) function.
(16)	DNET M HISTRY	Selects the screen for displaying communication histories related to the DeviceNet master function.
(17)	DNET S HISTRY	Selects the screen for displaying communication histories related to the DeviceNet slave function.

Graphic screen

The chapter selection soft keys that belong to the function key GRAPH and the function of each screen are described below.

When the graphic display function is enabled:

	(1)	(2)	(3)	(4)	(5)	
Page 1	PARAM	GRAPH			(OPRT)	+

Table 2.4.3 (f) Graphic

No.	Chapter menu	Description
(1)	PARAM	Selects the screen for setting graphic parameters.
(2)	GRAPH	Selects the screen for graphically displaying the tool path.

When the dynamic graphic display function is enabled:

	(1)	(2)	(3)	(4)	(5)	
Page 1	DRAW PARAM	PATH EXEC	ANIME EXEC	TOOL POS	(OPRT)	

Table 2.4.3 (g) Graphic

No.	Chapter menu	Description
(1)	DRAW PARAM	Selects the screen for setting drawing parameters.
(2)	PATH EXEC	Selects the screen for drawing tool paths.
(3)	ANIME EXEC	Selects the screen for drawing animation.
(4)	TOOL POS	Selects the screen for displaying tool positions.

FRYER MACHINE SYSTEMS INC
Robert Shensky

2.5 EXTERNAL I/O DEVICES

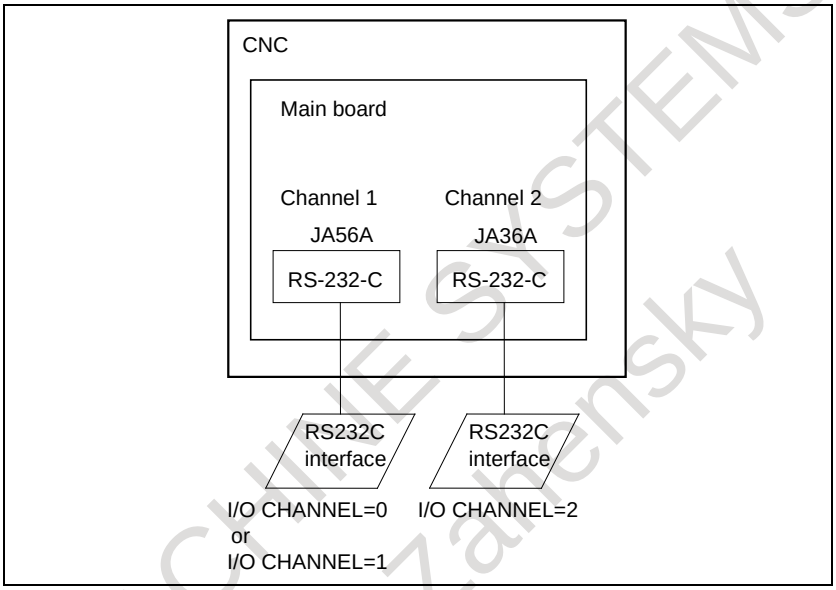
External I/O devices such as a memory card are available.
By using an external I/O device such as a memory card, the following data can be input or output:

1. Programs
2. Offset data
3. Parameters
4. Custom macro common variables

For how to input or output data and input data from or output it to a memory card, see Chapter, “DATA INPUT/OUTPUT”.

- Parameter setting

Before an external input/output device can be used, parameters must be set as follows.



This CNC has a total of two channels of RS-232C interfaces. It also has memory card interface and USB interface. The input/output device to be used is specified by setting the channel (interface) connected to that device in setting parameter I/O CHANNEL.
The specified data, such as a baud rate and the number of stop bits, of an input/output device connected to a specific channel must be set in parameters for that channel in advance. (These settings are not required for the memory card interface and USB interface.)
For channel 1, two combinations of parameters to specify the input/output device data are provided.
The following shows the interrelation between the RS-232C interface parameters for the channels.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

0020 I/O CHANNEL
or foreground input

Set channels to be used
for data input/output.

I/O CHANNEL (0 to 5)
=0 : Channel 1
=1 : Channel 1
=2 : Channel 2
=3 : Channel 3
:
:
:

Input/output to and from the memory
card interface, etc. is also possible.

When IO4 is set

0021 Foreground output

0022 Background input

0023 Background input

The channel setting is the same as
No.0020.

Input/output channel number (parameter No.0020)

↓

I/O CHANNEL=0
(Channel 1)
- 0101 Stop bit and other data
- 0102 Number specified for the input/output device
- 0103 Baud rate

I/O CHANNEL=1
(Channel 1)
- 0111 Stop bit and other data
- 0112 Number specified for the input/output device
- 0113 Baud rate

I/O CHANNEL=2
(Channel 2)
- 0121 Stop bit and other data
- 0122 Number specified for the input/output device
- 0123 Baud rate

:
:
:

I/O CHANNEL=5

FRYER MACHINE SYSTEMS INC
Rob Zahensky

3 MANUAL OPERATION

3.1 MANUAL REFERENCE POSITION RETURN

The tool is returned to the reference position as follows :
The tool is moved in the direction specified in bit 5 (ZMI) of parameter No.1006 for each axis with the reference position return switch on the machine operator's panel. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed. The rapid traverse rate and FL speed are specified in parameters Nos. 1424,1421, and 1425.
Four step rapid traverse override is effective during rapid traverse.
When the tool has returned to the reference position, the reference position return completion LED goes on. The tool generally moves along only a single axis, but can move along three axes simultaneously when specified so in bit 0 (JAX) of parameter No.1002.

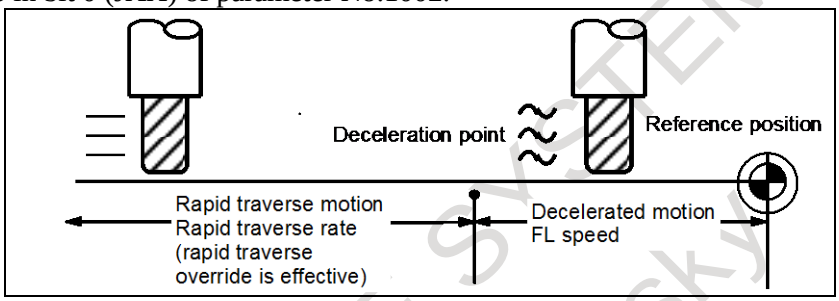


Fig. 3.1 (a) Manual reference position return

Procedure for manual reference position return

Procedure

1 Press the reference position return switch, one of the mode selection switches.
2 To decrease the feedrate, press a rapid traverse override switch.
3 Press the feed axis and direction selection switch corresponding to the axis and direction for reference position return. Continue pressing the switch until the tool returns to the reference position. The tool can be moved along three axes simultaneously when specified so in an appropriate parameter setting. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed set in a parameter. When the tool has returned to the reference position, the reference position return completion LED goes on.
4 Perform the same operations for other axes, if necessary.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

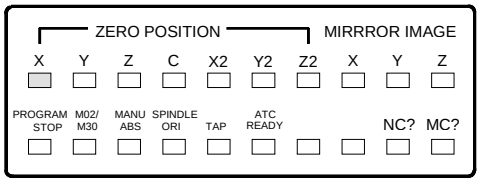


Fig. 3.1 (b)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

- Automatically setting the coordinate system

Bit 0 (ZPR) of parameter No.1201 is used for automatically setting the coordinate system. When ZPR is set, the coordinate system is automatically determined when manual reference position return is performed.

When α, β and γ are set in parameter 1250, the workpiece coordinate system is determined so that reference point on the tool holder or the position of the tip of the reference tool is X= α, Y = β, Z = γ when reference position return is performed. This has the same effect as specifying the following command for reference position return:

G92XαYβZγ;

However, when the workpiece coordinate system is enable (bit 0 (NWZ) of parameter No.8136 is 0), it is not able to use.

Limitation

- Moving the tool again

Once the reference position return completion LED lights at the completion of reference position return, the tool does not move unless the reference position return switch is turned off.

- Reference position return completion LED

The reference position return completion LED is extinguished by either of the following operations:

- Moving from the reference position.
- Entering an emergency stop state.

- The distance to return to reference position

For the distance (Not in the deceleration condition) to return the tool to the reference position, refer to the manual issued by the machine tool builder.

3.2 JOG FEED (JOG)

In the jog mode, pressing a feed axis and direction selection switch on the machine operator's panel continuously moves the tool along the selected axis in the selected direction.

The jog feedrate is specified in a parameter No.1423.

The jog feedrate can be adjusted with the jog feedrate override dial.

Pressing the rapid traverse switch moves the tool at the rapid traverse feedrate No. 1424 regardless of the position of the jog feedrate override dial. This function is called the manual rapid traverse.

Manual operation is allowed for one axis at a time. 3 axes can be selected at a time by bit 0 (JAX) of parameter No.1002.

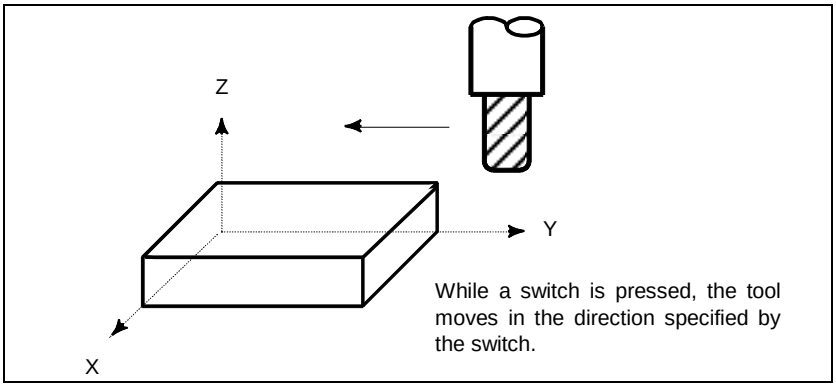


Fig. 3.2 Jog Feed (JOG)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Procedure for JOG feed

Procedure

1 Press the jog switch, one of the mode selection switches.
2 Press the feed axis and direction selection switch corresponding to the axis and the direction where the tool is to be moved. While the switch is pressed, the tool moves at the feedrate specified in the parameter No. 1423. The tool stops when the switch is released.
3 The jog feedrate can be adjusted with the jog feedrate override dial.
4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate while the rapid traverse switch is pressed. Rapid traverse override by the rapid traverse override switches is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

- Manual per revolution feed

The manual per revolution feed is enabled for jog feed by setting bit 4 (JRV) of parameter No.1402 to 1.
During the manual per revolution feed, the tool is jogged at the feedrate that is obtained by multiplying the spindle speed and the manual feedrate override value by the feed amount per revolution specified with the parameter No. 1423.
In the case of linear axis (metric input), during manual per revolution feed, the tool is jogged at the following feedrate:
Feed distance per rotation of the spindle (mm/rev) (specified with parameter No. 1423) × Manual feedrate override × actual spindle speed (rev/min).

Limitation

- Acceleration/deceleration for rapid traverse

Feedrate, time constant and method of automatic acceleration/ deceleration for manual rapid traverse are the same as G00 in programmed command.

- Change of modes

Changing the mode to the jog mode while pressing a feed axis and direction selection switch does not enable jog feed. To enable jog feed, enter the jog mode at first, then press a feed axis and direction selection switch.

- Rapid traverse prior to reference position return

If reference position return is not performed after power-on, pushing rapid traverse button does not actuate the rapid traverse but the remains at the JOG feedrate. This function can be disabled by setting bit 0 (RPD) of parameter No.1401.

- Without the position coder

Manual per revolution feed is not available at the setting of without the position coder (setting bit 0 (NPC) of parameter No.1402 to 1 or bit 1 (FPR) of parameter No.3729 to 1).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

3.3 INCREMENTAL FEED

In the incremental (INC) mode, pressing a feed axis and direction selection switch on the machine operator's panel moves the tool one step along the selected axis in the selected direction. The minimum distance the tool is moved is the least input increment. Each step can be 10, 100, or 1000 times the least input increment.
With using bit 2 (HNT) of parameter No. 7103, each step can be additionally 10 times the least input increment.
The feedrate set in parameter No. 1423 is applied.
By using the manual feedrate override signal, the feedrate can be increased or decreased.
The tool can also be moved at the rapid traverse rate by using the manual rapid traverse selection signal, independent of the manual feedrate override signal.

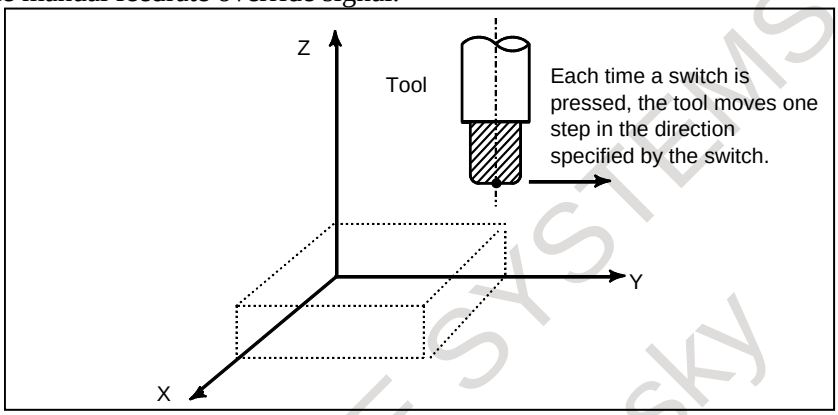


Fig. 3.3 Incremental feed

Procedure for incremental feed

Procedure

1 Press the INC switch, one of the mode selection switches.
2 Select the distance to be moved for each step with the magnification dial.
3 Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. Each time a switch is pressed, the tool moves one step. The feedrate is the same as the jog feedrate.
4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate.
Rapid traverse override by the rapid traverse override switch is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

- Manual per revolution feed

The manual per revolution feed is enabled for incremental feed by setting bit 4 (JRV) of parameter No.1402 to 1.
During the manual per revolution feed, incremental feed is performed at the feedrate that is obtained by multiplying the spindle speed and the manual feedrate override value by the feed amount per revolution specified with the parameter No. 1423.
In the case of linear axis (metric input), during manual per revolution feed, the tool is jogged at the following feedrate:
Feed distance per rotation of the spindle (mm/rev) (specified with parameter No. 1423) × Manual feedrate override × actual spindle speed (rev/min).

FRYER MACHINE SYSTEMS INC
Robert Zahensky

- Travel distance specified with a diameter

T

The distance the tool travels along the X-axis can be specified with a diameter.

3.4 MANUAL HANDLE FEED

In the handle mode, the tool can be minutely moved by rotating the manual pulse generator on the machine operator's panel. Select the axis along which the tool is to be moved with the handle feed axis selection switches.
The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment. One of four types of magnifiers selected with the manual handle feed move distance select signal can be applied. With bit 2 (HNT) of parameter No. 7103, the minimum distance can be further 10 times greater.
Up to three manual pulse generators can be used. (Up to three manual pulse generators can be moved at a time.)
If the option of Manual handle feed 4/5-units is enabled, up to five manual pulse generators can be used.

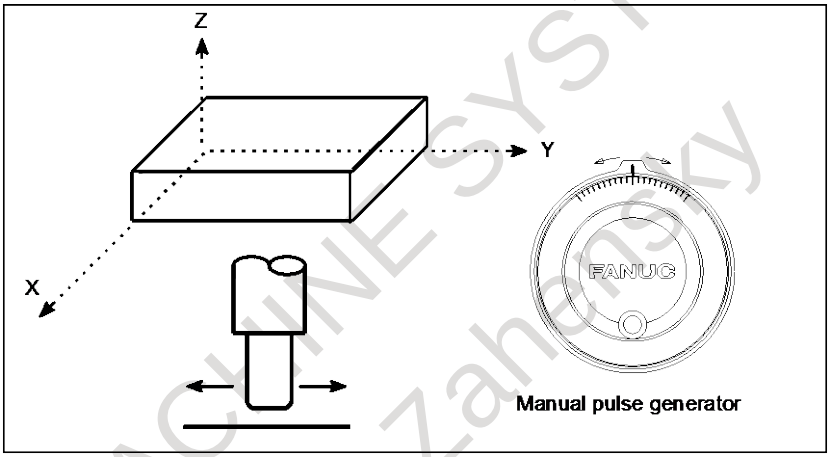


Fig. 3.4 (a) Manual handle feed

Procedure for manual handle feed

Procedure

1 Press the handle switch, one of the mode selection switches.
2 Select the axis along which the tool is to be moved by pressing a handle feed axis selection switch.
3 Select the magnification for the distance the tool is to be moved by pressing a handle feed magnification switch. The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment.
4 Move the tool along the selected axis by rotating the handle. Rotating the handle 360 degrees moves the tool the distance equivalent to 100 graduations.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

- Availability of manual pulse generator in Jog mode (JHD)

Bit 0 (JHD) of parameter No.7100 enables or disables the manual handle feed in the JOG mode.
When bit 0 (JHD) of parameter No.7100 is set 1, both manual handle feed and incremental feed are enabled.

FRYER MACHINE SYSTEMS INC
Rob Zahansky

- Manual handle feed in TEACH IN JOG mode (THD)

By setting bit 1 (THD) of parameter No. 7100, manual handle feed in TEACH IN JOG mode can be enabled or disabled.

- Superimposed feed of the jog feed and the manual handle feed (parameter FJH (No. 7107#1))

When bit 0 (JHD) of parameter No. 7100 is set to 1 or bit 1 (THD) of parameter No. 7100 is set to 1, the operation when the manual handle feed is executed while the jog feed is different according to the setting of the parameter FJH (No. 7107#1).
When bit 1 (FJH) of parameter No. 7107 is set to 0, the jog feed and the manual handle feed are superimposed.
When bit 1 (FJH) of parameter No. 7107 is set to 1, the jog feed takes precedence and the manual handle feed is ignored.

- When manual handle feed exceeding the rapid traverse rate is specified

The amount of pulses exceeding the rapid traverse rate can be saved by CNC as B. And amount of pulses B will be output as pulses C.

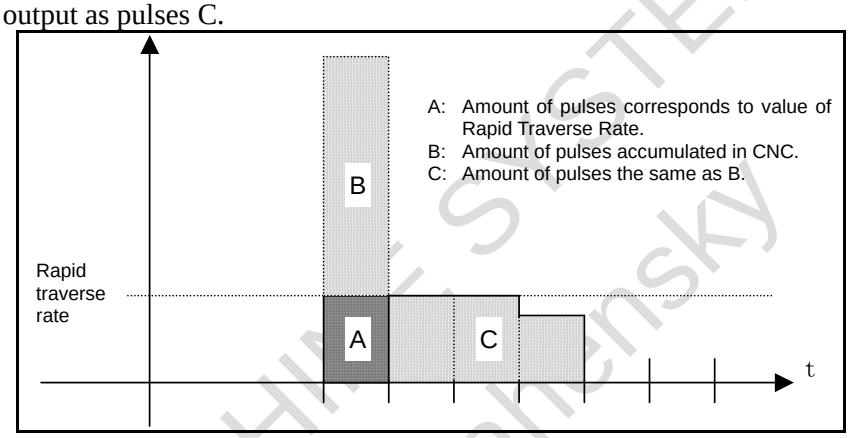


Fig. 3.4 (b) Amount of pulses output by CNC in Manual Handle Feed

Amount of pulses B is calculated in 2 cases as following:

1) In case of parameter No.7117 = 0
The feedrate is clamped at the Rapid Traverse Rate and generated pulses exceeding the Rapid Traverse Rate are ignored (B=0).

2) In case of parameter No.7117 > 0
The feedrate is clamped as the Rapid Traverse Rate, but the pulses exceeding the Rapid Traverse Rate is not ignored. Amount of pulses accumulated in CNC is calculated as following. (Although the rotation of manual pulse generator is stopped, if there is pulses accumulated in CNC, it will be output and the tool will move as long as amount of it.)

Magnification set by manual handle feed selection signals MP1 and MP2 is m, value of parameter No.7117 is n.

n < m: Clamping is set performed at value of parameter No.7117.
n ≥ m: Amount A+B, shown in Fig. 3.4 (c), which's value is multiple of m and small than n. As a result, clamping is performed as an integral multiple of the selected magnification.

FRYER MACHINE SYSTEMS INC
Rob Zalensky

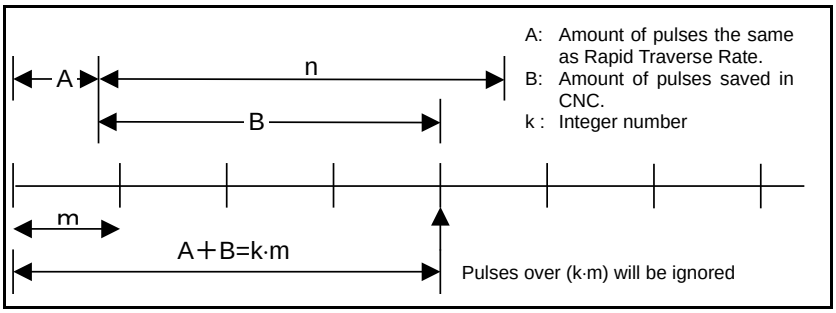


Fig. 3.4 (c) Amount of pulses exceeding the Rapid Traverse Rate (n ≥ m)

> **NOTE**
> Due to change of mode, clamping can be performed not as an integral multiple of the selected magnification.
> The distance the tool moves may not match the graduations on the manual pulse generator.

- Upper feedrate limit in manual handle feed

The upper feedrate limit depends on the input signal (maximum manual handle feedrate switch signal HNDLF) from the PMC as follows:

- When HNDLF is set to “0”, the feedrate is clamped to the manual rapid traverse rate (parameter No. 1424).
- When HNDLF is set to “1”, the feedrate is clamped to the feedrate set in parameter No. 1434.

- Movement direction of an axis to the rotation of MPG (HNGx)

Bit 0 (HNGx) of parameter No 7102 switches the direction of MPG in which the tool moves along an axis, corresponding to the direction in which the handle of the manual pulse generator is rotated.

This parameter is valid only for the following functions:

- Manual handle feed
- Manual handle interruption

Limitation

> ⚠ **WARNING**
> Rotating the handle quickly with a large magnification such as ×100 moves the tool too fast. The feedrate is clamped at the rapid traverse feedrate.

> **NOTE**
> Rotate the manual pulse generator at a rate of five rotations per second or lower. If the manual pulse generator is rotated at a rate higher than five rotations per second, the tool may not stop immediately after the handle is no longer rotated or the distance the tool moves may not match the graduations on the manual pulse generator.

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

3.5 MANUAL ABSOLUTE ON AND OFF

Whether the distance the tool is moved by manual operation is added to the coordinates can be selected by turning the manual absolute switch on or off on the machine operator's panel. When the switch is turned on, the distance the tool is moved by manual operation is added to the coordinates. When the switch is turned off, the distance the tool is moved by manual operation is not added to the coordinates.

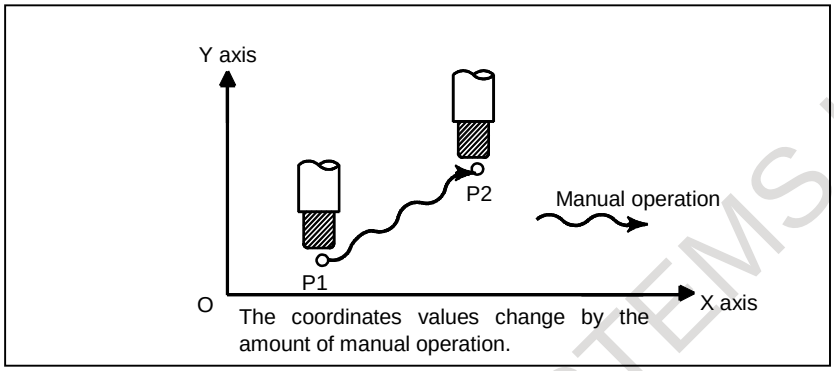


Fig. 3.5 (a) Coordinates with the switch ON

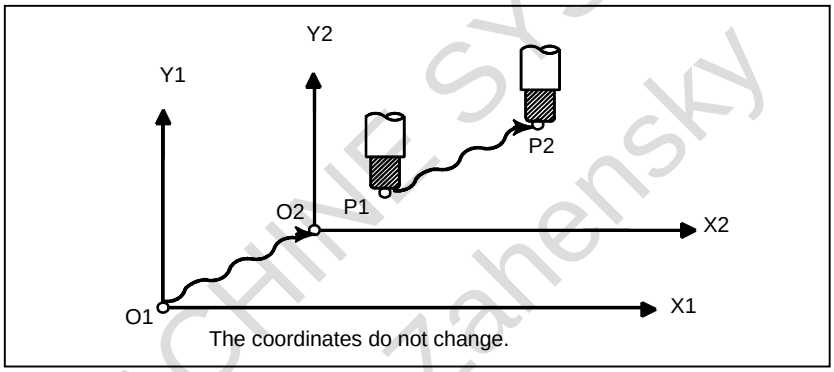


Fig. 3.5 (b) Coordinates with the switch OFF

Explanation

The following describes the relation between manual operation and coordinates when the manual absolute switch is turned on or off, using a program example.

```
G01G90  X100.0Y100.0F010 ;        <1>
        X200.0Y150.0     ;        <2>
        X300.0Y200.0     ;        <3>
```

Fig. 3.5 (c) Program example

The subsequent figures use the following notation:

→ Movement of the tool when the switch is on

- - - → Movement of the tool when the switch is off

The coordinates after manual operation include the distance the tool is moved by the manual operation. When the switch is off, therefore, subtract the distance the tool is moved by the manual operation.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Manual operation after the end of block

Coordinates when block <1> has been executed after manual operation (X-axis +20.0, Y-axis +100.0) at the end of movement of block <2>.

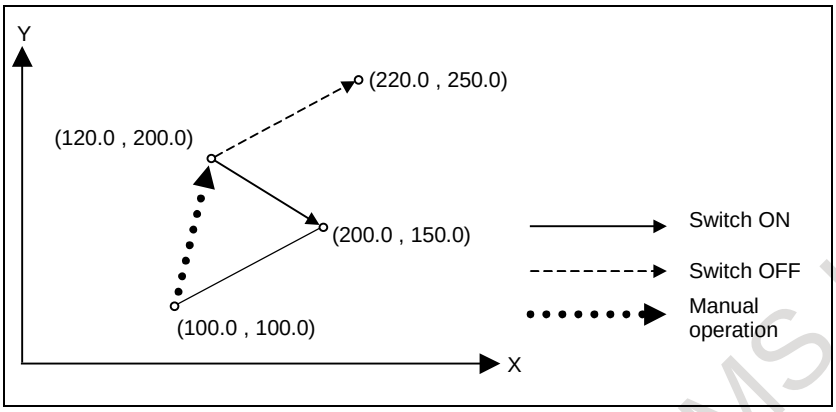


Fig. 3.5 (d) Manual operation after the end of block

- Manual operation after a feed hold

Coordinates when the feed hold button is pressed while block <2> is being executed, manual operation (Y-axis + 75.0) is performed, and the cycle start button is pressed and released.

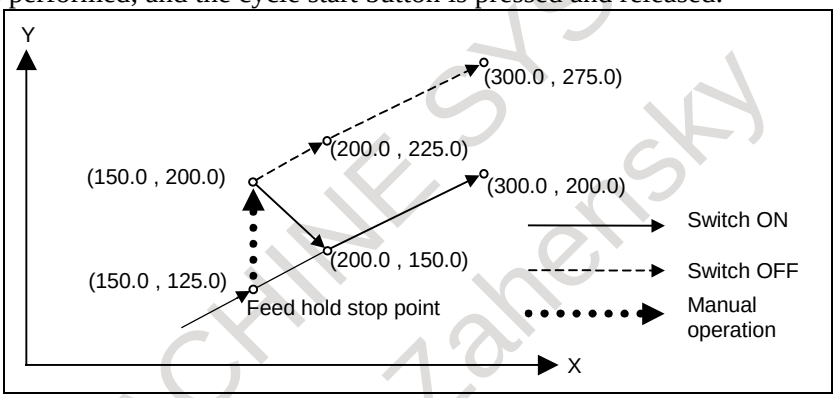


Fig. 3.5 (e) Manual operation after a feed hold

- When reset after a manual operation following a feed hold

Coordinates when the feed hold button is pressed while block <2> is being executed, manual operation (Y-axis +75.0) is performed, the control unit is reset with the RESET button, and block <2> is read again.

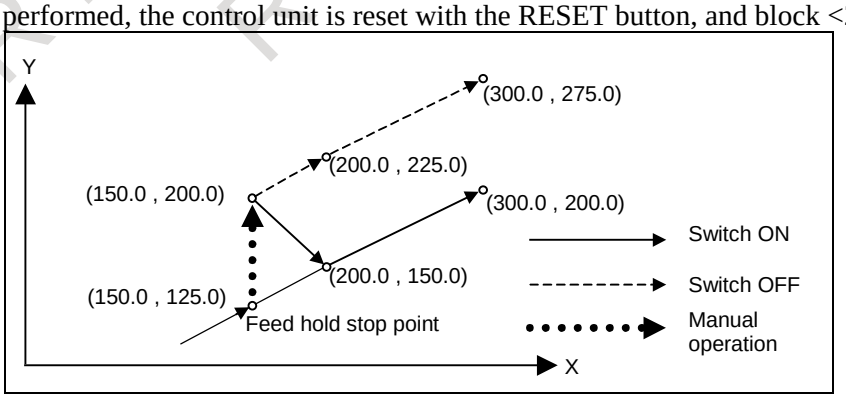


Fig. 3.5 (f) When reset after a manual operation following a feed hold

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- When a movement command in the next block is only one axis

When there is only one axis in the following command, only the commanded axis returns.

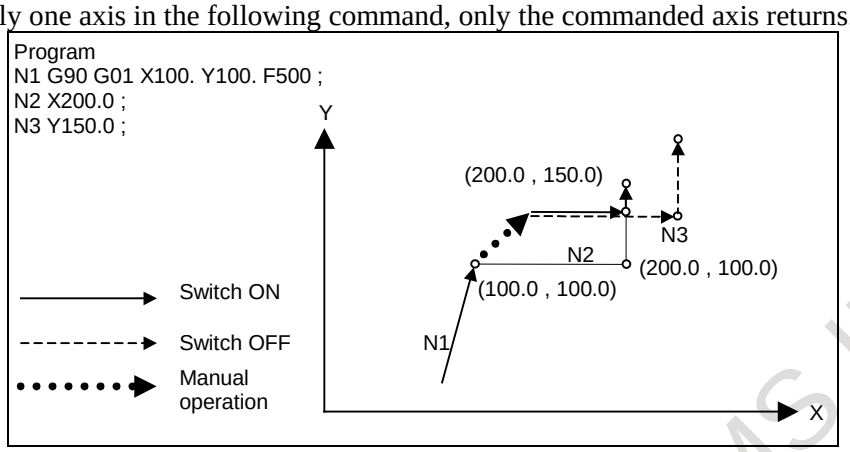


Fig. 3.5 (g) When a movement command in the next block is only one axis

- When the next move block is an incremental

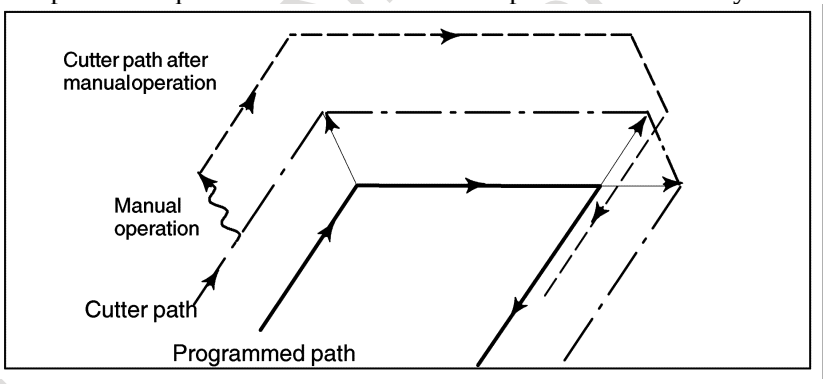
When the following commands are incremental commands, operation is the same as when the switch is OFF.

- Manual operation during cutter or tool nose radius compensation

- When the switch is OFF

  After manual operation is performed with the switch OFF during cutter or tool nose radius compensation, automatic operation is restarted then the tool moves parallel to the movement that would have been performed if manual movement had not been performed.

  The amount of separation equals to the amount that was performed manually.

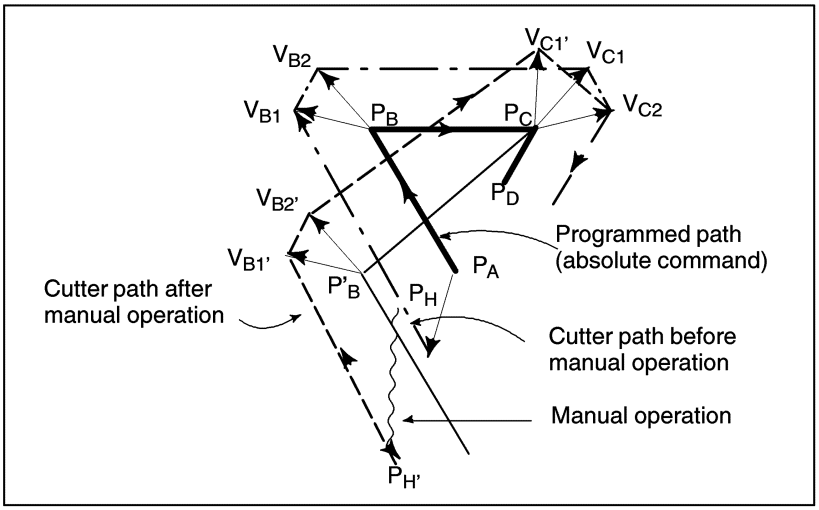


- When the switch is ON during cutter or tool nose radius compensation

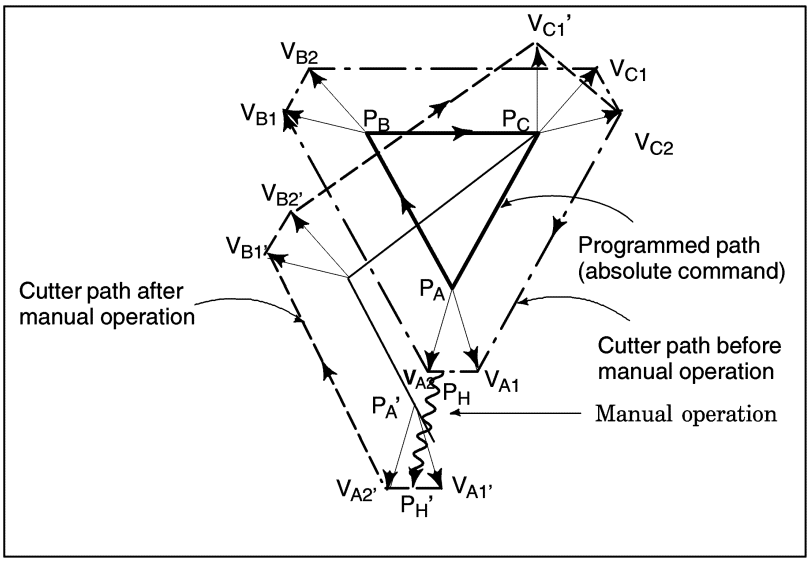
  Operation of the machine upon return to automatic operation after manual intervention with the switch is ON during execution with an absolute command program in the cutter or tool nose radius compensation mode will be described. The vector created from the remaining part of the current block and the beginning of the next block is shifted in parallel. A new vector is created based on the next block, the block following the next block and the amount of manual movement. This also applies when manual operation is performed during cornering.

FRYER SYSTEMS INC

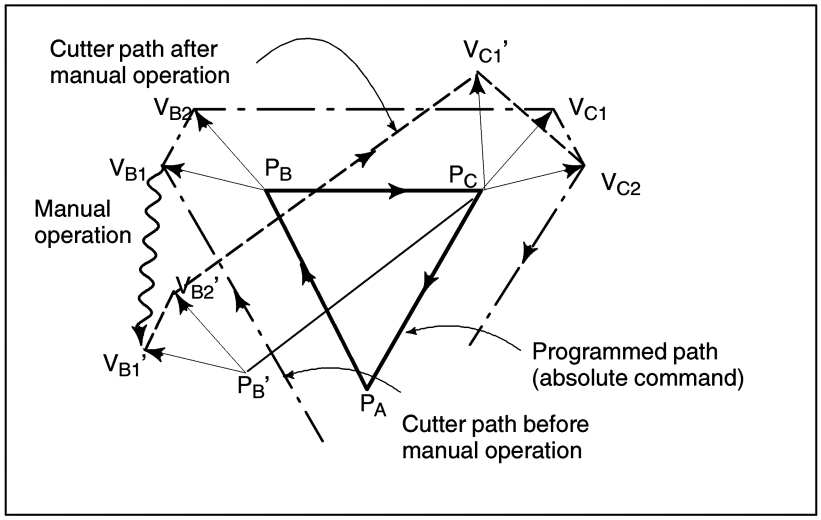
- Manual operation performed in other than cornering
  Assume that the feed hold was applied at point P_H while moving from P_A to P_B of programmed path P_A, P_B, and P_C and that the tool was manually moved to PH'. The block end point P_B moves to the point P_B' by the amount of manual movement, and vectors V_{B1} and V_{B2} at P_B also move to V_{B1}' and V_{B2}'. Vectors V_{C1} and V_{C2} between the next two blocks P_B - P_C and P_C - P_D are discarded and new vectors V_{C1}' and V_{C2}' (V_{C2}' = V_{C2} in this example) are produced from the relation between P_B' - P_C and P_C - P_D. However, since V_{B2}' is not a newly calculated vector, correct offset is not performed at block P_B' - P_C. Offset is correctly performed after P_C.



- Manual operation during cornering
  This is an example when manual operation is performed during cornering. V_{A2}', V_{B1}', and V_{B2}' are vectors moved in parallel with V_{A2}, V_{B1} and V_{B2} by the amount of manual movement. The new vectors are calculated from V_{C1} and V_{C2}. Then correct cutter or tool nose radius compensation is performed for the blocks following P_C.



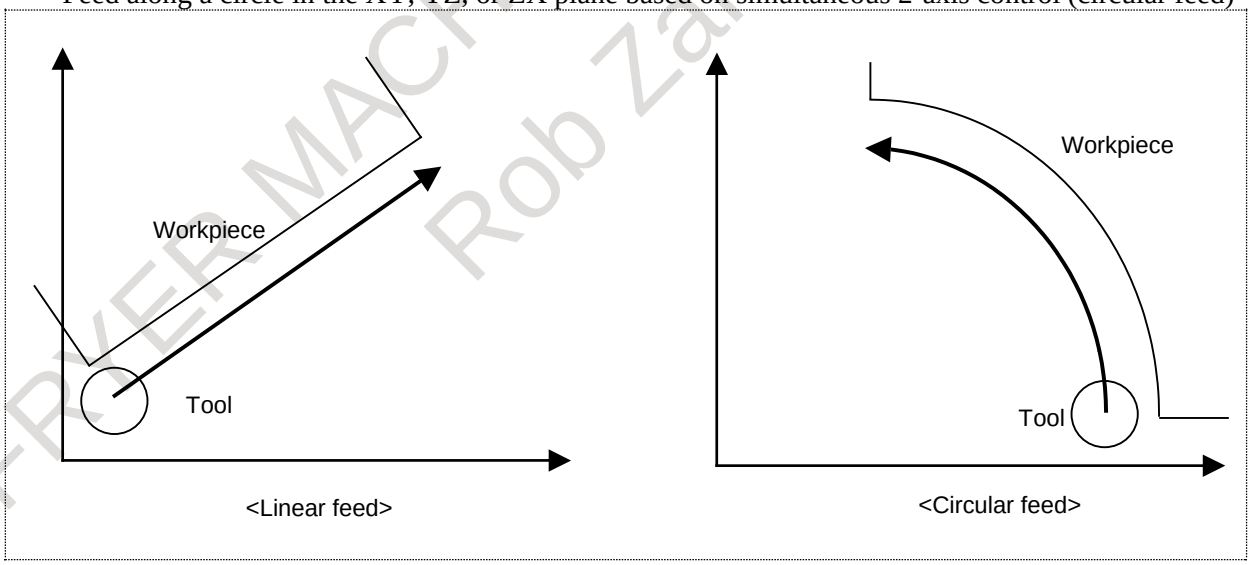
- Manual operation after single block stop
  Manual operation was performed when execution of a block was terminated by single block stop. Vectors V_{B1} and V_{B2} are shifted by the amount of manual operation. Sub-sequent processing is the same as case a described above. An MDI operation can also be intervened as well as manual operation. The movement is the same as that by manual operation.



3.6 MANUAL LINEAR/CIRCULAR INTERPOLATION

In manual handle feed or jog feed, the following types of feed operations are possible along with the conventional feed operation with simultaneous single-axis control (for X, Y, Z, or other axis).

- Feed along a tilted straight line in the XY, YZ, or ZX plane based on simultaneous 2-axis control (linear feed)
- Feed along a circle in the XY, YZ, or ZX plane based on simultaneous 2-axis control (circular feed)



NOTE

1 Two control axes should be included in the three standard axes.
2 In the following, only the case of the XY plane is described. If any other plane is used, replace XY with YZ or ZX. (Refer to the table below.)

XY plane	YZ plane	ZX plane
X	Y	Z
Y	Z	X

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanations

Procedure

1 For manual handle feed, select the manual handle feed mode. For jog feed, select the jog feed mode.

2 For manual handle feed, use the handle feed axis selection switch to select the feed axis (simultaneous 1-axis feed in the X-, Y-, or Z-axis or simultaneous 2-axes feed of the X- and Y-axes along a specified line or circle) used when the manual handle is turned.
For jog feed, use the feed axis direction selection switch to select the feed axis and its direction. The tool moves at the feedrate (jog feedrate) set in parameter No. 1423 in the specified axis direction or along the specified line or circle while the feed axis and its direction are selected.

3 For the manual handle, the tool moves in the specified axis direction when the manual handle is turned. The feedrate depends on the rotation speed of the manual handle. The amount of movement when the manual handle is turned for one pulse can be selected by the manual handle feed movement scale switch.
For jog feed, the manual feedrate override dial can be used to override the feedrate.

The above procedure is an example. For actual operation, refer to the manual provided by the machine tool builder.

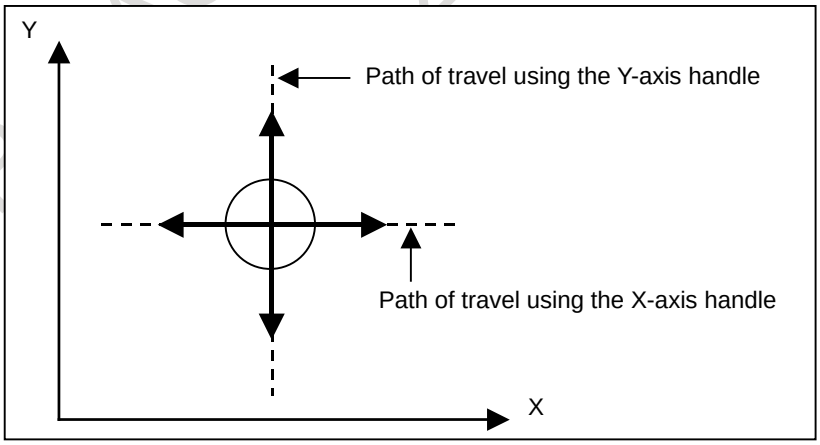
- Line/circle definition

A line or circle definition is not required for axis direction feed. For linear or circular feed, however, a line or circle must be defined. (For example, in the case of circular feed, the center and radius of the circle must be specified.)
Refer to the manual provided by the machine tool builder.

- Manual handle feed

In manual handle feed, the tool can be moved along a specified axis (X-axis, Y-axis, Z-axis, or N^{th}-axis), along a rotated straight line (linear feed), or along a circle (circular feed).

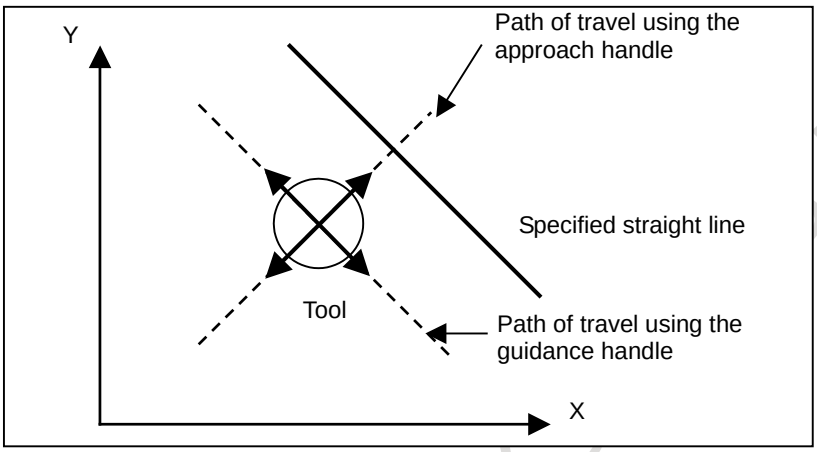
(1) Feed along a specified axis (simultaneous 1-axis control)
A single manual handle operation can move the tool along the desired axis, such as X-axis, Y-axis, or Z-axis, on a simultaneous 1-axis control basis.



Feed along a specified axis

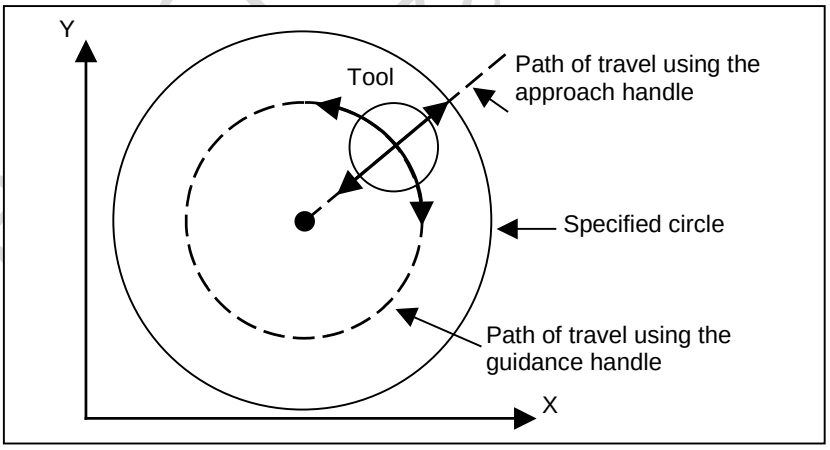
FRYER MACHINE SYSTEMS INC
Zahensky

(2) Linear feed (simultaneous 2-axis control)
A single manual handle operation can move the tool in parallel to the straight line on a simultaneous 2-axis control basis. This manual handle is referred to as the "guidance handle". Also, the tool can be moved at right angles to a specified straight line on a simultaneous 2-axis control basis. This manual handle is referred to as the "approach handle". When the guidance handle or approach handle is turned clockwise or counterclockwise, the tool travels forward or backward along the respective path.



Linear feed

(3) Circular feed (simultaneous 2-axis control)
A single manual handle operation can move the tool from the current position along a concentric circle that has the same center as a specified circle on a simultaneous 2-axis control basis. This manual handle is referred to as the guidance handle. Also, a single manual handle operation can move the tool along a straight line normal to a specified circle on simultaneous 2-axis control. This manual handle is referred to as the approach handle. When the guidance handle or approach handle is turned clockwise or counterclockwise, the tool travels forward or backward along the respective path.



Circular feed

- Feedrate for manual handle feed

The feedrate depends on the speed at which a manual handle is turned. The distance to be traveled by the tool (along a tangent in the case of linear or circular feed) per manual handle pulse can be selected using the manual handle feed travel distance magnification switch.
The approach handle clamp feedrate can be set using parameter No. 7160. The guidance handle clamp feedrate can be set using parameter No. 7161. Set the parameter No. 7117 to 0.

- Manual handle selection

Up to five manual handles to be connected. For information on specifying the usage of the manual handle connected to each interface (for feed in the axis direction, use as a guidance handle, or use as an approach handle), refer to the manual issued by the machine tool builder.

- Direction of movement using manual handles

When the tool is moved along a straight line or circle using the guidance handle or approach handle, the direction of the tool movement (for example, whether to make a clockwise or counterclockwise movement along a circle) when the handle is turned clockwise or counterclockwise can be specified.
Refer to the manual provided by the machine tool builder.

- Jog feed

In jog feed, the tool can be moved along a specified axis (X-axis, Y-axis, Z-axis, etc.), along a rotated straight line (linear feed), or along a circle (circular feed).

(1) Feed along a specified axis (simultaneous 1-axis control)
While a feed axis and its direction are specified with the feed axis direction selection switch, the tool moves in the specified axis direction. When the plane axis is fed, the feederate is dry run rate or the value of parameter No.1415. The feedrate can be overridden using the manual feedrate override dial.

NOTE
1 Two axes of the selected plane cannot be fed at the same time.
2 The manual rapid traverse is invalid.

When the axis excluding the plane axis is fed, the feederate is jog feedrate. The feedrate can be overridden using the manual feedrate override dial.

(2) Linear feed (simultaneous 2-axis control)
By defining a straight line, the tool can be moved as follows:
- While a feed axis and its direction are selected using the feed axis direction selection switch, the tool moves along a straight line parallel to the specified straight line on a simultaneous 2-axis control basis.
- While a feed axis and its direction are selected using the feed axis direction selection switch, the tool moves along a straight line perpendicular to the specified straight line on a simultaneous 2-axis control basis.

The feedrate in the tangential direction is dry run rate or the value of parameter No.1415. The feedrate can be overridden using the manual feedrate override dial. Manual rapid traverse is invalid.

(3) Circular feed (simultaneous 2-axis control)
By defining a circle, the tool can be moved as follows:
- While a feed axis and its direction are selected using the feed axis direction selection switch, the tool moves from the current position along the concentric circle that has the same center as the specified circle.
- While a feed axis and its direction are selected using the feed axis direction selection switch, the tool moves along a straight line normal to the specified circle.

The feedrate in the tangential direction is dry run rate or the value of parameter No.1415. The feedrate can be overridden using the manual feedrate override dial. Manual rapid traverse is invalid.

- Manual handle feed in JOG mode

Even in JOG mode, manual handle feed can be enabled using bit 0 (JHD) of parameter No. 7100. In this case, however, manual handle feed is enabled only when the tool is not moved using jog feed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Limitations

- Mirror image

The mirror image function is not available during the manual operation. (The manual operation can be executed when the mirror image switch is off and the mirror image setting is off.)

> **NOTE**
> If the tool is operated with the mirror image function activated, the normal operation will not be guaranteed.

- Rotation direction

When this function is effective, bit 0 (HNGx) of parameter No. 7102 is invalid which is used to change the axis movement direction with respect to the rotation direction of the manual pulse generator. (The rotation direction of the manual pulse generator is always the same as the axis movement direction; that is, the operation is the same as when bit 0 (HNGx) of parameter No. 7102 is set to "0".)

- The amount of the shift for manual handle feed

When this function is effective, parameters Nos. 12350 and 12351 used to determine the magnification of manual handle feed for each axis are invalid and the values of parameters Nos. 7113 and 7114 are used. (Parameters No. 7131 and No. 7132 are used for the second manual pulse generator, parameters Nos. 7133 and 7134 for the third, parameters Nos. 7135 and 7136 for the fourth, and parameters Nos. 7137 and 7138 for fifth.)

- Incremental feed

This function is invalid for the incremental feed.

- Synchronous/composite control

If the axis is set as the synchronous/composite control axis, this function is invalid.

3.7 RIGID TAPPING BY MANUAL HANDLE

For execution of rigid tapping, set rigid mode, then switch to handle mode and move the tapping axis with a manual handle. See the section of the rigid tapping to operate the rigid tapping by manual handle. For rigid tapping, refer to Section, "RIGID TAPPING" in the OPERATOR'S MANUAL (T series) or Section, "RIGID TAPPING" in the OPERATOR'S MANUAL (M series) and the relevant manual of the machine tool builder.

Rigid tapping by manual handle

Procedure

1 Stop the spindle and servo axes, then set MDI mode by pressing the MDI switch among the mode selection switches.

2 Enter and execute the following program:

Example 1) M series / T series (G code systems B, C)

```
M29 S100 ;
G91 G84 Z0 F1000 ;
```

Example 2) T series (G code system A)

```
M29 S100 ;
G84 W0 F1000 ;
```

The program above is required to determine a screw lead and set rigid tapping mode. In this program, a tapping axis must always be specified. Specify a value that does not operate the tapping axis. With the M series, specify G84 (G74). With the T series, specify G84 (G88).

FRYER MACHINE SYSTEMS INC
Rob Zarensky

> ⚠ **CAUTION**
> In this MDI programming, never specify commands to position the tool at a drilling position and at point R. Otherwise, the tool moves along an axis.

3 When the entered program is executed, rigid tapping mode is set.
4 After rigid mode is set upon completion of MDI program execution, switch to the handle mode by pressing the handle switch among the mode selection switches.

> ⚠ **CAUTION**
> At this time, never press the reset key. Otherwise, rigid mode is canceled.

5 To perform rigid tapping, select a tapping axis with the handle feed axis select switch, and move the tapping axis with the manual handle.

Explanation

- Manual rigid tapping

Manual rigid tapping is enabled by bit 0 (HRG) of parameter No. 5203 to 1.

- Cancellation of rigid mode

To cancel rigid mode, specify G80 as same the normal rigid tapping. When the reset key is pressed, rigid mode is canceled, but the canned cycle is not canceled.
When the rigid mode switch is to be set to off for rigid mode cancellation (when bit 2 (CRG) of parameter No. 5200 is set to 0), the G80 command ends after the rigid mode switch is set to off.

- Spindle rotation direction

The rotation direction of the spindle is determined by a specified tapping cycle G code and the setting of bit 1 (HRM) of parameter No. 5203. For example, when the HRM parameter is set to 0 in G84 mode, the spindle makes forward rotations as the tapping axis moves in the minus direction. (When the tapping axis moves in the plus direction, the spindle makes reverse rotations.)

- Arbitrary tapping axis

With the M series, an arbitrary tapping axis can be selected by setting bit 0 (FXY) of parameter No. 5101 to 1. With the T series, an arbitrary tapping axis can be selected not by using bit 0 (FXY) of parameter No. 5101 (but by using bit 0 (FXY) of parameter No. 5101 when the program format for FANUC Series 15 is specified). In this case, specify a G code for plane selection and the address of the tapping axis when setting the rigid tapping mode in the MDI mode.

- G84 command/same block specifying both M29 and G84

As the MDI program command for setting the rigid mode, G84 can be specified as a rigid tapping G code (by setting bit 0 (G84) of parameter No. 5200 to 1), or M29 and G84 can be specified in the same block.
Example 1: G91 G84 Z0 F1000 S100 ;
Example 2: G91 G84 Z0 F1000 M29 S100 ;

- Specification of manual handle feed faster than the rapid traverse rate

Set parameter No.7117 to 0 so that when manual handle feed is specified which is faster than the rapid traverse rate, the handle pulses beyond the rapid traverse rate are ignored.

- Program format for FANUC Series 15

By setting bit 1 (FCV) of parameter No. 0001 to 1, the program format for FANUC Series 15 can be used for specification.
Example 1: G91 G84.2 (G84.3) Z0 F1000 S100; (M series)
Example 2: G84.2 W0 F1000 S100; (T series : G code systems A)
Example 3: G91 G84.2 Z0 F1000 S100; (T series : G code system B, C)

FRYER MACHINE SYSTEMS INC Rob Zahensky

- Acceleration/deceleration type

When manual rigid tapping is executed, the acceleration/deceleration type and acceleration/deceleration time constant set in the rigid tapping parameters are valid.
The same settings are valid also for extraction.

- In the case of multi-spindle operation

In the case of multi-spindle operation, a spindle can be selected by specifying a P command as well as an S command.
Example : When the 2nd spindle is selected:

M29 S100 **P2**;
G91 G84 Z0 F1000 ;

Limitation

- Excessive error check

In manual rigid tapping, only an excessive error during movement is checked.

- Tool axis direction handle feed

Tool axis direction handle feed is disabled.

- Extraction override

In manual rigid tapping, the extraction override function is disabled, and the use of an acceleration/deceleration time constant for extraction is disabled.

- Number of repeats

In MDI programming, never specify K0 and L0, which are used to specify that the number of repeats is 0 and to disable the execution of a G84 block. If K0 or L0 is specified, rigid mode cannot be set.

- Positioning of the tool to a drilling position

When positioning the tool to a drilling position, select the X-axis or Y-axis with the axis select switch in handle mode. Never use the method of positioning to a drilling position in MDI mode or MEM mode. The method can operate the tapping axis.

- 3-dimensional rigid tapping

3-dimensional rigid tapping cannot be used in rigid tapping using the manual handle.

- Feed forward

In rigid tapping by manual handle, feed forward is disabled even if bit 2 (RFF) of parameter No. 5203 is set to 1 (Feed forward is enabled in rigid tapping).

- Constant surface speed control

If rigid tapping is commanded during constant surface speed control, alarm (PS0200), ”ILLEGAL S CODE COMMAND” is issued. Command rigid tapping after canceling constant surface speed control.

- Spindle Control with Servo Motor

Rigid tapping by manual handle cannot be used against the rigid tapping with servo motor.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

3.8 3-DIMENSIONAL MANUAL FEED

M

This function enables the use of the following functions.

- 3-dimensional manual feed
  - Tool axis direction handle feed/tool axis direction JOG feed/tool axis direction incremental feed
  - Tool axis right-angle direction handle feed/tool axis right-angle direction JOG feed/tool axis right-angle direction incremental feed
  - Tool tip center rotation handle feed/tool tip center rotation JOG feed/tool tip center rotation incremental feed
  - Table vertical direction handle feed/table vertical direction JOG feed/table vertical direction incremental feed
  - Table horizontal direction handle feed/table horizontal direction JOG feed/table horizontal direction incremental feed

A handle interrupt can be generated for each handle feed. Handle interrupts work according to the corresponding handle feed specifications described hereinafter unless otherwise noted.

- Screen display functions
  - Display of the coordinate of the tool tip
  - Display of pulse values
  - Display of the amount of machine axes movement

- Selecting a coordinate system when calculating the tool direction

If, in 3-dimensional manual feed, a workpiece coordinate system offset is set for a rotation axis, use bit 0 (CAC) of parameter No. 12319 to select whether to use values in the workpiece or machine coordinate system when calculating the tool direction.

- When the bit 0 (CAC) of parameter No. 12319 = 0:
  Values in the machine coordinate system are used.
  For those parameters for configuring the machine, Nos. 19680 to 19714, that depend on the coordinate on a rotation axis, set the values assumed when the machine coordinate on the rotation axis is 0.

- When the bit 0 (CAC) of parameter No. 12319 = 1:
  Values in the workpiece coordinate system are used.
  For those parameters for configuring the machine, Nos. 19680 to 19714, that depend on the coordinate on a rotation axis, set the values assumed when the workpiece coordinate on the rotation axis is 0.

For the functions below, values in the workpiece coordinate system of a rotation axis are used to calculate the tool direction and, therefore, if a workpiece coordinate system offset is set for a rotation axis, and 3-dimensional manual feed is to be used together with any of the functions below, set bit 0 (CAC) of parameter No. 12319 to 1.

- Tilted working plane indexing

(Example)

Related parameters:
No.19680=2 (Tool rotation type)
No.19682=3 (the master rotation axis (C-axis) is about the Z-axis)
No.19687=2 (the slave rotation axis (B-axis) is about the Y-axis)

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

No.19697=3 (the reference tool axis direction is the Z-axis direction)
No.19698=0 (angle RA when the reference tool axis direction is tilted)
No.19699=0 (angle RB when the reference tool axis direction is tilted)

Workpiece coordinate system offset:
B=10.0

Example 1:

The tool is assumed to face the Z-axis direction when it is in the following states.
Position in the workpiece coordinate system B= -10.0
Position in the machine coordinate system B=0.0
In this case, values in the machine coordinate system must be used to calculate the tool direction, set bit 0 (CAC) of parameter No. 12319 to 0.

Example 2:

The tool is assumed to face the Z-axis direction when it is in the following states.
Position in the workpiece coordinate system B=0.0
Position in the machine coordinate system B=10.0
In this case, values in the workpiece coordinate system must be used to calculate the tool direction, set bit 0 (CAC) of parameter No. 12319 to 1.

- 3-Dimensional manual feed by handle in JOG mode

Even in JOG feed mode, 3-Dimensional manual feed by handle feed can be enabled using bit 0 (JHD) of parameter No.7100.

- The manual handle interruption screen in Tilted working plane indexing mode

In Tilted working plane indexing mode, when 3-dimensional handle interrupt is executed in X,Y, and Z directions of the feature coordinate system (bit 0 (TWD) of parameter No.12320 is set to 1), it can be selected whether to display the travel distance in the manual handle interruption screen in the workpiece coordinate system or the feature coordinate system, by bit 3 (DMK) of parameter No.5402 and by bit 4 (MDT) of parameter No.12319.
In the case to display the travel distance in the feature coordinate system, set bit 3 (DMK) of parameter No.5402 to 0 and bit 4 (MDT) of parameter No.12319 to 1.

NOTE

1 A 3-dimensional handle interrupt must not be generated when a rotation axis command is being executed during automatic operation.
2 3-dimensional manual feed is disabled when the manual reference position return mode is selected.
3 If per-axis interlock is enabled to at least one of 3-dimensional manual feed axes, movement with manual feed is not performed.

3.8.1 Tool Axis Direction Handle Feed / Tool Axis Direction JOG Feed / Tool Axis Direction Incremental Feed

Overview

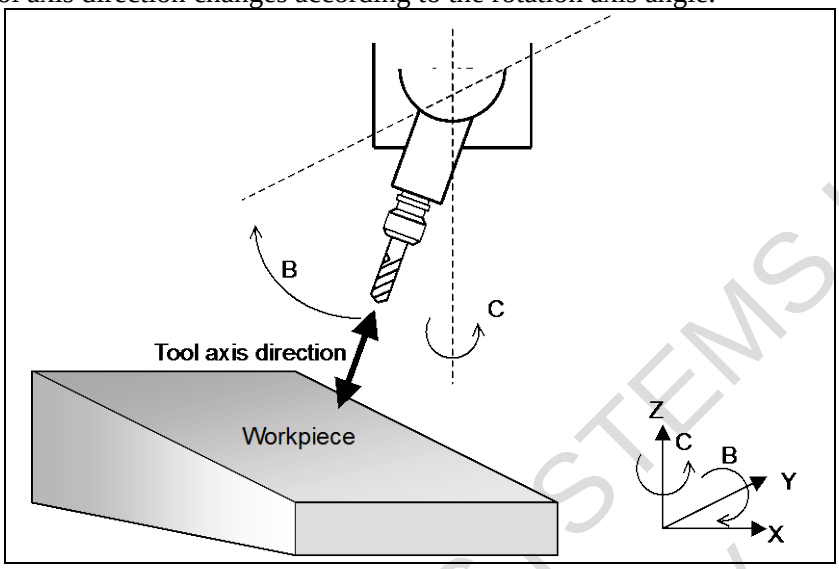
In the tool axis direction handle feed, tool axis direction JOG feed, and tool axis direction incremental feed, the tool or table is moved in the tool axis direction.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Explanation

- Tool axis direction

The tool axis direction that is taken when all the rotation axes for controlling the tool are at an angle of 0 degree is specified in parameters Nos.19697, 19698, and 19699. As the rotation axes for controlling the tool rotate, the tool axis direction changes according to the rotation axis angle.



- Tool axis direction feed in the tilted working plane indexing mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the tool axis direction feed in the tilted working plane indexing mode is assumed to be the Z direction in the feature coordinate system of the tilted working plane indexing.

- Tool axis direction handle feed

The tool axis direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.
<2> The tool axis direction feed mode signal (ALNGH) is set to "1" and the table base signal (TB_BASE) is set to "0".
<3> The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the tool axis direction handle feed mode effective is set in parameter No.12310.
<4> The value of parameter No.12310 matches the first manual handle feed axis selection signals (HS1A - HS1E).

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the tool axis direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.
If Manual handle feed maximum feedrate change signal HNDLF is "1", clamping is performed by the maximum handle feedrate (parameter No. 1434).

- Tool axis direction JOG feed/tool axis direction incremental feed

The tool axis direction JOG feed or tool axis direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.
<2> The tool axis direction feed mode signal (ALNGH) is set to "1" and the table base signal (TB_BASE) is set to "0".

FRYER MACHINE SYSTEMS INC
Rob Zahensky

<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the axis corresponding to the direction specified by parameter No.19697. (Even when the tool axis direction is slant because of the settings of parameters Nos.19698 and 19699, the signal that activates the tool axis direction JOG feed or tool axis direction incremental feed is determined by parameter No.19697 only.)
Ex.) Parameter No.19697 = 3 (+Z-axis direction); Z-axis is the 3rd axis.
- +J3 : Tool axis direction +
- -J3 : Tool axis direction -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate of a rotation axis is the jog feedrate of the axis to be rotated (parameter No. 1423). The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

3.8.2 Tool Axis Right-Angle Direction Handle Feed / Tool Axis Right-Angle Direction JOG Feed / Tool Axis Right-Angle Direction Incremental Feed

Overview

In the tool axis right-angle direction handle feed, tool axis direction JOG feed, or tool axis direction incremental feed, the tool or table is moved in the tool axis right-angle direction.
If bit 1 (FLL) of parameter No. 12320 is set to 1, the tool or table is moved in the latitude or longitude direction determined by the tool axis direction vector.

Explanation

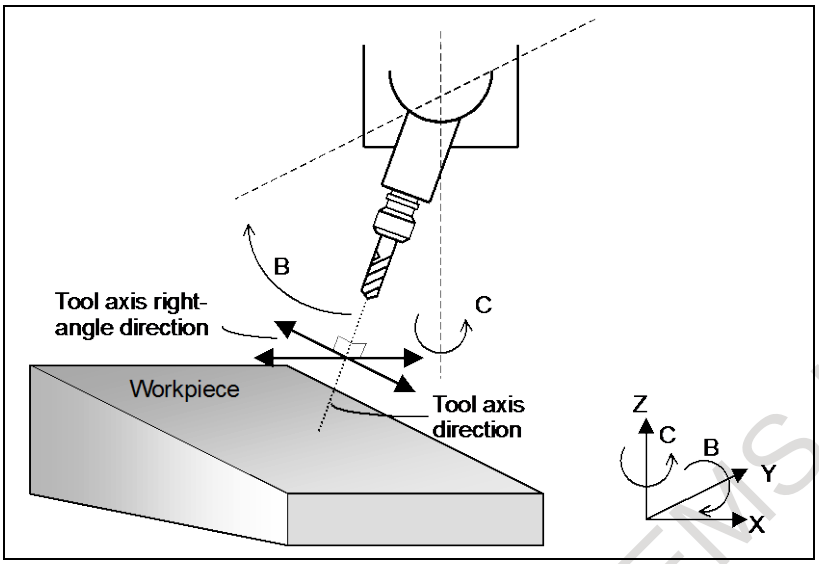
- Tool axis right-angle direction

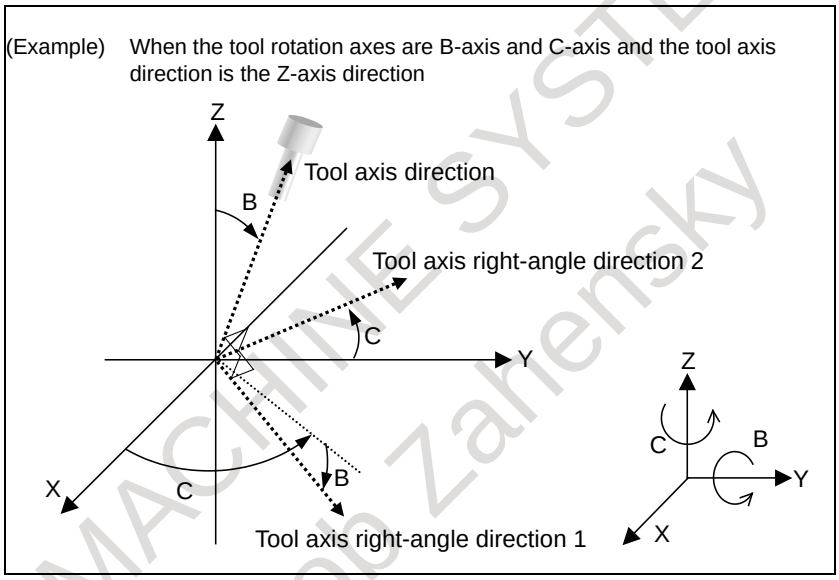
There are two tool axis right-angle directions, which are perpendicular to the tool axis direction (see the previous section).

Parameter No.19697	Tool axis right-angle direction 1	Tool axis right-angle direction 2
1 (The reference tool direction is +X.)	+Y direction	+Z direction
2 (The reference tool direction is +Y.)	+Z direction	+X direction
3 (The reference tool direction is +Z.)	+X direction	+Y direction

This table shows the tool axis right-angle directions that may be taken when the angles of all the rotation axes for controlling the tool are 0 degree and when parameters Nos.19698 and 19699 are both set to 0.
When the reference tool axis direction is inclined based on the settings of parameters Nos.19698 and 19699, the tool axis right-angle direction is also inclined as much.
As the rotation axes for controlling the tool rotate, the tool axis right-angle direction changes according to the rotation axis angle.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky





- Latitude and longitude directions

When bit 1 (FLL) of parameter No. 12320 is set to 1, the feed direction is defined as follows:

Let a vector perpendicular to a plane formed by the tool axis direction vector ($\vec{T}$) and normal axis direction vector ($\vec{P}$) (parameter No. 12321) be the tool axis right-angle direction 1 (longitude direction) vector ($\vec{R1}$). When tool axis right-angle direction 1 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Longitude direction feed)

Equation: $\vec{R1} = \vec{P} \times \vec{T}$

Let a vector perpendicular to the tool axis direction vector ($\vec{T}$) and tool axis right-angle direction 1 (longitude direction) vector ($\vec{R1}$) be the tool axis right-angle direction 2 (latitude direction) vector ($\vec{R2}$). When tool axis right-angle direction 2 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Latitude direction)

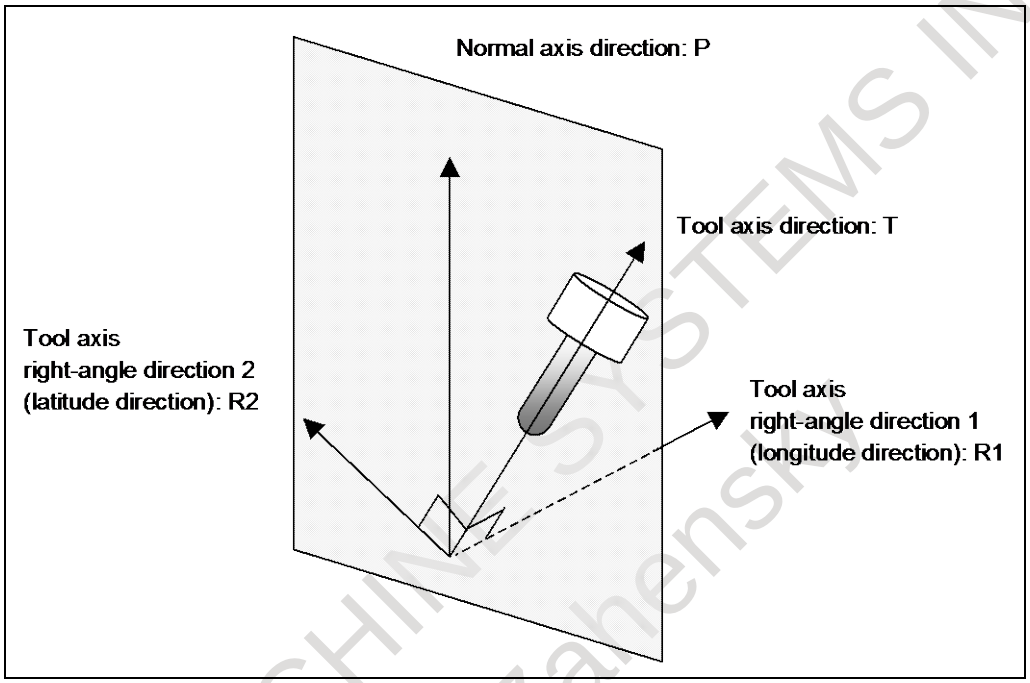
Equation: $\vec{R2} = \vec{T} \times \vec{R1}$

When the tool axis direction vector ($\vec{T}$) is parallel to the normal axis direction vector ($\vec{P}$) (parameter No. 12321) (when the angle between them is not greater than the setting of parameter No. 12322), tool axis right-angle direction 1 and tool axis right-angle direction 2 are assumed as follows:

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Parameter No.12321	Normal axis direction	Tool axis right-angle direction 1	Tool axis right-angle direction 2
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

If 0 is set in parameter No. 12321, the normal axis direction is set to the reference tool axis direction (parameter No. 19697).
If a value other than 0 to 3 is specified in parameter No. 12321, alarm PS5459, “MACHINE PARAMETER INCORRECT” is issued.



- Tool axis right-angle direction feed in the tilted working plane indexing mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the tool axis right-angle direction feed in the tilted working plane indexing mode is defined as follows:
Tool axis right-angle direction 1:
X direction in the feature coordinate system of the tilted working plane indexing
Tool axis right-angle direction 2:
Y direction in the feature coordinate system of the tilted working plane indexing

- Tool axis right-angle direction handle feed

The tool axis right-angle direction handle feed is enabled when the following four conditions are satisfied:
<1> Handle mode is selected.
<2> The tool axis right-angle direction feed mode signal (RGHTH) is set to "1" and the table base signal (TB_BASE) is set to "0".
<3> The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the tool axis right-angle direction handle feed mode effective is set in parameter No.12311 or No.12312.
<4> The value of parameter No.12311 or No.12312 matches the first manual handle feed axis selection signals (HS1A - HS1E).

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the tool axis right-angle direction by the amount of rotation.

FRYER MACHINE SYSTEMS INC
Rob Zabensky

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.
If Manual handle feed maximum feedrate change signal HNDLF is "1", clamping is performed by the maximum handle feedrate (parameter No. 1434).

- Tool axis right-angle direction JOG feed/tool axis right-angle direction incremental feed

The tool axis right-angle direction JOG feed or tool axis right-angle direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.

<2> The tool axis right-angle direction feed mode signal (RGHTH) is set to "1" and the table base signal (TB_BASE) is set to "0".

<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the axis corresponding to the direction that is perpendicular to the direction specified by parameter No.19697. (Even when the tool axis direction is slant because of the settings of parameters Nos.19698 and 19699, the signal that activates the tool axis right-angle direction JOG feed or tool axis right-angle direction incremental feed is determined by parameter No.19697 only.)

Ex.) Parameter No.19697=3 (+Z-axis direction); X-, Y-, and Z-axes are the 1st, 2nd, and 3rd axes respectively.

- +J1 : Tool axis right-angle direction 1 +
- -J1 : Tool axis right-angle direction 1 -
- +J2 : Tool axis right-angle direction 2 +
- -J2 : Tool axis right-angle direction 2 -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) for a driven feed axis direction selection signal. The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

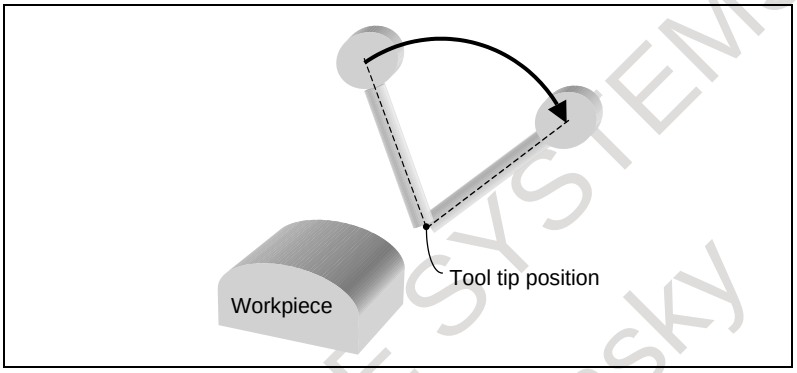
FRYER MACHINE SYSTEMS INC
Rob Zaremsky

3.8.3 Tool Tip Center Rotation Handle Feed / Tool Tip Center Rotation JOG Feed / Tool Tip Center Rotation Incremental Feed

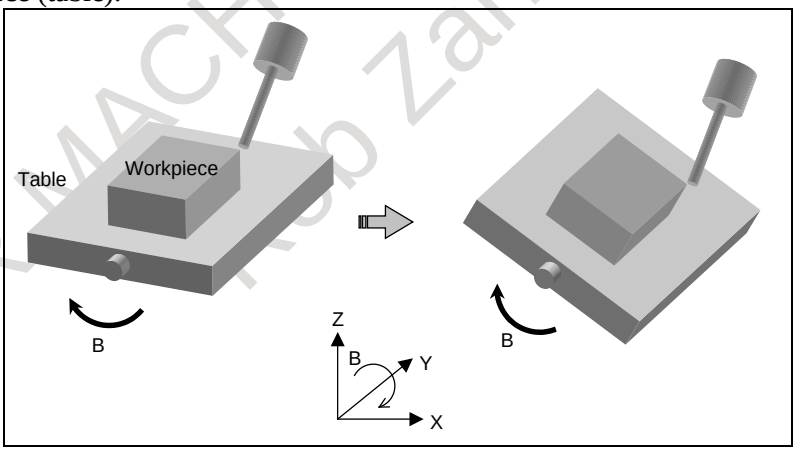
Overview

In the tool tip center rotation handle feed, tool tip center rotation JOG feed, and tool tip center rotation incremental feed, when a rotary axis is rotated by manual feed, the linear axes (X, Y, and Z axes) are moved so that turning the rotation axis does not change the relative relationship between the tool tip position and the workpiece (table).

- The following figure shows an example where the tool is rotated on the rotation axis. In this case, the linear axes are moved so that the position of the tool tip is not moved with respect to the workpiece.



- The following figure shows an example where the table is rotated on the rotation axis. As in the previous case, the linear axes are moved so that the position of the tool tip is not moved with respect to the workpiece (table).



- Tool tip center rotation handle feed

The tool tip center rotation handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.
<2> The tool tip center rotation feed mode signal (RNDH) is set to "1".
<3> The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the tool tip center rotation handle feed mode effective is set in parameter No.12313 or No.12314.
<4> The value of parameter No.12313 or No.12314 matches the first manual handle feed axis selection signals (HS1A - HS1E).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Amount of movement

When the manual pulse generator is rotated, the rotation axis is moved by the amount of rotation. The linear axes (X, Y, and Z axes) are moved so that turning the rotation axis does not change the relative relationship between the tool tip position and the workpiece.

Feedrate clamp

The feedrate is clamped so that the synthetic speed of the linear axes (in the tangential direction) does not exceed the manual rapid traverse rate (parameter No.1424) (of any moving linear axis). The feedrate is also clamped so that the speed of the rotation axis does not exceed the manual rapid traverse rate (parameter No.1424) (of that particular axis). Handle pulses generated while the clamp feedrate is exceeded are ignored.
If Manual handle feed maximum feedrate change signal HNDLF is "1", clamping is performed by the maximum handle feedrate (parameter No. 1434).

- Tool tip center rotation JOG feed/tool tip center rotation incremental feed

The tool tip center rotation JOG feed or tool tip center rotation incremental feed is enabled when the following three conditions are satisfied:
<1> JOG mode or incremental feed mode is selected.
<2> The tool tip center rotation feed mode signal (RNDH) is set to "1".
<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the rotation axis to be rotated.
Ex.) When the B-axis (4th axis) is rotated
- +J4 : Tool tip center rotation feed +
- -J4 : Tool tip center rotation feed -

Feedrate

Control is exerted so that the synthetic speed of the linear axes (in the tangential direction) is the dry run rate (parameter No.1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate of a rotation axis is the jog feedrate of the axis to be rotated (parameter No. 1423). The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the synthetic speed of the linear axes (in the tangential direction) does not exceed the manual rapid traverse rate (parameter No.1424) (of any moving linear axis). The feedrate is also clamped so that the speed of the rotation axis does not exceed the manual rapid traverse rate (parameter No.1424) (of that particular axis).

- Selection of the tool length offset value

The tool length in 3-dimensional manual feed is determined as explained below. (Table 3.8.3)
If bit 2 (LOD) of parameter No. 19746 is set to 0, the value set in parameter No. 12318 is assumed to be the tool length.
If the LOD parameter is set to 1, and the tool length offset function is performed, the offset data specified for the tool length offset is assumed to be the tool length.
If the LOD parameter is set to 1, and the tool length offset function is not performed, the tool length is determined as follows. If bit 3 (LOZ) of parameter No. 19746 is set to 0, the value set in parameter No. 12318 is assumed to be the tool length in 3-dimensional manual feed; if LOZ is set to 1, the tool length is assumed to be 0.

FLYER MACHINE SYSTEMS INC
Rob Zaremsky

Table 3.8.3 Tool length offset value in 3-dimensional manual feed

		Bit 2 (LOD) of parameter No. 19746		
		= 0	= 1	
			Tool length offset enabled	Tool length offset canceled
Bit 3 (LOZ) of parameter No. 19746	= 0	Parameter No. 12318	Offset data	Parameter No. 12318
	= 1			0

The tool length offset function is enabled when the following two conditions are both satisfied:
- The tool length offset function listed below is enabled (modal code of group 8 except G49)
  - G43/G44 : Tool length compensation
- The H/D code is other than 0.

If bit 6 (CLR) of parameter No. 3402 is set to 0 not to clear the tool length offset vector, G codes of group 8, and H codes at the time of a reset, the tool length offset status is maintained when a reset is made in the tool length offset mode.

3.8.4 Table Vertical Direction Handle Feed / Table Vertical Direction JOG Feed / Table Vertical Direction Incremental Feed

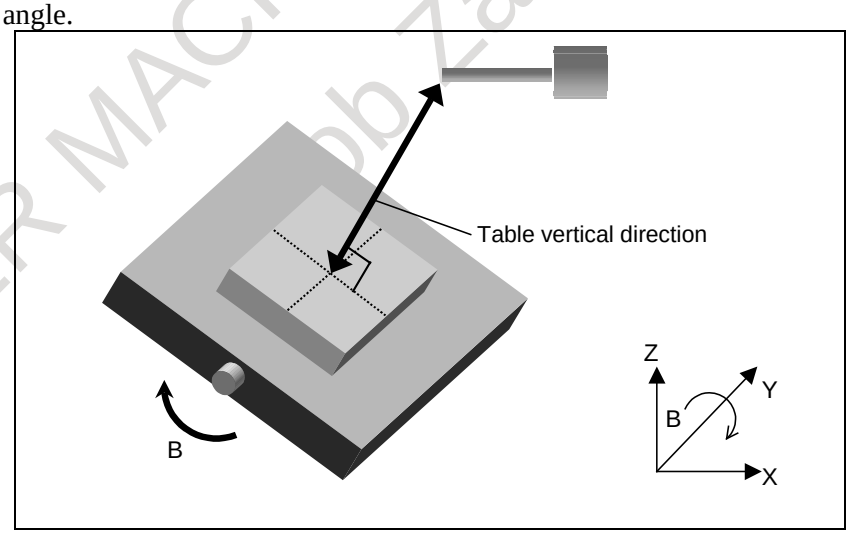
Overview

In the table vertical direction handle feed, table vertical direction JOG feed, and table vertical direction incremental feed, the tool is moved in the table vertical direction.

Explanation

- Table vertical direction

The table vertical direction is a direction vertical to the table. It is equal to the tool axis direction specified in parameter No.19697 when all of the rotation axes for controlling the table are at a an angle of 0 degree. When the rotation axes for controlling the table rotate, the table vertical direction changes according to the rotation axis angle.



- Table-based vertical direction feed in the tilted working plane indexing mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the table-based vertical direction feed in the tilted working plane indexing mode is assumed to be the Z direction in the feature coordinate system of the tilted working plane indexing.

FRYER MACHINE SYSTEMS INC
pb Zanensky

- Table vertical direction handle feed

The table vertical direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.

<2> Both the tool axis direction feed mode signal ALNGH and the table base signal TB_BASE are set to "1".

<3> The state of the first manual handle feed axis selection signals HS1A - HS1E to make the table vertical handle feed mode effective is set in parameter No.12310.

<4> The value of parameter No.12310 matches the first manual handle feed axis selection signals HS1A - HS1E.

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the table vertical direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.
If Manual handle feed maximum feedrate change signal HNDLF is "1", clamping is performed by the maximum handle feedrate (parameter No. 1434).

- Table vertical direction JOG feed/table vertical direction incremental feed

The table vertical direction JOG feed or table vertical direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.

<2> Both the tool axis direction feed mode signal ALNGH and the table base signal TB_BASE are set to "1".

<3> The feed axis direction selection signal [+Jn,-Jn (where n = 1 to the number of controlled axes)] is set to "1" for the axis corresponding to the direction specified by parameter No.19697.
Ex.) Parameter No.19697 = 3 (+Z-axis direction); Z-axis is the 3rd axis.
- +J3 : Table vertical direction +
- -J3 : Table vertical direction -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) for a driven feed axis direction selection signal. The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

3.8.5 Table Horizontal Direction Handle Feed / Table Horizontal Direction JOG Feed / Table Horizontal Direction Incremental Feed

Overview

In the table horizontal direction handle feed, table horizontal direction JOG feed, and table horizontal direction incremental feed, the tool is moved in the table horizontal direction.
If bit 1 (FLL) of parameter No. 12320 is set to 1, the tool or table is moved in the latitude or longitude direction determined by the table-based vertical direction vector.

TRYER MACHINE SYSTEMS INC
Rob Zalensky

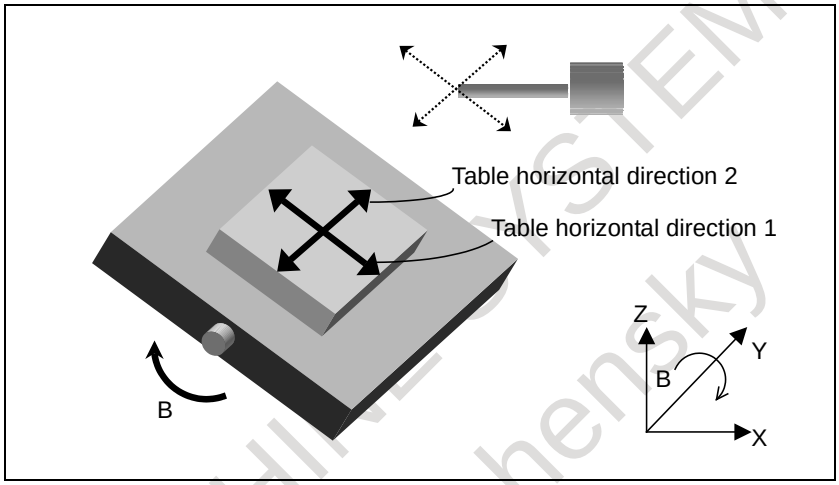
Explanation

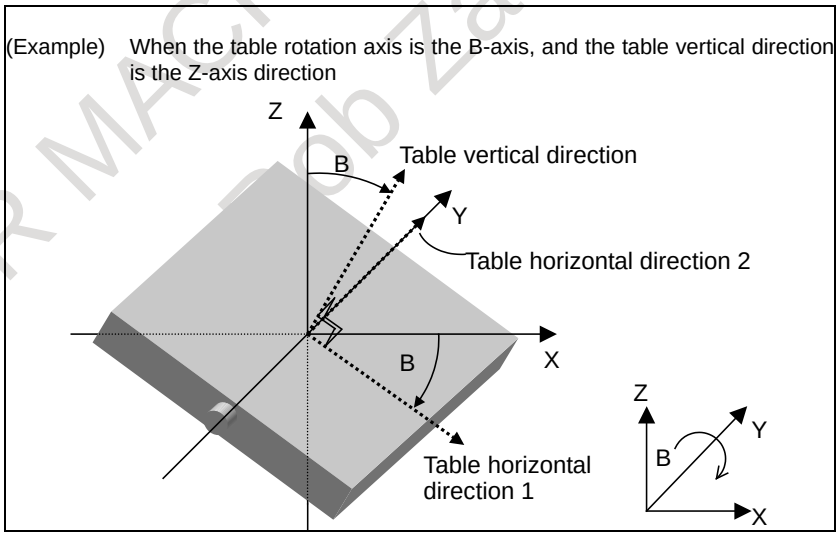
- Table horizontal direction

There are two table horizontal directions, which are perpendicular to the table vertical direction (see the previous section).

Parameter No.19697	Table horizontal direction 1	Table horizontal direction 2
1 (The reference tool direction is +X.)	+Y direction	+Z direction
2 (The reference tool direction is +Y.)	+Z direction	+X direction
3 (The reference tool direction is +Z.)	+X direction	+Y direction

This table shows the table horizontal directions that may be taken when the angles of all the rotation axes for controlling the table are 0 degree.
As the rotation axes for controlling the table rotate, the table horizontal direction changes according to the rotation axis angle.





- Latitude and longitude directions

When bit 1 (FLL) of parameter No. 12320 is set to 1, the feed direction is defined as follows:
Let a vector perpendicular to a plane formed by the table-based vertical direction vector ($\vec{T}$) and normal axis direction vector ($\vec{P}$) (parameter No. 12321) be the table-based horizontal direction 1 (longitude direction) vector ($\vec{R1}$). When tool axis right-angle direction 1 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Longitude direction feed)
Equation: $\vec{R1} = \vec{P} \times \vec{T}$

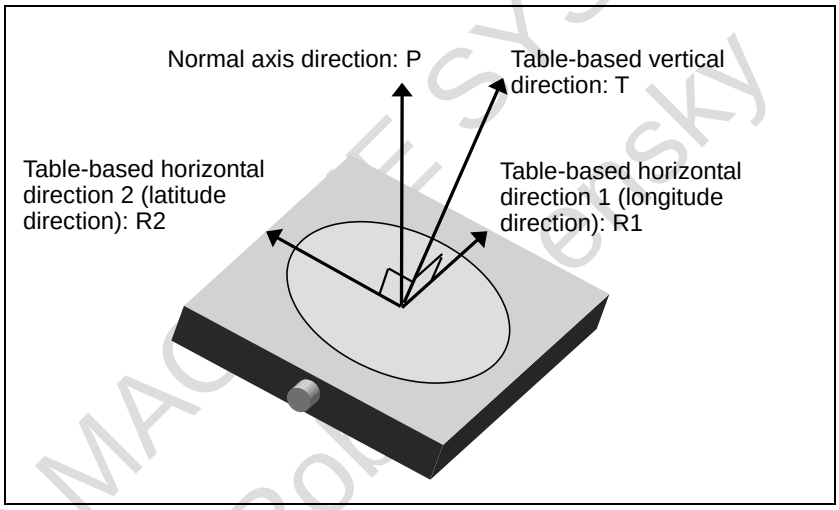
FRYER MACHINE SYSTEMS INC
Bob Zahensky

Let a vector perpendicular to the table-based vertical direction vector ($\vec{T}$) and table-based horizontal direction 1 (longitude direction) vector ($\vec{R1}$) be the table-based horizontal direction 2 (latitude direction) vector ($\vec{R2}$). When tool axis right-angle direction 2 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Latitude direction)
Equation: $\vec{R2} = \vec{T} \times \vec{R1}$

When table-based vertical direction vector ($\vec{T}$) is parallel to the normal axis direction vector ($\vec{P}$) (parameter No. 12321) (when the angle between them is not greater than the setting of parameter No. 12322), tool axis right-angle direction 1 and tool axis right-angle direction 2 are assumed as follows:

Parameter No. 12321	Normal axis direction	Table-based horizontal direction 1	Table-based horizontal direction 2
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

If 0 is set in parameter No. 12321, the normal axis direction is set to the tool axis direction.
If a value other than 0 to 3 is specified in parameter No. 12321, alarm PS5459, "MACHINE PARAMETER INCORRECT" is issued.



- Table-based horizontal direction feed in the tilted working plane indexing mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the table-based horizontal direction feed in the tilted working plane indexing mode is defined as follows:
Table-based horizontal direction 1:
X direction in the feature coordinate system of the tilted working plane indexing
Table-based horizontal direction 2:
Y direction in the feature coordinate system of the tilted working plane indexing

- Table horizontal direction handle feed

The table horizontal direction handle feed is enabled when the following four conditions are satisfied:
<1> Handle mode is selected.
<2> Both the tool axis right-angle direction feed mode signal RGHTH and the table base signal TB_BASE are set to 1.
<3> The state of the first manual handle feed axis selection signals HS1A - HS1E to make the table horizontal direction handle feed mode effective is set in parameter No.12311 or No.12312.
<4> The value of parameter No.12311 or No.12312 matches the first manual handle feed axis selection signals HS1A - HS1E.

FRYER MACHINE SYSTEMS INC Roland Jensky

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the table horizontal direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.
If Manual handle feed maximum feedrate change signal HNDLF is "1", clamping is performed by the maximum handle feedrate (parameter No. 1434).

- Table horizontal direction JOG feed/table horizontal direction incremental feed

The table horizontal direction JOG feed or table horizontal direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.
<2> Both the tool axis right-angle direction feed mode signal RGHTH and the table base signal TB_BASE are set to "1".
<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the axis corresponding to the direction that is perpendicular to the direction specified by parameter No.19697.
Ex.) Parameter No.19697 = 3 (+Z-axis direction); X-, Y-, and Z-axes are the 1st, 2nd, and 3rd axes respectively.
- +J1 : Table horizontal direction 1 +
- -J1 : Table horizontal direction 1 -
- +J2 : Table horizontal direction 2 +
- -J2 : Table horizontal direction 2 -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) of a driven feed axis direction selection signal. The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

Note

1 When a 3-dimensional handle interrupt is performed, rotation axis command execution must not be in progress in automatic operation.
2 When the manual reference position return mode is selected, 3-dimensional manual feed is not enabled.
3 If per-axis interlock is enabled to at least one of 3-dimensional manual feed axes, movement with manual feed is not performed.
4 When the offset value specified for the tool length offset function is used for tool center point rotation feed (when bit 2 (LOD) of parameter No. 19746 is set to 1), the controlled point should generally be shifted. (Set bit 5 (SVC) of parameter No. 19665 to 1.)
In this case, specify the tool length with a radius value.
5 The feed direction of a 3-dimensional handle interrupt is updated as follows.
(a) When bit 7 (PRA) of parameter No. 12319 is set to 0
It is updated when one of the following operation is executed.
- The mode of NC (ex, MEM or HANDLE) is changed.
- Reset is executed.
- The state of manual handle interrupt axis selection signals (HS1IA to HS1ID<Gn041.0 to 3>, HS1IE<Gn411.4> in case of manual pulse generator No.1) is changed.
- 3-dimensional manual feed turns enabled or disabled.

FRYER MACHINE SYSTEMS INC Rob Zalensky

(b) When bit 7 (PRA) of parameter No. 12319 is set to 1
It is always updated.

3.9 DISTANCE CODED LINEAR SCALE INTERFACE

Overview

The interval of each reference marks of distance coded linear scale are variable. Accordingly, if the interval is determined, the absolute position can be determined. The CNC measures the interval of reference marks by axis moving of short distance and determines the absolute position. Consequently the reference position can be established without moving to reference position.

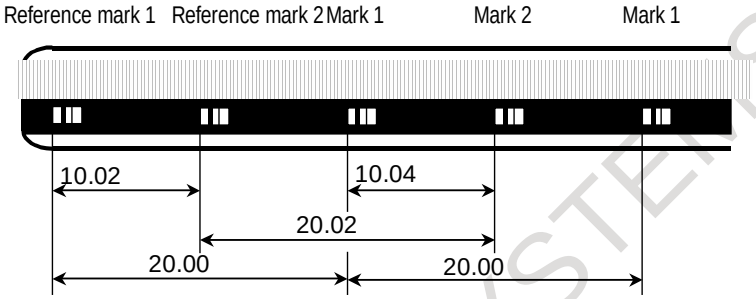


Fig. 3.9 Example of distance coded linear scale

3.9.1 Procedure for Reference Position Establishment

Procedure

(1) Select the JOG mode, and set the manual reference position return selection signal ZRN to "1".
(2) Set a direction selection signal(+J1,-J1,+J2,-J2,…) for a target axis.
(3) The axis is fed at a constant low speed (reference position return FL feedrate specified by parameter No.1425 setting).
(4) When a reference mark is detected, the axis stops, then the axis is fed at a constant low speed again.
(5) Above (4) is executed repeatedly until two, three or four reference marks are detected. And absolute position is determined and reference position establishment signal (ZRF1,ZRF2,ZRF3, …) turns to "1". (A number of reference marks is determined by bit 2 (DC2x) and 1 (DC4x) of parameter No.1802.)

The timing chart for this procedures is given below.

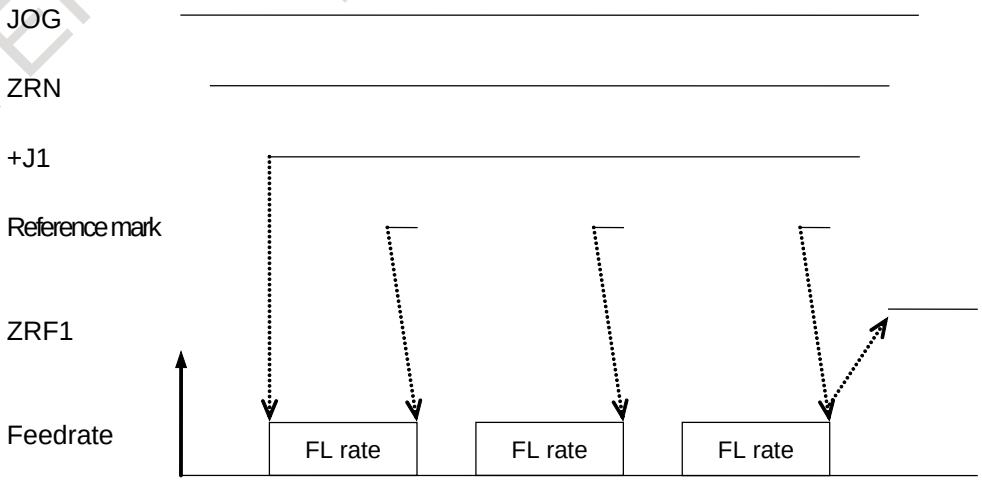


Fig. 3.9.1 Timing chart for reference position establishment

FRYER MACHINE SYSTEMS INC
Rob Zakensky

- Procedure for establishing a reference position through automatic operation

If an automatic reference position return (G28) is specified before a reference position is not established, steps (3) to (5) above are performed automatically.
After the reference position is established, the automatic reference position return is performed.

- Stopping the operation for establishing a reference position

The operation for establishing a reference position is stopped if any of
the following operations is performed in steps (3) to (5), described above.

- Reset
- Setting the feed axis direction selection signal (+J1, -J1, +J2, -J2, etc.) to 0

If any of the following operations is performed during the operation of automatic reference position return (G28) before a reference position is not established, the operation for establishing a reference position stops:

- Reset
- Performing feed hold during movement from an intermediate position

If the operation for establishing a reference position is stopped by an operation other than a reset, the operation for establishing a reference position must be reset and resumed.

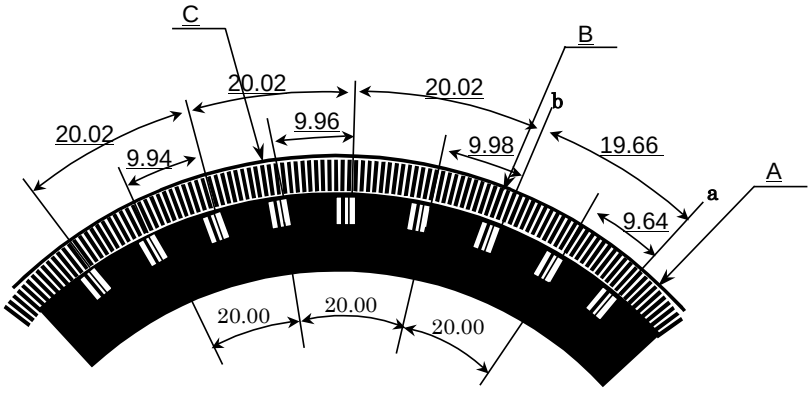
3.9.2 Reference Position Return

(1) When the reference position is not established and the axis moved by turning the feed axis direction signal (+J1,-J1,+J2,-J2,...) to "1" in REF mode, the reference position establishment procedure is executed.
(2) When the reference position is already established and the axis is moved by turning the feed axis direction signal (+J1,-J1,+J2,-J2,...) to "1" in REF mode, the axis is moved to the reference point without executing the reference position establishment procedure.
(3) When the reference position is not established and the reference position return command (G28) is executed, the reference position establishment procedure is executed. The next movement the axis depends on the setting of bit 0 (RFS) of parameter No. 1818.
(4) When the reference position is already established and the reference position command (G28) is executed, the movement of the axis depends on the setting of bit 1 (RF2) of parameter No.1818.

3.9.3 Distance Coded Rotary Encoder

In case of setting a rotary axis, if bit 3 (DCRx) of parameter No.1815 is set, the setting axis is regarded as being equipped with a distance coded rotary encoder.
In case of distance coded rotary encoder, the marker interval may be different from parameter setting value. (a-b section of the following figure) When the reference point return is executed through this section, it is not able to establish the reference point. Therefore, in case of distance coded rotary encoder, if the reference point return is started for B point from A point of below figure, the reference point is not established yet at B point. The reference point return is re-started for C point. The reference point return procedure is finished at C point.

FRYER MACHINE SYSTEMS INC
Rob Lehonsky



- When the reference point return procedure is executed, the coordinate value are rounded in 0 to 360 degree, even if Machine coordinate values are linear axis type.
- In case of distance coded rotary encoder, only the measurement by three points or four points is possible. (bit 2 (DC2) of parameter No.1802 is disregarded as 0.)

3.9.4 Axis Synchronization Control

Requirements when this function is used with axis synchronization control axes

When this function is used with axis synchronization control axes, the distance coded linear scale used for the master axis and that used for the slave axis must have reference marks placed at identical intervals.
(Set identical values in parameters Nos. 1821 and 1882 for both the master and slave axes.)
A distance coded linear scale can be installed on the master and slave axes either in the same or opposite direction.
This function does not work unless the use of this function is specified for both the master and slave axes (bit 2 (DCL) of parameter No. 1815 is 1).
Also, in all parameters related to this function, except parameter No. 1883, 1884 (distance from the scale zero point to reference position 1, 2), set identical values for both the master and slave axes.
If a parameter value for the master axis differs from the corresponding parameter value for the slave axis, alarm SV1051, “ILLEGAL SYNCHRONOUS AXIS” is issued.

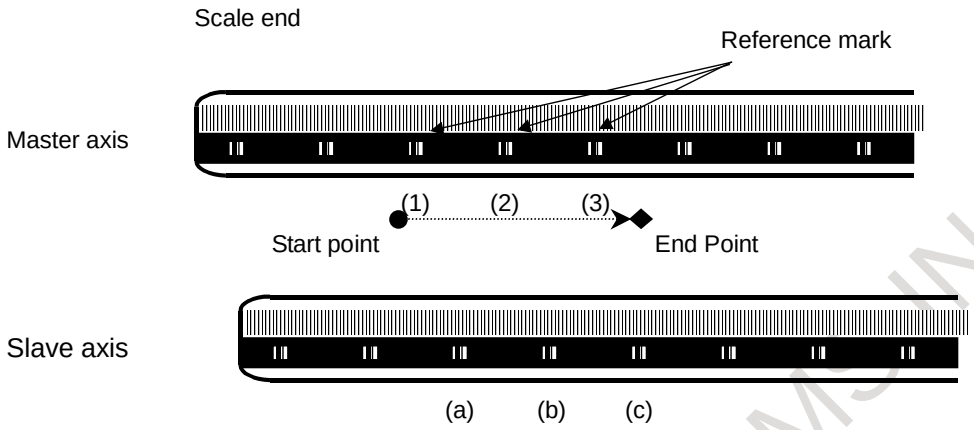
NOTE
When this function is used with axis synchronization control axes for which the operation mode is switched between synchronization operation and normal operation, this function is enabled only if the synchronization select signal (SYNC1, SYNC2, ...) is "1". (During establishment of a reference position, the synchronization select signal status must be maintained.)

Reference position establishment with axis synchronization control axes

With axis synchronization control axes, a reference position is established as follows. When a reference mark for the master or slave axis is detected, a stop takes place temporarily. Then, a feed operation is performed again at the reference position return FL feedrate. This sequence is repeated until a reference mark is detected three or four times for both the master and slave axes. Then the absolute position is calculated for both the master and slave axes, and the reference position establish signals ZRF1, ZRF2, ... are set to "1".
After the reference position has been established by the above operation, a synchronization error is corrected. (Checking for excessive synchronization error alarm 2 is made even during reference position establishment.)

FRYER MACHINE SYSTEMS INC
Rob Zaherensky

(Example of 3 points measurement system)



In the above example, the following sequence is executed.

a. When the reference mark (1) of the master axis is detected, both master axis and slave axis stop.
b. Both the axes begin to move again at a reference position return FL feedrate.
c. When the reference mark (a) of the slave axis is detected, both axes stop again.
d. Both the axes begin to move again at FL feedrate.
e. Both axes repeat the operation until all point ((2) -> (b) ->(3) -> (c)) are detected.
f. When the slave axis detects the third reference mark (c), both the axes end the reference position establishment.

NOTE

In case of this function is used with axis synchronization control axes, if the value of parameters Nos. 1883 and 1884 for both the master and slave axes is 0, the reference position is not established. Also, the reference position establish signals ZRF1, ZRF2, ... are set to "0".

3.9.5 PMC Axis Control

In PMC axis control, if the reference position return command (axis control command code 05H) is issued for an axis having a distance coded linear scale, reference position return is performed according to the reference position return sequence for the distance coded linear scale.
Specifically, the following operations take place:

Before reference position establishment	The reference position is established by detecting two, three or four reference marks. Movement to the reference position is not performed.
After reference position establishment	Positioning at the reference position is performed.

3.9.6 Angular Axis Control

There are the following limitations when the angular axis control is used.

- It is necessary to use the linear scale with the distance coded reference mark for both the perpendicular axis and the angular axis.
- When the reference point of the perpendicular axis is established, it is necessary to establish the reference point of the angular axis previously. When the reference point of the angular axis is not previously established, the alarm DS0020, “REFERENCE RETURN INCOMPLETE” is generated.
- During the reference point establishment operation of the angular axis, the command in the perpendicular axis is invalid in the manual reference point return.

FRYER MACHINE SYSTEMS INC
Rob Zamensky

3.9.7 Note

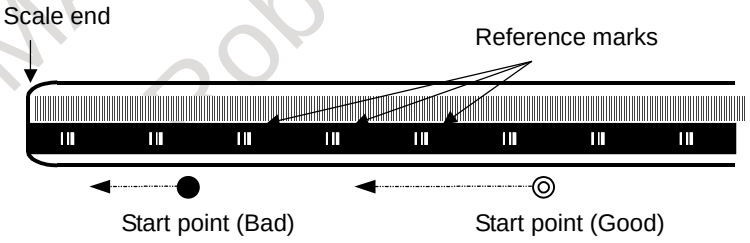
(1) In the case of the actual interval of reference marks is different from parameter setting value, the alarm DS1449, “REFERENCE MARKS ARE DIFFERENT FROM PARAMETER” occurs.

(2) This function is disabled if any of the following conditions is satisfied:
- Either parameter No.1821 (mark-1 interval) or parameter No.1882 (mark-2 interval) is set to 0.
- The setting of parameter No.1821 is greater than or equal to the setting of parameter No.1882.
- The difference between the settings made for parameters Nos. 1821 and 1882 is greater than or equal to twice either setting.
- The absolute-position detection function is enabled. (Bit 5 (APCx) of parameter No.1815 is set to 1.)

(3) A difference of parameters Nos.1821 and 1882 must be more than 4.
Example)
When the scale, which is that mark1 interval is 20.000mm and mark2 interval is 20.004mm, is used on IS-B machine :
When the detection unit of 0.001mm is selected, parameter No.1821 and No.1882 must be set 20000 and 20004, and the difference of them is 4.
To use such a scale, adjust the detection unit by modification of parameters No.1820 (CMR) and Nos.2084/2085(flexible feed gear) to make the difference of parameters Nos.1821 and 1882 more than 4 as following examples.
(a) Set the detection unit=0.0001mm, and set No.1821=200000, No.1882=200040
(b) Set the detection unit=0.0005mm, and set No.1821=40000, No.1882=40008

NOTE
When the detection unit is changed, parameters relating to the detection unit (such as the effective area and positional deviation limit) must also be changed accordingly.

(4) In this procedure, the axis does not stop until two, three or four reference marks are detected. If this procedure is started at the position near the scale end, CNC can not detect three or four reference marks and the axis does not stop until over travel alarm occurs. Care to start at the position that has enough distance from scale end.



(5) When the axis used this function, the following function can not be used.
- Absolute position detection (absolute pulse coder)

(6) If axial movement is made in the direction opposite to that of reference position return, the movement is reversed to the direction of reference position return after three or four reference marks have been detected. Steps (3) to (5) of the basic procedure for establishing a reference position are carried out to establish the reference position.

(7) Straightness compensation function
When the reference point establishment of moving axis is executed after the establishment of compensation axis, the compensation axis is moved by straightness compensation amount when the reference point of moving axis is established.

(8) The reference point establishment is not performed during synchronous control is activated.

(9) The reference point establishment is not performed during composite control is activated.

(10) The reference point establishment is not performed during superimposed control is activated.

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

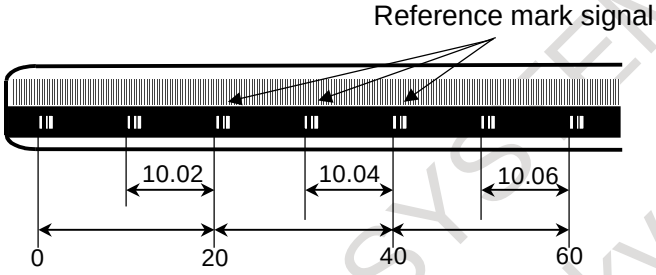
3.10 LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL)

Overview

By using High-resolution serial output circuit for the linear scale with distance-coded reference marks (serial), the CNC measures the interval of referenced mark by axis moving of short distance and determines the absolute position.
This function enables high-speed high-precision detection by using High-resolution serial output circuit.
It is available that using maximum stroke 30 meters length.

Explanation

The linear scale with distance-coded reference marks (serial) is combined the irregular reference marked linear scale with the High-resolution serial output circuit, it can detect the accurate position.



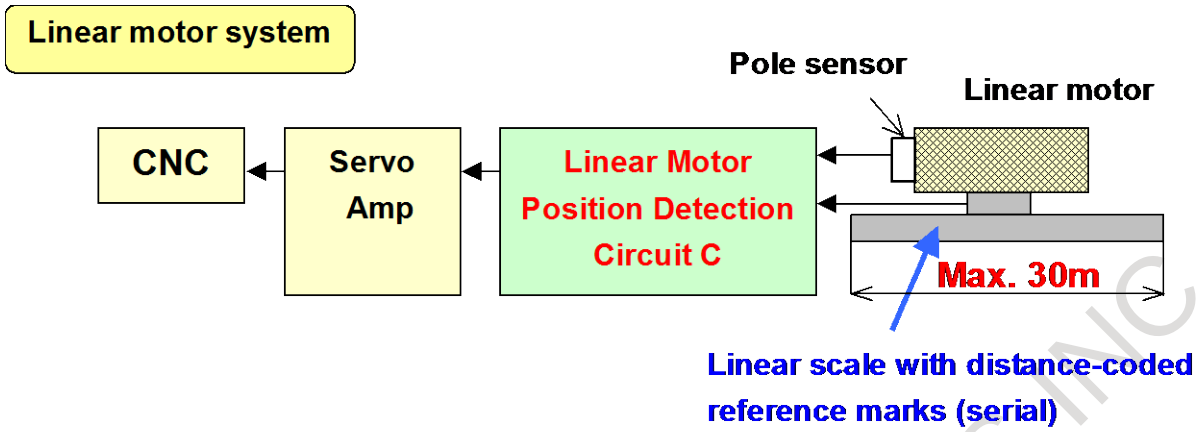
The CNC measures the interval of referenced mark by axis moving of short distance and determines the absolute position, because of the interval of each reference mark is different with regular interval.
It is not necessary that the axis is moved to the reference position for establishment of reference position.

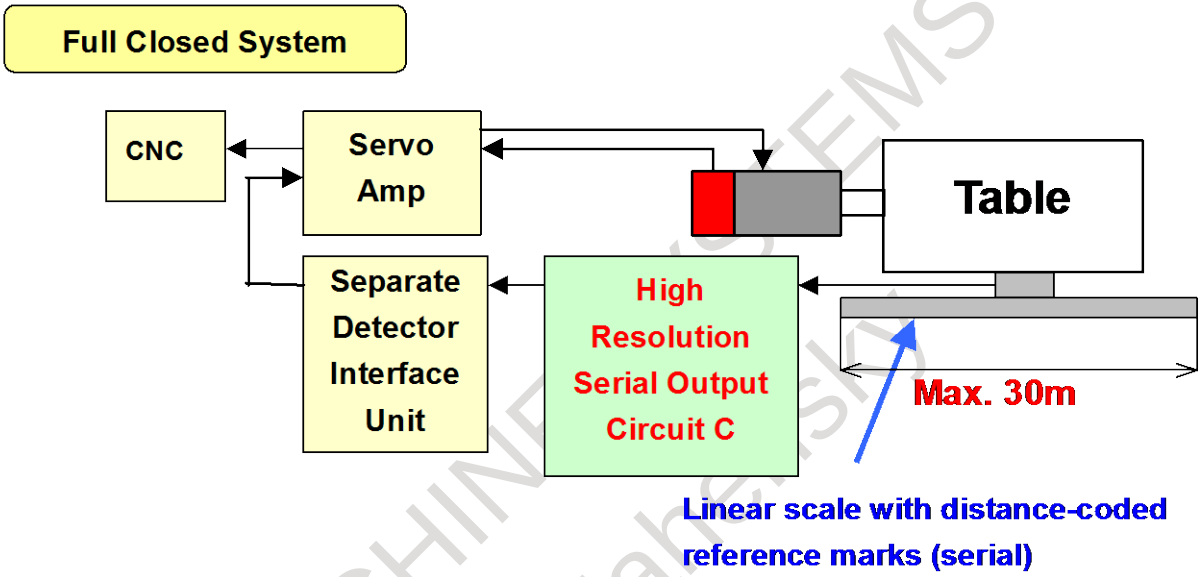
This function enables high-speed high-precision detection by using High-resolution serial output circuit.
It is available that using maximum stroke 30 meters length.

- Connection

It is available under linear motor system and full closed system.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

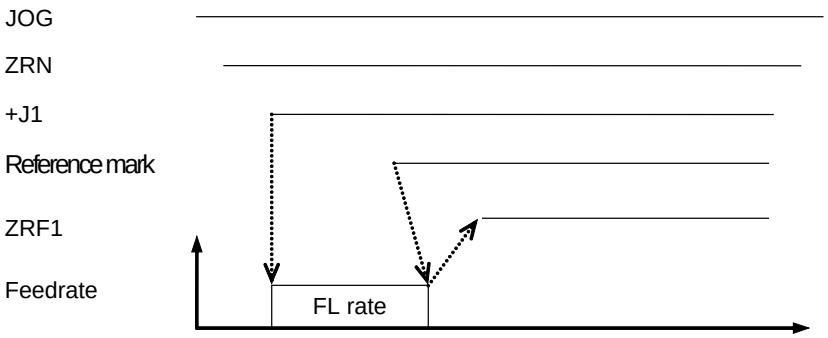




- **Procedure for reference position establishment through manual operation**

(1) Select the JOG mode, and set the manual reference position return selection signal ZRN to "1".
(2) Set a direction selection signal(+J1,-J1,+J2,-J2,…) for a target axis.
(3) The axis is fed at a constant low speed (reference position return FL feedrate specified by parameter No.1425 setting).
(4) When the absolute position of linear scale with distance-coded reference marks (serial) is detected, the axis stops. Then the absolute position of CNC is calculated and reference position establishment signal (ZRF1,ZRF2,ZRF3,…) turns to "1".

The timing chart for this procedures is given below.



FRYER MACHINE SYSTEMS INC Rob Zaharsky

- Procedure for reference position establishment through automatic operation

If an automatic reference position return (G28) is specified before a reference position is not established, steps (3) to (4) above are performed automatically.
After the reference position is established, the automatic reference position return is performed by setting of bit 0 (RFS) of parameter No. 1818.

- Stopping the operation for establishing a reference position

The operation for establishing a reference position is stopped if any of
the following operations is performed in steps (3) to (4), described above.

- Reset
- Setting the feed axis direction selection signal (+J1, -J1, +J2, -J2, etc.) to "0"
- Setting the Servo off signals (SVF1, SVF2, etc.) to "1"

If any of the following operations is performed during the operation of automatic reference position return (G28) before a reference position is not established, the operation for establishing a reference position stops:

- Reset
- Performing feed hold during movement from an intermediate position
- Setting the Servo off signals (SVF1, SVF2, etc.) to "1"

If the operation for establishing a reference position is stopped by an operation other than a reset, the operation for establishing a reference position must be reset and resumed.

- Establishing a reference position and moving to the reference position

By following operation, establishing a reference position and moving to the reference position is performed.

	Moving through manual operation in REF mode	Moving through automatic operation by automatic reference position return (G28)
The reference position is not established.	Establishing the reference position	Firstly, moving to the intermediate position, and establishing the reference position. Secondly, whether moving to the reference position or not is performed by setting bit 0 (RFS) of parameter No.1818.
The reference position is established.	Moving to the reference position	Whether moving to the intermediate position and the reference position or not is performed by setting bit 1 (RF2) of parameter No.1818.

- Feed axis synchronization control

In case of using the axis synchronization control, please confirm the following items.

- When this function is used with axis synchronization control axes, the linear scale with distance-coded reference marks (serial) used for the master axis and that used for the slave axis must have reference marks placed at identical intervals.
- The linear scale with distance-coded reference marks (serial) can be installed on the master and slave axes either in the same or opposite direction.
- To the parameters, which relate to this function (except No.1883, No.1884), the same value must be set for the master axis and for the slave axis.
- The linear scale with distance-coded reference marks (serial) should be applied for the master axis and the slave axis.
  If either of the master axis or the slave axis is not the linear scale with distance-coded reference marks (serial), alarm DS0018, "SERIAL DCL:MISMATCH(SSYNC CTRL)" occurs when reference position establishment is tried.
- During operating the establishment of reference position, the state of signal (SYNCn or SYNCJn) for selecting synchronized axis should be kept.

FRYER MACHINE SYSTEMS INC
Rob Zarnensky

Procedure for reference position establishment by axis synchronization control is as follows.

- Both of axes (master axis and slave axis) are fed on the reference position return FL feedrate until distance coded scales of both axes detect the absolute position.
- Then absolute position of both axes are calculated and Reference Position Establishment Signals (ZRF1,ZRF2,...) turn to "1".

- Angular axis control

In case of using the angular axis control, please confirm the following items.

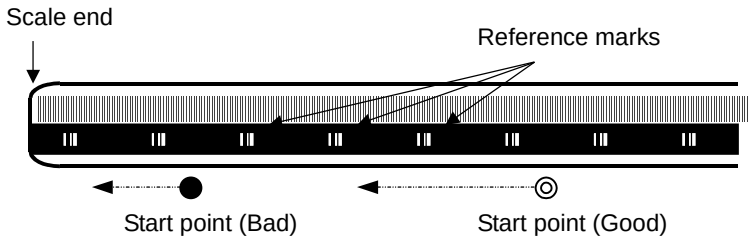
- It is necessary to use the linear scale with distance-coded reference marks (serial) for both the perpendicular axis and the angular axis.
  If not, the alarm DS0019, "SERIAL DCL:MISMATCH(ANGL-AXIS)" occurs when reference position establishment is commanded.
- When the reference point establishment of angular and perpendicular axes are tried, please set bit 2 (AZR) of parameter No. 8200 to 0 and input signal NOZAGC to "0".
  If not, the alarm DS0019 occurs when reference position establishment is commanded.
- When the reference point of the perpendicular axis is established, it is necessary to establish the reference point of the angular axis previously. When the reference point of the angular axis is not previously established, the alarm DS0020, "REFERENCE RETURN INCOMPLETE" occurs.
- On angular axis control, if you use automatic setting of parameter No.1883,1884 on reference point establishment, please establish reference point of perpendicular axis after reference point establishment and return of angular axis.

In manual reference position return, the perpendicular axis cannot be specified while the angular axis reference point is being established. The perpendicular axis, if specified, is ignored.

⚠ CAUTION

1 When the Linear scale with distance-coded reference marks (serial) is used, please set bit 3 (SDCx) of parameter No.1818 to 1.

2 On the Linear scale with distance-coded reference marks (serial), the axis does not stop until three reference marks are detected. If this procedure is started at the position near the scale end, CNC can not detect three reference marks and the axis does not stop until over travel alarm occurs. Please care to start at the position that has enough distance from scale end.
And if establishment of reference position is failed, the establishment is retried. Then axis does not stop until still more three reference marks are detected. So please set the maximum move amount (detection unit : parameter No.14010) not to reach the scale end.



3 On flexible synchronization control mode, reference position can't be established.

4 Straightness compensation function
When the reference point establishment of moving axis is executed after the establishment of compensation axis, the compensation axis is moved by straightness compensation amount when the reference point of moving axis is established.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

⚠ **CAUTION**

5 It is not available to use this function and the temporary absolute coordinate setting together.

6 Angular axis control cannot be performed together with synchronous control, composite control, or superposition control.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

4 AUTOMATIC OPERATION

Programmed operation of a CNC machine tool is referred to as automatic operation.

4.1 MEMORY OPERATION

Programs are registered in memory in advance. When one of these programs is selected and the cycle start switch on the machine operator's panel is pressed, automatic operation starts, and the cycle start LED goes on.
When the feed hold switch on the machine operator's panel is pressed during automatic operation, automatic operation is stopped temporarily. When the cycle start switch is pressed again, automatic operation is restarted.
When the RESET on the MDI unit is pressed, automatic operation terminates and the reset state is entered.
For the multi-path control, the programs for the multiple paths can be executed simultaneously so the multiple paths can operate independently at the same time.
The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

Memory operation

Procedure

1 Press the MEMORY mode selection switch.
2 Select a program from the registered programs. To do this, follow the steps below.
  2-1 Press the function key PROG to display the program screen.
  2-2 Press the address key P 0.
  2-3 Enter a program number using the numeric keys.
  2-4 Press the [O SRH] soft key.
3 For the multi-path control, select the path to be operated with the path selection switch on the machine operator's panel.
4 Press the cycle start switch on the machine operator's panel.
  Automatic operation starts, and the cycle start LED goes on. When automatic operation terminates, the cycle start LED goes off.
5 To stop or cancel memory operation midway through, follow the steps below.
  a. Stopping memory operation
     Press the feed hold switch on the machine operator's panel. The feed hold LED goes on and the cycle start LED goes off. The machine responds as follows:
     (i) When the machine was moving, feed operation decelerates and stops.
     (ii) When dwell was being performed, dwell is stopped.
     (iii) When M, S, or T was being executed, the operation is stopped after M, S, or T is finished.
     When the cycle start switch on the machine operator's panel is pressed while the feed hold LED is on, machine operation restarts.
  b. Terminating memory operation
     Press RESET on the MDI unit.
     Automatic operation is terminated and the reset state is entered.
     When a reset is applied during movement, movement decelerates then stops.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

- Memory operation

After memory operation is started, the following are executed:

(1) A one-block command is read from the specified program.
(2) The block command is decoded.
(3) The command execution is started.
(4) The command in the next block is read.
(5) Buffering is executed. That is, the command is decoded to allow immediate execution.
(6) Immediately after the preceding block is executed, execution of the next block can be started. This is because buffering has been executed.
(7) Hereafter, memory operation can be executed by repeating the steps (4) to (6).

- Stopping and terminating memory operation

Memory operation can be stopped using one of two methods: Specify a stop command, or press a key on the machine operator's panel.

- The stop commands include M00 (program stop), M01 (optional stop), and M02 and M30 (program end).
- There are two keys to stop memory operation: The feed hold key and reset key.

- Program stop (M00)

Memory operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged as in single block operation. The memory operation can be restarted by pressing the cycle start button. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

- Optional stop (M01)

Similarly to M00, memory operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel is set to ON. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

- Program end (M02, M30)

When M02 or M30 (specified at the end of the main program) is read, memory operation is terminated and the reset state is entered.
Since it depends on machine specification whether the active line returns to top of program or not when M02 or M30 is read, refer to the manual provided by the machine tool builder for details.

- Feed hold

When Feed Hold button on the operator's panel is pressed during memory operation, the tool decelerates to a stop at a time.

- Reset

Automatic operation can be stopped and the system can be made to the reset state by using

on the MDI unit or external reset signal. When reset operation is applied to the system during a tool moving status, the motion is slowed down then stops.

- Optional block skip

A block skip slash (/) must be specified at the beginning of a block.
Note that if a slash is placed elsewhere, the information from the slash to immediately before the EOB code is ignored.

FRYER MACHINE SYSTEMS INC
Rob Zaienskyj

- Cycle start for the multi-path control

For the multi-path control, a cycle start switch is provided for each path. This allows the operator to activate a single paths to operate them at the same time in memory operation or MDI operation. In general, select the path to be operated with the path selection switch on the machine operator's panel and then press the cycle start button to activate the selected path. (The procedure may vary with the machine tool builder. Refer to the appropriate manual issued by the machine tool builder.)

4.2 MDI OPERATION

In the MDI mode, a program consisting of up to 511 characters can be created in the same format as normal programs and executed from the MDI unit.
MDI operation is used for simple test operations.
The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

MDI Operation

Procedure

1 Select the MDI mode.
For the multi-path control, select the path for which a program is to be created with the path selection switch. Create a separate program for each path.

2 Press the function key PROG to select the program screen. The following screen appears:

MDI program screen

At this time, program number "O0000" is inserted automatically.

3 Prepare a program to be executed by an operation similar to normal program editing. M99 specified in the last block can return control to the beginning of the program after operation ends. Word insertion, modification, deletion, word search, address search, and program search are available for programs created in the MDI mode.

4 To entirely erase a program created in MDI mode, use one of the following methods:

a. Enter address key P 0, then press DELETE.

b. Alternatively, press RESET. In this case, set bit 7 (MCL) of parameter No. 3203 to 1 in advance.

FRYER SYSTEMS INC

5 To execute a program, set the cursor on the head of the program.
Push Cycle Start button on the operator's panel. By this action, the prepared program will start.
(For the multi-path control, select the path to be operated with the path selection switch on the machine operator's panel beforehand.) When the program end (M02, M30) or ER(%) is executed, the prepared program will be automatically erased and the operation will end.
By command of M99, control returns to the head of the prepared program.

6 To stop or terminate MDI operation in midway through, follow the steps below.

a. Stopping MDI operation
Press the feed hold switch on the machine operator's panel. The feed hold LED goes on and the cycle start LED goes off. The machine responds as follows:
(i) When the machine was moving, feed operation decelerates and stops.
(ii) When dwell was being performed, dwell is stopped.
(iii) When M, S, or T was being executed, the operation is stopped after M, S, or T is finished.
When the cycle start switch on the machine operator's panel is pressed, machine operation restarts.

b. Terminating MDI operation
Press the [RESET].
Automatic operation is terminated and the reset state is entered.
When a reset is applied during movement, movement decelerates then stops.

Explanation

The previous explanation of how to execute and stop memory operation also applies to MDI operation, except that in MDI operation, M30 does not return control to the beginning of the program (M99 performs this function).

- Erasing the program

Programs prepared in the MDI mode will be erased in the following cases:

- In MDI operation, if M02, M30 or ER(%) is executed.
- When bit 6 (MER) of parameter No. 3203 is set to 1, and the last block of the program is executed in single block operation

NOTE
In the two cases above, program erasure can be prevented by setting bit 6 (MKP) of parameter No. 3204 to 1.

- In MEM mode, if memory operation is performed.
- In EDIT mode, if any editing is performed.
- When the address key [P 0] and [DELETE] are pressed.
- Upon reset when bit 7 (MCL) of parameter No.3203 is set to 1

NOTE
Upon reset when the parameter MCL = 0, the cursor moves to the end of the program.

- Restart

If a program is not executed even once after the program is input, the program is executed from the beginning, regardless of where the cursor is placed. However, a program is executed starting at the beginning of the block where the cursor is place, if the program is stopped for a reason such as single block operation after restart of an MDI operation then is restarted after an editing operation.

FRYER MACHINE SYSTEMS INC
Rob Zalensky

⚠ CAUTION

When an MDI program is restarted, the program is executed starting at the beginning of the block where the cursor is placed, regardless of the cursor position in the block.

(Example)

When the cursor is placed on G90

:

G91 X100.0 G90Y200.0 Z300.0 ;

:

The program is executed starting at the beginning (namely, G91) of this block. So, the tool moves by 100.0 along the X-axis in the incremental programming, and moves to 200.0 and 300.0 along the Y-axis and Z-axis, respectively, in the absolute programming.

- Editing a program during MDI operation

A program can be edited during MDI operation. By setting bit 5 (MZE) of parameter No. 3203 to 1, editing can be disabled. However, even when bit 5 (MZE) of parameter No. 3203 is set to 1, editing can be enabled by resetting the operation.

- Absolute/incremental command

When bit 4 (MAB) of parameter No. 3401 is set to 1, the absolute/incremental programming of MDI operation does not depend on G90/G91. In this case, the incremental programming is set when bit 5 (ABS) of parameter No. 3401 is set to 0, and the absolute programming is set when bit 5 (ABS) of parameter No. 3401 is set to 1.

Bit 4 (MAB) of parameter No. 3401=0	Bit 4 (MAB) of parameter No. 3401=1	
	Parameter ABS =0	Parameter ABS =1
Absolute mode operation with G90 command, and incremental programming operation with G91 command	Incremental mode operation at all times, independent of G90/G91 command	Absolute mode operation at all times, independent of G90/G91 command

NOTE

When G code system A is used on a lathe system, the parameters MAB and ABS are invalid.

Limitation

- Program registration

Programs created in MDI mode cannot be registered.

- Number of characters in a program

A created program can consist of up to 511 characters including "O0000" automatically inserted.

- Subprogram nesting

The subprogram call command (M98) can be described in a program created in MDI mode. That is, programs that are registered in memory through MDI operation can be called and executed. The level of subprogram call nesting is the same as in MEM operation.

- Macro call

When the custom macro function is enabled, a macro program can be created and executed even in the MDI mode. Moreover, a macro program can be called for execution.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

NOTE

The GOTO statement, WHILE statement, and DO statement cannot be executed in a program created in the MDI mode. An alarm PS0377, "ILLEGAL MODE FOR GOTO/WHILE/DO" is issued.
When a program including those statements is to be executed, register the program in the program memory then call the program for execution.

4.3 DNC OPERATION

By activating automatic operation during the DNC operation mode (RMT), it is possible to perform machining (DNC operation) while a program is being read in via RS-232C interface. Available devices include a memory card, Embedded Ethernet and Data Server.
To use the DNC operation function, it is necessary to set the parameters related to the RS-232C interface in advance.
The procedure described below is just an example. For actual operation, refer to the relevant manual of the machine tool builder. For a Data Server, also refer to the “Fast Ethernet/Fast Data Server OPERATOR’S MANUAL”.

NOTE

DNC operation cannot be performed using a program in a USB memory.

DNC operation

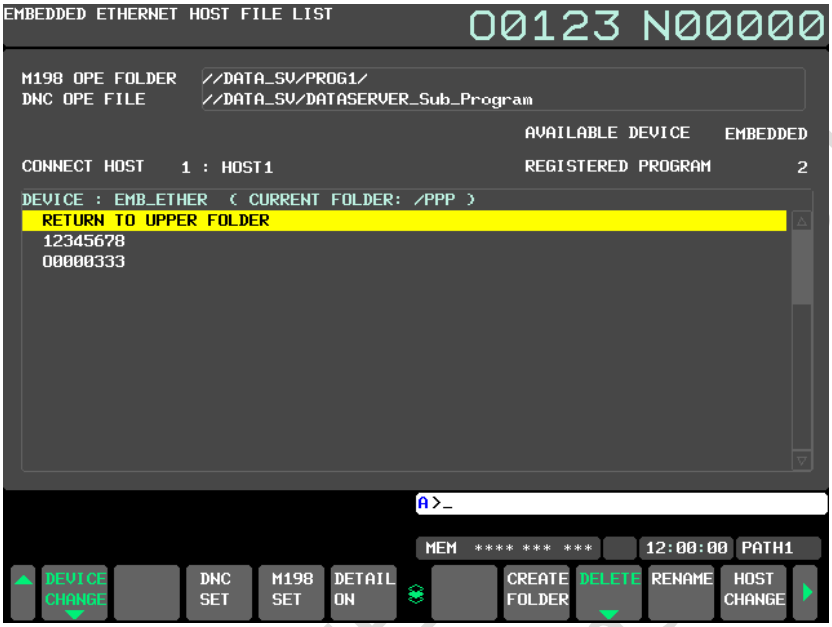
Procedure

1 Press the REMOTE switch on the machine operator's panel to enter the DNC mode.
2 Select the program to be executed.
The memory card (or floppy cassette):
- Selecting a DNC operation file
On the memory card (or floppy cassette) list screen, move the cursor to the file to be subjected to DNC operation and press "DNC SET" to select the file to be subjected to DNC operation. (The selected file is marked with "D".)
- Releasing a DNC operation file
On the memory card (or floppy cassette) list screen, move the cursor to the file for which DNC operation is released and press "DNC CLEAR" to release the DNC operation file. (The "D" mark for the file is removed.)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

The Embedded Ethernet:

- Selecting a DNC operation file
  On the Embedded Ethernet host file list screen, move the cursor to the file to be subjected to DNC operation and press "DNC SET" to select the file to be subjected to DNC operation.
- Releasing a DNC operation file
  On the Embedded Ethernet, DNC operation file is released, when the power is turned off.



3 Press the cycle start switch to execute the selected file. For details on the REMOTE switch, refer to the manual provided by the machine tool builder.

4 During DNC operation, executed programs are listed on the program check screen and program screen.

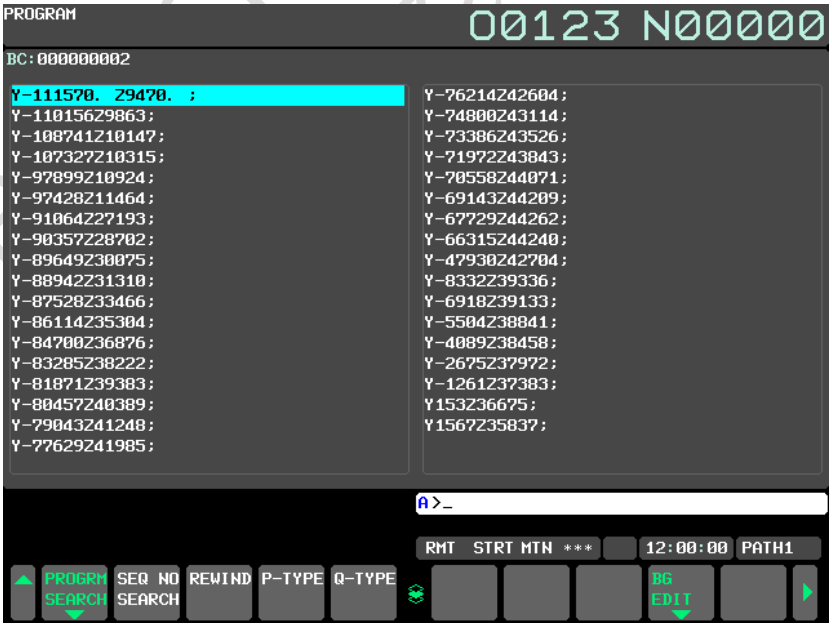


Fig. 4.3 (a) PROGRAM screen

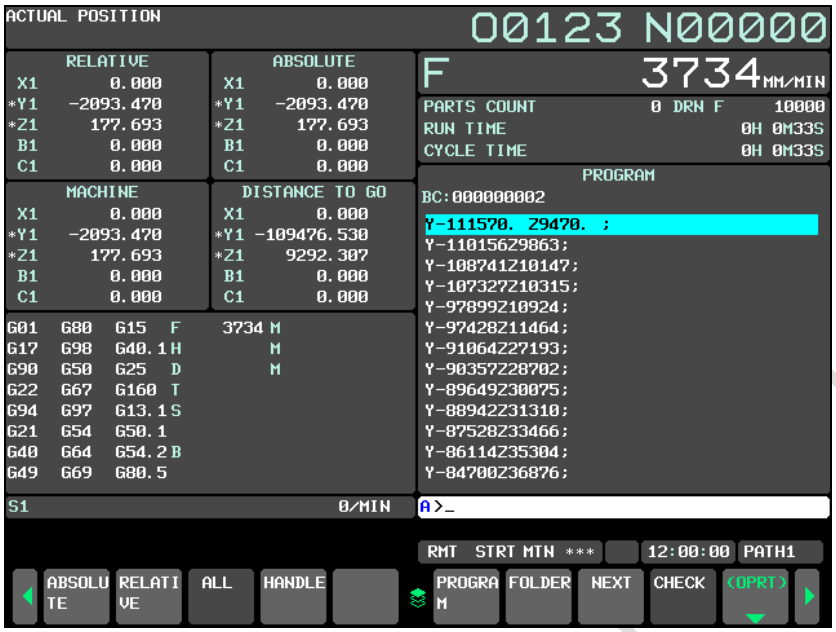


Fig. 4.3 (b) PROGRAM CHECK screen

NOTE

1 Before selecting a DNC operation file, be sure to release all schedule data. DNC operation and schedule operation cannot be specified at the same time.
2 A DNC operation file cannot be released during DNC operation.
3 To switch between devices when DNC settings are made, release the settings and then make the settings again.

Explanation

During DNC operation, subprograms and macro programs stored in memory can be called.

Limitation

- M198 (command for calling a program from within an external input/output unit)

In DNC operation, M198 cannot be executed. If M198 is executed, alarm PS0210, "CAN NOT COMMAND M198/M99" is issued.

- Custom macro

In DNC operation, custom macros can be specified, but no repeat instruction and branch instruction can be programmed. If a repeat instruction or branch instruction is executed, alarm PS0123, "ILLEGAL MODE FOR GOTO/WHILE/DO" is issued.

- M99

For returning from a subprogram or macro program to the calling program during DNC operation, the specification of a return command (M99P...) with a sequence number specified is not allowed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

4.4 SCHEDULE OPERATION

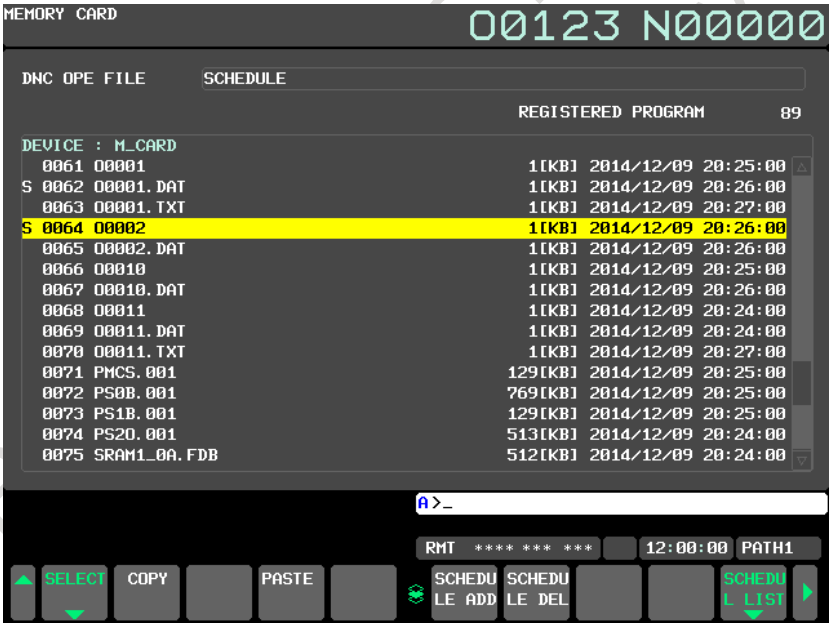
To perform schedule operation, select files (programs) registered in a memory card and specify the sequence of execution and the repetition count of each program.

NOTE
Schedule operation cannot be performed using a program in a USB memory.

Schedule operation

Procedure

1 Press the REMOTE switch on the machine operator's panel to enter the DNC mode.
2 Select the program to be subjected to schedule operation.
- Setting a schedule
  Move the cursor to the file to be subjected to schedule operation and press the soft key [SCHEDUL ADD]. The files registered as schedule data are marked with "S" on the left sides of their file names. The files are subjected to schedule operation in the order in which the files are selected. When settings are made this way, the repetition count of execution of each file is set to 1. To change the repetition count or execution order or check the schedule, press the soft key [SCHEDUL LIST] to display the schedule list screen.



[SCHEDUL ADD]
Adds the file at the cursor position as schedule data.

[SCHEDUL DEL]
Deletes the file at the cursor position from schedule data when the file is registered as schedule data.

[SCHEDUL LIST]
Lists the settings of schedule data to edit the repetition count or the like (see the next page for details).

- Editing a schedule
  To edit schedule data, press the soft key [SCHEDUL LIST] to display the schedule list screen, on which schedule data can be edited.

FRYER SYSTEMS INC

[FILE UP]	Moves the file at the cursor position up one line and moves the replaced file down one line.
[FILE DOWN]	Moves the file at the cursor position down one line and moves the replaced file up one line.
[DELETE]	Deletes the file at the cursor position and moves the files below the cursor up one line.
[INSERT]	Moves the files below the cursor down one line.
[ALL DELETE]	Deletes all records.

3 Press the cycle start switch to execute the selected files. For details on the REMOTE switch, refer to the manual provided by the machine tool builder.

NOTE

1 Before setting schedule operation, release DNC operation files in the MDI mode. DNC operation and schedule operation cannot be specified at the same time.
2 Before starting schedule operation, confirm that schedule data is set correctly on the schedule list screen.
3 Schedule data cannot be changed or edited during schedule operation. Before changing schedule data, make a reset to stop operation.
4 When M02 or M30 is not commanded at the end of the program that does the schedule operating, "%" is executed, and alarm PS5010, "END OF RECORD" is occurred.

Restrictions

- Repetition count

The maximum repetition count during schedule operation is 9999. When a negative value is specified, an endless loop (LOOP display) is assumed. The file for which 0 is set is skipped and processing proceeds to the next file.

- Number of registered files

The maximum number of programs that can be registered as schedule setting data is 20.

FRIEF MACHINE SYSTEMS INC
Rob Zahensky

- Selectable files

The files to be selected as schedule setting data must be registered in the same folder. (Files in the different folders cannot be selected.)

- M code

Even if a code other than M02 and M30 in the execution program is executed, the current count on the schedule execution status screen is not increased.

- Floppy disk folder display during execution of a file

During schedule operation, folders in a floppy disk cannot be displayed in a background edit.

- Intervention during automatic operation

Intervention in schedule operation cannot be performed during automatic operation.

- During multi-path control

The scheduling function cannot be used by multiple paths at the same time.

4.5 EXTERNAL SUBPROGRAM CALL (M198)

During memory operation, you can call and execute a subprogram registered in an external device (such as a Memory Card, Handy File, or Data Server) connected to the CNC.

NOTE
A program in a USB memory cannot be called using an external subprogram call (M198).

Format

M198 Pxxxxxxxx Lyyyyyyyy ;

↑ ↑

Pxxxxxxxx : Program number (or file number)
Lyyyyyyyy : Number of repetitive calls

- **If calling with a file name**

M198 <xxxx> Lyyyyyyyy ;

↑ ↑

xxxx : File name of an external subprogram
Lyyyyyyyy : Number of repetitive calls

When address L is omitted, the number of repetitive calls is assumed to be 1.

FS16 compatible command format
(The following command format is valid only when a 4-digit program number is used.)

M198 Pxxxx yyyy ;

↑ ↑

xxxx : Number of repetitive calls
yyyy : Program number (or file number)
When the number of repetitive calls is omitted, it is assumed to be 1.

FRYER MACHINE SYSTEMS INC Rob Zaherisky

Explanation

M code M198 specifies an external subprogram call. You can also call an external subprogram using an M code set in parameter No. 6030. (When an M code other than M198 is set as an M code for calling an external subprogram, M198 is executed as a normal M code.)

Specify a program number (file number) registered in an external device at address P. If the specified program number (file number) is not registered in the connected external device, an alarm PS1079, "PROGRAM FILE NOT FOUND" is issued.

If calling with a file name, the file name of an external subprogram must be specified immediately after M198.
If, in an external subprogram call specified with a file name, no program is registered in the connected external device, alarm SR1079 (alarm SR1966, "FILE NOT FOUND (MEMORY CARD)" in the case of a memory card) is issued.

Characters that can be used in the file name of an external subprogram and the number of characters in a file name are as described below.

- The following characters can be used in a file name:
  Alphabetical characters (uppercase and lowercase letters), numeric characters, and the four symbols below:
  - + _ .
  Because "." and ".." are reserved file names, they cannot be used.
- The maximum number of characters that can be used in a file name in each external device, as well as the alarms that are issued if an item out of the range (such as a character that cannot be specified) is specified, is as given below.

External device name	Number of characters	Alarm issued if an item out of the range is specified
Handy File	17	SR1079
FLOPPY CASSTTE	17	SR1079
Memory Card	12	SR1968
Data Server	32	PS0311

Example 1)

If using a Data Server as the external device, specify the command for calling the external subprogram located on the Data Server, as follows:

M198 <SMPL100**> L**10; Calls SAMPL100 ten times.

File name on the Data Server: SMPL100

```
%
<SMPL100>（Sample Program1）;
N1 G55;
N2 G90 G01 X0 Y0 Z30.0 F1000;
N3 G01 X0 Y0 Z0 F1000;
N4 ……;
```

FRYER MACHINE SYSTEMS INC
Rob Zahonsky

Example 2)

If using a memory card as the external device, specify the command for calling the external subprogram located on the memory card, as follows:

M198 <SMPL200**> L**20; Calls SAMPL200 twenty times.

File name on the memory card: SMPL200

```
%
O0200（Sample Program2）;
N1 G55;
N2 G90 G01 X0 Y0 Z30.0 F1000;
N3 G01 X0 Y0 Z0 F1000;
N4 ……;
```

NOTE

1 If the file name of an external subprogram differs from its program name, as in the example below, specify the file name (SMPL200) on the Memory Card.

2 If an MS-DOS format floppy disk is used with a Handy File, a file name can be specified as follows:

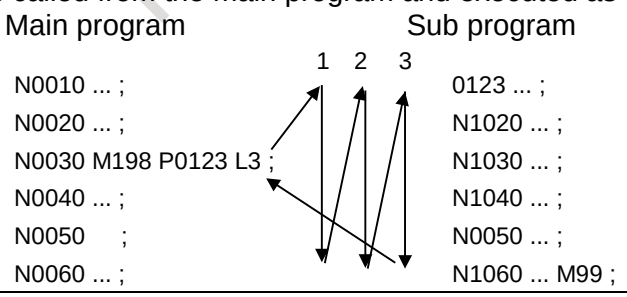
- A file name of up to 12 characters, 8 characters + "." + 3characters, can be specified. For an external subprogram call, there are cases in which no alarm is issued even if a call is made with a specification exceeding 12 characters, as in the following example.
  Example) M198 <ABCDEFGH>
  For this command, the file with the file name, "ABCDEFGH.DAT", is the one to be called.
  Even if <ABCDEFGHIJKLMN> is specified as a file name, the file with the first 8 characters + "DAT", i.e., "ABCDEFGH.DAT", is the one to be called.

Example 3)

M198 P0123 L3;

This command specifies that the subprogram having external subprogram number O0123 is to be called three times repeatedly.

The subprogram is called from the main program and executed as follows:



- **Program number call**
- **File name call**

You can also specify a subprogram call with its program number instead of the file number by the setting of bit 2 (SBP) of parameter No. 3404.

FRYER MACHINE SYSTEMS INC Rob Zuhensky

- Program number O8-digit

The called file name is as follows.

Table 4.5 (a) The file name called when instructing in M198

Device	Program number is 7-digit or less. (example: M198P1234567)	Program number is 8-digit. (example: M198P12345678)
When an external device is excluding a fast data server, or is a fast data server excluding in the storage mode.	“O” plus a 7-digit numeric (example: O1234567)	a 8-digit numeric (example: 12345678)
When an external device is a fast data server in the storage mode.	“O” plus a 8-digit numeric (example: O01234567)	a 8-digit numeric (example: 12345678)

- Input/Output file name for multi-path system

When bit 0 (MDP) of parameter No.138 is set to 1, the file name having extension with the path number can be called.
The extension of each path is as follows.

Table 4.5 (b) Extension of subprogram

Path number	Extension
1	Nothing
2	P-2
3	P-3
4	P-4
5	P-5
6	P-6
7	P-7
8	P-8
9	P-9
10	P10

Example) When "M198 P0123" is executed, the following file name is called.

Table 4.5 (c) File name called by M198

Path number	File name	
	Parameter MDP=0	Parameter MDP=1
1	O0123	O0123
2	O0123	O0123.P-2
3	O0123	O0123.P-3
4	O0123	O0123.P-4
5	O0123	O0123.P-5
6	O0123	O0123.P-6
7	O0123	O0123.P-7
8	O0123	O0123.P-8
9	O0123	O0123.P-9
10	O0123	O0123.P10

FRYER MACHINE SYSTEMS INC
Rob Zahenskij

NOTE

1 An external subprogram call can be specified during program operation in the MEM/MDI mode. However, in case of using with MDI mode, it is necessary to set the parameter bit 1 (MDE) of parameter No.11630 to 1.

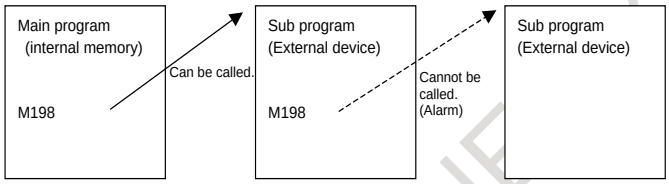
2 An external subprogram call is available for the following external devices:

External device name	Program number call	File number call	File name call
Handy File	Available	Available	Available
FLOPPY CASSTTE	Available	Available	Available
Memory Card	Available	Unavailable	Available
Data Server	Available	Unavailable	Available

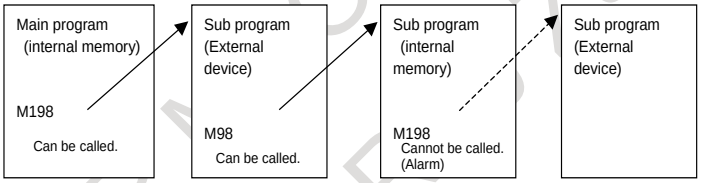
3 To perform a subprogram call using a Memory Card as the external device, set bit 7 (MNC) of parameter No. 138 to 1 and I/O channel (parameter No. 0020) to 4.
A program number call is always enabled regardless of the setting of bit 2 (SBP) of parameter No. 3404.

4 To perform a file name call using a Handy File or floppy cassette as the external device, set bit 2 (SBP) of parameter No. 3404 to 1.

5 called using another external device subprogram call. (An alarm PS1080, “DUPLICATE DEVICE SUB PROGRAM CALL” is issued.)



6 A subprogram registered in internal memory can be called from a subprogram called using an external device subprogram call. From the called subprogram in internal memory, another external device subprogram call cannot be performed. (An alarm (PS1080) is issued.)



7 A call using the external device subprogram call function is counted as one level of subprogram nesting.

8 In a multi-path system, an external device subprogram call cannot be performed simultaneously from multiple paths.
When external subprogram calls using the Data Server that are available in multi-path systems are used, however, external subprogram call commands using the Data Server can be specified simultaneously from multiple paths.

9 In an external subprogram call operation, custom macros can be specified, but repeat instruction and branch instruction cannot be programmed.

FRYER MACHINE SYSTEMS INC
F. Zahensky

4.6 EXTERNAL SUBPROGRAM CALLS USING THE DATA SERVER AVAILABLE IN MULTI-PATH SYSTEMS

Overview

In a multi-path system, external subprogram call commands using the Data Server can be specified simultaneously from multiple paths.

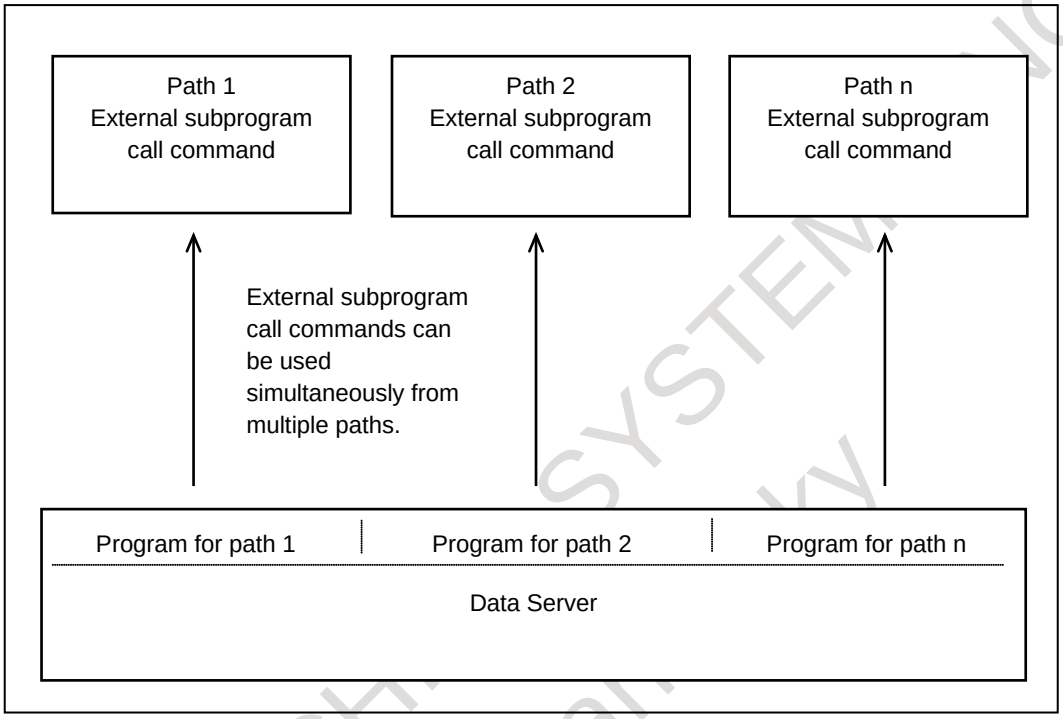


Fig. 4.6 (a) External subprogram calls specified simultaneously from multiple paths

Explanation

In a multi-path system, external subprogram call commands using the Data Server can be specified simultaneously from multiple paths.

・Example) Specifying an external subprogram call simultaneously from two paths

During the execution of an external subprogram command (M198) in path 1, another external subprogram call command (M198) can be executed from path 2, as shown below.

FRYER MACHINE SYSTEMS INC
Rob Zan...ky

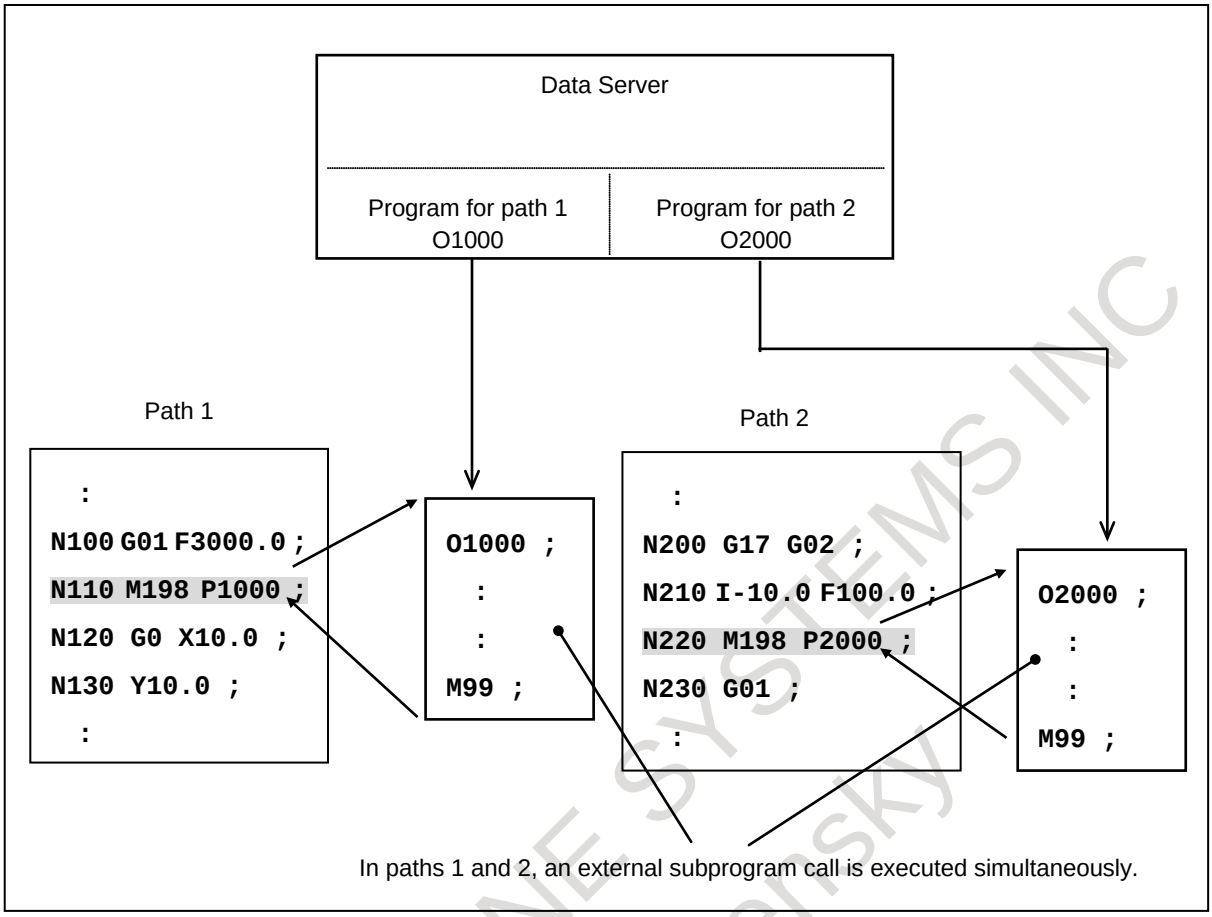


Fig. 4.6 (b) Specifying an external subprogram call in paths 1 and 2 simultaneously

Note

NOTE

1 This function is enabled only when the Data Server is selected as a foreground input device.
Other devices such as a memory card cannot be used to execute an external subprogram call command simultaneously in multiple paths.

2 DNC operation cannot be performed simultaneously in multiple paths.

3 The same external subprogram cannot be executed simultaneously in multiple paths.

4 Before specifying an external subprogram command, set a folder for performing an external subprogram call. For details, refer to the following manual:
”Fast Ethernet / Fast Data Server OPERATOR’S MANUAL (B-64014EN)”

5 To use this function, set bit 0 (ESB) of parameter No. 8107 to 1.

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

4.7 MANUAL HANDLE INTERRUPTION

By rotating the manual pulse generator in the automatic operation mode (manual data input, DNC operation, or memory operation) or in the memory editing mode, handle feed can be superimposed on movement by automatic operation. A handle interruption axis is selected using the manual handle interruption axis selection signals HSnIA to HSnIE.
The minimum unit of travel distance per scale division is the least input increment. One of four types of magnifiers selected with manual handle feed selection signals MP1 and MP2 can be applied. With bit 3 (HIT) of parameter No. 7103, the minimum unit of travel distance can be further 10 times greater.

A handle feed magnifier is selected using the manual handle feed amount selection signal. (See "MANUAL HANDLE FEED".)

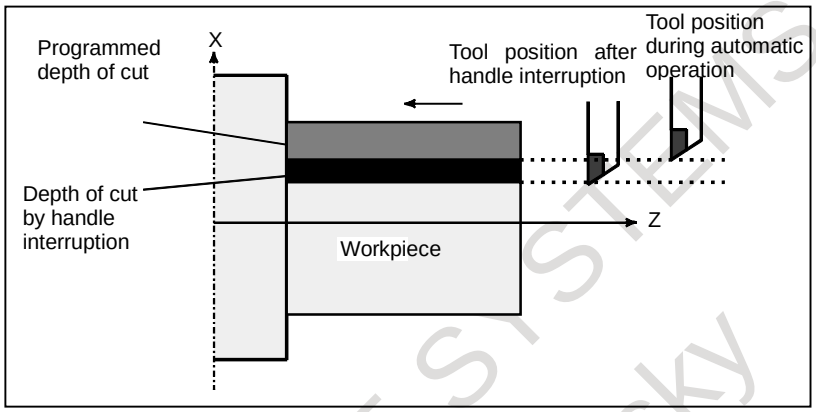


Fig. 4.7 Manual handle interruption

> ⚠ **WARNING**
> The travel distance per scale division by manual handle interruption is the least input increment as with manual handle feed. With a machine of metric input and inch output, for example, the travel distance per 254 scale divisions is 0.01 inch. With a machine of inch input and metric output, the travel distance per 100 scale divisions is 0.254 mm.

Explanation

- Interruption operation

1 When the handle interruption axis selection signal for a handle interruption axis is set to “1” in the automatic operation mode (manual data input, DNC operation, or memory operation) or in the memory editing mode, manual handle interruption can be performed by rotating the handle of the manual pulse generator.

> **NOTE**
> Even when the feedrate override signal sets 0%, manual handle interruption can be accepted.

2 For the method of selecting a manual handle interruption axis, refer to the relevant manual of the machine tool builder.

3 The feedrate during manual handle interruption is the sum of feedrate used for automatic operation and the feedrate used for movement by manual handle interruption. However, the feedrate during manual handle interruption is controlled so that it does not exceed the maximum allowable cutting feedrate for the axis.

FRYER MACHINE SYSTEMS INC
Rob Zakrensky

Example

Suppose that the maximum allowable cutting feedrate for an axis is 5 m/min, and that a movement is made in the + direction at 2 m/min along the axis. In this case, manual handle interruption can be accepted even when the manual pulse generator is rotated up to a speed equivalent to 3 m/min. Manual handle interruption by rotation in one direction can be accepted even when the manual pulse generator is rotated to a speed equivalent to 7 m/min.
If the manual pulse generator is rotated to a speed beyond the upper limits, those pulses from the manual pulse generator that correspond to the excess are lost, resulting in a mismatch between the scale mark of the manual pulse generator and the actually interrupted travel distance.

4 For a magnifier for manual handle interruption, refer to the relevant manual of the machine tool builder.

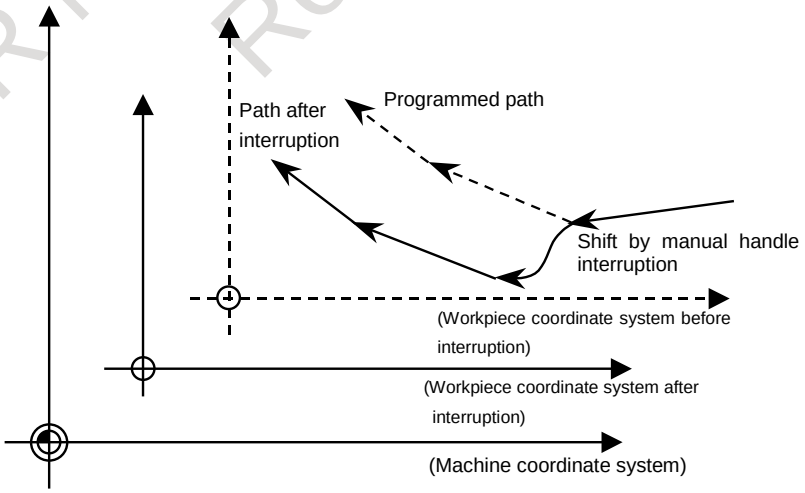
5 If the travel direction is reversed as a result of manual handle interruption, backlash compensation is performed. Pitch error compensation is performed for the position after interruption.

6 In manual handle interruption, only acceleration/deceleration for cutting feed is enabled. By setting bit 0 (MNJ) of parameter No. 1606 to 1, acceleration/deceleration for both of cutting feed and jog feed can be applied to manual handle interruption.

- Manual handle interruption and coordinate system

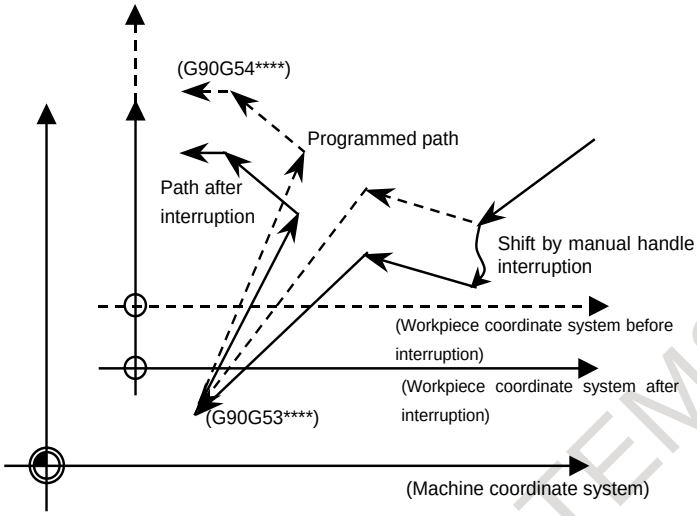
1 The amount of manual handle interruption shifts the workpiece coordinate systems and the local coordinate system. So, the machine moves, but the coordinates in the workpiece coordinate systems and the local coordinate system remain unchanged.
Regardless of which coordinate system is selected, all workpiece coordinate systems and the local coordinate system shift by the same amount.

- Absolute coordinates
  → Remain unchanged by handle interruption.
- Relative coordinates
  → Change by the amount of handle interruption.
- Machine coordinates
  → Change by the amount of handle interruption.



FRYER MACHINE SYSTEMS INC
Rob Zahensky

2 Even when manual handle interruption is performed, the machine coordinate system remains unchanged. The absolute command (G53) in the machine coordinate system is not affected by manual handle interruption.

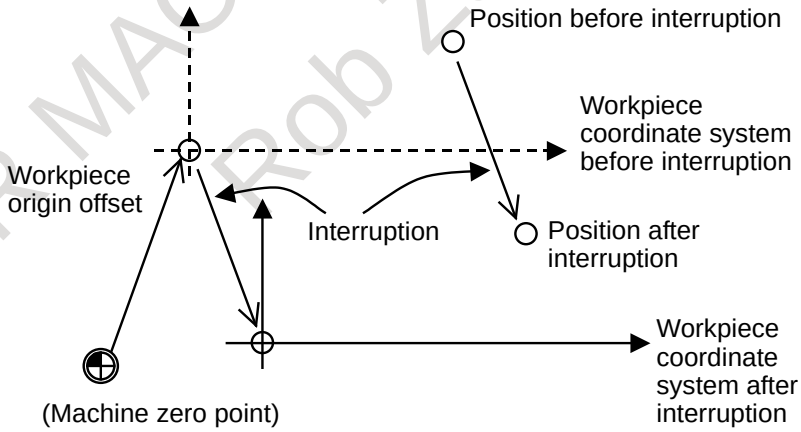


3 In automatic reference position return (G28), the end point (reference position) is not affected by manual handle interruption. However, the midpoint is in the workpiece coordinate system, so that the position shifted by the amount of interruption becomes the midpoint.

- Cancellation of the amount of interruption

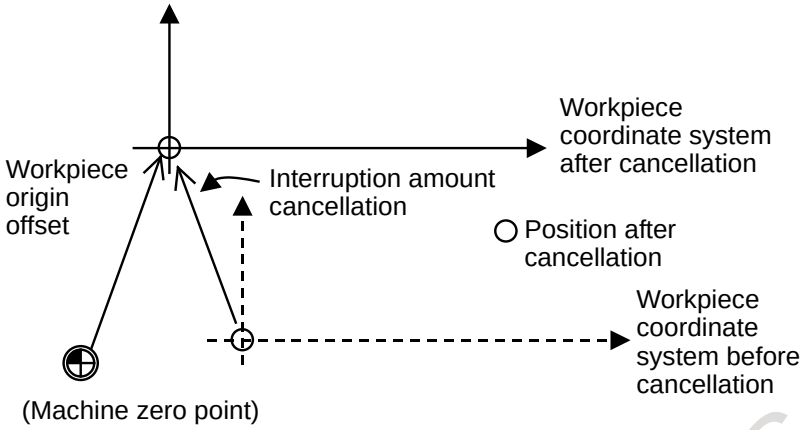
Operation by which the workpiece coordinate system shifted by manual handle interruption from the machine coordinate system is returned to the original workpiece coordinate system is referred to cancellation of the amount of interruption.
When the amount of interruption is canceled, the workpiece coordinate system is shifted by the amount of manual handle interruption, and the amount of interruption is reflected in the absolute coordinates.

Interruption shifts the workpiece coordinate system from the machine coordinate system.



By cancellation, the workpiece coordinate system returns to the state present before handle interruption.

FRYER MACHINE SYSTEMS INC
Rob Zahorsky



In the following cases, the amount of interruption is canceled:

- When a reset is made (when bit 1 (RTH) of parameter No. 7103 is set to 1)
- When emergency stop state is canceled (when bit 1 (RTH) of parameter No. 7103 is set to 1)
- When a manual reference position return operation is performed (when G28 is specified before a reference position is established)
- When a reference position is set without dogs
- When the workpiece coordinate system is preset

NOTE

When the amount of interruption is cleared using soft keys, only the indication of the amount of interruption becomes 0, and the workpiece coordinate system remains unchanged.

- Clearing the amount of interruption by soft keys

Clearing of the amount of interruption means that the indication of the amount of interruption by manual handle interruption is set to 0. The workpiece coordinate system does not change.
"Clearing all axes" or "Clearing any axis" is performed on the path for which the amount of manual handle interruption is indicated.
When bit 3 (HLC) of parameter No. 7100 is enabled, soft key [INTRPT CANCEL], which is used for this operation, appears. When HLC is disabled, soft key [INTRPT CANCEL] does not appear.

To select "Clearing all axes" or "Clearing any axis", follow the procedure below.

1 Press function key on the MDI unit.

2 Press soft key [HANDLE].

3 Press soft key [(OPRT)].

4 To prepare for "Clearing all axes" or "Clearing any axis", press soft key [INTRPT CANCEL].

To prepare for "Clearing all axes" or "Clearing any axis", perform one of the following.

- Clearing all axes
  Press soft key [INTRPT CANCEL] and then press soft key [ALL AXIS].

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Clearing any axis
  - Enter the axis name and then press soft key [INTRPT CANCEL].
  - Press soft key [INTRPT CANCEL], enter the axis name, and press soft key [EXEC].

If an incorrect axis name is entered, a warning message stating "FORMAT ERROR" appears.

- Relation with other functions

The following table indicates the relation between other functions and the movement by handle interruption.

Table 4.7 (a) Relation between other functions and the movement by handle interruption

Signals	Relation
Machine lock	Machine lock is effective. When machine lock is on, no movement is made due to handle interruption.
Interlock	Interlock is effective. When interlock is on, no movement is made due to handle interruption.
Mirror image	Mirror image is not effective. Interrupt functions on the plus direction by plus direction command, even if this signal turns on.

- Position display

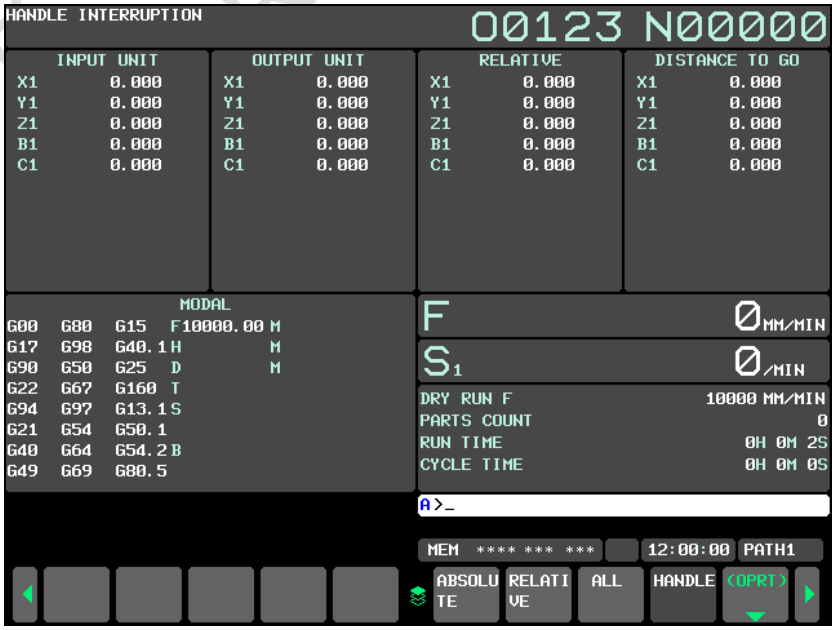
The following table shows the relation between various position display data and the movement by handle interruption.

Table 4.7 (b) relation between various position display data and the movement by handle interruption

Signals	Relation
Absolute coordinate value	Handle interruption does not change absolute coordinates.
Relative coordinate value	Relative coordinates are changed by the travel distance specified by handle interruption.
Machine coordinate value	Machine coordinates are changed by the travel distance specified by handle interruption.

- Travel distance display

Press the function key POS, then press the chapter selection soft key [HNDL]. The move amount by the handle interruption is displayed. The following 4 kinds of data are displayed concurrently.



FRYER MACHINE SYSTEMS INC
Rob Zaharsky

(a) INPUT UNIT:
Handle interruption move amount in input unit system
Indicates the travel distance specified by handle interruption according to the least input increment.

(b) OUTPUT UNIT :
Handle interruption move amount in output unit system
Indicates the travel distance specified by handle interruption according to the least command increment.

(c) RELATIVE:
Position in relative coordinate system
Indicates the present relative position where the amount of the handle interruption was included.

(d) DISTANCE TO GO:
The remaining travel distance in the current block has no effect on the travel distance specified by handle interruption.

The handle interruption move amount is cleared when the manual reference position return ends every axis.

- Display for five-axis systems or better

Systems having five or more axes provide the same display as the overall position display.

Note

NOTE

1 In a manual operation mode such as the jog feed mode, manual handle feed mode, or TEACH IN HANDLE mode, handle interruption cannot be performed.
2 During a machine lock or interlock, handle interruption does not cause movement.
3 Manual handle interruption is disabled for the axis in any of the following states.
- Follow-up state
- PMC axis control state
4 Manual handle interruption cannot be performed for the axis specified in the G00 mode. But manual handle interrupt is enabled within the rapid traverse rate of the specified axis by setting bit 1 (RMI) of parameter No.10480.

4.7.1 Manual Interruption of 3-dimensional Coordinate System Conversion

Overview

When the manual pulse generator is rotated in the 3-dimensional coordinate conversion mode, the travel distance specified by the manual pulse generator is superposed on the travel distance by automatic operation in the direction of the selected handle feed axis on the coordinate (program coordinate) system after 3-dimensional coordinate conversion.

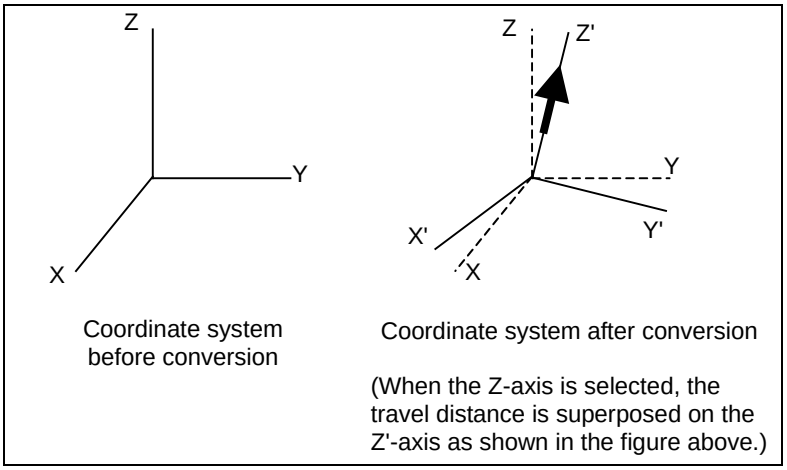


Fig. 4.7.1 Manual interruption of 3-dimensional coordinate system conversion

FRYER MACHINE SYSTEMS INC Rob Zaremsky

Explanation

- Interruption

This function performs interruption only when all of conditions 1 to 6 below are met. Otherwise, interruption does not occur.

1. During the automatic operation mode (MEM mode, MDI mode, or RMT mode)
2. During the 3-dimensional coordinate conversion mode
3. During the cutting mode (G01, G02, G03, or other G code for cutting is being executed.)
4. When interrupt is done to three axes of the target as 3-dimensional coordinate conversion
5. When 3-dimensional coordinate conversion manual interrupt enable/disable switch signal is set to “0”
6. When the tool axis direction, the tool axis right-angle direction, or the tool tip center rotation is not selected for the 3-dimensional manual feed

The feedrate superposed along the 3-dimensional coordinate conversion mode does not exceed the maximum cutting speed of each axis.

- Manual interruption and coordinate system

When this function operates, the absolute coordinate values do not include the travel distance by manual interruption. The absolute coordinate values are therefore not updated even when the manual pulse generator is rotated.
The machine coordinate values and relative coordinate values include the travel distance by manual interruption. However, the values are represented on the coordinate system before being converted. As shown in Fig. 4.7.1, the travel distance of the manual handle interruption is superposed along the Z'-axis. The travel distance is displayed using the X, Y, and Z coordinates of the coordinate system before being converted.
The travel distance by this function can be monitored on the manual handle interruption screen. Bit 3 (DMK) of parameter No. 5402 specifies whether the coordinate system (X, Y, Z) before being converted or the coordinate system (X', Y', Z') or after being converted is used to display the amount of interruption by the manual handle.

Note

Handle interruption is disabled during execution of a G68 or G69 block.

4.8 MIRROR IMAGE

During automatic operation, the mirror image function can be used for movement along an axis. In this case, the absolute coordinate is updated to the same direction as a programmed path, and the machine coordinate and the relative coordinate are updated to the opposite direction from a programmed path.
To use this function, set the mirror image switch to ON on the machine operator's panel, or set the mirror image setting to ON from the MDI unit.

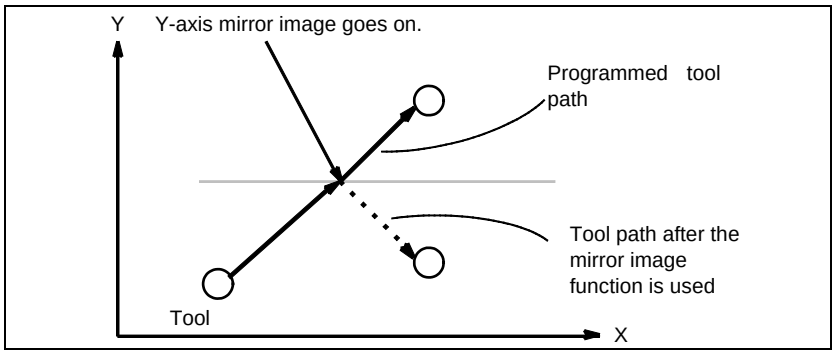


Fig. 4.8 (a) Mirror image

FRYER MACHINE SYSTEMS INC
Rob Zaborsky

Procedure for mirror image

Procedure

The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

1 Press the single block switch to stop automatic operation.
When the mirror image function is used from the beginning of operation, this step is omitted.

2 Press the mirror image switch for the target axis on the machine operator's panel.
Alternatively, turn on the mirror image setting by following the steps below:

2-1 Set the MDI mode.

2-2 Press the function key .

2-3 Press the [SETING] soft key for chapter selection to display the setting screen.

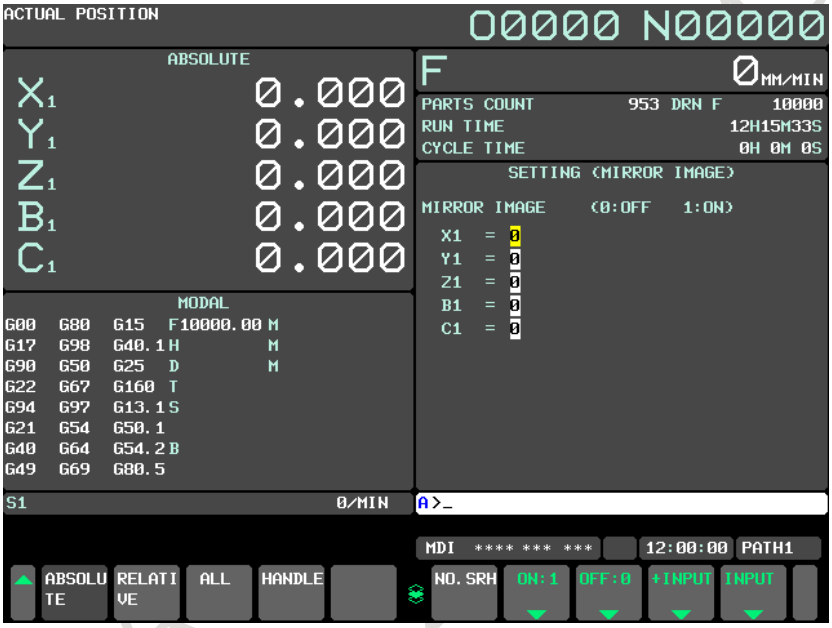


Fig. 4.8 (b) Setting screen

2-4 Move the cursor to the mirror image setting position, then set the target axis to 1.

3 Enter an automatic operation mode (MEM mode or MDI mode), then press the cycle start button to start automatic operation.

Explanation

- The mirror image function can also be turned on and off by setting bit 0 (MIRx) of parameter No.0012 to 1 or 0.
- For the mirror image switches, refer to the manual supplied by the machine tool builder.

- Shifting workpiece coordinate system

When operation using mirror image is executed after the workpiece coordinate system is specified, the workpiece coordinate system is shifted from the machine coordinate system.
This shifted amount can be cancelled by the following operation.

- Manual reference position return
- Workpiece coordinate system preset (G92.1, G50.3)
- Each axis workpiece coordinate system preset signals WPRST1-WPRST8

- Programmable mirror image (G50.1, G51.1)

When programmable mirror image (machining center system) and ordinary mirror image are specified at the same time, programmable mirror image is applied first.

- **Retrace**

Mirror image is enabled during reverse execution and forward reexecution of the retrace function. Therefore, it is necessary during the reverse execution and the forward reexecution to make mirror image in the same state as the forward execution.

Limitation

- **Manual operation and automatic reference position return**

Even if mirror image is applied, the following directions are not reversed.

- Direction of manual operation
- Direction of movement, from the intermediate position to the reference position during automatic reference position return

In these cases, the machine coordinate and the relative coordinate are updated to the same direction as a movement, and the absolute coordinate is updated to the opposite direction from a movement.

M

- **Single direction positioning**

Mirror image is not applied in parameter No.5440 set direction. Even in the mirror image mode, the direction of single direction positioning remains unchanged.

If positioning of linear interpolation type is used, and the state of mirror image when a single direction positioning block is looked ahead differs from the state of mirror image when the execution of the block is started, an alarm is issued. When switching mirror image in the middle of a program, disable looking ahead by specifying a non-buffering M code. Then, switch mirror image when there is no look-ahead block.

- **Boring cycle (G76 and G87)**

Even if mirror image is applied, shift direction for boring cycles (G76 and G87) is not reversed.

- **High precision oscillation function**

Never attempt to apply mirror image about the oscillation axis.

- **Manual linear/circular interpolation**

Mirror image is not available during the manual operation.

- **Program restart, Quick program restart**

The tool cannot be returned to a correct position when mirror image is used. However, P type return is possible for a block that switched between ON and OFF most recently or a subsequent block. In this case, mirror image signal status present when the program was interrupted must be maintained.

- **Manual intervention and return**

When performing manual intervention and return, never use mirror image.

- **Tool retract and recover**

When retracting the tool manually in the tool withdrawal mode, do not use mirror image.

- **Arbitrary speed threading**

Thread cannot be re-machined with mirror image applied.

- **General purpose retract**

The retract direction is the movement direction of the machine regardless of whether mirror image is valid or not. (A mirror image is not applied to the updating of absolute coordinates.)

FRYER MACHINE SYSTEMS INC
Rob Zahonsky

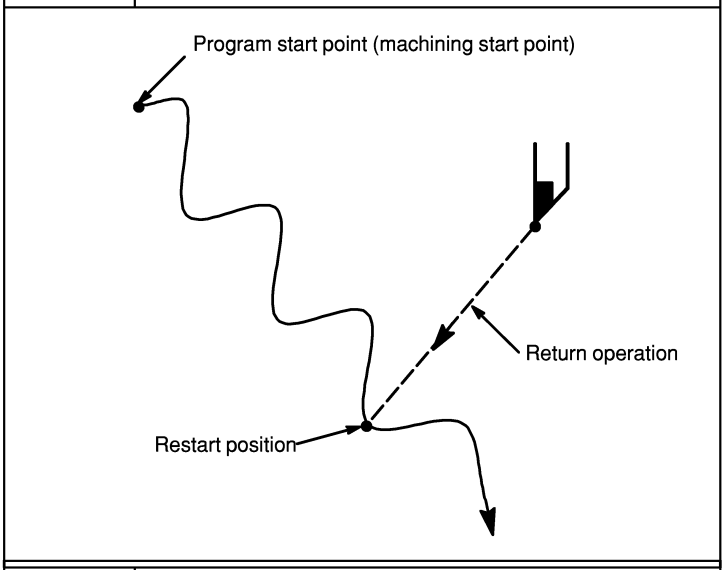
- **AI contour control**

In the speed control with the cutting feed, the travel direction on the Z-axis is determined with the appropriate NC command. Therefore, if mirror image is applied on the Z-axis, the direction on the Z-axis cannot be determined. When using the speed control with the cutting load, do not use mirror image.

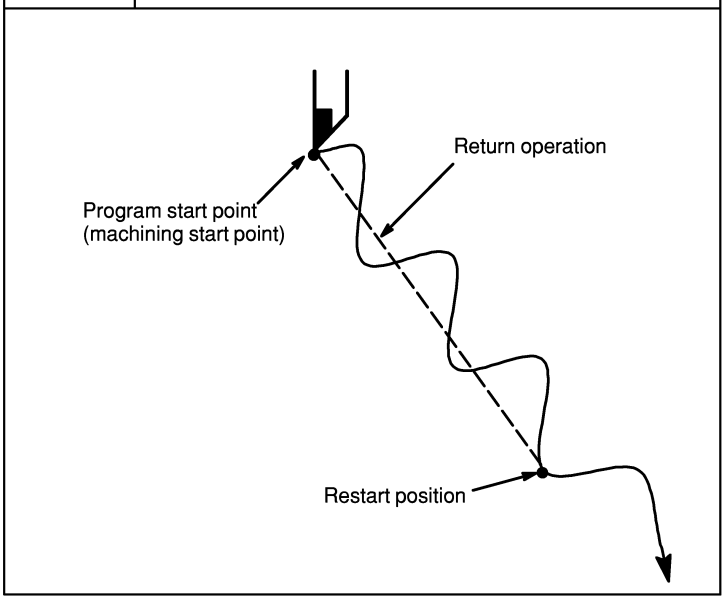
4.9 PROGRAM RESTART

This function specifies Sequence No. of a block to be restarted when a tool is broken down or when it is desired to restart machining operation after a day off, and restarts the machining operation from that block. It can also be used as a high-speed program check function.
There are two restart methods: the P-type method and Q-type method.

P TYPE	Operation can be restarted anywhere. This restart method is used when operation is stopped because of a broken tool.



Q TYPE	Before operation can be restarted, the machine must be moved to the programmed start point (machining start point)



FRYER INDUSTRIES INC.

Procedure for program restart by specifying a sequence number

Procedure 1

[P TYPE]

1 Retract the tool and replace it with a new one. When necessary, change the offset. (Go to procedure 2.)

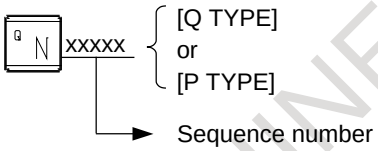
[Q TYPE]

1 When power is turned ON or emergency stop is released, perform all necessary operations at that time, including the reference position return.
2 Move the machine manually to the program starting point (machining start point), and keep the modal data and coordinate system in the same conditions as at the machining start.
3 If necessary, modify the offset amount. (Go to procedure 2.)

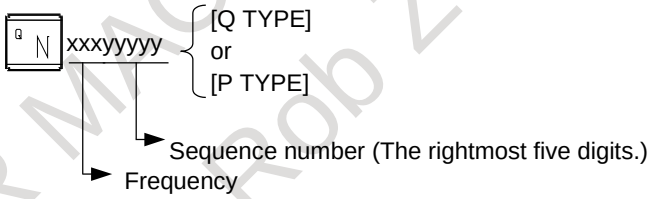
Procedure 2

[COMMON TO P TYPE / Q TYPE]

1 Turn the program restart switch on the machine operator's panel ON.
2 Press function key PROG to display the desired program.
3 Find the program head. Press RESET.
4 Enter the sequence number of the block to be restarted, then press the soft key [P TYPE] or [Q TYPE].



If the same sequence number appears more than once, the location of the target block must be specified. Specify a frequency and a sequence number.



FRYER MACHINE SYSTEMS INC
Rob Zahensky

5 The sequence number is searched for, and the program restart screen appears on the LCD display.

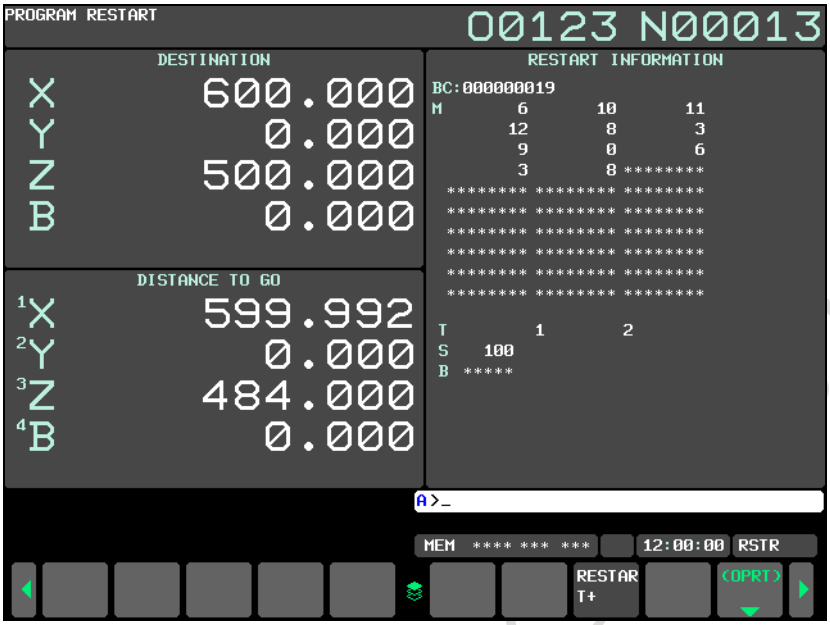


Fig. 4.9 (a) Program restart screen

DESTINATION shows the position at which machining is to restart.
DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart. A number to the left of each axis name indicates the order of axes (determined by parameter setting) along which the tool moves to the restart position.
The coordinates and amount of travel for restarting the program can be displayed for up to four axes. If your system supports five or more axes, pressing the [RSTR] soft key again displays the data for the fifth and subsequent axes.

M : Up to 30 most recently specified M codes. The maximum number of displayed M codes differs depending on the size of the display.
With 10.4"/15"/19" LCD/MDI unit : Up to 30 M codes

T : Two most recently specified T codes

S : Most recently specified S code

B : Most recently specified B code

Codes are displayed in the order in which they are specified. All codes are cleared by a program restart command or cycle start in the reset state.

6 Turn the program re-start switch OFF. At this time, the figure at the left side of axis name DISTANCE TO GO blinks.

7 Check the screen for the M, S, T, and B codes to be executed. If they are found, enter the MDI mode, then execute the M, S, T, and B functions. After execution, restore the previous mode. These codes are not displayed on the program restart screen.

8 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If such a possibility exists, move the tool manually to a position from which the tool can move to the machining restart position without encountering any obstacles.

9 Press the cycle start button. The tool moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified by parameter No. 7310 settings. Machining is then restarted.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Procedure for program restart by specifying a block Number

Procedure 1

[P TYPE]

1 Retract the tool and replace it with a new one. When necessary, change the offset. (Go to procedure 2.)

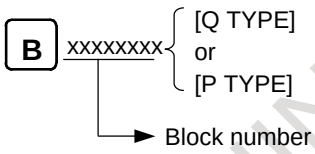
[Q TYPE]

1 When power is turned ON or emergency stop is released, perform all necessary operations at that time, including the reference position return.
2 Move the machine manually to the program starting point (machining start point), and keep the modal data and coordinate system in the same conditions as at the machining start.
3 If necessary, modify the offset amount. (Go to procedure 2.)

Procedure 2

[COMMON TO P TYPE / Q TYPE]

1 Turn the program restart switch on the machine operator's panel ON.
2 Press function key [PROG] to display the desired program.
3 Find the program head. Press [RESET].
4 Enter the number of the block to be restarted then press the soft key [P TYPE] or [Q TYPE]. The block number cannot exceed eight digits.



5 The block number is searched for, and the program restart screen appears on the LCD display.

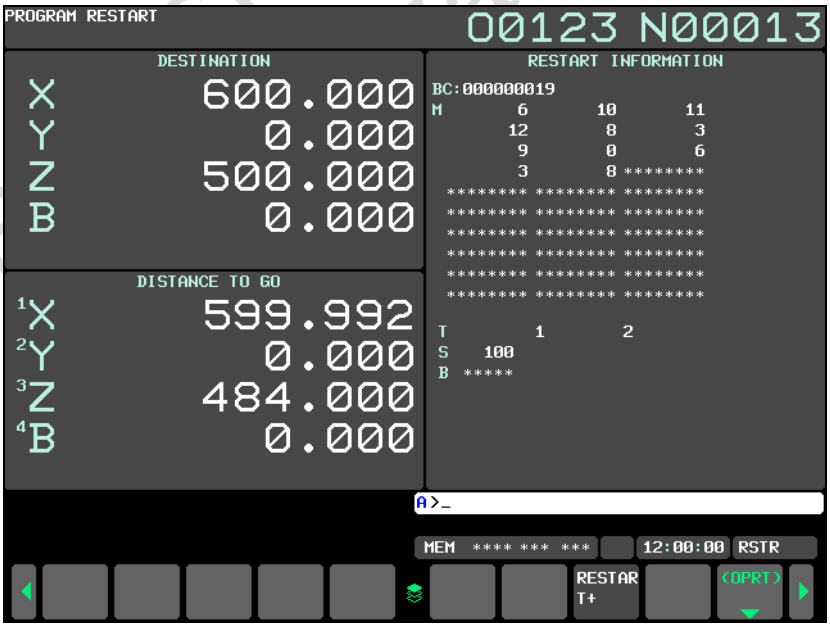


Fig. 4.9 (b) Program restart screen

DESTINATION shows the position at which machining is to restart.
DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart. A number to the left of each axis name indicates the order of axes (determined by parameter setting) along which the tool moves to the restart position.

FRYER MACHINE SYSTEMS INC
Kahensky

The coordinates and amount of travel for restarting the program can be displayed for up to four axes. If your system supports five or more axes, pressing the [RSTR] soft key again displays the data for the fifth and subsequent axes.

M : Up to 30 most recently specified M codes. The maximum number of displayed M codes differs depending on the size of the display.
With 10.4-/15-/19-inch LCD/MDI unit : Up to 30 M codes

T : Two most recently specified T codes

S : Most recently specified S code

B : Most recently specified B code

Codes are displayed in the order in which they are specified. All codes are cleared by a program restart command or cycle start in the reset state.

6 Turn the program re-start switch OFF. At this time, the figure at the left side of axis name DISTANCE TO GO blinks.

7 Check the screen for the M, S, T, and B codes to be executed. If they are found, enter the MDI mode, then execute the M, S, T, and B functions. After execution, restore the previous mode. These codes are not displayed on the program restart screen.

8 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If such a possibility exists, move the tool manually to a position from which the tool can move to the machining restart position without encountering any obstacles.

9 Press the cycle start button. The tool moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified by parameter No. 7310 settings. Machining is then restarted.

Outputting the M, S, T, and B codes for program restart

After the block to be restarted is searched for, you can perform the following operations:

1 Before the tool is moved to the machining restart position

<1> The most recently specified M, S, T, and B codes can automatically be output to the PMC.
The most recently specified S code is output as the maximum spindle speed when the S code is specified in the block containing G92 or as the specified spindle speed in other cases. As the most recently specified S code, only one S code is displayed on the program restart screen regardless of whether the S code is specified in the block containing G92.

<2> While the block to be restarted is being searched for, all sampled M codes and most recently specified S, T, and B codes can automatically be output to the PMC. Up to 35 M codes can be sampled. If the number of sampled M codes exceeds 35, the 35 most recently specified M codes are output to the PMC.

Switch between operations <1> and <2> using bit 6 (MOA) of parameter No. 7300.

2 Before the tool reaches the machining restart position

On the program restart screen, you can specify M, S, T, and B codes from the MDI unit in the MEM or DNC mode without changing the mode.

Outputting the most recently specified M, S, T, and B codes

When bit 7 (MOU) of parameter No. 7300 is set to 1, pressing the cycle start switch after searching for the block to be restarted automatically outputs the most recently specified M, S, T, and B codes to the PMC before the tool is moved to the machining restart position.

In the single block stop status, after the most recently specified M, S, T, and B codes are output, pressing the cycle start switch again moves the tool to the machining restart position.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Outputting all M codes and most recently specified S, T, and B codes

When bit 6 (MOA) of parameter No. 7300 is set to 1, pressing the cycle start switch after searching for the block to be restarted automatically outputs all M codes and most recently specified S, T, and B codes to the PMC before the tool is moved to the machining restart position.

(Example)

When M10, M11, M12, M13, M14, T0101, S1000, and B10 are sampled, a program is executed in the format shown below before the tool is moved to the machining restart position:

M10 T0101 S1000 B10 ;
M11 ;
M12 ;
M13 ;
M14 ;

Outputting M, S, T, and B codes on the program restart screen

When bit 7 (MOU) of parameter No. 7300 is set to 1, you can specify M, S, T, and B codes from the MDI unit in the MEM or DNC mode without changing the mode after searching for the block to be restarted until the tool reaches the machining restart position.

Procedure

1 When the block to be restarted is searched for using the program restart function, the program restart screen appears. When bit 7 (MOU) of parameter No. 7300 is set to 1, operation soft keys [OVERSTORE], [ERASE], and [INPUT] are displayed.

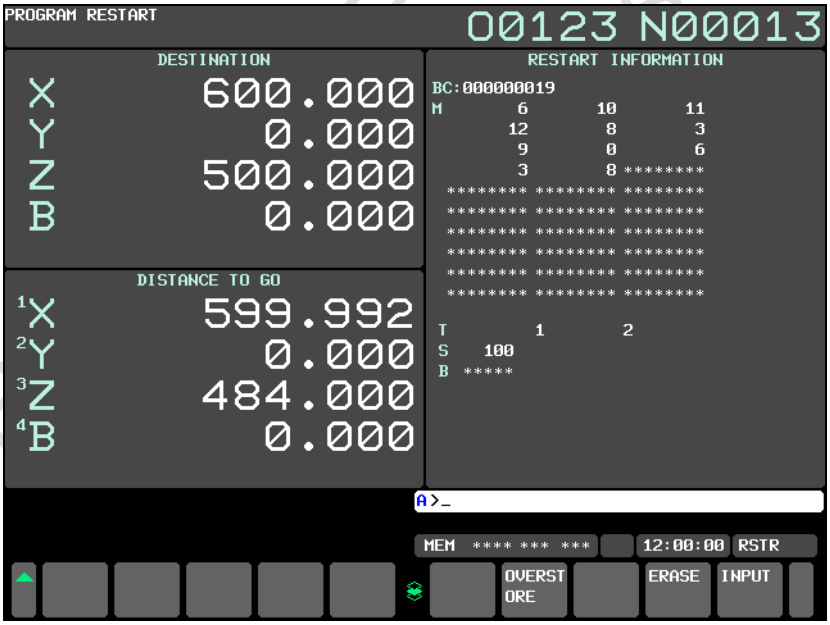


Fig. 4.9 (c) Program restart screen (outputting M, S, T, and B codes)

2 Before the tool reaches the machining restart position, pressing soft key [OVERSTORE] selects the over store mode. In the over store mode, data can be entered in the M, S, T, and B fields displayed in the (OVERSTORE) section.

To select the over store mode while the tool is moving to the machining restart position, hold restart operation by feed hold and press soft key [OVERSTORE].

Enter M, S, T, and B codes to be output in the (OVERSTORE) section from the MDI unit.

(Example)

To enter M10, S1000, T101, and B20 in the (OVERSTORE) section:

<1> Enter [M] [1] [0] from the MDI unit.

<2> Press the [INPUT] key.
You can also enter the S, T, and B codes by performing steps <1> and <2>.

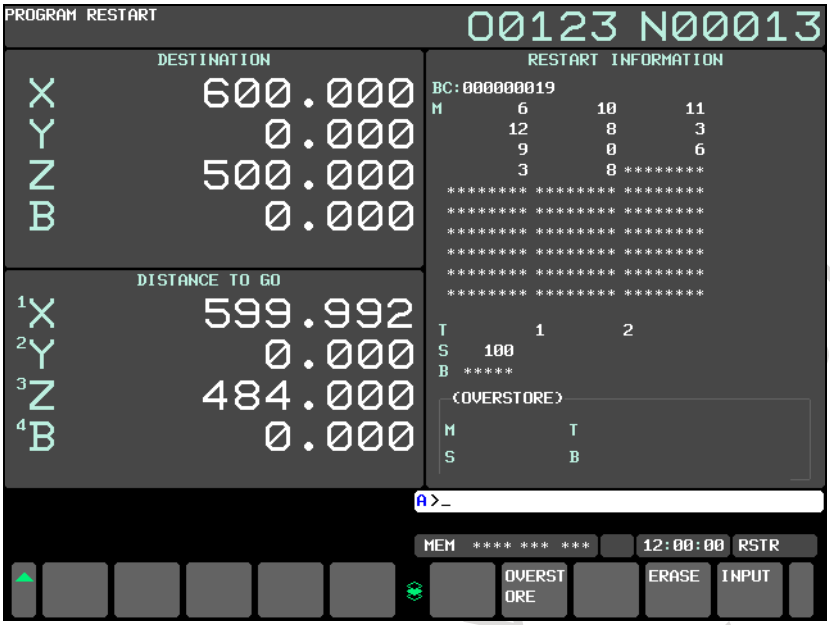


Fig. 4.9 (d) Program restart screen when M, S, T, and B codes are output

3 When values have been entered in the (OVERSTORE) section, pressing the cycle start switch outputs each code in the (OVERSTORE) section. The values in the (OVERSTORE) section are cleared.
4 To clear the values entered in the (OVERSTORE) section as M, S, T, and B codes, press soft key [ERASE]. All entered values are cleared.
5 Pressing soft key [OVERSTORE] again in the over store mode cancels the mode. Pressing the reset key also cancels the over store mode.
6 To continue with restart operation, cancel the over store mode and press the cycle start switch.

⚠ CAUTION
1 The M, S, T, and B codes specified in the over store mode are not displayed on the program restart screen.
2 In the over store mode, changing the operation mode to other than the MEM or DNC mode does not cancel the over store mode. In this case, no values can be entered in the (OVERSTORE) section.
3 In lathe systems, do not specify a T code in the over store mode. If a T code is specified, it is not executed.

Approach to the program restart position per arbitrary axis

This function achieves the following in a program restart:

- On the program restart screen, set the axis name of a single arbitrary axis by keying in or pressing a soft key. Then, perform a cycle start so that the target axis moves to the program restart position at the dry run speed, independently of the order set in parameter No. 7310.

To enable this function, set bit 7 (OAA) of parameter No. 11250 to 1.

TRYER MACHINE SYSTEMS INC Rob Zahensky

Procedure

The procedure for a program restart by using this function is described below.

Enter the state in which a program restart is possible, as described below.
Enter either MEM or DNC mode.

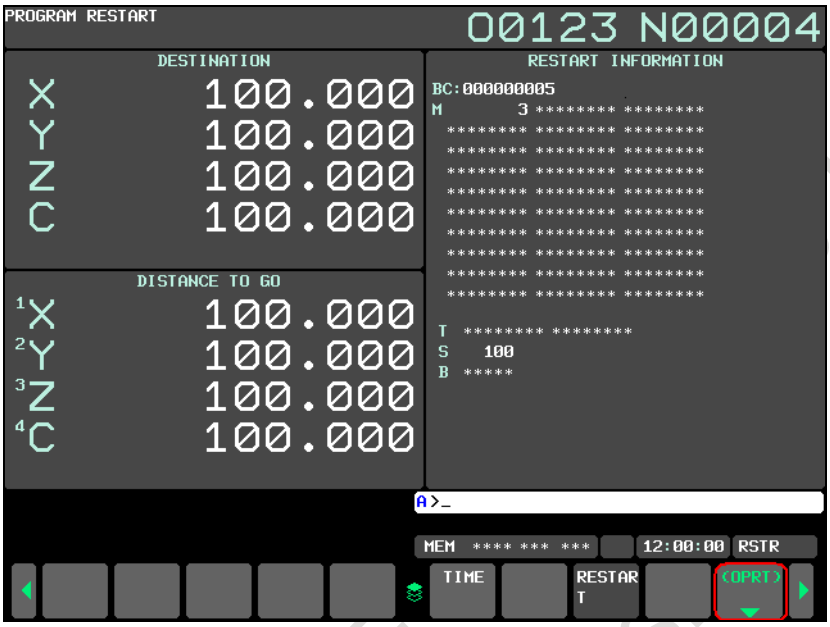


Fig. 4.9 (e) Program restart screen

Press the soft key [(OPRT)], then press the continuous menu key. Soft key [SET MV.AX] appears.

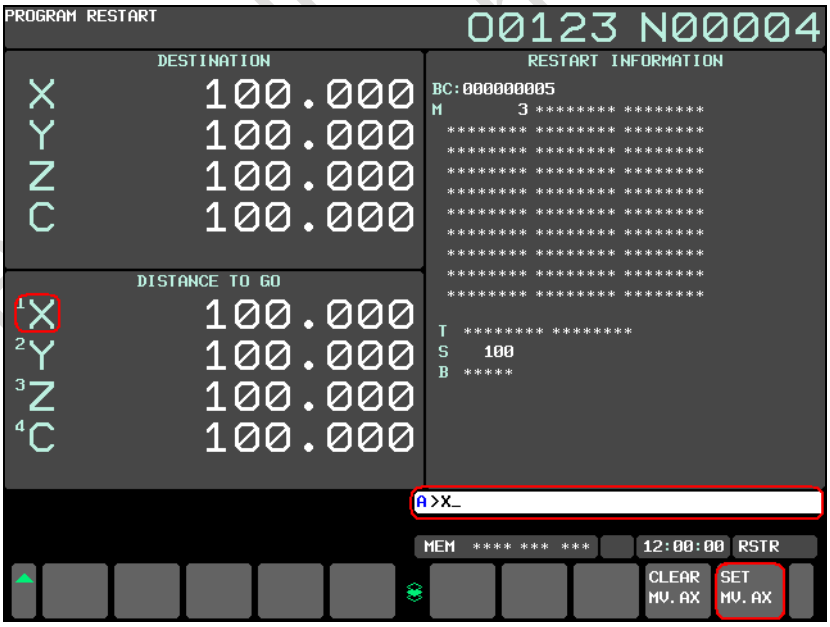


Fig. 4.9 (f) Program restart screen (movement axis setting)

Key in the axis name of the axis to move, and press the soft key [SET MV.AX], and the keyed-in axis name will flash.

NOTE

If an illegal axis is specified, the warning message, "THERE IS NO SPECIFIED NAME OF AXIS", appears.

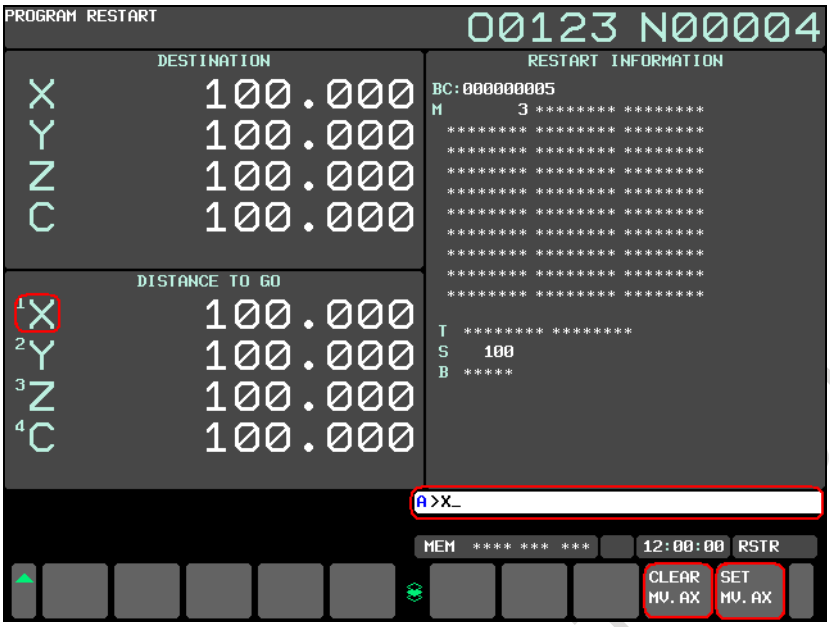


Fig. 4.9 (g) Program restart screen (movement axis setting, movement axis cancellation)

If wanting to cancel the axis that has been set and set another axis, key in this axis name, and press the soft key [SET MV.AX]. To move the axis, perform a cycle start, and the axis specified as described above will move to the program restart position at the dry run speed. When the movement of the axis starts, the axis name no longer flashes.

By pressing the soft key [CLEAR MV.AX], the axis that has been set as described above can be canceled. If the CNC mode is changed, the axis that has been set will be canceled (the axis name no longer flashes).

After the completion of the movement of all axes to the program restart position, perform a cycle start to restart machining.

If a cycle start is performed when the movement of all axes to the program restart position is not completed (or the axes are not yet moved) and no axis is set, the axes will move to the program restart position at the dry run speed. In this case, they will move in the order in which they are set in parameter No. 7310. Then, restart machining.

NOTE

1 If the soft key [CLEAR MV.AX] is pressed when no movement axis is set, the warning message, "THE MOVING AXIS HAS NOT BEEN SET", appears.

2 If the soft key [SET MV.AX] or [CLEAR MV.AX] is pressed in either of the states below, a warning message appears, making operation impossible.
- In a mode other than MEM and DNC modes
  Warning message: WRONG MODE
- Another axis is being moved, or restart coordinates are not set.
  Warning message: COMMAND ILLEGAL USE

Explanation

- Block number

When the CNC is stopped, the number of executed blocks is displayed on the program screen or program restart screen as BC. The operator can specify the number of the block from which the program is to be restarted, by referencing the number displayed on the screen. The displayed number indicates the number of the block that was executed most recently. For example, to restart the program from the block at which execution stopped, specify the displayed number, plus one.

FRYER MACHINE SYSTEMS INC
Ron Zahensky

The number of blocks is counted from the start of machining, assuming one NC line of a CNC program to be one block.

(Example 1)

CNC Program	Number of blocks
O 0001 ;	1
G90 G92 X0 Y0 Z0 ;	2
G01 X100.0 F100.0 ;	3
G03 X0 I-50.0 F50.0 ;	4
M30 ;	5

(Example 2)

CNC Program	Number of blocks
O 0001 ;	1
G90 G92 X0 Y0 Z0 ;	2
G90 G00 Z100.0 ;	3
G81 X100.0 Y0.0 Z120.0 R-80.0 F50.0 ;	4
#1=#1+1 ;	4
#2=#2+1 ;	4
#3=#3+1 ;	4
G00 X0 Z0 ;	5
M30 ;	6

Macro statements are not counted as blocks.

- Storing / clearing the block number

The block number is held in memory while no power is supplied. The number can be cleared by cycle start in the reset state.

- Block number when a program is halted or stopped

The program screen usually displays the number of the block currently being executed. When the execution of a block is completed, the CNC is reset, or the program is executed in single-block stop mode, the program screen displays the number of the program that was executed most recently. When a CNC program is halted or stopped by feed hold, reset, or single-block stop, the following block numbers are displayed:

Feed hold : Block being executed
Reset : Block executed most recently
Single-block stop : Block executed most recently

For example, when the CNC is reset during the execution of block 10, the displayed block number changes from 10 to 9.

- MDI intervention

When MDI intervention is performed while the program is stopped by single-block stop, the CNC commands used for intervention are not counted as a block.

- Block number exceeding eight digits

When the block number displayed on the program screen exceeds eight digits, the block number is reset to 0 and counting continues.

- Presetting the relative coordinate values after the search

If bit 3 (RPR) of parameter No.7301 is 1, relative coordinate values are preset by absolute coordinate values after the search in program restart.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Program restart for flexible synchronization control

Parameter setting

By setting bit 0 (FRS) of parameter No. 13421 to 1, a program restart is possible in a program that contains an M code command for turning the flexible synchronization control mode on/off.
If bit 0 (FRS) of parameter No. 13421 is 0, and an attempt is made to perform a program restart in a program that contains an M code command for turning the flexible synchronization control mode on/off, alarm PS5244, “TOO MANY DI ON” is issued.

Creating/executing a program

1. If this function is used, it is not possible to specify a slave axis during flexible synchronization control. If an attempt is made to specify a slave axis, alarm PS5376, “FSC SLAVE AXIS CAN NOT COMMANDED” is issued.

   (Example)

   C: master axis, A: slave axis
   O0001 ;
   N10 C0 A0 ;
   N20 M133 ; ← Turns synchronous control mode on
   N30 C10.0 A5.0 ; → Alarm PS5376
   N40 C20.0 ;
   N50 M136 ; ← Turns synchronous control mode off
   N60 G90 A20.0 ;
   N70 G91 A10.0 ;
   :

2. When flexible synchronization control is canceled, a workpiece coordinate system preset is automatically performed on the slave axis.

3. If, after flexible synchronization control is canceled, an incremental command is issued for the slave axis before an absolute command, alarm PS5377, “INVALID COMMAND AFTER FSC OFF” is issued.

   (Example)

   C: master axis, A: slave axis
   O0002 ;
   N10 C0 A0 ;
   N20 M133 ; ← Turns synchronous control mode on
   N30 C10.0 ;
   N40 C20.0 ;
   N50 M136 ;
   N60 G90 A20.0 ; ← Turns synchronous control mode off
   N70 G91 A10.0 ; ← The slave axis requires an absolute command.
   :

Operating a program restart

1. Before starting a program restart, make sure that the flexible synchronization control mode matches the synchronous control mode on the program command in the block in which the restart is desired (state of the M code command for turning the flexible synchronization control mode on/off). If they do not match, alarm PS5374, “FSC MODE MISMATCH IN RESTART” is issued. In the period from the beginning of the program to the time the M code for turning the flexible synchronization control mode on/ff is issued for the first time, synchronous control mode is regarded as being off on the program command.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

(Example)

```
C: master axis, A: slave axis
O0003 ;
N10 C90.0 ;
N20 A90.0 ;
N30 C0 A0 ;
N40 M133 ;        ← Turns svnchronous control mode on
N50 C10.0 ;       ← Restart block
N60 C20.0 ;
N70 M136 ;
N80 G90 A20.0 ;   ← Turns synchronous control mode off
N90 G91 A10.0 ;
  :
```

If the block in which to perform a restart is N50, it is necessary to enter the flexible synchronization control mode before performing a program restart.

2. During the execution of a program restart (period from the time the soft key [P TYPE] or [Q TYPE] is pressed until the end of the movement of all axes to the restart position), it is not possible to change the flexible synchronization control mode. If an attempt to change it, alarm PS5375, “FSC MODE CAN NOT CHANGED” is issued.

3. If a block under flexible synchronization control is specified as a restart block, the DISTANCE TO GO of the slave axis is 0. The restart position of the slave axis is "********".

4. If specifying a restart block after the cancellation of flexible synchronization control, specify a block after the absolute command for the axis that was the slave axis. If a block before the absolute command is specified, alarm PS5378, “INVALID RESTART BLOCK” is issued.

(Example)

```
C: master axis, A: slave axis
O0003 ;
N10 A90.0 ;
N20 C90.0 ;
N30 C0 A0 ;
N40 M133 ;        ← Turns synchronous control mode on
N50 C10.0 ;
N60 C20.0 ;
N70 M136 ;        ← Turns synchronous control mode off
N80 G90 A20.0 ;
N90 G91 A10.0 ;
  :
```

If N80 is specified as a restart block, alarm PS5378 is issued.
If N90 is specified as a restart block, the alarm is not issued because N90 is after the absolute command for the slave axis of N80.

Continuing flexible synchronization control in the event of the emergency stop or servo-off state

Usually, if the machine enters the emergency stop or servo-off state, flexible synchronization control is canceled. By setting bit 1 (FCN) of parameter No. 13421 to 1, it is now possible to continue flexible synchronization control. (This function can be used even if bit 0 (FRS) of parameter No. 13421 is 0.)

In this case, if the machine enters the emergency stop or servo-off state, the difference between the current machine position of the slave axis and the correct synchronous control position in relation to the master axis becomes the positional deviation of the slave axis. If the emergency stop or servo-off state is canceled, the slave axis moves to the correct synchronous control position.

FRYER MACHINE SYSTEMS INC Rob Zaherosky

Reference position return/machine coordinate system selection during flexible synchronization control

Usually, if any of G27, G28, G29, G30, and G53 is specified during flexible synchronization control, alarm PS0010, “IMPROPER G-CODE” is issued. If bit 2 (FRF) of parameter No. 13421 is set to 1, alarm PS5381, “INVALID COMMAND IN FSC MODE” is issued in either of the following cases. (This function can be used even if bit 0 (FRS) of parameter No. 13421 is 0.)

- G28 is issued for the master axis when the reference position is not established for the master axis under flexible synchronization control.
- If any of G27 to G30, and G53 is issued for the slave axis.

Program restart for the Cs contour controlled axis

Usually, before a program restart operation can be started, it is necessary to enter Cs contour control mode and establish the origin of the Cs contour controlled axis.

By using the Cs axis coordinate establishment function and setting bit 5 (CCS) of parameter No. 7300 to 1, however, it is no longer necessary to enter Cs contour control mode or establish the origin of the Cs contour controlled axis before starting a program restart operation.

In this case, it is possible to change to Cs contour control mode, if necessary, after the end of the search for the block subject to the restart.

If the system enters Cs contour control mode, the origin of the Cs contour controlled axis is established with the Cs axis coordinate establishment function. And, the "DISTANCE TO GO" of the Cs contour controlled axis is recalculated.

If the system is not in Cs contour control mode, the "DISTANCE TO GO" of the Cs contour control axis is 0. "********" appears for "DESTINATION".

NOTE

To use the Cs contour controlled axis coordinate establishment function, the reference position return must be performed on the Cs contour controlled axis at least once after the power is turned on.

Program restart for 3-dimensional coordinate conversion / Tilted working plane command

If the block subject to a restart is in 3-dimensional coordinate conversion mode G68 (machining center system)/G68.1 (lathe system) or Tilted working plane command mode, it is possible to select whether the movement of the axes to the restart position should be in the program coordinate system or the workpiece coordinate system, by using bit 1 (3DD) of parameter No. 7301. If parameter bit 3DD is 0, the axes move to the restart position one at a time, as seen from the program coordinate system, and if it is 1, they move to the restart position one at a time, as seen from the workpiece coordinate system.

The "DESTINATION" and "DISTANCE TO GO" displays follow the setting of 3DD.

The "ABSOLUTE" display follows the bit 6 (DAK) of parameter No. 3106.

Setting to prevent movement to restart position

When there are two or more axes which has the same setting value in parameter No.7310, the movement to the restart position is not executed in the second or more axis.

Limitation

- P type Restart

In the following conditions, P type restart cannot be performed:

- Automatic operation has not been performed since power-on.
- Automatic operation has not been performed since emergency stop was released.
- Automatic operation has not been performed since the coordinate system was changed or shifted (change of the external workpiece origin offset value).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

The block that can be restored properly by P type program restart is a block for which coordinate system setup or change was performed most recently before machining was interrupted.

- Restart block

The block where the program is to restart is not necessarily be the block at which the program was interrupted. You can restart the program from any block. For P-type restart, however, the block where the program is to restart must use the same coordinate system as when program execution was interrupted.

- Single block

When the single-block operation is enabled at the time of a movement to the restart point, a single-block stop occurs each time an axis operation takes place. In this case, no MDI operation is allowed.

- Manual intervention

During movement to the restart point, manual intervention is allowed for an axis for which a return operation has not yet been performed. However, manual operations do not cause any movement along axes for which a return operation has already been completed.

- MDI

When the search operation has ended, no move command can be specified by MDI before axis movement.

- Reset

Do not perform a reset operation during the time from the start of the search operation of the restart sequence until machining is restarted.
If a reset operation is performed, the restart steps must be performed again from the beginning.

- Feed hold

If a feed hold operation is performed during the search, the restart steps must be performed again from the beginning.

- Manual absolute

Every manual operation must be performed with the manual absolute mode turned on regardless of whether the manual operation is performed before or after machining.

- Reference position return

Unless an absolute position detector (absolute pulse coder) is provided, be sure to perform reference position return after power-up, then perform restart operation.

- Program restart switch

When the program restart switch is on, pressing the cycle start switch does not start operation.

- Blocks specifying a macro statement, macro call, and subprogram call

Blocks specifying a macro statement, macro call, and subprogram call are not searched for even when they have a sequence number. In such a case, search for a block previously preceding such a block.

- Interruption type custom macro

During movement to the machining restart point at a dry run feedrate, no interruption type custom macro can be started. If an interruption type custom macro is started, alarm DS0024, “UINT SIGNAL WAS ILLEGALLY INPUT” is issued.

- Index table indexing

When index table indexing is used for the machine, position the tool to the restart position before performing a program restart.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Commands that prevent program restart

Program restart cannot be performed for blocks placed in the following modes:

- Polygon turning (G50.2)
- Threading (G32,G33), Circular threading (G35,G36), Threading cycle (G92), Multiple-thread cutting cycle (G76)
- Polar coordinate interpolation (G12.1)
- Balance cutting (G68)
- Rigid tapping
- Spindle positioning

If any of the following commands is included between the beginning of a program and the block where the program is to restart, program restart cannot be performed:

- Workpiece coordinate system preset (G92.1,G50.3)
- Commands for enabling and disabling synchronous/composite control and superimposed control
- Commands for enabling and disabling axis synchronous control

- M, S, and T commands not usable in over store mode

The M, S, and T functions listed below, unlike the other M, S, and T functions, have special meanings within the CNC. These M, S, and T commands cannot be specified from the over store screen. To specify these commands, cancel the over store mode, and execute them in MDI operation.

Example:

- Spindle positioning
- Rigid tapping
- Tool management
- Tool offset for lathe system (T code)

Notes

(1) If bit 4 (INT) of parameter No. 13117 is 1, the interference check for cutter/tool nose radius compensation during a restart can be disabled.

(2) If bit 6 (SQB) of parameter No. 13117 is 1, a restart with a block number can be disabled. If B and a block number is specified as the block subject to a restart, a "format error" warning occurs.
If the actual execution of the conditional branch statement (if statement) of the custom macro program differs from its execution during a program restart, set bit 6 (SQB) of parameter No. 13117 to 1.

(3) If bit 7 (SQP) of parameter No. 13117 is 1, a P type restart is disabled. In this case, the soft key [P TYPE] no longer appears. If a P type program restart is not to be used, it is possible to disable the use of the P type by setting SQP to 1.

(4) A macro call block or a subprogram call block cannot be specified as the block subject to a restart.

(5) A tool direction control (G53.1) block cannot be specified as the block subject to a restart.

(6) The following system variables are not updated during a restart search:
#5021 to #5040 and #100051 to #100100 ABSMT
#5041 to #5060 and #100101 to #100150 ABSOT
#5061 to #5080 and #100151 to #100200 ABSKP

FRYER MACHINE SYSTEMS INC
Rob Zahonsky

⚠ **CAUTION**

As a rule, the tool cannot be returned to a correct position under the following conditions.

Special care must be taken in the following cases since none of them cause an alarm:

- Manual operation is performed when the manual absolute mode is OFF.
- Manual operation is performed when the machine is locked.
- When the mirror image is used. However, P type return is possible for a block that switched between ON and OFF most recently or a subsequent block. In this case, the mirror image signal status present when the program was interrupted must be maintained.
- When no coordinate system is set up at the beginning of a program in which main commands are executed in the incremental mode.
- When manual operation is performed in the course of axis movement for returning operation.
- When the program restart is commanded for a block between the block for skip cutting and subsequent absolute command block.
- When program restart is specified in the machine lock state, then the machine lock is canceled.
- When program restart specified for an intermediate block for a multiple repetitive canned cycle
- In general, when a coordinate system is set up, changed, or shifted after the search operation ends, the tool cannot be returned to a correct position.

Keep the following in mind when restarting a program including macro variables.

- Common variable
  When the program is restarted, the previous values are inherited as common variables without being preset automatically. Before restarting the program, initialize the appropriate variables to the original values used at start of the previous automatic operation.
- DI/DO
  At restart of the program, DI can be read by a system variable, but DO cannot be output.
- Clock
  When the program is being restarted, the clock time can be obtained by a system variable, but the time cannot be preset.
- Tool offset and workpiece origin offset
  When the program is being restarted, the offset can be read by a system variable, but change of the offset is allowed only for the Q type.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

4.9.1 Auxiliary Function Output in Program Restart Function

Overview

This function provides the following features for program restart:

- M/S/T/B codes found during a search through a block to be restarted for operation are output to the program restart screen and an MDI program. Then the M/S/T/B functions can be executed from the MDI program.
- On a system having the M-code grouping feature, M codes are grouped. When they are output to the above MDI program, only the M code specified last, among the M codes in the same group, is output to the program restart screen and MDI program.
- M codes for calling subprograms/custom macros and their arguments are also output to the MDI program.
- The order in which individual axes move the tool to the machining restart position can be not only conventionally parameter-set but also set from the program restart screen.

Explanation

Program restart procedure

Follow the procedure below to restart a program using this function:

1. Turn on the program restart switch.
2. Display a program you want to restart and search for the beginning of the program.
3. Enter the sequence number (Nxxxx) or block number (Bxxxx) of a block you want to restart, then press the soft key [P TYPE] or [Q TYPE].
4. Upon completion of searching the block, the program restart screen will be displayed.
5. The M/S/T/B codes found by the search are output to the MDI program. (For details, see "M/S/T/B code output to the MDI program".)
6. On the program restart screen, set the order in which individual axes moves the tool to the machining restart position.
   (For details, see "Specifying the order in which axes move the tool to the machining restart position".)
7. Turn off the program restart switch.
   At this time, the number to the left of each axis name under "DISTANCE TO GO" blinks.
8. Switch to MDI mode and check the MDI program. Fix the MDI program as required, then execute cycle start.
9. When the execution of the MDI program is completed, return to MEM mode.
10. Check whether the distance in "DISTANCE TO GO" is correct and whether the tool does not hit the workpiece or any other object when moving to the machining restart position. If the tool hits anything, move the tool manually to a position free from interference.
11. Press the cycle start button. The tool moves on each axis in the order set in step 6 to the machining restart position at the dry run feedrate, then restarts machining.

NOTE

1 In steps 4, 7, and 11, the warning "ORDINAL NUMBER ERROR (RESTART)" is displayed if the order in which individual axes move the tool is set incorrectly.
2 If the warning occurs in step 4 or 7, you can perform the step that follows.
3 If the warning occurs in step 11, cycle start cannot be executed.
In this case, correct the order in which axes move the tool to the program restart position, then perform step 11 again.

M/S/T/B code output to the MDI program

This function causes the CNC to automatically output to the MDI program the M/S/T/B codes found during a search for a block in which to restart operation.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE
1 Up to 50 M/S/T/B codes or macro call arguments can be output.
2 If there are no M/S/T/B codes to be output, the MDI program will not be created.
3 The number of characters that can be output to the MDI program is 512 including the program number (O0000) and EOB. If 512 characters are exceeded, only the latest command code of up to 512 characters is output.

- Codes that can be output to the MDI program

- S/T/B codes
  The code that appears last during a search is output. If, however, the T code is the same as the current modal T code, it is possible to specify whether to output it, by using bit 4 (MTO) of parameter No. 11250.

NOTE
1 In a lathe system, it is possible to specify whether to output T and B codes, using a parameter.
2 The S code of the clamp command of maximum spindle speed (G92S_; for the M series and G50S_; for the T series) is not output.
3 When high-precision spindle speed control is valid, if decimal fraction of S code is 0 or decimal fraction of S code is omitted, decimal fraction of S code output by this function is omitted.
  Example 1)
  When high-precision spindle speed control is valid, if "S123.0" is commanded, S code output by this function is "S123" (if "S123" is commanded, spindle rotates at a speed of 123.0[min^{-1}]).
  Example 2)
  When high-precision spindle speed control is valid, if "S123" is commanded, S code output by this function is "S123".
4 If extended spindle name is commanded, S code output by this function is “S_ P_”.

- M code
  For a system that has the M code grouping function, it is possible to group M codes from "M CODE GROUP" on the system screen.
  If grouped M codes are found, only the M code in the group that is specified last is output.
  All M codes that are not grouped are output. If, however, multiple M codes with the same number are specified, only the last single one is output.
  Subprogram call and custom macro call M codes are also output. Together with M codes, their arguments are also output to the MDI program. (For details, see "Output of subprogram/custom macro call M codes".)

NOTE
1 If multiple M codes are specified in a single block or if multiple MSTB codes are specified, it is possible to specify whether to output them to one block at a time or in the same block in the MDI program, by using bit 3 (MCO) of parameter No. 11250. (In either case, they are output in the MSTB order.)
2 If "0" or "1" is set as a group No., the M code is regarded as not belonging to any M code group.
3 Even if the option is enabled, it is possible to suppress a group check by setting bit 1 (MGC) of parameter No. 3400 to 1.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

(Example)

Set N100 in the program O100 below as a restart block.

M code group settings

- M10, M11 : Group 2
- M120, M220, M320 : Group 3
- M51 : Group 0 (no group)

```
O100
M10              -- Group 2
 :
M11              -- Group 2
 :
M120             -- Group 3
 :
M51              -- Group 0
 :
M10              -- Group 2
 :
M220             -- Group 3
 :
M51              -- Group 0
 :
M320             -- Group 3
 :
 :
N100 X10. Y10.   -- Restart block
```

M codes that are output

```
M10

M51

M320
```

Output of subprogram/custom macro call M codes

In the output of M/S/T/B codes to the MDI program, subprogram and custom macro call M codes are also output. Together with M codes, their arguments are also output to the MDI program.

NOTE

1. When the MDI program to which M/S/T/B codes are output is executed, the modal data and the workpiece coordinate system at the block position searched for during a program restart are effective. Thus, if a move command is to be executed in the program called from a custom macro call M code such as a tool change command, use the G53 (machine coordinate system selection) command so that it is not affected by the modal data or the workpiece coordinate system.
2. Those M/S/T/B codes that are specified in the subprogram and custom macro programs (including P-CODE calls) called with M codes are not output.
3. Waiting M codes are not output.
4. As a restart block, do not specify a block containing a subprogram call code.

- Subprogram call M codes

Subprogram call M codes specified with parameters Nos. 6044 to 6046 or Nos. 6071 to 6079 are output to the MDI program.

- Macro program call M codes

Macro program call M codes specified with parameters Nos. 6047 to 6049 or Nos.6080 to 6089 are output to the MDI program.

- Arguments of macro calls

Together with macro call M codes, their arguments are also output to the MDI program.

FRYER MACHINE SYSTEMS INC
Rob Zalensky

If the arguments of macro calls are to be output to the MDI program, and they are specified as macro variables or operations in the original program, the arguments are converted into numeric values.
(Example)

Original program	MDI program
#500 = 100.0 ; M20 A[#500/7] ;	M20 A 14.2857143 ;

NOTE

Conversion results are represented by 9-digit numbers, the decimal point, and the minus sign.
If a result cannot be represented by 9 digits, alarm PS5373, “ARGUMENT CONVERSION ERROR” is issued.
(Example)

In the example below, arguments cannot be converted into 9-digit numbers, so that alarm PS5373, “ARGUMENT CONVERSION ERROR” is issued.
M20 A[123456789.*10.0] ; (1234567890. , 10 digits)
M20 B[0.00000001/10.0] ; (0.000000001 , 10 digits)

Specification of the order of movement to the machining restart position

Usually, the order of movement to the machining restart position is the fixed one that is set in parameter No. 7310. This function enables changing of the order on the program restart screen when a program restart operation is performed.
The movement order can be changed after the end of the search for a program restart block until the start of the movement to the restart position.

- Procedure for specifying the order of movement to the program restart position

1. On the program restart screen, the cursor is displayed at the movement order display position to the left side of axis addresses.

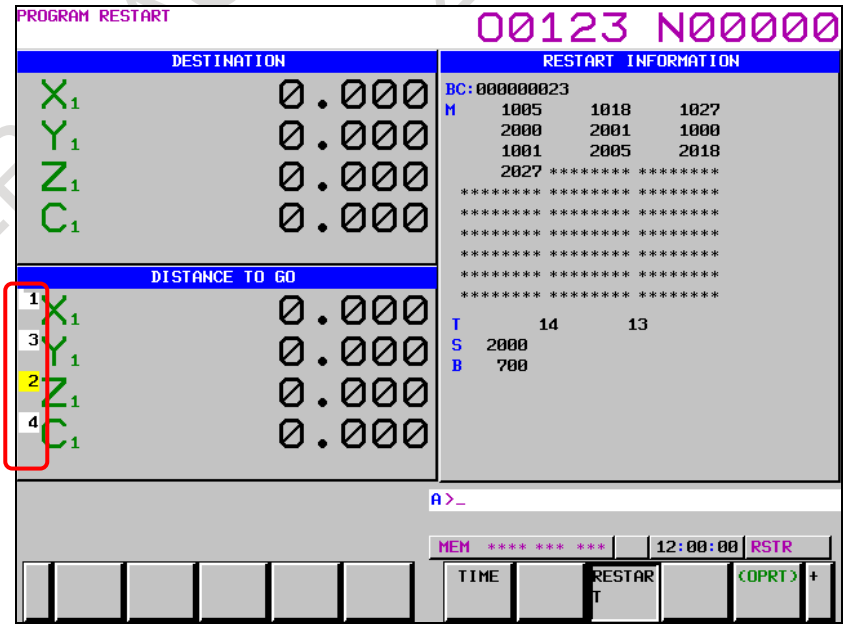


Fig. 4.9.1 Program restart screen

2. Using the up and down arrow keys, position the cursor on an axis for which to set the movement order, and enter a numeric value.

FRYER MACHINE SYSTEMS INC Zahonsky

NOTE

1 If the movement order is changed on the program restart screen, the value of parameter No. 7310 is also changed.

2 If an attempt is made to set a "0" or a value exceeding the number of controlled axes for an input value, the warning message, "DATA IS OUT OF RANGE", appears.

Note

If using this function, be sure to set bit 7 (MOU) of parameter No. 7300 to 0.

4.10 QUICK PROGRAM RESTART

This function restarts machining from a block by specifying the block on the relevant screen to restart machining after a tool is damaged or after holidays.

- For this function, a screen for restart operation is available. This screen is called “program restart setting screen”. To perform program restart, first specify a target block on the program restart setting screen, next press soft key [SEARCH EXEC]. Then, program restart is executed.
- On the program restart setting screen, information (including the program name and sequence number) on the last executed block is displayed as primary information. This information can be checked to see the machining interrupted point easily.
- On the program restart setting screen, in addition to the information on the last executed block, information on some blocks executed before that can be displayed. This information can be automatically memorized in CNC memory during the operation of a program. This memory is called “program restart memory”.
  Block information memorized in program restart memory is called “restart block information”.
- You can select an arbitrary block in restart block information and press soft key [SEARCH EXEC] to restart automatic operation from that block.

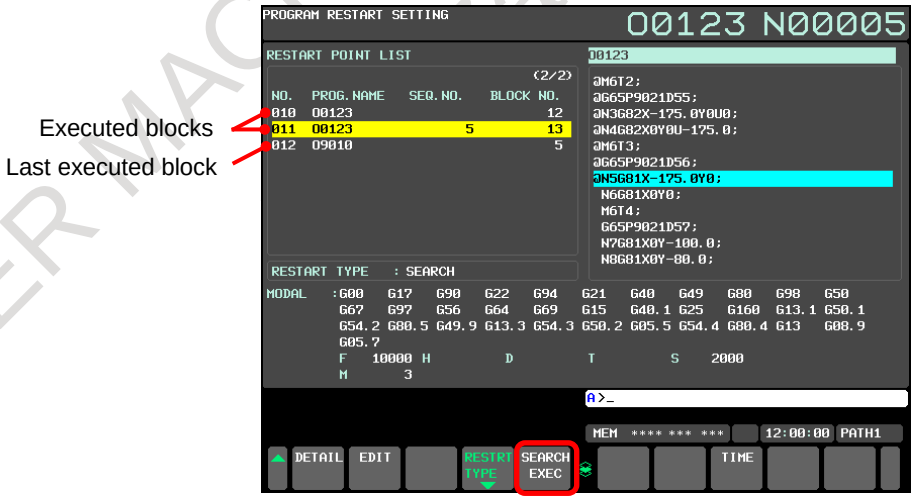


Fig. 4.10 (a) Program restart setting screen

- You can also restart a program after changing a block number or sequence number displayed on the program restart setting screen, using the MDI unit.
- To restore the machine to the suitable status (including the auxiliary function status and spindle status), use the “auxiliary function output in program restart” function. See Section, “Auxiliary Function Output in Program Restart Function” for details.
- For this function, either of the following two types of restart methods can be selected.

FRYER MACHINE SYSTEMS INC

1. Search method

   Use this type when it is difficult to restore the status of modal information and auxiliary functions manually. This type can be selected to restart a program not only from the block memorized in the program restart memory, but also from another block. When this method is selected, the modal information and position information of the block from which to restart the program can be restored automatically.

   Application cases : When the power is turned off after machining is interrupted
   When another workpiece is machined after machining is interrupted, and the restart block information on the previous workpiece is not memorized
   When the program is to be restarted from a block which is not memorized in the program restart memory

   Function : By using the "auxiliary function output in program restart" function, the auxiliary function status can be restored.
   The tool automatically moves to the restart position along each axis when the program is restarted.

   Operation : Restores modal information and coordinate values while performing the simulated execution of the program until the block specified in the restart block setting screen is found. (Updates modal information and coordinate values without moving the tool along any axis.)
   Mainly updates the following data:
   - Modal information including G, F, D, and H codes
   - Absolute coordinate system
   - Auxiliary function output in the program restart

2. Direct jump method

   Use this type when the operator restores the status of modal information and auxiliary functions manually and restarts the machining. This method can be selected only when the program is to be restarted from the block memorized in the program restart memory.
   When this method is selected, program restart is performed at high speed, though the modal information and position information of the restart block are not restored. Restart operation will be completed in a short time compared with the search method, in particular, when the restart block is in the middle of a long program.

   Application cases : When the operator can easily restore the machine status because changes in the machine status are few when, for example, machining is interrupted, and is to be restarted immediately)
   When a program is to be restarted from a block where operation can be started without restoring the machine status, such as a block on a machining process boundary

   Function : The "auxiliary function output in program restart" function cannot be used.
   The operator must determine whether to specify required auxiliary functions and specify them.
   The machine must be positioned at the restart position manually before operation is restarted because the tool does not move to the restart position along any axis when the program is restarted.
   Modal G codes and other modal information must be set to the suitable status by MDI operation before operation is restarted.

   Operation : Moves only the program cursor to the specified block at high speed without restoring modal information or coordinate values.

- Executing cycle start after pressing soft key [SEARCH EXEC] on the program restart setting screen restarts operation. When the search method is selected, the auxiliary function status is restored by the "auxiliary function output in program restart" function and the tool is moved to the restart position before operation is restarted.
- When soft key [TIME] is pressed, the elapsed time from the start of program to the restart block is displayed instead of the sequence number. And if [SEQ.NO] is pressed, sequence number is displayed again.

FRYER MACHINE SYSTEMS INC
Ron Zahorsky

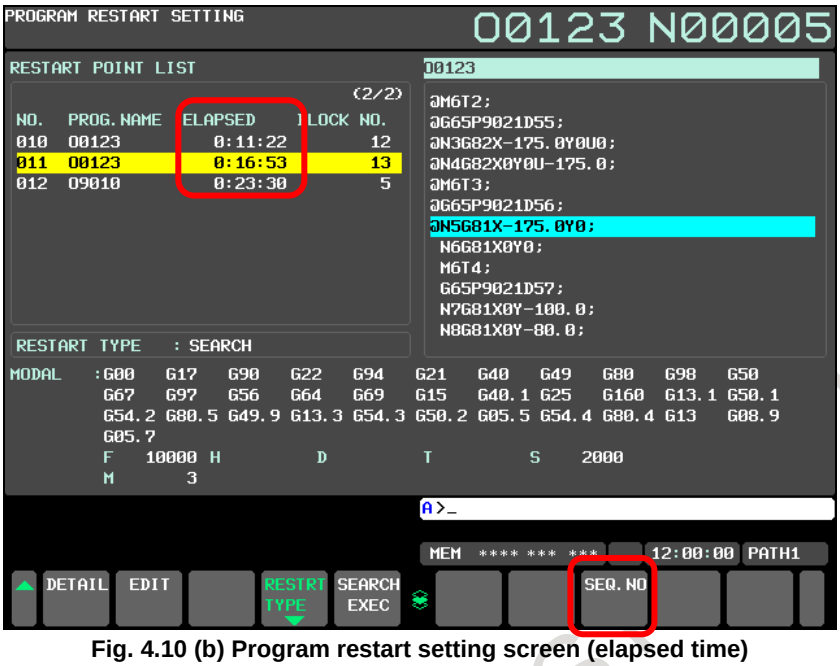


Fig. 4.10 (b) Program restart setting screen (elapsed time)

Procedure for restarting a program using the search method

Procedure 1

1. When power is turned on or emergency stop is released, perform all necessary operations at that time, including reference position return.
2. Confirm that the program to be restarted is selected. If you want to restart in the middle of a subprogram, confirm that the main program that calls the subprogram is selected.
3. Move the machine by manual operation to first the program starting point (position in which machining began first) and place the modal data and coordinate system in the same conditions as at the start of machining.
4. Modify the offset amount if necessary.
5. Go to procedure 2.

Procedure 2

1. Select the MEM or DNC mode.
2. Press function key [PROG]. Press soft key [RESTRT], then [RESTRT SETING]. The following program restart setting screen appears.
   The last item in the restart point list is the last block that was executed.
3. Go to procedure 3.

FRYER MACHINE SYS INC
Rob Zarensky

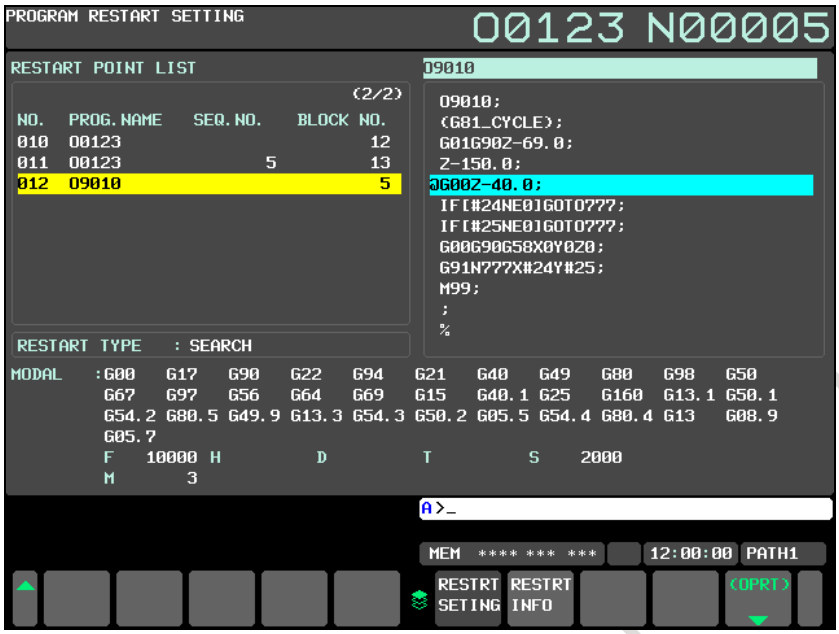


Fig. 4.10 (c) Program restart setting screen (restart point list)

> **NOTE**
> The program display shown on the right side of the screen in the figure above can be displayed only for the following cases:
> (1) Memory operation
> (2) Memory card program operation/editing
> (3) Operation of a program in the Data Server as a program by memory operation or execution of the program by an M98 subprogram call
> Accordingly, any program executed by an M198 external subprogram call or operated in the DNC mode is not displayed.

Procedure 3

1. Select a block from which to restart the program on the screen.
   To check detailed information of each block, press soft key, [(OPRT)], then [DETAIL]. Detailed information of the restart point pointed out by the cursor is displayed. To display the list screen again after checking detailed information, press soft key [LIST].
   - **When the block from which to restart the program is displayed**
     → Go to procedure 3-1.
   - **When a block near the block from which to restart the program is displayed**
     → Go to procedure 3-2.
   - **When neither of the above blocks is displayed**
     → Go to procedure 3-3.

Procedure 3-1 (When the block from which to restart the program is displayed)

1. Select a desired restart point using cursor keys [↓] [↑] and page keys [PAGE ↑] [PAGE ↓].
   (Go to procedure 4.)

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Fig. 4.10 (d)

Procedure 3-2 (When a block near the block from which to restart the program is displayed)

When a block near the block from which to restart the program is displayed as the restart point, specify the restart block by editing information of the restart point as follows:

1. Select a desired restart point using cursor keys ↓ ↑ and page keys ⇧PAGE PAGE⇩.
2. Press soft key [(OPRT)], then [EDIT].
   Move the cursor and edit data.
   Enter desired sequence number, block number, and repetition count. When a sequence number is assigned to the restart block, specify the sequence number and repetition count without specifying any block number.

Fig. 4.10 (e) Program restart setting screen (edit)

FRYEBOROVA MACHINE SYSTEMS CNC
Ing. Zdeněk Zahenský

NOTE

There are the following restrictions when two or more items are edited:

1 Among SEQUENCE NO., CURRENT BLOCK COUNT, and TOTAL BLOCK COUNT, only one item can be edited. Two or more items cannot be edited at a time. If one item is edited, the other two items become invalid.

2 In the REPEAT section, only one item can be edited. Two or more items cannot be edited at a time.
If one item is edited, the other two items become invalid.

The value in the CURRENT BLOCK COUNT field can be changed by selecting an arbitrary block in the program display window (on the right side of the screen) instead of entering the block number or sequence number. Follow the procedure below:

(1) Press the [→] key.

The cursor moves to the program display window (on the right side of the screen).

(2) Press the [↓] [↑] keys and select an arbitrary block from which to restart the program.

The value in the [CURRENT BLOCK COUNT] field in detailed information (on the left side of the screen) is automatically changed to the selected block.
It is possible ti search word by using soft key [SRH↑] and [SRH↓].

Fig. 4.10 (f) Specifying a block in the program display window

NOTE

This method is available for the following cases:

1. For restarting the program from a block in the memory operation mode
2. For restarting the program from a block in the memory card program operation/editing mode
3. For operating a program in the Data Server as a program by memory operation or executing it by an M98 subprogram call

Accordingly, this method is not available for a block in a program executed by an M198 external subprogram call or operated in the DNC mode.

3. Go to procedure 4.

Procedure 3-3 (When neither of the blocks is displayed)

Use this procedure when the block from which to restart the program is not displayed or if you want to restart another program that is not the interrupted program, from a desired block. Follow the procedure below:

1. Press soft key [(OPRT)] on the program restart setting screen. Then, press soft key [EDIT], then [NEW].
2. Enter the program name to restart. When restart is performed in the program which is called by M198 external subprogram or the program which is used in DNC mode, press soft key [DNC SELECT]. And enter file name.
3. Enter the sequence number or block number, and other data of the block from which to restart the program to specify the restart block.

Fig. 4.10 (g)

Enter a value in the relevant field as follows:

(1) To specify a block with a sequence number

When a sequence number is assigned to the block from which to restart the program, set the sequence number in the [SEQUENCE NO.] field.

When the same sequence number appears in the program more than once, set the number corresponding to the block in the [REPEAT (SEQUENCE NO.)] field. If this number is omitted, the block is assumed to be the first block having the relevant sequence number that appears in the program.

When the block from which to restart the program is in the subprogram, set the subprogram name to the item of "PROGRAM NAME". This setting is unnecessary if the block from which to restart the program is in the main program. The setting of the subprogram name can be omitted by setting a bit 4 (NPN) of parameter No.11251 to 1. The sequence number can be specified for NPN=1 without the distinction between the main program and the subprogram.

(2) To specify a block with a block number

When no sequence number is assigned to the block from which to restart the program, set the number of that block in the [CURRENT BLOCK COUNT] field.

Specify the block number with the number of lines (EOBs) from the beginning of the program. When the same block is executed more than once according to the GOTO statement, specify the number of repetitions that corresponds to that block in the [REPEAT (BLOCK COUNT)] field. If this number is omitted, the block is assumed to be the first block.

(3) To specify a block with the number of execution blocks

When the number of blocks executed from the beginning of the program is known, set the number in the [TOTAL BLOCK COUNT] field.

4. Go to procedure 4.

Procedure 4

1. Press soft keys [(OPRT)], [RESTRT TYPE], and [SEARCH] to select the search method as the restart type. “SEARCH” is displayed for [RESTART TYPE] on the screen.

Fig. 4.10 (h)

> **NOTE**
> In the initial status after power-on, the search method is selected as the restart type.

2. Press soft key [SEARCH EXEC]. The selected restart point is searched for. When the search ends, the program restart screen appears.
   For details of this screen, see step 5 in “Procedure 2” in “Procedure for program restart by specifying a sequence number” in Section, “PROGRAM RESTART”.
   When the CNC is not reset state and the soft key [SEARCH EXEC] is pressed, the warning "CNC MODE ERROR" is displayed.
3. Go to procedure 5.

Procedure 5

1. Set the order in which the tool moves to the machining restart position along axes on the program restart screen if necessary. The warning "ORDINAL NUMBER ERROR (RESTART)" is displayed if not correctly set.
2. Set the MDI mode and check the MDI program. After modifying the MDI program if necessary, execute cycle start.
3. After the MDI program has been executed, return the machine to the MEM mode.
4. Check whether the distance displayed under [DISTANCE TO GO] is correct. Also check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If such a possibility exists, move thee tool manually to a position from which the tool can move to the machining restart position without encountering any obstacles.
5. Press the cycle start button. The tool moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified in step 1.
   You can also move the tool to the machining restart position sequentially along axes by selecting an arbitrary axis at a time, independently of the order set in step 1. For details, see “Approach to the program restart position per arbitrary axis” in Section, “PROGRAM RESTART”.
   After the tool moves to the machining restart position along all axes, machining is restarted. Then, the program check screen automatically appears.

FRYER MACHINE S NC Rob Zaherisky

For details of steps 1 and 2 above, see Subsection, “Auxiliary Function Output in Program Restart Function”.

Procedure for restarting a program using the direct jump method

Procedure 1

1. When power is turned on or emergency stop is released, perform all necessary operations at that time, including reference position return.
2. Confirm that the program to be restarted is selected. If you want to restart in the middle of a subprogram, confirm that the main program that calls the subprogram is selected.

Procedure 2

1. Press function key PROG. Press soft key [RESTRT], then [RESTRT SETING]. The following program restart setting screen appears.
   The last item in the restart point list is the last block that was executed.
2. Check whether the block from which to restart the program is displayed. If the block is not displayed, the program cannot be restarted using the direct jump method. Use the search method to restart the program.

Fig. 4.10 (i)

3. Check the status of modal information and auxiliary functions in the block from which to restart the program. Restore the status by MDI operation if necessary.
4. Go to procedure 3.

FRYER SYSTEMS INC

Procedure 3

1. Select the MEM or DNC mode.
2. On the program restart setting screen, press soft keys [(OPRT)], [RESTRT TYPE], and [JUMP] to select the direct jump method as the restart type. “JUMP” is displayed for [RESTART TYPE] on the screen.

Fig. 4.10 (j)

3. Select the block from which to restart the program using the cursor and press soft key [SEARCH EXEC].
4. Go to procedure 4.

Procedure 4

1. The program cursor jumps to the specified block. On the program screen, check whether the cursor indicates the target block.
2. Check modal information and others again and specify the status by MDI operation if necessary.
3. Move the tool to the restart position by manual operation.
4. Return the machine to the MEM or DNC mode and execute cycle start.
   The program is executed from the cursor position.

Explanation

- Program restart memory

1. The following blocks are memorized in the program restart memory:
   a. Last executed block
   b. Blocks of positioning (When some blocks are consecutive, the first block is memorized)
   c. Blocks of subprogram calls (including M98, M198, G65, G, and M code macro calls and subprogram calls)
   d. Other blocks judged to be suitable for restarting a program (blocks for turning each mode on or off and others that function as a breakpoint in a program)
2. If the program restart memory becomes full, block information is deleted from the older one and the information on a new block is overwritten.
3. The program restart memory is cleared when operation is started in the MEM or RMT mode after a reset.
   The memory is also cleared when the program is changed and restart operation using the search method starts.

FRYER MACHINE INC Rob Zahenski

4. When a program is restarted with a new restart point specified (“Procedure 3-3” in “Procedure for restarting a program using the search method”), the information on the blocks memorized before restart operation is not cleared. After the program is restarted, block information after restart is added following the block information memorized before restart.

- Program restart using the search method

1. Any sequence number in the programmable parameter input (G10L52 or G10L50) mode is not searched for when a search is executed with specifying a sequence number.
2. When a short cut is specified for a rotary axis, the tool also uses a short cut to the machining restart position.
3. If the single-block operation is enabled when the tool moves to the machining restart position, a single-block stop occurs each time the tool moves along an axis. During this operation, however, no MDI operation is allowed.
4. While the tool is moving to the machining restart position, manual intervention is allowed for an axis along which the tool has not moved to the restart position. The tool is not moved manually along any axis along which it has moved, however.
5. During movement to the machining restart position, no interruption type custom macro can be started. If an interruption type custom macro is started, alarm DS0024, “UINT SIGNAL WAS ILLEGALLY INPUT” is issued.

- Procedure for restarting a program using the direct jump method

There are the following limitations.

1. A direct jump may not be able to be selected depending on the block. (Example: Block in the cutter or tool nose radius compensation mode)
   If an attempt is made to select “JUMP” as the restart type for such a block, the warning message, “ILLEGAL DATA”, is displayed and the setting is not accepted.
2. The direct jump method can be selected to restart a program from a block memorized in the program restart memory. If an attempt is made to use the direct jump method to restart a program from a block which is not memorized in the program restart memory, warning message “ILLEGAL DATA” is displayed.
3. When a direct jump method is selected, the status of items including position information, modal information, and macro variables is not restored. The “auxiliary function output in program restart” function is not also executed. For this reason, the operator must restore the status by manual or MDI operation with considering the status of the restart block.
4. In the following cases, a direct jump can be selected for a block in a subprogram or external subprogram call:
   (a) Subprogram call using M98
   (b) Subprogram call using G65
   (c) Block in a subprogram on the Data Server operating by M198
   (d) Block in a program on the Data Server operating in the DNC mode
   In other cases, a direct jump cannot be selected for a block in a subprogram or external subprogram call.
   That is, a direct jump cannot be selected for the following blocks:
   (e) Block in a subprogram specified by an M or G code or a program called by a macro call
   (f) Block operating in the DNC mode (other than the Data Server)
   (g) Block in an execution macro
5. Note that there are the following limitations for blocks described in c. and d. above:
   (1) It is not possible to jump to the right position when there is a branch instruction of the custom macro.
   (2) It may take 1 to a few minutes to move the cursor to the restart block in a long program longer than 20 MB.
6. The jump position is shifted if the number of lines in the program is changed by editing the program after interrupted.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Stop at the restart position

When bit 1 (RPS) of parameter No.7331 is set to 0, it enables to stop at the restart position by single block operation. When bit 1 (RPS) of parameter No.7331 is set to 1, it enables to stop at the restart position even for continuous operation.

- Execution macro (macro executor)

A program cannot be restarted from a block in an execution macro.

- Modal display on the restart block setting screen

The modal display on the restart block setting screen indicates the status before the execution of the memorized block. That is, data specified in the memorized block is not displayed.

- Reset

Do not perform reset operation from the start of a search for restart to the restart of machining.
If reset operation is performed, perform restart operation from the beginning again.

- Feed hold

If performing feed hold operation during search, perform restart operation from the beginning again.

- Manual absolute

Perform every manual operation with the manual absolute mode turned on regardless of whether the manual operation is performed before or after machining.

- Reference position return

When no absolute position detector (absolute pulse coder) is provided, be sure to perform reference position return after power-on, then perform restart operation.

- Cs contour control

For details, see Item, “Program restart for the Cs contour controlled axis” in Section, “PROGRAM RESTART”.

- Flexible synchronization control

Setting bit 0 (FRS) of parameter No. 13421 to 1 enables program restart in a program containing an M code for turning the flexible synchronization mode on or off.
For details, see Item, “Program restart for flexible synchronization control” in Section, “PROGRAM RESTART”.

- Tilted working plane indexing

To restart a program from a block in the tilted working plane indexing mode using a direct jump method, follow the procedure below:

1. Return the tool to the restart position.
2. Execute [SEARCH EXEC] using the direct jump method.
3. Specify the tilted working plane indexing mode using the MDI unit after the program has been restarted (blinking RSTR disappears).
4. If the program is restarted from block after tool axis direction control, please command tool axis direction control.
5. The present absolute coordinate is matched to the absolute coordinates of the restart block.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- **Other notes**

⚠ **WARNING**

In principle, the tool cannot return to a correct position in the following cases. Special care must be taken in the following cases since none of them cause an alarm.

- Manual operation is performed when the manual absolute mode is off.
- Manual operation is performed when the machine is locked.
- The mirror image is used.
- A program in which main commands are executed in the incremental mode does not begin with coordinate system setting.
- Manual intervention is performed while the tool is returning to the restart position along an axis.
- Program restart is specified for a block between the block for skip cutting and subsequent absolute command block.
- After program restart is specified in the machine lock state, the machine lock state is canceled.
- A program is restarted from a block in a multiple repetitive canned cycle with specifying a block or sequence number.
- In general, if a coordinate system is set up, changed, or shifted or a tool compensation value is changed, the tool cannot return to the correct position.

⚠ **CAUTION**

Keep the following in mind when restarting a program including a macro variable.

- Common variable
  When a program is restarted, the previous values are inherited as common variables without being preset automatically. For this reason, before restarting the program, initialize the required variables to the original values used at the start of the previous automatic operation.
- DI/DO
  During the restart of a program, DI can be read by a system variable, but DO cannot be output.
- Clock
  During the restart of a program, the clock time can be obtained by a system variable, but the time cannot be preset.

- **Relation to other functions**

The symbols in the table are described below:

A : Can restart a program from a block in the mode of each function.

NA : Cannot restart a program from a block in the mode of each function. Also cannot restart a program when any of these commands is specified in the first block of the program to the restart block.

*1 : There are some limitations. see Remarks.

*2 : Can restart a program after the status of items including modal information, macro variables, and auxiliary functions is restored by MDI operation or manual operation.

*3 : May restart a program using the “auxiliary function output in program restart” function in some cases.

*4 : Cannot restart a program from a block in the mode of each function.
Can restart a program from a block in which the mode is off when the program contains a block for turning the mode on or off.

FLYER MACHINE SYSTEMS INC
Rob Zaborsky

*5 : Coordinate values are calculated on the assumption that the block is executed to the end point. Therefore, if the program is restarted from a block without specifying an absolute command at least once following this command, the positioned coordinate values may be different from those actually executed.

From blocks from which a program can be restarted using the conventional program restart function, a program can also be restarted using the search method.

Table 4.10 Relation to other functions

Function name	Search method	Direct jump method	Remarks
Automatic operation (Memory operation)	A	A	
DNC operation	A	*1	Only the DNC operation of the Data Server is available on the direct jump method. When the program restart is executed excluding the Data Server, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
DNC operation with memory card	A	NA	In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Schedule function	NA	NA	In the search method, alarm SR1968, "ILLEGAL FILE NAME (MEMORY CARD)" is generated. In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Optional block skip	*1	*1	The optional block skip signal cannot be changed while executing the program restart. The program restart cannot be correctly executed, and restarts in a wrong block when the signal is changed.
Positioning	A	A	
Single direction positioning (G60)	A	A	
Linear interpolation	A	A	
Circular interpolation	A	A	
Helical interpolation	A	A	
Dwell	A	A	
Thread cutting, synchronous cutting	*4	*4	The thread shifts when restarting on the way of the threading blocks continuously commanded.
Multi threading	*4	*4	Same as the above-mentioned
Continuous threading	*4	*4	Same as the above-mentioned
Variable lead thread cutting	*4	*4	Same as the above-mentioned
Circular thread cutting	*4	*4	Same as the above-mentioned
Skip	*5	*2	
High-speed skip	*5	*2	
Multi-step skip	*5	*2	
Continuous high-speed skip	*5	*2	
Torque limit skip	*5	*2	
Polar coordinate interpolation	*4	*4	In the search method, tool position might shift by operating after the program restarts. In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.

FIXER MACHINE SYSTEMS INC
Rob Zaharchuk

Function name	Search method	Direct jump method	Remarks
Cylindrical interpolation	*4	*4	Alarm PS0176, "ILLEGAL G-CODE USE(G07.1 MODE)" is generated and restart is not available in both search methods and the direct jump methods.
Polar coordinate command	A	*4	In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Normal direction control	A	*4	In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Mirror image	NA	*2	In the search method, tool position might shift by operating after the program restarts.
Programmable mirror image	A	*4	In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Mirror image for double turret	A	*4	In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Scaling	A	*4	In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Figure copying	A	*4	In the direct jump method, the tool path might shift by operating after the program restarts.
Coordinate system rotation	A	*4	In the direct jump method, the tool path might shift by operating after the program restarts.
3-dimensional coordinate system conversion	A	*2	If G68(G68.1) is commanded with MDI operation and 3-dimensional coordinate system conversion mode is restored, the direct jump method is available.
Tilted working plane indexing	A	*2	See item “Tilted working plane indexing” for details when a direct jump method is selected. See item “Workpiece setting error compensation (or tilted working plane indexing + tool center point control)” to use tool center point control simultaneously.
Workpiece coordinate system	A	*2	
Workpiece coordinate system preset (G92.1/G50.3)	NA	*2	In the search method, tool position might shift by operating after the program restarts.
Coordinate system setting (G92/G50)	A	*2	
Reference position return	A	A	
Reference position return check	A	A	
2nd reference position return	A	A	
3rd/4th reference position return	A	A	
Tool length offset	A	*2	
Tool offset (G45 to G48)	A	*2	
Tool offset (T code on lathe system)	A	*2	
Tool radius / Tool nose radius compensation	A	*4	In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Tool offset for Milling and Turning function	A	*2	
Automatic tool length measurement (M series) / Automatic tool offset (T series)	*5	*2	
Plane selection	A	*2	

FRYER MACHINE SYSTEMS INC
Rob Zabensky

Function name	Search method	Direct jump method	Remarks
Balanced cutting	*4	*4	Synchronization between paths might shift by operating after the program restarts. In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Programmable data input	A	A	
Programmable parameter input	A	A	For the blocks between G10L52 (or G10L50) and G11, program restart with specifying a sequence number is disabled.
Sub program call (M98)	A	A	
Sub program call (M198)	A	*1	Only a program call on the Data Server is available for a direct jump method.
Inch/metric conversion	A	*2	
Direct drawing dimension programming	A	*4	
Chamfering/Corner R	A	*4	
Optional chamfering corner R	A	*4	
Custom macro	A	*2	
Interruption type custom macro	A	*2	
Canned cycle	*4	*4	Tool path might shift by operating after the program restarts.
Multiple repetitive cycles	*4	*4	Tool path might shift by operating after the program restarts.
Multiple repetitive cycles II	*4	*4	Tool path might shift by operating after the program restarts.
Canned cycles for grinding	*4,*5	*4	Tool path might shift by operating after the program restarts.
Canned cycle for drilling (except rigid tapping)	A	*4	In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Small-hole peck drilling cycle	*4	*4	In the search method, operation is restarted from the beginning of the hole machining. In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Rigid tapping	*4	*4	In the search method, an alarm PS0200, "ILLEGAL S CODE COMMAND" is generated after restart. In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
PMC axis control	*3	*2	
Real time custom macro	NA	*4	The real time custom macro is not executed while restarting. Therefore, the real time custom macro might not operate correctly after the program restarts. (both search methods and the direct jump methods)
Polygon turning	*4	*4	
Polygon turning by Cs contour control	*4	*4	
Polygon machining with two spindles	*4	*4	

FRYER MACHINE SYSTEMS INC
Rob Zaharaky

Function name	Search method	Direct jump method	Remarks
High-speed HRV3 control (G5.4)	*4	*2	On/off of this function is not restored as for the search method either. If G05.4Q1 or Q0 is commanded by MDI operation and the state is restored after restarted, operation can be restarted. (both search methods and the direct jump methods)
Optimum torque acceleration/deceleration	A	A	
Jerk control	A	A	
AI contour control I	A	A	
AI contour control II	A	A	
Exact stop mode	A	*2	
Tapping mode	A	*2	
Cutting mode	A	*2	
Auxiliary function	*3	*2	
2nd auxiliary function	*3	*2	
M code group check	*3	*2	
Tool management function	*3	*2	When the search method is selected and a tool exchange macro is executed after search, the amount of tool offset may be incorrect. (For a macro using a command such as H#8409D#8410)
Tool life management	*3	A	
Embedded macro	A	A	
Arbitrary angular axis control	A	A	
Axis synchronous control	NA	*4	In the search method, program restart can be restarted if the synchronization is always ON. If on/off of synchronization is changed by M code etc. in the program, the position of the master axis and the slave axis might shift after restart.
Synchronous/Composite control	NA	*4	If on/off is changed by M code etc. in the program, the position of the master axis and the slave axis might shift after restart.
Superimposed Control	NA	*4	If on/off is changed M code etc. in the program, the position of the master axis and the slave axis might shift after restart.
Synchronous/Composite/Superimposed control by program command	NA	*4	If on/off is changed in the program, the position of the master axis and the slave axis might shift after restart.
Flexible synchronization control	*1	*1	See item “Flexible synchronization control”.
Hobbing/Electric gear box	NA	*4	The machining is executed with the function is invalid after restarted.
Electric gear box 2 pair	NA	*4	Same as the above-mentioned
Electric gear box automatic phase synchronization	NA	*4	Same as the above-mentioned
Spindle electric gear box	NA	*4	Same as the above-mentioned
Machining condition selecting function	A	*2	
Spindle positioning	*4	*4	The spindle must be positioned to the restart position by MDI operation in advance.
Cs contouring control	*3	*2	See item “Cs contour control” for details when the search method is selected.
Spindle speed fluctuation detection	A	*2	
Spindle synchronous control	*3	*2	

FRYER MACHINE SYSTEMS INC
Rob Zabensky

Function name	Search method	Direct jump method	Remarks
Live tool control with servo motor	*3	*2	
Servo/spindle synchronous control	*3	*2	
Macro executor	*4	*4	A program cannot be restarted from a block in an execution macro.

- **Presetting the relative coordinate values after the search**

If bit 3 (RPR) of parameter No.7301 is 1, relative coordinate values are preset by absolute coordinate values after the search in quick program restart. However, this preset function is invalid in direct jump method of quick program restart.

4.10.1 Suppress Motion of Quick Program Restart

Overview

Suppress motion is a function which prohibits the approach to the start point of the specified restart block. In the conventional specification, the approach is performed to the start point of the restart block. To perform the approach to the end point of the restart block, the next block must be specified as the restart block. This function enables the tool to be moved to the end point of the specified restart block.
For example, when a block for approach is inserted in the machining program, the block for approach can be specified as the restart block and the tool can be moved to the end point of the block.

NOTE
An approach block is used for moving the tool closer to the workpiece from an arbitrary position (such as a tool change position). Normally, all axes used for the interrupted machining are specified. An approach block may consist of more than one block. In this case, select the first approach block as the restart block.

Example) Operation example

```
O1234;
:
N100 G91G28Z0;
N110 G28X0Y0;
N120 T01M6;
N130 G90G00X-50.0Y0;          Approach block for the X- and Y-axes
N140 Z-10.0;                  Approach block for the Z-axis
N150 G41G01X-50.0F1000.0D1;
N160 Y50.0;
:
```

When N130 is specified as the restart block, the tool moves from the current position to the end point of N130.

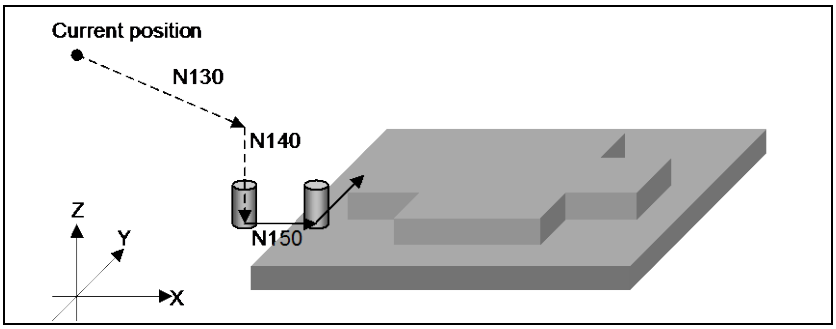


Fig. 4.10.1 (a) Operation during suppress motion

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

Explanation

Setting bit 5 (SPR) of parameter No. 11250 to 1 enables suppress motion to be selected.

- Operation method

1. On the program restart information screen, press soft key [(OPRT)].

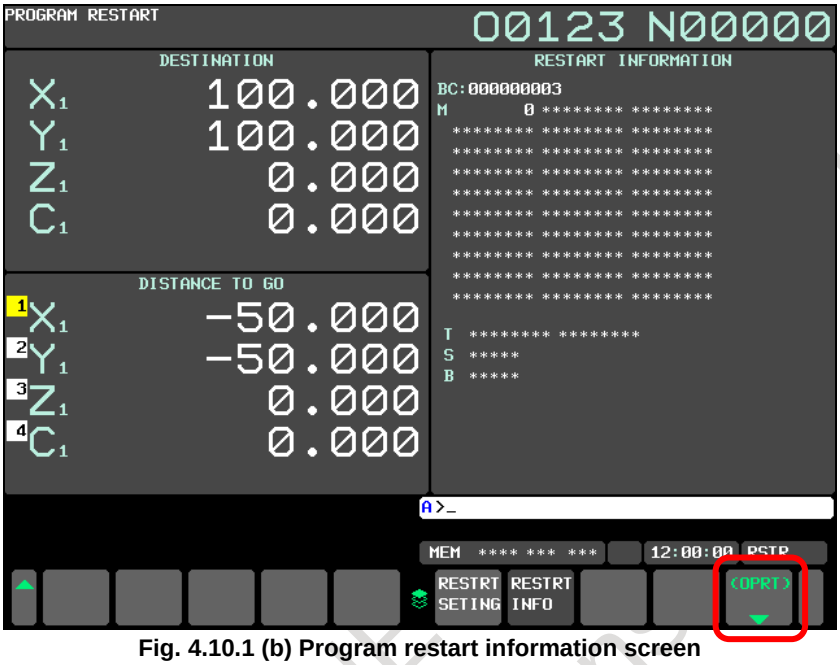


Fig. 4.10.1 (b) Program restart information screen

2. The following soft keys appear. Press the continuous menu key.

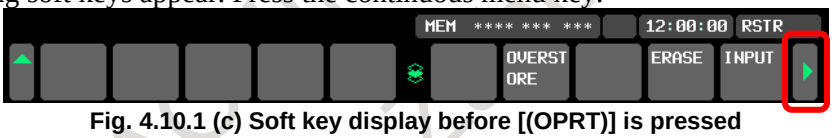


Fig. 4.10.1 (c) Soft key display before [(OPRT)] is pressed

3. Soft keys [SET SUPRES] and [CLEAR SUPRES] appear.

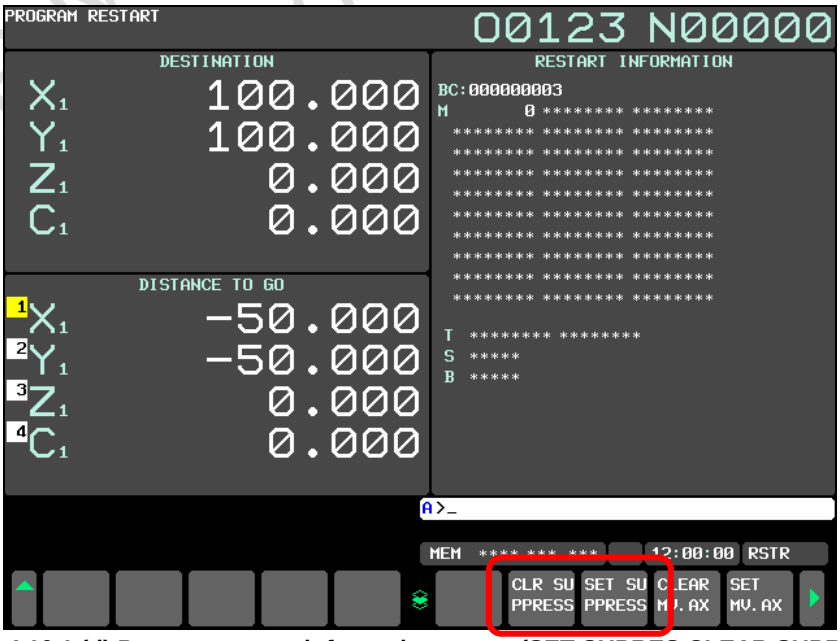


Fig. 4.10.1 (d) Program restart information screen(SET SUPRES,CLEAR SUPRES)

FRYER MACHINE TOOLS INC

4. To enable suppress motion, press soft key [SET SUPRES]. The number displayed on the left of each address changes to "--", the values under DESTINATION change to the current coordinates, and the values under DISTANCE TO GO change to 0.000.

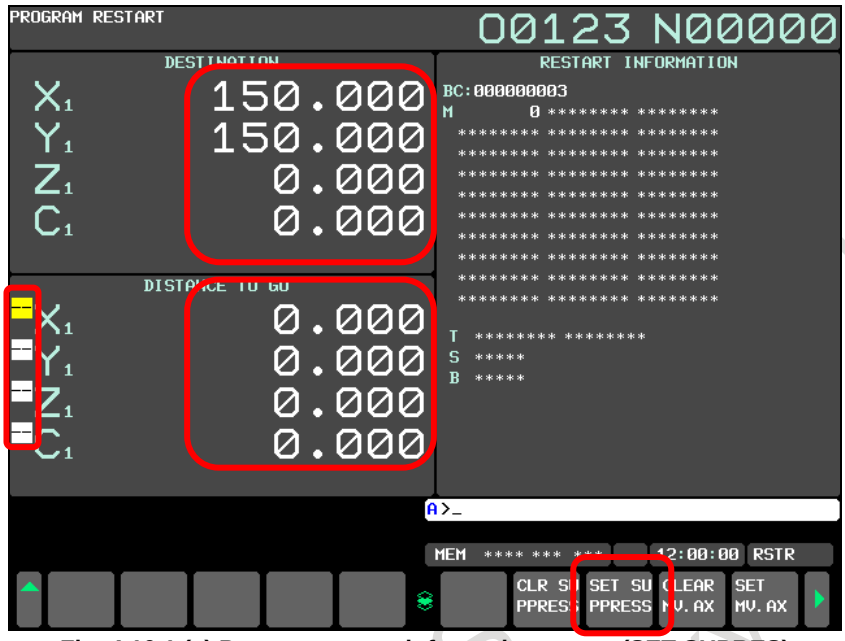


Fig. 4.10.1 (e) Program restart information screen(SET SUPRES)

5. To disable suppress motion, press soft key [CLEAR SUPRES]. The original numbers and values are displayed again.

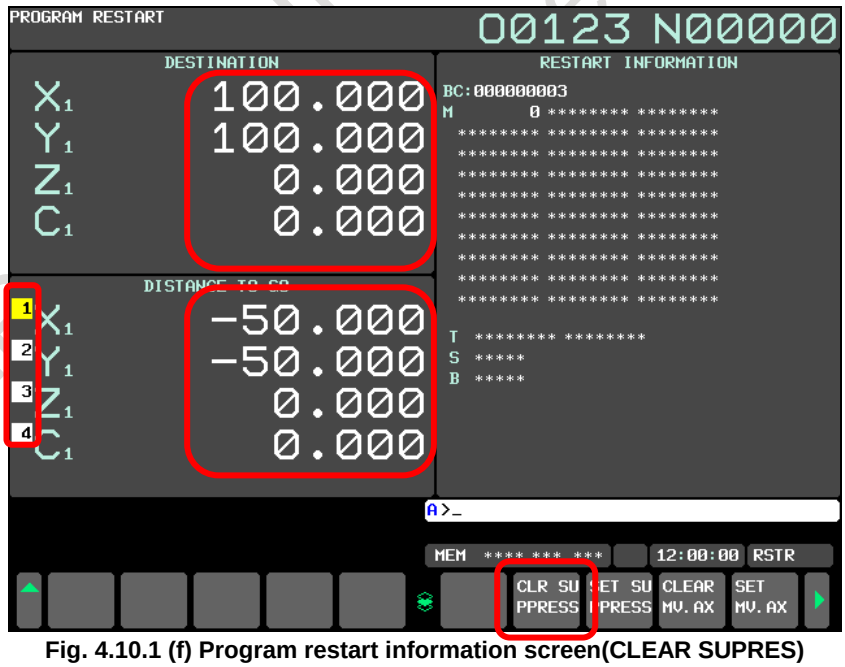


Fig. 4.10.1 (f) Program restart information screen(CLEAR SUPRES)

- Block for which suppress motion is available

To perform suppress motion, select a block which satisfies conditions (1) to (3) below:

(1) An absolute command is specified in the block.
If an incremental command is specified in the restart block, alarm PS1930, “ILLEGAL COMMAND AFTER RESTART”, occurs.

(2) The G00 or G01 command is specified in the block.
If a command other than G00 or G01 is specified in the block, alarm PS1930 occurs.

(3) The block is not a modal block listed below.
If the restart block is a modal block listed below, suppress motion is not available even when the G00 or G01 command is specified in that block. If suppress motion is performed for a modal block listed below, alarm PS1931, “ILLEGAL MODE AFTER RESTART”, occurs.
- Programmable mirror image
- 3-dimensional coordinate system conversion
- Coordinate system rotation
- Tilted working plane indexing
- Tilted working plane indexing by tool axis direction
- Tilted working plane indexing (incremental multiple command)
- Scaling
- Polar coordinate interpolation
- Normal direction control
- Canned cycle for drilling
- Canned cycle

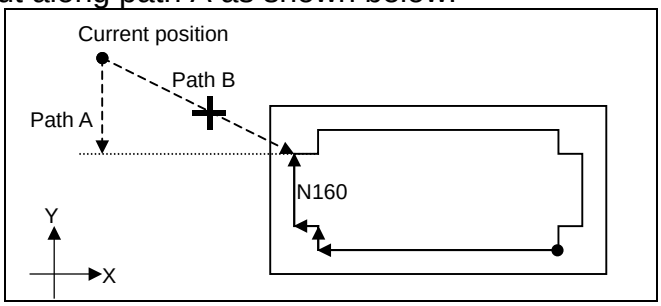
- Saving the suppress motion status

By setting bit 6 (SAV) of parameter No. 11250 to 1, the suppress motion status can be saved. With this function, once suppress motion setting is performed, the setting can be omitted after that.
The suppress motion status can be checked using parameter No. 7310. When suppress motion is used, a negative value is set in parameter No. 7310.
For an axis for which an invalid value is set in parameter No. 7310, the suppress motion status is not saved.

Caution

⚠ CAUTION

When suppress motion is used, the tool moves to the end point of the restart block only along the axis specified in the restart block. For this reason, the tool does not move to the end point of the restart block along any axis that is not specified in the restart block.
When N160 in the example is specified for the restart block, the tool moves not along path B, but along path A as shown below:



Along any axis that is not specified in the restart block, move the tool to a position where it will not hit anything when it moves during restart operation, in the manual mode in advance.

FRYER MACHINE SYSTEMS INC
Rob Zaherosky

Note

NOTE

1 While the tool is moving along another axis, suppress motion cannot be set or canceled. If suppress motion is set or canceled while the tool is moving along another axis, the warning message, “COMMAND ILLEGAL USE”, is displayed.

2 Performing approach for each arbitrary axis in the suppress motion status cancels the suppress motion.

4.10.2 Quick Program Restart for a Machining Cycle

When the MANUAL GUIDE *i* function is enabled, program operation can be restarted in the middle of a hole machining cycle of MANUAL GUIDE *i*.

You can specify the restart position of a hole machining cycle on the detailed machining cycle restart information screen to restart a program easily.

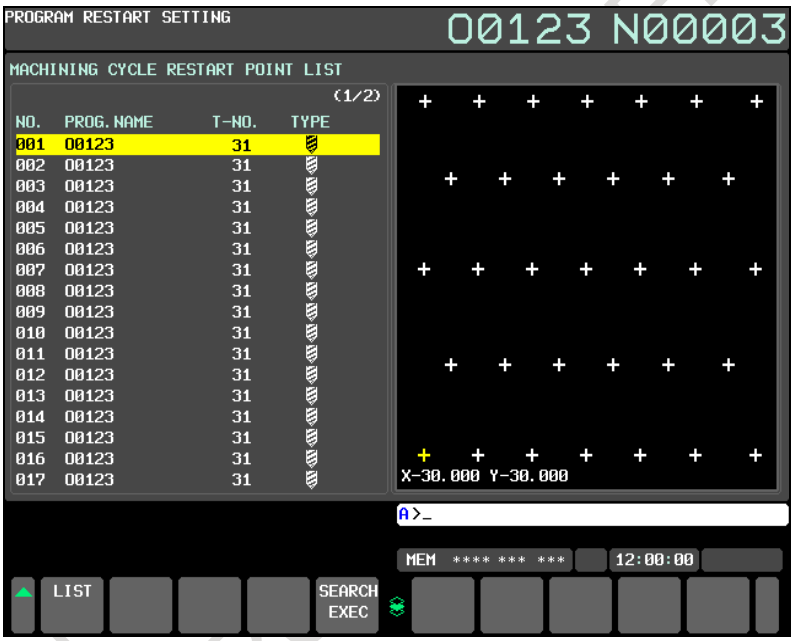


Fig. 4.10.2 (a) Detailed machining cycle restart information screen

NOTE

This function is available only for a hole machining cycle.
(This function is not available for Multi-path simultaneous program restart.)

FRYER SYSTEMS INC

Procedure for quick program restart for a machining cycle

Procedure

The procedure for quick program restart for a hole machining cycle is described below:

1. In the MEM or DNC mode, execute the automatic operation of a hole machining cycle of MANUAL GUIDE *i*.

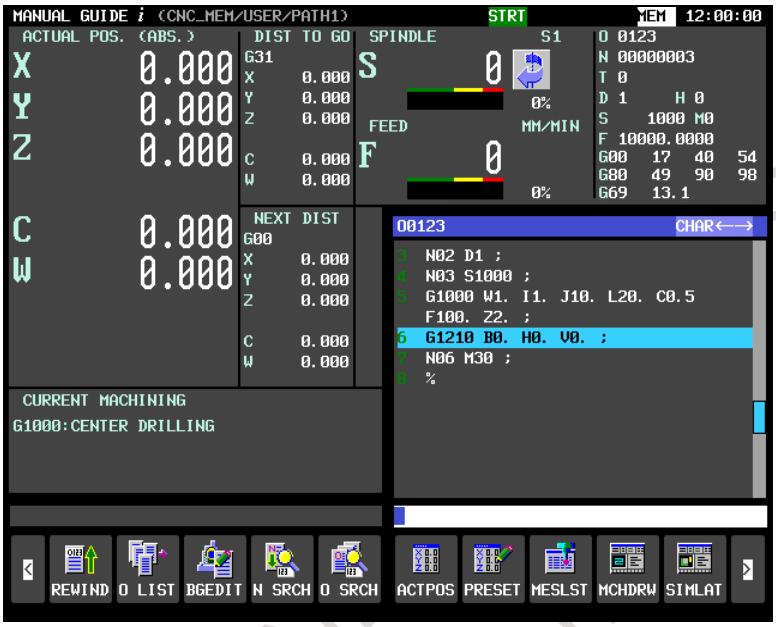


Fig. 4.10.2 (b) Base screen of MANUAL GUIDE *i* (1)

2. Interrupt or terminate the machining cycle, retract the tool, and replace it with a new one. Then, on the base screen, press soft key [RESTRT].

Fig. 4.10.2 (c) Base screen of MANUAL GUIDE *i* (2)

3. The CNC restart information list screen appears. Then, press soft key [MACHIN CYCLE].

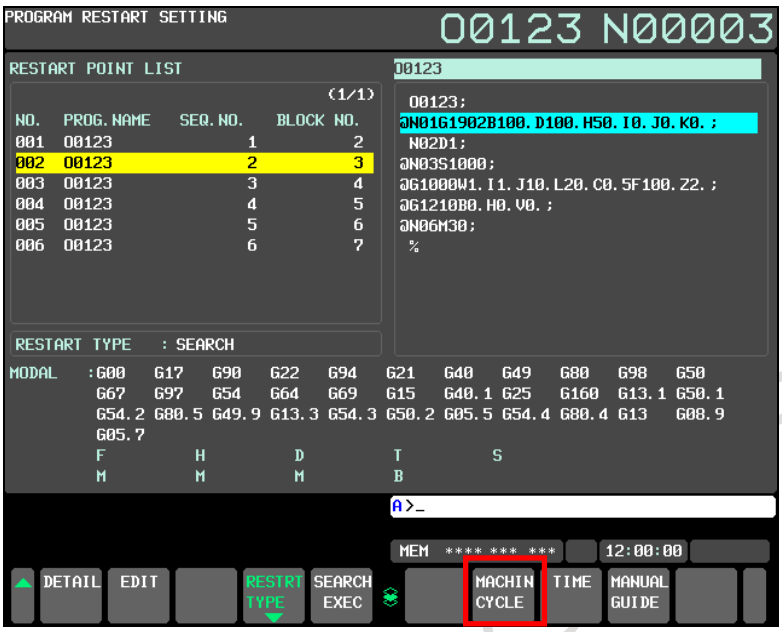


Fig. 4.10.2 (d) CNC restart information list screen

> **NOTE**
> Pressing soft key [MANUAL GUIDE] returns to the base screen of MANUAL GUIDE *i*. (when bit 2 (SMR) of parameter No. 11351 is set to 1)

4. The CNC detailed machining cycle restart information screen appears. The screen displays a list of restart points (up to 50 points) for the hole machining cycle on the left side and a preview window for the positions of the holes (each hole position is indicated with "+") on the right side. Move the cursor to the restart point of the hole position for restarting the cycle.

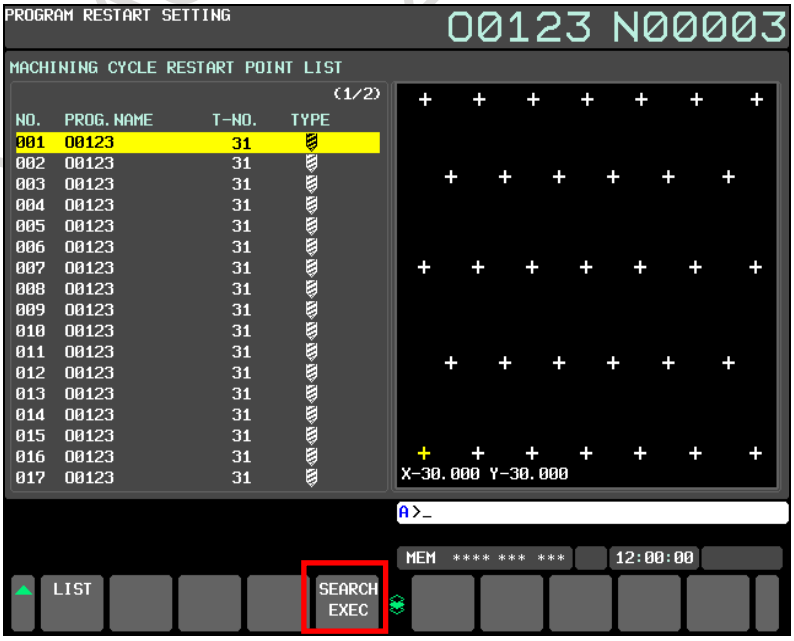


Fig. 4.10.2 (e) CNC detailed machining cycle restart information screen

FRYER MACHINE SYSTEMS INC Zahensky

5. Press soft key [SEARCH EXEC]. The machine takes a dry run to the restart point. At the same time, the program restart screen appears.

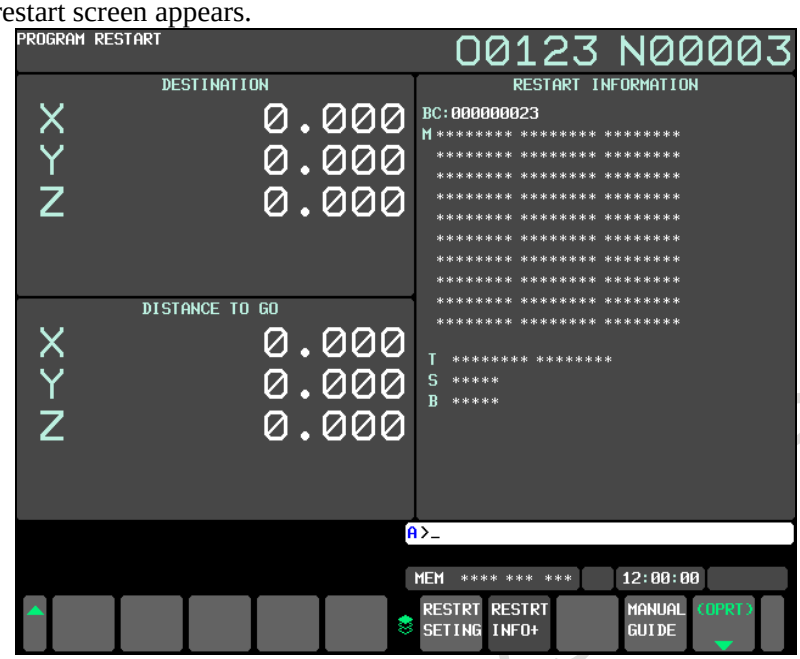


Fig. 4.10.2 (f) CNC program restart screen

> **NOTE**
> Pressing soft key [MANUAL GUIDE] returns to the base screen of MANUAL GUIDE *i*. (when bit 2 (SMR) of parameter No. 11351 is set to 1)

6. Then, press the cycle start button. The tool moves from the current tool position to the restart point along axes in the order specified in parameter No. 7310 at the dry run feedrate. After the tool moves until the values under DISTANCE TO GO become 0, machining is performed from the restart point.

Explanation

- CNC detailed machining cycle restart information screen

When soft key [MACHIN CYCLE] is pressed on the CNC restart information list screen, the following screen appears:

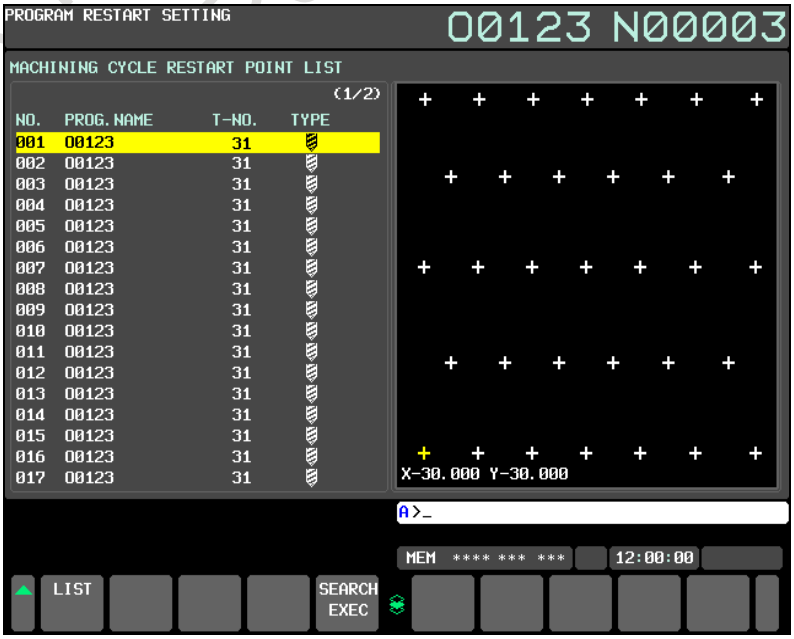


Fig. 4.10.2 (g) CNC detailed machining cycle restart information screen

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

The CNC detailed machining cycle restart information screen has the following soft keys.

Table 4.10.2 (a) Soft keys on the CNC detailed machining cycle restart information screen

Soft key	Description
LIST	Displays the CNC restart information list screen.
SEARCH EXEC	Performs a dry run to the restart point of the specified hole position and displays the CNC program restart screen.

This screen displays the following items.

Table 4.10.2 (b) Items displayed on the CNC detailed machining cycle restart information screen

Item	Description
NO.	Serial number assigned to each restart data item displayed in the list (up to three characters)
PROG. NAME	Execution program name (up to nine characters) If the program name consists of 10 or more characters, the first seven characters and "..." are displayed.
T-NO	T code or tool number (up to eight characters) (NOTE 1)
TYPE	Tool icon indicating a machining type. Each tool icon indicates a machining type as follows: : Center drilling or drilling : Boring, fine boring, or back boring : Tapping or thread milling : Reaming
Preview	"+" indicates the hole position of each machining profile. The hole position of the selected restart point is displayed in the cursor color (initial value: Yellow). The coordinates of the hole position are displayed at the bottom of the screen. (NOTE 2)

NOTE

1 The value displayed for T-NO differs depending on the settings of bit 2 (PCT) of parameter No. 3108 and bit 0 (DHN) of parameter No. 11320 as listed in the table below.

Table 4.10.2 (c) T-NO-related parameters

Bit 2 (PCT) of parameter No. 3108	Bit 0 (DHN) of parameter No. 11320	T-NO value
1	1	Spindle tool number (HD.T)
1	0	
0	1	
0	0	Modal T code When T0102 is specified in a lathe system, not compensation number 02, but tool number 1 is displayed. For a machine for which a tool in the standby position is specified, the number of the tool in the standby position is displayed for T-NO.

FRYER MACHINE SYSTEMS INC
Rob Zaherecky

NOTE

2 Color numbers specified on the CNC color setting screen are used for the preview display colors.

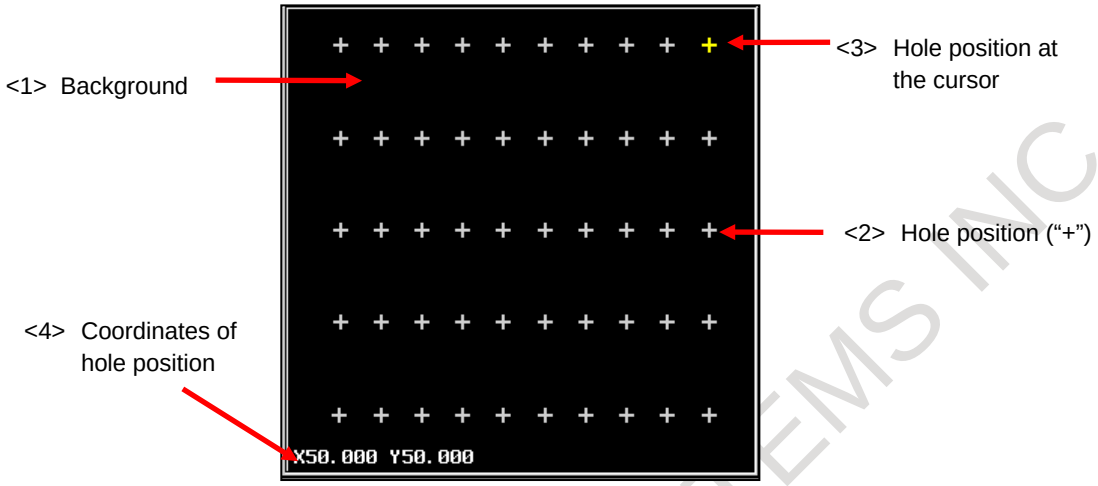


Fig. 4.10.2 (h) Preview screen configuration

Table 4.10.2 (d) Preview display colors

Preview display item	Color number	Initial value	Mainly used for
<1> Background	0	(fixed)	Soft key frame
<2> Hole position ("+")	15		Background color
<3> Hole position at the cursor	4		Cursor
<4> Coordinates of hole position	13		Alter possible data background

- Restart position of hole machining

The restart position of hole machining depends on "return mode (I)", input data of a hole machining cycle.

There are the following three return modes. The restart position is the block immediately before point R or I.

(1) [point I - 1]:

The tool returns to the point R level before moving from a hole to another hole. It returns to the point I level at the end of machining.

For the first hole position, machining restarts from the block immediately before point I (start point of operation 1). For the second or subsequent hole position, machining restarts from the hole position immediately before point R (start point of operation 5).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

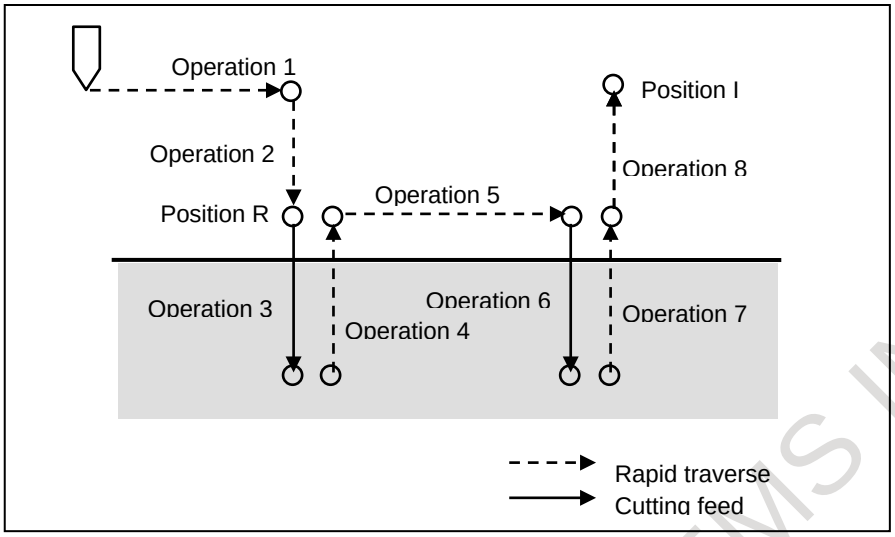


Fig. 4.10.2 (i) Machining operation for [point I - 1]

(2) [point I - 2]:
The tool returns to the point I level before moving from a hole to another hole, including the last operation.
For the first hole position, machining restarts from the block immediately before point I (start point of operation 1). For the second or subsequent hole position, machining restarts from the hole position immediately before point I (start point of operation 6).

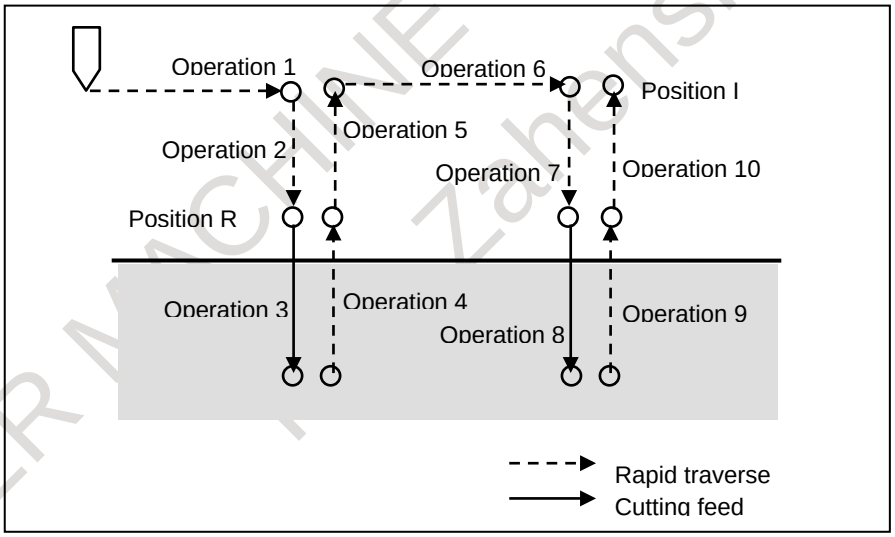


Fig. 4.10.2 (j) Machining operation for [point I - 2]

(3) [point I - 3]:
The tool returns to the point R level before moving from a hole to another hole, including the last operation.
For the first hole position, machining restarts from the block immediately before point R (start point of operation 2). For the second or subsequent hole position, machining restarts from the hole position immediately before point R (start point of operation 5).

FRYER MACHINE SYSTEMS INC
Zahensky

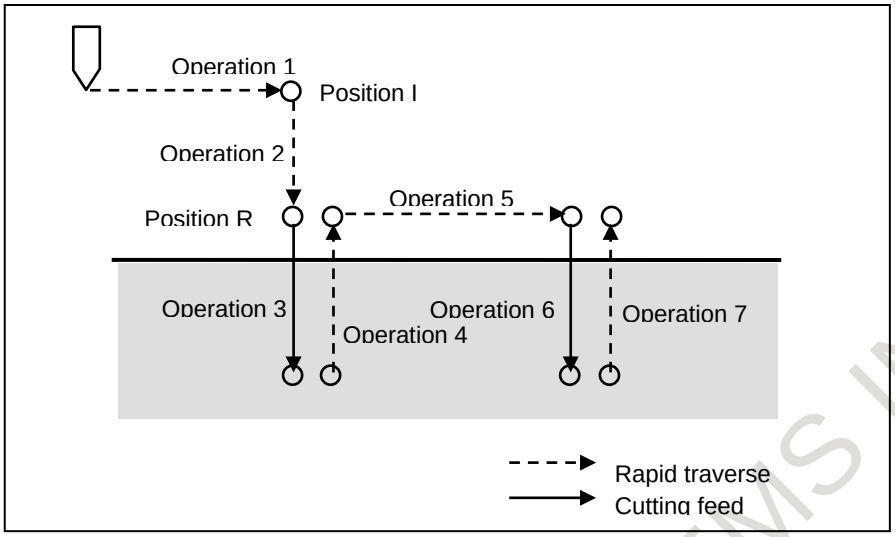


Fig. 4.10.2 (k) Machining operation for [point I - 3]

- Warning

Related warning messages are listed below.

Table 4.10.2 (e) Related warning messages

Message	Description
EXECUTING OPERATION	Soft key [RESTRT] was pressed during automatic operation. After the completion of automatic operation, press soft key [RESTRT].
IT IS NOT EXECUTED	No hole machining cycle has been operated. After performing automatic operation, press soft key [MACHIN CYCLE].

Notes

(1) This function is available only for a hole machining cycle. If an attempt is made to perform quick program restart for a machining cycle for other than hole machining, warning message “IT IS NOT EXECUTED” is displayed.
(2) Machining can be restarted only from an executed block. Hole position information for restart is stored when each block is executed. The memory in which the information is stored is called program restart memory.
(3) Information on up to 50 restart points can be stored in the program restart memory. For a multi-path system, information on up to 50 restart points are stored for each path.
(4) If the program restart memory becomes full, hole position information is deleted from the older one and the information on a new hole position is overwritten.
(5) The program restart memory is kept stored after power-on.
(6) The program restart memory is cleared when automatic operation is performed in the MEM or DNC mode after a reset.
(7) After execution, if the machining program is modified and program restart is performed, the restart position is changed. If the machining program is modified, be sure to operate the program, then perform program restart.
(8) The restart position of a hole machining cycle is the block immediately before point R or I.
(9) For a multi-path system, specify the restart position so that the waiting M codes for each path are consistent.
(10) Detailed machining cycle restart information screen is not displayed in Multi-path simultaneous program restart mode.

FRYER MACHINE SYSTEMS INC
Rod Zaherskv

4.10.3 Quick Program Restart for Multi Path System

Overview

When the machining was restarted by quick program restart in multi path system, it has been necessary to operate the program restart every one path.
In this function, when the program restart is operated in one path, the program restart is automatically operated as for other paths. As a result, the program restart can be executed by an easy operation in the machine with a multi path system.

Explanation

When the program restart is operated in certain one path, it is automatically operated in the multi path system as for other paths.
This operation is called "multi path quick program restart" ("multi path restart" at the following).
As for this function, only the program restart of the search method is possible. It cannot be used with the direct jump method.

Block that restart is possible

It is the following blocks that can restart the machining by the multi path restart.

(1) Block that is memorized in program restart memory, and was interrupted machining
(2) Block of waiting M code that was memorized in program restart memory, or arbitrary waiting M code specified on program restart setting screen

Fig.4.10.3 (a) Screen for specify waiting M code

(3) Block of arbitrary sequence number specified on program restart setting screen without using program restart memory

When the program restart is executed to the block like the above-mentioned in certain one path, an operation of the program restart is automatically started as for other paths. When an operation of the program restart is started, each path starts the search of a target block severally.

Operation when the block of above-mentioned (1) - (3) is specified is as follows.

(a) Block where machining was interrupted
It is searched for a block where interrupted of machining in each path.

FRYER SYSTEMS INC

(b) Block of the waiting M code
A block of the specified waiting M code is searched in each path, and the search stops in a block in which it was found. However, if the subprogram call is commanded in the same block as the waiting M code, the search stops at a first block of the subprogram. (Machining is restarted from a head of subprogram.)

(c) Block of arbitrary sequence number
A block of the specified sequence number is searched in each path, and the search stops in a block in which it was found. This method can be used in case that the same sequence number is put on the program in each path as a sign of the delimitation of the machining.

When the block that corresponds to above-mentioned (a)-(c) is specified as a restart block, the following method to specify are possible.

Table4.10.3 (a) Search that restart for multi path system is possible

Method to specify / Kind of restart block	Select in list or detail screen	Select in list or detail screen and edit	Direct specification by newly edit
(a) Block where machining was interrupted	Available	Not available	Not available
(b) Block of the waiting M code	Available	Not available	Available
(c) Block of arbitrary sequence number	Not available	Not available	Available

NOTE

1 Warning "CAN NOT RESTART IN MULTI PATH MODE" is displayed when multi path restart is executed with specifying blocks other than the above-mentioned (a)-(c).

2 No-wait signal can be used as well as normal automatic operation when searching a waiting M code. It is not searched in a path which the waiting M code is disable.
However, warning “CAN NOT RESTART IN MULTI PATH MODE” is displayed in the following cases.
- It is ignoring waiting in operation path.
- The path that can be searched by ignoring waiting is below one path.

3 When the alarm is issued in a path executing the multi path restart, alarm PS0060, "SEQUENCE NUMBER NOT FOUND" or alarm PS0071, "DATA NOT FOUND" is issued in other paths of the same group.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Procedure of multi path restart

(1) Restart from the list screen

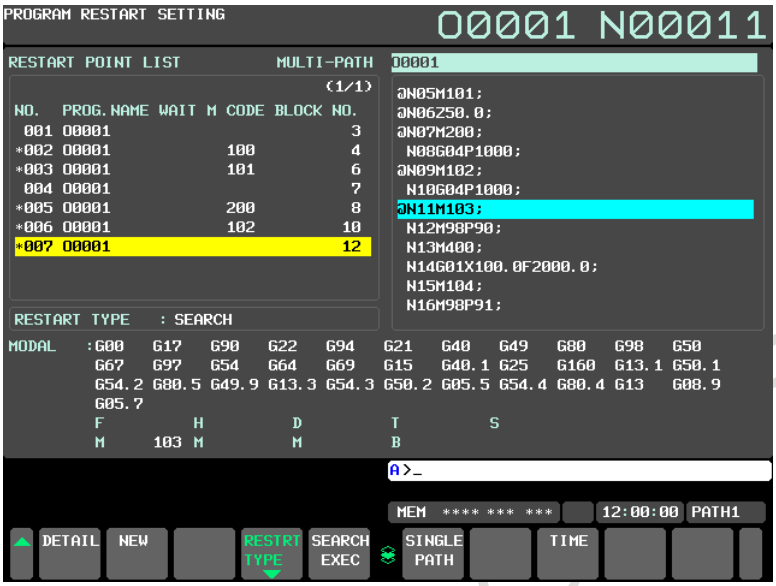


Fig.4.10.3 (b) List screen

1. Display the list screen.
2. Move the cursor to a restart point that is attached to the number of the sign of *.
3. Multi path restart is executed when soft key [SEARCH EXEC] is pressed.

NOTE

1 In the list screen, multi path restart can be executed only for the block where the program stopped at last or the block including waiting M code. * is displayed in the left of the list in the restart point that can be execute multi path restart. Warning "CAN NOT RESTART IN MULTI PATH MODE" is displayed when restarting for a restart point without *.

2 Alarm PS0071, "DATA NOT FOUND" is issued when the block of waiting M code is specified and the M code is not found in the program of each path.

3 In the path to which multi path restart is set, the waiting M code is displayed in stead of sequence number on the list screen. A sequence number is displayed on the list screen when the single path restart mode is selected or the multi path restart is not set.

FRYER MACHINE SYSTEMS INC Rob Zalensky

(2) Restart from detailed restart point screen

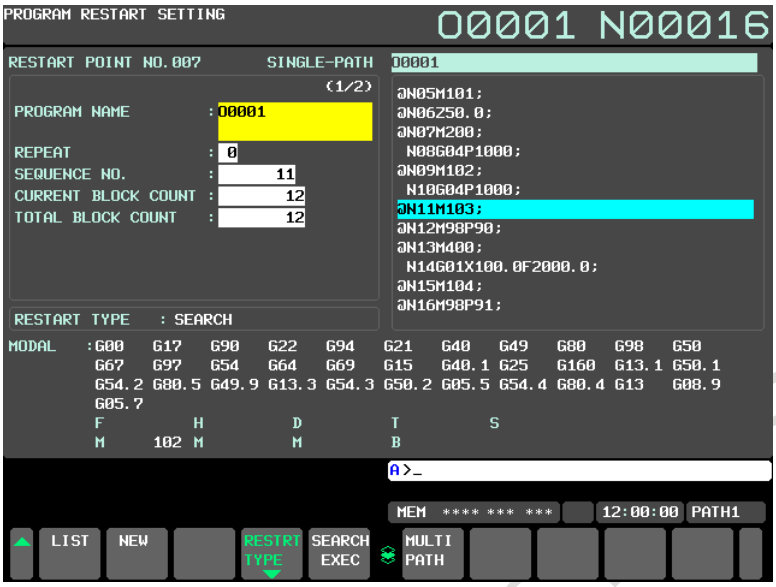


Fig.4.10.3 (c) Detail screen

1. Move the cursor to restart point that has adhered to number * in the restart point list screen, and press soft key [DETAIL].
2. Detailed restart point screen is displayed.
3, Multi path restart is executed when soft key [SEARCH EXEC] is pressed.

NOTE

1 When search is executed in restart point that cannot execute multi path restart, warning "CAN NOT RESTART IN MULTI PATH MODE" is displayed.

2 The waiting M code is displayed in a path that multi path restart is set. The waiting M code is not displayed when single path restart mode is selected or the multi path restart is not set.

(3) Restart from new restart point screen

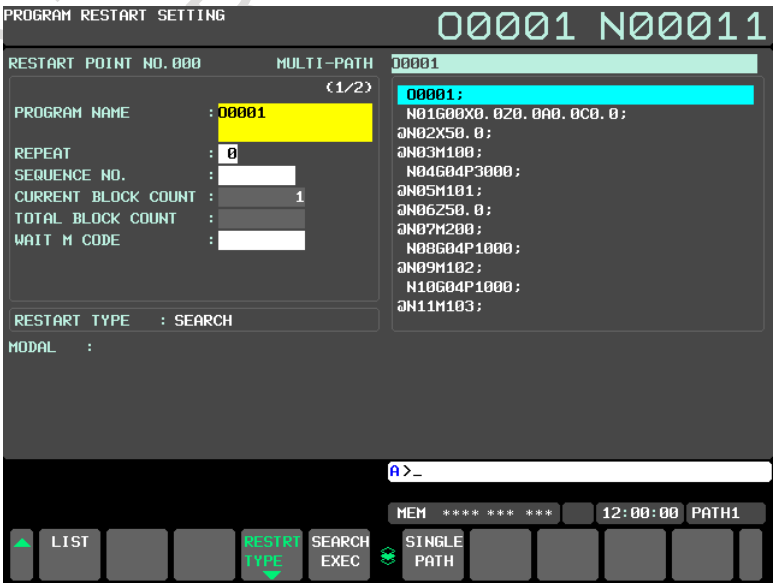


Fig.4.10.3 (d) New restart point screen

FRYER MACHINE SYS INC Bob Zaremsky

1. Press soft key [NEW] in the list screen.
2. New restart point screen is displayed.
3. Input either a waiting M code or a sequence number.
4. Multi path restart is executed when soft key [SEARCH EXEC] is pressed.

NOTE

1 If the restart is executed when the input waiting M code or the sequence number is not in the program of each path, alarm PS0060, "SEQUENCE NUMBER NOT FOUND" or alarm PS0071, "DATA NOT FOUND" is issued.
2 In the new restart point screen, it can not specify the restart block by block number or program display.
3 The waiting M code can be specified in the path which multi path restart is set. A waiting M code is not displayed when single path restart mode is selected or the multi path restart is not set.

Operation after search is completed

The target path of dry run movement after search is specified by bit 0 (MPD) of parameter No. 7331.

- When bit 0 (MPD) of parameter No. 7331 is set 0 :
  The order in the path which dry run movement is executed after search can be specified by parameter No.7338.
  It enters the state of STRT after the axes move to the machining restart position with dry run and the machining program becomes waiting for execution.
  When all paths of the same group enter the state of STRT, the waiting state is released and the machining program is executed.
  In the path that 0 is set to parameter No.7338, the dry run movement and the machining program are executed without waiting for other path.
- When bit 0 (MPD) of parameter No. 7331 is set 1 :
  Dry run is executed in the path that has been selected by the path selection signal.
  It is necessary to do cycle start in each path whenever the selected path is changed by the path selection signal.
  It enters the state of STRT after the axes move to the machining restart position with dry run and the machining program becomes waiting for execution.
  When all paths of the same group enter the state of STRT, the waiting state is released and the machining program is executed.

⚠CAUTION

1 In case of bit 0 (MPD) of parameter No. 7331 is set 0, when one path enters in the state of STRT, the movement of a dry run of next path is automatically started.
2 In case of parameter bit 0 (MPD) of parameter No. 7331 is set 1, the path has not been selected by the path selection signal does not accept the cycle start to a dry run movement after search.
3 Program restart under way signal becomes "1" until the movement of a dry run in all paths that belong to the same group is completed. However, when the movement of a dry run in the path is completed, signal becomes "0" as for the path set bit 0 (MPD) of parameter No. 7331 to 0 and parameter No.7338=0.

⚠ WARNING

The path of the same group to restart a multi path begins the execution of the machining program after waiting for other paths, but it does not synchronize with the path that is not the same group.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Operation procedure of dry run movement after restarting

When bit 0 (MPD) of parameter No. 7331 is set 0:

Table4.10.3 (b) Operating procedure when the order in path of dry run movement is specified

Operation sequence	Operation	State after operation		
		Path1	Path2	Path3
1	Cycle start for all paths	Dry run move STRT	-	-
2	-	STRT	-	Dry run move STRT
3	-	STRT	Dry run move STRT	STRT
4	-	Automatic operation restart	Automatic operation restart	Automatic operation restart

*It is assumed that the order in the path that moves by a dry run is path1 → path3 → path2.
STRT : Waiting for execution of machining program in STRT state

NOTE
1 In the single block operation, the automatic operation restarts at the same time in all paths when cycle start is done after single block stop after the movement of the last axis in path2 is completed.
2 In the continuous operation, operation sequence 1-4 is executed without stopping.

When bit 0 (MPD) of parameter No. 7331 is set 1:

Table4.10.3 (c) Operating procedure when the order in path of dry run movement is specified by the path selection signal

Operation sequence	Operation	State after operation		
		Path1	Path2	Path3
1	Select path1 by the path selection signal	-	-	-
2	Cycle start	Dry run move STRT	-	-
3	Select path3 by the path selection signal	STRT	-	
4	Cycle start	STRT	-	Dry run move STRT
5	Select path2 by the path selection signal	STRT	-	STRT
6	Cycle start	STRT	Dry run move STRT	STRT
7	-	Automatic operation restart	Automatic operation restart	Automatic operation restart

STRT : Waiting for execution of machining program in STRT state

⚠CAUTION
1 When all paths enter the STRT state, operation sequence 7 is begun.
2 Whenever the path is selected by the path selection signal, it is necessary to do the cycle start even if continuous operation.

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

Single path restart

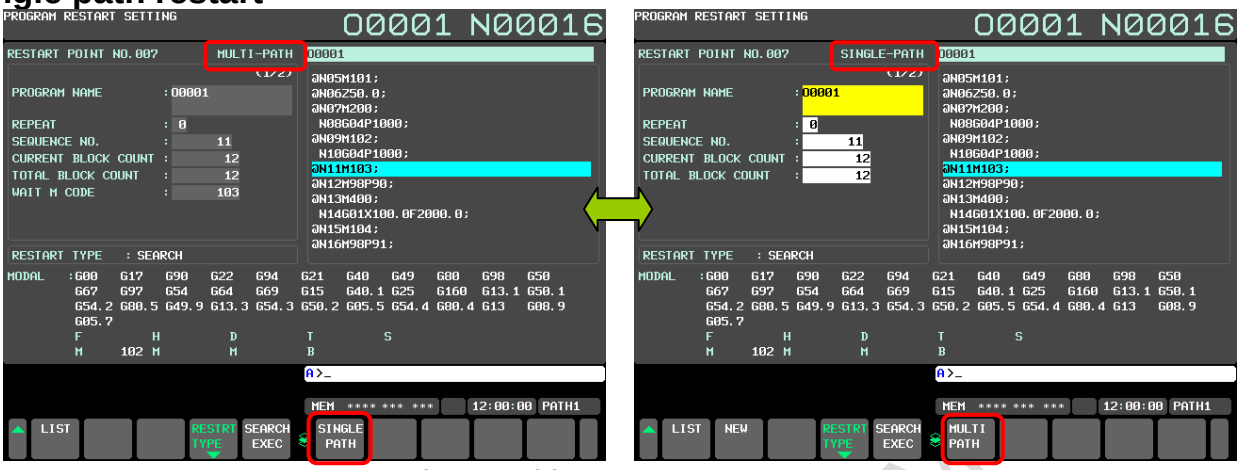


Fig. 4.10.3 (e) Program restart screen

When a soft key [SINGLE PATH] is pushed, it is change in the single path restart mode.
In the single path restart mode, it can be done the program restart operation only in the path under the display.
"SINGLE PATH" is displayed on the right of a left screen like showing in Fig. 4.10.3 (e) the single system restart mode inside.
The single path restart mode is canceled by either of the following operations.

- Program restart operation is executed.
- Another screen was selected.
- Soft key [MULTI PATH] is pushed.

Limitation

- Miscellaneous function

The miscellaneous function under the search is not output to PMC.
Please restore the state of the miscellaneous function by MDI operation before restarting the machining after the search ends.

- Synchronous / Composite / Superimposed control

On/off command of the synchronous control, the composite control, and the superimposed control by the miscellaneous function or G code under the search is invalid.
Please restore the state of the synchronous control, the composite control, and the superimposed control by MDI operation before restarting the machining after the search ends.

- Flexible path axis assignment

In the command of the flexible path axis assignment under the search, the flexible path axis assignment is not operated.
Please restore the state of the flexible path axis assignment by MDI operation before restarting the machining after the search ends.

FRYER MACHINE SYSTEM
Bob Zahensky

4.10.4 Improvement of Quick program restart for non-machining program

Overview

In Quick Program Restart function, when a program is executed in MEM or RMT mode, restart block information is automatically memorized in the program restart memory.
After a machining program is interrupted, another program, which is not related to machining, is sometimes executed for the purpose of tool retract and the preparation of the machining in MEM or RMT mode. In this case, restart the block information is newly memorized, and the restart block information on the machining program memorized till then is overwritten. (It is not overwritten when a non-machining program is executed in MDI mode.)
By this improvement, restart block information is not memorized in the case of a certain specific program. As a result, even when programs of tool retract and machining preparation are executed during the machining, restart block information related to machining can be kept.

Explanation

When the parameter to activate this function is set and the following programs are executed, restart block information is not memorized.

(1) Program with specific program number
Specify the range of the program number by the parameter No.7335 and No.7336. When the program with the program number specified here is selected as a main program, restart block information is not memorized.
Moreover, restart block information of a subprogram called from this main program is not memorized. When the program specified here is called from the main program whose restart block information is memorized, restart block information is memorized.

NOTE
The folder cannot be used. Restart block information is not memorized when the program number is within the range, regardless of a folder.

(2) One touch macro call
Restart block information is not memorized by setting bit 0 (OMC) of parameter No. 7330 to 1 while a one-touch macro is being executed.

Example

If parameter No.7335=7900 and No.7336=20 are set when program O7900-O7919 is used as a program for the tool retract, the relation between the main program number and the memorization of restart block information is as follows.

Main program number	Restart block information
O0001 — O7988	Memorized
O7900 — O7919	Not memorized
O7920 — O9999	Memorized

For instance, when O0001 is executed, restart block information about O0001 is memorized.
Restart block information about O0001 is kept, even if O0001 is interrupted on the way and O7901 for tool retract is selected as a main program and executed.
The program can be easily restarted by using kept restart block information if O0001 is selected again as a main program after the tool retract program is executed.
Restart block information about O0001 is overwritten, when O0001 is interrupted on the way and O0002 is newly executed as a main program, and restart block information on O0002 is memorized.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

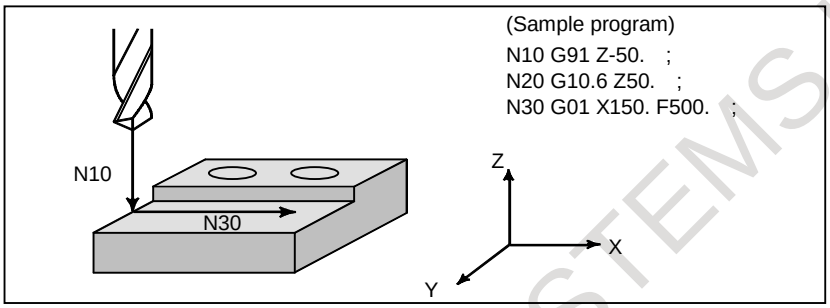
4.11 TOOL RETRACT AND RECOVER

The tool can be retracted from a workpiece to replace the tool, if damaged during machining, or to check the status of machining. Then, the tool can be returned to restart machining efficiently.

Procedure for tool retract and recover

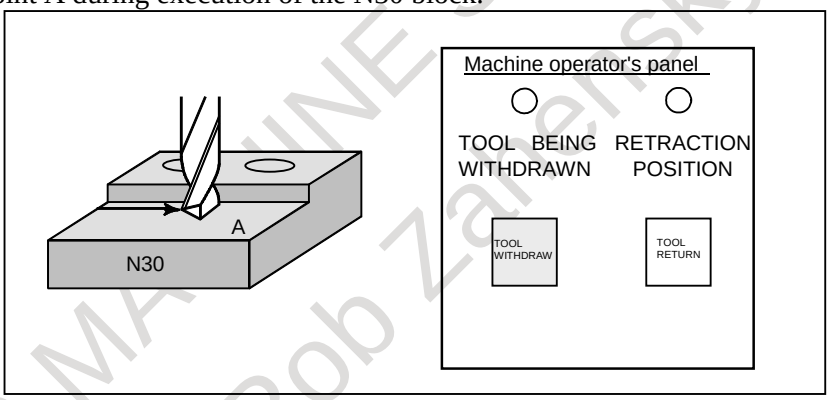
Procedure1 - Programming

Specify a retraction axis and distance in command G10.6IP_ beforehand. In the sample program below, the N20 block specifies that the Z-axis is the retraction axis and the retraction distance is to be 50 mm.

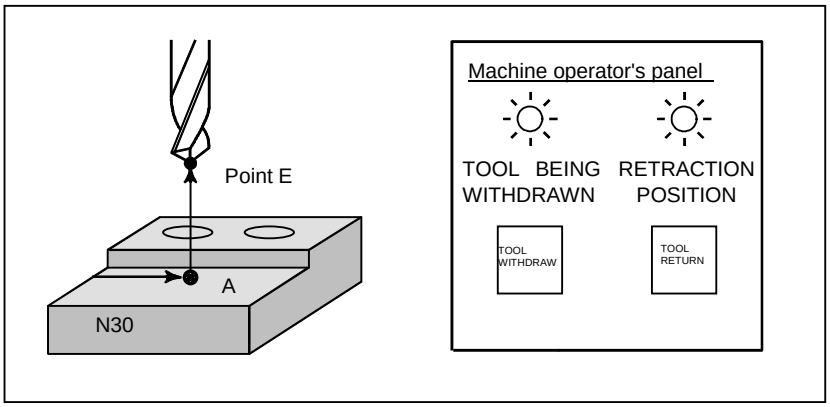


Procedure 2 - Retract

Suppose that the TOOL WITHDRAW switch on the machine operator's panel is turned on when the tool is positioned at point A during execution of the N30 block.



Next, the tool withdrawal mode is set and the TOOL BEING WITHDRAWN LED goes on. At this time, automatic operation is temporarily halted. The tool is then retracted by the programmed distance. If point A is the end point of the block, retraction is performed after automatic operation is stopped. Retraction is based on linear interpolation. The minimum feedrate among the parameter No.7042 values of moving axes is used for retraction. Upon completion of retraction, the RETRACT POSITION LED on the operator's panel goes on.



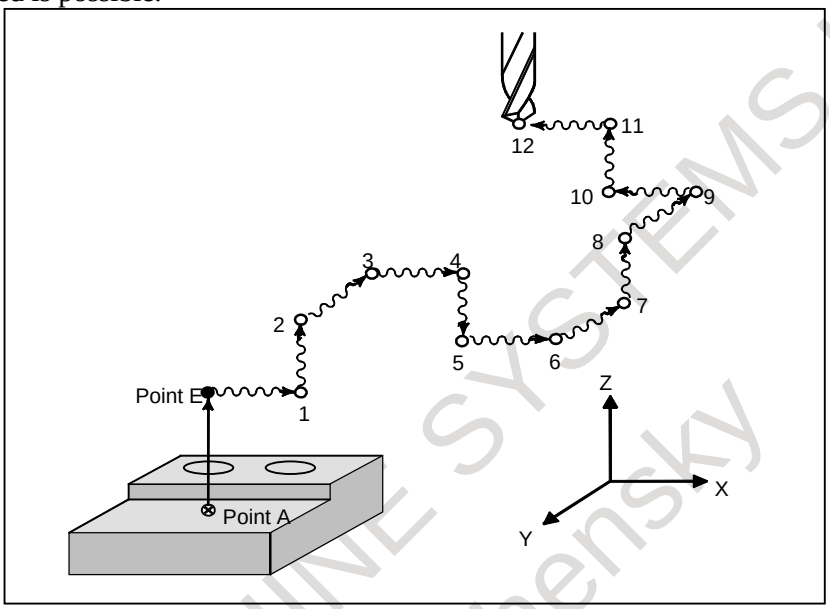
FRYER MACHINE SYSTEMS INC
Rob Zaremsky

During retraction, the screen displays PTRR and STRT.

- PTRR blinks in the field for indicating states such as the program editing status.
- STRT is displayed in the automatic operation status field.
- MTN is displayed in the field for indicating status such as movement along an axis.

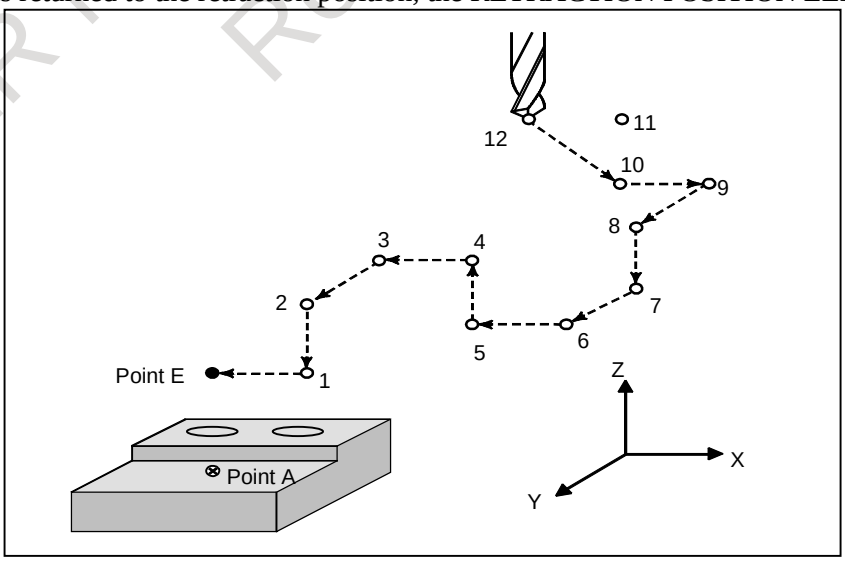
Procedure 3 - Withdrawal

Set the manual operation mode, then withdraw the tool. For manual operation, either jog feed incremental feed, or handle feed is possible.



Procedure 4 - Return

After withdrawing the tool and any additional operation such as replacing the tool, move the tool back to the previous retraction position. To return the tool to the retraction position, return the mode to automatic operation mode, then turn the TOOL RETURN switch on the operator's panel on then off again. The tool returns to the retraction position at the feedrate specified by parameter No.7042 for each axis.
When the tool has returned to the retraction position, the RETRACTION POSITION LED comes on.



FRYER MACHINE SYSTEMS INC
Rob Zahensky

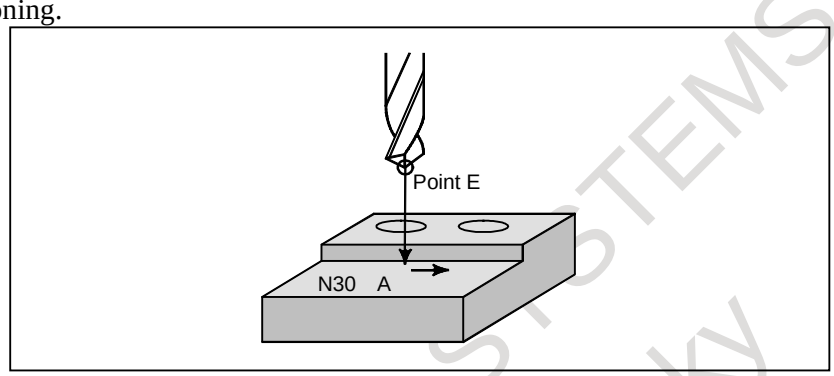
During return operation, the screen displays PTRR and MSTR.

```
MEM  MSTR MTN ***    12:00:00    PTRR
```

- PTRR blinks in the field for indicating states such as program editing status.
- MSTR is displayed in the automatic operation status field.
- MTN is displayed in the field for indicating states such as movement along an axis.

Procedure 5 - Repositioning

While the tool is at the retraction position (point E in the figure below) and the RETRACTION POSITION LED is on, press the cycle start switch. The tool is then repositioned at the point where retraction was started (i.e. where the TOOL WITHDRAW switch was turned on). Repositioning is based on linear interpolation. The minimum feedrate among the parameter No.7042 values of moving axes is used for repositioning.



Upon completion of repositioning, the tool withdraw mode is cancelled, and the TOOL BEING WITHDRAWN LED goes off and restart N30.

NOTE

1 If negative value or zero is set in parameter No.7042 of moving axes, retract, recovery and re-positioning are performed at the dry run feedrate.

2 Manual feedrate override signal is effective for the feedrate of parameter (No.7042).

Limitation

1 If the origin, presetting, workpiece origin offset value (or external workpiece origin offset value), or workpiece coordinate shift amount (for a lathe system) is changed after the retraction position is specified with G10.6 in the absolute mode, the change is not reflected in the retraction position. After such changes are made or the workpiece origin offset value (or external workpiece origin offset value) or workpiece coordinate shift amount (for a lathe system) is changed, respectively the retraction position with G10.6.

2 When retracting the tool manually in the tool withdrawal mode, do not use the machine lock, mirror-image, or scaling function.

3 Tool retract and recover cannot be performed along axes that are in the modes below.
- Superimposed control

⚠ WARNING

The retraction axis and retraction distance specified in G10.6 must be changed in an appropriate block according to the figure being machined. Be very careful when specifying the retraction distance; an incorrect retraction distance may damage the workpiece, machine, or tool.

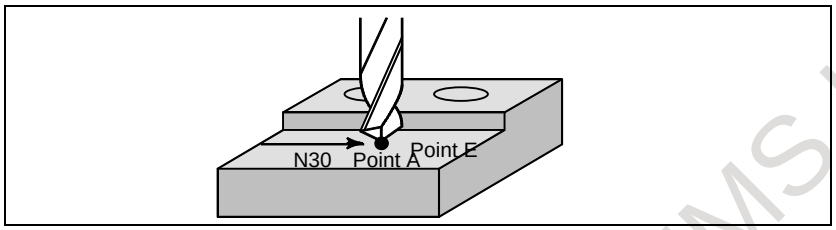
FRYER MACHINE SYSTEMS INC
Rob Zaremsky

4.11.1 Retract

Explanation

- When no retraction distance is specified

If no retraction distance or direction required for retraction are specified, retraction is not performed when the TOOL WITHDRAW switch on the operator's panel is turned on. Instead, the block being executed in automatic operation is interrupted (automatic operation is held or stopped). In this state, the tool can be withdrawn and returned.



- Retraction from the automatic operation hold or stop state

When the single block switch is turned on during automatic operation, or the TOOL WITHDRAW switch is turned on after the automatic operation hold or stop state is set by feed hold: Retraction is performed, then the automatic operation hold or stop state is set again.

- Stopping retraction

During retraction, feed hold operation is ignored. However, reset operation is enabled (retraction is stopped at reset). When an alarm is issued during retraction, the retraction is stopped immediately.

- Repositioning immediately after retraction

After retraction is completed, tool repositioning can be started without performing the withdraw and return operations.

4.11.2 Withdrawal

Explanation

- Axis selection

To move the tool along an axis, select the corresponding axis selection signal. Never specify axis selection signals for two or more axes at a time.

- Path memorization

When the tool is moved in manual operation along an axis, the control unit memorizes up to 20 paths of movements. If the tool is stopped after being moved along a selected axis and is then moved along another selected axis, the position where this switch takes place is memorized. After 20 paths have been memorized, the control unit does not memorize any additional switching points.

- Reset

Upon reset, memorized position data is lost and the tool withdraw mode is cancelled.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

4.11.3 Return

Explanation

- **Return path**

When there are more than 20 return paths, the tool first moves to the 20th position, then to the 19th position, then to the 18th position, and so forth until the retraction position is reached.

- **Single block**

The single block switch is enabled during return operation. If the single block switch is turned off, continuous return operation is performed. If the single block switch is turned off, the tool stops at each memorized position. In this case, return operation can be resumed by turning the TOOL RETURN switch on then off again.

- **Interruption of return operation**

When an alarm is issued during return operation, return operation stops.

- **Feed hold**

The feed hold function is enabled during return operation.

4.11.4 Repositioning

Explanation

- **Feed hold**

The feed hold function is disabled during repositioning.

- **Operation after completion of repositioning**

The operation after completion of repositioning depends on the automatic operation state present when the TOOL WITHDRAW switch is turned on.

(1) When automatic operation is being started
After completion of repositioning, the interrupted execution of the block is resumed.

(2) When automatic operation is held or stopped
After completion of repositioning, the tool stops once at the repositioned point, then the original automatic operation hold or stop state is set. When the cycle start switch is pressed, automatic operation is resumed.

4.11.5 Tool Retract and Recover for Threading

Explanation

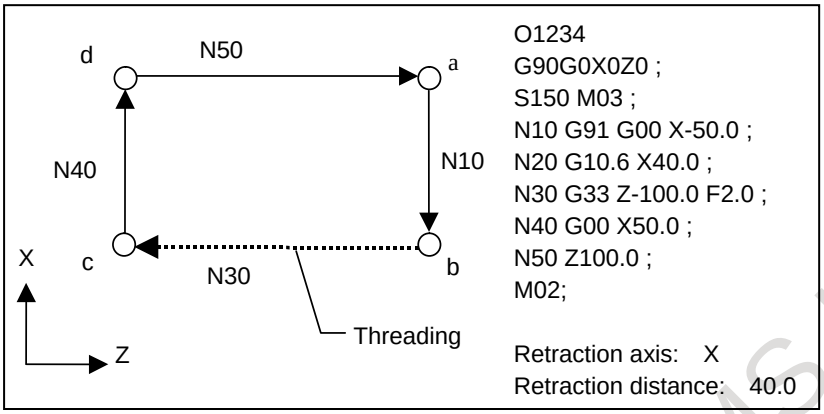
- **Differences between ordinary tool retract and recover and tool retract and recover for threading**

(1) During retraction, chamfering is performed between the specified retraction axis and threading axis.
(2) After retraction, one block that does not specify threading is executed and the tool stops.
(3) When the major axis for threading is specified as the retraction axis, retraction is not performed by turning the TOOL WITHDRAW switch on. In this case, after a block that does not specify threading is executed, an alarm PS0429, “ILLEGAL COMMAND IN G10.6” is issued and the tool stops.
(4) As repositioning, the tool is returned to the position specified in the first block that does not specify threading.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Operation procedure

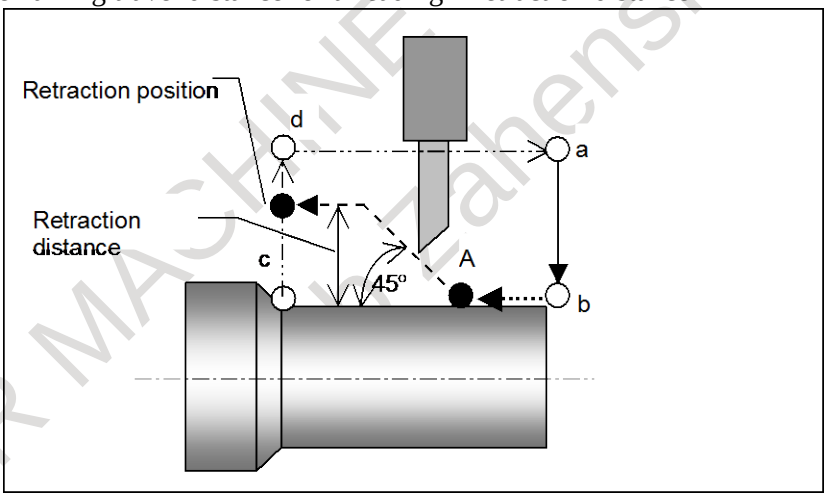
(1) Specify a retraction axis and retraction distance in command “G10.6IP- -;”.



(2) Turn the TOOL WITHDRAW switch during the execution of a threading command block.

(3) The tool withdrawal mode is set and retraction is performed. Chamfering at 45 degrees is performed between the retraction axis and major axis for threading using the retraction distance as the chamfering amount during retraction.
Details of retraction differ depending on whether the remaining travel distance for the threading command is smaller than the retraction distance when the TOOL WITHDRAW switch is turned on as follows:

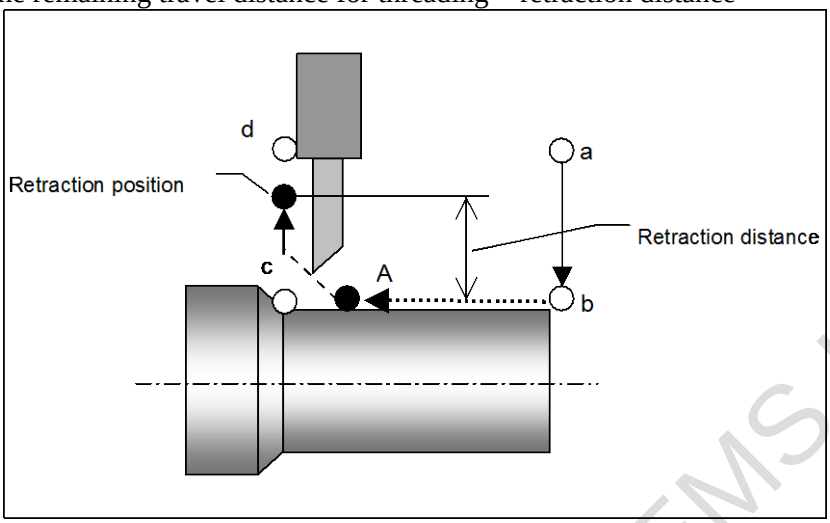
(a) When remaining travel distance for threading ≥ retraction distance



When the position where 45-degree chamfering by the retraction distance ends does not exceed the threading end position (c), the tool moves to the threading end position after the termination of chamfering.

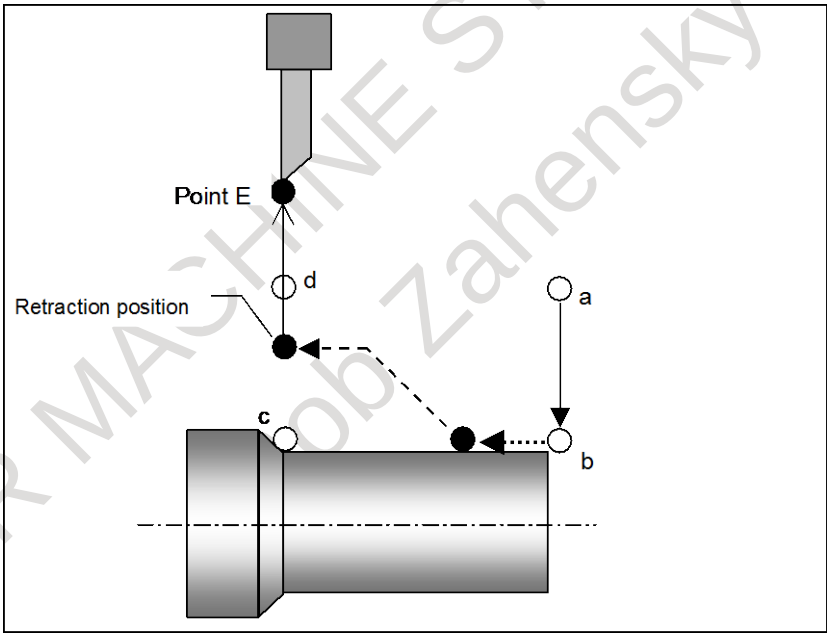
FRYER MACHINE SYSTEMS INC
Zahensky

(b) When the remaining travel distance for threading < retraction distance



When the position where 45-degree chamfering by the retraction distance ends exceeds the threading end position (c), the tool moves to the retraction position along the retraction axis after it reaches the threading end position.

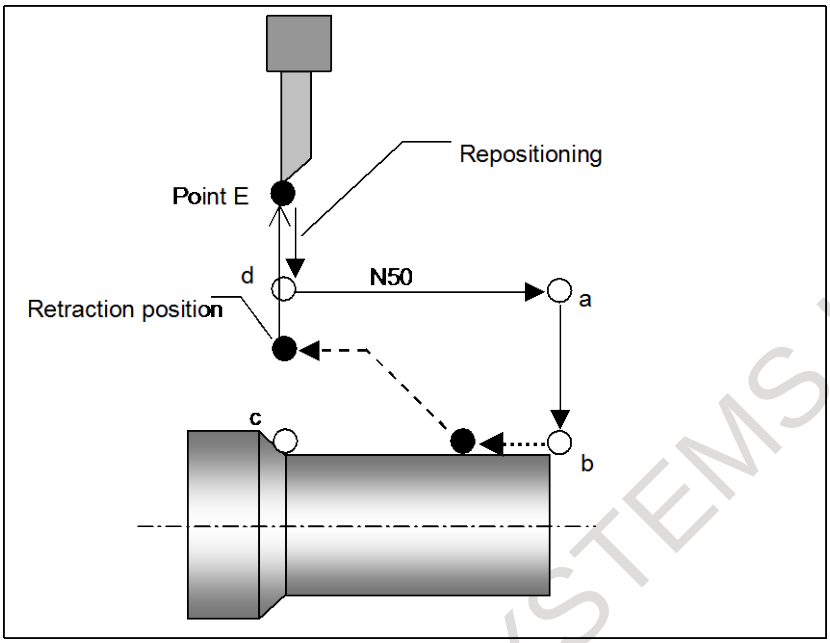
(4) After retraction is completed, the next block that does not specify threading is executed and the tool stops.



In this example, “X50.0” is specified in the first block that does not specify threading in the incremental mode, the tool moves to point E and stops.
If the major axis for threading is specified as the retraction axis, the block that does not specify threading is executed without performing retraction, an alarm PS0429 is issued, and the tool stops.

FRYER MACHINE SYSTEMS INC
Bob Zahensky

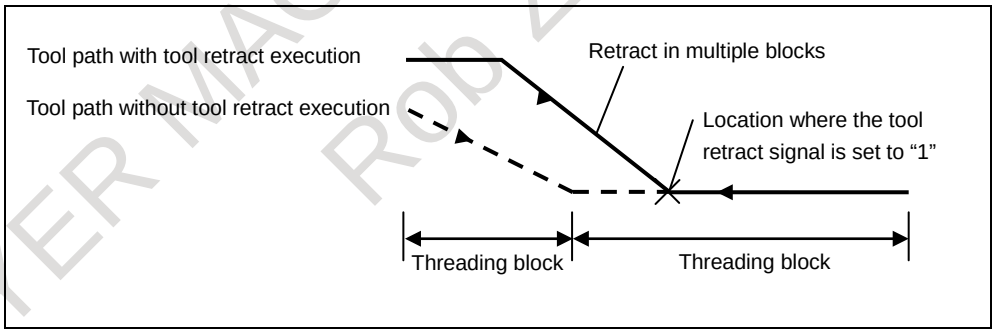
(5) As repositioning, the tool returns to the position specified in the first block that does not specify threading.



In this example, the repositioning position is point d. Automatic operation after repositioning starts at the N50 block.

NOTE

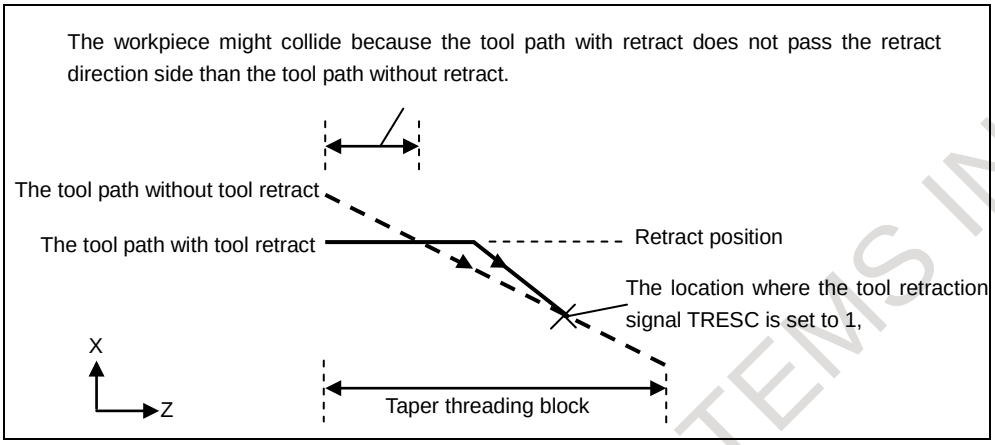
1 In threading cycle and multiple threading cycle, tool retract and recover is invalid.
2 In circular threading, tool retract and recover is invalid.
3 When dry run is executed, tool retract and recover is invalid.
4 In continuous threading blocks other than last threading block, if retract start at a location near the block end, retract is executed in multiple blocks.



FRYER MACHINE SYSTEMS INC
Rob Zakrevsky

NOTE

5 In taper threading, specify the retract position as the tool does not collide the workpiece surely because the tool might collide depending on specification of retract position as following figure.

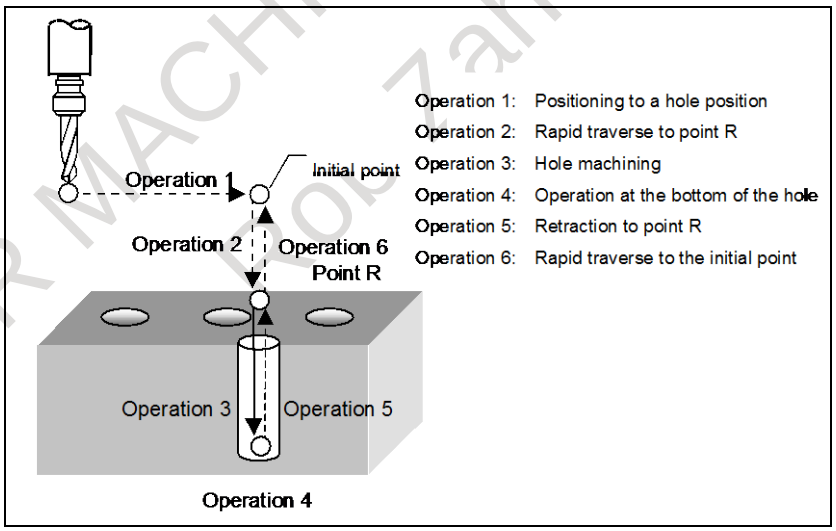


4.11.6 Operation Procedure for a Canned Cycle for Drilling

Explanation

- **Retract**

When the TOOL WITHDRAW switch is turned on during a canned cycle for drilling (abbreviated as a canned cycle below), retraction is performed depending on the cycle operation being executed at that time.



(1) During operation 1, the tool moves by the retraction distance specified in G10.6 in the same way as for ordinary retraction.
(2) During operation 2, the tool stops operation 2, moves to the initial point, and stops.
(3) During operation 3, the tool stops operation 3, executes cycle operations 4, 5, and 6 from that position, and stops at the initial point.

FRYER MACHINE SYSTEMS INC Rob Zahonsky

(4) During operation 4, 5, or 6, the tool continues the operation and stops at the initial point.
When the TOOL WITHDRAW switch is turned on during operation 2 to 6, the tool does not move according to the retraction specified in G10.6. After the TOOL WITHDRAW switch is turned on and the tool moves to the initial point, however, the tool withdrawal mode is set.
When the second or subsequent canned cycle is being executed and the TOOL WITHDRAW switch is turned on during operation 2 to 6, the retraction position differs depending on G98 (return to initial level) or G99 (return to point R level).
- G98 (return to initial level): The tool moves to the initial level.
- G99 (return to point R level):The tool moves to the point R level.

(5) During operation 2 to 6, the tool also moves to the initial point and stops when the TOOL WITHDRAW switch is turned on without the G10.6 command specified.

- Repositioning

When the tool is at the retraction position and the cycle start switch is pressed, repositioning is performed for the canned cycle.

(1) Repositioning performed when the TOOL WITHDRAW switch is turned on during operation 1
After the completion of repositioning, automatic operation is resumed in the same way as for ordinary repositioning.

(2) Repositioning performed when the TOOL WITHDRAW switch is turned on during operation 2
The canned cycle is reexecuted from operation 2.

(3) Repositioning performed when the TOOL WITHDRAW switch is turned on during operation 3
The canned cycle is reexecuted from operation 2.

(4) Repositioning performed when the TOOL WITHDRAW switch is turned on during operation 4, 5, or 6
The canned cycle is reexecuted for the same hole position from operation 2.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

4.12 MANUAL INTERVENTION AND RETURN

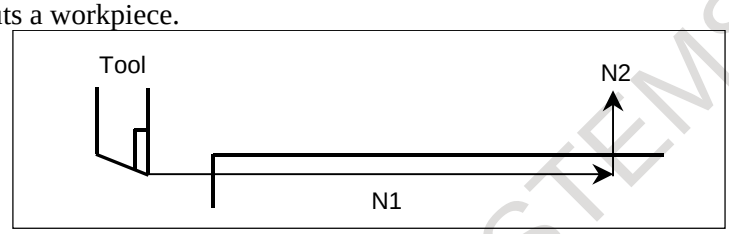
Overview

If you use feed hold to stop the tool from moving an axis during automatic operation and restarts the tool after manual intervention, for example, for checking a cutting surface, the tool can resume automatic operation after automatically returning to the pre-intervention position.

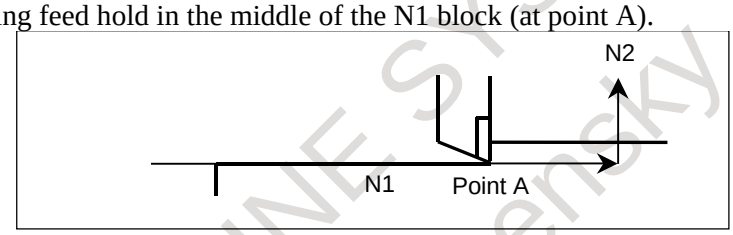
Explanation

Setting bit 0 (MIT) of parameter No. 7001 enables manual intervention and return.
The sequence of manual intervention and return is as Fig. 4.12 :

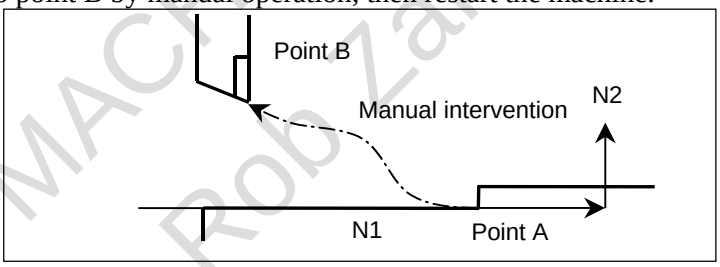
1. The N1 block cuts a workpiece.



2. Stop the tool using feed hold in the middle of the N1 block (at point A).



3. Retract the tool to point B by manual operation, then restart the machine.



4. After the tool automatically returns to point A by non-linear interpolation type positioning at the dry run feedrate, the remaining move command of the N1 block is executed.

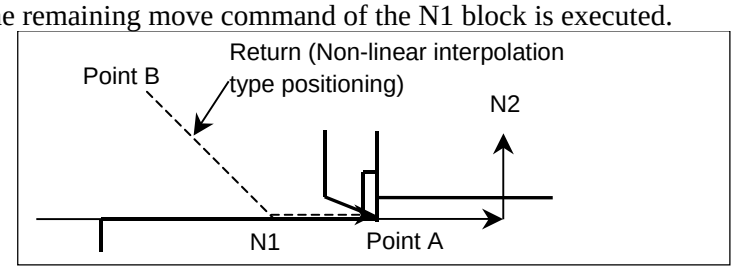
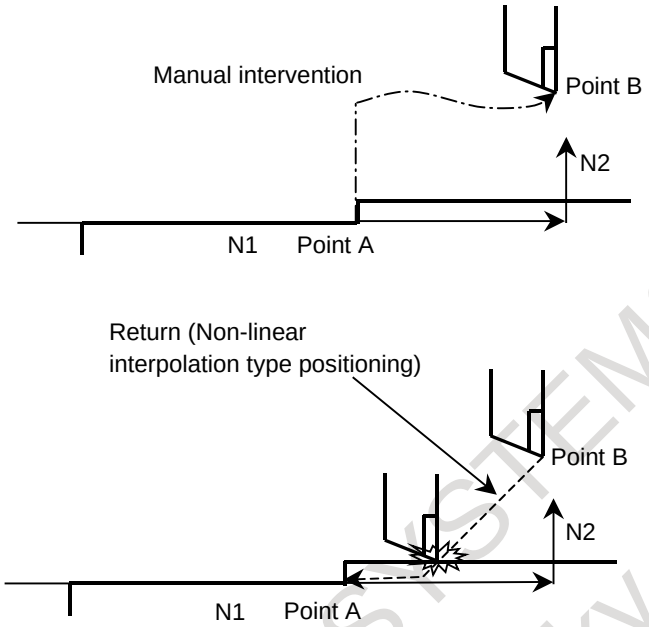


Fig. 4.12 Sequence of manual intervention and return

FRYER MACHINE SYSTEMS INC
Rob Zahensky

⚠ **WARNING**

Manual intervention must be performed correctly with meticulous care, following the machining direction and the shape of the workpiece, not to damage the workpiece, machine, and/or tool.



- Manual absolute on/off

In manual absolute off mode, the tool does not return to the stop position. Instead, the tool moves according to the manual absolute on/off function.

- Return feedrate

For the return operation, the dry run feedrate is used, and the jog feedrate override function is enabled. When manual rapid traverse rate selection signal RT is "1", the return feedrate is not dry run feedrate but rapid traverse rate.

- Return operation

Return operation is performed according to non-linear interpolation type positioning.

- Single block

If the single block stop switch is on during return operation, the tool stops at the stop position once and restarts movement when cycle start is executed again.

- Cancellation

If a reset, alarm, or emergency stop occurs during manual intervention or return operation, the manual intervention and return function is cancelled.

- MDI mode

The manual intervention and return function can be used in the MDI mode as well.

- Relation to other functions

In other functions such as canned cycle, the manual intervention and return function is effective.

- PMC axis control

If intervention is performed under PMC axis control after feed-holding of automatic operation, return operation does not apply to the axis moving under PMC axis control when the return operation starts.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

If the return operation is started with the axis under PMC axis control being stopped after having completed the PMC axis control command, however, the return operation is performed by the amount of movement by PMC axis control.
When PMC axis control is used for an axis unrelated to the program command, set bit 0 (NRT) of parameter No. 10410 to 1 to exclude the axis completely from return operation by the manual intervention and return function.

Limitations

- Enabling/disabling of manual intervention and return function

Manual intervention and return function is enabled in the feed hold state.
Manual intervention and return function is disabled in automatic operation stop state (single block stop, feed hold stop with no travel distance remaining, etc.) and the operation is performed according to the manual absolute on/off function.

- Offset

When the tool is replaced using manual intervention for a reason such as tool damage, even if the amount of offset is changed when restarting in the middle of the interrupted block, the change is not reflected in machining.

- Machine lock, mirror image, and scaling

When performing manual intervention and return, never use the machine lock, mirror image, or scaling function.

- 3-dimensional coordinate system conversion, and tilted working plane indexing

Alarm PS5219, “CAN NOT RETURN” is issued when manual intervention and the return is attempted during execution of 3-dimensional coordinate system conversion, or tilted working plane indexing.

- Parking

Do not switch the parking signal during manual intervention and return (for example, after feed hold stop). The tool may not return in the right position when the parking signal is switched during manual intervention and return.

- Manual intervention and return by multi-path system

If cycle start is executed at the same time on two or more paths in a multi-path system, the path that performed manual intervention does return operation. However, the path that did not perform manual intervention restarts processing without doing return operation.

- Axis for which return operation is not performed

When the axis is in the following conditions, return operation of the axis is not performed.
- Spindle positioning
- Tapping
- Slave axis of axis synchronous control
- Slave axis of synchronous control
- Spindle control by servo motor
- Polygon axis

NOTE
Under synchronous control, manual intervention and return can be performed for the slave axis only when bit 2 (PKUx) of parameter No. 8162 is 1 and the master axis is parking.

FRYER MACHINE SYSTEMS INC
Bob Zahensky

- Feed hold during return operation

Feed hold is not effective during return operation.

4.13 RETRACE

M

Overview

The tool can retrace the path along which the tool has moved so far (reverse execution). Furthermore, the tool can move along the retraced path in the forward direction (forward reexecution). After forward reexecution is performed until the tool reaches the position at which reverse execution started, machining is continued as programmed.

Procedure

- Forward execution → reverse execution

To perform forward execution of a program, set the “REVERSE” switch on the machine operator’s panel to off, then perform a cycle start operation. If the “REVERSE” switch on the machine operator’s panel is set to on, reverse execution or the end of reverse execution results.

To perform reverse execution of a program, use one of the following three methods:

(1) Set the “REVERSE” switch on the machine operator’s panel to on during forward execution of a block.

(2) Perform a single block stop operation during forward execution, then set the “REVERSE” switch on the machine operator’s panel to on.

(3) Perform a feed hold stop operation during forward execution, then set the “REVERSE” switch on the machine operator’s panel to on.

When method (1) is used, reverse execution starts after the end of the block being executed (after execution up to the single block stop position). Reverse execution does not start as soon as the “REVERSE” switch on the machine operator’s panel is set to on.

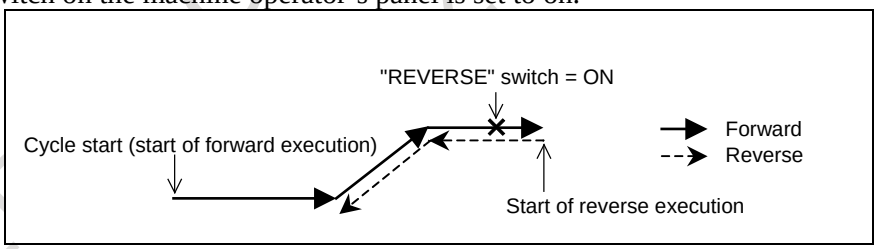


Fig. 4.13 (a)

When method (2) is used, performing a cycle start operation starts reverse execution from the position at which a single block stop takes place.

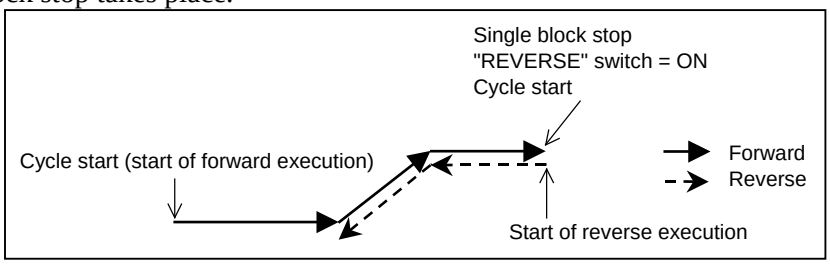


Fig. 4.13 (b)

FRYER MACHINE SYSTEMS INC
Zahensky

When method (3) is used, performing a cycle start operation starts reverse execution from the position at which a feed hold stop takes place.

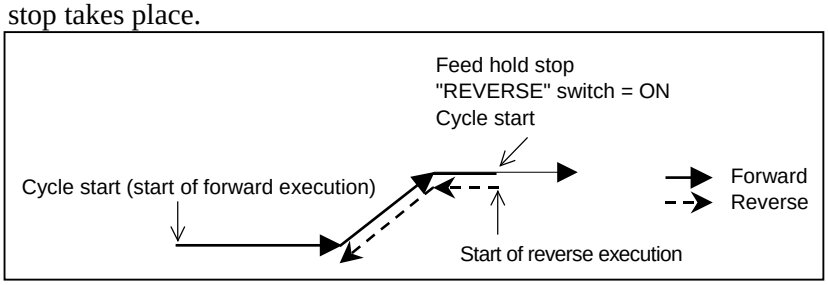


Fig. 4.13 (c)

- Reverse execution → forward reexecution

To perform forward reexecution of a program, use one of the following three methods:

(1) Set the “REVERSE” switch on the machine operator’s panel to off during reverse execution of a block.

(2) Set the “REVERSE” switch on the machine operator’s panel to off after a single block stop takes place during reverse execution.

(3) Set the “REVERSE” switch on the machine operator’s panel to off after a feed hold stop takes place during reverse execution.

When method (1) is used, forward reexecution starts after the block being executed ends (after execution up to the position at which a single block stop takes place). Forward reexecution does not start as soon as the “REVERSE” switch on the machine operator’s panel is set to off.

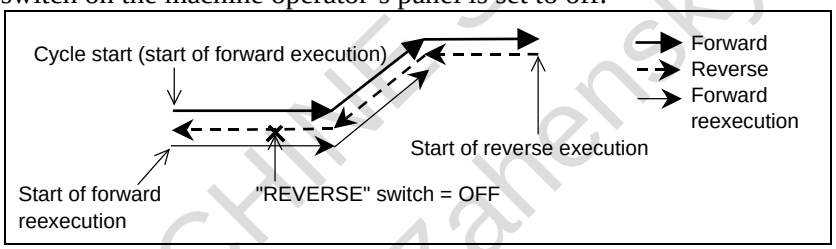


Fig. 4.13 (d)

When method (2) is used, performing a cycle start operation starts forward reexecution from the position at which a single block stop takes place.

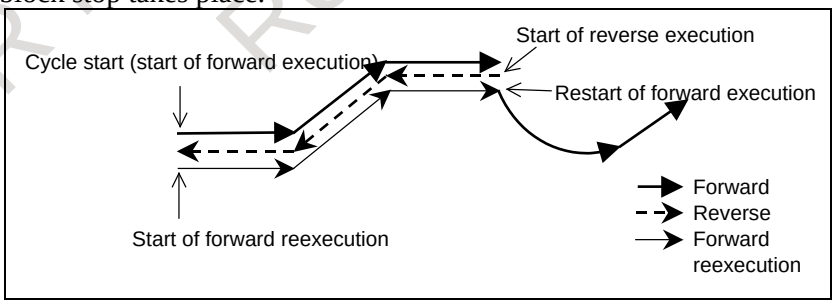


Fig. 4.13 (e)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

When method (3) is used, performing a cycle start operation starts forward reexecution from the position at which a feed hold stop takes place.

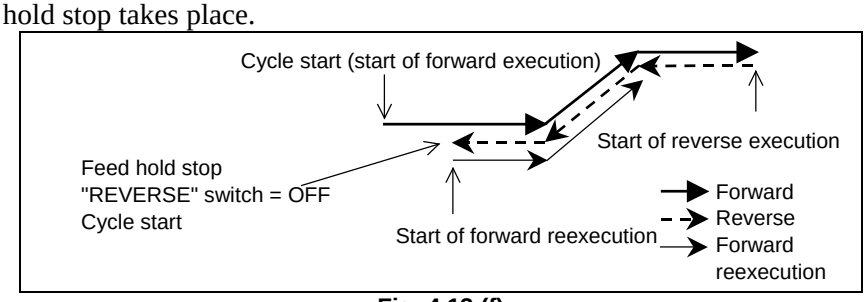


Fig. 4.13 (f)

- Reverse execution → end of reverse execution → forward reexecution

When a block to be executed is no longer present during reverse execution (when reverse execution has been performed up to the block where forward execution started, or when forward execution has not yet been performed), the reverse execution end state is entered, and operation stops.
Even when a cycle start operation is performed while the “REVERSE” switch on the machine operator’s panel is held on, operation is not performed, and the reverse execution end state is maintained. Forward reexecution (or forward execution) is started by setting the “REVERSE” switch on the machine operator’s panel to off then performing a cycle start operation.

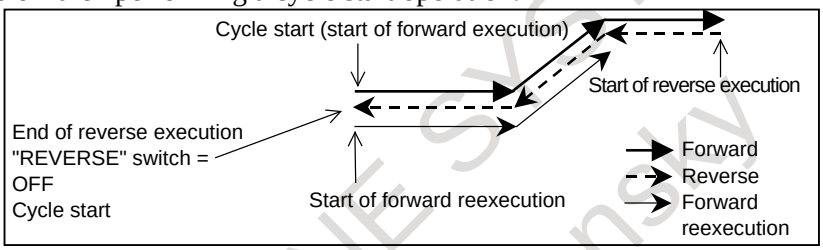


Fig. 4.13 (g)

- Forward reexecution → forward execution

After forward reexecution is performed up to the block at which reverse execution started, forward execution starts automatically, and commands are read from the program again and executed. No particular operation is required.

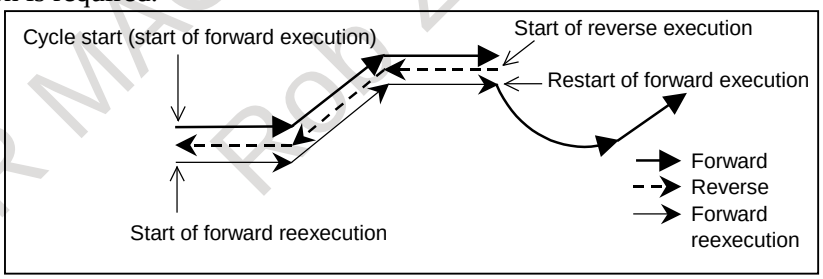


Fig. 4.13 (h)

If reverse execution was performed after feed hold stop, forward reexecution ends when the feed hold stop position is reached, then forward execution is performed. Also if single block operation was performed, forward reexecution ends at the single block stop position.

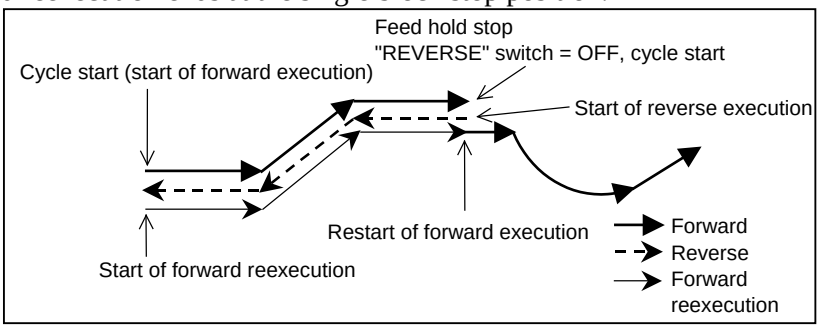


Fig. 4.13 (i)

FRYER MACHINE SYSTEMS INC
Bob Zahensky

Explanation

- Reverse execution and forward execution

Usually in automatic operation, a program is executed in the programmed order. This is called forward execution. This function allows a program executed by forward execution to be executed in the reverse direction. This is called reverse execution. Reverse execution allows the tool to retrace the path along which the tool has moved during forward execution.
Reverse execution of a program can be performed only for blocks that have been executed by forward execution.
Furthermore, in single block operation, reverse execution can also be performed on a block-by-block basis .

- Forward reexecution

Blocks that have been executed by reverse execution can be reexecuted in the forward direction up to the block from which reverse execution started. This is called forward reexecution. Forward reexecution allows the tool to retrace the same tool path as in forward execution until the position at which reverse execution started is reached.
After the block from which reverse execution started is reached, the program is executed again in the programmed order (forward execution).
Furthermore, in single block operation, forward reexecution can also be performed on a block-by-block basis.

- End of reverse execution

When a block to be executed is no longer present during reverse execution (when stored blocks have all been executed during reverse execution, or when forward execution has not yet been performed), operation stops. This is called the end of reverse execution.

- Status indication

During reverse execution, characters "RVRS" blink on the screen. During forward reexecution, characters "RTRY" blink to indicate that forward reexecution is in progress. The "RTRY" indication is kept blinking until the block at which reverse execution started is reached and normal operation starts (until forward execution is restarted).
When a block to be executed is no longer present during reverse execution, or if an attempt is made to perform reverse execution for a block that cannot be executed by reverse execution, characters "RVED" blink, notifying the user that reverse execution can no longer be performed.

- Number of blocks that can be executed by reverse execution

Up to about 100 blocks can be executed by reverse execution. Depending on the specified program, the maximum number of executable blocks may decrease.

- Reset

A reset operation (the RESET key on the MDI unit, the external reset signal, or the reset & rewind signal) clears the blocks stored for reverse execution.

- Feedrate

A feedrate to be applied during reverse execution can be specified in parameter No. 1414. If this parameter is set to 0, the feedrate in reverse execution is assumed to be the same as that in forward execution. Rapid traverse, however, is performed always at the rapid traverse rate, regardless of the setting of this parameter.
The feedrate in forward reexecution is always the same as that in forward execution.
In reverse execution or forward reexecution, feedrate override, rapid traverse override, and dry run are allowed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Start of reverse execution or forward reexecution after the end of a block

In a block for rapid traverse (G00), linear interpolation (G01), circular interpolation (G02, G03), dwelling (G04), skip cutting (G31), or an auxiliary function in an automatic operation mode (memory operation, part program operation, or MDI operation), reverse execution or forward reexecution can be started. However, reverse execution and forward reexecution do not start as soon as the reverse execution signal status is changed. When the block has ended, that is, after a movement, dwelling, or an auxiliary function is completed, reverse execution or forward reexecution starts.

- Start of reverse execution or forward reexecution after feed hold stop

When a feed hold stop operation is performed during execution of rapid traverse (G00), linear interpolation (G01), circular interpolation (G02, G03), or skip cutting (G31), then the reverse execution signal status is changed and operation is restarted, reverse execution or forward reexecution can be started immediately from the stop position. This cannot be performed when dwelling (G04) or an auxiliary function is being executed.

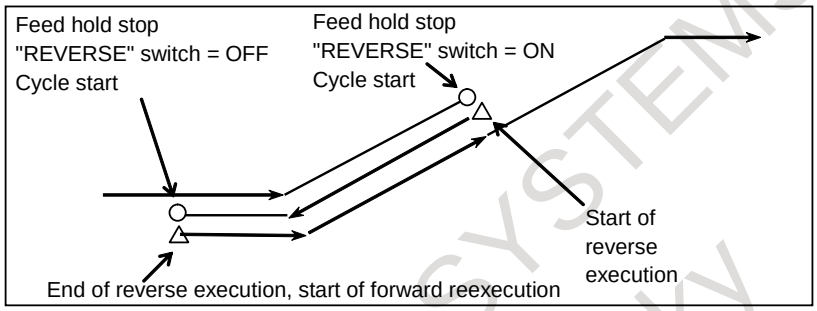


Fig. 4.13 (j)

When reverse execution is started after feed hold stop operation, the part from the start point of that block to the feed hold stop position is stored as one block. Therefore, when forward reexecution is performed with the single block switch set to 1, a single block stop takes place as soon as the position at which reverse execution started is reached.

- Start of reverse execution or forward reexecution after single block stop

After a single block stop takes place, reverse execution or forward reexecution can be started immediately when the reverse execution signal status is changed and restart operation is performed.

Limitation

- Blocks that cannot be executed by reverse execution

In the modes listed below, reverse execution cannot be performed.
When one of these commands appears during reverse execution, reverse execution ends immediately and "RVED" is displayed.

- Polar coordinate interpolation (G12.1, G13.1,G112,G113)
- Polar coordinate command (G16)
- Functions related thread cutting (G33,G34,G35,G36)
- Polygon turning (G50.2, G51.2)
- Tool axis direction control (G53.1)
- Single direction positioning (G60)
- Tapping mode (G63)
- Tapping cycle (G84,G74)
- Rigid tapping cycle (G84,G74,G84.2,G84.3)
- Fine boring cycle (G76)
- Back boring cycle (G87)

It is impossible to perform reverse execution for blocks specifying the commands listed below. If one of these commands appears during reverse execution, reverse execution ends immediately and "RVED" is displayed.

FRYER MACHINE SYSTEMS INC
Rob Zaharansky

Some of these commands turn a mode on and off. It is possible to start reverse execution and perform forward reexecution in a mode set by such a command. However, if a block that turns the mode on or off is reached during reverse execution, the reverse execution ends at that block, and "RVED" is displayed.

- Functions related AI contour control (G05,G05.1,G08)
- HRV3 on/off (G05.4)
- Inch/metric conversion (G20, G21)
- Stored stroke check on/off (G22, G23)
- Functions related reference position return (G27, G28, G29, G30)
- 3-dimensional coordinate system conversion (G68, G69)
- Feature coordinate system (G68.2)
- Figure copying (G72.1,G72.2)
- Index table indexing
- Cs contouring control
- Spindle positioning

- Manual intervention

To execute a program in the reverse direction after a feed hold stop or single block stop, when manual intervention is performed after the stop, make a return to the original position and then turn on the reverse signal. Movement by manual intervention is ignored during reverse execution and forward reexecution.
If manual intervention is performed during reverse execution or forward reexecution, the amount of manual intervention is added to the coordinate system at a restart after a stop due to a feed hold or single block during forward execution after the end of forward reexecution. Whether to add the amount of manual intervention follows the manual absolute switch.

- Single block stop position

A block that is internally generated by the control unit is also treated as one block during reverse execution.

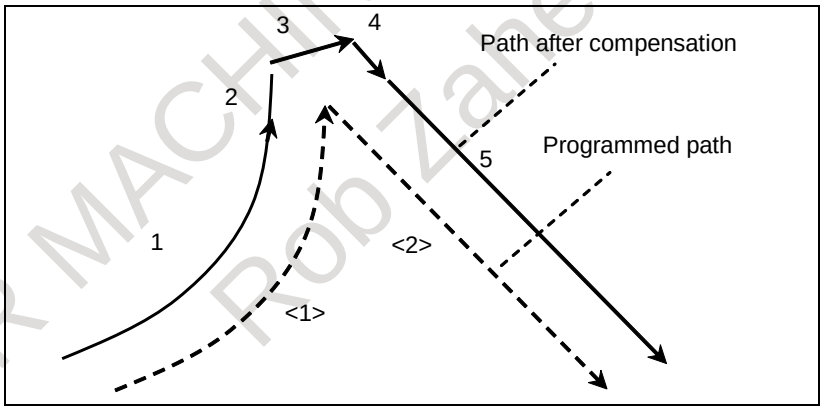


Fig. 4.13 (k) Path when cutter compensation is applied

In the above example, the program specifies two blocks, but in actual operation, move commands for five blocks are generated.
In such a case, positions at which a single block stop takes place may differ between forward execution and reverse execution.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Positioning (G00)

When non-linear type positioning is performed (bit 1 (LRP) of parameter No. 1401 is set to 0), the tool path in reverse execution and that in forward execution do not match. The tool path in forward reexecution is the same as that in forward execution.
When linear type positioning is performed (bit 1 (LRP) of parameter No. 1401 is set to 1), the tool path in reverse execution is the same as that in forward execution.

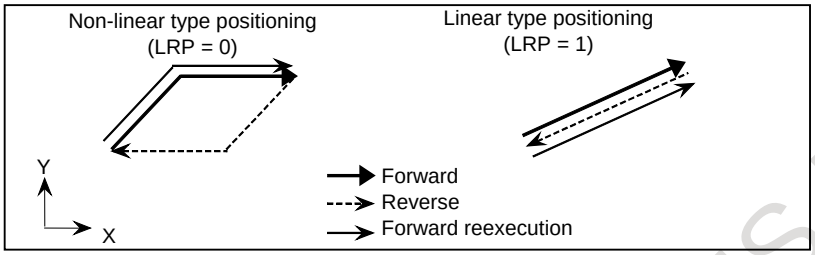


Fig. 4.13 (l)

- Dwell command (G04)

During reverse execution or forward reexecution, the dwell command (G04) is executed in the same way as in normal operation.

- Programmable data input (G10)

Tool compensation values, parameters, pitch error data, workpiece origin offsets, and tool life management values set or modified by programmable data input (G10) are ignored during reverse execution and forward reexecution.

- Skip function (G31) and automatic tool length measurement (G37)

The skip signal and the measurement position arrival signal are ignored during reverse execution and forward reexecution. During reverse execution and forward reexecution, the tool moves along the path that the tool has actually passed during forward execution.

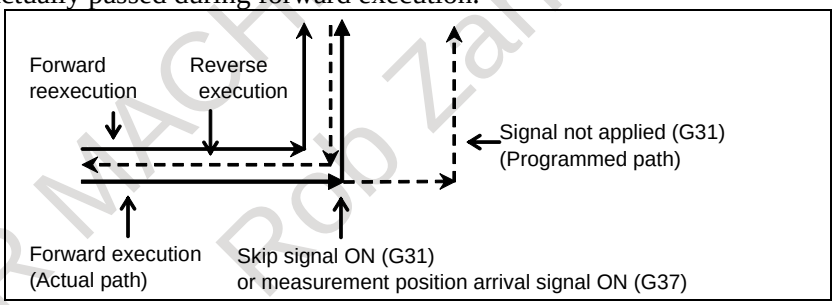


Fig. 4.13 (m)

- Setup of a coordinate system (G92, G54 to G59, G54.1P_, G52, and G92.1)

As for the display of the absolute coordinate value under backward movement, the value in the work coordinate system when backward movement begins is displayed. Therefore, when the command (G92, G54 to G59, G54.1P_, and G52) to which the work coordinate system is changed is passed, a different value might be displayed as forward movement. However, the actual machine position does not differ.

- Mirror image

When a block to which a mirror image is applied by programmable mirror image (G50.1, G51.1) is executed during reverse execution, the tool moves along the actual path resulting from the application of a mirror image in the reverse direction.
When a mirror image is applied to a block by setting or a machine signal, the block with the mirror image not applied is stored. Mirror image application by setting or a machine signal is enabled also during reverse execution and forward reexecution. Therefore, during reverse execution and forward reexecution, the mirror image by setting data or machine signal must be turned on and off so that this on/off status and the on/off status during forward execution match.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

- Changing offsets

Even when cutter compensation data or tool length offsets are changed during reverse execution or forward reexecution, the change in compensation or offset data does not become valid until forward reexecution ends and normal operation starts. Until then, the tool moves with the offset data that was applied when the block was executed for the first time during forward execution.

- Feedrate clamp

During reverse execution or forward reexecution, feedrate clamp is not performed with parameter No. 1420 (rapid traverse rate) or parameters Nos. 1430 and 1432 (maximum cutting feedrate). It is executed with parameter No. 1414 or at the feedrate assumed during forward execution.
If, for example, the parameters above are set to smaller values during reverse execution or forward reexecution, clamp is not performed with these values, but with parameter No. 1414 or at the feedrate assumed during forward execution.
For clamp at the feedrate assumed during backward execution or forward reexecution, change the feedrate with the external deceleration or override signal.

- Interrupt type custom macro

Do not initiate any interrupt during reverse execution.
Do not execute an interrupted block and the interrupt program in reverse execution.

- Tool management function

The tool life is not counted during reverse execution and forward reexecution.

- Inverse time feed (G93)

If a nonzero value is set as the feedrate to be applied during reverse execution in parameter No. 1414, a block that moves the tool by inverse time feed during forward execution is executed at the parameter-set feedrate (feed per minute) during reverse execution.
If the feedrate during reverse execution (parameter No. 1414) is not set (= 0), the same feedrate as applied during forward execution is used.

- Maximum spindle speed clamp (G92Sxxxx)

Clamping at a maximum spindle speed specified during reverse execution becomes valid. This means that if G92Sxxxx appears during reverse execution, the spindle speed is clamped at Sxxxx in the subsequent reverse execution. As a result, the clamp speed may differ between reverse execution and forward execution even when the same block is executed. The spindle speed is clamped when the G96 mode is set.

- Auxiliary functions

M, S, T, and the second auxiliary function (B function) are output directly also during reverse execution and forward reexecution.
When specified together with a move command in the same block, M, S, T, and the second auxiliary function (B function) are output with the move command at the same time during forward execution, reverse execution, and forward reexecution. Therefore, the output positions of M, S, T, and the second auxiliary function (B function) during reverse execution differ from those during forward execution and forward reexecution.

- Custom macro

Custom macro operations are ignored during reverse execution and forward reexecution.

- Execution macro (macro executor)

Macro executor operations are ignored during reverse execution and forward reexecution.

FAYER MACHINE SYSTEMS INC
Rob Zaharensky

- Tool retract and recover function

For retract operation and repositioning operation by the tool retract and recover function, reverse execution cannot be performed. Retract operation and repositioning operation are ignored during reverse execution and forward reexecution.

- AI contour control

When the reverse execution is started in AI contour control mode, a reverse execution ends immediately depending on a program then the backward movement is not possible.
During reverse execution and forward reexecution, the feedrate clamp function by acceleration under AI contour control is disabled.

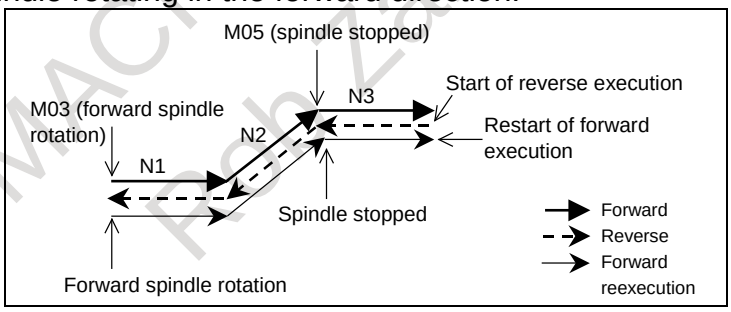
- Display

During reverse execution and forward reexecution, the modal display and the display of the currently executed program are not updated; information obtained at the start of reverse execution is maintained.

Warning

⚠ WARNING

1 Auxiliary functions are output directly even during reverse execution and forward reexecution. Accordingly, the execution status of an auxiliary function during forward execution may be reversed during reverse execution.
Example:
When forward rotation of the spindle (M03) and stop (M05) are specified
When N3 is executed during reverse execution, M05 is output. So, when N2 and N1 are executed during reverse execution, operation is performed with the spindle stopped.
When N1 is executed during forward reexecution, M03 is output. So, when N1 and N2 are executed during forward reexecution, operation is performed with the spindle rotating in the forward direction.



2 To perform reverse execution after a feed hold stop or single block stop operation, be sure to restore the original position if manual intervention has been performed after the stop , then set the "REVERSE" switch to on. Movements made by manual intervention are ignored during reverse execution and forward reexecution. (The same operation as in the manual absolute off state takes place.)
If manual intervention is performed during reverse execution or forward reexecution, the amount of manual intervention is added to the coordinate system at a restart after a stop due to a feed hold or single block during forward execution after the end of forward reexecution. Whether to add the amount of manual intervention follows the manual absolute switch.

FRYER MACHINE SYSTEMS INC
Rich Zaborsky

4.14 RETRACTION FOR RIGID TAPPING / RETRACTION FOR 3-DIMENSIONAL RIGID TAPPING

Overview

When rigid tapping is stopped, either as a result of a power off, an emergency stop or a reset, the tap may cut into the workpiece. The tap can subsequently be drawn out by using a PMC signal or program command.

This function automatically stores information relating to the tapping executed most recently. When a tap retraction signal is input or G30 is executed, the tap is removed from the hole, based on the stored information. The tap is pulled toward the R point. When a retract value is set in parameter No. 5382, the retraction distance can be increased by the value.

In the retraction for 3-dimensional rigid tapping, 3-dimensional rigid tapping or the rigid tapping in tilted working plane indexing mode is stopped, the tap may cut into the workpiece. The tap can subsequently be drawn out.

Format

Retraction for rigid tapping

G30 P99 M29 S_ ;

P99 : P99 is required to indicate that the G30 is retraction for rigid tapping.
M29 : When the rigid tapping mode specification M code is set in parameter No.5210, specify that value instead of M29. This directive is optional.
S_ : Spindle speed. Because of S code when rigid tapping is interrupted is used, S code is optional.

Retraction for rigid tapping during 3-dimensional coordinate system conversion mode

G68 X_ Y_ Z_ I_ J_ K_ R_ ; 3-dimensional coordinate system conversion (Note)

G30 P99 M29 S_ ; Retraction for rigid tapping

P99 : P99 is required to indicate that the G30 is retraction for rigid tapping.
M29 : When the rigid tapping mode specification M code is set in parameter No.5210, specify that value instead of M29. This directive is optional.
S_ : Spindle speed. Because of S code when rigid tapping is interrupted is used, S code is optional.

Retraction for rigid tapping during tilted working plane command mode

G68.2 X_ Y_ Z_ I_ J_ K_ R_ ; Tilted working plane command (Note)

G53.1 ; Tool axis direction control

G30 P99 M29 S_ ; Retraction for rigid tapping

P99 : P99 is required to indicate that the G30 is retraction for rigid tapping.
M29 : When the rigid tapping mode specification M code is set in parameter No.5210, specify that value instead of M29. This directive is optional.
S_ : Spindle speed. Because of S code when rigid tapping is interrupted is used, S code is optional.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

Retraction for rigid tapping by the G30 command is used when bit 1 (RG3) of parameter No. 5202 is 1.
If rigid tapping is interrupted due to a reset or emergency stop or if the power is turned off during rigid tapping, executing the command above in MEM or MDI operation causes the tapping axis to move to the R point while synchronized with the spindle, based on the rigid tapping command information in the machining program.
The command for retraction for rigid tapping is a one shot command.
The machining data for rigid tapping retraction is retained until the next rigid tapping command is specified, even after the power is turned off. Rigid tapping retraction can, therefore, be specified even after the power is turned off.
In the retraction for 3-dimensional rigid tapping, please command the 3-dimensional coordinate system conversion or tilted working plane command before the retraction for rigid tapping command. The command at this time, specify same arguments used when rigid tapping stops. Alarm PS5384, "RETRACT FOR RIGID CANNOT BE CMD." occurs if the specified arguments are different from those used at stopping.

NOTE
1 Retraction for rigid tapping by the G30 is not used when bit 1 (RG3) of parameter No. 5202 is 0.
2 In the retraction for rigid tapping by the G30, be sure to set bit 0 (G84) of parameter No. 5200 to 0.

Example

Machining program

```
M29 S1000 ;
G84 X20.0 Y20.0 R-10.0 Z-30.0 F500 ;
X50.0 Y50.0 ;
X100.0 Y100.0 ;
G80
```

Retraction program

```
G30 P99 M29 S1000 ;
G00 Z-10.0 ;
```

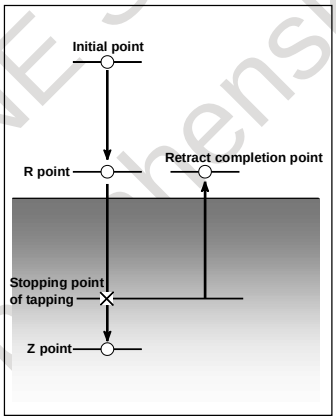


Fig. 4.14 Overview of retraction for rigid tapping function

Warning

⚠ WARNING
1 For rigid tapping retraction, the CNC internally activates a return program. Rigid tapping retraction may, therefore, cause some G codes or M/F/S/L codes to be overwritten (G80/G84/G74, G94/G95).
2 During rigid tapping retraction, do not change any setting affecting the travel distance (such as the increment system, input unit 10 time multiply, or calculator-type decimal notation) that was made when machining data was stored for rigid tapping retraction.

FRYER MACHINE SYSTEMS INC
Robert Jensky

Caution

⚠ CAUTION

1 If rigid tapping is stopped as a result of an emergency stop, the position on the tapping axis (Z-axis) is maintained but the spindle position is lost. In such a case, therefore, the positional relationship between the spindle and tapping axis is not guaranteed when operation is resumed.

2 Rigid tapping retraction is performed based on the tapping axis (Z-axis) commands accumulated for tapping. If rigid tapping is stopped as a result of an emergency stop, therefore, rigid tapping retraction may fail to draw the tapping tool completely out of the workpiece. In such a case, set retract value α (parameter No. 5382).

3 During rigid tapping retraction, switching the mode to the manual operation mode stops rigid tapping retraction, and decelerates and stops the machine.

4 If the power is turned off during rigid tapping motion by power failure or etc, when retraction for rigid tapping is executed after the power turned on, the tapping axis (Z-axis) may be retracted additionally by the accumulated distance of acceleration/deceleration.

Note

NOTE

1 Rigid tapping retraction is not performed if the input increment (inches or millimeters) selected when machining data is stored for rigid tapping retraction differs from that selected when rigid tapping retraction is executed.

2 Override can be applied to rigid tapping retraction when override during extraction in rigid tapping is validated. (bit 4 (DOV) of parameter No. 5200 is 1)

FRYER MACHINE SYSTEMS INC
Rob Zahrnsky

5 TEST OPERATION

The following functions are used to check before actual machining whether the machine operates as specified by the created program.

5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK

To display the change in the position without moving the tool, use machine lock.
There are two types of machine lock: all-axis machine lock, which stops the movement along all axes, and specified-axis machine lock, which stops the movement along specified axes only. In addition, auxiliary function lock, which disables M, S, T, and B (2nd auxiliary function) commands, is available for checking a program together with machine lock.

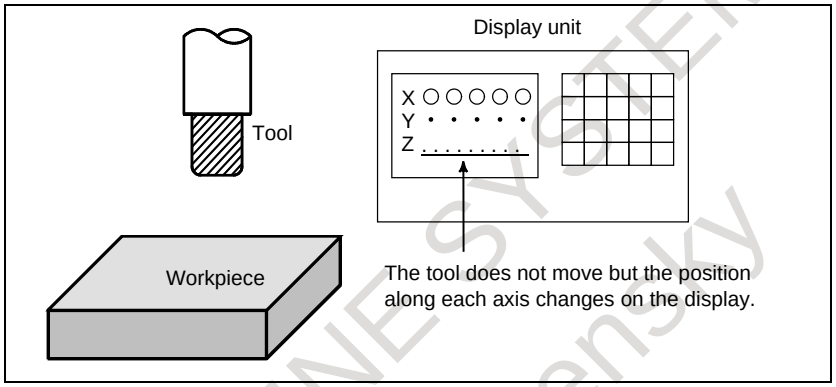


Fig. 5.1 Machine lock

5.1.1 Machine Lock

Procedure

Press the machine lock switch on the operator's panel. The tool does not move but the position along each axis changes on the display as if the tool were moving.
Some machines have a machine lock switch for each axis. On such machines, press the machine lock switches for the axes along which the tool is to be stopped. Refer to the appropriate manual provided by the machine tool builder for machine lock.

> ⚠ **CAUTION**
> The positional relationship between the workpiece coordinates and machine coordinates may differ before and after automatic operation using machine lock. In such a case, specify the workpiece coordinate system by using a coordinate setting command or by performing manual reference position return.

Limitation

- Automatic operation in the machine lock state (M, S, T, and B commands)

Machine lock applies only to move commands along controlled axes. Updating modal G codes or setting a coordinate system is performed normally. M, S, T, and B (2nd auxiliary function) commands are also performed normally.

- Reference position return under machine lock

When a G27, G28, or G30 command is issued in the machine lock state, the command is accepted but the tool does not move to the reference position and the reference position return LED does not go on.

FRYER MACHINE SYSTEMS INC Rob Zahensky

- **Selecting a machine coordinate system (G53)**

When a positioning of machine coordinate system (G53) is specified in the machine lock state, move commands are invalid and absolute coordinates and relative coordinates are not updated.
When machine lock is applied during G53 movement, output pulses (move commands) to the servo motors are stopped and machine coordinates are not updated. The absolute and relative coordinates are updated.

- **High precision oscillation function**

Do not switch machine lock signal to the oscillation axis during oscillation motion, otherwise an alarm PS5050 "ILL-COMMAND IN G81.1 MODE" is issued.

- **Manual intervention and return**

Do not perform manual intervention in the machine lock state, otherwise the tool might move to wrong position in return operation.

- **Tool retract and recover**

Do not perform manual retraction in the machine lock state, otherwise the tool might move to wrong position in recovery.

- **Program restart / Quick program restart**

The tool cannot be returned to a correct position under the following conditions.
- Manual operation is performed in the machine lock state.
- When program restart is specified in the machine lock state, then the machine lock is canceled.

- **Type for counting time of tool management function**

The count-up operation is not performed in the machine lock state.

- **High-speed program check function**

During the high-speed program check mode, all-axis machine lock signal MLK and each-axis machine lock signals MLK1 to MLK8 are invalid except for PMC control axis.

5.1.2 Auxiliary Function Lock

Procedure

Press the auxiliary function lock switch on the operator's panel. M, S, T, and B codes are disabled and not executed. Refer to the appropriate manual provided by the machine tool builder for auxiliary function lock.

Limitation

- **M codes not locked by auxiliary function lock**

M00, M01, M02, M30, M98, M99, and M198 (subprogram call function) commands are executed even in the auxiliary function lock state. M codes for calling a subprogram (parameters Nos. 6071 to 6079) and those for calling a custom macro (parameters Nos. 6080 to 6089) are also executed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

5.2 FEEDRATE OVERRIDE

A programmed feedrate can be reduced or increased by a percentage (%) selected by the override dial. This feature is used to check a program.
For example, when a feedrate of 100 mm/min is specified in the program, setting the override dial to 50% moves the tool at 50 mm/min.

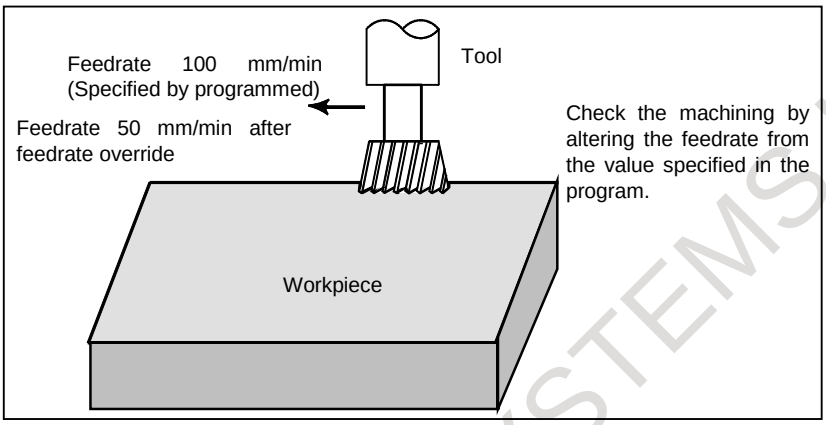


Fig. 5.2 Feedrate override

Feedrate override

Procedure

Set the feedrate override dial to the desired percentage (%) on the machine operator's panel, before or during automatic operation.
On some machines, the same dial is used for the feedrate override dial and jog feedrate dial. Refer to the appropriate manual provided by the machine tool builder for feedrate override.

Limitation

- **Override range**

The override that can be specified ranges from 0 to 254%. For individual machines, the range depends on the specifications of the machine tool builder.

- **Override during thread**

During the threading process, the override setting is ignored; it is always regarded as 100% during the process.

5.3 RAPID TRAVERSE OVERRIDE

An override can be applied to the rapid traverse rate. F0 is set by a parameter No. 1421.

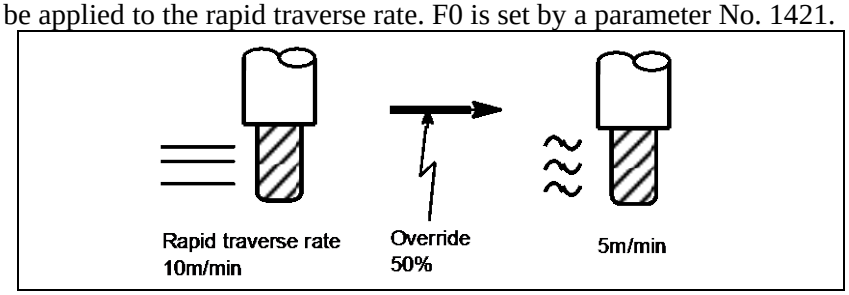


Fig. 5.3 Rapid traverse override

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Rapid traverse override

Procedure

Select the override by the rapid traverse override switch during rapid traverse. Refer to the appropriate manual provided by the machine tool builder for rapid traverse override.

Explanation

The following types of rapid traverse are available. Rapid traverse override can be applied for each of them.

(1) Rapid traverse by G00
(2) Rapid traverse during a canned cycle
(3) Rapid traverse in G27, G28, G29, G30, G53
(4) Manual rapid traverse
(5) Rapid traverse of manual reference position return

5.4 DRY RUN

The tool is moved at the feedrate specified by a parameter regardless of the feedrate specified in the program. This function is used for checking the movement of the tool under the state that the workpiece is removed from the table.

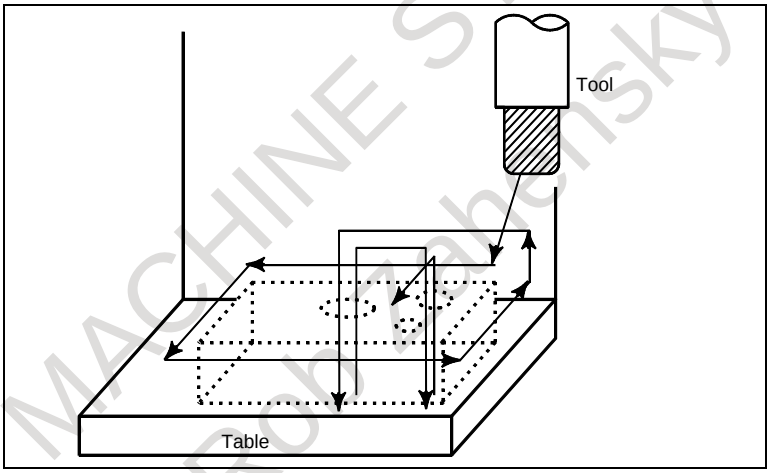


Fig. 5.4 Dry run

Dry run

Procedure

Press the dry run switch on the machine operator's panel during automatic operation.
The tool moves at the feedrate specified in a parameter. The rapid traverse switch can also be used for changing the feedrate.
Refer to the appropriate manual provided by the machine tool builder for dry run.

Explanation

- Dry run feedrate

The dry run feedrate changes as shown in the Table 5.4 according to the rapid traverse switch and parameters.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Table 5.4

Rapid traverse switch	Program command	
	Rapid traverse	Cutting feed
ON	Rapid traverse rate	Dry run feedrate × Jvmax(*2)
OFF	Dry run feedrate × JV, or rapid traverse rate (*1)	Dry run feedrate × JV(*2)

Max. cutting feedrate.....Setting by parameter No.1430
Rapid traverse rate.....Setting by parameter No.1420
Dry run feedrate .....Setting by parameter No.1410

(*1) Dry run feedrate × JV when bit 6 (RDR) of parameter No. 1401 is 1. Rapid traverse rate when parameter RDR is 0.
JV Jog feedrate override

(*2) Clamped to the maximum cutting feedrate
Jvmax Maximum value of jog feedrate override

5.5 SINGLE BLOCK

Pressing the single block switch starts the single block mode. When the cycle start button is pressed in the single block mode, the tool stops after a single block in the program is executed. Check the program in the single block mode by executing the program block by block.

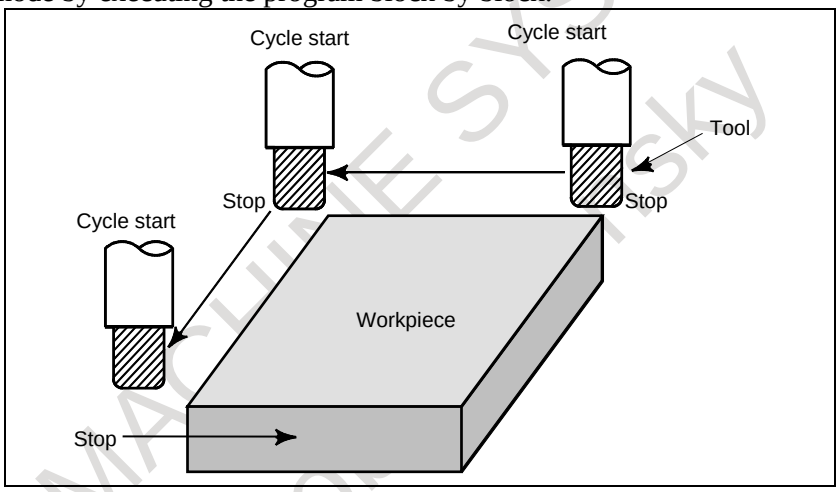


Fig. 5.5 (a) Single block

Single block

Procedure

(1) Turn the single block switch on the machine operator's panel on.
(2) Press the cycle start button. The machine executes one block and stops.
(3) Press the cycle start button again. The machine executes the next one block and stops.
Refer to the appropriate manual provided by the machine tool builder for single block execution.

Explanation

- Reference position return and single block

If G28, G29, and G30 are issued, the single block function is effective at the intermediate point.

- Single block during a canned cycle

In a canned cycle, the single block stop points are the end of <1>, <2>, and <6> shown below. When the single block stop is made after the point <1> or <2>, the feed hold LED lights.

FRYER MACHINE SYSTEMS INC

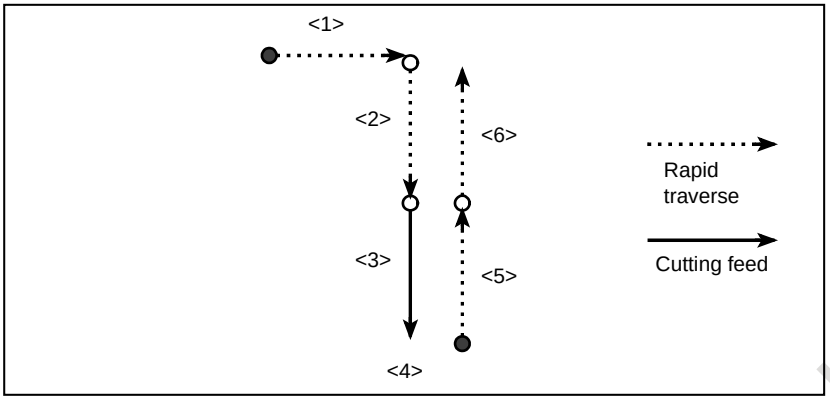


Fig. 5.5 (b) Single block during canned cycle

- Subprogram call and single block

Single block stop is not performed in a block containing M98P_;. M99; or G65.
However, single block stop is even performed in a block with M98P_ or M99 command, if the block contains an address other than O, N, P, L.

5.6 HIGH SPEED PROGRAM CHECK FUNCTION

When the cycle start button is pressed with the high speed program check mode set to be enabled, the program syntax and stroke limit are checked without axis movement.
A program check is performed at the maximum speed that can be specified by the CNC, regardless of the specified speed.

High speed program check

Procedure

(1) Perform position of the machine at the start position of the machining program in advance.
Set the workpiece coordinate system or tool compensation to the same state as in actual machining.
(2) Reset the CNC and press the high speed program check switch on the machine operator's panel to enter the high speed program check mode. In the high speed program check mode, "PRG-CHK" blinks at the lower-left of the CNC screen.
(3) Press the cycle start button to execute the program without axis movement. The feedrate is the maximum speed that can be specified by the CNC, regardless of the specified speed.
The program syntax and stroke limit are checked during execution of the program.
(4) Upon completion of the check, reset the CNC and release the high speed program check mode with the switch on the machine operator's panel.

Explanation

- Stroke Limit Check

Following the stroke limit check is executed without axes movements during the high speed program check mode.
- Stored stroke check 1
- Stored stroke check 2, 3
- Stroke limit check before move

If an alarm about the stroke limit occurs in the high speed program check mode, clear the alarm as described below.
(1) Change the mode to other than MEM or DNC.
(2) Reset the CNC.

FRYER MACHINE SYSTEMS INC
Bob Zahensky

> ⚠ **WARNING**
> 1 If the machine coordinate system is not established, a stroke limit check is not performed correctly. When a reference position return is completed after power-on, perform a high speed program check.
> 2 When the width of the prohibited area is small in a stored stroke limit 2 and 3 check, if a command to intersect the prohibited area in one block is issued, the stroke limit may not be detected. This means that a program not causing a stroke check alarm in a high speed program check may cause an alarm in actual operation.

- Program Check Feedrate

During the high speed program check mode, the feedrate of program execution becomes the maximum feedrate of CNC system regardless of the specified feedrate.
The feedrate clamp, override and dry run are not effective.
The execution speed of circular interpolation can be selected by bit 7 (PGF) of parameter No. 1490 from the maximum speed and dry run speed.

> **NOTE**
> 1 The execution time of dwell is the same as a normal operation.
> 2 The alarm PS011, “FEED ZERO (COMMAND)” occurs when there is no F command in a normal operation. However, it is executed at maximum speed without the alarm in the high speed program check mode even if there is no F command.

- Workpiece Coordinate System Preset

Upon completion of the high speed program check mode, the workpiece coordinate system is automatically preset. This enables program operation without making a reference position return upon completion of a program check. For details, refer to the manual provided by the machine tool builder.

> ⚠ **WARNING**
> When the coordinate system is set before the high speed program check mode is turned on by the work coordinate system setting G92 (machining center system and G code system B or C of the lathe system), G50 (G code system A of the lathe system) and the local coordinate system setting G52, their setting values are deleted by the execution of the work coordinate system preset.

- Machine Coordinate Display for Program Check

During the high speed program check mode, it is possible to change the display of machine coordinates from the actual machine coordinates to the machine coordinates for the program check by setting bit 7 (PGM) of parameter No.11320.
And it is possible to change the display of machine coordinate for each axis from the actual machine coordinate to the machine coordinate for the program check by setting bit 4 (PGA) of parameter No.3115.

- Tool life

If tool life management command or tool management command is specified during the high speed program check mode, tool life is not counted.

- Unavailable command

Following functions can not be specified during the high speed program check mode. If the following functions are specified, the alarm PS5364, “ILLEGAL COMMAND IN PROGRAM CHECK” occurs.
- Electronic gear box

POWER MACHINE SYSTEMS INC Rob Zahensky

- Commands not covered by a high speed program check

Auxiliary functions and other commands are not covered by a high speed program check. For details, refer to the manual provided by the machine tool builder.

5.7 MANUAL HANDLE RETRACE

Overview

In this function, the program can be executed both forward and backward with a manual handle (manual pulse generator) under automatic operation.
Therefore, errors of a program, interference, and so on can be checked easily by working a machine actually.

- Checking mode

In this mode, the program can be executed forward and backward and the program can be checked.
To change to the checking mode, it is necessary to change the mode to MEM mode, and the checking mode signal MMOD is set to "1". This function makes the data to execute the program backward when the program is executed forward in the checking mode.
To work a machine synchronizing with a pulse generated by a manual handle in the checking mode, the manual handle check signal MCHK is set to "1" in addition to the above-mentioned. As a result, it becomes possible to check the program with a manual handle.

NOTE
During the checking mode, it is not possible to change the parameter and offset.

- Forward movement with a manual handle

The "forward movement" is that the program is executed forward by turning a manual handle in the positive direction (when the manual handle check signal is set to "1".) or in no relation to rotation of a manual handle (when the manual handle check signal is set to "0".).
When the manual handle check signal is set to "1", the execution speed of the program is proportional to the number of rotations of a manual handle. The program is executed forward rapidly when a manual handle is turned to the positive direction rapidly. And, the program is executed forward slowly when a manual handle is turned to the positive direction slowly. The distance magnification traveled per pulse from manual handle can be switched as same as a usual manual handle feed function.
When the manual handle check signal is set to "0", the execution of the program is controlled as same as an automatic operation.

- Backward movement

The "backward movement " is that the program executed forward once is executed backward by turning a manual handle in the negative direction.
The program can be executed backward only for the block executed forward. And, the number of blocks for it is about 190 blocks. This block number changes by the content of the commanded program.
The program is executed backward rapidly when a manual handle is turned to the negative direction rapidly. And the program is executed backward slowly when a manual handle is turned to the negative direction slowly. The distance traveled per pulse from manual handle can switch magnification as well as a usual manual handle feed.

NOTE
In the system which exceeds 4 paths and 20 axes, the maximum number of blocks which can move backward becomes about 97 blocks.

FRIEP MACHINE SYSTEMS INC
Rob Zamensky

Explanation

- Control by the manual handle

Program execution start

The checking mode signal MMOD is set to "1" in MEM mode in order to change the checking mode. Then, the program execution is begun by turning ST signal from "1" to "0".
If the manual handle check signal MCHK is set to "1" at this time, the execution of the program is controlled by a manual handle. The program is executed synchronizing with rotation of a manual handle.
When a manual handle check signal MCHK is set to "0", it is controlled as usual execution.
When check mode signal MMOD is set to "1" during the operation of the program, it is enabled a check mode from the block that next buffering is done.
That is, even if check mode signal set to "1", check mode is not always enabled at once.
When check mode is enabled, check mode confirmation signal MMMOD is set to "1".

NOTE
After the signal MMOD is turned to "0" during the execution of the program, the program cannot be executed forward and backward.

Control with the manual handle

The value of the parameter No.6410 and the scale factors decide the moving speed of the machine by one pulse generated by a manual handle.
When a manual handle is turned, the actual movement speed of the machine is as follows.
[Feedrate command value] × [Number of the handle pulse per a second]× [Handle magnification]
× ([Parameter setting value]/100) × (8/1000) (mm/min or inch/min)

Example) When feedrate command value is 30mm/min, handle magnification is 100, parameter No.6410 is set to 1 and manual pulse generator is rotated at 100 pulse/rev, the feedrate of axis is decided as follow.
[Feedrate]=30[mm/min] × 100[pulse/s] × 100 × (1/100) × (8/1000)[s] =24[mm/min]

When the feedrate exceeds the override 100% feedrate by turning a manual handle rapidly, the feedrate is clamped at the speed of override 100%. That is, if the pulse of the following formula exceeds "1", the feedrate is clamped.
[Number of the handle pulse per a second]
× [Handle magnification] × ([Parameter setting value]/100) × (8/1000)
The rapid traverse feedrate is clamped at 10%. However, the feedrate of the rapid traverse is clamped at 100% when bit 0 (HDRPD) of parameter No.6400 is set to "1".
And if parameter No.6405 is set to an optional value, it can be clamped to override by nearly optional value.
When the parameter No.6405 is set to larger value than "100", it is clamped to nearly 100%.
When parameter No.6405 is set to "0", the setting of bit 0 (RPO) of parameter No.6400 becomes valid.
The single block signal and the feed hold signal are effective in the checking mode. When the execution of a program is stopped by the single block stop or the feed hold stop, it is necessary to turn ST signal from "1" to "0" in order to restart the program.
In the block with the movement and the block of dwell, the execution speed of the program can be controlled by turning a manual handle. As for the block of neither movement nor dwell such as the block of only address M, S, T, and F, the program advances to the following block even if a manual handle does not turn.
The rotation of the spindle does not synchronize with a pulse of a manual handle. During the checking mode, the spindle rotates at the specified rotation speed. As for the feed per revolution, a program is executed at the feedrate which was converted from the rotation speed of the spindle to the corresponding feed per minute inside CNC.

FRYER MACHINE SYSTEMS INC
Rob Zaranosky

NOTE
The manual handle used by this function is always the first. The 2nd and 3rd manual handle cannot be used in this function.

Forward movement and backward movement with a manual handle

The program is executed forward when a manual handle is turned to the positive direction. And, the program is executed backward when a manual handle is turned to the negative direction.
The program is executed backward if a manual handle is turned to the negative direction in executing the program forward.
When a manual handle keeps being turned in a negative direction, the program is executed backward and the execution stops in the block of O number. Then, if a manual handle is turned to the positive direction, the program is executed forward again.
Even if a manual handle controls the program execution, the program is executed forward in no relation to a pulse generated by a manual handle on setting the manual handle check signal to "0".
The rotation amount of the manual handle is not accepted as an amount of the movement while changing forward movement and backward movement.

Program end

When the block of M2 or M30 is executed, the manual handle retrace ends. It is not possible to execute the program backward from the block of M2 or M30.
When the execution of the program ends, RESET signal must be set to "1", and the checking mode signal and the manual handle check signal must be set to "0".
In 2 path control system, FIN signal must not be set to "1" when the block of M2 or M30 is executed in only one of paths. After the block of M2 or M30 has been executed in both paths, FIN signal is set to "1". (Except for the block of waiting M code is commanded before M2 or M30 in both paths.)

- Notice of the operation

- Dry-run can not operate during the checking mode. Dry-run signal must be set to "0".
- Automatic operation starts immediately with the feedrate commanded by the program, when the checking mode signal or the synchronous operation with handle signal is turned off during executing the program in the checking mode.
- The edit of the program and the change of the parameter and the offset must not be done.
- While the program is being executed backward or blocks once executed backward are being executed forward again, do not change the checking mode signal MMOD.
- The single block operation is also possible. However, even if the manual handle is operated in single block stop state, neither forward movement nor backward movement is done. If cycle start is done, the continuation of the program is executed in the same direction before a single block stop, and the manual handle becomes effective.

- Backward movement of each code

All modal information of G, T, S-code is memorized in executing the program forward. And, the memorized data of the modal G, T, S-code are used in executing the program backward.
As for M-codes, they are grouped and the modal information is managed by parameter No.6411 to 6490. Therefore, M-code can be executed backward according to the information. As for the modal information of the M-code, a change in each group is memorized in the execution data.
As for the codes except for G, M, S, and T, the same code is output between forward movement and backward movement.

- G-code

If G-code that changes modal information is commanded in backward movement, the modal information of previous block is executed.
Example)

```
N1G99;
N2G01X_F_;
```


FRYER MACHINE SYSTEMS INC
Rob Zahensky

N3X_Z_;
N4G98; ..... backward movement starts from this block
N5X_Y_Z_;

If backward movement starts from N4 block, the modal information is changed from G98 to G99 and G99 is executed from N3.
G-code with a movement is traced along the route opposite to forward movement.
G-code that can be command in executing the program backward is as follows.
The other G-codes cannot be command in executing the program backward.
The G-codes in the G-code system B and C also can be used.

Lathe system

G00	G01	G02	G03	G04	G10	G11
G22	G23	G25	G26	G27	G28	G30
G40	G41	G42	G50	G53	G61	G63
G64	G65	G68	G70	G71	G72	G73
G75	G80	G83	G85	G87	G89	G90
G94	G96	G97	G98	G99		

(G-code system A)

Machining center system

G00	G01	G02	G03	G04	G10	G11
G22	G23	G25	G26	G27	G28	G30
G40	G41	G42	G43	G44	G49	G53
G61	G63	G64	G65	G73	G76	G80
G81	G82	G83	G85	G86	G87	G88
G89	G82	G94	G95	G96	G97	

NOTE
1 In Small-Hole Pecking Drilling Cycle(G83), backward movement is prohibited.
2 In forward movement of Boring Cycle(G88), the sequence of actions at bottom of hole is shown as follows (dwell -> stop of spindle motor -> hold state). But in backward movement, that is (rotation of spindle -> hold state -> dwell after restart).
3 G68 for the lathe system means mirror image for double turret.

- M-code

If there is M-code in the same group is commanded in previous blocks, modal information of the M-code, commanded at the last in previous blocks, is output.
If no M-code is commanded in previous blocks, the M-code set to the first parameter in the same M-code group is output.
If M-code is not set to group M-code in parameter, the same M-code is output in backward movement.
If the bit 5 (RVN) of parameter No. 6400 is set to "1", the backward movement is prohibited when the M-code, which is not set to group M-code, is commanded in backward movement.

NOTE
When setting the bit 5(RVN) of parameter No.6400, backward movement prohibition is enabled except the M-code which was set in the grouping but backward movement can be enabled for the following M-code exceptionally.
1. Subprogram Call by M98/M99.
2. Subprogram Call using an M code
3. Macro Call using an M code
4. Waiting M code
5. M0

FRYER MACHINE SYSTEMS INC Rob Zahensky

Example)

Output of M-codes that are set to groups by parameters in backward movement

Setting of parameters:

Bit 2 (MC5) of parameter No.6400=1 and bit 3 (MC8) of parameter No.6400=0 (5 M-codes/group and 16 groups)

No.6411=100
No.6412=101
No.6413=102
No.6414=103
No.6415=104
} Group A

No.6416=200
No.6417=201
No.6418=202
No.6419=203
No.6420=204
} Group B

Program O10 is executed in forward movement from N1 to N15 and backward movement is executed from N15. In backward movement, the output of M-codes is shown as next table.

Table 5.7 (a)

	Forward movement	Backward movement
O10;		
N1G4X1.;		
N2M101;	M101	M100 (*1)
N3G4X1.;		
N4M204;	M204	M200 (*1)
N5G4X1.;		
N6M104;	M104	M101 (*2)
N7G4X1.;		
N8M300;	M300	M300
N9G4X1.;		
N10M200;	M200	M204 (*1)
N11G4X1.;		
N12M0;	M0	M0 (*3)
N13G4X1.;		
N14M102;	M102	M104 (*2)
N15G4X1.;		Backward movement starts from this block
M2;		

*1 No M-code in the same group is commanded before this block, so the M-code, which is set in the 1st. parameter of the same group, is output.

*2 M-code in the same group is commanded before this block, so the M-code, which is commanded at the last before this block, is output.

*3 M-code is not set to group M-code, so the same M-code is output.

- S and T-code

A modal value of the previous block is output.

When movement command and S-code or T-code is commanded in the same block, the timing of the output of the S-code and T-code is different. Because, the timing where S-code and T-code are output at the forward movement is different from that at the backward movement. By setting bit 7 (STO) of parameter No.6401 to "1", the timing of the output of S and T code at the forward movement is the same as the one at the backward movement.

Example)

T-code output timing at the backward movement

T-code is output as follows when the program proceeds backward after the forward movement to N8 block.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Table 5.7 (b)

	Forward movement	Backward movement	
		Parameter STO=0	Parameter STO=1
O1000;			
N1G98G00X0Z0;			Default T output
N2G00X-10.T11;	T11 output	Default T output	
N3G00X100.;			T11 output
N4G00X10.Z20.T22;	T22 output	T11 output	No T code output
N5G00X30.Z30.;			
N6G00X-10.Z-20.;			T22 output
N7G00X50.Z40.T33;	T33 output	T22 output	T33 output
N8G04X5.;		(Backward start)	(Backward start)
M30;			

The “Default T” means a T-code status at N1 block in forward movement. If the status is T0, “T0” signal is output as “Default T” in the backward movement.

The timing of T-code output of N7 and N8 in O1000 shown in the example above is as follows.

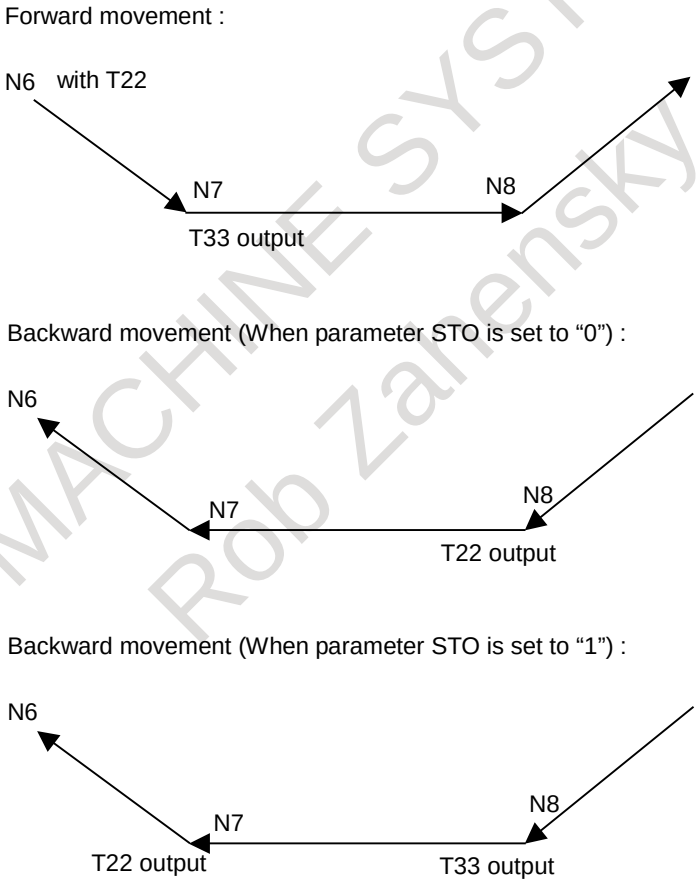


Fig. 5.7 (a)

- Direction change prohibition

The direction change prohibition is a state not changing the direction where the program is executed.
In the state, even if the rotating direction of a manual handle is reversed, the reversed rotation is ignored.
A manual handle must be rotate in the same direction as present direction for removing this state.
The direction change prohibition can be confirmed by output signal MNCHG.

It becomes the change prohibition state under the following condition.

- While the block with the code waiting for FIN is executing

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- After a block has done and until the next block begins to operate
- During thread cutting
- Modal G code of G68.1, G68 and G51.2
- The block with the axis that ends movement earlier in the block with G02 or nonlinear type position (G00) etc.
- During time waiting at the switching of blocks (for multi-path systems only. See "Time waiting in multi-path systems".)

- Backward movement prohibition

The backward movement prohibition is a state that the program cannot be executed from a certain block backward. In this state, the negative rotation of a manual handle is ignored, and the only positive rotation is effective. The program must be executed forward by rotating a manual handle in the positive direction for removing this state.
The backward movement prohibition can be confirmed by output signal MRVSP.
If the following blocks are executed in backward movement, backward movement is prohibited.

- Program number block of main program (except subprogram and macro program)
- Over the maximum number of the blocks for reverse movement
- The block including backward movement prohibition G-code (which is not described in the paragraph "G-code")
- The block which is executed while in modal including backward movement prohibition G-code (which is not described in the paragraph "G-code")

- Status display

In manual handle retrace, the status of manual handle retrace is displayed on clock display of CNC state display line. This status display is displayed during the execution of manual handle retrace. The clock is displayed usually.

When the all condition is filled, "M.H.RTR." is displayed on clock display of CNC state display line. This status is displayed by the color of color number 3 (INPUT KEY, O/N NO. and STATUS are the same color). The screen display is as shown in Fig. 5.7 (b). When the following conditions are not full, the clock is displayed.

(1) When bit 2 (CHS) of parameter No.6401 is set to "0":
  (a) Software option of handle manual retrace is enabled.
  (b) Status display disable/enable bit 6 (HST) of parameter No.6401 is set to "1".
  (c) Check mode confirmation signal MMMOD is set to "1".

(2) When bit 2 (CHS) of parameter No.6401 is set to "1":
  (a) Software option of handle manual retrace is enabled.
  (b) Status display disable/enable bit 6 (HST) of parameter No.6401 is set to "1".
  (c) Cycle start signal STL is set to "1".
  (d) Check mode signal MMOD is set to "1".
  (e) Manual handle check signal MCHK is set to "1".

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Fig. 5.7 (b) "M.H.RTR." status display

Besides, when reverse movement prohibition signal MRVSP is set to "1",
the "NO RVRS." is displayed. This status is displayed by blinking/reversing in the color of color number 1 (ALARM is the same color). The screen display is as shown in Fig. 5.7 (c). When reverse movement prohibition signal MRVSP is set to "0", the " M.H.RTR." is displayed again.

Fig. 5.7 (c) " NO RVRS." status display

Besides, when direction change prohibition signal MNCHG is set to "1" and the direction of program's execution is changed by manual handle, this status display changes from "M.H.RTR." to "NO.CHAG.".
This status is displayed by blinking/reversing in the color of color number 3 (INPUT KEY, O/N NO. and STATUS are the same color) . The screen display is shown as Fig. 5.7 (d). When the program is executed in the direction as the same as before by manual handle or direction change prohibition signal MNCHG is set to "0", the " M.H.RTR." is displayed again.
Moreover, when bit 1 (FWD) of parameter No. 6400 is set to "1" and the program is executed to change direction by manual handle, this status display changes from "M.H.RTR." to "NO.CHAG.".

NOTE

When the improvement of direction change movement in auxiliary function output block is enabled, the state of direction change prohibition signal MNCHG is set to "1" and direction change is possible. Therefore please note that the status display of the direction change prohibition is displayed though it is possible to direction change.

Fig. 5.7 (d) "NO.CHAG." status display

Limitation

- Movement in automatic operation by DNC operation mode(RMT)

In the automatic operation by DNC operation mode(RMT), the backward movement is prohibited though the forward movement is enable.

- Movement in subprogram operation by external subprogram call

In M198 or M-code for subprogram operation by external subprogram call (parameter No.6030), the backward movement is prohibited though the forward movement is enable.

- Movement command and M,S,T-code

When M,S,T-codes and movement commands are in the same block, the timing outputting codes changes between in forward movement and backward movement. Therefore, M, S, T-codes should be commanded in backward movement after confirming that distribution end signal DEN is set to "1".

Example of executing the following programs with lathe system

```
O0001 ;
M5 S0 F0 ;
G53 X0 Z0 ; ..................................................... (1)
G1 W100 M3 S100 F1. ; ........................................... (2)
G0 U50. W50. ; .................................................. (3)
M2 ;
```

FRYER MACHINE INC Rob Zaher

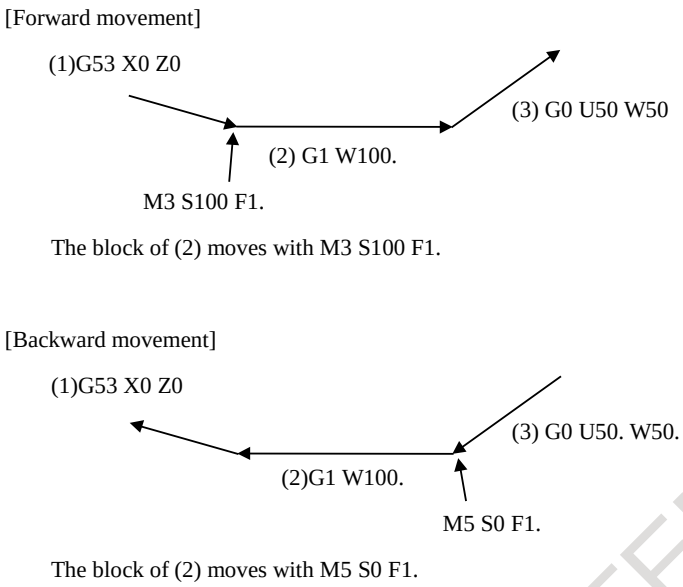


Fig. 5.7 (e)

- Non linear interpolation type positioning

In the non-linear interpolation type positioning, the route is different between forward movement and backward movement.

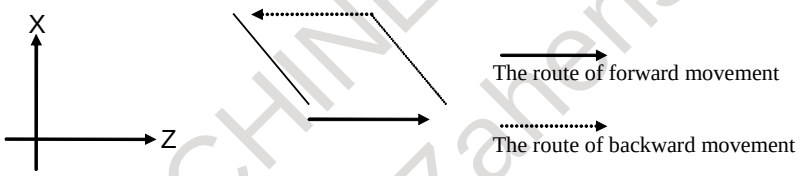


Fig. 5.7 (f)

Please use the interpolation type positioning to ward off danger. (Set the bit 1 (LRP) of parameter No.1401 to "1")
If the non-linear interpolation type positioning is used, backward movement is prohibited at the time when movement ends on any axis.

- Threading

When bit 6 (HRG) of parameter No.6403 is set to 0, threading (G32,G76,G84,G88,G92) is always executed at 100% override speed. That is to say, a pulse generated by a manual handle is ignored in executing a threading block. In thread cutting cycle, the pulse is invalid at the time actually cutting thread, but the one are valid in the other movements.

When bit 6 (HRG) of parameter No.6403 is set to 1, it is possible to move backward and re-forward in the block of threading. However, threading cycle (G92) and multiple threading cycle (G76) are excluded.

> ⚠ **WARNING**
> When the parameter HRG is set to 1, detach the work if you execute the program including the threading because the synchronization of the spindle and the feed axis is not maintained at threading block. The machining of work is impossible.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Rigid tapping (lathe system)

When bit 6 (HRG) of parameter No.6403 is set to 1, it is possible to move backward and re-forward in the block of rigid tapping. However, it is not possible to move backward by passing an on-off command of the rigid tapping mode.

> ⚠ **WARNING**
> When the parameter HRG is set to 1, detach the work if you execute the program including the rigid tapping. The machining of work is impossible.

- Tapping mode (G63)

When bit 6 (HRG) of parameter No.6403 is set to 1, it is possible to move backward and re-forward in the tapping mode.

> ⚠ **WARNING**
> When the parameter HRG is set to 1, detach the work if you execute the program including the tapping mode because the synchronization of the spindle and the feed axis is not maintained. The machining of work is impossible.

- Macro variable

In macro statement, the setting, operation, and so on of the macro variable is executed in only first forward movement. That is to say, the setting, operation, and so on of the macro variable is never executed in the block executing them once.

- Macro modal call

Manual handle retrace function is effective to macro modal call (G66,G66.1).

- Macro executor (execution macro)

Manual handle retrace function is effective while the execution macro of macro executor is called.

- Interruption type custom macro

Do not execute backward movement or re-forward movement after executing the interruption program of interruption type custom macro.

- Manual handle interrupt

Do not operate the manual handle interrupt simultaneously with manual handle retrace function.

- PMC axis control

The movement of PMC axis control cannot be controlled by this function.

- Multiple path simultaneous check in the multi-path system

When using the manual handle retrace function at the same time in multiple paths, the timing of block operation may slightly differ between these paths due to the repetition of forward and backward motion or differences in the rotation speed of the manual handle. To synchronize block operation between the paths, use the wait M code.

FRIEP MACHINE SYSTEMS INC
Rob Zahensky

- Time waiting in multi-path systems

In a multi-path system, control is performed so that during forward movement, the sum of the handle pulses input from the start to the end of each block is stored and during backward movement, the tool does not move to the previous block until as many handle pulses as those stored during forward movement are input. Any handle pulses that are input in an in-position check are also stored as part of the pulses mentioned above. If, therefore, the rotation speed of the handle (that is, the axis movement speed) during forward movement differs from that during backward movement, the time required for the in-position check varies, causing a difference between the sum of the handle pulses stored during forward movement and the sum of the handle pulses during backward movement. If this occurs, even if the block ends during movement (the distance-to-go display is 0), the tool may not move to the previous block until the handle is rotated in the backward movement direction for as many handle pulses as those stored during forward movement. Moreover, in such a case, a direction change is prohibited and, therefore, it is not possible to switch to forward movement until the tool moves to the previous block by continuing backward movement.

- Check of path unit by multi-path system

In the multi-path system, the program check of an arbitrary path is possible. In the path not to check, please select the mode excluding the MEM mode. Even if bit 4 (HMP) of parameter No.6400 is set to "1", it is possible to execute the check of forward movement, direction change and backward movement.
If waiting M code exist in the program, please set No-wait signal of two path NOWT, or No-wait signal of three path NMWT to "1".

- Modal display

In the backward movement with manual handle, the modal display is updated according to the operation condition of the program.

- Modal information

In the backward movement with manual handle, the state of modal information is updated according to the operation condition of the program.

- Change in operation mode

When you change to EDIT mode during the checking mode, the backward movement and the re-forward movement cannot be executed in the blocks which have been already executed.

- ON/OFF of Manual Handle Retrace mode

When check mode signal MMOD is set to "0" and handle available signal in checking mode MCHK is set to "0", the check mode might not be turned off at once.
Basically, in the middle of block, the check mode doesn't switch from ON to OFF or from OFF to ON. After the block is ended, the check mode switches from ON to OFF or from OFF to ON.

- AI contour control

If the checking mode signal MMOD is "1", AI contour control mode (G05.1Q1) is canceled. If AI contour control is canceled, forward movement and backward movement can be executed.
Furthermore, when check mode signal MMOD<Gn067.2> is set to 1 in AI contour control mode, it is enabled a check mode from the block that next buffering is done. Then, AI contour control is canceled.
When parameter FWD (No.6400#1) and parameter HAI (No.6403#7) are 1, AI contour control is effective in the checking mode. Even if it is already AI contour control mode, the checking mode is effective.

- Multi Spindle

During the backward movement, both TYPE-A and TYPE-B multi spindle control may not be operated exactly.

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

- Handle-synchronous feed

When manual handle retrace and a handle-synchronous feed are operated at the same time, forward movement and backward movement are done according to the operation of the first manual handle, and the feedrate of a handle-synchronous feed becomes effective.

- Execution of measurement G-code with the speed of override 100%

When bit 6 (MGO) of parameter No.6400 is set to "1", a handle pulse is invalid and it is always executed at 100% override. When bit 6 (MGO) of parameter No.6400 is set to "0", this function is invalid and a handle pulse is valid.
In the multi-path system, this function is not effective in the execution of another path and handle pulse is valid in another path. The measurement G code to which this function is effective is as follows.
(1) G31 for skip
(2) G31, G31P1, G31P2, G31P3, G31P4, G04, G04Q1, G04Q2, G04Q3 and G04Q4 for multistage skip
(3) G31P99 and G31P98 for torque limit skip

When the measurement by G36 and G37 (G37.1 and G37.2 when bit 3(G36) of parameter No.3405 is set to "1") of automatic tool compensation is executed, a handle pulse is invalid and it is always executed at 100% override regardless of setting bit 6 (MGO) of parameter No.6400. At the rapid traverse before measurement, handle pulse is valid.
When bit 7 (SKF) of parameter No.6200 is set to "0" and bit 2 (SFN) of parameter No.6207 is set to "0", handle pulse at G31 is invalid and it is always executed at 100% override regardless of setting bit 6 (MGO) of parameter No.6400.

When bit 7 (MG4) of parameter No.6400 is set to "1" and the software option of multistage skip is enabled and the setting of bits 0 (IS1) of parameter No. 6202 to bit 7 (DS8) of parameter No. 6206 is enabled, the backward movement prohibition is enabled in G04 block for multistage skip.
The G code to which this function is effective is as follows.
(1) G04, G04Q1, G04Q2, G04Q03 and G04Q4 for multistage skip

- Skip function, Multi-step skip function, High-speed skip function and Torque limit skip function

When bit 5 (HRF) of parameter No.6403 is set to 1, it is possible to move backward and re-forward in the block of G31.
During the backward movement or the re-forward movement, any skip signal and torque limit are ignored and it moves backward or re-forward on the path that moved first time forward movement.
The system variables #100151 to #100182 and #5061 to #5080 of custom macro are not updated during the backward movement and the re-forward movement.

- Reference position return check (G27)

The reference position return end signals ZP1 to ZP8 <Fn094> is not output in the backward movement.

- Polar coordinate interpolation
- Cylindrical interpolation
- 3-dimensional coordinate system conversion

It is not possible to move backward by passing an on-off block of these modes though it is possible to move backward and re-forward in these modes.

- Relation to another function

This function cannot coexist with the following functions
- Retrace

FRYER MACHINE SYSTEMS INC
Rob Zalinsky

5.7.1 Displaying Program Being Executed in Backward Movement

Overview

During executing the program backward by manual handle retrace, the block just before the block being executed can be displayed.
Therefore, it is possible to check what the block to be executed next is in backward movement.

Explanation

In displaying program being executed in backward movement, it is possible to select whether to display the block being executed or the block just before the block being executed at the start of the program.

Selecting method is as follows.

(1) To display the block being executed at the start of the program
Set bit 2 (RPD) of parameter No.11370 to 0.

(2) To display the block just before the block being executed at the start of the program
Set bit 2 (RPD) of parameter No.11370 to 1.
Furthermore, in a display containing look-ahead blocks (bit 1 (APD) of parameter No.11350 to 0), set bit 3 (FPD) of parameter No.11308 to 1.

When setting the case (2), the block just before the block selected by the cursor can be displayed.

- For a display containing look-ahead blocks (bit 1 (APD) of parameter No.11350 = 0)

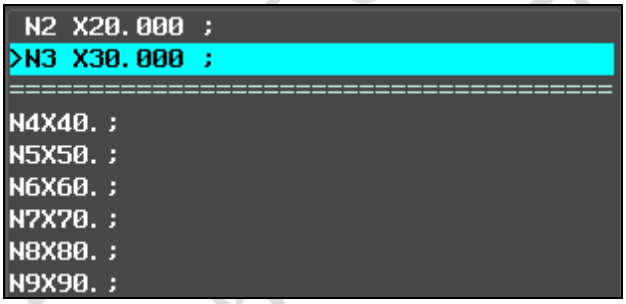


Fig. 5.7.1 (a) Displaying program being executed in backward movement

- For a text display (bit 1 (APD) of parameter No.11350 = 1)

Fig. 5.7.1 (b) Displaying program being executed in backward movement

FRYER MANUFACTURING SYSTEMS INC
Robert Lucky

5.8 AUXILIARY FUNCTION OUTPUT BLOCK REVERSE MOVEMENT FOR MANUAL HANDLE RETRACE

Overview

This function enables reverse movement during manual handle retrace even if a move command and an auxiliary function (M/S/T/B code) are specified in the same block.

Notes

NOTE

1 When a single auxiliary function is specified individually, carry out the regular auxiliary function completion sequence. Reverse movement becomes possible after moving to the next (or previous) block.
2 If a move command and an auxiliary function are specified in the same block and if no travel distance remains after movement, carry out the regular auxiliary function completion sequence. Reverse movement becomes possible after moving to the next (or previous) block.
3 To enable reverse movement using this function when an M code other than grouped M codes and a move command are specified in the same block, set bit 5 (RVN) of parameter No. 6400 to "0".
4 This function is not supported for M02 or M30.
5 If an auxiliary function is specified in a block having no move command, such as G70, carry out the regular auxiliary function completion sequence.
Reverse movement becomes possible after moving to the next (or previous) block.

5.9 MANUAL HANDLE RETRACE FUNCTION FOR MULTI-PATH

Overview

In the manual handle retrace function for multi-path systems, when the operation of re-forward movement is performed, the movement timing of all-path movement can be made identical to that of forward movement.

Explanation

In the manual handle retrace function for multi-path systems, when a certain path stopped with the backward movement prohibited and the other paths continue the backward movement, if re-forward movement is performed after that, all paths start the re-forward movement immediately, so the timing of block movement of each path is different from the original forward movement.
In this function, during re-forward movement, the path that stopped with backward movement prohibited doesn’t start re-forward movement immediately. After other paths reach the position the path stopped with backward movement prohibited, the re-forward movement of this path is performed. Therefore, in multi-path systems, it is possible to perform re-forward movement in which the timing of block movement of each path is made identical to that of forward movement.

NOTE

Even when this function is enabled, the timing of block movement of each path may differ slightly due to the repetition of forward and backward movement and the rotation speed of the manual handle. Therefore, when synchronization is necessary, use the waiting M code.

FERVER MACHINE SYSTEMS INC
Rob Zahensky

Example) Re-forward movement of manual handle retrace in a 2-path system

```
Path1            Path2
O1001            O1001
N1 G1 X5.0 F1    N1 G1 Z6.0 F1
N2 X12.0         N2 G31 Z14.0
N3 X22.0         N3 Z28.0
N4 X30.0         N4 Z36.0
N5 X34.5         N5 Z50.0
N6 X50.0         M30
M30
```

When the above programs are operated in each path, the operation states of individual steps (in forward, backward, and re-forward movement) are shown below.

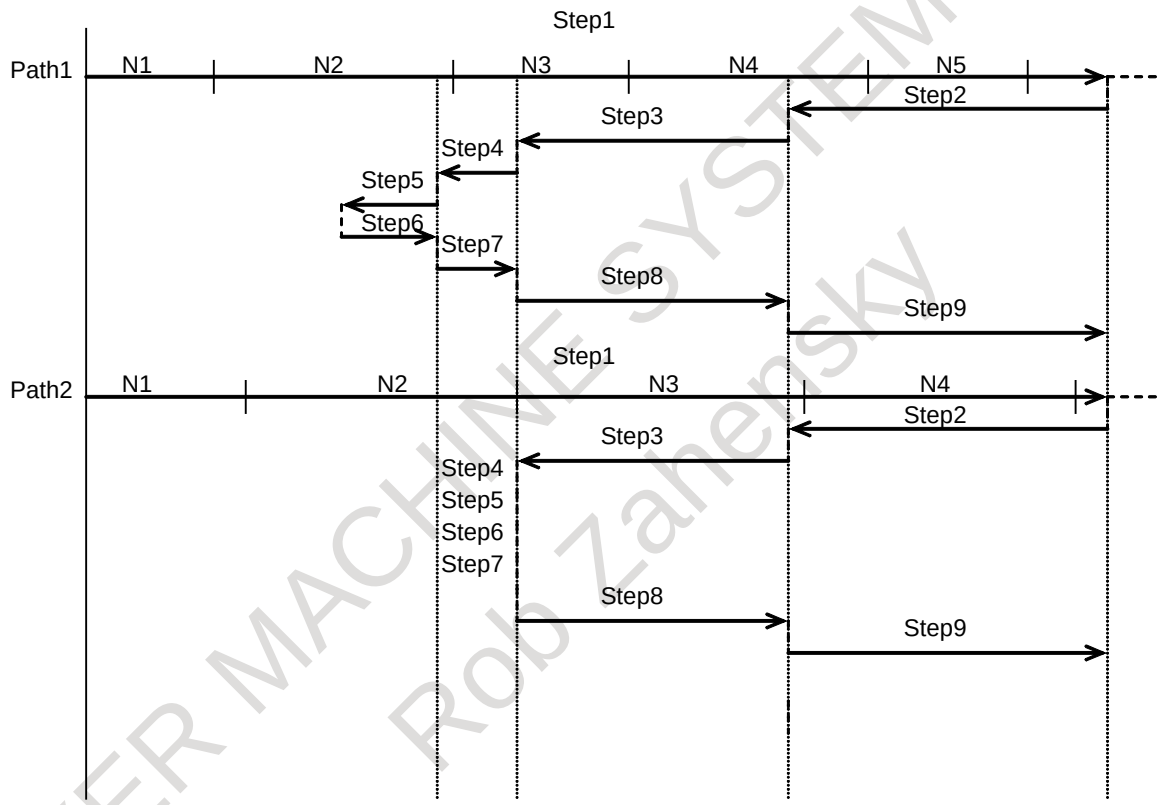


Fig. 5.9 (a)

STEP	1	2	3	4	5	6	7	8	9
Command state	Forward	Backward	Backward	Backward	Backward	Re-forward	Re-forward	Re-forward	Re-forward
Execution state in Path1	Forward	Backward	Backward	Backward	Backward	Re-forward	Re-forward	Re-forward	Re-forward
Execution state in Path2	Forward	Backward	Backward	Stop	Stop	Wait P1	Wait P1	Re-forward	Re-forward

However, when bit 4 (HMP) of parameter No. 6400 is 0 (even if a path is prohibited from changing movement direction, the other paths can still change it), if operation is performed in the condition shown in the example below, re-forward movement may not keep the same motion as in forward movement.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Example)

Conditions of the programs of the individual paths

Path1 In backward movement, there is a block that prohibits a change to forward movement.

Path2 In backward movement, there is neither block that prohibits backward movement nor block that prohibits a change to forward movement.

In the above conditions, the following operation is performed.

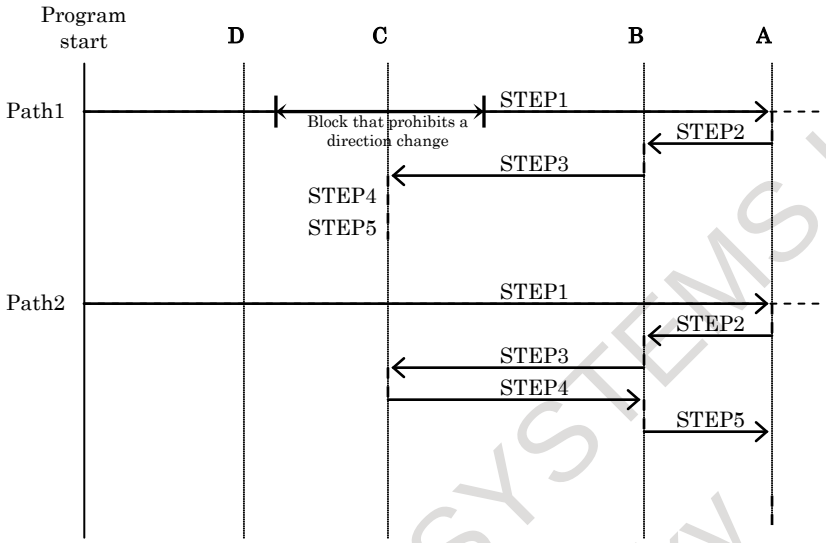


Fig. 5.9 (b)

STEP1: Forward movement is performed to position A in the figure.

STEP2, STEP3: Backward movement is performed to position C in the figure.

STEP4, STEP5: Re-forward movement is performed to position A in the figure. Since path 1 is prohibited from changing its direction, only path 2 performs re-forward movement.

To keep synchronization in re-forward movement, perform backward movement of path 1 until path 1 passes by block D, which prohibits a direction change, and start re-forward movement of path 1.
In this case, if bit 4 (HMP) of parameter No.6400 is set to 1 (if a path is prohibited from changing its movement direction, the other paths cannot also change their direction), the other paths cannot change their direction until path 1 passes by the position D.

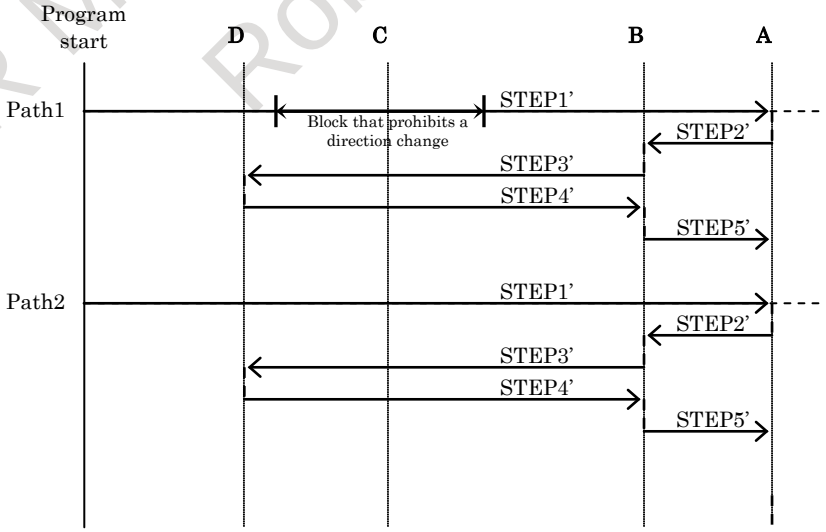


Fig. 5.9 (c)

STEP1': Forward movement is performed to position A in the figure.

STEP2',STEP3': Backward movement is performed to position D in the figure.

STEP4',STEP5: Re-forward movement is performed to position A in the figure.

FRYER MACHINE SYSTEMS INC Rob Zahensky

5.10 EXTENSION OF THE MANUAL HANDLE RETRACE FUNCTION

Overview

In manual handle retrace function, the following operations that were impossible so far become available.

(1) Forward movement of rigid tapping
(2) Forward movement of threading
(3) Forward movement of PMC axis control
(4) Backward movement of the orientation by the G00 command on the Cs contour control axis
(5) Backward movement of polygon turning with two spindles
(6) Backward movement of balance cutting

This function is included in the multi-path manual handle retrace function.

Explanation

- Rigid tapping

- When the bit 0 (HRA) of parameter No.6403 is set to 0 (Conventional specification)
  (a) When rigid tapping starts during forward movement, handle operation is disabled and rigid tapping is executed at a speed of 100% override.
  (b) Backward movement cannot be performed in the rigid tapping mode. In addition, when the rigid tapping block is reached after rigid tapping is completed in forward movement, backward movement cannot be done any more.
- When the bit 0 (HRA) of parameter No.6403 is set to 1 (Extensional specification)
  (a) In tapping during forward movement, the operation is performed at a speed according to handle operation.
  (b) Backward movement in rigid tapping is the same movement as in the case where bit 0 (HRA) of parameter No. 6403 is 0.

- Thread cutting

- When the bit 0 (HRA) of parameter No.6403 is set to 0 (Conventional specification)
  (a) When the threading block is reached during forward movement, handle operation become disabled and the operation is performed at a speed of 100% override.
  (b) Backward movement cannot be performed during threading. In addition, when the threading block is reached during backward movement after threading is completed in forward movement, backward movement cannot be done any more.
- When the bit 0 (HRA) of parameter No.6403 is set to 1 (Extensional specification)
  (a) In threading during forward movement, the operation is performed at a speed according to handle operation.
  (b) Backward movement in threading is the same movement as in the case where bit 0 (HRA) of parameter No. 6403 is 0.

> ⚠ **WARNING**
> In rigid tapping or threading during manual handle retrace, the workpiece cannot be actually machined.

- PMC axis control

- When the bit 1 (HRB) of parameter No.6403 is set to 0 (Conventional specification)
  (a) The PMC axis operates independently of handle operation.
- When the bit 1 (HRB) of parameter No.6403 is set to 1 (Extensional specification)
  (b) In PMC axis control during forward movement, the PMC axis is operated at a speed according to handle operation but, backward movement cannot be performed.

FRYER MACHINE SYSTEMS INC
Ron Zahensky

NOTE

1 Even when the axis specified by the program moves backward, the axis specified by PMC axis control cannot move backward. Therefore, the relative position between the axis specified by the program and the axis specified by PMC axis control changes when backward movement is performed while the axis that is specified by PMC axis control is operating. In this case, it is necessary to add the following PMC ladder process.
   - When the axis specified by PMC axis control is operating, prohibit the backward movement of the axis specified by the program using reverse movement prohibition signal MRVM.

2 Alarm PS0130, “NC AND PMC AXIS ARE CONFLICTED” occurs when one of the following operations is executed.
   (1) In the backward movement, when PMC axis control is specified for the axis specified by the program in the previous block.
   (2) During PMC axis control, when a direction change is performed for the axis specified by the program in the previous block.

3 In the skip (03h), continuous feed (06h), speed command (10h), torque control (11h), and auxiliary function (12h,13h,14h) of PMC axis control, even if bit 1 (HRB) of parameter No. 6403 is set to 1, the axis operates independently of handle operation.

⚠ **WARNING**

During the manual handle retrace, if a reset is made when a command by PMC axis control is not completed, the command by the program stops, but the command by PMC axis control continues. In this case, even if bit 1 (HRB) of parameter No. 6403 is set to 1, the command by PMC axis control operates independently of handle operation. Make a reset after confirming that the command by the PMC axis control is completed.

- Orientation of Cs contour control axis

- When the bit 2 (HRC) of parameter No.6403 is set to 0 (Conventional specification)
  (a) Backward movement cannot be performed in orientation movement by the first G00 command after the Cs contour control mode is turned on. Therefore, when the above G00 command is reached during backward movement, backward movement cannot be done any more.
  (b) Backward movement can be performed in the reference point return command (G28) of the Cs contour control axis, but the processing proceeds to the previous block without performing orientation movement.
- When the bit 2 (HRC) of parameter No.6403 is set to 1 (Extensional specification)
  (a) Backward movement can be performed in the orientation operation by the G00 command, but the processing proceeds to the previous block without performing orientation movement as in G28 command.
  (b) The reference point return command (G28) of the Cs contour control axis performs the same movement as in the case where bit 2 (HRC) of parameter No. 6403 is 0.

⚠ **WARNING**

The workpiece cannot be actually machined because the orientation movement of G00 and G28 is not executed during backward movement in Cs contour control.

FRYER MACHINE SYSTEMS INC Rob Zahensky

T

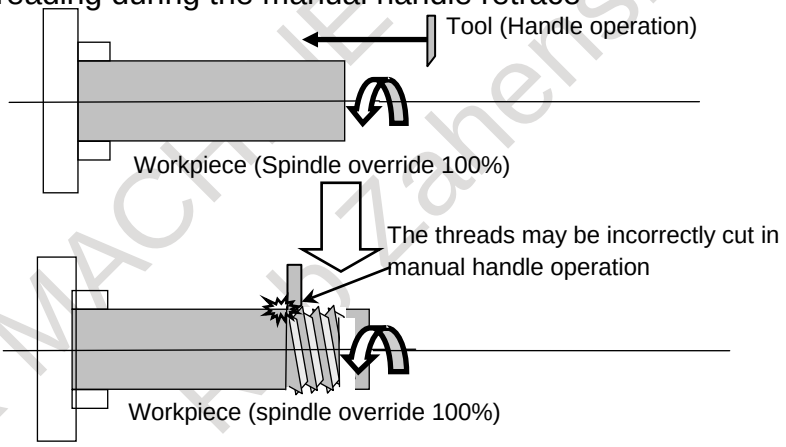
- Polygon machining with two spindles

- When the bit 3 (HRD) of parameter No.6403 is set to 0 (Conventional specification)
  - (a) Axes other than the polygonal synchronization axes operate at the speed according to handle operation even in the polygon machining mode. However, backward move cannot be performed in the block specified in the polygonal machining mode.
  - (b) When the polygon machining block (G50.2) is reached in backward movement after the polygon machining is completed in forward movement, backward movement cannot be done any more.
- When the bit 3 (HRD) of parameter No.6403 is set to 1 (Extensional specification)
  - (a) It is possible to move backward or re-forward in the block specified in the polygonal machining mode. When backward movement is performed during polygon machining, however, if the polygon machining-on block (G51.2) is reached, backward movement cannot be done any more.
  - (b) The operation when the polygon machining block (G50.2) is reached in backward movement after the polygon machining is completed in forward movement the same as in the case where bit 3 (HRD) of parameter No. 6403 is 0.

⚠ **WARNING**

In the threading and polygon machining between two spindles, the spindle operates at a speed of override 100% instead of the speed according to handle operation. Therefore, the workpiece cannot be actually machined.

(Example) Treading during the manual handle retrace



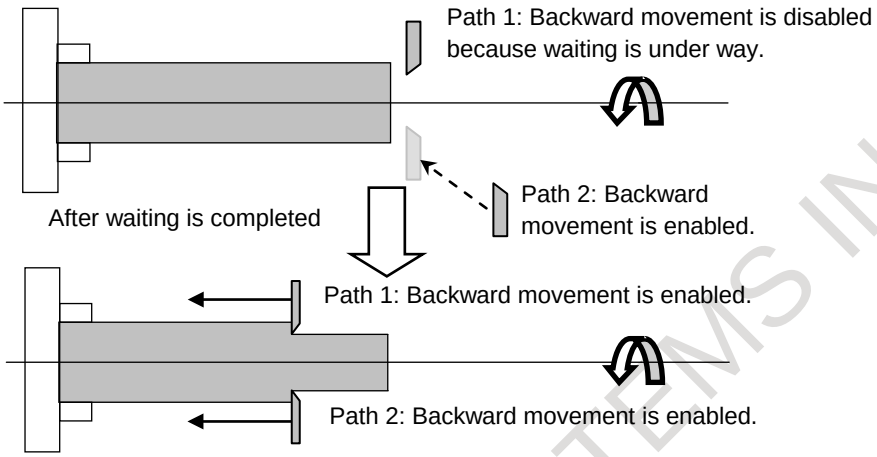
- Balanced cutting

- When the bit 4 (HRE) of parameter No.6403 is set to 0 (Conventional specification)
  - (a) In the balance cutting mode, axes operate at the speed according to handle operation, but backward movement cannot be performed.
  - (b) When the balance cutting block (G69) is reached in backward movement after the balance cutting is completed in forward movement, backward movement cannot be done any more.
- When the bit 4 (HRE) of parameter No.6403 is set to 1 (Extensional specification)
  - (a) It is possible to move backward or re-forward in the block specified in the balance cutting mode. When backward movement is performed in the balance cutting mode, if the balance cut-on block (G68) is reached, backward movement cannot be done any more.
  - (b) The operation when the balance cutting block (G69) is reached in the backward movement after the balance cutting was completed in forward movement is the same operation as in the case where bit 4 (HRE) of parameter No. 6403 is 0.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

1 In the balance cutting mode, waiting is done so that the block starts at the same time in each path. Backward movement cannot start during the waiting. After the waiting is completed and the next block starts, backward movement becomes possible.



2 In all paths that uses balance cutting, set bit 4 (HRE) of parameter No. 6403 to 1.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

6 SAFETY FUNCTIONS

To immediately stop the machine for safety, press the Emergency stop button. To prevent the tool from exceeding the stroke ends, Overtravel check and Stored stroke check are available. This chapter describes emergency stop, overtravel check, and stored stroke check.

6.1 EMERGENCY STOP

If you press Emergency Stop button on the machine operator's panel, the machine movement stops in a moment.

Fig. 6.1 Emergency stop

This button is locked when it is pressed. Although it varies with the machine tool builder, the button can usually be unlocked by twisting it.

Explanation

EMERGENCY STOP interrupts the current to the motor.
Causes of trouble must be removed before the button is released.

6.2 OVERTRAVEL

When the tool reaches the stroke end set by the machine tool limit switch, it decelerates and stops because of working the limit switch and an OVERTRAVEL alarm is displayed.

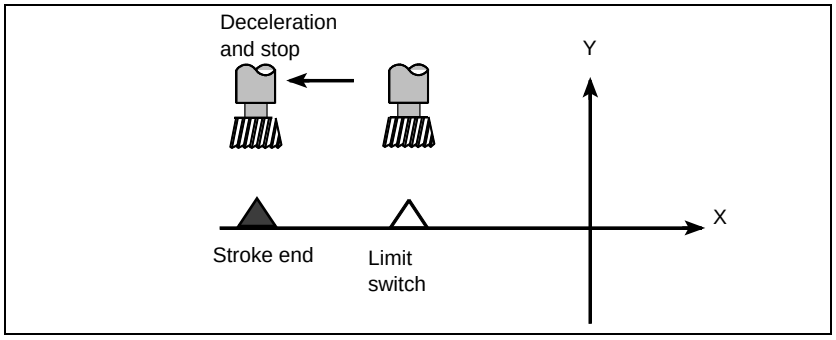


Fig. 6.2 Overtravel

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

- Overtravel during automatic operation

When the tool touches a limit switch along an axis during automatic operation, the tool is decelerated and stopped along all axes and an overtravel alarm is displayed.

- Overtravel during manual operation

In manual operation, the tool is decelerated and stopped only along the axis for which the tool has touched a limit switch. The tool still moves along the other axes.

- Releasing overtravel

Press the reset button to reset the alarm after moving the tool to the safety direction by manual operation. For details on operation, refer to the operator's manual of the machine tool builder.

Alarm

Alarm No.	Message	Description
OT0506	+ OVERTRAVEL (HARD)	The stroke limit switch in the positive direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.
OT0507	- OVERTRAVEL (HARD)	The stroke limit switch in the negative direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.

6.3 STORED STROKE CHECK

Three areas which the tool cannot enter can be specified with stored stroke check 1, stored stroke check 2, and stored stroke check 3.

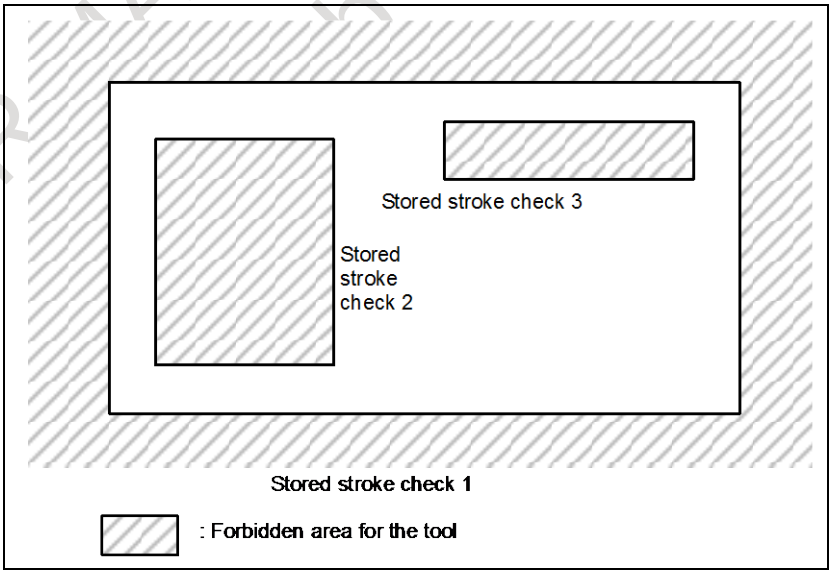


Fig. 6.3 (a) Stroke check

The following shows the areas which the tool cannot enter for each stored stroke check.

- Stored stroke check 1: Outside
- Stored stroke check 2: Outside or inside (switchable)
- Stored stroke check 3: Inside

FRYER MACHINE SYSTEMS INC
Zahensky

When the tool moves into the forbidden area, an alarm is displayed and the tool is decelerated and stopped.
When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.

Explanation

- Stored stroke check 1

Parameters (Nos. 1320, 1321 or Nos. 1326, 1327) set boundary. Outside the area of the set limits is a forbidden area. The machine tool builder usually sets this area as the maximum stroke.
When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.
At this time, a signal (overtravel alarm signal) can be output to the PMC if bit 6 (OTS) of parameter No. 1301 is set to 1. In addition, when the tool enters the forbidden area during manual operation, the signal (overtravel alarm signal) can be output to the PMC without generating the alarm by setting bit 1 (NAL) of parameter No. 1300 to 1. With this parameter setting, the alarm is generated when the tool enters the forbidden area during automatic operation.

⚠ CAUTION
1 If the two points for specifying a forbidden area are identical, all areas are handled as forbidden areas for stored stroke check 1.
2 The size of a forbidden area must be set carefully. If the size is set incorrectly, the stroke becomes infinite.

NOTE
1 Bit 7 (BFA) of parameter No. 1300 can be used to specify whether to stop the tool before it enters the forbidden area or when it enters the area.
2 The forbidden area must be specified with correct positional relationships. An incorrect area setting results in stroke infinity.

- Stored stroke check 1 area expansion

Stored stroke check 1 area expansion enables definition of eight areas using parameters Nos. 1320, 1321, 1326, 1327, 1350 to 1361. Stored stroke check 1 area expansion is an optional function.

NOTE
The basic notes are the same as those of stored stroke check 1.

- Stored stroke check 2

Parameters Nos. 1322, 1323 or commands set these boundaries. Inside or outside the area of the limit can be set as the forbidden area. Bit 0 (OUT) of parameter No. 1300 selects either inside or outside as the forbidden area.
In case of program command a G22 command forbids the tool to enter the forbidden area, and a G23 command permits the tool to enter the forbidden area.
Each of G22; and G23; should be commanded independently of another commands in a block.
The command in the Fig. 6.3 (b) creates or changes the forbidden area:

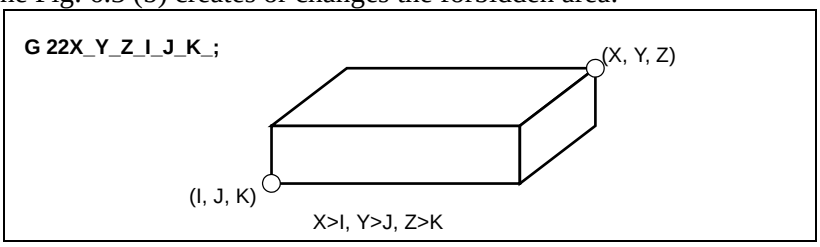


Fig. 6.3 (b) Creating or changing the forbidden area using a program

FRYER MACHINE SYSTEMS INC
Rob Zaranosky

When setting the area by parameters, points A (Parameter No.1322) and B (Parameter No.1323) in the Fig. 6.3 (c) must be set.

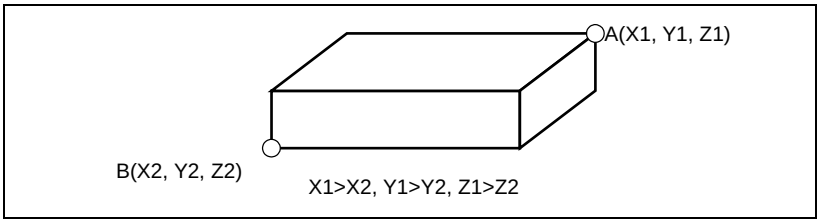


Fig. 6.3 (c) Creating or changing the forbidden area using a parameters

The values X1, Y1, Z1, X2, Y2, and Z2, which are set by parameters Nos. 1322 and 1323, must be specified by the distance from the machine coordinate system (machine unit). The values X, Y, Z, I, J, and K, which are set by a G22 command, must be specified by the distance in the least input increment (input unit).
When data is set by the program command, the command valued is converted in the machine increment and it set to the corresponding parameter (No.1322 and No.1323).

- Format

Stored stroke check 2 on

G22 X__ Y__ Z__ I__ J__ K__ ;

X, Y, Z:Coordinate of stored stroke check 2 in the positive direction on each axis
I, J, K: Coordinate of stored stroke check 2 in the negative direction on each axis

Use addresses X, Y, and Z or addresses I, J, and K. Addresses X and I specify the forbidden area related to the X-axis of the basic three axes. Addresses Y and J specify the forbidden area related to the Y-axis of the basic three axes. Addresses Z and K specify the forbidden area related to the Z-axis of the basic three axes. When addresses are omitted, a stroke check is made based on the values set by parameters.

Stored stroke check 2 off

G23 ;

- Stored stroke check 3

Set the boundary with parameters Nos. 1324 and 1325. The area inside the boundary becomes the forbidden area. The values X1, Y1, Z1, X2, Y2, and Z2 must be set as coordinates (machine unit) in the machine coordinate system.

⚠ CAUTION

1 If the two points for specifying a forbidden area are identical, all areas are handled as movable areas in case of stored stroke check 2 that the inside area is forbidden (bit 0 (OUT) of parameter No. 1300 = 0), or stored stroke check 3. And all areas except the specified point are handled as forbidden areas in case of stored stroke check 2 that the outside area is forbidden (the parameter OUT = 1).
2 Even if the two points for specifying a forbidden area are erroneously set, the rectangular parallelepiped having the points as vertices is assumed as a boundary.
3 Since an axis without the reference position return function has no forbidden areas, there are no alarms about forbidden areas for the axis.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Checkpoints in the forbidden area

The parameter setting or programmed value (XYZIJK) depends on which part of the tool or tool holder is checked for entering the forbidden area.
If point A (the top of the tool) is checked in Fig.6.3 (d) and Fig. 6.3 (e), the machine coordinates value that corresponds to distance "a" should be set as the data for the stored stroke limit function. If point B (the tool chuck) is checked, the machine coordinates value that corresponds to distance "b" must be set. When checking the tool tip (like point A), and if the tool length varies for each tool, setting the forbidden area for the longest tool requires no re-setting and results in safe operation.

- For machining center system

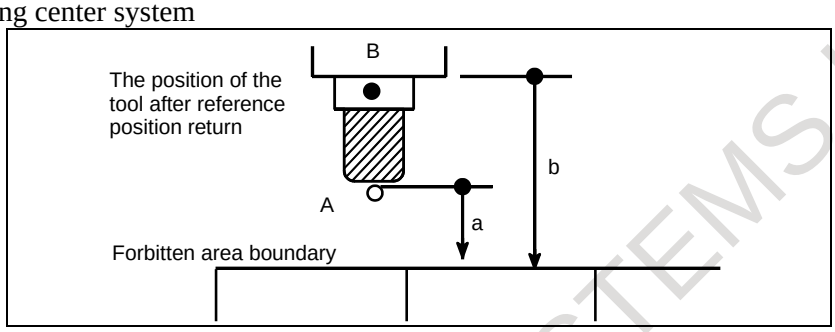


Fig.6.3 (d) Setting the forbidden area

- For lathe system

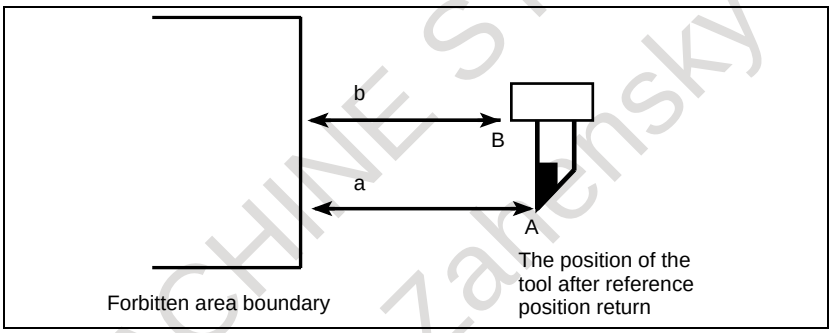


Fig. 6.3 (e) Setting the forbidden area

- Forbidden area overlapping

Area can be set in piles.

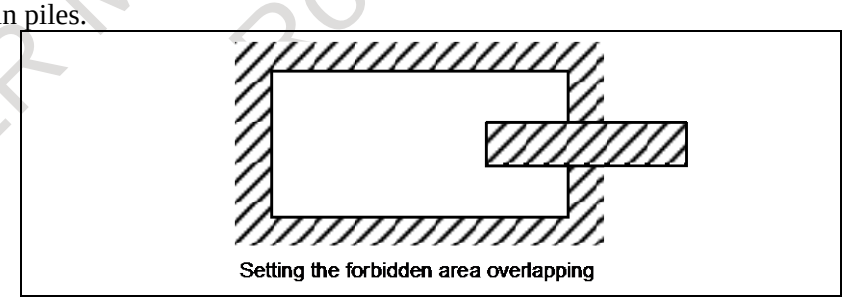


Fig. 6.3 (f) Setting the forbidden area overlapping

Unnecessary limits should be set beyond the machine stroke.

- Condition under which each check is enabled

Each check becomes effective after the power is turned on and manual reference position return or automatic reference position return by G28 has been performed. (In case not using absolute pulse coder)
After the power is turned on, if the reference position is in the forbidden area of each limit, an alarm is generated immediately. (Only in G22 mode for stored stroke check 2).

FRYER MACHINE SYSTEMS INC
Bob Zaharsky

- Releasing the alarms

If the enters a forbidden area and an alarm is generated, the tool can be moved only in the backward direction. To cancel the alarm, move the tool backward until it is outside the forbidden area and reset the system. When the alarm is canceled, the tool can be moved both backward and forward.

- Change from G23 to G22 in a forbidden area

When G23 is switched to G22 in the forbidden area, the following results.
<1> When the forbidden area is inside, an alarm is informed in the next move.
<2> When the forbidden area is outside, an alarm is informed immediately.

- Timing for displaying an alarm

In stored stroke check 1/2/3, parameter BFA (bit 7 of No. 1300) selects whether an alarm is displayed immediately before the tool enters the forbidden area or immediately after the tool has entered the forbidden area.

NOTE
1 Because checking is made with higher precision than the precision of indicated coordinates, an OT alarm may be issued even when coordinates on a boundary is indicated.

Alarm

Number	Message	Description
OT0500	+ OVERTRAVEL (SOFT 1)	A movement in the positive direction exceeded stored stroke check 1.
OT0501	- OVERTRAVEL (SOFT 1)	A movement in the negative direction exceeded stored stroke check 1.
OT0502	+ OVERTRAVEL (SOFT 2)	A movement in the positive direction exceeded stored stroke check 2.
OT0503	- OVERTRAVEL (SOFT 2)	A movement in the negative direction exceeded stored stroke check 2.
OT0504	+ OVERTRAVEL (SOFT 3)	A movement in the positive direction exceeded stored stroke check 3.
OT0505	- OVERTRAVEL (SOFT 3)	A movement in the negative direction exceeded stored stroke check 3.

6.4 STROKE LIMIT CHECK BEFORE MOVE

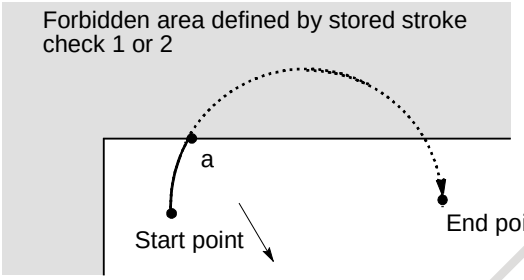
During automatic operation, before the movement specified by a given block is started, whether the tool enters the forbidden area defined by stored stroke check 1, 2, or 3 is checked by determining the position of the end point from the current position of the machine and a specified amount of travel. If the tool is found to enter the forbidden area defined by a stored stroke limit, the tool is stopped immediately upon the start of movement for that block, and an alarm is displayed.

FRYER MACHINE SYSTEMS INC
Rob Zaranosky

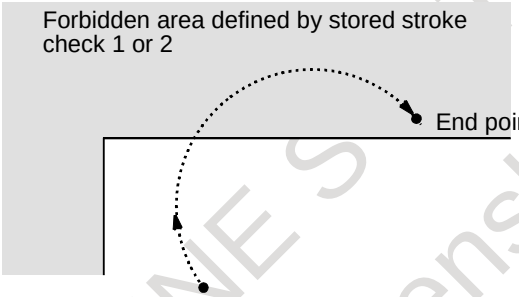
⚠ CAUTION

Whether the coordinates of the end point, reached as a result of traversing the distance specified in each block, are in a forbidden area is checked. In this case, the path followed by a move command is not checked. However, if the tool enters the forbidden area defined by stored stroke check 1, 2, or 3, an alarm is issued. (See the examples below.)

Example 1)

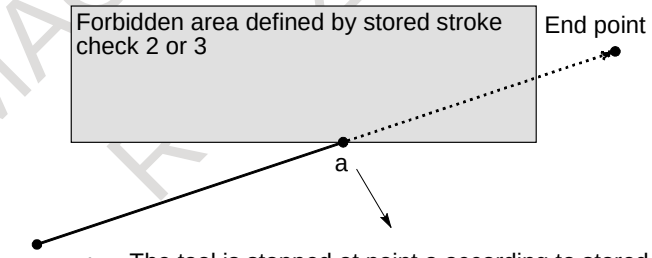


The tool is stopped at point a according to stored stroke check 1 or 2.

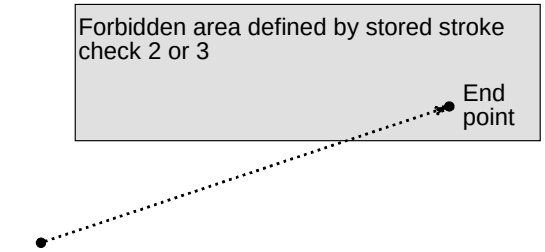


Immediately upon movement commencing from the start point, the tool is stopped to enable a stroke limit check before moving to be performed before movement.

Example 2)



The tool is stopped at point a according to stored stroke check 2 or 3.



Immediately upon movement commencing from the start point, the tool is stopped to enable a stroke limit check before moving to be performed before movement.

FRYER MACHINE SYSTEMS INC
R. Zarensky

Explanation

This function is enabled by setting bit 7 (PLC) of parameter No. 1301 to 1.
When a stroke limit check before moving is performed, whether to check the movement performed by a G31 (skip) block and G37 (automatic tool length measurement) block can be determined using (bit 2 (NPC) of parameter No. 1301).

Limitation

- Machine lock

If machine lock is applied at the start of movement, no stroke limit check made before movement is performed.

- G23

When stored stroke check 2 is disabled (G23 mode), no check is made to determine whether the tool enters the forbidden area defined by stored stroke check 2.

- Program restart

When a program is restarted, an alarm is issued if the restart position is within a forbidden area.

- A block consisting of multiple operations

If a block consisting of multiple operations (such as a canned cycle) is executed, an alarm is issued at the start point of any operation whose end point falls within an inhibited area.

- Cylindrical interpolation mode

In cylindrical interpolation mode, no check is made.

T

- Polar coordinate interpolation mode

In polar coordinate interpolation mode, no check is made.

- 3-dimensional coordinate system conversion

In 3-dimensional coordinate system conversion mode, no check is made.

- PMC axis control

No check is made for a movement based on PMC axis control.

Alarm

Number	Message	Description
OT0510	+ OVERTRAVEL (PRE-CHECK)	The block end point was found in the + side stroke limit prohibition area during a stroke check before movement. Modify the program.
OT0511	- OVERTRAVEL (PRE-CHECK)	The block end point was found in the - side stroke limit prohibition area during a stroke check before movement. Modify the program.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

6.4.1 Check of the Tool Path between Blocks by Stroke Limit Check before Move

Overview

Check of the tool path between blocks by Stroke limit check before move can be used.

Specification

By setting bit 0 (SBA) of parameter No.1302 to 1, whether the tool enters the forbidden area defined by Stored stroke limit 1, 2, or 3 is checked on the tool path of movement command in addition to checking the end point. (Hereinafter, this check is called “check of the tool path between blocks”.)
Check of the tool path between blocks is effective in Positioning (G00), Liner interpolation (G01), Circular interpolation (G02, G03), High-speed reference position return (G28, G30), or Selecting a machine coordinate system (G53).
If the tool enters the forbidden area defined by stored stroke check 1, 2, or 3, an alarm is issued when check of the tool path between blocks is not effective.

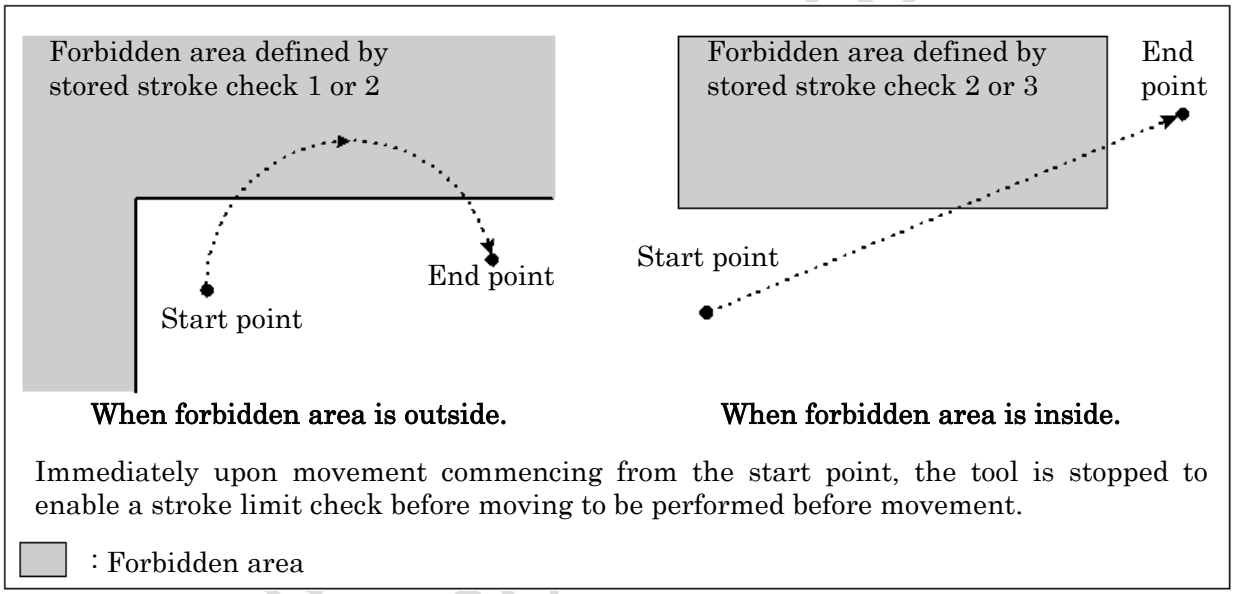


Fig. 6.4.1 (a) Check of the tool path between blocks

Explanation

Plane selection axes

When one axis or both axes among the selection axes of selected plane (G17, G18 or G19 plane) not exist (Hereinafter, this selection axes are called “plane selection axes”.), check of the tool path between blocks is not executed.

Target axis of check of the tool path between blocks

Check of the tool path between blocks is executed on the 3-dimension which is composed by the two axes of plane selection axes and the one axis other than plane selection axes (Hereinafter, this axis is called “plane non-selection axis”).

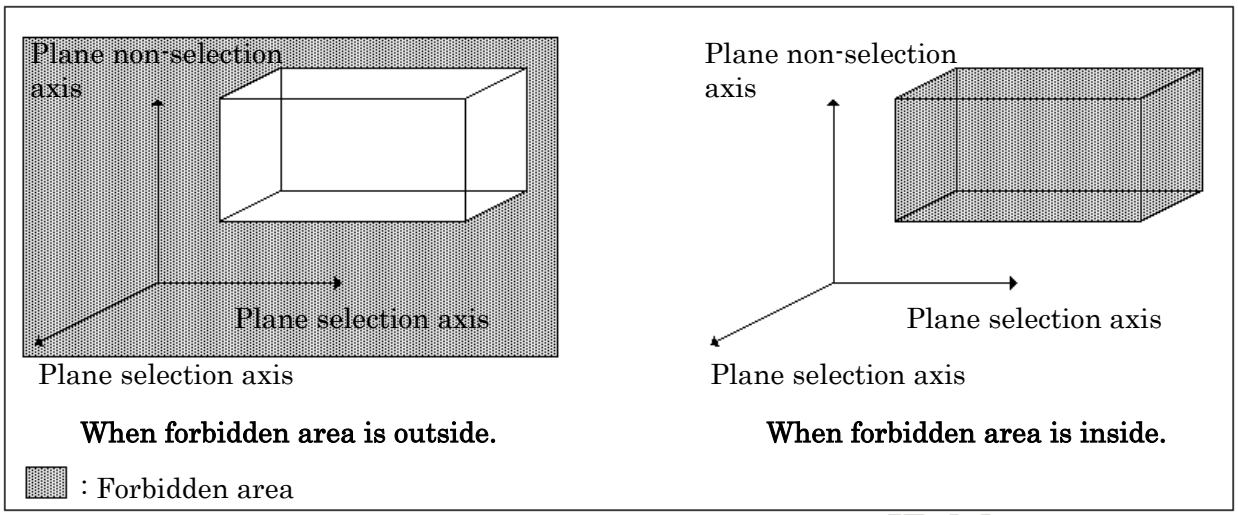


Fig. 6.4.1 (b) Target axis of check of the tool path between blocks

Normally, the basic axis other than plane selection axes in basic coordinate system is set to plane non-selection axis.

Example 1)
When X-Y plane (G17) is selected with the following axis configuration, plane selection axes are X axis and Y axis, and plane non-selection axis is Z axis. (Fig. 6.4.1 (c))
No.1022
(X)=1: X axis of the basic three axes
(Y)=2: Y axis of the basic three axes
(Z)=3: Z axis of the basic three axes
(W)=7: Axis parallel to Z axis
(A)=0: Rotary axis

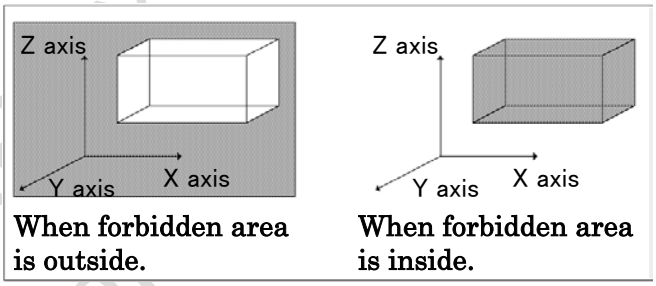


Fig. 6.4.1 (c) Target axis in Example 1

However, there are the following exceptions.

(1) The basic axis other than plane selection axes in basic coordinate system does not exist.

(1) - 1 When parallel axis does not exist.
In this case, check of the tool path between blocks is executed on the plane composed by plane selection axes.

Example 2)
When X-Y plane (G17) is selected with the following axis configuration, plane selection axes are X axis and Y axis, and plane non-selection axis does not exist. (Fig. 6.4.1 (d))
No.1022
(X)=1: X axis of the basic three axes
(Y)=2: Y axis of the basic three axes
(A)=0: Rotary axis

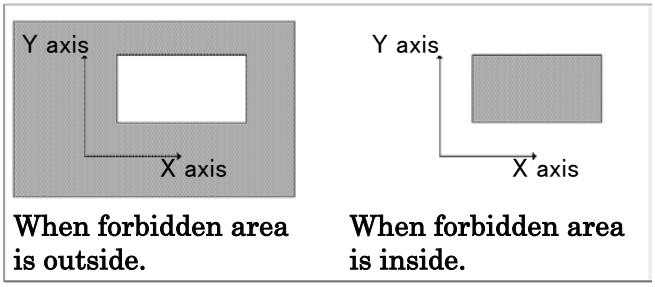


Fig. 6.4.1 (d) Target axis in Example 2

FRYER MACHINE SYSTEM Rob Za

(1) - 2 When the parallel axis exists.

In this case, the parallel axis of lowest number of intra-path relative axis number which is parallel to the basic axis other than plane selection axes is set to plane non-selection axis.

Example 3)
When X-Y plane (G17) is selected with the following axis configuration, plane selection axes are X axis and Y axis, and plane non-selection axis is W axis. (Fig. 6.4.1 (e))
No.1022
(X)=1: X axis of the basic three axes
(Y)=2: Y axis of the basic three axes
(W)=7: Axis parallel to Z axis
(A)=0: Rotary axis

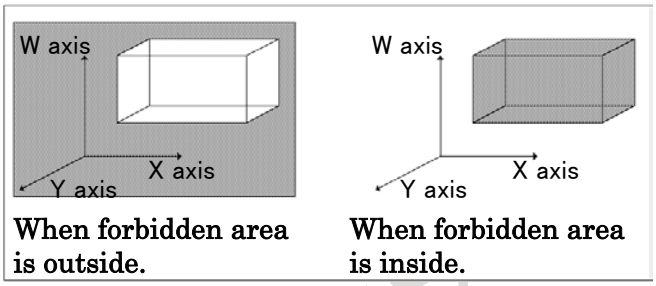


Fig. 6.4.1 (e) Target axis in Example 3

(2) In the state of selecting the plane which includes the plane non-selection axis direction, the axis movement had been specified.

In this case, the axis which was specified by the plane selection is set to plane non-selection axis. However, when the axis movement by "List of modes that cannot execute check of the tool path between blocks" or "List of commands that cannot execute check of the tool path between blocks" is specified, the axis which was specified by the plane selection is not set to plane non-selection axis. And, the basic axis other than plane selection axes in the basic coordination system is set to plane non-selection axis when G code group of plane selection G code clear by reset etc. (Example 4)

Example 4)
When U-Y plane (G17) is selected after specifying the axis movement in W-X plane (G18) with the following axis configuration (Program example 1), plane selection axes are U axis and Y axis, and plane non-selection axis is W axis. (Fig. 6.4.1 (f))
No.1022
(X)=1: X axis of the basic three axes
(Y)=2: Y axis of the basic three axes
(Z)=3: Z axis of the basic three axes
(U)=5: Axis parallel to X axis
(W)=7: Axis parallel to Z axis
(A)=0: Rotary axis

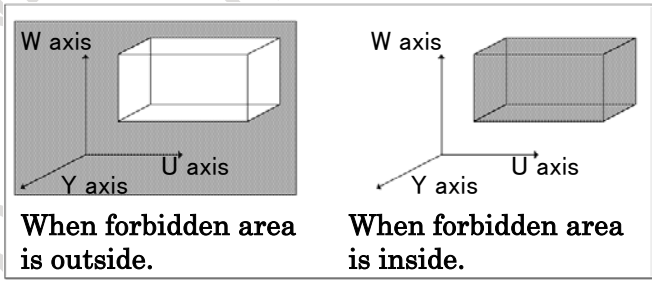


Fig. 6.4.1 (f) Target axis in Example 4 (U-Y-W)

FRYER MACHINE SYSTEMS
Robinsky

However, plane non-selection axis is Z axis in the following case. (Fig. 6.4.1 (g))

1. After selecting W-X plane (G18), U-Y plane (G17) is selected without specifying the axis movement (Program example 2).
2. After selecting W-X plane (G18), U-Y plane (G17) is selected after specifying only the axis movement by "List of modes that cannot execute check of the tool path between blocks" or "List of commands that cannot execute check of the tool path between blocks" (Program example 3).

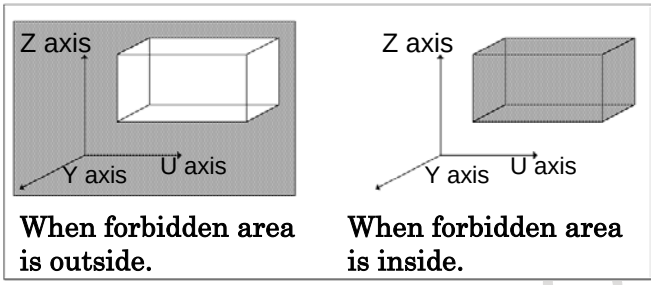


Fig. 6.4.1 (g) Target axis in Example 4 (U-Y-Z)

When G code group of plane selection G code clears by reset etc..

Afterwards, when G code group of plane selection G code clears by reset etc., plane selection axes are X axis and Y axis, and plane non-selection axis is Z axis. (Fig. 6.4.1 (h))

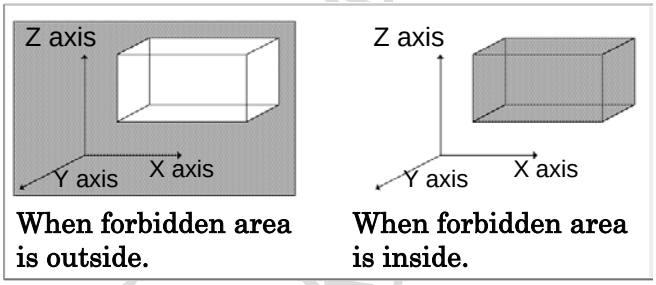


Fig. 6.4.1 (h) Target axis in Example 4 (Clear state)

```
<Program example 1>
O0001
G18W0X0
G01W10.0X10.0F1000.0 }  When axis movement of W axis is specified,
          :          }
G17U0Y0
          :          }  plane selection axes are U axis and Y axis, and plane
          :          }  non-selection axis is W axis.
```

```
<Program example 2>
O0001
G18W0X0
          :          }  When axis movement of W axis is not specified,
          :          }
G17U0Y0
          :          }  plane selection axes are U axis and Y axis, and plane
          :          }  non-selection axis is Z axis.
```

```
<Program example 3>
O0001
G18W0X0                  When axis movement of W axis is specified by G31 command
G31W100.0F100.0 }        in case of setting bit 2 (NPC) of parameter No.1301 to 1 or
G31X100.0F100.0 }        etc. ,
G17U0Y0
          :          }  plane selection axes are U axis and Y axis, and plane
          :          }  non-selection axis is Z axis.
```

NOTE

1 Check of the tool path between blocks is executed for forbidden area defined by stored stroke limit of plane selection axes and plane non-selection axis. Therefore, even if the axis other than plane selection axes and plane non-selection axis exceed the forbidden area defined by stored stroke limit in the tool path between blocks, the alarm is not generated by check of the tool path between blocks. (However, because stored stroke check of normal is effective, the alarm is generated when approaching the forbidden area defined by stored stroke limit during moving the axis.)

2 When the forbidden area is defined by inside of stored stroke limit, the alarm is generated if all axes to which stored stroke check is effective enter the forbidden area. However, in check of the tool path between blocks, the alarm is generated if plane selection axes and plane non-selection axis enters the forbidden area.

Limitation

Limitation of Stroke limit check before move

Limitation of Stroke limit check before move is effective for check of the tool path between blocks.

Non-linear type positioning

Check of the tool path between blocks is executed for the straight-line that connect the start position and end position of the block even if Positioning (G00), High-speed reference position return (G28, G30), or Selecting a machine coordinate system (G53) is move by non-linear type. Therefore, check of the tool path between blocks might not be able to be executed correctly, depending on the command.

Circular interpolation that is imperfect circle

When Circular interpolation that is imperfect circle is specified, check of the tool path between blocks is executed for the base circle. Therefore, check of the tool path between blocks might not be able to be executed correctly, depending on the shape of Circular interpolation.

Machine lock

For the axis which is Machine lock state when starting the block, check of the tool path between blocks is not executed.

Stored stroke check 2 off (G23)

When Stored stroke check 2 is invalid (G23 mode), check of the tool path between blocks for the forbidden area defined by Stored stroke limit 2 is not executed.

Arbitrary angular axis control

Stored stroke limit can be checked in Cartesian coordinate system by setting bit 0 (AOT), bit 1 (AO2) or bit 2 (AO3) of parameter No.8201 in Arbitrary angular axis control. By setting of checking stored stroke limit in Cartesian coordinate system, check of the tool path between blocks is effective. Check of the tool path between blocks by this stored stroke limit is not executed if angular axis or perpendicular axis to which stored stroke limit is checked by slanted coordinate system is set to plane selection axes or plane non-selection axis.

List of modes that cannot execute check of the tool path between blocks

Check of the tool path between blocks is not executed during the mode of following functions.

- Polar coordinate interpolation
- Cylindrical interpolation
- 3-dimensional coordinate conversion
- Skip function (However, the case of setting bit 2 (NPC) of parameter No.1301 to 0 is excluded.)

FRYER MACHINE SYSTEMS INC Rob Zabensky

- Automatic tool length measurement (M series) / automatic tool offset (T series) (However, the case of setting bit 2 (NPC) of parameter No.1301 to 0 is excluded.)

List of commands that cannot execute check of the tool path between blocks

Check of the tool path between blocks is not executed by the following commands.

- Movement command by Real time macro commands
- Movement command by PMC axis control
- Movement command for slave axis of Tandem control

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1302								SBA

[Input type] Parameter input
[Data type] Bit path

#0 SBA In Stroke limit check before move, check of the tool path between blocks for movement command is, :
0: Not executed.
1: Executed.

> **NOTE**
> 1 Check of the tool path between blocks is invalid for the functions to which Stroke limit check before move is invalid.
> 2 Limitation of Stroke limit check before move is valid for check of the tool path between blocks.

Alarm and message

Number	Message	Description
OT0510	+ OVERTRAVEL (PRE-CHECK)	The block end point or the tool path between blocks was found in the + side stroke limit forbidden area during Stroke limit check before move. Modify the program.
OT0511	- OVERTRAVEL (PRE-CHECK)	The block end point or the tool path between blocks was found in the - side stroke limit forbidden area during Stroke limit check before move. Modify the program.

FRYER MACHINE SYSTEMS INC
Rob Zaherecky

6.5 STROKE LIMIT AREA CHANGING FUNCTION

Overview

This function can be used to rewrite the parameters that set the + side coordinate value and the - side coordinate values of the stroke limits even when the axis is traveling. The new stroke limit range is enabled immediately after the parameters are rewritten. The rewritable parameters are shown below.

- Parameter No. 1320
  Coordinate value I of stored stroke check 1 in the positive direction on each axis
- Parameter No. 1321
  Coordinate value I of stored stroke check 1 in the negative direction on each axis
- Parameter No. 1322
  Coordinate value of stored stroke check 2 in the positive direction on each axis
- Parameter No. 1323
  Coordinate value of stored stroke check 2 in the negative direction on each axis
- Parameter No. 1324
  Coordinate value of stored stroke check 3 in the positive direction on each axis
- Parameter No. 1325
  Coordinate value of stored stroke check 3 in the negative direction on each axis
- Parameter No. 1326
  Coordinate value II of stored stroke check 1 in the positive direction on each axis
- Parameter No. 1327
  Coordinate value II of stored stroke check 1 in the negative direction on each axis

The machining cycle time can be reduced because this function can rewrite parameters even if some axes are moving.
This function is enabled by setting bit 0 (SLM) of parameter No.1312 to 1.
This function is optional.

Limitation

- Axis synchronous control

This function is disabled when the automatic slave axis parameter setting is enabled (bit 4 (SYP) of parameter No. 8303 = 1) in feed axis synchronous control. (Writing to the parameters is allowed only when all axes are stopped.)

Note

NOTE

If the parameters are rewritten so that the current position is included in a forbidden area during axis movement, the axis decelerates and stops, and an alarm is displayed. If an alarm occurs when the tool enters a forbidden area, the tool can move only opposite to the direction from which the tool came. After moving it outside the forbidden area in the opposite direction, release the alarm with a reset. When the alarm is released, the tool can move in both directions.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

6.6 STORED STROKE LIMIT RANGE SWITCHING FUNCTION BY SIGNAL

Overview

The range stored stroke limit can be switched by input signal of PMC. Therefore, the range stored stroke limit can easily be set again.

Note

NOTE

1 Stroke limit check before move also becomes effective in the switched area.
2 The range of stored stroke limit returns to the state set to the parameter at power on.
3 Stored stroke check 1 area expansion (coordinate value III - VIII) cannot be rewritten by this function.
4 When the range of stored stroke limit is switched by this function, even if the parameters (No.1320-No.1327) are rewritten by the G10 instruction, the PMC window, and the G22 instruction, it is not effective immediately.
5 Stroke limit area changing function is a function to rewrite the parameter (No.1320-No.1327). This function is a function that switches the range of stroke limit to the value that is set to data table (D) of PMC without rewriting the parameter (No.1320-No.1327).

6.7 WRONG OPERATION PREVENTION FUNCTIONS

An improper tool offset setting or an improper operation of the machine can result in the workpiece being cut inadequately or the tool being damaged. Also, if data is lost due to an operation mistake, it takes extra time to recover from the mistake.
The wrong operation prevention functions described below are meant to prevent the operator from performing any unintended operation (hereinafter referred to as an improper operation).

1 Functions that are used when data is set
- Data check to verify that the offset data is within the valid setting range
- Incremental input operation confirmation
- Prohibition of the absolute input by the soft key to prevent any improper absolute or incremental input operation
- Confirmation of any operation of deleting the program or all data
- Confirmation of a data update during the data setting process

2 Functions that are used when the program is executed
- Highlighting of updated modal information
- Display of the executed block status prior to the program execution
- Display of the axis status, such as the mirror image function enabled or the interlock function enabled
- Check for starting from the middle of the program
- Data check to verify that the offset data is within the effective setting range
- Maximum incremental value check

FRYER MACHINE SYSTEMS INC Rob Zahensky

6.7.1 Functions that are Used When Data is Set

The following functions are provided to prevent improper operations when data is set.
- Input data range check
- Confirmation of incremental input
- Prohibition of the absolute input by the soft key
- Confirmation of the deletion of the program
- Confirmation of the deletion of all data
- Confirmation of a data update during the data setting process

Set these functions on the operation confirmation function setting screen. For the input data range check, set a valid input data range, e.g. the upper and lower limits, for each input screen. For the other functions, specify whether to enable or disable them.
For information about how to display the individual setting screens, how to manipulate them, and other details, see the item “Operation confirmation setting screen” that describes the operation procedures.

6.7.1.1 Input data range check

This function allows an effective data range to be set and checks whether the input data is within the set range.

Input data range check

Explanation

- Outline of the input data range check

This function allows an effective data range to be set for the data of each input screen listed later and checks whether the input data is within the set range. If the input data is out of the effective data range, the warning message "DATA IS OUT OF RANGE" is displayed and the data is rejected.

For example, assume that the effective data range for a certain tool offset number is set to -200. to 200, and that you are going to input 100.[INPUT]. Even if you inadvertently press the 0 key one more time, resulting in 1000.[INPUT], the input of 1000. is not accepted.
The function detects a setting mistake and prevents the program from running with invalid data.

- Input screens for which this function is effective

- Tool compensation
- Workpiece origin offset

T

- Y-axis tool offset
- Workpiece shift

- Settings

To enable this function, set an effective data range for each input screen on the operation confirmation function setting screen. For information about how to display the individual setting screens, how to set data ranges, and other details, see the items that describe the setting of the data ranges.
If the set data range is invalid, no data input is accepted. Correct the data range setting, and then input data.

FRYER MACHINE SYSTEMS INC
Rob Zahansky

- Disabling the function

The input data range check is disabled if you make any of the following settings on the operation confirmation function setting screen.

- Both the upper and lower limit values for the tool offset number or workpiece coordinate system are 0.
- The upper and lower limit values for each offset are identical.

- Messages displayed during the input data range check

When the cursor moves into an input field of an input screen, one of the messages and warning messages in the Table 6.7.1.1 (a) or Table 6.7.1.1 (b) is displayed. No message is displayed when the input data range check is disabled.

When the set effective data range is valid

Table 6.7.1.1 (a) List of messages displayed 1

Input data status	Message	Color
The data in the input field is within the range.	Input range xxx - xxx	Black
The data in the input field is out of the range.	Input range xxx - xxx	Red

xxx: Upper and lower limit values

When the set effective data range is invalid

Table 6.7.1.1 (b) List of messages displayed 2

Range check status	Message	Color
Tool offset number overlap	NG SETTING (OFFSET NUM OVERLAP)	Red
Workpiece coordinate system overlap	NG SETTING (WORK COORD VAL OVERLAP)	Red
Invalid upper and lower limit values	NG SETTING (U-LMT AND L-LMT ILLEGAL)	Red

The message "NG SETTING (U-LMT AND L-LMT ILLEGAL)" is displayed in the following cases:

- The upper and lower limit values are reversed.
- The values are not effective (e.g., more pairs of offset numbers than allowed are set).
- Either of the tool offset numbers is 0.

- Range check for data changed by G10 or system variable

If the data changed by G10 or system variable is out of the effective data range, the alarm PS0334 "OFFSET DATA OUT OF RANGE" is displayed.

6.7.1.2 Confirmation of incremental input

This function displays a confirmation message when you attempt to input an incremental value by using the [+INPUT] soft key.

Confirmation of incremental input

Explanation

- Outline of the confirmation of incremental input

This function displays a confirmation message when you attempt to input an incremental value by using the [+INPUT] soft key in any of the input screens listed below. It lets you confirm whether you really want to change data or not before making that change.

For example, when you set 5.[+INPUT] for 10., the message "15. INPUT OK?" is displayed.
The function prevents improper absolute or incremental input operations.

> **NOTE**
> This function cannot be used to input two or more values consecutively by delimiting them by semicolons (;).

FRYER MACHINE SYSTEMS INC
Rob Zanenkov

- **Input screens for which this function is effective**
- Tool compensation
- Workpiece origin offset
- Settings
- Parameter
- Pitch error compensation

T

- Workpiece shift
- Y-axis tool offset
- Second tool geometry offset
- Chuck tail stock barrier
- Tool geometry data

- **Settings**

In the operation confirmation function setting screen, check or uncheck the "INCREMENTAL INPUT" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.7.1.3 Prohibition of the absolute input by the soft key

This function prohibits the absolute input using the [INPUT] soft key.

Prohibition of the absolute input by the soft key

Explanation

- **Outline of the prohibition of the absolute input by the soft key**

This function prohibits the absolute input by the [INPUT] soft key in the input screens listed later.
It prevents improper absolute or incremental input operations by requiring that the absolute input be made using the MDI key and that the incremental input be made using the [+INPUT] soft key.

- **Input screens for which this function is effective**
- Tool compensation
- Workpiece origin offset

T

- Y-axis tool offset
- Workpiece shift

- **Settings**

In the operation confirmation function setting screen, check or uncheck the "DISABLED SOFTKEY[INPUT] IN" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.7.1.4 Confirmation of the deletion of the program

This function displays the confirmation message "DELETE PROGRAM ?" when you attempt to delete the program.

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

Confirmation of the deletion of the program

Explanation

- Outline of the confirmation of the deletion of the program

When you attempt to delete the program, this function displays the confirmation message "DELETE PROGRAM ?" It lets you confirm whether you really want to delete the program or not before executing the deletion.
The function prevents the program from being deleted due to an improper operation.

- Settings

In the operation confirmation function setting screen, check or uncheck the "PROGRAM DELETE" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.7.1.5 Confirmation of the deletion of all data

This function displays the confirmation message "DELETE ALL DATA?" when you attempt to delete all data.

Confirmation of the deletion of all data

Explanation

- Outline of the confirmation of the deletion of all data

When you attempt to delete all data on the input screen described later, this function displays the confirmation message "DELETE ALL DATA?". It lets you confirm whether you really want to delete all data or not before executing the deletion.
The function prevents all data from being deleted due to an improper operation.

- Input screens for which this function is effective

- Tool compensation

T

- Y-axis tool offset

- Settings

In the operation confirmation function setting screen, check or uncheck the "ALL DATA DELETE" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.7.1.6 Confirmation of a data update during the data setting process

This function displays the [CAN] and [EXEC] soft keys for confirmation when you attempt to update the data of an input screen during the data setting process.

Confirmation of a data update during the data setting process

Explanation

- Outline of the confirmation of a data update during the data setting process

When you input data in input screen during the data setting process, this function displays the [CAN] and [EXEC] soft keys for confirmation. It lets you confirm whether you really want to update the data or not before executing the update.

TRYER MACHINE SYSTEMS INC
Rob Zaherlsky

The function prevents set values from being lost due to an improper operation.
If you input data using the [+INPUT] soft key when the confirmation of incremental input is enabled, a message is displayed to confirm the incremental input.

- Settings

In the operation confirmation function setting screen, check or uncheck the "INPUT IN SETTING" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.7.2 Functions that are Used when the Program is Executed

Overview

The following functions are provided to prevent improper operations when the program is executed.

- Display of updated modal information
- Start check signal
- Axis status display
- Confirmation of the start from a middle block
- Data range check
- Maximum incremental value check
- Warning indication during a reset in program operation

In the operation confirmation function setting screen, specify whether to enable or disable these functions individually.
To disable or enable "Warning indication during a reset in program operation", set bit 1 (WMD) of parameter No. 10334 instead of the wrong operation prevention function setting screen.
For information about how to display the setting screen, how to manipulate it, and other details, see the item “Operation confirmation setting screen” that describes the operation procedures.

6.7.2.1 Display of updated modal information

This function allows modal information updated by the NC command or RESET to be highlighted in the modal information display for the current block.

Display of updated modal information

Explanation

- Outline of the display of updated modal information

This function allows modal information updated by the NC command or RESET to be highlighted in the modal information display for the current block.
For example, when an absolute command has been changed to an incremental command or when the workpiece coordinate system has been initialized by RESET, the function displays the changed part of the data in an easy-to-recognize manner, in order to prevent improper operations during the execution of the program.

- Settings

In the operation confirmation function setting screen, check or uncheck the "UPDATE MODAL HIGHLIGHT DISPLAY" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

FRYER MACHINE SYSTEMS INC
Rob Zalensky

6.7.2.2 Start check signal

This function displays the remaining amount of travel and modal information of the block to be executed and puts the program to a temporary halt before the program is executed.

Start check signal

Explanation

- Outline of the start check signal

When a cycle start is made with the start check signal STCHK <G0408.0> set to 1, the function displays the remaining amount of travel and modal information of the block to be executed and puts the program to a temporary halt. Making the cycle start again resumes the execution of the program.
The function lets you check the status of the block before executing it, thus helping to prevent improper operations at the time of execution.
Using this function in combination with the updated modal information display function described in the preceding subsection makes it easier to check the status of the block to be executed.

- Settings

This function does not require any setting on the operation confirmation function setting screen.

6.7.2.3 Axis status display

This function displays the axis status to the left of the axis name in the coordinate display screen.

Axis status display

Explanation

- Outline of the axis status display

This function displays the axis status to the left of the axis name in the display of the machine coordinates, absolute coordinates, relative coordinates, and remaining travel amounts.
For example, when the mirror image function is enabled for the X1 axis, the absolute coordinates are displayed as follows.

```
ABSOLUTE
M  X1    10.000
   Y1    10.000
   Z1     0.000
```

By displaying the axis status as shown above, the function prevents improper operations at the time of execution.

- Axis status indication

The axis status is indicated as follows. These indications are listed in order of priority.

AXIS DETACH	: D
FLEXIBLE PATH AXIS ASSIGNMENT	: R
INTERLOCK	: I
MACHINE LOCK	: L
SERVO OFF	: S
TORQUE CONTROL	: T
AXIS MOVING	: *
MIRROR IMAGE	: M

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- **Settings**

In the operation confirmation function setting screen, check or uncheck the "AXIS STATUS DISPLAY" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.7.2.4 Confirmation of the start from a middle block

This function displays a confirmation message when you attempt to execute a memory operation with the cursor placed on a block in the middle of the program.

Confirmation of the start from a middle block

Explanation

- **Outline of the confirmation of the start from a middle block**

This function displays the confirmation message "START FROM MIDDLE OF PROG (START/RESET)" when you attempt to execute a memory operation with the cursor placed on a block in the middle of the program. It lets you confirm whether you really want to start execution from that block or not before executing the program.
The function prevents you from inadvertently making a cycle start from a block in the middle of the program.

- **Clearing confirmation message**

When the confirmation message is displayed and the cycle start is performed again, the confirmation message disappears and the automatic operation is restarted. And, when the confirmation message is displayed and the reset is performed, the confirmation message disappears and the reset state is entered.

- **Settings**

In the operation confirmation function setting screen, check or uncheck the "START FROM MIDDLE OF PROGRAM" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

- **Multi-path system**

The operation of the reconfirmation of the start from a middle block in a multi-path system is determined with the setting of bit 0 (MSC) of parameter No. 10335.
For example, if a cycle start is executed on two paths simultaneously in the program below, the operation is as given in Table 6.7.2.4.

Program for path 1

```
O1000 ;
G90 G28 X100.Y100. ;
G91 G00 X50. ;
  :
  :
  :
  :
M02
```

Cursor

Program for path 2

```
O1000 ;
G90 G28 X100.Y100. ;
G91 G00 X50. ;
  :
  :
  :
  :
M02
```

Table 6.7.2.4 Operation in a multi-path system

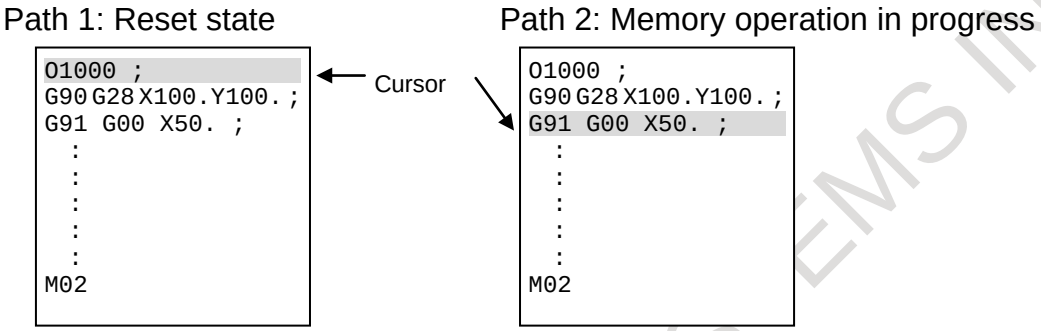
Bit 0 (MSC) of parameter No. 10335		Memory operation	Confirmation message
Path 1	Path 2		
0	0	Starts on path 1 only.	Displayed for both path.
0	1		
1	0		
1	1	Does not start on either path.	Displayed for both path.

FEDER MACHINE SYSTEMS INC Rob Zahensky

To reconfirm the start from a middle block between paths, set bit 0 (MSC) of parameter No. 10335 to 1 for each path.

NOTE

If bit 0 (MSC) of parameter No. 10335 for each path is 1, a cursor position check is not performed on the path on which memory operation is in progress. For example, in the case below, if a cycle start is executed on path 1, a cursor position check is not performed on path 2.



6.7.2.5 Data range check

This function lets you set an effective data range and check whether the data to be used for execution is within the set range.

Data range check

Explanation

- Outline of the data range check

This function lets you set an effective data range for each data item listed later and check whether the data to be used for execution is within the set range. If the data is out of the effective range, the alarm PS0334 "OFFSET DATA OUT OF EFFECTIVE RANGE" is displayed.
The function detects data setting mistakes and prevents the program from running with invalid data.

- Data for which this function is effective

- Tool compensation
- Workpiece origin offset

T

- Y-axis tool offset
- Workpiece shift

NOTE

To use this function, you need to set each effective data range correctly. For information about how to set the data ranges, see the item “Effective value range for each data”.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

6.7.2.6 Maximum incremental value check

This function checks the maximum incremental value specified for each axis by the NC command.

Maximum incremental value check

Explanation

- Outline of the maximum incremental value check

When the maximum incremental value is specified by the NC command described later, make sure that the absolute value of the travel distance by the incremental command does not exceed the specified value. If the specified value is exceeded, the alarm PS0337 "EXCESS MAXIMUM INCREMENTAL VALUE" is displayed.
A maximum incremental value can be specified on a per-axis basis and remains effective until 0 is set or the value is reset.
For example, when AI contour control is used, the function checks whether the amount of movement between blocks is kept to the specified value or less. Through this process, it detects erroneous program settings and prevents the program from running with invalid data.

- Format

The format of the NC command used to specify the maximum incremental value is as follows.

G91.1 IP_ ;
IP_ ; Maximum incremental value
(To cancel the maximum incremental value check, set 0.)

6.7.2.7 Warning display during a reset in program operation

When bit 6 (CLR) of parameter No. 3402 is 0, if a reset occurs during block execution in program operation, modal information returns to the state before block execution.
This function display a warning to notify the operator that modal information is not updated by information of the interrupted block.

Warning display during a reset in program operation

Explanation

- Overview of warning display during a reset in program operation

If a reset occurs during program operation, the warning "MODAL DATA IS CHANGED BY BLOCK STOP" is issued. To enable or disable the warning, set bit 1 (WMD) of parameter No. 10334.

- Warning occurrence condition

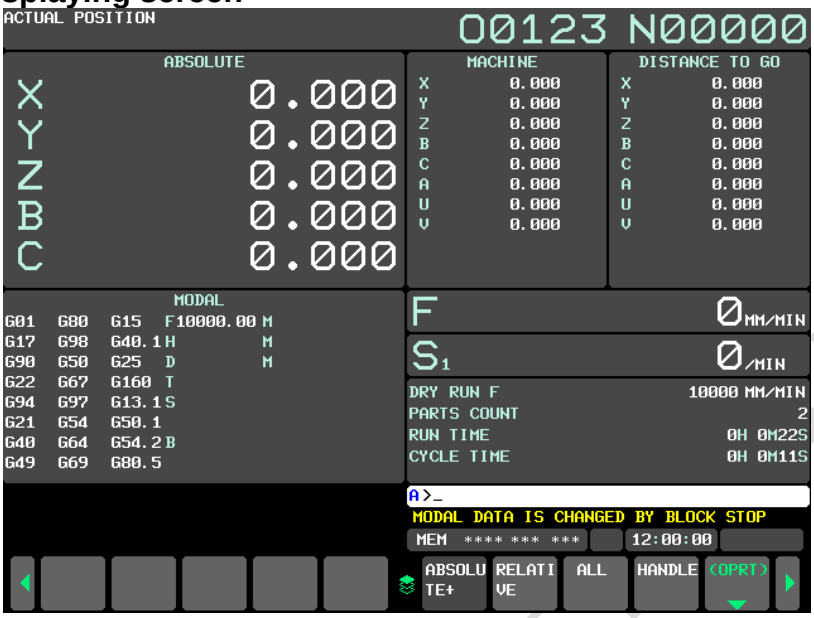
- When a reset occurs during program operation, if address G, F, H, D, T, S, M, or B (second auxiliary function) was changed.

- Warning clear condition

- When a reset is issued
- When either of MDI or Soft keys is pushed

FDYER MACHINE SYSTEMS INC
Rob Zahensky

- **Warning displaying screen**



6.7.3 Setting Screen

Overview

This section describes how to display the operation confirmation function setting screen and how to set the individual data items on this screen.

The operation confirmation function setting screen allows you to set the following items:

- Enabling or disabling each operation confirmation function
- Effective value range for the tool offset
- Effective value range for the workpiece origin offset

T

- Effective value range for the Y-axis tool offset
- Effective value range for the work shift

6.7.3.1 Operation confirmation function setting screen

This screen displays the enable/disable setting status of the following operation confirmation functions and lets you change their settings. (Hereinafter, the screen is referred to as the operation confirmation function setting screen.)

- Confirmation of incremental input
- Prohibition of the absolute input by the soft key
- Confirmation of the deletion of the program
- Confirmation of the deletion of all data
- Confirmation of a data update during the data setting process
- Display of updated modal information
- Axis status display
- Confirmation of the start from a middle block

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Displaying and setting the operation confirmation function setting screen

Procedure

1 Press the function key.

2 Press the continuous menu key at the right edge of the screen several times until the [GUARD] soft key is displayed.

3 Press the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).

4 If any screen other than the operation confirmation function setting screen is displayed, press the [GUARD] soft key. The operation confirmation function setting screen is displayed.

Fig. 6.7.3.1 Operation confirmation function setting screen

5 Move the cursor to the check box of the item you want to change the setting, by pressing the ↑, ↓, ←, and → keys.

6 Press the soft key [(OPRT)].

7 Press the soft key [ON:1] or [OFF:0]. When you press the [ON:1] soft key, a check mark (✓) appears in the corresponding check box, indicating that the function is enabled. When you press the [OFF:0] soft key, the check mark disappears from the check box, indicating that the function is disabled.

Explanation

- Items to be set

The Table 6.7.3.1 shows what is displayed for each item to be set and the corresponding functions. The items whose column of "Default" is marked by "○" is effective automatically after memory all clear.

Table 6.7.3.1

Displayed item	Default	Corresponding function
INCREMENTAL INPUT	○	Confirmation of incremental input
DISABLED SOFTKEY[INPUT] IN TOOL OFFSET, WORK SHIFT		Prohibition of the absolute input by the soft key (tool offset, Y-axis tool offset (lathe system), and work shift (lathe system))

Displayed item	Default	Corresponding function
DISABLED SOFTKEY[INPUT] IN WORK COORDINATES		Prohibition of the absolute input by the soft key (workpiece origin offset)
PROGRAM DELETE	○	Confirmation of the deletion of the program
ALL DATA DELETE	○	Confirmation of the deletion of all data
INPUT IN SETTING		Confirmation of a data update during the data setting process
UPDATE MODAL HIGHLIGHT DISPLAY	○	Display of updated modal information
AXIS STATUS DISPLAY	○	Axis status display
START FROM MIDDLE OF PROGRAM	○	Confirmation of the start from a middle block

6.7.3.2 Tool offset range setting screen

This screen displays the setting status of tool offset effective data ranges and lets you change their settings. (Hereinafter, the screen is referred to as the tool offset range setting screen.)
Up to 20 pairs of numbers can be specified to identify tool offset number ranges, and an effective offset value range can be defined for each of these 20 pairs.

Displaying and setting the tool offset range setting screen

Procedure

1 Press the function key.

2 Press the continuous menu key ▷ at the right edge of the screen several times until the [GUARD] soft key is displayed.

3 Press the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).

4 If any screen other than the tool offset range setting screen is displayed, press the [OFFSET] soft key. The tool offset range setting screen is displayed. What is displayed in this screen differs depending on the system configuration described later.

Fig. 6.7.3.2 Tool offset range setting screen

5 Press the soft key [(OPRT)].

FRYER MACHINE SYSTEMS INC
by Zarensky

6 Move the cursor to the item you want to set, by using the [PAGE ⇧] and [PAGE ⇩] keys, [⬆], [⬇], [⬅], and [➡] keys, or the [SWITCH] soft key.

7 Press the MDI key, enter necessary data, and then press the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- There is a tool offset number overlap.
- The upper and lower limit values are reversed.
- The values are not effective (e.g., more pairs of offset numbers than allowed are set).
- Either of the tool offset numbers is 0.

Also, the input data range check is invalidated in the following cases.

- Both the upper and lower limit values for the tool offset number are 0.
- The upper and lower offset limit values are identical.

Explanation

- System configuration

What to set differs for each of the following system configurations:

M

- Tool offset memory A
- Tool offset memory C

T

- Without geometry and wear offset
- With geometry and wear offset
- Tool offset for milling and turning function

M

- What to set with tool offset memory A

With tool offset memory A, an effective data range is specified using the four items in the Table 6.7.3.2 (a).

Table 6.7.3.2 (a)

Displayed item		What to set
RANGE	FROM	Specify a tool offset number range.
	TO	
-	LOW-LIMIT	Specify a valid tool offset value range in connection with a specified tool offset number range.
	UP-LIMIT	

- What to set with tool offset memory C

With tool offset memory C, an effective data range is specified using the ten items in the Table 6.7.3.2 (b).

Table 6.7.3.2 (b)

Displayed item			What to set
RANGE		FROM	Specify a tool offset number range.
		TO	
GEOM	LENGTH	LOW-LIMIT	Specify a valid tool offset value range for geometry length in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for geometry radius in connection with a specified tool offset number range.
		UP-LIMIT	

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Displayed item			What to set
WEAR	LENGTH	LOW-LIMIT	Specify a valid tool offset value range for wear length in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for wear radius in connection with a specified tool offset number range.
		UP-LIMIT	

In the case of this configuration, all the information needed to set an input data range cannot be displayed in a single screen page. Set the information while switching pages using the [SWITCH] soft key. The screen provides an indication that lets you know which part of the information is currently displayed.

T

- What to set without geometry/wear offset

Without geometry/wear offset, an effective data range is specified using the eight items in the Table 6.7.3.2 (c).

Table 6.7.3.2 (c)

Displayed item		What to set
RANGE	FROM	Specify a tool offset number range.
	TO	
X	LOW-LIMIT	Specify a valid tool offset value range for the X-axis in connection with a specified tool offset number range.
	UP-LIMIT	
Z	LOW-LIMIT	Specify a valid tool offset value range for the Z-axis in connection with a specified tool offset number range.
	UP-LIMIT	
RADIUS	LOW-LIMIT	Specify a valid tool offset value range for tool-nose radius in connection with a specified tool offset number range.
	UP-LIMIT	

NOTE
The radius-related items are not displayed if the cutter or tool nose radius compensation is not used.

- What to set with geometry/wear offset

With geometry/wear offset, an effective data range is specified using the 14 items in the Table 6.7.3.2 (d).

Table 6.7.3.2 (d)

Displayed item			What to set
RANGE		FROM	Specify a tool offset number range.
		TO	
GEOM	X	LOW-LIMIT	Specify a valid tool offset value range for the geometry X-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	Z	LOW-LIMIT	Specify a valid tool offset value range for the geometry Z-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for geometry tool-nose radius in connection with a specified tool offset number range.
		UP-LIMIT	
WEAR	X	LOW-LIMIT	Specify a valid tool offset value range for the wear X-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	Z	LOW-LIMIT	Specify a valid tool offset value range for the wear Z-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for wear tool-nose radius in connection with a specified tool offset number range.
		UP-LIMIT	

In the case of this system, all the information needed to set an input data range cannot be displayed in a single screen page. Set the information while switching pages using the [SWITCH] soft key. The screen provides an indication that lets you know which part of the information is currently displayed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE
The radius-related items are not displayed if the tool nose radius compensation option is not displayed.

- What to set with Tool offset for milling and turning function

With tool offset for milling and turning function, an effective data range is specified using the 14 items in the Table6.7.3.2 (e).

Table6.7.3.2 (e)

Displayed item			What to set
RANGE		FROM	Specify a tool offset number range.
		TO	
GEOM	X	LOW-LIMIT	Specify a valid tool offset value range for the geometry X-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	Z/LENGTH	LOW-LIMIT	Specify a valid tool offset value range for the geometry Z-axis or geometry length in connection with a specified tool offset number range.
		UP-LIMIT	
	NOSE R/ RAD/ CORNER R	LOW-LIMIT	Specify a valid tool offset value range for geometry tool-nose radius or geometry corner R in connection with a specified tool offset number range.
		UP-LIMIT	
WEAR	X	LOW-LIMIT	Specify a valid tool offset value range for the wear X-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	Z/LENGTH	LOW-LIMIT	Specify a valid tool offset value range for the wear Z-axis or wear length in connection with a specified tool offset number range.
		UP-LIMIT	
	NOSE R/ RAD/ CORNER R	LOW-LIMIT	Specify a valid tool offset value range for wear tool-nose radius or wear corner R in connection with a specified tool offset number range.
		UP-LIMIT	

In the case of this system, all the information needed to set an input data range cannot be displayed in a single screen page. Set the information while switching pages using the [SWITCH] soft key. The screen provides an indication that lets you know which part of the information is currently displayed.

NOTE
When the setting of cutter or tool nose radius compensation is valid (bit 7 (NCR) of parameter No. 8136 is set to 0), "NOSE R/RAD" items is displayed.

- Example of setting an input data range

For example, suppose that the following values are set with offset memory A.

FROM : TO LOW-LIMIT : UP-LIMIT
1 : 20 0.000 : 100.000

In this case, the tool offset input screen accepts only offset values from 0.000 to 100.000 for offset numbers 1 to 20.
If you attempt to input any other value, the warning message "DATA IS OUT OF RANGE" is displayed.

6.7.3.3 Workpiece origin offset range setting screen

This screen displays the setting status of workpiece origin offset and external workpiece origin offset effective data ranges and lets you change their settings. (Hereinafter, the screen is referred to as the workpiece origin offset range setting screen.)

Up to six pairs of values can be specified to identify workpiece coordinate ranges for the workpiece origin offset, and an effective offset value range can be defined for each of the axes of these six pairs.
As for the external workpiece origin offset, an effective offset value range can be specified for each axis.

FRYER MACHINE SYSTEMS INC
Rob Zaranosky

Displaying and setting the workpiece origin offset range setting screen

Procedure

1 Press the [OFS/SET] function key.

2 Press the continuous menu key [▷] at the right edge of the screen several times until the [GUARD] soft key is displayed.

3 Press the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).

4 If any screen other than the workpiece origin offset range setting screen is displayed, press the [WORK] soft key. The workpiece origin offset range setting screen is displayed.

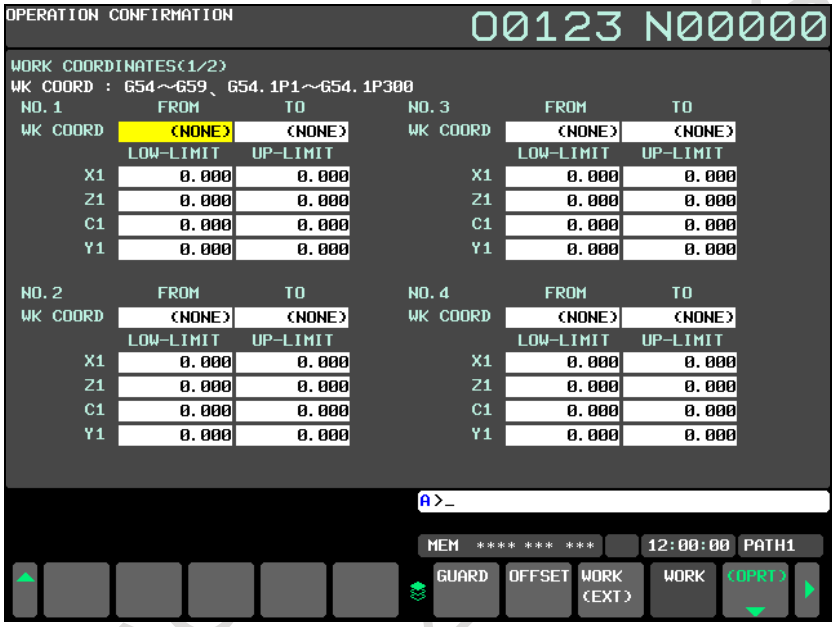


Fig. 6.7.3.3 Workpiece origin offset range setting screen

5 Press the soft key [(OPRT)].

6 Move the cursor to the item you want to set, by using the [PAGE ↑] and [PAGE ↓] keys, or [↑], [↓], [←], and [→] keys.

7 Press the MDI key, enter necessary data, and then press the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- There is a workpiece coordinate overlap.
- The upper and lower limit values are reversed.
- The values are not effective (e.g., an invalid workpiece coordinate system is set).
- The upper limit value is set for the workpiece coordinate system when 0 is set for the lower limit value.

Also, the input data range check is invalidated in the following cases.

- Both the upper and lower limit values for the workpiece coordinate system are 0.
- The upper and lower limit values for each offset are identical.

FRYER MACHINES INC. Rob Z

Explanation

- What to set for the workpiece origin offset

For the workpiece origin offset, an effective data range is specified using the four items in the Table 6.7.3.3 (a).

Table 6.7.3.3 (a)

Displayed item		What to set
RANGE	FROM	Specify a workpiece coordinate system range.
	TO	
AXIS NAME	LOW-LIMIT	Specify a valid offset value range in connection with a specified workpiece coordinate system range.
	UP-LIMIT	

- What to set for the external workpiece origin offset

For the external workpiece origin offset, an effective data range is specified using the two items in the Table 6.7.3.3 (b).

Table 6.7.3.3 (b)

Displayed item		What to set
AXIS NAME	LOW-LIMIT	Specify a valid external workpiece origin offset value range on each axis.
	UP-LIMIT	

6.7.3.4 Y-axis tool offset range setting screen

T

In the case of a lathe system or Tool offset for milling and turning function, this screen displays the setting status of Y-axis tool offset effective data ranges and lets you change their settings. (Hereinafter, the screen is referred to as the Y-axis tool offset range setting screen.)
Up to four pairs of values can be specified to identify Y-axis tool offset number ranges, and an effective offset value range can be defined for each of these four pairs.

Displaying and setting the Y-axis tool offset range setting screen

Procedure

1 Press the function key.

2 Press the continuous menu key at the right edge of the screen several times until the [GUARD] soft key is displayed.

3 Press the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).

4 If any screen other than the Y-axis tool offset range setting screen is displayed, press the [Y OFFSET] soft key. The Y-axis tool offset range setting screen is displayed. What is displayed in this screen differs depending on such factors as whether tool geometry/wear offsets are present.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Fig. 6.7.3.4 Y-axis tool offset range setting screen

5 Press the soft key [(OPRT)].

6 Move the cursor to the item you want to set, by using the [PAGE ↑] and [PAGE ↓] keys, [↑], [↓], [←], and [→] keys, or the [SWITCH] soft key.

7 Press the MDI key, enter necessary data, and then press the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- There is a tool offset number overlap.
- The upper and lower limit values are reversed.
- The values are not effective (e.g., more pairs of offset numbers than allowed are set).
- Either of the tool offset numbers is 0.

Also, the input data range check is invalidated in the following cases.

- Both the upper and lower limit values for the tool offset number are 0.
- The upper and lower offset limit values are identical.

Explanation

- What to set without geometry/wear offsets

Without geometry/wear offsets, an effective data range is specified using the four items in the Table 6.7.3.4 (a).

Table 6.7.3.4 (a)

Displayed item		What to set
RANGE	FROM	Specify a Y-axis tool offset number range.
	TO	
-	LOW-LIMIT	Specify a valid tool offset value range in connection with a specified Y-axis tool offset number range.
	UP-LIMIT	

- What to set with geometry/wear offsets

With geometry/wear offsets, an effective data range is specified using the six items in the Table 6.7.3.4 (b).

FRYER MACHINE SYSTEMS INC
Rob Zahenskyj

Table 6.7.3.4 (b)

Displayed item		What to set
RANGE	FROM	Specify a Y-axis tool offset number range.
	TO	
GEOM	LOW-LIMIT	Specify a valid tool offset value range for geometry in connection with a specified Y-axis tool offset number range.
	UP-LIMIT	
WEAR	LOW-LIMIT	Specify a valid tool offset value range for wear in connection with a specified Y-axis tool offset number range.
	UP-LIMIT	

6.7.3.5 Workpiece shift range setting screen

T

In the case of a lathe system, this screen displays the setting status of shift effective data ranges of workpiece shift coordinate systems and lets you change their settings. (Hereinafter, the screen is referred to as the workpiece shift range setting screen.)
An offset value range can be specified for each axis.

Displaying and setting workpiece shift input ranges

Procedure

1 Press the function key .

2 Press the continuous menu key at the right edge of the screen several times until the soft key [GUARD] is displayed.

3 Press the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).

4 If any screen other than the workpiece shift range setting screen is displayed, press the [WORK SHIFT] soft key. The workpiece shift range setting screen is displayed.

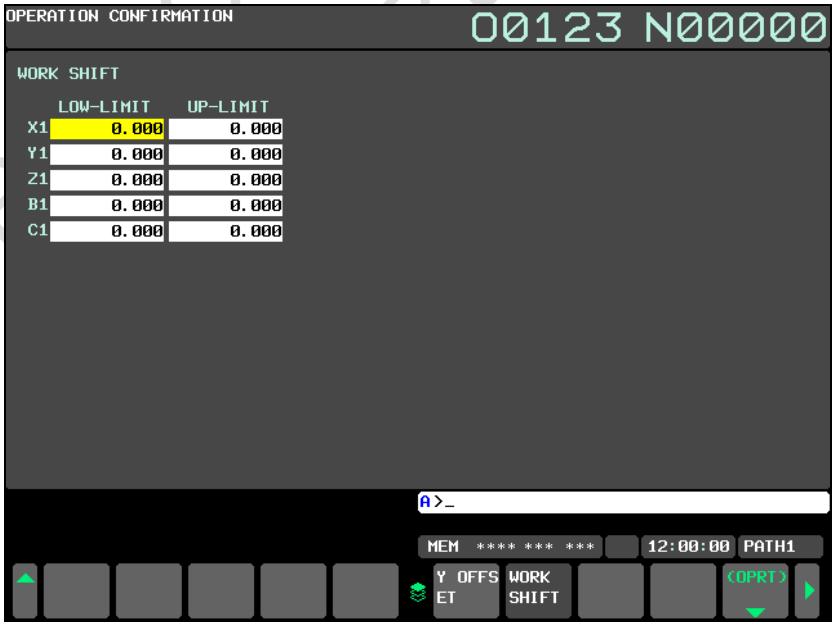


Fig. 6.7.3.5 Workpiece shift range setting screen

5 Press the soft key [(OPRT)].

FRYER MACHINE SYSTEMS INC
Zaharsky

6 Move the cursor to the item you want to set, by using the [PAGE ↑] and [PAGE ↓] keys, or [↑], [↓], [←], and [→] keys.

7 Press the MDI key, enter necessary data, and then press the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- The upper and lower limit values are reversed.

Also, the input data range check is invalidated in the following cases.

- The upper and lower offset limit values are identical.

Explanation

- What to set for the workpiece shift

For the workpiece shift, an effective data range is specified using the two items in the Table 6.7.3.5 .

Table 6.7.3.5

Displayed item		What to set
AXIS NAME	LOW-LIMIT	Specify a valid workpiece shift coordinate system shift value range on each axis.
	UP-LIMIT	

FRYER MACHINE SYSTEMS INC
Rob Zahensky

7 ALARM AND SELF-DIAGNOSIS FUNCTIONS

When an alarm occurs, the corresponding alarm screen appears to indicate the cause of the alarm. The causes of alarms are classified by error codes and number. Up to 60 previous alarms can be stored and displayed on the screen (alarm history display).
The system may sometimes seem to be at a halt, although no alarm is displayed. In this case, the system may be performing some processing. The state of the system can be checked using the self-diagnosis function.

7.1 ALARM DISPLAY

Explanation

- Alarm screen

When an alarm is issued, the display changes to the alarm screen. Two alarm screens "DETAIL" and "ALL PATH" are provided. You can choose one of the screens by pressing the corresponding soft key.

- Detail screen
  Alarm information for the currently selected path is displayed.

Fig. 7.1 (a) Alarm detail screen

FRYER SYSTEMS INC

- All path screen
  Alarm information for all paths is displayed sequentially from path 1.

Fig. 7.1 (b) All path screen

- Displaying an alarm screen

ALM is sometimes indicated in the bottom part of the screen display without displaying an alarm screen.

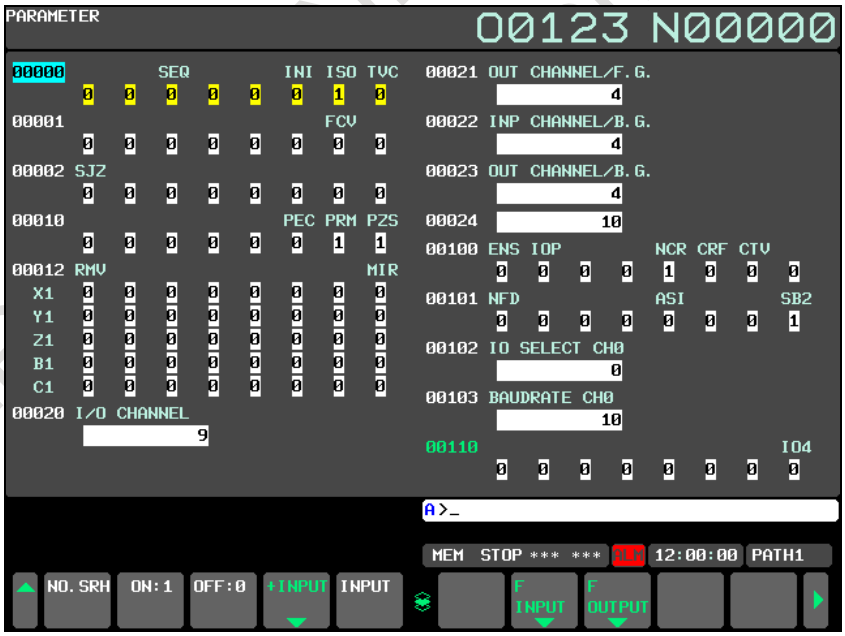


Fig. 7.1 (c) Parameter screen

FRYER ... INC

In this case, display the alarm screen by following the steps below.

1 Press the function key .
2 Press the soft key [ALARM].
3 Pressing the [ALARM] soft key changes the screen display to the "DETAIL" screen (or the alarm screen selected previously), and the [DETAIL] and [ALL PATH] soft keys appears.
   - Pressing the [DETAIL] soft key displays the "DETAIL" screen.
   - Pressing the [ALL PATH] soft key displays the "ALL PATH" screen.

   If the number of paths is 1, pressing the [ALARM] soft key displays the "DETAIL" screen, but the [ALARM] soft key indication remains unchanged.
4 You can change pages by using the page key.

- Releasing alarm

The cause of an alarm can be determined from the error code, number, and associated message. To release the alarm, generally correct the cause, then press the reset key.

- Error code and number

The type of an alarm is indicated by an error code and number.
Example: PS0010, SV0004, etc.
For details, see Appendix G, "ALARMS".

7.2 ALARM HISTORY DISPLAY

The CNC including the latest alarm are stored and displayed on the screen.
The display procedure is explained below.

NOTE

Alarms, operations, external operator messages, data change history data, etc. are stored in the same storage area. If the storage capacity is exceeded due to other history data, alarm history data may be erased. If, however, bit 2 (SAH) of parameter No. 11354 is 1, up to 50 alarms can be recorded regardless of the operation history state.
The maximum number of history data items that can be stored is as follows, where the history data is assumed to include alarm data only:
If bit 7 (HAL) of parameter No. 3196 is 1, about 4000 items.
If bit 7 (HAL) of parameter No. 3196 is 0, modal information and coordinates observed when an alarm is issued are also recorded, so that the maximum number varies from system to system.
(Example: For a 5-axis system, about 280 data items; for a 10-axis system, about 200 data items)
If bit 3 (EAH) of parameter No. 3112 is 1, however, macro and external alarm messages are also stored, so that the maximum number further decreases.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Alarm history display

Procedure

1 Press the function key .

2 Press the soft key [HISTRY].
An alarm history is displayed.
The following information is displayed:
<1> Date and time of alarm issuance
<2> Alarm type
<3> Alarm number
<4> Alarm message (sometimes not displayed depending on the alarm)
<5> Page No.

3 You can change pages by using the page key.

Fig. 7.2 Alarm history screen

FRYER MA... ...MS INC
Ro...

7.3 CHECKING BY DIAGNOSTIC DISPLAY

The system may sometimes seem to be at a halt, although no alarm has occurred. In this case, the system may be performing some processing. Diagnostic display can be used to check the system status.

Procedure for Diagnostic display

Procedure

1 Press the function key SYSTEM.

2 Press the soft key [DIAGNOSIS].

3 The diagnosis screen has more than 1 pages. Select the screen by the following operation.
   (1) Change the page by the page change key.
   (2) Method by soft key
      - Key input the number of the diagnosis data to be displayed.
      - Press the soft key [NO.SRH].

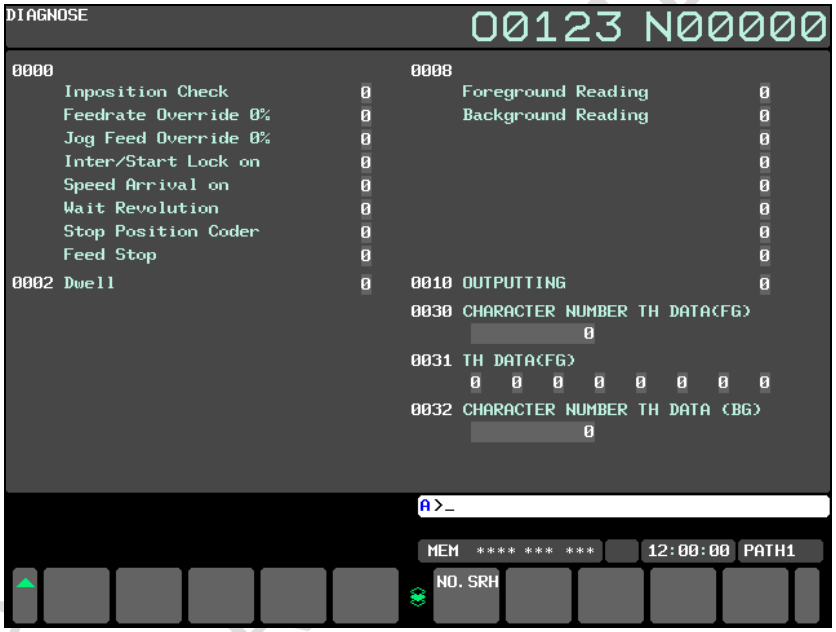


Fig. 7.3 Diagnostic display

FRYER SYSTEMS INC

7.4 RETURN FROM THE ALARM SCREEN

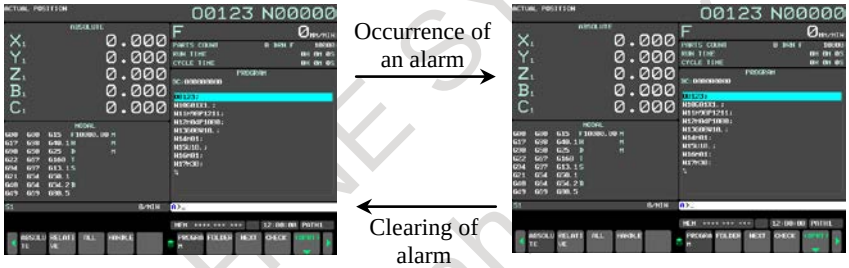
7.4.1 Return from the Alarm Screen

When alarms are cleared or function key [MESSAGE] is pressed on the alarm screen, the screen displayed before the alarm screen appears.
To enable this function, set bit 4 (ADC) of parameter No. 11302 is set to 1.

Switching between screens when alarms are cleared

When all alarms are cleared on the alarm screen, the screen displayed before the alarm screen appears again.
When the alarm screen was displayed automatically due to occurrence of an alarm, the screen displayed immediately before the alarm appears again.
When the alarm screen was displayed by pressing function key [MESSAGE] during occurrence of an alarm, the screen displayed immediately before the alarm appears again.

(Example)



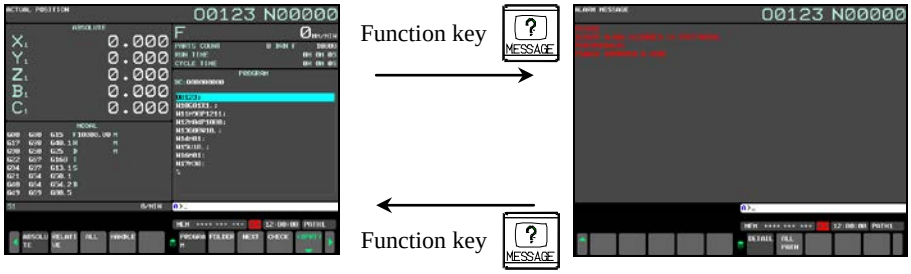
PROGRAM screen **ALARM screen**

> **NOTE**
> Even if alarms are cleared when the alarm screen is not displayed, the current screen is not changed.

Switching between screens by the function key

When function key [MESSAGE] is pressed on the alarm screen, the screen displayed before the alarm screen appears. Press function key [MESSAGE] to switch to the alarm screen for checking for alarms and then press function key [MESSAGE] to return to the previous screen.

(Example)



PROGRAM screen **ALARM screen**

FRYER MACHINE SYSTEMS INC Rob Zahn

If function key MESSAGE is pressed when the alarm screen was displayed automatically due to occurrence of an alarm, the screen displayed before the alarm screen appears again.

Restrictions

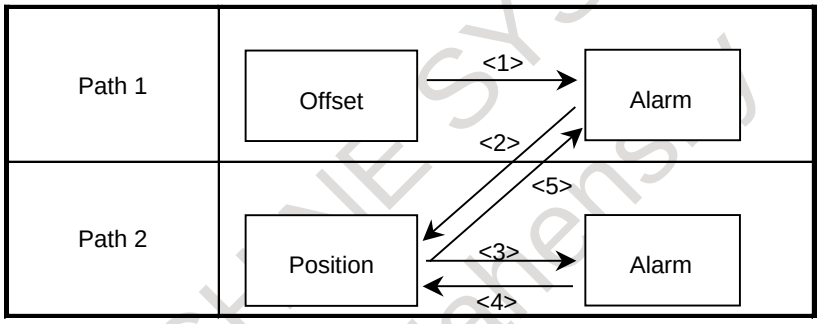
- Switching to the custom screen is not performed.
- The screens to which the alarm screen can be switched are only the screen selected by the chapter selection soft key.

7.4.2 Relationship with Other Functions

Relationship between the screen switching function and a return from the alarm screen during switching between paths

(1) When bit 5 (PSC) of parameter No. 3208 is set to 0, if paths are switched by the path switching signal, the screen last selected in the path appears again.
At this time, even if a return from the alarm screen to the previous screen is performed in one path, a return is not performed in the other path and the alarm screen remains displayed.

(Example)

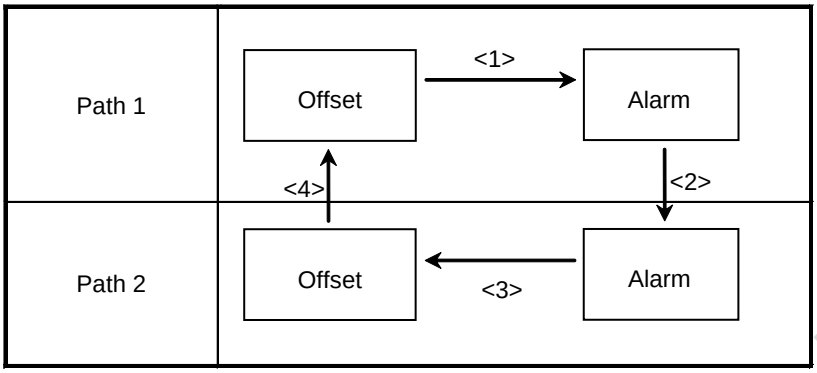


<1> When the message key is pressed on the offset screen of path 1, the alarm screen (path 1) appears.
<2> When switching to path 2 is performed on the alarm screen of path 1, the position screen of path 2 appears (when the screen last displayed in path 2 is the position screen).
<3> When the message key is pressed on the position screen of path 2, the alarm screen (path 2) appears.
<4> When the alarm is cleared or the message key is pressed on the alarm screen of path 2, a return to the position screen (path 2) is performed.
<5> When switching to path 1 is performed, the alarm screen (path 1) appears.

(2) When PSC is set to 1, if paths are switched by the path switching signal, the screen displayed immediately before the path switching appears again.
At this time, if a return from the alarm screen to the previous screen is performed in one path, the screen of the path in which a return was performed appears in the other path.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

(Example)



<1> When the message key is pressed on the offset screen of path 1, the alarm screen (path 1) appears.
<2> When switching to path 2 is performed on the alarm screen of path 1, the alarm screen (path 2) appears.
<3> When the alarm is cleared on the alarm screen of path 2, the offset screen (path 2) appears.
<4> When switching to path 1 is performed, the offset screen of path 1 appears.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

8 DATA INPUT/OUTPUT

Information stored in external I/O devices can be read into the CNC, and information can be written into external I/O devices.
External I/O devices include memory cards and USB memory that can be mounted to the memory card interface located on the left side of the display unit and personal computers and data servers that can be connected via embedded Ethernet.
The following types of data in the Table 8 (a) can be input and output.

Table 8 (a)

Data type	Default file name	Reference item
Program	ALL-PROG.TXT	8.2.1
Parameter	CNC-PARA.TXT	8.2.2
Offset data	TOOLOFST.TXT	8.2.3
Pitch error compensation data	PITCH.TXT	8.2.4
Custom macro common variable	MACRO.TXT	8.2.5
Workpiece coordinate system data	EXT_WKZ.TXT	8.2.6
Operation history data	OPRT_HIS.TXT	8.2.7
Tool management data (tool management function)	TOOL_MNG.TXT	8.2.8.1-2
Magazine data (tool management function)	MAGAZINE.TXT	8.2.8.3-4
Tool life status name (tool management function)	STATUS.TXT	8.2.8.5-6
Customize data name (tool management function)	CUSTOMIZ.TXT	8.2.8.7-8
Customize data displayed as tool management data (tool management expansion)	DISPCSTM.TXT	8.2.8.9-10
Spindle waiting position name (tool management expansion)	POSNAME.TXT	8.2.8.11-12
Decimal point position data of customize data (tool management expansion)	POINTPOS.TXT	8.2.8.13-14
Tool geometry data (tool management function for oversize tools)	TOOLGEOM.TXT	8.2.8.15-16
Tool life management data	TOOLLIFE.TXT	8.2.9
Maintenance information	MAINTINF.TXT	
Periodic maintenance data (periodic maintenance: status screen)	MAINTENA.TXT	
Second geometry tool offset	SEC_GEOM.TXT	
Real time custom macro variable	RTMMACRO.TXT	
System configuration data	SYS-CONF.TXT	
PMC signal protect	DIDOENBL.TXT	
Cause data (tool management tool attachment/detachment function)	TLCAUSE.TXT	
Attachment history data (tool management tool attachment/detachment function)	TLATAHIS.TXT	
Detachment history data (tool management tool attachment/detachment function)	TLDTAHIS.TXT	
Servo/spindle information	SV_SP_ID.TXT	
Machine system name data (periodic maintenance: machine system screen)	MAINTEMC.TXT	
Servo waveform diagnosis	WAVE-DGN.TXT	
Tool geometry data (interference check for each path)	TOOL-FRM.TXT	
P code variable	PCODE.TXT	
M code group data	MC-GROUP.TXT	

The above types of data can be input and output on the screens for displaying and setting those types of data.
If NC data such as programs and parameters is to be written to a memory card or USB memory, and if a file with the same name already exists, it is possible to select whether to overwrite the existing file or cancel writing with appropriate operation.

FRYER MACHINE SYSTEMS INC Rob Zaneski

This function is enabled by setting bit 1 (COW) of parameter No. 11308.

The external I/O device set in NC parameter No. 0020 is selected. See the Table 8 (b) below for details.

Table 8 (b)

Correspondence between settings and input/output units	
Setting	**Description**
0,1	RS-232C serial port 1
2	RS-232C serial port 2
4	Memory card interface
5	Data server interface
9	Embedded Ethernet interface
17	USB memory interface

NOTE

1 If the external input/output device is other than a data server interface or embedded Ethernet interface, a file name of up to 12 characters can be handled. NC programs on the memory of the CNC can each be assigned any file name of up to 32 characters, but the name is restricted to up to 12 characters in the input/output to and from external I/O devices.

2 If the external input/output device is a data server interface, embedded Ethernet interface, or USB memory interface, a file name of up to 32 characters can be handled.

3 When a memory card or USB memory interface is used as an external I/O device, up to 512 files can be handled.

8.1 OVERWRITING FILES ON A MEMORY CARD/USB MEMORY

Screen display

When an attempt is made to output NC data to a memory card, and if the specified file name or the default file name is the same as an existing file name on the memory card or USB memory, a confirmation message "OVERWRITE?" appears. (Bit 1 (COW) of parameter No. 11308 = 1)

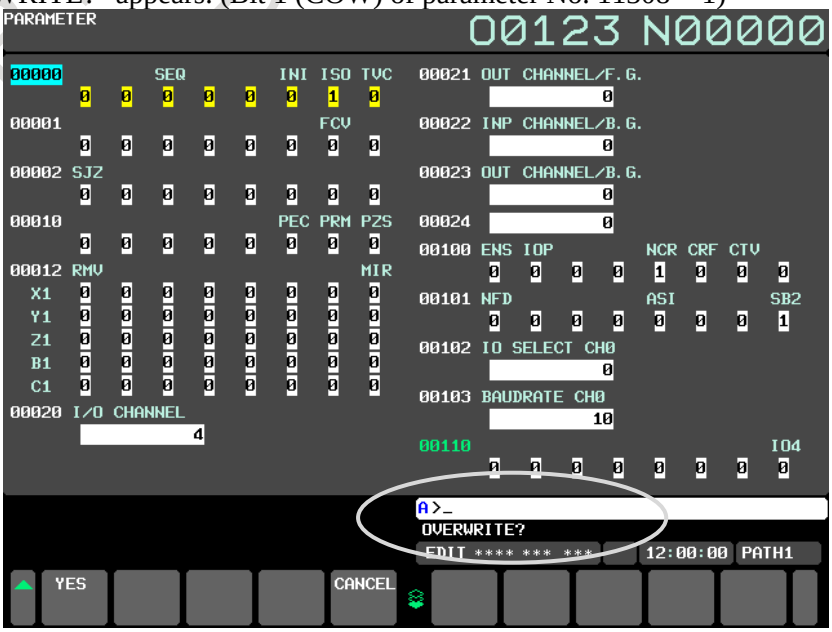


Fig. 8.1 (a) Screen display example

FRYER MACHINE SYSTEMS INC
Bob Zaharensky

Procedure

On the output screen for the desired function, perform the following operation.

1 Press the soft key [F OUTPUT].
2 Press the soft key [EXEC]. If a file with the same name does not exist on the memory card or USB memory, the file is output in this step.
3 If a file with the same name exists on the memory card or USB memory, soft keys [REWRITE] and [CAN] appear.
Pressing the soft key [REWRITE] causes the file to be overwritten.
Pressing the soft key [CAN] causes output to be canceled.

Example) Output from the parameter screen (for 10.4-inch display unit)

1 Press the function key SYSTEM.
2 Press the soft key [PARAMETER].
3 Enter the EDIT mode or state emergency stop.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT]. The soft key display switches from the one in Fig. 8.1 (b) to the one in Fig. 8.1 (c).
6 Press the soft key [EXEC]. Because no file name is specified, the file is output with a file name of CNC-PARA.TXT, but if a file with the same name exists on the memory card, the soft key display switches from the one in Fig. 8.1 (c) to the one in Fig. 8.1 (d), with a confirmation message appearing.
If a file with the same name does not exist on the memory card or USB memory, the file is output directly.
7 Pressing the soft key [REWRITE] causes the file to be overwritten.
Pressing the soft key [CAN] causes output to be canceled. If wishing to output the file after changing the file name, specify a file name after step 6, and perform step 7 again.

Fig. 8.1 (b) Soft key display before [F OUTPUT] is pressed

Fig. 8.1 (c) Soft key display after [F OUTPUT] is pressed

Fig. 8.1 (d) Soft key display after [EXEC] is pressed

Example) Output from the parameter screen (for 15/19-inch display unit)

1 Press the function key SYSTEM.
2 Press the vertical soft key [PARAMETER].
3 Enter the EDIT mode or state emergency stop.
4 Press the horizontal soft key [F OUTPUT]. The horizontal soft key display switches from the one shown in Fig. 8.1 (e) to the one in Fig. 8.1 (f).
5 Press the horizontal soft key [EXEC]. Because no file name is specified, the file is output with a file name of CNC-PARA.TXT, but if a file with the same name exists on the memory card or USB memory, the horizontal soft key display switches from the one in Fig. 8.1 (f) to the one in Fig. 8.1 (g), with a confirmation message appearing.
If a file with the same name does not exist on the memory card or USB memory, the file is output directly.

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

6 Pressing the horizontal soft key [REWRITE] causes the file to be overwritten.
Pressing the horizontal soft key [CAN] causes output to be canceled. If wishing to output the file after changing the file name, specify a file name after step 4, and perform step 5 again.

Fig. 8.1 (e) Soft key display before [F OUTPUT] is pressed

Fig. 8.1 (f) Soft key display after [F OUTPUT] is pressed

Fig. 8.1 (g) Soft key display after [EXEC] is pressed

> ⚠ **CAUTION**
> For a memory card, even if soft key [REWRITE] is pressed, warning message "OVER WRITE FAILED" is displayed and the output is canceled in the following cases. For a USB memory, alarm SR1925 or SR1928 is displayed and the output is canceled.
> - The file to be overwritten is for read only.
> - The memory card is pulled out while the confirmation message appears.

8.2 INPUT/OUTPUT ON EACH SCREEN

This section explains how to input and output data of the following types to and from each operation screen: program, parameter, offset, pitch error compensation, macro variable, workpiece coordinate system data, operation history, and tool management.

8.2.1 Inputting and Outputting a Program

8.2.1.1 Inputting a program

The following explains how to input a program from an external device to the memory of the CNC by using the program editing screen or program folder screen.

Inputting a program (for 10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key [PROG] to display the program editing screen or program folder screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the continuous menu key [▷] until soft key [F INPUT] appears.
Press the soft key [F INPUT].
6 Type the name of the file that you want to input.
Press the soft key [F SET]. To specify the program name to input, type the program name and press the soft key [P SET].

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

If the input file name is omitted, default input file name “ALL-PROG.TXT” is assumed.
For an explanation of the operations to be performed if an input file name [F SET] and an input program name [P SET] are omitted, see the
Table 8.2.1.1 (a).

7 If you input the program, press the soft key [EXEC].
If you do not input the program, press the soft key [CAN]. Soft keys return to the operation selection soft keys.
When the input operation starts, “INPUT” blinks in the lower right part of the screen. When the operation ends, the “INPUT” indication disappears.

Once read, the program is registered in the currently selected foreground folder.

NOTE
1 Foreground folders are displayed on the program folder screen. For an explanation of foreground folders, see II-12.1.3, and for how to change foreground folders, see III-11.6
2 When the programs and folders are inputted/outputted on the program list screen of folder tree display, set bit 7(FLI) of parameter No.11364 to 1.

Table 8.2.1.1 (a)

[F SET]	[P SET]	Input file name	Input program	Input program name
BLANK	BLANK	ALL-PROG.TXT	All programs in ALL-PROG.TXT	File name at the time the file is saved
BLANK	INPUT	ALL-PROG.TXT	First program in ALL-PROG.TXT	Program name set with [P SET]
INPUT	BLANK	File name set with [F SET]	All programs in the file specified with [F SET]	File name at the time the file is saved
INPUT	INPUT	File name set with [F SET]	First program in the file specified with [F SET]	Program name set with [P SET]

Inputting a program (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key PROG.
3 Press the vertical soft key [PROGRAM] or [FOLDER] to display programs or a program directory.
4 Press the EDIT switch on the machine operator’s panel.
5 Press the continuous menu key until horizontal soft key [F INPUT] appears.
Press the horizontal soft key [F INPUT].
6 Type the name of the file that you want to input.
Press the horizontal soft key [F SET]. To specify the program name to input, type the program name and press the horizontal soft key [P SET].
If the input file name is omitted, default input file name “ALL-PROG.TXT” is assumed.
For an explanation of the operations to be performed if an input file name [F SET] and an input program name [P SET] are omitted, see the Table 8.2.1.1 (b).
7 If you input the program, press the horizontal soft key [EXEC].
If you do not input the program, press the horizontal soft key [CAN]. Soft keys return to the operation selection soft keys.
When the input operation starts, “INPUT” blinks in the lower right part of the screen. When the operation ends, the “INPUT” indication disappears.

Once read, the program is registered in the currently selected foreground folder.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE
1 Foreground folders are displayed on the program folder screen. For an explanation of foreground folders, see II-12.1.3, and for how to change foreground folders, see III-11.6
2 When the programs and folders are inputted/outputted on the program list screen of folder tree display, set bit 7(FLI) of parameter No.11364 to 1.

Table 8.2.1.1 (b)

[F SET]	[P SET]	Input file name	Input program	Input program name
BLANK	BLANK	ALL-PROG.TXT	All programs in ALL-PROG.TXT	File name at the time the file is saved
BLANK	INPUT	ALL-PROG.TXT	First program in ALL-PROG.TXT	Program name set with [P SET]
INPUT	BLANK	File name set with [F SET]	All programs in the file specified with [F SET]	File name at the time the file is saved
INPUT	INPUT	File name set with [F SET]	First program in the file specified with [F SET]	Program name set with [P SET]

8.2.1.2 Outputting a program

A program stored in the memory of the CNC unit is output to an external device.

Outputting a program (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key PROG to display the program editing screen or program folder screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the continuous menu key ▷ until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
6 Type the program number to output and press the soft key [P SET]. To specify an output file name, type the output file name and press the soft key [F SET].
To output multiple programs at one time, type the start program number and the end program number separated by comma (,), and then press the soft key [P SET].
If, in this step, no output file name or no program name is specified, all programs in the foreground folders are output to "ALL-PROG.TXT".
For an explanation of the operations to be performed if an output file name [F SET] and an output program name [P SET] are omitted, see the Table 8.2.1.2 (a).
7 If you output the program, press the soft key [EXEC].
If you do not output the program, press the soft key [CAN]. Soft keys return to the operation selection soft keys.
When the output operation starts, "OUTPUT" blinks in the lower right part of the screen. When the operation ends, the "OUTPUT" indication disappears.

NOTE
1 Foreground folders are displayed on the program folder screen. For an explanation of foreground folders, see II-12.1.3, and for how to change foreground folders, see III-11.6
2 When the programs and folders are inputted/outputted on the program list screen of folder tree display, set bit 7(FLI) of parameter No.11364 to 1.

FRIEP MACHINE SYSTEMS INC
Rod Zahensky

Table 8.2.1.2 (a)

[F SET]	[P SET]	Output file name	Output program
BLANK	BLANK or (O-9999)	ALL-PROG.TXT	All programs in the foreground folders displayed in the program folder
BLANK	INPUT	Program name set with [P SET]	Program in the NC that is set with [P SET]
BLANK	Oxxxx,Oyyyy	ALL-PROG.TXT	Programs No.xxxx to No.yyyy that is set with [P SET]
INPUT	BLANK or (O-9999)	File name set with [F SET]	All programs in the foreground folders displayed in the program folder
INPUT	INPUT	File name set with [F SET]	Program in the NC that is set with [P SET]
INPUT	Oxxxx,Oyyyy	File name set with [F SET]	Programs No.xxxx to No.yyyy that is set with [P SET]

Outputting a program (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.

2 Press the function key [PROG].

3 Press the vertical soft key [PROGRAM] or [FOLDER] to display programs or a program directory.

4 Press the EDIT switch on the machine operator's panel.

5 Press the continuous menu key [▷] until horizontal soft key [F OUTPUT] appears.
Press the horizontal soft key [F OUTPUT].

6 Type the program number to output and press the horizontal soft key [P SET]. To specify an output file name, type the output file name and press the horizontal soft key [F SET].
To output multiple programs at one time, type the start program number and the end program number separated by comma (,), and then press the soft key [P SET].
If, in this step, no output file name or no program name is specified, all programs in the foreground folders are output to "ALL-PROG.TXT".
For an explanation of the operations to be performed if an output file name [F SET] and an output program name [P SET] are omitted, see the Table 8.2.1.2 (b).

7 If you output the program, press the horizontal soft key [EXEC].
If you do not output the program, press the horizontal soft key [CAN]. Soft keys return to the operation selection soft keys.
When the output operation starts, "OUTPUT" blinks in the lower right part of the screen. When the operation ends, the "OUTPUT" indication disappears.

NOTE
1 Foreground folders are displayed on the program folder screen. For an explanation of foreground folders, see II-12.1.3, and for how to change foreground folders, see III-11.6
2 When the programs and folders are inputted/outputted on the program list screen of folder tree display, set bit 7(FLI) of parameter No.11364 to 1.

Table 8.2.1.2 (b)

[F SET]	[P SET]	Output file name	Output program
BLANK	BLANK or (O-9999)	ALL-PROG.TXT	All programs in the foreground folders displayed in the program folder
BLANK	INPUT	Program name set with [P SET]	Program in the NC that is set with [P SET]
BLANK	Oxxxx,Oyyyy	ALL-PROG.TXT	Programs No.xxxx to No.yyyy that is set with [P SET]
INPUT	BLANK or (O-9999)	File name set with [F SET]	All programs in the foreground folders displayed in the program folder
INPUT	INPUT	File name set with [F SET]	Program in the NC that is set with [P SET]

FRYER MACHINE SYSTEMS INC
Rob Zahersky

[F SET]	[P SET]	Output file name	Output program
INPUT	Oxxxx,Oyyyy	File name set with [F SET]	Programs No.xxxx to No.yyyy that is set with [P SET]

8.2.1.3 Inputting and outputting of O8-digit

When the file name is not specified, the file name that is output and input is different according to the digit of the program number and the external input/output device.

- **Outputting program**

  The program is output with the following file name.

Table 8.2.1.3

Program number	External input/output device	File name
7-digit or less (Example: 12345)	Excluding Fast Data Server or Fast Data Server excluding in the storage mode	"O" plus a 7-digit numeric (Example: O0012345)
	Fast Data Server in the storage mode	"O" plus a 8-digit numeric (Example: O00012345)
8-digit (Example: 12345678)	ALL	a 8-digit numeric (Example: 12345678)

- **Inputting program**

  When the external input/output device is excluding Fast Data Server or Fast Data Server excluding in the storage mode, an input file name is "O"+7-digit numeric.
  When Fast Data Server in the storage mode, an input file name is "O"+8-digit numeric.

8.2.2 Inputting and Outputting Parameters

8.2.2.1 Inputting parameters

Parameters are loaded into the memory of the CNC unit from an external device. The input format is the same as the output format. When a parameter is loaded which has the same data number as a parameter already registered in the memory, the loaded parameter replaces the existing parameter.

Inputting parameters (for 10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key [OFS SET].
3 Press the continuous menu key [▷] until soft key [SETTING] appears.
  Press the soft key [SETTING].
4 Press the MDI switch on the machine operator's panel or enter state emergency stop.
5 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data.
  Alarm SW0100, "PARAMETER ENABLE SWITCH ON" appears.
6 Press the function key [SYSTEM].
7 Press the soft key [PARAMETER], then the parameter screen appears.
8 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
9 Press the soft key [(OPRT)].
10 Press the continuous menu key [▷] until soft key [F INPUT] appears.
  Press the soft key [F INPUT].
11 Type the name of the file that you want to input.
  If the input file name is omitted, default input file name "CNC-PARA.TXT" is assumed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12 Press the soft key [EXEC].
This starts reading the parameter, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the parameter, press the soft key [CAN].
13 Press the function key .
14 Press the soft key [SETTING].
15 Press the MDI switch on the machine operator’s panel or enter state emergency stop.
16 Enter 0 in response to the prompt for “PARAMETER WRITE” in setting data.
17 Turn the power of the CNC on again.

Inputting parameters (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [SETTING] appears.
Press the vertical soft key [SETTING].
4 Press the MDI switch on the machine operator’s panel or enter state emergency stop.
5 Enter 1 in response to the prompt for “PARAMETER WRITE” in setting data.
Alarm SW0100, “PARAMETER ENABLE SWITCH ON” appears.
6 Press the function key .
7 Press the vertical soft key [NEXT PAGE] until vertical soft key [PARAMETER] appears.
Press the vertical soft key [PARAMETER], then the parameter screen appears.
8 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
9 Press the horizontal soft key [F INPUT].
10 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “CNC-PARA.TXT” is assumed.
11 Press the horizontal soft key [EXEC].
This starts reading the parameter, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the parameter, press the horizontal soft key [CAN].
12 Press the function key .
13 Press the vertical soft key [SETTING].
14 Press the MDI switch on the machine operator’s panel or enter state emergency stop.
15 Enter 0 in response to the prompt for “PARAMETER WRITE” in setting data.
16 Turn the power of the CNC on again.

8.2.2.2 Outputting parameters

All parameters are output in a defined output format from the memory of the CNC to an external device.

Outputting parameters (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the soft key [PARAMETER], then the parameter screen appears.
4 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
5 Press the soft key [(OPRT)].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

6 Press the continuous menu key ▷ until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name “CNC-PARA.TXT” is assumed.
8 Press the soft key [EXEC].
This starts outputting the parameter, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting parameters (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key SYSTEM.
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [PARAMETER] appears.
Press the vertical soft key [PARAMETER], then the parameter screen appears.
4 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
5 Press the horizontal soft key [F OUTPUT].
6 Type the file name that you want to output.
If the file name is omitted, default file name “CNC-PARA.TXT” is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the parameter, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Explanation

- Suppressing output of parameters set to 0

When bit 1 (PRM) of parameter No. 0010 is set to 1, and soft key [EXEC] is pressed, the parameters in the Table 8.2.2.2 are not output:

Table 8.2.2.2

	Other than axis type	Axis type
Bit type	Parameter for which all bits are set to 0.	Parameter for an axis for which all bits are set to 0.
Value type	Parameter whose value is 0.	Parameter for an axis for which the value is 0.

8.2.3 Inputting and Outputting Offset Data

8.2.3.1 Inputting offset data

Offset data is loaded into the memory of the CNC from an external device. The input format is the same as for offset value output. When an offset value is loaded which has the same offset number as an offset number already registered in the memory, the loaded offset data replaces existing data.

Inputting offset data (for 10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key OFS SET.
3 Press the continuous menu key ▷ until soft key [OFFSET] appears.
Press the soft key [OFFSET].
4 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.

FRYER MACHINE SYSTEMS INC Rob Zahensky

5 Press the soft key [(OPRT)].
6 Press the continuous menu key ▷ until soft key [F INPUT] appears.
Press the soft key [F INPUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “TOOLOFST.TXT” is assumed.
8 Press the soft key [EXEC].
This starts reading the offset data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the offset data, press the soft key [CAN].

Inputting offset data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key OFS/SET.
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [OFFSET] appears.
Press the vertical soft key [OFFSET].
4 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
5 Press the horizontal soft key [F INPUT].
6 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “TOOLOFST.TXT” is assumed.
7 Press the horizontal soft key [EXEC].
This starts reading the offset data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the offset data, press the horizontal soft key [CAN].

8.2.3.2 Outputting offset data

All offset data is output in a defined output format from the memory of the CNC to an external device.

Outputting offset data (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key OFS/SET.
3 Press the continuous menu key ▷ until soft key [OFFSET] appears.
Press the soft key [OFFSET].
4 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key ▷ until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name “TOOLOFST.TXT” is assumed.
8 Press the soft key [EXEC].
This starts outputting the offset data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Outputting offset data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [OFFSET] appears.
Press the vertical soft key [OFFSET].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F OUTPUT].
6 Type the file name that you want to output.
If the file name is omitted, default file name "TOOLOFST.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the offset data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Explanation

- Output format

Output format is as follows:

M

- Tool compensation memory A (bit 6 (NGW) of parameter No.8136 = 1)

```
%
G10 G90 P01 R_ Q_
G10 G90 P02 R_ Q_
 :
G10 G90 P_ R_
%
```

Q_ : Virtual tool nose number (TIP). Not output when the virtual tool nose direction is not used.
P_ : Tool offset number (1 to the number of tool compensation pairs)
R_ : Tool compensation data. Output with a decimal point in the input unit used at output.

- Tool compensation memory C (bit 6 (NGW) of parameter No.8136 = 0)

```
%
G10 G90 L10 P01 R_ Q_
G10 G90 L11 P01 R_
G10 G90 L12 P01 R_
G10 G90 L13 P01 R_
G10 G90 L10 P02 R_ Q_
 :
G10 G90 L12 P_ R_
G10 G90 L13 P_ R_
%
```

L10 : Geometry compensation amount corresponding to the H code
L11 : Wear compensation amount corresponding to the H code
L12 : Geometry compensation amount corresponding to the D code
L13 : Wear compensation amount corresponding to the D code
Q_, P_, and R_ have the same meanings as for tool compensation memory A.

FRAMER MACHINE SYSTEMS INC
Rob Zahensky

T

The tool compensation amount and tool nose radius compensation amount are output in the following format.

```
%
G10 P01 X_ Z_ R_ Q_ Y_
G10 P02 X_ Z_ R_ Q_ Y_
 :
G10 P__ X_ Z_ R_ Q_ Y_
G10 P10001 X_ Z_ R_ Y_
G10 P10002 X_ Z_ R_ Y_
 :
G10 P100__ X_ Z_ R_ Y_
%
```

P_: Tool compensation number (1 to the number of tool compensation pairs)
Tool offset number: Specification of the tool compensation amount and tool wear compensation amount
10000 + tool offset number: Specification of the tool geometry compensation amount

X_: Tool compensation data (X). Output with a decimal point in the input unit used at output.

Z_: Tool compensation data (Z). Same as X_.

R_: Tool nose radius offset amount (R). The data format is the same as for X_.
When tool nose radius compensation is not provided, this item is not output.

Q_: Virtual tool nose number (TIP). When tool nose radius compensation is not provided, this item is not output.

Y_: Tool compensation data (Y). The data format is the same as for X_.
When no Y-axis offset is provided, this item is not output.

The second geometry tool offset data is output in the following format.

```
%
G10 P20001 X_ Z_ Y_
G10 P20002 X_ Z_ Y_
 :
G10 P200__ X_ Z_ Y_
%
```

P_ : Tool compensation number (1 to the number of tool compensation pairs)
Tool offset number: Specification of the tool compensation amount and tool wear compensation amount
20000 + tool offset number: Specification of the second tool geometry compensation amount

The other addresses are the same as for the tool compensation amount.

NOTE
The input format and output format do not depend on the G-code system A/B/C.

- Address name changing of output format

When bit 2 (OFN) of parameter No.11403 is set to 1, the specification address in "Y-axis offset and 4th/5th axis offset" can use address name 'A' or 'B' instead of default specification address 'Y', 'E' and 'F'.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

Example) When parameter is set to follow, the address is changed.
- No.5043 is set to 2 (Y-axis offset is the second axis)
- No.1020 (second axis) = 65 (Address name of Y-axis offset is 'A')
- Bit 2 (OFN) of No.11403 is set to 1 (Address depend on parameter No.1020)

```
%
G10 P01 X_ Z_ A_
G10 P02 X_ Z_ A_
 :
G10 P__ X_ Z_ A_
%
```

A_: Tool compensation data (Y-axis offset). The data format is the same as for X_. When no Y-axis offset is provided, this item is not output.
The other addresses are the same as for the tool compensation amount.

NOTE
1 The axis name that can be used as a specification address is only 'A', and 'B'. If specification address 'A' or 'B' is used, the address 'V' (incremental command of Y-axis offset) cannot be used.
2 When setting is the following condition, the default specification addresses 'Y', 'E', and 'F' are used.
  - Parameter No.1020 is set to axis name other than 'A(65)' or 'B(66)'.
  - An extended axis name is used.
3 When bit 2 (OFN) of parameter No.11403 is set to 1, conventional offset data cannot be read. If conventional offset data is read, Please set bit 2 (OFN) of parameter No.11403 to 0.

8.2.4 Inputting and Outputting Pitch Error Compensation Data

8.2.4.1 Inputting pitch error compensation data

Pitch error compensation data are loaded into the memory of the CNC from an external device. The input format is the same as the output format. When a pitch error compensation data is loaded which has the corresponding data number as a pitch error compensation data already registered in the memory, the loaded data replaces the existing data.

Inputting pitch error compensation data (for 10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key [OFS/SET].
3 Press the continuous menu key [▷] until soft key [SETTING] appears.
  Press the soft key [SETTING].
4 Press the MDI switch on the machine operator's panel or enter state emergency stop.
5 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data.
  Alarm SW0100, "PARAMETER ENABLE SWITCH ON" appears.
6 Press the function key [SYSTEM].
7 Press the continuous menu key [▷] until soft key [PITCH ERROR] appears.
  Press the soft key [PITCH ERROR].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

8 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
9 Press the soft key [(OPRT)].
10 Press the continuous menu key ▷ until soft key [F INPUT] appears.
Press the soft key [F INPUT].
11 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "PITCH.TXT" is assumed.
12 Press the soft key [EXEC].
This starts reading the pitch error compensation data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the pitch error compensation data, press the soft key [CAN].
13 Press the function key OFS/SET.
14 Press the soft key [SETTING].
15 Press the MDI switch on the machine operator's panel or enter state emergency stop.
16 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
17 Turn the power of the CNC on again.

Inputting pitch error compensation data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key OFS/SET.
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [SETTING] appears.
Press the vertical soft key [SETTING].
4 Press the MDI switch on the machine operator's panel or enter state emergency stop.
5 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data.
Alarm SW0100, "PARAMETER ENABLE SWITCH ON" appears.
6 Press the function key SYSTEM.
7 Press the vertical soft key [NEXT PAGE] until vertical soft key [PITCH ERROR] appears.
Press the vertical soft key [PITCH ERROR].
8 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
9 Press the horizontal soft key [F INPUT].
10 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "PITCH.TXT" is assumed.
11 Press the horizontal soft key [EXEC].
This starts reading the pitch error compensation data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the pitch error compensation data, press the horizontal soft key [CAN].
12 Press the function key OFS/SET.
13 Press the vertical soft key [SETTING].
14 Press the MDI switch on the machine operator's panel or enter state emergency stop.
15 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
16 Turn the power of the CNC on again.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

8.2.4.2 Outputting pitch error compensation data

All pitch error compensation data are output in a defined output format from the memory of the CNC to an external device.

Outputting pitch error compensation data (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key SYSTEM.
3 Press the continuous menu key ▷ until soft key [PITCH ERROR] appears.
Press the soft key [PITCH ERROR].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key ▷ until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name "PITCH.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the pitch error compensation data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting pitch error compensation data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key SYSTEM.
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [SETTING] appears.
Press the vertical soft key [PITCH ERROR].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F OUTPUT].
6 Type the file name that you want to output.
If the file name is omitted, default file name "PITCH.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the pitch error compensation data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.4.3 Input/output format of pitch error compensation data

Pitch error compensation data is input and output in the following input and output formats.

- Keywords

The following alphabets are used as keywords.
The numeric value following each keyword has the meaning listed below:

Table 8.2.4.3

Keyword	Meaning of the following numeric value
N	Pitch error compensation data number + 10000
Q	Data identification (1 : Parameter data, 0 : Pitch error compensation data)

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

Keyword	Meaning of the following numeric value
L	Input / Output format of pitch error compensation data (Without L : Incremental value type, L1 : Total value type)
P	Pitch error compensation data value

- Format

The input / output format of pitch error compensation data is changed depending on bit 0 (APE) of parameter No.3602.

In case of incremental value type (bit 0 (APE) of parameter No.3602 is set to 0).

N	*****	Q0	P	****	;

In case of total value type (bit 0 (APE) of parameter No.3602 is set to 1).

N	*****	Q0	L1	P	****	;

The 5-digit numeric value following N indicates a pitch error compensation data number to which a value of 10000 is added.
If the Bi-directional pitch error compensation is valid, the 5-digit numeric value following N indicates a pitch error compensation data number to which a value of 20000 is added.
Q0 indicates pitch error compensation data.
L1 indicates that the input / output format of pitch error compensation data is total value type. When the input / output format is incremental type (bit 0 (APE) of parameter No.3602 is set to 0), L1 is not to be specified. If the data with L1 is input when bit 0 (APE) of parameter No.3602 is set to 0, or the data without L1 is input when bit 0 (APE) of parameter No.3602 is set to 1, alarm SR1300 “ILLEGAL ADDRESS” is caused.
The numeric value following P indicates the value (integer value) of pitch error compensation data between -128 and 127.
The semicolon (;) indicates the end of block (LF in the ISO code or CR in the EIA code).

Example1 (bit 0 (APE) of parameter No.3602 is set to 0)

```
N10001Q0P100;
    Pitch error compensation data number  1
    Pitch error compensation data value   100
```

Example2 (bit 0 (APE) of parameter No.3602 is set to 1)

```
N10001Q0L1P100;
    Pitch error compensation data number  1
    Pitch error compensation data value   100
```

- Beginning and end of a record

A pitch error compensation data record begins with % and ends with %.

Example

```
% ;.............................................Beginning of record
N10000Q0P10;
N10001Q0P100;
   :
N11279Q0P0;
%.............................................End of record
```

When parameters and pitch error compensation data are integrated into one file, % is added to the beginning and end of the file.

FRYER MACHINE SYSTEMS INC
Ron Zahensky

> ⚠ **WARNING**
> 1 Do not fail to perform reference position return manually after changing the pitch error compensation data or backlash compensation data. Without this, the machine position can deviate from the correct position.
> 2 If the compensation data is changed when the input format is total value type(bit 0 (APE) of parameter No.3602 is set to 1), power must be turned off before operation is continued.

8.2.5 Inputting and Outputting Custom Macro Common Variables

8.2.5.1 Inputting custom macro common variables

The value of a custom macro common variable is loaded into the memory of the CNC from an external device. The same format used to output custom macro common variables is used for input.

Inputting custom macro common variables (for 10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the continuous menu key until soft key [MACRO] appears.
Press the soft key [MACRO].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key until soft key [F INPUT] appears.
Press the soft key [F INPUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "MACRO.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the custom macro common variables, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the custom macro common variables, press the soft key [CAN].

Inputting custom macro common variables (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [MACRO] appears.
Press the vertical soft key [MACRO].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F INPUT].
6 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "MACRO.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts reading the custom macro common variables, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the custom macro common variables, press the horizontal soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

- Common variables

The common variables (#500 to #549) can be input and output.
(When the addition of common variable is enabled, values from #500 to #999 can be input and output.)
#100 to #149 can be input when bit 3 (PV5) of parameter No.6001 is set to 1. (When the addition of common variable is enabled, values from #100 to #199 can be input and output.)

8.2.5.2 Outputting custom macro common variables

Custom macro common variables stored in the memory of the CNC can be output in a defined output format to an external device.

Outputting custom macro common variables (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the continuous menu key until soft key [MACRO] appears.
Press the soft key [MACRO].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name "MACRO.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the custom macro common variables, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting custom macro common variables (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [MACRO] appears.
Press the vertical soft key [MACRO].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F OUTPUT].
6 Type the file name that you want to output.
If the file name is omitted, default file name "MACRO.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the custom macro common variables, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Explanation

- Output format

The output format is as follows:
The values of custom macro variables are output in a bit-image hexadecimal representation of double-precision floating-point type data.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

```
%
G10L85P100(0000000000000000)
G10L85P101(0000000000000000)
G10L85P102(FFFFFFFFFFFFFFFF)
 :
G10L85P500(4024000000000000)
G10L85P501(4021000000000000)
G10L85P502(0000000000000000)
 :
SETVN500[ABC]
SETVN501[DEF]
SETVN502[GHI]
 :
M02
%
```

P_ : Macro variable number
SETVN_ [_] : Common variable name

NOTE
The conventional custom macro statement program format cannot be used for output.

By setting bit 0 (MCO) of parameter No.6019, it is possible to output macro variable numbers and variable data values as comments after normally output data.
The output comments do not affect data input.

- Common variable

The common variables (#500 to #549) can be input and output.
(When the addition of common variable is enabled, values from #500 to #999 can be input and output.)
#100 to #149 can be output when bit 3 (PV5) of parameter No.6001 is set to 1. (When the addition of common variable is enabled, values from #100 to #199 can be input and output.)

8.2.6 Inputting and Outputting Workpiece Coordinates System Data

8.2.6.1 Inputting workpiece coordinate system data

Coordinate system variable data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When coordinate system variable data with a data number corresponding to existing coordinate system variable data registered in the memory is loaded, the loaded coordinate system variable data replaces the existing coordinate system variable data.

Inputting workpiece coordinate system data (for 10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the continuous menu key until soft key [WORK] appears.
Press the soft key [WORK].
4 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
5 Press the soft key [(OPRT)].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

6 Press the continuous menu key ▷ until soft key [F INPUT] appears.
Press the soft key [F INPUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “EXT_WKZ.TXT” is assumed.
8 Press the soft key [EXEC].
This starts reading the workpiece coordinate system data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the workpiece coordinate system data, press the soft key [CAN].

Inputting workpiece coordinate system data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key OFS SET.
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [WORK] appears.
Press vertical soft key [WORK].
4 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
5 Press the horizontal soft key [F INPUT].
6 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “EXT_WKZ.TXT” is assumed.
7 Press the horizontal soft key [EXEC].
This starts reading the workpiece coordinate system data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the workpiece coordinate system data, press the horizontal soft key [CAN].

8.2.6.2 Outputting workpiece coordinate system data

All coordinate system variable data is output in the output format from the memory of the CNC to an external device.

Outputting workpiece coordinate system data (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key OFS SET.
3 Press the continuous menu key ▷ until soft key [WORK] appears.
Press the soft key [WORK].
4 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key ▷ until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name “EXT_WKZ.TXT” is assumed.
8 Press the soft key [EXEC].
This starts outputting the workpiece coordinate system data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Outputting workpiece coordinate system data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [WORK] appears.
Press vertical soft key [WORK].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F OUTPUT].
6 Type the file name that you want to output.
If the file name is omitted, default file name "EXT_WKZ.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the workpiece coordinate system data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.7 Inputting and Outputting Operation History Data

Only output operation is permitted on operation history data.
The output data is in text format. So, to reference the output data you must use an application that can handle text files on the personal computer.

8.2.7.1 Outputting operation history data

All operation history data is output in the output format form the memory of the CNC to an external device.

Outputting operation history data (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the continuous menu key until soft key [OPERAT HISTRY] appears.
Press the soft key [OPERAT HISTRY].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name "OPRT_HIS.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the operation history data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting operation history data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [OPERAT HISTRY] appears.
Press the vertical soft key [OPERAT HISTRY].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F OUTPUT].
6 Type the file name that you want to output.
If the file name is omitted, default file name "OPRT_HIS.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the operation history data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.7.2 Inputting operation history signal selection data

Operation history signal selection data is input from an external input/output device to the memory of the CNC. The input format is the same as the format in which operation history signal selection data is output.

Inputting operation history signal section data (for10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key SYSTEM.
3 Press the continuous menu key ▷ until soft key [OPERAT HISTRY] appears.
Press the soft key [OPERAT HISTRY].
4 Press the soft key [SIGNAL SELECT].
5 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
6 Press the soft key [(OPRT)].
7 Press the soft key [F INPUT].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "OHIS_SIG.TXT" is assumed.
9 Press the soft key [EXEC].
This starts reading the operation history signal selection data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the operation history signal selection data, press the soft key [CAN].

Inputting operation history signal section data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key SYSTEM.
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [OPERAT HISTRY] appears.
Press the vertical soft key [OPERAT HISTRY].
4 Press the vertical soft key [SIGNAL SELECT].
5 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
6 Press the horizontal soft key [F INPUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "OHIS_SIG.TXT" is assumed.
8 Press the horizontal soft key [EXEC].
This starts reading the operation history signal selection data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the operation history signal selection data, press the horizontal soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

8.2.7.3 Outputting operation history signal section data

Operation history signal selection data is output in the output format form the memory of the CNC to an external device.

Outputting operation history signal section data (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key SYSTEM.
3 Press the continuous menu key ▷ until soft key [OPERAT HISTRY] appears.
Press the soft key [OPERAT HISTRY].
4 Press the soft key [SIGNAL SELECT].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
6 Press the soft key [(OPRT)].
7 Press the soft key [F OUTPUT].
8 Type the file name that you want to output.
If the file name is omitted, default file name "OHIS_SIG.TXT" is assumed.
9 Press the soft key [EXEC].
This starts outputting the operation history signal selection data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting operation history signal section data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key SYSTEM.
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [OPERAT HISTRY] appears.
Press the vertical soft key [OPERAT HISTRY].
4 Press the vertical soft key [SIGNAL SELECT].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
6 Press the horizontal soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name "OHIS_SIG.TXT" is assumed.
8 Press the horizontal soft key [EXEC].
This starts outputting the operation history signal selection data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.7.4 Input/output format of operation history signal selection data

Operation history signal selection data is input and output in the following input and output formats.

- Keywords

The following alphabets are used as keywords.
The numeric value following each keyword has the meaning listed below:

Table 8.2.7.4

Keyword	Meaning of the following numeric value
N	Data number
L	PMC number

FRYER MACHINE SYSTEMS INC
Ron Zahensky

Keyword	Meaning of the following numeric value
X,Y,G,F,P	Signal address
Q	Bit state

- Format

Operation history signal selection data is output in the following format:

N	**	L	**	X,Y,G,F,P	*****	Q	********	;

The numeric value following the N represents an operation history signal selection data number.
The numeric value following the L represents a PMC number in a multipath PMC.
The numeric value following the X, Y, G, and F represents the signal address to be recorded in operation history. Here, P is a special keyword indicating that none is set.
The numeric value following the Q represents the bit position of the signal to be recorded in operation history.
The semicolon (;) indicates the end of block (LF in the ISO code or CR in the EIA code).

Example 1

N01 L01 G00043 Q00100111;

As operation history signal selection data number 1, bits 0, 1, 2, and 5 of G0043 of the first PMC are set.

Example 2

N02 L00 P00000 Q00000000;

For operation history signal selection data number 2, no signal selection data is set.

- Beginning and end of a record

A operation history signal selection data record begins with % and ends with %.

Example

```
% ; ..................................................Beginning of record
N01 L01 G00043 Q00100111;
N02 L01 F00001 Q00000001;
  :
N60 L00 P00000 Q00000000;
% ..................................................End of record
```

8.2.8 Inputting and Outputting Tool Management Data

NOTE

1 For multi-path systems, place all paths in the EDIT mode before performing input and output operations.
2 The format used is the same as the registration format of the G10 format.

8.2.8.1 Inputting tool management data

Tool management data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When tool management data with a data number corresponding to existing tool management data registered in the memory is loaded, the loaded tool management data replaces the existing tool management data.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Inputting tool management data (for 10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F INPUT].
6 Press the soft key [TOOL].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "TOOL_MNG.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the tool management data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the tool management data, press the soft key [CAN].

NOTE

1 When using oversize tool support of the tool management function, keep the following in mind.
- If a target tool is registered in a cartridge and interferes with other tools in registration or modification of tool geometry data of the tool management data, alarm PS5360, "TOOL INTERFERENCE CHECK ERROR" is issued. (The data is not input.)
- When restoring backup data in the state where all data related to the tool management function in the NC is cleared, restore tool geometry data, tool management data, and cartridge management table data in this order.

2 When bit 4 (OVI) of parameter No.13206 to 1, tool management data, tool offset data, and tool geometry size data can be input as one file.

Inputting tool management data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F INPUT].
7 Press the horizontal soft key [TOOL].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "TOOL_MNG.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the tool management data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the tool management data, press the horizontal soft key [CAN].

FRIER MACHINE SYSTEMS INC
Rob Zaharsky

NOTE

1 When using oversize tool support of the tool management function, keep the following in mind.
   - If a target tool is registered in a cartridge and interferes with other tools in registration or modification of tool geometry data of the tool management data, alarm PS5360, ”TOOL INTERFERENCE CHECK ERROR” is issued. (The data is not input.)
   - When restoring backup data in the state where all data related to the tool management function in the NC is cleared, restore tool geometry data, tool management data, and cartridge management table data in this order.

2 When bit 4 (OVI) of parameter No.13206 to 1, tool management data, tool offset data, and tool geometry size data can be input as one file.

8.2.8.2 Outputting tool management data

All tool management data is output in the output format from the memory of the CNC to an external device.

Outputting tool management data (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator’s panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [TOOL].
7 Type the file name that you want to output.
  If the output file name is omitted, default output file name “TOOL_MNG.TXT” is assumed.
8 Press the soft key [EXEC].
  This starts outputting the tool management data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
  To cancel the output, press the soft key [CAN].

NOTE

When bit 4 (OVI) of parameter No.13206 to 1, tool management data, tool offset data, and tool geometry size data can be output as one file. The format of tool offset data output using this function cannot be used with the programmable data input function.

Outputting tool management data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
  Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator’s panel.
6 Press the horizontal soft key [F OUTPUT].

FRYER MACHINE SYSTEMS Inc
Rob Zahensky

7 Press the horizontal soft key [TOOL].
8 Type the file name that you want to output.
If the output file name is omitted, default output file name “TOOL_MNG.TXT” is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the tool management data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

NOTE
When bit 4 (OVI) of parameter No.13206 to 1, tool management data, tool offset data, and tool geometry size data can be output as one file. The format of tool offset data output using this function cannot be used with the programmable data input function.

8.2.8.3 Inputting magazine data

Magazine data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When magazine data with a data number corresponding to existing magazine data registered in the memory is loaded, the loaded magazine data replaces the existing magazine data.

Inputting magazine data (for 10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator’s panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F INPUT].
6 Press the soft key [MAGAZINE].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “MAGAZINE.TXT” is assumed.
8 Press the soft key [EXEC].
This starts reading the magazine data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the magazine data, press the soft key [CAN].

NOTE
When using oversize tool support of the tool management function, keep the following in mind.
- If a tool interferes with other tools in registration in or modification to the cartridge management table, alarm PS5360, ”TOOL INTERFERENCE CHECK ERROR” is issued. (The data is not input.)
- When restoring backup data in the state where all data related to the tool management function in the NC is cleared, restore tool geometry data, tool management data, and cartridge management table data in this order.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Inputting magazine data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F INPUT].
7 Press the horizontal soft key [MAGAZINE].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "MAGAZINE.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the magazine data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the magazine data, press the horizontal soft key [CAN].

NOTE

When using oversize tool support of the tool management function, keep the following in mind.

- If a tool interferes with other tools in registration in or modification to the cartridge management table, alarm PS5360, "TOOL INTERFERENCE CHECK ERROR" is issued. (The data is not input.)
- When restoring backup data in the state where all data related to the tool management function in the NC is cleared, restore tool geometry data, tool management data, and cartridge management table data in this order.

8.2.8.4 Outputting magazine data

All magazine data is output in the output format from the memory of the CNC to an external device.

Outputting magazine data (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [MAGAZINE].
7 Type the file name that you want to output.
If the file name is omitted, default file name "MAGAZINE.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the magazine data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

Outputting magazine data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F OUTPUT].
7 Press the horizontal soft key [MAGAZINE].
8 Type the file name that you want to output.
If the file name is omitted, default file name "MAGAZINE.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the magazine data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.8.5 Inputting tool life status name data

Tool life status name data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When tool life status name data with a data number corresponding to existing tool life status name data registered in the memory is loaded, the loaded tool life status name data replaces the existing tool life status name data.

Inputting tool life status name data (for 10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F INPUT].
6 Press the soft key [STATUS].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "STATUS.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the tool life status name data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the tool life status name data, press the soft key [CAN].

Inputting tool life status name data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F INPUT].
7 Press the horizontal soft key [STATUS].

FRYER MACHINE SYSTEMS INC Rob Zahensky

8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “STATUS.TXT” is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the tool life status name data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the tool life status name data, press the horizontal soft key [CAN].

8.2.8.6 Outputting tool life status name data

All tool life status name data is output in the output format from the memory of the CNC to an external device.

Outputting tool life status name data (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator’s panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [STATUS].
7 Type the file name that you want to output.
If the file name is omitted, default file name “STATUS.TXT” is assumed.
8 Press the soft key [EXEC].
This starts outputting the tool life status name data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting tool life status name data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator’s panel.
6 Press the horizontal soft key [F OUTPUT].
7 Press the horizontal soft key [STATUS].
8 Type the file name that you want to output.
If the file name is omitted, default file name “STATUS.TXT” is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the tool life status name data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.8.7 Inputting name data of customize data

Name data of customize data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When name data of customize data with a data number corresponding to existing name data of customize data registered in the memory is loaded, the loaded name data of customize data replaces the existing data.

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

Inputting name data of customize data (for 10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F INPUT].
6 Press the soft key [CUSTOM].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "CUSTOMIZ.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the name data of customize data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the name data of customize data, press the soft key [CAN].

Inputting name data of customize data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F INPUT].
7 Press the horizontal soft key [CUSTOM].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "CUSTOMIZ.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the name data of customize data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the name data of customize data, press the horizontal soft key [CAN].

8.2.8.8 Outputting name data of customize data

All name data of customize data is output in the output format from the memory of the CNC to an external device.

Outputting name data of customize data (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [CUSTOM].
7 Type the file name that you want to output.
If the file name is omitted, default file name "CUSTOMIZ.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the name data of customize data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Outputting name data of customize data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F OUTPUT].
7 Press the horizontal soft key [CUSTOM].
8 Type the file name that you want to output.
If the file name is omitted, default file name "CUSTOMIZ.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the name data of customize data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.8.9 Inputting customize data displayed as tool management data

Customize data displayed as tool management data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When customize data displayed as tool management data with a data number corresponding to existing customize data displayed as tool management data registered in the memory is loaded, the loaded customize data replaces the existing data.

Inputting customize data displayed as tool management data (for10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key to display the tool management screen, magazine screen, or each tool data screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F INPUT].
6 Press the soft key [DSP-LAYOUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "DISPCSTM.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the customize data displayed as tool management data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the customize data displayed as tool management data, press the soft key [CAN].

Inputting customize data displayed as tool management data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .

FRYER MACHINE SYSTEMS INC Rob Zahensky

3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE], [EACH TOOL], or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F INPUT].
7 Press the horizontal soft key [DSP-LAYOUT].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "DISPCSTM.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the customize data displayed as tool management data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the customize data displayed as tool management data, press the horizontal soft key [CAN].

8.2.8.10 Outputting customize data displayed as tool management data

Customize data displayed as tool management data is output from the memory of the CNC to an external device in the output format.

Outputting customize data displayed as tool management data (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key to display the tool management screen, magazine screen, or each tool data screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [DSP-LAYOUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name "DISPCSTM.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the customize data displayed as tool management data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting customize data displayed as tool management data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE], [EACH TOOL], or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F OUTPUT].
7 Press the horizontal soft key [DSP-LAYOUT].

PRYER MACHINE SYSTEMS INC
Rob Zahensky

8 Type the file name that you want to output.
If the file name is omitted, default file name “DISPCSTM.TXT” is assumed.

9 Press the horizontal soft key [EXEC].
This starts outputting the customize data displayed as tool management data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.8.11 Inputting spindle waiting position name data

Spindle waiting position name data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When spindle waiting position name data with a data number corresponding to existing spindle waiting position name data registered in the memory is loaded, the loaded spindle waiting position name data replaces the existing data.

Inputting spindle waiting position name data (for 10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.

2 Press the function key to display the tool management screen, magazine screen, or each tool data screen.

3 Press the EDIT switch on the machine operator’s panel.

4 Press the soft key [(OPRT)].

5 Press the soft key [F INPUT].

6 Press the soft key [SPWP-NAME].

7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “POSNAME.TXT” is assumed.

8 Press the soft key [EXEC].
This starts reading the spindle waiting position name data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the spindle waiting position name data, press the soft key [CAN].

Inputting spindle waiting position name data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.

2 Press the function key .

3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].

4 Press the vertical soft key [MAGAZINE], [EACH TOOL], or [TOOL].

5 Press the EDIT switch on the machine operator’s panel.

6 Press the horizontal soft key [F INPUT].

7 Press the horizontal soft key [SPWP-NAME].

8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “POSNAME.TXT” is assumed.

9 Press the horizontal soft key [EXEC].
This starts reading the spindle waiting position name data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the spindle waiting position name data, press the horizontal soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

8.2.8.12 Outputting spindle waiting position name data

Spindle waiting position name data is output from the memory of the CNC to an external device in the output format.

Outputting spindle waiting position name data (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key to display the tool management screen, magazine screen, or each tool data screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [SPWP-NAME].
7 Type the file name that you want to output.
If the file name is omitted, default file name "POSNAME.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the spindle waiting position name data display, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting spindle waiting position name data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE], [EACH TOOL], or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F OUTPUT].
7 Press the horizontal soft key [SPWP-NAME].
8 Type the file name that you want to output.
If the file name is omitted, default file name "POSNAME.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the spindle waiting position name data display, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.8.13 Inputting decimal point position data of customize data

Decimal point position data of customize data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When decimal point position data of customize data with a data number corresponding to existing decimal point position data of customize data registered in the memory is loaded, the loaded decimal point position data of customize data replaces the existing data.

TRYER MACHINE SYSTEMS INC
Rob Zahensky

Inputting decimal point position data of customize data (for 10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key to display the tool management screen, magazine screen, or each tool data screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F INPUT].
6 Press the soft key [CSTM-DPOINT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "POINTPOS.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the decimal point position data of customize data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the decimal point position data of customize data, press the soft key [CAN].

Inputting decimal point position data of customize data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE], [EACH TOOL], or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F INPUT].
7 Press the horizontal soft key [CSTM-DPOINT].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "POINTPOS.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the decimal point position data of customize data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the decimal point position data of customize data, press the horizontal soft key [CAN].

8.2.8.14 Outputting decimal point position data of customize data

Decimal point position data of customize data is output from the memory of the CNC to an external device in the output format.

Outputting decimal point position data of customize data (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key to display the tool management screen, magazine screen, or each tool data screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [CSTM-DPOINT].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

7 Type the file name that you want to output.
If the file name is omitted, default file name “POINTPOS.TXT” is assumed.
8 Press the soft key [EXEC].
This starts outputting the decimal point position data of customize data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting decimal point position data of customize data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE], [EACH TOOL], or [TOOL].
5 Press the EDIT switch on the machine operator’s panel.
6 Press the horizontal soft key [F OUTPUT].
7 Press the horizontal soft key [CSTM-DPOINT].
8 Type the file name that you want to output.
If the file name is omitted, default file name “POINTPOS.TXT” is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the decimal point position data of customize data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.8.15 Inputting tool geometry data

Tool geometry data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When tool geometry data with a data number corresponding to existing tool geometry data registered in the memory is loaded, the loaded tool geometry data replaces the existing data.

Inputting tool geometry data (for 10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key to display the tool management screen, magazine screen, each tool data screen, or tool geometry data screen.
3 Press the EDIT switch on the machine operator’s panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F INPUT].
6 Press the soft key [TOOL- GEOM].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “TOOLGEOM.TXT” is assumed.
8 Press the soft key [EXEC].
This starts reading the tool geometry data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the tool geometry data, press the soft key [CAN].

FRYER MACHINE SYSTEMS INC
Ron Zahensky

NOTE
1 If the tool with a number of tool geometry data to be changed is registered to the magazine when an attempt is made to change the tool geometry data, an alarm PS5360, ”TOOL INTERFERENCE CHECK ERROR” is issued. (The data is not input.)
2 After data related to the tool management functions in the NC has all been cleared, restore backup data in the following order: Tool geometry data, tool management data, and magazine management table.

Inputting tool geometry data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE], [EACH TOOL], [TOOL], or [TOOL- GEOM].
5 Press the EDIT switch on the machine operator’s panel.
6 Press the horizontal soft key [F INPUT].
7 Press the horizontal soft key [TOOL- GEOM].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “TOOLGEOM.TXT” is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the tool geometry data, and “INPUT” blinks in the lower right part of the screen.
When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the tool geometry data, press the horizontal soft key [CAN].

NOTE
1 If the tool with a number of tool geometry data to be changed is registered to the magazine when an attempt is made to change the tool geometry data, an alarm PS5360, ”TOOL INTERFERENCE CHECK ERROR” is issued. (The data is not input.)
2 After data related to the tool management functions in the NC has all been cleared, restore backup data in the following order: Tool geometry data, tool management data, and magazine management table.

8.2.8.16 Outputting tool geometry data

Tool geometry data is output from the memory of the CNC to an external device in the output format.

Outputting tool geometry data (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key to display the tool management screen, magazine screen, each tool data screen, or tool geometry data screen.
3 Press the EDIT switch on the machine operator’s panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [TOOL- GEOM].

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

7 Type the file name that you want to output.
If the output file name is omitted, default output file name “TOOLGEOM.TXT” is assumed.
8 Press the soft key [EXEC].
This starts outputting the tool geometry data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting tool geometry data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE], [EACH TOOL], [TOOL], or [TOOL- GEOM].
5 Press the EDIT switch on the machine operator’s panel.
6 Press the horizontal soft key [F OUTPUT].
7 Press the horizontal soft key [TOOL- GEOM].
8 Type the file name that you want to output.
If the output file name is omitted, default output file name “TOOLGEOM.TXT” is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the tool geometry data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.9 Inputting and Outputting Tool Life Management Data

8.2.9.1 Inputting tool life management data

Tool life management data is loaded into the memory of the CNC from an external device such as a memory card.

Inputting tool life management data (for 10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator’s panel.
3 Press the function key .
4 Press the continuous menu key and chapter selection soft key [TOOL LIFE].
5 Press the soft key [(OPRT)].
When the tool life management (group editing) screen is displayed, press soft key [END] to display the tool life management (list) screen.
6 Press the soft key [F INPUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “TOOLLIFE.TXT” is assumed.
8 Press the soft key [EXEC].
This starts reading the tool life management data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahansky

NOTE
Before tool life management data is input, all tool life management data is erased.

Inputting tool life management data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator's panel.
3 Press the function key .
4 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL LIFE] appears.
5 Press the vertical soft key [TOOL LIFE].
When the tool life management (group editing) screen is displayed, press soft key [END] to display the tool life management (list) screen.
6 Press the horizontal soft key [F INPUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "TOOLLIFE.TXT" is assumed.
8 Press the horizontal soft key [EXEC].
This starts reading the tool life management data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

8.2.9.2 Outputting tool life management data

Tool life management data stored in the memory of the CNC is output to an external device such as a memory card.

Outputting tool life management data (for 10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator's panel.
3 Press the function key .
4 Press the continuous menu key and chapter selection soft key [TOOL LIFE].
5 Press the soft key [(OPRT)].
When the tool life management (group editing) screen is displayed, press soft key [END] to display the tool life management (list) screen.
6 Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the output file name is omitted, default output file name "TOOLLIFE.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the tool life management data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting tool life management data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator's panel.
3 Press the function key .

FRYER MACHINE SYSTEMS INC
Rob Zahensky

4 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL LIFE] appears.
5 Press the vertical soft key [TOOL LIFE].
When the tool life management (group editing) screen is displayed, press soft key [END] to display the tool life management (list) screen.
6 Press the horizontal soft key [F OUTPUT].
7 Type the file name that you want to output.
If the output file name is omitted, default output file name “TOOLLIFE.TXT” is assumed.
8 Press the horizontal soft key [EXEC].
This starts outputting the tool life management data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3 INPUT/OUTPUT ON THE ALL IO SCREEN

Just by using the ALL IO screen, you can input and output programs, parameters, offset data, pitch error compensation data, macro variables, workpiece coordinate system data, operation history data, and tool management data to an external I/O device.

⚠ CAUTION
While an external I/O device is being accessed, do not turn the power to the CNC off or remove the device, or the device may be damaged.

NOTE
The ALL IO screen can be operated only if a memory card or USB memory is selected as an external I/O device. Use NC parameter No.20 to select an external I/O device.

The following explains how to display the ALL IO screen:

Displaying the ALL IO screen (for 8.4/10.4-inch display unit)

Procedure

1 Press the function key SYSTEM.
2 Press the continuous menu key ▷ several times.
3 Press the soft key [ALL IO] to display the ALL IO screen.

The subsequent steps to select data from the ALL IO screen will be explained for each type of data.

Displaying the ALL IO screen (for 15/19-inch display unit)

Procedure

1 Press the function key SYSTEM.
2 Press the vertical soft key [NEXT PAGE] until vertical soft key [ALL I/O] appears.
3 Press the vertical soft key [ALL I/O] to display the ALL IO screen.

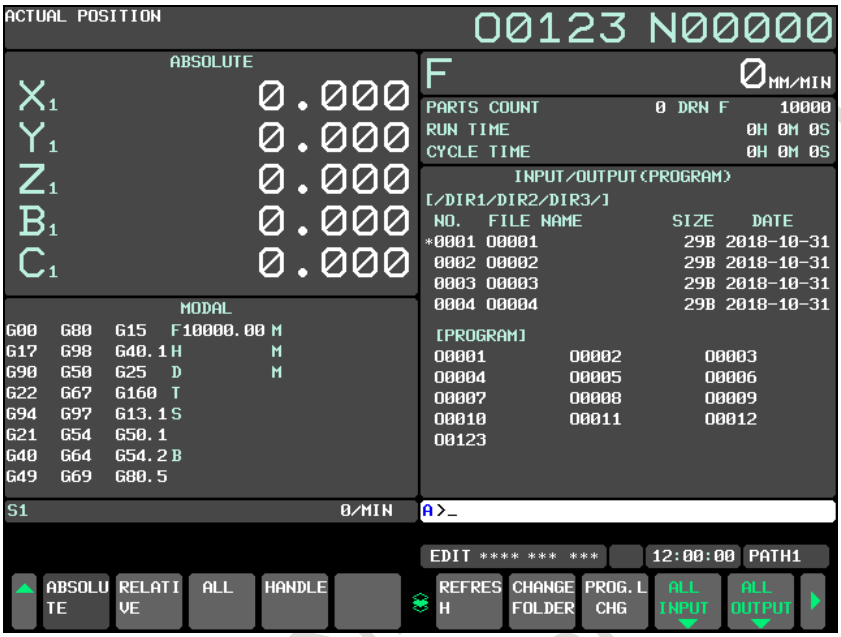
The subsequent steps to select data from the ALL IO screen will be explained for each type of data.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

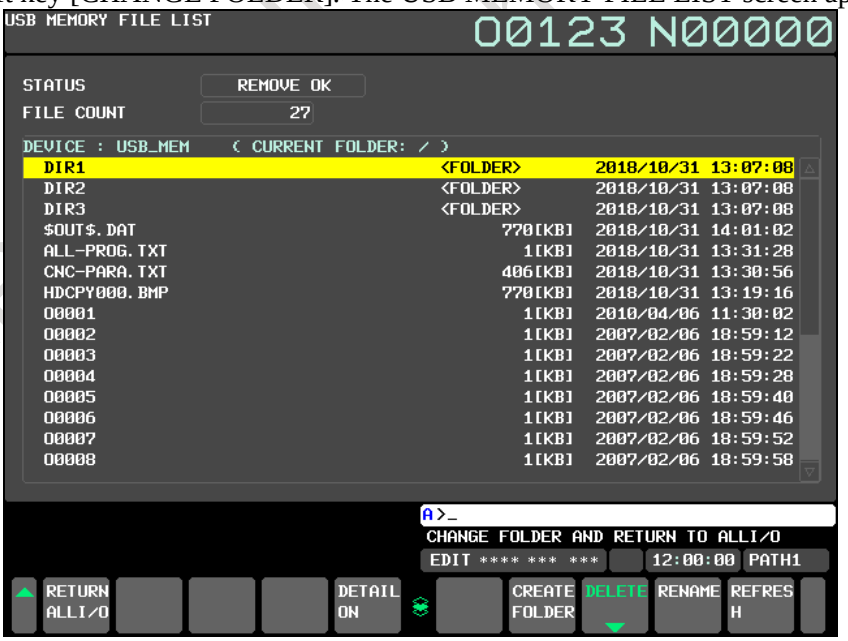
Procedure for changing the target folder for inputting and outputting data (for 8.4/10.4-inch display unit)

The last folder displayed on the USB MEMORY FILE LIST screen is treated as the default target folder for inputting and outputting data in a USB memory. If you want to input data from or output it to another folder, perform the following operation.

1 Display the ALL IO screen.



2 Press the soft key [CHANGE FOLDER]. The USB MEMORY FILE LIST screen appears.



3 Display a folder you want to use for inputting and outputting data.

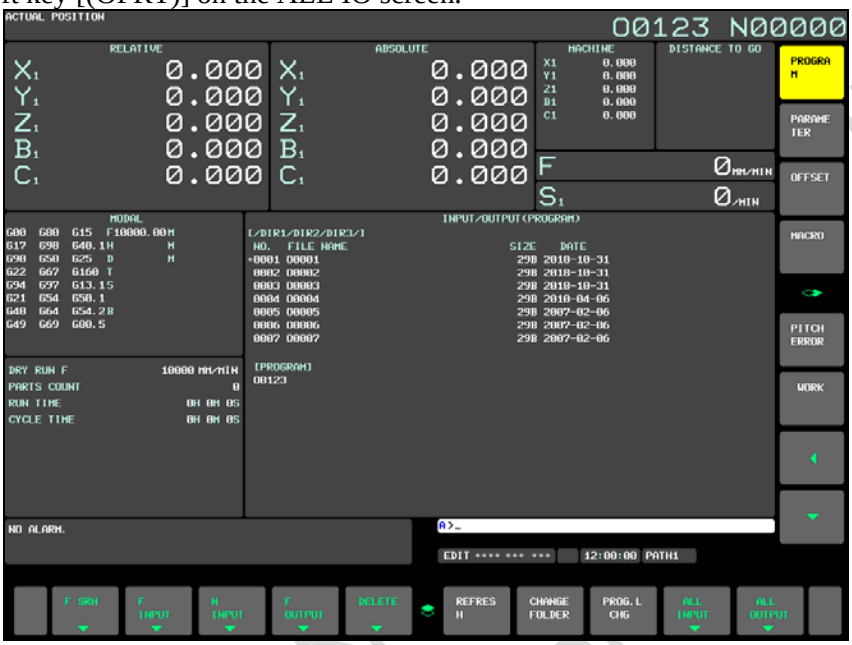
4 Press the soft key [RETURN ALL I/O]. The ALL IO screen appears again. The folder displayed on the list screen is used as the target folder for inputting and outputting data.

FRYER MINNESOTA INC

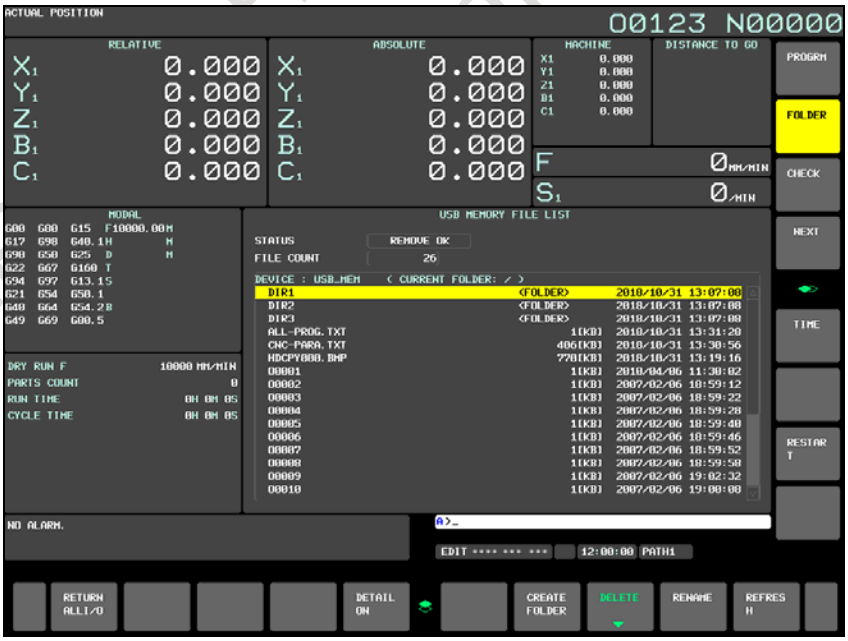
Procedure for changing the target folder for inputting and outputting data (for 15/19-inch display unit)

The last folder displayed on the USB MEMORY FILE LIST screen is treated as the default target folder for inputting and outputting data in a USB memory. If you want to input data from or output it to another folder, perform the following operation.

1 Press the soft key [(OPRT)] on the ALL IO screen.



2 Press the horizontal soft key [CHANGE FOLDER]. The USB MEMORY FILE LIST screen appears.



3 Display a folder you want to use for inputting and outputting data.

4 Press the soft key [RETURN ALL I/O]. The ALL IO screen appears again. The folder displayed on the list screen is used as the target folder for inputting and outputting data.

8.3.1 Inputting/Outputting a Program

A program can be input and output using the ALL IO screen.

Inputting a program (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [PROGRAM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "ALL-PROG.TXT" is assumed.
See the Table 8.3.1 (a) for details.
6 Set the program number to be used after the input.
Type a program number, and press the soft key [P SET].
If no program number is specified here, the program number in the file is adopted as is.
7 Press the soft key [EXEC].
This starts reading the program, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Table 8.3.1 (a)

[F SET]	[P SET]	Input file name	Input program	Input program name
BLANK	BLANK	ALL-PROG.TXT	All programs in ALL-PROG.TXT	File name at the time the file is saved
BLANK	INPUT	ALL-PROG.TXT	First program in ALL-PROG.TXT	Program name set with [P SET]
INPUT	BLANK	File name set with [F NAME]	All programs in the file specified with [F NAME]	File name at the time the file is saved
INPUT	INPUT	File name set with [F NAME]	First program in the file specified with [F NAME]	Program name set with [P SET]

Inputting a program (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PROGRAM] appears.
Press the vertical soft key [PROGRAM].
2 Press the EDIT switch on the machine operator's panel.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name "ALL-PROG.TXT" is assumed.
See the Table 8.3.1 (b) for details.
5 Set the program number to be used after the input.
Type a program number, and press the horizontal soft key [P SET].
If no program number is specified here, the program number in the file is adopted as is.
6 Press the horizontal soft key [EXEC].
This starts reading the program, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Table 8.3.1 (b)

[F SET]	[P SET]	Input file name	Input program	Input program name
BLANK	BLANK	ALL-PROG.TXT	All programs in ALL-PROG.TXT	File name at the time the file is saved
BLANK	INPUT	ALL-PROG.TXT	First program in ALL-PROG.TXT	Program name set with [P SET]
INPUT	BLANK	File name set with [F NAME]	All programs in the file specified with [F NAME]	File name at the time the file is saved
INPUT	INPUT	File name set with [F NAME]	First program in the file specified with [F NAME]	Program name set with [P SET]

Outputting a program (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [PROGRAM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the program that you want to output.
Type a program number, and press the soft key [P SET].
If O-9999 is typed, all programs in the memory are output.
If the start program number and the end program number separated by comma (,) is typed, multiple programs in specified range are output at one time.
6 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
When no file name is set, the output file name is assumed to be "O-number" if a single program number is specified; if O-9999 is specified, the output file name is assumed to be "ALL-PROG.TXT".
See the Table 8.3.1 (c) for details.
(For an explanation of foreground folders, see Section, "Default Folders", and for how to change foreground folders, see Section, "SELECTING A DEFAULT FOLDER")
7 Press the soft key [EXEC].
This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Table 8.3.1 (c)

[F SET]	[P SET]	Output file name	Output program
BLANK	BLANK or (O-9999)	ALL-PROG.TXT	All programs in the foreground folders displayed in the program folder
BLANK	INPUT	Program name set with [P SET]	Program in the NC that is set with [P SET]
BLANK	Oxxxx,Oyyyy	ALL-PROG.TXT	Programs No.xxxx to No.yyyy that is set with [P SET]
INPUT	BLANK or (O-9999)	File name set with [F NAME]	All programs in the foreground folders displayed in the program folder
INPUT	INPUT	File name set with [F NAME]	Program in the NC that is set with [P SET]
INPUT	Oxxxx,Oyyyy	File name set with [F SET]	Programs No.xxxx to No.yyyy that is set with [P SET]

Outputting a program (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PROGRAM] appears.
Press the vertical soft key [PROGRAM].
2 Press the EDIT switch on the machine operator's panel
3 Press the horizontal soft key [F OUTPUT].

FRYER MACHINE SYSTEMS INC Rob Zaremsky

4 Set the program that you want to output.
Type a program number, and press the horizontal soft key [P SET].
If O-9999 is typed, all programs in the memory are output.
If the start program number and the end program number separated by comma (,) is typed, multiple programs in specified range are output at one time.

5 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
When no file name is set, the output file name is assumed to be “O-number” if a single program number is specified; if O-9999 is specified, the output file name is assumed to be “ALL-PROG.TXT”.
See the Table 8.3.1 (d) for details.
(For an explanation of foreground folders, see Section, “Default Folders”, and for how to change foreground folders, see Section, “SELECTING A DEFAULT FOLDER”)

6 Press the horizontal soft key [EXEC].
This starts outputting the program, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Table 8.3.1 (d)

[F SET]	[P SET]	Output file name	Output program
BLANK	BLANK or (O-9999)	ALL-PROG.TXT	All programs in the foreground folders displayed in the program folder
BLANK	INPUT	Program name set with [P SET]	Program in the NC that is set with [P SET]
BLANK	Oxxxx,Oyyyy	ALL-PROG.TXT	Programs No.xxxx to No.yyyy that is set with [P SET]
INPUT	BLANK or (O-9999)	File name set with [F NAME]	All programs in the foreground folders displayed in the program folder
INPUT	INPUT	File name set with [F NAME]	Program in the NC that is set with [P SET]
INPUT	Oxxxx,Oyyyy	File name set with [F SET]	Programs No.xxxx to No.yyyy that is set with [P SET]

8.3.2 Inputting/Outputting all Programs and Folders

All programs and folders can be input and output using the ALL IO screen.

Inputting all programs and folders (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [PROGRAM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel.
Select EDIT mode for all paths.
3 Press the soft key [(OPRT)].
4 Press the soft key [ALL INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name “ALL-FLDR.TXT” is assumed.
6 Press the soft key [EXEC].
This starts reading the program, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

FRYER MACHINE SYSTEMS INC Rob Zahansky

Inputting all programs and folders (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PROGRAM] appears.
Press the vertical soft key [PROGRAM].
2 Press the EDIT switch on the machine operator's panel.
Select EDIT mode for all paths.
3 Press the horizontal soft key [ALL INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name "ALL-FLDR.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the program, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting all programs and folders (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [PROGRAM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [ALL OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "ALL-FLDR.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting all programs and folders (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PROGRAM] appears.
Press the vertical soft key [PROGRAM].
2 Press the EDIT switch on the machine operator's panel.
3 Press the horizontal soft key [ALL OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name "ALL-FLDR.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

8.3.3 Inputting and Outputting Parameters

Parameters can be input and output using the ALL IO screen.

Inputting parameters (for 8.4/10.4-inch display unit)

Procedure

1 Press the function key .
2 Press the continuous menu key until soft key [SETTING] appears.
Press the soft key [SETTING].
3 Press the MDI switch on the machine operator's panel or enter state emergency stop.
4 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data. Alarm SW0100, "PARAMETER ENABLE SWITCH ON" appears.
5 Press the soft key [PARAMETER] on the ALL IO screen.
6 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
7 Press the soft key [(OPRT)].
8 Press the soft key [N INPUT].
9 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "CNC-PARA.TXT" is assumed.
10 Press the soft key [EXEC].
This starts reading the parameter, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].
11 Press the function key .
12 Press the soft key [SETTING].
13 Press the MDI switch on the machine operator's panel or enter state emergency stop.
14 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
15 Turn the power of the CNC on again.

Inputting parameters (for 15/19-inch display unit)

Procedure

1 Press the function key .
2 Press the vertical soft key [NEXT PAGE] until vertical soft key [SETTING] appears.
Press the vertical soft key [SETTING].
3 Press the MDI switch on the machine operator's panel or enter state emergency stop.
4 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data. Alarm SW0100, "PARAMETER ENABLE SWITCH ON" appears.
5 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PARAMETER] appears.
Press the vertical soft key [PARAMETER].
6 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
7 Press the horizontal soft key [N INPUT].
8 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name "CNC-PARA.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the parameter, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rod Zuhensky

10 Press the function key .
11 Press the vertical soft key [SETTING].
12 Press the MDI switch on the machine operator's panel or enter state emergency stop.
13 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
14 Turn the power of the CNC on again.

Outputting parameters (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [PARAMETER] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name "CNC-PARA.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the parameter, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting parameters (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PARAMETER] appears.
Press the vertical soft key [PARAMETER].
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name "CNC-PARA.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the parameter, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3.4 Inputting and Outputting Offset Data

Offset data can be input and output using the ALL IO screen.

Inputting offset data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [OFFSET] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "TOOLOFST.TXT" is assumed.

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

6 Press the soft key [EXEC].
This starts reading the offset data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting offset data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [OFFSET] appears.
Press the vertical soft key [OFFSET].
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name “TOOLOFST.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the offset data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting offset data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [OFFSET] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name “TOOLOFST.TXT” is assumed.
6 Press the soft key [EXEC].
This starts outputting the offset data, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting offset data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [OFFSET] appears.
Press the vertical soft key [OFFSET].
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name “TOOLOFST.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the offset data, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

8.3.5 Inputting/Outputting Pitch Error Compensation Data

Pitch error compensation data can be input and output using the ALL IO screen.

Inputting pitch error compensation data (for 8.4/10.4-inch display unit)

Procedure

1 Press the function key .
2 Press the continuous menu key until soft key [SETTING] appears.
Press the soft key [SETTING].
3 Press the MDI switch on the machine operator's panel or enter state emergency stop.
4 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data. Alarm SW0100, "PARAMETER ENABLE SWITCH ON" appears.
5 Press the soft key [PITCH] on the ALL IO screen.
6 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
7 Press the soft key [(OPRT)].
8 Press the soft key [N INPUT].
9 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "PITCH.TXT" is assumed.
10 Press the soft key [EXEC].
This starts reading the pitch error compensation data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].
11 Press the function key .
12 Press the soft key [SETTING].
13 Press the MDI switch on the machine operator's panel or enter state emergency stop.
14 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
15 Turn the power of the CNC on again.

Inputting pitch error compensation data (for 15/19-inch display unit)

Procedure

1 Press the function key .
2 Press the vertical soft key [NEXT PAGE] until vertical soft key [SETTING] appears.
Press the vertical soft key [SETTING].
3 Press the MDI switch on the machine operator's panel or enter state emergency stop.
4 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data. Alarm SW0100, "PARAMETER ENABLE SWITCH ON" appears.
5 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PITCH ERROR] appears.
Press the vertical soft key [PITCH ERROR].
6 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
7 Press the horizontal soft key [N INPUT].
8 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name "PITCH.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the pitch error compensation data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

10 Press the function key .
11 Press the vertical soft key [SETTING].
12 Press the MDI switch on the machine operator's panel or enter state emergency stop.
13 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
14 Turn the power of the CNC on again.

Outputting pitch error compensation data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [PITCH] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name "PITCH.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the pitch error compensation data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting pitch error compensation data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PITCH ERROR] appears.
Press the vertical soft key [PITCH ERROR].
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name "PITCH.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the pitch error compensation data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3.6 Inputting/Outputting Custom Macro Common Variables

Custom macro common variables can be input and output using the ALL IO screen.

Inputting custom macro common variables (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [MACRO] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "MACRO.TXT" is assumed.

FRYER MACHINE SYSTEMS INC
Rob Zahenskyj

6 Press the soft key [EXEC].
This starts reading the custom macro common variables, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting custom macro common variables (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [MACRO] appears.
Press the vertical soft key [MACRO].
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name “MACRO.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the custom macro common variables, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting custom macro common variables (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [MACRO] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name “MACRO.TXT” is assumed.
6 Press the soft key [EXEC].
This starts outputting the custom macro common variables, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting custom macro common variables (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [MACRO] appears.
Press the vertical soft key [MACRO].
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name “MACRO.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the custom macro common variables, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

8.3.7 Inputting and Outputting Workpiece Coordinates System Data

Workpiece coordinates system data can be input and output using the ALL IO screen.

Inputting workpiece coordinate system data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [WORK] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "EXT_WKZ.TXT" is assumed.
6 Press the soft key [EXEC].
This starts reading the workpiece coordinate system data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting workpiece coordinate system data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [WORK] appears.
Press vertical soft key [WORK].
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name "EXT_WKZ.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the workpiece coordinate system data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting workpiece coordinate system data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [WORK] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name "EXT_WKZ.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the workpiece coordinate system data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahonsky

Outputting workpiece coordinate system data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [WORK] appears.
Press vertical soft key [WORK].
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name "EXT_WKZ.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the workpiece coordinate system data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3.8 Inputting and Outputting Operation History Data

Operation history data can be output on the ALL IO screen (with function key SYSTEM for 15/19-inch display unit).
Only output operation is permitted for operation history data.
The output data is of text format. So, to reference the output data you must use an application that can handle text files on the personal computer.

Outputting operation history data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [OPERAT HISTRY] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name "OPRT_HIS.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the operation history data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting operation history data (for 15/19-inch display unit)

Procedure

1 Press the function key SYSTEM.
2 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [OPERAT HISTRY] appears.
Press the vertical soft key [OPERAT HISTRY].
3 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
4 Press the horizontal soft key [F OUTPUT].
5 Set the file name to be output.
If the file name is omitted, default file name "OPRT_HIS.TXT" is assumed.

PROVEN MACHINE SYSTEMS INC Rob Zahensky

6 Press the horizontal soft key [EXEC].
This starts outputting the operation history data, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears. To cancel the output, press the horizontal soft key [CAN].

8.3.9 Inputting and Outputting Tool Management Data

Tool management data can be input and output using the ALL IO screen.

NOTE
1 For multi-path systems, place all paths in the EDIT mode before performing input and output operations.
2 The format used is the same as the registration format of the G10 format.

Inputting tool management data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [TOOL] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name “TOOL_MNG.TXT” is assumed.
6 Press the soft key [EXEC].
This starts reading the tool management data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting tool management data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL] appears.
Press the vertical soft key [TOOL].
2 Press the EDIT switch on the machine operator’s panel.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name “TOOL_MNG.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the tool management data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting tool management data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [TOOL] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the output file name is omitted, default output file name “TOOL_MNG.TXT” is assumed.
6 Press the soft key [EXEC].
This starts outputting the tool management data, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting tool management data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL] appears.
Press the vertical soft key [TOOL].
2 Press the EDIT switch on the machine operator’s panel.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the output file name is omitted, default output file name “TOOL_MNG.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the tool management data, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Inputting magazine data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [MAGAZINE] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name “MAGAZINE.TXT” is assumed.
6 Press the soft key [EXEC].
This starts reading the magazine data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting magazine data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [MAGAZINE] appears.
Press the vertical soft key [MAGAZINE].
2 Press the EDIT switch on the machine operator’s panel.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name “MAGAZINE.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the magazine data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Outputting magazine data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [MAGAZINE] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name "MAGAZINE.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the magazine data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting magazine data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [MAGAZINE] appears.
Press the vertical soft key [MAGAZINE].
2 Press the EDIT switch on the machine operator's panel.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name "MAGAZINE.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the magazine data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Inputting tool life status name data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [STATUS] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "STATUS.TXT" is assumed.
6 Press the soft key [EXEC].
This starts reading the tool life status name data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting tool life status name data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [STATUS] appears.
Press the vertical soft key [STATUS].
2 Press the EDIT switch on the machine operator's panel.
3 Press the horizontal soft key [N INPUT].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name “STATUS.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the tool life status name data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting tool life status name data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [STATUS] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name “STATUS.TXT” is assumed.
6 Press the soft key [EXEC].
This starts outputting the tool life status name data, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting tool life status name data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [STATUS] appears.
Press the vertical soft key [STATUS].
2 Press the EDIT switch on the machine operator’s panel.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name “STATUS.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the tool life status name data, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Inputting name data of customize data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [CUSTOM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name “CUSTOMIZ.TXT” is assumed.
6 Press the soft key [EXEC].
This starts reading the name data of customize data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Inputting name data of customize data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [CUSTOM] appears.
Press the vertical soft key [CUSTOM].
2 Press the EDIT switch on the machine operator's panel.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name "CUSTOMIZ.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the name data of customize data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting name data of customize data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [CUSTOM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name "CUSTOMIZ.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the name data of customize data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting name data of customize data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [CUSTOM] appears.
Press the vertical soft key [CUSTOM].
2 Press the EDIT switch on the machine operator's panel.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name "CUSTOMIZ.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the name data of customize data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3.10 Inputting and Outputting All Tool Management Data at a Time

All data used by the tool management function is classified into the following two types: "all tool data" and "all custom data" and each type of data is input and output at a time on the ALL IO screen.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Explanation

- Input/output data

Each type of data of the tool management function is input from and output to an external input/output device. The following tables list the types of output data and the data output order.

Table 8.3.10 (a) All tool data

Output order	Output data
1	Tool geometry data
2	Tool management data
3	Magazine management table
4	Magazine property data
5	Pot property data
6	Tool offset data
7	Tool geometry size data

Table 8.3.10 (b) All custom data

Output order	Output data
1	Name data of customize data
2	Tool life status name data
3	Customize data displayed as tool management data
4	Spindle and waiting position name data
5	Decimal point position data of customize data
6	Removal reason data
7	Number of decimal places of magazine property customize data
8	Number of decimal places of pot property customize data
9	Name data of magazine property customize data
10	Name data of pot property customize data
11	Name data of tool type numbers

NOTE

1 For a multi-path system, place all paths in the EDIT mode before inputting and outputting data.
2 After the following tool management data is input normally, turn the power off:
- Customize data displayed as tool management data
- Decimal point position data of customize data
- Number of decimal places of magazine property customize data
- Number of decimal places of pot property customize data

3 If a target tool is registered in a magazine and interferes with other tools in registration or modification of tool geometry data, alarm PS5360, ”TOOL INTERFERENCE CHECK ERROR” is issued. In this case, erase the tool geometry data, tool management data, and tool magazine table data in the NC or edit input data, then input all data at a time.
4 If a function related to input tool management data is disabled or the input data format is invalid, input operation is stopped.
5 When creating or editing input data, do not change the order in which data is output at a time.
6 The tool offset data output using this function cannot be used by the programmable data input function.
7 For a multi-path system, do not input data output for another path.

FRYER MACHINE SYSTEMS INC
Bob Zaremsky

Inputting all tool data at a time (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [ALL TOOL] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
6 Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "AL_TLMGR.TXT" is assumed.
7 Press the soft key [EXEC].
This starts reading the data, and "INPUT" blinks in the lower right part of the screen.
When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting all tool data at a time (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [ALL TOOL] appears.
2 Press the vertical soft key [ALL TOOL].
3 Press the EDIT switch on the machine operator's panel.
4 Press the horizontal soft key [N INPUT].
5 Set the name of the file that you want to input.
6 Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "AL_TLMGR.TXT" is assumed.
7 Press the soft key [EXEC].
This starts reading the data, and "INPUT" blinks in the lower right part of the screen.
When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Outputting all tool data at a time (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [ALL TOOL] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
6 Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default output file name "AL_TLMGR.TXT" is assumed.
7 Press the soft key [EXEC].
This starts outputting the data, and "OUTPUT" blinks in the lower right part of the screen.
When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Outputting all tool data at a time (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [ALL TOOL] appears.
2 Press the vertical soft key [ALL TOOL].
3 Press the EDIT switch on the machine operator's panel.
4 Press the horizontal soft key [F OUTPUT].
5 Set the file name to be output.

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

6 Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default output file name “AL_TLMGR.TXT” is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the data, and “OUTPUT” blinks in the lower right part of the screen.
When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Inputting all custom data at a time (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [ALL CUSTOM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
6 Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name “AL_TLCSM.TXT” is assumed.
7 Press the soft key [EXEC].
This starts reading the data, and “INPUT” blinks in the lower right part of the screen.
When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting all custom data at a time (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [ALL CUSTOM] appears.
2 Press the vertical soft key [ALL CUSTOM].
3 Press the EDIT switch on the machine operator’s panel.
4 Press the horizontal soft key [N INPUT].
5 Set the name of the file that you want to input.
6 Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name “AL_TLCSM.TXT” is assumed.
7 Press the soft key [EXEC].
This starts reading the data, and “INPUT” blinks in the lower right part of the screen.
When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

Outputting all custom data at a time (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [ALL CUSTOM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
6 Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default output file name “AL_TLCSM.TXT” is assumed.
7 Press the soft key [EXEC].
This starts outputting the data, and “OUTPUT” blinks in the lower right part of the screen.
When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Outputting all custom data at a time (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [ALL CUSTOM] appears.
2 Press the vertical soft key [ALL CUSTOM].
3 Press the EDIT switch on the machine operator's panel.
4 Press the horizontal soft key [F OUTPUT].
5 Set the file name to be output.
6 Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default output file name "AL_TLCSM.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the data, and "OUTPUT" blinks in the lower right part of the screen.
When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3.11 Inputting and Outputting Workpiece Setting Error Compensation Value

Workpiece setting error compensation value can be input and output using the ALL IO screen.

Inputting workpiece setting error compensation value (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [WORK SET ER] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "WSEC_VAL.TXT" is assumed.
6 Press the soft key [EXEC].
This starts reading the workpiece setting error compensation value, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting workpiece setting error compensation value (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [WORK SET ER] appears.
Press the vertical soft key [WORK SET ER].
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name "WSEC_VAL.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the workpiece setting error compensation value, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

FRYER MACHINE SYSTEMS INC
Rob Zahenskyj

Outputting workpiece setting error compensation value (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [WORK SET ER] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default output file name "WSEC_VAL.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the workpiece setting error compensation value, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].
7 To cancel the output, press the soft key [CAN].

Outputting workpiece setting error compensation value (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [WORK SET ER] appears.
Press the vertical soft key [WORK SET ER].
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default output file name "WSEC_VAL.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the workpiece setting error compensation value, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3.12 File Format and Error Messages

Explanation

- **File format**

All files that are read from and written to an external device are of text format. The format is described below.
A file starts with % or LF, followed by the actual data. A file always ends with %. In a read operation, data between the first % and the next LF is skipped. Each block ends with an LF, not a semicolon (;).

- LF: 0A (hexadecimal) of ASCII code
- When a file containing lowercase letters, kana characters, and several special characters (such as $, \, and !) is read, those letters and characters are ignored.
  Example)

```
%
O0001(MEMORY CARD SAMPLE FILE)
G17 G49 G97
G92 X-11.3 Y2.33
  :
  :
M30
%
```

- ASCII code is used for input/output, regardless of the setting parameter (ISO/EIA).

FRYER MACHINE SYSTEMS INC
Rob Zahansky

- Bit 3 (NCR) of parameter No. 0100 can be used to specify whether the end of block code (EOB) is output as "LF" only, or as "LF, CR, CR."

Limitation

- Memory card specification

Use memory cards that comply with PCMCIA Ver2.0 or JEIDA Ver4.1.

- Attribute memory

Memory cards that have no attribute memory or contain no device information in the attribute memory cannot be used.

- Flash ROM card

Flash ROM cards can be used for reading only.

8.4 EMBEDDED ETHERNET OPERATIONS

8.4.1 Host File List Display

> ⚠ **CAUTION**
> Before performing FTP communication using the Embedded Ethernet for the first time, consult with your network administrator, carefully set a network address and other items, and conduct communication tests thoroughly. Any error in settings such as a network address setting can lead to an adverse influence such as a communication failure on the entire network.
> Be very careful about any communication failure. If the Ethernet is used on a network involves with a communication failure, a communication failure intermittently occurs in the Ethernet, which may cause a CNC system error.

> **NOTE**
> 1 With the FTP File Transfer function (FTP client), a single CNC can connect only one FTP server.
> 2 When a lot of files are registered in the host computer, the display processing of host file list display might become slow. In this case, please divide the folder that registers the files.

Host file list display (for 8.4/10.4-inch display unit)

A list of the files held on the host computer is displayed.

Procedure

1 Press the function key [PROG].
2 Press the soft key [FOLDER]. The program list screen appears. (If the soft key does not appear, press the continuous menu key [▷].)
3 Press the soft key [(OPRT)].
4 Press the soft key [DEVICE CHANGE].
5 Press the soft key [EMB ETHER], and the EMBEDDED ETHERNET HOST FILE LIST screen appears. When DNC operation from the Embedded Ethernet is valid (bit 1 (EDE) of parameter is 1), M198 OPE FOLDER and DNC OPE FILE are display.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

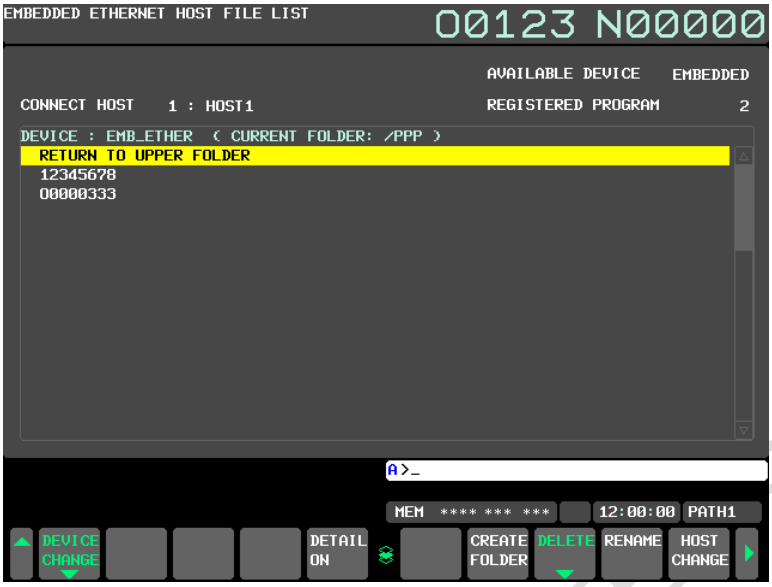


Fig. 8.4.1 (a) Embedded Ethernet host file list screen

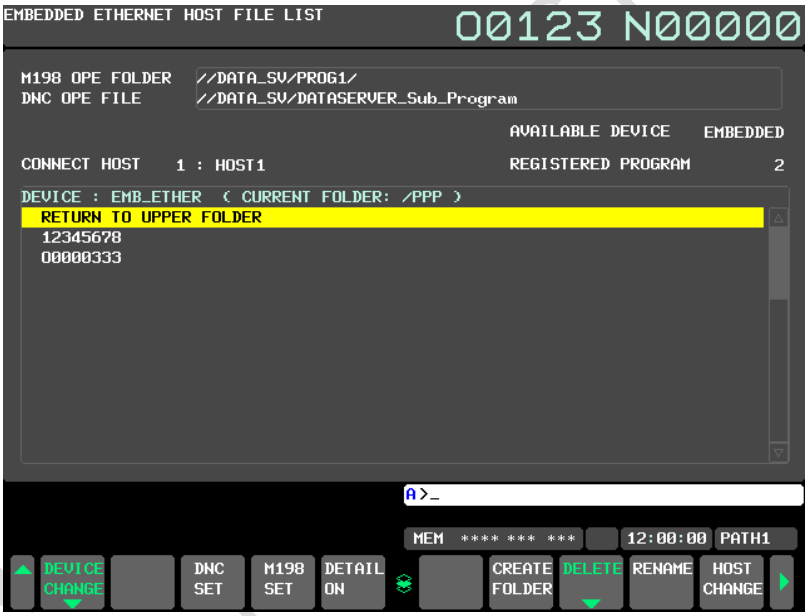


Fig. 8.4.1 (b) Embedded Ethernet host file list screen
(DNC operation from the Embedded Ethernet is valid)

NOTE

1 When using the FTP file transfer function, check that the valid device is the embedded Ethernet port.
The three conditions below determine a connection destination on the host file list screen:
(1) Check that the valid device is the embedded Ethernet port. Make a selection in "DEVICE SELECTION" on the Ethernet setting screen.
(2) Set connection destinations on the FTP transfer screen. The FTP transfer screen is displayed by pressing soft key [FTP TRANS] on the Ethernet setting screen.
(3) A host computer can be selected from connection destinations 1, 2, and 3. Make a selection according to soft key [HOST CHANGE] described later.

2 File names including 2-bytes characters cannot be displayed correctly.

6 When a list of files is larger than one page, the screen display can be switched using the page keys [PAGE ⇧] [PAGE ⇩].

7 Press the soft key [REFRESH] to update the screen display.

8 Press the soft key [DETAIL OFF] to display the host file list (file names only) screen. Press the soft key [DETAIL ON] to display the host file list (detail) screen.

9 When selecting a folder with cursor and pressing the MDI key [INPUT], the selected folder is changed to the new current folder.

Host file list display (for 15/19-inch display unit)

A list of the files held on the host computer is displayed.

Procedure

1 Press the function key [PROG].

2 Press the vertical soft key [FOLDER]. The program list screen appears.

3 Press the continuous menu key [▷] until horizontal soft key [DEVICE CHANGE] appears. Press the horizontal soft key [DEVICE CHANGE].

4 Press the horizontal soft key [EMB ETHER], and the EMBEDDED ETHERNET HOST FILE LIST screen appears. When DNC operation from the Embedded Ethernet is valid (bit 1 (EDE) of parameter is 1), M198 OPE FOLDER and DNC OPE FILE are display.

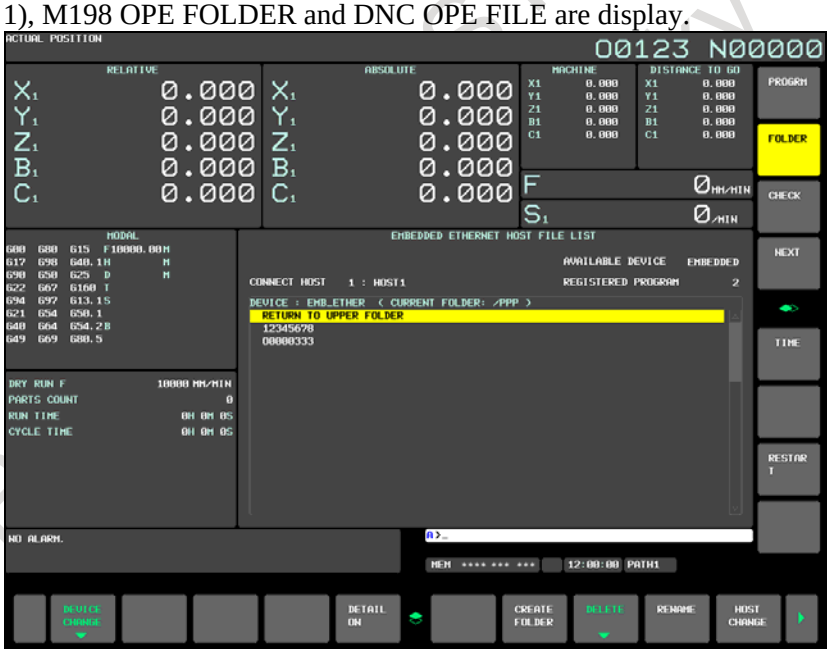


Fig. 8.4.1 (c) Embedded Ethernet host file list screen

FRYER SYSTEMS INC

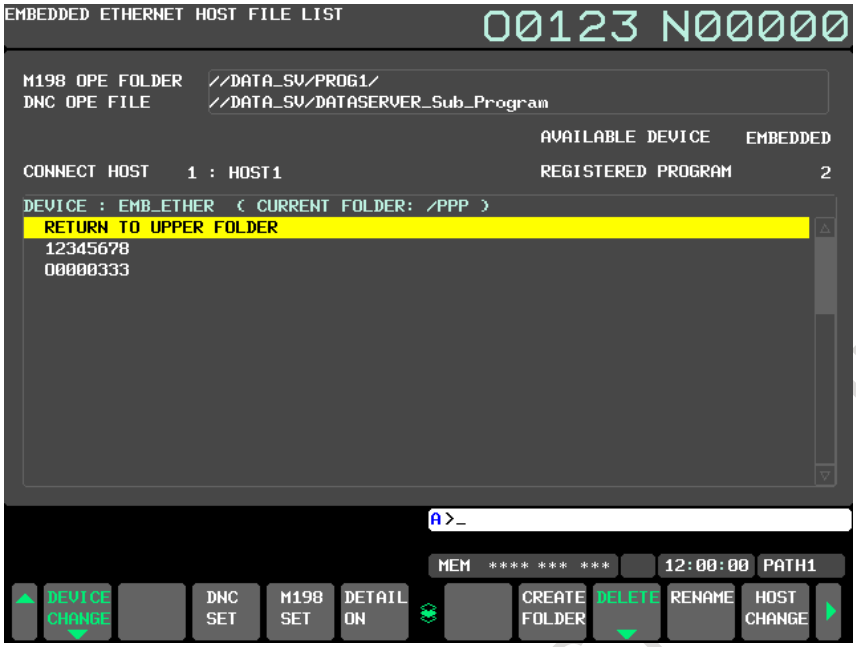


Fig. 8.4.1 (d) Embedded Ethernet host file list screen
(DNC operation from the Embedded Ethernet is valid)

NOTE

1 When using the FTP file transfer function, check that the valid device is the embedded Ethernet port.
The three conditions below determine a connection destination on the host file list screen:
(1) Check that the valid device is the embedded Ethernet port. Make a selection in "DEVICE SELECTION" on the Ethernet setting screen.
(2) Set connection destinations on the FTP transfer screen. The FTP transfer screen is displayed by pressing soft key [FTP TRANS] on the Ethernet setting screen.
(3) A host computer can be selected from connection destinations 1, 2, and 3. Make a selection according to soft key [HOST CHANGE] described later.

2 File names including 2-bytes characters cannot be displayed correctly.

5 When a list of files is larger than one page, the screen display can be switched using the page keys [PAGE ↑] [PAGE ↓].

6 Press the horizontal soft key [REFRESH] to update the screen display.

7 Press the horizontal soft key [DETAIL OFF] to display the host file list (file names only) screen. Press the horizontal soft key [DETAIL ON] to display the host file list (detail) screen.

8 When selecting a folder with cursor and pressing the MDI key [INPUT], the selected folder is changed to the new current folder.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

Display item (common to 8.4/10.4/15/19-inch display units)

M198 OPE FOLDER

Display a folder for M198-based subprogram calling, when external device subprogram call from the embedded Ethernet is valid.

DNC OPE FILE

Display a file name used when DNC operation is performed, when DNC operation from the Embedded Ethernet is valid.

AVAILABLE DEVICE

"EMBEDDED" or "PCMCIA" is displayed.

CONNECT HOST

The currently selected connection host number and host name are displayed.

REGISTERED PROGRAM

The number of files registered in the work folder of the connected host is displayed. Up to 8 digits can be displayed.

CURRENT FOLDER

The current folder name of the connected host is displayed.
If the folder-path is long compared with the display-item, characters: "…" and only the first and last ten letters of the folder name are displayed.

FILE LIST

There is no distinction between file names and folder names.
Although the maximum number of displayed characters is 127, characters are displayed as far as they can be displayed in one line.

Operation list (common to 8.4/10.4/15/19-inch display units)

DNC SET

Specifies the file for which to perform DNC operation, when DNC operation from the Embedded Ethernet is valid.

M198 SET

Specifies the folder for which to perform M198 subprogram calling, when external device subprogram call from the embedded Ethernet is valid.

DETAIL ON, DETAIL OFF

The screen display can be switched between the display of file names only and the display of details.

REFRESH

Display data can be updated.

F INPUT

A file can be input from the host computer to program memory. This item is displayed only when 9 is set as the I/O device number of the CNC.

F OUTPUT

A file can be output from program memory to the host computer. This item is displayed only when 9 is set as the I/O device number of the CNC.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

DELETE
A folder or file can be deleted from the host computer.

RENAME
A file or folder on the host computer can be renamed.

CREATE FOLDER
A folder can be created on the host computer.

HOST CHANGE
The connected host can be changed.

8.4.2 FTP File Transfer Function

The operation of the FTP file transfer function is described below.

NC program input (for 8.4/10.4-inch display unit)

A file (NC program) stored on the host computer can be input into program memory.

> ⚠ **CAUTION**
> If bit 2 (REP) of parameter No.3201 is set to 1, when an NC program having the same file name as an NC program to be input is already present in program memory, the existing NC program is overwritten.

> **NOTE**
> NC programs cannot be input simultaneously with an NC program output.

Procedure

1 Display the EMBEDDED ETHERNET HOST FILE LIST screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the continuous menu key ▷ until soft key [F INPUT] appears.
Press the soft key [F INPUT].
5 Position the cursor on the file to input, and press [F GET] or type the file name to input.
Press the soft key [F SET].
If the input file name is omitted, default input file name "ALL-PROG.TXT" is assumed.
6 Type the program name, and press the soft key [P SET].
For an explanation of the operations to be performed if an input file name [F SET] and an input program name [P SET] are omitted, see the Table 8.4.2 (a).
7 Press the soft key [EXEC].
This starts reading the program, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Table 8.4.2 (a)

[F SET]	[P SET]	Input file name	Input program	Input program name
BLANK	BLANK	ALL-PROG.TXT	All programs in ALL-PROG.TXT	File name at the time the file is saved
BLANK	INPUT	ALL-PROG.TXT	First program in ALL-PROG.TXT	Program name set with [P SET]
INPUT	BLANK	File name set with [F SET]	All programs in the file specified with [F SET]	File name at the time the file is saved
INPUT	INPUT	File name set with [F SET]	First program in the file specified with [F SET]	Program name set with [P SET]

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NC program input (for 15/19-inch display unit)

A file (NC program) stored on the host computer can be input into program memory.

> ⚠ **CAUTION**
> If bit 2 (REP) of parameter No.3201 is set to 1, when an NC program having the same file name as an NC program to be input is already present in program memory, the existing NC program is overwritten.

> **NOTE**
> NC programs cannot be input simultaneously with an NC program output.

Procedure

1 Display the EMBEDDED ETHERNET HOST FILE LIST screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the continuous menu key ▷ until horizontal soft key [F INPUT] appears.
Press the horizontal soft key [F INPUT].
4 Position the cursor on the file to input, and press [F GET] or type the file name to input.
Press the horizontal soft key [F SET].
If the input file name is omitted, default input file name "ALL-PROG.TXT" is assumed.
5 Type the program name, and press the horizontal soft key [P SET].
For an explanation of the operations to be performed if an input file name [F SET] and an input program name [P SET] are omitted, see the Table 8.4.2 (b).
6 Press the horizontal soft key [EXEC].
This starts reading the program, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Table 8.4.2 (b)

[F SET]	[P SET]	Input file name	Input program	Input program name
BLANK	BLANK	ALL-PROG.TXT	All programs in ALL-PROG.TXT	File name at the time the file is saved
BLANK	INPUT	ALL-PROG.TXT	First program in ALL-PROG.TXT	Program name set with [P SET]
INPUT	BLANK	File name set with [F SET]	All programs in the file specified with [F SET]	File name at the time the file is saved
INPUT	INPUT	File name set with [F SET]	First program in the file specified with [F SET]	Program name set with [P SET]

NC program output (for 8.4/10.4-inch display unit)

A file (NC program) stored in program memory can be output to the host computer.

> **NOTE**
> NC programs cannot be output simultaneously with an NC program input.

Procedure

1 Display the EMBEDDED ETHERNET HOST FILE LIST screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the continuous menu key ▷ until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
5 Type the program name to output, and press the soft key [P SET]. To specify an output file name, type the output file name and press the soft key [F SET].

FRYER MACHINE SYSTEMS INC
Rob Zarensky

If, in this step, no output file name or no program name is specified, all programs in the foreground folders are output to “ALL-PROG.TXT”.
For an explanation of the operations to be performed if an output file name [F SET] and an output program name [P SET] are omitted, see the Table 8.4.2 (c).
(For an explanation of foreground folders, see Section, “Default Folders”, and for how to change foreground folders, see Section, “SELECTING A DEFAULT FOLDER”)

6 Press the soft key [EXEC].
This starts outputting the program, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output of the program, press the soft key [CAN].

Table 8.4.2 (c)

[F SET]	[P SET]	Output file name	Output program
BLANK	BLANK or (O-9999)	ALL-PROG.TXT	All programs in the foreground folders displayed in the program folder
BLANK	INPUT	Program name set with [P SET]	Program in the NC that is set with [P SET]
INPUT	BLANK or (O-9999)	File name set with [F SET]	All programs in the foreground folders displayed in the program folder
INPUT	INPUT	File name set with [F SET]	Program in the NC that is set with [P SET]

NC program output (for 15/19-inch display unit)

A file (NC program) stored in program memory can be output to the host computer.

NOTE
NC programs cannot be output simultaneously with an NC program input.

Procedure

1 Display the EMBEDDED ETHERNET HOST FILE LIST screen.
2 Press the EDIT switch on the machine operator’s panel.
3 Press the continuous menu key ▷ until horizontal soft key [F OUTPUT] appears.
Press the horizontal soft key [F OUTPUT].
4 Type the program name to output, and press the horizontal soft key [P SET]. To specify an output file name, type the output file name and press the horizontal soft key [F SET].
If, in this step, no output file name or no program name is specified, all programs in the foreground folders are output to “ALL-PROG.TXT”.
For an explanation of the operations to be performed if an output file name [F SET] and an output program name [P SET] are omitted, see the Table 8.4.2 (d).
(For an explanation of foreground folders, see Section, “Default Folders”, and for how to change foreground folders, see Section, “SELECTING A DEFAULT FOLDER”)
5 Press the horizontal soft key [EXEC].
This starts outputting the program, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Table 8.4.2 (d)

[F SET]	[P SET]	Output file name	Output program
BLANK	BLANK or (O-9999)	ALL-PROG.TXT	All programs in the foreground folders displayed in the program folder
BLANK	INPUT	Program name set with [P SET]	Program in the NC that is set with [P SET]
INPUT	BLANK or (O-9999)	File name set with [F SET]	All programs in the foreground folders displayed in the program folder
INPUT	INPUT	File name set with [F SET]	Program in the NC that is set with [P SET]

FRYER MACHINE SYSTEMS INC
Rob Zahensky

8.4.3 DNC Operation File Set

The operation of the DNC operation file set is described below.

1 Display the embedded Ethernet host file list screen.
2 By using cursor keys, select a file to be used for DNC operation.
3 Press soft key [DNC SET].

8.4.4 M198 Folder Set

The operation of the folder for M198-based subprogram calling set is described below.

1 Move to the folder containing a file to be called by M198-based subprogram calling.
2 Press soft key [M198 SET].
3 The specified folder

8.5 SCREEN HARD COPY FUNCTION

Overview

This function converts screen information displayed on the CNC into bit map format data and output it to a memory card or USB memory. Once output, bit map format data can be displayed and edited on a personal computer. This function can be enabled by setting bit 7 (HDC) of parameter No. 3301.

Explanation

- Start/cancellation methods

The screen hard copy function is started by pressing and holding down key [SHIFT] for five seconds. The function can be canceled by pressing key [CAN].

- Acquisition and output of screen data

When started, the screen hard copy function starts acquiring screen data. As soon as it has acquired it, the function outputs bit map format data to the memory card inserted into the LCD unit. While screen data is being acquired, the screen remains stationary for a few seconds. Acquired screen data can be output from the memory card screen. Also, while screen data is being output, "OUTPUT" blinks in the status display.

- Screen data file names

Bit map format screen data files created by this function are assigned the names below:

In case of parameter bit 4(HCN) of parameter No.3301 is 0, file name starts with “HDCPY000.BMP” and increments the number to 999 as following.
"HDCPY000.BMP" (Name of a file to be first output to a memory card etc. after power-on)
"HDCPY001.BMP" (Name of a file to be second output to a memory card etc. after power-on)
:
:
"HDCPY999.BMP" (Name of a file to be 1000 output to a memory card etc. after power-on)
If the screen hard copy function is executed after a file with "HDCPY999.BMP" is output, the file name returns to "HDCPY000.BMP". However, if a file with the same file name as the one to be output when the screen hard copy function is executed exists on the memory card, alarm SR1973, “FILE ALREADY EXIST” is issued. If the capacity of the memory card is exceeded, alarm SR1962, “CARD FULL (MEMORY CARD)” is issued.
In either case, screen data is not output, so that either the existing file needs to be deleted or the memory card needs to be replaced with a new one.

FRYER MACHINE SYSTEMS INC
Ron Zahensky

In case of using a USB memory, if the name of a file to be output during execution of screen hard copy already exists in the memory, the file is overwritten. If the capacity of the USB memory is exceeded, alarm SR1932, “DEVICE FULL(USB MEMORY)” is issued. In this case, please delete unnecessary files or use a new USB memory.

In case of parameter HCN is 1, file name id defined by the date and the execute number of screen hard copy function on that day as following.
"YYMMDD00.BMP" (Name of a file to be first output to a memory card etc. on that day)
"YYMMDD01.BMP" (Name of a file to be second output to a memory card etc. on that day)
:
:
"YYMMDD99.BMP" (Name of a file to be 100 output to a memory card etc. on that day)
(YY(00~99) means the year, MM(01~12) means the month and DD(01~31) means the day)
(Example)
If screen hard copy is executed on 2013/04/17 in the first time, bit map format file is created with the name of “13041700.BMP”.
If the screen hard copy function is executed after a file with “YYMMDD99.BMP” is output, the file name returns to "YYMMDD00.BMP". However, if a file with the same file name as the one to be output when the screen hard copy function is executed exists on the memory card, system searches number which does not exist and output the file with that number. If all number files (00~99) exist on the memory card, alarm SR1973, “FILE ALREADY EXIST” is issued. If the capacity of the memory card is exceeded, alarm SR1962, “CARD FULL (MEMORY CARD)” is issued.
In either case, screen data is not output, so that either the existing file needs to be deleted or the memory card needs to be replaced with a new one.
In case of using a USB memory, if the name of a file to be output during execution of screen hard copy already exists in the memory, system searches the file number which does not exist on the memory and output the file with that number, same as memory card. If all number files (00~99) exist in the memory, the file is overwritten. If the capacity of the USB memory is exceeded, alarm SR1932, “DEVICE FULL(USB MEMORY)” is issued. In this case, please delete unnecessary files or use a new USB memory.

Limitation

- Screens whose hard copies cannot be made

Hard copies of the BOOT screen, the IPL screen, and the system alarm screen cannot be made.

- Foreground I/O devices

During DNC operation, for example, screen data cannot be output while a foreground I/O device is in use.

- Canceling the hard copy function

If the hard copy function is canceled before a hard copy is completed, an incomplete bit map file of data that has been output is created.

- CNC screen display function

If CNC screen display function is working, the screen hard copy function is disabled. In this case, bit map format data can be made by screen copy function of CNC screen display function. Please refer to the operator’s manual of CNC screen display function.

- Output folder

The screen data is output to the root folder of the memory card or the USB memory.

FLYER MACHINE SYSTEMS INC
Rob Zaleski

8.6 USB FUNCTION

Overview

Setting NC parameter No. 20 to 17 selects a USB memory as the data input/output destination.
Basic specifications of the USB function are listed below.

- The USB function is conforming to USB Ver. 1.0.
- The USB function supports the FAT12, FAT16, FAT32, and VFAT file systems.
- It can handle up to six folder levels under the root folder.
- Up to 32 ASCII code characters (alphanumeric characters) are available for a file or folder name.

⚠ WARNING
A commercially available USB memory does not have sufficient reliability in the FA environment. When a part program, a parameter, etc. are input or output to a USB memory, those data might be damaged and there is a possibility that a serious accident or loss occurs.
Therefore, set bit 0 (ISU) of parameter No.11505 to input or output data to a USB memory using ISO codes unless data is input using ASCII codes.

⚠ CAUTION
Do not turn off CNC or do not remove the USB memory during accessing the USB memory. The USB memory may be damaged.

NOTE
1 There is a USB memory not designed for the long preservation of data. Therefore, do not use a USB memory by the long preservation purpose of data, and back up important data to the personal computer.
2 When empty capacity of the USB memory decreases or the number of files in the USB memory increases, it might take time to the input or output operation. In this case, input or output after deleting an unnecessary file in the USB memory.
3 Any USB memory that is not physically formatted cannot be formatted on the CNC. Format a USB memory to be used for the first time with FAT or FAT32 without specifying the Quick Format option on the personal computer.

Restrictions

There are the following restrictions on the USB function.

- Use a USB memory conforming to USB Ver. 1.0.

NOTE
The USB memory that has the capacity less than a hundred GB on the market can be used. However, the operation of all USB memories on the market cannot be guaranteed. There is a USB memory with slow access speed, and even if the USB memories of the same model that are made by the same manufacturer are used, the operation might be different.
Before using a USB memory, fully check its operation.

- Only one USB memory can be recognized. (No USB hub can be recognized.)
- The maximum number of files and folders in a folder is 512.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- A non-ASCII character (alphanumeric character) is not used in a file or folder name.
- DNC operation cannot be performed using a program in a USB memory.
- Schedule operation cannot be performed using a program in a USB memory.
- A program in a USB memory device cannot be called using an external subprogram call (M198).
- "Memory Card Program Operation/Editing" cannot be applied to a USB memory.

8.7 NC DATA OUTPUT FUNCTION

Overview

This function outputs the following various data into memory card or USB memory by one operation.
- SRAM data
- User files (made by user such as PMC ladder)
- Text data (Parameter, program, and so on)

This function enables to back up the system data easily.
Moreover, the contents of SRAM data can be confirmed immediately because both SRAM data and text data are output.
NC data output signal ALLO is set to “1” from the beginning to the end of outputting text data.

NOTE
1 When USB memory is used, set root folder as data output destination.
2 It takes a time to output data.
3 The output file name is a default file name. If a file with the same name already exists in the external I/O device, the file is overwritten. It is recommended to clear the external I/O device before using this function.

8.7.1 All Data Output Screen

This function outputs on all data output screen of ALL IO.
To display all data output screen, set bit 0 (BOP) of parameter No.313 to 1.

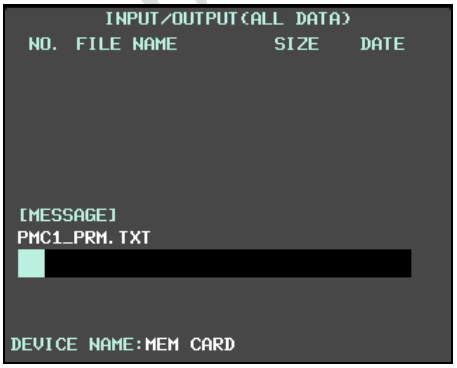


Fig 8.7.1 All data output screen

Displaying all data output screen

Procedure

1 Press the soft key [ALL DATA] on the ALL IO screen.

Explanation

The following items are displayed on all data output screen
- File information of external I/O device
  File number, name, size (byte) and creation date are displayed.

FRYER MACHINE SYSTEMS INC
b Zahenisky

- Message
  Confirmation message and progress bar are displayed.
- Device name
  The name of I/O device (memory card or USB memory) selected by parameter (No.20) is displayed.
  If the other devices are selected, “ILLEGAL” is displayed.

8.7.1.1 Output operation

For output operation of this function, cancel emergency stop state and switch to EDIT mode.
And, set the following parameters.

- Parameter (No.20) = 4 or 17 Memory card: 4, USB memory: 17

In multi-path system, set the following parameter.

- Bit 0 (MDP) of parameter No.138 = 1 A path number is added to the extensions of input/output files.

Moreover, when bit 0 (IO4) of parameter No.110 is 1 (separate control of I/O channel numbers is performed), set the following parameters.

- Parameter (No.20) = Parameter (No.21) and Parameter (No.22) = Parameter (No.23)

Output operation of NC data output function

Procedure

1 Display all data output screen.
2 Press the soft key [(OPRT)].
3 Press the soft key [F OUTPUT].
The message “THIS OPERATION NEEDS POWER OFF. AND SAME NAME FILE IS OVERWRITTEN. EXECUTE?” is displayed in message area. To cancel the output, press the soft key [CAN].
4 Press the soft key [EXEC].
This starts outputting text data and NC data output signal ALLO is set to “1”.
The key operation is invalid while outputting data. (Example: If the function key PROG is pressed, it is not switched to the program screen.)
The progress bar shows the progress report of output.

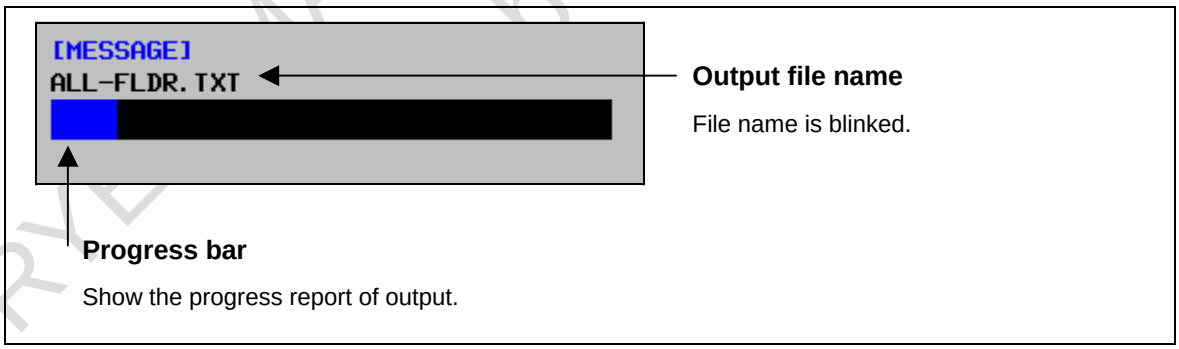


Fig8.7.1.1 (a) Progress bar display

5 When output of all text data is finished, the message” POWER MUST BE OFF. TURN ON THE POWER AGAIN TO SAVE SRAM AND USER FILE.” is displayed and NC data output signal ALLO is set to “0”.

6 Turn off the power then turn on the power again with external I/O device mounted.
When a display is PANELi, PC, or a display unit with Windows CE, operate either of the following operation.
- Set "the display of the BOOT/IPL screen during the next startup of the CNC" before turning off the power. (For HSSB connection)
- Set the rotary switch to "F" after turning off the power.

FRYE MACHINE SYSTEMS INC
Joseph Zahensky

- Turn on while still pressing MDI key 6 and 7 after turning off the power. (For a display unit with Windows CE)

Refer to "APPENDIX Option Setting Screen" in MAINTENACE MANUAL (B-64695EN) for "the display of the BOOT/IPL screen during the next startup of the CNC".
Refer to "APPENDIX CHANGING START SEQUENCES" in MAINTENACE MANUAL for the rotary switch on stand-alone type CNC.

7 At power-up, SRAM data and user files are output. The progress report of output is displayed while outputting.

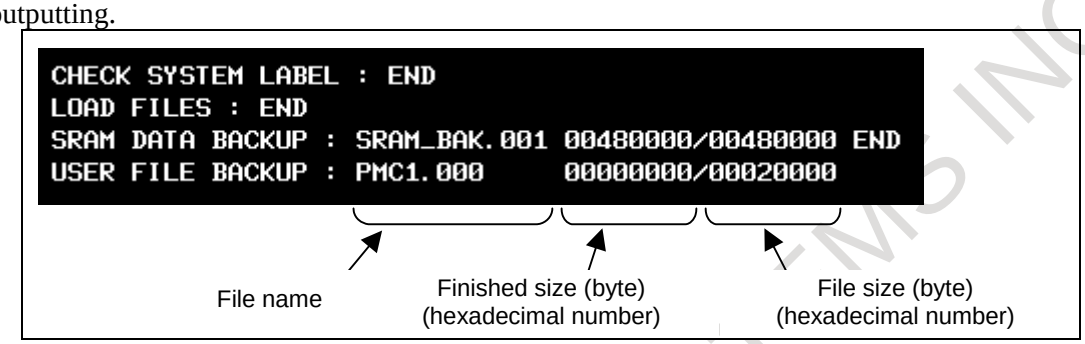


Fig8.7.1.1 (b) Output of SRAM data and user files

8 After finishing output of SRAM data and user files, CNC starts up.

8.7.1.2 Format

Memory card and USB memory can be formatted.
When format is executed, all data in memory card or USB memory is cleared.

> **NOTE**
> It is recommended to format memory card and USB memory before executing output operation of NC data output function.

Format operation

Procedure

1 Display all data output screen.
2 Press the soft key [(OPRT)].
3 Press the soft key [FORMAT].
The message “FORMAT OK?” is displayed in message area.
To cancel the format, press the soft key [CAN].
4 Press the soft key [EXEC].
The message “FORMAT EXECUTING” is displayed in message area.
5 After finishing format, the message “COMPLETE” is displayed.

8.7.2 Output Directory

8.7.2.1 LCD mounted type CNC

The output files by this function are output to the current directory.

8.7.2.2 Stand-alone type CNC (without personal computer function)

The output files by this function are output to the current directory.

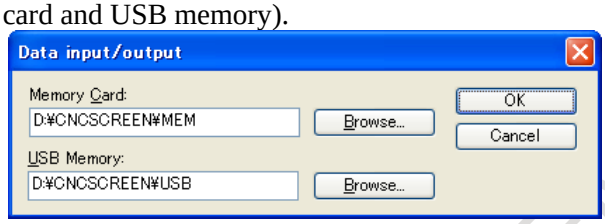
FRYER MACHINE SYSTEMS INC
Rob Zahensky

8.7.2.3 Stand-alone type CNC (with personal computer function / HSSB connection)

Set the following output directory before using this function.
The I/O directory is decided as follows:

Text data

1. The case that bit 0 (PCM) of parameter No.300 is set to 1
   Text data is output to the I/O directory specified by CNC screen display function (PC side) (Common to memory card and USB memory).

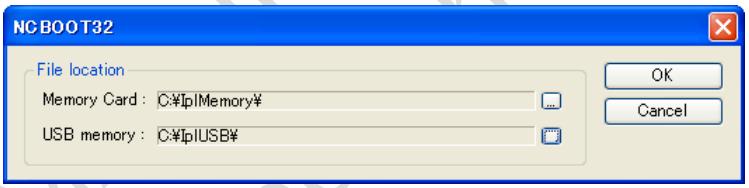


   Setting procedure)
   Start up CNC screen display function and select "data I/O" in toolbar "file".

2. The case that bit 0 (PCM) of parameter No.300 is set to 0
   Memory card: Text data is output to a memory card on the stand-alone type unit (NC side).
   USB memory: Text data is output to the I/O directory specified by CNC screen function (PC side).

SRAM data and User files

SRAM data and user files are output to the I/O directory specified by NCBOOT (PC side) (Common to memory card and USB memory).



(Setting procedure)
Select a "Setting" in the pop-up menu in IPL monitor.
Refer to "APPENDIX F. MAINTENACE OF PERSONAL COMPUTER FUNCTIONS (BOOT-UP AND IPL)" in MAINTENACE MANUAL (B-64695EN) for displaying IPL monitor.

Supported edition

This application is supported in the following editions.
- CNC screen display function (HSSB version): Edition 4.2 or later
- NCBOOT (HSSB version): Edition 4.6 or later

This function cannot be operated normally if one of these applications is not supported.

NOTE
1 SRAM data and user files are output to the I/O directory specified by NCBOOT (PC side) regardless of bit 0 (PCM) of parameter No.300.
2 When the I/O directory specified by CNC screen display function (PC side) is different from that specified by NCBOOT (PC side), the output directories of "Text data" and "SRAM data and User files" differ from each other.

FRYER MACHINE SYSTEMS INC Robert Chensky

8.7.2.4 Stand-alone type CNC (with personal computer function / Ethernet connection)

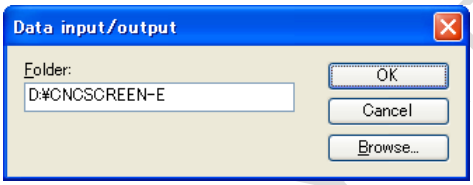
NOTE
USB function cannot be used on stand-alone type CNC with personal computer function (Ethernet connection).

Set the following output directory before using this function.
The I/O directory is decided as follows:

Text data

Memory card:

- Bit 0 (PCM) of parameter No.300 is set to 1
  Text data is output to the I/O directory specified by CNC screen display function (PC side).



Setting procedure)
Start up CNC screen display function and select "data I/O" in toolbar "file".

- Bit 0 (PCM) of parameter No.300 is set to 0
  Text data is output to a memory card on the stand-alone type unit (NC side).

USB memory:
Not available.

SRAM data and User files

Memory card:
SRAM data and user files are output to a memory card on the stand-alone type unit (NC side).
The I/O directory specified by NCBOOTE is ignored.

USB memory:
Not available.

NOTE
SRAM data and user files are output to a memory card on the stand-alone type unit (NC side) regardless of bit 0 (PCM) of parameter No.300.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

8.7.3 List of Output Data

8.7.3.1 Text data

Text data shown in Table 8.7.3.1 (a) are output.
In case of data for each path, text data for all paths are output.

Table 8.7.3.1 (a) Text data list

Data kind	File name	Path	Optional function
Tool offset	TOOLOFST.TXT	○	
Parameter	CNC_PARA.TXT		
PMC parameter	PMCn_PRM.TXT (n: path number)		
PMC parameter (Dual Check Safety)	PMCS_PRM.TXT		○
Program	ALL-FLDR.TXT		
Pitch error compensation data	PITCH.TXT		○
Custom macro common variable	MACRO.TXT	○	○
Workpiece coordinate system data	EXT_WKZ.TXT	○	○
M code group data	MC-GROUP.TXT	○	○
Operation history data	OPRT_HIS.TXT		
Maintenance information	MAINTINF.TXT		
Periodic maintenance data	MAINTENA.TXT		
Second geometry tool offset	SEC_GEOM.TXT	○	○
Real time custom macro variable	RTMMACRO.TXT	○	○
System configuration data	SYS-CONF.TXT		
PMC signal protect	DIDOENBL.TXT	○	○
Attachment history data (tool management tool attachment/detachment function)	TLATAHIS.TXT		○
Detachment history data (tool management tool attachment/detachment function)	TLDTAHIS.TXT		○
Servo/spindle information	SV_SP_ID.TXT		
Tool geometry data (Interference check for each path)	TOOL-FRM.TXT	○	○
Machine system name data	MAINTEMC.TXT		
P code variable	PCODE.TXT	○	○
Tool geometry size data	TOOLSIZE.TXT	○	○
CNC ID information	CNCIDNUM.TXT		
Operation history signal section data	OHIS_SIG.TXT		
System alarm information	SYS_ALM.TXT		
All tool data (Tool management function)	AL_TLMGR.TXT	○	○
All custom data (Tool management function)	AL_TLCSM.TXT		○
Tool life management data	TOOLLIFE.TXT	○	○
Machine state history data	MSRHSTRY.TXT		○
Network setting information	NETWORK.MEM		

NOTE
1 The kinds of output text data are depended on the option setting.
2 The text data which is output by this function can be input on the screen of each function. Program file “ALL-FLDR.TXT” can be input by using the operation of inputting all programs and folders on the ALL IO screen.

In case of data for each path, the extension shown in Table 8.7.3.1 (b) is added.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Table 8.7.3.1 (b) Additional extension to path data

Path number	Extension
1	TXT
2	P-2

8.7.3.2 SRAM data and User files

SRAM data and all data in FROM are output by this function.
The examples of the output file are as follows.

Table 8.7.3.2 User file list

File kind	File name	Remarks
Option information	OPRM_INF.000	
PMC sequence program	PMCn.000	n: Path number
PMC sequence program for Dual Check Safety	PMCS.000	
PMC multi-language message	MnPMCMSG	n: Path number
C-language executer application data	CEX_nnM.000	nn: Either 05, 10, 15, 20, 25, 30, 35, 40, 50, 60, 70, 80, 90 or A0
C-language executer user data file	CEXnxxxx.000	n: numeric character x: alphanumeric character
Macro executer application data	PDnnxxxx.000	n: P-CODE number xxxx: Data size
FANUC PICTURE application data	FPF_20M.000	
FANUC PICTURE screen data	FPF0xx.000	xx: Data size (unit of 128KB, decimal number)
Machine operation menu data	EXSFTKEY.000	
I/O configuration	IOCONF.000	
Machine alarm diagnosis data	TDG-DATA.000	

NOTE
The kinds of output user files are depended on the option setting.

8.7.4 List of Message

8.7.4.1 Warning message

The following message is displayed, when an error occurs by an operation on all data output screen.

Table 8.7.4.1 Warning message list

Warning message	Description
MEMORY CARD IS NOT READY.	The memory card is not mounted. Mount the memory card.
USB MEMORY IS NOT MOUNTING	The USB memory is not mounted. Mount the USB memory.
THE MACHINE IS UNDER USE	The I/O device is used by other functions.
CHANGE TO EDIT MODE	Select the EDIT mode.
PLEASE RELEASE EMERGENCY STOP	Cancel emergency stop state for all paths.
PARAMETER SETTING ERROR	The I/O device could not be accessed. Set the following parameters. Parameter (No.0020) = 4 or 17 Parameter IO4(No.110#0) = 0 Parameter MDP(No.138#0) = 1
OUTPUT ERROR(PMC PARAM	The error occurred while outputting PMC parameter.
FORMAT ERROR	The format cannot be executed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

8.7.4.2 Error message

If the error occurred while outputting SRAM data or user files, the following error message is displayed on IPL screen.

Table 8.7.4.2 Error message list

Error message	Description
DEVICE NOT READY	The I/O device is not ready.
DEVICE FULL	The capacity of the I/O device is insufficient.
DEVICE PROTECTED	The I/O device is write–protected.
DEVICE NOT MOUNTED	The I/O device could not be mounted.
FILE PROTECTED	The file is write–protected.
CANCELED	The output operation is canceled.
DIRECTORY FULL	The file could not be generated in the root directory for the I/O device. Reduce the number of files in root directory.

8.8 INPUT/OUTPUT FILE NAME FOR MULTI-PATH SYSTEM

This function is used to add the path number to the extension of the input/output file.
This function can manage the files easily for each path.
This function is available when bit 0 (MDP) of parameter No.138 is set to 1.

> **NOTE**
> This function is available only when input/output is executed by not specifying the file name but using default file name.
> If the file name is specified, this function is unavailable. In that case, the specified file name is used for input/output and the path number is not added.

Explanation

- About extension

When bit 0 (MDP) of parameter No.138 is set to 1, the path number is added to the extension of the default file name.
The extension of each path is as follows.

Table 8.8 (a) Extension of default file name

Path number	Extension
1	TXT
2	P-2

> **NOTE**
> The extension of the loader path 1 is “PL1” and the loader path 2 is ”PL2”.

Example) In case of offset data, the default file name is as follows.

Table 8.8 (b) Input/output filename of offset data

Path number	Default file name	
	Parameter MDP=0	Parameter MDP=1
1	TOOLOFST.TXT	TOOLOFST.TXT
2	TOOLOFST.TXT	TOOLOFST.P-2

FRYER MACHINE SYSTEMS INC
Rob Zalensky

- Target data

This function is available only for path individual data. The path number is not added for path common data by this function.

- Part program

This function is available for the part program.
The extension of each path is as follows.

Table 8.8 (c) Extension of part program

Path number	Extension
1	Nothing
2	P-2

NOTE
The extension of the loader path 1 is “PL1” and the loader path 2 is ”PL2”.

Example) When the part program “O0001” is input/output, the file name is as follows.

Table 8.8 (d) Input/output file name of part program

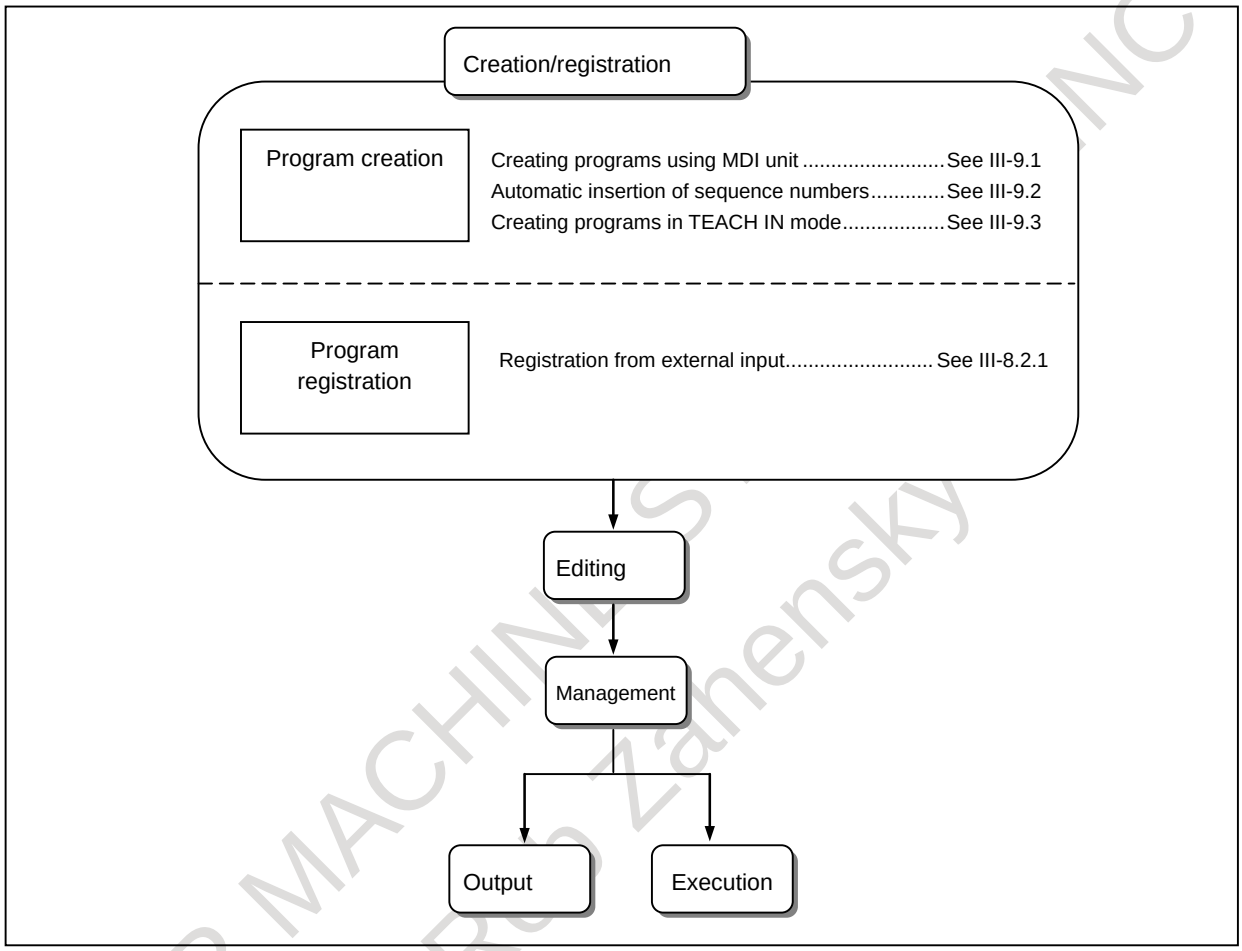
Path number	File name	
	Parameter MDP=0	Parameter MDP=1
1	O0001	O0001
2	O0001	O0001.P-2

NOTE
If the file name is specified, the path number is not added.
For example, when “O0001” is output by using [F SET] on second path, the output file name is “O0001”. It is not “O0001.P-2”.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

9 CREATING PROGRAMS

This chapter explains how to create programs by MDI of the CNC.
This chapter also explains automatic insertion of sequence numbers and how to create programs in TEACH IN mode.



9.1 CREATING PROGRAMS USING THE MDI UNIT

Programs can be created in the EDIT mode using functions described in Section, ”EDITING PROGRAMS”.

Procedure for Creating Programs Using the MDI Unit

1 Enter the EDIT mode.
2 Press the PROG key.
3 Press address key P 0 and enter the program number.
4 Press the editing key INSERT or INPUT.
5 Create a program using the program editing functions described in III-10.

Explanation

- Comments in a program

Comments can be written in a program using the control in/out codes.
Example) O0001 (TEST PROGRAM) ;
M08 (COOLANT ON) ;

- When the editing key INSERT or INPUT is pressed after the control-out code "(", comments, and control-in code ")" have been typed, the typed comments are registered.
- When the editing key INSERT or INPUT is pressed midway through comments, to enter the rest of comments later, the data typed before the editing key INSERT or INPUT is pressed may not be correctly registered (not entered, modified, or lost) because the data is subject to an entry check which is performed in normal editing.
- Control-out code "(" or Control-in code ")" cannot be registered by itself.

9.2 AUTOMATIC INSERTION OF SEQUENCE NUMBERS

Sequence numbers can be automatically inserted in each block when a program is created using the MDI keys in the EDIT mode.
Set the increment for sequence numbers in parameter No. 3216.

Procedure for automatic insertion of sequence numbers

Procedure

1 Enter 1 in response to the prompt for "SEQUENCE NO." in setting data. (see Subsection, “Displaying and Entering Setting Data”).

2 Enter the EDIT mode.

3 Press PROG key to display the program screen.

4 Search for or register the number of a program to be edited and move the cursor to the EOB (;) of the block after which automatic insertion of sequence numbers is started.
When a program number is registered and an EOB (;) is entered with the editing key INSERT or INPUT, sequence numbers are automatically inserted starting with 0. Change the initial value, if required, according to step 10, then skip to step 7.

5 Press address key N and enter the initial value of N.

6 Press the editing key INSERT or INPUT.

7 Enter each word of a block.

8 Press EOB key.

9 Press the editing key INSERT or INPUT. The EOB is registered in memory and sequence numbers are automatically inserted. For example, if the initial value of N is 10 and the parameter for the increment is set to 2, N12 inserted and displayed below the line where a new block is specified.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

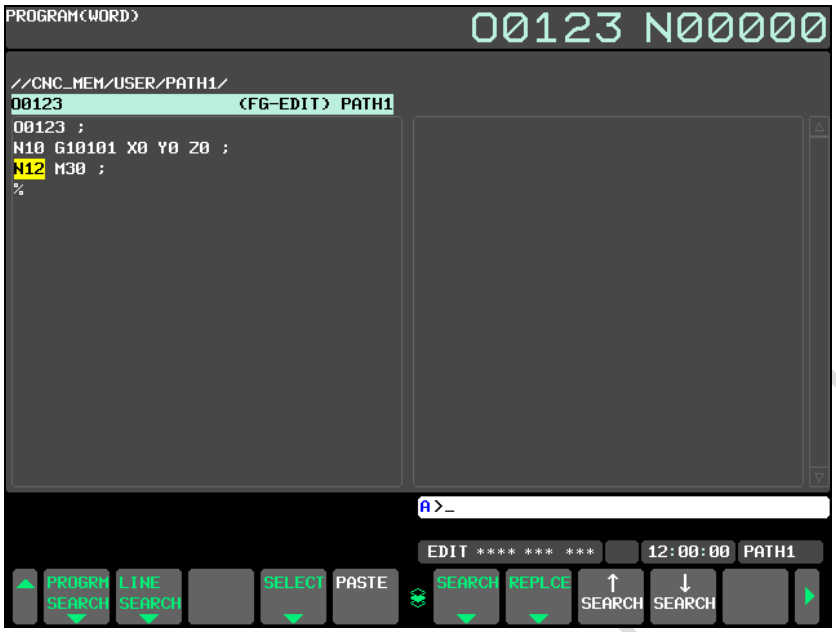


Fig. 9.2

10 In the example above, if N12 is not necessary in the next block, pressing the DELETE key after N12 is displayed deletes N12.
If wishing to insert N100, not N12, into the next block, type N100 immediately after N12 is displayed, and press CALC ALTER. This causes N100 to be registered and the initial value to be changed to 100.

9.3 CREATING PROGRAMS IN TEACH IN MODE (PLAYBACK)

In the TEACH IN JOG or TEACH IN HANDLE mode, you can create a program while inserting the coordinate of the current position along each axis in the absolute coordinate system when the tool is moved by manual operation into the program.
You can input the words other than axis names in the same way as in the EDIT mode.

Program screen in the TEACH IN JOG mode

Displayed items

In the TEACH IN JOG or TEACH IN HANDLE MODE, the following program screen is displayed.
On the left of the screen, the coordinates of the current position in the absolute and relative coordinate systems are displayed; on the right of the screen, the contents of a program are displayed. You can create a program while checking the current position by manual operation.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Fig. 9.3 (a) Program screen in the TEACH IN JOG mode

Inputting the coordinates of the current position

You can use the following procedure to insert the coordinate of the current position along each axis in the absolute coordinate system:

1 Select the TEACH IN JOG mode or TEACH IN HANDLE mode.

2 Press PROG key to display the program screen. Search for or register the number of a program to be edited and move the cursor to the position where the current position along each axis is to be inserted.

3 Move the tool to the desired position with jog or handle.

4 Key in the axis name of an axis along which you want to insert the coordinate of the current position.

5 Press the editing key INSERT or INPUT. Then, the coordinate of the current position along the specified axis is inserted in the program.
(Example) X10.521 Coordinate of the current position
X10521 Data inserted in the program

Example

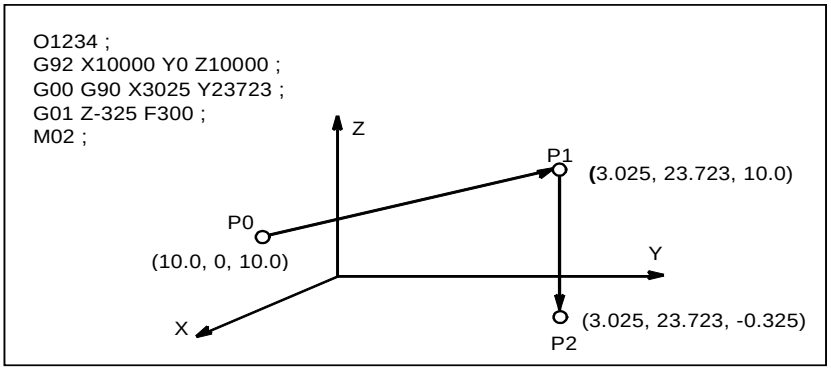


Fig. 9.3 (b)

1 Select the TEACH IN HANDLE mode.

2 Make positioning at position P0 by the manual pulse generator.

FRYER MACHINE SYSTEMS INC
Rob Zaherlsky

3 Select the program screen.

4 Enter program number O1234 as follows:

[O] [1] [2] [3] [4] [INSERT]

This operation input program number O1234 in memory.
Next, press the following keys:

[EOB] [INSERT]

An EOB (;) is entered after program number O1234.

5 Enter the P0 machine position for data of the first block as follows:

[G] [9] [2] [INSERT] [X] [INSERT] [Y] [INSERT] [Z] [INSERT] [EOB] [INSERT]

This operation registers G92X10000Y0Z10000; in program.

6 Position the tool at P1 with the manual pulse generator.

7 Enter the P1 machine position for data of the second block as follows:

[G] [0] [0] [G] [9] [0] [INSERT] [X] [INSERT] [Y] [INSERT] [EOB] [INSERT]

This operation input G00G90X3025Z23723; in program.

8 Position the tool at P2 with the manual pulse generator.

9 Enter the P2 machine position for data of the third block as follows:

[G] [0] [1] [INSERT] [Z] [INSERT] [F] [3] [0] [0] [INSERT] [EOB] [INSERT]

This operation input G01Z -325F300; in program.

10 Input M02; in program as follows:

[M] [0] [2] [INSERT] [EOB] [INSERT]

This completes the registration of the sample program.

Explanation

- Registering a position with compensation

When an axis name and a numeric value are keyed in and the editing key [INSERT] or [INPUT] is pressed, the value keyed in is added to the absolute coordinate of the current position and the coordinate is inserted. This operation allows insertion of a corrected value for the absolute coordinate position.
When an extended axis name is used and the name ends with a numeric value, insert an equal sign (=) between the axis name and numeric value.

- Registering commands other than position commands

Commands to be entered before and after a position command must be entered before and after the machine position is registered, by using the same operation as program editing in EDIT mode.

- Calculator-type input

When the calculator-type input format is disabled (bit 0 (DPI) of parameter No. 3401 is set to 0), the coordinate of the current position is inserted into the program in least input increments. When the calculator-type input format is enabled (the bit is set to 1), the coordinate is inserted with a decimal point.

(Example)

Coordinate of the current position	X10.521

At this time, the X-axis coordinate is inserted into the program as follows:

When the calculator-type input format is disabled	X10521
When the calculator-type input format is enabled	X10.521

FRYER MACHINE SYSTEMS INC
Rob Zahensky

9.4 CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION

NC programs can be created on a block-by-block basis, viewing the displayed G code menu screen and G code details screen.

Procedure for Conversational Programming with Graphic Function

Procedure 1: Creating a program

1 Enter the EDIT mode.

2 Press function key [PROG]. If no program is registered, the following screen is displayed.

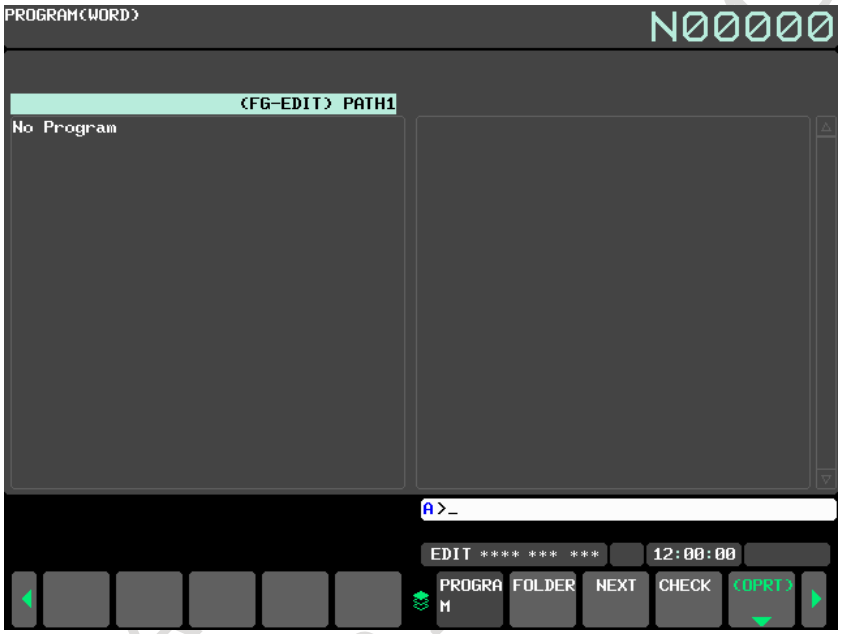


Fig. 9.4 (a) Program screen (with no program registered)

If a program is registered, the program currently selected is displayed.

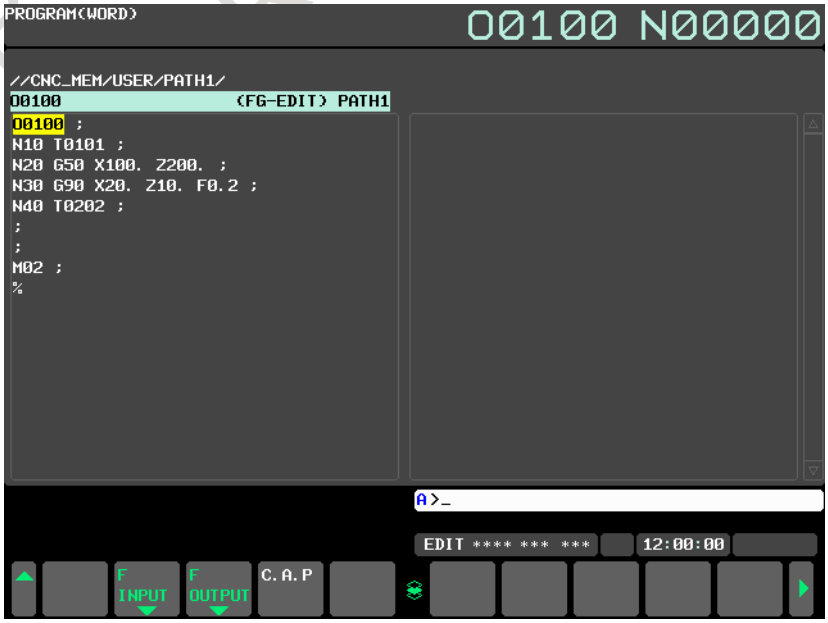


Fig. 9.4 (b) Program screen (with a program registered)

FRYER MA ROD IS INC

3 Key in the program number of a program to be registered after keying in address O, then press [INSERT] key. For example, when a program with program number 10 is to be registered, key in O10 , then press [INSERT] key. This registers a new program O0010.

4 Press the [C.A.P] soft key. The following G code menu is displayed on the screen. (If the [C.A.P] soft key is not displayed, hold down the rightmost continuous menu key [▷] or the leftmost return menu key [◁] until the [C.A.P] soft key is displayed.)

Fig. 9.4 (c) G code menu screen

5 Select the G code corresponding to a function to be programmed by a cursor. When the positioning function is desired, for example, the G code menu lists the function with the G code G00. So select G00. If the screen does not indicate a function to be programmed, press the [PAGE ⇧] or [PAGE ⇩] page key to display the next G code menu screen. Repeat this operation until a desired function appears. If a desired function is not a G code, select “STANDARD FORMAT”.

6 Press the soft key [SELECT] or [INPUT] key to display a detailed screen for a selected G code. The figure below shows an example of detailed screen for G00.

FRYER MACHINE INC Bob Zaine

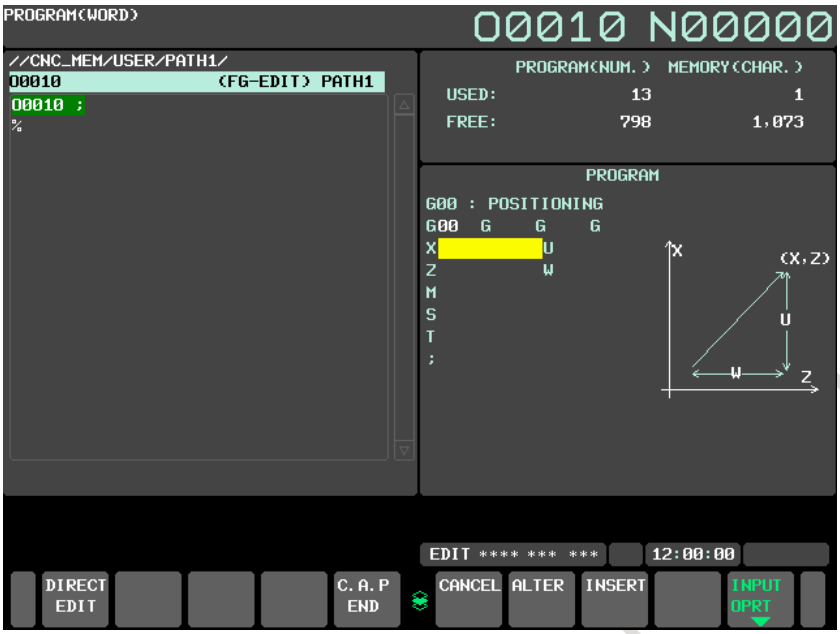


Fig. 9.4 (d) G code details screen (G00)

When “STANDARD FORMAT” is selected, the standard details screen is displayed.

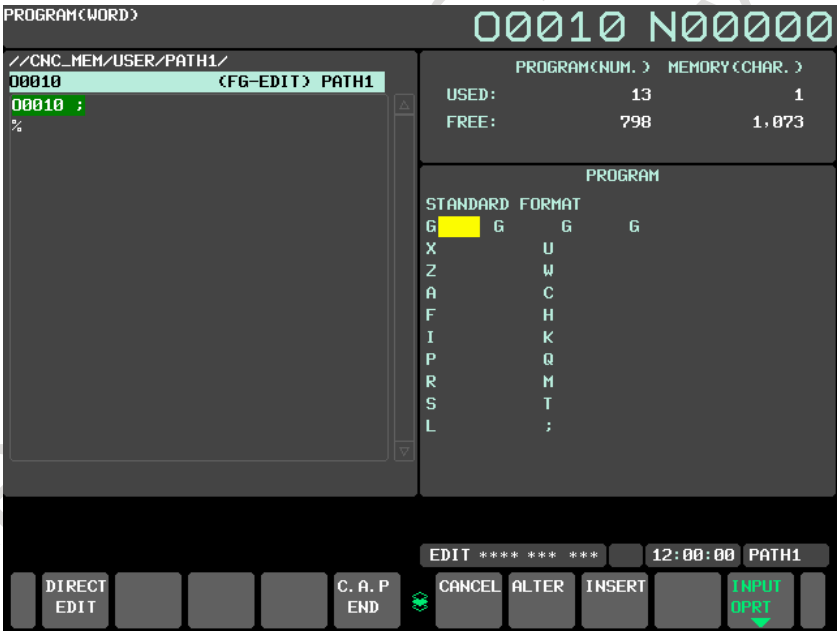


Fig. 9.4 (e) G code details screen (standard details screen)

7 Move the cursor to the block to be modified on the program screen. If the address where the cursor is placed on the program side is present on the figure side, the address on the figure side blinks.

8 Enter numeric data by pressing the numeric keys and press the [INPUT] soft key or INPUT key. This completes the input of one data item.

9 By repeating steps 7 and 8 above, enter all data necessary for the G code keyed in step 5 above.

10 Press the [INSERT] soft key or the INSERT key. This completes the registration of data of one block in program memory. On the screen, the G code menu screen is displayed, allowing the user to enter data for another block. Repeat the procedure starting with 5 as required.

11 After registering all programs, press the [C.A.P. END] soft key. The registered programs are converted to the conversational format and displayed.

12 Press the [RESET] key to return to the program head.

Procedure 2: Modifying a block

1 Move the cursor to the block to be modified on the program screen and press the [C.A.P] soft key. Or, press the [C.A.P] soft key first to display the conversational screen, then press the [PAGE ↑] or [PAGE ↓] page key until the block to be modified is displayed.

2 When data other than a G code is to be altered, just move the cursor to the data and key in a desired value, then press the [INPUT] soft key or [INPUT] key.

3 When a G code is to be altered, press the menu return key [◁] and the soft key [G.MENU]. Then the G code menu appears. Select a desired G code by the cursor, then key in the value. For example, to specify a cutting feed, since the G code menu indicates G01, select G01. Then press the soft key [SELECT] or the [INPUT] key. The detailed screen of the G code is displayed, so enter the data.

4 After data is changed completely, press the [ALTER] soft key or the [CALC ALTER] key. This operation replaces an entire block of a program.

Procedure 3: Inserting a block

1 On the conversational screen, display the block immediately before a new block is to be inserted, by using the [PAGE ↑] or [PAGE ↓] page key. On the program screen, move the cursor with the page keys and cursor keys to immediately before the point where a new block is to be inserted.

2 Press the soft key [G.MENU] to display the G code menu. Then enter new block data.

3 When input of one block of data is completed in step 2, press the [INSERT] soft key or the [INSERT] key. This operation inserts a block of data.

Procedure 4: Deleting a block

1 On the conversational screen, display the contents of a block to be deleted, then press the [DELETE] soft key or the [DELETE] key.

2 The [YES] and [NO] soft keys are displayed. Press the [YES] soft key.

3 The contents of the block displayed are deleted from program memory. Then the contents of the next block are displayed on the conversational screen.

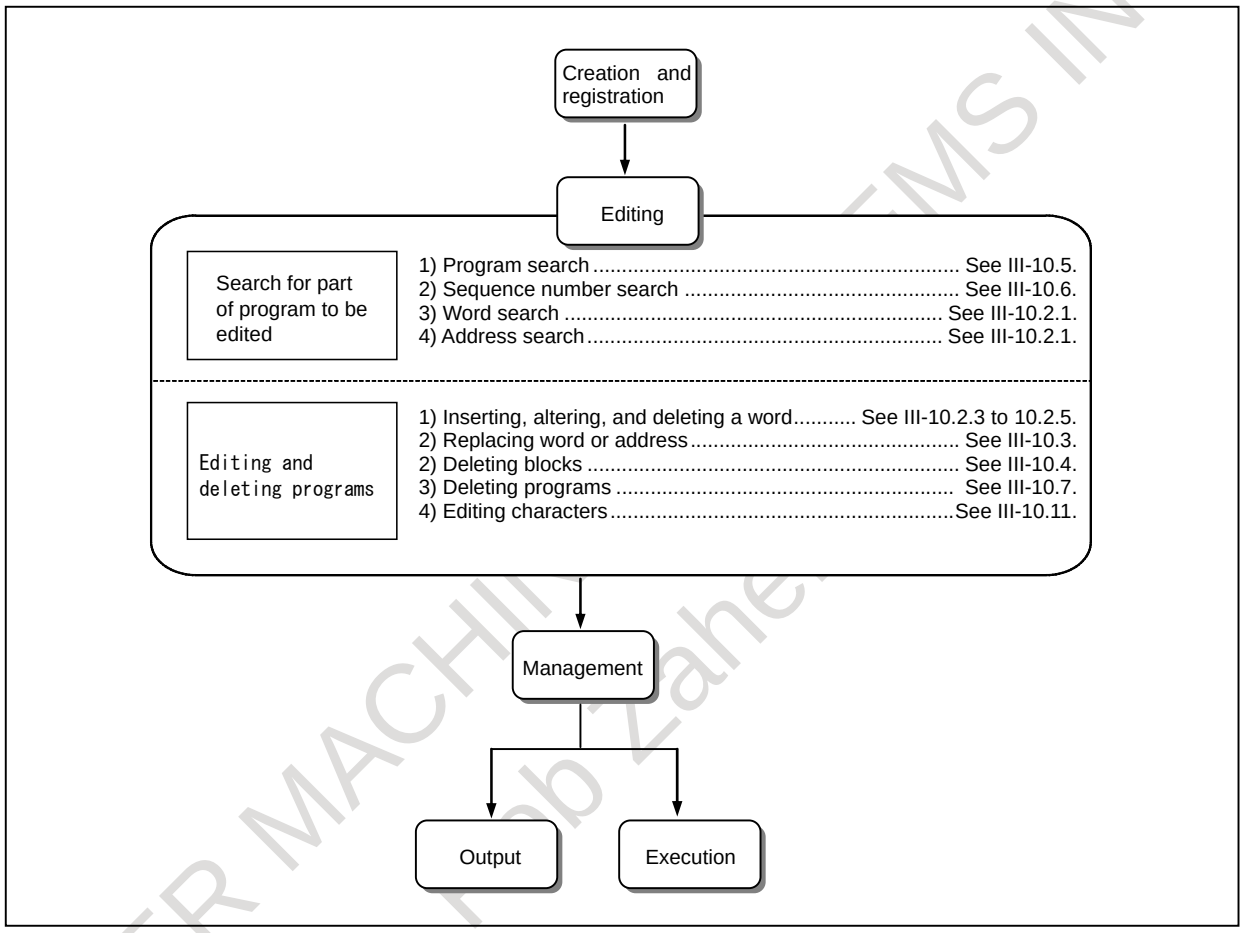
Notes

With a machining center system, the names of the basic three axes are X, Y, and Z at all times.
With a lath system, the names of the basic two axes are X and Z at all times.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

10 EDITING PROGRAMS

This chapter describes how to edit programs registered in the CNC. Editing includes the insertion, modification, and deletion of words. Editing also includes deletion of the entire program and automatic insertion of sequence numbers. In addition, PC-like program text copy and move operations are possible. This chapter also describes program search, sequence number search, word search, and address search, which are performed before editing the program.



FRYER MACHINE SYSTEMS INC
Rob Zaher

10.1 EDIT DISABLE ATTRIBUTE AND DISPLAY DISABLE ATTRIBUTE

Before a program can be edited, the edit disable attribute and display disable attribute must be removed.
The edit disable attribute and display disable attribute can be set for each program and folder.
Programs with the edit disable attribute or display disable attribute, and programs in folders with the edit disable attribute or display disable attribute cannot be edited.

Procedure for removing the edit disable attribute

1 Select EDIT mode.
2 Press the function key PROG.
3 Press the soft key [FOLDER].
4 Press the soft key [(OPRT)].
5 Press the soft key [DETAIL ON].
6 Move the cursor to the program or folder from which you want to remove the edit disable attribute.
7 Press the soft key [CHANGE ATTR].
8 Press the soft key [EDIT ENABLE].
9 Press the soft key [END].

⚠ CAUTION
1 After completing editing, set the edit disable attribute as necessary.
2 To set the edit disable attribute, follow the same procedure as for removing the attribute.
In the step 8, press the soft key [EDIT DISABL].

Procedure for removing the display disable attribute

1 Select MDI mode.
2 Set the same value as the password (parameter No. 3210 (PSW)) in parameter No. 3211 (KEY) to release protection of programs.
3 Select EDIT mode.
4 Press the function key PROG.
5 Press the soft key [FOLDER].
6 Press the soft key [(OPRT)].
7 Press the soft key [DETAIL ON].
8 Move the cursor to the program or folder from which you want to remove the display disable attribute.
9 Press the soft key [CHANGE ATTR].
10 Press the soft key [DISP ENABLE].
11 Press the soft key [END].

⚠ CAUTION
1 After completing editing, set the display disable attribute as necessary.
2 To set the display disable attribute, perform steps 1 to 9 of the procedure for removing the attribute and press soft key [DISP DISABL] in step 10. Then, set parameter No. 3210 (PSW) to a value other than 0 that is not the value set in parameter No. 3211 (KEY). The program or folder is not displayed in any list screen.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

10.2 INSERTING, ALTERING AND DELETING A WORD

This section outlines the procedure for inserting, altering, and deleting a word in a program registered in memory.

Procedure for inserting, altering and deleting a word

1 Select EDIT mode.

2 Press the function key PROG.

3 Select a program to be edited.
If a program to be edited is selected, perform the operation 4.
If a program to be edited is not selected, search for the program number.

4 Move the cursor to the position to be edited, with one of the methods below.
- Scan method
- Word search method
- Address search method

For each method, refer to– Subsection, “Word Search”.

5 Perform an operation such as altering, inserting, or deleting a word.

Explanation

- Concept of word and editing unit

A word is an address followed by a number. With a custom macro, the concept of word is ambiguous.
So the editing unit is considered here.
The editing unit is a unit subject to alteration or deletion in one operation. In one scan operation, the cursor indicates the start of an editing unit.
An insertion is made after an editing unit.
Definition of editing unit
- Program portion from an address to immediately before the next address
- An address is an alphabet, IF, WHILE, GOTO, END, DO=, or ; (EOB).

According to this definition, a word is an editing unit.
The word "word," when used in the description of editing, means an editing unit according to the precise definition.

> ⚠ **CAUTION**
> When a change, insertion, or deletion was performed on data of a program by pausing machining with the single block stop, feed hold, or other operations during execution of a program, be sure to return the cursor to the original position before restarting the program. To execute the program with the cursor positioned at another position, be sure to make a reset.
> Otherwise, the program may not be executed as expected from the program shown on the screen after machining restarts.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

10.2.1 Word Search

A word can be searched for by merely moving the cursor through the text (scanning), by word search, or by address search.

Procedure for scanning a program

1 Press the cursor key [→].
The cursor moves forward word by word; the cursor is displayed at a selected word.

2 Press the cursor key [←].
The cursor moves backward word by word; the cursor is displayed at a selected word.
Example: Scanning Z1250.0

Fig. 10.2.1 (a)

3 Holding down the cursor key [→] or [←] scans words continuously.

4 The first word of the next block is searched for when the cursor key [↓] is pressed.

5 The first word of the previous block is searched for when the cursor key [↑] is pressed.

6 Holding down the cursor key [↓] or [↑] moves the cursor to the head of a block continuously.

7 Pressing the page key [PAGE ↓] displays the next page and searches for the first word of the page.

8 Pressing the page key [↑ PAGE] displays the previous page and searches for the first word of the page.

9 Holding down the page key [PAGE ↓] or [↑ PAGE] displays one page after another.

Procedure for searching a word

Example of searching for S12

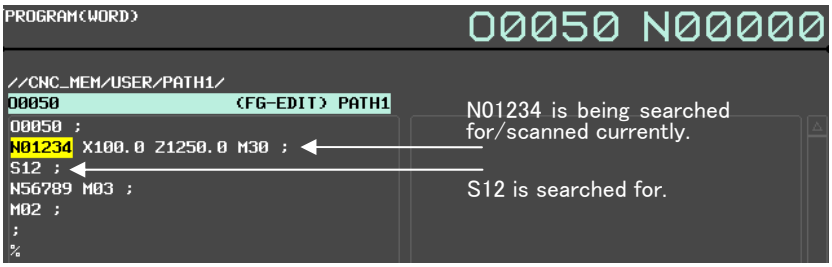


Fig. 10.2.1 (b)

1 Press soft key [SEARCH].

2 Key in address [S].

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

3 Key in [1] [2] .
- S12 cannot be searched for if only S1 is keyed in.
- S09 cannot be searched for by keying in only S9.
  To search for S09, be sure to key in S09.

4 Pressing soft key [↓ SEARCH] starts searching forward from the cursor position. Pressing [↑ SEARCH] starts searching backward.

5 To search for the same word successively, press [↓ SEARCH] or [↑ SEARCH].

Procedure for searching an address

Example of searching for M03

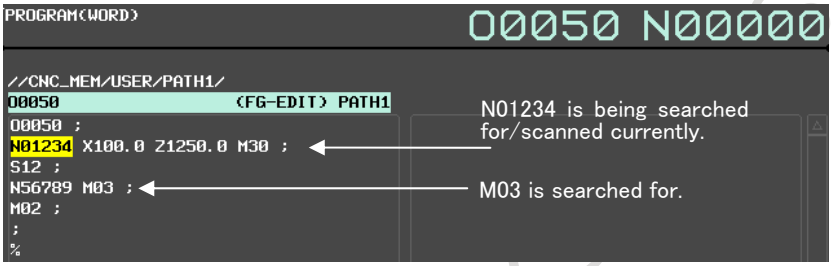


Fig. 10.2.1 (c)

1 Pressing the [SEARCH] key

2 Key in address [M] .

3 Pressing soft key [↓ SEARCH] starts searching forward from the cursor position. Pressing [↑ SEARCH] starts searching backward.

4 To search for the same word successively, press [↓ SEARCH] or [↑ SEARCH].

10.2.2 Heading a Program

The cursor can be jumped to the top of a program. This function is called heading the program pointer. This section describes the four methods for heading the program pointer.

Procedure for heading a program

Method 1

1 Press [RESET] key when the program screen is selected in EDIT mode. When the cursor has returned to the start of the program, the contents of the program are displayed from its start on the screen.

Method 2 Search for the program number.

1 When the program screen is selected in MEMORY or EDIT mode, enter a program number or program name.
  When entering a program number, press address key [O] then type the program number.

2 Press soft key [PROGRM SEARCH].

Method 3

1 Select the program screen or program check screen in MEMORY mode.

2 Press soft key [(OPRT)].

3 Press soft key [REWIND].

FRYER MACHINE SYSTEMS INC Rob Zahensky

Method 4

1 Select the program screen in EDIT mode.
2 Press soft key [(OPRT)].
3 Press soft key [LINE SEARCH].
4 Press soft key [TOP].

10.2.3 Inserting a Word

Procedure for inserting a word

1 Search for or scan the word immediately before a word to be inserted.
2 Key in an address to be inserted.
3 Key in data.
4 Press the editing key [INSERT] or [INPUT].

Example of Inserting T15

1 Search for or scan Z1250.

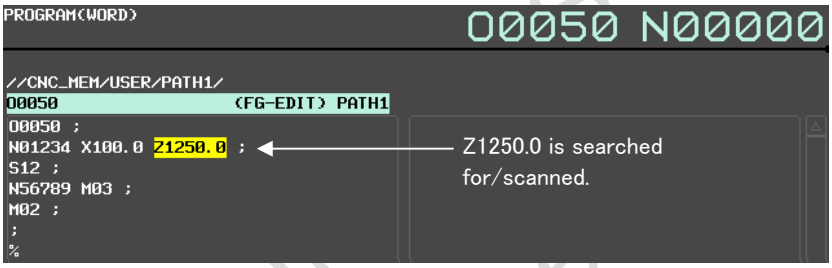


Fig. 10.2.3 (a)

2 Key in [T] [1] [5].
3 Press the editing key [INSERT] or [INPUT].

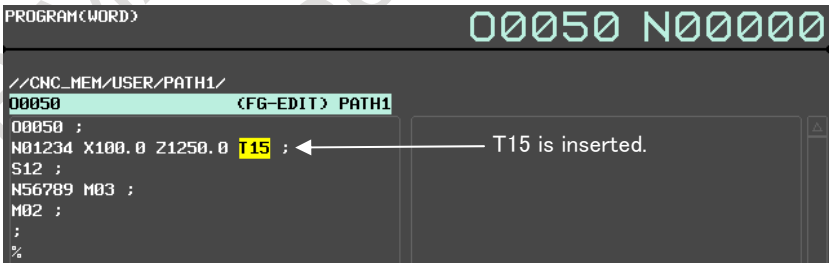


Fig. 10.2.3 (b)

10.2.4 Altering a Word

Procedure for altering a word

1 Search for or scan a word to be altered.
2 Key in an address to be inserted.
3 Key in data.
4 Press the editing key [CALC ALTER].

FRYER MACHINE SYSTEMS INC

Example of changing T15 to M15

1 Search for or scan T15.

Fig. 10.2.4 (a)

2 Key in [M] [1] [5] .

3 Press the editing key [CALC ALTER].

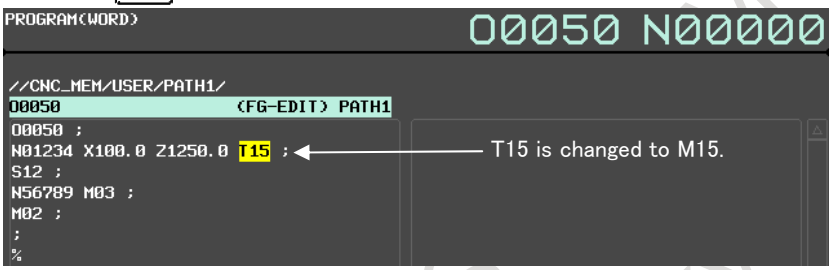


Fig. 10.2.4 (b)

10.2.5 Deleting a Word

Procedure for deleting a word

1 Search for or scan a word to be deleted.

2 Press the editing [DELETE].

Example of deleting X100.0

1 Search for or scan X100.0.

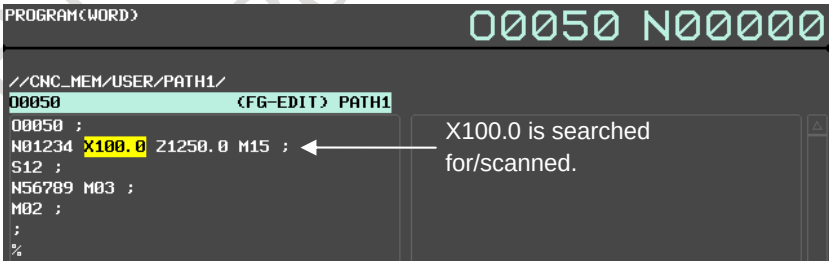


Fig. 10.2.5 (a)

2 Press the editing [DELETE].

Fig. 10.2.5 (b)

FRYER MACHINE SYSTEMS INC
Bob Zahensky

10.3 REPLACING A WORD OR ADDRESS

Replace the specified word or address in a program at a time.
The following methods are available: Replacement after checking each found character string and batch replacement without checking any found character string.

Procedure for replacing a word or address

- Replacing a word or address after checking each found character string

1 Move the cursor to the position from which to start replacement.
2 Press soft key [REPLCE].
3 Enter a word or address before replacement.
4 Press soft key [BEFORE].
5 Enter a word or address after replacement.
6 Press soft key [AFTER].
7 Press soft key [↑ SEARCH] or [↓ SEARCH]. The word or address is searched for in the direction of the arrow.
8 Press soft key [REPLCE] to replace the character string at the cursor and search for the match.
If you want to search for the next match without replacing the character string at the cursor, press soft key [↑ SEARCH] or [↓ SEARCH].
9 Repeat step 8 until required replacement is terminated.
To terminate replacement during operation, press soft key [END].

- Replacing a word or address without checking any found character string

1 Move the cursor to the position from which to start replacement.
2 Press soft key [REPLCE].
3 Enter a word or address before replacement.
4 Press soft key [BEFORE].
5 Enter a word or address after replacement.
6 Press soft key [AFTER].
7 Press soft key [↑ SEARCH] or [↓ SEARCH]. The word or address is searched for in the direction of the arrow.
8 Press soft key [REPLCE ALL].
9 To cancel replace all operation, press soft key [NO].
To execute replace all operation, press soft key [YES]. The word or address is searched for from the cursor position in the direction of the arrow in the soft key pressed in step 7 and all found character strings are replaced at a time. Replace all operation is performed to the beginning of the program for upward replacement or to the end of the program for downward replacement.

NOTE
1 Any character string containing EOB cannot be specified as the character string before or after replacement.
2 Pressing soft key [REPLCE ALL] during replacement with checking each found character string allows replacement without checking the following found character strings to start.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

10.4 DELETING BLOCKS

A block or blocks can be deleted in a program.

10.4.1 Deleting a Block

The portion from the current word position to the next EOB is deleted. The cursor is then placed in the word next to the deleted EOB.

Procedure for deleting a block

1 Search for or scan address N for a block to be deleted.

2 Press the [EOB] key.

3 Press the editing key [DELETE].

Example of deleting a block of N01234

1 Search for or scan N01234.

Fig. 10.4.1 (a)

2 Press the [EOB] key.

3 Press the editing key [DELETE].

Fig. 10.4.1 (b)

10.4.2 Deleting Multiple Blocks

The several blocks in the forward direction from the current word position up to the EOB of the farthest of those blocks are deleted. The cursor is then placed in the word next to the deleted EOB.

Procedure for deleting blocks

1 Search for or scan a word in the first block of a portion to be deleted.

2 Press the [EOB] key as many times as the number of blocks that you want to delete.

3 Press the editing key [DELETE].

FRYER MACHINE SYSTEMS INC
b Zahe sky

Example of deleting blocks from N01234 to the EOB of a block which is two blocks ahead

1 Search for or scan N01234.

```
O0050 ;
N01234 Z1250.0 M15 ;
S12 ;
N05678 M03 ;
M02 ;
%
```

N01234 is searched for/scanned.

Fig. 10.4.2 (a)

2 Press [EOB] [EOB].

3 Press the editing key [DELETE].

```
O0050 ;
N05678 M03 ;
M02 ;
%
```

Fig. 10.4.2 (b)

10.5 PROGRAM SEARCH

When memory holds multiple programs, a program can be searched for. There are four methods as follows.

Procedure for program search

Procedure

Method 1

1 Select EDIT or MEMORY mode.

2 Press function key [PROG] to display the program screen.

3 Enter a program number or program name. When entering a program number, press address key [P 0] then type the program number.

4 Press soft key [PROGRM SEARCH]. Alternatively, press the cursor key [↓] or [↑].

5 Upon completion of search operation, the program number searched for is displayed in the upper-right corner of the screen
If the program is not found, alarm PS0071 occurs.

Method 2

1 Select EDIT or MEMORY mode.

2 Press function key [PROG] to display the program screen.

3 Press soft key [PROGRM SEARCH].

4 Then, press soft key [PREV PROGRM] or [NEXT PROGRM].
Pressing [PREV PROGRM] searches for the previous program on the folder, and pressing [NEXT PROGRM] searches for the next program.

FRYER MACHINE SYSTEMS INC Rob Zahensky

Method 3 This method searches for the program number (0001 to 0031) corresponding to a signal on the machine tool side to start automatic operation. Refer to the relevant manual prepared by the machine tool builder for detailed information on operation.

1 Select MEMORY mode.
2 Set the reset state.
   - The reset state is the state where the LED for indicating that automatic operation is in progress is off. (Refer to the relevant manual of the machine tool builder.)
3 Set the program number selection signal on the machine tool side to a number from 01 to 31.
4 Press the cycle start button.
   - When the signal on the machine tool side represents 00, program number search operation is not performed.
   - If the program corresponding to a signal on the machine tool side is not registered, alarm DS0059 is raised.

Method 4 When bit 4 (PGS) of parameter No. 11308 is 1, search for an O number program by specifying digits only.

1 Select EDIT or MEMORY mode.
2 Press function key [PROG] to display the program screen.
3 Enter the digits of a program number. Address key [P O] need not be pressed before the digits.
4 Press soft key [PROGRM SEARCH]. Alternatively, press the cursor key [↓] or [↑].

Example
- When bit 4 (PGS) of parameter No. 11308 is 1
  Specify "1000" → Search for program O1000.
  Specify "ABC" → Search for program ABC.
- When bit 4 (PGS) of parameter No. 11308 is 0
  Specify "1000" → Search for program 1000.
  Specify "ABC" → Search for program ABC.

When bit 4 (PGS) of parameter No. 11308 is 1, it is not possible to search for a program with any name consisting of digits only.

10.6 SEQUENCE NUMBER SEARCH

Sequence number search operation is usually used to search for a sequence number in the middle of a program so that execution can be started or restarted at the block of the sequence number.

Example) Sequence number 02346 in a program (O0002) is searched for.

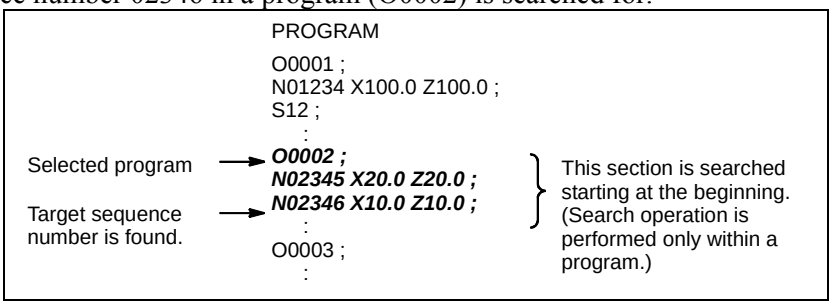


Fig. 10.6 (a)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Procedure for sequence number search

Procedure

1 Select MEMORY mode.

2 Press function key [PROG].

3 If the program contains a sequence number to be searched for, perform the operations 4 to 7 below.
If the program does not contain a sequence number to be searched for, select the program number of the program that contains the sequence number to be searched for.

4 Press address key [N].

5 Key in a sequence number to be searched for.

6 Press soft key [N SRH].

7 Upon completion of search operation, the sequence number searched for is displayed in the upper-right corner of the screen.
If the specified sequence number is not found in the program currently selected, alarm PS0060 occurs.

Explanation

- Operation during Search

Those blocks that are skipped do not affect the CNC. This means that the data in the skipped blocks such as coordinates and M, S, and T codes does not alter the CNC coordinates and modal values.
So, in the first block where execution is to be started or restarted by using a sequence number search command, be sure to enter required M, S, and T codes and coordinates. A block searched for by sequence number search usually represents a point of shifting from one process to another. When a block in the middle of a process must be searched for to restart execution at the block, specify M, S, and T codes, G codes, coordinates, and so forth as required from the MDI after closely checking the machine tool and NC states at that point.

- Checking during search

During search operation, the following checks are made:

- Optional block skip

Limitation

- Searching in sub-program

During sequence number search operation, M98Pxxxx (subprogram call) is not executed. So an alarm PS0060 is raised if an attempt is made to search for a sequence number in a subprogram called by the program currently selected.

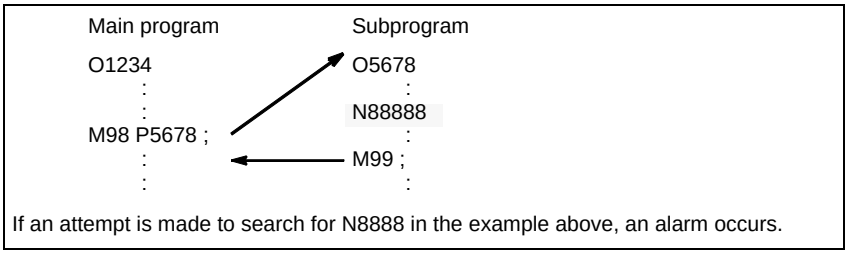


Fig. 10.6 (b)

FRYER MACHINE SYSTEMS INC
Rob Zaheresky

10.7 DELETING PROGRAMS

Programs registered in memory can be deleted, either one program by one program or all at once.

10.7.1 Deleting One Program

A single program in the folder containing the program currently being edited can be deleted.

Procedure for deleting one program

1 Select the EDIT mode.
2 Press function key [PROG] to display the program screen.
3 Press address key [P 0].
4 Key in a desired program number.
5 Press the editing key [DELETE]. The program with the entered program number is deleted.

10.7.2 Deleting All Programs

All programs in the folder containing the program currently being edited can be deleted.

Procedure for deleting all programs

1 Select the EDIT mode.
2 Press function key [PROG] to display the program screen.
3 Press address key [P 0].
4 Key in -9999.
5 Press editing key [DELETE] to delete all programs.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

10.8 EDITING OF CUSTOM MACROS

Unlike ordinary programs, custom macro programs are modified, inserted, or deleted based on editing units.
Custom macro words can be entered in abbreviated form.
Comments can be entered in a program.
Refer to the Section, "CREATING PROGRAMS USING THE MDI UNIT" for the comments of a program.

Explanation

- Editing unit

When editing a custom macro already entered, the user can move the cursor to each editing unit that starts with any of the following characters and symbols:

(a) Address
(b) # located at the start of the left side of a substitution statement
(c) /, (,=, and ;
(d) First character of IF, WHILE, GOTO, END, DO, POPEN, BPRNT, DPRNT and PCLOS

On the screen, a blank is placed before each of the above characters and symbols.
Example) Head positions where the cursor is placed

```
N001 X-#100 ;
#1 =123 ;
N002 /2 X[12/#3] ;
N003 X-SQRT[#3/3*[#4+1]] ;
N004 X-#2 Z#1 ;
N005 #5 =1+2-#10 ;
IF[#1NE0] GOTO10 ;
WHILE[#2LE5] DO1 ;
#[200+#2] =#2*10 ;
#2 =#2+1 ;
END1 ;
```

- Abbreviations of custom macro word

When a custom macro word is altered or inserted, the first two characters or more can replace the entire word.
Namely,

WHILE → WH	GOTO → GO	XOR → XO	AND → AN	SIN → SI	ASIN → AS
COS → CO	ACOS → AC	TAN → TA	ATAN → AT	SQRT → SQ	ABS → AB
BCD → BC	BIN → BI	FIX → FI	FUP → FU	ROUND → RO	END → EN
POPEN → PO	BPRNT → BP	DPRNT → DP	PCLOS → PC	EXP → EX	THEN → TH

Example) Keying in
WH [AB [#2] LE RO [#3]] has the same effect as
WHILE [ABS [#2] LE ROUND [#3]]
The program is also displayed in this way.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

10.9 CURSOR MOVEMENT LIMITATIONS ON PROGRAM EDITING

If a program is to be edited while it is stopped or halted, limitations are imposed so that already executed blocks and buffered blocks cannot be edited. Modifiable blocks are those at and below the cursor position at the time the system switches to the edit mode (EDIT). In the remainder of this manual, the area of editable blocks is called the edit-enabled area, while the area of executed blocks and buffered blocks is called the edit-disabled area.
These limitations are applied when the automatic operation status signal <F0000.7> is ON and when a single block execution of MDI operation is edited, and are also applied to TEACH IN HNDL or TEACH IN JOG mode.
This function is enabled when bit 3 (FPD) of parameter No. 11308 is 1.

Explanation

- States where cursor movement limitations are imposed

- EDIT mode while automatic operation is stopped or halted
- MDI mode and when a single block is set
- TEACH IN HNDL or TEACH IN JOG mode (only when the playback function is enabled)

- Cursor movement

Operations that cause the cursor to move to blocks above the edit mode start position are ignored.

- ↑
- ←
- PAGE UP
- CAN (character editing)

- Search and replacement

Operations that cause the cursor to move to the edit-disabled area are disabled.

- Top line search: Results in a search of the top line of the edit-enabled block.
- Specified-line search (character editing): Made through the edit-enabled area.
  If, however, an attempt is made to search through the edit-disabled area, warning "PROGRAM IN OPERATION" is issued, and the operation is disabled.
- Upward search: Made through the edit-enabled area.
  If attempted at the beginning of the edit-enabled block, this operation is ignored.
- Upward search (replacement): Made through the edit-enabled area.
  If attempted at the beginning of the edit-enabled block, this operation is ignored.
- Upward candidate search: Made through the edit-enabled area.
  If attempted at the beginning of the edit-enabled block, this operation is ignored.
- Character string + ↑ (search): Typing the character string to search for and pressing the ↑ key searches for the character string.
  This search is made through the edit-enabled area. If attempted at the beginning of the edit-enabled block, this operation is ignored.
- Replace all: This operation is disabled.

- Reset

The program execution status is canceled, and the cursor movement limitations are also canceled.

- Program search

Program search operations are disabled in following screens.

- Program Editing screen(Word editing/Character editing/Program contents display/Playback)
- Program Check screen
  Warning "PROGRAM IN OPERATION" is issued, and the operation is disabled.

FRIEDMACHINE SYSTEMS INC
Rob Zahensky

- Program List screen
  When bit 5 (PES) of parameter No. 11302 is 1, only the cursor movement to selected program is executed. Setting to the main program and changing to editing screen are not executed.

- Time insertion

When the machining time stamp function is enabled, soft key [INSERT TIME] does not appear in the state in which the cursor movement limitations are imposed.

10.10 PASSWORD FUNCTION

The password function locks bit 4 (NE9) of parameter No. 3202, which protects programs with program Nos. O9000 to O9999 and programs and folders having the edit/display disable attribute, according to the settings in two parameters, PASSWORD (parameter No. 3210) and KEYWD (parameter No. 3211). In the locked state, parameter NE9 cannot be set to 0. In this case, the protection of programs with program Nos. O9000 to O9999 and programs and folders having the edit/display disable attribute cannot be released unless a correct keyword is set.
A locked state means that the value set in the parameter PASSWD differs from the value set in the parameter KEYWD. The values set in these parameters are not displayed. The locked state is released when the value already set in the parameter PASSWD is also set in parameter KEYWD. When 0 is displayed in parameter PASSWD, parameter PASSWD is not set.

Procedure for locking and unlocking

Locking

1 Set the MDI mode.
2 Enable parameter writing (Section, “Displaying and Entering Setting Data”). At this time, alarm PS0100 is issued on the CNC.
3 Set parameter No. 3210 (PASSWD). At this time, the locked state is set.
4 Disable parameter writing.
5 Press the RESET key to release the alarm state.

Unlocking

1 Set the MDI mode.
2 Enable parameter writing (Section, “Displaying and Entering Setting Data”). At this time, alarm SW0100 is issued on the CNC.
3 In parameter No. 3211 (KEYWD), set the same value as set in parameter No. 3210 (PASSWD) for locking. At this time, the locked state is released.
4 Set bit 4 (NE9) of parameter No.3202 to 0.
5 Disable parameter writing.
6 Press the RESET key to release the alarm state.
7 Subprograms from program Nos. 9000 to 9999 can now be edited.

Explanation

- Setting parameter PASSWD

The locked state is set when a value is set in the parameter PASSWD. However, note that parameter PASSWD can be set only when the locked state is not set (when PASSWD = 0, or PASSWD = KEYWD). If an attempt is made to set parameter PASSWD in other cases, a warning is given to indicate that writing is disabled. When the locked state is set (when PASSWD ≠ 0 and PASSWD ≠ KEYWD), parameter NE9 is automatically set to 1. If an attempt is made to set NE9 to 0, a warning is given to indicate that writing is disabled.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Changing parameter PASSWD

Parameter PASSWD can be changed when the locked state is released (when PASSWD = 0, or PASSWD = KEYWD). After step 3 in the procedure for unlocking, a new value can be set in the parameter PASSWD. From that time on, this new value must be set in parameter KEYWD to release the locked state.

- Setting 0 in parameter PASSWD

When 0 is set in the parameter PASSWD, the number 0 is displayed, and the password function is disabled. In other words, the password function can be disabled by either not setting parameter PASSWD at all, or by setting 0 in parameter PASSWD after step 3 of the procedure for unlocking. To ensure that the locked state is not entered, care must be taken not to set a value other than 0 in parameter PASSWD.

- Re-locking

After the locked state has been released, it can be set again by setting a different value in parameter PASSWD, or by turning the power to the NC off then on again to reset parameter KEYWD.

⚠ CAUTION

1 Once the locked state is set, parameter NE9 cannot be set to 0 and parameter PASSWD cannot be changed until the locked state is released or the memory all-clear operation is performed. Special care must be taken in setting parameter PASSWD.

2 The edit/display disable attribute cannot be set unless parameter PASSWD is set.

3 In the locked state, programs with the edit/display disable attribute are treated as follows:
- The presence of the programs is hidden. This means that these programs are not displayed on screens such as the program folder screen. These programs cannot be edited either.
- These programs cannot be selected as a main program. They can be called as subprograms.

4 In the unlocked state, programs with the edit/display disable attribute are treated in the same manner as ordinary programs.

5 The programs in a folder having the edit/display disable attribute are also treated as described in Caution 3 and 4 above.

6 In the locked state, the folders in a folder having the edit/display disable attribute are treated as follows:
- The presence of the folders is hidden. This means that these folders are not displayed on screens such as the program folder screen.

7 In the unlocked state, the folders in a folder having the edit/display disable attribute are treated in the same manner as ordinary folders.

FRYER MACHINE SYSTEMS INC
Rob Zarnsky

10.11 EDITING PROGRAM CHARACTERS

This section describes how to edit programs registered in the CNC.
Editing operations include character insertion, modification, deletion, and replacement.
While program word editing is performed by recognizing program words, program character editing is performed on a character-by-character basis.

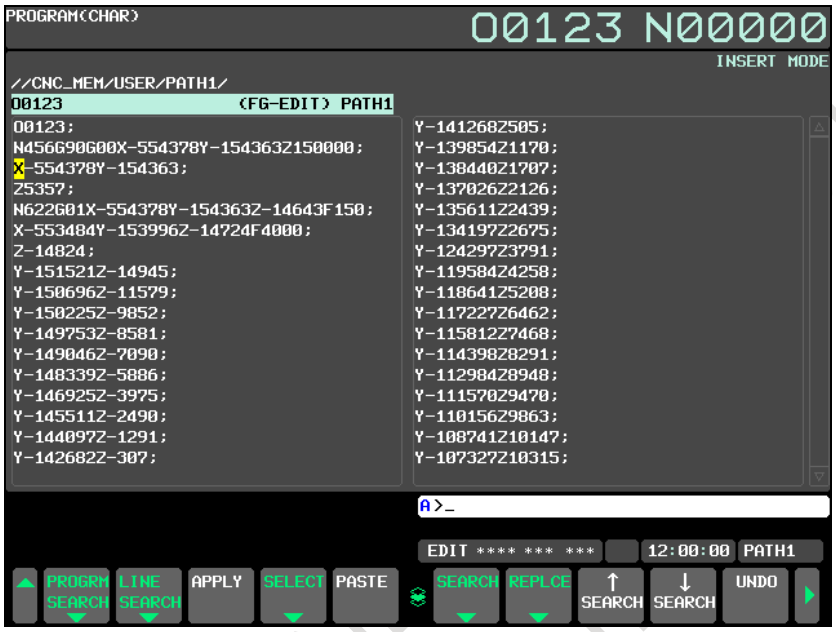


Fig. 10.11 Character editing

Explanation

- Edit unit

Character editing is performed on a character-by-character basis. Select between character editing and word editing according to what to edit.

- Line

A range delimited by an EOB is specified as a line.
In character editing, edited data is saved on a line-by-line basis.
When one program line contains many characters, it extends over multiple lines on the screen, but these lines are counted as one program line.

- Line splitting

When the cursor is on a line during line editing, pressing edit key [INPUT] causes the line to be split into two, one before the cursor and the other after the cursor.

Pressing edit key [CAN] immediately after the line is split causes the lines to return to a single one.

Alternatively, moving the cursor to the EOB of the first line resulting from the split and pressing edit key [DELETE] causes the lines to return to a single one.

- Line merging

If the EOB of a line is deleted, the line is combined with the next line to form a single line.

- Maximum number of characters in one line

One line can consist of up to 140 characters.

FRYER MACHINE INC
Rob Zaher

- Line number

The number of lines are counted starting with the starting line of a program, which is counted as the first line. Even when a line wraps around to the next and subsequent lines, these lines are counted as a single line.

- Clipboard

The clipboard is an area to store characters when a cut or copy operation is performed. The area has a capacity for holding approximately 4000 characters.
The characters that are cut or copied by one cut or copy operation are stored. The most recently stored character string can be used by the paste function.
The information stored by the paste operation is kept unchanged until another cut or copy operation is performed.
The characters stored in the clipboard are maintained until the power to the CNC is turned off unless the clipboard is updated again.

- Undo function

The function for undoing operations performed in program editing undoes operations sequentially, starting with the one performed last. It can undo only those operations that update character strings.
A single undo operation undoes a single input or deletion operation.
If data updated with replace operations is to be undone, a single undo operation undoes the replacement of a single item.
Operations of the cursor keys [→] [←] [↑] [↓] and the page keys [PAGE ↓] [↑ PAGE] cannot be undone with this function because they do not update character strings.

- Example of the undo function

1 Suppose that the following character string is present before modifications:
N110AX[#AXIS3]=100.0;

2 Edit key [DELETE] is pressed five times.
X[#AXIS3]=100.0;

3 Soft key [UNDO] is pressed five times. Then, the original character string is restored as follows:
AX[#AXIS3]=100.0;
0AX[#AXIS3]=100.0;
10AX[#AXIS3]=100.0;
110AX[#AXIS3]=100.0;
N110AX[#AXIS3]=100.0;

- Input modes for editing

Input modes for program editing include insert mode and overwrite mode.
To switch between the input modes, press soft key [INPUT MODE].
Initially, insert mode is set.

- Insert mode
  In insert mode, an entered character is inserted between the current cursor position and the preceding character position.
  1234567890
  When the cursor is placed at 6, and X is entered, the following results:
  12345X67890

- Overwrite mode
  In overwrite mode, the character at the cursor position is replaced by an entered character.
  1234567890
  When the cursor is placed at 6, and X is entered, the following results:
  12345X7890

FRYER MACHINE SYSTEMS INC
Rob Zahenskyj

- Restrictions on editing

O numbers and file names cannot be edited.
EOR (%) cannot be deleted.

- Line editing and automatic saving

When a line is edited, the line is displayed in the update color, blue (which can be changed with color settings), to indicate that the line has been edited. When the line is displayed in the update color, the edit has not yet been saved.
When the cursor is moved to a line preceding or succeeding the edited line, the edit is automatically saved, and the line display changes from the update color to the normal color.
Alternatively, pressing soft key [SAVE] enables the edit to be saved.

- Function for automatically saving an edited line vs. function for undoing editing operations (UNDO function)

When an edit operation is undone, the line is displayed in the update color, and the edit is not saved any longer. If the cursor moves to another line when the operation is undone, the edit is automatically saved.

10.11.1 Available Keys

The available keys are as follows:

- Cursor keys

Cursor keys [→], [←], [↑], and [↓] move the cursor.

- Editing key [DELETE]

Deletes the character at the cursor position.

- Editing key [CAN]

Deletes the character immediately before the cursor position.
When the cursor is at the beginning of a line, the character at the end of the previous line is deleted.

- Editing key [INPUT]

Causes a line change.

- Page change keys

Pages are changed using page key [PAGE ↓] or [↑ PAGE]

Page key [PAGE ↓] moves to the next page.

When page key [PAGE ↓] is pressed on the last page, the cursor moves to the last character position of the last line (the % position).

Page key [↑ PAGE] moves to the previous page.

When page key [↑ PAGE] is pressed on the first page, the cursor moves to the first character position of the first line.

- Character keys

Characters are entered using these keys.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

10.11.2 Input Mode

Input modes include insert mode and overwrite mode.

Changing input mode

To switch between the input modes, use soft key [INPUT MODE].
Pressing soft key [INPUT MODE] causes insert mode and overwrite mode to be selected alternately.
The current mode is indicated in the lower right part of the editing screen.

10.11.3 Line Number Display

This function is used to display a program with line numbers. Pressing soft key [LINE NUMBER] displays a program with line numbers. Pressing soft key [LINE NUMBER] again causes the line numbers to disappear.

10.11.4 Search

A program is searched for a character string.

Search

Procedure

1 Press soft key [SEARCH].
2 A search character string input area is displayed. Enter the character string to search for. To cancel the search, press soft key [CANCEL].
3 Upward search operation
Pressing soft key [UP] causes the program to be searched in the upward direction (toward the top), starting at the current cursor position.
When a search character string is found, the cursor moves to the character string.
Pressing soft key [UP] again causes the program to be searched for the next candidate.
4 Downward search operation
Pressing soft key [DOWN] causes the program to be searched in the downward direction (toward the bottom), starting at the current cursor position.
When a search character string is found, the cursor moves to the character string.
Pressing soft key [DOWN] again causes the program to be searched for the next candidate.

10.11.5 Replacement

A character string in a program is replaced with a specified character string.

Replacement

Procedure

1 Press soft key [REPLCE].
2 Search and replacement character string input areas are displayed. Enter a search character string (character string to be replaced) and a replacement character string (character string to replace the former character string).
To move between the search character string input area and the replacement character string input area, use the cursor keys [↓] and [↑].
If a search character string is input, but no replacement character string is input, the former character string will be deleted.
To cancel the replacement, press soft key [CANCEL].

TRYER MACHINE SYSTEMS INC
Rob Zahensky

3 Replacement operations
Replacement operations include an operation for searching for a search character string and an operation for actually replacing it.
To perform an operation for searching for a search string, use soft key [↑ SEARCH] or [↓ SEARCH].
To perform an operation for actually replacing it, use [REPLCE] or [REPLCE ALL].
- Soft key [↑ SEARCH]
  Searches the program in the upward direction (toward the top), starting at the current cursor position.
  When a search character string is found, the cursor moves to the character string.
  Pressing soft key [↑ SEARCH] again causes the program to be searched for the next candidate.
- Soft key [↓ SEARCH]
  Searches the program in the downward direction (toward the bottom), starting at the current cursor position.
  When a search character string is found, the cursor moves to the character string.
  Pressing soft key [↓ SEARCH] again causes the program to be searched for the next candidate.
- Soft key [REPLCE]
  Replaces the character string found with a search operation with a replacement character string.
- Soft key [REPLCE ALL]
  Replace all instances of the search character string in the program text with the replacement character string.
  Pressing this soft key causes message "ARE YOU SURE YOU WANT TO EXECUTE?" to appear and also soft key [YES] and soft key [NO] to appear. Pressing soft key [YES] causes all replacements to be executed.
  Pressing soft key [NO] cancels the replacement operation, without executing the replacements.

10.11.6 Reversing Edit Operations (Undo Function)

Edit operations performed on a program can be undone sequentially starting with the one performed last.

Reversing edit operations (undo function)

Procedure

1 Pressing soft key [UNDO] once undoes a single operation.
When there are no longer any operation to be undone, pressing this soft key has no effect.

10.11.7 Copy

A selected character string can be copied to the clipboard.

Copying

Procedure

1 Move the cursor to the beginning of the character string to select.
2 Press soft key [SELECT].
3 Using the cursor keys [→] [←] [↑] [↓] and the page keys [PAGE ↓] [↑ PAGE], move the cursor to the end of the character string to select.
At this time, the background of the selected character string is displayed in the selection color (cursor color).
4 Press soft key [COPY] to copy the selected character string to the clipboard.
To cancel the copying, press soft key [CANCEL].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

10.11.8 Cut

A selected character string can be deleted.
When deleted, the character string is copied to the clipboard.

Cut

Procedure

1 Move the cursor to the beginning of the character string to select.
2 Press soft key [SELECT].
3 Using the cursor keys [→] [←] [↑] [↓] and the page keys [PAGE ↓] [↑ PAGE], move the cursor to the end of the character string to select.
At this time, the background of the selected character string is displayed in the selection color (cursor color).
4 Press soft key [CUT] to delete the selected character string.
To cancel the deletion, press soft key [CANCEL].

10.11.9 Paste

The character string stored in the clipboard can be inserted at the current cursor position.
After pasted, the character string remains stored in the clipboard, so that the character string can be pasted as many times as necessary.

Paste

Procedure

1 Move the cursor to the position at which you want to paste a character string.
2 Press the soft key [PASTE].

10.11.10 Saving

Moving the cursor to a line preceding or succeeding the edited line causes the edit to be automatically saved, but it is possible to save the edit by pressing soft key [SAVE], if necessary. For a program consisting of only a single line, in particular, it is necessary to press soft key [SAVE] to save the edit.

Saving

Procedure

1 Press the soft key [SAVE].

10.11.11 Creation

A new program can be created.

Creation

Procedure

1 Press soft key [NEW PROGRM].
2 A program name input area appears.
3 Enter the name of a program to be created.
4 Press the soft key [EXECUTE]. This creates a new program and displays the editing screen.
To cancel the creation of a new program, press soft key [CANCEL].

FRYER MACHINE SYSTEMS INC
Rob Zaranensky

10.11.12 Line Search

The cursor can be moved to a specified line.
The cursor can be moved to a line with a specified line number, to the first line of the program, and to the last line of the program.

Line search

Procedure

Movement to a line with a specified line number
1 Press soft key [LINE SEARCH].
2 A line number input area appears.
3 Enter a line number. The first line is regarded as having a line number of 1.
4 Press soft key [LINE NUMBER], and the cursor moves to the specified line.

Movement to the first line
1 Press soft key [LINE SEARCH].
2 Press soft key [TOP], and the cursor moves to the first line of the program.

Movement to the last line
1 Press soft key [LINE SEARCH].
2 Press soft key [BOTTOM], and the cursor moves to the last line of the program.

10.12 KEYS AND PROGRAM ENCRYPTION

Overview

Program contents can be protected by setting parameters for encryption and for the program security range.

Explanation

1 Security with a password and a security range
When the password and security range parameters are specified, the display, editing, and input/output operations are disabled for the programs within the security range.
This prevents the custom macro programs created by the machine tool builder from being accidentally changed or deleted by the end user. This also provides security because the contents of the programs are not displayed.
2 Input/output of encrypted programs
The programs in the security range can be encrypted before being output. Once encrypted, the programs cannot be decrypted. In addition, encrypted programs can be input directly.

- Locking/unlocking

When the programs in the security range are secured, the program memory is said to be locked.
If they are not secured, the program memory is said to be unlocked.

- Parameter

- Parameter PASSWORD (No.3220)
Set the password required to lock the program memory. A password other than 0 can be set. The password is not displayed. A password can be set when no password is set (PASSWORD = 0) or the program memory is unlocked.
- Parameter KEY (No.3221)
When KEY is set equal to PASSWORD, the program memory is unlocked. The value set for the parameter is never displayed. Each time the CNC is started, 0 is set for the parameter. This means that whenever the CNC is started, a lock is provided if a password is set (PASSWORD ≠ 0).

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Minimum value (in parameter No.3222) and maximum value (in parameter No.3223) of the program security range
  Set the desired security range. The value set for MINIMUM must not be greater than that set for MAXIMUM. The programs in the range of MINIMUM to MAXIMUM are secured. If MINIMUM is set to 0, it is regarded as being 9000. If MAXIMUM is set to 0, it is regarded as being 9999.
  A range can be set when no password is set or the program memory is unlocked.

NOTE

1 For security, the values set for PASSWORD and KEY are not displayed. For the same reason, PASSWORD, MINIMUM, and MAXIMUM can be specified only when no password is set or the program memory is unlocked. Set a password, taking great care to avoid a situation where the program memory cannot be unlocked because a password is set incorrectly.

2 The [+INPUT] key used to specify PASSWORD and KEY behave in the same way as the [INPUT] soft key.

  Example: When 99 is set for KEY, pressing [1] and [+INPUT] changes the value to 1.

3 These four parameters cannot be output externally. Also, even when these parameters are input by the parameter input operation, they are ignored.

- Inputting/outputting and collating programs

When a program is encrypted, a password is output. The password is used to load the program.
The following output operations are enabled for programs outside the security range if the locked state is present or for programs within the security range if the unlocked state is present:

Outputting all programs

Locked/unlocked	Results
Locked	All the programs outside the security range are output in the normal way.
Unlocked	All the programs within the security range are encrypted and output.
Password not set	All the programs in the program memory are output in the normal way.

Outputting a single program

Locked/unlocked	Results
Locked	If a program is outside the security range, it is output in the normal way. If it is within the security range, a warning "PROTECTED" is issued.
Unlocked	If a program is outside the security range, it is output in the normal way. If it is within the security range, it is encrypted and output.
Password not set	The program is output in a normal way.

Outputting specified multiple programs

Locked/unlocked	Results
Locked	When all of the specified programs fall outside the protected range, they are output as usual. When all of the specified programs are within the security range, warning message "PROTECTED" is issued. When some of the specified program numbers are outside the security range, and the others are within the security range, only the programs outside the security range are output normally. When the programs to be output are not within the specified range, warning message "THE WRONG DATA IS USED" is issued.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Locked/unlocked	Results
Unlocked	When all of the specified programs fall outside the protected range, they are output as usual. When all the specified programs fall within the protected range, they are output as encoded programs. When some of the specified program numbers are outside the security range, and the others are within the security range, only the programs within the security range are encoded and output. When the programs to be output are not within the specified range, warning message "THE WRONG DATA IS USED" is issued.
Password not set	The program is output in a normal way.

Inputting an un-encrypted program

Locked/unlocked	Results
Locked	When the program to be input is outside the security range, it is input normally. When the program to be input is within the security range, warning message "PROTECTED" is issued.
Unlocked, or password not set	The program is input.

Inputting an encrypted program

Password set in the system and password of the program	Results
Password set in the system	Warning “WRITE PROTECT” is issued.
Password for the program	When the program is within the security range, it is input normally. When the program is outside the security range, warning message "WRITE PROTECTED" is issued.
Password not set in the system	The program is input. The PSW in the file is set for parameter No. 3220. This applies if the program is in the security range. If the program is outside the security range, warning message "PROTECTED" is issued.

Collating a program with an encrypted program

In the unlocked state, the following takes place:

Password set in the system and password of the program	Results
Password set in the system	Alarm SR0075 “PROTECT” is issued.
Password set in the system = Password of the program, or password not set in the system	The program is collated.

In the locked state, program collation cannot be performed.

NOTE

To encrypt programs, set parameter ISO (bit 1 of No.0000) to 1 (to specify that the output code is ISO).

- Program display

1 On the program folder screen, all program numbers are displayed together with comments.

2 In the locked state, the programs within the security range are not displayed on the program screen. In the unlocked state, the programs within the security range are displayed in the same way as normal programs.

FRIED MACHINE SYSTEMS INC Rob Zahensky

NOTE

Immediately after switched from the unlocked state to the locked state or from the locked state to the unlocked state, programs are displayed in the previous state. They are displayed in the new state when a program search or other operation is performed.

- Editing and deleting programs

When the program memory is locked, the programs within the security range cannot be edited or deleted. When the program memory is locked, an attempt to delete all programs results in only those programs outside the security range being deleted.
In the locked state, the programs in the security range cannot be selected as a main program. If an attempt is made to select such a program on the program folder screen as a main program, warning message "SETTING A MAIN PROGRAM FAILED" is issued.

- Searching for programs

In the locked state, a program search is performed within the protected range as described below.

1 When no program number is specified, programs within the protected range are skipped.
2 When an attempt is made to search for a program within the security range by specifying a program number, the search operation is ignored, resulting in warning message "Program is protected".

10.13 SIMULTANEOUS EDITING OF MULTIPATH PROGRAMS

Simultaneous editing of multi-path programs enables simultaneous editing of programs for multiple paths on a single screen.
This function is enabled when bit 0 (DHD) of parameter No. 3106 is 1.

Explanation

- Procedure

1 Set the paths subject to simultaneous editing, with multi-path simultaneous display parameters Nos. 13131 and 13132.
(For details of the parameters, refer to the Parameter Manual (B-64700EN).)
2 Place the paths subject to simultaneous editing in EDIT mode.
3 Press function key PROG.
4 Press soft key [PROGRAM] to display the program editing screen.

- Screen display

Fig. 10.13 (a) and Fig. 10.13 (b) show examples of performing simultaneous editing of multi-path programs.
Above each program is a status line on which three items of information are displayed: the program name, "FG-EDIT", which indicates that the program is being edited in the foreground, and the path name.
For the program currently being edited, its status line is displayed in reverse video.

FRYER MACHINE SYSTEMS INC
Rob Zaharisky

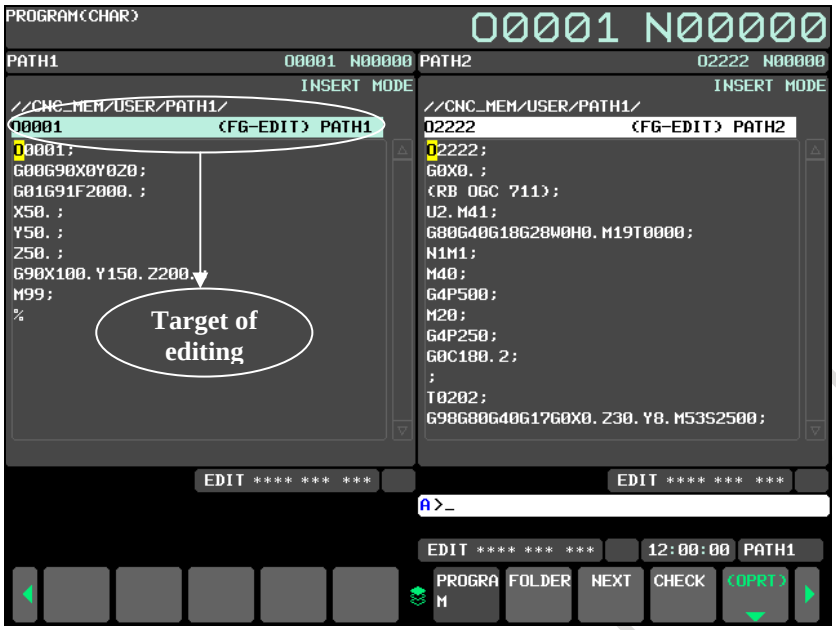


Fig. 10.13 (a) Simultaneous editing of multi-path programs screen (10.4-inch display unit)

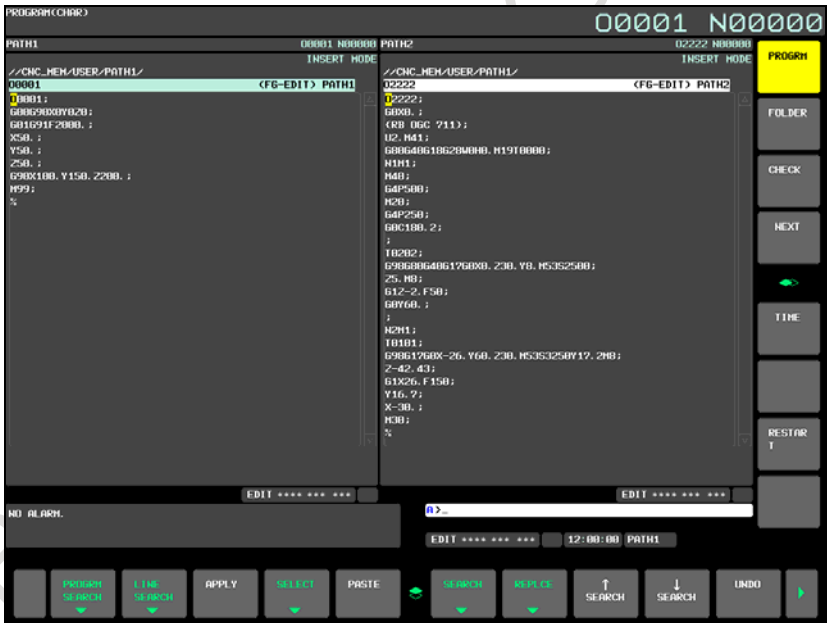


Fig. 10.13 (b) Simultaneous editing of multi-path programs screen (15-inch display unit)

- Modes

When the paths to be displayed simultaneously are in either EDIT or MEM mode, the multi-path programs are displayed simultaneously on the program screen. Selecting the path for which EDIT mode is selected enables program editing to be performed.

Fig. 10.13 (c) shows an example in which both EDIT and MEM modes are selected.

For path 1, MEM mode (execution status screen) is selected, and for path 2, EDIT mode (editing screen) is selected.

Soft keys are switched according to the mode for the currently selected path.

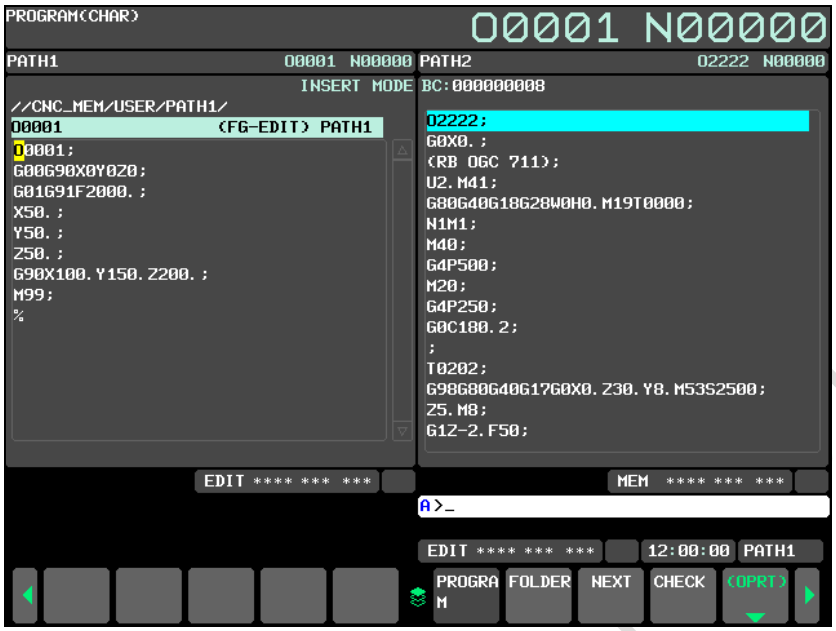


Fig. 10.13 (c) Screen on which both MEM and EDIT modes are selected

- Switching the path subject to editing

The path selected with the path selection signal is subject to editing.

- Maximum number of paths that can be subject to editing simultaneously

The maximum number of paths that can be subject to editing simultaneously on each display unit is as given in the Table 10.13.

Table 10.13

Display unit size	Maximum number of paths that can be subject to editing simultaneously
10.4-inch	3
15 or 19-inch	4

- Conditions under which simultaneous editing is not usable

Simultaneous editing of multi-path programs is disabled in the following cases:

- The program screen is selected as a small screen.
- The paths to be displayed simultaneously include a path for which a mode other than EDIT and MEM modes is selected.
- The virtual MDI key function is enabled.

If, in the state of simultaneous editing of multi-path programs, background editing is started, background editing is performed on all screens. Simultaneous editing of multi-path programs and background editing cannot be used at the same time.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

10.14 MULTI-PATH EDITING FUNCTION

10.14.1 Overview

When the program of the path to be edited is scrolled in the simultaneous editing of multi-path programs screen, other path programs that are displayed on the same screen can be scrolled simultaneously.

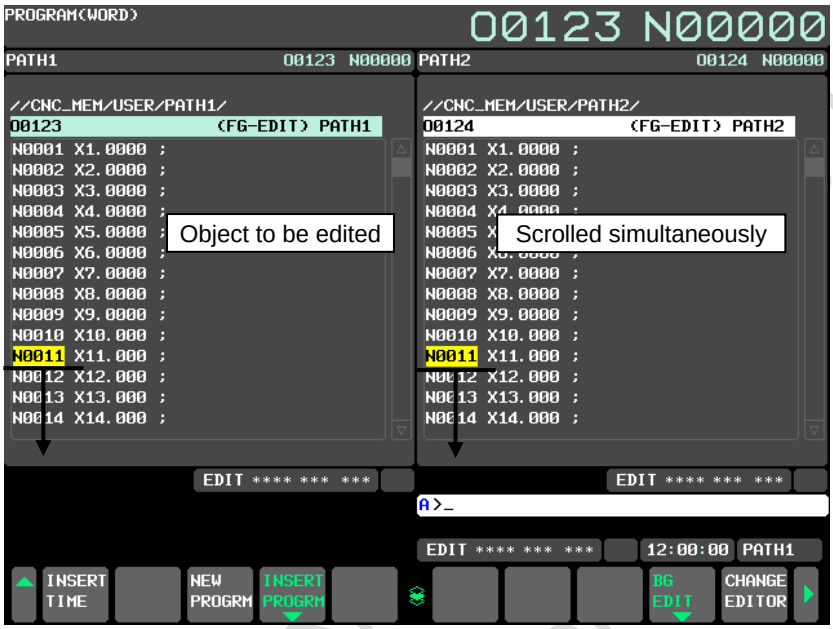


Fig. 10.14.1 Synchronous scroll function

This function provides the simultaneous scroll mode in which all programs being edited simultaneously are scrolled and the single scroll mode in which only the program to be edited is scrolled. It is possible to switch between these modes easily through soft key operation.

In the simultaneous scroll mode, when a waiting M-code comes due to a scroll, waiting is performed by stopping the scroll of the path until the arrival of the same waiting M-code of other paths. It is, therefore, possible to edit the program while confirming the waiting of each path.
Moreover, the waiting search function enables the cursor positions of all paths being edited simultaneously to be moved to specified waiting M-code simultaneously.

10.14.2 Details

Switching between simultaneous and single scroll modes

The following conditions are necessary for switching between simultaneous scroll mode and single scroll mode on the program editing screen.

- Bit 0 (DHD) of parameter No. 3106 for enabling the simultaneous multi-path program editing function is 1.
- There are two or more paths in the simultaneous editing mode.
- All paths being edited simultaneously are in EDIT mode.
- All program editing screens for simultaneous editing are in word editor mode.

NOTE
1 The single scroll mode is selected when the power is turn on.
2 If the above conditions are not satisfied, the scroll mode is automatically switched to single scroll mode.

Procedure for switching to the simultaneous scroll mode

The procedure for switching to the simultaneous scroll mode is as described below.

1 Press function key PROG.
2 Press soft key [PROGRAM] to display the program editing screen.
3 Press soft key [(OPRT)].
4 Press continuous menu key [+] several times until oft key [SYNC SCROLL] appears.
5 Press soft key [SYNC SCROLL].
6 The screen changes to the simultaneous scroll mode. At this time, the display of the soft key [SYNC SCROLL] changes to [SINGLE SCROLL].

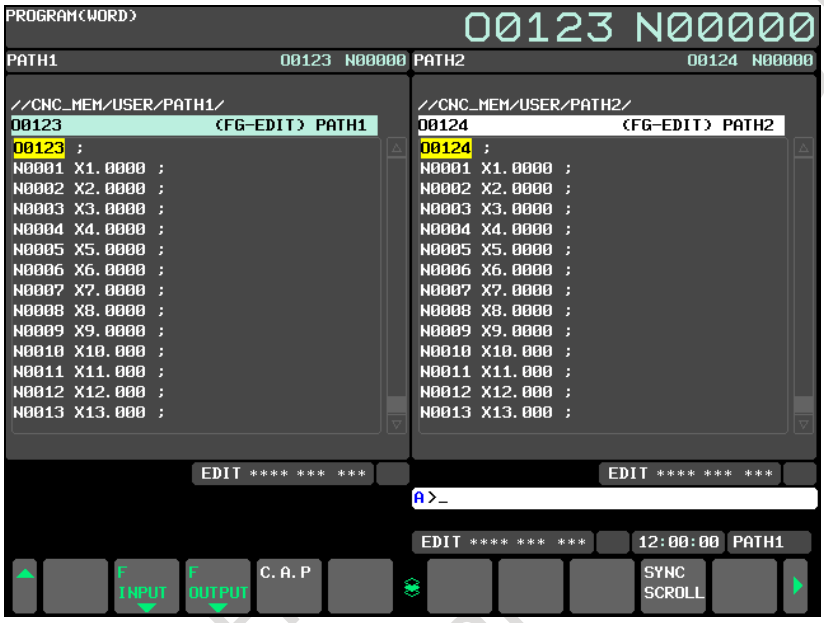


Fig. 10.14.2 (a) Multi-path editing screen

Procedure for switching to the single scroll mode

The procedure for switching to the single scroll mode is as described below.

1 Press function key PROG.
2 Press soft key [PROGRAM] to display the program editing screen.
3 Press soft key [(OPRT)].
4 Press continuous menu key [+] several times until oft key [SINGLE SCROLL] appears.
5 Press soft key [SINGLE SCROLL].
6. The screen changes to the single scroll mode. At this time, the display of the soft key [SINGLE SCROLL] changes to [SYNC SCROLL].

Automatic simultaneous scroll mode change

When the parameter DAS (No.11349#5) is set to 1, the scroll mode is simultaneous scroll mode after power in turn on. The scroll mode becomes a single scroll mode in the following situations. It automatically returns to the simultaneous scroll mode when the following situations are canceled.

- There are no two paths or more in the simultaneous editing mode.
- One or more paths are not EDIT modes.
- One or more programs of simultaneous editing are in character editing mode.
- During background editing mode.

TRYER MACHINERY INC Rob Zak

Cursor movement operations

In the simultaneous scroll mode, pressing the cursor movement keys ↑ ↓ causes the cursors to move up and down in all programs being edited simultaneously.

If the cursor movement keys ← → are pressed to move within the same block, the cursors for the paths other than the one to be edited do not move.

If the cursor movement keys ← → are pressed to move the cursor to the previous and next blocks, the cursors for the paths not to be edited also move.

Those programs for paths that are not displayed on the screen do not scroll simultaneously.

Page change operation

In the simultaneous scroll mode, pressing the page change keys PAGE ↓ ↑ PAGE causes the page to change in all programs being edited simultaneously.

Scroll waiting with a waiting M-code

In the simultaneous scroll mode, if the cursor moves to a block that contains a waiting M-code, the system enters the scroll waiting state.

In the scroll waiting state, the cursor cannot be moved in the direction in which the cursor has moved to the block that contains a waiting M-code until the cursors of all programs subject to waiting move to the same waiting M-code.

In the scroll waiting state, the cursor color changes to blue as shown in Fig. 10.14.2 (b).

NOTE

A waiting M code is not ignored due to the waiting ignore signal.

Example:

The cursor cannot be moved in the down direction if pressing the cursor key ↓ causes the system to enter the scroll waiting state. The cursor can move in the up direction.

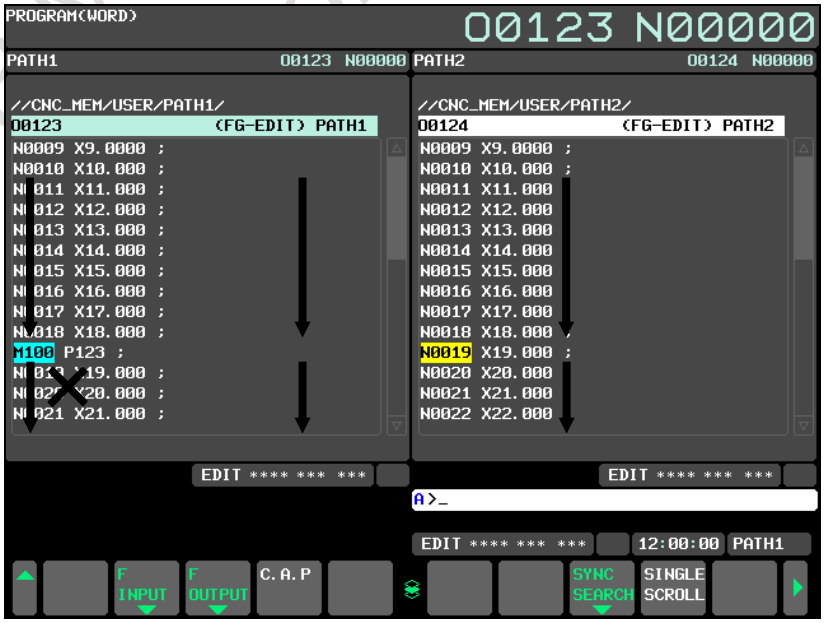


Fig. 10.14.2 (b) Waiting caused by pressing the cursor key

FRYER MACHINE SYSTEMS INC
Jacob Zahensky

Similarly, if the cursor is moved to a block that contains a waiting M-code by pressing a page change key, the system enters the scroll waiting state in the block that contains a waiting M-code.

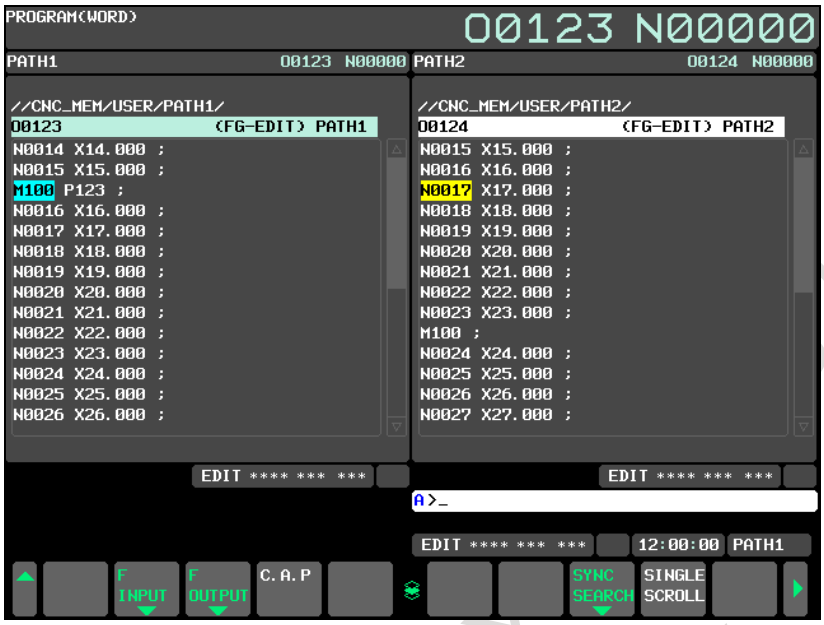


Fig. 10.14.2 (c) Scroll waiting caused by pressing a page change key

Completion of scroll waiting

When the cursors move to the same waiting M-code in all programs subject to waiting, scroll waiting is completed, so that scrolling can be continued.

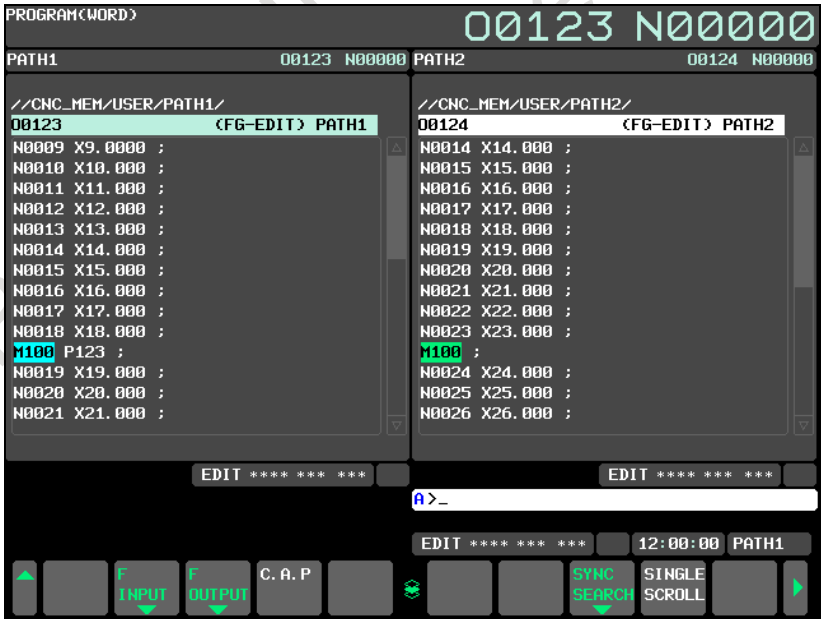


Fig. 10.14.2 (d) All programs displayed simultaneously are in the scroll waiting state

Release of scroll waiting

If, in the scroll waiting state, the cursor for the program for another path is moved to the beginning or end of the program, a confirmation message and soft keys are displayed as shown in Fig. 10.14.2 (e).
To release the scroll waiting state, press the soft key [EXEC].
To cancel the release of the scroll waiting state, press the soft key [CAN].

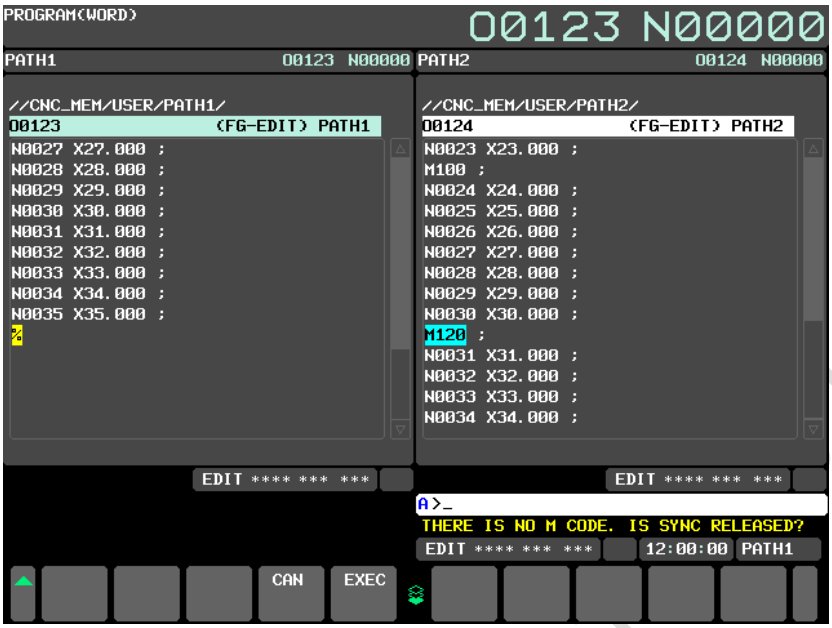


Fig. 10.14.2 (e) Display of the confirmation message for the release of the scroll waiting state

Waiting M-code search

By doing a waiting M-code search, it is possible to simultaneously move the cursors to the blocks that contain specified waiting-M code in all programs being edited simultaneously.

To do a waiting M-code search, press the soft key [SYNC SEARCH] in the simultaneous scroll mode. The soft keys shown in Fig. 10.14.2 (f) are displayed.

Fig. 10.14.2 (f) Soft keys for doing a waiting M-code search

[PREVI SYNC]	Searches for a waiting M-code in the up direction, starting at the cursor position in the program to be edited. The cursors of the paths specified for waiting move to the same waiting M-code.
[NEXT SYNC]	Searches for a waiting M-code in the down direction, starting at the cursor position in the program to be edited. The cursors of the paths specified for waiting move to the same waiting M-code.
[MCODE NUMBER]	Searches for a specified waiting M-code. Only an M-code or an M-code number can be specified. The cursors move to the specified waiting M-code in all programs being edited simultaneously.
[TOP]	In all programs being edited simultaneously, moves the cursors to the beginning of the programs.
[BOTTOM]	In all programs being edited simultaneously, moves the cursors to the end of the programs.

If, as a result of a waiting M-code search, no waiting M-code is found, the cursor moves to the end of the program. In this case, the warning, "SYNC POINT IS NOT FOUND.", is displayed.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Highlight of Waiting M-code

When bit 1 (SMH) of parameter No.24308 is 1, waiting M code is highlighted.

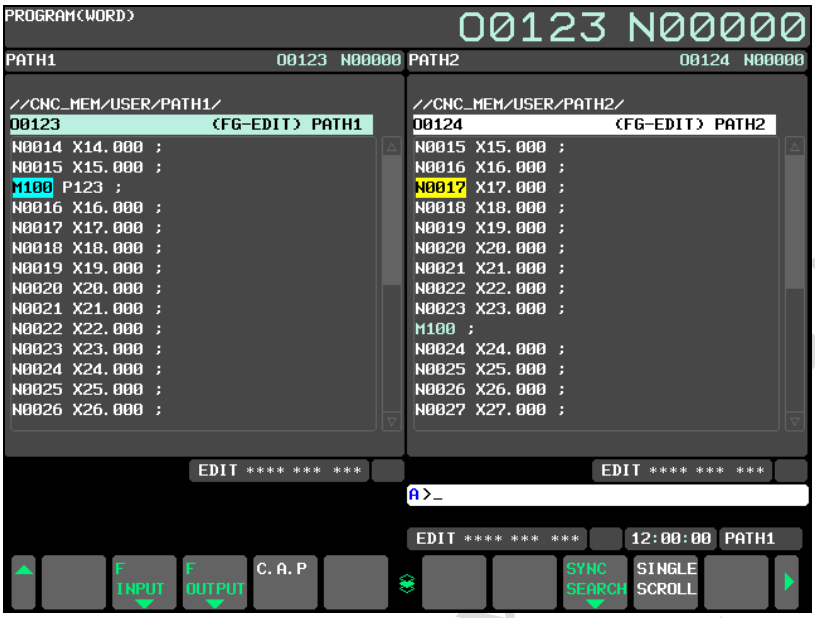


Fig. 10.14.2 (g) Highlight of Waiting M-code

Restrictions

The simultaneous scroll function cannot be used in character editing or in background editing.

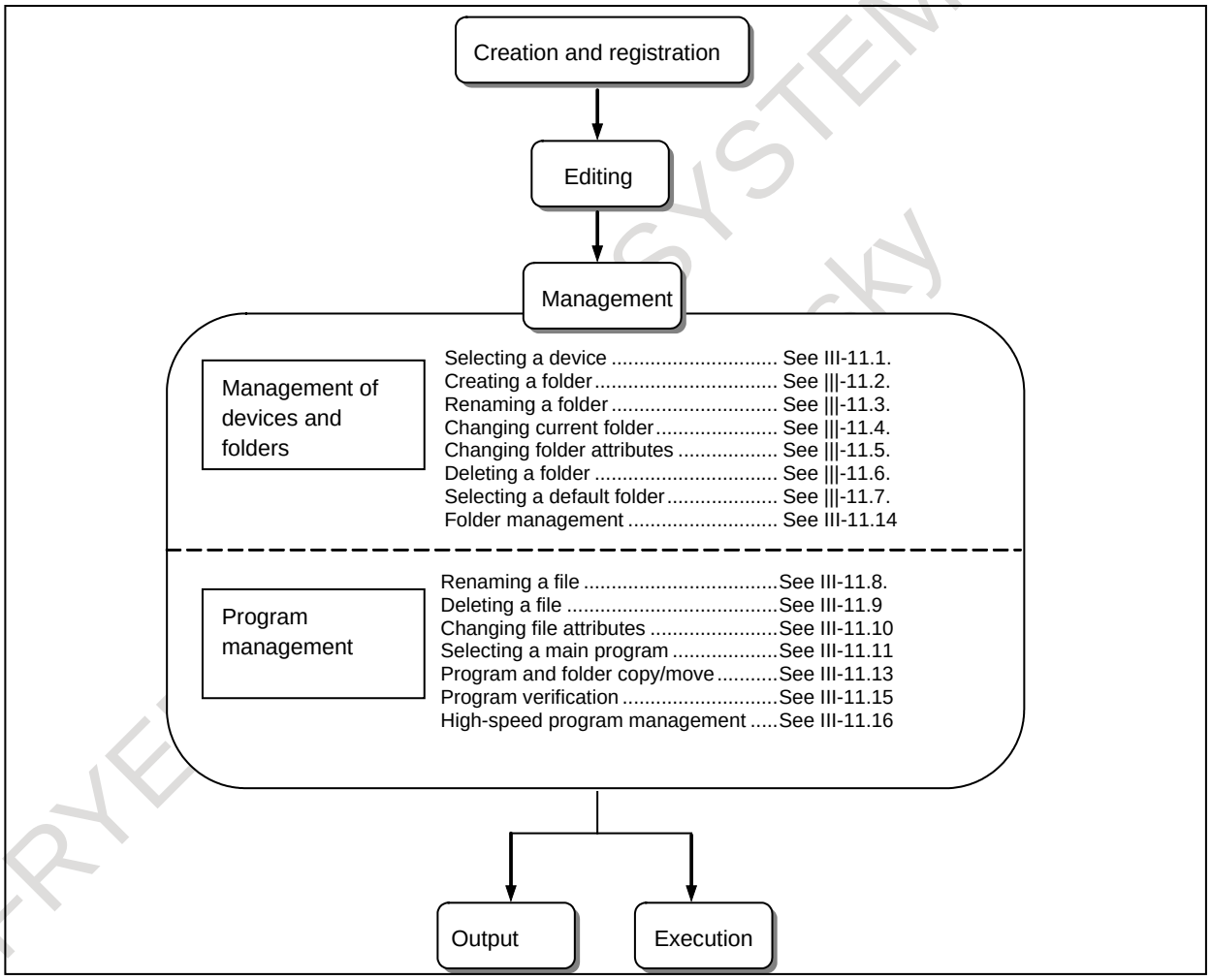
FRYER MACHINE SYSTEMS INC
Rob Zahensky

11 PROGRAM MANAGEMENT

Program management functions are classified into the following two types:
- Functions for folders
- Functions for programs

Functions for folders include creation, deletion, change of names and attributes, and so on.
Functions for programs include main program selection, deletion, change of names and attributes, and so on.

About program management, refer to "PROGRAM MANAGEMENT" in PROGRAMMING additionally.



FRYE SYSTEMS INC

11.1 SELECTING A DEVICE

When the fast data server function is provided, a program storage device can be selected. This section explains the selection procedure.

Procedure for selecting a device

1 Press the function key PROG.
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [DEVICECHANGE].
5 Press the soft key for the desired device.
Devices that can be selected are listed below (Table 11.1):

Table 11.1

Soft key	Device name
CNC MEM	CNC part program storage memory
MEM CARD	Program storage file (named "FANUCPRG.BIN")
EMB ETHER	Embedded Ethernet
DTSVR	Data Server
DTSVR HOST	Data Server Host
MEMORY CARD	Memory card
FLOPPY	Floppy cassette (Handy File)
USB MEM	USB memory

11.1.1 Selecting a Memory Card Program as a Device

Overview

By selecting a memory card including a program storage file (named "FANUCPRG.BIN") as a device, memory operation can be performed with the program in the program storage file selected as the main program.
In addition, the content of a program storage file can be displayed on the program list screen or a program in a program storage file can be edited on the program edit screen.
A program storage file can be created using a memory card program tool (A08B-9010-J700#ZZ11) on a commercially available personal computer. To be used, the program storage file must be written to a memory card formatted in FAT16 format.
(A program held in a program storage file is hereinafter referred to as a memory card program. Moreover, a memory card storing a program storage file is referred to as a program storage memory card.)

NOTE
Any program in a USB memory cannot be selected or edited.

Procedure for selecting a device

1 Press the function key PROG.
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [DEVICECHANGE].
5 Press the soft key [MEMCARD].

FRYER MACHINE SYSTEMS INC
Rob Zamensky

NOTE
1 A FAT16-formatted memory card containing the program storage file FANUCPRG.BIN is recognized as a program storage memory card.
2 For a program storage memory card containing more than 63 folders and programs, the function for extending the number of memory card program registrations is required. The function for extending the number of memory card program registrations is applicable to a program storage memory card containing a maximum of 1000 folders and programs.

Procedure for removing a device

When a program storage memory card is replaced or a memory card is used for normal usage such as data input/output, clear the recognition of the program storage memory card with removal operation.

1 Press the function key .
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [DEVICECHANGE].
5 Press the soft key [DETACH].

NOTE
1 This soft key appears when the CNC recognizes a program storage memory card during device change operation.
2 This operation is enabled only in EDIT mode or MEM mode.
When a memory card program is selected in the main programs of multiple paths in a multi-path control system, set the modes of all paths to EDIT mode or MEM mode.
3 When the default folder is a folder in a program storage file, it is changed to "//CNC_MEM/" by a removal operation.
4 When the main program is a memory card program, the main program enters the unselected state by a removal operation.

Explanation

- About operation

A memory card program can be selected as a main program to perform memory operation.
Memory operation has the following features:
- Subprogram call nesting is allowed.
- Macro program call nesting is allowed.
- In a custom macro, a control command using a GOTO statement/WHILE statement can be specified.
- With the T series, a multiple repetitive canned cycle for turning can be specified.

NOTE
To use the function for using custom macros, enable custom macros (bit 5 (NMC) of parameter No.8135 is set to 0).

- Selection as a main program

As a main program to be automatically executed in the MEM mode, a memory card program can be selected.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- **Sub program (call using M98/G72.1/G72.2)**
- **Macro program (call using G65/G66/G66.1/M96)**

The following subprogram/macro program held in the same folder as containing the main program is called:

- Sub program call (M98)
- Macro call (Simple macro call G65 / Modal call G66, G66.1)
- Macro interrupt (M96)
- Figure copying (G72.1, G72.2)

If the program cannot be found in the same folder as the main program, the following folder is searched:

- Common program folder on the CNC_MEM device (CNC program storage memory)

- **Sub program (call using M code/S code/T code/particular address/the second auxiliary function)**
- **Macro program (call using G code/M code)**

The following subprogram/macro program calls a program from the CNC_MEM device (CNC program storage memory):

- Subprogram call using M code/S code/T code/particular addresses/the second auxiliary function
- Macro call using G code/M code

The folders set as search targets are searched in the following order, and the first found program is called:

1 Common program folder among the initial folders
2 MTB-dedicated folder 2 among the initial folders
3 MTB-dedicated folder 1 among the initial folders
4 System folder among the initial folders

Search target folders are set in parameter No. 3457.

NOTE

For a memory card program, subprogram call using M code/S code/T code/particular addresses/the second auxiliary function or macro call using G code/M code can be specified. However, a program on the CNC_MEM device (CNC program storage memory) is called.

- **External program number search / External workpiece number search**

A program on a program storage memory card can be searched for with the external program number search function or external workpiece number search function.

Limitation

For a memory card program, M198 cannot be specified. Moreover, no memory card program can be called from a program on the CNC_MEM device (CNC program storage memory) by specifying M198.

When a setting is made to enable an external device subprogram call from a memory card (M198) or DNC operation from a memory card (bit 7 (MNC) of parameter No. 0138 = 1), the content of program storage file cannot be displayed during automatic operation.

When a program storage memory card is selected, the memory card cannot be used for the ordinary purposes listed below. To use a memory card in such a case, perform a "removal" operation to cancel the recognition of the program storage memory card.

- ALL I/O screen
  Display of the contents of a memory card, and input/output data to and from a memory card
- PMC data I/O screen
  Display of the contents of a memory card, and input/output to and from a memory card
- Program directory screen
  Input/output program data to and from a memory card

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- External device subprogram call (M198) operation
  Subprogram call (M198) with a memory card set as an external device
- DNC operation
  DNC operation from a memory card

⚠ CAUTION

1 Do not remove the memory card when a program that specifies a write to the memory card is being edited. The data can be destructed.
2 If an editing operation is completed, the results of editing are preserved even when the power to the CNC is turned off.
3 When removing the memory card, be sure to perform a "removal" operation. If the memory card is removed without performing a "removal" operation and an attempt is made to access the memory card, the alarm (SR1964) or alarm (IO1030) is issued.
  If the card is removed inadvertently, insert the card again and perform a "removal" operation.
  When an alarm is issued, perform the following operation:
  - When the alarm (SR1964) is issued
    Reset the alarm after performing a "removal" operation.
  - When the alarm (IO1030) is issued
    The alarm can be reset only by turning off the power to the CNC.
4 There are cases in which when a memory card is replaced with another, the CNC cannot detect the replacement. Thus, it is risky to replace a memory card without performing a "removal" operation, and this should never be attempted.

- Operation of creation, edition, and management of a program

When “memory card program as a device” is selected, operation of creation, edition, and management of a program is below (Table 11.1.1):

Table 11.1.1

Item	Usable
Creation of a program	Unusable
Edition prohibition attribute	Unusable
Inserting, alteration, and deletion a Word	Usable
Deletion of a block	Usable
Program search	Usable
Sequence number search	Usable
Deletion of a program	Unusable
Editing a custom macro	Usable
Password function	Unusable
Editing a program character	Usable
Program and folder copy/move function	Usable (Note)
Key and program encryption	Unusable
Selecting a device	Usable
Creation of a folder	Unusable
Renaming of a folder	Unusable
Changing the attribute of a folder	Unusable
Deletion of a folder	Unusable
Selecting a default folder	Usable
Renaming of a file	Unusable
Deletion of a file	Unusable
Changing the attribute of a file	Unusable
Selecting a main program	Usable
Input/output of program	Unusable

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

In memory card program device, programs are copied from memory card program device to CNC built-in memory device or to data server device, or copied from CNC built-in memory device or from data server device to memory card program device.

11.1.2 Selecting a From Cassette as a Device

The Handy File can be connected to the NC to display a list of programs in a floppy cassette.

Procedure for selecting a device

1 Set the relevant parameters so that communication with the Handy File is enabled.
2 Press the function key PROG.
3 Press the soft key [FOLDER].
4 Press the soft key [(OPRT)].
5 Press the soft key [DEVICECHANGE].
6 Press the soft key [FLOPPY].

Limitation

- Floppy disk directory display during schedule operation

During schedule operation, directories in a floppy disk cannot be displayed in a background edit.

- Operation of creation, edition, and management of a program

When “floppy cassette as a device” is selected, operation of creation, edition, and management of a program is below (Table 11.1.2):

Table 11.1.2

Item	Usable
Creation of a program	Unusable
Edition prohibition attribute	Unusable
Inserting, alteration, and deletion a Word	Unusable
Deletion of a block	Unusable
Program search	Usable
Sequence number search	Unusable
Deletion of a program	Usable
Editing a custom macro	Unusable
Password function	Unusable
Editing a program character	Unusable
Program and folder copy/move function	Unusable
Key and program encryption	Unusable
Selecting a device	Usable
Creation of a folder	Unusable
Renaming of a folder	Unusable
Changing the attribute of a folder	Unusable
Deletion of a folder	Unusable
Selecting a default folder	Unusable
Renaming of a file	Unusable
Deletion of a file	Usable
Changing the attribute of a file	Unusable
Selecting a main program	Unusable
Input/output of program	Usable

FANUC MACHINE SYSTEMS INC
Rob Zaherosky

11.1.3 Selecting a USB Memory as a Device

When a commercially available USB memory is inserted into the USB memory interface of the CNC, the CNC recognizes the USB memory as an external input/output device.

The following operations can be performed for a USB memory:
- Listing the files and folders in the USB memory
- Renaming and creating a file or folder in the USB memory
- Inputting and outputting an NC program between the CNC part program storage memory and USB memory

Procedure for selecting a device

1 Press the function key PROG.
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [DEVICECHANGE].
5 Press the soft key [USB MEM].

Limitation

There are the following restrictions on the USB function.
(1) Only one USB memory can be recognized. (No USB hub can be recognized.)
(2) Up to 32 characters can be used for a file or folder name.
If a file or folder name is longer than 32 characters, the 32nd character is replaced with a tilde (~) and the subsequent characters are omitted when displayed. All operations including management operation and input/output cannot be performed for a file or folder whose name is omitted with a tilde (~).
(3) The maximum number of folder levels is 6 (excluding the root folder).
(4) The maximum number of files and folders in a folder is 512.
(5) If a non-ASCII character (alphanumeric character) is used in a file or folder name, it cannot be displayed correctly.

For detailed specifications of and restrictions on a USB memory, see the section related to the USB function.

NOTE
1 Remove a USB memory only when “REMOVE OK” is displayed as the USB memory status, or data may be damaged.
2 When the free space of the USB memory is a little or when a large amount of file is in the USB memory, it might take time to operate the USB memory. In the default configuration, warning "TIME-OUT" is displayed after 30 seconds passed, and the operation of the USB memory stops. When the period of the time-out is changed, set parameter No.11376 .

- Operation of creation, edition, and management of a program

When “USB memory as a device” is selected, operation of creation, edition, and management of a program is below (Table 11.1.3):

Table 11.1.3

Item	Usable
Creation of a program	Unusable
Edition prohibition attribute	Unusable

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Item	Usable
Inserting, alteration, and deletion a Word	Unusable
Deletion of a block	Unusable
Program search	Usable
Sequence number search	Unusable
Deletion of a program	Usable
Editing a custom macro	Unusable
Password function	Unusable
Editing a program character	Unusable
Program and folder copy/move function	Usable (Note)
Key and program encryption	Unusable
Selecting a device	Usable
Creation of a folder	Usable
Renaming of a folder	Usable
Changing the attribute of a folder	Unusable
Deletion of a folder	Usable
Selecting a default folder	Unusable
Renaming of a file	Usable
Deletion of a file	Usable
Changing the attribute of a file	Unusable
Selecting a main program	Unusable
Input/output of program	Usable

NOTE
In USB device, programs are copied from USB device to CNC built-in memory device or from CNC built-in memory device to USB device.

11.2 CREATING A FOLDER

This section explains the procedure for creating a folder.

Procedure for creating a folder

1 Select EDIT mode.
2 Press the function key [PROG].
3 Move to the folder in which you want to create a folder.
Use the cursor keys [↓] and [↑] to move among folders. After selecting the folder, press the [INPUT] key.
4 Press the soft key [(OPRT)].
5 Type folder name.
6 Press the soft key [CREATE FOLDER].

NOTE
1 Each folder name must be unique within the same folder.
2 Each time a folder is created, the number of programs that can be registered decreases by one.
3 Depending on the operation status and protection status, a folder cannot sometimes be created.

FRYER MACHINE SYSTEMS Inc
Rob Zahensky

11.3 RENAMING A FOLDER

This section explains the procedure for renaming a folder.

Procedure for renaming a folder

1 Select EDIT mode.
2 Press the function key [PROG].
3 Press the soft key [FOLDER].
4 Select the folder that you want to rename.
  To select a folder, use the cursor keys [↓] and [↑].
5 Press the soft key [(OPRT)].
6 Type new folder name.
7 Press the soft key [RENAME].

NOTE
1 You cannot rename the initial folders.
2 Each folder name must be unique within the same folder.
3 Depending on the operation status and protection status, a folder cannot sometimes be renamed.

11.4 CHANGING CURRENT FOLDER

Current folder is selected folder in program contents display and program folder screen.

Procedure for changing current folder in folder screen

Changing to the upper folder

1 Press the function key [PROG].
2 Press the soft key [FOLDER].
3 Move the cursor to “RETURN TO UPPER FOLDER”.
4 Press the MDI key [INPUT].

Changing to the under folder

1 Press the function key [PROG].
2 Press the soft key [FOLDER].
3 Move the cursor to the folder to select.
4 Press the MDI key [INPUT].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Procedure for changing current folder in folder tree display.

In folder tree display, current folder is changed by moving cursor on the tree.

Changing to the upper folder

1 Press the function key [PROG].
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [TREE LIST].
5 Press the cursor key [←].

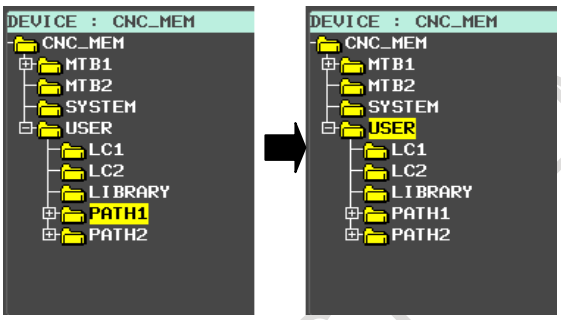


Fig.11.4 (a)

Changing to the under folder

1 Press the function key [PROG].
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [TREE LIST].
5 Press the cursor key [→].

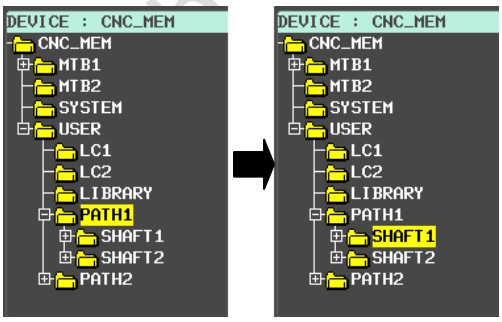


Fig.11.4 (b)

FRYER MACHINE SYSTEMS INC
Zahensky

Changing to the next folder

1 Press the function key [PROG].
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [TREE LSIT].
5 Press the cursor key [↑] or [↓].

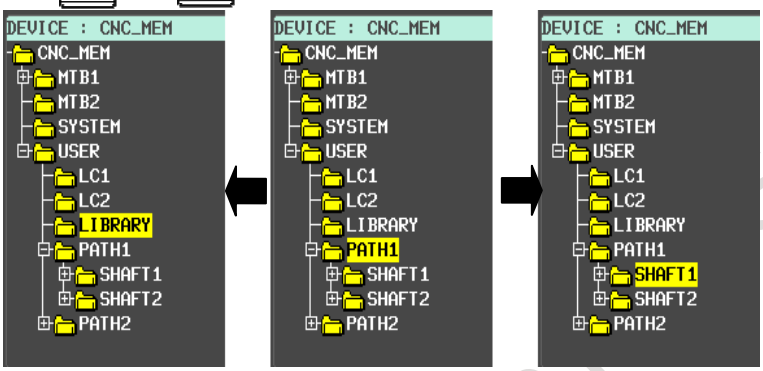


Fig.11.4 (c)

11.5 CHANGING FOLDER ATTRIBUTES

This section explains the procedure for changing the attribute of a folder (edit disable or edit/display disable).

Procedure for changing folder attributes

1 Select EDIT mode.
2 Press the function key [PROG].
3 Press the soft key [FOLDER].
4 Select the folder of which attribute is to be changed.
  To select a folder, use the cursor keys [↓] and [↑].
5 Press the soft key [(OPRT)].
6 Press the soft key [DETAILON].
7 Press the soft key [CHANGE ATTR].
  - To disable editing, press the soft key [EDIT DISABL].
  - To enable editing, press the soft key [EDIT ENABLE].
  - To disable editing and display, press the soft key [DISP DISABL].
  - To enable editing and display, press the soft key [DISP ENABLE].

NOTE
1 Depending on the operation status and protection status, the attribute of a folder cannot sometimes be changed.
2 When the edit disable attribute is set for a folder, editing of folders and files in that folder is disabled.
3 When the edit/display disable attribute is set for a folder, editing and display of folders and files in that folder is disabled, and these folders and files are not displayed.
4 The items that can be set vary depending on the status of parameters and so on.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

11.6 DELETING A FOLDER

This section explains the procedure for deleting a folder.

Procedure for deleting a folder

1 Select EDIT mode.
2 Press the function key [PROG].
3 Press the soft key [FOLDER].
4 Select the folder that you want to delete.
  To select a folder, use the cursor keys [↓] and [↑].
5 Press the soft key [(OPRT)].
6 Press the soft key [DELETE].
  - To perform the deletion, press the soft key [EXEC].
  - To cancel the deletion, press the soft key [CANCEL].

NOTE
1 You cannot delete the initial folders.
2 A folder cannot be deleted unless the folder is empty.
  (An empty folder means that the folder contains neither folders nor files.)
3 If a folder contains a folder or file having the edit/display disable attribute, the folder may seem to be empty when displayed, but the folder is not actually empty, so it cannot be deleted.
4 Depending on the operation status and protection status, a folder cannot sometimes be deleted.

11.7 SELECTING A DEFAULT FOLDER

This section explains the procedure for selecting a foreground or background default folder.
What is a default folder is described in "PROGRAM MANAGEMENT" in PROGGRAMMING.

Procedure for selecting a default folder

1 Select EDIT mode.
2 Press the function key [PROG].
3 Press the soft key [FOLDER].
4 Move to the folder that you want to select as the default folder.
  Use the cursor keys [↓] and [↑] to move among folders. After selecting the folder, press the [INPUT] key.
5 Press the soft key [(OPRT)].
6 - To select the foreground, press the soft key [FORE CHANGE].
  - To select the background, press the soft key [BACK CHANGE].

NOTE
1 When the foreground or background default folder is not set, the path folder, which is an initial folder, is assumed.

FRYER MACHINE SYSTEMS INC
Ron Zahensky

NOTE
2 The settings for the foreground and background default folders are stored in the default folder setting file.
3 When a program file, program folder, or program folder management file is cleared, the default folder setting file is also cleared at the same time.

11.8 RENAMING A FILE

This section explains the procedure for renaming a file.

Procedure for renaming a file

1 Select EDIT mode.
2 Press the function key [PROG].
3 Press the soft key [FOLDER].
4 Move to the folder containing the file that you want to rename.
Use the cursor keys [↓] and [↑] to move among folders. After selecting the folder, press the [INPUT] key.
5 Select the file that you want to rename.
To select a file, use the cursor keys [↓] and [↑].
6 Press the soft key [(OPRT)].
7 Type new file name.
8 Press the soft key [RENAME].

NOTE
1 Each file name must be unique within the same file.
2 When the assigned file name cannot be treated as a program number, the program in the file is restricted as follows:
- Specification by program number is impossible.
(Such as a subprogram call)
- Information output by program number is impossible.
3 Depending on the operation status and protection status, a file cannot sometimes be renamed.

11.9 DELETING A FILE

This section explains the procedure for deleting a file.

Procedure for deleting a file

1 Select EDIT mode.
2 Press the function key [PROG].
3 Press the soft key [FOLDER].
4 Move to the folder containing the file that you want to delete.
Use the cursor keys [↓] and [↑] to move among folders. After selecting the folder, press the [INPUT] key.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

5 Select the file that you want to delete.

To select a file, use the cursor keys ↓ and ↑.

6 Press the soft key [(OPRT)].

7 Press the soft key [DELETE].

- To perform the deletion, press the soft key [EXEC].
- To cancel the deletion, press the soft key [CANCEL].

NOTE

Depending on the operation status and protection status, a file cannot sometimes be deleted.

11.10 CHANGING FILE ATTRIBUTES

This section explains the procedure for changing the attribute of a file (edit disable, edit/display disable, encoding, or protection of data at eight levels).

Procedure for selecting the attribute of a file

1 Select EDIT mode.

2 Press the function key PROG.

3 Press the soft key [FOLDER].

4 Move to the folder containing the file of which attribute is to be changed.

Use the cursor keys ↓ and ↑ to move among folders. After selecting the folder, press the INPUT key.

5 Select the file of which attribute is to be changed.

To select a file, use the cursor keys ↓ and ↑.

6 Press the soft key [(OPRT)].

7 Press the soft key [DETAILON].

8 Press the continuous menu key ▷ until the desired soft key appears.

9
- To disable editing, press the soft key [EDIT DISABL].
- To enable editing, press the soft key [EDIT ENABLE].
- To disable editing and display, press the soft key [DISP DISABL].
- To enable editing/display, press the soft key [DISP ENABLE].
- To set encoding, press the soft key [ENCODESET].
- To cancel encoding, press the soft key [ENCODE RESET].
- To change the change protection level, type a change protection level, then press the soft key [CHANGELEVEL].
- To change the output protection level, type an output protection level, then press soft key [OUT LEVEL].

NOTE

1 Depending on the operation status and protection status, a file cannot sometimes be deleted.

2 The items that can be set vary depending on the status of options, parameters, and so on.

3 To change the display disable attribute, release protection of programs in advance. Set the same value as the password (parameter No. 3210 (PSW)) in parameter No. 3211 (KEY) to release the protection.

FRYER MACHINE SYSTEMS INC Rob Zahensky

11.11 SELECTING A MAIN PROGRAM

This section explains the procedure for selecting a main program.

Procedure for selecting a main program

1 Select EDIT mode.

2 Press the function key [PROG].

3 Press the soft key [FOLDER].

4 Move to the folder containing the file that you want to use as a main program.

Use the cursor keys [↓] and [↑] to move among folders. After selecting the folder, press the [INPUT] key.

5 Select the file that you want to use as the main program.

To select a file, use the cursor keys [↓] and [↑].

6 Press the soft key [(OPRT)].

7 Press the soft key [MAIN PROGRM].

"@" is displayed at the left of the program that has been selected as the main program. (Fig.11.11)

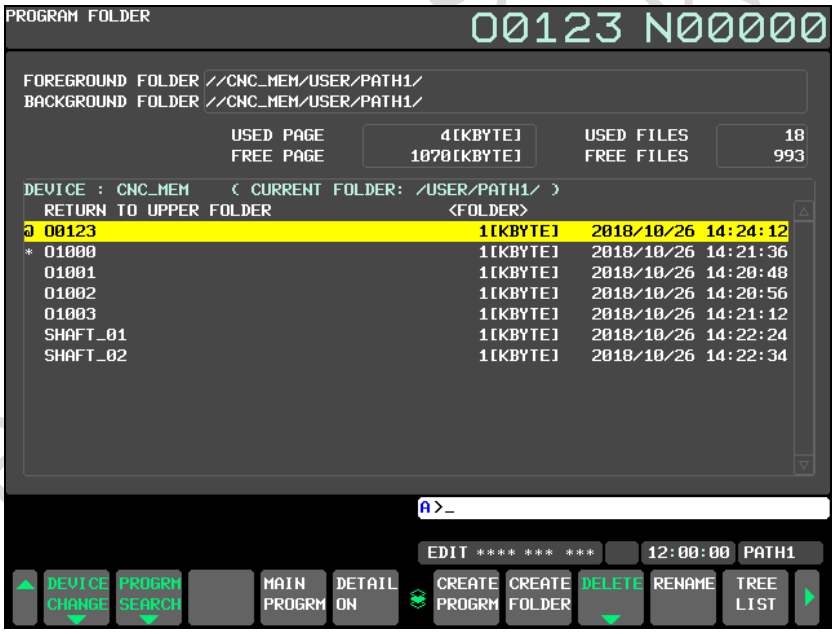


Fig.11.11 Selecting a main program (Program folder screen)

NOTE

1 Depending on the operation status and protection status, the main program cannot sometimes be selected.

2 In the multi-path system, "*" is displayed at the left of the program that has been selected as the main program in other path. (Fig.11.11 (a))

FRYER SYSTEMS INC

11.12 MAKING A PROGRAM COMPACT

This section explains the procedure for making a program compact.

Procedure for selecting a main program

1 Select EDIT mode.
2 Press the function key [PROG].
3 Press the soft key [FOLDER].
4 Move to the folder containing the file of the program that you want to make compact.
Use the cursor keys [↓] and [↑] to move among folders. After selecting the folder, press the [INPUT] key.
5 Select the file of the program that you want to make compact.
To select a file, use the cursor keys [↓] and [↑].
6 Press the soft key [(OPRT)].
7 Press the soft key [PROGRM CNDENS].
When a program is made compact, the program size is updated.

NOTE
1 Depending on the operation status and protection status, a program cannot sometimes be made compact.
2 Only programs on the CNC_MEM device can be made compact.
3 If bit 2 (CND) of parameter No. 3202 is set to 1, soft key [PROGRM CNDENS] is displayed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

11.13 PROGRAM AND FOLDER COPY/MOVE FUNCTION

Overview

Programs and folders can be copied or moved from folder to folder.

Procedure for copying a program or a folder

1. Press the function key [PROG].
2. Press the chapter selection soft key [FOLDER].
   The program folder screen in Fig.11.13 appears:

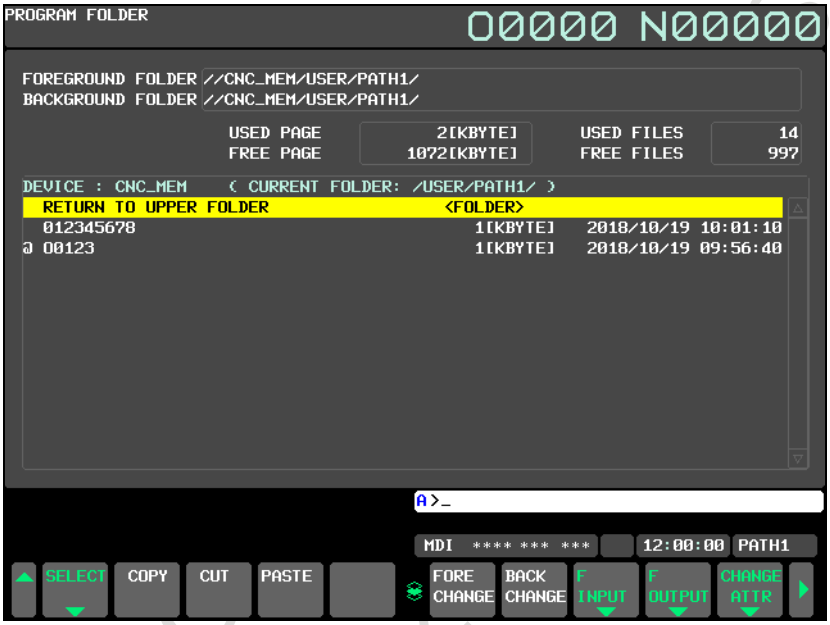


Fig.11.13 Program folder screen

3. Display the folder from which you want to copy or move the program or the folder. Move the cursor to the folder on the screen and press the key [INPUT] to move to the folder. Move the cursor to "RETURN TO UPPER FOLDER" on the screen and press the key [INPUT] to move the upper folder.
4. Press the soft key [(OPRT)].
5. Press the continuous menu key [▷] until the soft key [COPY] appears.
6. Move the cursor to the program or the folder that you want to select.
7. Press the soft key [COPY]. The program or folder is selected, and the background color becomes selected color (default: blue). If you want to move the program or the folder, press the soft key [CUT].
8. Display the folder to which you want to copy or move the program or folder.
9. Press the soft key [PASTE]. If you want to rename the program or folder, type new name before pressing [PASTE].
10. To copy a file or folder from the USB memory to the Data Server, press soft key [PROGRAM] or [DATA] to specify the attribute (program or data) of the copy destination file after pressing soft key [PASTE] of procedure 9. (Note 1) To rename and copy the file or folder, enter the new file or folder name in the key-in buffer before pressing soft key [PROGRAM] or [DATA].

FRYER MACHINES INC Rob Z

Procedure for copying some programs or folders

Selection by range

1. Press the function key PROG.
2. Press the chapter selection soft key [FOLDER].
3. Display the folder from which you want to copy or move the program or the folder. Move the cursor to the folder on the screen and press the key INPUT to move to the folder. Move the cursor to "RETURN TO UPPER FOLDER" on the screen and press the key INPUT to move the upper folder.
4. Press the soft key [(OPRT)].
5. Press the continuous menu key ▷ until the soft key [SELECT] appears.
6. Press the soft key [SELECT].
7. Press the soft key [RANGE SELECT]. In addition, move the cursor up and down to select the programs and the folders. The background color of selected programs and folders becomes selected color (default: blue).
8. If the selected range is determined, press the soft key [COPY]. If you want to move the program or folder, press the soft key [CUT].
9. Change the current folder to the destination folder.
10. Press the soft key [PASTE].
11. To copy a file or folder from the USB memory to the Data Server, press soft key [PROGRAM] or [DATA] to specify the attribute (program or data) of the copy destination file.(Note 1)

Selection by each program or folder

1. Press the function key PROG.
2. Press the chapter selection soft key [FOLDER].
3. Display the folder from which you want to copy or move the program or the folder. Move the cursor to the folder on the screen and press the key INPUT to move to the folder. Move the cursor to "RETURN TO UPPER FOLDER" on the screen and press the key INPUT to move the upper folder.
4. Press the soft key [(OPRT)].
5. Press the continuous menu key ▷ until the soft key [COPY] appears.
6. Press the soft key [SELECT].
7. Move the cursor to program or folder to be copied.
8. Press the soft key [SELECT].
9. Repeat 7th and 8th operation until select is ended.
10. When selected programs or folders are determined, press the soft key [COPY]. If you want to move the program or folder, press the soft key [CUT].
11. Change the current folder to the destination folder.
12. Press the soft key [PASTE].
13. To copy a file or folder from the USB memory to the Data Server, press soft key [PROGRAM] or [DATA] to specify the attribute (program or data) of the copy destination file.(Note 1)

> ⚠ **WARNING**
> When a file containing multiple programs is copied from the USB memory to the Data Server, the file is not divided into the programs and the programs are not registered individually on the Data Server. The programs are registered as one series of programs. Be sure to copy a file containing one program. Operating the machine after copying a file containing multiple programs is dangerous because the machine may behave unexpectedly.

FRIER MACHINE SYSTEMS INC Rob Zahenskyj

NOTE

1 The program (text file) attribute can be specified only when bit 1 (DUC) of parameter No. 24309 is set to 1. On the Data Server, various types of data can be saved in addition to text format NC programs. To handle text format NC programs, set the program (text file) attribute. To handle NC programs for NC data other than text format NC programs, set the data (binary file) attribute.
2 To copy multiple files from the USB memory to the Data Server, do not select program files and data files simultaneously. When multiple files are selected, the specified attribute is set for all selected files. When a folder is selected, the specified attribute is also set for all files in the folder.
3 If the system is placed in other than the EDIT mode or emergency stop state during copy operation between the Data Server and USB memory, the copy operation is stopped. Do not change the mode during copy operation.
4 Files or folders may not be able to be copied, depending on the status such as operation state or protection state.
5 Programs or folders up to 10 can be selected.

About same name programs and folders

When the same name of program or folder exists at the move destinations, confirmation message “OVERWRITE? (program or folder name)” is displayed.

- When the soft key [YES] is pressed, the program/folder is overwritten. If the same name programs/folders appear again, the confirmation message is displayed.
- When the soft key [NO] is pressed, copying or moving operation of the program/folder is skipped. If the same name programs/folders appear again, the confirmation message is displayed.
- When the soft key [YES ALL] is pressed, the program/folder is overwritten. Even if the same name programs/folders appear again, they are also overwritten.
- When soft key [NO ALL] is pressed, copying or moving operation of program/folder is skipped. If the same name programs/folders appear again, they are also skipped.
- When the soft key [CANCEL] or [<] is pressed, copying or moving operation is canceled.

During the overwrite confirmation is displayed and the screen is changed, the move operation is canceled.

Deselecting programs/folders

When either of the following operations is performed, the selection state of program/folder is released.

- Pressing of soft key [CANCEL] after performing a selecting operation.
- Change of the source folder before the selection state is fixed.

Restrictions

If the following conditions are corresponded, copy and move operation is invalid.

- The program data protection is enable.(The memory protect signal KEY3 is OFF)
- The top of the origin folder of copy/move or destination folder of copy/move has the edit disable attribute.
- The copy and move operation is performed to sub-folder of the copy/move origin folder.
- The copy destination program storage capacity is insufficient.

Programs and folders, which correspond to the following conditions, cannot be copied and moved.

- The copy/move origin or destinations programs/folders are protected by the edit disable attribute.
- The copy/move origin or destinations programs/folders are protected by 8 level data protection function.
- The copy/move destinations program is selected as a main program.
- The depth of the path is exceeded 6 classes after the folders are copied/moved.

FRYER MACHINE SYSTEMS INC Rob Zahensky

11.13.1 Copy and movement between different devices

Overview

It is enable to copy and move the files between different devices.
The files can be copied/moved in a same way.

Restrictions

About file operation restrictions on each device

A copy or movement between the following devices is possible.

Table11.13.1 Copy and movement between different devices

		Destination			
		CNC built-in memory	Data server	Memory card	USB
Origin	CNC built-in memory	A	A	B	B
	Data server	A	A	C	A
	Memory card program	A	A	C	C
	Memory card	B	C	C	C
	USB	B	A	C	C

A : Files and folders can be copied and moved
B : Files can be copied and moved. Folders cannot be copied and moved.
(NOTICE: Memory card program device can be only copied.)
C : Programs and folders cannot be copied and moved.

About Memory card program, refer to "Selecting a Memory Card Program as a Device" in OPERATION.

NOTE

1 Memory card program device has following restrictions.
(a) Programs cannot be moved in memory card program device.
(b) Memory card program device cannot be used with memory card device.
2 Memory card device has following restrictions.
(a) Files cannot be moved.
(b) When the programs are copied, place all paths in the EDIT mode.
(c) When the programs are copied from CNC built-in memory device to memory card device, the behavior follows the setting of bit 1 (COW) of parameter No.11308.
(d) When the programs are copied from memory card device to CNC built-in memory device, the behavior follows the setting of bit 2 (REP) of parameter No.3201.
(e) Memory card device cannot be used with memory card program device.
3 USB device has following restrictions.
(a) Files cannot be moved.
(b) When the programs are copied, place all paths in the EDIT mode.
4 Copying between data server and USB device has following restrictions.
(a) Files cannot be moved.
(b)Please change all path to EDIT mode or emergency stop when you copy files.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Copy of file including some programs

Copy other than between Data server and USB memory

When the file including some program is copied, don't input the file name to the key input buffer.
When some programs are included in the file of the copy origin, if the character string is input to the key input buffer, only the program of the head in the file is read to CNC MEM with program name input to the key input buffer.

Copy between Data server and USB memory

Even if the file includes two or more programs, it is copied as one file. Please select the soft key [DATA] at the paste.

File name on memory card

When the program with 8 characters or more name of the program in the program memory on CNC is output, it is output on the memory card as follows.

For eight digits O numbers ('O'+ numeric parts)

- The numeric parts are 8 digits or more :
  The file name is changed to eight numerical value digits except first 'O'.
  Example) O12345678 → 12345678
- The numeric parts are less than 8 digits :
  One digit is deleted '0' and program name is changed to seven digits of 'O' + numerical value.
  Example) O00012345 → O0012345

For arbitrary program name (9 characters or more)

- Program name is changed to the file name that rounds down since the ninth character.
  Example) 123456789 → 12345678

The file name output on the memory card might overlap by the above operation. In this case, it is output according to the setting of bit 1 (COW) of parameter No.11308.

11.13.2 Copy of program file with memory card device

Overview

In the program list screen, the program file can be input and output between the memory card connected as an external input-output device and program memory (CNC MEM) on CNC by the same operation as the copy of the program file.

NOTE

1 This function targets the memory card by the PCMCIA interface connected as an external input-output device.
2 Change to the following parameter setting to copy the program file between the memory cards.
  - Parameter No.20-23 are set to 4.
  - Bit 0 (IO4) of parameter No.0110 is set to 0.

  When Bit 0 (IO4) of parameter No.0110 is set to 1, and sets the numbers of the input-output device other than the memory card to parameter No.20-23, warning " CHANGE EXTERNAL DEVICE TO MEMORYCARD" is occurred, and the file with the memory card cannot be copied.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

・ CNC mode when the copy is operated

Change to the EDIT mode when you copy the file between different devices.
Moreover, the multi path system must change the entire path to the EDIT mode.
Warning "CHANGE TO EDIT MODE" is occurred excluding the EDIT mode.

・Selection of file

The file is selected according to the program file selection operation that becomes effective because of the bit 4 (PCB) of parameter No.11374 is set to 0.
The file that can be selected as a copy target is only a file in the same folder.

・Copy of program file

When the program file is copied from the program memory to the memory card, the selected program is output to the memory card.
When the program file is copied from the memory card to the program memory, the selected file is read to the program memory.

・File that can be copied

The file that can be output to the memory card by the copy operation is only a program file on the CNC-MEM device. Therefore, the file that exists in other devices except the CNC-MEM and folder cannot be output.

NOTE
Even if the folder has been selected when outputting it to the memory card, the folder is not output.

・Overwrite of program file

The file is overwritten according to the program file overwriting operation that becomes effective because of the bit 4 (PCB) of parameter No.11374 is set to 0.

NOTE
Operation when the file is pasted on the memory card operates according to the setting of bit 1 (COW) of parameter No.11308.
Operation when the file is pasted on the program memory of CNC operates according to the setting of bit 2 (REP) of parameter No.3201.

Restrictions

・File operation restrictions on devices

A copy or movement between the following devices is possible.

- CNC MEM
- MEM CARD (binary format)
  (However, this device can only be selected as the copy source folder from which a file is read.)
- Data server
- Memory card
  (However, This device can do only the copy operation with the CNC MEM device.
  Moreover, this device cannot be used together with MEM CARD.)

For devices other than the above, a copy or movement between them is impossible.
If an attempt is made to perform a copy or movement, the warning message "CAN NOT COPY/MOVE" appears.

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

・Copy of file including some programs

Don’t input anything to the key input buffer when you copy the file including some programs.
When some programs are included in the file of the copy origin, if the character string is input to the key input buffer, only the program of the head in the file is read to CNC MEM with program name input to the key input buffer.

・File name on memory card

When the program with 8 characters or more name of the program in the program memory on CNC is output, it is output on the memory card as follows.

For eight digits O numbers ('O'+ numeric parts)

- The numeric parts are 8 digits or more：
  The file name is changed to eight numerical value digits except first 'O'.
  Example) O12345678 → 12345678
- The numeric parts are less than 8 digits：
  One digit is deleted '0' and program name is changed to seven digits of 'O' + numerical value.
  Example) O00012345 → O0012345

For arbitrary program name (9 characters or more)

- Program name is changed to the file name that rounds down since the ninth character.
  Example) 123456789 → 12345678

The file name output on the memory card might overlap by the above operation. In this case, it is output according to the setting of bit 1 (COW) of parameter No.11308.

11.13.3 Deleting files in memory card device

Overview

The file on the memory card can be deleted according to the program file deletion operation that becomes effective because of the bit 4 (PCB) of parameter No.11374 is set to 0.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

11.14 FOLDER MANAGEMENT

This function restricts the folder operation in program management. As a result, operator can manage the program without the complex folder operation. Following two specifications can be selected by the parameter.

1) The folders under the path folders can be used for program management.
2) Only the path folders can be used for program management.

Path folders are the one of the initial folders. The folder exists under //CNC_MEM/USER, and the number is same to the system path. Please refer to "PROGRAM MANAGEMENT" in PROGRAMMING for details.

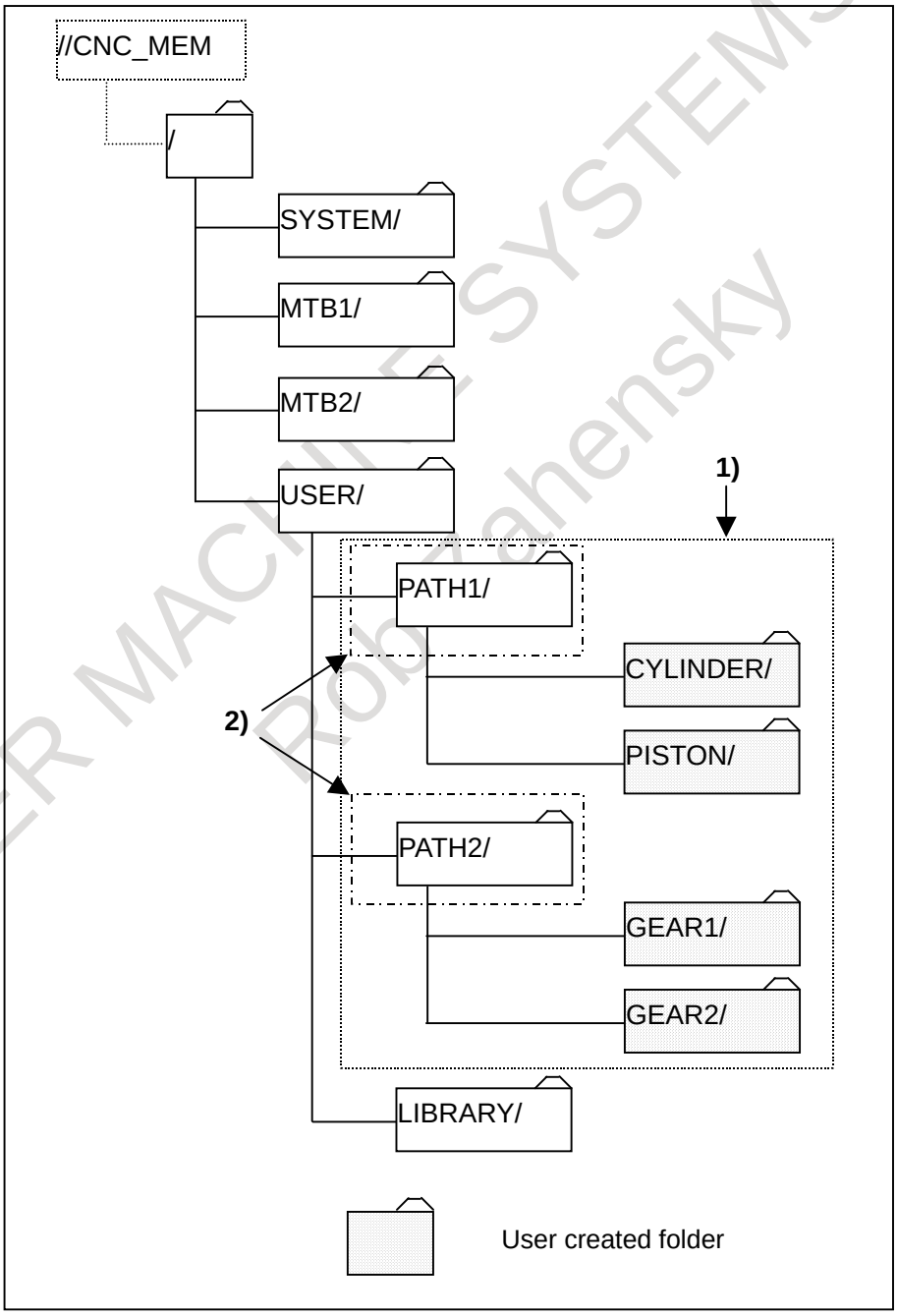


Fig. 11.14 Folder that can be used by each setting

FRYER MACHINE SYSTEMS INC
Rob Fahensky

11.14.1 Program Management under the Path Folder

If parameter FPF (No.11302#6) is set to 1, the operation on the program folder screen becomes as follows. At this time, it is necessary to set parameter CFP (No.11304#7) to 0.

Table 11.14.1 Restriction of operation

Operation		Working
Moving of the folder	Under the path folder	Possible
	Upper from the path folder	Impossible
Creating of user folder		Possible
Changing of foreground default folder		Possible
Changing of background default folder		Possible

Display on program folder screen

Only the folder under the path folder corresponding to selected path can be displayed.
Warning message “CAN’T MOVE TO FOLDER” is displayed and cannot move when the folder is operated to move upper from the path folder.

Setting of foreground/background default folder

Only the folder under the path folder corresponding to selected path can be set to foreground/background default folder.

11.14.2 Program Management only in the Path Folder

If parameter CFP (No.11304#7) is set to 1, the operation on the program folder screen becomes as follows. At this time, the setting of parameter FPF (No.11302#6) is invalid.

Table 11.14.2 Restriction of operation

Operation		Working
Moving of the folder	Under the path folder	Impossible
	Upper from the path folder	Impossible
Creating of user folder		Impossible
Changing of foreground default folder		Impossible
Changing of background default folder		Impossible

Display on program folder screen

Only the path folder corresponding to selected path can be displayed. “RETURN TO UPPER FOLDER” is not displayed. (Refer to “1)” in Fig. 11.14.2 (a) and Fig. 11.14.2 (b).)
The subfolder that has already been made under the path folder before the parameter CFP(No.11304#7) is made effective can be displayed. But it is impossible to move to the subfolder.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

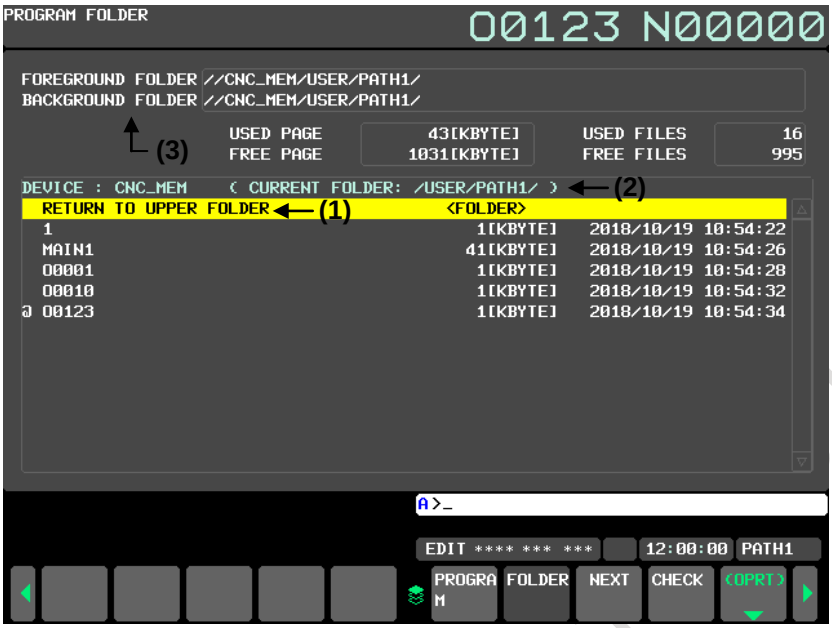


Fig. 11.14.2 (a) Program folder screen (parameter CFP(No.11304#7)=0)

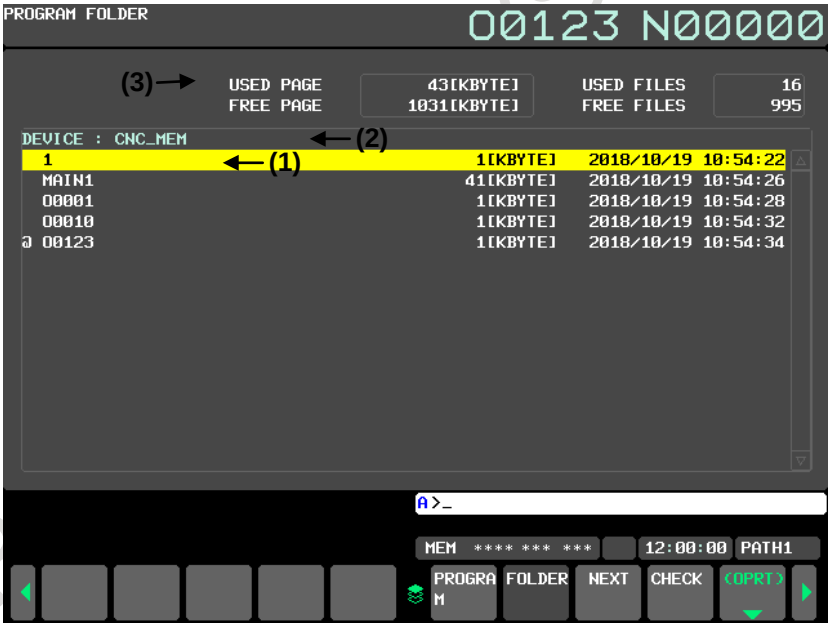


Fig. 11.14.2 (b) Program folder screen (parameter CFP(No.11304#7)=1)

Display of device name and current folder

The current folder name is not displayed though the device name is displayed so far. (Refer to “2)” in Fig. 11.14.2 (a) and Fig. 11.14.2 (b).)

Display and setting of foreground/background default folder

Foreground/background default folders are not displayed. (Refer to “3)” in Fig. 11.14.2 (a) and Fig. 11.14.2 (b).)

Moreover, foreground/background default folders cannot be set.

> ⚠ **CAUTION**
> Set foreground/background default folder to each path folder before the parameter CFP(No.11304#7) is made effective.

Soft key that is not displayed

The following soft key is not displayed, and cannot be operated.

- CREATE FOLDER 1)
- FORE CHANGE 2)
- BACK CHANGE 3)

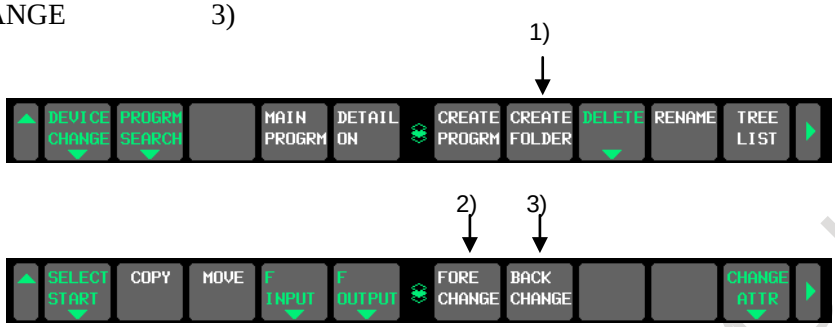


Fig11.14.2 (c) Soft key display for CNC_MEM device

11.14.3 Folder for Subprogram/Macro Calls

In the following subprogram call/macro call, the folder /USER/LIBRARY, /MTB12, /MTB2 and /SYSTEM are searched before the folder where there is the main program are searched.
There is a possibility being executed for the subprogram that the operator doesn't intend if the subprogram is registered to the above-mentioned folder even if the display and the operation are limited by this function.

- Subprogram call based on an M code
- Subprogram call based on a particular address
- Subprogram call based on a second auxiliary function code
- Macro call based on an S code
- Macro call based on a T code
- Macro call based on a G code
- Macro call based on an M code
- One-touch macro call

By setting 1 to the parameter SCC (No.3457#6), the folder where there is the main program is added to the top of search order.

1) Folder where the main program is stored. ← Added.
2) Common program folder of initial folder
3) MTB-dedicated folder 2 of initial folder
4) MTB-dedicated folder 1 of initial folder
5) System folder of initial folder

The folders of 2) though 5) can be excluded from search target folders by setting the parameters LIB(No.3457#0), MC2(No.3457#1), MC1(No.3457 #2), and SYS(No.3457#3).

FRYER MACHINE SYSTEMS INC
Bob Zahensky

11.15 PROGRAM VERIFICATION

This function enables to verify the differences of the programs in the I/O device and the programs in the CNC built-in memory.

Sequence of program verification

1. Make sure the I/O device is ready for input and output.
2. Press the EDIT switch on the machine operator's panel.
3. The program data protection is enabled on the machine operator's panel.(Turn off the memory protect signal KEY3)
4. Press the function key [PROG] and display the program list screen.
5. Press the soft key [(OPRT)].
6. Press the continuous menu key [▷].
7. Press the soft key [F INPUT].
8. Set the file name in the I/O devices.
   Input the file name and press the soft key [F NAME].
   If the file name is not set, "ALL-PROG.TXT" is set.
9. Set the program name in the NC memory.
   Input program name and press the soft key [P SET].
   If the program name is not specified, the program names in the file which is set in above '8' are set.
10. Press the soft key [EXEC].
    Program verification is started and the "COMPARE" is blinked in lower right. When verification finishes, "COMPARE" disappears.
    If the program in the I/O device and the program in the CNC built-in memory are same, the message "PROGRAM HAS BEEN VERIFIED" is displayed.
    If they are different, the alarm (PS0079) "PROGRAM NOT MATCH" is issued

NOTE
If 8 level data protection function is valid, this function cannot be used.

Explanation

When one program is registered in a file

If one program is registered in the file, the program in the file and the program that is specified by [P SET] are verified.
If the program in the CNC built-in memory is not specified by [P SET], the program whose program name is same as program in the file are verified.

When some programs are registered in a file

Each program in the file which is specified by [F SET] and the program whose program name is same as the program name in the file are verified. When all verifications succeed, the message "PROGRAM HAS BEEN VERIFIED" is displayed.

NOTE
When the program is specified by [P SET], the alarm (PS0079) is issued as long as all of programs in the file are same as the program which is set by [P SET].

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

11.16 HIGH-SPEED PROGRAM MANAGEMENT

When many kinds of machining are arranged, this function can achieve the short processing time when many programs are registered collectively or all programs are deleted. The program is not saved automatically to nonvolatile memory at registration, modification, or deletion of program.

If the power supply is turned off without saving, the changed programs data is not saved. Please execute following sequence of program save when the programs are saved.

Sequence of program save

1. The program edit screen or the program folder screen is displayed.
2. Press soft key [(OPRT)] and the continuous menu key several times. Soft key [ALL SAVE] appears.

   [Program edit screen]

   [Program folder screen]

3. Press soft key [ALL SAVE]. The following softkey is displayed.

4. When the soft key [EXEC] is pressed, the saving program is begun. When the soft key [CAN] is pressed, the saving program is discontinued.
5. While saving changes, the status "ALLSAVE" flashes.
6. When the status "ALLSAVE" disappears, saving is completed.

NOTE

1 There are no changes about the method of displaying programs and the editing programs.

2 In cases that this function is enabled, if the power supply is turned off without saving, the changed programs data is not saved.
The program selection by the following operation also is not saved.
- MDI operation
- External workpiece number search
- External program number search

Please perform saving operation by program edit screen, program folder screen, signal or the application by FOCAS2/ C Language Library, if it is necessary to preserve the changed programs data/ the program selection after the power supply is turned off.

3 When the automatic operation is executing in any path, the save operation is not executed.

4 If the power supply is turned off during saving the programs, all programs are deleted. In this case, an alarm PS0519, “PROGRAM FILES ARE BROKEN AND CLEARED” occurs when the power supply is turned on next.

5 This function is effective only on CNC program storage memory (CNC_MEM device).

FRYER MACHINE SYSTEMS INC
Rob Zaherky

12 SETTING AND DISPLAYING DATA

To operate a CNC machine tool, various data must be set on the MDI unit for the CNC. The operator can monitor the state of operation with data displayed during operation.
This chapter describes how to display and set data for each function.

Explanation

- Screen transition chart

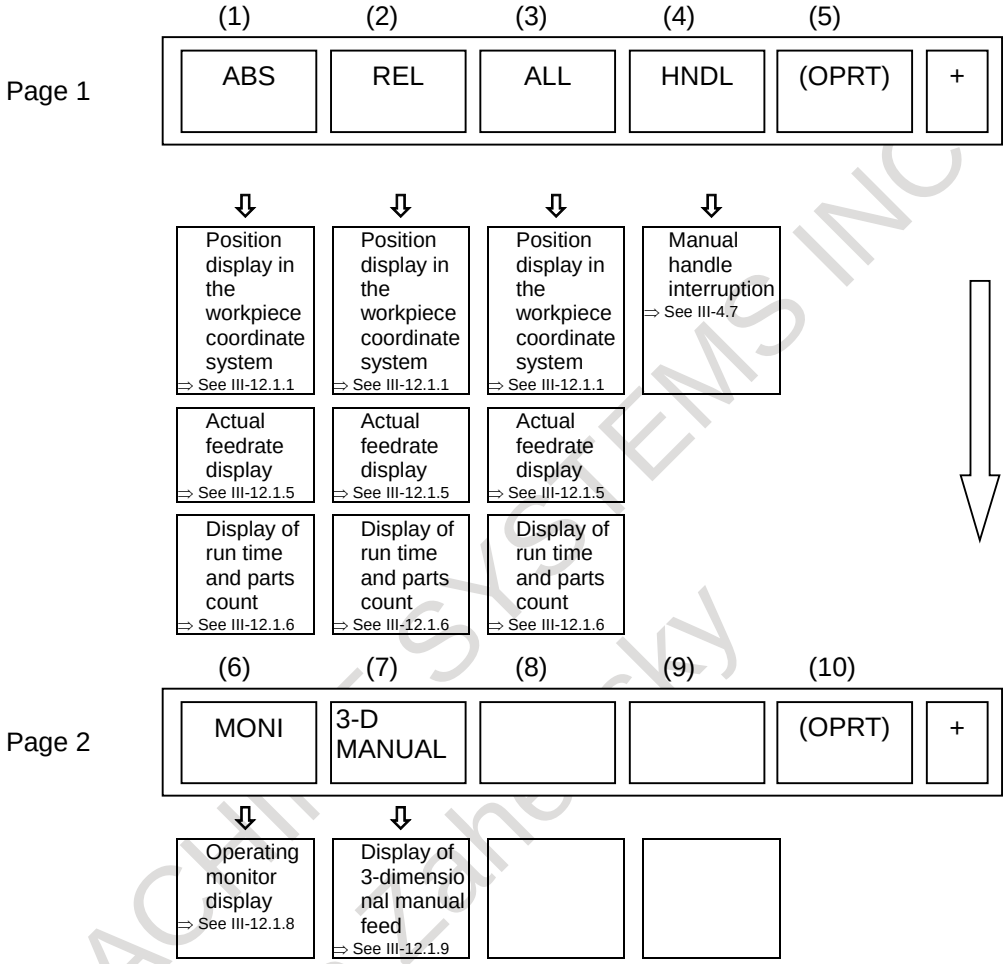
The screen transition for when each function key on the MDI unit is pressed is shown below. The subsections referenced for each screen are also shown. See the appropriate subsection for details of each screen and the setting procedure on the screen. See other chapters for screens not described in this chapter.
See Chapter, “ALARM AND SELF-DIAGNOSIS FUNCTIONS” for the screen that appears when function key MESSAGE is pressed. In general, function key CUSTOM1 or CUSTOM2 is prepared by the machine tool builder and used for macros. Refer to the manual issued by the machine tool builder for the screen that appears when function key CUSTOM1 or CUSTOM2 is pressed.

- Data protection key

The machine may have a data protection key to protect part programs, tool compensation values, setting data, and custom macro variables. Refer to the manual issued by the machine tool builder for where the data protection key is located and how to use it.

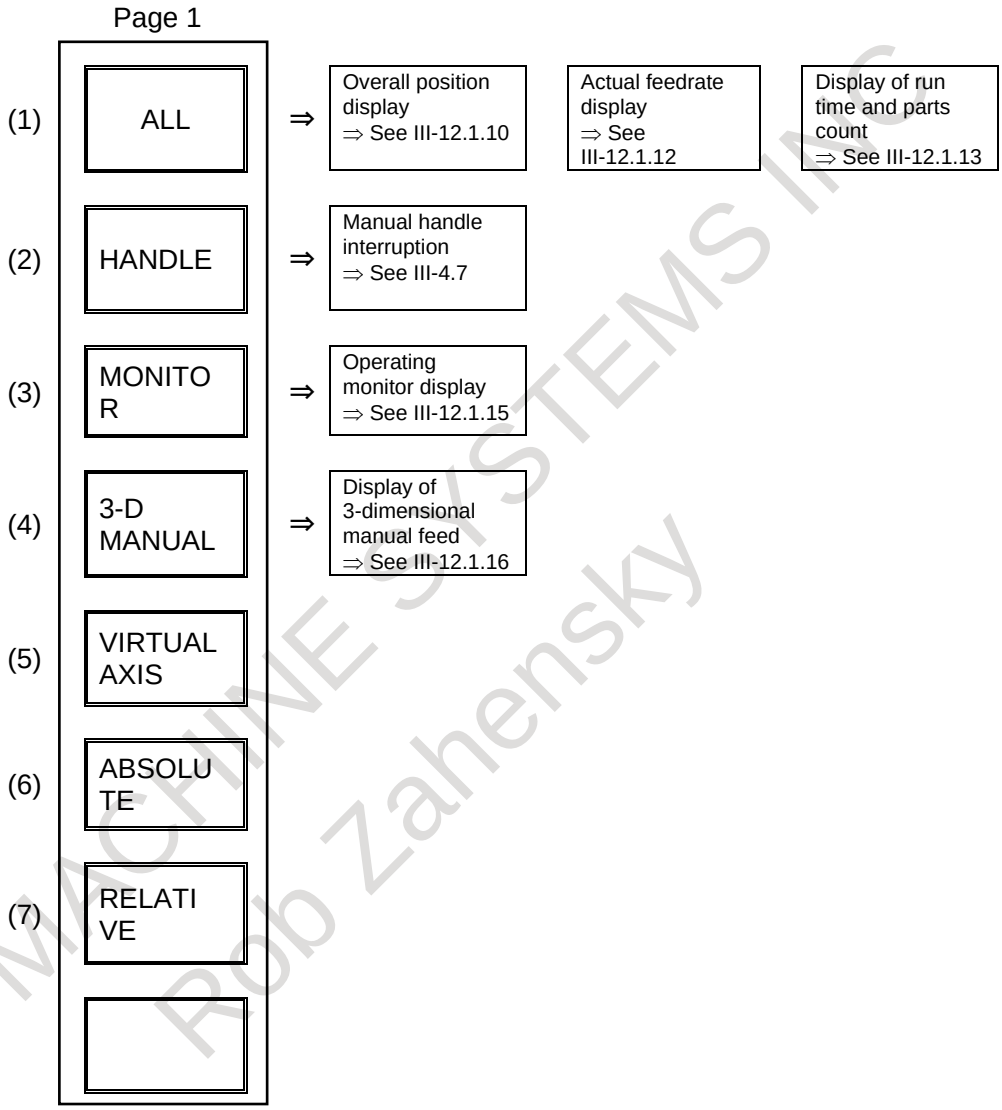
FRYER MACHINE SYSTEMS INC
Rob Zahensky

Screen displayed when the function key POS is pressed (for 8.4/10.4-inch display unit)



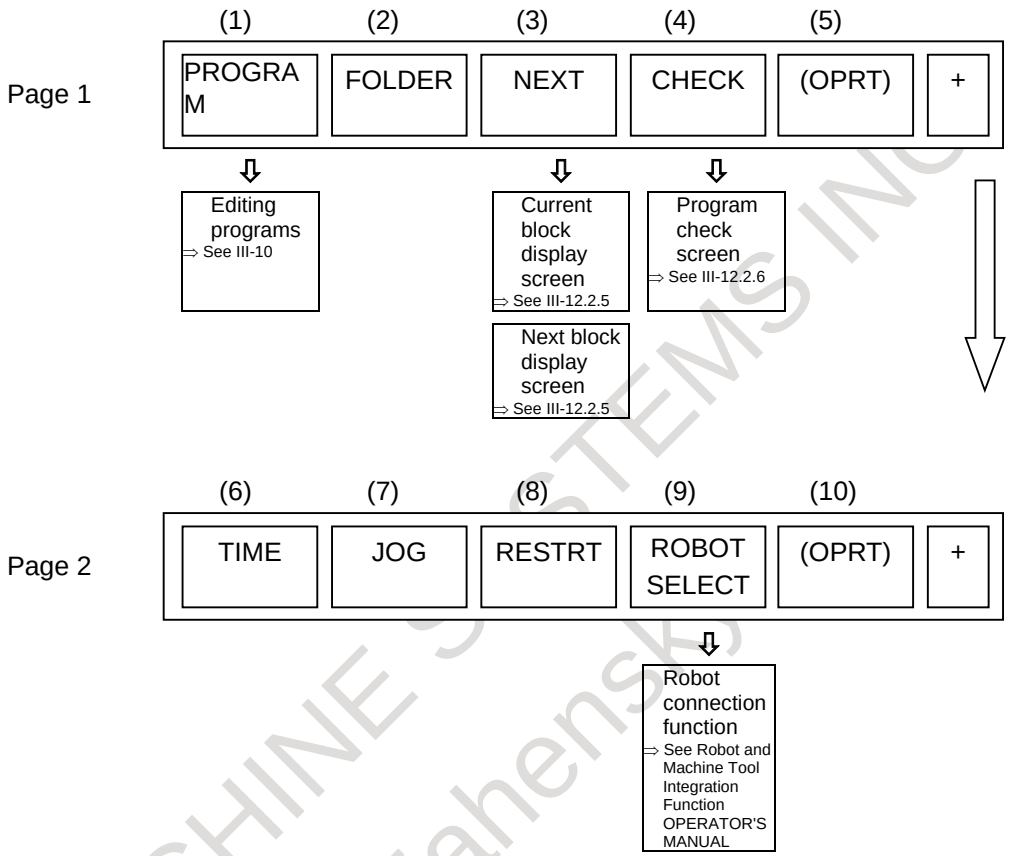
FRYER MACHINE SYSTEMS INC
Robert Zahansky

Screen displayed when the function key POS is pressed (for 15/19-inch display unit)



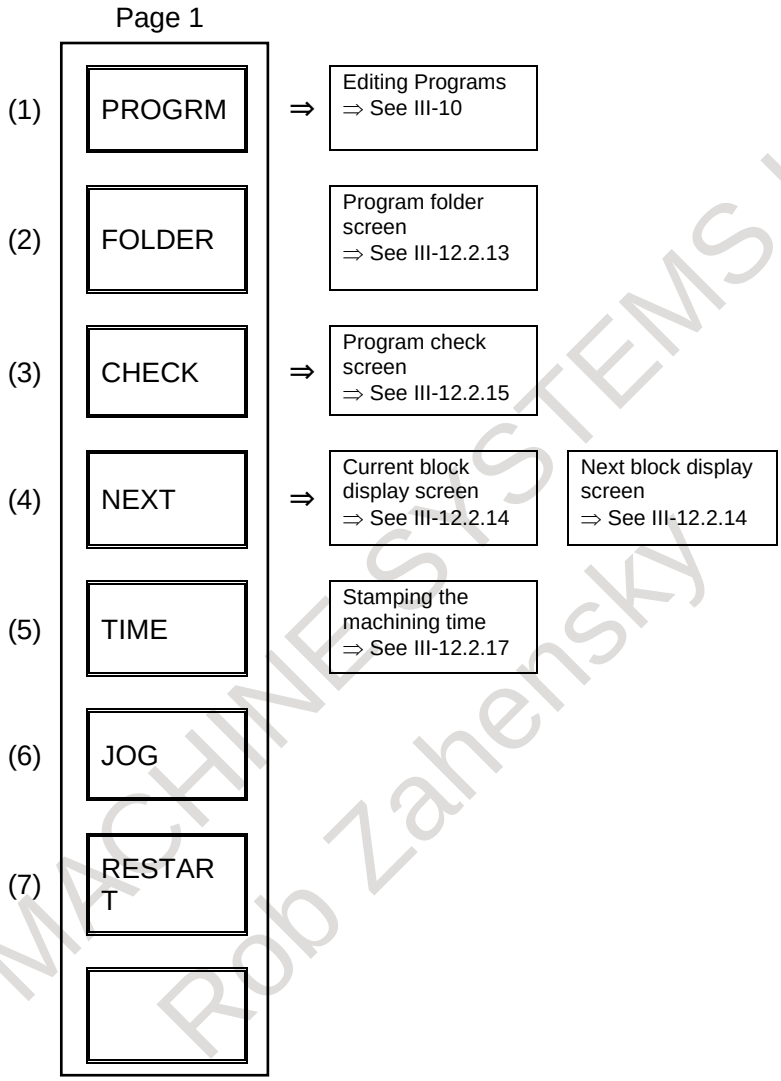
FRYER MACHINE SYSTEMS INC
Rob Zahensky

Screen displayed when the function key PROG is pressed (for 8.4/10.4-inch display unit)



FRYER MACHINE SYSTEMS INC
Rob Zahenski

Screen displayed when the function key PROG is pressed (for 15/19-inch display unit)



FRYER MACHINE SYSTEMS INC
Rob Zahensky

Page 2

(8) ROBOT SELECT ⇒ Robot connection function
⇒ See Robot and Machine Tool Integration Function OPERATOR'S MANUAL

(9)

(10)

(11)

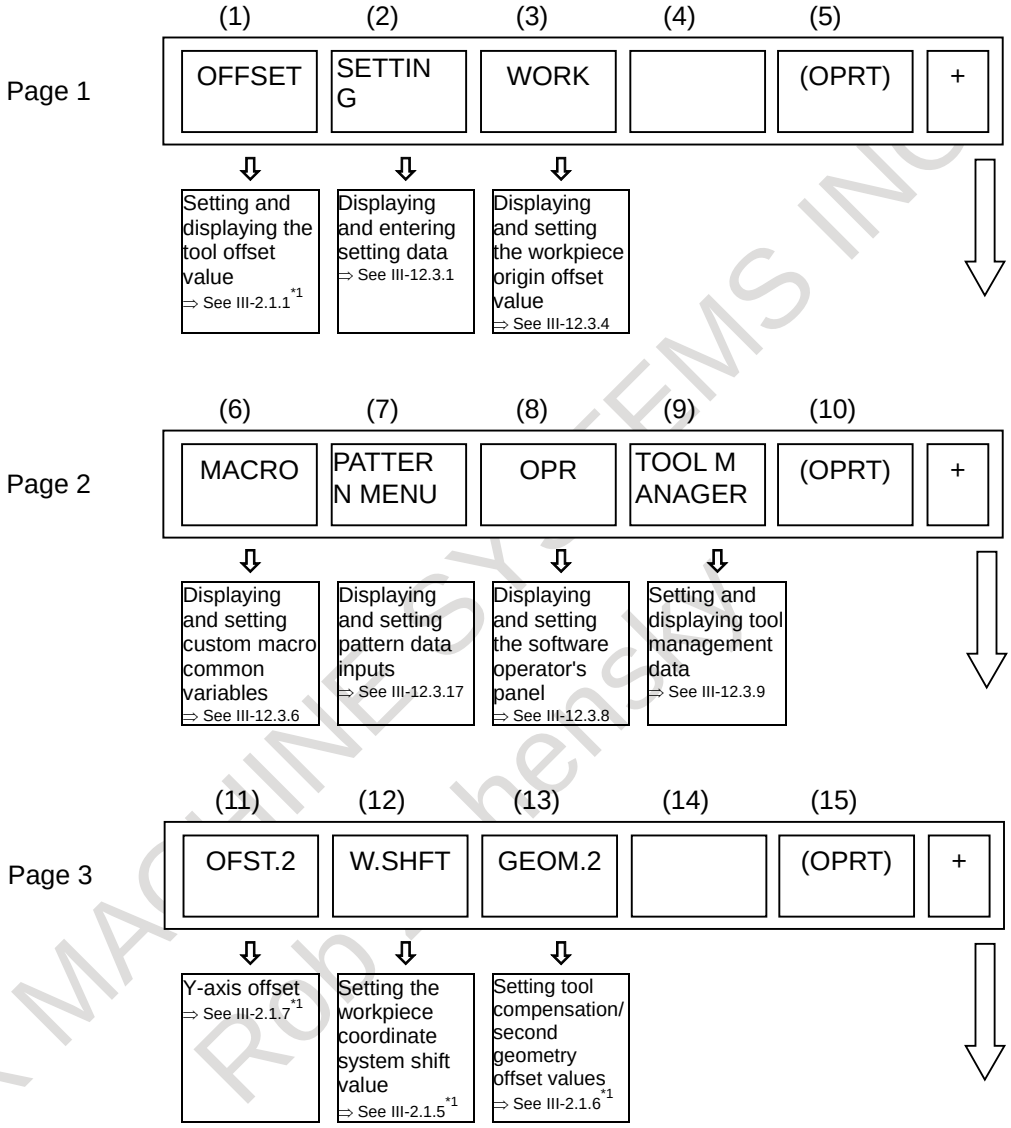
(12)

(13)

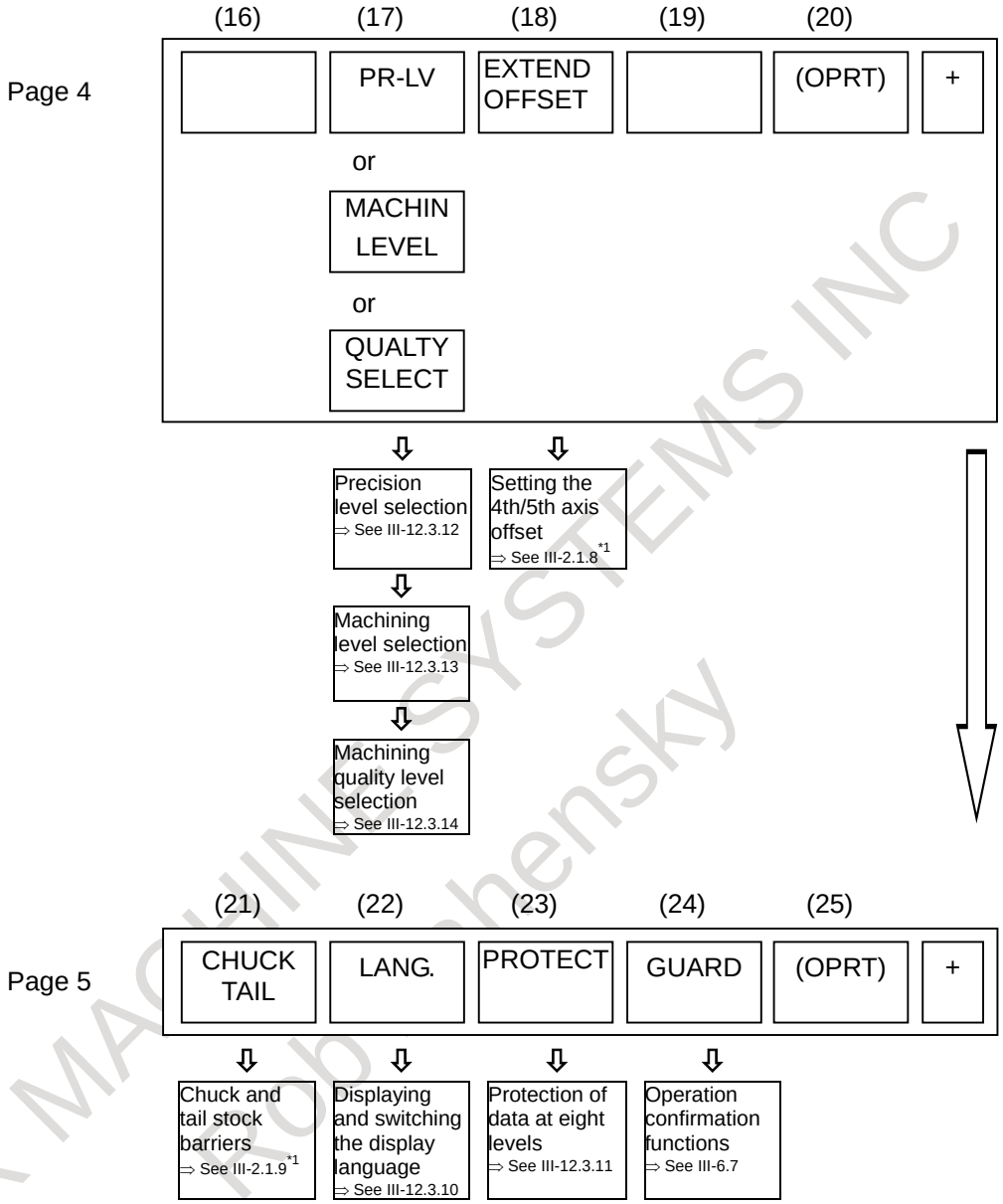
(14)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Screen displayed when the function key is pressed (for 8.4/10.4-inch display unit)



FRYER MACHINE SYSTEMS INC
Rob Chensky



FRYER MACHINE SYSTEMS INC
Robochensky

Screen displayed when the function key is pressed (for 15/19-inch display unit)

Page 1

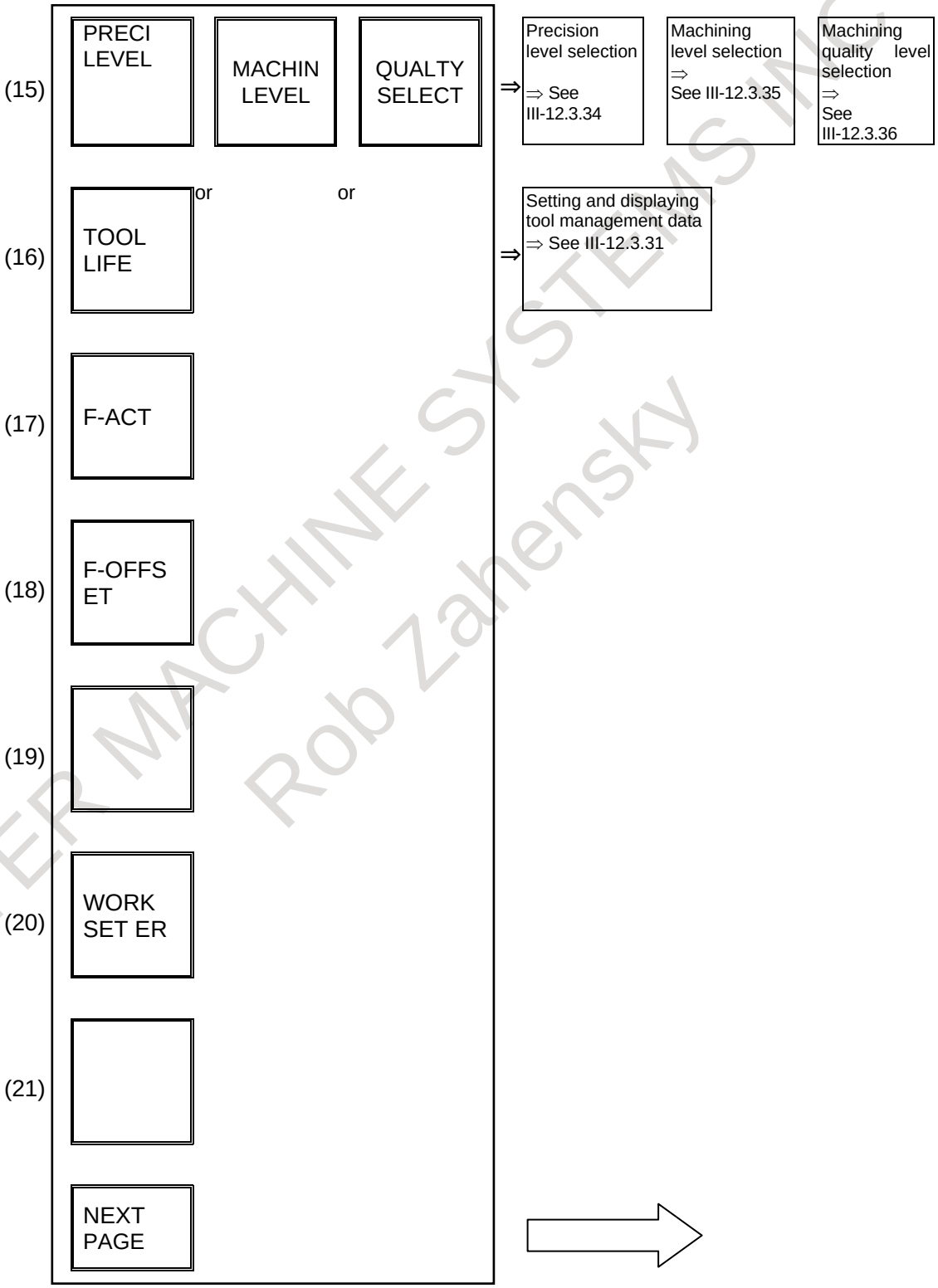
(1)	OFFSET	⇒	Setting and displaying the tool offset value ⇒ See III-2.1.1 *1
(2)	SETTIN G	⇒	Displaying and entering setting data ⇒ See III-12.3.23
(3)	WORK	⇒	Displaying and setting the workpiece origin offset value ⇒ See III-12.3.26
(4)	MACRO	⇒	Displaying and setting custom macro common variables ⇒ See III-12.3.28
(5)	OPERAT PANEL	⇒	Displaying and setting the software operator's panel ⇒ See III-12.3.30
(6)	Y OFFS ET	⇒	Y-axis offset ⇒ See III-2.1.7 *1
(7)	WORK SHIFT	⇒	Setting the workpiece coordinate system shift value ⇒ See III-2.1.5 *1
	NEXT PAGE		

Page 2

(8)	2ND GEOM	⇒	Setting tool compensation/second geometry offset values ⇒ See III-2.1.6 *1
(9)	TOOL FORM		
(10)	CHOPPI NG		
(11)	CHUCK TAIL	⇒	Chuck and tail stock barriers ⇒ See III-2.1.9 *1
(12)	LANGUA GE	⇒	Displaying and switching the display language ⇒ See III-12.3.32
(13)	PROTEC T	⇒	Protection of data at eight levels ⇒ See III-12.3.33
(14)	GUARD	⇒	Operation confirmation functions ⇒ See III-6.7
	NEXT PAGE		

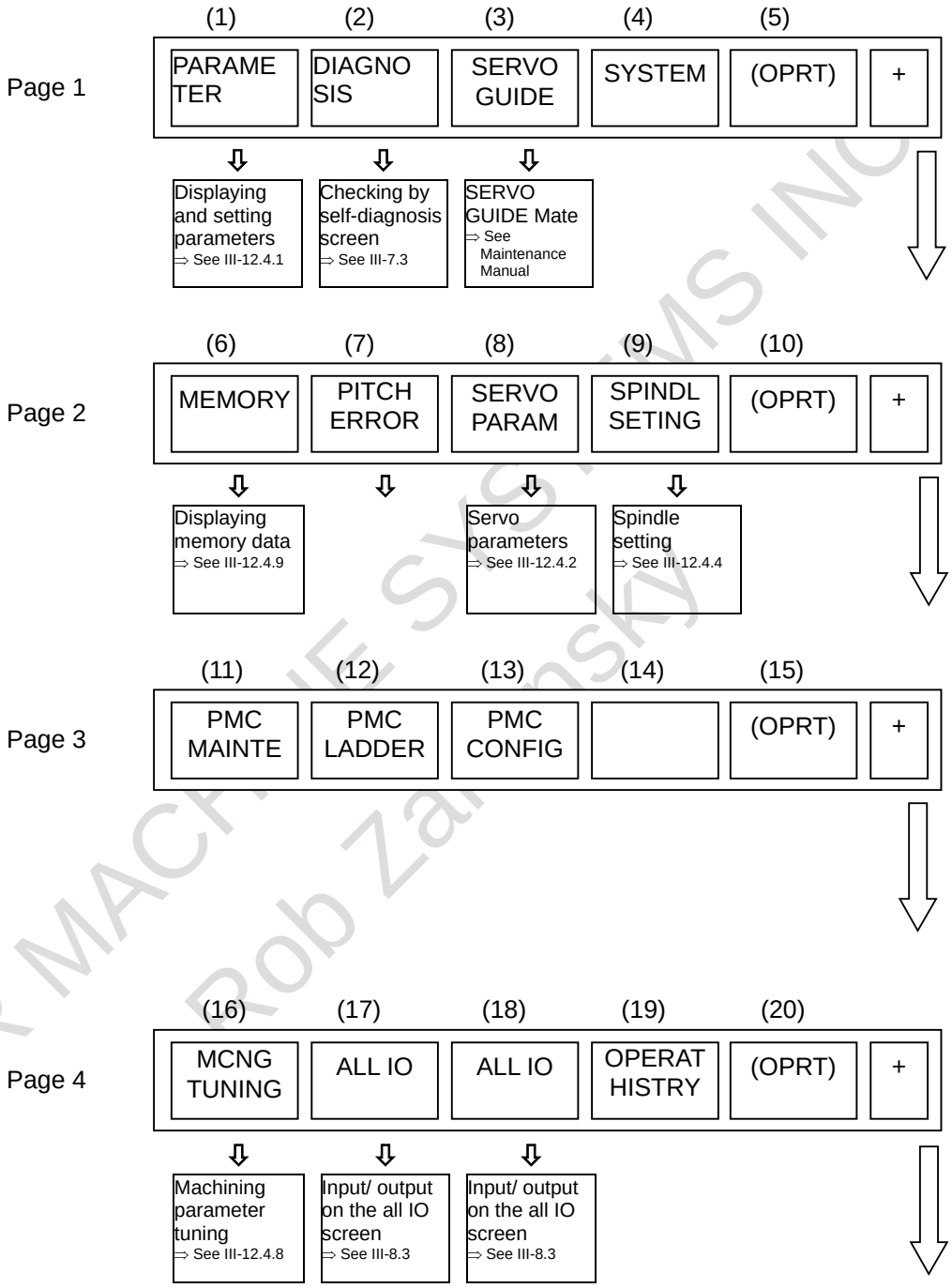
FRYER MACHINE SYSTEMS INC
Rob Zahensky

Page 3

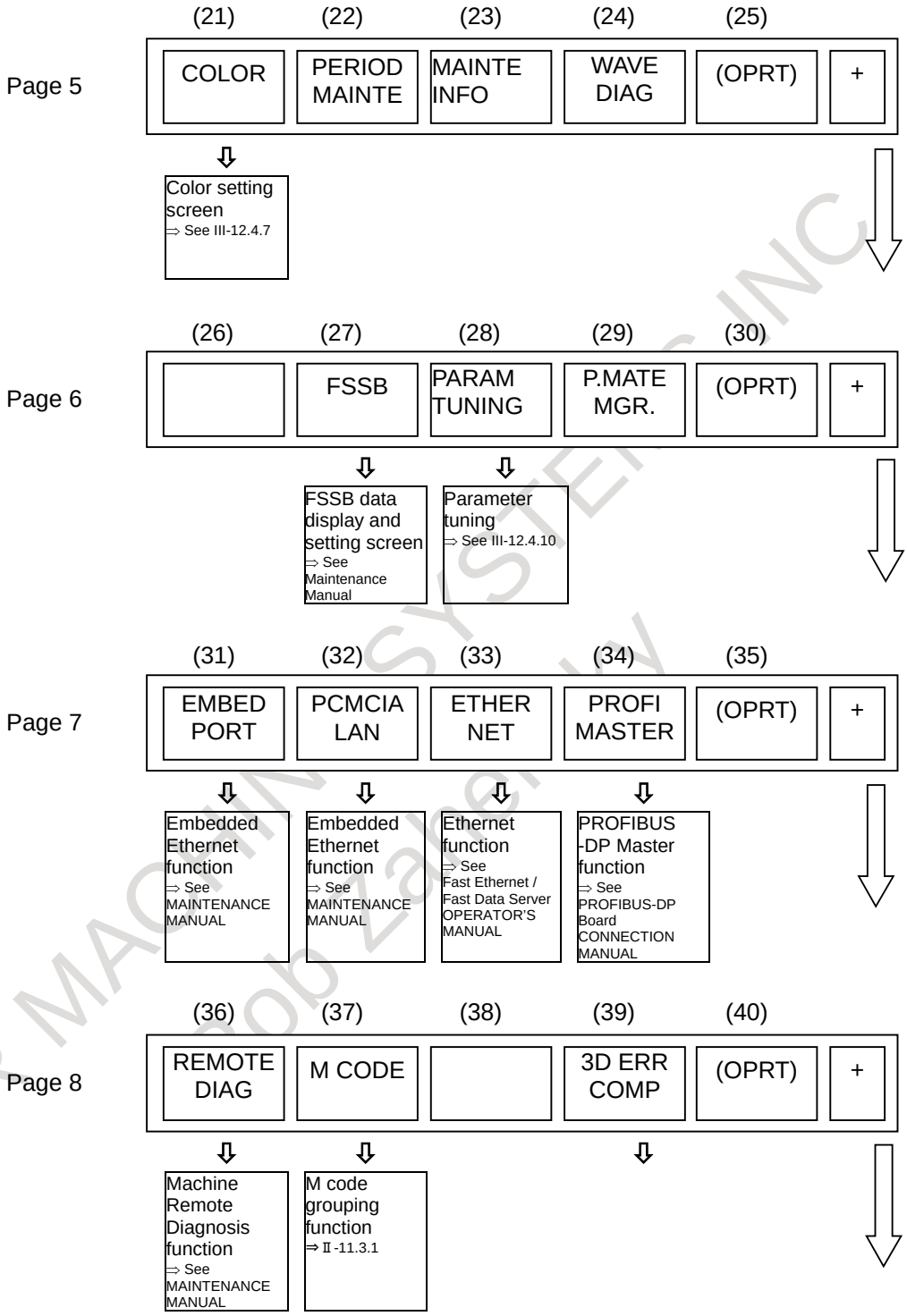


FRYER MACHINE SYSTEMS INC
Rob Zahensky

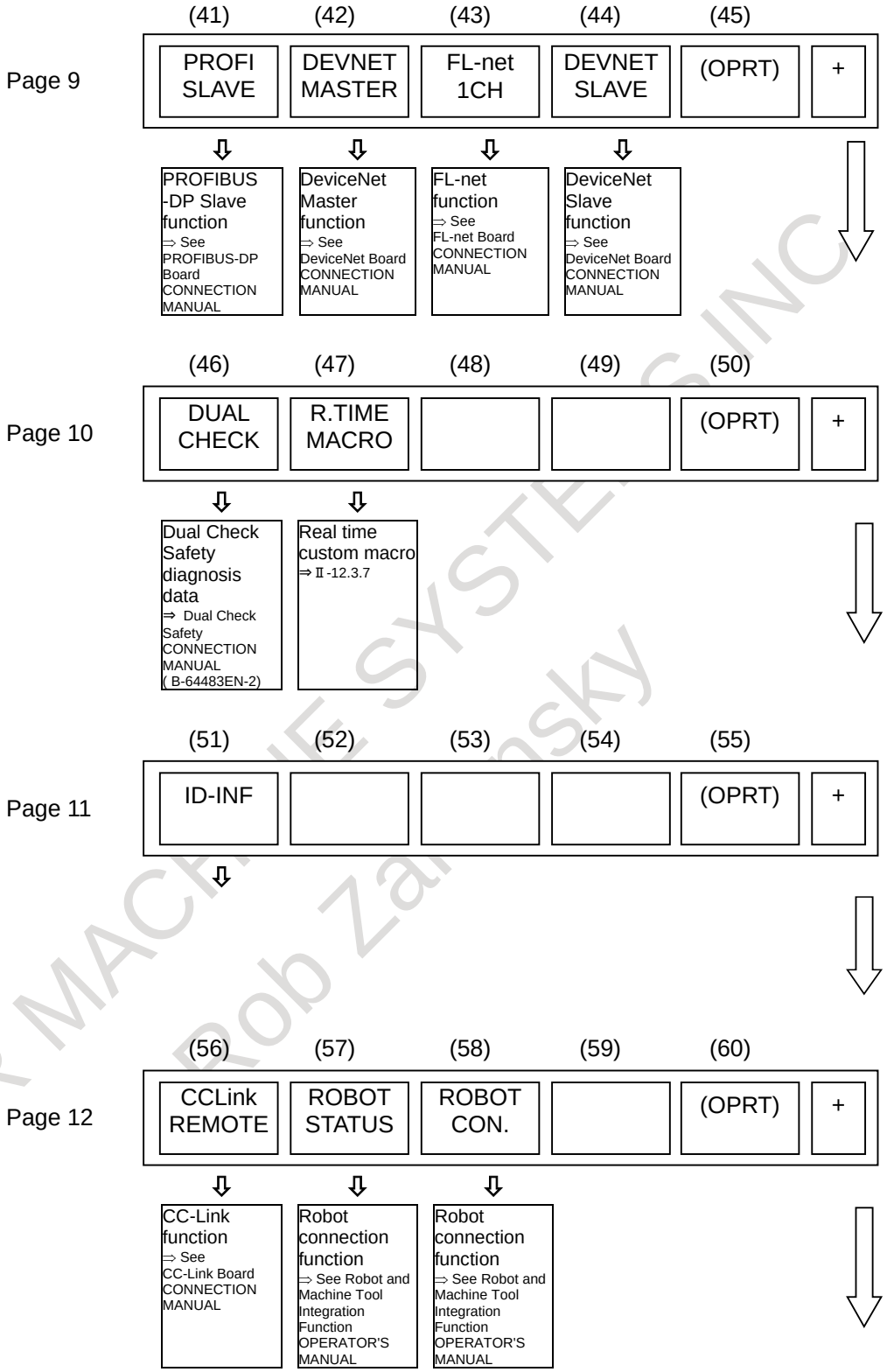
Screen displayed when the function key SYSTEM is pressed (for 8.4/10.4-inch display unit)



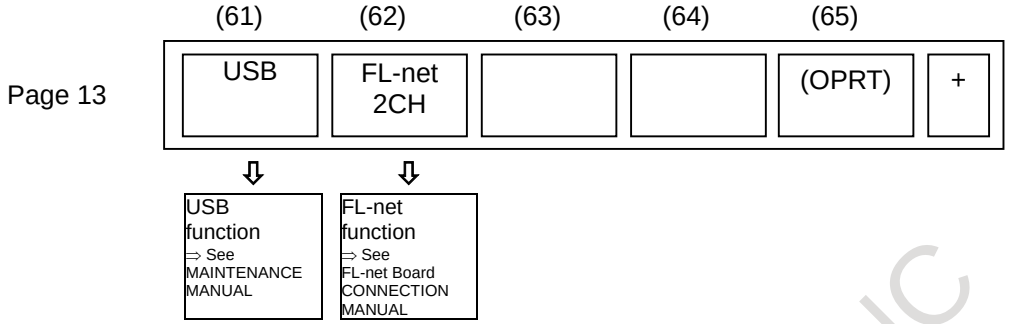
FRYER MACHINE SYSTEMS INC
Rob Zaranosky



FRYER MACHINE SYSTEMS INC
Bob Zaherly



FRYER MACHINE SYSTEMS INC
Rob Zaranosky

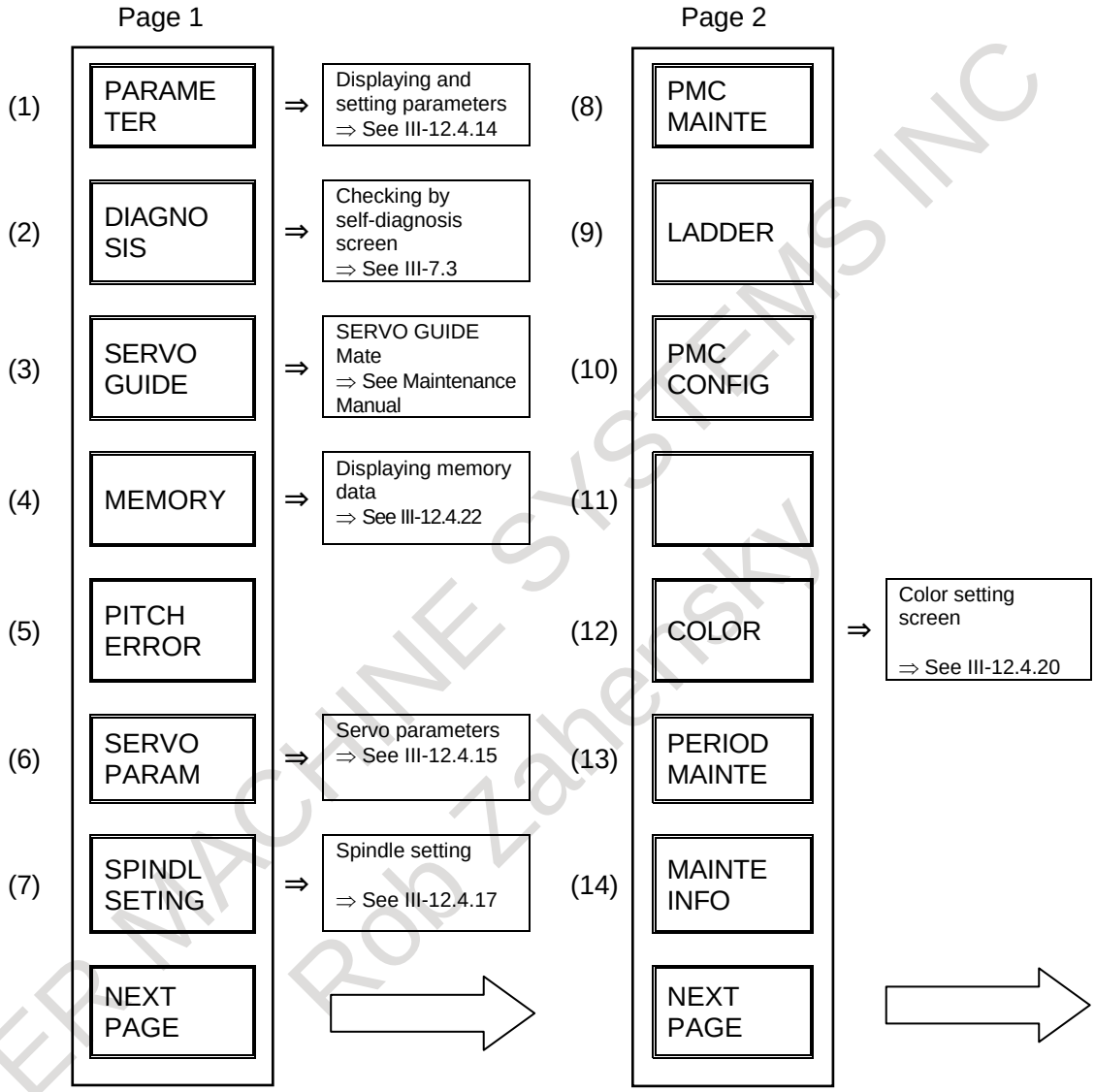


NOTE

For information about a dedicated screen for each path control type in the lathe system/machining center system, refer to the manuals:
*1: OPERATOR'S MANUAL (T series) (B-64694EN-1)
*2: OPERATOR'S MANUAL (M series) (B-64694EN-2)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Screen displayed when the function key SYSTEM is pressed (for 15/19-inch display unit)



FRYER MACHINE SYSTEMS INC
Rob Zaherisky

Page 3

(15)	FSSB	⇒	FSSB data display and setting screen ⇒ See Maintenance Manual
(16)	MCHN TUNING	⇒	Machining parameter tuning ⇒ See III-12.4.21
(17)	WAVE DIAG		
(18)	OPERAT HISTRY		
(19)	ALL IO	⇒	Input/ output on the all IO screen ⇒ See III-8.3
(20)	ALL IO	⇒	Input/ output on the all IO screen ⇒ See III-8.3
(21)	PARAM TUNING	⇒	Parameter tuning ⇒ See III-12.4.23
	NEXT PAGE		

Page 4

(22)	M CODE GROUP	⇒	M code grouping function ⇒ See II-11.3.1
(23)	EMBED PORT	⇒	Embedded Ethernet function ⇒ See MAINTENANCE MANUAL
(24)	PCMCIA LAN	⇒	Embedded Ethernet function ⇒ See MAINTENANCE MANUAL
(25)	ETHER NET	⇒	Ethernet function ⇒ See Fast Ethernet / Fast Data Server OPERATOR'S MANUAL
(26)	PROFI MASTER	⇒	PROFIBUS-DP Master function ⇒ See PROFIBUS-DP Board CONNECTION MANUAL
(27)	PROFI SLAVE	⇒	PROFIBUS-DP Slave function ⇒ See PROFIBUS-DP Board CONNECTION MANUAL
(28)	3D ERR COMP		
	NEXT PAGE		

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

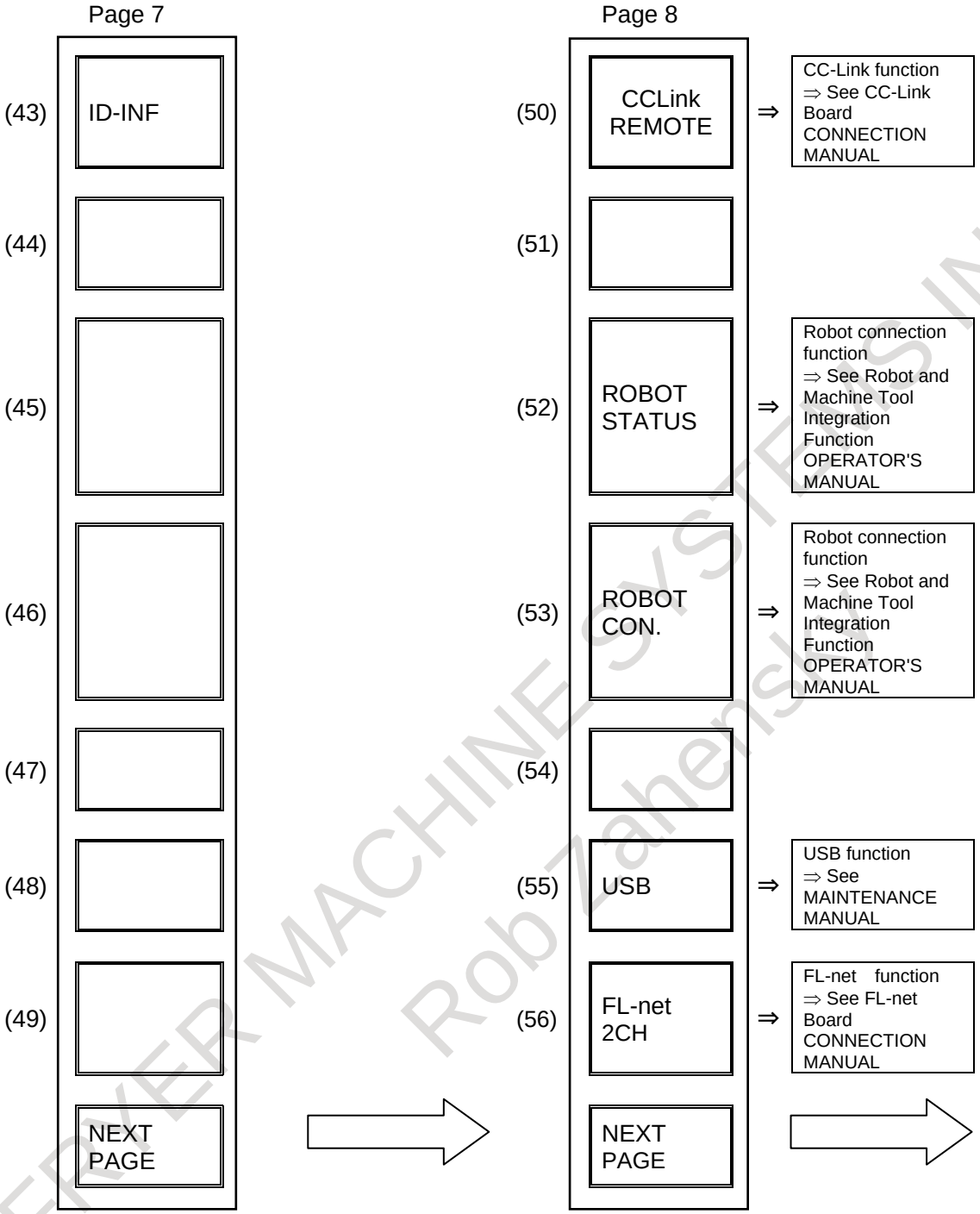
Page 5

(29)	P.MATE MGR.		
(30)	SYSTEM		
(31)	REMOTE DIAG	⇒	Machine Remote Diagnosis function ⇒ See Fast Ethernet / Fast Data Server OPERATOR'S MANUAL
(32)	DUAL CHECK	⇒	Dual Check Safety diagnosis data ⇒ See Dual Check Safety CONNECTION MANUAL (B-64483EN-2)
(33)	R.TIME MACRO	⇒	Displaying and setting real time custom macro data ⇒ See III-12.3.29
(34)	HI-PIT CH ERR		
(35)	PARTS LEARN		
	NEXT PAGE		

Page 6

(36)			
(37)	DEVNET MASTER	⇒	DeviceNet Master function ⇒ See DeviceNet Board CONNECTION MANUAL
(38)	FL-net 1CH	⇒	FL-net function ⇒ See FL-net Board CONNECTION MANUAL
(39)	DEVNET SLAVE	⇒	DeviceNet Slave function ⇒ See DeviceNet Board CONNECTION MANUAL
(40)	CUSTOM BOARD		
(41)			
(42)			
	NEXT PAGE		

FRYER MACHINE SYSTEMS INC
Rob Zahensky



FIXER MACHINE SYSTEMS INC
Rob Zahenski

NOTE

For information about a dedicated screen for each path control type in the lathe system/machining center system, refer to the manuals:

*1: OPERATOR’S MANUAL (T series) (B-64694EN-1)

*2: OPERATOR’S MANUAL (M series) (B-64694EN-2)

12.1 SCREENS DISPLAYED BY FUNCTION KEY POS

Section 12.1, "SCREENS DISPLAYED BY FUNCTION KEY POS", consists of the following subsections:

Screens of an 8.4/10.4-inch display unit

Press function key POS to display the current position of the tool.

The following three screens are used to display the current position of the tool:

- Current position display screen for the workpiece coordinate system.
- Current position display screen for the relative coordinate system.
- Current overall position display screen.

The above screens can also display the feedrate, run time, and the number of parts. In addition, a floating reference position can be set on these screens.

Function key POS can also be used to display the load on the servo motor and spindle motor and the rotation speed of the spindle motor (operating monitor display).

Function key POS can also be used to display the screen for displaying the distance moved by handle interruption. See Section, “MANUAL HANDLE INTERRUPTION” for details on this screen.

On any of the position display screens, the status (such as D, I, L, S, *, or M) of an axis is indicated on the left of the axis name to prevent wrong operations. See "Axis status display" in Section, "WRONG OPERATION PREVENTION FUNCTIONS" for details.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.1.1 Position Display in the Workpiece Coordinate System

Displays the current position of the tool in the workpiece coordinate system. The current position changes as the tool moves. The least input increment is used as the unit for numeric values. The title at the top of the screen indicates that absolute coordinates are used.

Display procedure for the current position screen in the workpiece coordinate

Procedure

1 Press function key [POS].
2 Press soft key [ABSOLUTE].
3 Press soft key [ABSOLUTE] again. The sixth and subsequent axes are displayed.

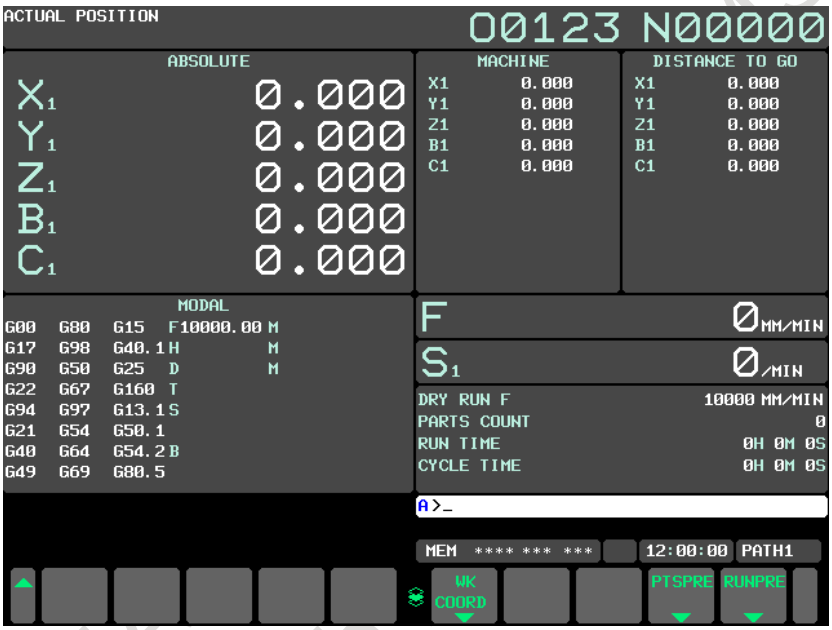


Fig. 12.1.1 (a) Current position (absolute) screen (M series)(10.4-inch display unit)

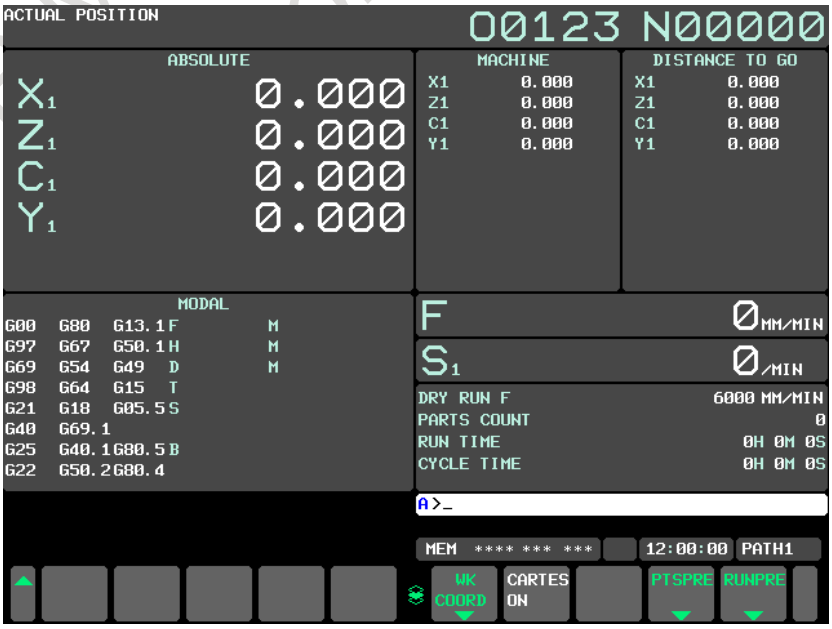


Fig. 12.1.1 (b) Current position (absolute) screen (T series)(10.4-inch display unit)

FRYER MACHINE TOOLS INC

Explanation

- Display including compensation values

M

Bits 6 (DAL) and 7 (DAC) of parameter No. 3104 can be used to select whether the displayed values include tool length compensation and cutter compensation.

T

Bit 1 (DAP) of parameter No. 3129 and bit 7 (DAC) of parameter No. 3104 can be used to select whether the displayed values include tool offset and tool nose radius compensation.

12.1.2 Position Display in the Relative Coordinate System

Displays the current position of the tool in a relative coordinate system based on the coordinates (see Explanation) set by the operator. The current position changes as the tool moves. The increment system is used as the unit for numeric values.
The title at the top of the screen indicates that relative coordinates are used.

Display procedure for the current position screen with the relative coordinate system

Procedure

1 Press function key .

2 Press soft key [RELATIVE].

3 Press soft key [RELATIVE] again. The sixth and subsequent axes are displayed.

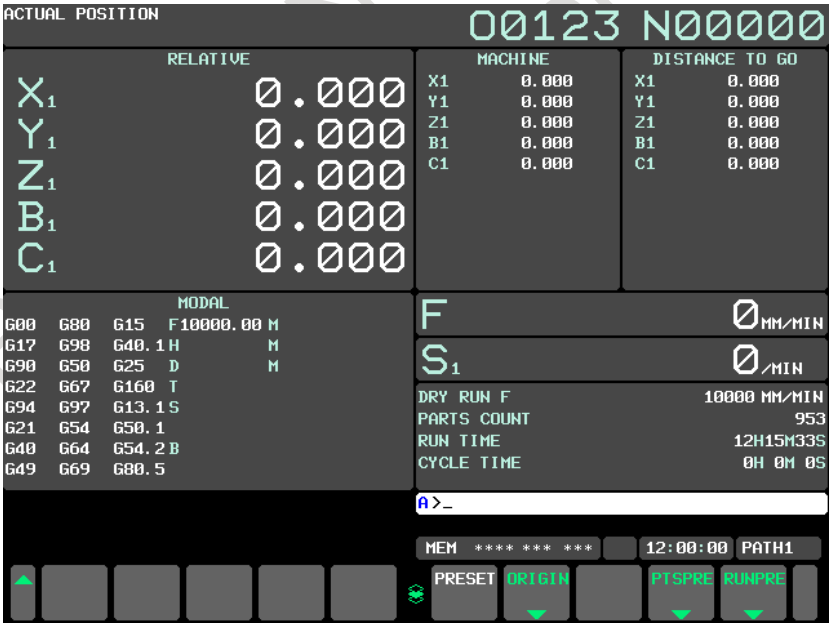


Fig. 12.1.2 (a) Current position (relative) screen (M series)(10.4-inch display unit)

FRYER ... NE SYSTEMS INC
...nsky

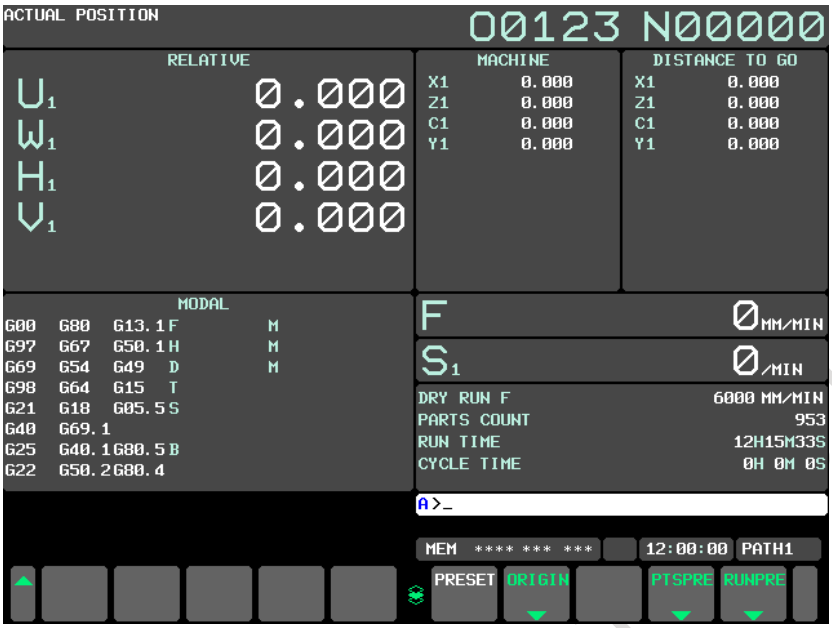


Fig. 12.1.2 (b) Current position (relative) screen (T series) (10.4-inch display unit)

See Explanation for the procedure for setting the coordinates.

Explanation

- Setting the relative coordinates

The current position of the tool in the relative coordinate system can be reset to 0 or preset to a specified value as follows.
Normal decimal point input and Pocket calculator type decimal point input can be selected by bit 0 (DPI) of parameter.No.3401.

Resetting relative coordinates to 0

When all axes are reset to 0

1 Press function key [POS].

2 Press chapter selection key [RELATIVE] to display the relative coordinate screen.

3 Press soft key [(OPRT)].

4 Press soft key [ORIGIN].

5 Press soft key [ALL AXIS].
The relative coordinates of all axes are reset to 0.

When a specified axis is reset to 0

1 Press function key [POS].

2 Press chapter selection key [RELATIVE] to display the relative coordinate screen.

3 Press soft key [(OPRT)].

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

4 Press soft key [ORIGIN].

5 Input the axis name of the resetting axis.
The axis name blinks.
6 Press soft key [EXEC].
The relative coordinate is reset to 0.

Presetting relative coordinates

Procedure

1 Press function key [POS].
2 Press chapter selection key [RELATIVE] to display the relative coordinate screen.
3 Press soft key [(OPRT)].

4 Input the axis name of the presetting axis.
The axis name blinks.
5 Input the coordinates value, and press soft key [PRESET].
The relative coordinate is preset by input coordinate value.

NOTE
1 When the preset of the specified axis is done in the following states, warning is displayed.
- When only the numerical value is input or the character that is not the axis name is input
  Waning massage: “THERE IS NO SPECIFIED NAME OF AXIS”
- When input form is except for “axis name + numerical value” is done
  Waning massage: ”FORMAT ERROR”

2 When axis name display is changed by parameter No.3132, specify the changed axis name.

- Display including compensation values

M

Bits 4 (DRL) and 5 (DRC) of parameter No. 3104 can be used to select whether the displayed values include tool length compensation and cutter compensation.

T

Bit 0 (DRP) of parameter No. 3129 and bit 5 (DRC) of parameter No. 3104 can be used to select whether the displayed values include tool offset and tool nose radius compensation.

- Presetting by setting a coordinate system

M

Bit 3 (PPD) of parameter No. 3104 can be used to specify whether the position indication values in the absolute coordinate system are preset as those in the relative coordinate system during coordinate system setting or manual reference position return by the G92 command.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

T

Bit 3 (PPD) of parameter No. 3104 can be used to specify whether the position indication values in the absolute coordinate system are preset as those in the relative coordinate system during coordinate system setting or manual reference position return by the G50 command (for G code system A) or G92 command (for G code system B or C).

If parameter bit PPD is set to 0, by setting bit 5 (PWR) of parameter No. 11277 to 1, it is possible to preset the relative coordinates at power-on in the machine coordinate system.

12.1.3 Overall Position Display

Displays the following positions on a screen : Current positions of the tool in the workpiece coordinate system, relative coordinate system, and machine coordinate system, and the remaining distance.
The relative coordinates can also be set on this screen. See Subsection, “Position Display in the Relative Coordinate System” for the procedure.

Procedure for displaying overall position display screen

Procedure

1 Press function key POS.
2 Press soft key [ALL].
3 Press soft key [ALL] again. The sixth and subsequent axes are displayed.

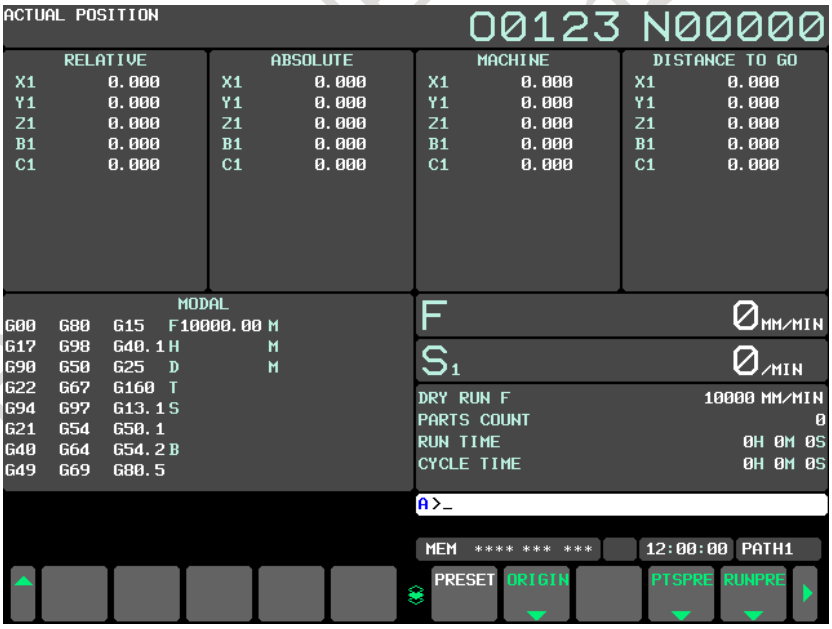


Fig. 12.1.3 (a) Current position (overall) screen (M series) (10.4-inch display unit)

FRYER SYSTEMS INC
Ryosky

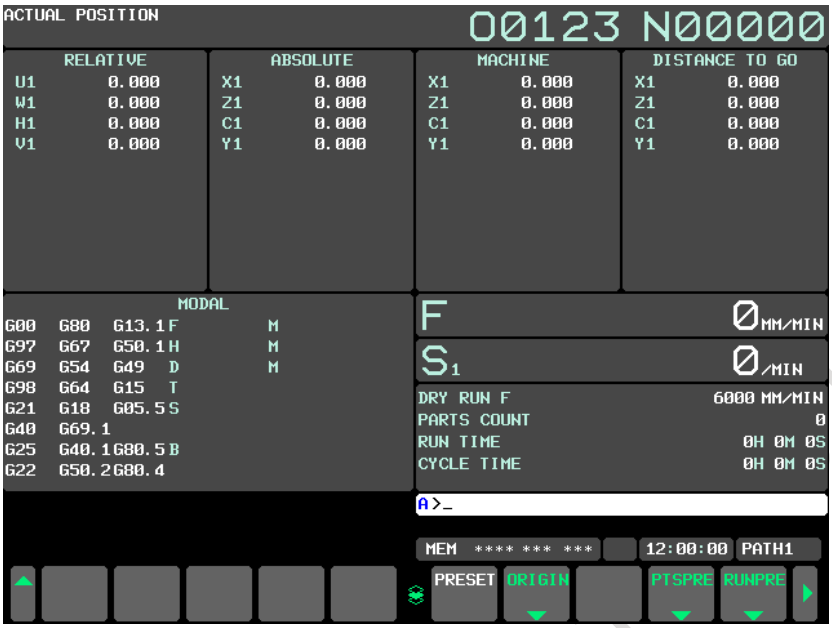


Fig. 12.1.3 (b) Current position (overall) screen (T series) (10.4-inch display unit)

Explanation

- Coordinate display

The current positions of the tool in the following coordinate systems are displayed at the same time:

- Current position in the relative coordinate system (relative coordinate)
- Current position in the workpiece coordinate system (absolute coordinate)
- Current position in the machine coordinate system (machine coordinate)
- Distance to go (distance to go)

- Distance to go

The distance remaining is displayed in the MEM or MDI mode.
The distance the tool is yet to be moved in the current block is displayed.

- Machine coordinate system

The least command increment is used as the unit for values displayed in the machine coordinate system. However, the least input increment can be used by setting bit 0 (MCN) of parameter No. 3104.

- Resetting the relative coordinates

The total position display screen also supports the resetting of the relative coordinates to 0 or presetting of them to specified values. See the procedure for resetting the relative coordinates described in Subsection, “Position Display in the Relative Coordinate System”.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.1.4 Workpiece Coordinate System Preset

If a workpiece coordinate system has been shifted with manual intervention or any other operation, an MDI operation can be performed to preset the system to a workpiece coordinate system that is offset by a workpiece origin offset, from the machine zero point that has been set before the shifting.
A command (G92.1) can be programmed to preset a workpiece coordinate system. (See Subsection, “Workpiece Coordinate System Preset (G92.1)”.)

Procedure for the workpiece coordinate system preset

Procedure

When all axes are performed to preset

1 Press function key [POS].
2 Press soft key [ALL] or [ABSOLUTE].
3 Press soft key [(OPRT)].
4 Press the continuous menu key [▷] until soft key [WK COORD] appears.

5 Press soft key [WK COORD].

6 Press soft key [ALL AXIS].

When a specified axis is performed to preset

1 Press function key [POS].
2 Press soft key [ALL] or [ABSOLUTE].
3 Press soft key [(OPRT)].
4 Press the continuous menu key [▷] until soft key [WK COORD] appears.

5 Press soft key [WK COORD].

6 Enter the axis name ([X], [&Y], ...) and zero ([P 0]).
7 Press soft key [EXEC].

Explanation

- Operation mode

This function can be executed when the reset state or automatic operation stop state is entered, regardless of the operation mode.

- Presetting relative coordinates

As with absolute coordinates, bit 3 (PPD) of parameter No. 3104 is used to specify whether to preset relative coordinates (RELATIVE).

FRYER MACHINE SYSTEMS INC
RO: Zahensky

12.1.5 Actual Feedrate Display

The actual feedrate on the machine can be displayed on a current position display screen or program check screen.
The Actual Feedrate Display is switched to feed per minute / feed per revolution by combination of bit 3 (GSC) of parameter No.3107, bit 5 (FSS) of parameter No.3191, operation status and G code modal.

GSC	FSS	Operation status	Modal G code	Display unit
0	-	-	-	Feed per minute
1	0	Manual feed Manual per revolution feed Rapid traverse Dry run	-	Feed per minute
		Manual feed Manual per revolution feed Rapid traverse other than dry run	M series :G93, G94 T series :G98(G code system A) G93, G94(G code system B,C)	Feed per minute
			M series :G95 T series :G99 (G code system A) G95 (G code system B,C)	Feed per revolution
	1	-	-	Feed per revolution

Feed per minute and feed per revolution example are as follows.
<Feed per minute>

<Feed per revolution>

The display of inch and millimeter is switched by the bit2(INI) of parameter No.0 of input unit.
Relationship between the bit2(INI) of parameter No.0 of input unit and the display unit of feed par minute / feed per revolution are shown in the following table.

	Feed par minute	Feed per revolution
mm (INI=0)	MM/MIN	MM/REV
inch (INI=1)	INCH/M	INCH/R

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Display procedure for the actual feedrate on the current position display screen

Procedure

1 Press the function key [POS] to display a current position display screen. At the location indicated by [ ], an actual feedrate is displayed.

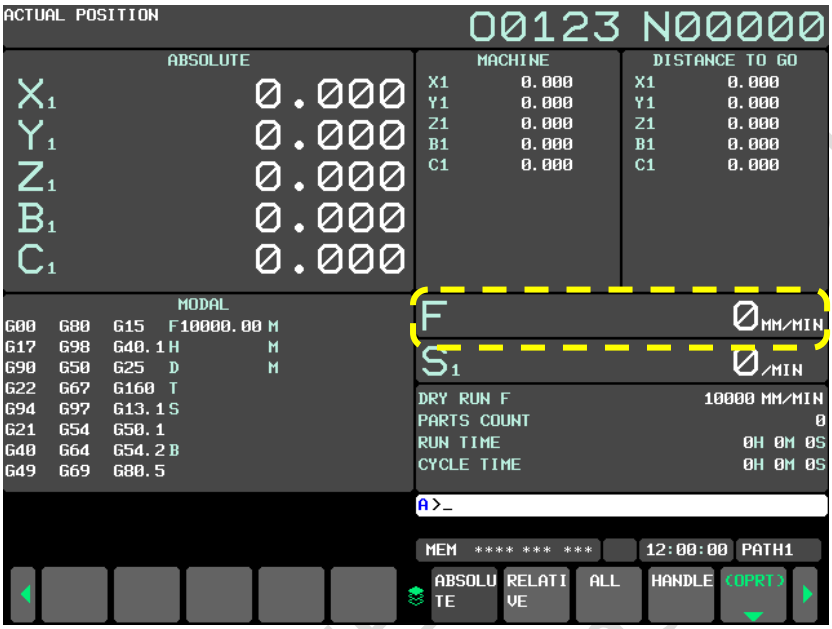


Fig. 12.1.5 (a) Current position (absolute) screen (10.4-inch display unit)

The actual feedrate is displayed in units of millimeter/min or inch/min (depending on the specified least input increment) under the display of the current position.

Explanation

- Actual feedrate value

The actual rate (per minuts) is calculated by the following expression:

$$Fact = \sqrt{\sum_{i=1}^{n} (fi)^2}$$

n : Number of axes
fi : Cutting feed rate in the tangential direction of each axis or rapid traverse rate
Fact : Actual feedrate displayed

The display unit:

mm/min (metric input)
inch/min (Inch input, two digits below the decimal point are displayed.)

- Actual feedrate display of rotary axis

In the case of movement of rotary axis, the speed is displayed in units of deg/min but is displayed on the screen in units of input system at that time. For example, when the rotary axis moves at 50 deg/min, the following is displayed: 50 mm/min (in metric input) or 0.50 inch/min (in inch input).

- Actual feedrate display on the other screen

The program check screen also displays the actual feedrate.

- Number of decimal places

The number of decimal places of feed per minute is set by the parameter No.3135.
The number of decimal places of feed per revolution feedrate is as follows.

In case of mm input, number of decimal places displayed 2 digits
In case of inch input, number of decimal places displayed 3 digits
* Digit number is fixed. It is not possible to be changed by parameters etc.

FRYER MACHINE INC Rob Zahen

12.1.6 Display of Run Time and Parts Count

The run time, cycle time, and the number of machined parts are displayed on the current position display screens.

Procedure for displaying run time and parts count on the current position display screen

Procedure

1 Press the function key POS to display a current position display screen. At the location indicated by ⬚, a run time and parts count are displayed.

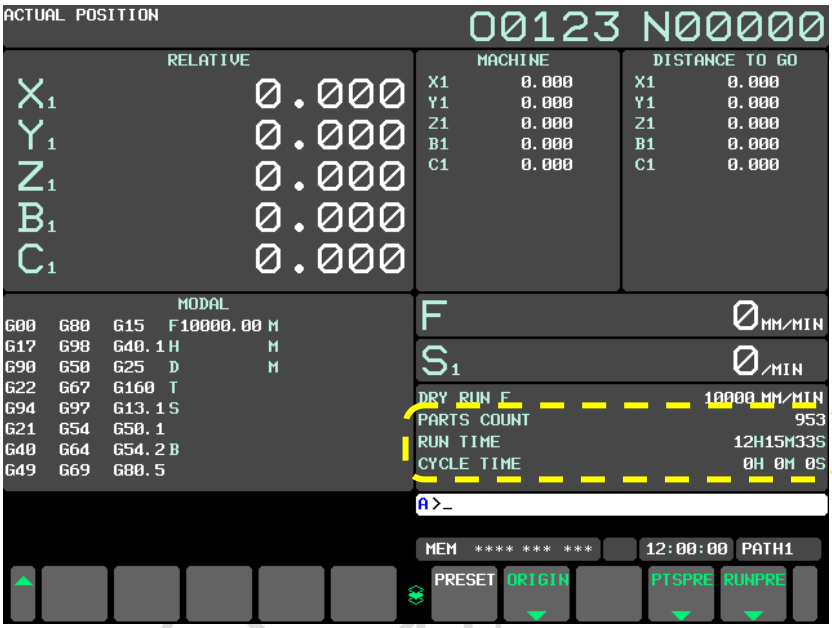


Fig. 12.1.6 (a) Current position (relative) screen (M series) (10.4-inch display unit)

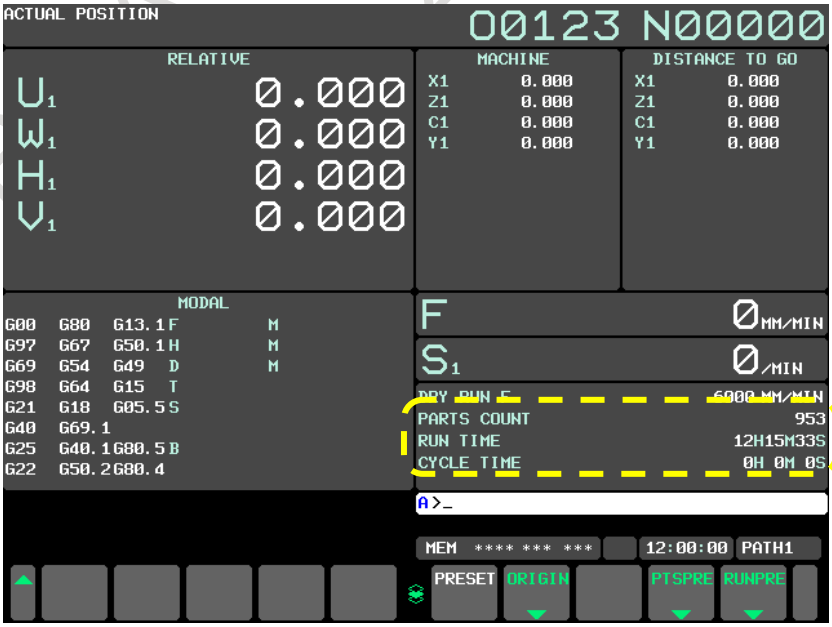


Fig. 12.1.6 (b) Current position (relative) screen (T series) (10.4-inch display unit)

The number of machined parts (PART COUNT), run time (RUN TIME), and cycle time (CYCLE TIME) are displayed under the current position.

Explanation

- PART COUNT

Indicates the number of machined parts. The number is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed.

- RUN TIME

Indicates the total run time during automatic operation, excluding the stop and feed hold time.

- CYCLE TIME

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- Display on the other screen

Details of the run time and the number of machined parts are displayed on the setting screen. See Subsection, “Displaying and Setting Run Time, Parts Count, and Time”.

- Parameter setting

The number of machined parts and run time cannot be set on current position display screens. They can be set by parameters Nos. 6711, 6751, and 6752 or on the setting screen.

- Incrementing the number of machined parts

Bit 0 (PCM) of parameter No. 6700 is used to specify whether the number of machined parts is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed, or only each time an M code specified by parameter No. 6710 is executed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.1.7 Operating Monitor Display

The load meter for a servo axis can be displayed. Also, the load meter and speed meter for a serial spindle can be displayed.
To enable this function, bit 5 (OPM) of parameter No. 3111 must be set to 1.

Procedure for displaying the operating monitor

Procedure

1 Press function key [POS].

2 Press continuous menu key [▷].

3 Press soft key [MONITOR].

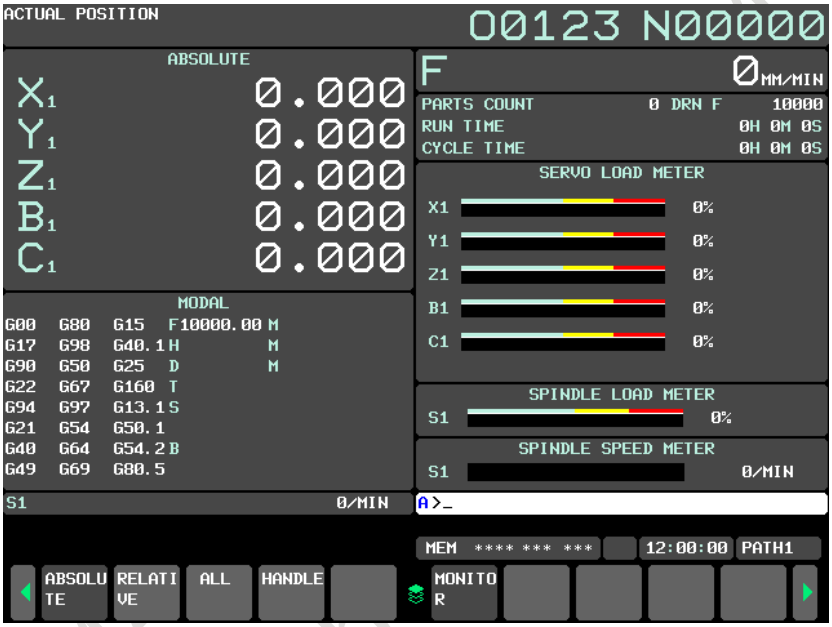


Fig. 12.1.7 (a) Operating monitor (M series) (10.4-inch display unit)

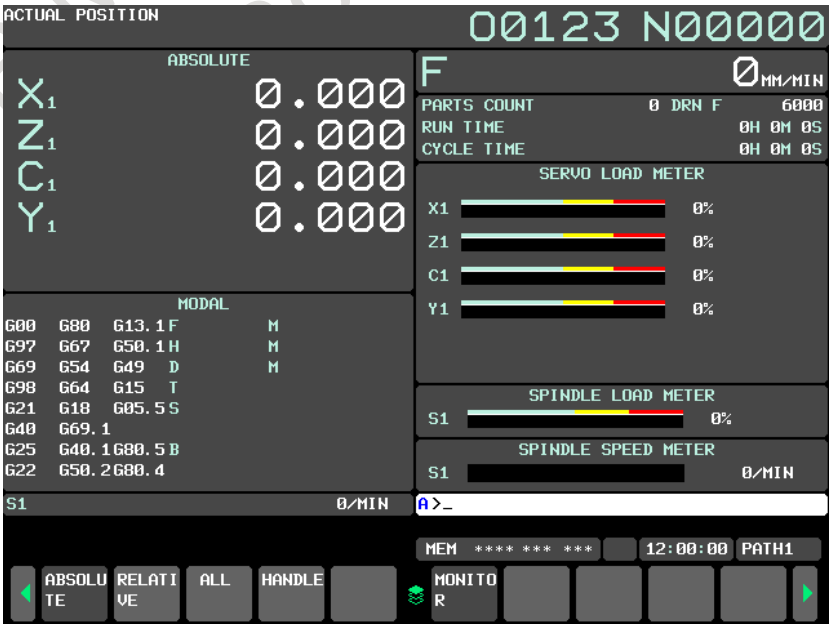


Fig. 12.1.7 (b) Operating monitor (T series) (10.4-inch display unit)

FRYER MACHINE TOOLS INC

Explanation

- Display of the servo axes

Servo axis load meters as many as the maximum number of controlled axes of the path can be displayed. One screen displays load meters for up to five axes at a time. By pressing the soft key [MONITOR], load meters for the 6th axis and up are displayed.

- Display of the spindle axes

When serial spindles are used, the reading on the load meter and speedometer can be displayed only for the main serial spindle.

- Unit of graph

The bar graph for the load meter shows load up to 200% (only a value is displayed for load exceeding 200%).
The bar graph for the speedometer shows the ratio of the current spindle speed to the maximum spindle speed (100%).

- Load meter

The reading on the load meter depends on servo parameter No. 2086 and spindle parameter No. 4127.

- Speedometer

Although the speedometer normally indicates the speed of the spindle motor, it can also be used to indicate the speed of the spindle by setting bit 6 (OPS) of parameter No. 3111 to 1.
The spindle speed to be displayed during operation monitoring is calculated from the speed of the spindle motor (see the formula below).
The spindle speed can therefore be displayed, during operation monitoring, even when no position coder is used.
To display the correct spindle speed, however, the maximum spindle speed for each gear (spindle speed at each gear ratio when the spindle motor rotates at the maximum speed) must be set in parameters Nos. 3741 to 3744.
The input of the clutch and gear signals for the first serial spindle is used to determine the gear which is currently selected. Control the input of the CTH1A and CTH2A signals according to the gear selection, by referring to the table below.
(Formula for calculating the spindle speed to be displayed)

$$\text{Spindle speed displayed during operation monitoring} = \frac{\text{Speed of spindle motor}}{\text{Maximum speed of spindle motor}} \times \text{Maximum spindle speed with the gear being used}$$

The following table lists the correspondence between clutch and gear selection signals CTH1A and CTH2A, used to determine the gear being used, and parameters:

CTH1A	CTH2A	Parameter	Serial spindle specification
0	0	=No.3741 (Maximum spindle speed with gear 1)	HIGH
0	1	=No.3742 (Maximum spindle speed with gear 2)	MEDIUM HIGH
1	0	=No.3743 (Maximum spindle speed with gear 3)	MEDIUM LOW
1	1	=No.3744 (Maximum spindle speed with gear 4)	LOW

The speed of the spindle motor and spindle can be displayed, during operation monitoring, only for the first serial spindle and the spindle switching axis for the first serial spindle. It cannot be displayed for the second spindle.

- Color of graph

If the value of a load meter exceeds 100%, the bar graph turns purple.

FLYER MACHINE SYSTEMS INC
Rob Zaremsky

12.1.8 Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount)

M

The absolute coordinates of the tool tip, the number of pulses, and a machine axis move amount based on 3-dimensional manual feed are displayed.

Displaying the screen for 3-dimensional manual feed

Procedure

1 Press function key [POS].

2 Press the continuous menu key [▷] several times to display the soft key [3-D MANUAL].

3 Press the soft key [3-D MANUAL] to display the screen of 3-dimensional manual feed.

Fig. 12.1.8 (a) 3-dimensional manual feed screen(10.4-inch display unit)

Explanation

- Tool tip position

The addresses of the three basic machine configuration axes for performing 3-dimensional manual feed and the current position of the tool tip are displayed.

- Tool axis reference (number of pulses)

TD

The amount of travel in the tool axis direction in tool axis direction handle feed, tool axis direction jog feed, or tool axis direction incremental feed is displayed.
The unit is the least input increment of the axis in the direction specified by parameter No. 19697.

R1

The amount of travel in the first axis direction in tool axis right-angle direction handle feed, tool axis right-angle direction jog feed, or tool axis right-angle direction incremental feed is displayed.
The unit is the least input increment of the axis in the first axis direction normal to the direction specified by parameter No. 19697.

R2

The amount of travel in the second axis direction in tool axis right-angle direction handle feed, tool axis right-angle direction jog feed, or tool axis normal direction incremental feed is displayed.
The unit is the least input increment of the axis in the second axis direction normal to the direction specified by parameter No. 19697.

- Tool tip center (number of pulses)

C1

The angular displacement in tool tip center rotation handle feed, tool tip center rotation jog feed, or tool tip center rotation incremental feed for rotation of the first rotary axis is displayed. The unit is the least input increment of the first rotary axis.

C2

The angular displacement in tool tip center rotation handle feed, tool tip center rotation jog feed, or tool tip center rotation incremental feed for rotation of the second rotary axis is displayed. The unit is the least input increment of the second rotary axis.

- Table reference (number of pulses)

VR

The amount of travel in the table reference vertical direction in table reference vertical direction handle feed, table reference vertical direction jog feed, or table reference vertical direction incremental feed is displayed.
The unit is the least input increment of the axis in the direction specified by parameter No. 19697.

H1

The amount of travel in the first axis direction in table reference horizontal direction handle feed, table reference horizontal direction jog feed, or table reference horizontal direction incremental feed is displayed.
The unit is the least input increment of the axis in the first axis direction normal to the direction specified by parameter No. 19697.

H2

The amount of travel in the second axis direction in table reference horizontal direction handle feed, table reference horizontal direction jog feed, or table reference horizontal direction incremental feed is displayed.
The unit is the least input increment of the axis in the second axis direction normal to the direction specified by parameter No. 19697.

- Amounts of machine axis travel

The addresses of machine configuration axes used for 3-dimensional manual feed and the sum of the amount of travel of each axis used for 3-dimensional manual feed are displayed.
The values of the basic three axes (X-axis, Y-axis, and Z-axis), the first rotary axis, and the second rotary axis are displayed in this order.
For the definition of the first rotary axis and second rotary axis, see the description of parameter No. 19680.
When bit 0 (CLR) of parameter No. 13113 is set to 1, the displayed data is cleared by a reset.

- Absolute coordinates, machine coordinates

The absolute coordinates and machine coordinates of all axes are displayed. If too many axes are involved for display on one screen, the remaining axes can be displayed by pressing the soft key [3-D MANUAL] for page feed.

FRYER MACHINE SYSTEMS INC
Rob Zaherosky

- **F (feedrate)**
  - When bit 3 (CFD) of parameter No. 13113 is set to 0
    The composite feedrate at a control point on a linear axis or rotary axis is displayed.
  - When bit 3 (CFD) of parameter No. 13113 is set to 1
    The feedrate of the tool tip is displayed.

Operation

The display of the number of pulses can be cleared by soft key operations.

1 Press soft key [(OPRT)].

2 Select the soft key corresponding to a function subject to clearing of the amount of travel.
Pressing the rightmost soft key displays the second page.

3 Press soft key [ERASE] to clear the amount of travel of the specified function. Press soft key [CAN] to cancel erase operation.

Screens of a 15/19-inch display unit

Press function key POS to display the current overall position display screen that shows the current position of the tool.
The above screen can also display the feedrate, run time, and the number of parts. In addition, a floating reference position can be set on this screen.

Function key POS can also be used to display the load on the servo motor and spindle motor and the rotation speed of the spindle motor (operating monitor display).

Function key POS can also be used to display the screen for displaying the distance moved by handle interruption. See Section, “MANUAL HANDLE INTERRUPTION” for details on this screen.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.1.9 Overall Position Display (15/19-inch Display Unit)

Displays the following positions on a screen : Current positions of the tool in the workpiece coordinate system, relative coordinate system, and machine coordinate system, and the remaining distance.
The relative coordinates can also be set on this screen. (Refer to Explanation.)

Procedure for displaying overall position display screen

Procedure

1 Press function key [POS].

2 Press vertical soft key [ALL].

Fig. 12.1.9 (a) Current position screen (M series) (15-inch display unit)

Fig. 12.1.9 (b) Current position screen (T series) (15-inch display unit)

FRYER MACHINERY INC

Explanation

- Coordinate display

The current positions of the tool in the following coordinate systems are displayed at the same time:

- Current position in the relative coordinate system (relative coordinate)
- Current position in the work coordinate system (absolute coordinate)
- Current position in the machine coordinate system (machine coordinate)
- Distance to go (distance to go)

- Distance to go

The distance remaining is displayed in the MEM or MDI mode.
The distance the tool is yet to be moved in the current block is displayed.

- Machine coordinate system

The least command increment is used as the unit for values displayed in the machine coordinate system. However, the least input increment can be used by setting bit 0 (MCN) of parameter No. 3104.

- Setting the relative coordinates

The current position of the tool in the relative coordinate system can be reset to 0 or preset to a specified value as follows.
Normal decimal point input and Pocket calculator type decimal point input can be selected by bit 0 (DPI) of parameter.No.3401.

Resetting relative coordinates to 0

When all axes are reset to 0

1 Press function key

.

2 Press horizontal soft key [ORIGIN].

3 Press horizontal soft key [ALL AXIS].
The relative coordinates of all axes are reset to 0.

When a specified axis is reset to 0

1 Press function key

.

2 Press horizontal soft key [ORIGIN].

3 Input the axis name of the resetting axis.
The axis name blinks.

4 Press horizontal soft key [EXEC].
The relative coordinate is reset to 0.

FRYER MACHINE SYSTEMS INC
Rob Zarensky

Presetting relative coordinates

Procedure

1 Press function key [POS].

2 Input the axis name of the presetting axis.
The axis name blinks.

3 Input the coordinates value, and press horizontal soft key [PRESET].
The relative coordinate is preset by input coordinate value.

NOTE

1 When the preset of the specified axis is done in the following states, warning is displayed.
- When only the numerical value is input or the character that is not the axis name is input
  Waning massage: “THERE IS NO SPECIFIED NAME OF AXIS”
- When input form is except for “axis name + numerical value” is done
  Waning massage: “FORMAT ERROR”

2 When axis name display is changed by parameter No.3132, specify the changed axis name.

- Display including compensation values

M

The following parameters can be used to determine whether the displayed values include tool length compensation and cutter compensation.
- Workpiece coordinate system: Bits 6 (DAL) and 7 (DAC) of parameter No. 3104
- Relative coordinate system: Bits 4 (DRL) and 5 (DRC) of parameter No. 3104

T

The following parameters can be used to determine whether the displayed values include tool offset and tool nose radius compensation.
- Workpiece coordinate system: Bit 1 (DAP) of parameter No. 3129 and bit 7 (DAC) of parameter No. 3104
- Relative coordinate system: Bit 0 (DRP) of parameter No. 3129 and bit 5 (DRC) of parameter No. 3104

- Presetting by setting a coordinate system

M

Bit 3 (PPD) of parameter No. 3104 can be used to specify whether the position indication values in the absolute coordinate system are preset as those in the relative coordinate system during coordinate system setting or manual reference position return by the G92 command.

T

Bit 3 (PPD) of parameter No. 3104 can be used to specify whether the position indication values in the absolute coordinate system are preset as those in the relative coordinate system during coordinate system setting or manual reference position return by the G50 command (for G code system A) or G92 command (for G code system B or C).

FRYER MACHINE SYSTEMS INC
Rob Zelensky

12.1.10 Workpiece Coordinate System Preset (15/19-inch Display Unit)

If a workpiece coordinate system has been shifted with manual intervention or any other operation, an MDI operation can be performed to preset the system to a workpiece coordinate system that is offset by a workpiece origin offset, from the machine zero point that has been set before the shifting.
A command (G92.1) can be programmed to preset a workpiece coordinate system. (See Subsection, “Workpiece Coordinate System Preset (G92.1)”.)

Procedure for the workpiece coordinate system preset

Procedure

When all axes are performed to preset

1 Press function key [POS].

2 Press vertical soft key [ALL].

3 Press soft key [WK COORD].

4 Press soft key [ALL AXIS].

When a specified axis is performed to preset

1 Press function key [POS].

2 Press vertical soft key [ALL].

3 Press soft key [WK COORD].

4 Enter the axis name ([X], [Y], ...) and zero ([0]).

5 Press soft key [EXEC].

Explanation

- Operation mode

This function can be executed when the reset state or automatic operation stop state is entered, regardless of the operation mode.

- Presetting relative coordinates

As with absolute coordinates, bit 3 (PPD) of parameter No. 3104 is used to specify whether to preset relative coordinates (RELATIVE).

FRYER MACHINE SYSTEMS INC
Rob Zehensky

12.1.11 Actual Feedrate Display (15/19-inch Display Unit)

The actual feedrate on the machine (per minute) can be displayed on a current position display screen or program check screen.

Display procedure for the actual feedrate on the current position display screen

Procedure

1 Press the function key POS to display a current position display screen. At the location indicated by ⬚, an actual feedrate is displayed.

Fig. 12.1.11 (a) Current position screen (M series) (15-inch display unit)

Fig. 12.1.11 (b) Current position screen (T series) (15-inch display unit)

The actual feedrate is displayed in units of millimeter/min or inch/min (depending on the specified least input increment) under the display of the current position.

Explanation

- Actual feedrate value

The actual rate is calculated by the following expression:

$$Fact = \sqrt{\sum_{i=1}^{n} (fi)^2}$$

n : Number of axes
fi : Cutting feed rate in the tangential direction of each axis or rapid traverse rate
Fact : Actual feedrate displayed

The display unit:

mm/min (metric input)
inch/min (Inch input, two digits below the decimal point are displayed.)

- Actual feedrate display of feed per revolution

In the case of feed per revolution and thread cutting, the actual feedrate displayed is the feed per minute rather than feed per revolution.

- Actual feedrate display of rotary axis

In the case of movement of rotary axis, the speed is displayed in units of deg/min but is displayed on the screen in units of input system at that time. For example, when the rotary axis moves at 50 deg/min, the following is displayed: 50 mm/min (in metric input) or 0.50 inch/min (in inch input).

- Actual feedrate display on the other screen

The program check screen also displays the actual feedrate.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.1.12 Display of Run Time and Parts Count (15/19-inch Display Unit)

The run time, cycle time, and the number of machined parts are displayed on the current position display screens.

Procedure for displaying run time and parts count on the current position display screen

Procedure

1 Press the function key POS to display a current position display screen. At the location indicated by ⬚, a run time and parts count are displayed.

Fig. 12.1.12 (a) Current position screen (M series) (15-inch display unit)

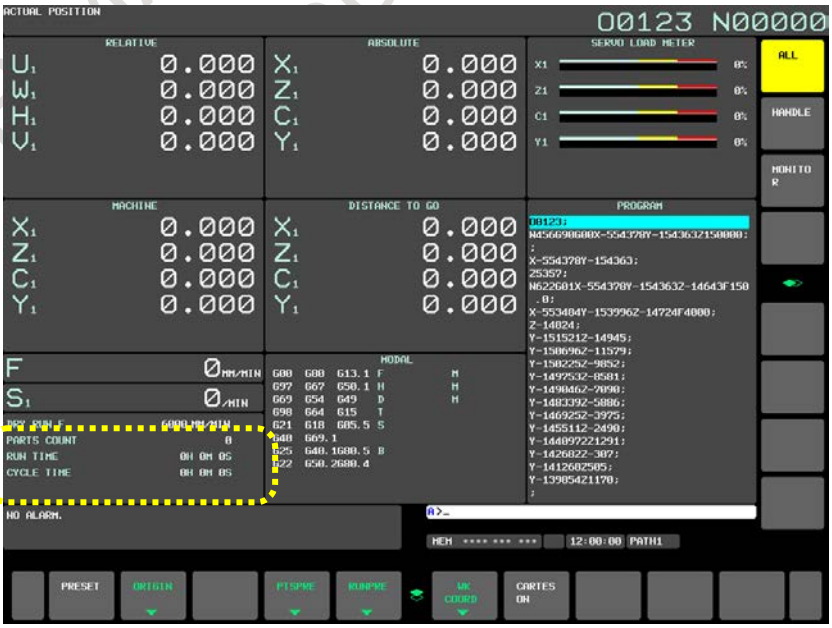


Fig. 12.1.12 (b) Current position screen (T series) (15-inch display unit)

FRYER MACHINES INC

The number of machined parts (PART COUNT), run time (RUN TIME), and cycle time (CYCLE TIME) are displayed under the current position.

Explanation

- PART COUNT

Indicates the number of machined parts. The number is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed.

- RUN TIME

Indicates the total run time during automatic operation, excluding the stop and feed hold time.

- CYCLE TIME

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- Display on the other screen

Details of the run time and the number of machined parts are displayed on the setting screen. See Subsection, “Displaying and Setting Run Time, Parts Count, and Time (15/19-inch Display Unit)”.

- Parameter setting

The number of machined parts and run time cannot be set on current position display screens. They can be set by parameters Nos. 6711, 6751, and 6752 or on the setting screen.

- Incrementing the number of machined parts

Bit 0 (PCM) of parameter No. 6700 is used to specify whether the number of machined parts is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed, or only each time an M code specified by parameter No. 6710 is executed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.1.13 Operating Monitor Display (15/19-inch Display Unit)

The load meter for each servo axis can be displayed. Also, the load meter and speed meter for a serial spindle can be displayed.
To enable this function, bit 5 (OPM) of parameter No. 3111 must be set to 1.

Procedure for displaying the operating monitor

Procedure

1 Press function key .

2 Press vertical soft key [MONITOR].

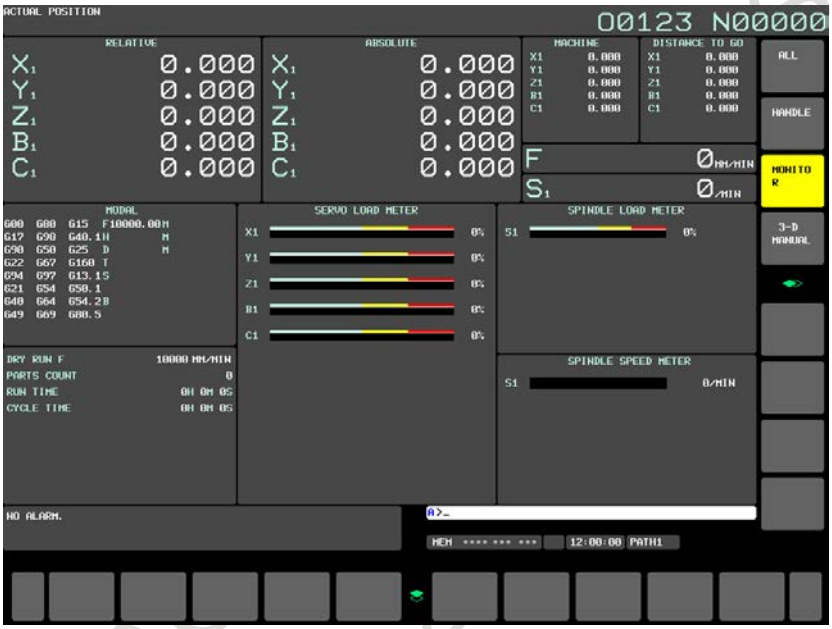


Fig. 12.1.13 (a) Operating monitor (M series) (15-inch display unit)

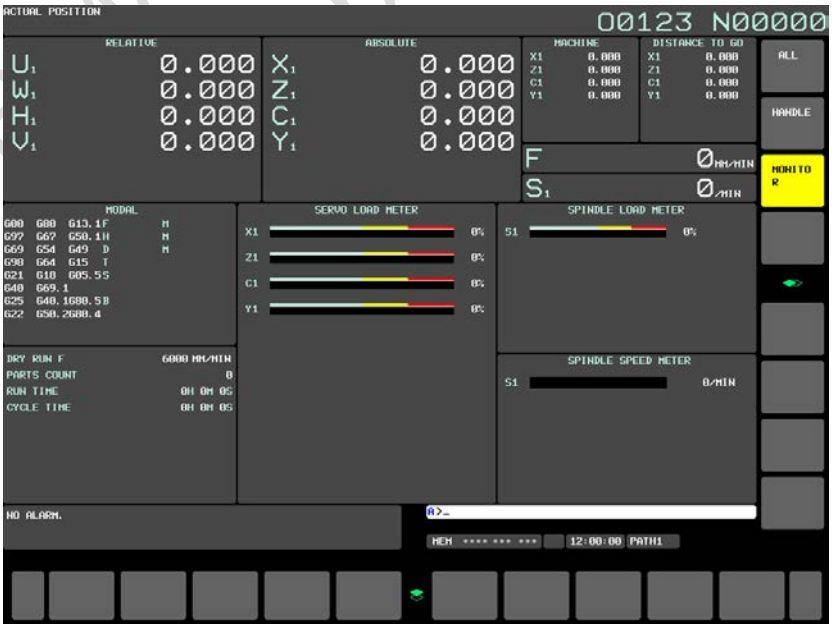


Fig. 12.1.13 (b) Operating monitor (T series) (15-inch display unit)

FRYER MACHINE TOOLS INC

Explanation

- Display of the servo axes

Servo axis load meters as many as the maximum number of controlled axes of the path can be displayed. One screen displays load meters for up to five axes at a time. By pressing the vertical soft key [MONITOR], load meters for the 6th axis and up are displayed.

- Display of the spindle axes

When serial spindles are used, the reading on the load meter and speedometer can be displayed only for the main serial spindle.

- Unit of graph

The bar graph for the load meter shows load up to 200% (only a value is displayed for load exceeding 200%).
The bar graph for the speedometer shows the ratio of the current spindle speed to the maximum spindle speed (100%).

- Load meter

The reading on the load meter depends on servo parameter No. 2086 and spindle parameter No. 4127.

- Speedometer

Although the speedometer normally indicates the speed of the spindle motor, it can also be used to indicate the speed of the spindle by setting bit 6 (OPS) of parameter No. 3111 to 1.
The spindle speed to be displayed during operation monitoring is calculated from the speed of the spindle motor (see the formula below).
The spindle speed can therefore be displayed, during operation monitoring, even when no position coder is used.
To display the correct spindle speed, however, the maximum spindle speed for each gear (spindle speed at each gear ratio when the spindle motor rotates at the maximum speed) must be set in parameters Nos. 3741 to 3744.
The input of the clutch and gear signals for the first serial spindle is used to determine the gear which is currently selected. Control the input of the CTH1A and CTH2A signals according to the gear selection, by referring to the table below.
(Formula for calculating the spindle speed to be displayed)

$$\text{Spindle speed displayed during operation monitoring} = \frac{\text{Speed of spindle motor}}{\text{Maximum speed of spindle motor}} \times \text{Maximum spindle speed with the gear being used}$$

The following table lists the correspondence between clutch and gear selection signals CTH1A and CTH2A, used to determine the gear being used, and parameters:

CTH1A	CTH2A	Parameter	Serial spindle specification
0	0	=No.3741 (Maximum spindle speed with gear 1)	HIGH
0	1	=No.3742 (Maximum spindle speed with gear 2)	MEDIUM HIGH
1	0	=No.3743 (Maximum spindle speed with gear 3)	MEDIUM LOW
1	1	=No.3744 (Maximum spindle speed with gear 4)	LOW

The speed of the spindle motor and spindle can be displayed, during operation monitoring, only for the first serial spindle and the spindle switching axis for the first serial spindle. It cannot be displayed for the second spindle.

- Color of graph

If the value of a load meter exceeds 100%, the bar graph turns red.

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

12.1.14 Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount) (15/19-inch Display Unit)

M

The absolute coordinates of the tool tip, the number of pulses, and a machine axis move amount based on 3-dimensional manual feed are displayed.

Displaying the screen for 3-dimensional manual feed

Procedure

1 Press function key POS.

2 Press the vertical soft key [3-D MANUAL] to display the screen of 3-dimensional manual feed.

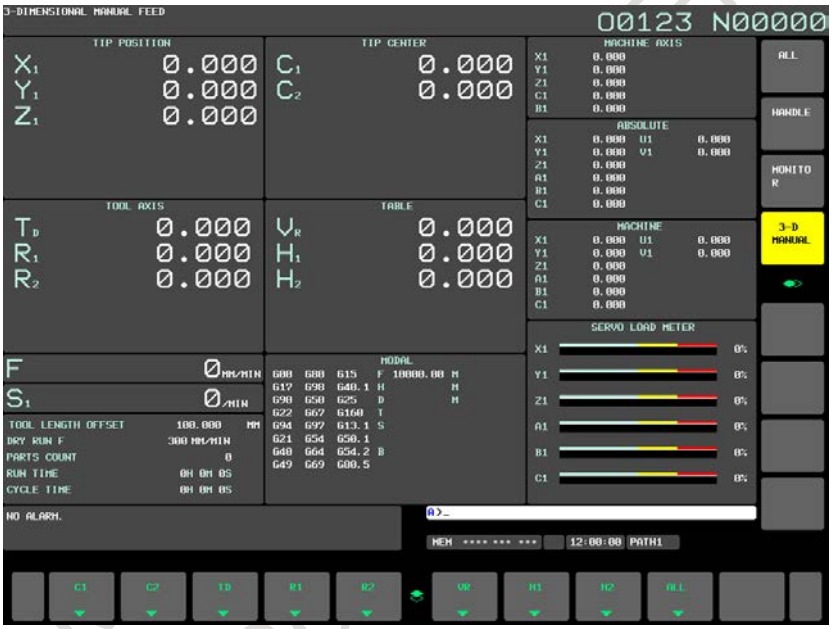


Fig. 12.1.14 (a) 3-dimensional manual feed screen(15-inch display unit)

Explanation

- Tool tip position

The addresses of the three basic machine configuration axes for performing 3-dimensional manual feed and the current position of the tool tip are displayed.

- Tool axis reference (number of pulses)

TD

The amount of travel in the tool axis direction in tool axis direction handle feed, tool axis direction jog feed, or tool axis direction incremental feed is displayed.
The unit is the least input increment of the axis in the direction specified by parameter No. 19697.

R1

The amount of travel in the first axis direction in tool axis right-angle direction handle feed, tool axis right-angle direction jog feed, or tool axis right-angle direction incremental feed is displayed.
The unit is the least input increment of the axis in the first axis direction normal to the direction specified by parameter No. 19697.

R2

The amount of travel in the second axis direction in tool axis right-angle direction handle feed, tool axis right-angle direction jog feed, or tool axis normal direction incremental feed is displayed.
The unit is the least input increment of the axis in the second axis direction normal to the direction specified by parameter No. 19697.

- Tool tip center (number of pulses)

C1

The angular displacement in tool tip center rotation handle feed, tool tip center rotation jog feed, or tool tip center rotation incremental feed for rotation of the first rotary axis is displayed. The unit is the least input increment of the first rotary axis. For the definition of the first rotary axis, see the description of parameter No. 19680.

C2

The angular displacement in tool tip center rotation handle feed, tool tip center rotation jog feed, or tool tip center rotation incremental feed for rotation of the second rotary axis is displayed. The unit is the least input increment of the second rotary axis. For the definition of the second rotary axis, see the description of parameter No. 19680.

- Table reference (number of pulses)

VR

The amount of travel in the table reference vertical direction in table reference vertical direction handle feed, table reference vertical direction jog feed, or table reference vertical direction incremental feed is displayed.
The unit is the least input increment of the axis in the direction specified by parameter No. 19697.

H1

The amount of travel in the first axis direction in table reference horizontal direction handle feed, table reference horizontal direction jog feed, or table reference horizontal direction incremental feed is displayed.
The unit is the least input increment of the axis in the first axis direction normal to the direction specified by parameter No. 19697.

H2

The amount of travel in the second axis direction in table reference horizontal direction handle feed, table reference horizontal direction jog feed, or table reference horizontal direction incremental feed is displayed.
The unit is the least input increment of the axis in the second axis direction normal to the direction specified by parameter No. 19697.

- Amounts of machine axis travel

The addresses of machine configuration axes used for 3-dimensional manual feed and the sum of the amount of travel of each axis used for 3-dimensional manual feed are displayed.
The values of the basic three axes (X-axis, Y-axis, and Z-axis), the first rotary axis, and the second rotary axis are displayed in this order.
For the definition of the first rotary axis and second rotary axis, see the description of parameter No. 19680.
When bit 0 (CLR) of parameter No. 13113 is set to 1, the displayed data is cleared by a reset.

- Absolute coordinates, machine coordinates

The absolute coordinates and machine coordinates of all axes are displayed. If too many axes are involved for display on one screen, the remaining axes can be displayed by pressing the vertical soft key [3-D MANUAL] for page feed.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- **F (feedrate)**
  - When bit 3 (CFD) of parameter No. 13113 is set to 0
    The composite feedrate at a control point on a linear axis or rotary axis is displayed.
  - When bit 3 (CFD) of parameter No. 13113 is set to 1
    The feedrate of the tool tip is displayed.

Operation

The display of the number of pulses can be cleared to 0 with horizontal soft keys.

1 Select the horizontal soft key corresponding to a function for which the display of the amount of travel is to be cleared.

2 Press horizontal soft key [ERASE] to clear the display of the amount of travel of the specified function, or press horizontal soft key [CAN] to cancel the operation.

12.2 SCREENS DISPLAYED BY FUNCTION KEY PROG

Screens of a10.4-inch display unit

This section describes the screens displayed by pressing function key PROG. The screens include a program editing screen, program folder list display screen, and screens for displaying the command states of the program currently being executed.

1. Program screen
2. Program folder screen
3. Next block display screen
4. Program check screen

On the program screen, you edit the program that is currently selected, and display the block that is currently executed during program operation. In MDI mode, you also edit an MDI operation program, and display the block that is currently executed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.2.1 Program Contents Display

Displays the program currently being executed in MEM mode.

Displaying the program being executed

Procedure

1 Press function key to display the program screen.

2 Press chapter selection soft key [PROGRAM].
The cursor is positioned at the block currently being executed.

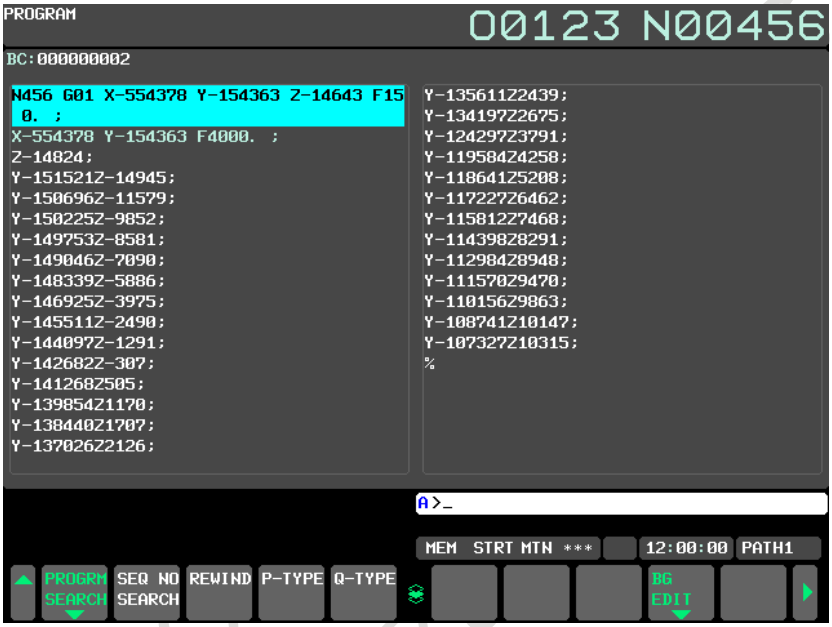


Fig. 12.2.1 (a) Screen for displaying the program being executed (10.4-inch display unit)

12.2.1.1 Displaying the executed block (BC)

Overview

When the program being executed is displayed, one executed block can be displayed. This function can be enabled by setting bit 3 (FPD) of parameter No. 11308.

The program screen shows the following:
- Executed block
- Block being executed
- Look-ahead block(*)
- Non-look-ahead block

(*) For details on look-ahead, see the explanation "Program look-ahead" below.

The executed block is displayed on the top of the screen, and ">" is shown in front of the block being executed, across which the cursor is placed. Below the block being executed, look-ahead blocks are displayed. Below them, non-look-ahead blocks are displayed.
There is a dividing line between the look-ahead blocks and the non-look-ahead blocks.

FRYER MACHINES INC Rob Z

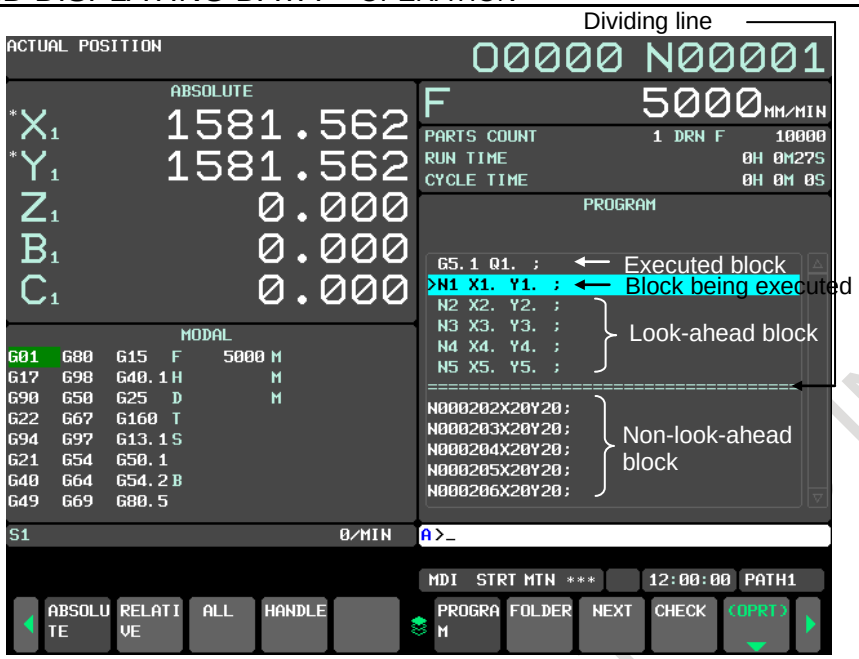


Fig. 12.2.1.1 (a) Screen for displaying the program being executed (MDI mode)

Fig. 12.2.1.1 (b) Screen for displaying the program being executed (MEM mode)

Explanation

- Program look-ahead

When an automatic operation cycle starts, a program is executed as described below.

<1> The command of one block is read from the specified program.
<2> The read command is analyzed and converted to executable data.
<3> The command of the block is executed.

During execution of the command, steps <1> and <2> are performed for the next block in advance. Upon completion of the current block, execution of the next block can be started immediately. This processing is referred to as look-ahead or buffering.

The number of look-ahead blocks depends on the processing status or parameters.

12.2.1.2 Text display

Overview

Setting bit 1 (APD) of parameter No. 11350 can select whether to display the contents of the running NC program in the look-ahead or text mode.

In the look-ahead mode (APD = 0):

Displays look-ahead blocks and non-look-ahead blocks in the running NC program.

In the text mode (APD = 1):

Displays blocks in the running NC program that is in the program memory. The displayed blocks include the block just before the block being executed.

(Example)

Fig. 12.2.1.2 (a) shows the display in the look-ahead mode and Fig. 12.2.1.2 (b) shows the display in the text mode when block N4 in program O0099 shown below is being executed.

```
O0099
N1 G91 G00 X100.
N2 Z100.
N3 #501 = 100.
N4 Y-100.
IF[#1 GT 10] GOTO 7
N5 Y#501 (COMMENT)
N6 M98 P9100
N7 G90 X100
N8 G300
N9 G00 X0Y0
N10 M30
```

Running program

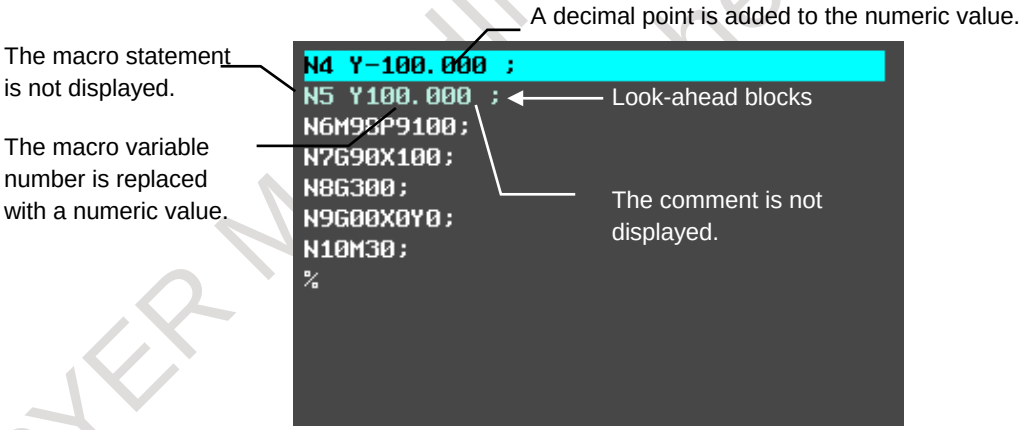


Fig. 12.2.1.2 (a) Display when block N4 is being executed (look-ahead mode)

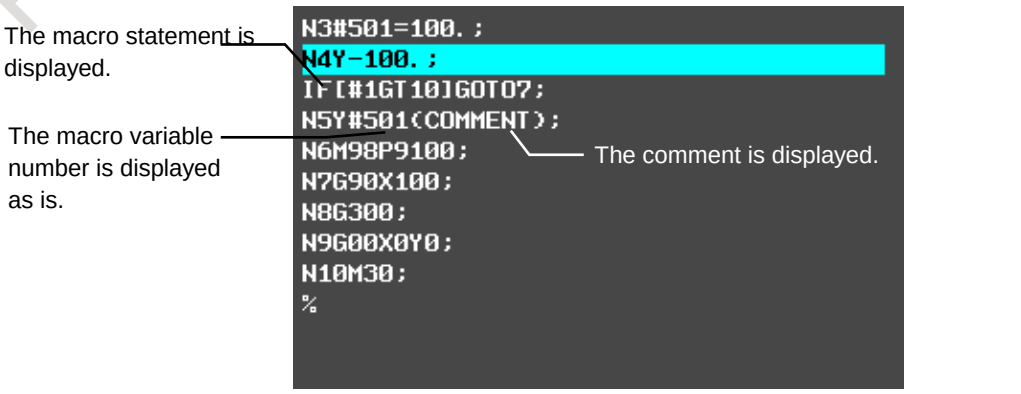


Fig. 12.2.1.2 (b) Display when block N4 is being executed (text mode)

External subprogram call

Setting bit 7 (DPD) of parameter No. 11356 can select whether to display the contents of the running external subprogram by DNC operation or M198 in the look-ahead or text mode.
However, text mode is effective when the number of look-ahead blocks is only less than 5.
When the number of look-ahead blocks is 5 or more, look-ahead mode is effective.

12.2.1.3 Call stack display

Overview

In the call stack display, the program name and nest level of subprogram from main program to current executing subprogram is displayed. When the device to which program is stored is a CNC build-in memory, the program comment is displayed in addition to a program name and a nest level. Moreover, the folder name where the program is stored is displayed in the active program now.

Call stack display

Procedure

1 Press the function key [PROG].
2 Press the soft key [CHECK].
3 Press the soft key [(OPRT)].
4 Press the soft key [▷] until soft key [CALL STK ON] appears.

Fig.12.2.1.3 (a) Program check screen (10.4-inch) (Non call stack display)

FRYER SYSTEMS INC

5 Press the soft key [CALL STK ON].

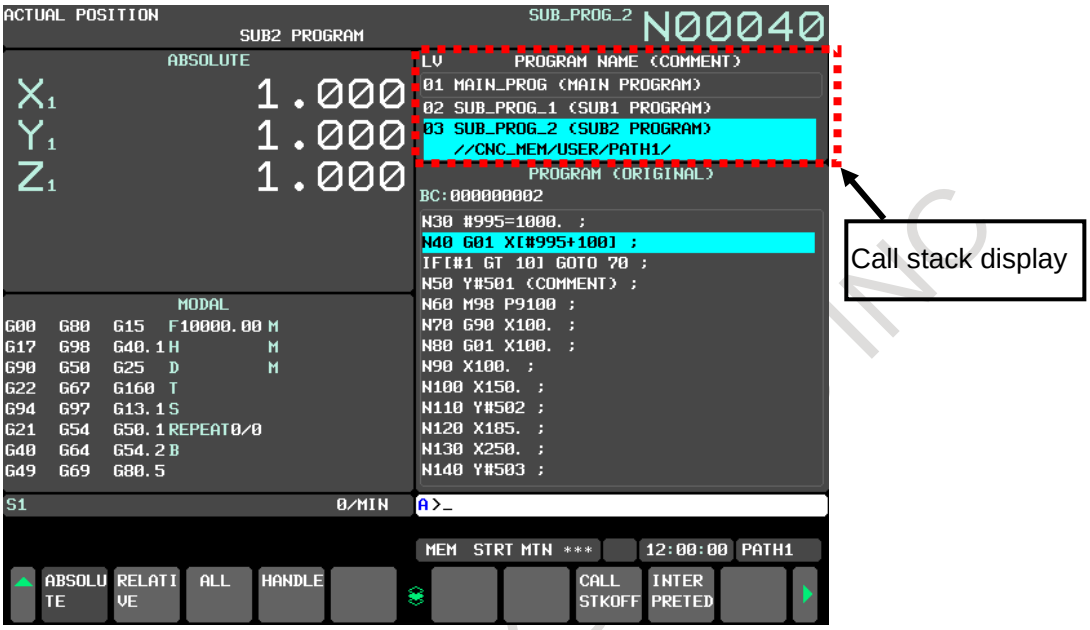


Fig.12.2.1.3 (b) Program check screen (10.4-inch) (Call stack display)

A part of the program check screen changes into the call stack.
Moreover, softkey [CALL STK ON] changes into softkey [CALL STKOFF].

6 The screen returns to Fig.12.2.1.3 (a) by pressing the soft key [CALL STKOFF].

Explanation

In the call stack, nest level and program name and program comment of subprogram from main program to program being executed is displayed. Moreover, about the program being executed, in addition to program name and program comment, the folder information is displayed.
The following items are displayed on the screen.

Table12.2.1.3 Item of call stack display

Content of display	Explanation
NEST LEVEL	Level of subprogram calling from main program to program being executed is displayed by the number. The main program is displayed in the line of nest level 01.
PROGRAM NAME	The name of subprogram called from the main program is displayed. And when all calling program cannot be displayed within the area, it is displayed from the one with large number of nest level. Moreover, the folder where the program is stored is displayed for the active program. The head of the folder is a device name.
PROGRAM COMMENT	In case of CNC build-in memory, Program comment is displayed in parentheses at the right of program name.

FRYER MACHINE SYSTEMS INC
Jacob Zahersky

Example of screen

When the block of the cursor position is executed by the subprogram call from the main program that exists in the figure below,

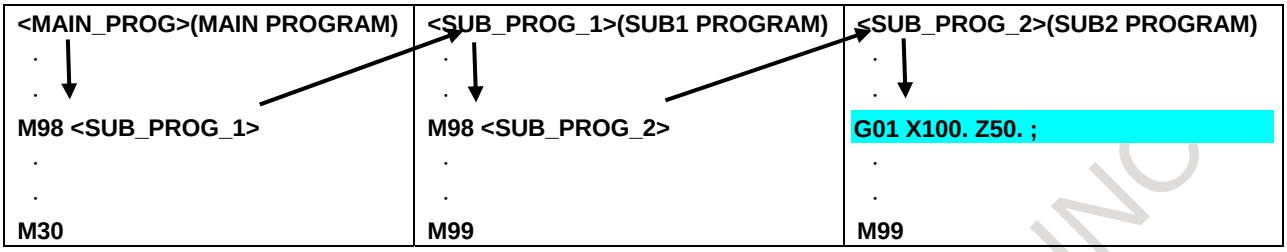


Fig.12.2.1.3 (c) Example of subprogram call

The call stack is displayed as shown in the figure below.

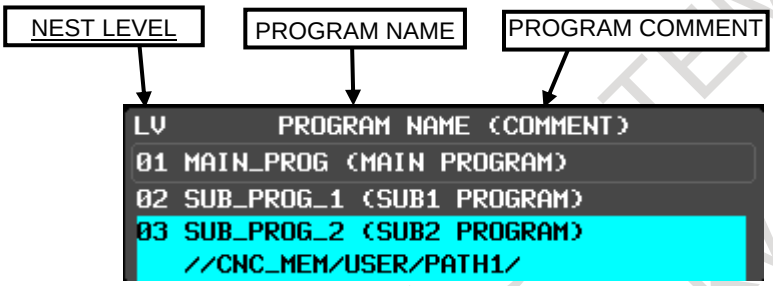


Fig.12.2.1.3 (d) Call stack display

NOTE

1 When the storing number of program name and program comment and the folder information exceed display area, the exceed part is displayed by the "…".
2 As for MDI program, the program name is displayed as O0000 in the call stack.
3 As for External program, the folder where the program is stored is not displayed and the top of program path name is displayed as "//DNC" in the call stack.
4 The programs of external subprogram call (M198) and execution macro are not displayed in the call stack.
5 In the multi-path system simultaneous display, the call stack display can not be performed on the same screen. However, if single-path display mode is set (bit 3 (MTS) of parameter No.11355#3 is 1), the call stack can be displayed.

12.2.1.4 Display interpreted and original programs on the same screen

Overview

The input custom macro programs before conversion (original) and after conversion (interpreted) are displayed on the program check screen at the same time.
If bit 5 (EPC) of parameter No.11356 is set to 1, this function becomes effective.

FRYER MACHINE SYSTEMS INC Rob Zhonsky

Display interpreted and original programs on the same screen

Procedure

1 Press the function key [PROG].

2 Press the soft key [CHECK].

3 Press the soft key [(OPRT)].

4 Press the soft key [▷] until soft key [INTERPRETED] appears.

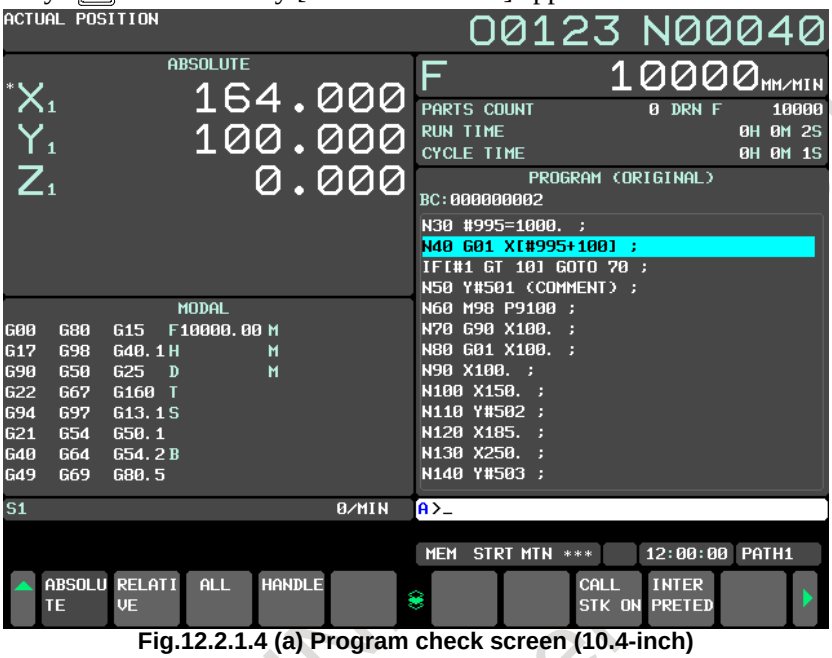


Fig.12.2.1.4 (a) Program check screen (10.4-inch)
(Non display interpreted and original programs on the same screen)

5 Press the soft key [INTERPRETED].
The original program display is switched to the simultaneous display of interpreted program and original program.
And the soft key [INTERPRETED] is changed to [ORIGINAL].

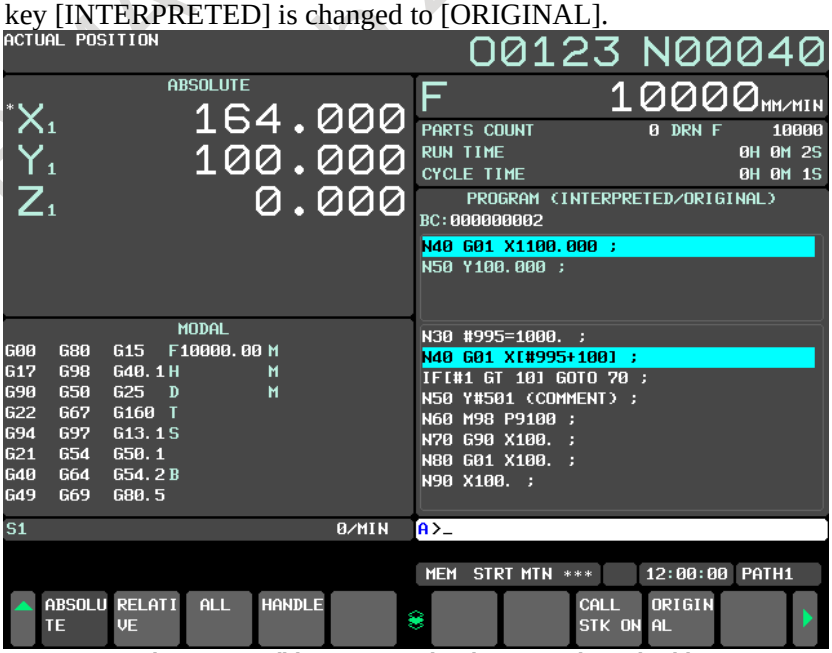


Fig.12.2.1.4 (b) Program check screen (10.4-inch)
(Display interpreted and original programs on the same screen)

6 The screen returns to Fig.12.2.1.4 (a) by pressing the soft key [ORIGINAL].

Explanation

- For interpreted program

The program after the custom macro program is calculated is displayed.
Both the block being executed now and the block executed next to the executing block are displayed.
When the block executed next is a call command of the subprogram such as M98, called program is displayed.

- For original program

The program before the custom macro program is calculated is displayed.
The blocks displayed before and after the active block are displayed in the order of registration in the memory.
The block is not necessarily a block executed last time, or, the block is not necessarily a block that will be executed next time either.

The difference point of the original program and the interpreted program is as follows.

Table12.2.1.4 The difference point of the original program and the interpreted program

	Comment	Macro program	Macro Variable	Decimal Point	Subprogram call
Interpreted Program	Not displayed	Not displayed	Value stored in macro variable	Least increment system	Not displayed
(Example)	G01	(Not displayed)	G01 X123.456 ;	G01 X1.000 ;	(Not displayed)
Original Program	Displayed	Displayed	Macro variable	The decimal point position is the same as the input.	Displayed
(Example)	G01 (COMMENT)	IF [#1 GT 10] GOTO 20	G01 X#100 ;	G01 X1. ;	M98P1000;

Example of screen

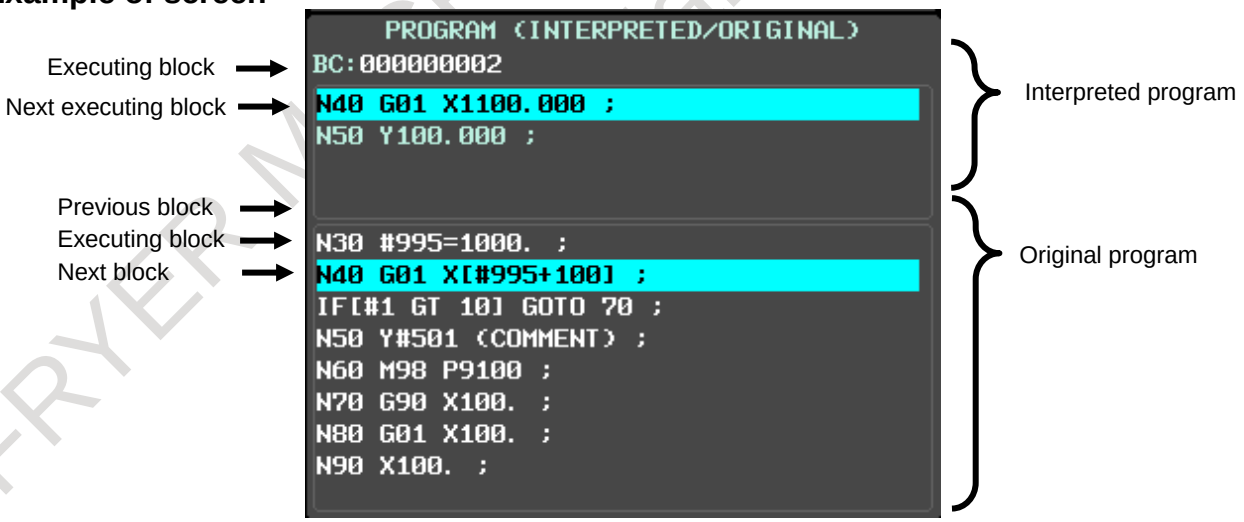


Fig.12.2.1.4 (c) Display interpreted and original programs on the same screen

NOTE

1 In case of the program stored in internal memory (CNC MEM) and the program in memory card program device (MEMCARD), the interpreted and original programs are displayed on the same screen.
2 Protected programs and execution macro programs are not displayed.
3 The interpreted program is displayed in the area of the original program display during DNC operation.

FRYER MACHINE SYSTEMS INC

NOTE

4 In operating of single block, the cursor shows the executed block. The cursor doesn't move to the next block.

5 In the multi-path system simultaneous display, the display interpreted and original programs can not be performed on the same screen.

Restrictions

This function cannot be used at the same time with the following functions.

- Manual Handle Retrace
- Retrace

12.2.2 Editing a Program

A program can be edited in the EDIT mode.
Two modes of editing are available. One mode is word editing, which performs word-by-word editing. The other is character editing, which performs character-by-character editing.
For program creation and editing operation, see Chapter III-9, "CREATING PROGRAMS" and Chapter III-10, "EDITING PROGRAMS".

Displaying the program editing screen

Procedure

1 Press function key PROG to display the program screen.

2 Press chapter selection soft key [PROGRAM].

- Word editing

Editing operations such as text insertion, modification, and deletion, and cursor movements are performed on a word-by-word basis.

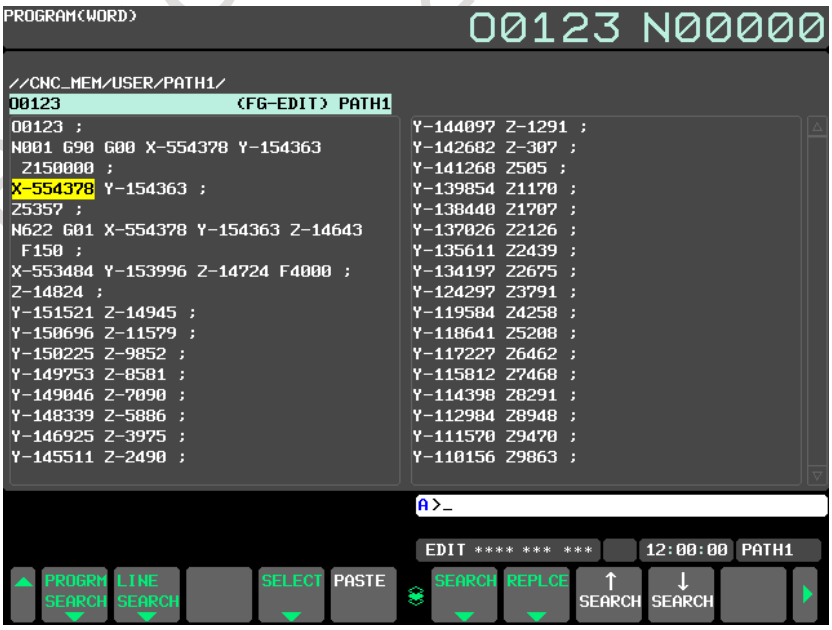


Fig. 12.2.2 (a) Program word editing screen (10.4-inch display unit)

FRYER CHINE SYSTEMS INC
Zahensky

- Character editing

Program editing operations and cursor movements are performed on a character-by-character basis as with a general text editor.
Text is input directly to the cursor position instead of using the key input buffer.

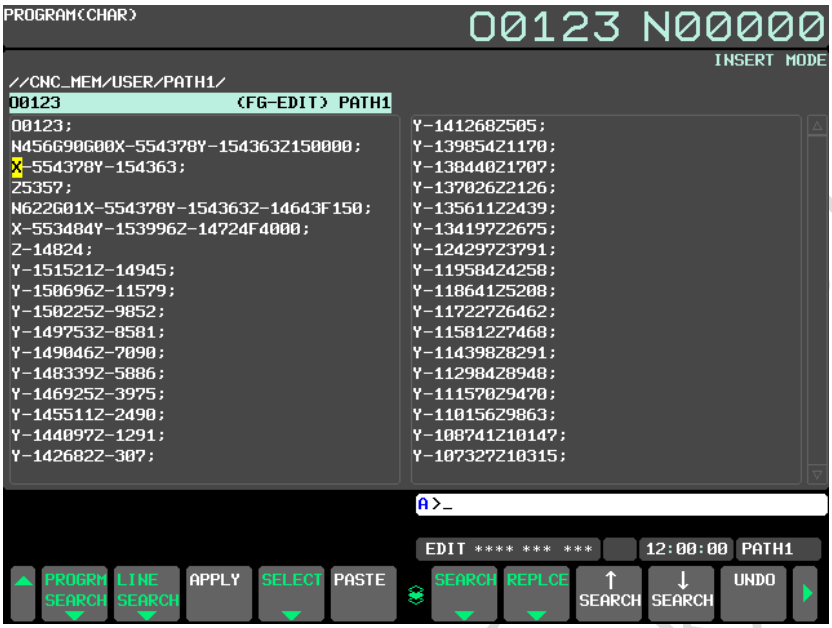


Fig. 12.2.2 (b) Program character editing screen (10.4-inch display unit)

Switching between program editing modes

You can switch between word editing and character editing with soft keys.

Procedure

1 Press function key PROG to display the program screen.
2 Press chapter selection soft key [PROGRAM].
3 Press operation soft key [(OPRT)].
4 Pressing operation soft key [CHANGE EDITOR] switches the editing mode between word editing and character editing.

FRYER MACHINE INC
Rob Zahensky

12.2.3 Program Screen for MDI Operation

During MDI operation or editing of an MDI operation program in the MDI mode, the program currently being executed mode is displayed.
For MDI operation, see Section III-4.2, “MDI Operation”.

Procedure for displaying the program screen for MDI operation

Procedure

1 Press function key .

2 Press chapter selection soft key [PROGRAM].
The program input from the MDI is displayed.

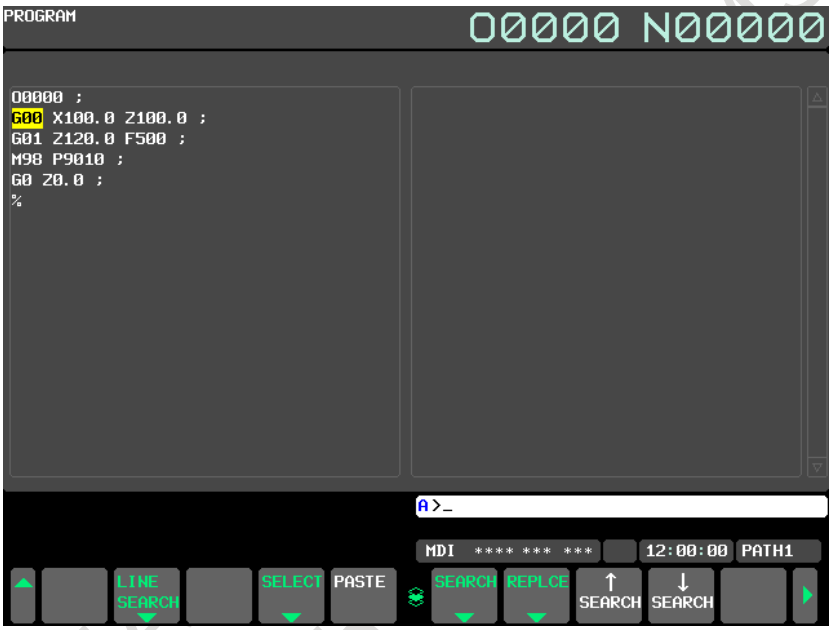


Fig. 12.2.3 (a) MDI operation program screen (10.4-inch display unit)

FRYER MA... Rob... ...S INC

12.2.4 Program Folder Screen

A list of programs registered in the program memory is displayed.
For the program folder screen, see Chapter III-11, “PROGRAM MANAGEMENT”.

Displaying the program folder screen

Procedure

1 Press function key [PROG].

2 Press chapter selection soft key [FOLDER].

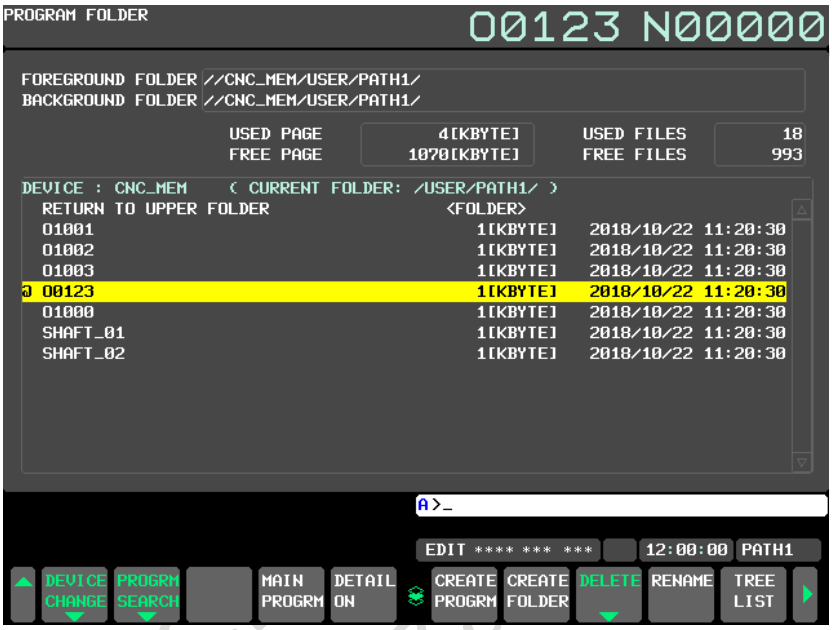


Fig. 12.2.4 (a) Program folder screen (10.4-inch display unit)

Displaying comments on the program folder screen

The comments of the first block of the program are displayed by setting bit 1 (COL) of parameter No. 11351 to 1.

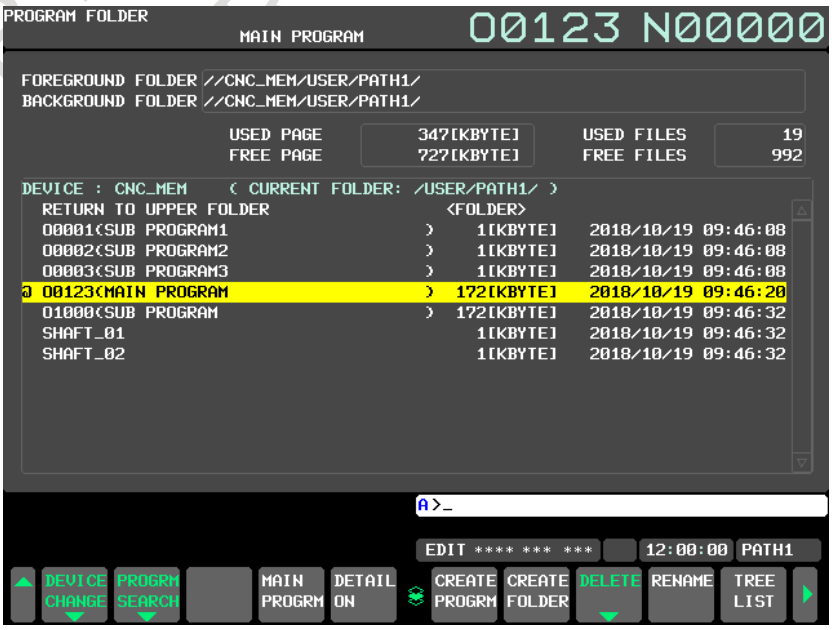


Fig.12.2.4 (b) Displaying comments on the program folder screen

12.2.4.1 Split display on the program folder screen

Overview

On the program folder screen, the folder information display can be split into two folder information views, upper and lower, as shown in the Fig. 12.2.4.1 (a). This function is enabled by setting bit 1 (MLD) of parameter No. 11318 to 1.

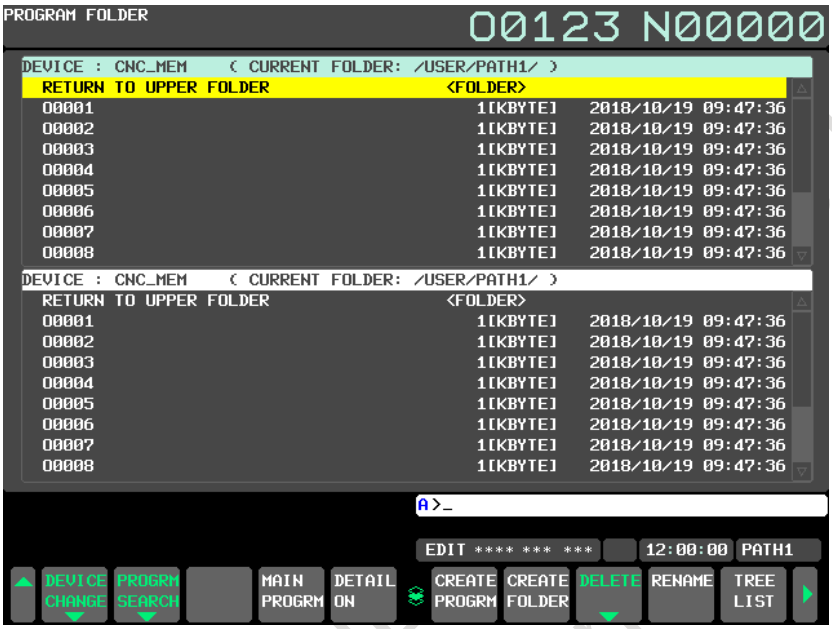
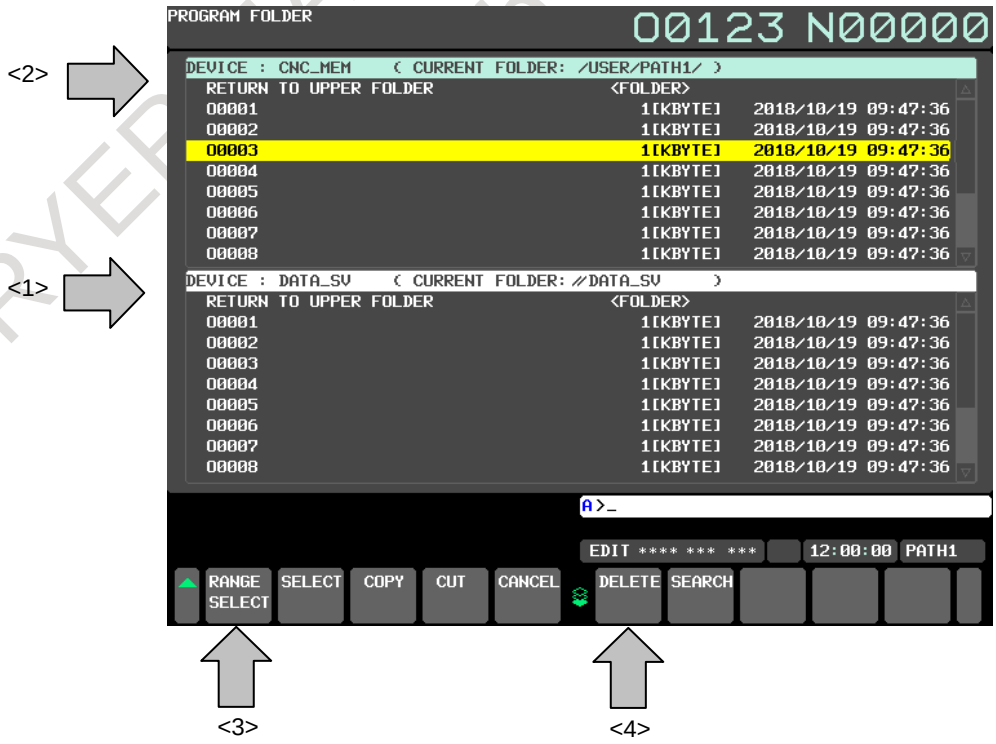


Fig. 12.2.4.1 (a) Program folder screen (split-display screen)(10.4-inch display unit)

On the split folder display, file information about different sets of devices or folders can be shown in different views at the same time, so you can copy or move files easily.

Example of copying a file from the CNC MEM to the Data Server



Procedure

<1> Change to a destination folder on the Data Server.
<2> Change to a folder on the CNC_MEM that contains a file you want to copy.
<3> Select the file.
<4> Copy the file.

Procedure for switching to split display

Procedure

1 Press function key [PROG] to display the program folder screen.
2 Press soft key [(OPRT)].
3 Press continuous menu key [▷] twice.
Soft key [MULTI LIST] appears.

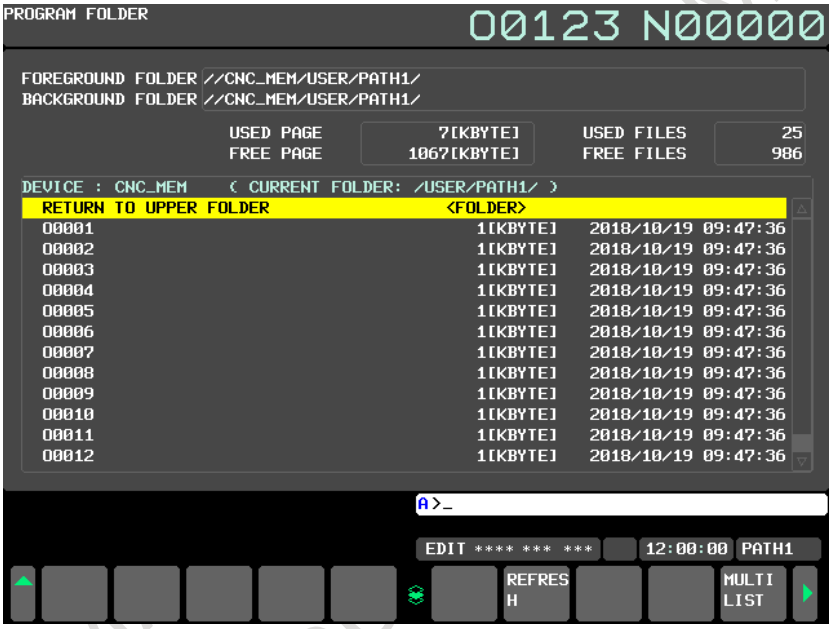


Fig. 12.2.4.1 (b) Program folder screen (normal screen)(10.4-inch display unit)

FRYER MACHINE TOOLS INC

4 Press soft key [MULTI LIST].
The folder information display is split into two folder views, upper and lower, which show the same folder information. Immediately after the splitting, the upper folder view becomes active for operations. At the same time, soft key [SINGLE LIST] appears.

Fig. 12.2.4.1 (c) Program folder screen (split-display screen)(10.4-inch display unit)

5 Pressing soft key [SINGLE LIST] takes you back to the original program folder display.
Then, the folders are shown which have been selected in the active view for operations on the split display.

Procedure for switching between active folder views for file operations on the split display

On the split folder display, you can switch between active folder views for file operations as described below.
In the active folder view for file operations, the cursor is displayed on the file list.

Procedure

1 When the upper folder view is active for file operations and you press MDI key [SHIFT] and cursor key [↓], the lower folder view becomes active for file operations.

2 When the lower folder view is active for file operations and you press MDI key [SHIFT] and cursor key [↑], the upper folder view becomes active for file operations.

NOTE

- In step 1, even if you press MDI key [SHIFT] and cursor key [↑] when the upper folder view is active for file operations, the lower folder view becomes active for file operations.
- In step 2, even if you press MDI key [SHIFT] and cursor key [↓] when the lower folder view is active for file operations, the upper folder view becomes active for file operations.

FRYER MACHINE - Rob Zahenski INC

Explanation

- File operations on the split display

The following operations are allowed on the split display.

<1> You can select and display devices or folders individually in each folder view.

<2> You can copy or move files between folders selected in different folder views.
(For details, see the section on operations of copying/moving between devices.)

In the selected folder view, you can perform the same operations as on the normal screen.

- Path switching

When path switching is performed while the split display is provided for a multi-path system, the folder view in which the CNC_MEM is selected as a device is changed to a view of the last selected folder for each path on the CNC_MEM.

- Device information display

The device information as shown in the normal program folder screen (portion in a dotted box in the Fig. 12.2.4.1 (d)) is not shown on the split display (Fig. 12.2.4.1 (c)).

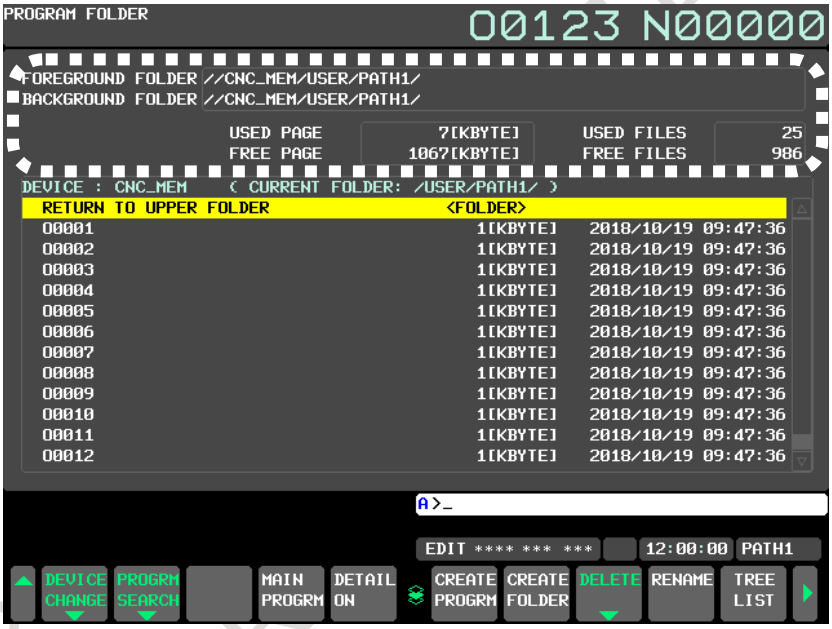


Fig. 12.2.4.1 (d) Program folder screen (normal screen)(10.4-inch display unit)

You should see the device information on the normal screen.

Limitation

- Device that can be selected in both views at the same time on the split display

The same device of any of the following types can be selected in both views at the same time on the split display.

- CNC MEM
- MEM CARD(binary format)
- Data Server

For other types, the same device cannot be selected for display in the folder views at the same time.

If any other type of device is selected when split display starts, the upper and lower folder views cannot show the same sets of folders due to the above limitation. Therefore, the lower folder view shows the folder selected as a foreground folder.

- **Display units that support the split display**

The split display is supported on 10.4-inch, 15-inch, and 19-inch display units.

12.2.4.2 Folder tree display

Overview

The folder tree can be displayed on program folder screen. By using this function, it is easy to manage programs of each folder.
The folder tree is available in the program memory device (CNC_MEM), the USB device, the memory card program device (MEMCARD), and the data server device.

Explanation

The folder tree is displayed the left side of the screen, and the list of folders and programs is displayed the right side on the screen.
On the folder tree screen, current folder rank and selected folder are displayed.
On the program list screen, programs and folder of selected folder are displayed.

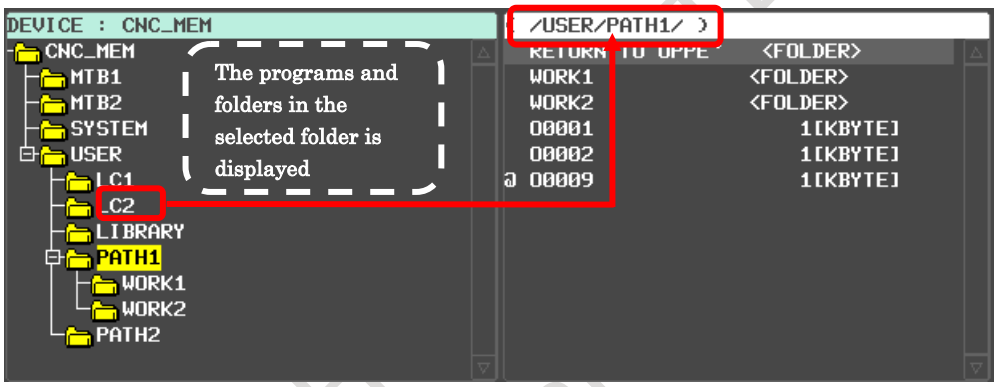


Fig.12.2.4.2 (a) Program folder tree display

Procedure for displaying folder tree

1. Press function key .
2. Press chapter selection soft key [FOLDER].
3. Press the soft key [(OPRT)].
4. Press the soft key [TREE LIST].
5. The folder tree like the above is displayed.
6. If the press the soft key [LIST], the program list screen without folder tree is displayed.

FRYER MACH... Rob Za... TEMS INC

Procedure for changing folder

In the folder tree screen, a folder is changed by moving the cursor.

Selecting upper folder: Press the cursor key ←.

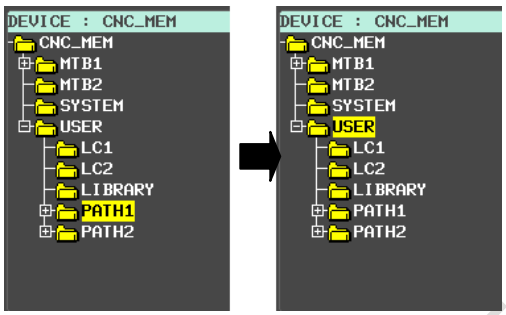


Fig.12.2.4.2 (b)

Select under folder: Press the cursor key →.

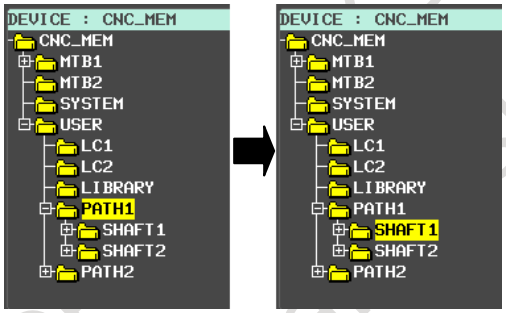


Fig.12.2.4.2 (c)

Select next or back folder: Press the cursor key ↑ or ↓.

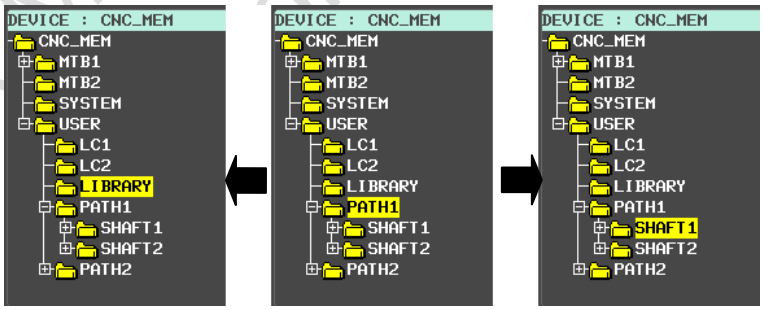


Fig.12.2.4.2 (d)

FRYER MACHINE SYSTEMS INC

12.2.5 Next Block Display Screen

Displays the block currently being executed and the block to be executed next.

Procedure for displaying the next block display screen

Procedure

1 Press function key .

2 Press chapter selection soft key [NEXT].
The G codes, addresses, command values specified in the block currently being executed and the next block are displayed.

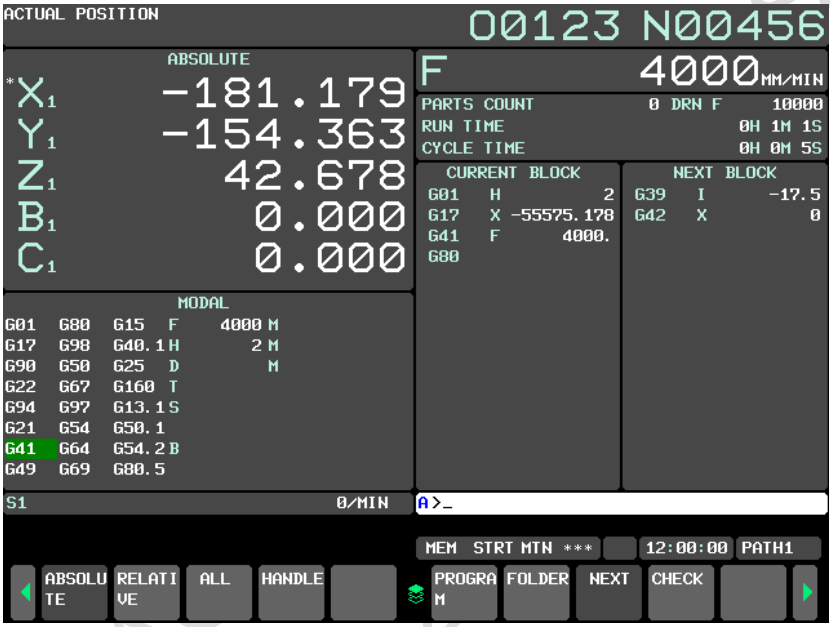


Fig. 12.2.5 (a) Next block display screen (10.4-inch display unit)

FRYER MACHINES INC
Robo

12.2.6 Program Check Screen

Displays the program currently being executed, current position of the tool, and modal data.

Procedure for displaying the program check screen

Procedure

1 Press function key [PROG].

2 Press chapter selection soft key [CHECK].
The program currently being executed, current position of the tool, and modal data are displayed.

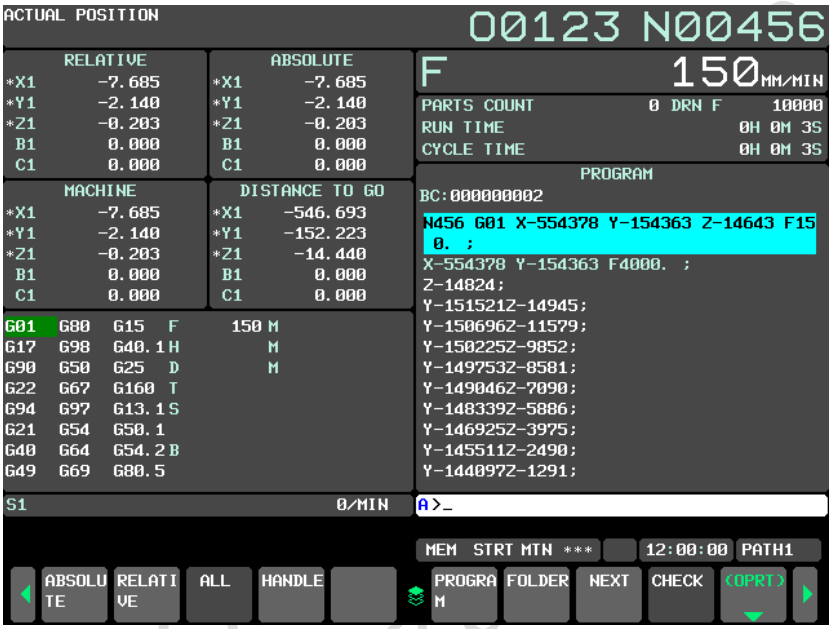


Fig. 12.2.6 (a) Program check screen (10.4-inch display unit)

Explanation

- Program display

The program currently being executed is displayed.
The block being executed is displayed in reverse video.

- Current position display

The current position in the relative coordinate system, workpiece coordinate system, and machine coordinate system, and the remaining amount of travel are displayed.

- Modal G codes

Up to 24 modal G codes are displayed.

FR TEP MAC Rob Z INC

12.2.6.1 Modal G codes display in 3 columns on program check screen

Overview

Modal G codes can be displayed up to 12 codes in 3 columns on program check screen when the virtual MDI function is used.

Explanation

When this function is effective, modal G codes can be displayed up to 12 codes in 3 columns on program check screen when the virtual MDI function is used.
When this function is disabled, modal G codes can be displayed up to 8 codes in 2 columns.
To enable this function, please set following parameters.

- Modal G codes can be displayed in 3 columns (bit 5 (M3D) of parameter No.24308 is set to 1)
- Extended axis name disabled (bit 0 (EEA) of parameter No.1000 is set to 0)
- Call stack display disabled (bit 5 (CDE) of parameter No.11374 is set to 1)
- Display interpreted and original programs on the same screen is disabled (bit 5 (EPC) of parameter No.11356)

Virtual MDI key

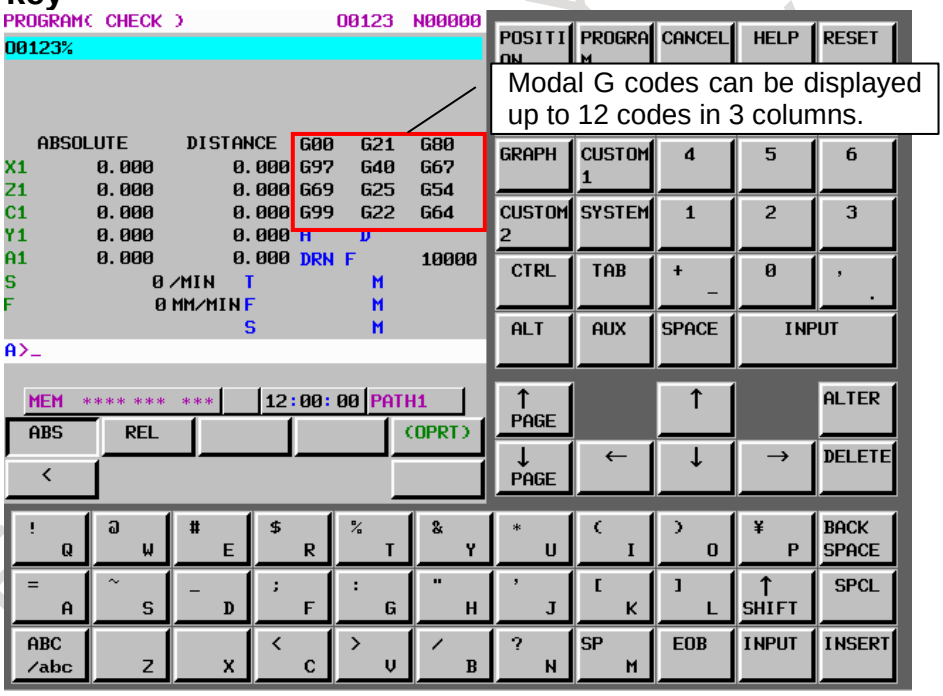


Fig.12.2.6.1 (a) Modal G codes displayed in 3 columns (Virtual MDI key)

FRYE SYSTEMS INC

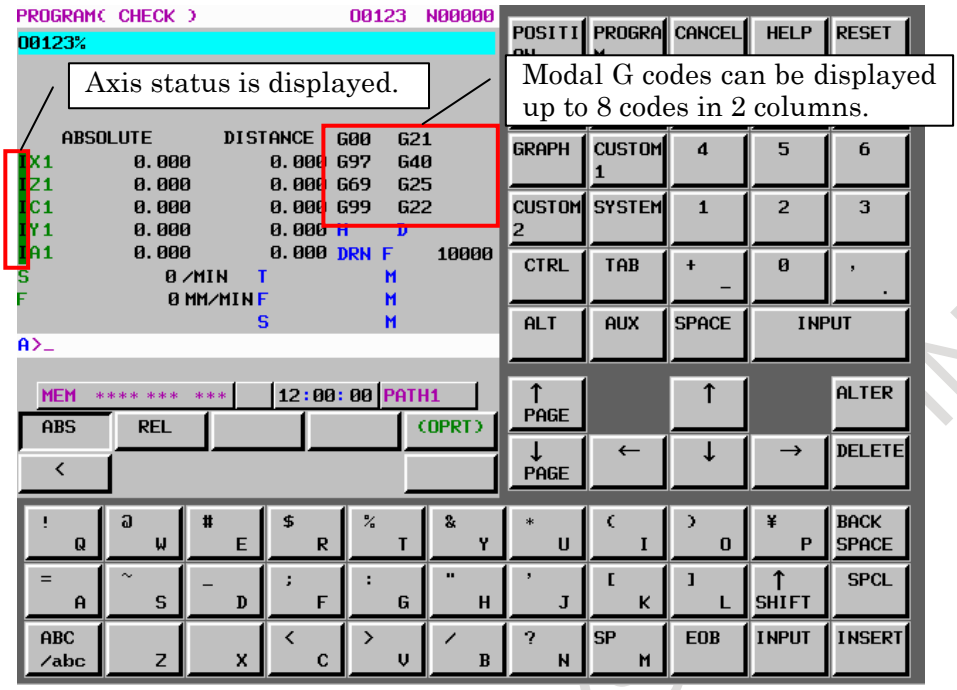


Fig.12.2.6.1 (b) Modal G codes displayed in 2 columns (Virtual MDI key)

Restrictions

When this function is effective, there are following restrictions.

- Axis status is not displayed
- Expanded axis name, call stack display, and display interpreted and original programs on the screen are not available.

12.2.6.2 Current block screen

Overview

Current block and the modal command are displayed when the virtual MDI function is used.
Current block is displayed if following parameter set to 1.

Bit 6 (CBD) of parameter No.24308

Displaying current block

Procedure

1 Press function key [PROG].

2 Press soft key [CURRNT].

G code, address, command value, and modal value of the block currently being executed are displayed.

FRYER MACHINE SYSTEMS INC
Rod Zahensky

Virtual MDI

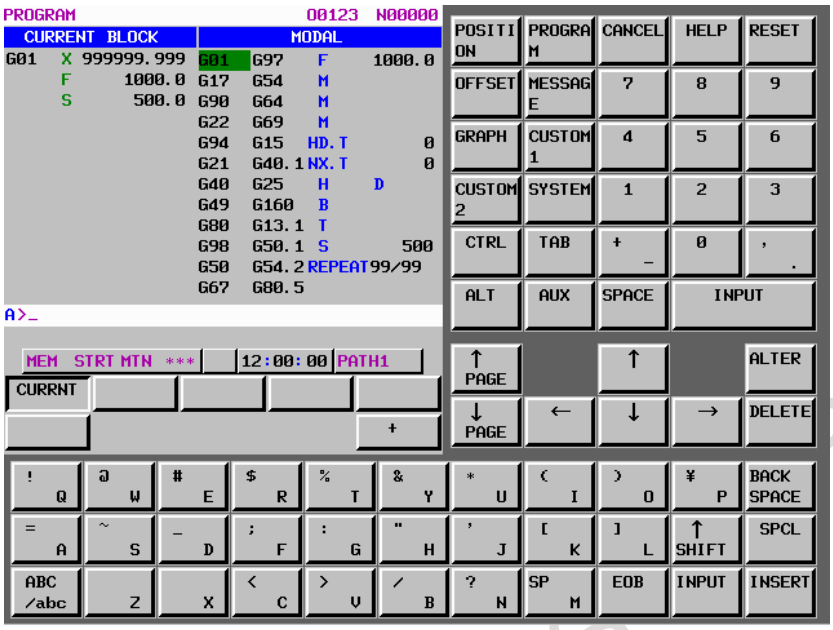


Fig.12.2.6.2 (a) Current block (Virtual MDI key)

Explanation

Modal information displayed on current block screen differs for each system configuration.

M

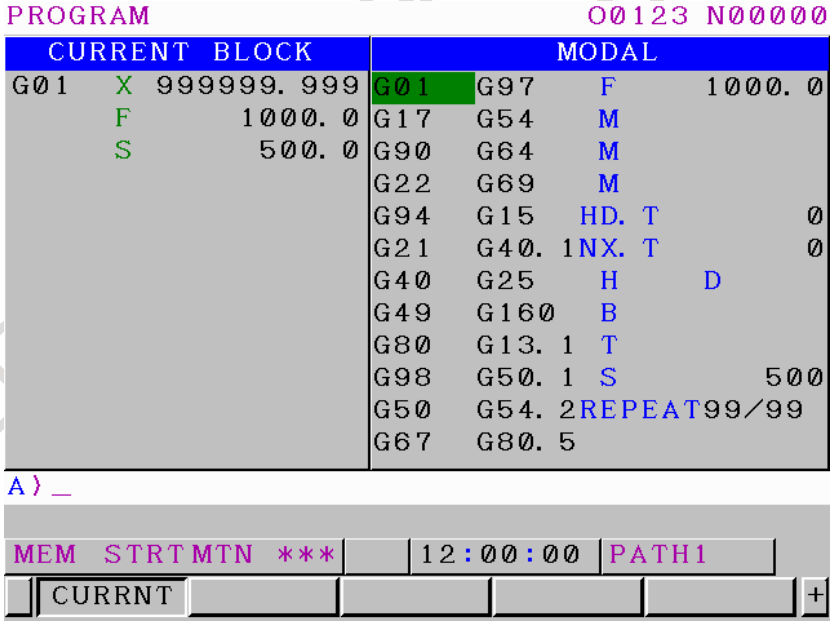


Fig.12.2.6.2 (b) M

T

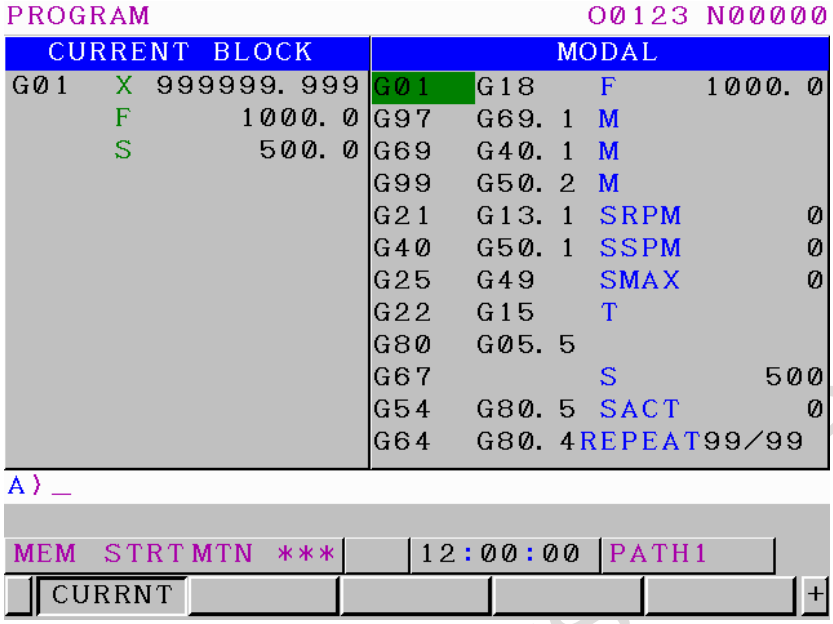


Fig.12.2.6.2 (c) T

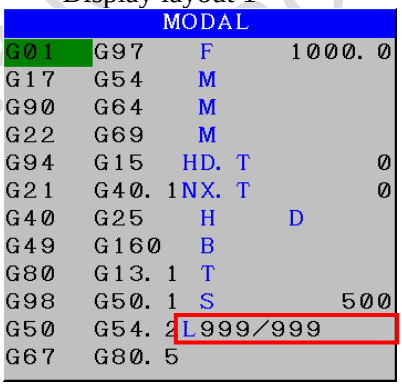
Number of repetitive calls

When M98 is called, number of repetitive call of the subprogram is displayed.
Title and number of repetitive calls changes as follows by number of calls of command.

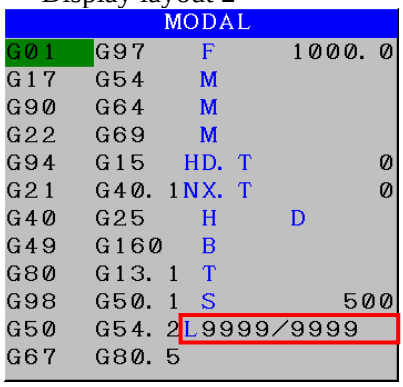
Table12.2.6.2 (a) Title and number of repetitive calls

Specified number of repetitive calls	Number of repetitive calls	Title	Display layout
0 to 99	Displayed	REPEAT	1
100 to 9999	Displayed	L	2
10000 to 999999	Not displayed	REPEAT	3
1000000 to 99999999	Not displayed	L	4

Display layout 1



Display layout 2



FRYER MACHINE SYSTEMS INC
Bob Zaremsky

Display layout 3

```
MODAL
G01  G97    F      1000. 0
G17  G54    M
G90  G64    M
G22  G69    M
G94  G15   HD. T         0
G21  G40. 1NX. T         0
G40  G25    H      D
G49  G160   B
G80  G13. 1 T
G98  G50. 1 S          500
G50  G54. 2L9999999
G67  G80. 5
```

Display layout 4

```
MODAL
G01  G97    F      1000. 0
G17  G54    M
G90  G64    M
G22  G69    M
G94  G15   HD. T         0
G21  G40. 1NX. T         0
G40  G25    H      D
G49  G160   B
G80  G13. 1 T
G98  G50. 1 S          500
G50  G54. 2L99999999
G67  G80. 5
```

Fig 12.2.6.2 (d) Title and number of repetitive calls

Changing of modal information

Following modal information can be displayed.
These are displayed in the same display area.
If they are multiple display setting, they are displayed according to priority.

- HD.T/NX.T
- A/C code
- 4th ,5th M code
- SRPM/SSPM/SMAX

Priority of modal information display

Priority is as follows.

Table 12.2.6.2 (b) Priority of modal information displaying

Priority	M	T
1	HD.T/NX.T	4th, 5th M code
2	A/C code	SRPM/SSPM/SMAX
3	4th, 5th M code	–

HD.T/NX.T, T code (M)

HD.T/NX.T/T code of modal information can display according to following parameters.

Table 12.2.6.2 (c) Display layout of HD.T/NX.T/T code

DHN(No.11320#0)	PCT(No.3108#2)	HD.T/NX.T	T code	Display layout
1	1	Display	Display	1
1	0	Display	Display	
0	1	Display	Not display	2
0	0	Not display	Display	3

Display layout 1

```
MODAL
G00  G97    F
G17  G54    M
G90  G64    M
G22  G69    M
G94  G15   HD. T         0
G21  G40. 1NX. T         0
G40  G25    H      D
G49  G160   B
G80  G13. 1 T
G98  G50. 1 S
G50  G54. 2REPEAT99/99
G67  G80. 5
```

Display layout 2

```
MODAL
G00  G97    F
G17  G54    M
G90  G64    M
G22  G69    M
G94  G15   HD. T         0
G21  G40. 1NX. T         0
G40  G25    H      D
G49  G160   B
G80  G13. 1
G98  G50. 1 S
G50  G54. 2REPEAT99/99
G67  G80. 5
```

Display layout 3

```
MODAL
G00  G97    F
G17  G54    M
G90  G64    M
G22  G69    M
G94  G15
G21  G40. 1
G40  G25    H      D
G49  G160   B
G80  G13. 1 T
G98  G50. 1 S
G50  G54. 2REPEAT99/99
G67  G80. 5
```

Fig. 12.2.6.2 (e) HD.T/NX.T, T code

FRYER MACHINE SYSTEMS INC
Rob Zahansky

B code (M)

If second auxiliary function is enabled(parameter No.8132#2(BCD)), B code of modal information is displayed.

Table 12.2.6.2 (d) Display layout of A/C code

Second auxiliary function	B code	Display layout
Enabled	Displayed	1
Disabled	Not displayed	2

Display layout 1

```
MODAL
G00  G97   F
G17  G54   M
G90  G64   M
G22  G69   M
G94  G15
G21  G40. 1
G40  G25   H     D
G49  G160  B
G80  G13. 1 T
G98  G50. 1 S
G50  G54. 2REPEAT99/99
G67  G80. 5
```

Display layout 2

```
MODAL
G00  G97   F
G17  G54   M
G90  G64   M
G22  G69   M
G94  G15
G21  G40. 1
G40  G25   H     D
G49  G160
G80  G13. 1 T
G98  G50. 1 S
G50  G54. 2REPEAT99/99
G67  G80. 5
```

Fig.12.2.6.2 (f) B code

M code (M/T)

Number of displayed M code is follows the number of M code that can be commanded within one block. Number of M codes that can be commanded within one block depends on bit 5 (MSB) of parameter No.11630 and bit 7 (M3B) of parameter No.3404.

Table12.2.6.2 (e) Number of M code

M5B(No.11630#5)	M3B(No.3404#7)	Number of M code	Display layout
1	1	5	1
1	0	5	
0	1	3	2
0	0	1	3

Display layout 1

```
MODAL
G00  G97   F
G17  G54   M
G90  G64   M
G22  G69   M
G94  G15   M
G21  G40. 1 M
G40  G25   H     D
G49  G160  B
G80  G13. 1 T
G98  G50. 1 S
G50  G54. 2REPEAT99/99
G67  G80. 5
```

Display layout 2

```
MODAL
G00  G97   F
G17  G54   M
G90  G64   M
G22  G69   M
G94  G15
G21  G40. 1
G40  G25   H     D
G49  G160  B
G80  G13. 1 T
G98  G50. 1 S
G50  G54. 2REPEAT99/99
G67  G80. 5
```

Display layout 3

```
MODAL
G00  G97   F
G17  G54   M
G90  G64
G22  G69
G94  G15
G21  G40. 1
G40  G25   H     D
G49  G160  B
G80  G13. 1 T
G98  G50. 1 S
G50  G54. 2REPEAT99/99
G67  G80. 5
```

Fig. 12.2.6.2 (g) M code

FIBER MACHINE SYSTEMS INC Rob Zuhensky

SRPM/SSPM/SMAX(T)

If constant surface speed control function is enabled(bit 0 (SSC) of parameter No.8133 is 1), modal information SRPM/SSPM/SMAX can be displayed by bit 7 (CSS) of parameter No.24314.

Table 12.2.6.2 (f) SRPM/SSPM/SMAX display

Constant surface speed control	CSS(No.24314#7)	SRPM/SSPM/SMAX	Display layout
Enabled	1	Displayed	1
Enabled	0	Not Displayed	2
Disabled	1	Not Displayed	
Disabled	0	Not Displayed	

Display layout 1

```
                MODAL
G00   G18     F
G97   G69. 1  M
G69   G40. 1  M
G99   G50. 2  M
G21   G13. 1  SRPM        0
G40   G50. 1  SSPM        0
G25   G49     SMAX        0
G22   G15     T
G80   G05. 5
G67           S
G54   G80. 5  SACT        0
G64   G80. 4REPEAT99/99
```

Display layout 2

```
                MODAL
G00   G18     F
G97   G69. 1  M
G69   G40. 1  M
G99   G50. 2  M
G21   G13. 1
G40   G50. 1
G25   G49
G22   G15     T
G80   G05. 5
G67           S
G54   G80. 5  SACT        0
G64   G80. 4REPEAT99/99
```

Fig. 12.2.6.2 (h) SRPM/SSPM/SMAX

FRYER MACHINE SALES INC
Rob Zahensky

Spindle command with a decimal point

If spindle command with a decimal point is effective, actual spindle speed (SACT) of modal, and S code are displayed with a decimal point.
Spindle command with a decimal point is enable by following conditions.

- Number having decimal points is enabled.
- Bit 3 (SDP) of parameter No.3798) is set to 1.

S code on current block screen which independent on spindle command with a decimal point is displayed exactly as specified by the program command.

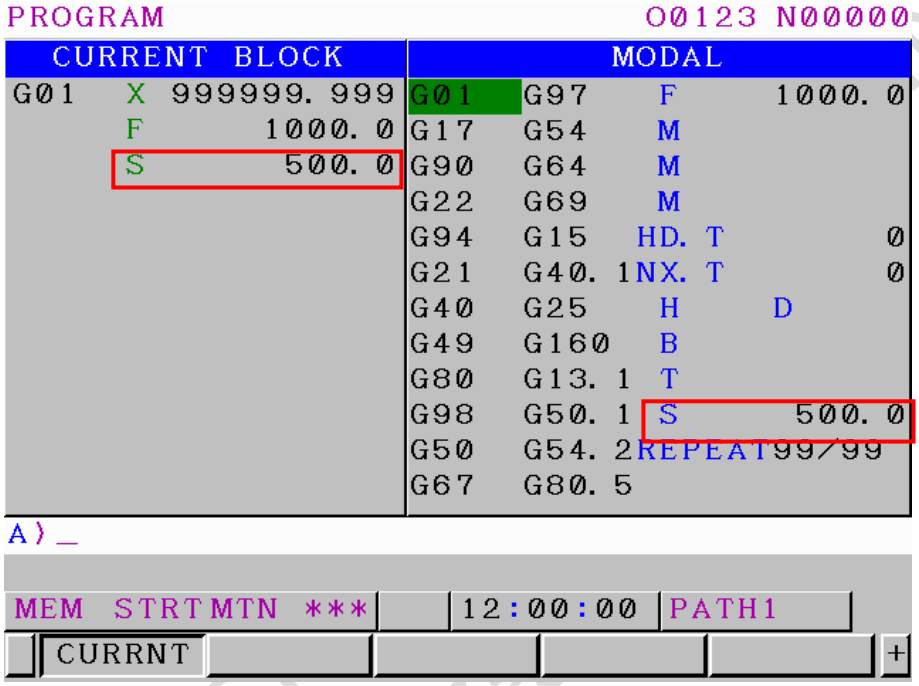


Fig. 12.2.6.2 (i) Spindle command with a decimal point (M)

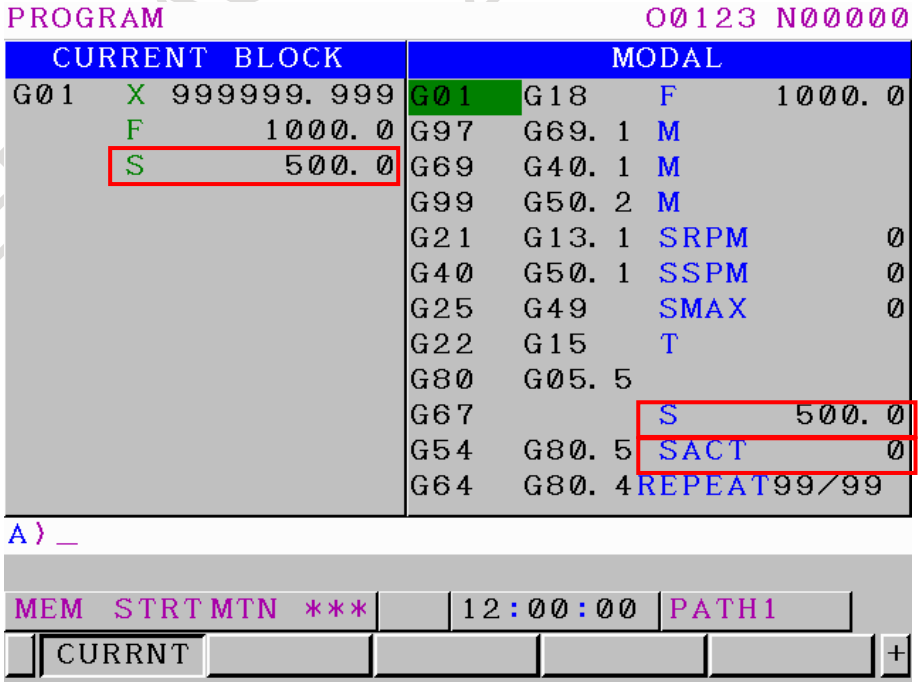


Fig. 12.2.6.2 (j) Spindle command with a decimal point（T）

Next block screen

If setting of current block bit 6 (CBD) of parameter No.24308 is 1, G code on next block screen displayed in black letters.
There is no space between G code and address.

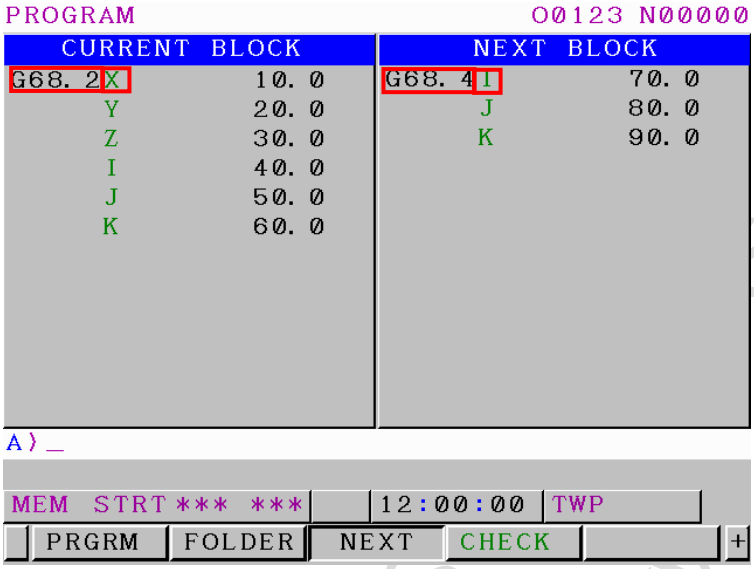


Fig. 12.2.6.2 (k) Next block screen (Bit 6 (CBD) of parameter No.24308 is set to 1)

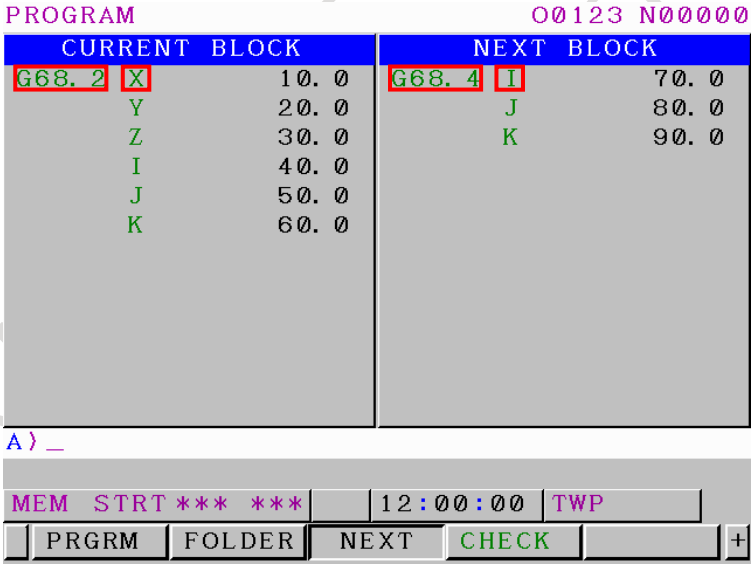


Fig. 12.2.6.2 (l) Next block screen (Bit 6 (CBD) of parameter No.24308 is set to 0)

Restrictions

When the display unit is 10.4/15/19-inch, current block screen does not exist.

12.2.7 Background Editing

Editing one program during execution of another program is referred to as background editing. You can perform the same edit operations in the background as those in normal editing (foreground editing).

On a 10.4-inch or 15-inch display, you can perform background editing for the programs displayed in multiple tiled windows. You can switch from a program displayed in one of the tiled windows to another to perform copy, paste, and other edit operations, which allows you to edit programs efficiently.
You can edit one program simultaneously on a three programs on a 10.4-inch display.

Function

- Background editing

A program other than the currently selected program can be edited. Background editing can be performed in any mode.

- EDIT mode and reference mode

When background editing is started in the EDIT mode, the program can be edited. When background editing is started in the reference mode, the program is opened in the read-only mode and cannot be edited.
While a program opened in the EDIT mode cannot be run, a program opened in the reference mode can be run.

- Program selected in the foreground

If the program selected in the foreground is specified as a program to be edited in the background, background editing is started in the read-only mode. The text at an arbitrary position of the program currently being executed can be checked.

- Switching from the program directory screen to the editing screen

By selecting a program from the program folder screen, background editing can be started.
On the program folder screen, move the cursor to a program to be edited in the background, then press the MDI key INPUT. The screen display switches to the background editing screen.
Background editing can be started without entering a program name.

- Background editing on a multi-path system

Background editing does not depend on the path.
Even if the path to be operated is changed using the path selection signal, the background editing currently being executed is just continued.

NOTE

The following programs cannot be edited in the background. They can be opened only in the read-only mode.
- Running program
- Main program
- Program with the edit disable attribute

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Display

When background editing starts, the ordinary program editing screen switches to the background editing screen.
When two or more programs are edited in the background, the screen is split to display these programs. For a 10.4-inch display, up to three programs can be edited at the same time.

- Word editing

Fig. 12.2.7 (a) shows background word editing performed simultaneously for two programs (right and left programs). On the status line at the top of the window for each program, the program name and "BG-EDIT" (indicating background editing in progress) are displayed.
The status line of the program being edited is displayed in reverse video.

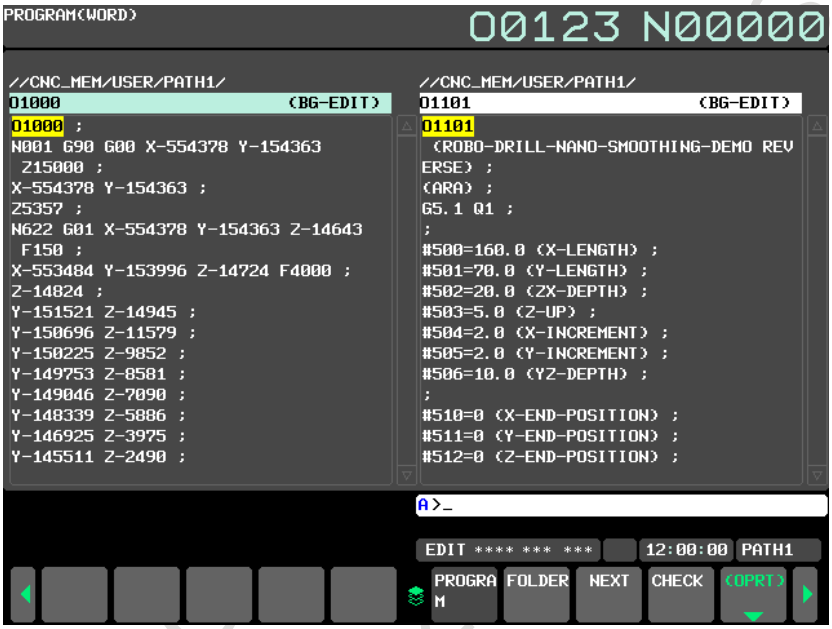


Fig. 12.2.7 (a) Background editing screen (word editing) (10.4-inch display unit)

FRYER MACHINE TOOLS INC
Rob

- Character editing

Fig. 12.2.7 (b) shows background character editing performed simultaneously for two programs (right and left programs). Similarly to word editing, at the top of the window for each program, the status line is displayed. In addition, the current input mode (INSERT MODE or OVERWRITE MODE) is displayed at the upper right of the screen for character editing.
The status line of the program being edited is displayed in reverse video.

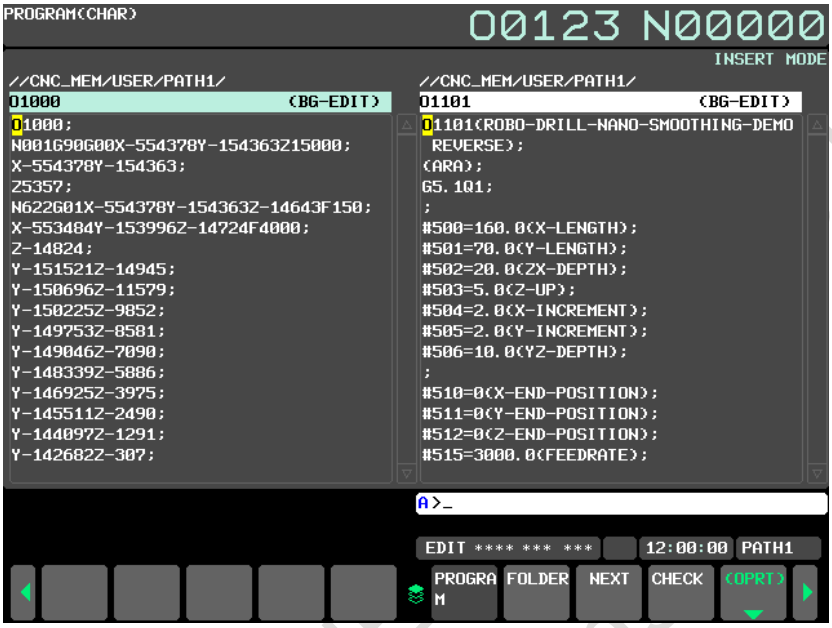


Fig. 12.2.7 (b) Background editing screen (character editing) (10.4-inch display unit)

- Editing status

The following items are displayed on the status line and program editing area according to the background editing status.

Editing status	Displayed items
No program selected	(BG-EDIT) “NO PROGRAM” is displayed in the editing area.
Program opened	program-name + (BG-EDIT)
Read-only program opened	program-name + (BG:READ ONLY) The contents of the program are displayed in green.

Starting background editing from the editing screen

Procedure (word editing)

Method 1

1 Press function key [PROG].
2 Press soft key [PROGRAM].
3 Enter a program name.
4 Press soft key [BG EDIT].

NOTE
1 Background editing is performed in the EDIT mode.
2 If a nonexistent program is specified, a new program is created.

FRYER MACHINE INC Rob Zaher

Method 2

1 Press function key PROG.
2 Press soft key [PROGRAM].
3 Press soft key [(OPRT)], then soft key [BG EDIT].
4 Enter a program name.
5 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode.

NOTE
If a nonexistent program is specified,
- Pressing soft key [EDIT EXEC] causes a new program to be created.
- Pressing soft key [REF EXEC] causes the warning message “SPECIFIED PROGRAM NOT FOUND” to appear. A new program is not created.

Method 3

1 Press function key PROG.
2 Press soft key [PROGRAM].
3 Press soft key [(OPRT)], then soft key [BG EDIT].
4 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode. At this time, “NO PROGRAM” is displayed on the screen.
5 Enter a program name.
6 Press soft key [PROGRM SEARCH].

NOTE
If a nonexistent program is specified, the warning message “SPECIFIED PROGRAM NOT FOUND” is displayed. A new program is not created.

Procedure (character editing)

Method 1

1 Press function key PROG.
2 Press soft key [PROGRAM].
3 Press soft key [(OPRT)], then soft key [BG EDIT].
4 Enter a program name.
5 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode.

NOTE
If a nonexistent program is specified,
- Pressing soft key [EDIT EXEC] causes a new program to be created.
- Pressing soft key [REF EXEC] causes the warning message “SPECIFIED PROGRAM NOT FOUND” to appear. A new program is not created.

FRIEP MACHINE SYSTEMS INC
Rob Zahensky

Method 2

1 Press function key [PROG].
2 Press soft key [PROGRAM].
3 Press soft key [(OPRT)], then soft key [BG EDIT].
4 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode. At this time, "NO PROGRAM" is displayed on the screen.
5 Press soft key [PROGRM SEARCH].
6 Enter a program name.
7 Press soft key [PROGRM NAME].

NOTE
If a nonexistent program is specified, the warning message "SPECIFIED PROGRAM NOT FOUND" is displayed. A new program is not created.

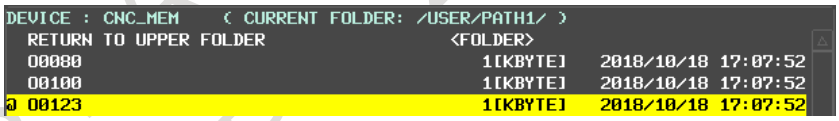
When background editing has already started and editing start operation is performed again, the program being edited and newly specified program are edited simultaneously.

Starting background editing from the program folder screen

By selecting a program from the program folder screen, background editing can be started. The cursor is used to select a program. You do not need to enter a program name.

Procedure

1 Press function key [PROG].
2 Press soft key [FOLDER].
3 Move the cursor to a program to be edited.
4 Press the MDI key [INPUT]. The screen display is switched to the background editing screen.



NOTE
When background editing is started from the program folder screen, the EDIT mode is set.
For the following programs, the reference mode is set, however:
- Running program
- Main program
- Program with the edit disable attribute

FRYER MACHINE SYSTEMS INC
R. Zahensky

Background editing operation

- Editing operation

The same editing operations as performed in the foreground can be performed.

- Switching from a program to another for editing

To switch from a program to another for editing when editing multiple programs simultaneously in the background, press [SHIFT] key, then [←] or [→].

- Editing operation in the read-only mode

Cursor movement and page switching are possible. However, program editing is disabled.

- Copying and pasting text between programs

Text in a program selected and copied during program editing is kept held after the program to be edited is changed. You can paste text in a program onto another program.

Background editing end operation

Background editing can be ended using the procedure described below.
The procedure for ending background editing of one program and that for ending all background editing of multiple programs are shown below.

- Ending editing of one program

1 Press function key [PROG].
2 Press soft key [PROGRAM].
3 Select a program for which to end editing.
4 Press soft key [(OPRT)], then soft key [BG END].

- Ending editing of all programs

1 Press function key [PROG].
2 Press soft key [PROGRAM].
3 Press soft key [(OPRT)], then soft key [BG ALL END].

To return to ordinary foreground editing, end all background editing. If at least one program is being edited in the background, you cannot return to foreground editing.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.2.8 Stamping the Machining Time

The execution times of the most recently executed ten programs can be displayed in hours, minutes, and seconds.
The calculated machining time can be inserted as a comment of the program to check the machining time on the program folder screen.

Procedure for stamping the machining time

Procedure

- Displaying the machining time

1 Press function key [PROG].

2 Press the continuous menu key [▷] several times to display soft key [TIME].

3 Press soft key [TIME]. The machining time display screen appears.

Fig. 12.2.8 (a) Machining time display screen (10.4-inch display unit)

FRYER SYSTEMS INC

- Calculating the machining time

1 Select the memory operation mode, then press the RESET key.

2 Select the program screen, then select a program of which machining time you want to calculate.

3 Execute the program to perform actual machining.

4 When the RESET key is pressed or, M02 or M30 is executed, the machining time count operation stops. When the machining time display screen is selected, the program number of the stopped main program and its machining time are displayed.

5 The Fig. 12.2.8 (b) shows the screen when the machining times of the ten main programs O0020, O0040, …, and O0200 are displayed and the screen when the machining time of O0220 is newly calculated after that.

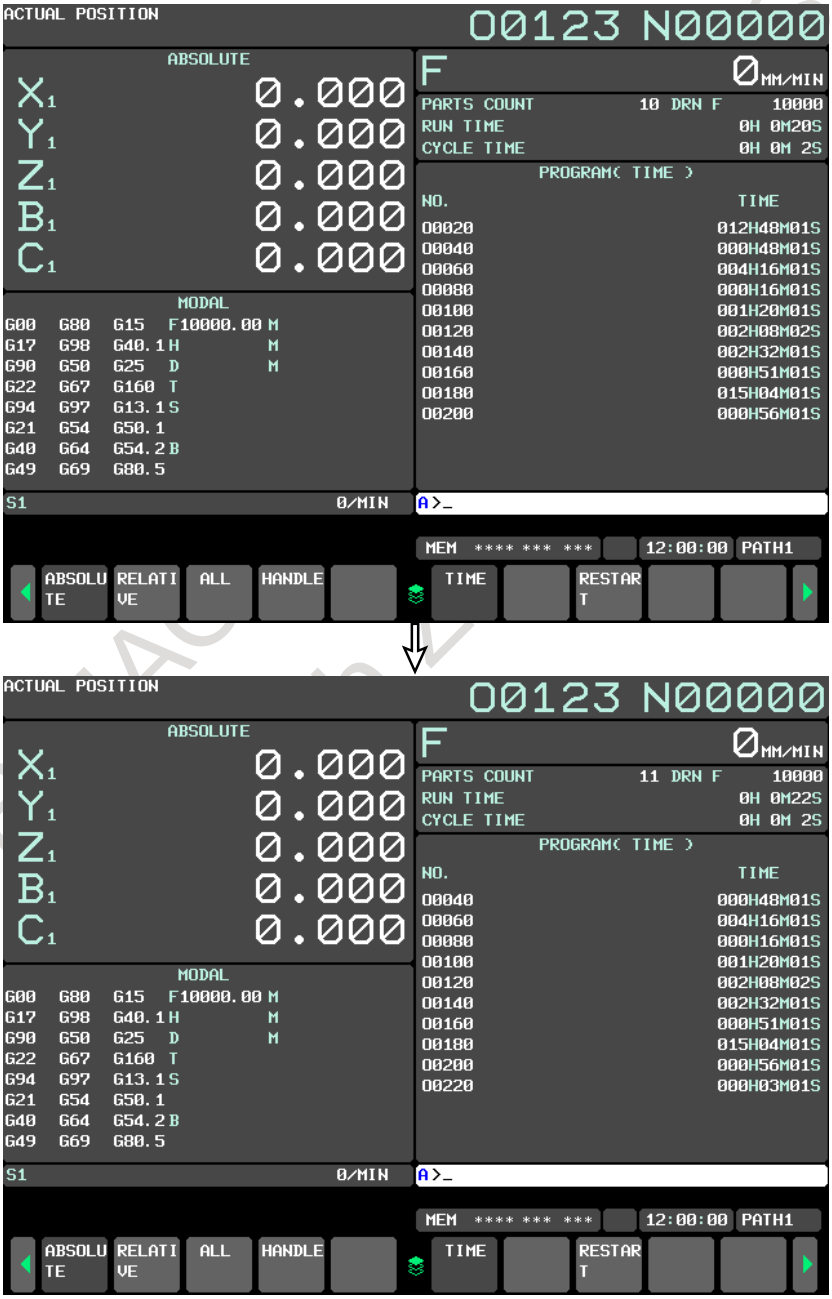


Fig. 12.2.8 (b) Stamping the machining time (10.4-inch display unit)

FRYER MACHINE TOOLS INC

Procedure for inserting the machining time on the program screen

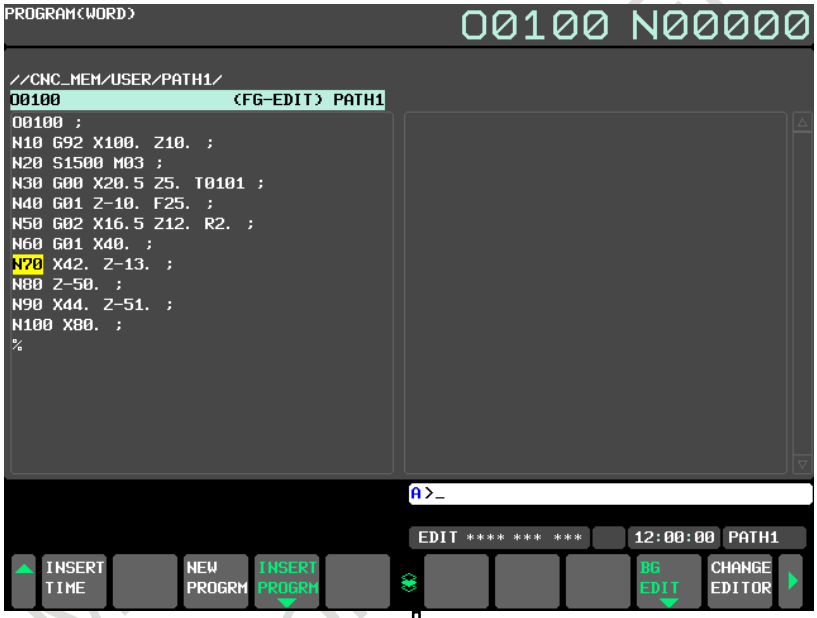
Procedure

You can display the machining time of a program as a comment of the program. The procedure is shown below:

1 To insert the calculated machining time of a program as a comment, the machining time of the program must be displayed on the machining time display screen. Before stamping the machining time of the program, check that the program number is displayed on the machining time display screen.

2 Set the edit mode or background editing state and select the program screen. Then, select a program of which machining time you want to insert.

3 Suppose that the machining time of O0100 is displayed on the machining time display screen.
Press soft key [(OPRT)] to display the operation soft keys. Then, press the continuous menu key ▷ several times to display soft key [INSERT TIME]. Press soft key [INSERT TIME]. The beginning of the program is displayed and the machining time of the program is inserted after the program name.



Press soft key [INSERT TIME].

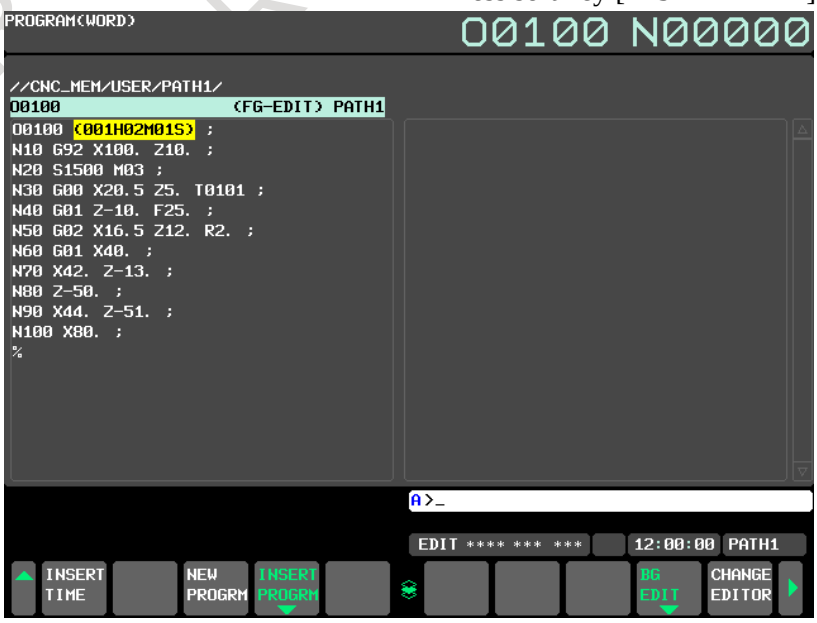


Fig. 12.2.8 (c) Program screen (10.4-inch display unit)

4 If a comment is written in the block containing the program number of a program of which machining time is to be inserted, the machining time is inserted after the comment.

Fig. 12.2.8 (d) Program screen (10.4-inch display unit)

FRYER SYSTEMS INC

Display on the program folder screen

The machining time of a program inserted in the program as a comment is displayed after the existing comment of the program on the program folder screen.

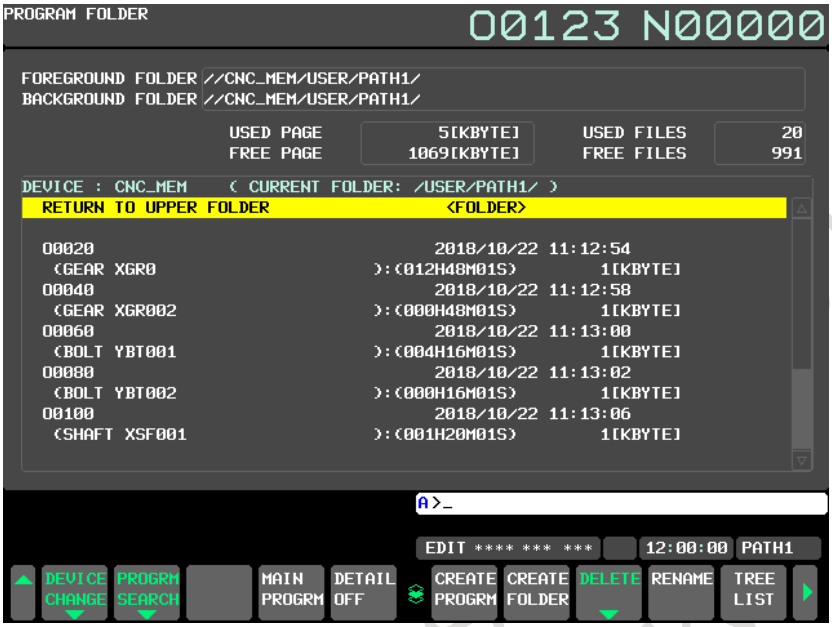


Fig. 12.2.8 (e) Program folder screen (10.4-inch display unit)

Explanation

- Machining time

The machining time is counted from the initial start after a reset in the memory operation mode to the next reset. If a reset is not performed during operation, the machining time is counted from the start to M02 or M30. The time during which operation is held is not counted, but the time used to wait for the completion of the M, S, T, and/or B functions is counted, however.

- Stamping the machining time

You can insert (stamp) the displayed machining time in a program stored in memory as a comment. The machining time is inserted as a comment after the program number.

- Program folder

You can display the machining time inserted after a program number on the program folder screen. This operation lets you know the machining time of each program to use the time as useful reference data when planning processes on site.

Limitation

- Alarm

The execution of a program may be held by an alarm during the machining time count operation. In this case, the machining time is counted until the alarm is released by a reset.

- M02

It may be specified that M02 does not reset the CNC, but returns completion signal FIN to the CNC to restart the program from the beginning successively (bit 5 (M02) of parameter No. 3404 is set to 0). In this case, when M02 returns completion signal FIN, the machining time count operation stops.

- Stamping the machining time

When an attempt is made to stamp the machining time of a program in the program, the machining time of the program may not be displayed on the machining time display screen. In this case, the machining time cannot be inserted in the program even if soft key [INSERT TIME] is pressed.

- Correcting the machining time

If an incorrect machining time is calculated (such as when a reset is made during the execution of a program), reexecute the program to calculate the correct machining time. The same program number may be displayed in two or more lines on the machining time display screen. In this case, the latest calculated machining time is inserted in the program.

- States of the stamped machining time

In the following states, the stamped machining time is displayed on the program folder screen as shown below.

1 When the comment of a program is longer than 16 characters
The 17th and subsequent characters of the comment are discarded and the machining time display field is left blank.

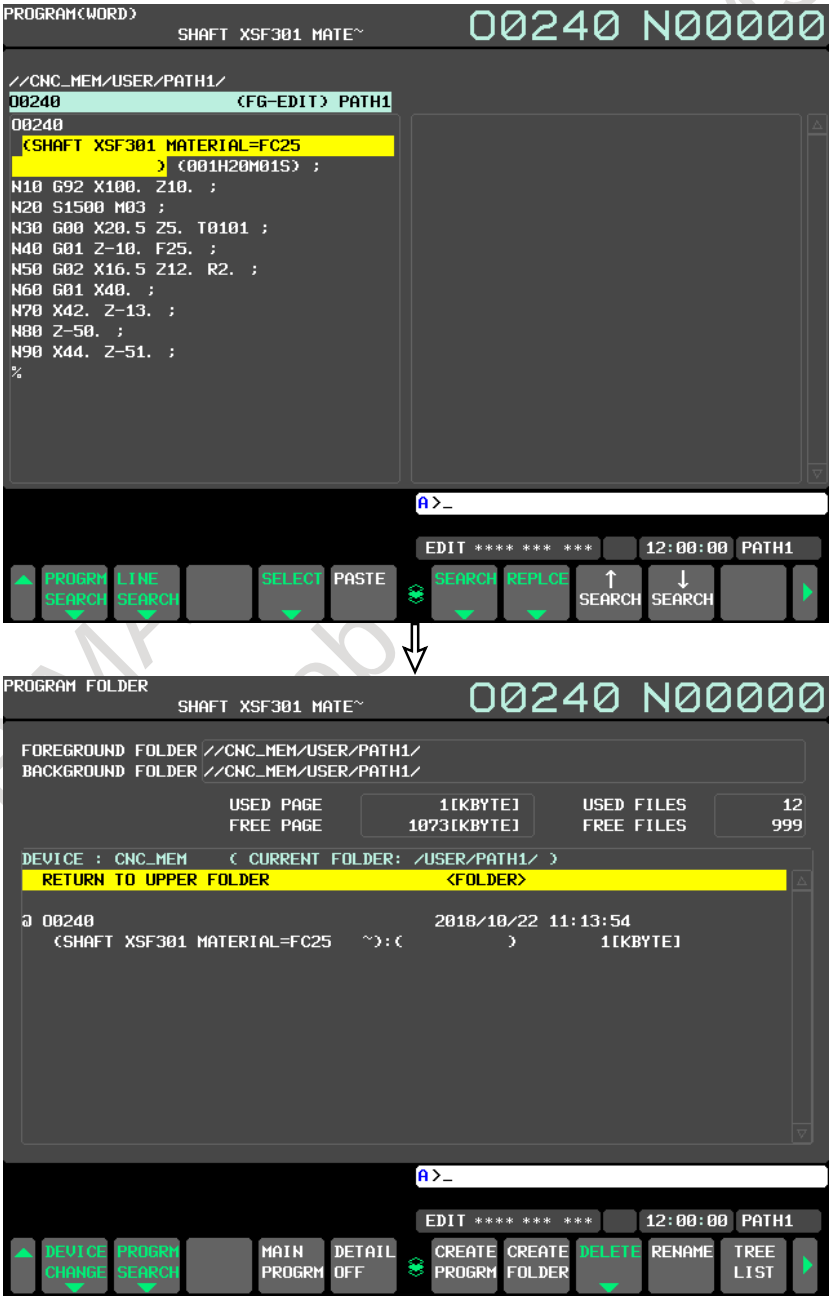


Fig. 12.2.8 (f) When the comment of a program is longer than 16 characters (10.4-inch display unit)

2 When two or more machining times are stamped
The first machining time is displayed.

Fig. 12.2.8 (g) When two or more machining times are stamped (10.4-inch display unit)

FRYER'S INC

3 When the format of an inserted machining time is not “hhhHmmMssS” (H following a 3-digit number, M following a 2-digit number, and S following a 2-digit number, in this order)
The machining time display field is left blank.

Fig. 12.2.8 (h) When the format of an inserted machining time is not “hhhHmmMssS” (H following a 3-digit number, M following a 2-digit number, and S following a 2-digit number, in this order) (10.4-inch display unit)

12.2.9 Screen for Assistance in Entering Tilted Working Plane Indexing

The screens for assistance in entering tilted working plane indexing (hereinafter referred to as guidance screens) include a command type selection screen and a tilted working plane data setting screen. The command type selection screen is used to select a tilted working plane indexing. The tilted working plane data setting screen is used to set specified tilted working plane data required for the selected command. By making settings and performing operations on these guidance screens, a tilted working plane indexing block can be created.
The created block is reflected as a new insertion to a program being edited or as a modification to an existing block.
This function can be enabled by setting bit 1 (GGD) of parameter No. 11304 to 1.

Creation of a new block

The following describes the procedure for creating a tilted working plane indexing block on guidance screens and for inserting the block to a program being edited on a program editing screen.

1 On a program editing screen, display a program to which you want to insert a tilted working plane indexing block.
The foreground editing screen, background editing screen, or MDI editing screen should be displayed.

- Displaying the foreground editing screen
  <1> Select the EDIT mode.
  <2> Press function key PROG.
  <3> Press soft key [PROGRAM].

- Displaying the background editing screen
  <1> Press function key PROG.
  <2> Press soft key [FOLDER].
  <3> Press any of the cursor keys to move the cursor to a program to be edited in the background.
  <4> Press INPUT key.

- Displaying the MDI editing screen
  <1> Select the MDI mode.
  <2> Press function key PROG.
  <3> Press soft key [PROGRAM].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

The program editing screen is displayed.

Fig.12.2.9 (a)

2 Press any of the cursor keys to move the cursor to a position where you want to insert a block.
Note that a block created on the guidance screens is inserted after the block at the cursor position. (If the block at the cursor position includes a tilted working plane indexing, the existing block is modified. See "Modification to an existing block" below.)

3 Press soft key [(OPRT)].

4 Press continuous menu key ▷ several times, and then press soft key [GUIDANCE].
The command type selection screen is displayed.

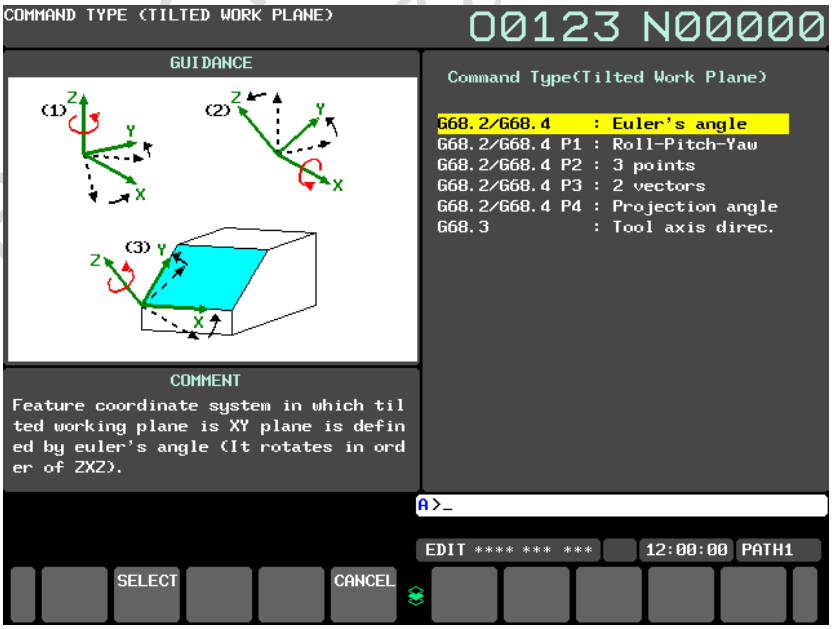


Fig.12.2.9 (b)

FRYER MACHINE SYSTEMS INC Zahensky

5 Select a command type with any of the cursor keys, and then press soft key [SELECT].
The tilted working plane data setting screen is displayed.

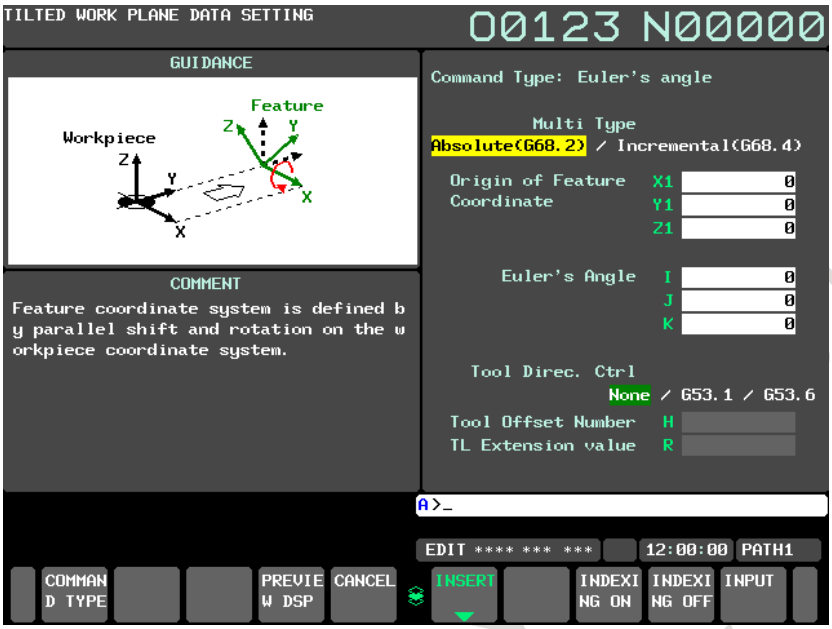


Fig.12.2.9 (c)

6 Enter command data for the setting items.
7 Press soft key [INSERT].

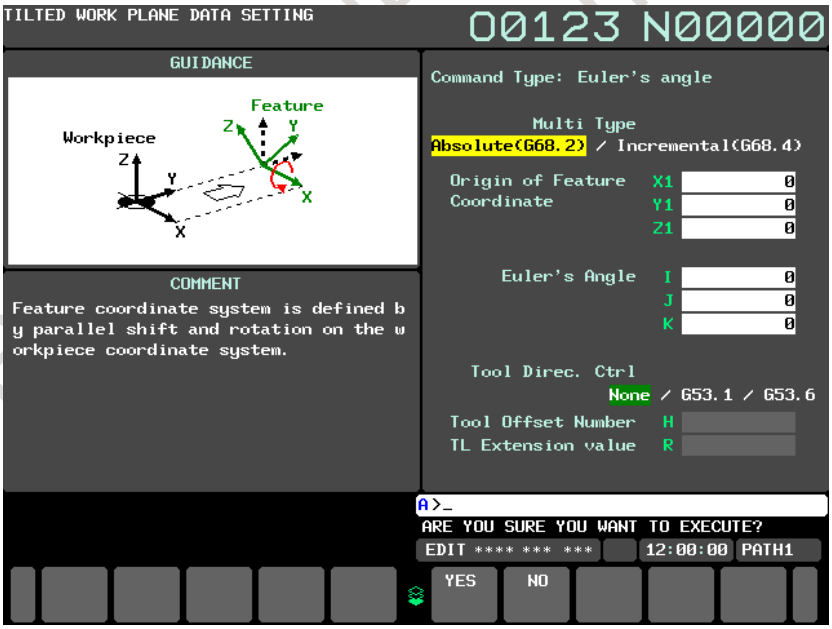


Fig.12.2.9 (d)

8 Press soft key [YES].
This takes you back to the program editing screen, where the new block is inserted after the block at the cursor position.

Modification to an existing block

The following describes the procedure for replacing a block in a program being edited on a program editing screen, with a tilted working plane indexing block created on a guidance screen.

1 On a program editing screen, display a program to be edited.
(For the procedure for displaying a program editing screen, see step 1 in "Creation of a new block".)
The program editing screen is displayed.

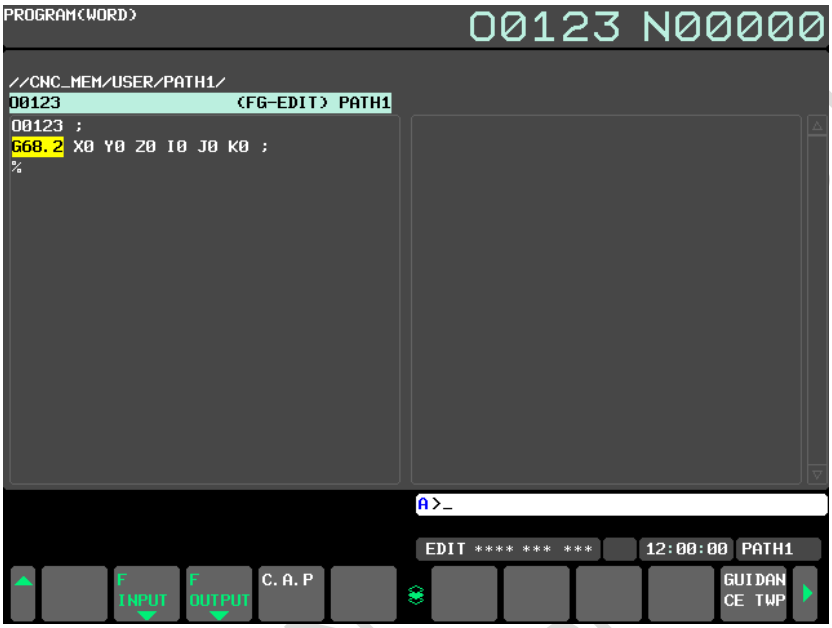


Fig.12.2.9 (e)

2 Press any of the cursor keys to move the cursor to a block to be modified.
For a command extending over more than one block, move the cursor to the first block.

3 Press soft key [(OPRT)].

4 Press continuous menu key ▷ several times, and then press soft key [GUIDANCE TWP]. The tilted working plane data setting screen is displayed.

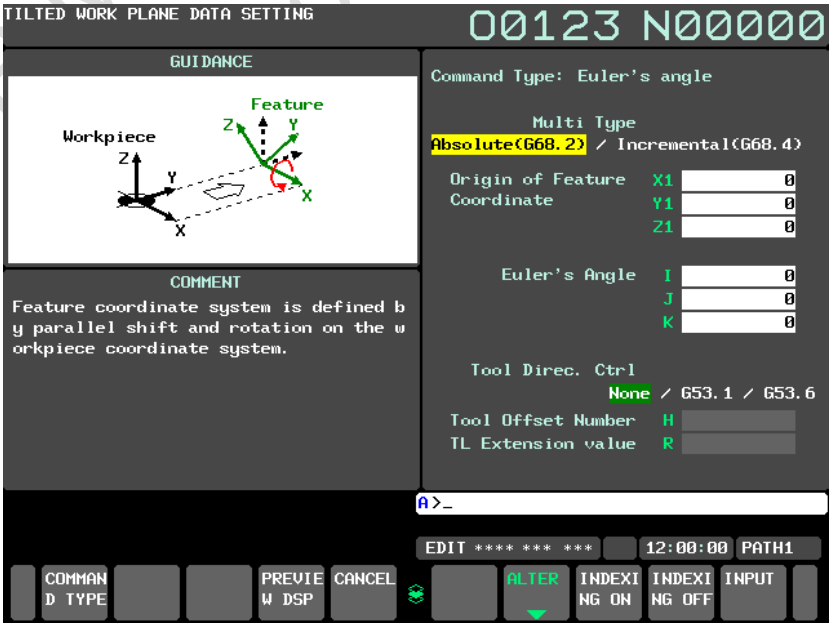


Fig.12.2.9 (f)

FRYER MACHINE INC Rob Zaher

5 Enter command data for setting items to be modified.
6 Press soft key [ALTER].

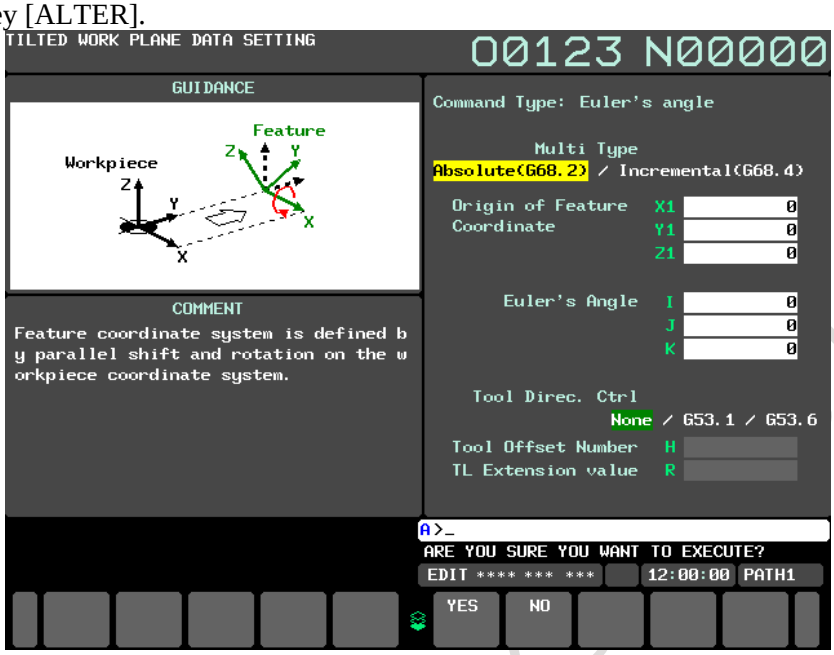


Fig.12.2.9 (g)

7 Press soft key [YES].

This takes you back to the program editing screen, where the block at the cursor position is replaced.

Guidance screen cancellation

Pressing soft key [CANCEL] on a guidance screen takes you back to the program editing screen. At this time, the data that has been set on the guidance screen is discarded.

NOTE

1 In addition to the above operation, the following operations also cancel a guidance screen. The data that has been set on the guidance screen is discarded.
- When bit 7 (CPG) of parameter No. 11302 is 1 (setting for automatically switching between program-related screens according to CNC mode), the CNC mode is changed.
- When a guidance screen is displayed from the foreground editing screen, the CNC mode is changed to a mode other than EDIT, TJOG, or THND.
- When a guidance screen is displayed from the MDI editing screen, the CNC mode is changed to a mode other than MDI.
- When a guidance screen is displayed on a 15/19-inch display unit, the screen is switched by a vertical soft key.
- The screen is switched by an MDI key.
- The screen is switched by a path select signal.
- An event that causes screen switching has occurred, including occurrence of an alarm, display of an operator message, or signal-based display of a screen (such as a tool compensation screen, workpiece shift screen, workpiece coordinate system setting screen, or C Language Executor screen).

2 If MDI key [PROG] is pressed after switching from a guidance screen to another screen, a program editing screen is displayed, instead of the guidance screen.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Notes

- Conditions under which soft key [GUIDANCE TWP] is displayed
  Soft key [GUIDANCE TWP] is displayed on a program editing screen under the following conditions:
  1 Foreground editing screen
    - The CNC mode is EDIT, TJOG, or THND.
    - The editing and display are not prohibited for a program to be edited.
  2 Background editing screen
    - A program to be edited is not in reference mode.
    - The editing and display are not prohibited for a program to be edited.
  3 MDI editing screen
    - The CNC mode is MDI.

- Screen displayed when soft key [GUIDANCE TWP] is pressed
  When soft key [GUIDANCE TWP] is pressed, which guidance screen is displayed is automatically determined according to the conditions described below.
  1 When a tilted working plane indexing is not included in a block at the cursor position on a program editing screen
    The command type selection screen is displayed. A block created on the guidance screens is inserted after the block at the cursor position on the program editing screen.
  2 When a tilted working plane indexing is included in a block at the cursor position on a program editing screen
    The tilted working plane data setting screen is displayed, which shows the data for the tilted working plane indexing in the block at the cursor position on the program editing screen.
    When a block of a tool axis direction control command exists immediately after the tilted working plane indexing, the command data for the block is also shown. The tilted working plane indexing in the block at the cursor position on the program editing screen is replaced with a block created on the guidance screen.

NOTE
If the CNC is in the reset state or emergency stop state when soft key [GUIDANCE TWP] is pressed on the foreground editing screen or MDI editing screen, the warning "PROGRAM READ FAILED" appears, and the operation cannot be continued. (Soft keys other than [CANCEL] are not displayed.) Press soft key [CANCEL] to return to the program editing screen, and then press soft key [GUIDANCE TWP] again.

FRYER MACHINE SYSTEMS INC
Rob Zaranski

12.2.9.1 Command type selection screen

The command type selection screen is used to select the type of a tilted working plane indexing you want to insert into a program to be edited. One of the following command types can be selected:

- G68.2 / G68.4 (Euler's angle)
- G68.2 / G68.4 P1 (Roll-Pitch-Yaw angle)
- G68.2 / G68.4 P2 (3 points specification)
- G68.2 / G68.4 P3 (2 vectors specification)
- G68.2 / G68.4 P4 (Projection angle)
- G68.3 (Tool Axis Direction)

G68.2 is an absolute command, and G68.4 is an incremental command.

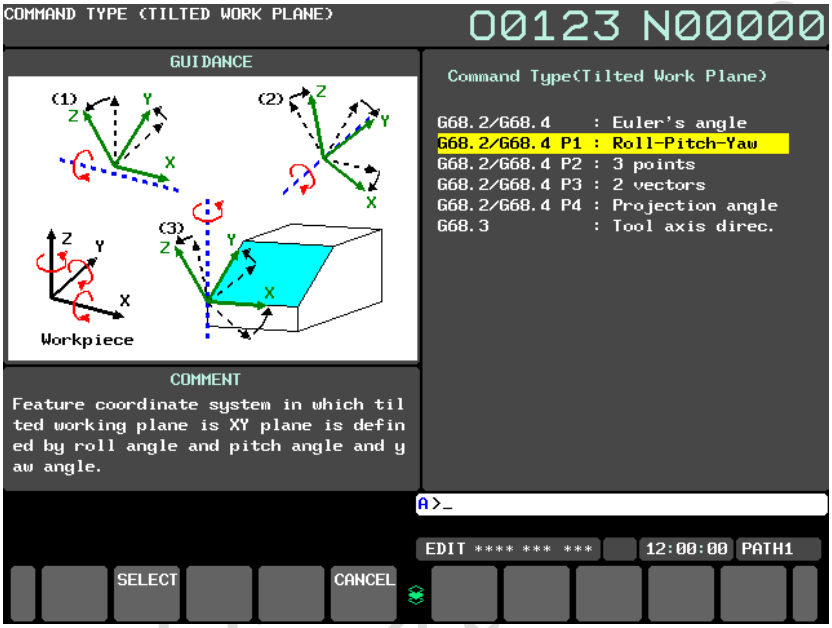


Fig. 12.2.9.1 (a) Command type selection screen (10.4-inch display unit)

Selection of a command type

1. Press cursor key ↑ or ↓ to move the cursor to a command type you want to select. As the cursor is moved, the figure corresponding to the command type at the cursor position is displayed.
2. When soft key [SELECT] is pressed, the command type at the cursor position is accepted, and the tilted working plane data setting screen is displayed.

> **NOTE**
> If the warning "PROGRAM READ FAILED" appears when the command type selection screen is displayed, the operation cannot be continued. (Soft keys other than [CANCEL] are not displayed.) Press soft key [CANCEL] to return to the program editing screen, and then press soft key [GUIDANCE TWP] again.

FRYER MACHINERY INC Rob Z

12.2.9.2 Tilted working plane data setting screen

The tilted working plane data setting screen is used to set specified tilted working plane data required for a tilted working plane indexing of the type that has been selected on the command type selection screen or has been selected when soft key [GUIDANCE TWP] is pressed.
Different types of tilted working plane data setting screen are provided for different command types. For details of each type of tilted working plane data setting screen, see "Details of the tilted working plane data setting screen".

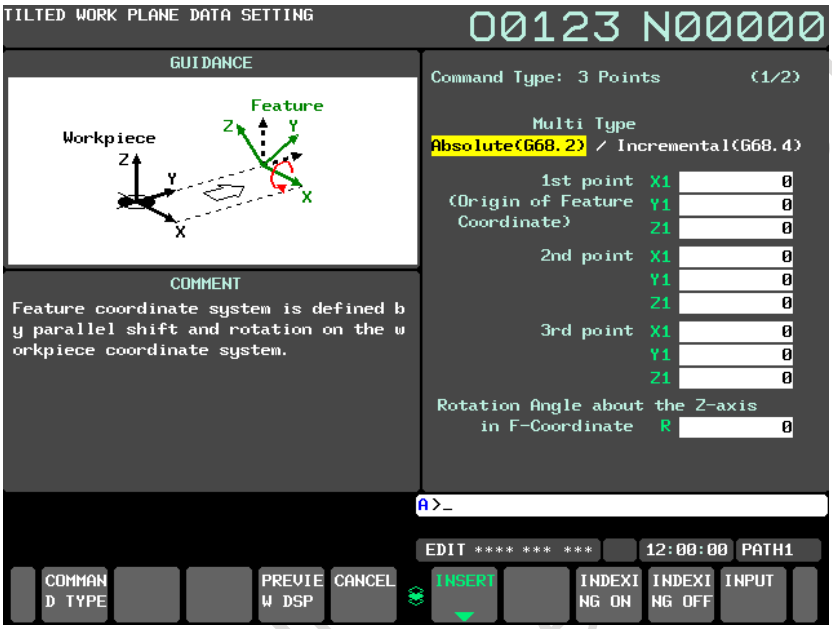


Fig. 12.2.9.2 (a) Tilted working plane data setting screen-3 points specification(10.4-inch display unit)

Display of the tilted working plane data setting screen

When a new block is created, the initial values are set in the setting and selection items.
When an existing block is modified, the command data for the block at the cursor position on the program editing screen is reflected in the setting and selection items.

NOTE

1 For existing block modification, if the guidance screen is displayed when the cursor is placed in the middle of multiple blocks for a command, the parameters for the block(s) before the cursor are not reflected in the setting and selection items, but only the parameters for the block(s) after the cursor are reflected. If no data is to be reflected in a setting or selection item, the initial value is set in the item.

2 For existing block modification, only the commands with parameter-specified axis names are reflected in the setting items on the screen. (The values specified in data for commands with incorrect axis names are not reflected in the setting items.)

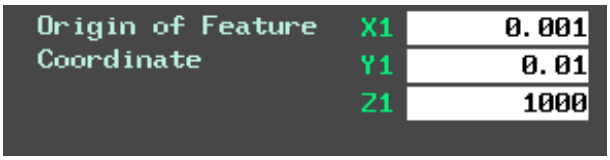
FRYER MACHINERY INC Rob Zahn

Command data input

- Item for which to enter a value

  Press cursor key ↑ or ↓ to move the cursor to an item you want to set.

  Enter a value, and then press the INPUT key or soft key [INPUT].



Example)

When the origin of a feature coordinate system is set as shown above, addresses X, Y, and Z are specified as follows:

G68.2 X0.001 Y0.01 Z1000 ...

For two items "Tool Offset Number" and "From TCP to center", the state with no command can be set by deleting the specified values according to the following procedure.

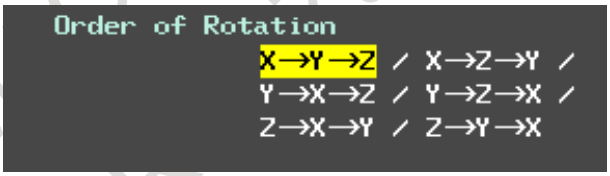
1 Press cursor key ↑ or ↓ to move the cursor to the item "Tool Offset Number" or "From TCP to Center".

2 Press INPUT or soft key [INPUT] without entering anything.

- Item to be selected from a list

  1 Press cursor key ↑ or ↓ to move the cursor to an item you want to set.

  2 Press cursor key ← or → to move the cursor to an item you want to select.



Example)

Order of rotation for Roll-Pitch Yaw angle

Insertion of a block

If the guidance screen is displayed when the block at the cursor position on the program editing screen does not include a tilted working plane indexing, soft key [INSERT] is displayed on the tilted working plane data setting screen. An operation for inserting a tilted working plane indexing block is described below.

For warnings that may be issued at the time of block insertion, see "Limitation".

1 Press soft key [INSERT].
The confirmation message "ARE YOU SURE YOU WANT TO EXECUTE?" is displayed, and soft keys [YES] and [NO] are displayed.

2 Press soft key [YES].
A block is created from the command type and command data and then is inserted after the block at the cursor position in the program to be edited. After block insertion, the guidance screen is closed and the program editing screen is displayed.
Pressing soft key [NO] takes you back to the tilted working plane data setting screen.

FRYER MACHINE SYSTEMS INC
R. Zahonsky

Example)
G00 X0.; When the guidance screen is displayed and the 3-point specification is selected as a
↓ command type for block insertion, a created block is inserted after the block at the cursor position.
G00 X0.;
G68.2 P2 Q0...
G68.2 P2 Q1...
G68.2 P2 Q2...
G68.2 P2 Q3...

Replacement of a block

If the guidance screen is displayed when the block at the cursor position on the program editing screen includes a tilted working plane indexing, soft key [ALTER] is displayed on the tilted working plane data setting screen. An operation for replacing a tilted working plane indexing block is described below.
For warnings that may be issued at the time of block replacement, see "Limitation".

1 Press soft key [ALTER].
The confirmation message "ARE YOU SURE YOU WANT TO EXECUTE?" is displayed, and soft keys [YES] and [NO] are displayed.
2 Press soft key [YES].
A block is created from the command type and command data and then replaces the tilted working plane indexing in the block at the cursor position on the program editing screen. After block replacement, the guidance screen is closed and the program editing screen is displayed.
Pressing soft key [NO] takes you back to the tilted working plane data setting screen.

NOTE
1 If a block to be replaced includes a command other than a tilted working plane indexing, the command is deleted during block replacement. However, only the sequence number at the beginning is preserved.
2 If the guidance screen is displayed when the cursor is placed in the middle of multiple blocks for a command, the block(s) before the cursor are not replaced. These blocks remain unchanged after replacement.
3 If soft key [GUIDANCE TWP] is pressed when the cursor is placed on a block of a tool axis direction control command, the command type selection screen is displayed in new insertion mode. A block created on the guidance screens is inserted after the block of the tool axis direction control command.

Limitation

The following lists the warnings that may be issued at the time of block insertion or replacement.
If a warning is displayed, return to the program editing screen with soft key [CANCEL] and press soft key [GUIDANCE TWP] again, or eliminate the cause of the warning and retry the operation.

Warning	Description
“PROGRAM WRITE FAILED”	- The guidance screen was displayed from the foreground editing screen or MDI editing screen, and a block insertion or replacement operation was performed when the CNC was in the reset state or emergency stop state.
“PROGRAM CANNOT BE WRITTEN”	- A block insertion or replacement operation was performed after a program to be edited was updated due to program downloading by an external application while the guidance screen was displayed. - A block insertion or replacement operation was performed after the start of the main program was located by a reset while the guidance screen was displayed.

FRYER MACHINE SYSTEMS INC
Bob Zaremsky

Warning	Description
“WRITE PROTECT”	- A block insertion or replacement operation was performed when the editing or display was prohibited for a program to be edited. - A block insertion or replacement operation was performed when the memory protection signal (KEY signal) for program registration or editing was off. - A block insertion or replacement operation was performed when the operation level of the 8-level data protection function was lower than the protection level of a part program editing operation.

Display of the command type selection screen

When soft key [COMMAND TYPE] is pressed, the command type selection screen is displayed. When the command type is changed on the command type selection screen, the values set on the tilted working plane data setting screen are cleared.

When a tilted working plane indexing is included in the block at the cursor position on the program editing screen, you can display the guidance screen, change the command type on the command type selection screen, and then perform block replacement. In this case, the block at the cursor position on the program editing screen is replaced with a block created from the command type and command data that have been set.

12.2.9.3 Details of the tilted working plane data setting screen

The following six tilted working plane indexing are supported.

For details of the commands, see Section, "TILTED WORKING PLANE INDEXING".

- G68.2 / G68.4 (Euler’s angle)
- G68.2 / G68.4 P1 (Roll-Pitch-Yaw angle)
- G68.2 / G68.4 P2 (3 points specification)
- G68.2 / G68.4 P3 (2 vectors specification)
- G68.2 / G68.4 P4 (Projection angle)
- G68.3 (Tool Axis Direction)

G68.2 is an absolute command, and G68.4 is an incremental command.

G68.2 / G68.4(Euler’s angle)

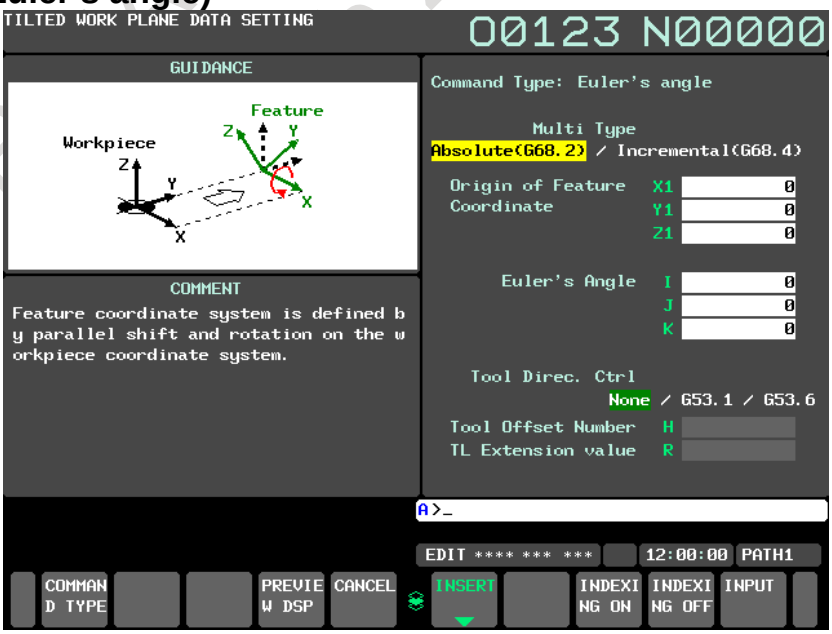


Fig. 12.2.9.3 (a) Tilted working plane data setting screen-Euler’s angle(10.4-inch display unit)

FRYER ACME SYSTEMS INC
Zohensky

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Euler's angle
  I : Specify an angle of rotation around the Z-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).
  This rotation determines coordinate system 1 (X1-Y1-Z1) from the workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type) (X-Y-Z).
  J : Specify an angle of rotation around the X-axis of coordinate system 1.
  This rotation determines coordinate system 2 (X2-Y2-Z2) from coordinate system 1 (X1-Y1-Z1).
  K : Specify an angle of rotation around the Z-axis of coordinate system 2.
  After this rotation, a feature coordinate system is obtained by shifting the origin of the workpiece coordinate system by the coordinates specified in "Origin of Feature Coordinate".

G68.2 / G68.4(Roll-Pitch-Yaw angle)

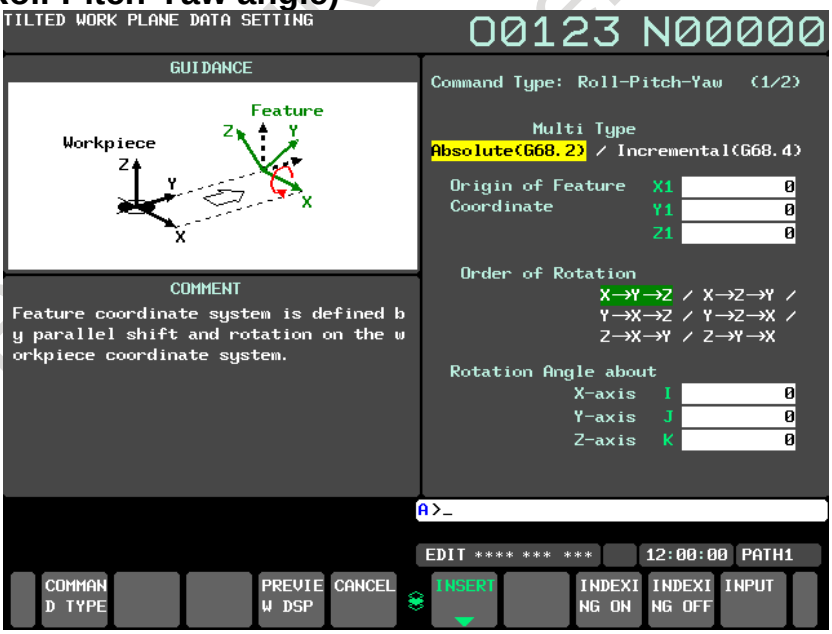


Fig. 12.2.9.3 (b) Tilted working plane data setting screen- Roll-Pitch-Yaw angle(10.4-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Order of Rotation
  Select an order in which the X-axis, Y-axis, and Z-axis are rotated in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). The selectable rotation orders are as follows:

	1st rotation axis	2nd rotation axis	3rd rotation axis
X→Y→Z	X-axis	Y-axis	Z-axis
X→Z→Y	X-axis	Z-axis	Y-axis
Y→X→Z	Y-axis	X-axis	Z-axis
Y→Z→X	Y-axis	Z-axis	X-axis
Z→X→Y	Z-axis	X-axis	Y-axis
Z→Y→X	Z-axis	Y-axis	X-axis

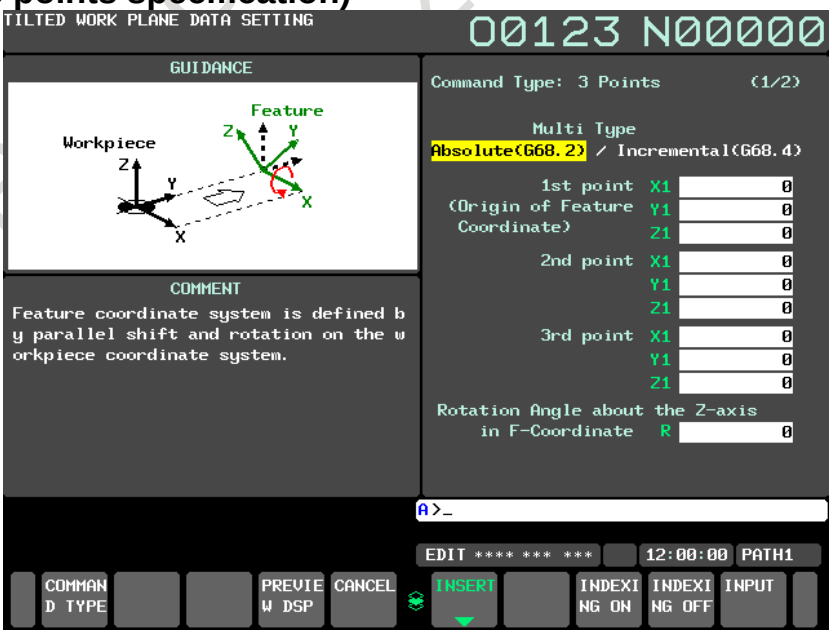
- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about X-axis
  Specify an angle of rotation around the X-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about Y-axis
  Specify an angle of rotation around the Y-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about Z-axis
  Specify an angle of rotation around the Z-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

G68.2 / G68.4(3 points specification)



FRYER MACHINE SYSTEMS INC
Zahensky

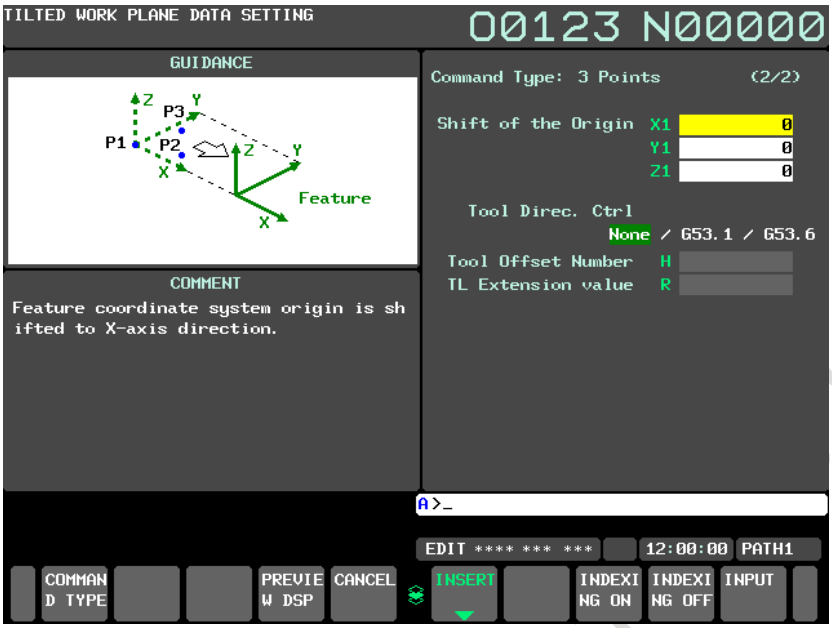


Fig. 12.2.9.3 (c) Tilted working plane data setting screen- 3 points specification(10.4-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Shift of the Origin
  Specify, in a feature coordinate system, an amount of shift from the feature coordinate system origin specified for the 1st point (point P1).

- Rotation Angle about the Z-axis in F-Coordinate
  Specify an angle of rotation around the Z-axis of a feature coordinate system. The direction of rotation angle R is positive when a rotation is made clockwise as viewed in the Z-axis direction of the feature coordinate system.

- 1st point (Origin of Feature Coordinate)
  Specify the origin (X, Y, and Z of point P1) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- 2nd point
  Specify the coordinates (X, Y, and Z of point P2) of the 2nd point as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). The 1st point and 2nd point determine the X-axis of the feature coordinate system.

- 3rd point
  Specify the coordinates (X, Y, and Z of point P3) of the 3rd point as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). Among the directions normal to the X-axis, a direction with a smaller angle relative to the P1 → P3 vector is the Y-axis of the feature coordinate system.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

G68.2 / G68.4(2 vectors specification)

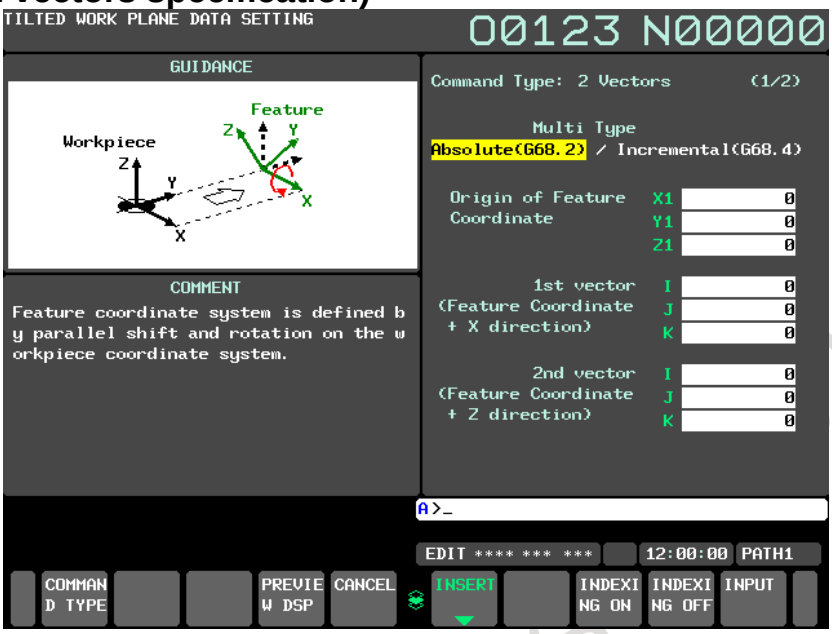


Fig. 12.2.9.3 (d) Tilted working plane data setting screen-2 vectors specification(10.4-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- V1 (X’) vector
  Specify the X-axis direction vector of a feature coordinate system as values in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- V2 (Z’) vector
  Specify the Z-axis direction vector of a feature coordinate system as values in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

FRYER MACHINE SYS INC
Rob Zahensky

G68.2 / G68.4(Projection angle)

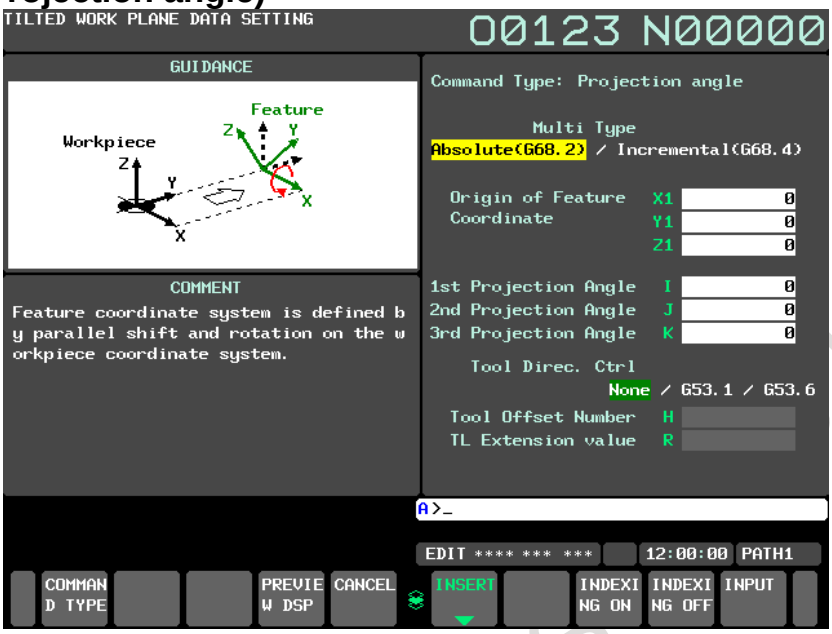


Fig. 12.2.9.3 (e) Tilted working plane data setting screen-Projection angle(10.4-inch display unit)

- Multi Type

  Absolute:

  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.

  Incremental:

  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate

  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Projection angle

  I : Specify a projection angle relative to the X-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

  J : Specify a projection angle relative to the Y-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

  K : Specify an angle of rotation around the Z-axis of a feature coordinate system.

FRYER MACHINE SYSTEMS INC
Rob Zahonsky

G68.3(Tool Axis Direction)

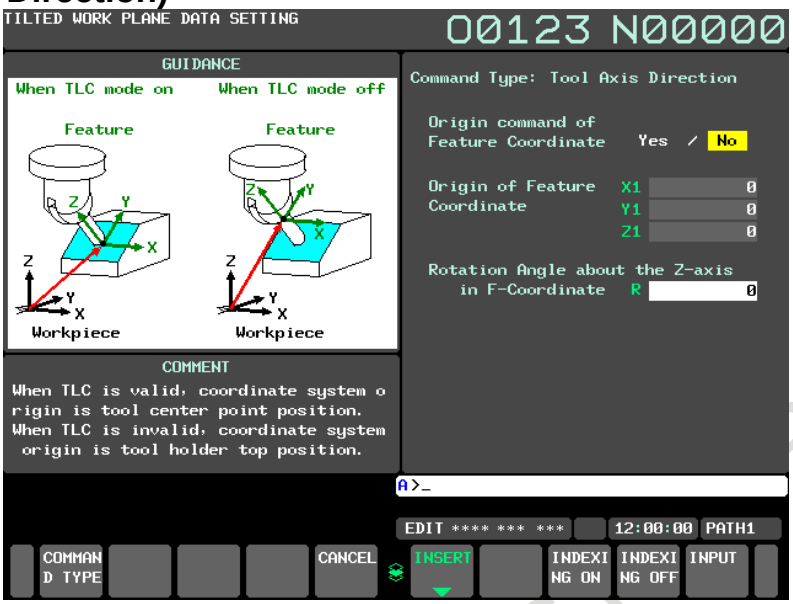


Fig. 12.2.9.3 (f) Tilted working plane data setting screen-Tool Axis Direction(10.4-inch display unit)
(When "No" is selected in "Origin command of Feature Coordinate")

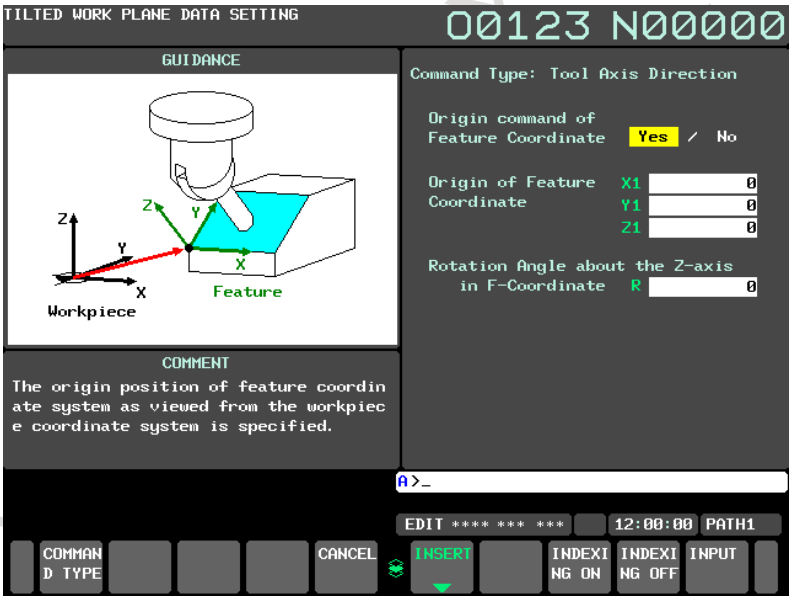


Fig. 12.2.9.3 (g) Tilted working plane data setting screen-Tool Axis Direction(10.4-inch display unit)
(When "Yes" is selected in "Origin command of Feature Coordinate")

- Origin command of Feature Coordinate
  Select whether to specify a feature coordinate system origin.
  Yes : A feature coordinate system origin is specified.
  No : A feature coordinate system origin is not specified.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system.
  This setting cannot be made if "No" is selected in "Origin command of Feature Coordinate".

- Rotation Angle about the Z-axis in F-Coordinate
  Specify an angle of rotation around the Z-axis of a feature coordinate system. The direction of rotation angle R is positive when a rotation is made clockwise as viewed in the Z-axis direction of the feature coordinate system.

12.2.9.4 Tilted Working Plane Indexing during manual operation

On the guidance screen, tilted working plane indexing in current executed path is executable and cancellable.

How to execute tilted working plane indexing

1 Input definition of tilted working plane on the guidance screen.

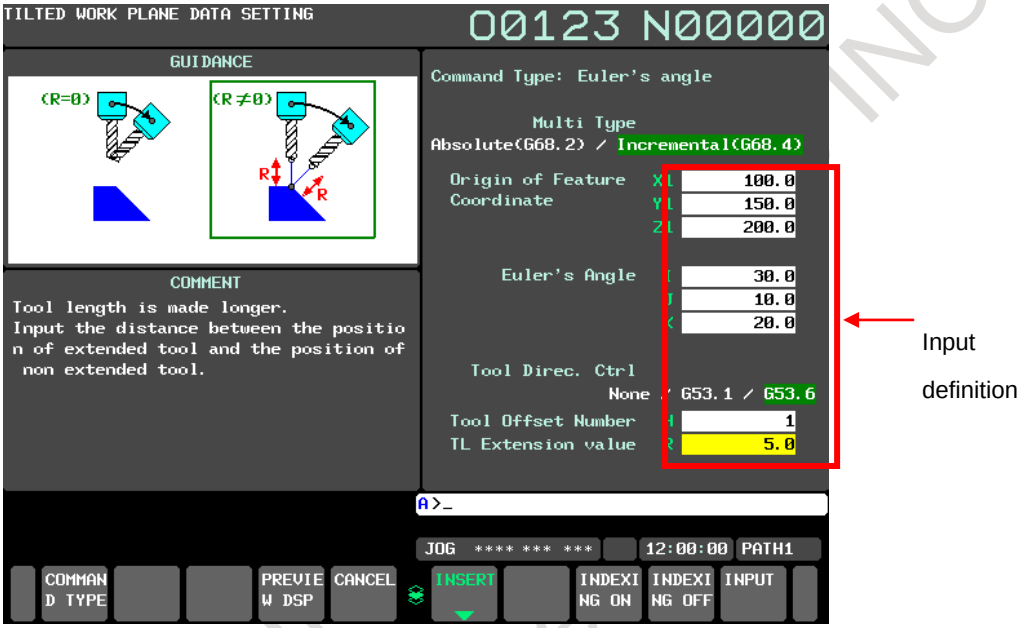


Fig. 12.2.9.4 (a) Guidance screen

2 Press soft key [INDEXING ON].

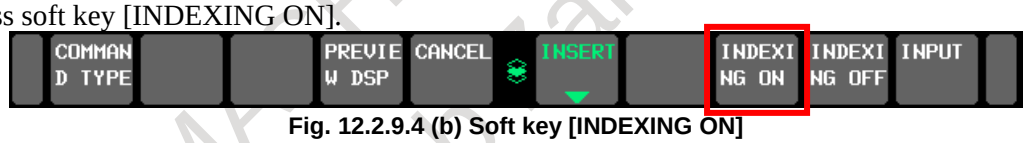


Fig. 12.2.9.4 (b) Soft key [INDEXING ON]

NOTE
If the case of incremental (G68.4), value set to guidance screen is cleared when soft key [INDEXING ON] is pressed.

How to cancel tilted working plane indexing

1 Press soft key [INDEXING OFF] on the guidance screen.

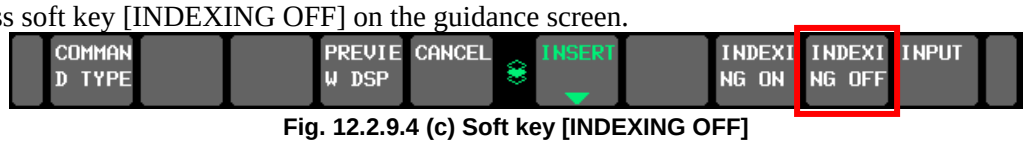


Fig. 12.2.9.4 (c) Soft key [INDEXING OFF]

NOTE
If soft key is pressed in the following conditions, warning occurs.
- Not during cancelling tool length compensation
- Neither in MDI, JOG nor HND mode

FRYER MACHINE Robotics INC

12.2.9.5 Limitation

This function is supported on 10.4-inch, 15-inch, and 19-inch display units.

Screens of a 15/19-inch display unit

This section describes the screens displayed by pressing function key PROG. The screens include a program editing screen, program folder list display screen, and screens for displaying the command states of the program currently being executed.

1. Program screen
2. Program folder screen
3. Next block display screen
4. Program check screen

On the program screen, you edit the program that is currently selected, and display the block that is currently executed during program operation. In MDI mode, you also edit an MDI operation program, and display the block that is currently executed.

12.2.10 Program Contents Display (15/19-inch Display Unit)

Displays the program currently being executed in MEM mode.

Displaying the program being executed

Procedure

1 Press function key PROG to display the program screen.

2 Press vertical soft key [PROGRAM].
The cursor is positioned at the block currently being executed.

Fig. 12.2.10 (a) Screen for displaying the program being executed (15-inch display unit)

FRYER MACHINE SYSTEMS INC
Zahensky

12.2.10.1 Displaying the executed block (BC)

For an explanation of how to display the executed block, see Subsection, "Displaying the executed block".

12.2.10.2 Call stack display (15/19-inch display unit)

Overview

In the call stack display, the program name and nest level of subprogram from main program to current executing subprogram is displayed. When the device to which program is stored is a CNC build-in memory, the program comment is displayed in addition to a program name and a nest level. Moreover, the folder name where the program is stored is displayed in the active program now.

Call stack display

Procedure

1 Press the function key .

2 Press the vertical soft key [CHECK].

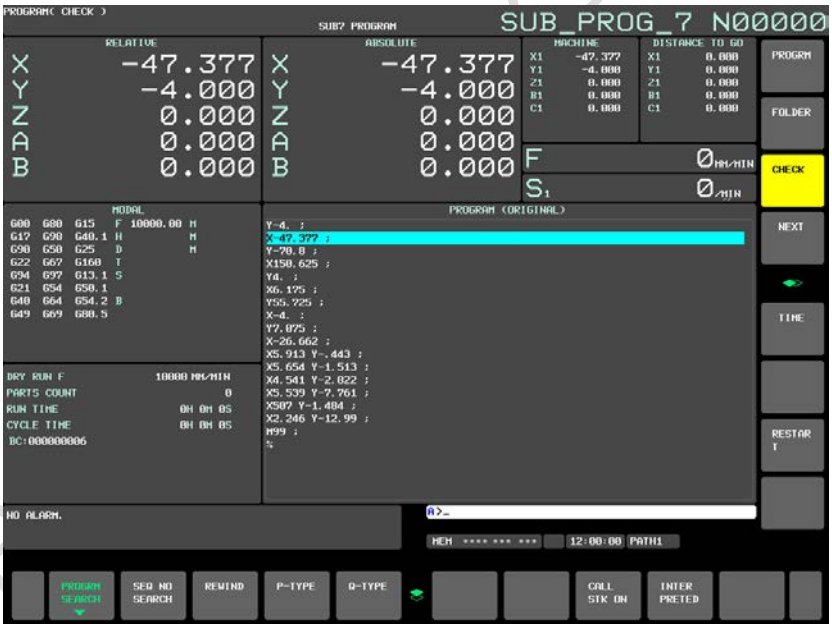


Fig.12.2.10.2 (a) Program check screen (15-inch) (Non call stack display)

FRYER SYSTEMS INC

3 Press the horizontal soft key [CALL STK ON].
The call stack is displayed on the program check screen.
And the display of the horizontal soft key changes into [CALL STKOFF].

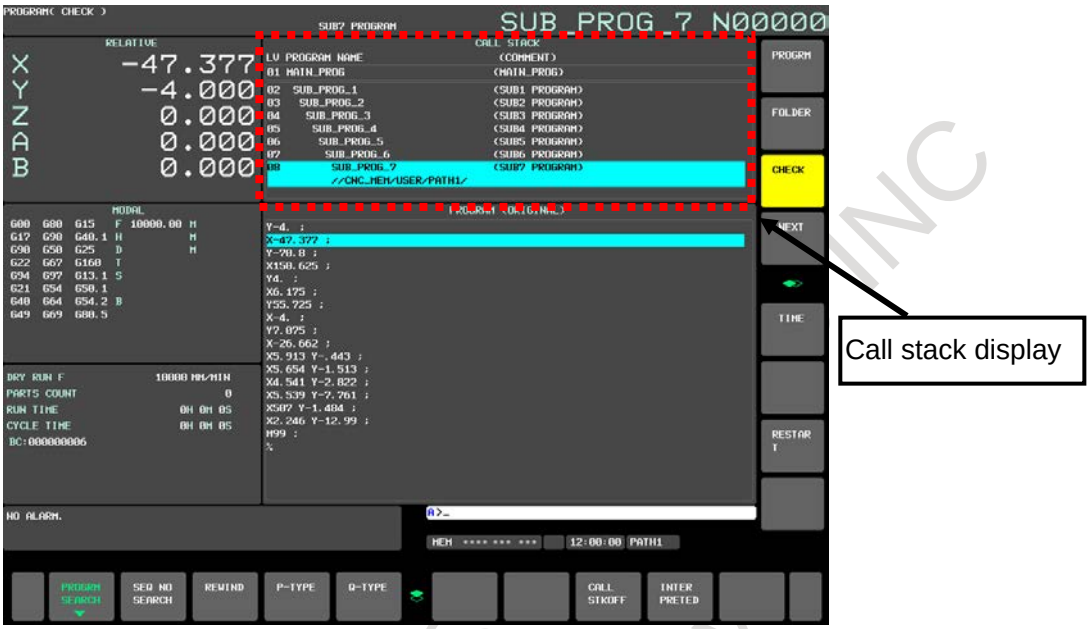


Fig.12.2.10.2 (b) Program check screen (15-inch) (Call stack display)

4 The screen returns to Fig.12.2.10.2(a) by pressing the horizontal soft key [CALL STKOFF].

Explanation

In the call stack, nest level and program name and program comment of subprogram from main program to program being executed is displayed. Moreover, about the program being executed, in addition to program name and program comment, the folder information is displayed.
The following items are displayed on the screen.

Table12.2.10.2 Item of call stack display

Content of display	Explanation
NEST LEVEL	Level of subprogram calling from main program to program being executed is displayed by the number. The main program is displayed in the line of nest level 01.
PROGRAM NAME	The name of subprogram called from the main program is displayed. And when all calling program cannot be displayed within the area, it is displayed from the one with large number of nest level. Moreover, the folder where the program is stored is displayed for the active program. The head of the folder is a device name.
PROGRAM COMMENT	In case of CNC build-in memory, Program comment is displayed in parentheses at the right of program name.

FRYER MACHINE S INC
Rob Zahensky

Example of screen

When the block of the cursor position is executed by the subprogram call from the main program that exists in the figure below,

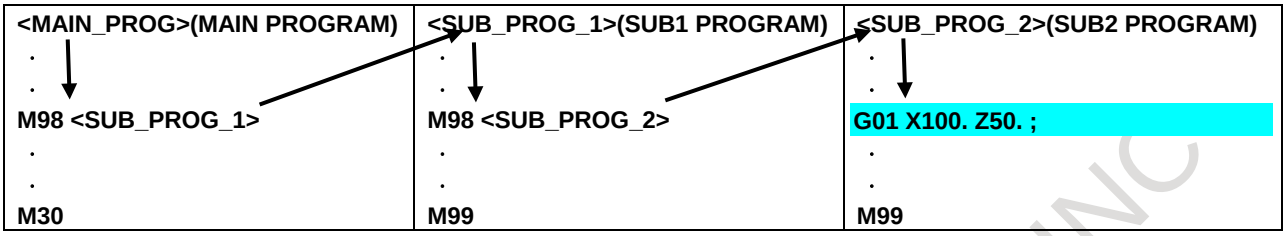


Fig.12.2.10.2 (c) Example of subprogram call

The call stack is displayed as shown in the figure below.

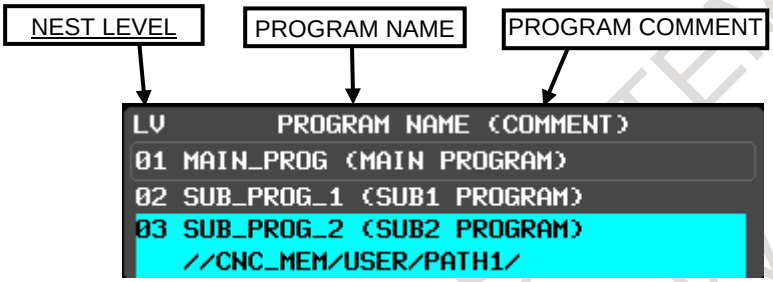


Fig.12.2.10.2 (d) Call stack display

NOTE

1 When the storing number of program name and program comment and the folder information exceed display area, the exceed part is displayed by the "…".
2 As for MDI program, the program name is displayed as O0000 in the call stack.
3 As for External program, the folder where the program is stored is not displayed and the top of program path name is displayed as "//DNC" in the call stack.
4 The programs of external subprogram call (M198) and execution macro are not displayed in the call stack.
5 In the multi-path system simultaneous display, the call stack display can not be performed on the same screen. However, if single-path display mode is set (bit 3 (MTS) of parameter No.11355#3 is 1), the call stack can be displayed.

FRYER MACHINE SYSTEMS INC
Rob Zakrensky

12.2.10.3 Display interpreted and original programs on the same screen (15/19-inch display unit)

Overview

The input custom macro programs before conversion (original) and after conversion (interpreted) are displayed on the program check screen at the same time.
If bit 5 (EPC) of parameter No.11356 is set to 1, this function becomes effective.

Display interpreted and original programs on the same screen

Procedure

1 Press the function key PROG.

2 Press the vertical soft key [CHECK].

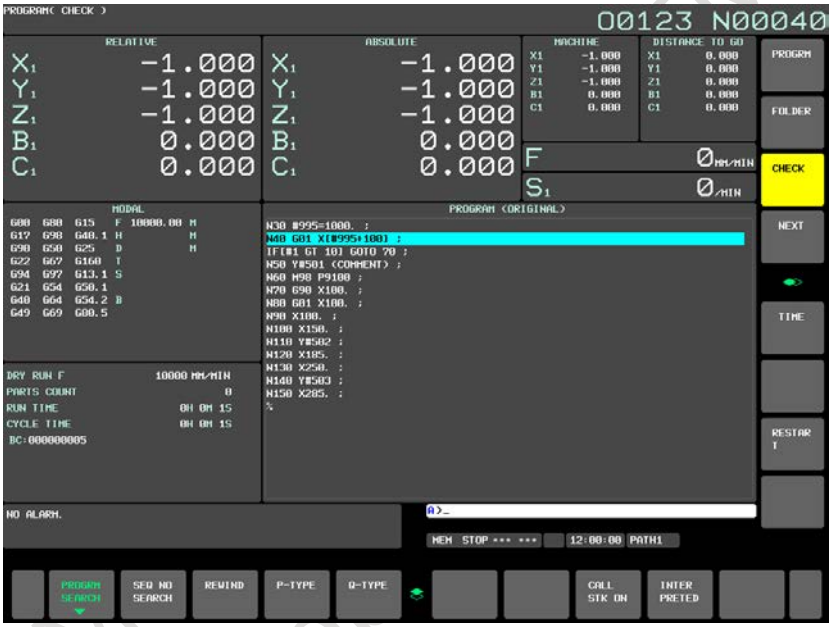


Fig.12.2.10.3 (a) Program check screen (15-inch)
(Non display interpreted and original programs on the same screen)

FRYER MACHINE TOOLS INC

3 Press the horizontal soft key [INTERPRETED].
The original program display is switched to the simultaneous display of interpreted program and original program. And the horizontal soft key [INTERPRETED] is changed to [ORIGINAL].

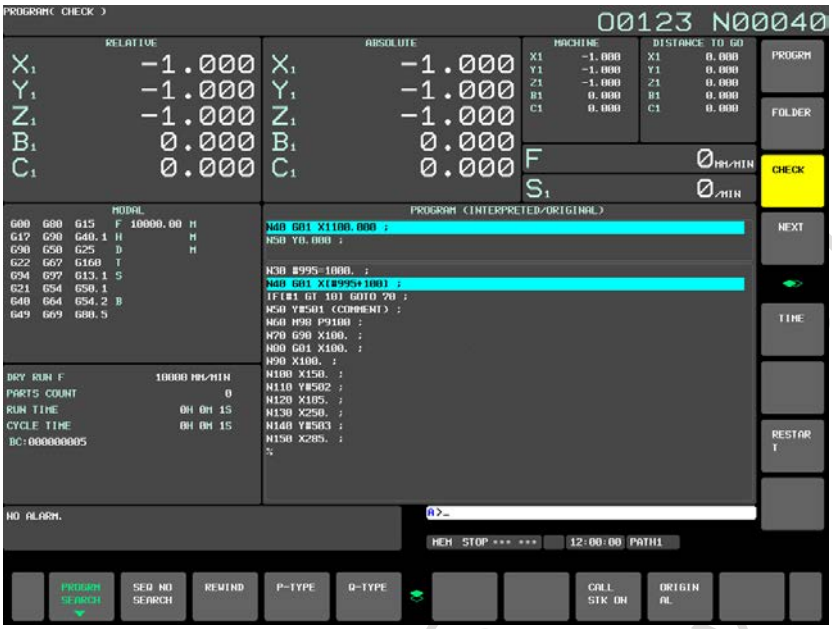


Fig.12.2.10.3 (b) Program check screen (15-inch)
(Display interpreted and original programs on the same screen)

4 The screen returns to Fig.12.2.10.3(a) by pressing the horizontal soft key [ORIGINAL].

Explanation

- For interpreted program

The program after the custom macro program is calculated is displayed.
Both the block being executed now and the block executed next to the executing block are displayed.
When the block executed next is a call command of the subprogram such as M98, called program is displayed.

- For original program

The program before the custom macro program is calculated is displayed.
The blocks displayed before and after the active block are displayed in the order of registration in the memory.
The block is not necessarily a block executed last time, or, the block is not necessarily a block that will be executed next time either.

The difference point of the original program and the interpreted program is as follows.

Table12.2.10.3 The difference point of the original program and the interpreted program

	Comment	Macro program	Macro Variable	Decimal Point	Subprogram call
Interpreted Program	Not displayed	Not displayed	Value stored in macro variable	Least increment system	Not displayed
(Example)	G01	(Not displayed)	G01 X123.456 ;	G01 X1.000 ;	(Not displayed)
Original Program	Displayed	Displayed	Macro variable	The decimal point position is the same as the input.	Displayed
(Example)	G01 (COMMENT)	IF [#1 GT 10] GOTO 20	G01 X#100 ;	G01 X1. ;	M98P1000;

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Example of screen

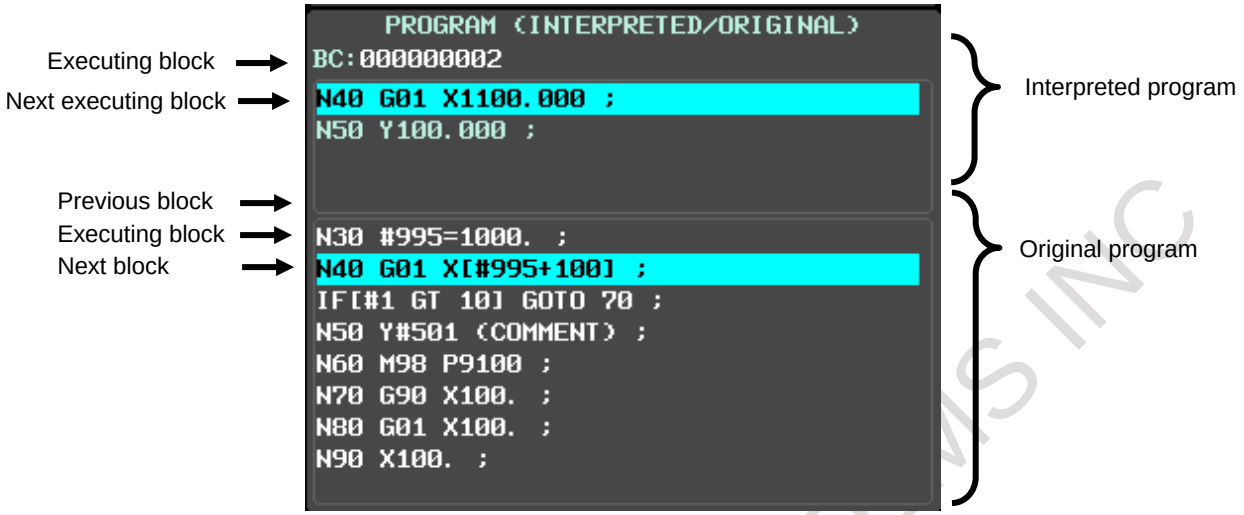


Fig.12.2.10.3 (c) Display interpreted and original programs on the same screen

NOTE

1 In case of the program stored in internal memory (CNC MEM) and the program in memory card program device (MEMCARD), the interpreted and original programs are displayed on the same screen.
2 Protected programs and execution macro programs are not displayed.
3 The interpreted program is displayed in the area of the original program display during DNC operation.
4 In operating of single block, the cursor shows the executed block. The cursor doesn't move to the next block.
5 In the multi-path system simultaneous display, the display interpreted and original programs can not be performed on the same screen.

Restrictions

This function cannot be used at the same time with the following functions.

- Manual Handle Retrace
- Retrace

12.2.11 Editing a Program (15/19-inch Display Unit)

A program can be edited in the EDIT mode.
Two modes of editing are available. One mode is word editing, which performs word-by-word editing. The other is character editing, which performs character-by-character editing.
For program creation and editing operation, see Chapter, "CREATING PROGRAMS " and Chapter , "EDITING PROGRAMS".

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

Displaying the program editing screen

Procedure

1 Press function key to display the program screen.

2 Press vertical soft key [PROGRAM].

- Word editing

Editing operations such as text insertion, modification, and deletion, and cursor movements are performed on a word-by-word basis.

Fig. 12.2.11 (a) Program word editing screen (15-inch display unit)

- Character editing

Program editing operations and cursor movements are performed on a character-by-character basis as with a general text editor.

Text is input directly to the cursor position instead of using the key input buffer.

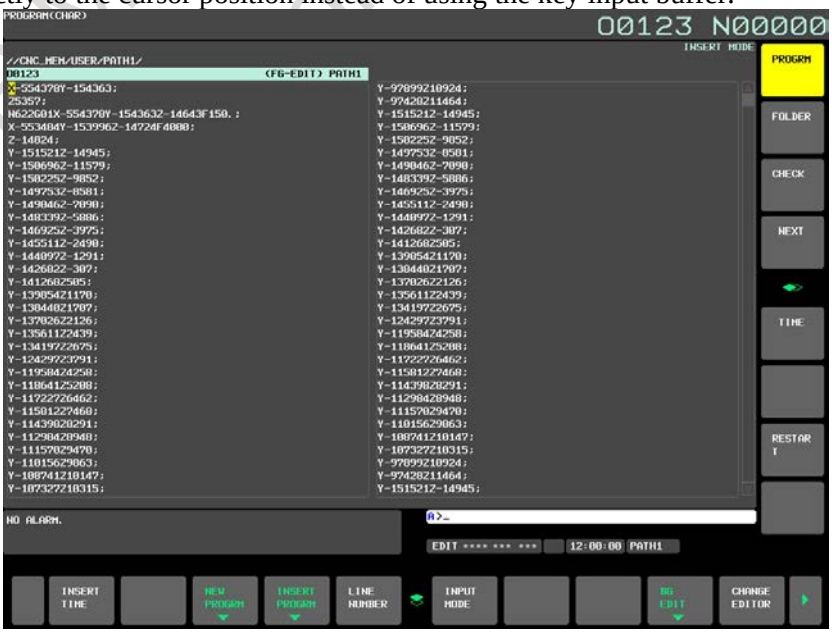


Fig. 12.2.11 (b) Program character editing screen (15-inch display unit)

FRYER MACHINERY INC

Switching between program editing modes

You can switch between word editing and character editing with horizontal soft keys.

Procedure

1 Press function key PROG to display the program screen.
2 Press vertical soft key [PROGRAM].
3 Press continuous menu key to display horizontal soft key [CHANGE EDITOR].
4 Pressing horizontal soft key [CHANGE EDITOR] switches the editing mode between word editing and character editing.

12.2.12 Program Screen for MDI Operation (15/19-inch Display Unit)

During MDI operation or editing of an MDI operation program in the MDI mode, the program currently being executed mode is displayed.
For MDI operation, see Section, “MDI Operation”.

Procedure for displaying the program screen for MDI operation

Procedure

1 Press function key PROG.
2 Press vertical soft key [PROGRAM].
The program input from the MDI is displayed.

Fig. 12.2.12 (a) MDI operation program screen (15-inch display unit)

FRYER ... NE SYSTEMS INC
...nsky

12.2.13 Program Folder Screen (15/19-inch Display Unit)

A list of programs registered in the program memory is displayed.
For the program folder screen, see Chapter, “PROGRAM MANAGEMENT”.

Displaying the program folder screen

Procedure

1 Press function key [PROG].

2 Press vertical soft key [FOLDER].

Fig. 12.2.13 (a) Program folder screen (15-inch display unit)

Displaying comments on the program folder screen

The comments of the first block of the program are displayed by setting bit 1 (COL) of parameter No. 11351 to 1.

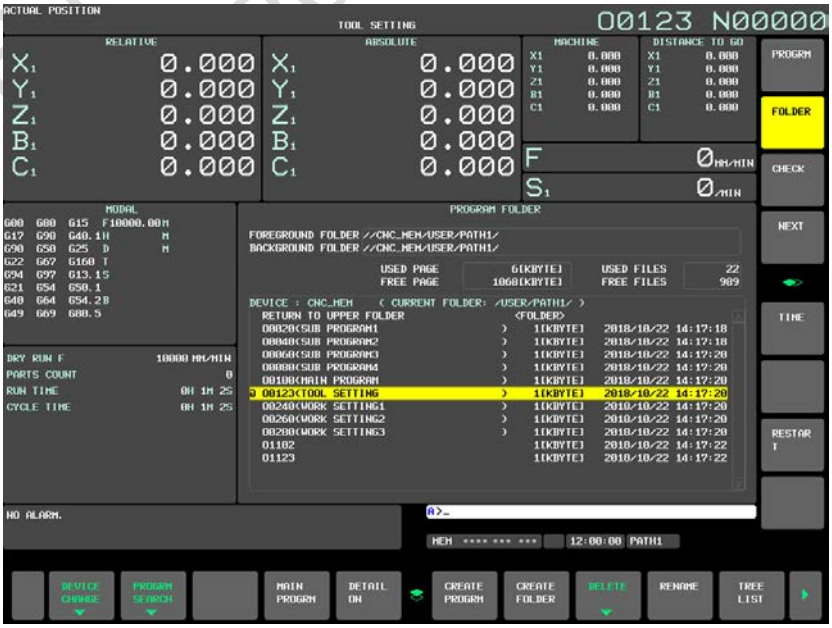


Fig.12.2.13 (b) Displaying comments on the program folder screen

FRYER MACHINE INC

12.2.13.1 Split display on the program folder screen

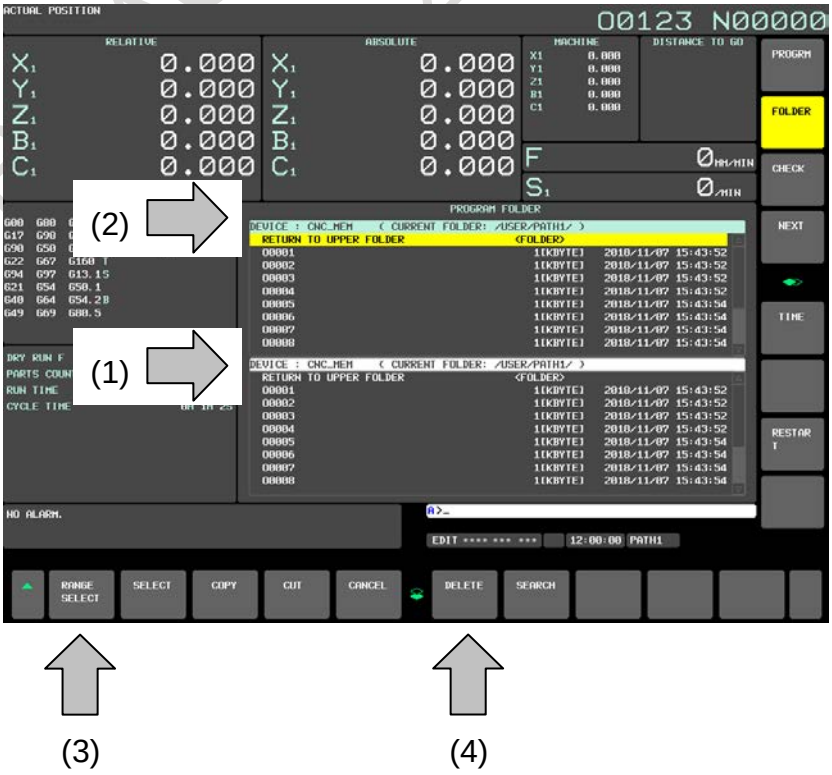
Overview

On the program folder screen, the folder information display can be split into two folder information views, upper and lower, as shown in the Fig. 12.2.13.1 (a).

Fig. 12.2.13.1 (a) Program folder screen (split-display screen)(15-inch display unit)

On the split folder display, file information about different sets of devices or folders can be shown in different views at the same time, so you can copy or move files easily.

Example of copying a file from the CNC_MEM to the Data Server



FRYER MACHINE INC Zanen

Procedure

<1> Change to a destination folder on the Data Server.
<2> Change to a folder on the CNC_MEM that contains a file you want to copy.
<3> Select the file.
<4> Copy the file.

Procedure for switching to split display

Procedure

1 Press function key [PROG] to display the program folder screen.
2 Press horizontal soft key [(OPRT)].
3 Press continuous menu key [▷] twice.
Horizontal soft key [MULTI LIST] appears.

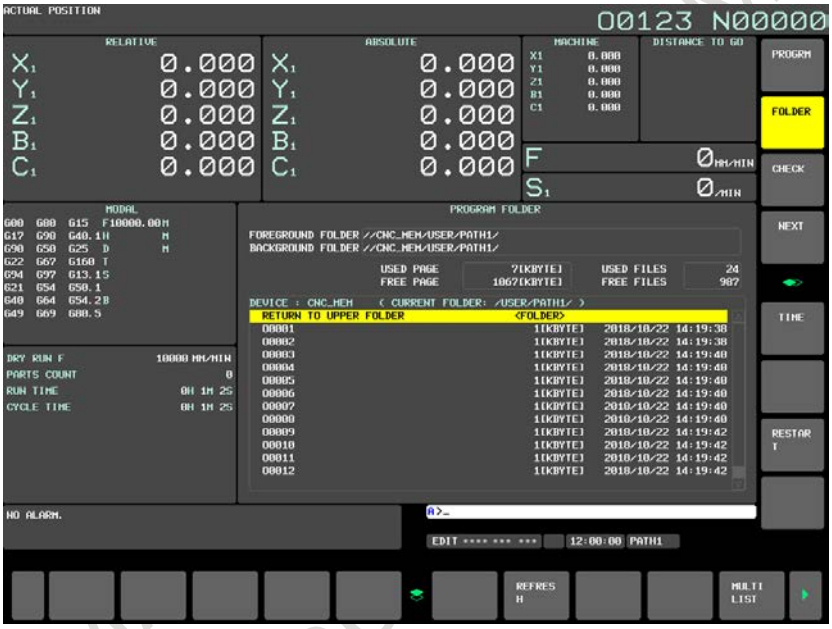


Fig. 12.2.13.1 (b) Program folder screen normal screen)(15-inch display unit)

FRYER MS INC

4 Press horizontal soft key [MULTI LIST].
The folder information display is split into two folder views, upper and lower, which show the same folder information. Immediately after the splitting, the upper folder view becomes active for operations. At the same time, horizontal soft key [SINGLE LIST] appears.

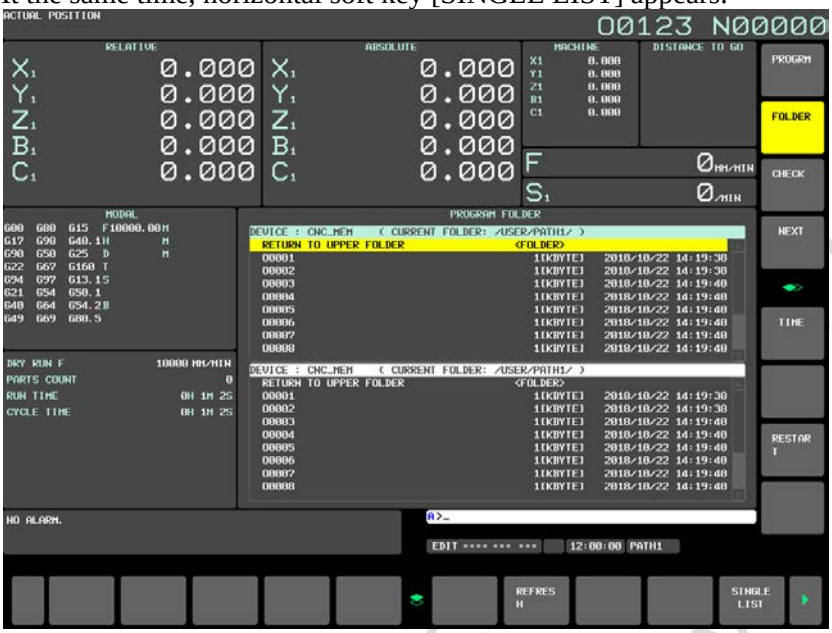


Fig. 12.2.13.1 (c) Program folder screen (split-display screen)(15-inch display unit)

5 Pressing horizontal soft key [SINGLE LIST] takes you back to the original program folder display. Then, the folders are shown which have been selected in the active view for operations on the split display.

Procedure for switching between active folder views for file operations on the split display

On the split folder display, you can switch between active folder views for file operations as described below.
In the active folder view for file operations, the cursor is displayed on the file list.

Procedure

1 When the upper folder view is active for file operations and you press MDI key [SHIFT] and cursor key [↓], the lower folder view becomes active for file operations.

2 When the lower folder view is active for file operations and you press MDI key [SHIFT] and cursor key [↑], the upper folder view becomes active for file operations.

NOTE

1 In step 1, even if you press MDI key [SHIFT] and cursor key [↑] when the upper folder view is active for file operations, the lower folder view becomes active for file operations.

2 In step 2, even if you press MDI key [SHIFT] and cursor key [↓] when the lower folder view is active for file operations, the upper folder view becomes active for file operations.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Explanation

- File operations on the split display

The following operations are allowed on the split display.

<1> You can select and display devices or folders individually in each folder view.

<2> You can copy or move files between folders selected in different folder views.
(For details, see the section on operations of copying/moving between devices.)

In the selected folder view, you can perform the same operations as on the normal screen.

- Path switching

When path switching is performed while the split display is provided for a multi-path system, the folder view in which the CNC MEM is selected as a device is changed to a view of the last selected folder for each path on the CNC MEM.

- Device information display

The device information as shown in the normal program folder screen (portion in a dotted box in the Fig. 12.2.13.1 (d)) is not shown on the split display (Fig. 12.2.13.1 (c)).

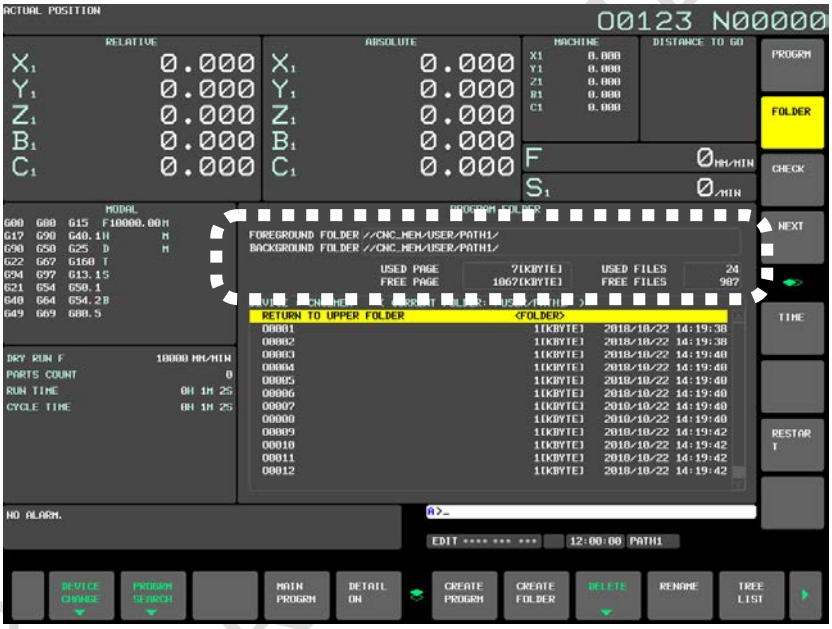


Fig. 12.2.13.1 (d) Program folder screen (normal screen)(15-inch display unit)

You should see the device information on the normal screen.

Limitation

- Device that can be selected in both views at the same time on the split display

The same device of any of the following types can be selected in both views at the same time on the split display.

- CNC MEM
- MEM CARD(binary format)
- Data Server

For other types, the same device cannot be selected for display in the folder views at the same time.

If any other type of device is selected when split display starts, the upper and lower folder views cannot show the same sets of folders due to the above limitation. Therefore, the lower folder view shows the folder selected as a foreground folder.

FRYER PRODUCTS SYSTEMS INC

- **Display units that support the split display**

The split display is supported on 10.4-inch, 15-inch, and 19-inch display units.

12.2.13.2 Folder tree display (15/19-inch display unit)

This is same as the 10.4-inch display units. Refer to Subsection, “Folder tree display”.

12.2.14 Next Block Display Screen (15/19-inch Display Unit)

Displays the block currently being executed and the block to be executed next.

Procedure for displaying the next block display screen

Procedure

1 Press function key PROG.

2 Press vertical soft key [NEXT].
The G codes, addresses, command values specified in the block currently being executed and the next block are displayed.

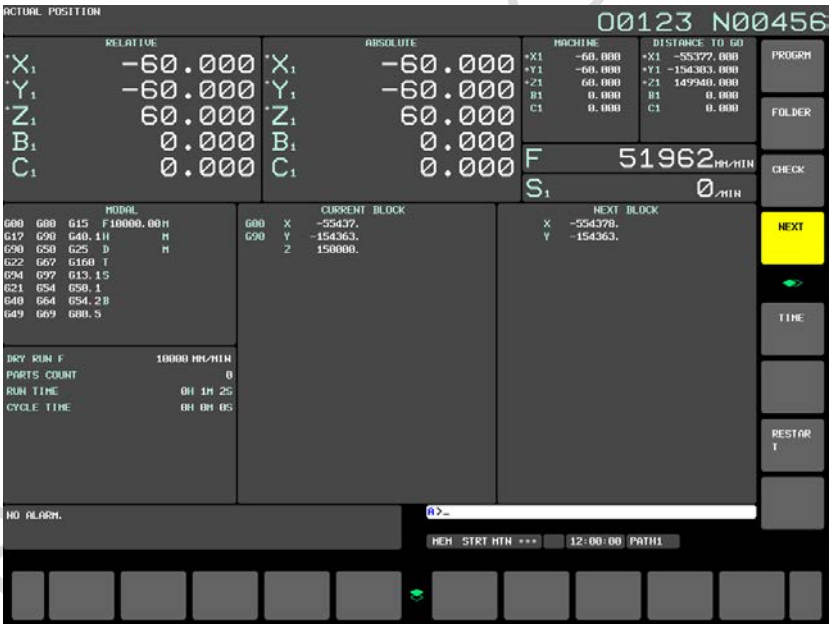


Fig. 12.2.14 (a) Next block display screen (15-inch display unit)

FRYER SYSTEMS INC

12.2.15 Program Check Screen (15/19-inch Display Unit)

Displays the program currently being executed, current position of the tool, and modal data.

Procedure for displaying the program check screen

Procedure

1 Press function key .

2 Press vertical soft key [CHECK].
The program currently being executed, current position of the tool, and modal data are displayed.

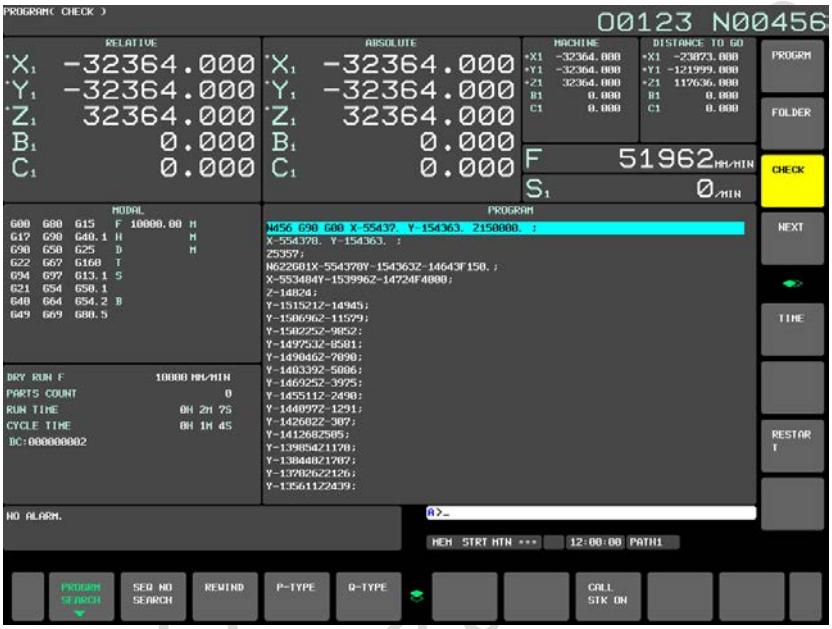


Fig. 12.2.15 (a) Program check screen (15-inch display unit)

Explanation

- Program display

The program currently being executed is displayed.
The block being executed is displayed in reverse video.

- Current position display

The current position in the relative coordinate system, workpiece coordinate system, and machine coordinate system, and the remaining amount of travel are displayed.

- Modal G codes

Up to 24 modal G codes are displayed.

FRYER MACHINE INC Rob Z

12.2.16 Background Editing (15/19-inch Display Unit)

Editing one program during execution of another program is referred to as background editing. You can perform the same edit operations in the background as those in normal editing (foreground editing).

On a 15-inch or 19-inch display, you can perform background editing for the programs displayed in multiple tiled windows. You can switch from a program displayed in one of the tiled windows to another to perform copy, paste, and other edit operations, which allows you to edit programs efficiently.
You can edit four programs simultaneously.

Function

- Background editing

A program other than the currently selected program can be edited. Background editing can be performed in any mode.

- EDIT mode and reference mode

When background editing is started in the EDIT mode, the program can be edited. When background editing is started in the reference mode, the program is opened in the read-only mode and cannot be edited.
While a program opened in the EDIT mode cannot be run, a program opened in the reference mode can be run.

- Program selected in the foreground

If the program selected in the foreground is specified as a program to be edited in the background, background editing is started in the read-only mode. The text at an arbitrary position of the program currently being executed can be checked.

- Switching from the program directory screen to the editing screen

By selecting a program from the program folder screen, background editing can be started.
On the program folder screen, move the cursor to a program to be edited in the background, then press the MDI key INPUT. The screen display switches to the background editing screen.
Background editing can be started without entering a program name.

- Background editing on a multi-path system

Background editing does not depend on the path.
Even if the path to be operated is changed using the path selection signal, the background editing currently being executed is just continued.

NOTE

The following programs cannot be edited in the background. They can be opened only in the read-only mode.
- Running program
- Main program
- Program with the edit disable attribute

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Display

When background editing starts, the ordinary program editing screen switches to the background editing screen.
When two or more programs are edited in the background, the screen is split to display these programs. For a 15-inch or 19-inch display unit, up to four programs can be edited at the same time.

- Word editing

Fig. 12.2.16 (a) shows background word editing performed simultaneously for two programs (right and left programs). On the status line at the top of the window for each program, the program name and "BG-EDIT" (indicating background editing in progress) are displayed.
The status line of the program being edited is displayed in reverse video.

Fig. 12.2.16 (a) Background editing screen (word editing) (15-inch display unit)

FRYER MACHINE TOOLS INC
Rob

- Character editing

Fig. 12.2.16 (b) shows background character editing performed simultaneously for two programs (right and left programs). Similarly to word editing, at the top of the window for each program, the status line is displayed. In addition, the current input mode (INSERT MODE or OVERWRITE MODE) is displayed at the upper right of the screen for character editing.
The status line of the program being edited is displayed in reverse video.

Fig. 12.2.16 (b) Background editing screen (character editing) (15-inch display unit)

- Editing status

The following items are displayed on the status line and program editing area according to the background editing status.

Editing status	Displayed items
No program selected	(BG-EDIT) “NO PROGRAM” is displayed in the editing area.
Program opened	program-name + (BG-EDIT)
Read-only program opened	program-name + (BG:READ ONLY) The contents of the program are displayed in green.

Starting background editing from the editing screen

Procedure (word editing)

Method 1

1 Press function key [PROG].
2 Press soft key [PROGRAM].
3 Enter a program name.
4 Press soft key [BG EDIT].

NOTE
1 Background editing is performed in the EDIT mode.
2 If a nonexistent program is specified, a new program is created.

FRYER MACHINE INC Rob Zaheer

Method 2

1 Press function key PROG.
2 Press soft key [PROGRAM].
3 Press soft key [BG EDIT].
4 Enter a program name.
5 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode.

NOTE
If a nonexistent program is specified,
- Pressing soft key [EDIT EXEC] causes a new program to be created.
- Pressing soft key [REF EXEC] causes the warning message “SPECIFIED PROGRAM NOT FOUND” to appear. A new program is not created.

Method 3

1 Press function key PROG.
2 Press soft key [PROGRAM].
3 Press soft key [BG EDIT].
4 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode. At this time, “NO PROGRAM” is displayed on the screen.
5 Enter a program name.
6 Press soft key [PROGRM SEARCH].

NOTE
If a nonexistent program is specified, the warning message “SPECIFIED PROGRAM NOT FOUND” is displayed. A new program is not created.

Procedure (character editing)

Method 1

1 Press function key PROG.
2 Press soft key [PROGRAM].
3 Press soft key [BG EDIT].
4 Enter a program name.
5 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode.

NOTE
If a nonexistent program is specified,
- Pressing soft key [EDIT EXEC] causes a new program to be created.
- Pressing soft key [REF EXEC] causes the warning message “SPECIFIED PROGRAM NOT FOUND” to appear. A new program is not created.

FRIEP MACHINE SYSTEMS INC
Rob Zahensky

Method 2

1 Press function key .
2 Press soft key [PROGRAM].
3 Press soft key [BG EDIT].
4 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode. At this time, “NO PROGRAM” is displayed on the screen.
5 Press soft key [PROGRM SEARCH].
6 Enter a program name.
7 Press soft key [PROGRM NAME].

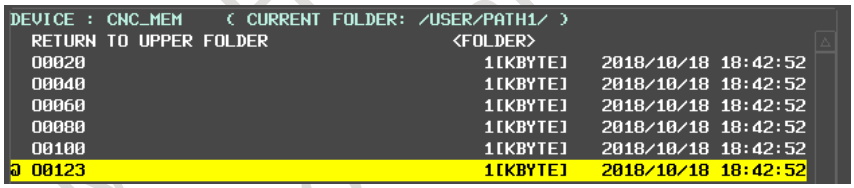
NOTE
If a nonexistent program is specified, the warning message “SPECIFIED PROGRAM NOT FOUND” is displayed. A new program is not created.

Starting background editing from the program folder screen

By selecting a program from the program folder screen, background editing can be started. The cursor is used to select a program. You do not need to enter a program name.

Procedure

1 Press function key .
2 Press vertical soft key [FOLDER].
3 Move the cursor to a program to be edited.
4 Press the MDI key . The screen display is switched to the background editing screen.



NOTE
When background editing is started from the program folder screen, the EDIT mode is set.
For the following programs, the reference mode is set, however:
- Running program
- Main program
- Program with the edit disable attribute

FRYER MACHINE SYSTEMS INC
Roshensky

Background editing operation

- Editing operation

The same editing operations as performed in the foreground can be performed.

- Switching from a program to another for editing

To switch from a program to another for editing when editing multiple programs simultaneously in the background, press [SHIFT] key, then [←] or [→].

- Editing operation in the read-only mode

Cursor movement and page switching are possible. However, program editing is disabled.

- Copying and pasting text between programs

Text in a program selected and copied during program editing is kept held after the program to be edited is changed. You can paste text in a program onto another program.

Background editing end operation

Background editing can be ended using the procedure described below.
The procedure for ending background editing of one program and that for ending all background editing of multiple programs are shown below.

- Ending editing of one program

1 Press function key [PROG].
2 Press vertical soft key [PROGRAM].
3 Select a program for which to end editing.
4 Press horizontal soft key [BG END].

- Ending editing of all programs

1 Press function key [PROG].
2 Press vertical soft key [PROGRAM].
3 Press horizontal soft key [BG ALL END].

To return to ordinary foreground editing, end all background editing. If at least one program is being edited in the background, you cannot return to foreground editing.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.2.17 Stamping the Machining Time (15/19-inch Display Unit)

The execution times of the most recently executed ten programs can be displayed in hours, minutes, and seconds.
The calculated machining time can be inserted as a comment of the program to check the machining time on the program folder screen.

Procedure for stamping the machining time

Procedure

- Displaying the machining time

1 Press function key .

2 Press vertical soft key [TIME]. The machining time display screen appears.

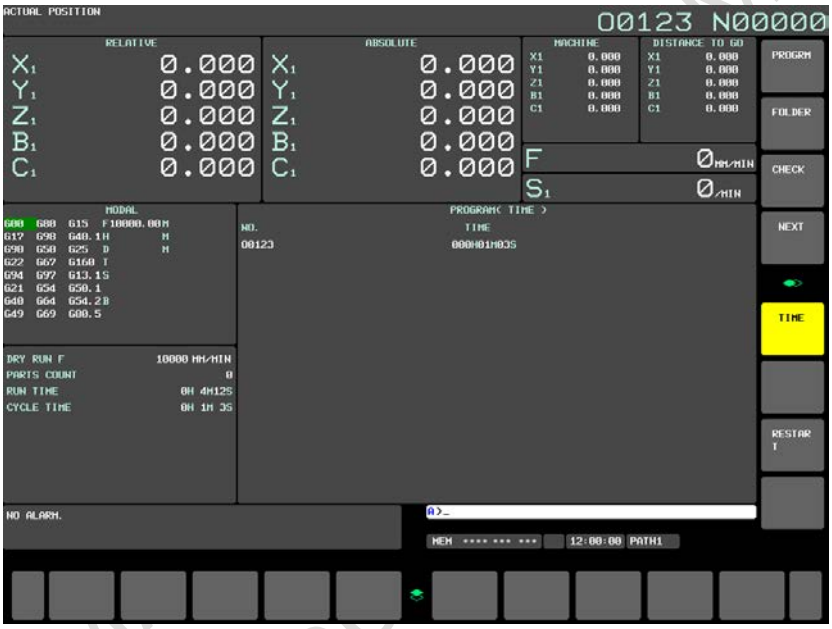


Fig. 12.2.17 (a) Machining time display screen (15-inch display unit)

FRYER MAHAMS INC

- Calculating the machining time

1 Select the memory operation mode, then press the RESET key.

2 Select the program screen, then select a program of which machining time you want to calculate.

3 Execute the program to perform actual machining.

4 When the RESET key is pressed or, M02 or M30 is executed, the machining time count operation stops. When the machining time display screen is selected, the program number of the stopped main program and its machining time are displayed.

5 The Fig. 12.2.17 (b) shows the screen when the machining times of the ten main programs O0020, O0040, …, and O0200 are displayed and the screen when the machining time of O0220 is newly calculated after that.

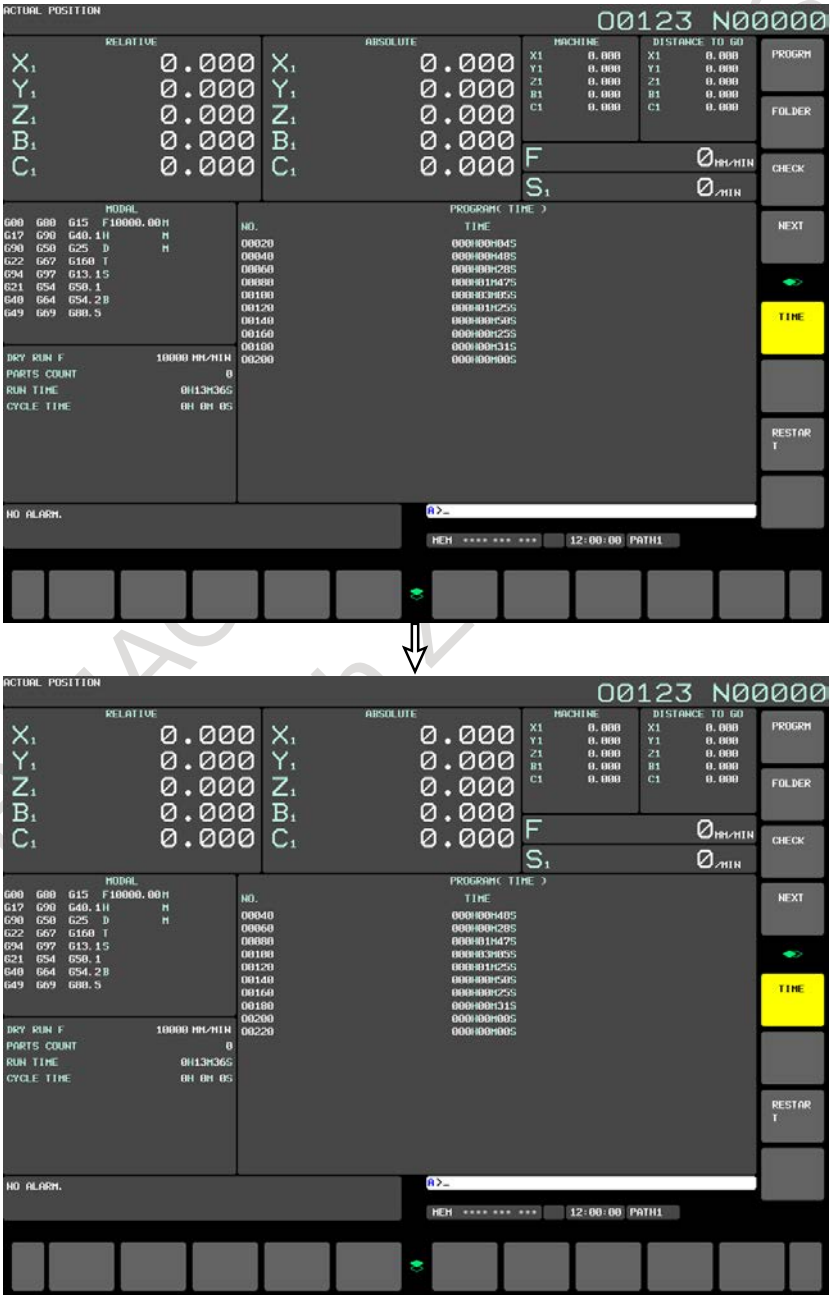


Fig. 12.2.17 (b) Stamping the machining time (15-inch display unit)

Procedure for inserting the machining time on the program screen

Procedure

You can display the machining time of a program as a comment of the program. The procedure is shown below:

1 To insert the calculated machining time of a program as a comment, the machining time of the program must be displayed on the machining time display screen. Before stamping the machining time of the program, check that the program number is displayed on the machining time display screen.

2 Set the edit mode or background editing state and select the program screen. Then, select a program of which machining time you want to insert.

3 For example, the machining time of O0100 is displayed on the machining time display screen. Press continuous menu key ▷ until horizontal soft key [INSERT TIME] appears. When horizontal soft key [INSERT TIME] is pressed, the start of the program is located, and the machining time of the program is inserted after the program name.

Press soft key [INSERT TIME].

Fig. 12.2.17 (c) Program screen (15-inch display unit)

FRYER MACHINE TOOLS INC

4 If a comment is written in the block containing the program number of a program of which machining time is to be inserted, the machining time is inserted after the comment.

Press soft key [INSERT TIME].

Fig. 12.2.17 (d) Program screen (15-inch display unit)

FRYER SYSTEMS INC

Display on the program folder screen

The machining time of a program inserted in the program as a comment is displayed after the existing comment of the program on the program folder screen.

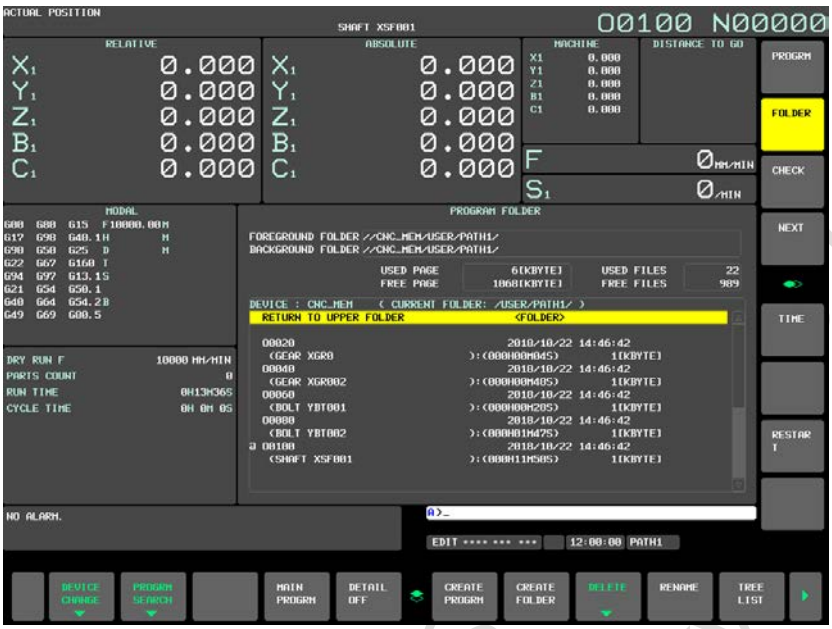


Fig. 12.2.17 (e) Program folder screen (15-inch display unit)

Explanation

- Machining time

The machining time is counted from the initial start after a reset in the memory operation mode to the next reset. If a reset is not performed during operation, the machining time is counted from the start to M02 or M30. The time during which operation is held is not counted, but the time used to wait for the completion of the M, S, T, and/or B functions is counted, however.

- Stamping the machining time

You can insert (stamp) the displayed machining time in a program stored in memory as a comment. The machining time is inserted as a comment after the program number.

- Program folder

You can display the machining time inserted after a program number on the program folder screen. This operation lets you know the machining time of each program to use the time as useful reference data when planning processes on site.

Limitation

- Alarm

The execution of a program may be held by an alarm during the machining time count operation. In this case, the machining time is counted until the alarm is released by a reset.

- M02

It may be specified that M02 does not reset the CNC, but returns completion signal FIN to the CNC to restart the program from the beginning successively (bit 5 (M02) of parameter No. 3404 is set to 0). In this case, when M02 returns completion signal FIN, the machining time count operation stops.

- Stamping the machining time

When an attempt is made to stamp the machining time of a program in the program, the machining time of the program may not be displayed on the machining time display screen. In this case, the machining time cannot be inserted in the program even if horizontal soft key [INSERT TIME] is pressed.

FRYER MACHINE SERVICE INC
Rob Zahensky

- Correcting the machining time

If an incorrect machining time is calculated (such as when a reset is made during the execution of a program), reexecute the program to calculate the correct machining time. The same program number may be displayed in two or more lines on the machining time display screen. In this case, the latest calculated machining time is inserted in the program.

- States of the stamped machining time

In the following states, the stamped machining time is displayed on the program folder screen as shown below.

1 When the comment of a program is longer than 16 characters
The 17th and subsequent characters of the comment are discarded and the machining time display field is left blank.

Fig. 12.2.17 (f) When the comment of a program is longer than 16 characters (15-inch display unit)

FRYER MACHINE TOOLS INC

2 When two or more machining times are stamped
The first machining time is displayed.

Fig. 12.2.17 (g) When two or more machining times are stamped (15-inch display unit)

FRYER SYSTEMS INC

3 When the format of an inserted machining time is not “hhhHmmMssS” (H following a 3-digit number, M following a 2-digit number, and S following a 2-digit number, in this order)
The machining time display field is left blank.

Fig. 12.2.17 (h) When the format of an inserted machining time is not “hhhHmmMssS” (H following a 3-digit number, M following a 2-digit number, and S following a 2-digit number, in this order) (15-inch display unit)

12.2.18 Screen for Assistance in Entering Tilted Working Plane Indexing (15/19-inch Display Unit)

The screens for assistance in entering tilted working plane indexing (hereinafter referred to as guidance screens) include a command type selection screen and a tilted working plane data setting screen. The command type selection screen is used to select a tilted working plane indexing. The tilted working plane data setting screen is used to set specified tilted working plane data required for the selected command. By making settings and performing operations on these guidance screens, a tilted working plane indexing block can be created.
The created block is reflected as a new insertion to a program being edited or as a modification to an existing block.
This function can be enabled by setting bit 1 (GGD) of parameter No. 11304 to 1.

Creation of a new block

The following describes the procedure for creating a tilted working plane indexing block on guidance screens and for inserting the block to a program being edited on a program editing screen.

1 On a program editing screen, display a program to which you want to insert a tilted working plane indexing block.
The foreground editing screen, background editing screen, or MDI editing screen should be displayed.

- Displaying the foreground editing screen
  <1> Select the EDIT mode.
  <2> Press function key PROG.
  <3> Press vertical soft key [PROGRAM].

- Displaying the background editing screen
  <1> Press function key PROG.
  <2> Press soft key [FOLDER].
  <3> Press any of the cursor keys to move the cursor to a program to be edited in the background.
  <4> Press INPUT key.

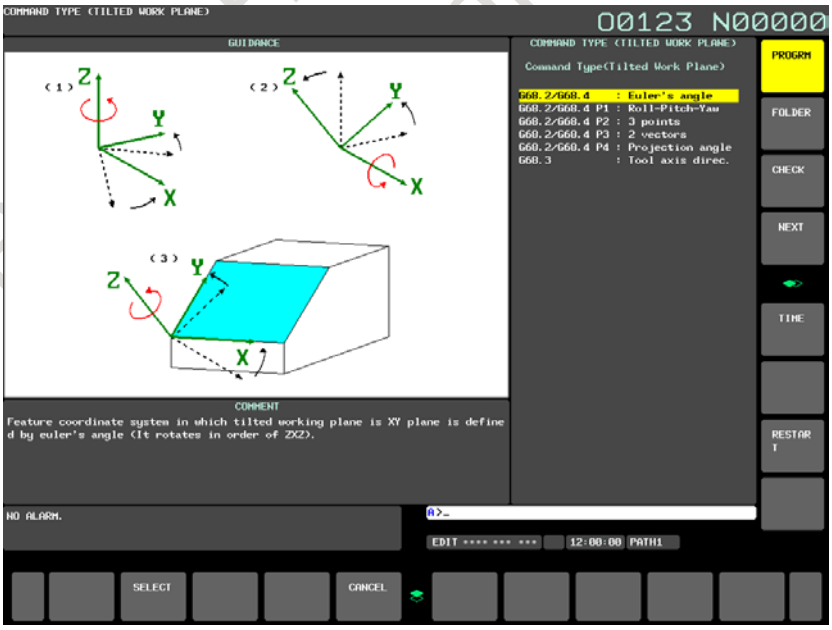
- Displaying the MDI editing screen
  <1> Select the MDI mode.
  <2> Press function key PROG.
  <3> Press vertical soft key [PROGRAM].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

The program editing screen is displayed.

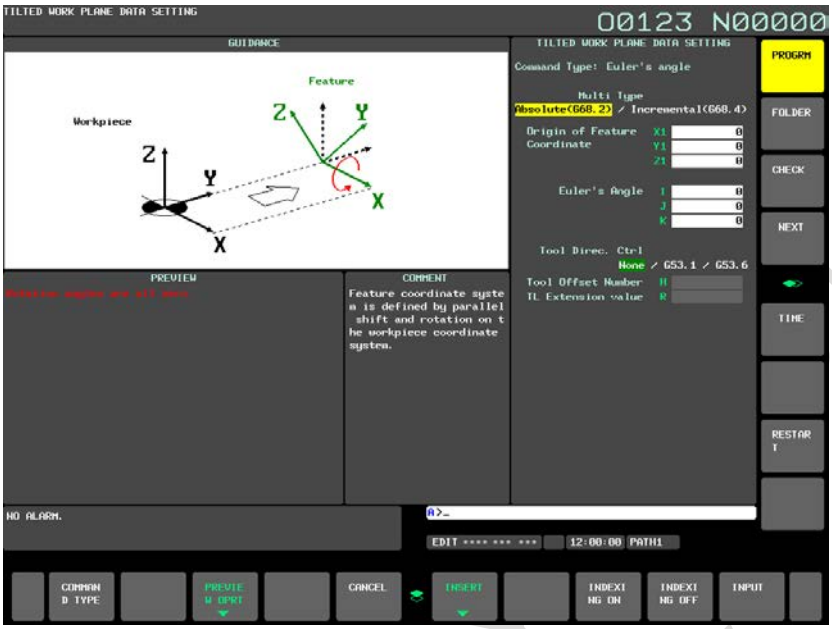
2 Press any of the cursor keys to move the cursor to a position where you want to insert a block.
Note that a block created on the guidance screens is inserted after the block at the cursor position. (If the block at the cursor position includes a tilted working plane indexing, the existing block is modified. See "Modification to an existing block" below.)

3 Press continuous menu key ▷ several times, and then press horizontal soft key [GUIDANCE TWP].
The command type selection screen is displayed.

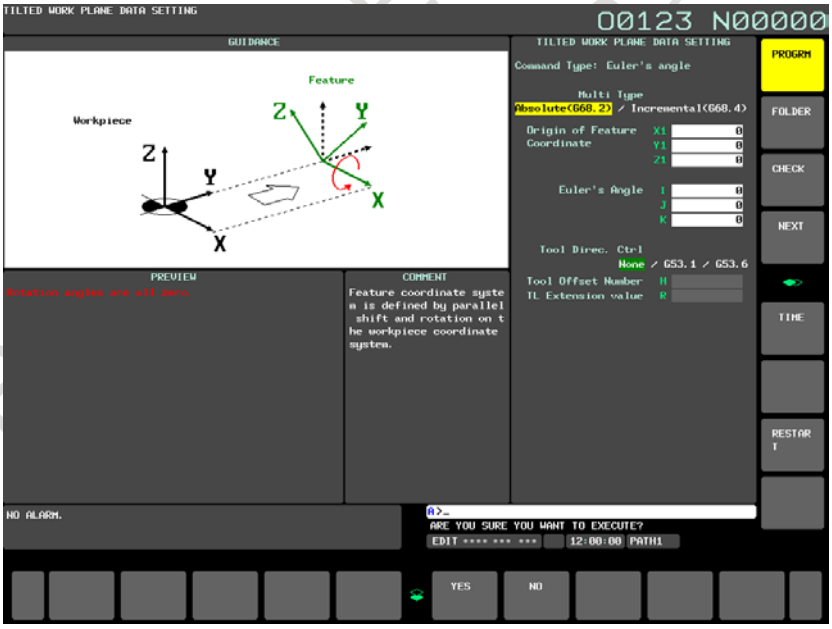


FRYER MACHINE SYSTEMS INC
Chernyshensky

4 Select a command type with any of the cursor keys, and then press horizontal soft key [SELECT].
The tilted working plane data setting screen is displayed.



5 Enter command data for the setting items.
6 Press horizontal soft key [INSERT].



7 Press horizontal soft key [YES].
This takes you back to the program editing screen, where the new block is inserted after the block at the cursor position.

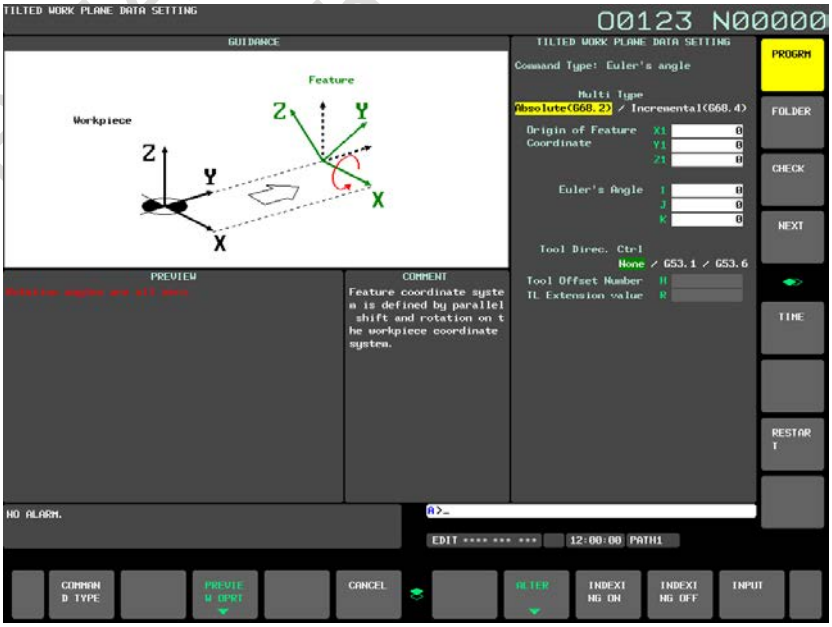
Modification to an existing block

The following describes the procedure for replacing a block in a program being edited on a program editing screen, with a tilted working plane indexing block created on a guidance screen.

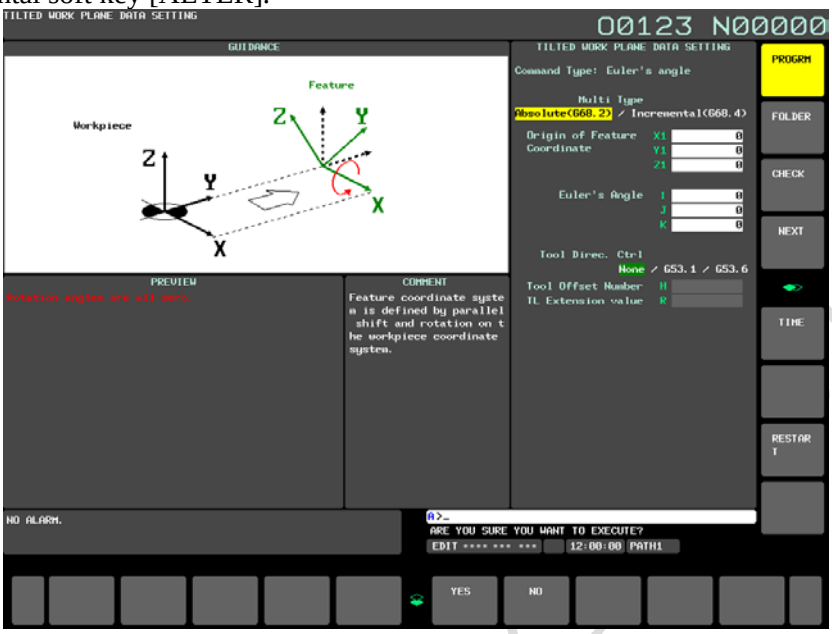
1 On a program editing screen, display a program to be edited.
(For the procedure for displaying a program editing screen, see step 1 in "Creation of a new block".)
The program editing screen is displayed.

2 Press any of the cursor keys to move the cursor to a block to be modified.
For a command extending over more than one block, move the cursor to the first block.

3 Press continuous menu key ▷ several times, and then press horizontal soft key [GUIDANCE TWP]. The tilted working plane data setting screen is displayed.



4 Enter command data for setting items to be modified.
5 Press horizontal soft key [ALTER].



6 Press horizontal soft key [YES].
This takes you back to the program editing screen, where the block at the cursor position is replaced.

Guidance screen cancellation

Pressing horizontal soft key [CANCEL] on a guidance screen takes you back to the program editing screen. At this time, the data that has been set on the guidance screen is discarded.

NOTE

1 In addition to the above operation, the following operations also cancel a guidance screen. The data that has been set on the guidance screen is discarded.
- When bit 7 (CPG) of parameter No. 11302 is 1 (setting for automatically switching between program-related screens according to CNC mode), the CNC mode is changed.
- When a guidance screen is displayed from the foreground editing screen, the CNC mode is changed to a mode other than EDIT, TJOG, or THND.
- When a guidance screen is displayed from the MDI editing screen, the CNC mode is changed to a mode other than MDI.
- When a guidance screen is displayed on a 15/19-inch display unit, the screen is switched by a vertical soft key.
- The screen is switched by an MDI key.
- The screen is switched by a path select signal.
- An event that causes screen switching has occurred, including occurrence of an alarm, display of an operator message, or signal-based display of a screen (such as a tool compensation screen, workpiece shift screen, workpiece coordinate system setting screen, or C Language Executor screen).

2 If MDI key [PROG] is pressed after switching from a guidance screen to another screen, a program editing screen is displayed, instead of the guidance screen.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Notes

- Conditions under which horizontal soft key [GUIDANCE TWP] is displayed
  Horizontal soft key [GUIDANCE TWP] is displayed on a program editing screen under the following conditions:
  1 Foreground editing screen
    - The CNC mode is EDIT, TJOG, or THND.
    - The editing and display are not prohibited for a program to be edited.
  2 Background editing screen
    - A program to be edited is not in reference mode.
    - The editing and display are not prohibited for a program to be edited.
  3 MDI editing screen
    - The CNC mode is MDI.

- Screen displayed when horizontal soft key [GUIDANCE TWP] is pressed
  When horizontal soft key [GUIDANCE TWP] is pressed, which guidance screen is displayed is automatically determined according to the conditions described below.
  1 When a tilted working plane indexing is not included in a block at the cursor position on a program editing screen
    The command type selection screen is displayed. A block created on the guidance screens is inserted after the block at the cursor position on the program editing screen.
  2 When a tilted working plane indexing is included in a block at the cursor position on a program editing screen
    The tilted working plane data setting screen is displayed, which shows the data for the tilted working plane indexing in the block at the cursor position on the program editing screen. When a block of a tool axis direction control command exists immediately after the tilted working plane indexing, the command data for the block is also shown. The tilted working plane indexing in the block at the cursor position on the program editing screen is replaced with a block created on the guidance screen.

NOTE
If the CNC is in the reset state or emergency stop state when horizontal soft key [GUIDANCE TWP] is pressed on the foreground editing screen or MDI editing screen, the warning "PROGRAM READ FAILED" appears, and the operation cannot be continued. (Horizontal soft keys other than [CANCEL] are not displayed.) Press horizontal soft key [CANCEL] to return to the program editing screen, and then press horizontal soft key [GUIDANCE TWP] again.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

12.2.18.1 Command type selection screen

The command type selection screen is used to select the type of a tilted working plane indexing you want to insert into a program to be edited. One of the following command types can be selected:

- G68.2 / G68.4 (Euler's angle)
- G68.2 / G68.4 P1 (Roll-Pitch-Yaw angle)
- G68.2 / G68.4 P2 (3 points specification)
- G68.2 / G68.4 P3 (2 vectors specification)
- G68.2 / G68.4 P4 (Projection angle)
- G68.3 (Tool Axis Direction)

G68.2 is an absolute command, and G68.4 is an incremental command.

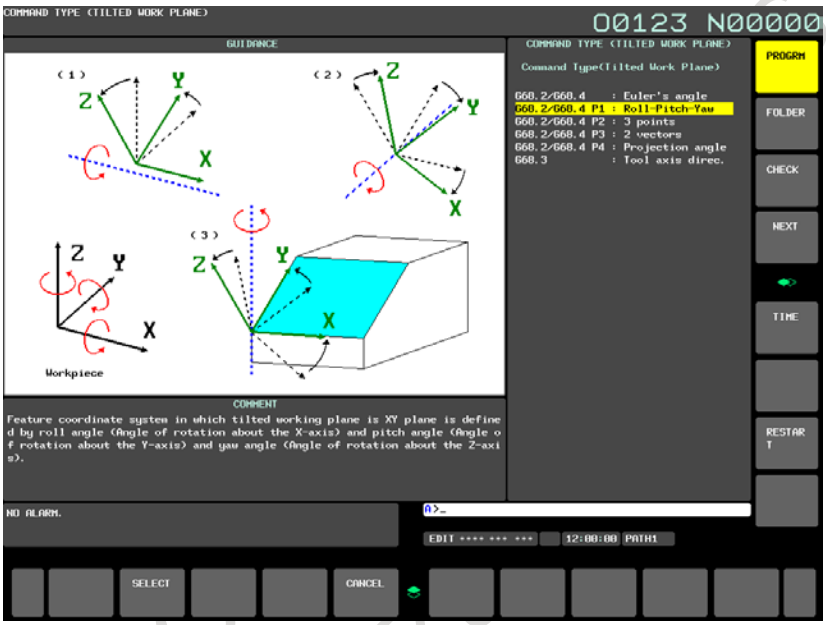


Fig. 12.2.18.1 (a) Command type selection screen (15-inch display unit)

Selection of a command type

1 Press cursor key ⬆ or ⬇ to move the cursor to a command type you want to select. As the cursor is moved, the figure corresponding to the command type at the cursor position is displayed.

2 When horizontal soft key [SELECT] is pressed, the command type at the cursor position is accepted, and the tilted working plane data setting screen is displayed.

NOTE

If the warning "PROGRAM READ FAILED" appears when the command type selection screen is displayed, the operation cannot be continued. (Horizontal soft keys other than [CANCEL] are not displayed.) Press horizontal soft key [CANCEL] to return to the program editing screen, and then press horizontal soft key [GUIDANCE TWP] again.

FRYER MACHINERY Rob ZUMBRUNNEN INC

12.2.18.2 Tilted working plane data setting screen

The tilted working plane data setting screen is used to set specified tilted working plane data required for a tilted working plane indexing of the type that has been selected on the command type selection screen or has been selected when horizontal soft key [GUIDANCE TWP] is pressed.
Different types of tilted working plane data setting screen are provided for different command types. For details of each type of tilted working plane data setting screen, see "12.2.18.3 Details of the tilted working plane data setting screen".

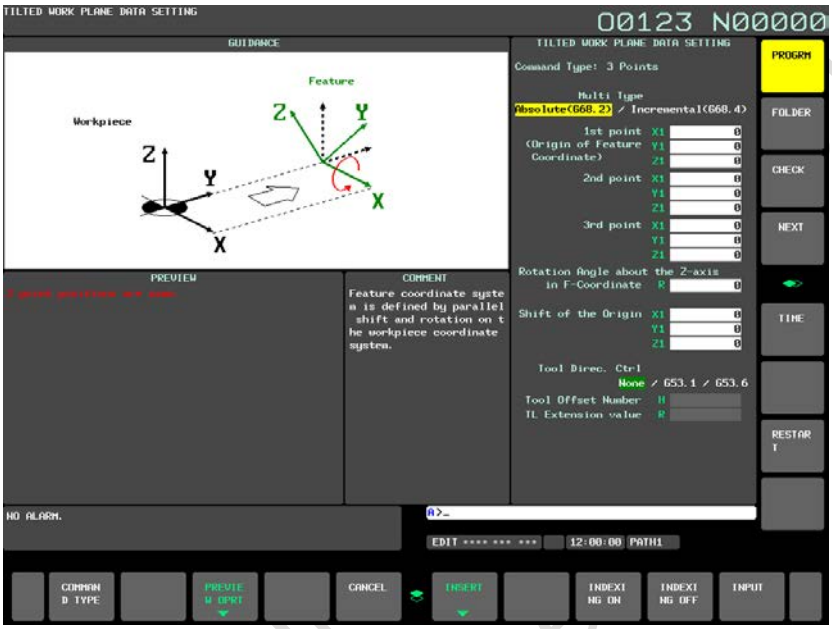


Fig. 12.2.18.2 (a) Tilted working plane data setting screen-3 points specification(15-inch display unit)

Display of the tilted working plane data setting screen

When a new block is created, the initial values are set in the setting and selection items.
When an existing block is modified, the command data for the block at the cursor position on the program editing screen is reflected in the setting and selection items.

NOTE

1 For existing block modification, if the guidance screen is displayed when the cursor is placed in the middle of multiple blocks for a command, the parameters for the block(s) before the cursor are not reflected in the setting and selection items, but only the parameters for the block(s) after the cursor are reflected. If no data is to be reflected in a setting or selection item, the initial value is set in the item.

2 For existing block modification, only the commands with parameter-specified axis names are reflected in the setting items on the screen. (The values specified in data for commands with incorrect axis names are not reflected in the setting items.)

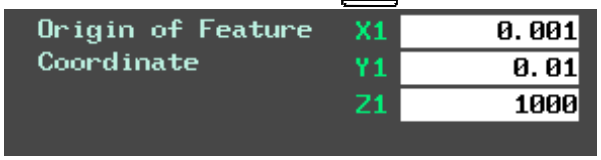
FRYER MACHINE INC Rob Zahn

Command data input

- Item for which to enter a value

  Press cursor key ↑ or ↓ to move the cursor to an item you want to set.

  Enter a value, and then press the INPUT key or horizontal soft key [INPUT].



Example)

When the origin of a feature coordinate system is set as shown above, addresses X, Y, and Z are specified as follows:

G68.2 X0.001 Y0.01 Z1000 ...

For two items "Tool Offset Number" and "From TCP to center", the state with no command can be set by deleting the specified values according to the following procedure.

1 Press cursor key ↑ or ↓ to move the cursor to the item "Tool Offset Number" or "From TCP to Center".

2 Press INPUT or horizontal soft key [INPUT] without entering anything.

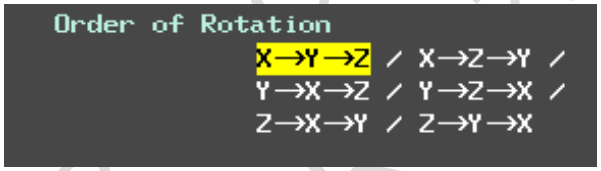
- Item to be selected from a list

  1 Press cursor key ↑ or ↓ to move the cursor to an item you want to set.

  2 Press cursor key ← or → to move the cursor to an item you want to select.

Example)

Order of rotation for Roll-Pitch Yaw angle



Insertion of a block

If the guidance screen is displayed when the block at the cursor position on the program editing screen does not include a tilted working plane indexing, horizontal soft key [INSERT] is displayed on the tilted working plane data setting screen. An operation for inserting a tilted working plane indexing block is described below.

For warnings that may be issued at the time of block insertion, see "Limitation".

1 Press horizontal soft key [INSERT].
The confirmation message "ARE YOU SURE YOU WANT TO EXECUTE?" is displayed, and horizontal soft keys [YES] and [NO] are displayed.

2 Press horizontal soft key [YES].
A block is created from the command type and command data and then is inserted after the block at the cursor position in the program to be edited. After block insertion, the guidance screen is closed and the program editing screen is displayed.
Pressing horizontal soft key [NO] takes you back to the tilted working plane data setting screen.

FRYER MACHINE SYSTEMS INC
Roman Zahorsky

Example)

```
G00 X0.;
   ↓
G00 X0.;
G68.2 P2 Q0...
G68.2 P2 Q1...
G68.2 P2 Q2...
G68.2 P2 Q3...
```

When the guidance screen is displayed and the 3-point specification is selected as a command type for block insertion, a created block is inserted after the block at the cursor position.

Replacement of a block

If the guidance screen is displayed when the block at the cursor position on the program editing screen includes a tilted working plane indexing, horizontal soft key [ALTER] is displayed on the tilted working plane data setting screen. An operation for replacing a tilted working plane indexing block is described below.
For warnings that may be issued at the time of block replacement, see "Limitation".

1 Press horizontal soft key [ALTER].
The confirmation message "ARE YOU SURE YOU WANT TO EXECUTE?" is displayed, and horizontal soft keys [YES] and [NO] are displayed.
2 Press horizontal soft key [YES].
A block is created from the command type and command data and then replaces the tilted working plane indexing in the block at the cursor position on the program editing screen. After block replacement, the guidance screen is closed and the program editing screen is displayed.
Pressing horizontal soft key [NO] takes you back to the tilted working plane data setting screen.

NOTE
1 If a block to be replaced includes a command other than a tilted working plane indexing, the command is deleted during block replacement. However, only the sequence number at the beginning is preserved.
2 If the guidance screen is displayed when the cursor is placed in the middle of multiple blocks for a command, the block(s) before the cursor are not replaced. These blocks remain unchanged after replacement.
3 If horizontal soft key [GUIDANCE TWP] is pressed when the cursor is placed on a block of a tool axis direction control command, the command type selection screen is displayed in new insertion mode. A block created on the guidance screens is inserted after the block of the tool axis direction control command.

Limitation

The following lists the warnings that may be issued at the time of block insertion or replacement.
If a warning is displayed, return to the program editing screen with horizontal soft key [CANCEL] and press horizontal soft key [GUIDANCE TWP] again, or eliminate the cause of the warning and retry the operation.

Warning	Description
“PROGRAM WRITE FAILED”	• The guidance screen was displayed from the foreground editing screen or MDI editing screen, and a block insertion or replacement operation was performed when the CNC was in the reset state or emergency stop state.
“PROGRAM CANNOT BE WRITTEN”	• A block insertion or replacement operation was performed after a program to be edited was updated due to program downloading by an external application while the guidance screen was displayed. • A block insertion or replacement operation was performed after the start of the main program was located by a reset while the guidance screen was displayed.

FRYER MACHINE SYSTEMS INC
Rob Zakhensky

Warning	Description
“WRITE PROTECT”	• A block insertion or replacement operation was performed when the editing or display was prohibited for a program to be edited. • A block insertion or replacement operation was performed when the memory protection signal (KEY signal) for program registration or editing was off. • A block insertion or replacement operation was performed when the operation level of the 8-level data protection function was lower than the protection level of a part program editing operation.

Display of the command type selection screen

When horizontal soft key [COMMAND TYPE] is pressed, the command type selection screen is displayed. When the command type is changed on the command type selection screen, the values set on the tilted working plane data setting screen are cleared.

When a tilted working plane indexing is included in the block at the cursor position on the program editing screen, you can display the guidance screen, change the command type on the command type selection screen, and then perform block replacement. In this case, the block at the cursor position on the program editing screen is replaced with a block created from the command type and command data that have been set.

12.2.18.3 Details of the tilted working plane data setting screen

The following six tilted working plane indexing are supported.

For details of the commands, see II-22.3, "TILTED WORKING PLANE INDEXING".

- G68.2 / G68.4 (Euler’s angle)
- G68.2 / G68.4 P1 (Roll-Pitch-Yaw angle)
- G68.2 / G68.4 P2 (3 points specification)
- G68.2 / G68.4 P3 (2 vectors specification)
- G68.2 / G68.4 P4 (Projection angle)
- G68.3 (Tool Axis Direction)

G68.2 is an absolute command, and G68.4 is an incremental command.

G68.2 / G68.4(Euler’s angle)

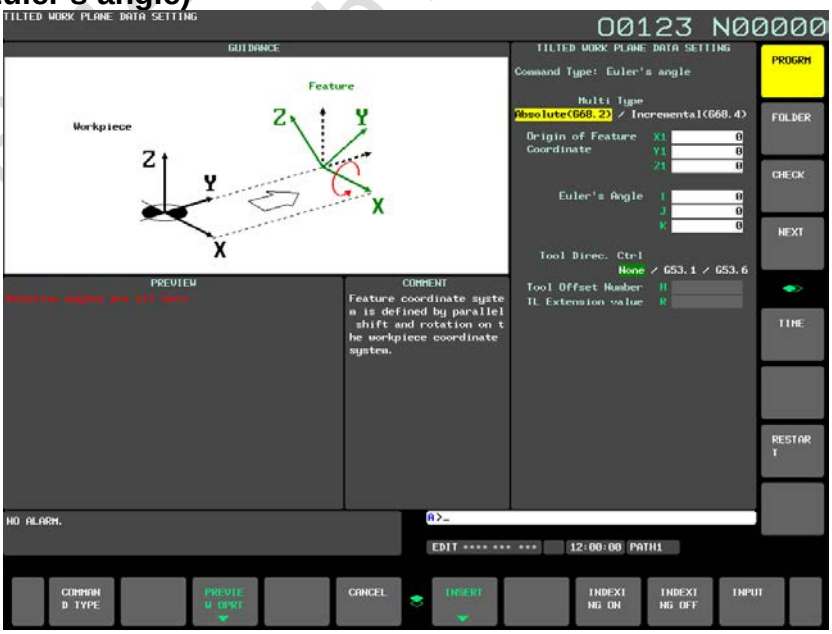


Fig. 12.2.18.3 (a) Tilted working plane data setting screen-Euler’s angle(15-inch display unit)

FRYER MACHINE SYSTEMS INC
Zahensky

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Euler's angle
  I : Specify an angle of rotation around the Z-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).
  This rotation determines coordinate system 1 (X1-Y1-Z1) from the workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type) (X-Y-Z).
  J : Specify an angle of rotation around the X-axis of coordinate system 1.
  This rotation determines coordinate system 2 (X2-Y2-Z2) from coordinate system 1 (X1-Y1-Z1).
  K : Specify an angle of rotation around the Z-axis of coordinate system 2.
  After this rotation, a feature coordinate system is obtained by shifting the origin of the workpiece coordinate system by the coordinates specified in "Origin of Feature Coordinate".

G68.2 / G68.4(Roll-Pitch-Yaw angle)

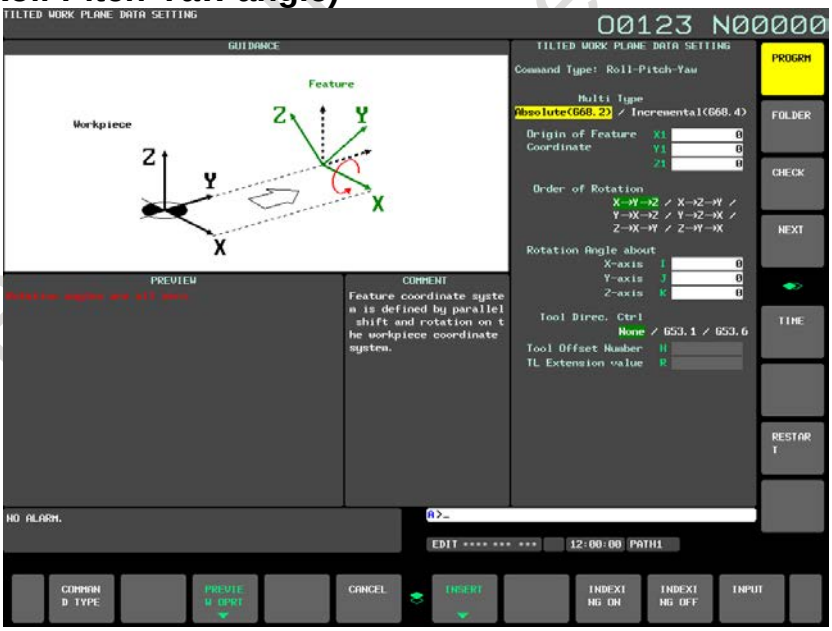


Fig. 12.2.18.3 (b) Tilted working plane data setting screen- Roll-Pitch-Yaw angle(15-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

FRYER MINE SYSTEMS INC
ensky

- Order of Rotation
  Select an order in which the X-axis, Y-axis, and Z-axis are rotated in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). The selectable rotation orders are as follows:

	1st rotation axis	2nd rotation axis	3rd rotation axis
X→Y→Z	X-axis	Y-axis	Z-axis
X→Z→Y	X-axis	Z-axis	Y-axis
Y→X→Z	Y-axis	X-axis	Z-axis
Y→Z→X	Y-axis	Z-axis	X-axis
Z→X→Y	Z-axis	X-axis	Y-axis
Z→Y→X	Z-axis	Y-axis	X-axis

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about X-axis
  Specify an angle of rotation around the X-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about Y-axis
  Specify an angle of rotation around the Y-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about Z-axis
  Specify an angle of rotation around the Z-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

G68.2 / G68.4(3 points specification)

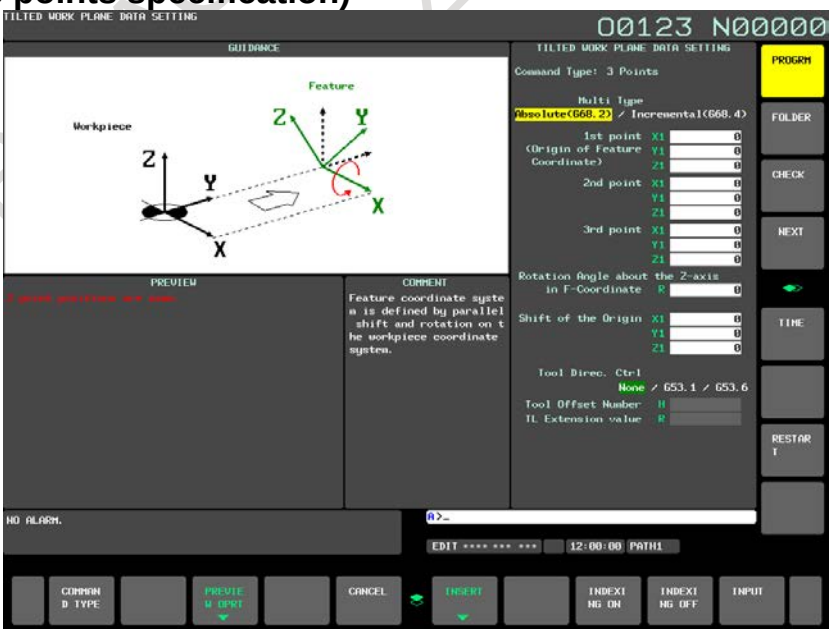


Fig. 12.2.18.3 (c) Tilted working plane data setting screen- 3 points specification(15-inch display unit)

FRYER MACHINE SYSTEMS INC
Zaharensky

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Shift of the Origin
  Specify, in a feature coordinate system, an amount of shift from the feature coordinate system origin specified for the 1st point (point P1).

- Rotation Angle about the Z-axis in F-Coordinate
  Specify an angle of rotation around the Z-axis of a feature coordinate system. The direction of rotation angle R is positive when a rotation is made clockwise as viewed in the Z-axis direction of the feature coordinate system.

- 1st point (Origin of Feature Coordinate)
  Specify the origin (X, Y, and Z of point P1) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- 2nd point
  Specify the coordinates (X, Y, and Z of point P2) of the 2nd point as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). The 1st point and 2nd point determine the X-axis of the feature coordinate system.

- 3rd point
  Specify the coordinates (X, Y, and Z of point P3) of the 3rd point as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). Among the directions normal to the X-axis, a direction with a smaller angle relative to the P1 → P3 vector is the Y-axis of the feature coordinate system.

G68.2 / G68.4(2 vectors specification)

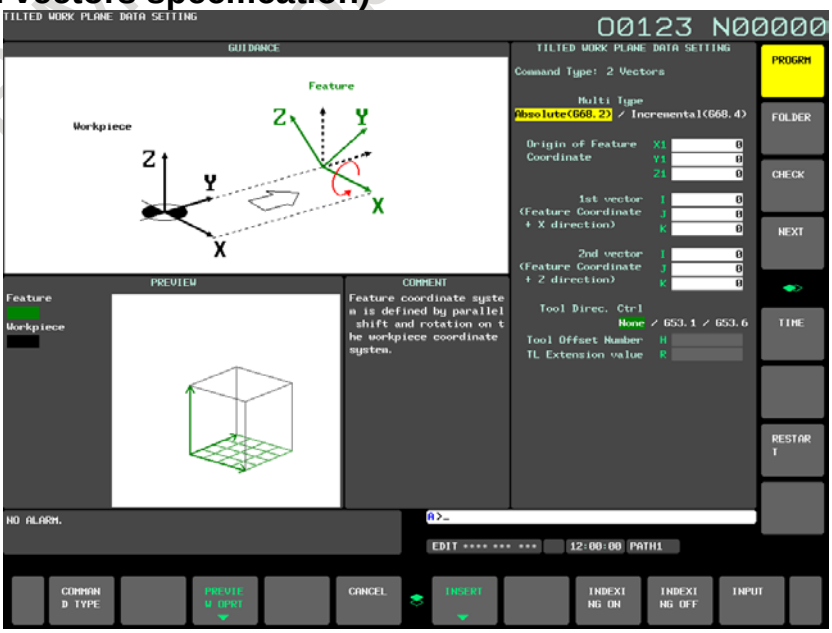


Fig. 12.2.18.3 (d) Tilted working plane data setting screen-2 vectors specification(15-inch display unit)

FRYER MACHINE SYSTEMS INC
Lab Zahensky

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- V1 (X’) vector
  Specify the X-axis direction vector of a feature coordinate system as values in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- V2 (Z’) vector
  Specify the Z-axis direction vector of a feature coordinate system as values in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

G68.2 / G68.4(Projection angle)

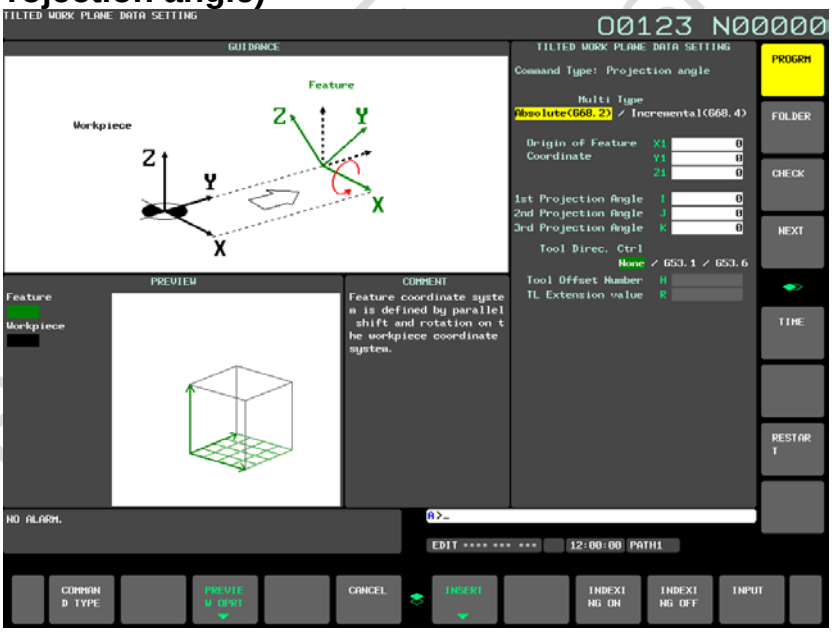


Fig. 12.2.18.3 (e) Tilted working plane data setting screen-Projection angle(15-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

FRYER SYSTEMS INC
sky

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Projection angle
  I : Specify a projection angle relative to the X-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).
  J : Specify a projection angle relative to the Y-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).
  K : Specify an angle of rotation around the Z-axis of a feature coordinate system.

G68.3(Tool Axis Direction)

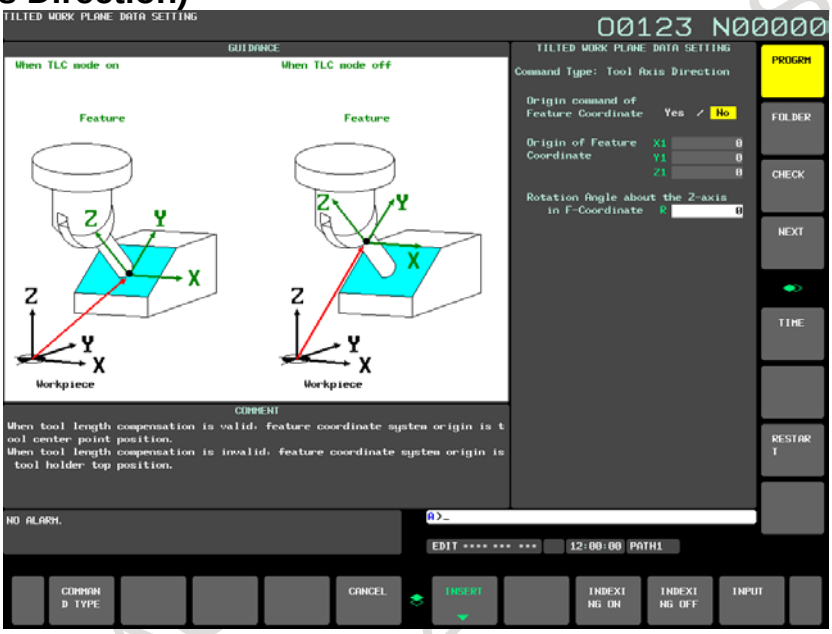


Fig. 12.2.18.3 (f) Tilted working plane data setting screen-Tool Axis Direction(15-inch display unit)
(When "No" is selected in "Origin command of Feature Coordinate")

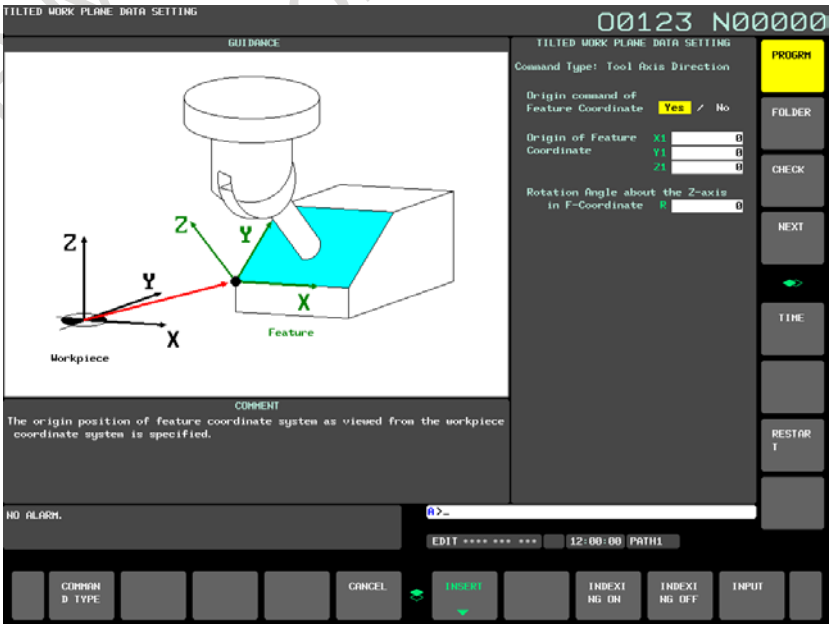


Fig. 12.2.18.3 (g) Tilted working plane data setting screen-Tool Axis Direction(15-inch display unit)
(When "Yes" is selected in "Origin command of Feature Coordinate")

- Origin command of Feature Coordinate
  Select whether to specify a feature coordinate system origin.
  Yes : A feature coordinate system origin is specified.
  No : A feature coordinate system origin is not specified.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system.
  This setting cannot be made if "No" is selected in "Origin command of Feature Coordinate".

- Rotation Angle about the Z-axis in F-Coordinate
  Specify an angle of rotation around the Z-axis of a feature coordinate system. The direction of rotation angle R is positive when a rotation is made clockwise as viewed in the Z-axis direction of the feature coordinate system.

12.2.18.4 Limitation

This function is supported on 10.4-inch, 15-inch, and 19-inch display units.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3 SCREENS DISPLAYED BY FUNCTION KEY

Screens of a 10.4-inch display unit

Press function key to display or set tool compensation values and other data.

This section describes how to display or set the following data:

1. Tool compensation value
2. Settings
3. Sequence number comparison and stop
4. Run time and part count
5. Workpiece origin offset value
6. Custom macro common variables
7. Software operator's panel
8. Tool management data
9. Display language switching
10. Protection of data at eight levels
11. Precision level selection
12. Tool life management data
13. Pattern data input

The software operator's panel, display language switching, precision level selection, and pattern data input depend on the specifications of the machine tool builder. See the manual issued by the machine tool builder for details.

12.3.1 Displaying and Entering Setting Data

Data such as the TV check flag and punch code is set on the setting data screen. On this screen, the operator can also enable/disable parameter writing, enable/disable the automatic insertion of sequence numbers in program editing, and perform settings for the sequence number comparison and stop function.
See III-9.2 for automatic insertion of sequence numbers.
See III-12.3.2 for the sequence number comparison and stop function. This subsection describes how to set data.

Procedure for setting the setting data

Procedure

1 Select the MDI mode.

2 Press function key .

3 Press soft key [SETTING] to display the setting data screen.
This screen consists of several pages.
Press page key or until the desired screen is displayed.
An example of the setting data screen is shown below.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Fig. 12.3.1 (a) SETTING (HANDY) screen (10.4-inch display unit)

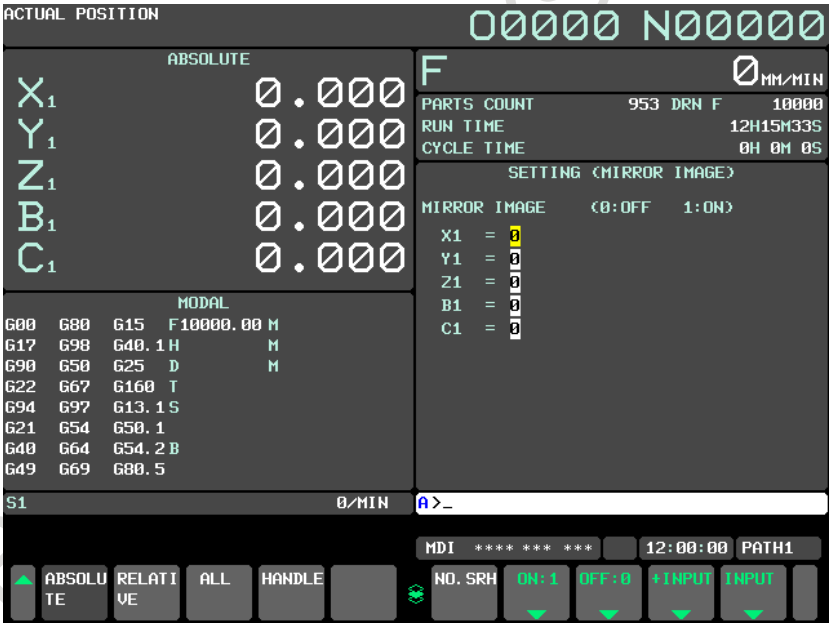
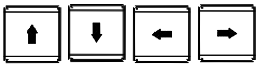


Fig. 12.3.1 (b) SETTING (MIRROR IMAGE) screen (10.4-inch display unit)

4 Move the cursor to the item to be changed by pressing cursor keys  .

5 Enter a new value and press soft key [INPUT].

Explanation

- PARAMETER WRITE

Setting whether parameter writing is enabled or disabled.
0 : Disabled
1 : Enabled

- TV CHECK

Setting to perform TV check.
0 : No TV check
1 : Perform TV check

- OUTPUT CODE

Setting code when data is output through RS-232C interface.
0 : EIA code output
1 : ISO code output

- INPUT UNIT

Setting a program input unit, inch or metric system
0 : Metric
1 : Inch

- I/O CHANNEL

Using channel of RS-232C interface.
0 : Channel 0
1 : Channel 1
2 : Channel 2

- SEQUENCE NO.

Setting of whether to perform automatic insertion of the sequence number or not at program edit in the EDIT mode.
0 : Does not perform automatic sequence number insertion.
1 : Perform automatic sequence number insertion.

- PROGRAM FORMAT

Setting of whether to use the Series 15 format.
0: Uses the standard format.
1: Uses the Series 15 format.
For the Series 15 format, refer to Chapter, “MEMORY OPERATION USING Series 15 FORMAT” in the OPERATOR’S MANUAL (T series : B-64694EN-1) or Chapter, “MEMORY OPERATION USING Series 15 FORMAT” in the OPERATOR’S MANUAL (M series : B-64694EN-2).

- SEQUENCE STOP

Setting the sequence number with which the operation stops for the sequence number comparison and stop function and the number of the program to which the sequence number belongs

T

- CCR / DDDP

This setting item is displayed only when the chamfering or corner R and the direct drawing dimension programming are enabled. These functions cannot be use at same time. Setting of which function is enabled.
0:Chamfering or corner R is enabled.
1:Direct drawing dimensions programming function is enabled.

- MIRROR IMAGE

Setting of mirror image ON/OFF for each axes.
0 : Mirror image off
1 : Mirror image on

- Others

Page key [PAGE ⇧] or [PAGE ⇩] can also be pressed to display the SETTING (TIMER) screen. See III-12.3.3 for this screen.

FIXER MACHINE SYSTEMS INC
Rob Zahonsky

12.3.2 Sequence Number Comparison and Stop

If a block containing a specified sequence number appears in the program being executed, operation enters single block mode after the block is executed.

Procedure for sequence number comparison and stop

Procedure

1 Select the MDI mode.

2 Press function key [OFS/SET].

3 Press chapter selection soft key [SETTING].

4 Press page key [PAGE ⇧] or [PAGE ⇩] several times until the following screen is displayed.

Fig. 12.3.2 (a) SETTING (HANDY) screen (10.4-inch display unit)

5 Enter in (PROGRAM NO.) for SEQUENCE STOP the number (1 to 99999999) of the program containing the sequence number with which operation stops.

6 Enter in (SEQUENCE NO.) for SEQUENCE STOP (with eight or less digits) the sequence number with which operation is stopped.

7 When automatic operation is executed, operation enters single block mode at the block containing the sequence number which has been set.

Explanation

- Sequence number after the program is executed

After the specified sequence number is found during the execution of the program, the sequence number setting for sequence number compensation and stop becomes “-1”.

- Exceptional blocks

If the predetermined sequence number is found in a block in which all commands are those to be processed within the CNC control unit, the execution does not stop at that block.

[Example]
```
N1 #1=1 ;
N2 IF[#1 EQ 1]GOTO 08 ;
N3 GOTO 09 ;
N4 M98 P1000 ;
N5 M99 ;
```

FRYER MACHINE TOOLS INC

In the example shown above, if the predetermined sequence number is found, the execution of the program does not stop.

- Stop in the canned cycle

If the predetermined sequence number is found in a block which has a canned cycle command, the execution of the program stops after the return operation is completed.

- When the same sequence number is found several times in the program

If the predetermined sequence number appears twice or more in a program, the execution of the program stops after the block in which the predetermined sequence number is found for the first time is executed.

- Block to be repeated a specified number of times

If the predetermined sequence number is found in a block which is to be executed repeatedly, the execution of the program stops after the block is executed specified times.

12.3.3 Displaying and Setting Run Time, Parts Count, and Time

Various run times, the total number of machined parts, number of parts required, and number of machined parts can be displayed.
This data can be set by parameters or on this screen (except for the total number of machined parts and the time during which the power is on, which can be set only by parameters).
This screen can also display the clock time. The time can be set on the screen.

Procedure for Displaying and Setting Run Time, Parts Count and Time

Procedure

1 Select the MDI mode.

2 Press function key [OFS/SET].

3 Press chapter selection soft key [SETTING].

4 Press page key [PAGE ↑] or [PAGE ↓] several times until the following screen is displayed.

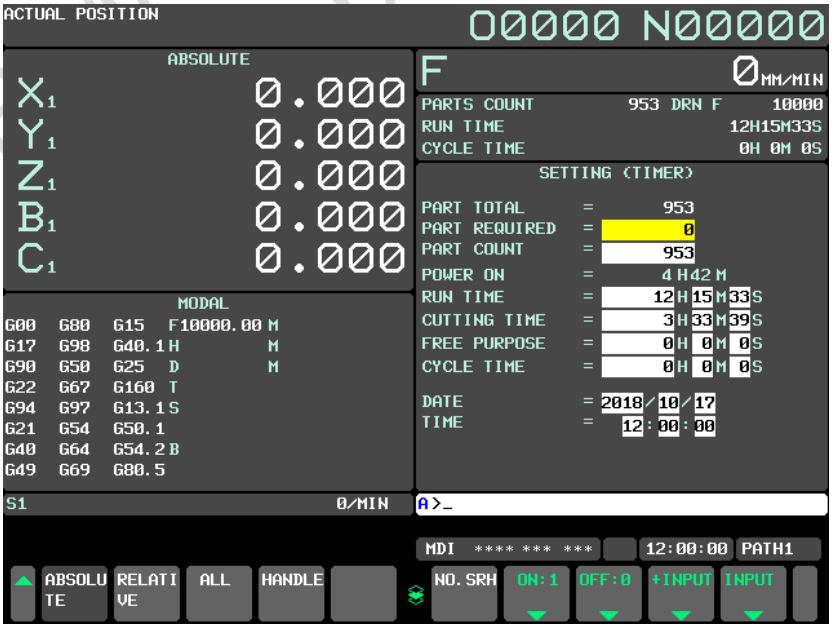


Fig. 12.3.3 (a) SETTING (TIMER) screen (10.4-inch display unit)

FRYER MACHINE SYSTEMS INC
Jacob Zahensky

5 To set the number of parts required, move the cursor to PARTS REQUIRED and enter the number of parts to be machined.
6 To set the clock, move the cursor to DATE or TIME, enter a new date or time, then press soft key [INPUT].

Explanation

- PARTS TOTAL

This value is incremented by one when M02, M30, or an M code specified by parameter No. 6710 is executed. This value cannot be set on this screen. Set the value in parameter No. 6712.

- PARTS REQUIRED

It is used for setting the number of machined parts required.
When the “0” is set to it, there is no limitation to the number of parts.
Also, its setting can be made by the parameter No 6713.

- PARTS COUNT

This value is incremented by one when M02, M30, or an M code specified by parameter No. 6710 is executed. The value can also be set by parameter No. 6711. In general, this value is reset when it reaches the number of parts required. Refer to the manual issued by the machine tool builder for details.

- POWER ON

Displays the total time which the power is on. This value cannot be set on this screen but can be preset in parameter No. 6750.

- RUN TIME

Indicates the total run time during automatic operation, excluding the stop and feed hold time.
This value can be preset in parameter No. 6751 or No. 6752.

- CUTTING TIME

Displays the total time taken by cutting that involves cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03). This value can be preset in parameter No. 6753 or No. 6754.

- FREE PURPOSE

This value can be used, for example, as the total time during which coolant flows. Refer to the manual issued by the machine tool builder for details.

- CYCLE TIME

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- DATA and TIME

Displays the current date and time. The date and time can be set on this screen.

- Usage

When the command of M02 or M30 is executed, the total number of machined parts and the number of machined parts are incremented by one. Therefore, create the program so that M02 or M30 is executed every time the processing of one part is completed. Furthermore, if an M code set to the parameter No. 6710 is executed, counting is made in the similar manner. Also, it is possible to disable counting even if M02 or M30 is executed (bit 0 (PCM) of parameter No. 6700 is set to 1). For details, see the manual issued by machine tool builders.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Limitation

- Run time and part count settings

Negative value cannot be set. Also, the setting of “M” and “S” of run time is valid from 0 to 59.
Negative value may not be set to the total number of machined parts.

- Time settings

Neither negative value nor the value exceeding the value in the following table can be set.

Table 12.3.3

Item	Maximum value	Item	Maximum value
Year	2096	Hour	23
Month	12	Minute	59
Day	31	Second	59

12.3.4 Displaying and Setting the Workpiece Origin Offset Value

Displays the workpiece origin offset for each workpiece coordinate system (G54 to G59, G54.1 P1 to G54.1 P48 and G54.1 P1 to G54.1 P300) and external workpiece origin offset.
The workpiece origin offset value and external workpiece origin offset value can be set on this screen.

Procedure for displaying and setting the workpiece origin offset value

Procedure

1 Press function key .

2 Press chapter selection soft key [WORK].
The workpiece coordinate system setting screen is displayed.

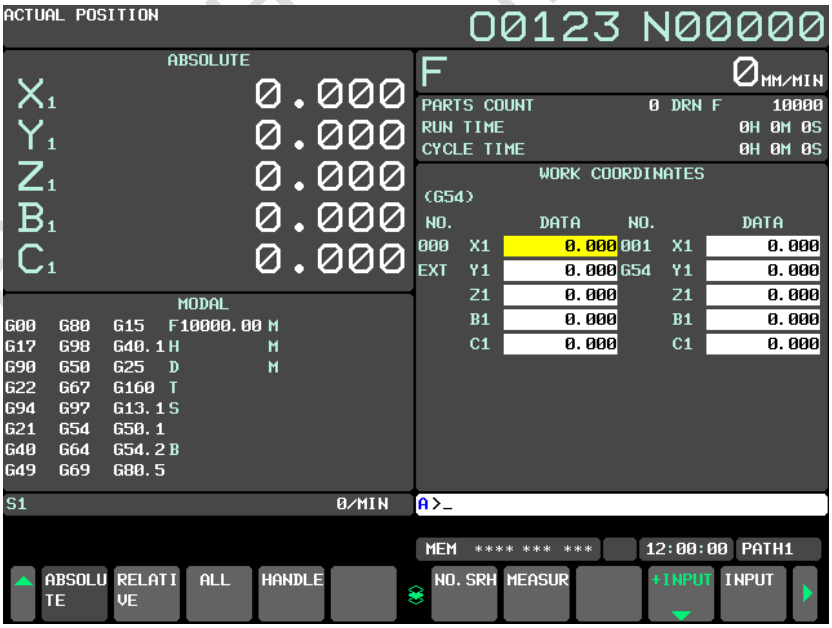


Fig. 12.3.4 (a) WORK COORDINATES screen (10.4-inch display unit)

FRYER MACHINE SYSTEMS INC
hensky

3 The screen for displaying the workpiece origin offset values consists of two or more pages. Display a desired page in either of the following two ways:
   - Press the page key [PAGE ⇧] or [PAGE ⇩].
   - Enter the workpiece coordinate system number (0 : external workpiece origin offset, 1 to 6: workpiece coordinate systems G54 to G59, P1 to P48 : workpiece coordinate systems G54.1 P1 to G54.1 P48, P1 to P300 : workpiece coordinate systems G54.1 P1 to G54.1 P300) and press operation selection soft key [NO.SRH].
4 Turn off the data protection key to enable writing.
5 Move the cursor to the workpiece origin offset to be changed.
6 Enter a desired value by pressing numeric keys, then press soft key [INPUT]. The entered value is specified in the workpiece origin offset value. Or, by entering a desired value with numeric keys and pressing soft key [+INPUT], the entered value can be added to the previous offset value.
7 Repeat steps 5 and 6 to change other offset values.
8 Turn on the data protection key to disable writing.

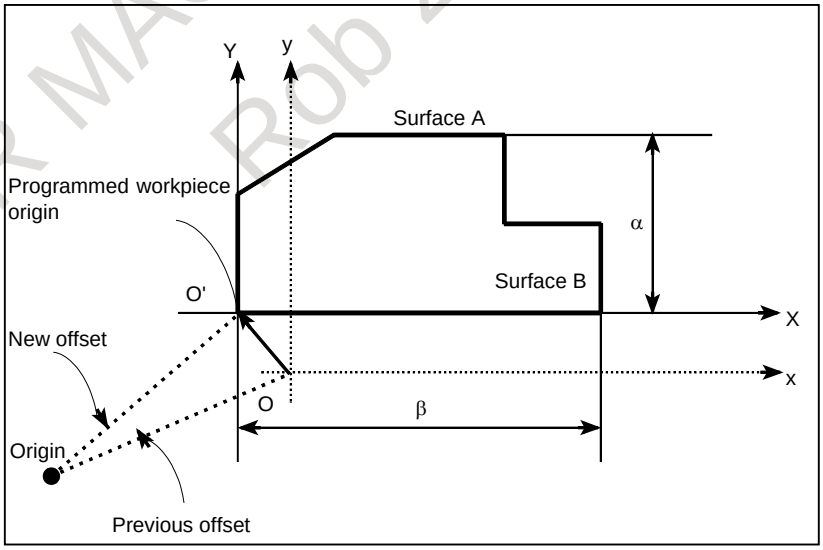
NOTE
When the axis name set in parameter No.1020 is not valid, the workpiece origin offset value of the axis should be set to 0.

12.3.5 Direct Input of Workpiece Origin Offset Value Measured

This function is used to compensate for the difference between the programmed workpiece coordinate system and the actual workpiece coordinate system. The measured offset for the origin of the workpiece coordinate system can be input on the screen so that the command values match the actual dimensions.
Selecting the new coordinate system matches the programmed coordinate system with the actual coordinate system.

Procedure for direct input of workpiece origin offset value measured

Procedure



FRYER MACHINE SYSTEMS INC
Rob Zaherisky

1 When the workpiece is shaped as shown above, position the reference tool manually until it touches surface A of the workpiece.
2 Retract the tool without changing the Y coordinate.
3 Measure distance α between surface A and the programmed origin of the workpiece coordinate system as shown above.
4 Press function key .
5 To display the WORK COORDINATES screen, press the chapter selection soft key [WORK].

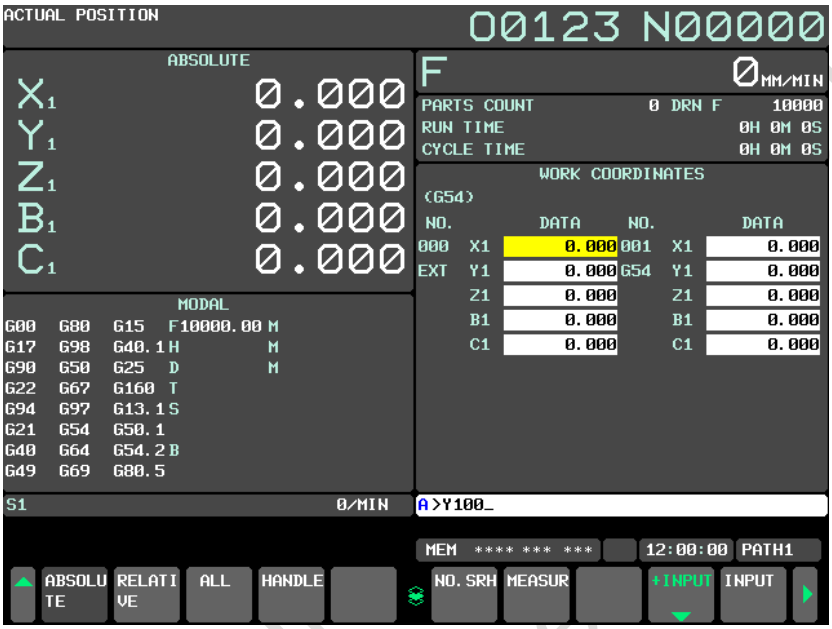


Fig. 12.3.5 (a) WORK COORDINATES screen (10.4-inch display unit)

6 Position the cursor to the workpiece origin offset value to be set.
7 Press the address key for the axis along which the offset is to be set (Y-axis in this example).
8 Enter the measured value (α) then press soft key [MEASUR].
9 Move the reference tool manually until it touches surface B of the workpiece.
10 Retract the tool without changing the X coordinate.
11 Enter the measured value of distance (β) to surface B as an X value as with steps 7 and 8.

Limitation

- Consecutive input

Offsets for two or more axes cannot be input at the same time.

- During program execution

This function cannot be used while a program is being executed.

FRYER MACHINE INC
Rondzahn

12.3.6 Displaying and Setting Custom Macro Common Variables

Displays common variables (#100 to #149 or #100 to #199, and #500 to #531 or #500 to #999) on the screen.
The values for variables can be set on this screen.
Relative coordinates can also be set to variables.

Procedure for displaying and setting custom macro common variables

Procedure

1 Press function key [OFS/SET].

2 Press the continuous menu key [▷], chapter selection soft key [MACRO], and soft key [(OPRT)] in the stated order.
The following screen is displayed.

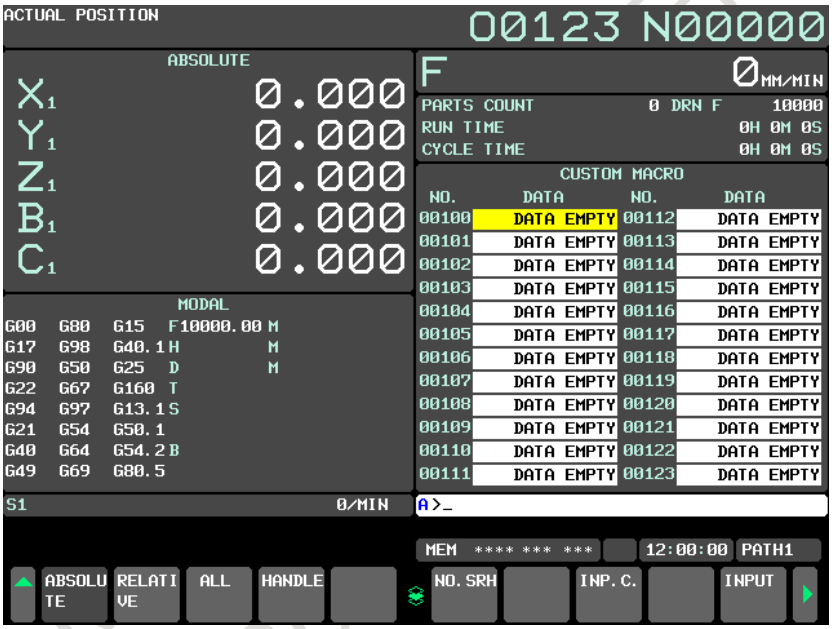


Fig. 12.3.6 (a) CUSTOM MACRO screen (10.4-inch display unit)

3 Move the cursor to the variable number to set using either of the following methods:
- Enter the variable number and press soft key [NO.SRH].
- Move the cursor to the variable number to set by pressing page keys [PAGE ⇧] and/or [PAGE ⇩] and cursor keys [⬆], [⬇], [⬅], and/or [➡].

4 Enter data with numeric keys and press soft key [INPUT].

5 To set a relative coordinate in a variable, press address key [X], [Y], or [Z], then press soft key [INP.C.].

6 To set a blank in a variable, just press soft key [INPUT]. The value field for the variable becomes blank.

FRYER SYSTEMS INC

Explanation

If the value of a variable produced by an operation is not displayable, an indication below is provided.
When the significant number of digits is 12 (with bit 0 (F16) of parameter No. 6008 set to 0):

Variable value range	Variable value indication
0 < Variable value < +0.00000000001	+Underflow
0 > Variable value > -0.00000000001	-Underflow
Variable value > 999999999999	+Overflow
Variable value < -999999999999	-Overflow

When the significant number of digits is 8 (with bit 0 (F16) of parameter No. 6008 set to 1):

Variable value range	Variable value indication
0 < Variable value < +0.0000001	+Underflow
0 > Variable value > -0.0000001	-Underflow
Variable value > 99999999	+Overflow
Variable value < -99999999	-Overflow

12.3.7 Displaying and Setting Real Time Custom Macro Data

Real time macro variables (RTM variables) are dedicated to real time custom macros.
RTM variables are divided into temporary real time macro variables (temporary RTM variables) and permanent real time macro variables (permanent RTM variables).
The values of permanent RTM variables are kept stored after the power is turned off.
Temporary RTM variables are cleared to 0 when the power is turned off.
System variables (DI/DO variables) dedicated to real time custom macros are used to read and write PMC interface signals. Data can be read and written in bit and byte units. Before writing a signal, release the protection of the corresponding variable on the PMC signal protect screen.

Displaying and setting real time custom macro variables

Procedure

1 Press function key SYSTEM.

2 Press the continuous menu key ▷ several times, then press chapter selection soft key [R.TIME MACRO]. The following screen appears:

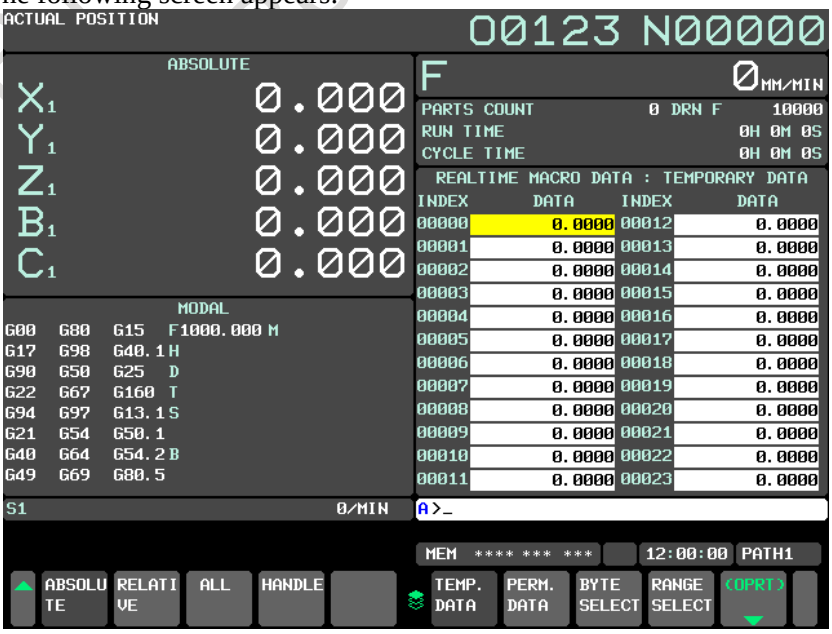


Fig. 12.3.7 (a) REAL TIME MACRO screen (10.4-inch display unit)

FRYER MACHINE SYSTEMS INC
Bob Zahensky

3 To display or set real time custom macro variables whose values are not stored at power-off, press soft key [TEMP. DATA].

4 To display or set real time custom macro variables whose values are stored at power-off, press soft key [PERM. DATA].

5 Move the cursor to the number of a real time custom macro variable you want to set using either of the following methods:

- Enter the number of a real time custom macro variable and press soft key [NO. SRH].
- Move the cursor to the number of a real time custom macro variable you want to set by pressing page keys [PAGE ⇧] and/or [PAGE ⇩] and cursor keys [↑], [↓], [←], and/or [→].

6 Enter data.

Displaying and setting DI/DO variables

Procedure

For setting in byte units:

1 Press function key [SYSTEM].

2 Press the continuous menu key [▷] several times, then press chapter selection soft key [R.TIME MACRO]. The following screen appears:

3 Press soft key [BYTE SELECT]. The following screen appears:

Fig. 12.3.7 (b) BYTE SELECT screen (10.4-inch display unit)

4 Move the cursor to the number of a DI/DO variable you want to set using either of the following methods:

- Enter the number and press soft key [NO. SRH].
- Move the cursor to a desired number by pressing page keys [PAGE ⇧] and/or [PAGE ⇩] and cursor keys [↑], [↓], [←], and/or [→].

5 Enter data.

FRYER SYSTEMS INC

12.3.8 Displaying and Setting the Software Operator's Panel

Operations on the MDI unit can substitute for the functions of switches on the machine operator's panel. This means that a mode selection, jog feed override selection, and so forth can be made with operations on the MDI unit, eliminating the need to use the corresponding switches on the machine operator's panel. Jog feed can also be performed using numeric keys.

Procedure for displaying and setting the software operator's panel

Procedure

1 Press function key [OPS SET].

2 Press the continuous menu key [▷], then press chapter selection soft key [OPR].

3 The screen consists of several pages. Press page key [↑ PAGE] or [PAGE ↓] until the desired screen is displayed.

Fig. 12.3.8 (a) Page 1 of Software Operator's Panel screen (without the manual handle feed function) (10.4-inch display unit)

Fig. 12.3.8 (b) Page 1 of Software Operator's Panel screen (with the manual handle feed function) (10.4-inch display unit)

FRYER MACHINE SYSTEMS INC

Fig. 12.3.8 (c) Page 2 of Software Operator's Panel screen (10.4-inch display unit)

4 Move the cursor to the desired switch by pressing cursor key ↑ or ↓.

5 Press the cursor key ← or → to match the mark ■ to an arbitrary position and set the desired condition.

6 Press one of the following arrow keys to perform jog feed. Press the 5 key together with an arrow key to perform jog rapid traverse.

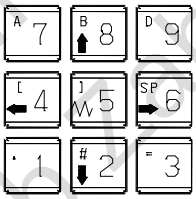


Fig. 12.3.8 (d) MDI arrow keys

Explanation

- Valid operations

The valid operations on the software operator's panel are shown below. Whether to use each group can be chosen using parameter No. 7200. Those groups that are not used are not displayed on the software operator's panel.

Group1 : Mode selection
Group2 : Selection of jog feed axis, Manual rapid traverse
Group3 : Selection of manual pulse generator feed axis, selection of manual pulse magnification
Group4 : Jog feederate, feederate override, rapid traverse override
Group5 : Optional block skip, single block, machine lock, dry run
Group6 : Protect key
Group7 : Feed hold

- Screens on which jog feed is valid

When the LCD indicates other than the software operator's panel screen and self-diagnosis screen, jog feed is not conducted even if the arrow key is pushed. This function cannot be used when the virtual MDI key function is used.

FRYER MACHINE SYSTEMS INC Rob Zanensky

- Jog feed and arrow keys

The feed axis and direction corresponding to the arrow keys can be set with parameters Nos. 7210 to 7217.

- General purpose switches

For the meanings of these switches, refer to the manual issued by machine tool builder.

12.3.9 Setting and Displaying Tool Management Data

The tool management function totally manages tool information including tool offsets and tool life information.
This function provides a magazine screen and tool management screen. This subsection explains how to set these screens.

12.3.9.1 Displaying and setting magazine screen

1 Press function key .

2 Press chapter selection soft key [TOOL MANAGER].
Alternatively, press function key several times until the tool management screen appears.
Press soft key [MAGAZINE].

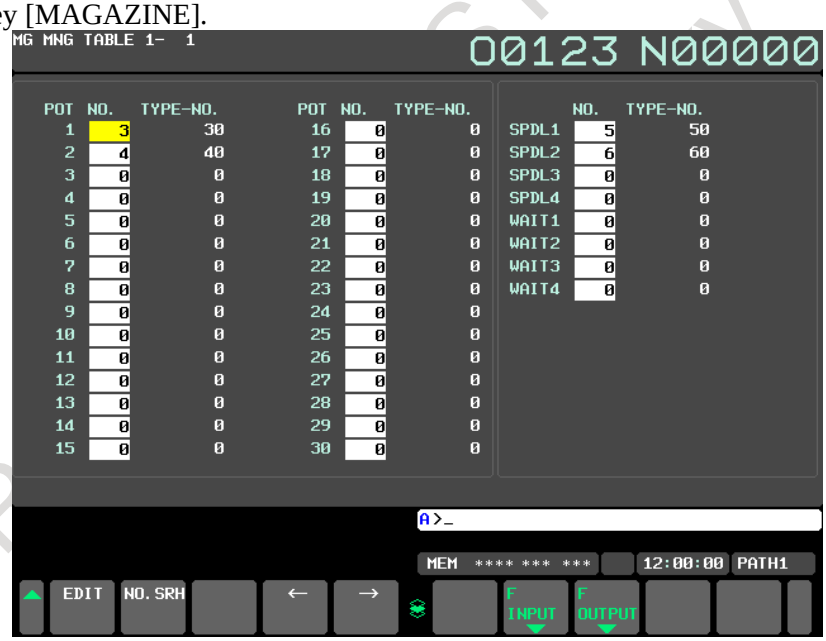


Fig. 12.3.9.1 (a) Tool management magazine screen (10.4-inch display unit)

3 By using the page keys, cursor keys, and soft keys [←] and [→], move the cursor to the pot No. position of the magazine for which you want to set or modify data.
Alternatively, type the number of the tool management data you wan to set or modify, and press soft key [NO. SRH].

4 Press soft key [EDIT].

5 To set the tool management data number of a pot, type the tool management data number, then press soft key [INPUT].
To delete the tool management data number set for a pot, follow the steps below.
<1> Press soft key [ERASE].
<2> Press soft key [CURDAT]. To delete the tool management data numbers registered for all pots, press soft key [ALL].
<3> Press soft key [EXEC]. Alternatively, type 0.

6 To end the edit operation, press soft key [EXIT].
This returns the screen display to the conventional tool management screen.

Explanation

- Another method

Magazine data can be input/output also by using external I/O devices.
See III-8, “DATA I/O”.

- Displayed data

POT : As many pots as specified in parameter No. 13222, 13227, 13232, or 13237 (settable for each magazine) are displayed. The starting pot number can be set for each magazine in parameter No. 13223, 13228, 13233, or 13238.

NO. : Tool management data numbers are displayed.
In the following cases, tools cannot be set in magazines:
1. A tool management data number beyond the range from 0 to (the number of valid pairs in parameter No. 13220) is set.
2. Tool management data is invalid (bit 1 of tool information is 0).
3. The tool management data number to be set is already set for another pot.

TYPE NO. : Tool type numbers corresponding to tool management data numbers are displayed.
Values cannot be set on this screen.

SPDL : The tool management data numbers and tool type numbers of spindle positions are displayed.

WAIT : The tool management data numbers and tool type numbers of wait positions are displayed.

- Edit mode

To edit data, press soft key [EDIT] to enter tool management data edit mode.

- Tool management data number

Tool management data numbers can optionally be extended to 64, 240, or 1000 pairs. In addition, the number of tools used can be set in parameter No. 13220.

- Display of the spindle positions and wait positions

The spindle position and wait position can be set for each path.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.9.2 Displaying and setting tool management screen

Procedure

1 Press function key .

2 Press chapter selection soft key [TOOL MANAGER].

Alternatively, press function key several times until the tool management screen appears.

3 Press soft key [TOOL MANAGER].

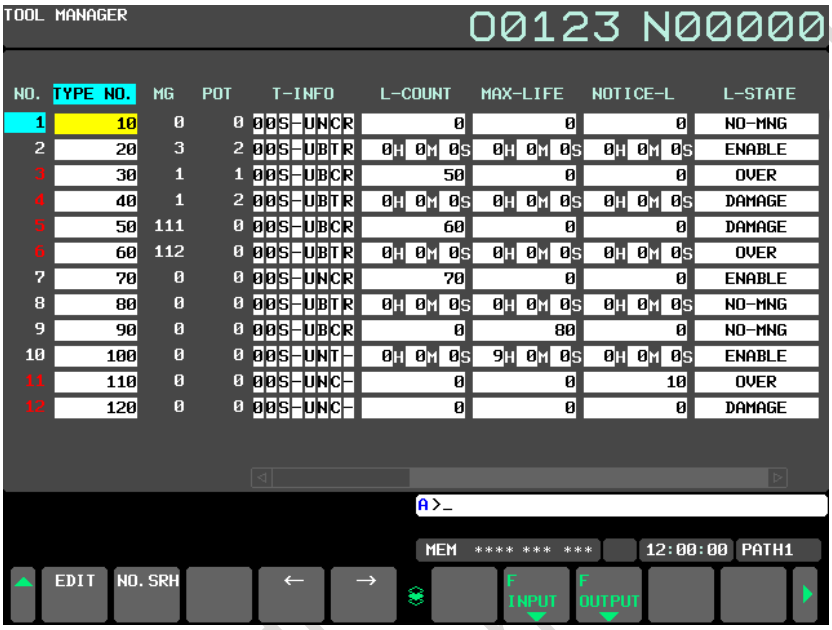


Fig. 12.3.9.2 (a) Tool management data screen (10.4-inch display unit)

4 By using the page keys, cursor keys, and soft keys [←] and [→], move the cursor to the position of the tool information of the tool number for which you want to set or modify data.
Alternatively, type the tool number of the data you want to set or modify, and press soft key [NO. SRH].

5 Press soft key [EDIT].

6 To set tool data, type a desired value, then press soft key [INPUT].
To delete tool information, follow the steps below.
<1> In step 4, move the cursor to the tool information you want to erase.
<2> Press soft key [ERASE].
<3> Press soft key [CURDAT]. To delete entire tool information, press soft key [ALL].
<4> Press soft key [EXEC].

7 To end the edit operation, press soft key [EXIT].
This returns the screen display to the conventional tool management screen.

FRYER MACH. INC Rob Zak

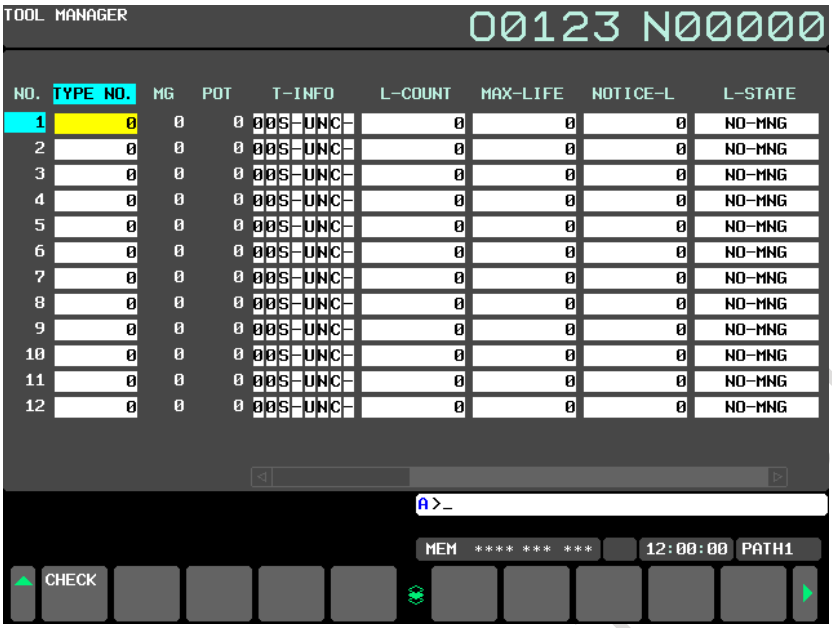


Fig. 12.3.9.2 (b) Tool management data screen (check function) (10.4-inch display unit)

8 When soft key [CHECK] is pressed, if there are tools with the same number but with different count types (count and time), the cursor moves to the tool type number of the smallest tool management number in the tool type numbers and a warning message appears.
The warning messages issued by the check function are shown below.
<1> For the same type of tools with different count types
L-COUNT TYPE MISMATCH:XXXXXXXX
<2> For the same type of tools with the same count type
L-COUNT TYPE MATCH

Explanation

- Another method

Tool management data, customize data, and names set for tool states can be input/output also by using external I/O devices.
See III-8, “DATA I/O”.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Displayed information

- Life information

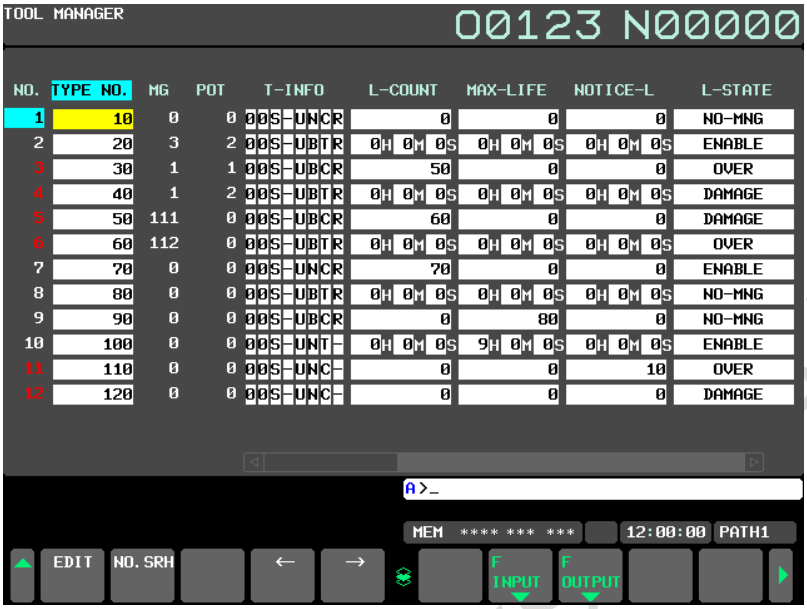


Fig. 12.3.9.2 (c) Tool management data life status screen(10.4-inch display unit)

NO. : Tool management data numbers are displayed. These numbers can be displayed but cannot be set. The tool management data number of edited data is kept blinking until tool management data edit mode ends. Tool management data numbers set or deleted on the magazine screen also blink on the tool management screen.

TYPE NO. : Tool type numbers are displayed.
A value from 0 to 99,999,999 can be set.

MG : The magazine number assigned to each tool is indicated. These numbers can be displayed but cannot be set.

POT : The pot number assigned to each tool is indicated. These numbers can be displayed but cannot be set.

T-INFO : The following five types of information are displayed sequentially from the right:

- Tool management data: Valid (R)/invalid (-))
- Life count type: Time (T)/count (C)
- Tool type: Oversize tool (B)/normal tool (N)
- Data access: Locked (L)/unlocked (U)
- When a tool is not under life management:
  Included in the tools to be searched (1)/not included in the tools to be searched (0)

NOTE

1 The tool types and data access information vary depending on the specifications defined by the machine tool builder.

2 The same type of tools must have the same life count type.

L-COUNT : The number of use times/use period of time of each tool is indicated.
Up to 99,999,999 times or 999 hours 59 minutes 59 seconds can be set.

MAX-LIFE : The maximum life value of each tool is indicated.
Up to 99,999,999 times or 999 hours 59 minutes 59 seconds can be set.

NOTICE-L : Noticed life value of a tool
Up to 99,999,999 times or 999 hours 59 minutes 59 seconds can be set.

L-STATE : Current tool state
One of the four states, including invalid (0), present (1, 2), not present (3), and broken (4), is indicated. The numbers in parentheses are data values used when these states are input in MDI.

FRYER MACHINE SYSTEMS INC
Fred Zahensky

- Spindle speed/feedrate

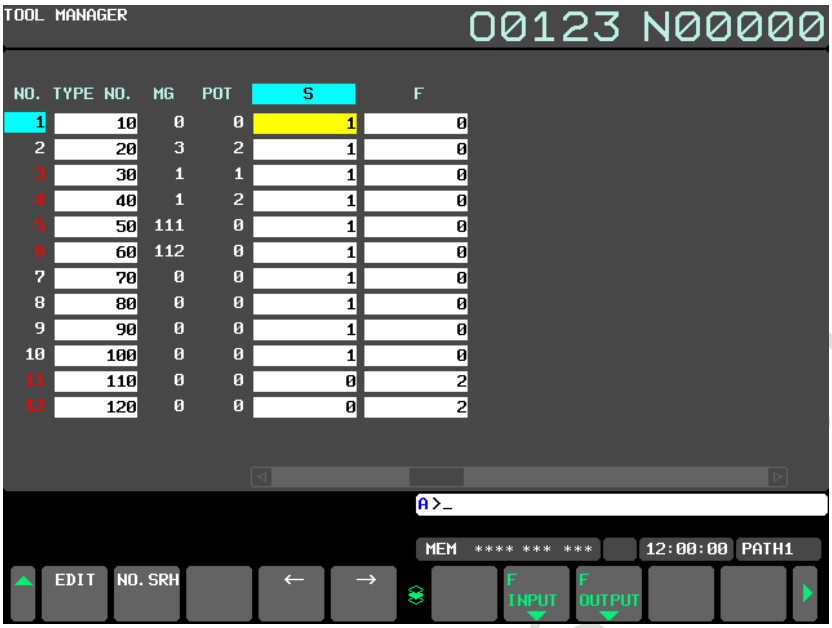


Fig. 12.3.9.2 (d) Tool management data spindle speed/feedrate screen (10.4-inch display unit)

S : Spindle speed.
A value from 0 to 99,999 can be set.

F : Feedrate.
A value from 0 to 99,999,999 can be set.

- Tool offset information

Fig. 12.3.9.2 (e) Tool management data tool offset screen (10.4-inch display unit)

H : Tool length compensation number (for machining center systems only).
A value from 0 to 999 can be set.

D : Cutter compensation number (for machining center systems only).
A value from 0 to 999 can be set.

TG : Tool geometry compensation number (for lathe systems only).
A value from 0 to 999 can be set.

TW : Tool wear compensation number (for lathe systems only).
A value from 0 to 999 can be set.

The displayed offset data is determined depending on the option configuration and parameter settings (bits 1(DCR), 2(DOY), 4(DO2), 6(DOT), and 7(DOM) of parameter No. 13202).

- Customize information

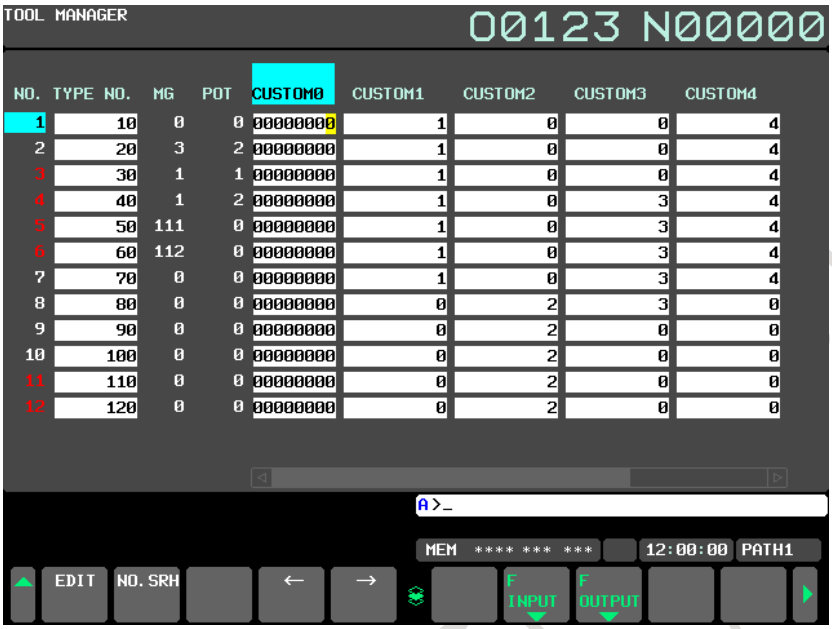


Fig. 12.3.9.2 (f) Tool management data customize data screen (10.4-inch display unit)

CUSTOM0 : Bit-type customize information.
For each bit, 1 or 0 can be input.

CUSTOM1 to 4 : Customize information. Any value from -99,999,999 to 99,999,999 can be set.

CUSTOM5 to 20 : Customize information. These items are displayed only when customize data extension option (5 to 20) of the tool management function is enabled. Any value from -99,999,999 to 99,999,999 can be set.

CUSTOM21 to 40 : Customize information. These items are displayed only when customize data extension option (5 to 40) of the tool management function is enabled. Any value from -99,999,999 to 99,999,999 can be set.

- Tool management data number

Tool management data numbers can optionally be extended to 64, 240, or 1000 pairs. The number of tools used can be set in parameter No. 13220.

- Edit mode

To edit data, press soft key [EDIT] to enter tool management data edit mode.

- Life count override

When the period of time is set as the tool life count type, the life count can be overridden if bit 2 (LFV) of parameter No. 6801 is set to 1. Specify an override value by using a switch on the machine operator's panel.

Example)

When cutting is performed for 10 minutes with an override of 0.1, one minute is counted in the tool life counter.

- Tool management extension function

When tool management extension functions are enabled, you can use the following functions in addition to the tool management functions:

- A value with a decimal point can be set as customize data.
  The maximum number of decimal places can be set to 7 by G10 data input and file input.

FRYER MACHINE S INC Rob Zahensky

- Various types of tool management data can be protected using KEY signals.
  An attempt to enter a value to a protected data causes a warning message “WRITE PROTECT.”
- Tool life count period selection
  Bit 5 of tool information is used to make a life count period selection.

Item		Description
Data length		1 byte (flag data)
#5	REV	0: A life count period of 1 sec is used. 1: A life count period of 8 msec is used.

Range of count is as follows.

1sec : 0 to 3,599,999 sec (999 hours 59 minutes 59 seconds)
8msec : 0 to 3,599,992 ms (59 minutes 59 seconds 992 milliseconds)

12.3.9.3 Each tool data screen

Each tool data screen

Procedure

1 Press function key .

2 Press chapter selection soft key [TOOL MANAGER].
Alternatively, press function key several times until the tool management screen appears.

3 Press soft key [EACH TOOL]. The each tool data screen appears.

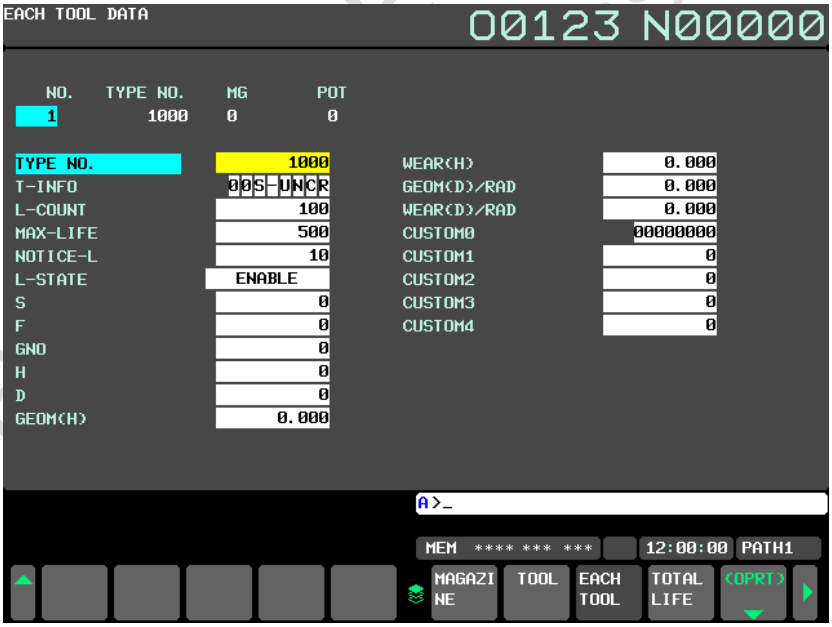


Fig. 12.3.9.3 (a) Each tool data screen (10.4-inch display unit)

FRYER SYSTEMS INC

Explanation

- Header

The following four data items are displayed: NO., TYPE NO., MG, and POT.
When the data table of a tool extends over two or more pages, the same header is displayed on these pages.

- Data table

The data table shows data items related to a tool at a time.
The data items are displayed from the upper left to the lower left, the upper right, and the lower right in ascending order of screen display position numbers specified using the customize function. The number of digits displayed for one data item is fixed to 11.
Up to 24 data items are displayed on one page. (12 data items × 2 columns)
When more than 24 data items are set for a tool, the 25th and subsequent data items are displayed on the next page. (Up to three pages)
When a data item is set as a screen element of the tool management data screen twice or more using the tool management data display customize function (one of the tool management extension functions), only the data item with the smaller display position number is displayed. When no value is set for a data item, the field for the data item is not displayed and the next data item is displayed.

Key operations

- MDI key operations

Numeral keys Inputs a numeric value.

PAGE ⇧ Displays the previous page for the same tool.

PAGE ⇩ Displays the next page for the same tool.

⬆ Moves the cursor to the previous data item.

When the cursor is on the top row of the data table, it moves to the bottom row of the left column. When the cursor is on the first data item, it moves to the last data item.

⬇ Moves the cursor to the next data item.

When the cursor is on the bottom row of the data table, it moves to the top row of the right column. When the cursor is on the last data item, it moves to the first data item.

⬅ Moves the cursor left on the screen.

When the cursor is on the left column of the data table, it moves to the right column on the row immediately above. When the cursor is on the first data item, it moves to the last data item.

➡ Moves the cursor right on the screen.

When the cursor is on the right column of the data table, it moves to the left column on the row immediately below. When the cursor is on the last data item, it moves to the first data item.

- Soft key operations

Soft key [EDIT]
Enters the management data edit mode.

Soft key [NO.SRH]
Displays data with a tool management data number when pressed after the tool management data number is entered.

Soft key [PREV.TOOL]
Returns to the previous tool management data number.

Soft key [NEXT.TOOL]
Proceeds to the next tool management number.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Operation in the management data edit mode

To edit data, press soft key [EDIT] to enter the management data edit mode.

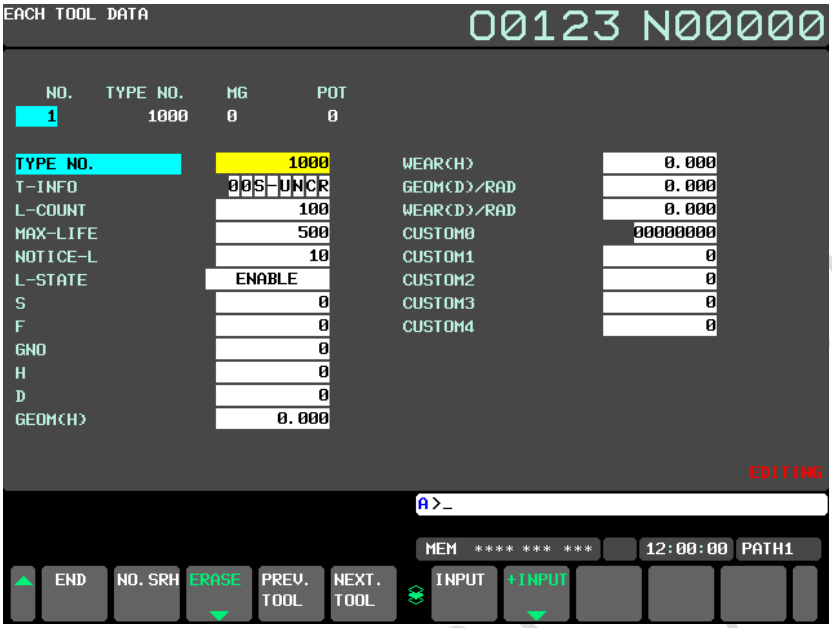


Fig. 12.3.9.3 (b) Each tool data screen (10.4-inch display unit)

In the management data edit mode, “EDITING” is displayed at the lower right of the screen.
In addition to the above key operations, the following key operations are available in the management data edit mode:

Actually writes a value entered using numeric keys.

Soft key [END]

Ends the management data edit mode.

Operations for editing tool information are the same as those on the tool management data screen.

FRYER MACHINE SALES INC
Rob Zahensky

12.3.9.4 Displaying the total life of tools of the same type

Total life data screen

Procedure

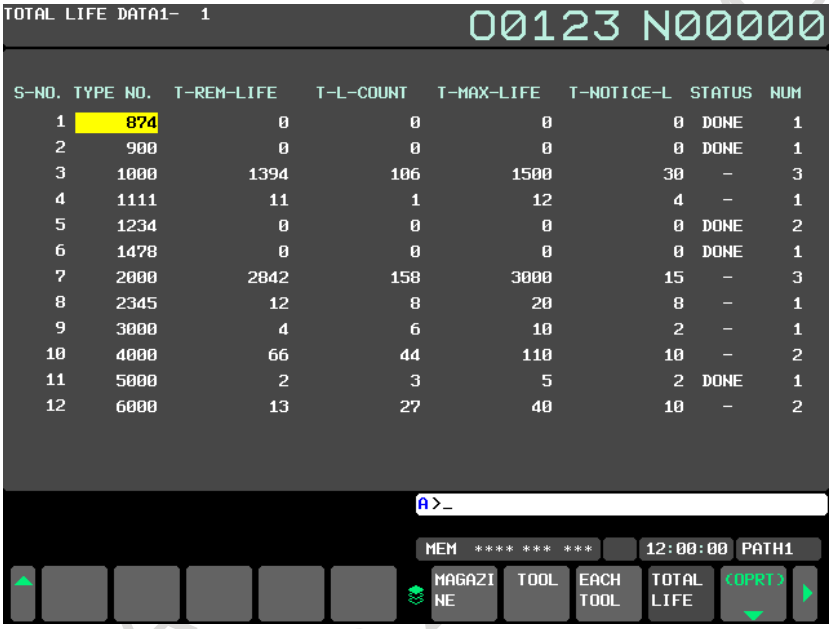
1 Press function key .

2 Press chapter selection soft key [TOOL MANAGER].

Alternatively, press key several times until the tool management screen appears.

3 Press soft key [TOTAL LIFE]. The total life data screen appears.

4 Press soft key [(OPRT)], then soft key [CHANGE] to switch the display between total life data of the count counting type tools and that of the time counting type tools.



S-NO.	TYPE NO.	T-REM-LIFE	T-L-COUNT	T-MAX-LIFE	T-NOTICE-L	STATUS	NUM
1	874	0	0	0	0	DONE	1
2	900	0	0	0	0	DONE	1
3	1000	1394	106	1500	30	-	3
4	1111	11	1	12	4	-	1
5	1234	0	0	0	0	DONE	2
6	1478	0	0	0	0	DONE	1
7	2000	2842	158	3000	15	-	3
8	2345	12	8	20	8	-	1
9	3000	4	6	10	2	-	1
10	4000	66	44	110	10	-	2
11	5000	2	3	5	2	DONE	1
12	6000	13	27	40	10	-	2

Fig. 12.3.9.4 (a) Count display (10.4-inch display unit)

S-NO.	TYPE NO.	T-REM-LIFE	T-L-COUNT	T-MAX-LIFE	T-NOTICE-L	STATUS	NUM
1	874	0H 0M 0S	0H 0M 0S	0H 0M 0S	0H 0M 0S	DONE	1
2	900	0H 0M 0S	0H 0M 0S	0H 0M 0S	0H 0M 0S	DONE	1
3	1000	0H 5M45S	0H 3M15S	0H 9M 0S	0H 0M15S	-	3
4	1111	0H35M37S	1H24M23S	2H 0M 0S	0H40M 0S	DONE	1
5	1234	0H 0M 0S	0H 0M 0S	0H 0M 0S	0H 0M 0S	DONE	2
6	1478	0H 0M 0S	0H 0M 0S	0H 0M 0S	0H 0M 0S	DONE	1
7	2000	2H58M20S	0H 1M40S	3H 0M 0S	0H 0M30S	-	3
8	2345	0H 9M59S	1H50M 1S	2H 0M 0S	0H10M 0S	DONE	1
9	3000	0H 0M 0S	0H 0M10S	0H 0M10S	0H 0M 5S	DONE	1
10	4000	2H 3M11S	0H 1M49S	2H 5M 0S	0H 6M 0S	-	2
11	5000	0H14M25S	0H 0M35S	0H15M 0S	0H 2M 0S	-	1
12	6000	0H56M47S	0H 3M13S	1H 0M 0S	0H 2M 0S	-	2

Fig. 12.3.9.4 (b) Time display (10.4-inch display unit)

FRYER MACHINERY INC

Displayed information

S-NO.	:	Sequential number of each tool type
TYPE NO.	:	Tool type number
T-REM-LIFE	:	Total of remaining life values of tools with the same tool type number
T-L-COUNT	:	Total of used counts/times of tools with the same tool type number
T-MAX-LIFE	:	Total of maximum life values of tools with the same tool type number
NUM	:	Number of tools with the same tool type number
T-NOTICE-L	:	Total of notice life values of tools with the same tool type number when the tool life arrival signal is to be output for each tool type
STATUS	:	Whether the tool life arrival signal has been output when the signal is to be output for each tool type Either of the two states (UNDONE and DONE) is displayed.

When bit 3 (ETE) of parameter No. 13200 is set to 0 and bit 2 (TRT) of parameter No. 13200 is set to 1, the tool life arrival signal is to be output for each tool type number and T-NOTICE-L and STATUS are displayed.

Key operations

- MDI key operations

Displays the previous page. The cursor moves to the last data item on that page.

Displays the next page. The cursor moves to the first data item on that page.

Moves the cursor up on the screen.

Moves the cursor down on the screen.

Disabled.

Disabled.

- Soft key operations

Soft key [DETAIL]
Displays the detailed life data screen.

Soft key [UPDATE]
Updates data displayed on the total life data screen.

Soft key [CHANGE]
Switch the display between the count and time types.

Soft key [T-TYPENO. SRH]
Moves the cursor to a tool type number when pressed after the number is entered.

Soft key [T-ASCE -SORT]
Sorts data in ascending order of tool type numbers.

Soft key [T-DESC -SORT]
Sorts data in descending order of tool type numbers.

Soft key [R-ASCE -SORT]
Sorts data in ascending order of remaining life values.

Soft key [R-DESC -SORT]
Sorts data in descending order of remaining life values.

NOTE

1 After soft key [T-ASCE -SORT], [T-DESC -SORT], [R-ASCE -SORT], or [R-DESC -SORT] is pressed, the cursor is positioned at the top of page 1 of the total life data screen.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

2 When the power is turned on, data of the count counting type is displayed in ascending order of tool type numbers. When the display type is changed or data is sorted in a different order, the status is kept.

3 If soft key [DETAIL] is pressed when no data is displayed on the total life data screen, the warning message “NO DETAILED LIFE DATA SCREEN” is output.

4 If an unregistered tool type is specified and soft key [T-TYPENO. SRH] is pressed, the warning message “UNREGISTERED NUMBER” is output.

Detailed life data screen

Procedure

1 Press function key .

2 Press chapter selection soft key [TOOL MANAGER].

Alternatively, press several times until the tool management screen appears.

3 Press soft key [TOTAL LIFE]. The total life data screen appears.

4 Press soft key [(OPRT)], then soft key [DETAIL]. The detailed life data screen appears.

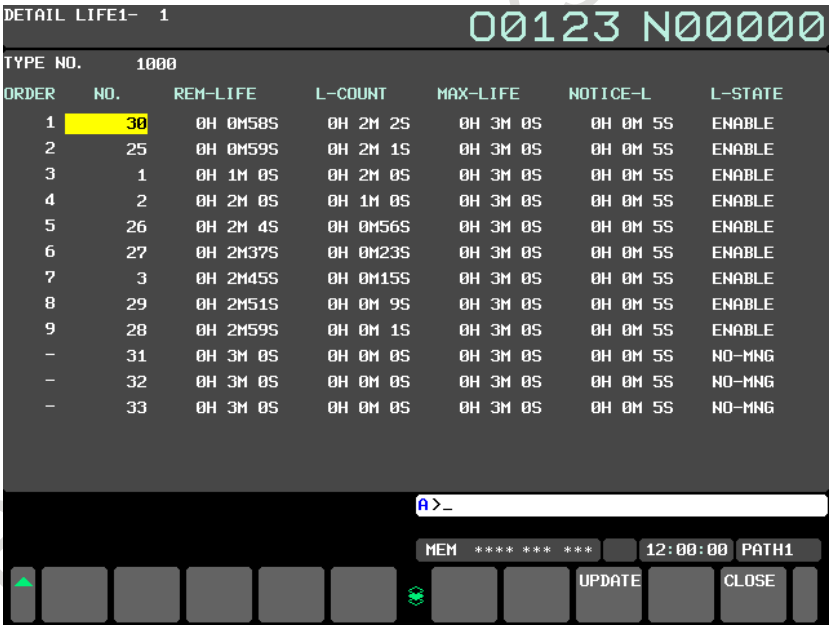


Fig. 12.3.9.4 (c) Detailed life data screen (10.4-inch display unit)

- Displayed information

TYPE NO. : Tool type number

ORDER : Sequential number in ascending order of remaining life times or the order in which the customize data is set.

NO. : Tool management data number

REM-LIFE : Remaining life value obtained by subtracting the life count value from the maximum life value

L-COUNT : Total used count or time of the tool

MAX-LIFE : Maximum life value of the tool

NOTICE-L : Notice life value of the tool

STATUS : Life status of the tool
One of the four states (NO-MNG (0), ENABLE (1, 2), NONE (3), and DAMAGED (4)) is displayed.

FANUC SYSTEMS INC

Key operations

- MDI key operations

Displays the previous page.

Displays the next page.

Moves the cursor up on the screen. The cursor moves to the last data item on that page.

Moves the cursor down on the screen. The cursor moves to the first data item on that page.

Disabled.

Disabled.

- Soft key operations

Soft key [UPDATA]

Updates data displayed on the detailed life data screen.

The cursor is positioned at the beginning of page 1 after this soft key is pressed.

Soft key [CLOSE]

Closes the detailed life data screen and returns to the total life data screen.

NOTE

When soft key [CLOSE] is pressed and the total life data screen is displayed again, the cursor on the total life screen data is positioned as follows.

1 When the tools with the tool type number displayed when the detailed data screen is closed are registered as tool management data, the cursor on the total life screen is positioned at that tool type number.

2 When the tools with the tool type number displayed when the detailed data screen is closed are not registered as tool management data, the cursor on the total life data screen is positioned at the first total life data.

FRYER MACHINE SYSTEMS INC
Rob Zaharansky

12.3.9.5 Tool geometry data screen

Tool geometry data screen

Procedure

1 Press function key .

2 Press chapter selection soft key [TOOL MANAGER].

Alternatively, press several times until the tool management screen appears.

3 Press continuous menu key , then soft key [TOOL-GEOM]. The tool geometry data screen appears.

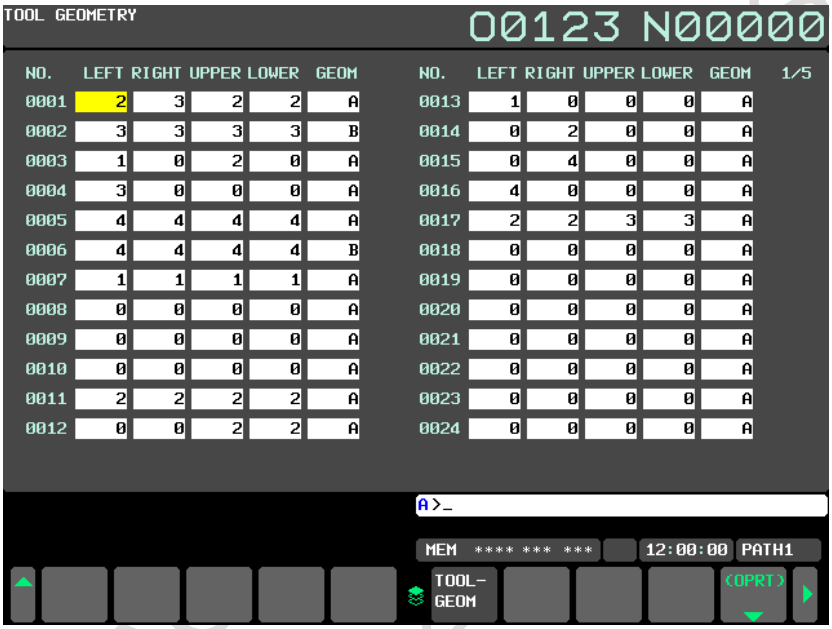


Fig. 12.3.9.5 (a) Tool geometry data screen (10.4-inch display unit)

- Displayed item

NO. : Tool geometry number
Up to 20 numbers can be displayed.

LEFT : Sets the number of pots that are to be occupied on the left of the reference pot.
A value between 0 and 4 can be set.

RIGHT : Sets the number of pots that are to be occupied on the right of the reference pot.
A value between 0 and 4 can be set.

UPPER : Sets the number of pots that are to be occupied above the reference pot.
A value between 0 and 4 can be set. (Use this item when the magazine is of the matrix type.)

LOWER : Sets the number of pots that are to be occupied under the reference pot.
A value between 0 and 4 can be set. (Use this item when the magazine is of the matrix type.)

Key operations

- Operations in the standard mode

MDI key operations

Numeral keys Inputs a numeric value.

Moves the cursor up on the screen.

Moves the cursor down on the screen.

FRYER MACHINES INC Rob

Moves the cursor left on the screen.

Moves the cursor right on the screen.

Soft key operations

Soft key [NO.SRH]

Moves the cursor on a tool geometry number when pressed after the number is entered.

Soft key [EDIT]

Enters the data edit mode.

Soft key [F INPUT]

Inputs data related to the tool management functions.
This key is available only in the standard mode.
Put the NC in the EDIT mode.

Soft key [F OUTPUT]

Outputs data related to the tool management functions.
This key is available only in the standard mode.
Put the NC in the EDIT mode.

In the management data edit mode, the following key operations are available in addition to the above key operations.

- Operations in the edit mode

To edit data, press soft key [EDIT] to enter the edit mode. In the edit mode, “EDITING” is displayed at the lower right of the screen.

MDI key operation

INPUT Actually writes a value entered using numeric keys.

Soft key operation

Soft key [END]

Ends the data edit mode.

Example

Set the edit mode. When the tool geometry with tool geometry number 1 occupies 1 pot in the left direction, 0.5 pots in the right direction, and 1.5 pots in the down direction, set data as shown in the figure below:

FRYER MACHINE SYSTEMS INC
Rob Zahensky

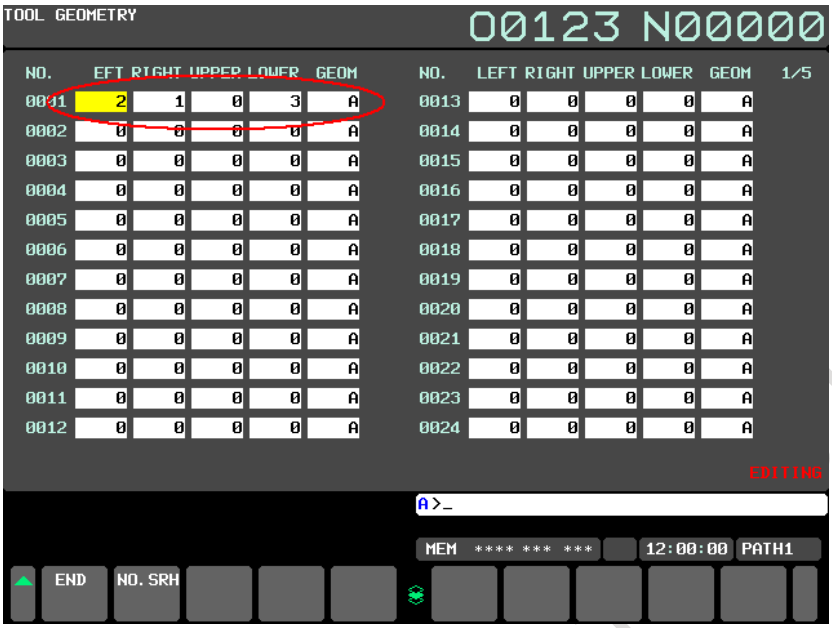


Fig. 12.3.9.5 (b) Example of setting data on the tool geometry data screen (10.4-inch display unit)

- Display of occupied pots in the magazine management table

Each pot occupied by a tool stored in another pot is indicated with an asterisk (*).

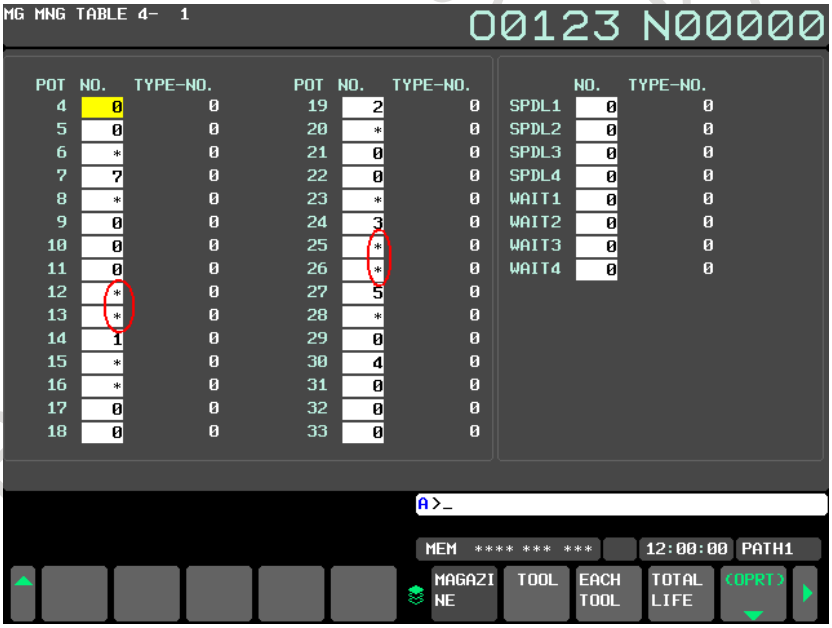


Fig. 12.3.9.5 (c) Magazine management table (10.4-inch display unit)

If a tool to be registered for a magazine is determined to interfere with another tool, the warning message “TOOL INTERFERENCE CHECK ERROR:xxxx,xxxx” is displayed. xxxx indicates the tool number of each of the two tools. If a tool is determined to interfere with multiple tools, only the first tool detected by the NC interference check is displayed.

If a tool is determined to interfere with the frame of a magazine, the tool number of the tool to be stored and “FRAME” are displayed.

FRYER SYSTEM INC

- Search for an empty pot for an oversize tool

In the management data edit mode, press soft key [(OPRT)], then continuous menu key ▷ displays the soft keys for searching for an empty pot for an oversize tool.

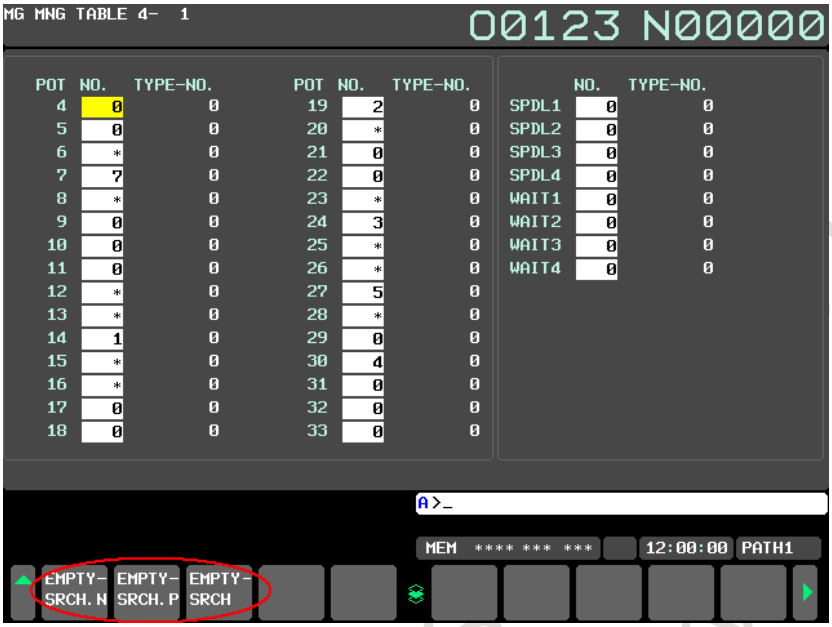


Fig. 12.3.9.5 (d) Searching for an empty pot for an oversize tool (10.4-inch display unit)

Enter the tool geometry number in the key-in buffer and press a search soft key. The cursor moves to an empty pot fit for the geometry.

EMPTY-SRCH.N : Searches for an empty pot forward.
EMPTY-SRCH.P : Searches for an empty pot backward.
EMPTY-SRCH : Searches for the pot nearest to the current position.

- Tool management screen

You can use bit 2 of tool information to switch between an oversize tool and normal tool. For an oversize tool, set a tool geometry number fit for the tool.

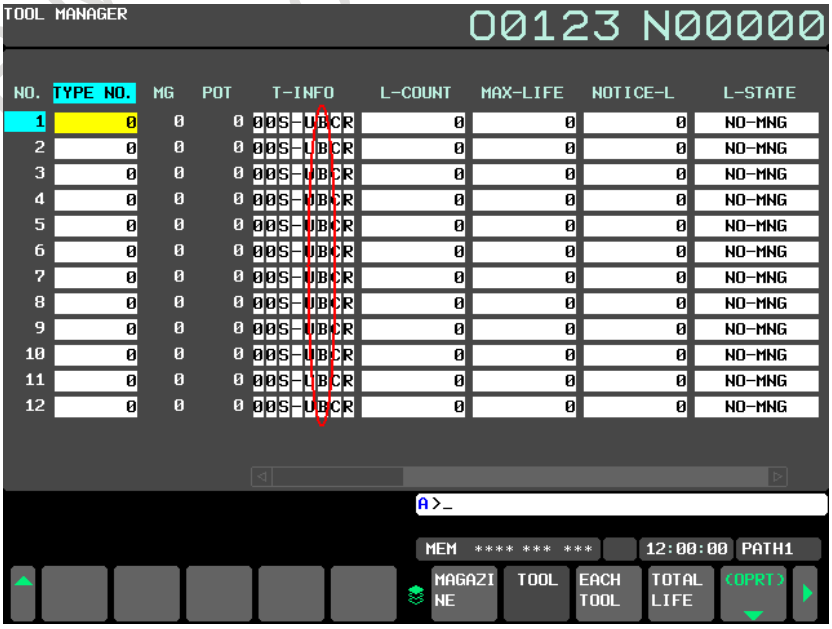


Fig. 12.3.9.5 (e) Bit for switching between a normal tool and oversize tool (10.4-inch display unit)

FRYER MACHINE S INC
Jacob Zahensky

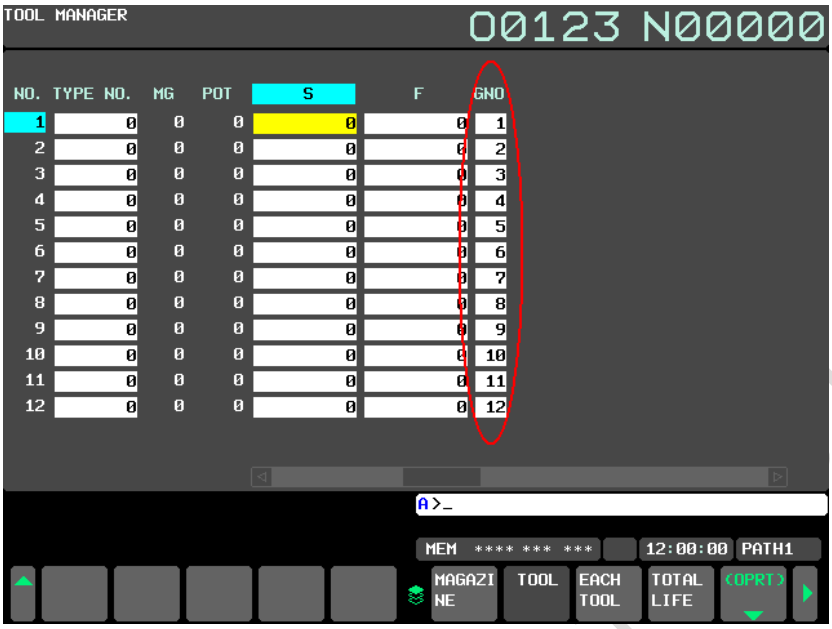


Fig. 12.3.9.5 (f) Tool geometry number (10.4-inch display unit)

12.3.10 Displaying and Switching the Display Language

The language used for display can be switched to another language.
A display language can be set using a parameter. However, by modifying the setting of the display language on this screen, the display language can be switched without turning off then on the power.

Displaying and setting the display language

Procedure

1 Press function key .

2 Press the continuous menu key ▷ several times.

3 Press soft key [LANGUAGE] to display the language screen.

Fig. 12.3.10 (a) LANGUAGE screen (10.4-inch display unit)

FRYER MACHINE SYSTEMS INC
ob Zahensky

4 Press page key [PAGE ⇧] or [PAGE ⇩], then press cursor keys [⬆] or [⬇] to move the cursor to a desired display language.

5 Press operation soft key [APPLY]. The display language is switched to the selected language. The language specified on this screen continues to be used if the power is turned off then back on.

Explanation

- Language switching

The language screen can be displayed if bit 0 (NLC) of parameter No. 3280 is set to 0.

- Selectable languages

The display languages selectable on this screen are as follows:

1. English
2. Japanese
3. German
4. French
5. Chinese (Traditional)
6. Chinese (Simplified)
7. Italian
8. Korean
9. Spanish
10. Dutch
11. Danish
12. Portuguese
13. Polish
14. Hungarian
15. Swedish
16. Czech
17. Russian
18. Turkish
19. Bulgarian
20. Rumanian
21. Slovak
22. Finnish
23. Vietnamese
24. Indonesian
25. Slovenian

Among the languages listed above, English and other usable languages are displayed on the screen as a list of switchable languages.

Limitation

- Language parameter modification on the parameter screen

Which language to use for display is specified with parameter No. 3281. This parameter can be modified using the parameter screen as well. However, if a modification is made on the parameter screen, the new setting is not reflected until “APPLY” operation is performed on the language screen or the power is turned on again. If an invalid value is set in parameter No. 3281 on the parameter screen, the screen is displayed in English after the power is turned on again.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.11 Protection of Data at Eight Levels

You can set eight CNC and PMC operation levels and one of eight protection levels for each type of CNC and PMC data.
When an attempt is made to change CNC and PMC data or output it to an external unit, the operation level is compared with the protection level to determine whether to allow the change or the external output.

12.3.11.1 Operation level setting

You can set eight CNC and PMC operation levels.

Displaying and setting the operation level setting

Procedure

1 Press function key .

2 Press the continuous menu key several times until soft key [PROTECT] is displayed.

3 Press soft key [PROTECT].
The operation level setting screen shown below is displayed.

Fig. 12.3.11.1 (a) Operation level setting screen (10.4-inch display unit)

4 Key in the password for an operation level to be set/modified, then press soft key [INPUT PASSWD].

5 To return the operation level to 0, 1, 2, or 3, press soft key [CANCEL PASSWD].

Explanation

- **Operation level setting**

To select operation level 0 to 3, use the corresponding memory protection key signal.
To select operation level 4 to 7, use the corresponding password.

Table 12.3.11.1 Operation level setting

Operation level	Setting	Sample grouping
7 (high)	Password	-
6	Password	MTB

FRYER SYSTEMS INC

Operation level	Setting	Sample grouping
5	Password	Dealer and integrator
4	Password	End user
3	Memory protection key signal	User level (level 1)
2	Memory protection key signal	User level (level 2)
1	Memory protection key signal	User level (level 3)
0 (low)	Memory protection key signal	User level (level 4)

When operation level 4 to 7 is set, the operation level remains unchanged until the password is cleared. (The operation level also remains unchanged if the power is turned off.)
Operation level 7 is reserved for CNC and PMC maintenance.

NOTE
When a password is being entered, an asterisk (*) is displayed instead of each entered character.

12.3.11.2 Password modification

The current operation level is displayed.
The password for each of operation levels 4 to 7 can be modified.

Displaying and setting the password modification screen

Procedure

1 Press function key .
2 Press the continuous menu key several times until soft key [PROTECT] is displayed.
3 Press soft key [PROTECT].
4 Press soft key [PASSWORD].
The PASSWORD CHANGE screen shown below is displayed.

Fig. 12.3.11.2 (a) PASSWORD CHANGE screen (10.4-inch display unit)

5 Key in an operation level whose password is to be modified, then press soft key [INPUT].

FRYER MACHINE SYSTEMS INC
Zahensky

6 Key in the current password for the operation level whose password is to be modified, then press soft key [INPUT].
7 Key in a new password, then press soft key [INPUT].
8 Key in the new password again for confirmation, then press soft key [INPUT].
9 Press soft key [PASSWD CHANGE].
10 To clear the password, press soft key [PASSWD CLEAR].

Explanation

Up to eight characters (only uppercase alphabetic characters and numeric characters) can be input.

NOTE

1 For a password, consisting of three to eight characters, the following characters are available:
- Uppercase alphabetic characters
- Numeric characters

2 When a password is being entered, an asterisk (*) is displayed instead of each entered character.
3 Whether a password can be changed at the current operation level is determined as follows:
- Password of an operation level higher than the current operation level
  Cannot be changed.
- Password of the current operation level
  Can be changed.
- Password of an operation level lower than the current operation level
  Can be changed (only to the initial password).

4 The set password is not displayed.
Be careful not to forget the password.

12.3.11.3 Protection level setting

The current operation level is displayed.
The change protection level and output protection level of each data item are displayed.
The change protection level and output protection level of each data item can be changed.

Confirmation based on protection level setting

Procedure

1 Press function key .
2 Press the continuous menu key several times until soft key [PROTECT] is displayed.
3 Press soft key [PROTECT].
4 Press soft key [DATA LEVEL] to change the protection level of CNC data or press soft key [PMC LEVEL] to change the protection level of PMC data.

FRYER MACHINE SYSTEMS INC
Rob Zahersky

The following protection level change screen is displayed.

Fig. 12.3.11.3 (a) Protection level change screen (10.4-inch display unit)

5 Move the cursor to the change level or output level of a desired data item.
6 Key in a new desired level, then press soft key [INPUT].

> **NOTE**
> When the protection level of PMC data is set, soft key [SWITCH PMC] is used to switch between PMC paths to be set, for multi-path PMC.

Explanation

When the protection level of a data item is higher than the current operation level, the protection level of the data item cannot be changed.

The protection level of a data item cannot be changed to a protection level higher than the current operation level.

For each of the following types of data, you can set a data protection level. There are the following two types of data protection levels:

- Change protection level
  Sets the protection level used when data is changed.
- Output protection level
  Sets the protection level used when data is output to an external unit.

As a protection level, you can set a value of 0 (low) to 7 (high).

Table 12.3.11.3 (a) Protection level of each type of data

Type of data	Initial protection level	
	Change	Output
Custom macro variable data <CUSTOM MACRO> (including variable data dedicated to the macro executor)	0	0
Periodical maintenance data <PERIODICAL MAINTENANCE DATA>	0	0
Tool management data <TOOL LIFE TIME DATA>	0	0
Tool offset data <TOOL OFFSET DATA> (For each type when tool geometry compensation and tool wear compensation are treated differently)	0	0
Clock data <TIME>	0	0
Workpiece origin shift amount data <WORK ZERO SHIFT>	0	0

FRYER MACHINE SYSTEMS INC
Rob Zanensky

Type of data	Initial protection level	
	Change	Output
Workpiece origin offset data <WORK ZERO OFFSET>	0	0
Ethernet setting data <ETHER NET SETTING>	0	0
Parameter data <PARAMETER DATA>	4	0
Settings <SETTING DATA>	0	0
Pitch error compensation data <PITCH ERROR DATA>	4	0
Parameter data for Power Mate CNC manager function <PARAMETER DATA FOR POWER MATE CNC MANAGER>	0	0
Each part program <PROGRAMS FOR EACH PARTS>	0	0
Part program editing operation <CHANGE PROGRAMS FOR EACH PARTS>	0	0
Absolute coordinate preset operation <PRESET OF ABSOLUTE AXIS DATA>	0	0

Table 12.3.11.3 (b) Protection level of PMC data

Type of data	Initial protection level	
	Change	Output
Composition parameter	0	0
Setting (online)	0	0
Setting (each path)	0	0
Sequence program	0	0
PMC parameter	0	0
Timer	0	0
Counter	0	0
Keep relay	0	0
Keep relay (system)	0	0
Data table	0	0
Data table control	0	0
PMC memory	0	0
I/O configuration	0	0
I/O Link group selection	0	0
Registration of I/O device	0	0

NOTE

1 For some types of data, the output function is not provided.
2 When the protection level of data is higher than the current operation level, the protection level cannot be changed.
3 The protection level of data cannot be changed to a level higher than the current operation level.
4 Settable types of data increase or decrease, depending on the option configuration.
5 For details on the protection level of PMC data, refer to “PMC Programming Manual (B-64513EN)”.
6 Data related to tool information on the EACH TOOL DATA screen for the tool management function is protected by protecting tool offset data.
7 The type of tool offset data put in effect varies depending on the tool compensation value memory used.
8 To change the protection level for each part program, do so on the PROGRAM FOLDER screen rather than on the PROTECT LEVEL screen.
9 Part program editing includes program editing for the MDI mode.
10 Presetting absolute coordinates causes preset workpiece coordinate system values to be protected.

FRYER MACHINE SYSTEMS INC
Bob Zahensky

NOTE

11 During tool offset data input/output, if any tool offset data type is not allowed to be changed or output to the outside, it is processed as follows:
- Input : Any data type other than those whose change is not allowed is changed.
- Output : Any data type other than those whose change is not allowed is output.

12.3.11.4 Setting the change protection level and output protection level of a program

The display/operations indicated below can be performed from the directory screen.
The change protection level and output protection level of each part program are displayed.
The change protection level and output protection level of each part program can be changed.

Setting the change protection level and output protection level of a program

Procedure

1 Press function key .

2 Press soft key [FOLDER].
The program folder screen shown below is displayed.

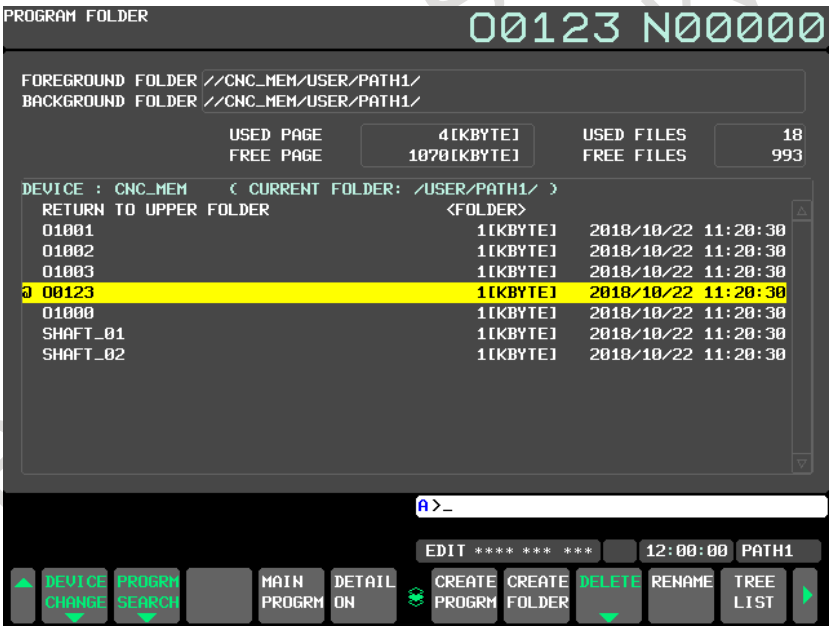


Fig. 12.3.11.4 (a) Program folder screen (10.4-inch display unit)

3 Press soft key [(OPRT)].
4 Press soft key [DETAIL ON]. The screen display switches to the detail display screen.
5 Move the cursor to a desired program.
6 Press the continuous menu key several times until [CHANGE ATTR] is displayed.
7 Press soft key [CHANGE ATTR].
8 Key in a new desired level, then press soft key [CHANGE LEVEL].
9 To change the output protection level, key in a new desired level, then press soft key [OUT LEVEL].

FRYER SYSTEMS INC

Explanation

The change protection level (0 to 7) and output protection level (0 to 7) are displayed as “CHANGE PROTECTION LEVEL VALUE/OUTPUT PROTECTION LEVEL”.

NOTE

1 When the protection level of data is higher than the current operation level, the protection level cannot be changed.
2 The protection level of data cannot be changed to a level higher than the current operation level.
3 A protection level can be set only for those part programs that are held on the “CNC_MEM” device.

12.3.12 Precision Level Selection

An intermediate precision level between the parameters for emphasis on velocity (precision level 1) and the parameters for emphasis on precision (precision level 10) set on the machining parameter tuning screen (AI contour) can be selected. As shown in the Fig. 12.3.12 (a), since the levels are proportionally linear, by selecting an intermediate level, optimal parameters can be automatically calculated to perform machining.

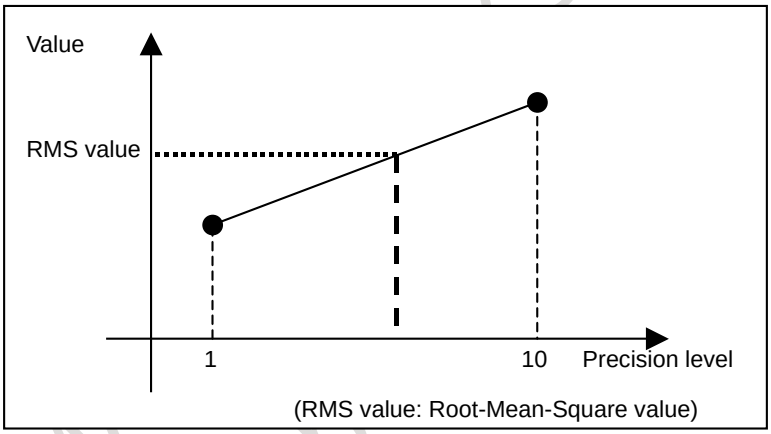


Fig. 12.3.12 (a) Image of “level”

FRYER MA ROY SYSTEMS INC

Procedure for precision level selection

1 Select the MDI mode.

2 Press function key .

3 Press soft key [PRECI LEVEL].

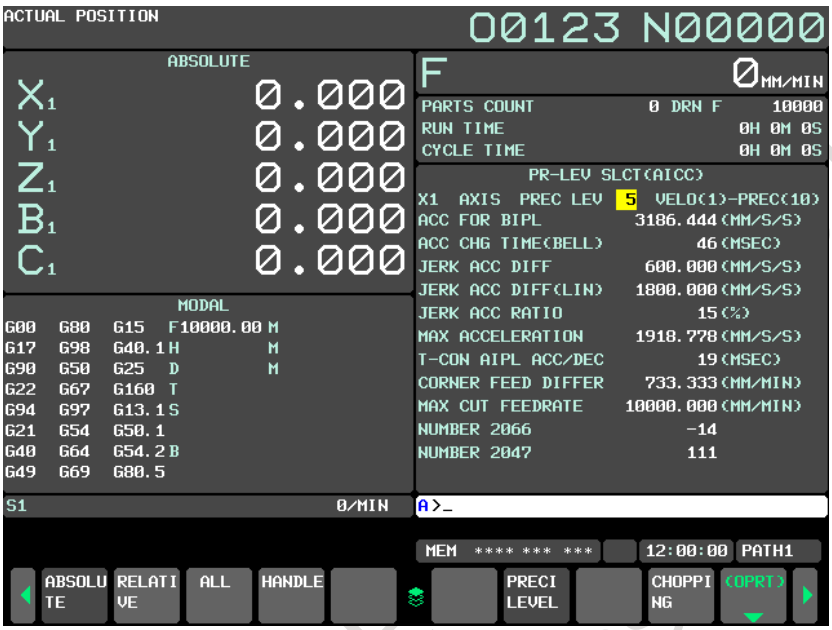


Fig. 12.3.12 (b) Precision level selection screen (10.4-inch display unit)

4 To change the precision level, key in a desired precision level (1 to 10), then press the INPUT key on the MDI unit.

5 When the precision level is changed, a RMS value is obtained from the velocity-emphasized parameter set and precision-emphasized parameter set for parameter modification.
For the modified parameters, see the description of the machining parameter tuning.

6 If there is an axis in addition to the currently displayed axes, press page key PAGE or PAGE several times to display the screen for the axis.

FRYER MACHINE INC
Rob Zaharic

12.3.13 Machining Level Selection

12.3.13.1 Smoothing level selection

An intermediate smoothing level between the parameters for smoothing level 1 and the parameters for smoothing level 10 set on the machining parameter tuning screen (smoothing) can be selected. As shown in the Fig. 12.3.13.1 (a), since the levels are proportionally linear, by selecting an intermediate level, optimal parameters can be automatically calculated to perform machining.

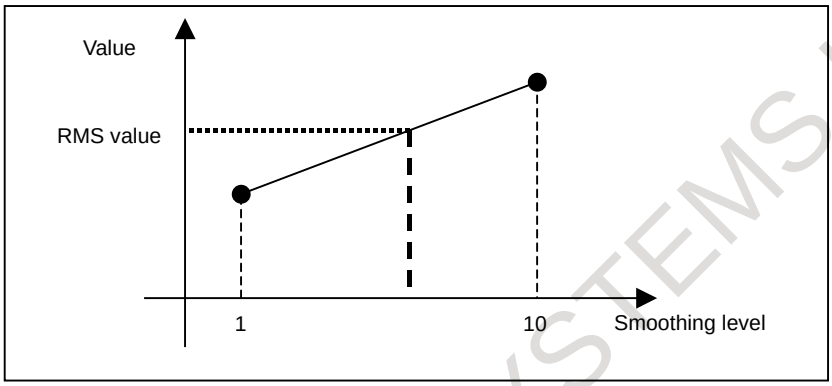


Fig. 12.3.13.1 (a) Image of "level"

Procedure for smoothing level selection

1 Select the MDI mode.

2 Press function key .

3 Press soft key [PRECI LEVEL].

4 Press soft key [SMOOTH LEVEL].

Fig. 12.3.13.1 (b) Smoothing level selection screen (10.4inch)

5 To change the smoothing level, key in a desired smoothing level (1 to 10), then press the key on the MDI unit.

FRYER MACHINE SYSTEMS INC
Zahensky

6 When the smoothing level is changed, a RMS value is obtained from the smoothing level 1 parameter set and smoothing level 10 parameter set for parameter modification.
For the modified parameters, see the description of the machining parameter tuning.

7 If there is an axis in addition to the currently displayed axes, press page key PAGE or PAGE several times to display the screen for the axis.

12.3.13.2 Precision level selection

For details of precision level selection, See Subsection, “Precision Level Selection”.

12.3.14 Displaying and Setting Tool Life Management Data

Displaying tool life management data on a screen enables the current status of tool life management to be grasped. Also on the screen, tool life management data can be edited. The screen is either:

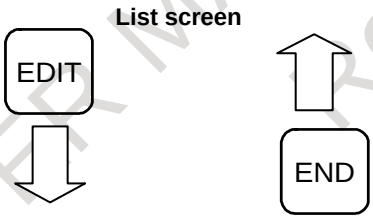
- Tool life management (list screen) or
- Tool life management (group editing screen)

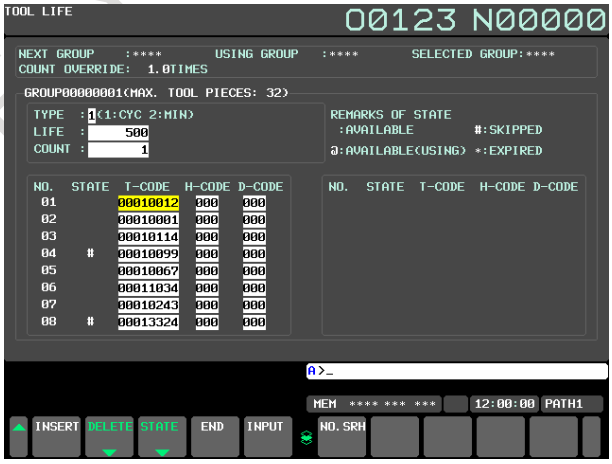
Overview

Tool life management (list screen)
Displayed items:
- NEXT GROUP - USING GROUP
- SELECTED GROUP - COUNT OVERRIDE
- GROUP NO. - COUNT TYPE
- LIFE - LIFE COUNT
- TOOL MANAGEMENT STATUS - TOOL NO.
- GROUP TO BE CHANGE

Functions:
- Searching for groups
- Clearing execution data
- Setting the tool life counter

List screen





Group editing screen

Tool life management (group editing screen)
Displayed items:
- NEXT GROUP - USING GROUP
- SELECTED GROUP - GROUP NO.
- MAX.TOOL PIECES - COUNT TYPE
- LIFE - LIFE COUNT
- T-CODE - H-CODE and D-CODE
- TOOL MANAGEMENT STATUS

Functions:
- Setting tool data - Deleting group data
- Setting a tool status - Deleting tool data
- Selecting tool skip - Searching for groups
- Setting a life count type, life value, and tool life counter

Pressing horizontal soft key [EDIT] or [END] switches from the list screen to the group editing screen or vice versa.

FAYER MATE SYSTEMS INC Rob Zaiss

M

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1), the following parameters can be used to display arbitrary groups and remaining set values.

- If bit 5 (TGN) of parameter No. 6802 = 1
  An arbitrary group is displayed on the list and group editing screens.
- If bit 3 (GRP) of parameter No. 6802 = 1
  Remaining set values are displayed on the group editing screen. The group editing screen always displays H and D codes.

T

The T series is provided with two tool change types, turret type and ATC type. For the T series, displays vary depending on which tool change type is in use.

If the turret type is in use (bit 3 (TCT) of parameter No. 5040 = 0)

- Neither the H code nor the D code is used; neither is displayed.
- No arbitrary group can be used; no arbitrary group is displayed.

If the ATC type is in use (bit 3 (TCT) of parameter No. 5040 = 1)

- The D code is displayed on the group editing screen.
- If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1) and bit 5 (TGN) of parameter No. 6802 = 1, arbitrary groups are displayed on the list and group editing screens.

If the tool life management B function is enabled (bit 4 (LFB) of parameter No.6805 = 1) and bit 3 (GRP) or parameter No. 6802 = 1, remaining set values are displayed on the group editing screen.

12.3.14.1 Tool life management (list screen)

This screen can display the life management status of all tools in tool groups and whether the life of the tool groups has expired. It also enables you to set tool life counters and clear execution data.

Displays on the list screen

Procedure

1 Press function key .

2 Press the continuous menu key ▷ several times to display soft key [TOOL LIFE].

3 Press soft key [TOOL LIFE].

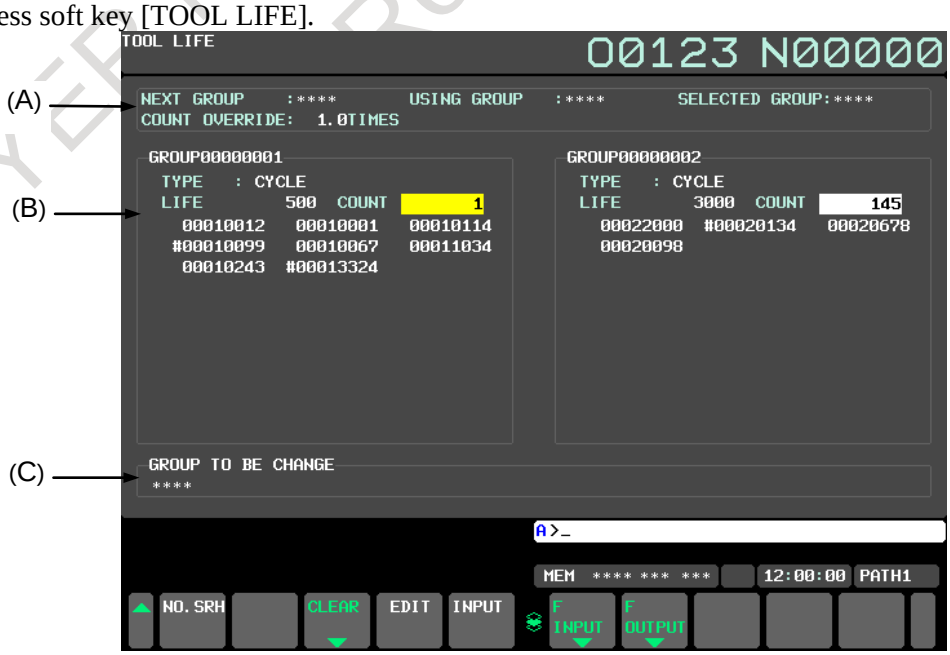


Fig. 12.3.14.1 (a) Displaying tool life management (list screen) (10.4-inch display unit)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Contents of (A)

(A) displays tool group numbers and an override value. If there is no tool group to display, ****, instead of tool group numbers, is displayed.

NEXT GROUP : Tool group number for which life counting is started by the next M06 command.
USING GROUP : Tool group number for which life counting is currently under way.
SELECTED GROUP : Tool group number for which life counting is currently under way or life counting has been performed most recently.
COUNT OVERRIDE : "1.0TIMES" is displayed if the tool life counter override signal is disabled (bit 2 (LFV) of parameter No. 6801 = 0).

NOTE
If arbitrary group numbers are enabled, NEXT GROUP, USING GROUP, and SELECTED GROUP are each represented with an arbitrary group number rather than the tool group number.

- Contents of (B)

(B) displays the set life value, the current content of the tool life counter, and the registered tool numbers (in the order they are used) for each tool group. If the life count type is specification by duration, the measurement unit used in displaying and specifying set life values and tool life counter values is selected according to the setting of bit 0 (FCO) of parameter No. 6805 as listed below.

Bit 0 (FCO) of parameter No. 6805	0	1
Measurement unit used in displaying and specifying set life values and tool life counter values	1 minute	0.1 minutes

The following table lists the prefixes used with tool numbers.

Tool status	Tool in use	Tool not in use
Life remaining	@	No indication
Skip	#	#
Life expired	*	*

NOTE
1 The tool life counter indicates the count value for the tool indicated with @.
2 If bit 3 (EMD) of parameter No. 6801 = 0, a tool number remains prefixed with @ even if the life of the tool has expired until another tool is selected.
3 If bit 3 (EMD) of parameter No. 6801 =1, the following differences can occur depending on the type of the life counter in use.
- If the life counter type is specification by duration, the prefix is changed to "*" (life expired) as soon as the tool life expires.
- If the life counter type is specification by count, the counter is incremented by one at the end of a program (such as M02 or M30). So, the prefix is not changed to "*" (life expired) even if the tool life counter matches the life value. The symbol "*" (life expired) appears when the tool life count is incremented after the CNC is reset.

4 If bit 2 (ETE) of parameter No. 6804 = 1, the symbol "*" indicating the expiration of the life of the last tool appears on the tool life management screen when the life counter for the last tool of the tool group of interest matches the life value. This way, the information about the last tool in the FOCAS2 or PMC window indicates that the life of the tool has expired if the tool change signal TLCH <Fn064.0> is 1.

FRYER MACHINE SYSTEMS INC
Rob Zahenski

If arbitrary group numbers are enabled, an arbitrary group number is displayed in the parentheses beside the tool group number. If no arbitrary group number is specified, “********” is displayed instead.

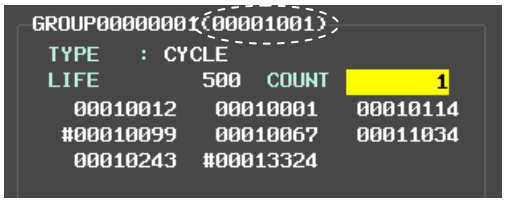


Fig. 12.3.14.1 (b) Displaying arbitrary group numbers

Arbitrary group numbers are enabled by setting the following parameters.

M

- The tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1).
- Arbitrary group numbers are enabled (bit 5 (TGN) of parameter No. 6802 = 1).

T

- The current tool change type is ATC (bit 3 (TCT) of parameter No. 5040 = 1).
- The tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1).
- Arbitrary group numbers are enabled (bit 5 (TGN) of parameter No. 6802 = 1).

- Contents of (C)

(C) displays tool group numbers for which a tool change signal has been issued.
If there are so many tool group numbers that all the numbers cannot be displayed, some are omitted, and “>>“ is displayed instead.
If there is no tool group number in need of change, “****” is displayed.

> **NOTE**
> If arbitrary group numbers are enabled, a tool group number in need of change is represented with an arbitrary group number rather than the tool group number.

Setting data on the list screen

Tool life management data can be specified in the reset state (both the OP and RST signals are “0”). However, setting bit 1 (TCI) of parameter No. 6804 to 1 enables tool life management data to be specified even during automatic operation (the OP signal is “1”).

> **NOTE**
> As for USING GROUP or NEXT GROUP settings:
> 1) During automatic operation (OP signal = “1” and bit 1 (TCI) of parameter No. 6804 = 1), only the tool life counter can be changed.
> 2) In the reset state (OP signal = “0” and RST signal “0”), the following editing operation stops life management because it cannot be continued.
> - Clearing execution data

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Procedure

- Setting the tool life counter

The tool life counter can be set with a value, using the following methods.

Method 1

1 Place the cursor on the tool life counter for a desired tool group.
2 Enter the value from the keypad.
3 Press soft key [INPUT].

Method 2

1 Place the cursor on the tool life counter for a desired tool group.
2 Enter the value from the keypad.
3 Press INPUT key.

- Clearing execution data

All existing execution data for a tool group selected by the cursor can be cleared as follows:

1 Place the cursor on the tool group whose execution data you want to clear.
2 Press soft key [ERASE].
3 Press soft key [EXEC].

> **NOTE**
> Setting bit 4 (GRS) of parameter No. 6800 to 1 enables execution data for all registered tool groups to be cleared.

- Selecting tool groups

Tool groups can be selected using the following methods.

Method 1

1 Enter a tool group number from the keypad.
2 Press soft key [NO.SRH].

> **NOTE**
> If arbitrary group numbers are enabled, a tool group is selected by searching for an arbitrary group number rather than the tool group number.

Method 2

1 Press page key PAGE or PAGE to display desired groups.
2 Press cursor key ← or → to move the cursor to the desired group at either the left or right.

- Switching to the group editing screen

Switch to tool life management (group editing screen).

1 Move the cursor to the tool group you want to edit.
2 Press soft key [EDIT].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.14.2 Tool life management (group editing screen)

On this screen, it is possible to edit tool life management data (such as tool life value, tool life counter, and tool data) for the tool group of interest.

Displaying the group editing screen

Procedure

1 Place the cursor on the tool group you want to edit from the list screen.
2 Press soft key [(OPRT)].
3 Press soft key [EDIT].

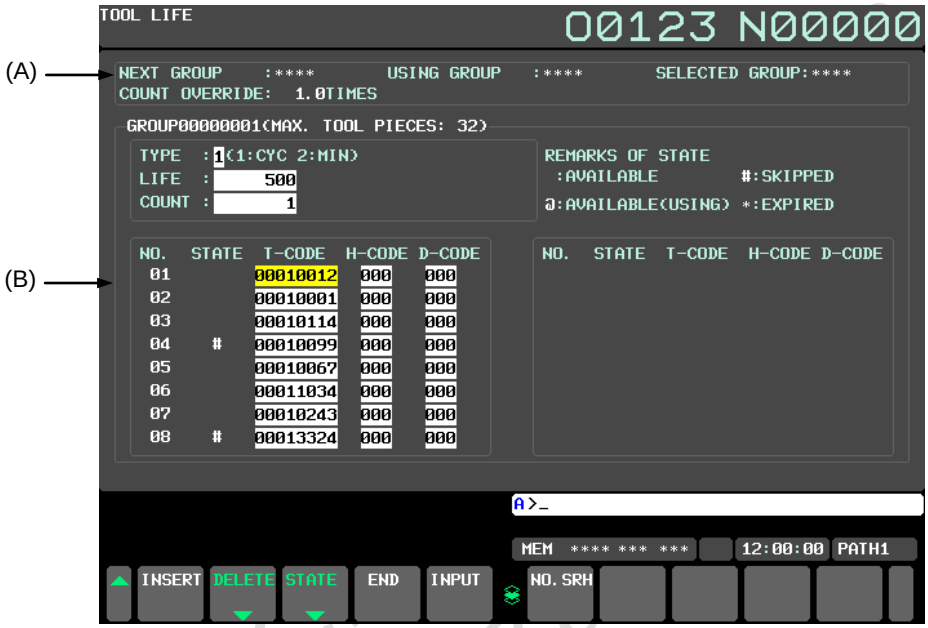


Fig. 12.3.14.2 (a) Displaying tool life management (group editing screen) (10.4-inch display unit)

> **NOTE**
> If no tool is registered with a tool group, none of a life count type, a life value, and a tool life counter value is displayed for the tool group.

- Contents of (A)

Like the counterpart of the list screen, (A) of the editing screen displays the next tool group number and override value. If there is no appropriate tool group, **** is displayed instead.

NEXT GROUP : Tool group for which life counting is started by the next M06 command.
USING GROUP : Tool group number for which life counting is currently under way.
SELECTED GROUP : Tool group number for which life counting is currently under way or life counting has been performed most recently.
COUNT OVERRIDE : “1.0TIMES” is displayed if the tool life counter override signal is disabled (bit 2 (LFV) of parameter No. 6801 = 0).

> **NOTE**
> If arbitrary group numbers are enabled, NEXT GROUP, USING GROUP, and SELECTED GROUP are each represented with an arbitrary group number rather than the tool group number.

- Contents of (B)

(B) details tool life management data related to a selected tool group as follows:

TYPE :

1	Specification by count
2	Specification by duration

LIFE : Tool life value

COUNT : Tool life counter

STATE :

Tool status	Tool in use	Tool not in use
Life remaining	@	No indication
Skip	#	#
Life expired	*	*

T-CODE : Tool number

M

H-CODE : Tool length compensation specification code

D-CODE : Cutter compensation specification code

T

H-CODE : No display.

D-CODE : Tool offset value specification code
if ATC type is in use (bit 3 (TCT) of parameter No. 5040 = 1)

NOTE

1 The tool life counter indicates the count value for the tool indicated with @.

2 If bit 3 (EMD) of parameter No. 6801 = 0, a tool number remains prefixed with @ even if the life of the tool has expired until another tool is selected.

3 If bit 3 (EMD) of parameter No. 6801 = 1, the following differences can occur depending on the type of the life counter in use.
- If the life counter type is specification by duration, the prefix is changed to “*” (life expired) as soon as the tool life expires.
- If the life counter type is specification by count, the counter is incremented by one at the end of a program (such as M02 or M30). So, the prefix is not changed to “*” (life expired) even if the tool life counter matches the life value. The symbol “*” (life expired) appears when the tool life count is incremented after the CNC is reset.

4 If bit 2 (ETE) of parameter No. 6804 = 1, the symbol “*” indicating the expiration of the life of the last tool appears on the tool life management screen when the life counter for the last tool of the tool group of interest matches the life value. This way, the information about the last tool in the FOCAS2 or PMC window indicates that the life of the tool has expired if the tool change signal TLCH <Fn064.0> is 1.

Displaying arbitrary groups and remaining set values

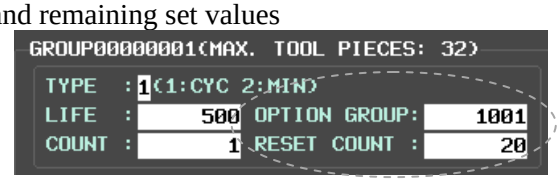


Fig. 12.3.14.2 (b) Displaying arbitrary group numbers and remaining set values on the editing screen

FRYER MACHINE SYSTEMS INC
Rob Zahensky

M

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1) and the following parameters are enabled, arbitrary groups and remaining set values are displayed.
OPTION GROUP : Arbitrary group number (if bit 5 (TGN) of parameter No. 6802 = 1)
RESET COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1)

T

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1)
Turret type (bit 3 (TCT) of parameter No. 5040 = 0)
OPTION GROUP : No display.
REST COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1)
ATC type (bit 3 (TCT) of parameter No. 5040 = 1)
OPTION GROUP : Arbitrary group number (if bit 5 (TGN) of parameter No. 6802 = 1)
RESET COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1)

Setting data on the group editing screen

Tool life management data can be specified in the reset state (both the OP and RST signals are “0”). However, setting bit 1 (TCI) of parameter No. 6804 to 1 enables tool life management data to be specified even during automatic operation (the OP signal is “1”).
The available editing operations are listed below.

M

Items that can be edited	Mode
Setting a life count type, life value, tool life counter, and tool data (T code, H code, and D code)	All modes
Adding tool numbers (T code)	MDI
Deleting all tool group data at a time	MDI
Deleting tool data (status, T code, H code, and D code)	MDI
Selecting tool skip	MDI
Specifying to clear tool data (life re-set)	MDI

T

Items that can be edited	Mode
Setting a life count type, life value, tool life counter, and tool data (T code and D code) However, the D code can be edited only if the ATC type is in use (bit 3 (TCT) of parameter No. 5040 = 1).	All modes
Adding tool numbers (T code)	MDI
Deleting all tool group data at a time	MDI
Deleting tool data (status, T code, and D code)	MDI
Selecting tool skip	MDI
Specifying to clear tool data (life re-set)	MDI

If no tool is registered with a tool group, none of a life count type, a tool life value, and a tool life counter value can be set for the tool group. First of all, add a tool number (T code).

FLYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

1 As for USING GROUP or NEXT GROUP editing:
   <1> During automatic operation (OP signal = “1” and bit 1 (TCI) of parameter No. 6804 = 1), only the tool life counter can be changed.
   <2> In the reset state (OP signal = “0” and RST signal “0”), the following editing operation stops life management because it cannot be continued.
      - Adding tool numbers (T code)
      - Deleting all tool group data at a time
      - Deleting tool data (status, T code, H code, and D code)
2 The following editing operations may set the tool change signal to “1”.
   - Selecting tool skip for the last tool.
   - Deleting tool numbers, resulting in any tool other than those whose life has expired or who have been skipped being not found in the tool group of interest.
3 The following editing operations may reset the tool change signal to “0”.
   - Adding tool numbers, leading to tools whose life has not expired being set in the tool group of interest.
   - Selecting tool clear.

Procedure

- Setting a life count type, tool life value, tool life counter, tool data, arbitrary group number, and remaining set value

Setting a life count type, tool life value, tool life counter, tool data, arbitrary group number, and remaining set value

Method 1

1 Place the cursor on the desired item.
2 Enter a value from the keypad.
3 Press soft key [INPUT].

Method 2

1 Place the cursor on the desired item.
2 Enter a value from the keypad.
3 Press

.

NOTE

1 Changing a tool life value or tool life counter does not affect the tool status or tool change signal.
2 Changing a life count type causes the tool life value and too life counter to be reset to 0.

M

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1) and the following parameters are enabled, arbitrary groups and remaining set values can be specified

OPTION GROUP : Arbitrary group number (if bit 5 (TGN) of parameter No. 6802 = 1)
RESET COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

T

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1)
Turret type (bit 3 (TCT) of parameter No. 5040 = 0)
OPTION GROUP : Cannot be set.
RESET COUNT : It is possible to set a remaining set value to be used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1).
ATC type (bit 3 (TCT) of parameter No. 5040 = 1)
OPTION GROUP : Arbitrary group numbers can be set (if bit 5 (TGN) of parameter No. 6802 = 1).
RESET COUNT : It is possible to set a remaining set value to be used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1).

- Adding tool numbers

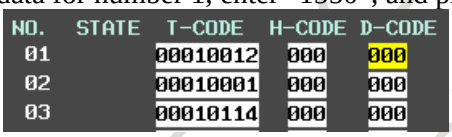
Tool numbers can be added to a tool group as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) just before a tool number to be added.
3 Enter the tool number from the key pad.
4 Press soft key [INSERT].

(Example)

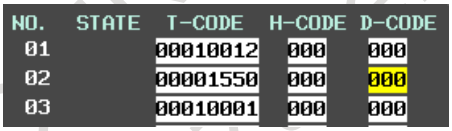
Adding tool number 1550 between numbers 1 and 2 (for the M series)

1 Move the cursor to the data for number 1, enter “1550”, and press [INSERT].



NO.	STATE	T-CODE	H-CODE	D-CODE
01		00010012	000	000
02		00010001	000	000
03		00010114	000	000

2 The entered T code 1550 is inserted in the position of number 2. The H and D codes are reset to 0.



NO.	STATE	T-CODE	H-CODE	D-CODE
01		00010012	000	000
02		00001550	000	000
03		00010001	000	000

- Deleting all tool group data at a time

All tool group data can be deleted at a time as follows:

1 Select the MDI mode.
2 Select the tool group from which you want to delete all tool data at a time.
3 Press soft key [DELETE].
4 Press soft key [GROUP].
5 Press soft key [EXEC].

- Deleting tool data

Tool data can be deleted from a tool group as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) you want to delete.
3 Press soft key [DELETE].
4 Press soft key [<CURS>].

NOTE

1 Deleting all tools from a tool group is equivalent to deleting the tool group itself.
2 Deleting a tool indicated with @ (in use) results in @ being moved to the previous tool whose life has expired or which has been skipped.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Selecting tool skip

Tool data can be placed in a skip state as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) of a tool you want to skip.
3 Press soft key [STATE].
4 Press soft key [SKIP].

- Specifying to clear tool data (life re-set)

Tool data state can be cleared as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) for a tool you want to clear.
3 Press soft key [STATE].
4 Press soft key [CLEAR].

- Selecting a tool group

A tool group can be selected as follows:

Method 1

1 Enter a tool group number from the keypad.
2 Press soft key [NO.SRH].

NOTE

If arbitrary group numbers are enabled, a tool group is selected by searching for an arbitrary group number rather than the tool group number.

Method 2

1 Press page key [PAGE ↑] or [PAGE ↓] to display the target tool group.

- Switching to the list screen

The tool life management (list screen) can be resumed as follows:

1 Press soft key [END].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.15 Displaying and Setting Pattern Data Inputs

Described below are a method for displaying machining menus (pattern menus) created by machine tool builders and a method for setting them. The descriptions are based on an example. For actual pattern menus and pattern data, refer to manuals released by respective machine tool builders.

Displaying pattern data and pattern menus

Given below is a procedure for displaying pattern menus.

1 Press function key [QPS SET].

2 Press the continuous menu key [▷].

3 Press soft key [PATTERN MENU].
The pattern menu screen shown below appears.

Fig. 12.3.15 (a) Pattern menu screen (10.4-inch display unit)

On this screen, a pattern to be used can be selected.
The following two methods can be used to select patterns.

- Using the cursor

  1 Move the cursor to a pattern name you want to select, using cursor key [↑] or [↓], and then press soft key [SELECT] or [INPUT].

- Specifying a pattern number

  1 Enter a number displayed at the left side of a pattern name, and press soft key [SELECT] or [INPUT].
  The custom macro screen (pattern data screen) shown below appears.

FRYER MACHINE TOOLS INC

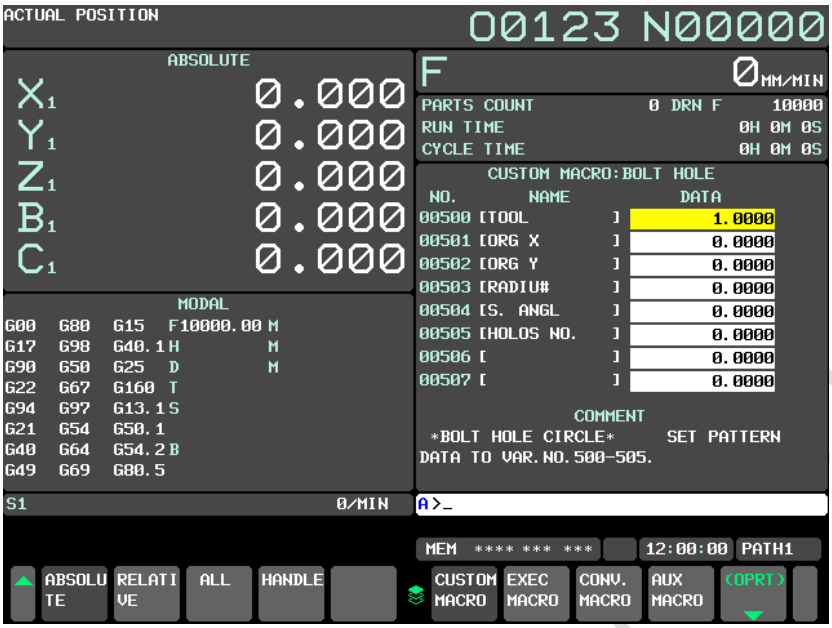


Fig. 12.3.15 (b) Custom macro screen (pattern data) (10.4-inch display unit)

2 Enter the necessary pattern data, and press INPUT.

3 After entering all necessary data, select the MEMORY mode and press the cycle start button. Machining begins.

Explanation

- **Explanations about the pattern menu screen**

HOLE PATTERN

An arbitrary character string consisting of 12 or less characters can be displayed as a menu title.

BOLT HOLE

An arbitrary character string consisting of 10 or less characters can be displayed as a pattern name.

Machine tool builders should program menu title and pattern name character strings, using custom macros, and save them in program memory.

- **Explanations about the custom macro screen (pattern data screen)**

BOLT HOLE

An arbitrary character string consisting of 12 or less characters can be displayed as a pattern data title.

TOOL

An arbitrary character string consisting of 10 or less characters can be displayed as a variable name.

BOLT HOLE CIRCLE

The comment text consists of nine 12-character blocks, or the comment can be up to 12 lines (10.4-inch display unit) with one block counted as one line.

Machine tool builders should program variable name and comment text character strings, using custom macros, and save them in program memory.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.16 Setting and Displaying Data When the Tool Offset for Milling and Turning Function Is Enabled

T

The tool offset for milling and turning function enables offset data to be displayed and operated on a tool offset screen common to milling and turning systems.

Setting and displaying tool offset values

Procedure

1 Press function key .

In multi-path control, select the path for which to set tool offset values, using the path selection switch in advance.

2 Press chapter selection soft key [OFFSET].

Alternatively, press function key several times until the tool offset screen appears.

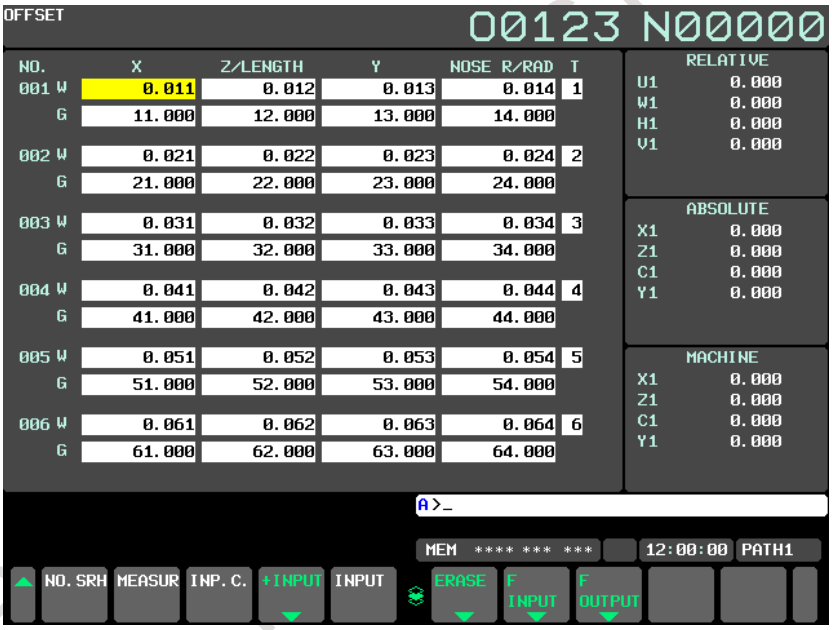


Fig. 12.3.16 (a) Tool offset screen (10.4-inch display unit)

X	: Set the X-axis tool offset value.
Z/LENGTH	: Set the Z-axis tool offset value or tool length compensation value.
Y	: Set the Y-axis tool offset value.
NOSE R/RAD	: Set the tool nose radius compensation value or tool radius compensation value.
T	: Set the virtual tool nose direction.

3 Press page and cursor keys to move the cursor to the offset value you want to set or change.
Alternatively, enter the number of the offset value you want to set or change and press soft key [NO.SRH].

4 To set the offset value, enter the value and press soft key [INPUT].
To change the offset value, enter a value to be incremented to or decremented from the current setting and press soft key [+INPUT].
Alternatively, enter a new offset value and press soft key [INPUT].

FRYER SYSTEMS INC

Explanation

- Decimal point programming

The offset value can be entered with a decimal point.

- Another setting method

The tool offset value can be input/output also by using external I/O devices.
See Chapter 8, “DATA I/O”, in Part III.
The tool offset value can also be set by using the direct input of offset value measured B, tool offset value counter input, and other functions.

- Number of tool offsets

128 tool offsets are available as standard.
The number of tool offsets can be optionally expanded to 200.
In a multi-path system, one of the above numbers of tool offsets can be selected for each path.

- Disabling offset value input

By setting bits 0 (WOF) and 1 (GOF) of parameter No. 3290, the input of tool wear compensation values and tool geometry compensation values is disabled.
The input of the tool offset values in any specified range from the MDI can be disabled by setting the number of the first target tool offset value in parameter No. 3294 and the number of target tool offset values starting from the first value in parameter No. 3295.
If offset values are entered continuously, they are handled as follows:

1) If an offset value that can be input is input, then an offset value that cannot be input is input, a warning is output, but the offset value in the input enable range is set.
2) If an offset value that cannot be input is input, then an offset value that can be input is input, a warning is output, and the offset values are not set.

- NOSE R/RAD, T display

When the tool nose radius compensation is disabled (bit 7 (NCR) of parameter No. 8136 is 1), NOSE R/RAD, T are not displayed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.17 Eco Setting Screen

Select a desired Eco level before starting machining, after comparing data of power consumption and cycletime for each Eco level.

Displaying Eco setting screen

Procedure

1 Press function key .

2 Press the continuous menu key several times to display soft key [ECO].

3 Press soft key [ECO].

4 Press soft key [ECO SET].

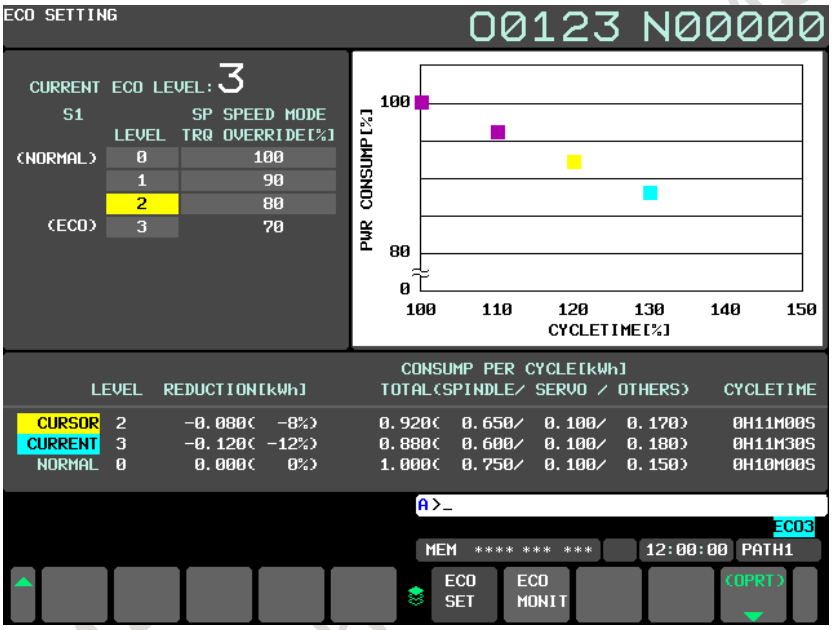


Fig. 12.3.17 (a) Eco setting screen

Explanation

- **Display for select an Eco level**

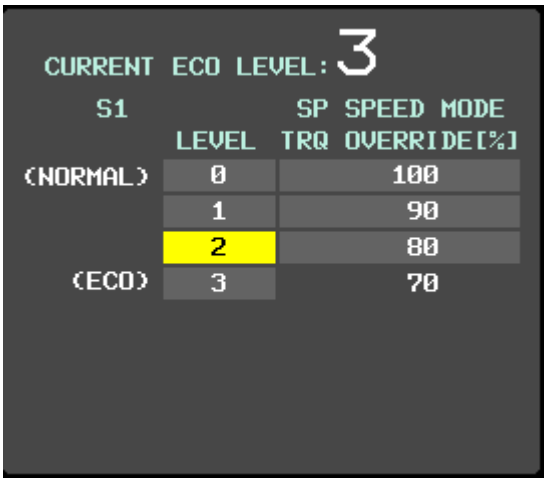


Fig. 12.3.17 (b) Display for select an Eco level

Select an Eco level here.

FRYER MA ROBOMS INC

The selected Eco level for the current path is also displayed.

Items	Contents
Current Eco level	The selected Eco level is displayed.
Level	Available Eco levels are displayed. Level 0 means normal machining. Highlight a desired Eco level and press operation menu soft key [SELECT], and the selected Eco level is changed.
Torque override	Torque overrides for each level are displayed. Torque overrides for spindle speed control mode, spindle synchronization control mode and rigid tapping mode are displayed. Since only one mode can be displayed at a time, pressing soft keys [SPNDL SPEED][SPNDL SYNC][RIGID TAP] change displayed mode. Note that, when each function is disabled, torque overrides of spindle synchronization control mode or rigid tapping mode are not displayed. If multiple spindles belong to the current path, pressing MDI keys PAGE ⇧ PAGE ⇩ change spindles, whose torque overrides are displayed.

NOTE

When an invalid value (out of 0–100%) is specified in the parameter No.11397, 11398 or 5260, torque overrides are displayed as “100” in red, excluding the torque override for Eco level 0.

- **Graphic display for data**

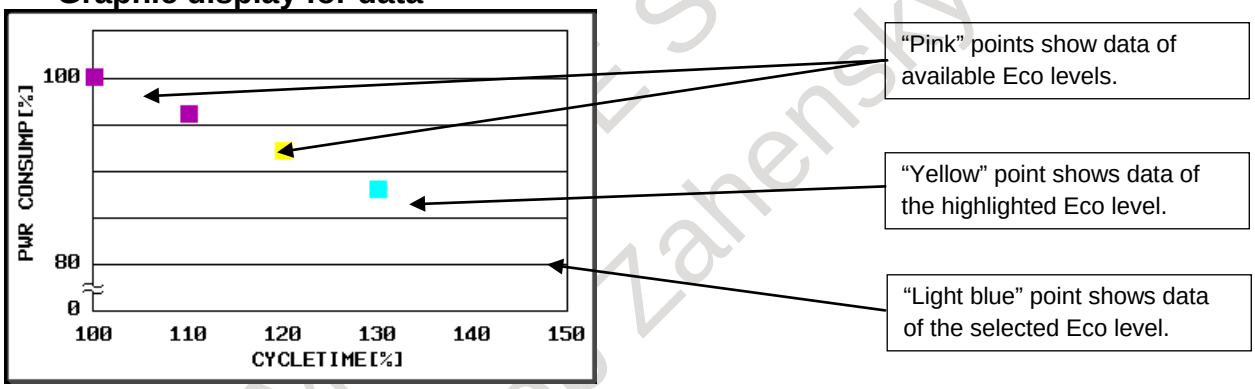


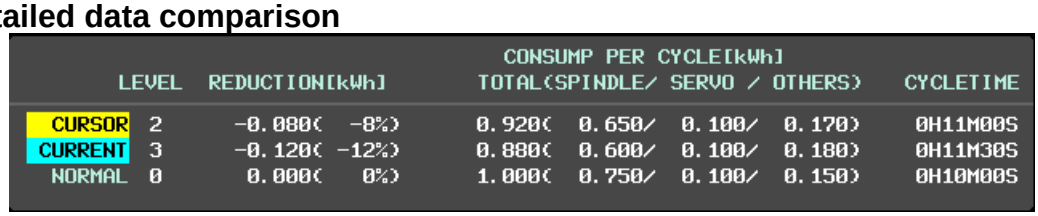
Fig. 12.3.17 (c) Graph for eco setting

Data of each Eco level are displayed in power consumption vs. cycletime.
Data of the current path are displayed.

NOTE

Nothing is displayed without data of Eco level 0.

- **Detailed data comparison**



	LEVEL	REDUCTION[kWh]	CONSUMP PER CYCLE[kWh] TOTAL(SPINDLE/ SERVO / OTHERS)	CYCLETIME
CURSOR	2	-0.080(-8%)	0.920(0.650/ 0.100/ 0.170)	0H11M00S
CURRENT	3	-0.120(-12%)	0.880(0.600/ 0.100/ 0.180)	0H11M30S
NORMAL	0	0.000(0%)	1.000(0.750/ 0.100/ 0.150)	0H10M00S

Fig. 12.3.17 (d) Comparison of data for eco setting

Detailed data of power consumption and cycletime of each Eco level can be compared here in order to select a desired Eco level.
Data of the current path are displayed.

FRYER MACHINE SYSTEMS INC Rob Zahensky

Items	Contents
Level	Eco levels are displayed.
Reduction	Reductions in power consumption per cycle are displayed. Reductions are calculated by using power consumption at normal machining (Eco level 0) as the base.
Power consumptions per cycle (spindle/servo/others)	Total power consumptions per cycle are displayed. Power consumptions in spindles, servos and others (peripheral devices) are also displayed separately.
Cycletime	Cycletimes are displayed.

NOTE
1 Power consumption data obtained by actual machining is displayed. Therefore, 0 is displayed at the first machining or soon after deleting data. Though these data depends on parameter settings, machining circumstances and so on, soon after changing settings, data obtained before the change is displayed.
2 Here, a “cycle” means one automatic operation.

Operation on Eco setting screen

Procedure

- Select of an Eco level

Select a desired Eco level with comparing among data of power consumption and cycletime at each Eco level.

1 Select MDI mode.
2 Highlight a desired Eco level.
3 Press soft key [(OPRT)].
4 Press soft key [SELECT] on operation menu.
The selected Eco level is displayed in “CURRENT ECO LEVEL” and the select is completed.

NOTE
Select an Eco level for each path when the select is available for each path (bit 1 (EEP) of the parameter No.24303 is set to 1).

- Deleting data of power consumption and cycletime at each Eco level

Disused data of power consumption and cycletime can be deleted individually or in a batch.
Both individual and batch deletes are effective only on data of the current path.

1 Highlight an Eco level whose data is going to be deleted.
Batch delete does not require this operation.
2 Press soft key [(OPRT)].
3 Press soft key [DELETE] or [DELETE ALL] on operation menu.
If [DELETE] is pressed, individual data is deleted.
If [DELETE ALL] is pressed, all data of the current path are deleted.
Select menu [EXEC]/[CANCEL] is displayed.
4 Press soft key [EXEC].
Data is deleted.

- Switching displayed unit of power consumption

Displayed unit of power consumption can be switched to “kWh” or “Wh”.
Soft key [kWh/Wh] switches the unit to “kWh” or “Wh”.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Switching spindle mode whose torque overrides are displayed

The spindle mode whose torque overrides are displayed can be switched by soft keys [SPNDL SPEED], [SPNDL SYNC] and [RIGID TAP].
When each function is disabled, torque overrides of spindle synchronization control mode or rigid tapping mode are not displayed.

- Switching spindle whose torque overrides are displayed

When multiple spindles belong to the current path, MDI keys [PAGE ⇧] [PAGE ⇩] can switch the spindle whose torque overrides are displayed.

- Change of torque overrides

Torque override for Eco level 3 can be changed.

1 Set bit 0 (PWE) of the parameter No.8900 to 1.
2 Highlight the torque override for Eco level 3.
3 Select MDI mode or enter state emergency stop for all paths.
4 Release spindle synchronization mode for all paths.
5 Input value by MDI keys.
Available values are integers from 1 to 100.
6 Press soft key [INPUT] or MDI key [INPUT].
Torque overrides for each Eco level are changed.

⚠ CAUTION
Machine or work can be damaged by out-of-synchronization when torque overrides of spindles in synchronization control mode differ from each other. This is because this function differs the time constant for acceleration/deceleration in spindle synchronization mode.
Set a common torque override for spindles belonging to the same group of synchronization.

NOTE
1 Use this operation only for deciding the point of the least power consumption.Please refer to “Energy Saving Level Selecting Function “ of CONNECTION MANUAL (FUNCTION) (B-64693EN-1) for details.
2 When select of an Eco level is available for each path (bit 1 (EEP) of the parameter No.24303 is set to 1), operations 3 and 4 are needed only for the current path.
3 When bit 0 (PWE) of the parameter No.8900 is set to 1, the torque override cannot be highlighted.

FRYER MACHINE SYSTEMS INC
Rob Zamorsky

12.3.18 Eco Monitoring Screen

Power consumption of machine can be observed comprehensively, such as effects of eco-machining and time transition of power consumption.

Displaying Eco monitoring screen

Procedure

1 Press function key .

2 Press the continuous menu key several times to display soft key [ECO].

3 Press soft key [ECO].

4 Press soft key [ECO MONIT].

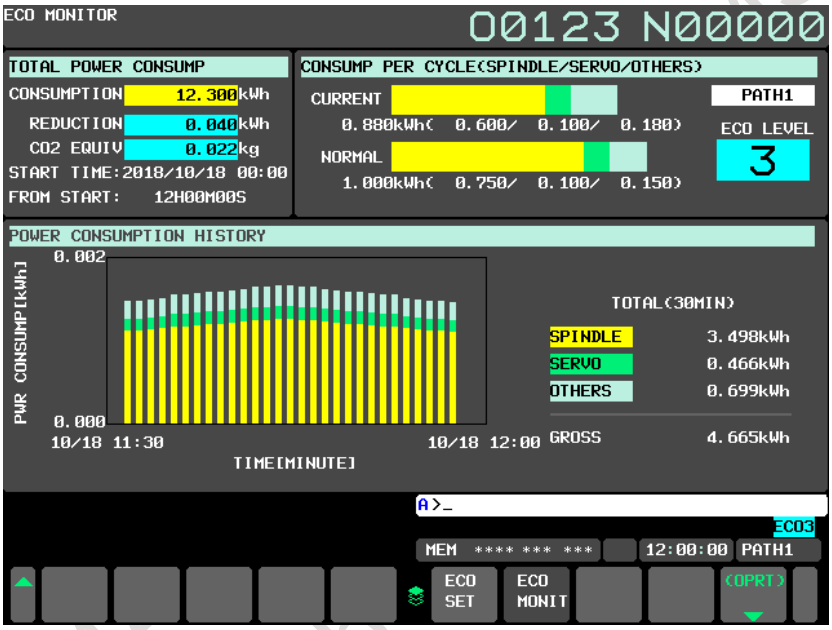


Fig. 12.3.18 (a) Eco monitoring screen

Explanation

- Display of total power consumption

Total power consumption of a machine is displayed, which has been accumulated since the last clear operation of total power consumption. This is not cleared even if power is off.

Items	Contents
Consumption	Accumulated power consumption of a whole machine is displayed.
Reduction	Difference between power consumption of normal machining and that of eco-machining is accumulated. The accumulated data of all paths are added. See “Example” below.
CO2 equivalent	Carbon-dioxide emission equivalent to “Reduction” is displayed.

NOTE

The conversion factor for CO2 equivalent is specified by the parameter No.11399.

- Power consumption per cycle (spindle/servo/others)

Power consumption per cycle at the current Eco level is displayed in bar graphs, compared with that of normal machining (Eco level 0).

Items	Contents
Current	The power consumption of a running cycle is displayed and updated in real time. When nothing is operated, the power consumption of the last cycle is displayed. With multiple paths, summed data of all paths as well as data of the current path can be displayed. Note that the summed data can include only data of paths where both “Normal” and “Current” have non-zero values. Soft keys [EACH PATH] and [ALL PATH] switch the displayed data.
Normal	The power consumption of the last cycle at Eco level 0 is displayed. With multiple paths, summed data of all paths as well as data of the current path can be displayed. Note that the summed data can include only data of paths where both “Normal” and “Current” have non-zero values. Soft keys [EACH PATH] and [ALL PATH] switch the displayed data.
Eco level	The selected Eco level is displayed. When eco-machining is ineffective, “OFF” is displayed. The Eco level of the current path is displayed.

NOTE

When select of an Eco level is available for each path (bit 1 (EEP) of the parameter No.24303 is set to 1), “Eco level” is not displayed during displaying summed data of all paths.

- History of power consumption

Time transition of power consumption is displayed in bar graphs, with power consumption categorized into that in spindles, servos and others.

Items	Contents
History of power consumption	Unit selected on operation menu decides displayed values as follows

Units	Displayed values
MINUTE	Power consumption for a minute is displayed in a bar graph. Up to 30 minutes’ power consumption can be displayed.
30MIN	Power consumption for 30 minutes is displayed in a bar graph. Up to 15 hours’ power consumption can be displayed.
HOUR	Power consumption for an hour is displayed in a bar graph. Up to 30 hours’ power consumption can be displayed.
DAY	Power consumption for a day is displayed in a bar graph. Up to 30 days’ power consumption can be displayed.

Items	Contents
TOTAL	Total power consumption in each unit is displayed as follows.

Units	Contents
MINUTE	Power consumption for last 30 minutes is displayed.
30MIN	Power consumption for last 15 hours is displayed.
HOUR	Power consumption for last 30 hours is displayed.
DAY	Power consumption for last 30 days is displayed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

1 Power consumption data obtained by actual machining are displayed. Therefore, 0 is displayed at the first machining or soon after deleting data. Though these data depend on parameter settings, machining circumstances and so on, soon after changing settings, data obtained before the change are displayed.

2 Difference between power consumption of normal machining and that of the current Eco level is added to “Reduction” in total power consumption at every cycle end. When no normal cycle finishes after changing parameter settings, machining programs and so on, notice that reduction calculated from power consumption data of old settings is added to “Reduction”. In addition, with no data of normal machining, nothing is added to “Reduction”. See also “Example” below.

Example

The values of “Consumption” and “Reduction” are calculated according to the examples below:

1 The case that power consumption of Eco level 0 is equal to 10kWh:

A Given that 8kWh has been consumed at Eco level 2 after clearing total power consumptions.

- “Consumption” = 0 + 8 = 8kWh
- “Reduction” = 0 + (10 - 8) = 2kWh

B Given that, following A. above, 7kWh has been consumed at Eco level 3.

- “Consumption” = 8 + 7 = 15kWh
- “Reduction” = 2 + (10 - 7) = 5kWh

2 The case that a cycle with Eco level 0 has not finished since main program was changed.

A Given that, following 1.B. above, operating new program has consumed 12kWh at Eco level 2 (Since main program has been changed, power consumption can be larger than that at Eco level 0.).

- “Consumption” = 15 + 12 = 27kWh
- “Reduction” = 5 + (10 - 12) = 3kWh

(This is calculated by using data of Eco level 0 obtained before main program was changed.)

3 The case that there is no data of Eco level 0

A Given that, following 2.A above, data of Eco level 0 has been deleted on Eco setting screen and then, 7kWh has been consumed at Eco level 3.

- “Consumption” = 27 + 7 = 34kWh
- “Reduction” = 3kWh (Nothing is added without data of Eco level 0.)

With multiple paths, the summation of all paths can be displayed in “Consumption” and “Reduction”, these values of each path calculated like examples above.
For example, when consumption and reduction of path 1 are equal to 10kWh and 2kWh, and those of path 2 are equal to 15kWh and 3kWh, displayed values are as follows:

- “Consumption” = 10 + 15 = 25kWh
- “Reduction” = 2 + 3 = 5kWh.

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

Operation on Eco setting screen

Procedure

- Switching displayed time unit of history of power consumption

Displayed time unit of history of power consumption can be switched into every minute, every 30 minutes, every hour and every day.

1 Press soft key [(OPRT)].
2 Press one of soft keys [MINUTE], [30MIN], [HOUR] and [DAY].
Displayed time unit changes.

- Turning on/off of eco-machining

Eco-machining and normal machining can be switched by soft key [ECO OFF]/[ECO ON].
Given that Eco level 3 is selected, for example,

- Normal machining is selected by pressing soft key [ECO OFF]. Here, the selected Eco level does not change (Eco level 3 is kept).
- After the operation above, eco-machining of Eco level 3 is selected by pressing soft key [ECO ON].

1 Select MDI mode for all paths.
2 Release spindle synchronization mode for all paths.
3 Press soft key [(OPRT)].
4 Press soft key [ECO OFF]/[ECO ON].
In case of turning off eco-machining, Eco indicator “ECOX” in Warning area is turned off. Note that the current Eco level displayed on Eco setting screen does not change because the selected Eco level does not change in this operation.
In case of turning on eco-machining, Eco indicator “ECOX” in Warning area is turned on. Note that nothing is displayed when Eco level 0 is selected.

NOTE
When select of an Eco level is available for each path (bit 1 (EEP) of the parameter No.24303 is set to 1), operations 1 and 2 are needed only for the current path.

- Switching displayed unit of power consumption

Displayed unit of power consumption can be switched to “kWh” or “Wh”.
Soft key [kWh/Wh] switches the unit to “kWh” or “Wh”.

- Clearing total power consumption

Displayed total power consumption can be set to 0 and new accumulation is started.

1 Press soft key [(OPRT)].
2 Press soft key [TOTAL CLEAR].
Select menu [EXEC]/[CANCEL] is displayed.
3 Press soft key [EXEC].
Displayed total power consumptions are set to 0 and “start time” is set to current time

- Switching displayed data in “Power consumption per cycle”

With multiple paths, displayed data in “Power consumption per cycle” can be switched into data of each path or summed data of all paths.
Soft key [EACH PATH]/[ALL PATH] can switch the data.

1 Press soft key [(OPRT)].
2 Press soft key [EACH PATH]/[ALL PATH].
Data in “Power consumption per cycle” is changed.

FRIEFMACHINE SYSTEMS INC
Rob Zahensky

Limitation

Since this function uses clock, change in clock causes following results.
Note that change in clock requires power off.

- **Total power consumption**

"From start" changes according to change in clock, though "Consumption", "Reduction", "CO2 equivalent" and "Start time" does not change.

- **History of power consumption**

If clock changes backward, history of power consumption is entirely set to 0.
If clock changes forward, obtained power consumption and time do not change, though power consumption of jumped period is set to 0. If clock is advanced by an hour, for example, power consumption of the jumped hour is considered as 0.

Screens of a 15/19-inch display unit

12.3.19 Displaying and Entering Setting Data (15/19-inch Display Unit)

Data such as the TV check flag and punch code is set on the setting data screen. On this screen, the operator can also enable/disable parameter writing, enable/disable the automatic insertion of sequence numbers in program editing, and perform settings for the sequence number comparison and stop function.
See Section, "AUTOMATIC INSERTION OF SEQUENCE NUMBERS" for automatic insertion of sequence numbers.
See Subsection, "Sequence Number Comparison and Stop" for the sequence number comparison and stop function. This subsection describes how to set data.

Setting the setting data

Procedure

1 Select the MDI mode.

2 Press function key .

3 Press vertical soft key [SETTING] to display the setting data screen.
This screen consists of several pages.
Press page key or until the desired screen is displayed.
An example of the setting data screen is shown below.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

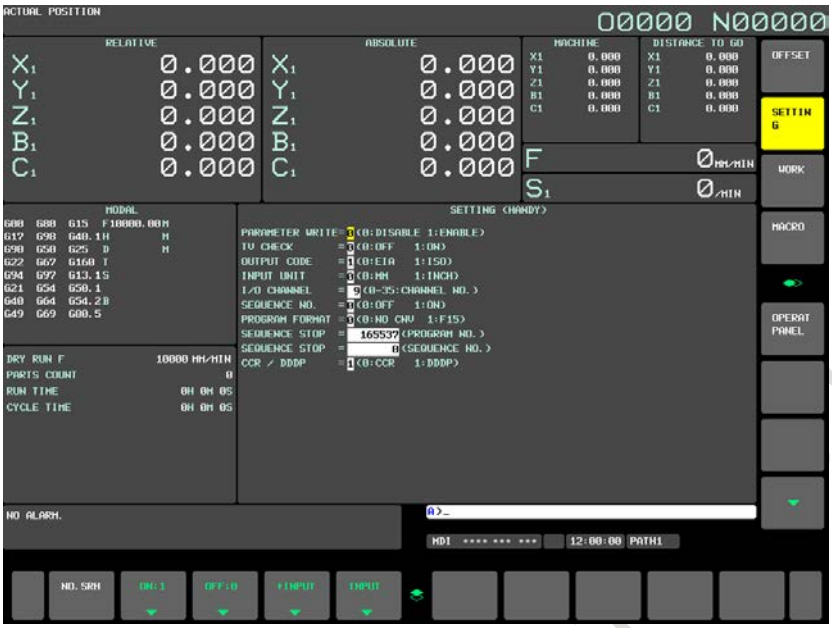


Fig. 12.3.19 (a) SETTING (HANDY) screen (15-inch display unit)

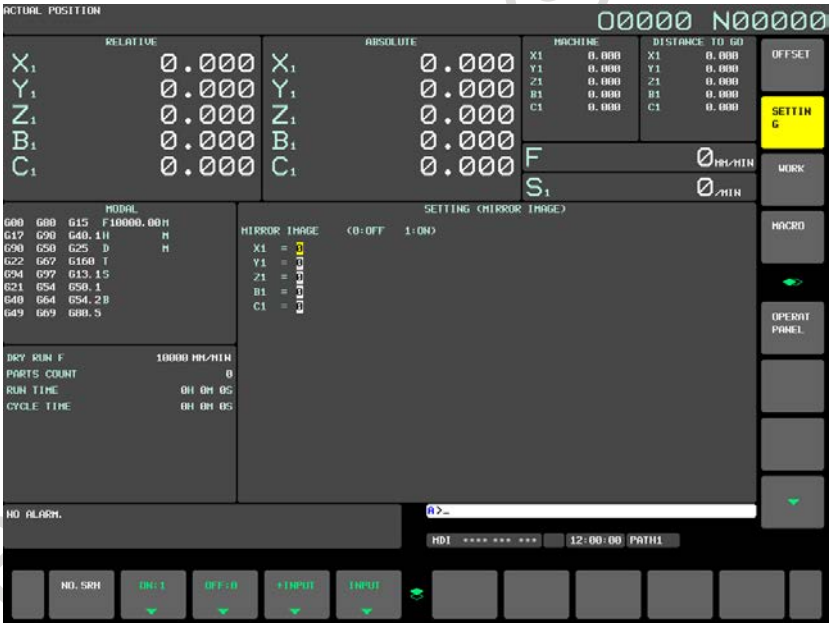


Fig. 12.3.19 (b) SETTING (MIRROR IMAGE) screen (15-inch display unit)

4 Move the cursor to the item to be changed by pressing cursor keys ↑ ↓ ← → .

5 Enter a new value and press horizontal soft key [INPUT].

FRYER SYSTEMS INC

Explanation

- PARAMETER WRITE

Setting whether parameter writing is enabled or disabled.
0 : Disabled
1 : Enabled

- TV CHECK

Setting to perform TV check.
0 : No TV check
1 : Perform TV check

- PUNCH CODE

Setting code when data is output through RS-232C interface.
0 : EIA code output
1 : ISO code output

- INPUT UNIT

Setting a program input unit, inch or metric system
0 : Metric
1 : Inch

- I/O CHANNEL

Using channel of RS-232C interface.
0 : Channel 0
1 : Channel 1
2 : Channel 2

- SEQUENCE NO.

Setting of whether to perform automatic insertion of the sequence number or not at program edit in the EDIT mode.
0 : Does not perform automatic sequence number insertion.
1 : Perform automatic sequence number insertion.

- PROGRAM FORMAT

Setting of whether to use the Series 15 format.
0: Uses the standard format.
1: Uses the Series 15 format.
For the Series 15 format, refer to Chapter, “MEMORY OPERATION USING Series 15 FORMAT” in the OPERATOR’S MANUAL (T series : B-64694EN-1) or Chapter, “MEMORY OPERATION USING Series 15 FORMAT” in the OPERATOR’S MANUAL (M series : B-64694EN-2).

- SEQUENCE STOP

Setting the sequence number with which the operation stops for the sequence number comparison and stop function and the number of the program to which the sequence number belongs.

T

- CCR / DDDP

This setting item is displayed only when the chamfering or corner R and the direct drawing dimension programming option are enabled. These functions cannot be use at same time. Setting of which function is enabled.
0:Chamfering or corner R is enabled.
1:Direct drawing dimensions programming function is enabled.

FRIEDMACHINE SYSTEMS INC
Rob Zahensky

- MIRROR IMAGE

Setting of mirror image ON/OFF for each axes.
0 : Mirror image off
1 : Mirror image on

- Others

Page key [PAGE ⇧] or [PAGE ⇩] can also be pressed to display the SETTING (TIMER) screen. See Subsection, “Displaying and Setting Run Time, Parts Count, and Time (15/19-inch Display Unit)” for this screen.

12.3.20 Sequence Number Comparison and Stop (15/19-inch Display Unit)

If a block containing a specified sequence number appears in the program being executed, operation enters single block mode after the block is executed.

Sequence number comparison and stop

Procedure

1 Select the MDI mode.
2 Press function key [SET OFS].
3 Press vertical soft key [SETTING].
4 Press page key [PAGE ⇧] or [PAGE ⇩] several times until the screen in the Fig. 12.3.20 (a) is displayed.

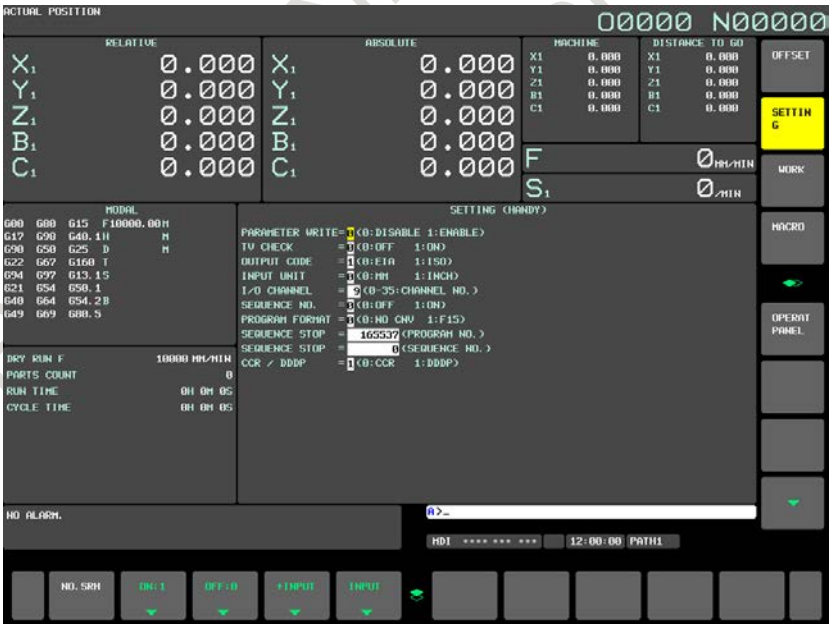


Fig. 12.3.20 (a) SETTING (HANDY) screen (15-inch display unit)

5 Enter in (PROGRAM NO.) for SEQUENCE STOP the number (1 to 99999999) of the program containing the sequence number with which operation stops.
6 Enter in (SEQUENCE NO.) for SEQUENCE STOP (with eight or less digits) the sequence number with which operation is stopped.
7 When automatic operation is executed, operation enters single block mode at the block containing the sequence number which has been set.

FRYER ... NE SYSTEMS INC ... nsky

Explanation

- Sequence number after the program is executed

After the specified sequence number is found during the execution of the program, the sequence number set for sequence number compensation and stop is decremented by one.

- Exceptional blocks

If the predetermined sequence number is found in a block in which all commands are those to be processed within the CNC control unit, the execution does not stop at that block.

[Example]
```
N1 #1=1 ;
N2 IF[#1 EQ 1]GOTO 08 ;
N3 GOTO 09 ;
N4 M98 P1000 ;
N5 M99 ;
```

In the example shown above, if the predetermined sequence number is found, the execution of the program does not stop.

- Stop in the canned cycle

If the predetermined sequence number is found in a block which has a canned cycle command, the execution of the program stops after the return operation is completed.

- When the same sequence number is found several times in the program

If the predetermined sequence number appears twice or more in a program, the execution of the program stops after the block in which the predetermined sequence number is found for the first time is executed.

- Block to be repeated a specified number of times

If the predetermined sequence number is found in a block which is to be executed repeatedly, the execution of the program stops after the block is executed specified times.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.21 Displaying and Setting Run Time, Parts Count, and Time (15/19-inch Display Unit)

Various run times, the total number of machined parts, number of parts required, and number of machined parts can be displayed.
This data can be set by parameters or on this screen (except for the total number of machined parts and the time during which the power is on, which can be set only by parameters).
This screen can also display the clock time. The time can be set on the screen.

Displaying and Setting Run Time, Parts Count and Time

Procedure

1 Select the MDI mode.

2 Press function key [OFS/SET].

3 Press vertical soft key [SETTING].

4 Press page key [PAGE ↑] or [PAGE ↓] several times until the screen in the Fig. 12.3.21 (a) is displayed.

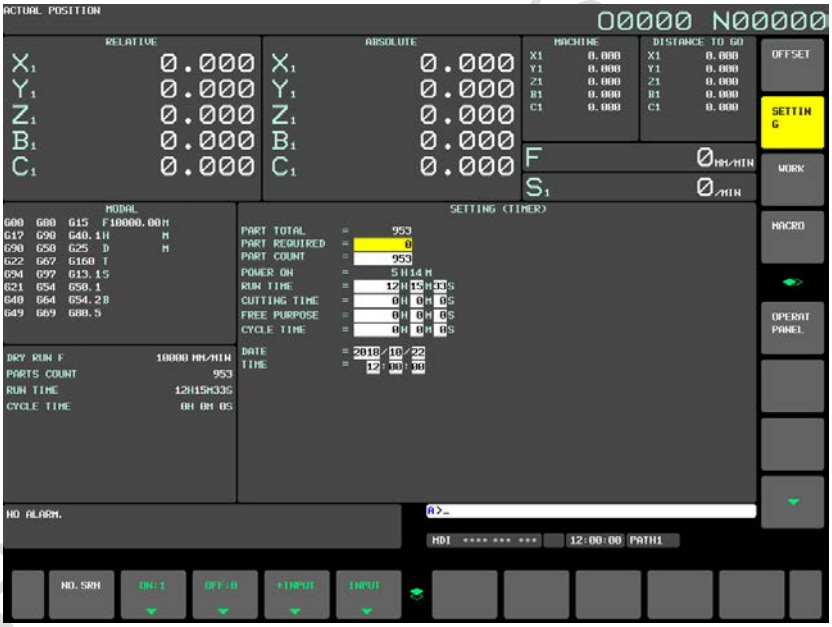


Fig. 12.3.21 (a) SETTING (TIMER) screen (15-inch display unit)

5 To set the number of parts required, move the cursor to PARTS REQUIRED and enter the number of parts to be machined.

6 To set the clock, move the cursor to DATE or TIME, enter a new date or time, then press horizontal soft key [INPUT].

Explanation

- PARTS TOTAL

This value is incremented by one when M02, M30, or an M code specified by parameter No. 6710 is executed. This value cannot be set on this screen. Set the value in parameter No. 6712.

- PARTS REQUIRED

It is used for setting the number of machined parts required.
When the "0" is set to it, there is no limitation to the number of parts.
Also, its setting can be made by the parameter No 6713.

FRYER SYSTEMS INC

- PARTS COUNT

This value is incremented by one when M02, M30, or an M code specified by parameter No. 6710 is executed. The value can also be set by parameter No. 6711. In general, this value is reset when it reaches the number of parts required. Refer to the manual issued by the machine tool builder for details.

- POWER ON

Displays the total time which the power is on. This value cannot be set on this screen but can be preset in parameter No. 6750.

- RUN TIME

Indicates the total run time during automatic operation, excluding the stop and feed hold time.
This value can be preset in parameter No. 6751 or No. 6752.

- CUTTING TIME

Displays the total time taken by cutting that involves cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03). This value can be preset in parameter No. 6753 or No. 6754.

- FREE PURPOSE

This value can be used, for example, as the total time during which coolant flows. Refer to the manual issued by the machine tool builder for details.

- CYCLE TIME

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is turned off.

- DATA and TIME

Displays the current date and time. The date and time can be set on this screen.

- Usage

When the command of M02 or M30 is executed, the total number of machined parts and the number of machined parts are incremented by one. Therefore, create the program so that M02 or M30 is executed every time the processing of one part is completed. Furthermore, if an M code set to the parameter No. 6710 is executed, counting is made in the similar manner. Also, it is possible to disable counting even if M02 or M30 is executed (bit 0 (PCM) of parameter No. 6700 is set to 1). For details, see the manual issued by machine tool builders.

Limitation

- Run time and part count settings

Negative value cannot be set. Also, the setting of "M" and "S" of run time is valid from 0 to 59.
Negative value may not be set to the total number of machined parts.

- Time settings

Neither negative value nor the value exceeding the value in the following table can be set.

Table 12.3.21

Item	Maximum value	Item	Maximum value
Year	2096	Hour	23
Month	12	Minute	59
Day	31	Second	59

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.22 Displaying and Setting the Workpiece Origin Offset Value (15/19-inch Display Unit)

Displays the workpiece origin offset for each workpiece coordinate system (G54 to G59, G54.1 P1 to G54.1 P48 and G54.1 P1 to G54.1 P300) and external workpiece origin offset.
The workpiece origin offset value and external workpiece origin offset value can be set on this screen.

Displaying and setting the workpiece origin offset value

Procedure

1 Press function key .

2 Press vertical soft key [WORK].
The workpiece coordinate system setting screen is displayed.

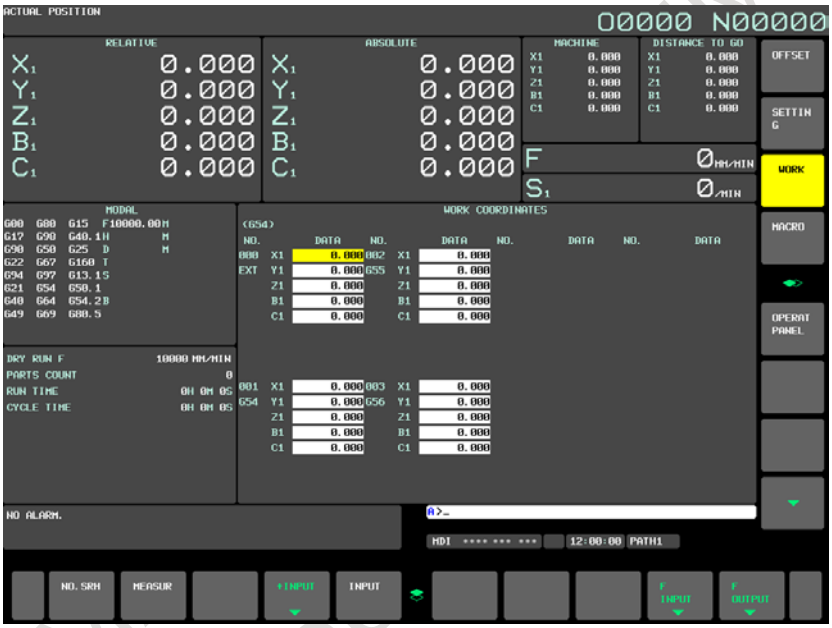


Fig. 12.3.22 (a) WORK COORDINATES screen (15-inch display unit)

3 The screen for displaying the workpiece origin offset values consists of two or more pages.
Display a desired page in either of the following two ways:

- Press the page key or .
- Enter the workpiece coordinate system number (0 : external workpiece origin offset, 1 to 6: workpiece coordinate systems G54 to G59, P1 to P48 : workpiece coordinate systems G54.1 P1 to G54.1 P48, P1 to P300 : workpiece coordinate systems G54.1 P1 to G54.1 P300) and press horizontal soft key [NO.SRH].

4 Turn off the data protection key to enable writing.

5 Move the cursor to the workpiece origin offset to be changed.

6 Enter a desired value by pressing numeric keys, then press horizontal soft key [INPUT]. The entered value is specified in the workpiece origin offset value. Or, by entering a desired value with numeric keys and pressing horizontal soft key [+INPUT], the entered value can be added to the previous offset value.

7 Repeat steps 5 and 6 to change other offset values.

8 Turn on the data protection key to disable writing.

FRYER MACHINE TOOLS INC

12.3.23 Direct Input of Workpiece Origin Offset Value Measured (15/19-inch Display Unit)

This function is used to compensate for the difference between the programmed workpiece coordinate system and the actual workpiece coordinate system. The measured offset for the origin of the workpiece coordinate system can be input on the screen such that the command values match the actual dimensions. Selecting the new coordinate system matches the programmed coordinate system with the actual coordinate system.

Direct input of workpiece origin offset value measured

Procedure

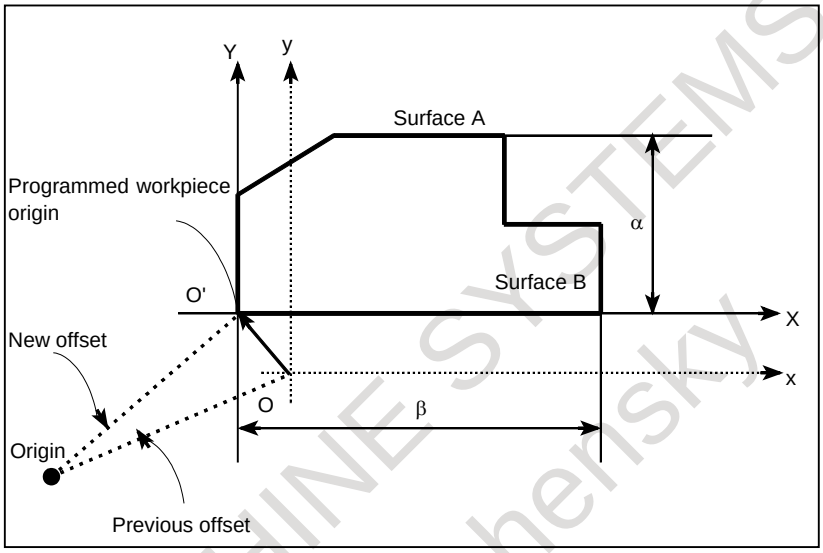


Fig. 12.3.23 (a)

1 When the workpiece is shaped as shown Fig. 12.3.23 (a), position the reference tool manually until it touches surface A of the workpiece.
2 Retract the tool without changing the Y coordinate.
3 Measure distance α between surface A and the programmed origin of the workpiece coordinate system as shown Fig. 12.3.23 (a).
4 Press function key .
5 To display the WORK COORDINATES screen, press vertical soft key [WORK].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

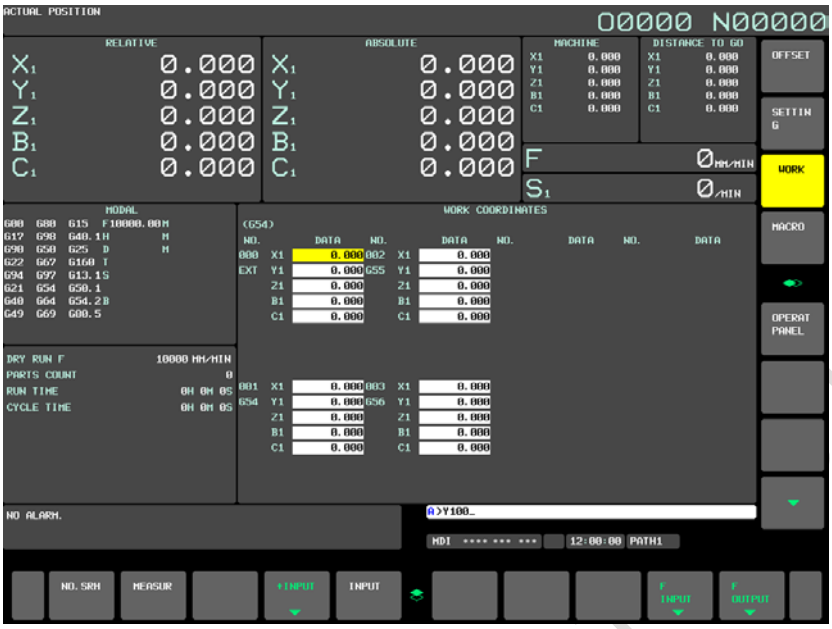


Fig. 12.3.23 (b) WORK COORDINATES screen (15-inch display unit)

6 Position the cursor to the workpiece origin offset value to be set.
7 Press the address key for the axis along which the offset is to be set (Y-axis in this example).
8 Enter the measured value (α) then press horizontal soft key [MEASUR].
9 Move the reference tool manually until it touches surface B of the workpiece.
10 Retract the tool without changing the X coordinate.
11 Enter the measured value of distance (β) to surface B as an X value as with steps 7 and 8.

Limitation

- Consecutive input

Offsets for two or more axes cannot be input at the same time.

- During program execution

This function cannot be used while a program is being executed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.24 Displaying and Setting Custom Macro Common Variables (15/19-inch Display Unit)

Displays common variables (#100 to #149 or #100 to #199, and #500 to #531 or #500 to #999) on the screen.
The values for variables can be set on this screen.
Relative coordinates can also be set to variables.

Displaying and setting custom macro common variables

Procedure

1 Press function key [SET].

2 Press vertical soft key [MACRO].
The screen in the Fig. 12.3.24 (a) is displayed.

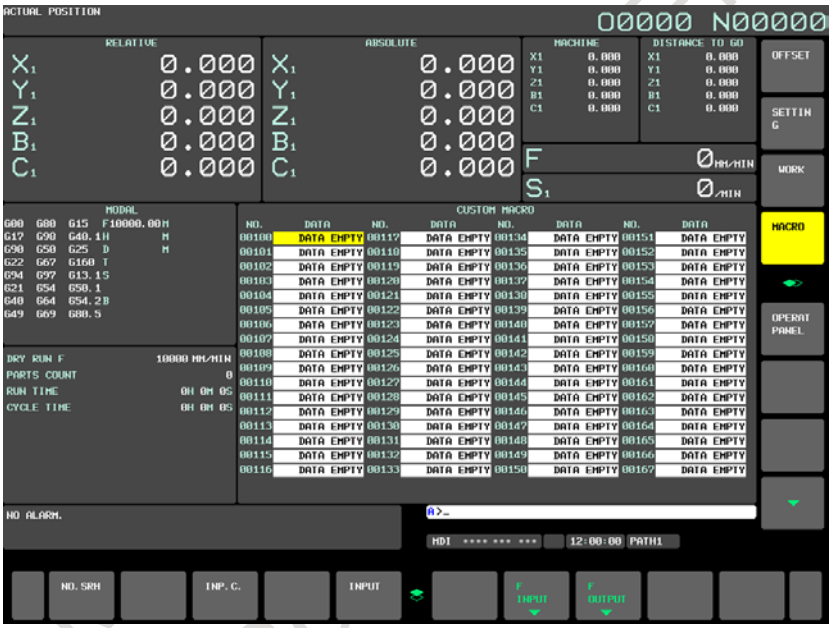


Fig. 12.3.24 (a) CUSTOM MACRO screen (15-inch display unit)

3 Move the cursor to the variable number to set using either of the following methods:
- Enter the variable number and press horizontal soft key [NO.SRH].
- Move the cursor to the variable number to set by pressing page keys [PAGE] and/or [PAGE] and cursor keys [↑], [↓], [←], and/or [→].

4 Enter data with numeric keys and press horizontal soft key [INPUT].

5 To set a relative coordinate in a variable, press address key [X], [Y], or [Z], then press horizontal soft key [INP.C.].

6 To set a blank in a variable, just press horizontal soft key [INPUT]. The value field for the variable becomes blank.

FRYER SYSTEMS INC

Explanation

If the value of a variable produced by an operation is not displayable, an indication below is provided.

When the significant number of digits is 12 (with bit 0 (F16) of parameter No. 6008 set to 0):

Variable value range	Variable value indication
0 < Variable value < +0.00000000001	+Underflow
0 > Variable value > -0.00000000001	-Underflow
Variable value > 999999999999	+Overflow
Variable value < -999999999999	-Overflow

When the significant number of digits is 8 (with bit 0 (F16) of parameter No. 6008 set to 1):

Variable value range	Variable value indication
0 < Variable value < +0.0000001	+Underflow
0 > Variable value > -0.0000001	-Underflow
Variable value > 99999999	+Overflow
Variable value < -99999999	-Overflow

12.3.25 Displaying and Setting Real Time Custom Macro Data (15/19-inch Display Unit)

Real time macro variables (RTM variables) are dedicated to real time custom macros.
RTM variables are divided into temporary real time macro variables (temporary RTM variables) and permanent real time macro variables (permanent RTM variables).
The values of permanent RTM variables are kept stored after the power is turned off.
Temporary RTM variables are cleared to 0 when the power is turned off.

System variables (DI/DO variables) dedicated to real time custom macros are used to read and write PMC interface signals. Data can be read and written in bit and byte units. Before writing a signal, release the protection of the corresponding variable on the PMC signal protect screen.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Displaying and setting real time custom macro variables

Procedure

1 Press function key [SYSTEM].

2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [R.TIME MACRO]. The screen shown below appears.

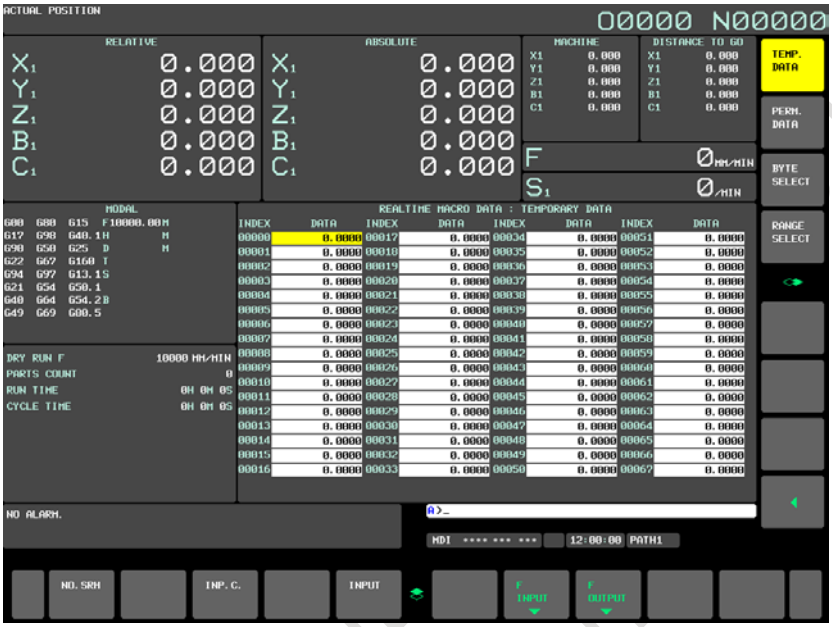


Fig. 12.3.25 (a) REAL TIME MACRO screen (15-inch display unit)

3 To display or set real time custom macro variables of which values are not stored at power-off, press vertical soft key [TEMP. DATA].

4 To display or set real time custom macro variables of which values are stored at power-off, press vertical soft key [PERM. DATA].

5 Move the cursor to the number of a real time custom macro variable you want to set using either of the following methods:

- Enter the number of a real time custom macro variable and press horizontal soft key [NO. SRH].
- Move the cursor to the number of a real time custom macro variable you want to set by pressing page keys [PAGE] and/or [PAGE] and cursor keys [↑], [↓], [←], and/or [→].

6 Enter data.

FRYER MACHINE INC Rob Zaher

Displaying and setting DI/DO variables

Procedure

For setting in byte units:

1 Press function key [SYSTEM].

2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [R.TIME MACRO].

3 Press vertical soft key [BYTE SELECT]. The following screen appears:

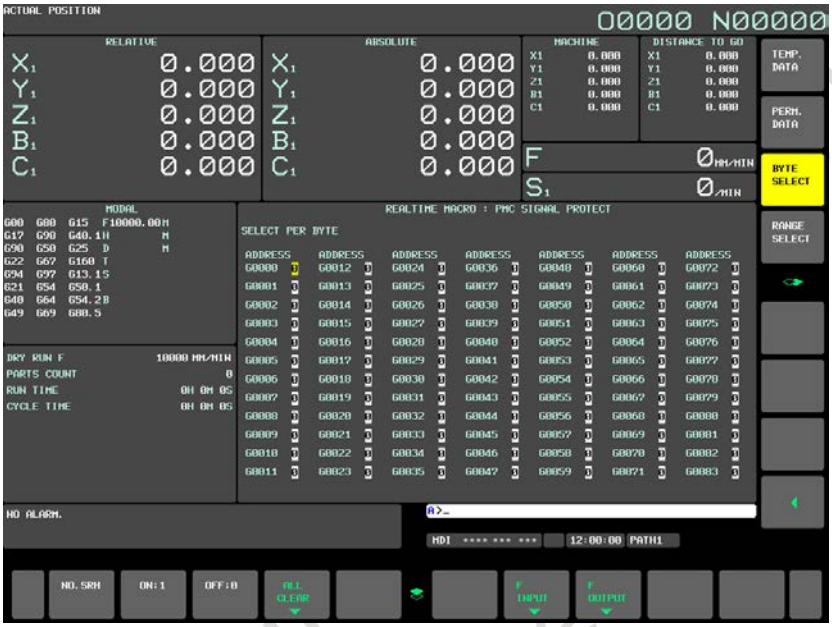


Fig. 12.3.25 (b) BYTE SELECT screen (15-inch display unit)

4 Move the cursor to the number of a DI/DO variable you want to set using either of the following methods:

- Enter the number and press horizontal soft key [NO. SRH].
- Move the cursor to a desired number by pressing page keys [PAGE] and/or [PAGE] and cursor keys [↑], [↓], [←], and/or [→].

5 Enter data.

FRYER MACHINE INC Rob Zahn

12.3.26 Displaying and Setting the Software Operator's Panel (15/19-inch Display Unit)

Operations on the MDI unit can substitute for the functions of switches on the machine operator's panel. This means that a mode selection, jog feed override selection, and so forth can be made with operations on the MDI unit, eliminating the need to use the corresponding switches on the machine operator's panel. Jog feed can be performed using numeric keys.

Displaying and setting the software operator's panel

Procedure

1 Press function key [OFS/SET].

2 Press vertical soft key [NEXT PAGE] and then vertical soft key [OPERAT PANEL].

3 The screen consists of several pages. Press page key [PAGE ⇧] or [PAGE ⇩] until the desired screen is displayed.

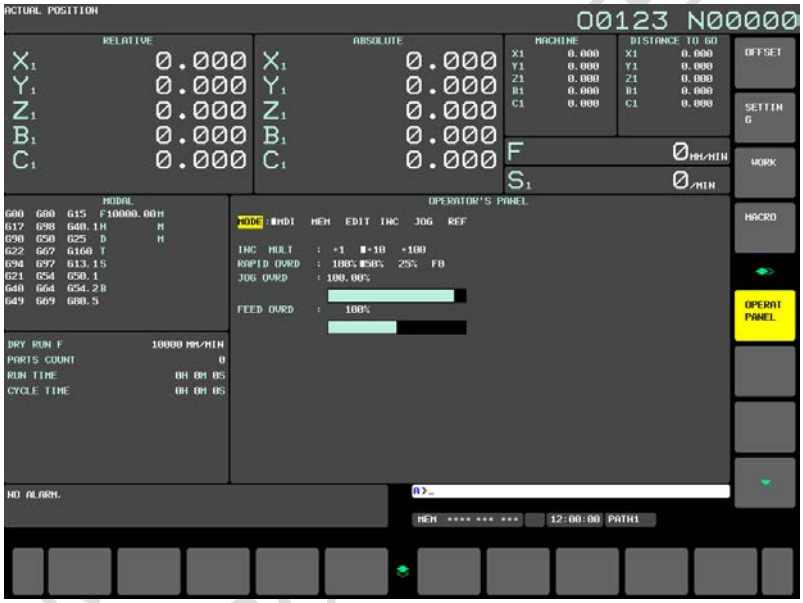


Fig. 12.3.26 (a) Without the manual handle feed function (15-inch display unit)

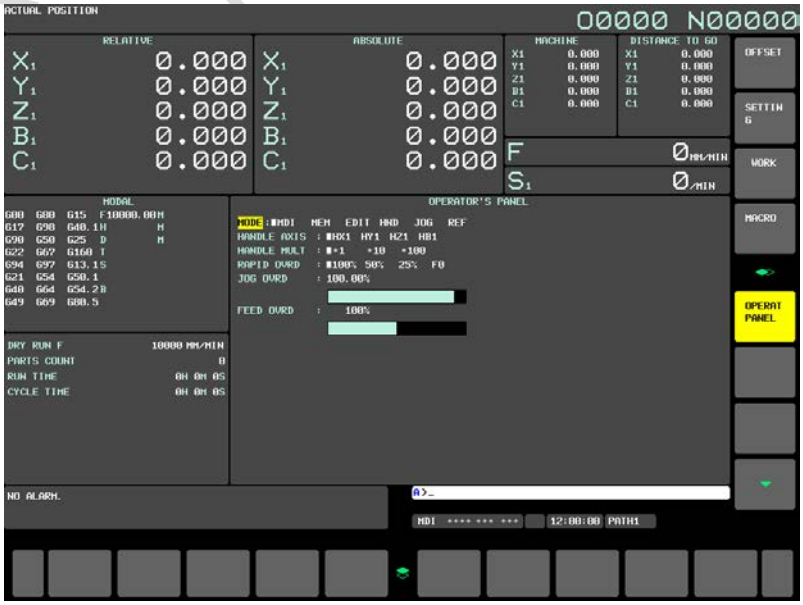


Fig. 12.3.26 (b) With the manual handle feed function (15-inch display unit)

FRYER SYSTEMS INC

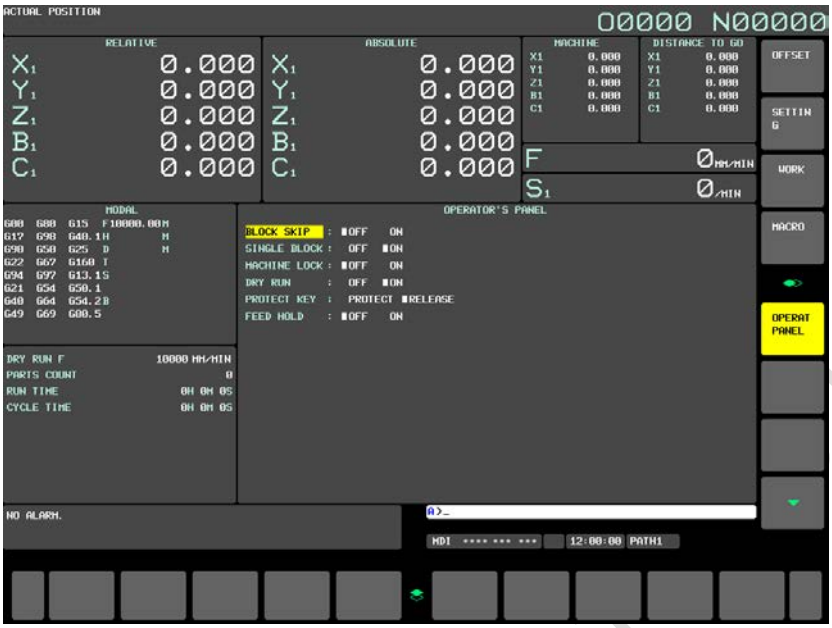


Fig. 12.3.26 (c) (15-inch display unit)

4 Move the cursor to the desired switch by pressing cursor key ↑ or ↓.

5 Push the cursor key ← or → to match the mark ■ to an arbitrary position and set the desired condition.

6 Press one of the following arrow keys (Fig. 12.3.26 (d)) to perform jog feed. Press the 5 key together with an arrow key to perform jog rapid traverse.

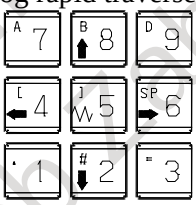


Fig. 12.3.26 (d) MDI arrow keys

Explanation

- Valid operations

The valid operations on the software operator's panel are shown below. Whether to use each group can be chosen using parameter No. 7200. Those groups that are not used are not displayed on the software operator's panel.

Group1 : Mode selection
Group2 : Selection of jog feed axis, Manual rapid traverse
Group3 : Selection of manual pulse generator feed axis, selection of manual pulse magnification
Group4 : Jog federate, federate override, rapid traverse override
Group5 : Optional block skip, single block, machine lock, dry run
Group6 : Protect key
Group7 : Feed hold

- Screens on which jog feed is valid

When the screen indicates other than the software operator's panel screen and self-diagnosis screen, jog feed is not conducted even if the arrow key is pressed. This function cannot be used when the virtual MDI key function is used.

- Jog feed and arrow keys

The feed axis and direction corresponding to the arrow keys is set with parameters Nos. 7210 to 7217.

FRYER MACHINE SYSTEMS INC Rob Zahensky

- **General purpose switches**

For the meanings of these switches, refer to the manual issued by machine tool builder.

12.3.27 Setting and Displaying Tool Management Data (15/19-inch Display Unit)

The tool management function totally manages tool information including tool offsets and tool life information.
This function provides a magazine screen and tool management screen. This subsection explains how to set these screens.

12.3.27.1 Displaying and setting magazine screen (15/19-inch display unit)

Procedure

1 Press function key [OFS/SET].

2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [TOOL MANAGER]. The screen shown below appears.

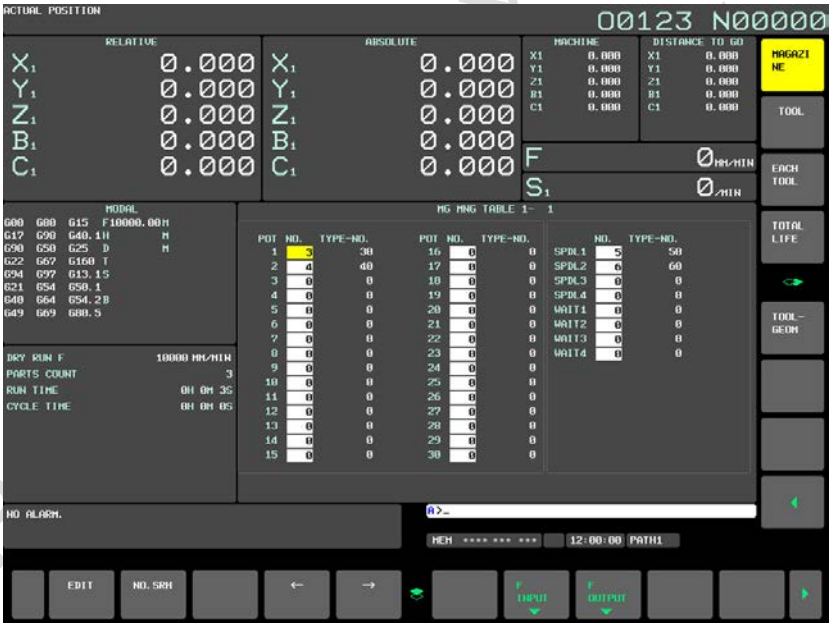


Fig. 12.3.27.1 (a) Tool management magazine screen (15-inch display unit)

3 By using the page keys [PAGE ⇧] [PAGE ⇩], cursor keys [⬆] [⬇] [⬅] [➡], and horizontal soft keys [←] and [→], move the cursor to the pot No. position of the magazine for which you want to set or modify data.
Alternatively, type the number of the tool management data you wan to set or modify, and press horizontal soft key [NO. SRH].

4 Press horizontal soft key [EDIT].

FRYER SYSTEMS INC

5 To set the tool management data number of a pot, type the tool management data number, then press MDI key INPUT.

To delete the tool management data number set for a pot, follow the steps below.

<1> Press horizontal soft key [ERASE].

<2> Press horizontal soft key [CURDAT]. To delete the tool management data numbers registered for all pots, press horizontal soft key [ALL].

<3> Press horizontal soft key [EXEC].

6 To end the edit operation, press horizontal soft key [EXIT].

Explanation

- Another method

Magazine data can be input/output also by using external I/O devices.
See Section "DATA I/O".

- Displayed data

POT : As many pots as specified in parameter No. 13222, 13227, 13232, or 13237 (settable for each magazine) are displayed. The starting pot number can be set for each magazine in parameter No. 13223, 13228, 13233, or 13238.

NO. : Tool management data numbers are displayed.
In the following cases, tools cannot be set in magazines:
1. A tool management data number beyond the range from 0 to (the number of valid pairs in parameter No. 13220) is set.
2. Tool management data is invalid (bit 1 of tool information is 0).
3. The tool management data number to be set is already set for another pot.

TYPE NO. : Tool type numbers corresponding to tool management data numbers are displayed.
Values cannot be set on this screen.

SPINDLE : The tool management data numbers and tool type numbers of spindle positions are displayed.

WAIT : The tool management data numbers and tool type numbers of wait positions are displayed.

- Edit mode

To edit data, press horizontal soft key [EDIT] to enter tool management data edit mode.

- Tool management data number

Tool management data numbers can optionally be extended to 64, 240, or 1000 pairs. In addition, the number of tools used can be set in parameter No. 13220.

- Display of the spindle positions and wait positions

The spindle position and wait position can be set for each path.

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

12.3.27.2 Displaying and setting tool management screen (15/19-inch display unit)

Procedure

1 Press function key [OFS/SET].

2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [TOOL MANAGER].

3 Press vertical soft key [TOOL].

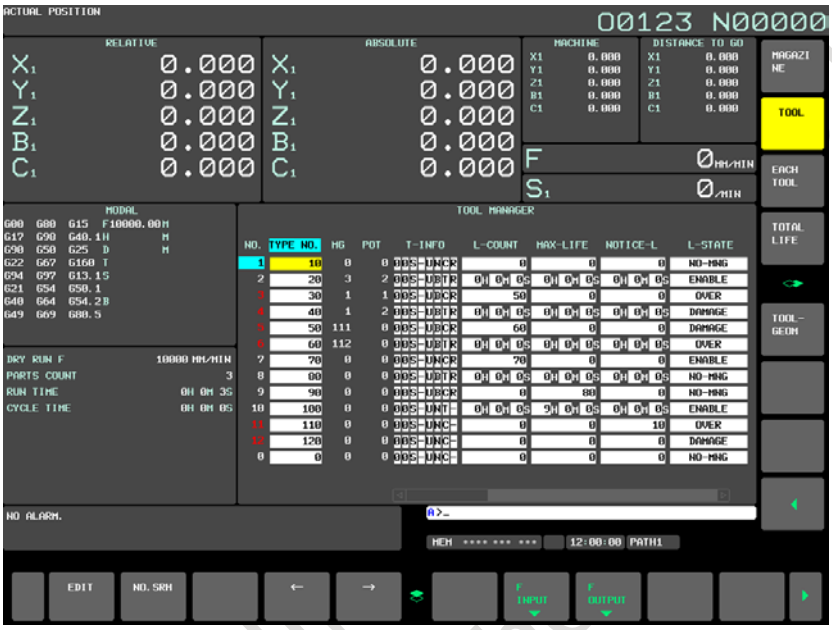


Fig. 12.3.27.2 (a) Tool management data screen (15-inch display unit)

4 By using the page keys [PAGE ↑] [PAGE ↓], cursor keys [↑] [↓] [←] [→], and horizontal soft keys [←] and [→], move the cursor to the position of the tool information of the tool number for which you want to set or modify data.
Alternatively, type the tool number of the data you want to set or modify, and press horizontal soft key [NO. SRH].

5 Press horizontal soft key [EDIT].

6 To set tool data, type a desired value, then press MDI key [INPUT].

To delete tool information, follow the steps below.

<1> In step 4, move the cursor to the tool information you want to erase.

<2> Press horizontal soft key [ERASE].

<3> Press horizontal soft key [CURDAT]. To delete entire tool information, press horizontal soft key [ALL].

<4> Press horizontal soft key [EXEC].

7 To end the edit operation, press horizontal soft key [EXIT].

FRYER MACHINE INC Rob Fryer

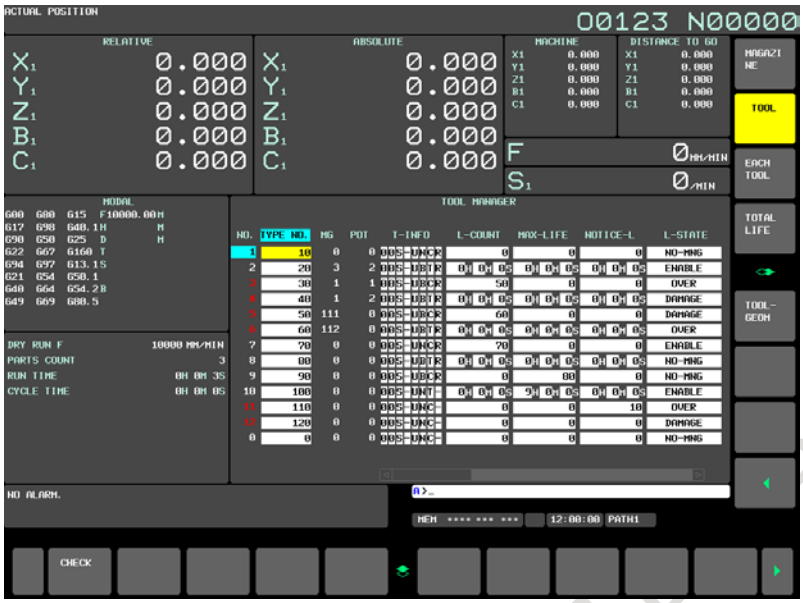


Fig. 12.3.27.2 (b) Tool management data screen (check function) (15-inch display unit)

8 When horizontal soft key [CHECK] is pressed, if there are tools with the same number but with different count types (count and time), the cursor moves to the tool type number of the smallest tool management number in the tool type numbers and a warning message appears.
The warning messages issued by the check function are shown below.
<1> For the same type of tools with different count types
L-COUNT TYPE MISMATCH:XXXXXXXX (XXXXXXXX= tool type number)
<2> For the same type of tools with the same count type
L-COUNT TYPE MATCH

Explanation

- Another method

Tool management data, customize data, and names set for tool states can be input/output also by using external I/O devices.
See Section "DATA I/O".

- Displayed information

- Life information

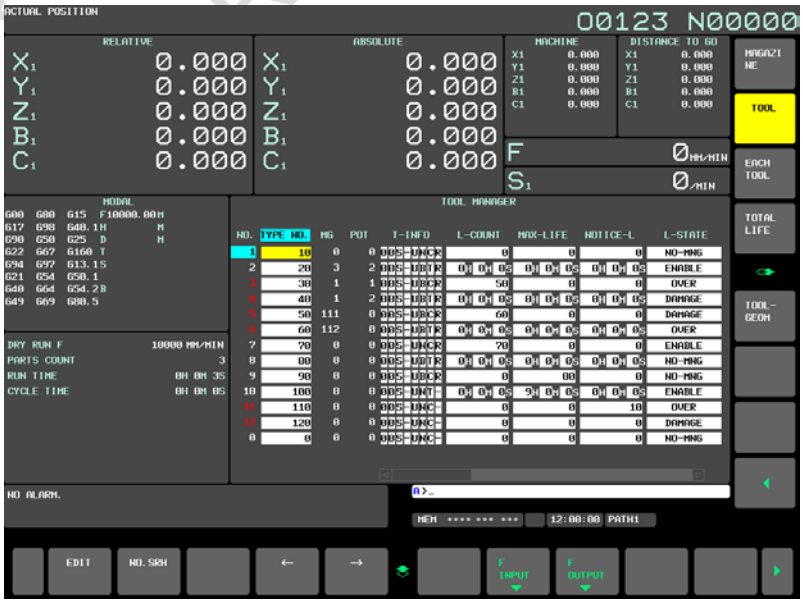


Fig. 12.3.27.2 (c) Tool management data life status screen(15-inch display unit)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NO. : Tool management data numbers are displayed. These numbers can be displayed but cannot be set. The tool management data number of edited data is kept blinking until tool management data edit mode ends. Tool management data numbers set or deleted on the magazine screen also blink on the tool management screen.

TYPE NO. : Tool type numbers are displayed.
A value from 0 to 99,999,999 can be set.

MG : The magazine number assigned to each tool is indicated. These numbers can be displayed but cannot be set.

POT : The pot number assigned to each tool is indicated. These numbers can be displayed but cannot be set.

T-INFO : The following five types of information are displayed sequentially from the right:
- Tool management data: Valid (R)/invalid (-))
- Life count type: Time (T)/count (C)
- Tool type: Oversize tool (B)/normal tool (N)
- Data access: Locked (L)/unlocked (U)
- When a tool is not under life management:
  Included in the tools to be searched (1)/not included in the tools to be searched (0)

NOTE
1 The tool types and data access information vary depending on the specifications defined by the machine tool builder.
2 The same type of tools must have the same life count type.

L-COUNT : The number of use times/use period of time of each tool is indicated.
Up to 99,999,999 times or 999 hours 59 minutes 59 seconds can be set.

MAX-LIFE : The maximum life value of each tool is indicated.
Up to 99,999,999 times or 999 hours 59 minutes 59 seconds can be set.

NOTICE-L : Noticed life value of a tool
Up to 99,999,999 times or 999 hours 59 minutes 59 seconds can be set.

L-STATE : Current tool state
One of the four states, including invalid (0), present (1, 2), not present (3), and broken (4), is indicated. The numbers in parentheses are data values used when these states are input in MDI.

- Spindle speed/feedrate

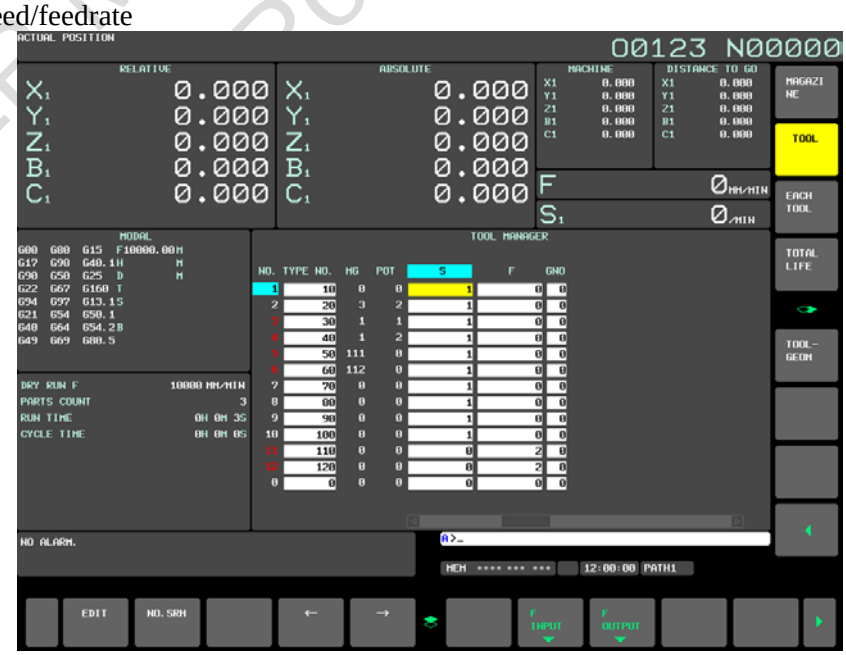


Fig. 12.3.27.2 (d) Tool management data spindle speed/feedrate screen (15-inch display unit)

FRYER MACHINE SYSTEMS INC
Bob Zaharsky

S : Spindle speed.
A value from 0 to 99,999 can be set.
F : Feedrate.
A value from 0 to 99,999,999 can be set.

- Tool offset information

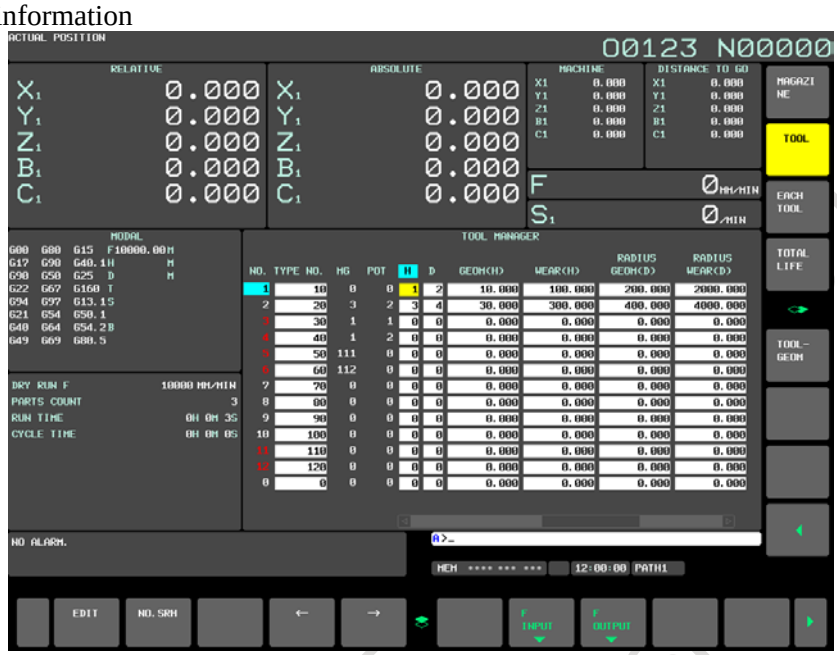


Fig. 12.3.27.2 (e) Tool management data tool offset screen (15-inch display unit)

H : Tool length compensation number (for machining center systems only).
A value from 0 to 999 can be set.
D : Cutter compensation number (for machining center systems only).
A value from 0 to 999 can be set.
TG : Tool geometry compensation number (for lathe systems only).
A value from 0 to 999 can be set.
TW : Tool wear compensation number (for lathe systems only).
A value from 0 to 999 can be set.

The displayed offset data is determined depending on the option configuration and parameter settings (bits 1(DCR), 2(DOY), 4(DO2), 6(DOT), and 7(DOM) of parameter No. 13202).

FRYER MACHINE Rob Zahensky INC

- Customize information

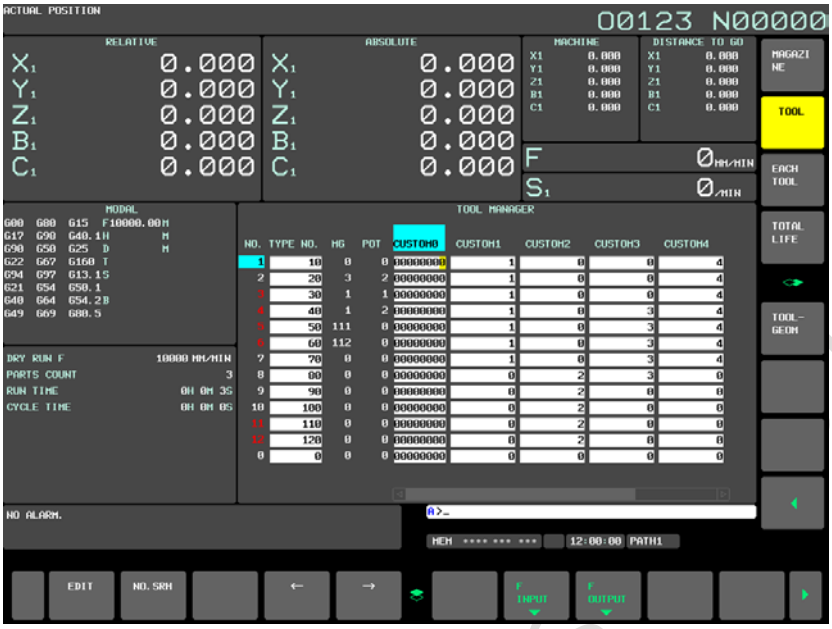


Fig. 12.3.27.2 (f) Tool management data customize data screen (15-inch display unit)

CUSTOM0 : Bit-type customize information.
For each bit, 1 or 0 can be input.

CUSTOM1 to 4 : Customize information. Any value from -99,999,999 to 99,999,999 can be set.

CUSTOM5 to 20 : Customize information. These items are displayed only when customize data extension option (5 to 20) of the tool management function is enabled. Any value from -99,999,999 to 99,999,999 can be set.

CUSTOM21 to 40 : Customize information. These items are displayed only when customize data extension option (5 to 40) of the tool management function is enabled. Any value from -99,999,999 to 99,999,999 can be set.

- Tool management data number

Tool management data numbers can optionally be extended to 64, 240, or 1000 pairs. The number of tools used can be set in parameter No. 13220.

- Edit mode

To edit data, press horizontal soft key [EDIT] to enter tool management data edit mode.

- Life count override

When the period of time is set as the tool life count type, the life count can be overridden if bit 2 (LVF) of parameter No. 6801 is set to 1. Specify an override value by using a switch on the machine operator's panel.

Example)

When cutting is performed for 10 minutes with an override of 0.1, one minute is counted in the tool life counter.

- Tool management extension function

When tool management extension functions are enabled, you can use the following functions in addition to the tool management functions:

- A value with a decimal point can be set as customize data.
  The maximum number of decimal places can be set to 7 by G10 data input and file input.
- Various types of tool management data can be protected using KEY signals.
  An attempt to enter a value to a protected data causes a warning message “WRITE PROTECT.”

FRYER MACHINE SYSTEMS INC
Rob Zakensky

- Tool life count period selection
  Bit 5 of tool information is used to make a life count period selection.

Item		Description
Data length		1 byte (flag data)
#5	REV	0: A life count period of 1 sec is used. 1: A life count period of 8 msec is used.

Range of count is as follows.

1sec : 0 to 3,599,999 sec (999 hours 59 minutes 59 seconds)
8msec : 0 to 3,599,992 ms (59 minutes 59 seconds 992 milliseconds)

12.3.27.3 Each tool data screen (15/19-inch display unit)

Each tool data screen

Procedure

1 Press function key .

2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [TOOL MANAGER].

3 Press vertical soft key [EACH TOOL]. The each tool data screen appears.

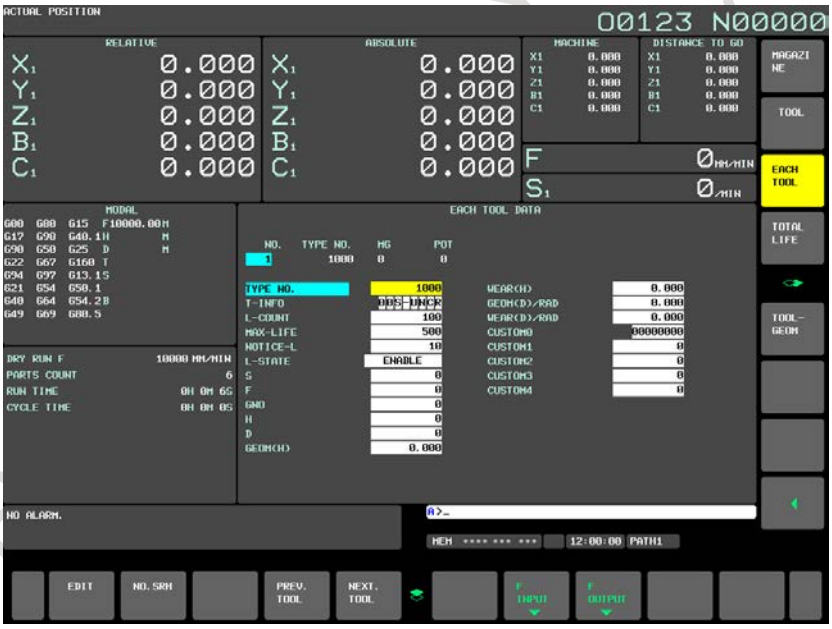


Fig. 12.3.27.3 (a) Each tool data screen (15-inch display unit)

FRYER SYSTEMS INC

Explanation

- Header

The following four data items are displayed: NO., TYPE NO., MG, and POT.
When the data table of a tool extends over two or more pages, the same header is displayed on these pages.

- Data table

The data table shows data items related to a tool at a time.
The data items are displayed from the upper left to the lower left, the upper right, and the lower right in ascending order of screen display position numbers specified using the customize function. The number of digits displayed for one data item is fixed to 11.
Up to 24 data items are displayed on one page. (12 data items × 2 columns)
When more than 24 data items are set for a tool, the 25th and subsequent data items are displayed on the next page. (Up to three pages)
When a data item is set as a screen element of the tool management data screen twice or more using the tool management data display customize function (one of the tool management extension functions), only the data item with the smaller display position number is displayed. When no value is set for a data item, the field for the data item is not displayed and the next data item is displayed.

Key operations

- MDI key operations

Numeral keys Inputs a numeric value.

PAGE ↑ Displays the previous page for the same tool.

PAGE ↓ Displays the next page for the same tool.

↑ Moves the cursor to the previous data item.

When the cursor is on the top row of the data table, it moves to the bottom row of the left column. When the cursor is on the first data item, it moves to the last data item.

↓ Moves the cursor to the next data item.

When the cursor is on the bottom row of the data table, it moves to the top row of the right column. When the cursor is on the last data item, it moves to the first data item.

← Moves the cursor left on the screen.

When the cursor is on the left column of the data table, it moves to the right column on the row immediately above. When the cursor is on the first data item, it moves to the last data item.

→ Moves the cursor right on the screen.

When the cursor is on the right column of the data table, it moves to the left column on the row immediately below. When the cursor is on the last data item, it moves to the first data item.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Soft key operations

Horizontal soft key [EDIT]

Enters the management data edit mode.

Horizontal soft key [NO.SRH]

Displays data with a tool management data number when pressed after the tool management data number is entered.

Horizontal soft key [PREV.TOOL]

Returns to the previous tool management data number.

Horizontal soft key [NEXT.TOOL]

Proceeds to the next tool management number.

Horizontal soft key [F INPUT]

Inputs data related to the tool management function.
Can be used only in the standard mode.
Requires placing the NC in the EDIT mode.

Horizontal soft key [F OUTPUT]

Writes data related to the tool management function.
Can be used only in the standard mode.
Requires placing the NC in the EDIT mode.

Operation in the management data edit mode

To edit data, press horizontal soft key [EDIT] to enter the management data edit mode.

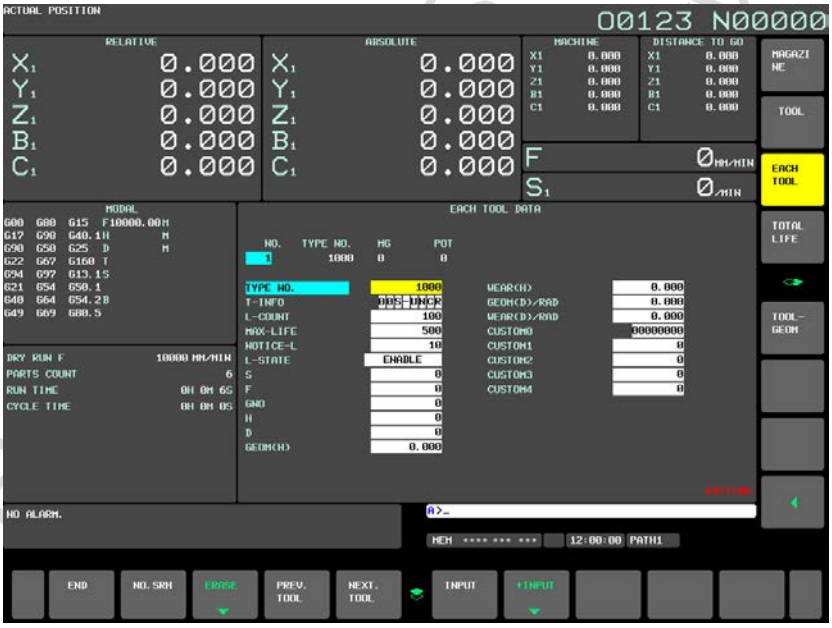


Fig. 12.3.27.3 (b) Each tool data editing screen (15-inch display unit)

In the management data edit mode, “EDITING” is displayed at the lower right of the screen.
In the management data edit mode, the above key operations (Fig. 12.3.27.3 (b)), in addition to those mentioned above except [F INPUT] and [F OUTPUT], can be performed.

Actually writes a value entered using numeric keys.

Horizontal soft key [END]

Ends the management data edit mode.

Operations for editing tool information are the same as those on the tool management data screen.

FRYER SYSTEMS INC

12.3.27.4 Displaying the total life of tools of the same type (15/19-inch display unit)

Total life data screen

Procedure

1 Press function key .

2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [TOOL MANAGER].

3 Press vertical soft key [TOTAL LIFE]. The total life data screen appears.

4 Using horizontal soft key [CHANGE] can switch total tool life data displays between specification by count and specification by duration.

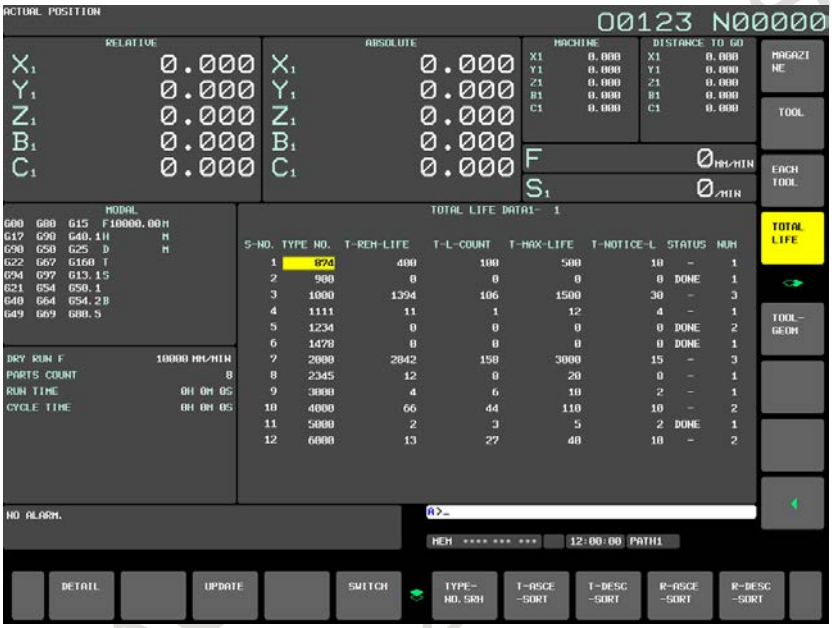


Fig. 12.3.27.4 (a) Count display (15-inch display unit)

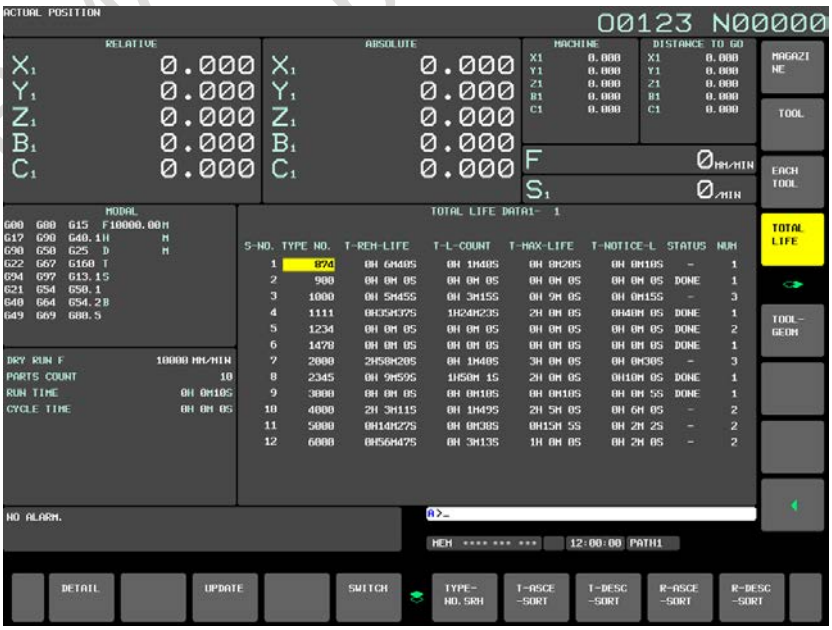


Fig. 12.3.27.4 (b) Time display (15-inch display unit)

FRYER MACHINES INC

Displayed information

S-NO. : Sequential number of each tool type
TYPE NO. : Tool type number
T-REM-LIFE : Total of remaining life values of tools with the same tool type number
T-L-COUNT : Total of used counts/times of tools with the same tool type number
T-MAX-LIFE : Total of maximum life values of tools with the same tool type number
T-NOTICE-L : Total of notice life values of tools with the same tool type number when the tool life arrival signal is to be output for each tool type
STATUS : Whether the tool life arrival signal has been output when the signal is to be output for each tool type
Either of the two states (UNDONE and DONE) is displayed.
NUM : Number of tools with the same tool type number

When bit 3 (ETE) of parameter No. 13200 is set to 0 and bit 2 (TRT) of parameter No. 13200 is set to 1, the tool life arrival signal is to be output for each tool type number and T-NOTICE-L and STATUS are displayed.

Key operations

- MDI key operations

PAGE ⇧ Displays the previous page. The cursor moves to the last data item on that page.

PAGE ⇩ Displays the next page. The cursor moves to the first data item on that page.

⬆ Moves the cursor up on the screen.

⬇ Moves the cursor down on the screen.

⬅ Disabled.

➡ Disabled.

- Soft key operations

Horizontal soft key [DETAIL]	Displays the detailed life data screen.
Horizontal soft key [UPDATE]	Updates data displayed on the total life data screen.
Horizontal soft key [CHANGE]	Switch the display between the count and time types.
Horizontal soft key [T-TYPENO. SRH]	Moves the cursor to a tool type number when pressed after the number is entered.
Horizontal soft key [T-ASCE -SORT]	Sorts data in ascending order of tool type numbers.
Horizontal soft key [T-DESC -SORT]	Sorts data in descending order of tool type numbers.
Horizontal soft key [R-ASCE -SORT]	Sorts data in ascending order of remaining life values.
Horizontal soft key [R-DESC -SORT]	Sorts data in descending order of remaining life values.

NOTE

1. After horizontal soft key [T-ASCE -SORT], [T-DESC -SORT], [R-ASCE -SORT], or [R-DESC -SORT] is pressed, the cursor is positioned at the top of page 1 of the total life data screen.
2. When the power is turned on, data of the count counting type is displayed in ascending order of tool type numbers. When the display type is changed or data is sorted in a different order, the status is kept.
3. If horizontal soft key [DETAIL] is pressed when no data is displayed on the total life data screen, the warning message “NO DETAILED LIFE DATA SCREEN” is output.
4. If an unregistered tool type is specified and horizontal soft key [T-TYPENO. SRH] is pressed, the warning message “UNREGISTERED NUMBER” is output.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Detailed life data screen

Procedure

1 Press function key .

2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [TOOL MANAGER].

3 Press vertical soft key [TOTAL LIFE]. The total life data screen appears.

4 Selecting horizontal soft key [DETAIL] causes the detailed life data screen to appear.

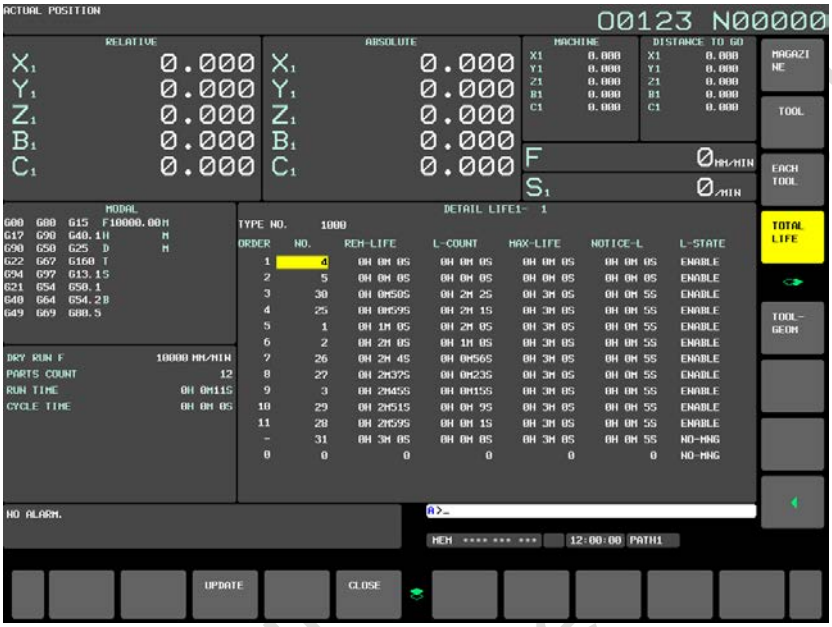


Fig. 12.3.27.4 (c) Detailed life data screen (15-inch display unit)

- Displayed information

TYPE NO. : Tool type number

ORDER : Sequential number in ascending order of remaining life times or the order in which the customize data is set.

NO. : Tool management data number

REM-LIFE : Remaining life value obtained by subtracting the life count value from the maximum life value

L-COUNT : Total used count or time of the tool

MAX-LIFE : Maximum life value of the tool

NOTICE-L : Notice life value of the tool

L-STATE : Life status of the tool
One of the four states (NO-MNG (0), ENABLE (1, 2), NONE (3), and DAMAGED (4)) is displayed.

Key operations

- MDI key operations

PAGE ↑ Displays the previous page.

PAGE ↓ Displays the next page.

↑ Moves the cursor up on the screen. The cursor moves to the last data item on that page.

↓ Moves the cursor down on the screen. The cursor moves to the first data item on that page.

Disabled.

Disabled.

- **Soft key operations**

Horizontal soft key [UPDATE]

Updates data displayed on the detailed life data screen.

The cursor is positioned at the beginning of page 1 after this soft key is pressed.

Horizontal soft key [CLOSE]

Closes the detailed life data screen and returns to the total life data screen.

NOTE

When horizontal soft key [CLOSE] is pressed and the total life data screen is displayed again, the cursor on the total life screen data is positioned as follows.

- When the tools with the tool type number displayed when the detailed data screen is closed are registered as tool management data, the cursor on the total life screen is positioned at that tool type number.
- When the tools with the tool type number displayed when the detailed data screen is closed are not registered as tool management data, the cursor on the total life data screen is positioned at the first total life data.

12.3.27.5 Tool geometry data screen (15/19-inch display unit)

Tool geometry data screen

Procedure

1 Press function key .

2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [TOOL MANAGER].

3 Selecting vertical soft key [TOOL-GEOM] causes the tool geometry data screen to appear.

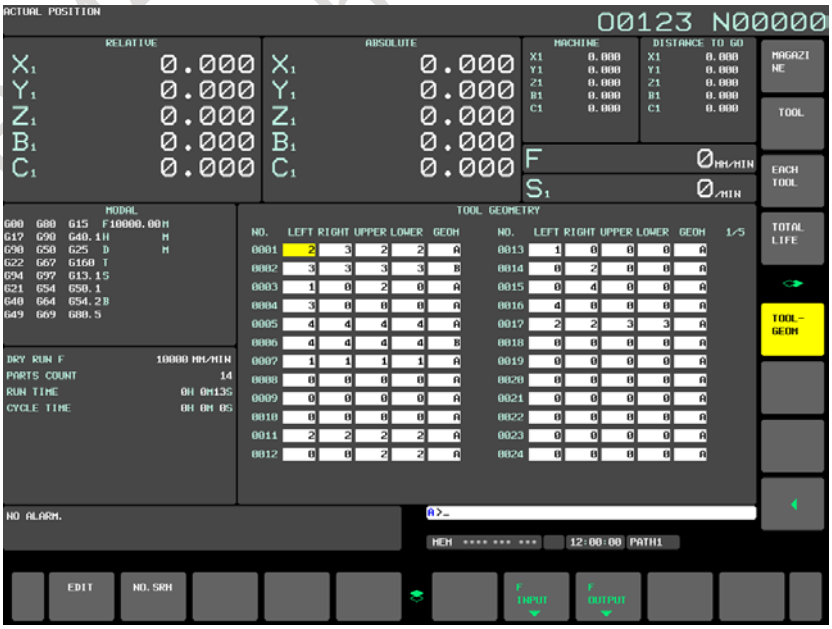


Fig. 12.3.27.5 (a) Tool geometry data screen (15-inch display unit)

FRYER MACHINE SYSTEMS INC
Joseph Zahensky

- Displayed item

NO. : Tool geometry number
Up to 20 numbers can be displayed.
LEFT : Sets the number of pots on the left of the reference pot that are to be occupied.
A value between 0 and 4 can be set.
RIGHT : Sets the number of pots on the right of the reference pot that are to be occupied.
A value between 0 and 4 can be set.
UPPER : Sets the number of pots above the reference pot that are to be occupied.
A value between 0 and 4 can be set. (Use this item when the magazine is of the matrix type.)
LOWER : Sets the number of pots under the reference pot that are to be occupied.
A value between 0 and 4 can be set. (Use this item when the magazine is of the matrix type.)

Key operations

- Operations in the standard mode

MDI key operations

Numeral keys Inputs a numeric value.

Moves the cursor up on the screen.

Moves the cursor down on the screen.

Moves the cursor left on the screen.

Moves the cursor right on the screen.

Soft key operations

Horizontal soft key [NO.SRH] Moves the cursor on a tool geometry number when pressed after the number is entered.
Horizontal soft key [EDIT] Enters the data edit mode.
Horizontal soft key [F INPUT] Inputs data related to the tool management functions.
This key is available only in the standard mode.
Put the NC in the EDIT mode.
Horizontal soft key [F OUTPUT] Outputs data related to the tool management functions.
This key is available only in the standard mode.
Put the NC in the EDIT mode.

In the management data edit mode, the above key operations (Fig. 12.3.27.5 (a)), in addition to [NO.SRH] mentioned above, can be performed.

- Operations in the edit mode

To edit data, press horizontal soft key [EDIT] to enter the edit mode. In the edit mode, “EDITING” is displayed at the lower right of the screen.

MDI key operation

Actually writes a value entered using numeric keys.

Soft key operation

Horizontal soft key [END]
Ends the data edit mode.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Example

Set the edit mode. When the tool geometry with tool geometry number 1 occupies 1 pot in the left direction, 0.5 pots in the right direction, and 1.5 pots in the down direction, set data as shown in the Fig. 12.3.27.5 (b):

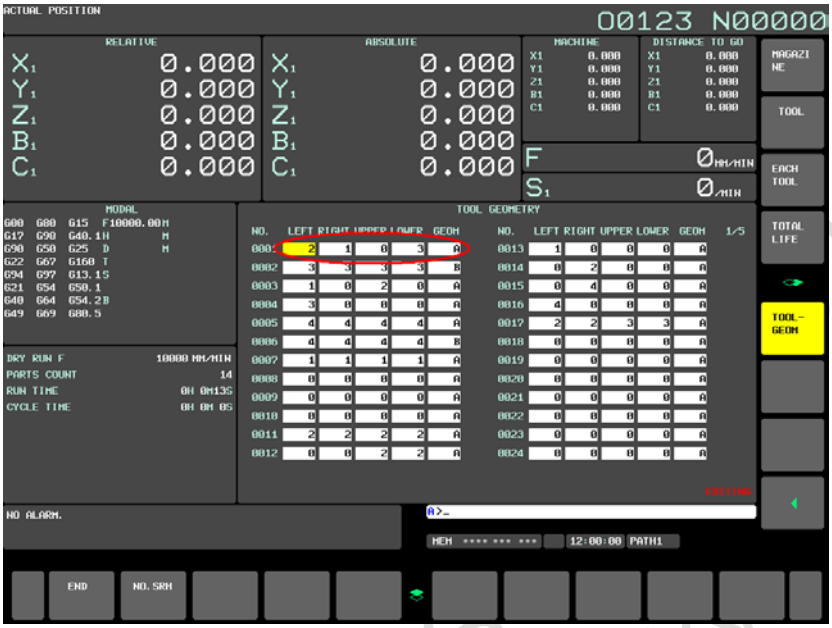


Fig. 12.3.27.5 (b) Example of setting data on the tool geometry data screen (15-inch display unit)

- Display of occupied pots in the magazine management table

Each pot occupied by a tool stored in another pot is indicated with an asterisk (*).

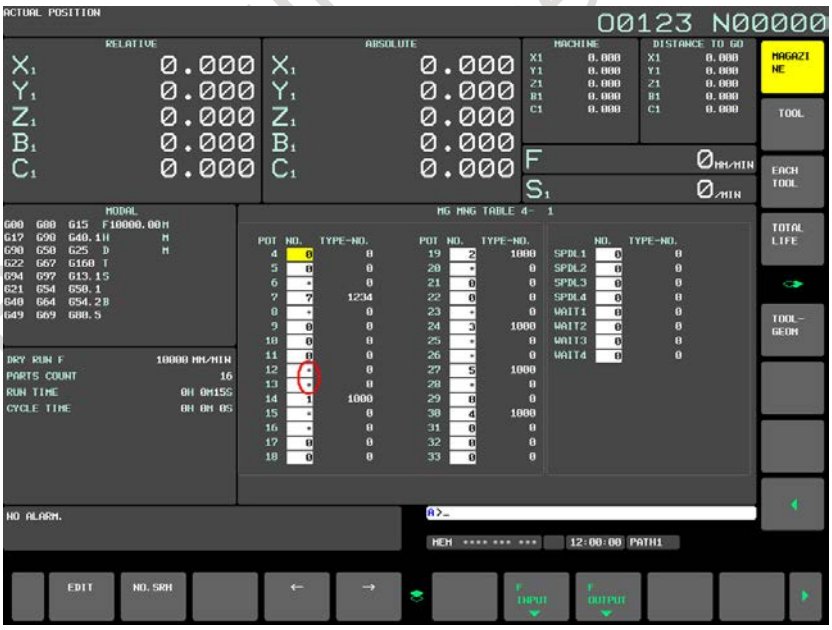


Fig. 12.3.27.5 (c) Magazine management table (15-inch display unit)

If a tool to be registered for a magazine is determined to interfere with another tool, the warning message “TOOL INTERFERENCE CHECK ERROR:xxxx,xxxx” is displayed. xxxx indicates the tool number of each of the two tools. If a tool is determined to interfere with multiple tools, only the first tool detected by the NC interference check is displayed.

If a tool is determined to interfere with the frame of a magazine, the tool number of the tool to be stored and “FRAME” are displayed.

- Search for an empty pot for a oversize tool

During the management data editing, a horizontal soft key for searching for empty oversize tool pots is displayed.

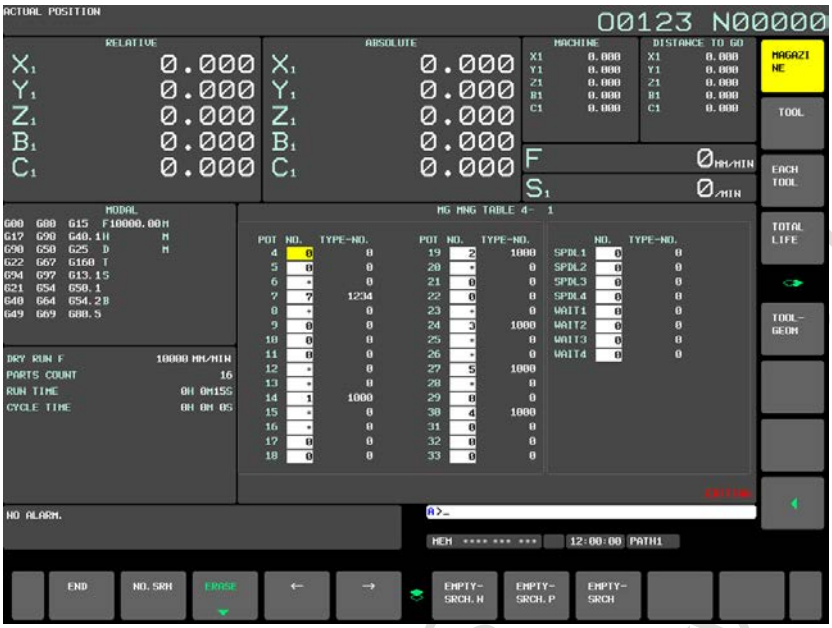


Fig. 12.3.27.5 (d) Searching for an empty pot for a oversize tool (15-inch display unit)

Enter the tool geometry number in the key-in buffer and press a search horizontal soft key. The cursor moves to an empty pot fit for the geometry.

EMPTY-SRCH.N : Searches for an empty pot forward.
EMPTY-SRCH.P : Searches for an empty pot backward.
EMPTY-SRCH : Searches for the pot nearest to the current position.

- Tool management screen

You can use bit 2 of tool information to switch between a oversize tool and normal tool. For a oversize tool, set a tool geometry number fit for the tool.

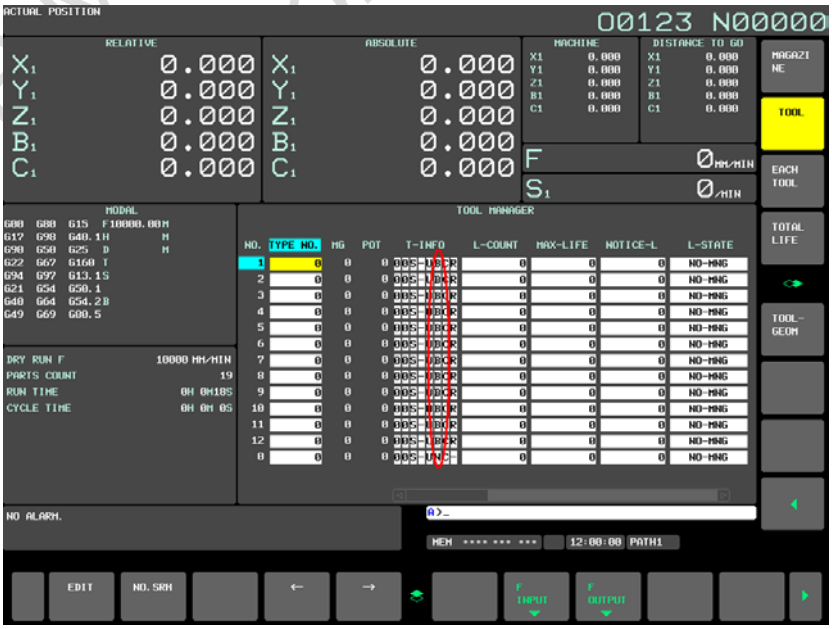


Fig. 12.3.27.5 (e) Bit for switching between a normal tool and oversize tool (15-inch display unit)

FRYER MACHINE S INC
ob Zahensky

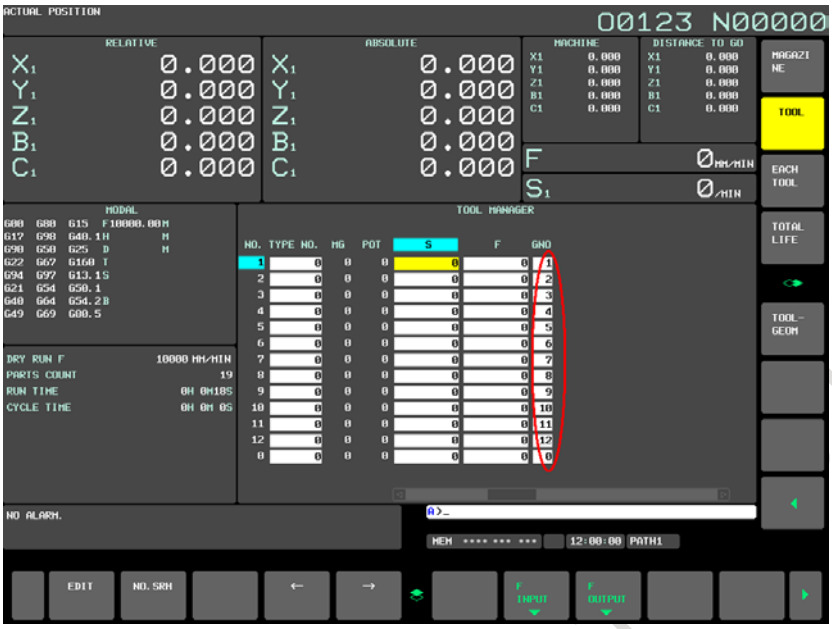


Fig. 12.3.27.5 (f) Tool geometry number (15-inch display unit)

12.3.28 Displaying and Switching the Display Language (15/19-inch Display Unit)

The language used for display can be switched to another language.
A display language can be set using a parameter. However, by modifying the setting of the display language on this screen, the display language can be switched without turning off then on the power.

Displaying and setting the display language

Procedure

1 Press function key .

2 Press vertical soft key [NEXT PAGE] several times to display soft key [LANGUAGE].

3 Press vertical soft key [LANGUAGE] to display the language screen.

Fig. 12.3.28 (a) LANGUAGE screen (15-inch display unit)

FRYER MACHINE SYSTEMS INC
Jacob Zahensky

4 Press page key [PAGE ⇧] or [PAGE ⇩], then press cursor key [⬆] or [⬇] to move the cursor to a desired display language.

5 Press horizontal soft key [APPLY]. The display language is switched to the selected language. The language specified on this screen continues to be used if the power is turned off then back on.

Explanation

- Language switching

The language screen can be displayed if bit 0 (NLC) of parameter No. 3280 is set to 0.

- Selectable languages

The display languages selectable on this screen are as follows:

1. English
2. Japanese
3. German
4. French
5. Chinese (Traditional)
6. Chinese (Simplified)
7. Italian
8. Korean
9. Spanish
10. Dutch
11. Danish
12. Portuguese
13. Polish
14. Hungarian
15. Swedish
16. Czech
17. Russian
18. Turkish
19. Bulgarian
20. Rumanian
21. Slovak
22. Finnish
23. Vietnamese
24. Indonesian
25. Slovenian

Among the languages listed above, English and other usable languages are displayed on the screen as a list of switchable languages.

Limitation

- Language parameter modification on the parameter screen

Which language to be used for display is specified with parameter No. 3281. This parameter can be modified using the parameter screen as well. However, if a modification is made on the parameter screen, the new setting is not reflected until "APPLY" operation is performed on the language screen or the power is turned on again. If an invalid value is set in parameter No. 3281 on the parameter screen, the screen is displayed in English after the power is turned on again.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.29 Protection of Data at Eight Levels (15/19-inch Display Unit)

You can set eight CNC and PMC operation levels and one of eight protection levels for each type of CNC and PMC data.
When an attempt is made to change CNC and PMC data or output it to an external unit, the operation level is compared with the protection level to determine whether to allow the change or external output.

12.3.29.1 Operation level setting (15-inch display unit)

You can set eight CNC and PMC operation levels.

Displaying and setting the operation level setting screen

Procedure

1 Press function key .

2 Press vertical soft key [NEXT PAGE] several times to display soft key [PROTECT].

3 Press vertical soft key [PROTECT].
The operation level setting screen (Fig. 12.3.29.1) shown below is displayed.

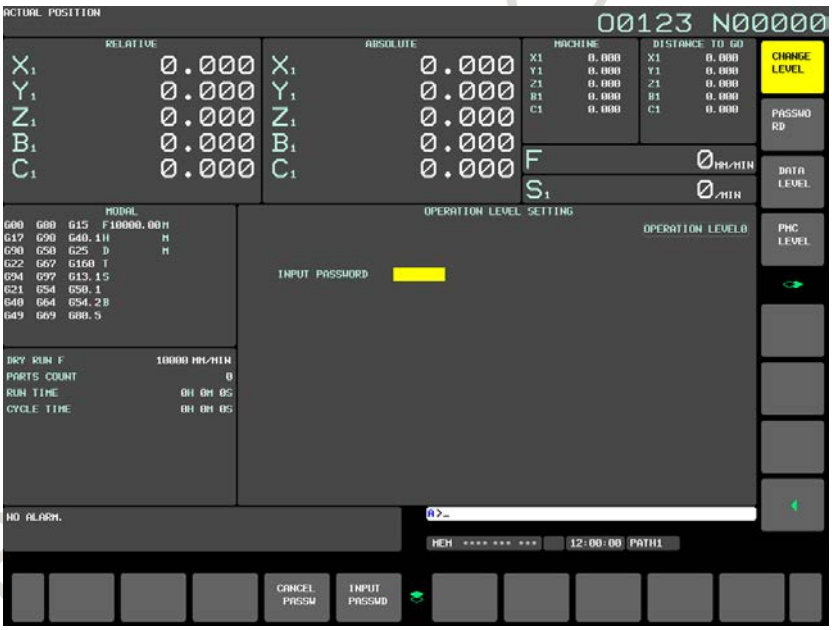


Fig. 12.3.29.1 (a) Operation level setting screen (15-inch display unit)

4 Key in the password for an operation level to be set/modified, then press horizontal soft key [INPUT PASSWD].

5 To return the operation level to 0, 1, 2, or 3, press horizontal soft key [CANCEL PASSWD].

Explanation

- Operation level setting

To select operation level 0 to 3, use the corresponding memory protection key signal.
To select operation level 4 to 7, use the corresponding password.

Table 12.3.29.1 Operation level setting

Operation level	Setting	Sample grouping
7 (high)	Password	-
6	Password	MTB
5	Password	Dealer and integrator

Operation level	Setting	Sample grouping
4	Password	End user
3	Memory protection key signal	User level (level 1)
2	Memory protection key signal	User level (level 2)
1	Memory protection key signal	User level (level 3)
0 (low)	Memory protection key signal	User level (level 4)

When operation level 4 to 7 is set, the operation level remains unchanged until the password is cleared. (The operation level also remains unchanged if the power is turned off.)
Operation level 7 is reserved for CNC and PMC maintenance.

NOTE
When a password is being entered, an asterisk (*) is displayed instead of each entered character.

12.3.29.2 Password modification (15/19-inch display unit)

The current operation level is displayed.
The password for each of operation levels 4 to 7 can be modified.

Displaying and setting the password modification screen

Procedure

1 Press function key .
2 Press vertical soft key [NEXT PAGE] several times to display soft key [PROTECT].
3 Press vertical soft key [PROTECT].
4 Press vertical soft key [PASSWORD].
The PASSWORD CHANGE screen (Fig. 12.3.29.2 (a)) shown below is displayed.

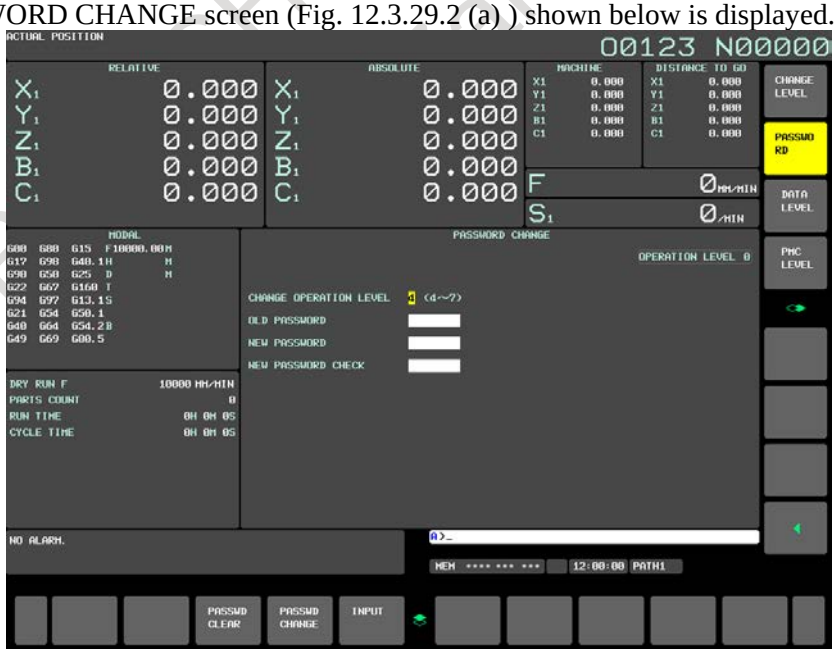


Fig. 12.3.29.2 (a) PASSWORD CHANGE screen (15-inch display unit)

5 Key in an operation level whose password is to be modified, then press horizontal soft key [INPUT].
6 Key in the current password for the operation level whose password is to be modified, then press horizontal soft key [INPUT].
7 Key in a new password, then press horizontal soft key [INPUT].
8 Key in the new password again for confirmation, then press horizontal soft key [INPUT].

FRYER LINE SYSTEMS INC
Chehensky

9 Press horizontal soft key [PASSWD CHANGE].
10 To clear the password, press horizontal soft key [PASSWD CLEAR].

Explanation

Up to eight characters (only uppercase alphabetic characters and numeric characters) can be input.

NOTE

1 For a password, consisting of three to eight characters, the following characters are available:
   - Uppercase alphabetic characters
   - Numeric characters
2 When a password is being entered, an asterisk (*) is displayed instead of each entered character.
3 Whether a password can be changed at the current operation level is determined as follows:
   - Password of an operation level higher than the current operation level
     Cannot be changed.
   - Password of the current operation level
     Can be changed.
   - Password of an operation level lower than the current operation level
     Can be changed (only to the initial password).
4 The set password is not displayed.
   Be careful not to forget the password.

12.3.29.3 Protection level setting (15/19-inch display unit)

The current operation level is displayed.
The change protection level and output protection level of each data item are displayed.
The change protection level and output protection level of each data item can be changed.

Confirmation based on protection level setting

Procedure

1 Press function key .
2 Press vertical soft key [NEXT PAGE] several times to display soft key [PROTECT].
3 Press vertical soft key [PROTECT].
4 Press vertical soft key [DATA LEVEL] to change the protection level of CNC data or press vertical soft key [PMC LEVEL] to change the protection level of PMC data.
   The following protection level change screen (Fig. 12.3.29.3 (a)) is displayed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

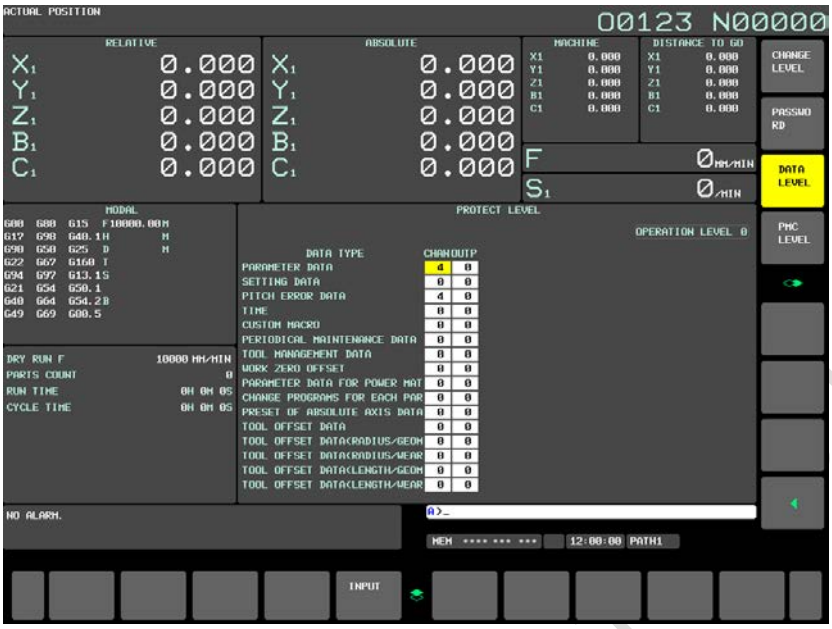


Fig. 12.3.29.3 (a) Protection level change screen (15-inch display unit)

5 Move the cursor to the change level or output level of a desired data item.
6 Key in a new desired level, then press horizontal soft key [INPUT].

> **NOTE**
> When the protection level of PMC data is set, soft key [SWITCH PMC] is used to switch between PMC paths to be set, for multi-path PMC.

Explanation

When the protection level of a data item is higher than the current operation level, the protection level of the data item cannot be changed.
The protection level of a data item cannot be changed to a protection level higher than the current operation level.
For each of the following types of data, you can set a data protection level. There are the following two types of data protection levels:

- Change protection level
  Sets the protection level used when data is changed.
- Output protection level
  Sets the protection level used when data is output to an external unit.

As a protection level, you can set a value of 0 (low) to 7 (high).

Table 12.3.29.3 (a) Protection level of each type of data

Type of data	Initial protection level	
	Change	Output
Custom macro variable data <CUSTOM MACRO> (including variable data dedicated to the macro executor)	0	0
Periodical maintenance data <PERIODICAL MAINTENANCE DATA>	0	0
Tool management data <"TOOL LIFE TIME DATA>	0	0
Tool offset data <TOOL OFFSET DATA> (For each type when tool geometry compensation and tool wear compensation are treated differently)	0	0
Clock data <TIME>	0	0
Workpiece origin shift amount data <WORK ZERO SHIFT>	0	0
Workpiece origin offset data <WORK ZERO OFFSET>	0	0
Ethernet setting data <ETHER NET SETTING>	0	0

FRYER MACHINE SYSTEMS INC
Rob Zaherski

Type of data	Initial protection level	
	Change	Output
Parameter data <PARAMETER DATA>	4	0
Settings <SETTING DATA>	0	0
Parameter data for Power Mate CNC manager function <PARAMETER DATA FOR POWER MATE CNC MANAGER>	0	0
Each part program <PROGRAMS FOR EACH PARTS>	0	0
Part program editing operation <CHANGE PROGRAMS FOR EACH PARTS>	0	0
Absolute coordinate preset operation <PRESET OF ABSOLUTE AXIS DATA>	0	0

Table 12.3.29.3 (b) Protection level of PMC data

Type of data	Initial protection level	
	Change	Output
Composition parameter	0	0
Setting (online)	0	0
Setting (each path)	0	0
Sequence program	0	0
PMC parameter	0	0
Timer	0	0
Counter	0	0
Keep relay	0	0
Keep relay (system)	0	0
Data table	0	0
Data table control	0	0
PMC memory	0	0
I/O configuration	0	0
I/O Link group selection	0	0
Registration of I/O device	0	0

NOTE

1 For some types of data, the output function is not provided.
2 When the protection level of data is higher than the current operation level, the protection level cannot be changed.
3 The protection level of data cannot be changed to a level higher than the current operation level.
4 Settable types of data increase or decrease, depending on the option configuration.
5 For details on the protection level of PMC data, refer to “PMC Programming Manual (B-64513EN)”.
6 Data related to tool information on the EACH TOOL DATA screen for the tool management function is protected by protecting tool offset data.
7 The type of tool offset data put in effect varies depending on the tool compensation value memory used.
8 To change the protection level for each part program, do so on the PROGRAM FOLDER screen rather than on the PROTECT LEVEL screen.
9 Part program editing includes program editing for the MDI mode.
10 Presetting absolute coordinates causes preset workpiece coordinate system values to be protected.
11 During tool offset data input/output, if any tool offset data type is not allowed to be changed or output to the outside, it is processed as follows:
- Input : Any data type other than those whose change is not allowed is changed.
- Output : Any data type other than those whose change is not allowed is output.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.29.4 Setting the change protection level and output protection level of a program (15/19-inch display unit)

The display/operations indicated below can be performed from the program folder screen.
The change protection level and output protection level of each part program are displayed.
The change protection level and output protection level of each part program can be changed.

Setting the change protection level and output protection level of a program

Procedure

1 Press function key [PROG].

2 Press vertical soft key [FOLDER].

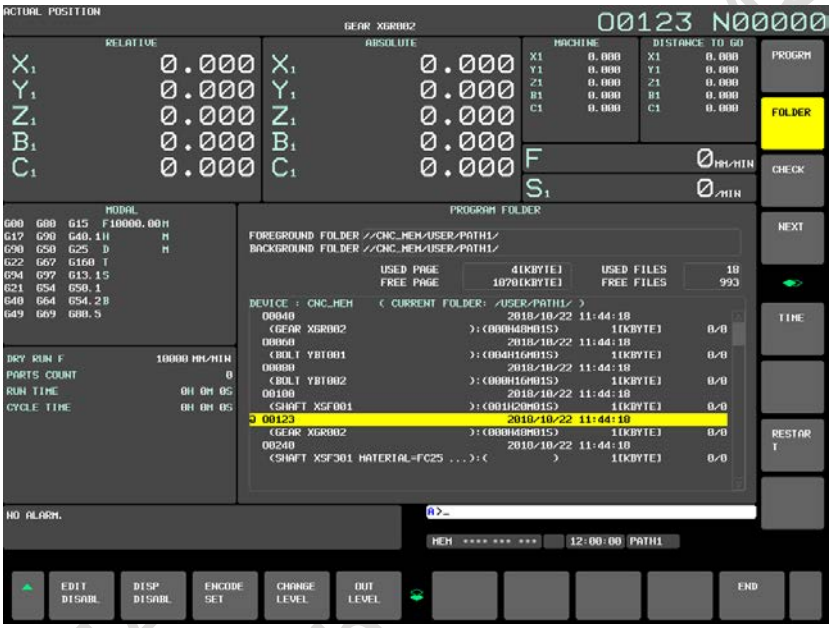


Fig. 12.3.29.4 (a) Program directory screen (15-inch display unit)

3 Press horizontal soft key [DETAIL ON] to switch to detail displays.

4 Move the cursor to a desired program.

5 Press the continuous menu key [▷] to display horizontal soft key [CHANGE ATTR].

6 Press horizontal soft key [CHANGE ATTR].

7 Key in a new desired level, then press horizontal soft key [CHANGE LEVEL].

8 To change the output protection level, key in a new desired level, then press horizontal soft key [OUT LEVEL].

Explanation

The change protection level (0 to 7) and output protection level (0 to 7) are displayed as "CHANGE PROTECTION LEVEL VALUE/OUTPUT PROTECTION LEVEL".

NOTE

1 When the protection level of data is higher than the current operation level, the protection level cannot be changed.

2 The protection level of data cannot be changed to a level higher than the current operation level.

3 A protection level can be set only for those part programs that are held on the "CNC_MEM" device.

12.3.30 Precision Level Selection (15/19-inch Display Unit)

An intermediate precision level between the parameters for emphasis on velocity (precision level 1) and the parameters for emphasis on precision (precision level 10) set on the machining parameter tuning screen (AI contour) can be selected. As shown in the Fig. 12.3.30 (a), the levels are proportionally linear, and an intermediate level can be selected so that optimal parameters can be automatically calculated to perform machining.

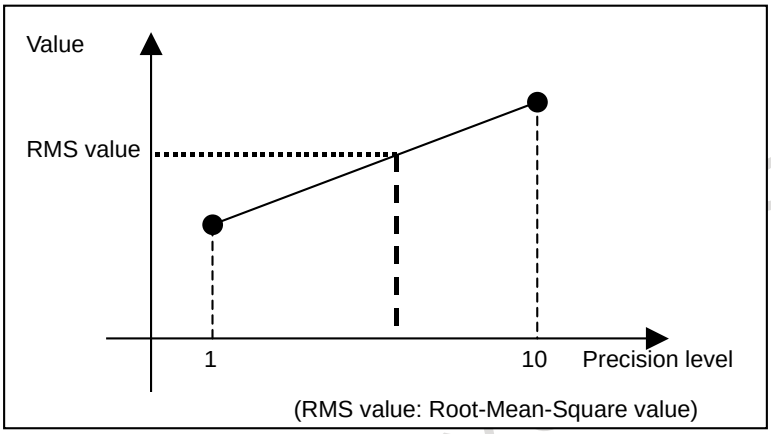


Fig. 12.3.30 (a) Image of "level"

Procedure for precision level selection

1 Select the MDI mode.

2 Press function key .

3 Press vertical soft key [NEXT PAGE] several times to display vertical soft key [PRECI LEVEL].

4 Press vertical soft key [PRECI LEVEL].

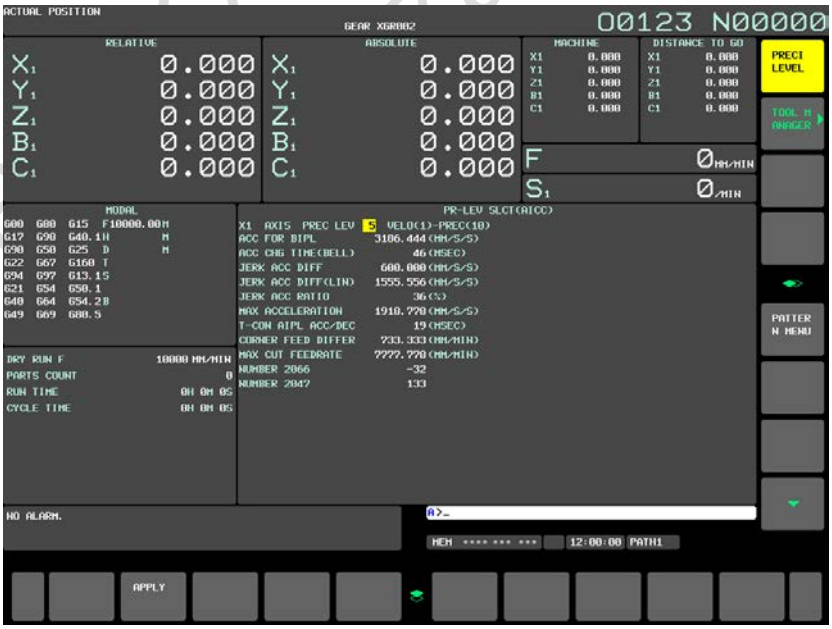


Fig. 12.3.30 (b) Precision level selection screen

5 To change the precision level, key in a desired precision level (1 to 10), then press MDI key .

6 When the precision level is changed, a RMS value is obtained from the velocity-emphasized parameter set and precision-emphasized parameter set for parameter modification.
For the modified parameters, see the description of the machining parameter tuning.

7 If there is an axis in addition to the currently displayed axes, press page key

or

several times to display the screen for the axis.

12.3.31 Machining Level Selection (15/19-inch Display Unit)

12.3.31.1 Smoothing level selection

An intermediate smoothing level between the parameters for smoothing level 1 and the parameters for smoothing level 10 set on the machining parameter tuning screen (smoothing) can be selected. As shown in the Fig. 12.3.31.1 (a), the levels are proportionally linear, and an intermediate level can be selected so that optimal parameters can be automatically calculated to perform machining.

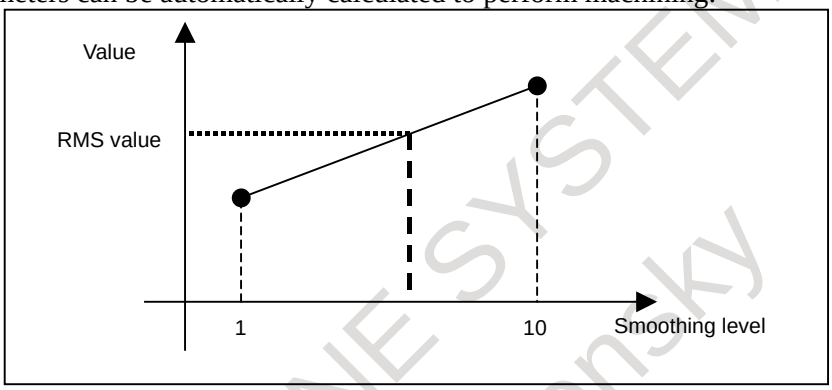


Fig. 12.3.31.1 (a) Image of "level"

Procedure for smoothing level selection

1 Select the MDI mode.

2 Press function key

.

3 Press vertical soft key [NEXT PAGE] several times to display vertical soft key [MACHIN LEVEL].

4 Press vertical soft key [MACHIN LEVEL] to display vertical soft key [SMOOTH LEVEL].

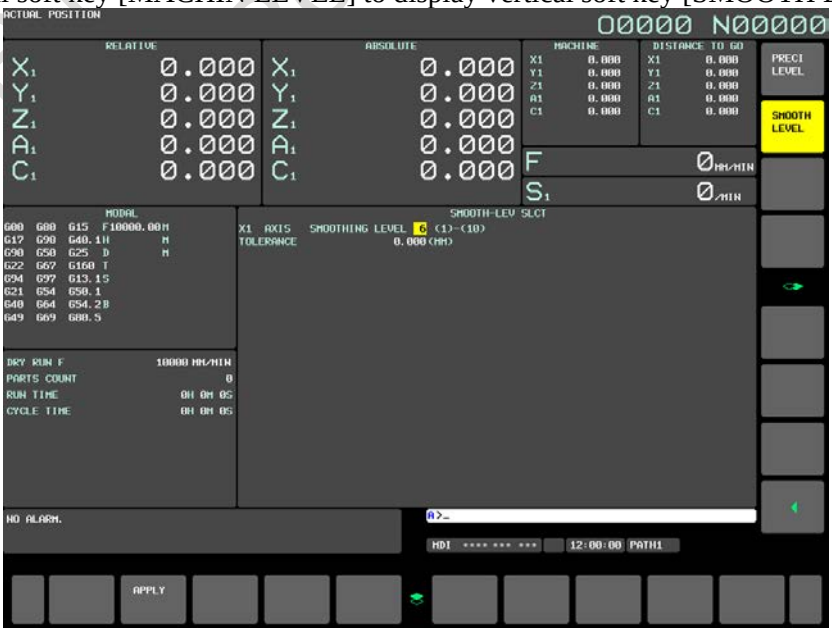


Fig. 12.3.31.1 (b) Smoothing level selection screen

FRYER MACHINE SYSTEMS INC
Jacob Zahensky

5 To change the smoothing level, key in a desired smoothing level (1 to 10), then press MDI key INPUT .

6 When the smoothing level is changed, a RMS value is obtained from the smoothing level 1 parameter set and smoothing level 10 parameter set for parameter modification.
For the modified parameters, see the description of the machining parameter tuning.

7 If there is an axis in addition to the currently displayed axes, press page key PAGE or PAGE several times to display the screen for the axis.

12.3.31.2 Precision level selection

For details of precision level selection, See Subsection, “Precision Level Selection (15/19-inch Display Unit)”.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.32 Displaying and Setting Tool Life Management Data (15/19-inch Display Unit)

Displaying tool life management data on a screen enables the current status of tool life management to be grasped. Also on the screen, tool life management data can be edited. The screen is either:

- Tool life management (list screen) or
- Tool life management (group editing screen)

Overview

List screen

Tool life management (list screen)
Displayed items:
- NEXT GROUP
- USING GROUP
- SELECTED GROUP
- COUNT OVERRIDE
- GROUP NO.
- COUNT TYPE
- LIFE
- LIFE COUNT
- TOOL MANAGEMENT STATUS
- REGISTERED TOOL NUMBER
- GROUP TO BE CHANGE

Functions:
- Searching for groups
- Clearing execution data
- Setting the tool life counter

EDIT

END

Group editing screen

Tool life management (group editing screen)
Displayed items:
- NEXT GROUP
- USING GROUP
- SELECTED GROUP
- GROUP NO.
- MAX.TOOL PIECES
- COUNT TYPE
- LIFE
- LIFE COUNT
- T-CODE
- TOOL MANAGEMENT STATUS
- H-CODE and D-CODE
- TOOL NO.

Functions:
- Setting tool data
- Deleting group data
- Setting a tool status
- Deleting tool data
- Selecting tool skip
- Searching for groups
- Setting a life count type, life value, and tool life counter

Pressing horizontal soft key [EDIT] or [END] switches from the list screen to the group editing screen or vice versa.

M

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1), the following parameters can be used to display arbitrary groups and remaining set values.

- If bit 5 (TGN) of parameter No. 6802 = 1
  An arbitrary group is displayed on the list and group editing screens.
- If bit 3 (GRP) of parameter No. 6802 = 1
  Remaining set values are displayed on the group editing screen. The group editing screen always displays H and D codes.

T

The T series is provided with two tool change types, turret type and ATC type. For the T series, displays vary depending on which tool change type is in use.
If the turret type is in use (bit 3 (TCT) of parameter No. 5040 = 0)

- Neither the H code nor the D code is used; neither is displayed.
- No arbitrary group can be used; no arbitrary group is displayed.

If the ATC type is in use (bit 3 (TCT) of parameter No. 5040 = 1)

- The D code is displayed on the group editing screen.
- If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1) and bit 5 (TGN) of parameter No. 6802 = 1, arbitrary groups are displayed on the list and group editing screens.

If the tool life management B function is enabled (bit 4 (LFB) of parameter No.6805 = 1) and bit 3 (GRP) or parameter No. 6802 = 1, remaining set values are displayed on the group editing screen.

12.3.32.1 Tool life management (list screen) (15/19-inch display unit)

This screen can display the life management status of all tools in tool groups and whether the life of the tool groups has expired. It also enables you to set tool life counters and clear execution data.

Displays on the list screen

Procedure

1 Press function key .
2 Press vertical soft key [NEXT PAGE] several times to display soft key [TOOL LIFE].
3 Press vertical soft key [TOOL LIFE].

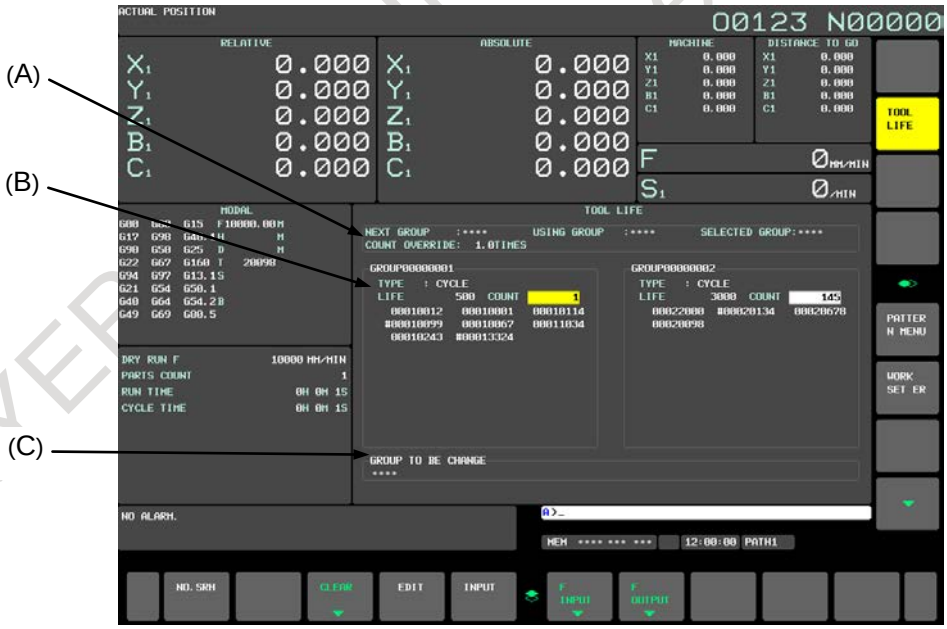


Fig. 12.3.32.1 (a) Displaying tool life management (list screen) (15-inch display unit)

FRYER ... NE SYSTEMS INC ... ensky

- Contents of (A)

(A) displays tool group numbers and an override value. If there is no tool group to display, ****, instead of tool group numbers, is displayed.

NEXT GROUP : Tool group number for which life counting is started by the next M06 command.
USING GROUP : Tool group number for which life counting is currently under way.
SELECTED GROUP : Tool group number for which life counting is currently under way or life counting has been performed most recently.
COUNT OVERRIDE : "1.0TIMES" is displayed if the tool life counter override signal is disabled (bit 2 (LFV) of parameter No. 6801 = 0).

NOTE
If arbitrary group numbers are enabled, NEXT GROUP, USING GROUP, and SELECTED GROUP are each represented with an arbitrary group number rather than the tool group number.

- Contents of (B)

(B) displays the set life value, the current content of the tool life counter, and the registered tool numbers (in the order they are used) for each tool group. If the life count type is specification by duration, the measurement unit used in displaying and specifying set life values and tool life counter values is selected according to the setting of bit 0 (FCO) of parameter No. 6805 as listed below.

Bit 0 (FCO) of parameter No. 6805	0	1
Measurement unit used in displaying and specifying set life values and tool life counter values	1 minute	0.1 minutes

The following table lists the prefixes used with tool numbers.

Tool status	Tool in use	Tool not in use
Life remaining	@	No indication
Skip	#	#
Life expired	*	*

NOTE
1 The tool life counter indicates the count value for the tool indicated with @.
2 If bit 3 (EMD) of parameter No. 6801 = 0, a tool number remains prefixed with @ even if the life of the tool has expired until another tool is selected.
3 If bit 3 (EMD) of parameter No. 6801 =1, the following differences can occur depending on the type of the life counter in use.
- If the life counter type is specification by duration, the prefix is changed to "*" (life expired) as soon as the tool life expires.
- If the life counter type is specification by count, the counter is incremented by one at the end of a program (such as M02 or M30). So, the prefix is not changed to "*" (life expired) even if the tool life counter matches the life value. The symbol "*" (life expired) appears when the tool life count is incremented after the CNC is reset.

4 If bit 2 (ETE) of parameter No. 6804 = 1, the symbol "*" indicating the expiration of the life of the last tool appears on the tool life management screen when the life counter for the last tool of the tool group of interest matches the life value. This way, the information about the last tool in the FOCAS2 or PMC window indicates that the life of the tool has expired if the tool change signal TLCH is "1".

FRYER MACHINE SYSTEMS INC
Rob Zahenski

If arbitrary group numbers are enabled, an arbitrary group number is displayed in the parentheses beside the tool group number. If no arbitrary group number is specified, “********” is displayed instead.

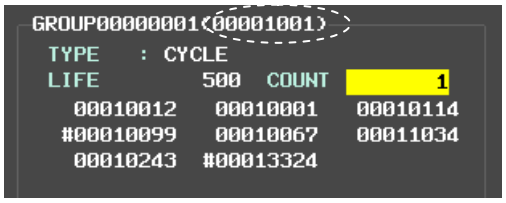


Fig. 12.3.32.1 (b) Displaying arbitrary group numbers

Arbitrary group numbers are enabled by setting the following parameters.

M

- The tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1).
- Arbitrary group numbers are enabled (bit 5 (TGN) of parameter No. 6802 = 1).

T

- The current tool change type is ATC (bit 3 (TCT) of parameter No. 5040 = 1).
- The tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1).
- Arbitrary group numbers are enabled (bit 5 (TGN) of parameter No. 6802 = 1).

- Contents of (C)

(C) displays tool group numbers for which a tool change signal has been issued.
If there are so many tool group numbers that all the numbers cannot be displayed, some are omitted, and “>>“ is displayed instead.
If there is no tool group number in need of change, “****” is displayed.

NOTE
If arbitrary group numbers are enabled, a tool group number in need of change is represented with an arbitrary group number rather than the tool group number.

Setting data on the list screen

Tool life management data can be specified in the reset state (both the OP and RST signals are “0”). However, setting bit 1 (TCI) of parameter No. 6804 to 1 enables tool life management data to be specified even during automatic operation (the OP signal is “1”).

NOTE
As for USING GROUP or NEXT GROUP settings:
1) During automatic operation (OP signal = “1” and bit 1 (TCI) of parameter No. 6804 = 1), only the tool life counter can be changed.
2) In the reset state (OP signal = “0” and RST signal “0”), the following editing operation stops life management because it cannot be continued.
   - Clearing execution data

FRYER MACHINE SYSTEMS INC
Bob Zaharisky

Procedure

- Setting the tool life counter

The tool life counter can be set with a value, using the following methods.

Method 1

1 Place the cursor on the tool life counter for a desired tool group.
2 Enter the value from the keypad.
3 Press horizontal soft key [INPUT].

Method 2

1 Place the cursor on the tool life counter for a desired tool group.
2 Enter the value from the keypad.
3 Press INPUT key.

- Clearing execution data

All existing execution data for a tool group selected by the cursor can be cleared as follows:

1 Place the cursor on the tool group whose execution data you want to clear.
2 Press horizontal soft key [ERASE].
3 Press horizontal soft key [EXEC].

NOTE
Setting bit 4 (GRS) of parameter No. 6800 to 1 enables execution data for all registered tool groups to be cleared.

- Selecting tool groups

Tool groups can be selected using the following methods.

Method 1

1 Enter a tool group number from the keypad.
2 Press soft key [NO.SRH].

NOTE
If arbitrary group numbers are enabled, a tool group is selected by searching for an arbitrary group number rather than the tool group number.

Method 2

1 Press page key PAGE or PAGE to display desired groups.
2 Press cursor key ← or → to move the cursor to the desired group at either the left or right.

- Switching to the group editing screen

Switch to tool life management (group editing screen).

1 Move the cursor to the tool group you want to edit:
2 Press horizontal soft key [EDIT].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.32.2 Tool life management (group editing screen) (15/19-inch display unit)

On this screen, it is possible to edit tool life management data (such as tool life value, tool life counter, and tool data) for the tool group of interest.

Displaying the group editing screen

Procedure

1. Place the cursor on the tool group you want to edit from the list screen.
2. Press horizontal soft key [EDIT].

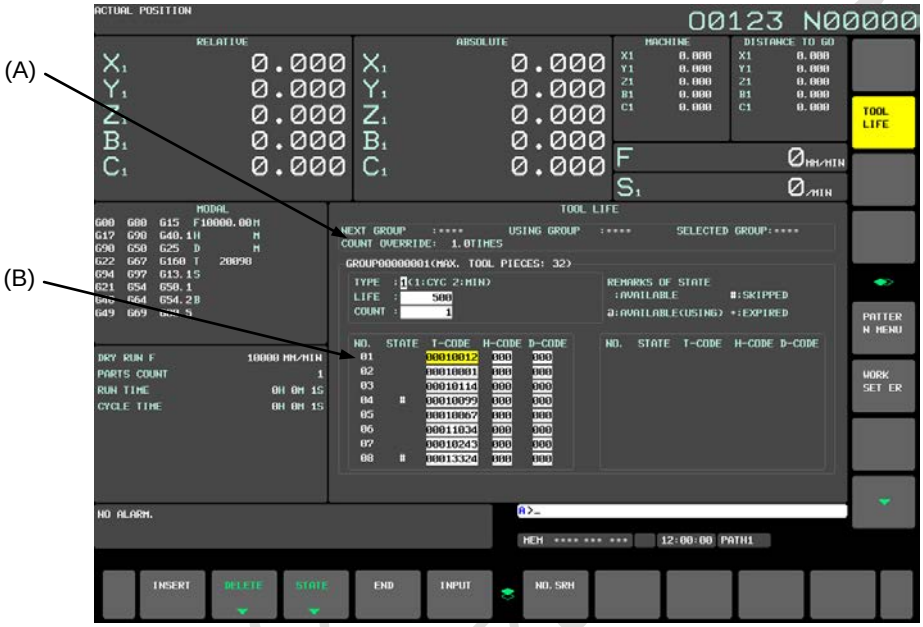


Fig. 12.3.32.2 (a) Displaying tool life management (group editing screen) (15-inch display unit)

> **NOTE**
> If no tool is registered with a tool group, none of a life count type, a life value, and a tool life counter value is displayed for the tool group.

- Contents of (A)

Like the counterpart of the list screen, (A) of the editing screen displays the next tool group number and override value. If there is no appropriate tool group, **** is displayed instead.

NEXT GROUP : Tool group for which life counting is started by the next M06 command.
USING GROUP : Tool group number for which life counting is currently under way.
SELECTED GROUP : Tool group number for which life counting is currently under way or life counting has been performed most recently.
COUNT OVERRIDE : “1.0TIMES” is displayed if the tool life counter override signal is disabled (bit 2 (LFV) of parameter No. 6801 is 0).

> **NOTE**
> If arbitrary group numbers are enabled, NEXT GROUP, USING GROUP, and SELECTED GROUP are each represented with an arbitrary group number rather than the tool group number.

- Contents of (B)

(B) details tool life management data related to a selected tool group as follows:

TYPE :

1	Specification by count
2	Specification by duration

LIFE : Tool life value
COUNT : Tool life counter

STATE :

Tool status	Tool in use	Tool not in use
Life remaining	@	No indication
Skip	#	#
Life expired	*	*

T-CODE : Tool number

M

H-CODE : Tool length compensation specification code
D-CODE : Cutter compensation specification code

T

H-CODE : No display.
D-CODE : Tool offset value specification code
if ATC type is in use (bit 3 (TCT) of parameter No. 5040 = 1)

NOTE

1 The tool life counter indicates the count value for the tool indicated with @.
2 If bit 3 (EMD) of parameter No. 6801 is 0, a tool number remains prefixed with @ even if the life of the tool has expired until another tool is selected.
3 If bit 3 (EMD) of parameter No. 6801 is 1, the following differences can occur depending on the type of the life counter in use.
- If the life counter type is specification by duration, the prefix is changed to “*” (life expired) as soon as the tool life expires.
- If the life counter type is specification by count, the counter is incremented by one at the end of a program (such as M02 or M30). So, the prefix is not changed to “*” (life expired) even if the tool life counter matches the life value. The symbol “*” (life expired) appears when the tool life count is incremented after the CNC is reset.

4 If bit 2 (ETE) of parameter No. 6804 is 1, the symbol “*” indicating the expiration of the life of the last tool appears on the tool life management screen when the life counter for the last tool of the tool group of interest matches the life value. This way, the information about the last tool in the FOCAS2 or PMC window indicates that the life of the tool has expired if the tool change signal TLCH is “1”.

Displaying arbitrary groups and remaining set values

Fig. 12.3.32.2 (b) Displaying arbitrary group numbers and remaining set values on the editing screen

FRYER MACHINE SYSTEMS INC
Rob Zakensky

M

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 is 1) and the following parameters are enabled, arbitrary groups and remaining set values are displayed.
OPTION GROUP : Arbitrary group number (if bit 5 (TGN) of parameter No. 6802 is 1)
REST COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 is 1)

T

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 is 1)
Turret type (bit 3 (TCT) of parameter No. 5040 is 0)
OPTION GROUP : No display.
REST COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 is 1)

ATC type (bit 3 (TCT) of parameter No. 5040 is 1)
OPTION GROUP : Arbitrary group number (if bit 5 (TGN) of parameter No. 6802 is 1)
REST COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 is 1)

Setting data on the group editing screen

Tool life management data can be specified in the reset state (both the OP and RST signals are “0”). However, setting bit 1 (TCI) of parameter No. 6804 to 1 enables tool life management data to be specified even during automatic operation (the OP signal is “1”).
The available editing operations are listed below.

M

Items that can be edited	Mode
Setting a life count type, life value, tool life counter, and tool data (T code, H code, and D code)	All modes
Adding tool numbers (T code)	MDI
Deleting all tool group data at a time	MDI
Deleting tool data (status, T code, H code, and D code)	MDI
Selecting tool skip	MDI
Specifying to clear tool data (life re-set)	MDI

T

Items that can be edited	Mode
Setting a life count type, life value, tool life counter, and tool data (T code and D code) However, the D code can be edited only if the ATC type is in use (bit 3 (TCT) of parameter No. 5040 is 1).	All modes
Adding tool numbers (T code)	MDI
Deleting all tool group data at a time	MDI
Deleting tool data (status, T code, and D code)	MDI
Selecting tool skip	MDI
Specifying to clear tool data (life re-set)	MDI

If no tool is registered with a tool group, none of a life count type, a tool life value, and a tool life counter value can be set for the tool group. First of all, add a tool number (T code).

FOYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

1 As for USING GROUP or NEXT GROUP editing:
  <1> During automatic operation (OP signal is “1” and bit 1 (TCI) of parameter No. 6804 is 1), only the tool life counter can be changed.
  <2> In the reset state (OP signal is “0” and RST signal is “0”), the following editing operation stops life management because it cannot be continued.
    - Adding tool numbers (T code)
    - Deleting all tool group data at a time
    - Deleting tool data (status, T code, H code, and D code)
2 The following editing operations may set the tool change signal to “1”.
  - Selecting tool skip for the last tool.
  - Deleting tool numbers, resulting in any tool other than those whose life has expired or who have been skipped being not found in the tool group of interest.
3 The following editing operations may reset the tool change signal to “0”.
  - Adding tool numbers, leading to tools whose life has not expired being set in the tool group of interest.
  - Selecting tool clear.

Procedure

- Setting a life count type, tool life value, tool life counter, tool data, arbitrary group number, and remaining set value

Setting a life count type, tool life value, tool life counter, tool data, arbitrary group number, and remaining set value

Method 1

1 Place the cursor on the desired item.
2 Enter a value from the keypad.
3 Press horizontal soft key [INPUT].

Method 2

1 Place the cursor on the desired item.
2 Enter a value from the keypad.
3 Press

.

NOTE

1 Changing a tool life value or tool life counter does not affect the tool status or tool change signal.
2 Changing a life count type causes the tool life value and too life counter to be reset to 0.

M

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 is 1) and the following parameters are enabled, arbitrary groups and remaining set values can be specified

OPTION GROUP : Arbitrary group number (if bit 5 (TGN) of parameter No. 6802 is 1)
REST COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 is 1)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

T

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 is 1)
Turret type (bit 3 (TCT) of parameter No. 5040 is 0)
OPTION GROUP : Cannot be set.
REST COUNT : It is possible to set a remaining set value to be used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 is 1).
ATC type (bit 3 (TCT) of parameter No. 5040 is 1)
OPTION GROUP : Arbitrary group numbers can be set (if bit 5 (TGN) of parameter No. 6802 is 1).
REST COUNT : It is possible to set a remaining set value to be used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 is 1).

- Adding tool numbers

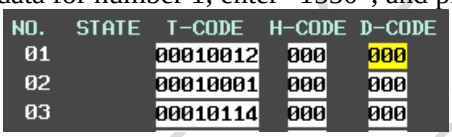
Tool numbers can be added to a tool group as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) just before a tool number to be added.
3 Enter the tool number from the key pad.
4 Press horizontal soft key [INSERT].

(Example)

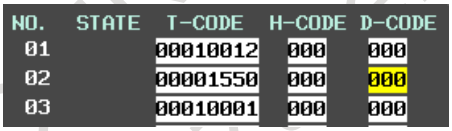
Adding tool number 1550 between numbers 1 and 2 (for the M series)

1 Move the cursor to the data for number 1, enter “1550”, and press [INSERT].



NO.	STATE	T-CODE	H-CODE	D-CODE
01		00010012	000	000
02		00010001	000	000
03		00010114	000	000

2 The entered T code 1550 is inserted in the position of number 2. The H and D codes are reset to 0.



NO.	STATE	T-CODE	H-CODE	D-CODE
01		00010012	000	000
02		00001550	000	000
03		00010001	000	000

- Deleting all tool group data at a time

All tool group data can be deleted at a time as follows:

1 Select the MDI mode.
2 Select the tool group from which you want to delete all tool data at a time.
3 Press horizontal soft key [DELETE].
4 Press horizontal soft key [GROUP].
5 Press horizontal soft key [EXEC].

- Deleting tool data

Tool data can be deleted from a tool group as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) you want to delete.
3 Press horizontal soft key [DELETE].
4 Press horizontal soft key [<CURS>].

NOTE

1 Deleting all tools from a tool group is equivalent to deleting the tool group itself.
2 Deleting a tool indicated with @ (in use) results in @ being moved to the previous tool whose life has expired or which has been skipped.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

- Selecting tool skip

Tool data can be placed in a skip state as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) for a tool you want to skip.
3 Press horizontal soft key [STATE].
4 Press horizontal soft key [SKIP].

- Specifying to clear tool data (life re-set)

Tool data state can be cleared as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) for a tool you want to clear.
3 Press horizontal soft key [STATE].
4 Press horizontal soft key [CLEAR].

- Selecting a tool group

A tool group can be selected as follows:

Method 1

1 Enter a tool group number from the keypad.
2 Press horizontal soft key [NO.SRH].

NOTE

If arbitrary group numbers are enabled, a tool group is selected by searching for an arbitrary group number rather than the tool group number.

Method 2

1 Press page key

or

to display the target tool group.

- Switching to the list screen

The tool life management (list screen) can be resumed as follows:

1 Press horizontal soft key [END].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.33 Displaying and Setting Pattern Data Inputs (15/19-inch Display Unit)

Described below are a method for displaying machining menus (pattern menus) created by machine tool builders and a method for setting them. The descriptions are based on an example. For actual pattern menus and pattern data, refer to manuals released by respective machine tool builders.

Displaying pattern data and pattern menus

Given below is a procedure for displaying pattern menus.

1 Press function key [OFS/SET].

2 Press vertical soft key [NEXT PAGE] several times.

3 Press vertical soft key [PATTERN MENU].
The pattern menu screen (Fig. 12.3.33 (a)) shown below appears.

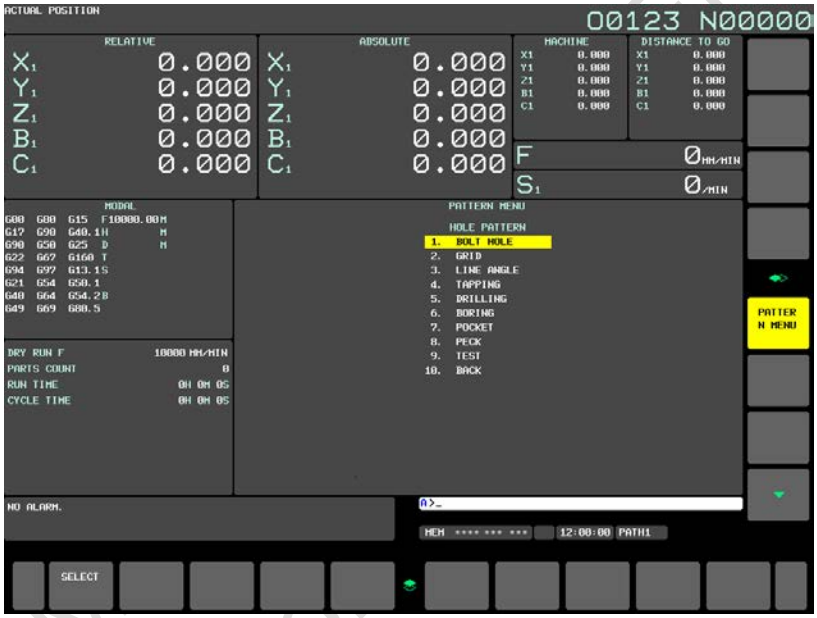


Fig. 12.3.33 (a) Pattern menu screen (15-inch display unit)

On this screen, a pattern to be used can be selected.
The following two methods can be used to select patterns.

- Using the cursor

  1 Move the cursor to a pattern name you want to select, using cursor key [↑] or [↓], and then press soft key [SELECT] or [INPUT].

- Specifying a pattern number

  1 Enter a number displayed at the left side of a pattern name, and press soft key [SELECT] or [INPUT].
  The custom macro screen (pattern data screen) (Fig. 12.3.33 (b)) shown below appears.

FRYER INDUSTRIAL SYSTEMS INC

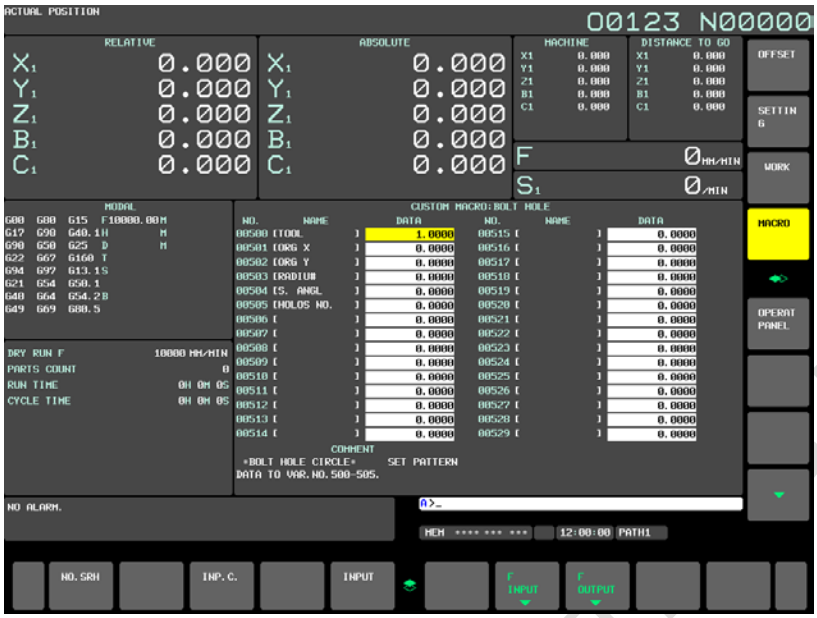


Fig. 12.3.33 (b) Custom macro screen (pattern data) (15-inch display unit)

2 Enter the necessary pattern data, and press INPUT.

3 After entering all necessary data, select the MEM mode and press the cycle start button. Machining begins.

Explanation

- Explanations about the pattern menu screen

HOLE PATTERN

An arbitrary character string consisting of 12 or less characters can be displayed as a menu title.

BOLT HOLE

An arbitrary character string consisting of 10 or less characters can be displayed as a pattern name.

Machine tool builders should program menu title and pattern name character strings, using custom macros, and save them in program memory.

- Explanations about the custom macro screen (pattern data screen)

BOLT HOLE

An arbitrary character string consisting of 12 or less characters can be displayed as a pattern data title.

TOOL.

An arbitrary character string consisting of 10 or less characters can be displayed as a variable name.

BOLT HOLE CIRCLE

The comment text consists of nine 12-character blocks, or the comment can be up to 12 lines (10.4-inch display unit) with one block counted as one line.

Machine tool builders should program variable name and comment text character strings, using custom macros, and save them in program memory.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.3.34 Setting and Displaying Data When the Tool Offset for Milling and Turning Function Is Enabled (15/19-inch Display Unit)

T

Setting and displaying tool offset values (15/19-inch display unit)

Procedure

1 Press function key .

In multi-path control, select the path for which to set tool offset values, using the path selection switch in advance.

2 Press chapter selection vertical soft key [OFFSET].

Alternatively, press function key several times until the tool offset screen appears.

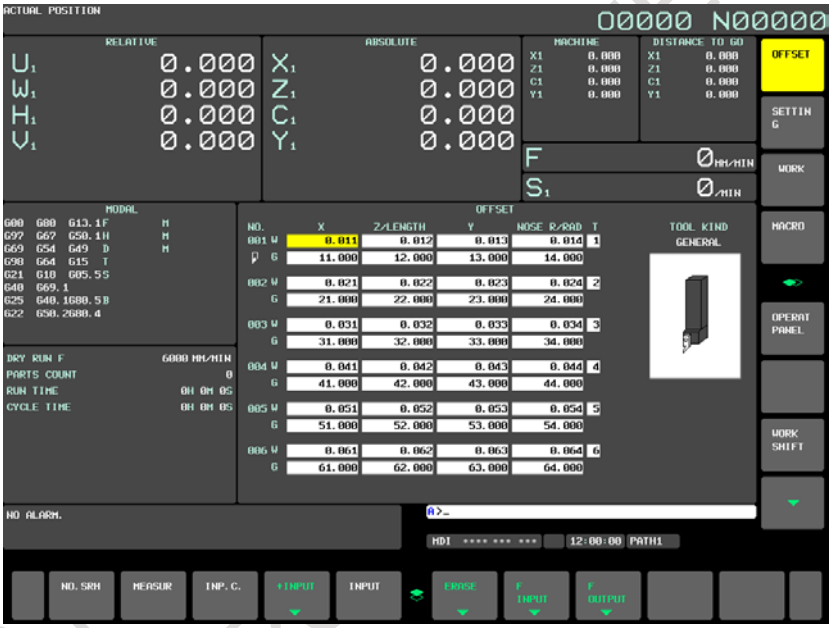


Fig. 12.3.34 (a) Tool offset screen (15-inch display unit)

X : Set the X-axis tool offset value.
Z/LENGTH : Set the Z-axis tool offset value or tool length compensation value.
Y : Set the Y-axis tool offset value.
NOSE R/RAD : Set the tool nose radius compensation value or tool radius compensation value.
T : Set the virtual tool nose direction.

3 Press page and cursor keys to move the cursor to the offset value you want to set or change.
Alternatively, enter the number of the offset value you want to set or change and press horizontal soft key [NO.SRH].

4 To set the offset value, enter the value and press horizontal soft key [INPUT].
To change the offset value, enter a value to be incremented to or decremented from the current setting and press horizontal soft key [+INPUT].
Alternatively, enter a new offset value and press horizontal soft key [INPUT].

FRYER PRODUCTS SYSTEMS INC

Explanation

- Decimal point programming

The offset value can be entered with a decimal point.

- Another setting method

The tool offset value can be input/output also by using external I/O devices.
See Chapter, “DATA I/O”.
The tool offset value can also be set by using the direct input of offset value measured B, tool offset value counter input, and other functions.

- Number of tool offsets

128 tool offsets are available as standard.
In a multi-path system, one of the above numbers of tool offsets can be selected for each path.

- Disabling offset value input

By setting bits 0 (WOF) and 1 (GOF) of parameter No. 3290, the input of tool wear compensation values and tool geometry compensation values is disabled.
The input of the tool offset values in any specified range from the MDI can be disabled by setting the number of the first target tool offset value in parameter No. 3294 and the number of target tool offset values starting from the first value in parameter No. 3295.
If offset values are entered continuously, they are handled as follows:

1) If an offset value that can be input is input, then an offset value that cannot be input is input, a warning is output, but the offset value in the input enable range is set.
2) If an offset value that cannot be input is input, then an offset value that can be input is input, a warning is output, and the offset values are not set.

12.3.35 Eco Setting Screen (15/19-inch Display Unit)

Select a desired Eco level before starting machining, after comparing data of power consumption and cycletime for each Eco level.

Displaying Eco setting screen

Procedure

1 Press function key .
2 Press [NEXT PAGE] several times to display soft key [ECO].
3 Press vertical soft key [ECO].
4 Press vertical soft key [ECO SET].

FRYER MACHINE SYSTEMS INC
Rob Zalensky

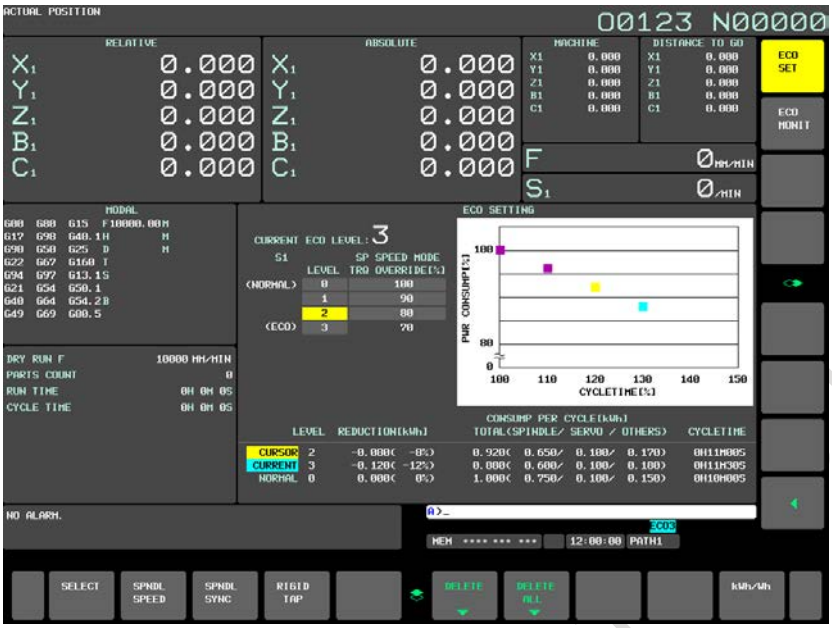


Fig. 12.3.35 (a) Eco setting screen

Explanation

- **Display for select an Eco level**

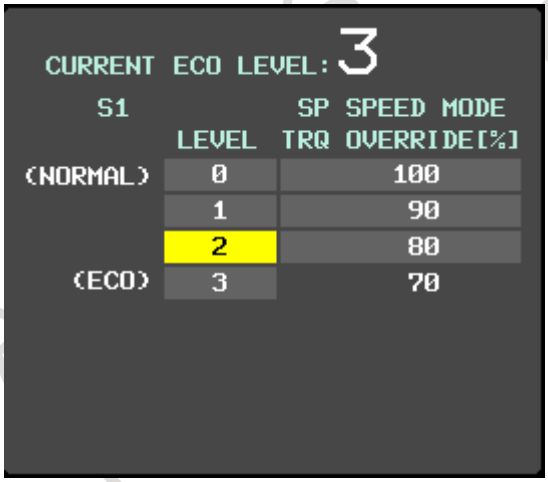


Fig. 12.3.35 (b) Display for select an Eco level

Select an Eco level here.
The selected Eco level for the current path is also displayed.

Items	Contents
Current Eco level	The selected Eco level is displayed.
Level	Available Eco levels are displayed. Level 0 means normal machining. Highlight a desired Eco level and press operation menu soft key [SELECT], and the selected Eco level is changed.

Items	Contents
Torque override	Torque overrides for each level are displayed. Torque overrides for spindle speed control mode, spindle synchronization control mode and rigid tapping mode are displayed. Since only one mode can be displayed at a time, pressing soft keys [SPNDL SPEED][SPNDL SYNC][RIGID TAP] changes displayed mode. In case of parameter SYC(No.8133#4)=0, torque overrides of spindle synchronization control mode are not displayed. And in case of parameter NRG(No.8135#3)=1, torque overrides of rigid tapping mode are not displayed. If multiple spindles belong to the current path, pressing MDI keys [PAGE ⇧] [PAGE ⇩] change spindles, whose torque overrides are displayed.

NOTE
When an invalid value (out of 0–100%) is specified in the parameter No.11397, 11398 or 5260, torque overrides are displayed as “100” in red, excluding the torque override for Eco level 0.

- **Graphic display for data**

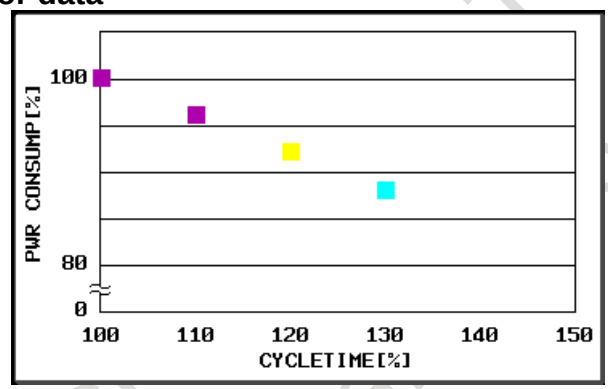


Fig. 12.3.35 (c) Graph for eco setting

Data of each Eco level are displayed in power consumption vs. cycletime.
Data of the current path are displayed.

- “Light blue” point shows data of the selected Eco level.
- “Yellow” point shows data of the highlighted Eco level.
- “Pink” points show data of available Eco levels.

NOTE
Nothing is displayed without data of Eco level 0.

- **Detailed data comparison**

	LEVEL	REDUCTION[kWh]	CONSUMP PER CYCLE[kWh] TOTAL(SPINDLE/ SERVO / OTHERS)	CYCLETIME
CURSOR	2	-0.080(-8%)	0.920(0.650/ 0.100/ 0.170)	0H11M00S
CURRENT	3	-0.120(-12%)	0.880(0.600/ 0.100/ 0.180)	0H11M30S
NORMAL	0	0.000(0%)	1.000(0.750/ 0.100/ 0.150)	0H10M00S

Fig. 12.3.35 (d) Comparison of data for eco setting

Detailed data of power consumption and cycletime of each Eco level can be compared here in order to select a desired Eco level.
Data of the current path are displayed.

Items	Contents
Level	Eco levels are displayed.
Reduction	Reductions in power consumption per cycle are displayed. Reductions are calculated by using power consumption at normal machining (Eco level 0) as the base.
Power consumptions per cycle (spindle/servo/others)	Total power consumptions per cycle are displayed. Power consumptions in spindles, servos and others (peripheral devices) are also displayed separately.
Cycletime	Cycletimes are displayed.

NOTE
1 Power consumption data obtained by actual machining is displayed. Therefore, 0 is displayed at the first machining or soon after deleting data. Though these data depends on parameter settings, machining circumstances and so on, soon after changing settings, data obtained before the change is displayed.
2 Here, a “cycle” means one automatic operation.

Operation on Eco setting screen

Procedure

- Select of an Eco level

Select a desired Eco level with comparing among data of power consumption and cycletime at each Eco level.

1 Select MDI mode.
2 Highlight a desired Eco level.
3 Press horizontal soft key [SELECT].
The selected Eco level is displayed in “CURRENT ECO LEVEL” and the select is completed.

NOTE
Select an Eco level for each path when the select is available for each path (bit 1 (EEP) of the parameter No.24303 is set to 1).

- Deleting data of power consumption and cycletime at each Eco level

Disused data of power consumption and cycletime can be deleted individually or in a batch.
Both individual and batch deletes are effective only on data of the current path.

1 Highlight an Eco level whose data is going to be deleted.
Batch delete does not require this operation.
2 Press horizontal soft key [DELETE] or [DELETE ALL] on operation menu.
If [DELETE] is pressed, individual data is deleted.
If [DELETE ALL] is pressed, all data of the current path are deleted.
Select menu [EXEC]/[CANCEL] is displayed.
3 Press horizontal soft key [EXEC].
Data is deleted.

- Switching displayed unit of power consumption

Displayed unit of power consumption can be switched to “kWh” or “Wh”.
Soft key [kWh/Wh] switches the unit to “kWh” or “Wh”.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Switching spindle mode whose torque overrides are displayed

The spindle mode whose torque overrides are displayed can be switched by soft keys [SPNDL SPEED], [SPNDL SYNC] and [RIGID TAP].
In case of parameter SYC(No.8133#4)=0, torque overrides of spindle synchronization control mode are not displayed. And in case of parameter NRG(No.8135#3)=1, torque overrides of rigid tapping mode are not displayed.

- Switching spindle whose torque overrides are displayed

When multiple spindles belong to the current path, MDI keys [PAGE ↑] [PAGE ↓] can switch the spindle whose torque overrides are displayed.

- Change of torque overrides

Torque override for Eco level 3 can be changed.

1 Set bit 0 (PWE) of the parameter No.8900 to 1.
2 Highlight the torque override for Eco level 3.
3 Select MDI mode or enter state emergency stop for all paths.
4 Release spindle synchronization mode for all paths.
5 Input value by MDI keys.
Available values are integers from 1 to 100.
6 Press soft key [INPUT] or MDI key [INPUT].
Torque overrides for each Eco level are changed.

> ⚠ **CAUTION**
> Machine or work can be damaged by out-of-synchronization when torque overrides of spindles in synchronization control mode differ from each other. This is because this function differs the time constant for acceleration/deceleration in spindle synchronization mode.
> Set a common torque override for spindles belonging to the same group of synchronization.

> **NOTE**
> 1 Use this operation only for deciding the point of the least power consumption. Please refer to “Energy Saving Level Selecting Function “ of CONNECTION MANUAL (FUNCTION) (B-64693EN-1) for details.
> 2 When select of an Eco level is available for each path (bit 1 (EEP) of the parameter No.24303 is set to 1), operations 3 and 4 are needed only for the current path.
> 3 When bit 0 (PWE) of the parameter No.8900 is set to 1, the torque override cannot be highlighted.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

12.3.36 Eco Monitoring Screen (15/19-inch Display Unit)

Power consumption of machine can be observed comprehensively, such as effects of eco-machining and time transition of power consumption.

Displaying Eco monitoring screen

Procedure

1 Press function key .
2 Press [NEXT PAGE] several times to display soft key [ECO].
3 Press vertical soft key [ECO].
4 Press vertical soft key [ECO MONIT].

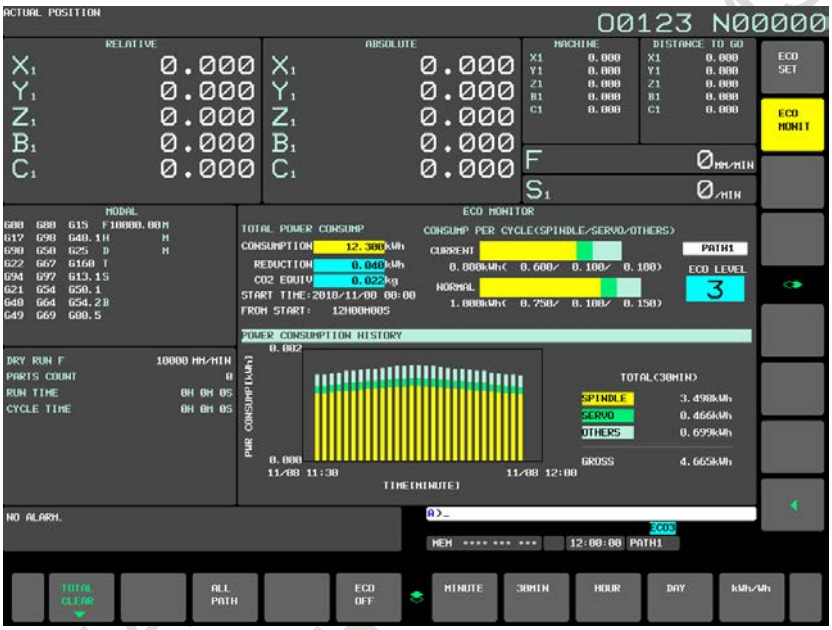


Fig. 12.3.36 (a) Eco monitoring screen

Explanation

- **Display of total power consumption**

Total power consumption of a machine is displayed, which has been accumulated since the last clear operation of total power consumption. This is not cleared even if power is off.

Items	Contents
Consumption	Accumulated power consumption of a whole machine is displayed.
Reduction	Difference between power consumption of normal machining and that of eco-machining is accumulated. The accumulated data of all paths are added. See “Example” below.
CO2 equivalent	Carbon-dioxide emission equivalent to “Reduction” is displayed.

NOTE
The conversion factor for CO2 equivalent is specified by the parameter No.11399.

FRYER MACHINE ROBOTICS INC

- Power consumption per cycle (spindle/servo/others)

Power consumption per cycle at the current Eco level is displayed in bar graphs, compared with that of normal machining (Eco level 0).

Items	Contents
Current	The power consumption of a running cycle is displayed and updated in real time. When nothing is operated, the power consumption of the last cycle is displayed. With multiple paths, summed data of all paths as well as data of the current path can be displayed. Note that the summed data can include only data of paths where both “Normal” and “Current” have non-zero values. Soft keys [EACH PATH] and [ALL PATH] switch the displayed data.
Normal	The power consumption of the last cycle at Eco level 0 is displayed. With multiple paths, summed data of all paths as well as data of the current path can be displayed. Note that the summed data can include only data of paths where both “Normal” and “Current” have non-zero values. Soft keys [EACH PATH] and [ALL PATH] switch the displayed data.
Eco level	The selected Eco level is displayed. When eco-machining is ineffective, “OFF” is displayed. The Eco level of the current path is displayed.

NOTE

When select of an Eco level is available for each path (bit 1 (EEP) of the parameter No.24303 is set to 1), “Eco level” is not displayed during displaying summed data of all paths.

- History of power consumption

Time transition of power consumption is displayed in bar graphs, with power consumption categorized into that in spindles, servos and others.

Items	Contents
History of power consumption	Unit selected on operation menu decides displayed values as follows

Units	Displayed values()
MINUTE	Power consumption for a minute is displayed in a bar graph. Up to 30 minutes’ power consumption can be displayed.
30MIN	Power consumption for 30 minutes is displayed in a bar graph. Up to 15 hours’ power consumption can be displayed.
HOUR	Power consumption for an hour is displayed in a bar graph. Up to 30 hours’ power consumption can be displayed.
DAY	Power consumption for a day is displayed in a bar graph. Up to 30 days’ power consumption can be displayed.

Items	Contents
TOTAL	Total power consumption in each unit is displayed as follows.

Units	Contents
MINUTE	Power consumption for last 30 minutes is displayed.
30MIN	Power consumption for last 15 hours is displayed.
HOUR	Power consumption for last 30 hours is displayed.
DAY	Power consumption for last 30 days is displayed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

1 Power consumption data obtained by actual machining are displayed. Therefore, 0 is displayed at the first machining or soon after deleting data. Though these data depend on parameter settings, machining circumstances and so on, soon after changing settings, data obtained before the change are displayed.

2 Difference between power consumption of normal machining and that of the current Eco level is added to “Reduction” in total power consumption at every cycle end. When no normal cycle finishes after changing parameter settings, machining programs and so on, notice that reduction calculated from power consumption data of old settings is added to “Reduction”. In addition, with no data of normal machining, nothing is added to “Reduction”. See also “Example” below.

Example

The values of “Consumption” and “Reduction” are calculated according to the examples below:

1 The case that power consumption of Eco level 0 is equal to 10kWh:

A Given that 8kWh has been consumed at Eco level 2 after clearing total power consumptions.

- “Consumption” = 0 + 8 = 8kWh
- “Reduction” = 0 + (10 - 8) = 2kWh

B Given that, following A. above, 7kWh has been consumed at Eco level 3.

- “Consumption” = 8 + 7 = 15kWh
- “Reduction” = 2 + (10 - 7) = 5kWh

2 The case that a cycle with Eco level 0 has not finished since main program was changed.

A Given that, following 1.B. above, operating new program has consumed 12kWh at Eco level 2 (Since main program has been changed, power consumption can be larger than that at Eco level 0.).

- “Consumption” = 15 + 12 = 27kWh
- “Reduction” = 5 + (10 - 12) = 3kWh

(This is calculated by using data of Eco level 0 obtained before main program was changed.)

3 The case that there is no data of Eco level 0

A Given that, following 2.A above, data of Eco level 0 has been deleted on Eco setting screen and then, 7kWh has been consumed at Eco level 3.

- “Consumption” = 27 + 7 = 34kWh
- “Reduction” = 3kWh (Nothing is added without data of Eco level 0.)

With multiple paths, the summation of all paths can be displayed in “Consumption” and “Reduction”, these values of each path calculated like examples above.
For example, when consumption and reduction of path 1 are equal to 10kWh and 2kWh, and those of path 2 are equal to 15kWh and 3kWh, displayed values are as follows:

- “Consumption” = 10 + 15 = 25kWh
- “Reduction” = 2 + 3 = 5kWh.

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

Operation on Eco setting screen

Procedure

- Switching displayed time unit of history of power consumption

Displayed time unit of history of power consumption can be switched into every minute, every 30 minutes, every hour and every day.

Press one of horizontal soft keys [MINUTE], [30MIN], [HOUR] and [DAY].
Displayed time unit changes.

- Turning on/off of eco-machining

Eco-machining and normal machining can be switched by soft key [ECO OFF]/[ECO ON].
Given that Eco level 3 is selected, for example,

- Normal machining is selected by pressing soft key [ECO OFF]. Here, the selected Eco level does not change (Eco level 3 is kept).
- After the operation above, eco-machining of Eco level 3 is selected by pressing soft key [ECO ON].

1 Select MDI mode for all paths.
2 Release spindle synchronization mode for all paths.
3 Press horizontal soft key [ECO OFF]/[ECO ON].
In case of turning off eco-machining, Eco indicator “ECOX” in Warning area is turned off. Note that the current Eco level displayed on Eco setting screen does not change because the selected Eco level does not change in this operation.
In case of turning on eco-machining, Eco indicator “ECOX” in Warning area is turned on. Note that nothing is displayed when Eco level 0 is selected.

NOTE
When select of an Eco level is available for each path (bit 1 (EEP) of the parameter No.24303 is set to 1), operations 1 and 2 are needed only for the current path.

- Switching displayed unit of power consumption

Displayed unit of power consumption can be switched to “kWh” or “Wh”.
Soft key [kWh/Wh] switches the unit to “kWh” or “Wh”.

- Clearing total power consumption

Displayed total power consumption can be set to 0 and new accumulation is started.

1 Press horizontal soft key [TOTAL CLEAR].
Select menu [EXEC]/[CANCEL] is displayed.
2 Press soft key [EXEC].
Displayed total power consumptions are set to 0 and “start time” is set to current time

- Switching displayed data in “Power consumption per cycle”

With multiple paths, displayed data in “Power consumption per cycle” can be switched into data of each path or summed data of all paths.
Soft key [EACH PATH]/[ALL PATH] can switch the data.

Press horizontal soft key [EACH PATH]/[ALL PATH].
Data in “Power consumption per cycle” is changed.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Restriction

Since this function uses clock, change in clock causes following results.
Note that change in clock requires power off.

- **Total power consumption**

"From start" changes according to change in clock, though "Consumption", "Reduction", "CO2 equivalent" and "Start time" does not change.

- **History of power consumption**

If clock changes backward, history of power consumption is entirely set to 0.
If clock changes forward, obtained power consumption and time do not change, though power consumption of jumped period is set to 0. If clock is advanced by an hour, for example, power consumption of the jumped hour is considered as 0.

12.4 SCREENS DISPLAYED BY FUNCTION KEY SYSTEM

When the CNC and machine are connected, parameters must be set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor or other parts.
This chapter describes how to set parameters on the MDI unit. Parameters can also be set with external input/output devices such as the memory card (see Chapter, "DATA INPUT/OUTPUT").
In addition, pitch error compensation data used for improving the precision in positioning with the ball screw on the machine can be set or displayed by the operations under function key SYSTEM.
See Chapter, "ALARM AND SELF-DIAGNOSIS FUNCTIONS" for the diagnosis screens displayed by pressing function key SYSTEM.

Screens of an 8.4/10.4-inch display unit

12.4.1 Displaying and Setting Parameters

When the CNC and machine are connected, parameters are set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor. The setting of parameters depends on the machine. Refer to the parameter list prepared by the machine tool builder.
Normally, the user need not change parameter setting.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Procedure for displaying and setting parameters

Procedure

1 Set 1 for PARAMETER WRITE to enable writing. See the procedure for enabling/disabling parameter writing described below.

2 Press function key [SYSTEM].

3 Press chapter selection soft key [PARAM] to display the parameter screen.

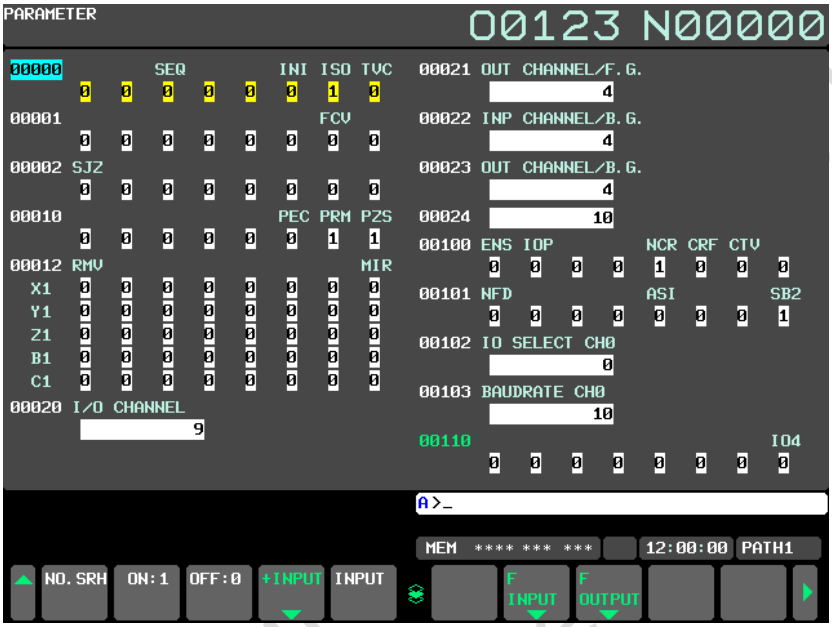


Fig. 12.4.1 (a) PARAMETER screen (10.4-inch display unit)

4 Move the cursor to the parameter number to be set or displayed in either of the following ways:

- Enter the parameter number and press soft key [NO.SRH] .
- Move the cursor to the parameter number using the page keys, [PAGE] and [PAGE], and cursor keys, [↑], [↓], [←], and [→].

5 To set the parameter, enter a new value with numeric keys and press soft key [INPUT]. The parameter is set to the entered value and the value is displayed.

6 Set 0 for PARAMETER WRITE to disable writing.

FRYER MACHINE INC
Rob Zahn

Procedure for enabling/displaying parameter writing

Procedure

1 Select the MDI mode or enter state emergency stop.

2 Press function key .

3 Press soft key [SETTING] to display the setting screen.

Fig. 12.4.1 (b) SETTING screen (10.4-inch display unit)

4 Move the cursor to PARAMETER WRITE using cursor keys.

5 Press soft key [(OPRT)], then press [ON:1] to enable parameter writing.
At this time, the CNC enters the alarm state SW0100.

6 After setting parameters, return to the setting screen. Move the cursor to PARAMETER WRITE and press soft key [(OPRT)] , then press [OFF:0].

7 Press the RESET key to release the alarm condition.

When an alarm PW0000 occurred, however, it is not released unless the power is turned off and back on.

Explanation

- Setting parameters with external input/output devices

See Chapter, “DATA INPUT/OUTPUT” for setting parameters with external input/output devices such as the memory card.

- Parameters that require turning off the power

Some parameters are not effective until the power is turned off and on again after they are set. Setting such parameters causes alarm PW0000. In this case, turn off the power, then turn it on again.

- Parameter list

Refer to the Parameter Manual (B-64700EN) for the parameter list.

- Setting data

Some parameters can be set on the setting screen if the parameter list indicates "Setting entry is acceptable". Setting 1 for PARAMETER WRITE is not necessary when these parameters are set on the setting screen.

12.4.2 Servo Parameters

This subsection describes the initialization of digital servo parameters performed, for example, at the time of field tuning of the machine tool.

Procedure for servo parameter setting

Procedure

1 Turn on the power in the emergency stop state.
2 Set bit 0 (SVS) of parameter No. 3111 to 1 to display servo setting and tuning screens.
3 Press function key [SYSTEM], continuous menu key [▷], then soft key [SERVO PARAM].
4 Press soft key [SERVO PARAM] to select the servo parameter screen.
The following screen (Fig. 12.4.2) appears:

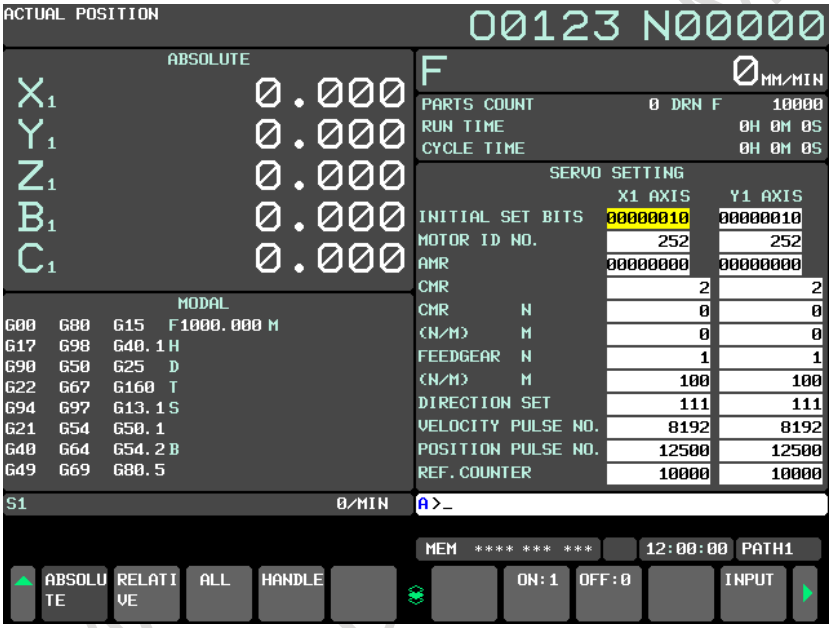


Fig. 12.4.2 (a) Servo parameter setting screen (10.4-inch display unit)

5 With the page keys and cursor keys, move the cursor to the position of data to be set or modified.
6 Key in a desired value then press soft key [INPUT].

FRYER MACHINE TOOLS INC

12.4.3 Servo Tuning

Data related to servo tuning is displayed and set.

Procedure for servo tuning

Procedure

1 Turn on the power in the emergency stop state.
2 Set bit 0 (SVS) of parameter No. 3111 to 1 to display servo setting and tuning screens.
3 Press function key [SYSTEM], continuous menu key [▷], then soft key [SERVO PARAM].
4 Press soft key [SERVO TUNE] to select the servo tuning screen.
5 Use the cursor and page keys to enter necessary data for initial setting. The following screen (Fig. 12.4.3) appears:

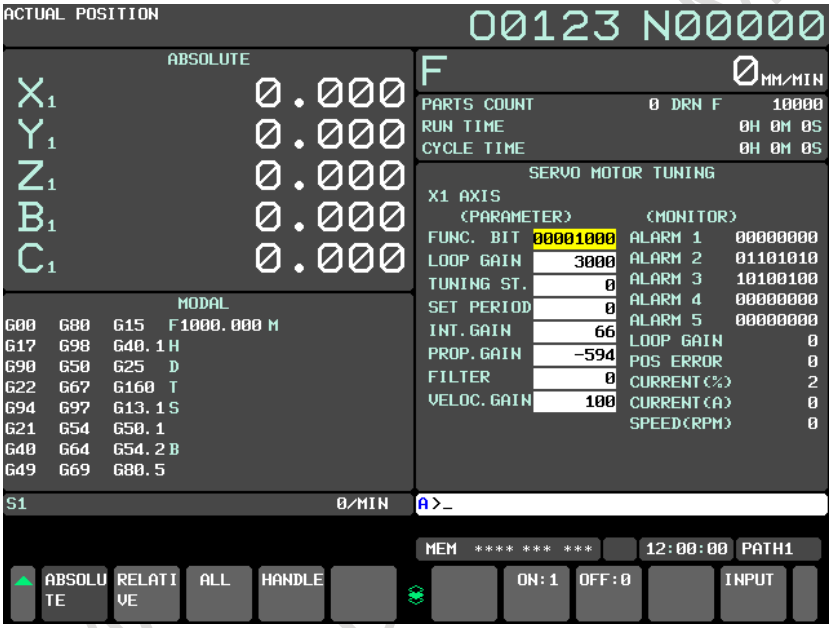


Fig. 12.4.3 (a) Servo tuning screen (10.4-inch display unit)

6 With the page keys and cursor keys, move the cursor to the position of data to be set or modified.
7 Key in a desired value then press soft key [INPUT].

FRYER MFG. PRODUCTS INC

12.4.4 Spindle Setting

Parameters related to spindles are set and displayed. In addition to the parameters, related data can be displayed. Screens for spindle setting, spindle tuning, and spindle monitoring are provided.

Setting spindle parameters

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display spindle setting and tuning screens.

2 Press function key [SYSTEM], continuous menu key [▷], then soft key [SPINDL SETING].

3 Press soft key [SPINDL SETING] to select the spindle setting screen.

4 The following screen (Fig. 12.4.4) appears:

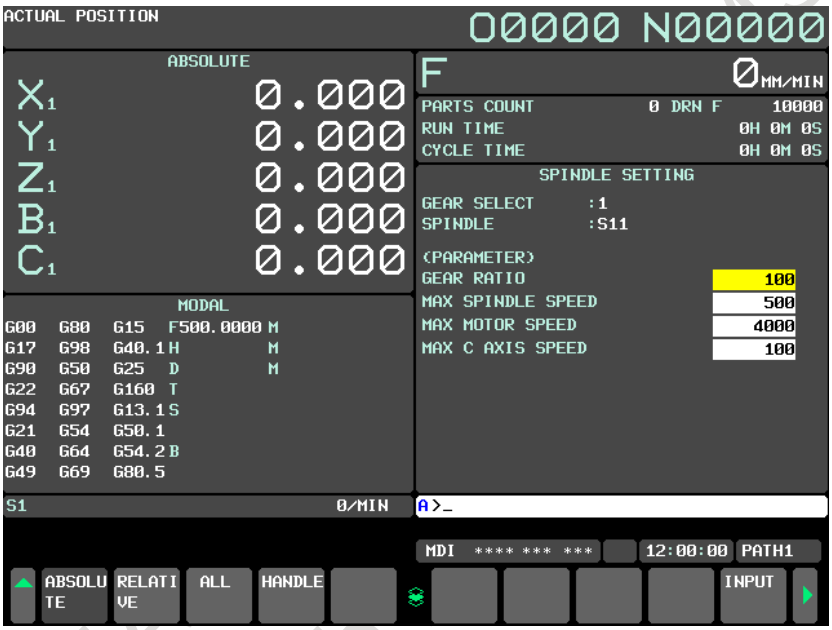


Fig. 12.4.4 (a) Spindle setting screen (10.4-inch display unit)

4 With the page keys and cursor keys, move the cursor to the position of data to be set or modified.

5 Key in a desired value then press soft key [INPUT].

FRYER MACHINE TOOLS INC

12.4.5 Spindle Tuning

Spindle tuning data is displayed and set.

Setting for spindle tuning

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display spindle setting and tuning screens.

2 Press function key [SYSTEM], continuous menu key [▷], then soft key [SPINDL SETING].

3 Press soft key [SPINDL TUNE] to select the spindle tune screen.

4 The following screen (Fig. 12.4.5 (a)) appears:

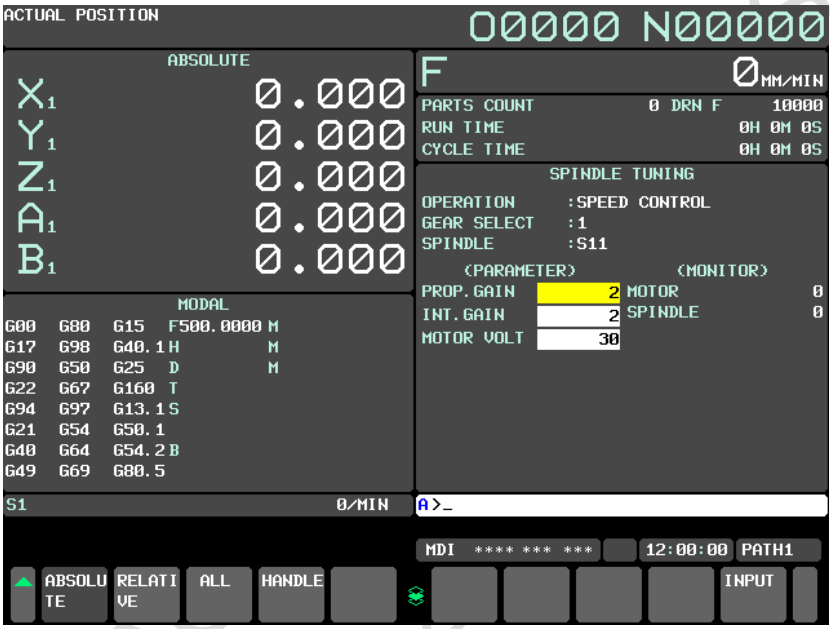


Fig. 12.4.5 (a) Spindle tuning screen (10.4-inch display unit)

5 With the page keys and cursor keys, move the cursor to the position of data to be set or modified.

6 Key in a desired value then press soft key [INPUT].

FRYER MA... Rob ... S INC

12.4.6 Spindle Monitor

Spindle-related data is displayed.

Displaying the spindle monitor

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display spindle setting and tuning screens.

2 Press function key [SYSTEM], continuous menu key [▷], then soft key [SPINDL SETING].

3 Press soft key [SP.MON] to select the spindle monitor screen.

4 The following screen (Fig. 12.4.6) appears:

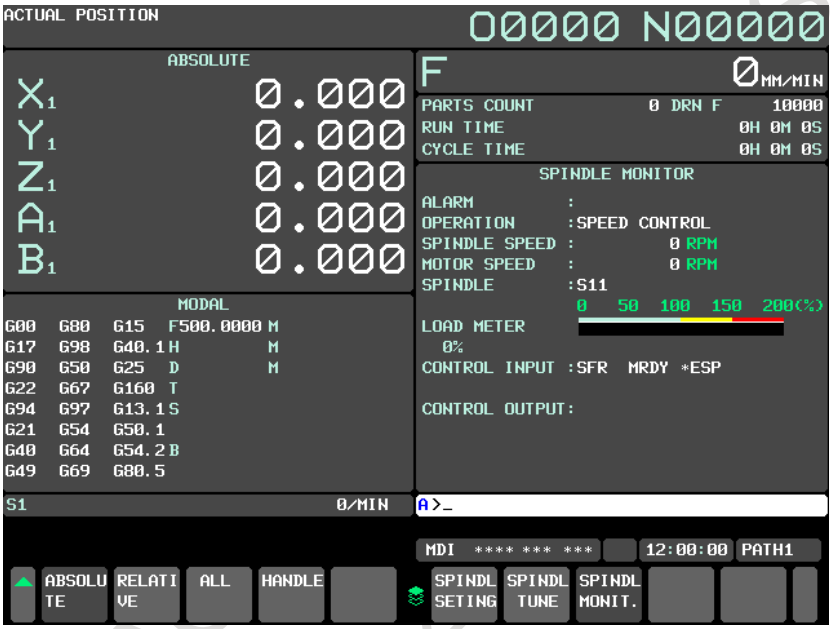


Fig. 12.4.6 (a) Spindle monitor screen (10.4-inch display unit)

FRYER MACHINES INC
Rob

12.4.7 Color Setting Screen

Screen colors can be set on the color setting screen.

Displaying the color setting screen

Procedure

1 Press function key SYSTEM.

2 Press the continuous menu key ▷ several times to display soft key [COLOR].

3 Press soft key [COLOR] to display the color setting screen.

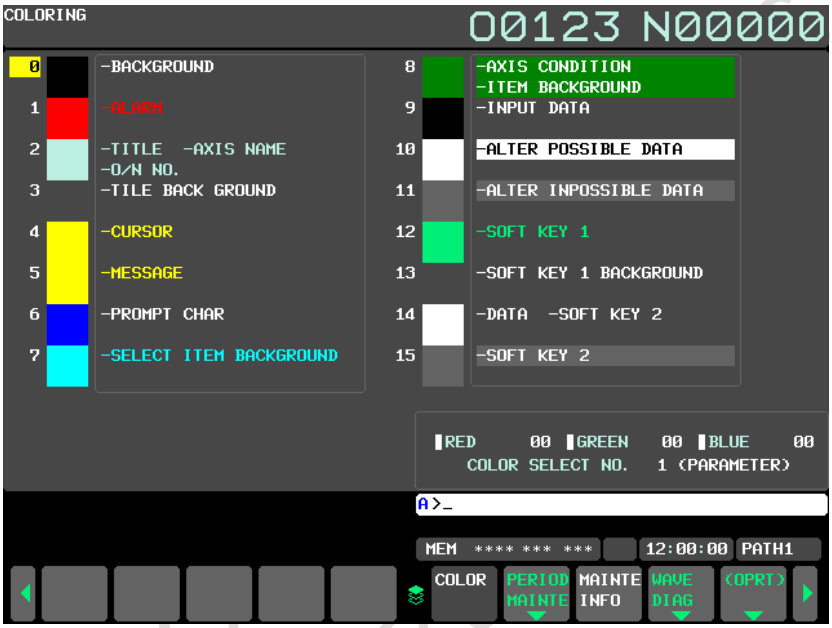


Fig. 12.4.7 (a) Color setting screen (10.4-inch display unit)

Procedure for operating the color setting screen

- Modifying the color (color palette values)

1 Press soft key [(OPRT)]. The soft key display changes to the following operation soft keys:

2 Move the cursor to a color number whose color palette values are to be modified.
The current color palette value set for each primary color is displayed.

3 Select a primary color whose setting is to be modified, with the corresponding operation soft key [RED], [GREEN], or [BLUE].
More than one primary color can be selected at the same time.
Each time operation soft key [RED], [GREEN], or [BLUE] is pressed, the operation soft key toggles between selection and deselection.
(When operation soft keys [RED], [GREEN], and [BLUE] are not displayed, press the rightmost soft key to display the operation soft keys.)

4 Select operation soft key [BRIGHT] or [DARK] to modify the brightness of the selected prime color(s).

FRYER MAC... INC

- Storing the color (color palette values)

The set color palette values can be stored.

1 Press operation soft key [COLOR1], [COLOR2], or [COLOR3] to select a storage area.
(When operation soft keys [COLOR1], [COLOR2], and [COLOR3] are not displayed, press the rightmost soft key to display the operation soft keys.)

COLOR1 Standard color data parameters Nos. 6581 to 6595
COLOR2 Parameters Nos. 10421 to 10435
COLOR3 Parameters Nos. 10461 to 10475

2 Press operation soft key [MEMORY]. The operation soft key display switches to the following:

3 Pressing operation soft key [EXEC] stores the current color palette settings in the selected area.
Pressing operation soft key [CAN] or the leftmost key does not store the current color palette settings in the selected area.

- Calling the color (color palette values)

1 Press operation soft key [COLOR1], [COLOR2], or [COLOR3] to select a storage area where color palette values are stored.
(When operation soft keys [COLOR1], [COLOR2], and [COLOR3] are not displayed, press the rightmost soft key to display the operation soft keys.)

2 Press operation soft key [RECALL]. The operation soft key display switches to the following:

3 Pressing operation soft key [EXEC] calls the color palette values from the selected area to enable the color to be modified. This operation is invalid if no color palette values are stored.
Pressing operation soft key [CAN] or the leftmost key does not call the color palette values from the selected area.

NOTE

1 Immediately after the power is turned on, the settings of COLOR1 (parameters) are used for display. If nothing is stored for COLOR1, the default colors are used for display.
2 Do not modify the color setting data parameters directly by MDI key input. When modifying the standard color data, be sure to perform a storage operation on the color setting screen.

FRYER MACHINE SYSTEMS INC
Rob Zaleski

12.4.8 Machining Parameter Tuning

12.4.8.1 Machining parameter tuning (AI contour)

In AI contour control, by setting a velocity-emphasized parameter set and precision-emphasized parameter set of machining condition selection function and setting the precision level matching a machining condition such as rough machining or finish machining on the precision level setting screen or by programming, the parameters suitable for the condition can be automatically calculated to perform machining.
On this screen, the parameter sets for emphasis on velocity (precision level 1) and emphasis on precision (precision level 10) can be set.
Set the following parameters:

- Acceleration rate of look-ahead acceleration/deceleration before interpolation
- Acceleration change time (bell-shaped)
- Allowable acceleration rate
- Acceleration rate of acceleration/deceleration after interpolation
- Corner speed difference
- Maximum feedrate
- Items that can be set freely (2 items)

M

- Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control
- Allowable acceleration change value for each axis in acceleration change under jerk control in successive linear interpolation operations
- Ratio of the change time of the rate of change of acceleration in look-ahead smooth bell-shaped acceleration/deceleration before interpolation

For details of each parameter, see the descriptions of AI contour control and jerk control (M system).
By setting bit 0 (MPR) of parameter No. 13601 to 1, this screen can be hidden.
For the method of setting a precision level, see the description of the precision level selection screen in Subsection 12.3.12, “Precision Level Selection”.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Procedure for machining parameter tuning

Procedure

1 Set the MDI mode.

2 Press function key [SYSTEM].

3 Press soft key [MCHN TUNING] to display the machining parameter tuning screen.

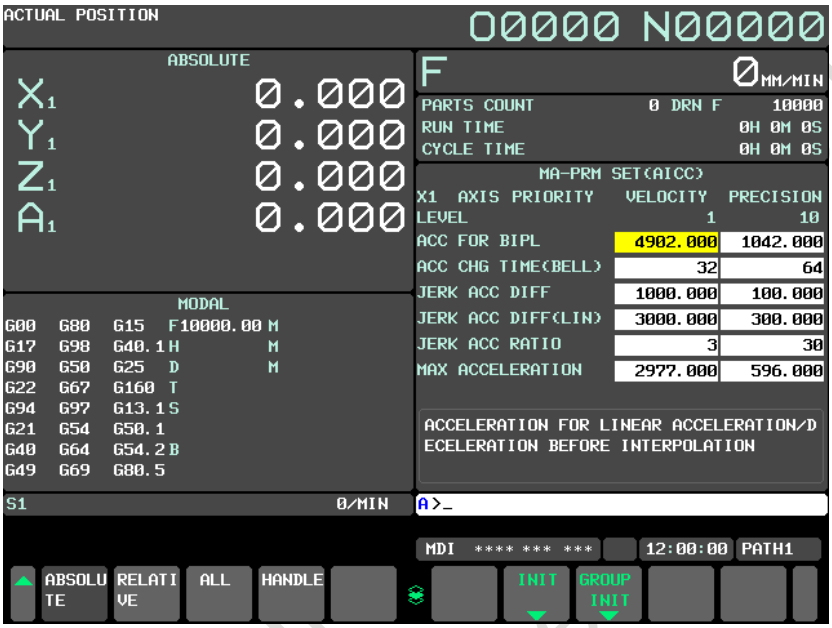


Fig. 12.4.8.1 (a) Machining parameter tuning screen (AI contour) (10.4-inch display unit)

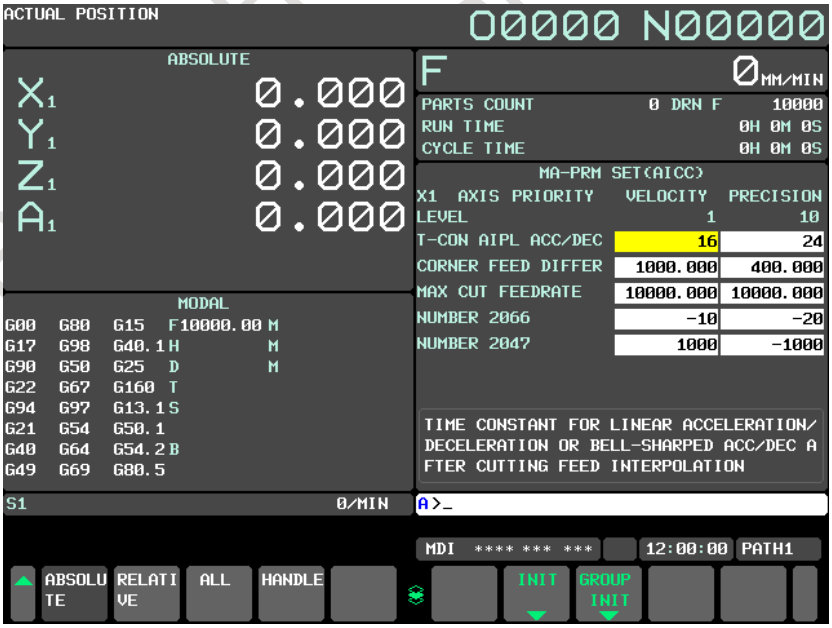


Fig. 12.4.8.1 (b) Machining parameter tuning screen (AI contour) (10.4-inch display unit)

4 Move the cursor to the position of a parameter to be set, as follows:
Press page key [PAGE ↑] or [PAGE ↓], and cursor keys [↑], [↓], [←] and /or [→] to move the cursor to the parameter.

5 Key in desired data then press the [INPUT] key on the MDI unit.

6 When data is input, a RMS value is found according to the precision level parameters. (The precision level can be changed on the precision level selection screen or parameter setting screen.) If a RMS value calculation fails, a warning (indicating that automatic setting failed) is displayed.
7 Repeat steps 4 and 5 until all machining parameters are set.
8 In addition to the setting method described above, a parameter setting method using soft keys is available. Pressing soft key [INIT] displays the standard value (recommended by FANUC) of the item selected by the cursor in the key input buffer. Pressing soft key [EXEC] initializes the item to the standard value. Pressing soft key [GROUP INIT] initializes all items of a group (emphasis on velocity or emphasis on precision) selected by the cursor to the standard values.

The Table 12.4.8.1 (a) indicates the initial settings.

Table 12.4.8.1 (a) Initial settings

Setting item	AI contour control		Unit
	Emphasis on velocity (LV1)	Emphasis on precision (LV10)	
Acceleration rate of look-ahead acceleration/deceleration before interpolation <ACC FOR BIPL>	4902.000	1042.000	mm/sec^2
Acceleration change time (bell-shaped) <ACC CHG TIME(BELL)>	32	64	msec
Allowable acceleration change value <JERK ACC DIFF>	0	0	mm/sec^2
Allowable acceleration change value in successive linear interpolation operations <JERK ACC DIFF(LIN)>	0	0	mm/sec^2
Ratio of the change time of the jerk control <JERK ACC RATIO>	0	0	%
Allowable acceleration rate <MAX AACELERATION>	2977.000	596.000	mm/sec^2
Time constant for acceleration/deceleration after interpolation <T-CON AIPL ACC/DEC>	24	24	msec
Corner speed difference <CORNER FEED DIFFER>	1000	400	mm/min
Maximum cutting speed <MAX CUT FEEDRATE>	10000	10000	mm/min

Explanation

- Look-ahead acceleration/deceleration before interpolation

Set an acceleration rate for a linear portion in look-ahead acceleration/deceleration before interpolation.
Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13610 (velocity-emphasized parameter)
Parameter No. 13611 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1660: Maximum allowable acceleration rate for each axis in look-ahead acceleration/deceleration before interpolation

- Acceleration change time (bell-shaped)

Set a time constant for a bell-shaped portion in acceleration/ deceleration before look-ahead interpolation.
Unit of data: ms

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13612 (velocity-emphasized parameter)
Parameter No. 13613 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1772: Time constant for bell-shaped look-ahead acceleration/deceleration before interpolation of constant acceleration time type

FRYER MACHINE SYSTEMS INC
Rob Zahensky

> ⚠ **CAUTION**
> A set time constant is applied to all axes. So, a modification made to this item changes the settings for all axes.

M

- Allowable acceleration change value in velocity control based on acceleration change under jerk control

Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

Set an allowable acceleration change value per ms for each axis in velocity control based on acceleration change under jerk control.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13614 (velocity-emphasized parameter)
Parameter No. 13615 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1788: Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control

> **NOTE**
> This setting item is displayed only when the jerk control function is enabled.

M

- Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations

Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

Set an allowable acceleration change value per ms for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13616 (velocity-emphasized parameter)
Parameter No. 13617 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1789: Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations

> ⚠ **CAUTION**
> 1 For an axis with 0 set in this parameter, the parameters (allowable acceleration change value in velocity control based on acceleration change under jerk control: No. 13614, No.13615) are valid.
> 2 For an axis with 0 set in the parameter (allowable acceleration change value in velocity control based on acceleration change under jerk control: No. 13614, No.13615), velocity control based on acceleration change is disabled, so that this parameter has no effect.

> **NOTE**
> This setting item is displayed only when the jerk control function is enabled.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

M

- Ratio of the change time of the jerk control in look-ahead smooth bell-shaped acceleration/deceleration before interpolation

Unit of data: %

Set the ratio (in %) of the change time of jerk control to the change time of acceleration in look-ahead smooth bell-shaped acceleration/deceleration before interpolation.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13618 (velocity-emphasized parameter)
Parameter No. 13619 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1790: Ratio of the change time of the jerk control in look-ahead smooth bell-shaped acceleration/deceleration before interpolation

NOTE
This setting item is displayed only when the jerk control function is enabled.

- Allowable acceleration rate

Set an allowable acceleration rate in acceleration-based speed determination.
Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13620 (velocity-emphasized parameter)
Parameter No. 13621 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1735: Allowable acceleration rate for each axis applicable to the deceleration function based on acceleration in circular interpolation
Parameter No. 1737: Allowable acceleration rate for each axis applicable to the deceleration function based on acceleration in AI contour control

⚠ CAUTION
When bit 0 (MCR) of parameter No. 13600 is set to 1, the deceleration function based on acceleration in circular interpolation is not set.

- Time constant for acceleration/deceleration after interpolation

Set a time constant for acceleration/deceleration after interpolation.
Unit of data: ms

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13622 (velocity-emphasized parameter)
Parameter No. 13623 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1769: Time constant for acceleration/deceleration after cutting feed interpolation

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Corner speed difference

Set an allowable corner speed difference used for speed determination.
Unit of data: mm/sec, inch/sec, deg/sec (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13624 (velocity-emphasized parameter)
Parameter No. 13625 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1783: Allowable speed difference for each axis in automatic corner deceleration based on speed difference

- Maximum cutting speed

Set a maximum cutting speed for each axis.
Unit of data: mm/sec, inch/sec, deg/sec (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13626 (velocity-emphasized parameter)
Parameter No. 13627 (precision-emphasized parameter)

Moreover, the following parameter is also set from the precision level:
Parameter No. 1432: Maximum cutting feedrate for each axis in the AI contour control mode

- Arbitrary items

Two arbitrary parameters can be registered. Each item can correspond to a CNC parameter or servo parameter. A parameter number corresponding to each item is to be specified with parameters.

As indicated below (Table 12.4.8.1 (b)), set the parameters for corresponding parameter numbers, velocity-emphasized parameters (precision level 1), and precision-emphasized parameters (precision level 10).

Table 12.4.8.1 (b) Parameters related to arbitrary items

	Corresponding parameter number	Setting of velocity-emphasized (precision level 1) value	Setting of precision-emphasized (precision level 10) value
Arbitrary item 1	No.13628	No.13630	No.13632
Arbitrary item 2	No.13629	No.13631	No.13633

- Display
  Tuning target parameter numbers are displayed.

> ⚠ **CAUTION**
> As arbitrary items, the numbers of the following parameters cannot be specified:
> - Bit parameter
> - Spindle parameters (Parameters Nos. 4000 to 4799)
> - Real-type parameter
> - Power-off parameter
> - Nonexistent parameter

FRYER MACHINE SYSTEMS INC
Rob Zalensky

12.4.9 Displaying Memory Data

The contents of the CNC memory can be displayed starting at a specified address.

Displaying memory data

Procedure

1 Set bit 0 (MEM) of parameter No. 8950 to 1 to display the memory contents display screen.

2 Press function key [SYSTEM].

3 Press the continuous menu key [▷], then press chapter selection soft key [MEMORY].
The following screen (Fig. 12.4.9) appears:

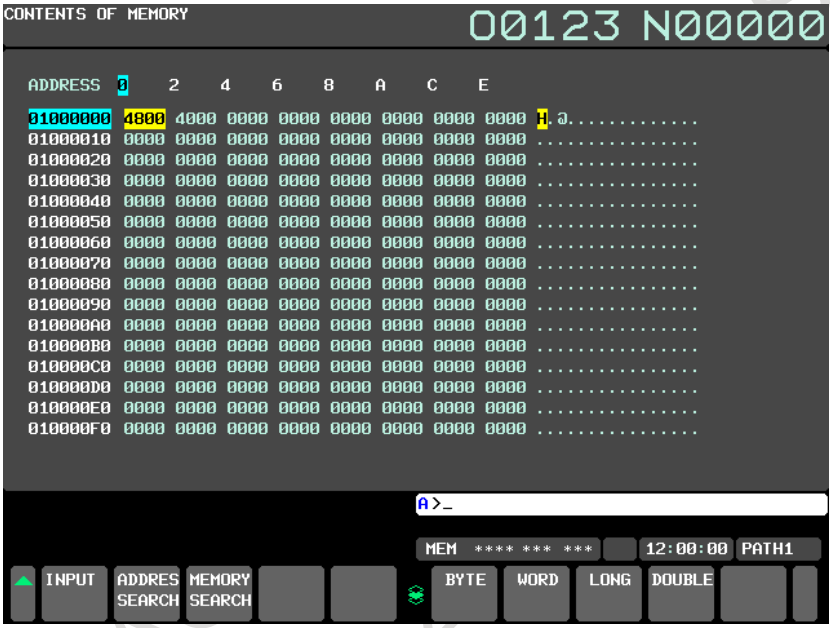


Fig. 12.4.9 (a) Memory contents display screen (10.4-inch display unit)

4 Key in a desired address (hexadecimal) then press soft key [ADDRES SEARCH]. Starting at the specified address, 256-byte data is displayed.
(Example: When you input 100000 then press [ADDRES SEARCH], data is displayed starting at 100000H.)

5 Display data can be switched by using page keys [PAGE ↑] and [PAGE ↓] and cursor keys [↑], [↓], [←], and/or [→].

6 By pressing soft key [BYTE], [WORD], [LONG], or [DOUBLE], select a data type to be displayed.

Explanation

A memory data display format can be selected from the following four options:

Byte display (1 byte in hexadecimal)
Word display (2 bytes in hexadecimal)
Long display (4 bytes in hexadecimal)
Double display (8 bytes in decimal: Double precision floating-point display)

One screen displays 256-byte memory data.

FRYER MACHINES INC Rob

NOTE

1 When an address is input, "H" for representing "hexadecimal" need not be specified at the end of the address. If H is added, a warning is displayed to indicate that the format is invalid.

2 When word display is selected as the display format, an input address is rounded to a multiple of 2 bytes. When long display or double display is selected, an input address is rounded to a multiple of 4 bytes.

⚠ **WARNING**

1 If a memory address that must not be accessed in address search is input, a system alarm is issued.
When making an address search, check that the address is accessible and that the address is input correctly.

2 This function is designed for maintenance, and must not be used by general users.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.4.10 Parameter Tuning Screen

The parameter tuning screen is a screen for parameter setting and tuning designed to achieve the following:

1 The minimum required parameters that must be set when the machine is started up are collectively displayed to facilitate start-up of the machine.
2 The servo tuning screen, spindle tuning screen, and machining parameter tuning screen are displayed for smooth tuning.

The parameter tuning screen consists of a menu screen and several setting screens.

12.4.10.1 Displaying the menu screen and selecting a menu item

The parameter tuning menu screen displays the following items:

[START UP]
- SYSTEM SETTING
- AXIS SETTING
- FSSB (SV AMP)
- FSSB (SP AMP)
- FSSB (AXIS)
- SERVO SETTING
- SPINDLE SETTING
- MISCELLANY

[TUNING]
- SERVO TUNING
- SPINDLE TUNING
- AICC TUNING

On the parameter tuning menu screen, one of the displayed items can be selected to display the corresponding screen. From each setting screen, you can return to the menu screen by performing a soft key operation.

NOTE
1 Some items may not be displayed, depending on the system configuration.
2 When bit 0 (SVS) of parameter No. 3111 is set to 0, "SERVO SETTING" and "SERVO TUNING" are not displayed. When bit 1 (SPS) of parameter No. 3111 is set to 0, "SPINDLE TUNING" is not displayed.

Displaying the menu screen and selecting a setting screen

Procedure

1 Set the MDI mode.
2 Switch the setting of "PARAMETER WRITE" to "ENABLED". For details, see the procedure for "PARAMETER WRITE" in Subsection, “Displaying and Setting Parameters”.
3 Press function key [SYSTEM].
4 Press the continuous menu key [▷] several times.
5 Press soft key [PARAM TUNING] to display the menu screen for parameter tuning .

FRYER MACHINE SYSTEMS INC
Rob Zahensky

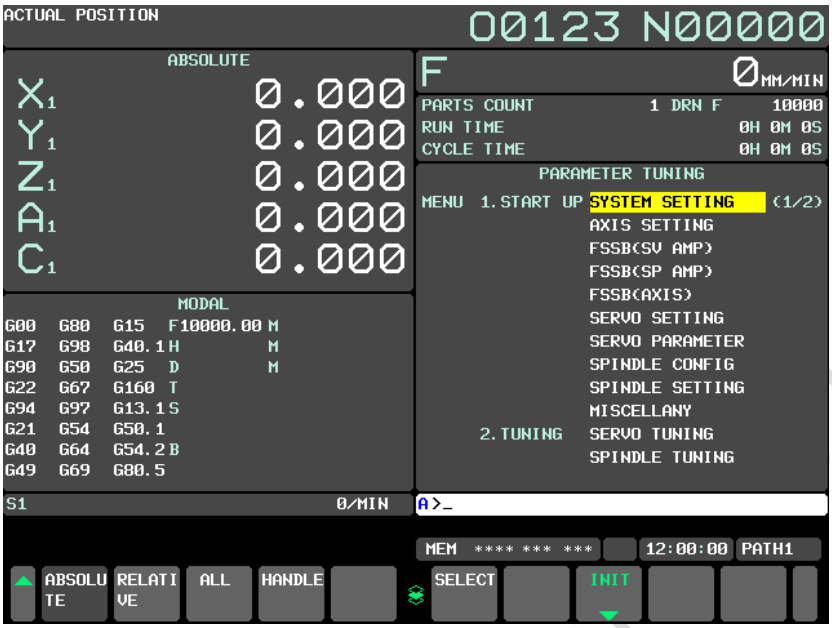


Fig. 12.4.10.1 (a) Menu screen for parameter tuning (10.4-inch display unit)

6 Move the cursor to a desired item by pressing cursor key ↑ or ↓.

7 Press soft key [SELECT]. The screen display switches to the selected screen.

Returning to the menu screen

Procedure

1 Press soft key [SELECT] on the parameter tuning menu screen described in Subsection, “Displaying the menu screen and selecting a menu item”. The screen and soft keys shown below (Fig. 12.4.10.1 (b) are displayed. (The screen below (Fig. 12.4.10.1 (b) is displayed when "AXIS SETTING" is selected.)

Fig. 12.4.10.1 (b) Axis setting screen (10.4-inch display unit)

FRYER MACHINE SYSTEMS INC Zahensky

2 Press the continuous menu key ▷ several times.
3 Press soft key [MENU].
The screen display returns to the parameter tuning menu screen.
4 Upon completion of parameter setting, switch the setting of "PARAMETER WRITE" to "DISABLED".

NOTE
Some setting screens can also be displayed by a chapter selection soft key. If a screen is selected using a chapter selection soft key, however, you cannot return to the parameter tuning menu screen.

Explanation

- Items displayed with [START UP]

The items of [START UP] indicate the screens for setting the minimum required parameters for starting up the machine.

Table 12.4.10.1 (a) Items displayed with [START UP]

Display item	Description
SYSTEM SETTING	Screen for setting CNC parameters related to the entire system configuration
AXIS SETTING	Screen for setting CNC parameters related to axes, coordinates, feedrate, and acceleration/deceleration
FSSB (SV AMP)	FSSB servo amplifier setting screen
FSSB (SP AMP)	FSSB spindle amplifier setting screen
FSSB (AXIS)	FSSB axis setting screen
SERVO SETTING	Servo setting screen
SPINDLE SETTING	Screen for setting spindle-related parameters
MISCELLANY	Screen for setting parameters related to the allowable number of M code digits and whether to display the servo setting and spindle tuning screens

NOTE
Some items may not be displayed, depending on the system configuration.

- Items displayed with [TUNING]

The items of [TUNING] indicate the screens for servo, spindle, and high-speed high-precision machining tuning.

Table. 12.4.10.1 (b) Items displayed with [TUNING]

Display item	Description
SERVO TUNING	Servo tuning screen
SPINDLE TUNING	Spindle tuning screen
AICC TUNING	Machining parameter tuning screen

NOTE
1 Some items may not be displayed, depending on the system configuration.
2 When bit 0 (SVS) of parameter No. 3111 is set to 0, "SERVO SETTING" and "SERVO TUNING" are not displayed. When bit 1 (SPS) of parameter No. 3111 is set to 0, "SPINDLE TUNING" is not displayed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.4.10.2 Parameter tuning screen (system setting)

This screen enables the parameters related to the entire system configuration to be displayed and modified. The parameters can be initialized to the standard values (recommended by FANUC).

Display and setting

Procedure

1 Move the cursor to [SYSTEM SETTING] by pressing cursor key ⬆ or ⬇ on the parameter tuning menu screen described in Subsection, “Displaying the menu screen and selecting a menu item”.

2 Press soft key [SELECT]. The screen display switches to the screen and soft keys shown below (Fig. 12.4.10.2).

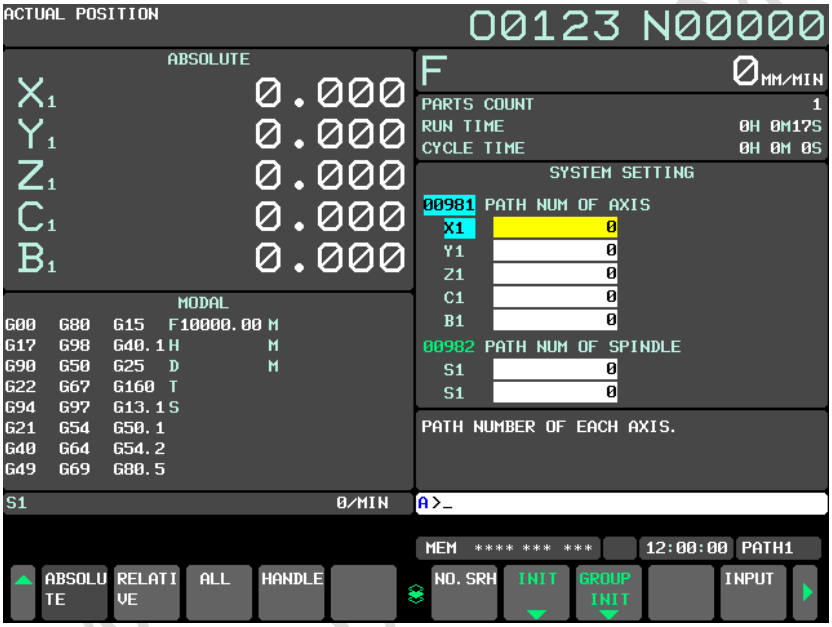


Fig. 12.4.10.2 (a) Parameter tuning screen (system setting) (10.4-inch display unit)

3 Move the cursor to a parameter number to be set or displayed, according to one of the methods below.

- Enter the parameter number and press soft key [NO.SRH] .
- Move the cursor to the desired parameter number by pressing page key ⬆PAGE or PAGE⬇, and cursor keys ⬆, ⬇, ⬅, and/or ➡.

  A brief description of the parameter where the cursor is placed is provided at the bottom of the screen. However, no description is provided when the cursor is placed on multiple bits for bit parameters.

4 Input desired data then press the INPUT key on the MDI unit to set the parameter.

5 Press soft key [INIT]. The standard value (recommended by FANUC) for the item selected by the cursor is displayed in the key input buffer. Pressing soft key [EXEC] in this state initializes the item to the standard value.

6 Press soft key [G_INIT]. A message asking whether to set the group standard values is displayed on the screen. Pressing soft key [EXEC] in this state inputs all of the standard values of the group.

NOTE

1 If the cursor is placed on a parameter that has no standard value assigned, no standard value is input even when [INIT] is pressed.
2 When the cursor is placed on multiple bits for bit parameters, the multiple bits can be input simultaneously. When [INIT] is pressed in this state, the key input buffer displays the standard values for the bits where the cursor is placed. If a bit has no standard value assigned, "*" is displayed for the bit, and no value is input for the bit.
3 When [G_INIT] is pressed, those parameters that have no standard values assigned are not initialized.

12.4.10.3 Parameter tuning screen (axis setting)

This screen enables the CNC parameters related to axes, coordinates, feedrate, and acceleration/deceleration to be displayed and set. The parameters displayed can be divided into four groups:

(Basic) group : The parameters related to basic settings are displayed.
(Coordinate) group : The parameters related to coordinates are displayed.
(Feedrate) group : The parameters related to feedrate are displayed.
(Acceleration/deceleration) group : The parameters related to acceleration/deceleration are displayed.

The parameters can be initialized to the standard values (recommended by FANUC). For the display and setting procedure, see the Subsection, “Parameter tuning screen (system setting)”.

Fig. 12.4.10.3 (a) Parameter tuning screen (axis setting) (10.4-inch display unit)

FRYER SYSTEMS INC

12.4.10.4 Displaying and setting the FSSB servo amplifier setting screen

From the parameter tuning screen, the FSSB servo amplifier setting screen can be displayed. For details of the FSSB servo amplifier setting screen, see the description of the FSSB servo amplifier setting screen in Subsection, “FSSB Setting” in the Connection Manual (Function) (B-64693EN-1).

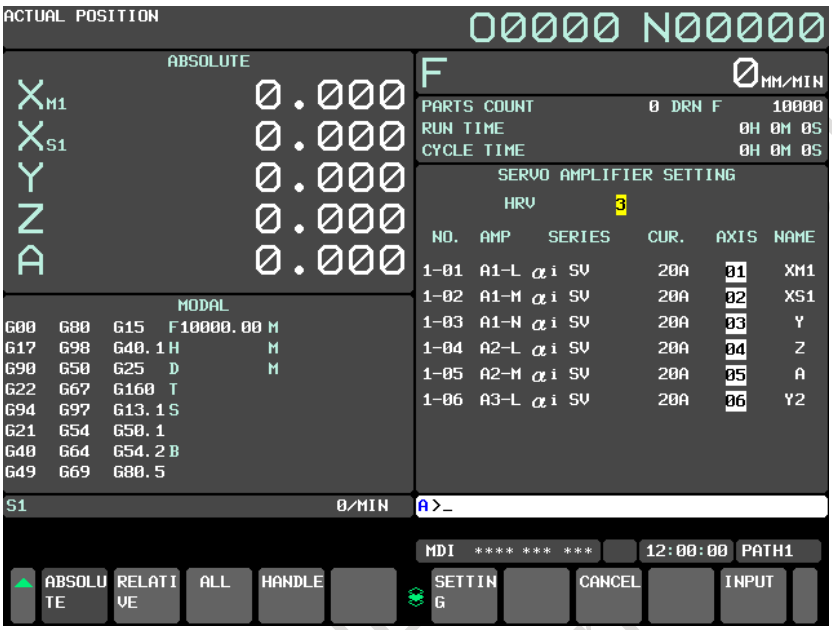


Fig. 12.4.10.4 (a) FSSB servo amplifier setting screen (10.4-inch display unit)

12.4.10.5 Displaying and setting the FSSB spindle amplifier setting screen

From the parameter tuning screen, the FSSB spindle amplifier setting screen can be displayed. For details of the FSSB spindle amplifier setting screen, see the description of the FSSB spindle amplifier setting screen in Subsection, “FSSB Setting” in the Connection Manual (Function) (B-64693EN-1).

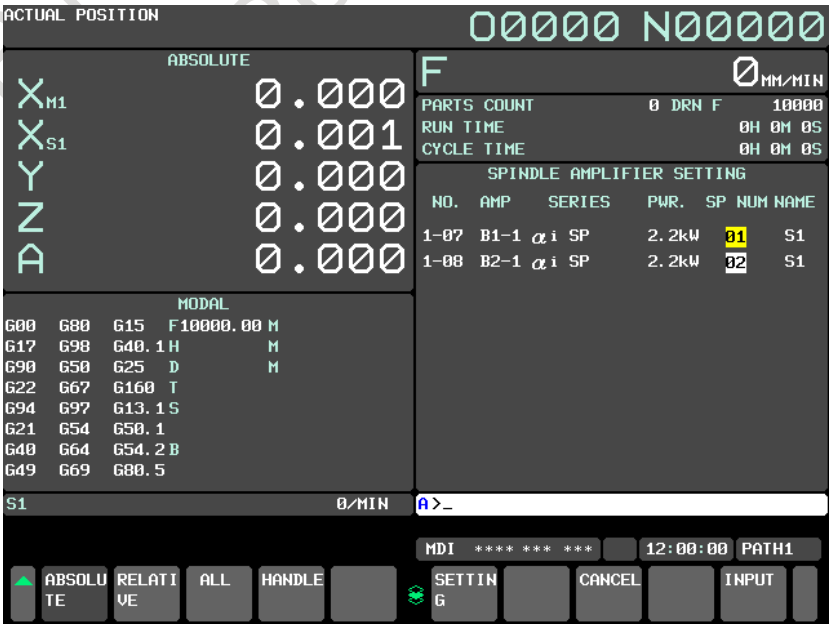


Fig. 12.4.10.5 (a) FSSB spindle amplifier setting screen (10.4-inch display unit)

12.4.10.6 Displaying and setting the FSSB axis setting screen

From the parameter tuning screen, the FSSB axis setting screen can be displayed. For details of the FSSB axis setting screen, see the description of the FSSB axis setting screen in Subsection, “FSSB Setting” in the Connection Manual (Function) (B-64693EN-1).

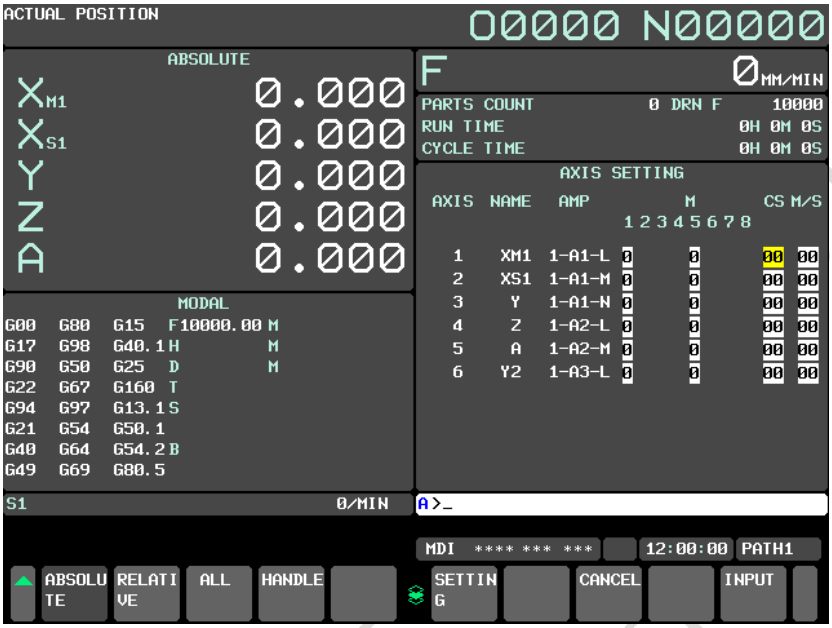


Fig. 12.4.10.6 (a) FSSB axis setting screen (10.4-inch display unit)

12.4.10.7 Displaying and setting the servo setting screen

From the parameter tuning screen, the servo setting screen can be displayed. For details of the servo setting screen, see the Subsection, “Servo Parameters”.

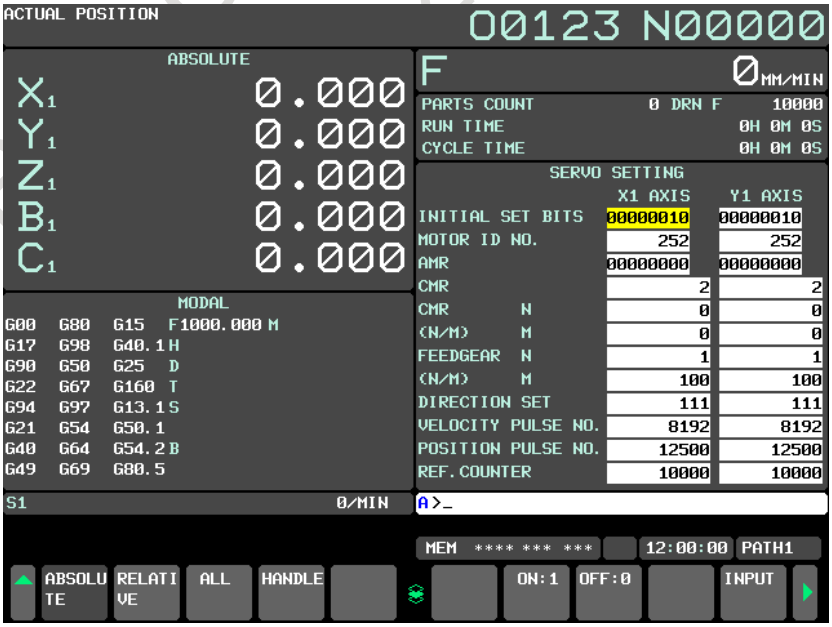


Fig. 12.4.10.7 (a) Servo setting screen (10.4-inch display unit)

FRYER CHINE Zahnsysteme INC

12.4.10.8 Parameter tuning screen (spindle setting)

The spindle-related parameters can be displayed and modified.
For the display and setting procedure, see the Subsection, “Parameter tuning screen (system setting)”.

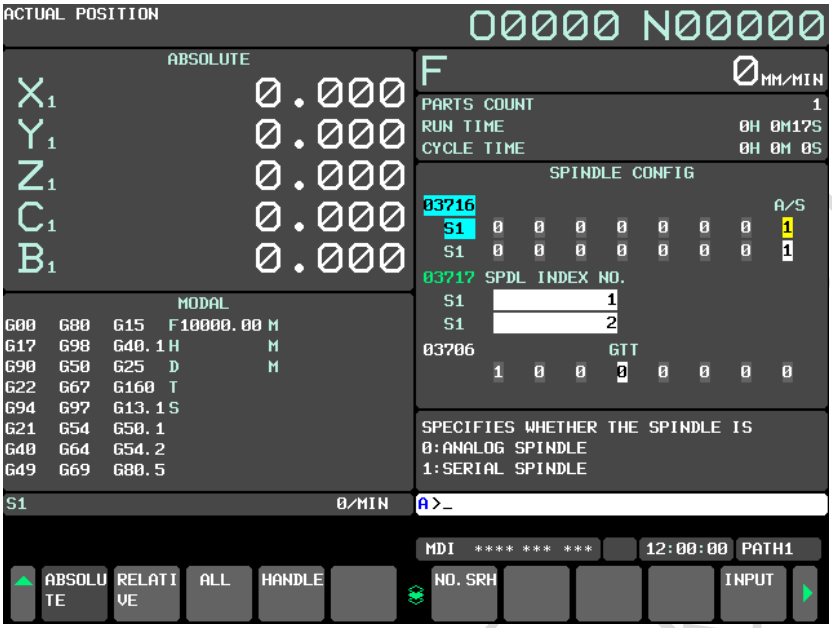


Fig. 12.4.10.8 (a) Parameter tuning screen (spindle setting) (10.4-inch display unit)

12.4.10.9 Parameter tuning screen (miscellaneous settings)

The parameters related to the allowable number of M code digits and whether to display the servo setting and spindle tuning screens can be displayed and modified. Moreover, the parameters can be initialized to the standard values (recommended by FANUC).
For the display and setting procedure, see the Subsection, “Parameter tuning screen (system setting)”.

Fig. 12.4.10.9 (a) Parameter tuning screen (Miscellaneous setting) (10.4-inch display unit)

12.4.10.10 Displaying and setting the servo tuning screen

From the parameter tuning screen, the servo tuning screen can be displayed. For details of the servo tuning screen, see the Subsection, “Servo Tuning”.

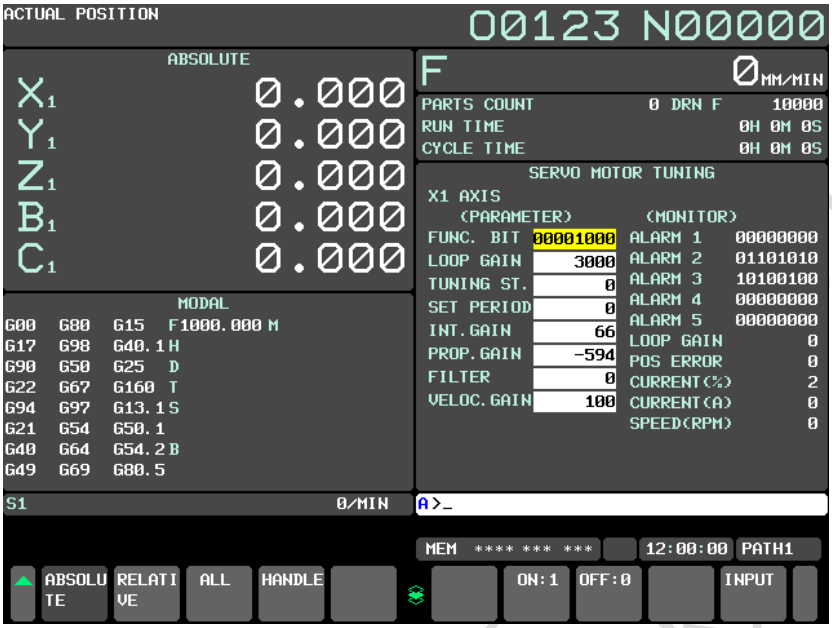


Fig. 12.4.10.10 (a) Servo tuning screen (10.4-inch display unit)

12.4.10.11 Displaying and setting the spindle tuning screen

From the parameter tuning screen, the spindle tuning screen can be displayed. For details of the spindle tuning screen, see the Subsection, “Spindle Tuning”.

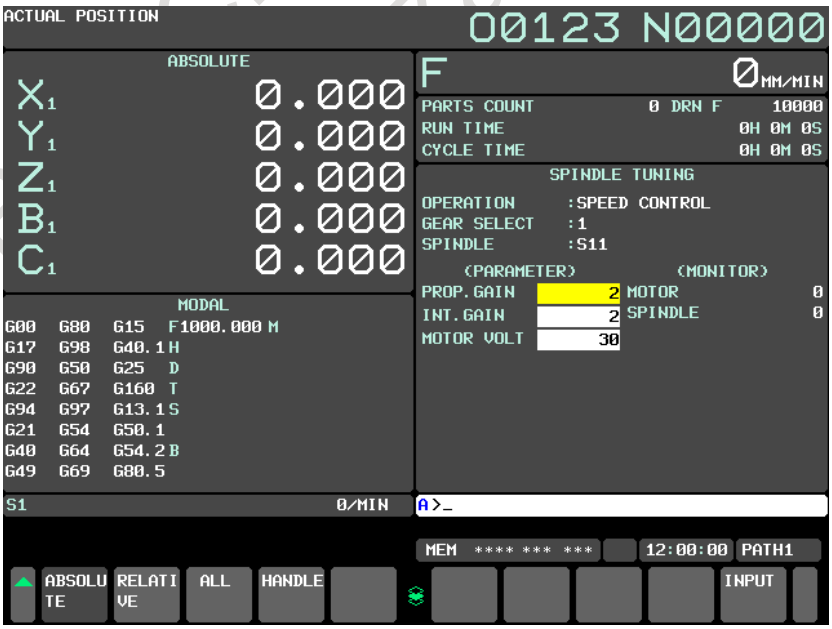


Fig. 12.4.10.11 (a) Spindle tuning screen (10.4-inch display unit)

FRYER MACHINE INC Zahensk

12.4.10.12 Displaying and setting the machining parameter tuning screen

From the parameter tuning screen, the machining parameter tuning screen can be displayed. For details of the machining parameter tuning screen, see the Subsection, “Machining Parameter Tuning”.

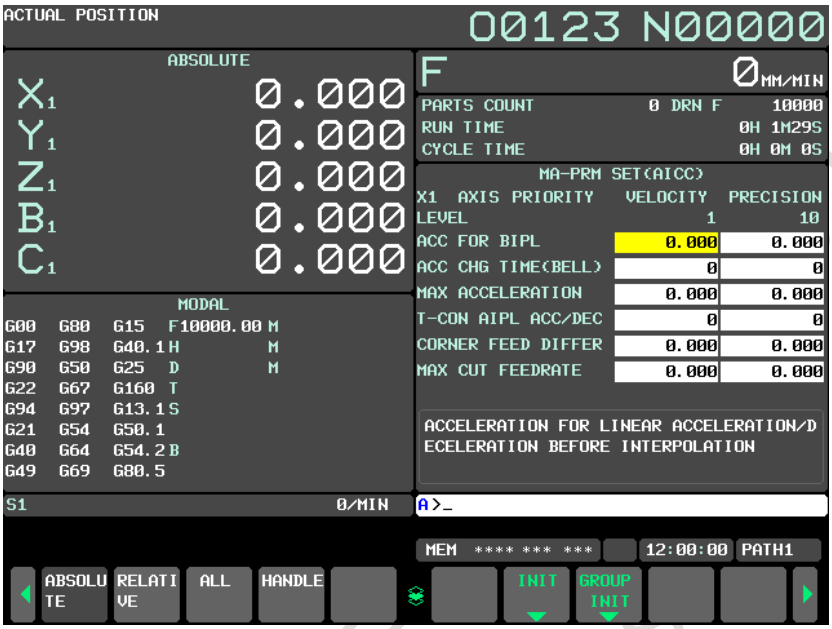


Fig. 12.4.10.12 (a) Machining parameter tuning screen (10.4-inch display unit)

Explanation

- Parameters displayed for parameter tuning

Table 12.4.10.12 (a) Parameters displayed for parameter tuning (1)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
SYSTEM SETTING	System setting	981		Sets the path of each axis.	
		982		Sets the path of each spindle.	
		983		Sets the T series/M series of each path. 0:T series / 1:M series	
		3021		Sets the G/F signal address of each axis.	*1
		3022		Sets the G/F signal address of each spindle.	*2
		3006#0	GDC	The deceleration signal for reference position return is: 0:X signal / 1:G signal	1
		3008#2	XSG	The signal address allocated to address X is: 0:Fixed / 1:Set by the parameter	1
		3013		Address allocated to the deceleration signal for reference position return	*3
		3014		Bit position allocated to the deceleration signal for reference position return	*4

*1 : When intra-path axis number ≤ 8, (path number - 1)*10+(intra-path axis number - 1)
When intra-path axis number ≥ 9, no standard value is available.
Example) When path 1 has 9 axes, and path 2 has 3 axes:
0,1,...,7,(none) for axes of path 1, 10,11,12 for axes of path 2

FRYER MACHINE INC Rob Zahens

*2 : When intra-path spindle number ≤ 4, (path number - 1)*10+(intra-path spindle number - 1)
When intra-path spindle number ≥ 5, no standard value is available.
Example) When path 1 has 5 spindles, and path 2 has 1 spindle:
0,1,...,4,(none) for spindles of path 1, 10 for spindle of path 2

*3 : When (path number ≤ 3) and (intra-path axis number ≤ 8)
Axes of path 1: 9
Axes of path 2: 7
Axes of path 3: 10
Other axes: No standard value is available.
Example) When path 1 has 9 axes, and path 2 has 3 axes:
9,9,9,9,9,9,9,9,(none) for axes of path 1, 7,7,7 for axes of path 2

*4 : When (path number ≤ 3) and (intra-path axis number ≤ 8)
(Intra-path axis number - 1)
Other axes: No standard value is available.
Example) When path 1 has 9 axes, and path 2 has 3 axes:
0,1,...,7,(none) for axes of path 1, 0,1,2 for axes of path 2

Table 12.4.10.12 (b) Parameters displayed for parameter tuning (2)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
SPINDLE SETTING	Spindle setting	3716#0	A/S	Sets the type of spindle motor: 0:Analaog / 1:Serial.	
		3717		Sets an amplifier number to be assigned to each spindle.	
		3706#4	GTT	Specifies a spindle gear selection method. 0:M type / 1:T type	
		3718		Sets a suffix to be added to spindle speed display on a screen such as the position display screen.	
		3735		Sets the minimum clamp speed of the spindle motor.	
		3736		Sets the maximum clamp speed of the spindle motor.	
		3741		Sets the maximum spindle speed for gear 1.	
		3742		Sets the maximum spindle speed for gear 2.	
		3743		Sets the maximum spindle speed for gear 3.	
		3744		Sets the maximum spindle speed for gear 4.	
		3772		Sets the maximum speed of the spindle. When 0 is set, the speed is not clamped.	
		4133		Sets the motor model code of the serial spindle. (This setting is not required for an analog spindle.)	
		4019#7	***	The parameters for the serial spindle are: 0:Not automatically set / 1:Automatically set (This setting is not required for an analog spindle.)	

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Table 12.4.10.12 (c) Parameters displayed for parameter tuning (3)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
AXIS SETTING	Basic	1001#0	INM	Least command increment on linear axes: 0:Metric (millimeter machine) / 1:Inch (inch machine)	
		1005#0	ZRNx	When an automatic operation (other than G28) is executed before reference position return: 0:Alarm is issued (PS224) / 1:No alarm is issued.	0
		1005#1	DLZx	Reference position return without dogs: 0:Disabled / 1:Enabled	
		1006#0	ROTx	Setting of linear axes or rotary axis: 0:Linear axis / 1:Rotation axis	
		1006#3	DIAx	Setting of the amount of travel: 0:Radius specification / 1: Diameter specification	
		1006#5	ZMIx	Reference position return direction: 0:Plus direction / 1:Minus direction	
		1008#0	ROAx	Rotation axis roll-over function: 0:Disabled / 1:Enabled	1
		1008#2	RRLx	With the amount of travel per revolution, relative coordinates are: 0:Not rounded / 1:Rounded	1
		1013#0	ISAx	Sets the least input increment and least command increment: 0:IS-B / 1:IS-A	
		1013#1	ISCx	Sets the least input increment and least command increment: 0:IS-B / 1:IS-C	
		1020		Program axis name	*1
		1022		Sets each axis in the basic coordinate system.	*2
		1023		Servo axis number	
		1815#1	OPTx	A separate pulse coder is: 0:Not used / 1:Used	
		1815#4	APZx	The correspondence between machine positions and absolute-position detector positions is: 0:Not established / 1:Established	
		1815#5	APCx	The position detector used is: 0:Other than an absolute-position detector / 1:Absolute-position detector	
		1825		Servo loop gain	
		1826		Effective area	
		1828		Positional deviation limit during travel	
		1829		Positional deviation limit during stop	500

*1 : For the M series: 88(X),89(Y),90(Z) sequentially starting with the first axis (no standard value for the fourth axis and up)
For the T series: 88(X),90(Z) (no standard value for the third axis and up)

*2 : For the M series: 1,2,3 sequentially starting with the first axis (no standard value for the fourth axis and up)
For the T series: 1,3 (no standard value for the third axis and up)

FRYER MACHINE SYSTEMS INC
Rob Zarensky

Table 12.4.10.12 (d) Parameters displayed for parameter tuning (4)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
AXIS SETTING	Coordinate	1240		Machine coordinate of the first reference position	
		1241		Machine coordinate of the second reference position	
		1260		Amount of travel per revolution of a rotary axis	360.000
		1320		Coordinate of the boundary of stored stroke check 1 in the positive direction	
		1321		Coordinate of the boundary of stored stroke check 1 in the negative direction	
	Feedrate	1401#6	RDR	For a rapid traverse command, dry run is: 0:Disabled / 1:Enabled	0
		1410		Dry run feedrate	
		1420		Rapid traverse rate	
		1421		Rapid traverse override F0 rate	
		1423		Jog feedrate	
		1424		Manual rapid traverse rate	
		1425		FL feedrate for reference position return	
		1428		Reference position return feedrate	
		1430		Maximum cutting feedrate	
	Acceleration/deceleration	1610#0	CTLx	Acceleration/deceleration in cutting feed or dry run during cutting feed: 0: Exponential acceleration/deceleration is applied / 1: Linear acceleration/deceleration after interpolation is applied	
		1610#4	JGLx	Acceleration/deceleration for jog feed is: 0: Exponential acceleration/deceleration / 1: Same as acceleration/deceleration for cutting feed (The settings of bit 1 (CTBx) and bit 0 (CTLx) of parameter No. 1610 are followed.)	
		1620		Time constant for linear acceleration/deceleration for rapid traverse	
		1622		Time constant for acceleration/deceleration for cutting feed	
		1623		FL feedrate for acceleration/deceleration after interpolation for cutting feed	
		1624		Time constant for acceleration/deceleration for jog feed	
		1625		FL feedrate for exponential acceleration/deceleration for jog feed	
MISCELLANY	MISC	3030		Allowable number of digits of an M code	3
		3111#0	SVS	The servo setting screen is: 0:Not displayed / 1:Displayed	1
		3111#1	SPS	The spindle setting screen is: 0:Not displayed / 1:Displayed	1

FRYER MACHINE SYSTEMS INC
Rob Zykowsky

12.4.11 Periodic Maintenance Screen

Periodic maintenance screens are used for managing consumables (such as the backlight of a LCD unit and backup batteries). By setting the name of a consumable, its life time, and the method for counting down the life time, the life of the consumable can be counted down in the appropriate method set for the consumable, and the remaining life time can be displayed.
With these screens, the user can easily manage consumables that require periodic replacement.
The total number of item that can be managed is 10 in all path.

Explanation

There are four periodic maintenance screens: the status screen, the setting screen, the machine menu screen, and the NC menu screen.

Status screen : Item names, remaining times, and count statuses are displayed, and item names are set.
Setting screen : Life times, remaining times, count types (count-down method), and path number (in case of multi-path system) are set.
Machine menu screen : The names of consumables in the machine can be registered.
NC menu screen : The names of consumables in the NC are already registered.

Using periodic maintenance screens

<1> Referencing a periodic maintenance screen
Display the status screen. For the meanings of items on the status screen, see Item, “**Status screen**”.

<2> Adding the name of a new consumable item to a periodic maintenance screen or editing an existing consumable item on the screen
The name of a consumable item can be added or edited on the machine menu screen. For details, see Item, “**Machine menu screen**”.

<3> Adding or editing a life time, remaining time, and a method of counting down the remaining time for a consumable on a periodic maintenance screen
A life time and remaining time can be added or edited on the setting screen. For details, see Item, “**Setting screen**”.

<4> Newly displaying the item name and remaining time of a consumable on a periodic maintenance screen.

1 Setting an item name
Select the item name of a consumable to be displayed from the machine menu screen or NC menu screen, or enter the name using the MDI keys. For the procedure, see Item, “**Item name**” in “**Status screen**”.

2 Setting a life time, remaining time, and count type
Select the life time, remaining time, and count type of a consumable to be displayed from the setting screen. For the procedure, see Item, “**Remaining time**” in “**Status screen**”.

Procedure for displaying a periodic maintenance screen

1 Press function key [SYSTEM].
2 Press the continuous menu key [▷] several times to display [PERIOD MAINTE].
3 Press soft key [PERIOD MAINTE] to display the periodic maintenance screen.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Status screen

When soft key [STATUS] is pressed, the status screen is displayed. The status screen shows the item names, count statuses, and remaining times of managed consumables.

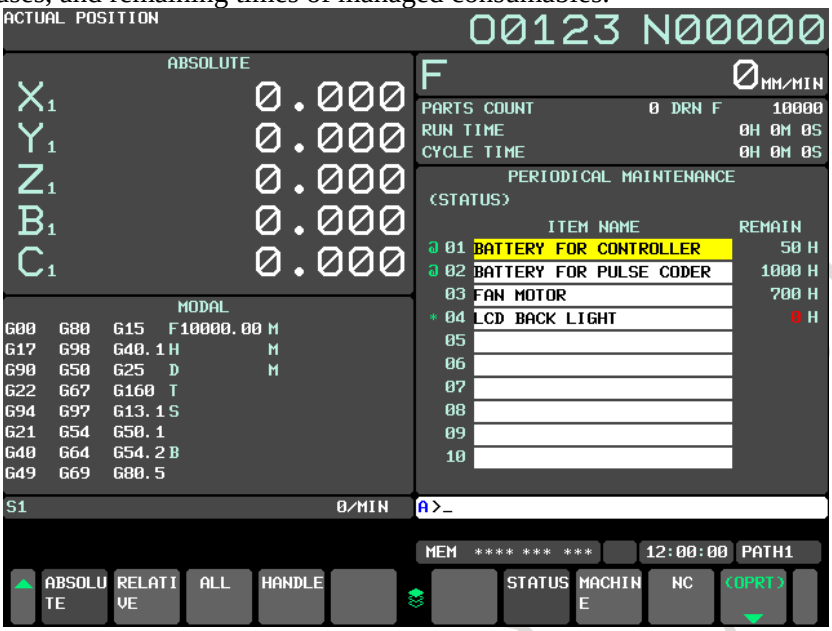


Fig. 12.4.11 (a) Status screen

- Item name

As the item name, set the name of a consumable to be managed by periodic maintenance. To set an item name, select a name from the machine menu screen or NC menu screen, or directly enter the name using the MDI keys.

Setting an item name from a menu screen

1 On the status screen, move the cursor to a target item name, press soft key [(OPRT)], then press soft key [ENTRY].
2 Press soft key [MACHINE] or [NC] to display the machine menu screen or NC menu screen.
3 Move the cursor to an existing item name on the menu screen, press soft key [(OPRT)], and press soft key [SELECT] then [EXEC] successively.
4 The screen display returns to the status screen, and the item name selected on the menu screen is added to the status screen.

Initially, there is no item name set on the machine menu screen, so item names must be registered in advance. For the registration method, see the description of the procedure for registering item names on the machine menu screen.

Setting an item name using the MDI keys

1 Press soft key [(OPRT)].
2 Type the alphanumeric characters to be input, then press soft key [INPUT].
3 The entered item name is registered on the status screen.

When soft key [+INPUT] is pressed instead of soft key [INPUT], the entered characters can be added to an existing item name.
When typing 2-byte characters, type "*" before and after the character codes. The 2-byte character codes must conform to FANUC codes. (See Appendix, "FANUC 2-BYTE CHARACTER CODE TABLE".)
An item name to be registered must be up to 24 characters long if it consists of alphanumeric characters only; or it must be up to 12 characters long if it consists of 2-byte characters only.
Example: To register "LCDバックライト", enter the following:

>LCD*110E10F410CC114010B610FE*_

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE
1 An asterisk "*" is used as a control code, so it cannot be used in item names. In addition, characters "[", "]", "(", and ")" cannot be used in item names.
2 When an item name consisting of both alphanumeric characters and 2-byte characters is registered, the warning "DATA OUT OF RANGE" may be output.

Deleting an item name
To delete a registered item name, move the cursor to the item name, press soft key [ERASE], then press soft key [EXEC].
When an item name is deleted, its life time, remaining time, and count type are deleted at the same time.

- Remaining time

As the remaining time, the period of time left until the time for replacement is reached by count down operation is displayed.
If the percentage of the remaining time to the life time has reached to the value (%) specified by parameter No. 8911, or smaller, the remaining time is displayed in red.
Even after the life time has expired, count down operation continues.

Setting the remaining time
1 On the status screen, place the cursor on an item for which the remaining time is to be set (the item name must have been set in advance).
2 Press soft key [(OPRT)], then press soft key [CHANGE].
3 The screen display changes to the setting screen.
4 Set the life time, remaining time, and count type. For the setting method and other information, see Item, "**Setting screen**".

NOTE
On the status screen, the remaining time and life time cannot be set.
These items must be set on the setting screen.

- Count status

The count status is indicated on the left side of the item number as follows:

Indication	Count status
Blank	Counting stopped
@	Counting in progress
*	Life expired

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Setting screen

On the setting screen, the life time, remaining time, and count type of a managed consumable are set.

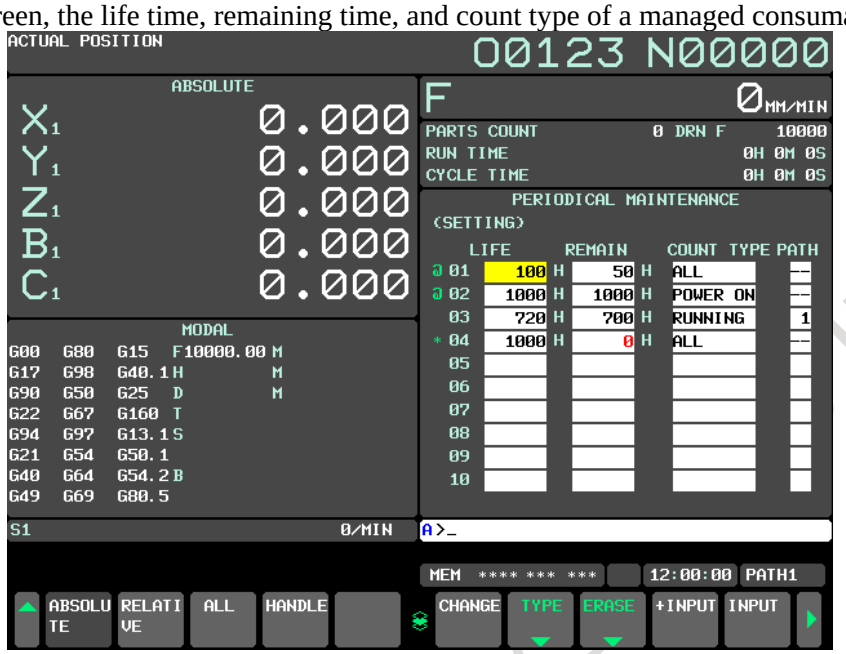


Fig. 12.4.11 (b) Setting screen

Display procedure

1 When the status screen is displayed, press soft key [(OPRT)].
2 Press soft key [CHANGE].

- Life time

Set the life time of a consumable.

Move the cursor to an existing item, type a life time, then press soft key [INPUT] (or the [INPUT] key). The life time is then set, and the same value is set also as the remaining time.
At this time, "------" is indicated in the count type field.
When soft key [+INPUT] is pressed, the entered value can be added to the life time already set. The same value as the added value is added also to the remaining time.
A value ranging from 0 to 65535 (in hours) can be set.

NOTE

1 If a setting operation is attempted when the item name is not registered, the warning "EDIT REJECTED" is issued.
2 If a value exceeding the valid range is entered, the warning "DATA IS OUT OF RANGE" is issued.
3 If soft key [ERASE] or [TYPE] is pressed, the warning "EDIT REJECTED" is issued.

- Remaining time

The period of time left until the time for replacement is reached by count down operation is indicated.
If the percentage of the remaining time to the life time has reached the value (%) specified by parameter No. 8911, or smaller, the remaining time is displayed in red.
Even after the life time has expired, count down operation continues.
Move the cursor to the remaining time of a target registered number, type a remaining time, then press soft key [INPUT] (or the [INPUT] key). The remaining time is then set.
When soft key [+INPUT] is pressed, the entered value can be added to the remaining time already set.
A value ranging from 0 to (life time) can be set.
When soft key [ERASE] then soft key [EXEC] are pressed, the same value as the life time is set.

FRYER MACHINE SYSTEMS INC
Robzanensky

NOTE
1 If a setting operation is attempted when the item name or life time is not registered, the warning “EDIT REJECTED” is issued.
2 When a value exceeding the valid range is entered, the warning “DATA IS OUT OF RANGE” is issued.
3 If horizontal soft key [TYPE] is pressed, the warning “EDIT REJECTED” is issued.

- Count type

As the count type, select the way of counting.
Place the cursor on the count type of a target registration number, then press soft key [TYPE]. Count types are displayed as soft keys as shown below. Select one of these soft keys, then press soft key [EXEC].

Soft key	Meaning	Indication
[NO CNT]	Counting is not performed (stopped)	— — — — — —
[ALL]	Counting is performed at all times	All times
[POWER ON]	Counting is performed when power is on.	When power is on
[RUN]	Counting is performed when operation is in progress.	When operation is in progress
[CUT]	Counting is performed when cutting is being performed.	When cutting is being performed

NOTE
1 If a setting operation is attempted when the item name or life time is not registered, the warning “EDIT REJECTED” is issued.
2 Soft keys [INPUT] and [+INPUT] have no effect.
3 When counting is performed at all times, a 24-hour error is generated in leap year.
4 If soft key [ERASE] is pressed, the warning “EDIT REJECTED” is issued.

- Path number

When [RUN] or [CUT] is set to the count type in multi path, path number can be set.
Make the cursor to the count type of a target registration number, then input path number and press soft key [INPUT].
The setting range is 0-(max path number).
When the specified path is operating or cutting, counting is performed.
If 0 is set, when either of path is operating or cutting, counting is performed.

NOTE
1 If [NO CNT], [ALL] or [POWER ON] is set to count type, “--” is displayed. In this case, if the setting is operated, the warning “EDIT REJECTED” is issued.
2 If the setting is beyond the valid data range, the warning “DATA IS OUT OF RANGE” is issued.
3 If soft key [TYPE] is pressed, the warning “EDIT REJECTED” is issued.
4 If soft key [ERASE] is pressed, the warning “EDIT REJECTED” is issued. The soft key [+INPUT] is invalid.

Registration from a program

The item name, life time, remaining time, count type and path number can be registered in the status screen and the setting screen by executing the program of the following format.

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

Format

G10 L60 Px [n] Aa Rr Qpt

X : Registration number

n : Item name, format: [alphanumeric-characters*2-byte-characters*alphanumeric-characters]

a : life time

r : remaining time

p : path number

t : count type

The meaning of the value of the count type is as follow.

0 : NO COUNT

1 : ALL TIMES

2 : POWER ON

3 : RUNNING

4 : CUTTING

The range of the path number is 0 – (max path number).

EX.)

Q24 : The path number is 2, and the count type is “CUTTING”

Q103 : The path number is 10, and the count type is “RUNNING”

Machine menu screen

On the machine menu screen, the names of consumables of the machine are registered. From this screen, item names can be added to the status screen. For the method of addition to the status screen, see the description of the status screen.

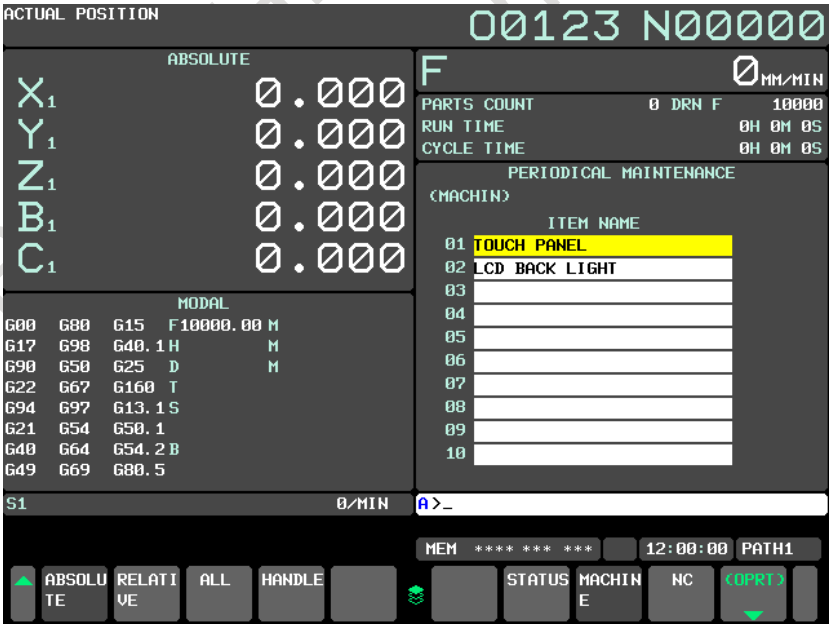


Fig. 12.4.11 (c) Machine menu screen

- Displaying the screen

1 When the status screen is displayed, press soft key [MACHINE].

On the machine menu screen, item names can be registered using one of the following two methods:

- Registration from a program
- Registration using the MDI keys

FRYER MACHINE SYSTEMS INC Chensky

- Registration from a program

By executing a program having the following format, an item name can be registered in the menu of the machine:

Format

G10 L61 Px [n]

X Registration number

n Item name, format: [alphanumeric-characters*2-byte-characters*alphanumeric-characters]

- Registration using the MDI keys

An item name can be registered on the menu screen of the machine by typing the item name in the format shown below on the machine menu screen, then pressing soft key [INPUT] (or the INPUT key).

When soft key [+INPUT] is pressed, the typed characters can be added to an already registered item name.

Format

Alphanumeric-characters*2-byte-characters*alphanumeric-characters

Two-byte character codes must conform to FANUC codes. (See Appendix, "FANUC 2-BYTE CHARACTER CODE TABLE".)
When typing 2-byte characters, type an asterisk "*" before and after the character codes. An item name to be registered must be up to 24 characters long if it consists of alphanumeric characters only; or it must be up to 12 characters long if it consists of 2-byte characters only.

Example:

To register "LCD バックライト", enter the following:
>LCD*110E10F410CC114010B610FE*_

NOTE

1 An asterisk "*" is used as a control code, so it cannot be used in item names. In addition, characters "[", "]", "(", and ")" cannot be used in item names.
2 When an item name consisting of both alphanumeric characters and 2-byte characters is registered, the warning “DATA IS OUT OF RANGE” may be output.
3 When a blank item name is selected on the machine screen, the warning “EDIT REJECTED” is issued.

To delete a registered item name, move the cursor to the item name, press soft key [ERASE], then press soft key [EXEC].

FREE MACHINE SYSTEMS INC
Rob Zalonsky

NC menu screen

From this screen, an item name can be registered on the status screen. For the method of registration to the status screen, see the description of the status screen.

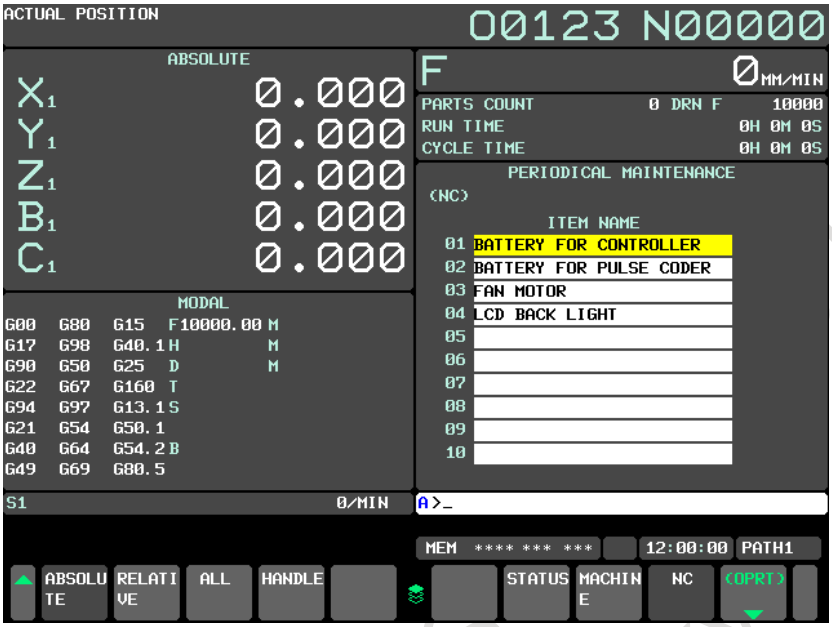


Fig. 12.4.11 (d) NC menu screen

- Displaying the screen

1 When the status screen is displayed, press soft key [NC].

> **NOTE**
> On the NC screen, the registration, deletion, and I/O of item names cannot be performed.
> When a blank item name is selected, a blank is set.

12.4.12 System Configuration Screen

The system configuration screen provides information about the types of installed hardware and software.

Procedure for displaying the screen

Procedure

1 Press the function key [SYSTEM] to display a screen that shows parameters and other information.

2 Press soft key [SYSTEM]. The system configuration screen is displayed.

There are two types of system configuration screens: the hardware configuration screen and the software configuration screen. The screen display can switch between these screens by using [PAGE ↑] and [PAGE ↓]. When soft key [SERVO INFO] or [SPINDLE INFO] is pressed, information about the connected servo system or spindles is displayed.

FRYER MACHINE SYSTEMS INC Rob Zahensky

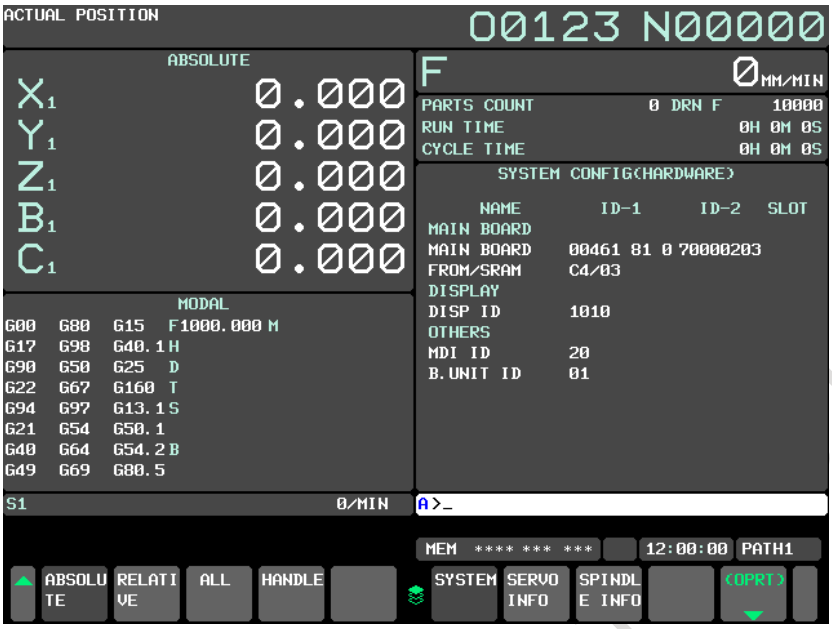


Fig. 12.4.12 (a) System configuration screen

Hardware configuration screen

This screen shows the names and IDs of the hardware used by the NC.

Fig. 12.4.12 (b) Hardware configuration screen

Software configuration screen

This screen shows the names and series/editions of the software used by the NC.

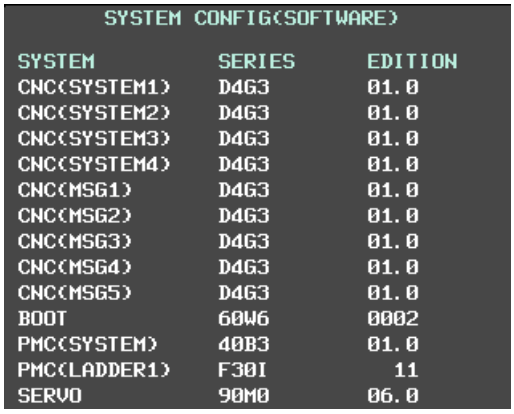


Fig. 12.4.12 (c) Software configuration screen

Servo information screen

When a servo system is connected to the NC, ID information of the connected servo devices (servo motors and servo amplifiers) can be displayed on the NC.

Displaying the screen

1 When the system configuration screen is displayed, press soft key [SERVO INFO].
2 The servo information screen is displayed.

```
          SERVO INFORMATION
   X1  AXIS
SERVO MOTOR SPEC A06B-2212-B002
SERVO MOTOR S/N  C151F3475
PULSECODER SPEC. A860-2051-T321
PULSECODER S/N   14121115
SV SPEC.         A06B-6240-H305
SV S/N           V14Z43833
SV SOFT EDITION  9H0015.0
PS SPEC.         A06B-6200-H008
PS S/N           V14Z39904
PS SOFT EDITION  9G0019.0
```

Fig. 12.4.12 (d) Servo information screen

Spindle information screen

When a spindle system is connected to the NC, the ID information of the connected spindle devices (spindle motors and spindle amplifiers) can be displayed on the NC.

Displaying the screen

1 When the system configuration screen is displayed, press soft key [SPINDLE INFO].
2 The spindle information screen is displayed.

```
          SPINDLE INFORMATION
   S1
MAINSP MOTOR SPC A06B-1401-B105
MAINSP MOTOR S/N V00000001
SP SPEC.         A06B-6220-H002#H600
SP S/N           V14Y02875
PS SPEC.         A06B-6200-H008
PS S/N           V14Z39904
PS SOFT EDITION  9G0019.0
```

Fig. 12.4.12 (e) Spindle information screen

FRYER MACHINE SYSTEMS INC
Zahensky

12.4.13 Power Consumption Monitoring Screen

The electric power data of servo axis and spindle axis consumption and regeneration can be displayed.

Display of power consumption monitoring screen

1 Press function key [SYSTEM].

2 Press continuous menu key [▷] several times until soft key [POWER MONIT] appears.

4 Press soft key [POWER MONIT].

Fig. 12.4.13 (a) Power consumption monitoring screen for 10.4-inch display unit

Operation of power consumption monitoring screen

Switch of display axis

When information on all axes is not displayed, the page is switched with the MDI key <PageUp> or <PageDown>.

Clear cumulative power consumption value

1 Press soft key [(OPRT)].

2 Press soft key [TOTAL CLEAR].
Soft key [EXEC] and [CAN] appear.

3 Press soft key [EXEC].
Cumulative power consumption and cumulative time start from 0.
To quit clear operation, press soft key [CAN].

Explanation

TIME

Integrating time of power consumption is displayed.

Axis name

Axis name of servo and spindle is displayed. "ALL" means the total amount of power consumption.

FRYER MACHINE Rob Zak INC

CONSUMP

Integral power consumption is displayed.

REGEN

Integral power regeneration is displayed.

NET

Integral net amount of power consumption is displayed.
The net amount of power consumption = power consumption – power regeneration.

PRESENT

Present net power consumption is displayed. A negative value is displayed about the net power consumption while the power is regenerating like the axis is decelerating, etc.

Bar-graph

Current net power consumption is displayed as a bar-graph.
The scale of bar-graph can be specified in the parameter (No.11392 in case of the servo, No.11393 in case of the spindle).
The scale of the entire power consumption bar-graph can be specified in the parameter (No.11371).
The bar expands from the center to right while consuming the electric power. The bar expands from the center to left while regenerating the electric power.

Unit of data

When the cumulative power consumption value can no longer be displayed in kWh, unit of data is switched to MWh automatically

Screens of a 15/19-inch display unit

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.4.14 Displaying and Setting Parameters (15/19-inch Display Unit)

When the CNC and machine are connected, parameters are set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor. The setting of parameters depends on the machine. Refer to the parameter list prepared by the machine tool builder. Normally, the user need not change parameter setting.

Procedure for displaying and setting parameters

Procedure

1 Set 1 for PARAMETER WRITE to enable writing. See the procedure for enabling/disabling parameter writing described below.

2 Press function key [SYSTEM].

3 Press vertical soft key [PARAM] to display the parameter screen.

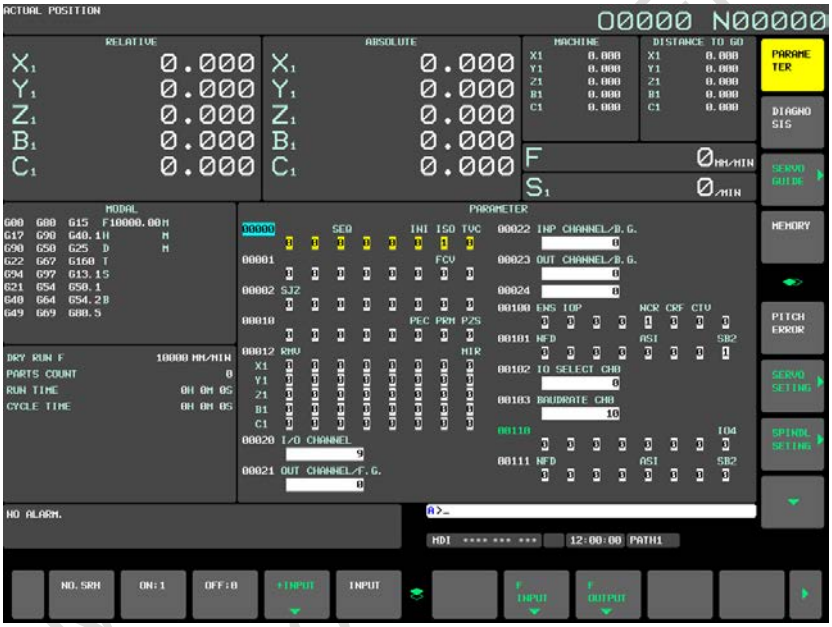


Fig. 12.4.14 (a) PARAMETER screen (15-inch display unit)

4 Move the cursor to the parameter number to be set or displayed in either of the following ways:
- Enter the parameter number and press horizontal soft key [NO.SRH] .
- Move the cursor to the parameter number using the page keys, [PAGE ⇧] and [PAGE ⇩], and cursor keys, [⬆], [⬇], [⬅], and [➡].

5 To set the parameter, enter a new value with numeric keys and press horizontal soft key [INPUT]. The parameter is set to the entered value and the value is displayed.

6 Set 0 for PARAMETER WRITE to disable writing.

FRYER INDUSTRIAL SYSTEMS INC

Procedure for enabling/displaying parameter writing

Procedure

1 Select the MDI mode or enter state emergency stop.

2 Press function key .

3 Press vertical soft key [SETTING] to display the setting screen.

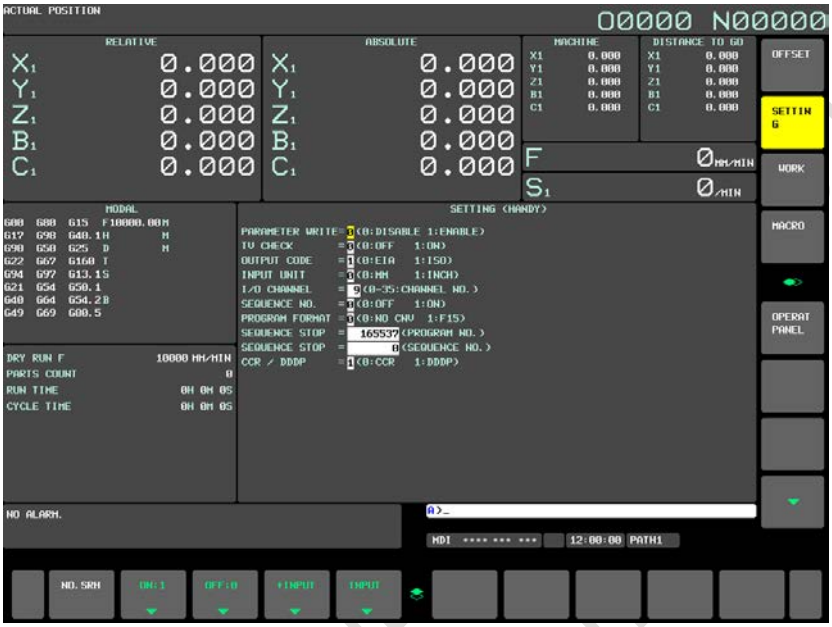


Fig. 12.4.14 (b) SETTING screen (15-inch display unit)

4 Move the cursor to PARAMETER WRITE using cursor keys.

5 Press horizontal soft key [ON:1] to enable parameter writing.
At this time, the CNC enters the alarm state SW0100.

6 After completing parameter setting, move the cursor to PARAMETER WRITE on the setting screen again, and press horizontal soft key [OFF:0].

7 Depress the RESET key to release the alarm condition.

When an alarm PW0000 occurred, however, it is not released unless the power is turned off and back on.

Explanation

- Setting parameters with external input/output devices

See Chapter, “DATA INPUT/OUTPUT” for setting parameters with external input/output devices such as the memory card.

- Parameters that require turning off the power

Some parameters are not effective until the power is turned off and on again after they are set. Setting such parameters causes alarm PW0000. In this case, turn off the power, then turn it on again.

- Parameter list

Refer to the Parameter Manual (B-64700EN) for the parameter list.

- Setting data

Some parameters can be set on the setting screen if the parameter list indicates "Setting entry is acceptable". Setting 1 for PARAMETER WRITE is not necessary when three parameters are set on the setting screen.

FRYER MACHINE INC Rob Zaher

12.4.15 Servo Parameters (15/19-inch Display Unit)

This subsection describes the initialization of digital servo parameters performed, for example, at the time of field tuning of the machine tool.

Procedure for servo parameter setting

Procedure

1 Turn on the power in the emergency stop state.

2 Set bit 0 (SVS) of parameter No. 3111 to 1 to display servo setting and tuning screens.

3 Press function key [SYSTEM] then vertical soft key [SERVO PARAM]. The following screen appears:

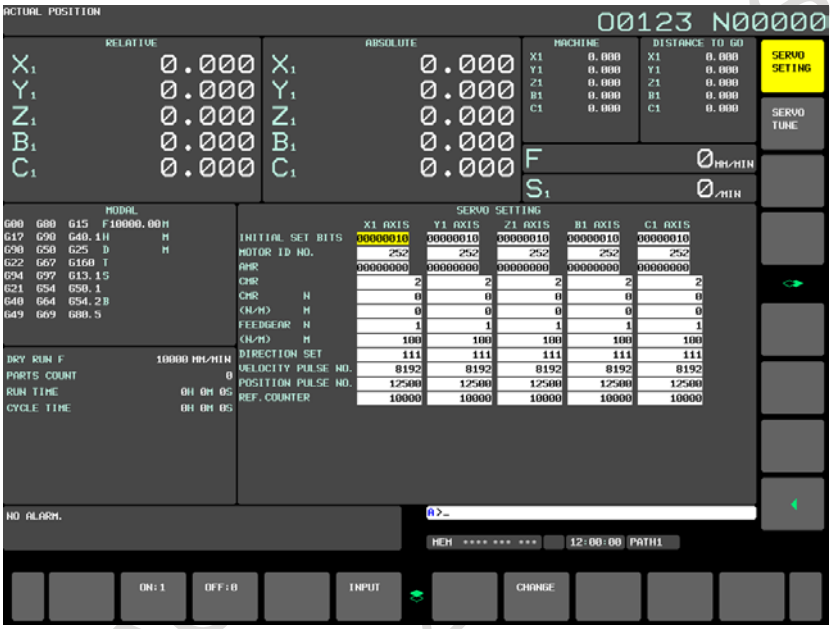


Fig. 12.4.15 (a) Servo parameter setting screen (15-inch display unit)

4 With the page keys [PAGE ↑] [PAGE ↓] and cursor keys [↑] [↓] [←] [→], move the cursor to the position of data to be set or modified.

5 Key in a desired value then press horizontal soft key [INPUT].

FRYER MACHINES INC

12.4.16 Servo Tuning (15/19-inch Display Unit)

Data related to servo tuning is displayed and set.

Procedure for servo tuning

Procedure

1 Turn on the power in the emergency stop state.

2 Set bit 0 (SVS) of parameter No. 3111 to 1 to display servo setting and tuning screens.

3 Press function key [SYSTEM] then vertical soft key [SERVO PARAM].

4 Press vertical soft key [SERVO TUNE] to select the servo tuning screen. The servo tuning screen appears:

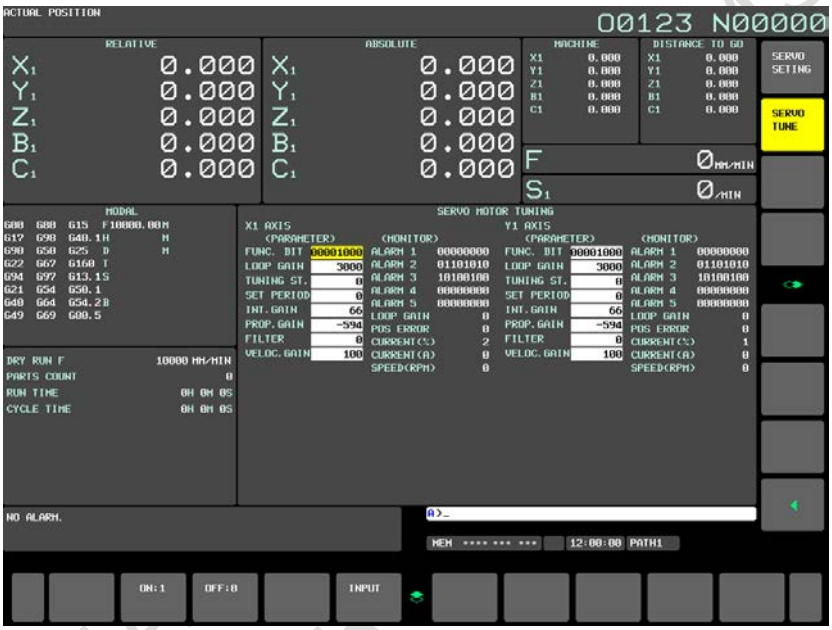


Fig. 12.4.16 (a) Servo tuning screen (15-inch display unit)

5 With the page keys [PAGE] [PAGE] and cursor keys [↑] [↓] [←] [→], move the cursor to the position of data to be set or modified.

6 Key in a desired value then press horizontal soft key [INPUT].

FRYER MACHINE PRODUCTS INC

12.4.17 Spindle Setting (15/19-inch Display Unit)

Parameters related to spindles are set and displayed. In addition to the parameters, related data can be displayed. Screens for spindle setting, spindle tuning, and spindle monitoring are provided.

Setting spindle parameters

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display spindle setting and tuning screens.

2 Press function key [SYSTEM] then vertical soft key [SPINDL SETING].

3 Press vertical soft key [SPINDL SETING] to select the spindle setting screen.

4 The spindle setting screen appears:

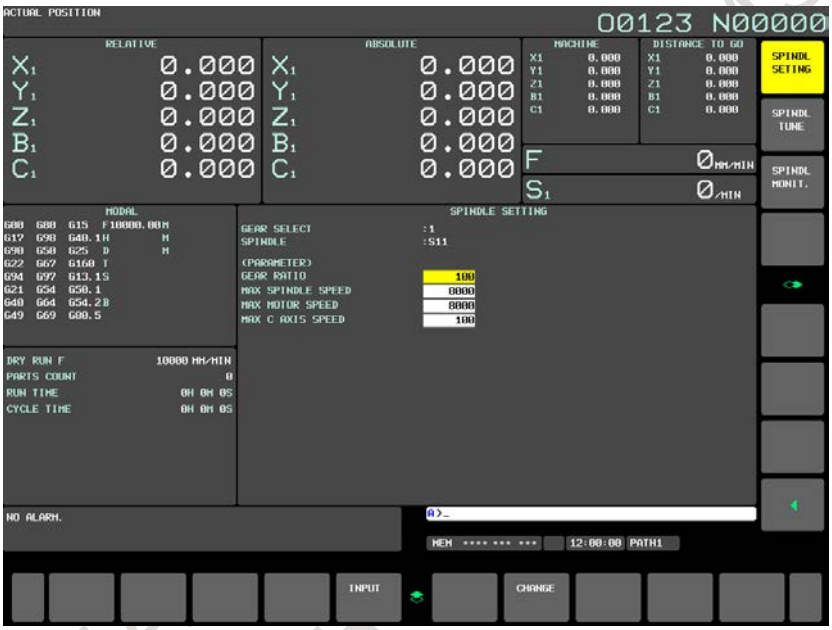


Fig. 12.4.17 (a) Spindle setting screen (15-inch display unit)

5 With the page keys [PAGE ⇧] [PAGE ⇩] and cursor keys [⬆] [⬇] [⬅] [➡], move the cursor to the position of data to be set or modified.

6 Key in a desired value then press horizontal soft key [INPUT].

FRYER MACHINE PRODUCTS INC

12.4.18 Spindle Tuning (15/19-inch Display Unit)

Spindle tuning data is displayed and set.

Setting for spindle tuning

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display spindle setting and tuning screens.

2 Press function key [SYSTEM] then vertical soft key [SPINDL SETING].

3 Press vertical soft key [SPINDL TUNE] to select the spindle tuning screen.

4 The spindle tuning screen appears:

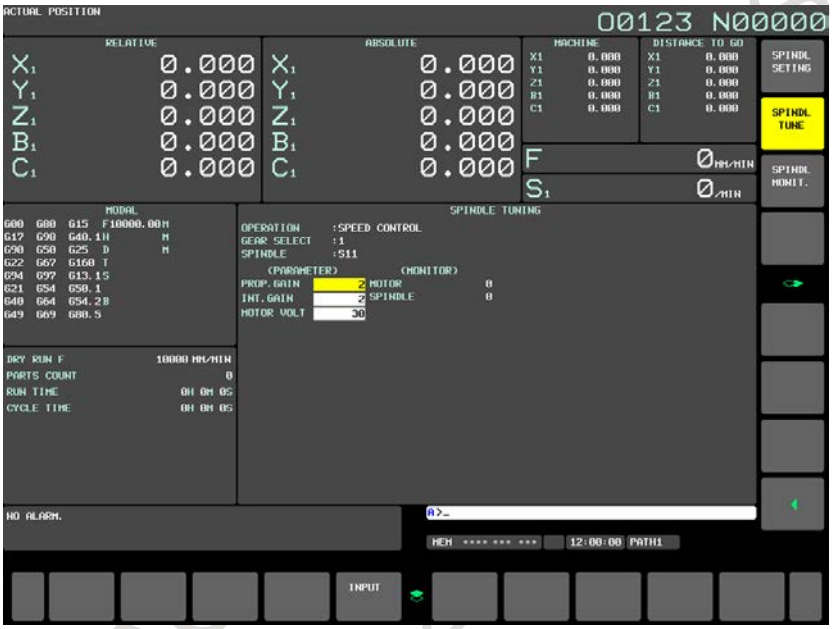


Fig. 12.4.18 (a) Spindle tuning screen (15-inch display unit)

5 With the page keys [PAGE] [PAGE] and cursor keys [↑] [↓] [←] [→], move the cursor to the position of data to be set or modified.

6 Key in a desired value then press horizontal soft key [INPUT].

FRYER MACHINERY Robots INC

12.4.19 Spindle Monitor (15/19-inch Display Unit)

Spindle-related data is displayed.

Displaying the spindle monitor

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display spindle setting and tuning screens.

2 Press function key [SYSTEM] then vertical soft key [SPINDL SETING].

3 Press vertical soft key [SPINDL MONIT.] to select the spindle monitor screen.

4 The spindle monitor screen appears:

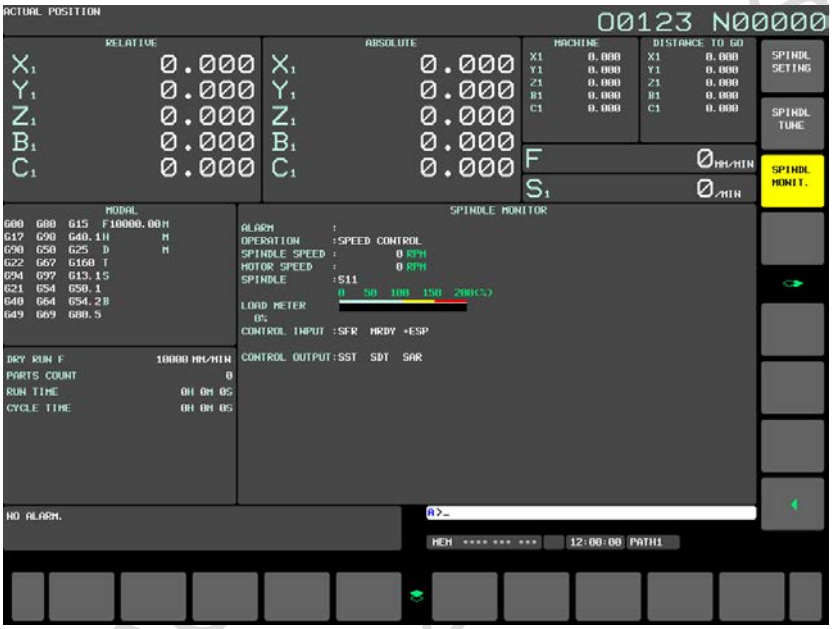


Fig. 12.4.19 (a) Spindle monitor screen (15-inch display unit)

FRYER MACHINERY INC
Rob

12.4.20 Color Setting Screen (15/19-inch Display Unit)

Screen colors can be set on the color setting screen.

Displaying the color setting screen

Procedure

1 Press function key .

2 Press vertical soft key [NEXT PAGE] several times to display vertical soft key [COLOR].

3 Press vertical soft key [COLOR] to display the color setting screen.

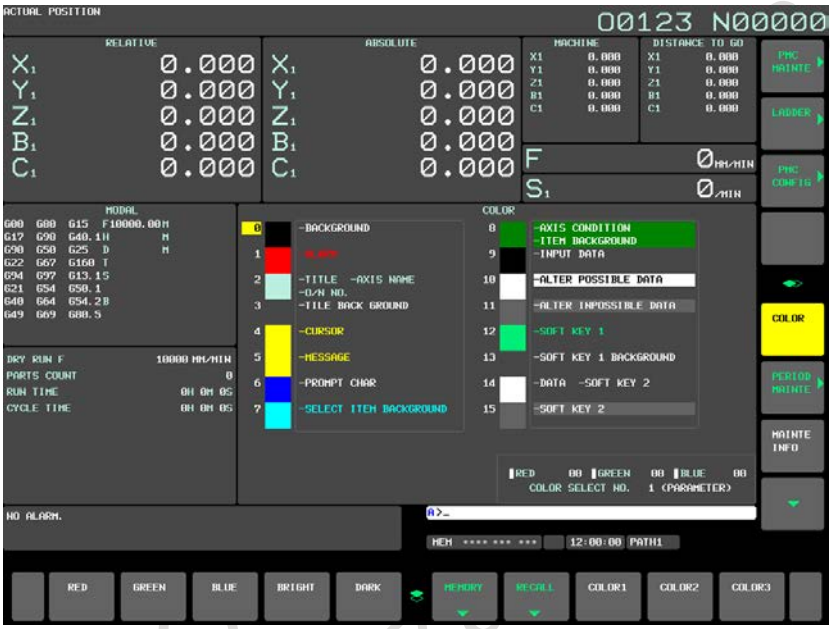


Fig. 12.4.20 (a) Color setting screen (15-inch display unit)

Procedure for operating the color setting screen

- Modifying the color (color palette values)

1 The following horizontal soft keys are displayed:

2 Move the cursor to a color number whose color palette values are to be modified.
The current color palette value set for each primary color is displayed.

3 Select a primary color whose setting is to be modified, with the corresponding horizontal soft key [RED], [GREEN], or [BLUE].
More than one primary color can be selected at the same time.
Each time horizontal soft key [RED], [GREEN], or [BLUE] is pressed, the operation soft key toggles between selection and deselection.

4 Select horizontal soft key [BRIGHT] or [DARK] to modify the brightness of the selected prime color(s).

- Storing the color (color palette values)

The set color palette values can be stored.

1 Press horizontal soft key [COLOR1], [COLOR2], or [COLOR3] to select a storage area.

FRYER MACHINE Pub ZERO INC

COLOR1 Standard color data parameters Nos. 6581 to 6595
COLOR2 Parameters Nos. 10421 to 10435
COLOR3 Parameters Nos. 10461 to 10475

2 Press horizontal soft key [MEMORY]. The horizontal soft key display switches to the following:

3 Pressing horizontal soft key [EXEC] stores the current color palette settings in the selected area.
Pressing horizontal soft key [CAN] or the leftmost key does not store the current color palette settings in the selected area.

- Calling the color (color palette values)

1 Press horizontal soft key [COLOR1], [COLOR2], or [COLOR3] to select a storage area where color palette values are stored.

2 Press horizontal soft key [RECALL]. The horizontal soft key display switches to the following:

3 Pressing horizontal soft key [EXEC] calls the color palette values from the selected area to enable the color to be modified. This operation is invalid if no color palette values are stored.
Pressing horizontal soft key [CAN] or the leftmost key does not call the color palette values from the selected area.

NOTE

1 Immediately after the power is turned on, the settings of COLOR1 (parameters) are used for display. If nothing is stored for COLOR1, the default colors are used for display.
2 Do not modify the color setting data parameters directly by MDI key input. When modifying the standard color data, be sure to perform a storage operation on the color setting screen.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.4.21 Machining Parameter Tuning (15/19-inch Display Unit)

12.4.21.1 Machining parameter tuning (AI contour)

In AI contour control, by setting a velocity-emphasized parameter set and precision-emphasized parameter set of machining condition selection function and setting the precision level matching a machining condition such as rough machining or finish machining on the precision level setting screen or by programming, the parameters suitable for the condition can be automatically calculated to perform machining.

On this screen, the parameter sets for emphasis on velocity (precision level 1) and emphasis on precision (precision level 10) can be set.

Set the following parameters:

- Acceleration rate of acceleration/deceleration before interpolation
- Acceleration change time (bell-shaped)
- Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control
- Allowable acceleration change value for each axis in acceleration change under jerk control in successive linear interpolation operations
- Ratio of the change time of the rate of change of acceleration in smooth bell-shaped acceleration/deceleration before interpolation
- Allowable acceleration rate
- Acceleration rate of acceleration/deceleration after interpolation
- Corner speed difference
- Maximum feedrate
- Items that can be set freely (2 items)

For details of each parameter, see the descriptions of AI contour control and jerk control (M Series).

By setting bit 0 (MPR) of parameter No. 13601 to 1, this screen can be hidden.

For the method of setting a precision level, see the description of the precision level selection screen in Subsection, “Precision Level Selection”.

FRYER MACHINE SYSTEMS INC
Rob Zamonsky

Procedure for machining parameter tuning

Procedure

1 Set the MDI mode.

2 Press function key [SYSTEM].

3 Press vertical soft key [NEXT PAGE] several times to display vertical soft key [MCHN TUNING]. The machining parameter tuning screen appears:

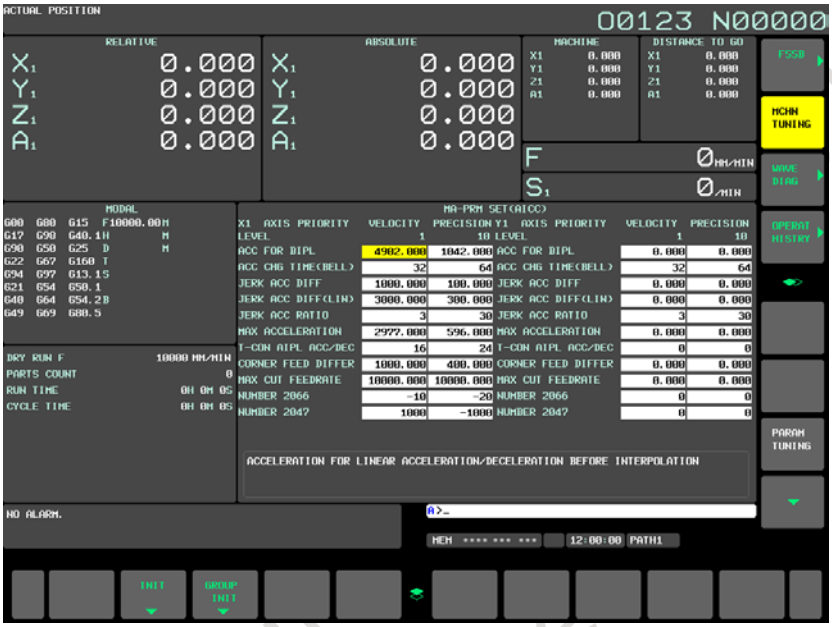


Fig. 12.4.21.1 (a) Machining parameter tuning screen (AI contour) (15-inch display unit)

4 Move the cursor to the position of a parameter to be set, as follows:
Press page key [PAGE ↑] or [PAGE ↓], and cursor keys [↑], [↓], [←] and /or [→] to move the cursor to the parameter.

5 Key in desired data then press the [INPUT] key on the MDI unit.

6 When data is input, a RMS value is found according to the precision level parameters. (The precision level can be changed on the precision level selection screen or parameter setting screen.) If a RMS value calculation fails, a warning (indicating that automatic setting failed) is displayed.

7 Repeat steps 4 and 5 until all machining parameters are set.

8 In addition to the setting method described above, a parameter setting method using horizontal soft keys is available. Pressing horizontal soft key [INIT] displays the standard value (recommended by FANUC) of the item selected by the cursor in the key input buffer. Pressing horizontal soft key [EXEC] initializes the item to the standard value. Pressing horizontal soft key [GROUP INIT] initializes all items of a group (emphasis on velocity or emphasis on precision) selected by the cursor to the standard values.

FRYER MACHINE INC Rob Zahra

The table below indicates the initial settings.

Table 12.4.21.1 (a) Initial settings

Setting item	AI contour control		Unit
	Emphasis on velocity (LV1)	Emphasis on precision (LV10)	
Acceleration rate of acceleration/deceleration before interpolation <ACC FOR BIPL>	4902.000	1042.000	mm/sec^2
Acceleration change time (bell-shaped) <ACC CHG TIME(BELL)>	32	64	msec
Allowable acceleration change value <JERK ACC DIFF>	0	0	mm/sec^2
Allowable acceleration change value in successive linear interpolation operations <JERK ACC DIFF(LIN)>	0	0	mm/sec^2
Ratio of the change time of the jerk control <JERK ACC RATIO>	0	0	%
Allowable acceleration rate <MAX AACELERATION>	2977.000	596.000	mm/sec^2
Time constant for acceleration/deceleration after interpolation <T-CON AIPL ACC/DEC>	24	24	msec
Corner speed difference <CORNER FEED DIFFER>	1000	400	mm/min
Maximum cutting speed <MAX CUT FEEDRATE>	10000	10000	mm/min

Explanation

- Look-ahead acceleration/deceleration before interpolation

Set an acceleration rate for a linear portion in look-ahead acceleration/deceleration before interpolation.
Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13610 (velocity-emphasized parameter)
Parameter No. 13611 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1660: Maximum allowable acceleration rate for each axis in acceleration/deceleration before interpolation

- Acceleration change time (bell-shaped)

Set a time constant for a bell-shaped portion in look-ahead acceleration/ deceleration before interpolation.
Unit of data: ms

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13612 (velocity-emphasized parameter)
Parameter No. 13613 (precision-emphasized parameter)

When bit 7 of parameter No. 13600 is set to 1, the parameter set on the machining parameter tuning screen is reflected in the following parameters for which the valid data range is extended to the range from 0 to 200:
Parameter No. 13662 (velocity-emphasized parameter)
Parameter No. 13663 (precision-emphasized parameter)
Moreover, the following parameter is also set according to the precision level:
Parameter No. 1772: Time constant for look-ahead bell-shaped acceleration/deceleration before interpolation of constant acceleration time type

> ⚠ **CAUTION**
> A set time constant is applied to all axes. So, a modification made to this item changes the settings for all axes.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Allowable acceleration change value in velocity control based on acceleration change under jerk control

Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

Set an allowable acceleration change value per ms for each axis in velocity control based on acceleration change under jerk control.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13614 (velocity-emphasized parameter)
Parameter No. 13615 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1788: Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control

NOTE
This setting item is displayed only when the jerk control function (M Series) is enabled.

- Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations

Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

Set an allowable acceleration change value per ms for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13616 (velocity-emphasized parameter)
Parameter No. 13617 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1789: Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations

⚠ CAUTION
1 For an axis with 0 set in this parameter, the parameters (allowable acceleration change value in velocity control based on acceleration change under jerk control: No. 13614, No.13615) are valid.
2 For an axis with 0 set in the parameter (allowable acceleration change value in velocity control based on acceleration change under jerk control: No. 13614, No.13615), velocity control based on acceleration change is disabled, so that this parameter has no effect.

NOTE
This setting item is displayed only when the jerk control function (M Series) is enabled.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Ratio of the change time of the jerk control in smooth bell-shaped acceleration/deceleration before interpolation

Unit of data: %

Set the ratio (in %) of the change time of jerk control to the change time of acceleration in smooth bell-shaped acceleration/deceleration before interpolation.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13618 (velocity-emphasized parameter)
Parameter No. 13619 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:

Parameter No. 1790: Ratio of the change time of the jerk control in smooth bell-shaped acceleration/deceleration before interpolation

NOTE
This setting item is displayed only when the jerk control function (M Series) is enabled.

- Allowable acceleration rate

Set an allowable acceleration rate in acceleration-based speed determination.
Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13620 (velocity-emphasized parameter)
Parameter No. 13621 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:

Parameter No. 1735: Allowable acceleration rate for each axis applicable to the deceleration function based on acceleration in circular interpolation

Parameter No. 1737: Allowable acceleration rate for each axis applicable to the deceleration function based on acceleration in AI contour control

⚠ CAUTION
When bit 0 (MCR) of parameter No. 13600 is set to 1, the deceleration function based on acceleration in circular interpolation is not set.

- Time constant for acceleration/deceleration after interpolation

Set a time constant for acceleration/deceleration after interpolation.
Unit of data: ms

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13622 (velocity-emphasized parameter)
Parameter No. 13623 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:

Parameter No. 1769: Time constant for acceleration/deceleration after cutting feed interpolation

FRYER MACHINE SYSTEMS INC
Bob Zahensky

- Corner speed difference

Set an allowable corner speed difference used for speed determination.
Unit of data: mm/sec, inch/sec, deg/sec (machine unit)
The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13624 (velocity-emphasized parameter)
Parameter No. 13625 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1783: Allowable speed difference for each axis in automatic corner deceleration based on speed difference

- Maximum cutting speed

Set a maximum cutting speed for each axis.
Unit of data: mm/sec, inch/sec, deg/sec (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13626 (velocity-emphasized parameter)
Parameter No. 13627 (precision-emphasized parameter)

Moreover, the following parameter is also set from the precision level:
Parameter No. 1432: Maximum cutting feedrate for each axis in the AI contour control mode

- Arbitrary items

Two arbitrary parameters can be registered. Each item can correspond to a CNC parameter or servo parameter. A parameter number corresponding to each item is to be specified with parameters.

As indicated below, set the parameters for corresponding parameter numbers, velocity-emphasized parameters (precision level 1), and precision-emphasized parameters (precision level 10).

Table 12.4.21.1 (b) Parameters related to arbitrary items

	Corresponding parameter number	Setting of velocity-emphasized (precision level 1) value	Setting of precision-emphasized (precision level 10) value
Arbitrary item 1	No.13628	No.13630	No.13632
Arbitrary item 2	No.13629	No.13631	No.13633

- Display
  Tuning target parameter numbers are displayed.

⚠ CAUTION
As arbitrary items, the numbers of the following parameters cannot be specified:
- Bit parameter
- Spindle parameters (Parameters Nos. 4000 to 4799)
- Real-type parameter
- Power-off parameter
- Nonexistent parameter

FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.4.22 Displaying Memory Data (15/19-inch Display Unit)

The contents of the CNC memory can be displayed starting at a specified address.

Displaying memory data

Procedure

1 Set bit 0 (MEM) of parameter No. 8950 to 1 to display the memory contents display screen.

2 Press function key [SYSTEM].

3 Press vertical soft key [MEMORY].
The following screen appears:

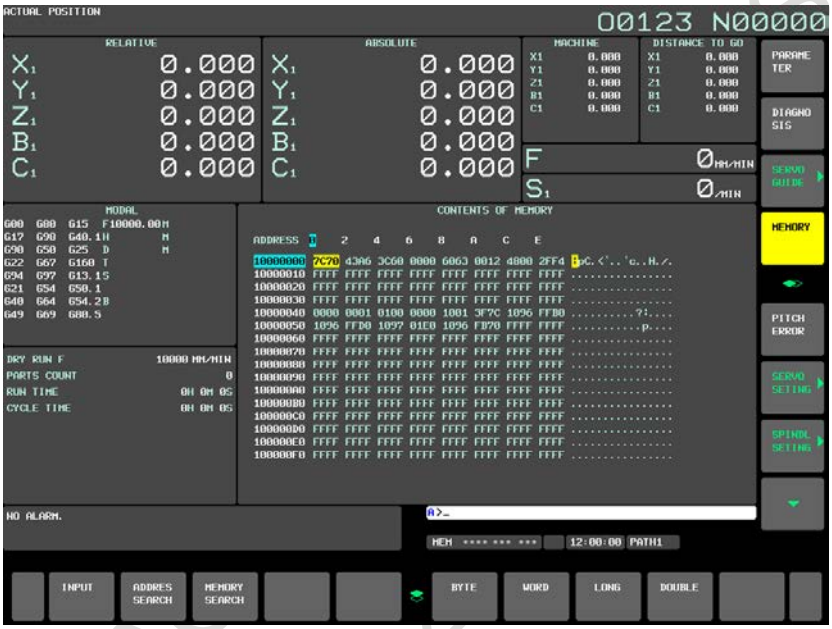


Fig. 12.4.22 (a) Memory contents display screen (15-inch display unit)

4 Key in a desired address (hexadecimal) then press horizontal soft key [ADDRES SEARCH]. Starting at the specified address, 256-byte data is displayed.
(Example: When you input 100000 then press [ADDRES SEARCH], data is displayed starting at 100000H.)

5 Display data can be switched by using page keys [PAGE] and [PAGE] and cursor keys [↑], [↓], [←], and/or [→].

6 By pressing horizontal soft key [BYTE], [WORD], [LONG], or [DOUBLE], select a data type to be displayed.

Explanation

A memory data display format can be selected from the following four options:

Byte display (1 byte in hexadecimal)
Word display (2 bytes in hexadecimal)
Long display (4 bytes in hexadecimal)
Double display (8 bytes in decimal: Double precision floating-point display)

One screen displays 256-byte memory data.

NOTE
1 When an address is input, "H" for representing "hexadecimal" need not be specified at the end of the address. If H is added, a warning is displayed to indicate that the format is invalid.
2 When word display is selected as the display format, an input address is rounded to a multiple of 2 bytes. When long display or double display is selected, an input address is rounded to a multiple of 4 bytes.

⚠ **WARNING**
1 If a memory address that must not be accessed in address search is input, a system alarm is issued.
When making an address search, check that the address is accessible and that the address is input correctly.
2 This function is designed for maintenance, and must not be used by general users.

12.4.23 Parameter Tuning Screen (15/19-inch Display Unit)

The parameter tuning screen is a screen for parameter setting and tuning designed to achieve the following:
1 The minimum required parameters that must be set when the machine is started up are collectively displayed to facilitate start-up of the machine.
2 The servo tuning screen, spindle tuning screen, and machining parameter tuning screen are displayed for smooth tuning.
The parameter tuning screen consists of a menu screen and several setting screens.

12.4.23.1 Displaying the menu screen and selecting a menu item (15/19-inch display unit)

The parameter tuning menu screen displays the following items:

[START UP]
- SYSTEM SETTING
- AXIS SETTING
- FSSB (SV AMP)
- FSSB (SP AMP)
- FSSB (AXIS)
- SERVO SETTING
- SPINDLE SETTING
- MISCELLANY

[TUNING]
- SERVO TUNING
- SPINDLE TUNING
- AICC TUNING

On the parameter tuning menu screen, one of the displayed items can be selected to display the corresponding screen. From each setting screen, you can return to the menu screen by performing a horizontal soft key operation.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

1 Some items may not be displayed, depending on the system configuration.
2 When bit 0 (SVS) of parameter No. 3111 is set to 0, "SERVO SETTING" and "SERVO TUNING" are not displayed. When bit 1 (SPS) of parameter No. 3111 is set to 0, "SPINDLE TUNING" is not displayed.

Displaying the menu screen and selecting a setting screen

Procedure

1 Set the MDI mode.
2 Switch the setting of "PARAMETER WRITE" to "ENABLED". For details, see the procedure for "PARAMETER WRITE" in Subsection, “Displaying and Setting Parameters (15/19-inch Display Unit)”.
3 Press function key [SYSTEM].
4 Press vertical soft key [NEXT PAGE] several times then press vertical soft key [PARAM TUNING]. The following screen appears:

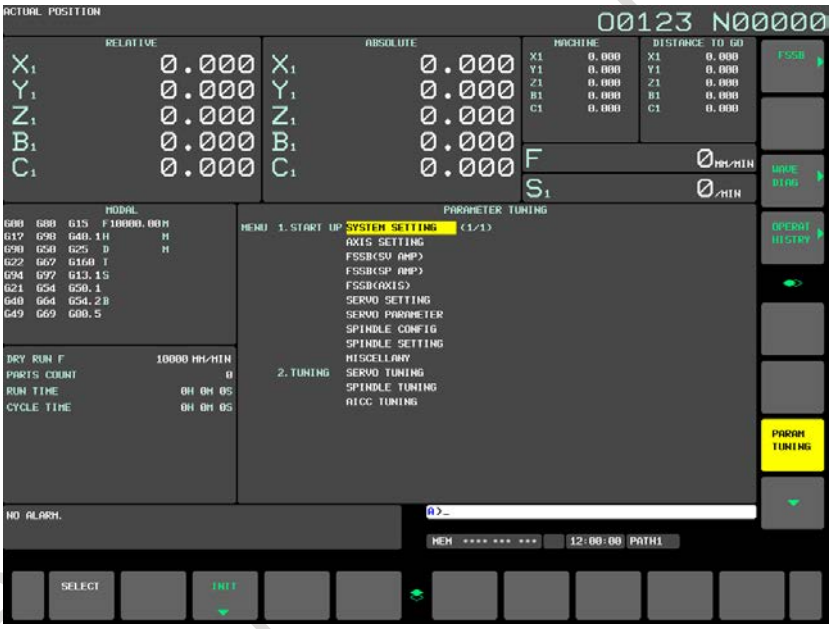


Fig. 12.4.23.1 (a) Parameter tuning menu screen (15-inch display unit)

5 Move the cursor to a desired item by pressing cursor key [↑] or [↓].
6 Press horizontal soft key [SELECT]. The screen display switches to the selected screen.

FPYER TEMS INC

Returning to the menu screen

Procedure

1 Press horizontal soft key [SELECT] on the parameter tuning menu screen described in Subsection III-12.4.27.1. The screen and soft keys shown below (Fig. 12.4.23.1 (b)) are displayed. (The screen below (Fig. 12.4.23.1 (b)) is displayed when "AXIS SETTING" is selected.)

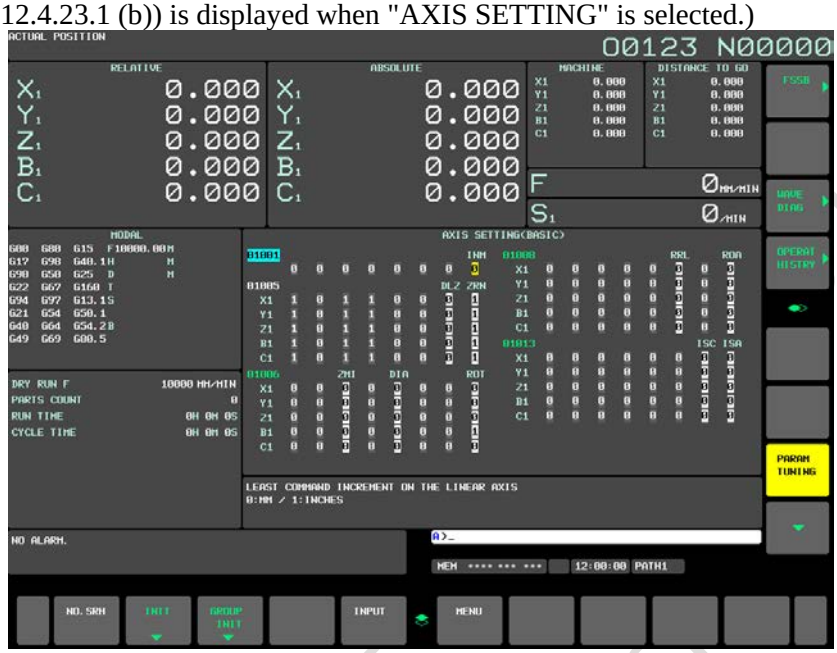


Fig. 12.4.23.1 (b) Axis setting screen (15-inch display unit)

2 Press horizontal soft key [MENU].
The screen display returns to the parameter tuning menu screen.

3 Upon completion of parameter setting, switch the setting of "PARAMETER WRITE" to "DISABLED".

> **NOTE**
> Part of the setting screens can be displayed also by using a vertical soft key for chapter selection. When these screens are displayed using the vertical soft key for chapter selection, it becomes impossible to return to the parameter tuning menu screen.

Explanation

- Items displayed with [START UP]

The items of [START UP] indicate the screens for setting the minimum required parameters for starting up the machine.

Table 12.4.23.1 (a) Items displayed with [START UP]

Display item	Description
SYSTEM SETTING	Screen for setting CNC parameters related to the entire system configuration
AXIS SETTING	Screen for setting CNC parameters related to axes, coordinates, feedrate, and acceleration/deceleration
FSSB (SV AMP)	FSSB servo amplifier setting screen
FSSB (SP AMP)	FSSB spindle amplifier setting screen
FSSB (AXIS)	FSSB axis setting screen
SERVO SETTING	Servo setting screen
SPINDLE SETTING	Screen for setting spindle-related parameters
MISCELLANY	Screen for setting parameters related to the allowable number of M code digits and whether to display the servo setting and spindle tuning screens

FRYER MACHINE INC Rob Zarensky

NOTE
Some items may not be displayed, depending on the system configuration.

- Items displayed with [TUNING]

The items of [TUNING] indicate the screens for servo, spindle, and high-speed high-precision machining tuning.

Table 12.4.23.1 (b) Items displayed with [TUNING]

Display item	Description
SERVO TUNING	Servo tuning screen
SPINDLE TUNING	Spindle tuning screen
AICC TUNING	Machining parameter tuning screen

NOTE
1 Some items may not be displayed, depending on the system configuration.
2 When bit 0 (SVS) of parameter No. 3111 is set to 0, "SERVO SETTING" and "SERVO TUNING" are not displayed. When bit 1 (SPS) of parameter No. 3111 is set to 0, "SPINDLE TUNING" is not displayed.

12.4.23.2 Parameter tuning screen (system setting) (15/19-inch display unit)

This screen enables the parameters related to the entire system configuration to be displayed and modified. The parameters can be initialized to the standard values (recommended by FANUC).

Display and setting

Procedure

1 Move the cursor to [SYSTEM SETTING] by pressing cursor key ↑ or ↓ on the parameter tuning menu screen described in Subsection, “Displaying the menu screen and selecting a menu item (15/19-inch display unit)”.

2 Press horizontal soft key [SELECT]. The screen display switches to the screen and horizontal soft keys shown below.

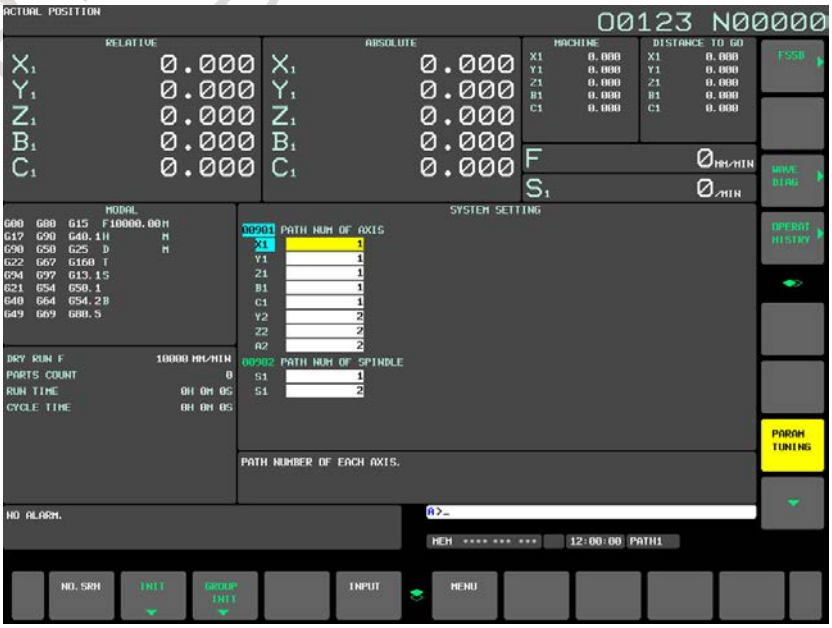


Fig. 12.4.23.2 (a) Parameter tuning screen (system setting) (15-inch display unit)

FRYER MACHINE SYSTEMS INC
Bob Zahenisky

3 Move the cursor to a parameter number to be set or displayed, according to one of the methods below.
- Input the desired parameter number, then press horizontal soft key [NO.SRH].
- Move the cursor to the desired parameter number by pressing page key [PAGE ↑] or [PAGE ↓], and cursor keys [↑], [↓], [←], and/or [→].

  A brief description of the parameter where the cursor is placed is provided at the bottom of the screen. However, no description is provided when the cursor is placed on multiple bits for bit parameters.

4 Input desired data then press the [INPUT] key on the MDI unit to set the parameter.

5 Press horizontal soft key [INIT]. The standard value (recommended by FANUC) for the item selected by the cursor is displayed in the key input buffer. Pressing horizontal soft key [EXEC] in this state initializes the item to the standard value.

6 Press horizontal soft key [GROUP INIT]. A message asking whether to set the group standard values is displayed on the screen. Pressing horizontal soft key [EXEC] in this state inputs all of the standard values of the group.

NOTE

1 If the cursor is placed on a parameter that has no standard value assigned, no standard value is input even when [INIT] is pressed.

2 When the cursor is placed on multiple bits for bit parameters, the multiple bits can be input simultaneously. When [INIT] is pressed in this state, the key input buffer displays the standard values for the bits where the cursor is placed. If a bit has no standard value assigned, "*" is displayed for the bit, and no value is input for the bit.

3 When [GROUP INIT] is pressed, those parameters that have no standard values assigned are not initialized.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

12.4.23.3 Parameter tuning screen (axis setting) (15/19-inch display unit)

This screen enables the CNC parameters related to axes, coordinates, feedrate, and acceleration/deceleration to be displayed and set. The parameters displayed can be divided into four groups:

(Basic) group : The parameters related to basic settings are displayed.
(Coordinate) group : The parameters related to coordinates are displayed.
(Feedrate) group : The parameters related to feedrate are displayed.
(Acceleration/deceleration) group : The parameters related to acceleration/deceleration are displayed.

The parameters can be initialized to the standard values (recommended by FANUC). For the display and setting procedure, see the description of the parameter tuning screen (axis setting) in Subsection, “Parameter tuning screen (system setting) (15/19-inch display unit)”.

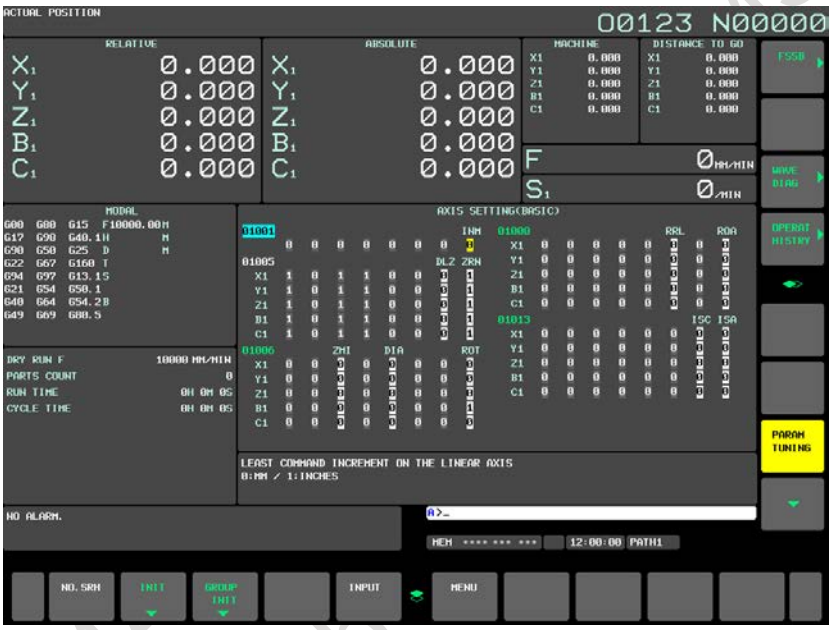


Fig. 12.4.23.3 (a) Parameter tuning screen (axis setting) (15-inch display unit)

FRYER MACHINE TOOLS INC

12.4.23.4 Displaying and setting the FSSB servo amplifier setting screen (15/19-inch display unit)

From the parameter tuning screen, the FSSB servo amplifier setting screen can be displayed. For details of the FSSB servo amplifier setting screen, see the description of the FSSB servo amplifier setting screen in Subsection, “FSSB Setting” in the Connection Manual (Function) (B-64693EN-1).

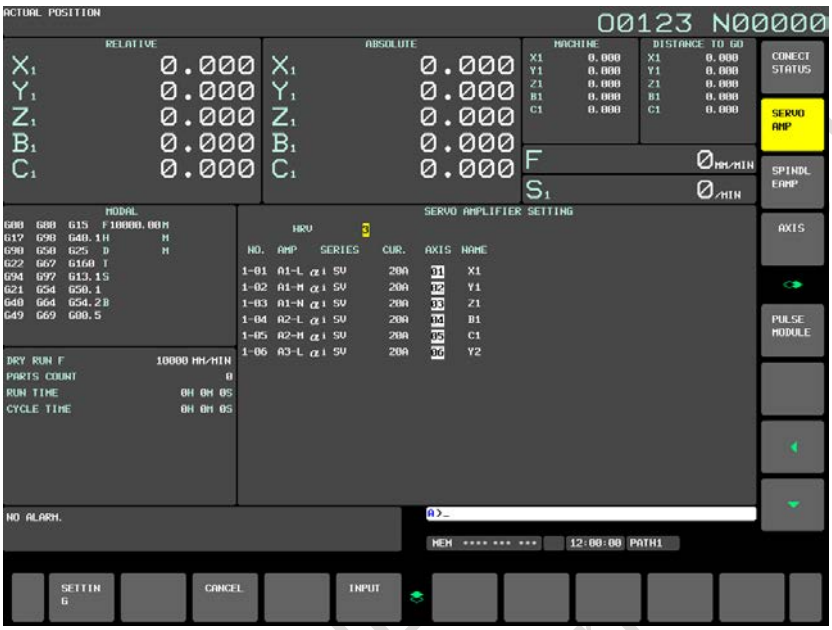


Fig. 12.4.23.4 (a) FSSB servo amplifier setting screen (15-inch display unit)

12.4.23.5 Displaying and setting the FSSB spindle amplifier setting screen (15/19-inch display unit)

From the parameter tuning screen, the FSSB spindle amplifier setting screen can be displayed. For details of the FSSB spindle amplifier setting screen, see the description of the FSSB spindle amplifier setting screen in Subsection, “FSSB Setting” in the Connection Manual (Function) (B-64693EN-1).

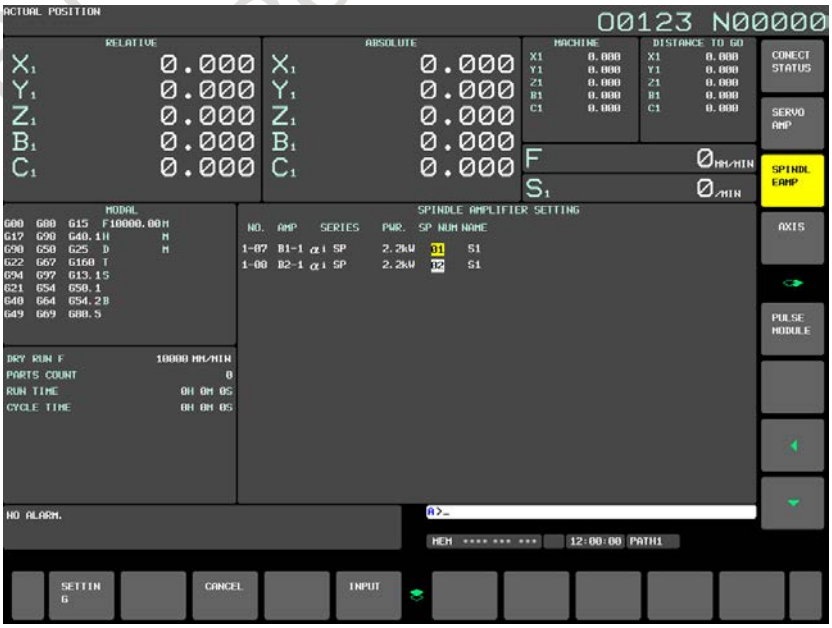


Fig. 12.4.23.5 (a) FSSB spindle amplifier setting screen (15-inch display unit)

FRYER MACHINE INC

12.4.23.6 Displaying and setting the FSSB axis setting screen (15/19-inch display unit)

From the parameter tuning screen, the FSSB axis setting screen can be displayed. For details of the FSSB axis setting screen, see the description of the FSSB axis setting screen in Subsection, “FSSB Setting” in the Connection Manual (Function) (B-64693EN-1).

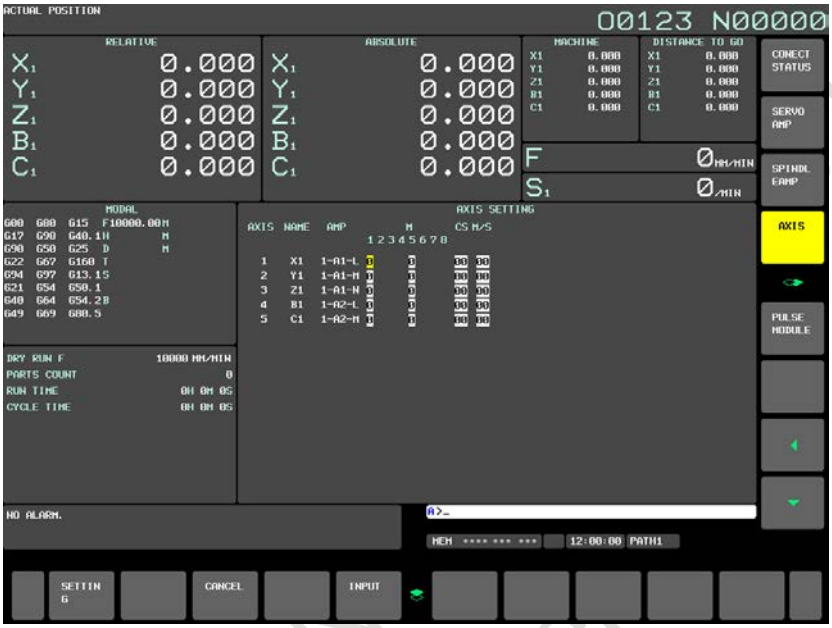


Fig. 12.4.23.6 (a) FSSB axis setting screen (15-inch display unit)

12.4.23.7 Displaying and setting the servo setting screen (15/19-inch display unit)

From the parameter tuning screen, the servo setting screen can be displayed. For details of the servo setting screen, see the Subsection, “Servo Parameters (15/19-inch Display Unit)”.

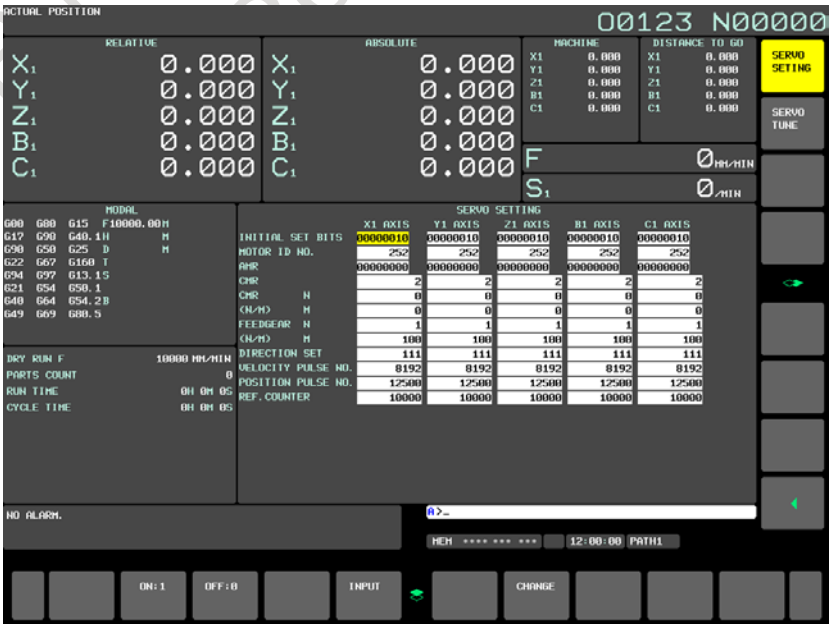


Fig. 12.4.23.7 (a) Servo setting screen (15-inch display unit)

12.4.23.8 Parameter tuning screen (spindle setting) (15/19-inch display unit)

The spindle-related parameters can be displayed and modified.
For the display and setting procedure, see the Subsection, “Parameter tuning screen (system setting) (15/19-inch display unit)”.

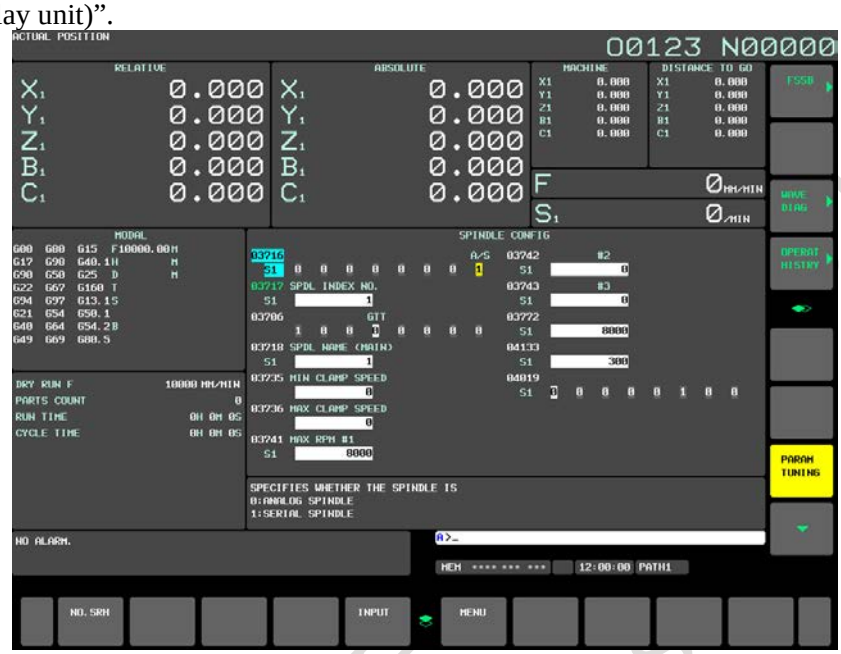


Fig. 12.4.23.8 (a) Parameter tuning screen (spindle setting) (15-inch display unit)

12.4.23.9 Parameter tuning screen (miscellaneous settings) (15/19-inch display unit)

The parameters related to the allowable number of M code digits and whether to display the servo setting and spindle tuning screens can be displayed and modified. Moreover, the parameters can be initialized to the standard values (recommended by FANUC).
For the display and setting procedure, see the Subsection, “Parameter tuning screen (system setting) (15/19-inch display unit)”.

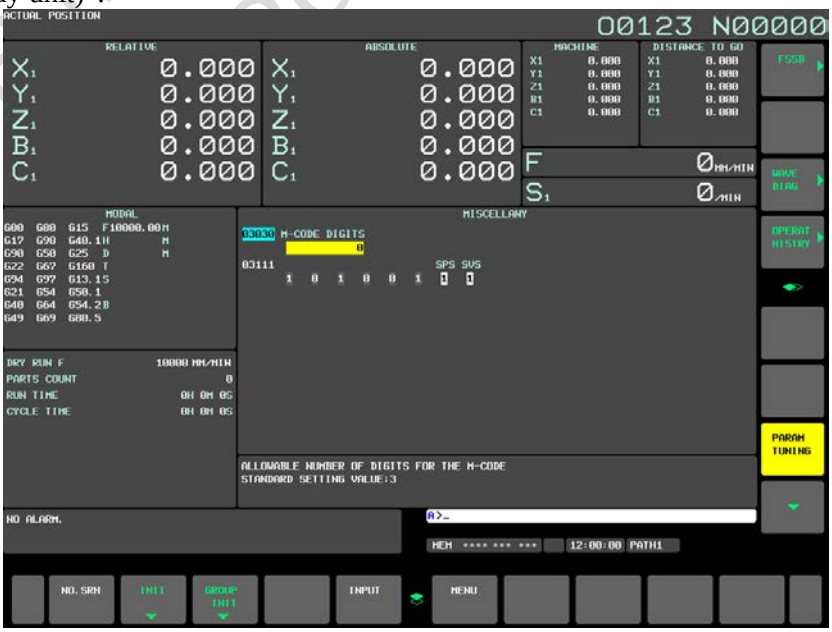


Fig. 12.4.23.9 (a) Parameter tuning screen (Miscellaneous setting) (15-inch display unit)

FRYER MACHINE INC

12.4.23.10 Displaying and setting the servo tuning screen (15/19-inch display unit)

From the parameter tuning screen, the servo tuning screen can be displayed. For details of the servo tuning screen, see the Subsection, “Servo Tuning (15/19-inch Display Unit)”.

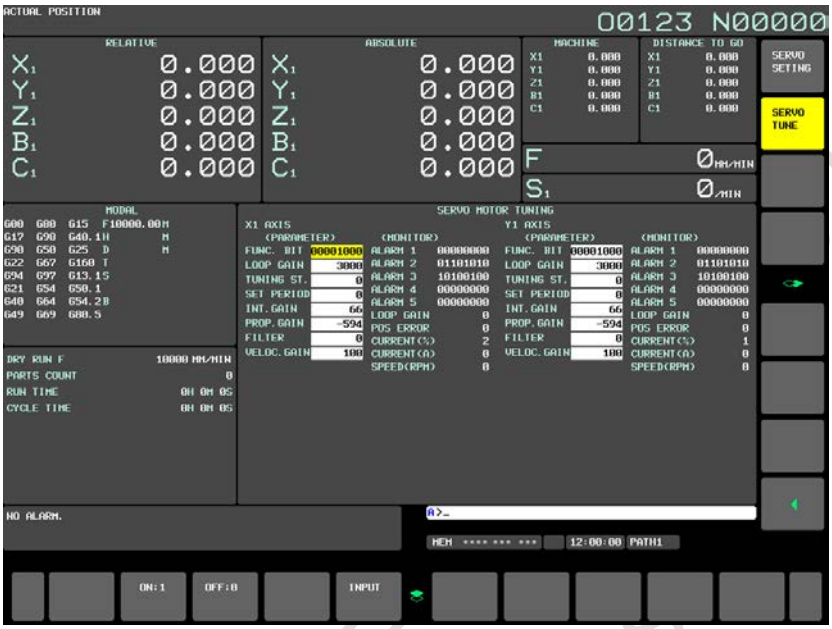


Fig. 12.4.23.10 (a) Servo tuning screen (15-inch display unit)

12.4.23.11 Displaying and setting the spindle tuning screen (15/19-inch display unit)

From the parameter tuning screen, the spindle tuning screen can be displayed. For details of the spindle tuning screen, see the Subsection, “Spindle Tuning (15/19-inch Display Unit)”.

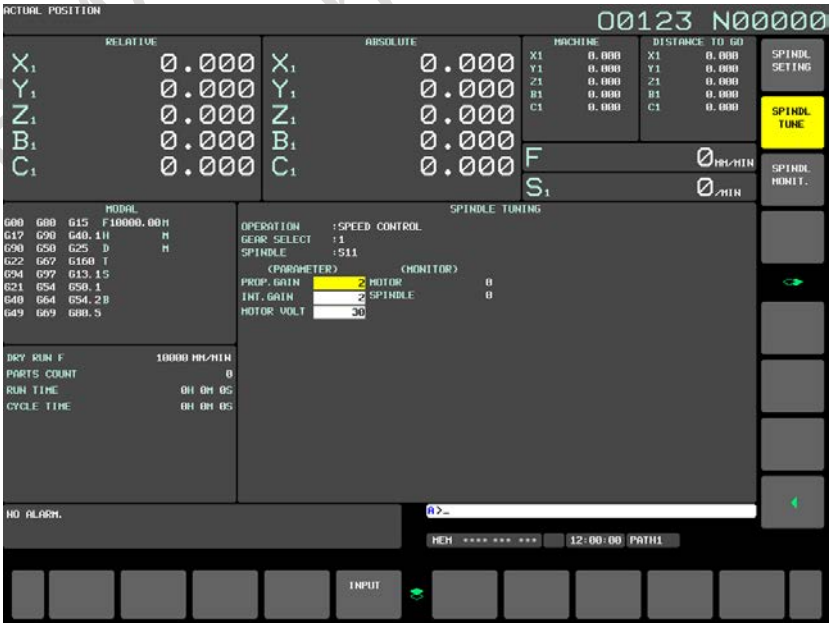


Fig. 12.4.23.11 (a) Spindle tuning screen (15-inch display unit)

FRYER MACHINE INC

12.4.23.12 Displaying and setting the machining parameter tuning screen (15/19-inch display unit)

From the parameter tuning screen, the machining parameter tuning screen can be displayed. For details of the machining parameter tuning screen, see the Subsection, “Machining Parameter Tuning (15/19-inch Display Unit)”.

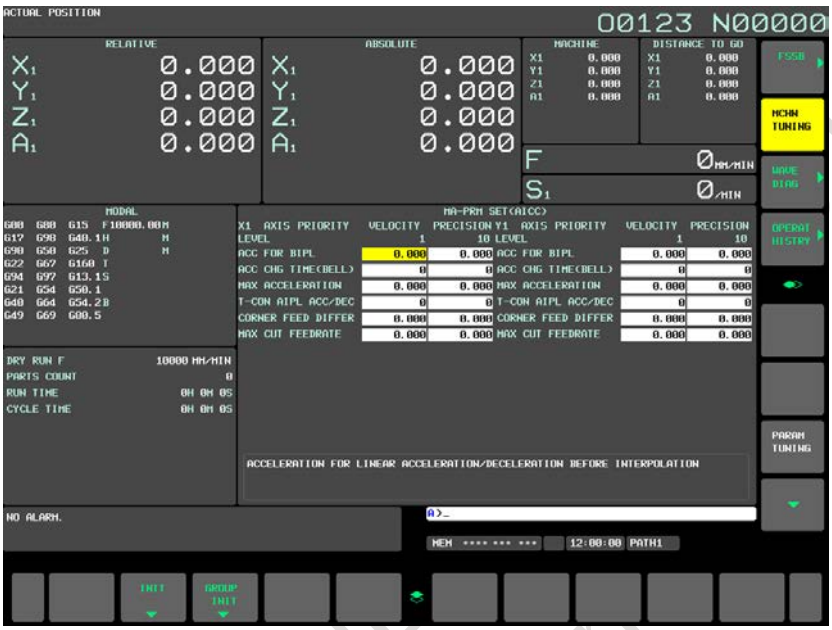


Fig. 12.4.23.12 (a) Machining parameter tuning screen (15-inch display unit)

Explanation

- Parameters displayed for parameter tuning

Table 12.4.23.12 (a) Parameters displayed for parameter tuning (1)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
SYSTEM SETTING	System setting	981		Sets the path of each axis.	
		982		Sets the path of each spindle.	
		983		Sets the T series/M series of each path. 0:T series / 1:M series	
		3021		Sets the G/F signal address of each axis.	*1
		3022		Sets the G/F signal address of each spindle.	*2
		3006#0	GDC	The deceleration signal for reference position return is: 0:X signal / 1:G signal	1
		3008#2	XSG	The signal address allocated to address X is: 0:Fixed / 1:Set by the parameter	1
		3013		Address allocated to the deceleration signal for reference position return	*3
		3014		Bit position allocated to the deceleration signal for reference position return	*4

*1 : When intra-path axis number ≤ 8, (path number - 1)*10+(intra-path axis number - 1)
When intra-path axis number ≥ 9, no standard value is available.
Example) When path 1 has 9 axes, and path 2 has 3 axes:
0,1,...,7,(none) for axes of path 1, 10,11,12 for axes of path 2

FRYER MACHINE INC Rob Zaher

*2 : When intra-path spindle number ≤ 4, (path number - 1)*10+(intra-path spindle number - 1)
When intra-path spindle number ≥ 5, no standard value is available.
Example) When path 1 has 5 spindles, and path 2 has 1 spindle:
0,1,...,4,(none) for spindles of path 1, 10 for spindle of path 2

*3 : When (path number ≤ 3) and (intra-path axis number ≤ 8)
Axes of path 1: 9
Axes of path 2: 7
Axes of path 3: 10
Other axes: No standard value is available.
Example) When path 1 has 9 axes, and path 2 has 3 axes:
9,9,9,9,9,9,9,9,(none) for axes of path 1, 7,7,7 for axes of path 2

*4 : When (path number ≤ 3) and (intra-path axis number ≤ 8)
(Intra-path axis number - 1)
Other axes: No standard value is available.
Example) When path 1 has 9 axes, and path 2 has 3 axes:
0,1,...,7,(none) for axes of path 1, 0,1,2 for axes of path 2

Table 12.4.23.12 (b) Parameters displayed for parameter tuning (2)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
SPINDLE SETTING	Spindle setting	3716#0	A/S	Sets the type of spindle motor: 0:Analaog / 1:Serial.	
		3717		Sets a motor number to be assigned to each spindle.	
		3706#4	GTT	Specifies a spindle gear selection method. 0:M type / 1:T type	
		3718		Sets a suffix to be added to spindle speed display on a screen such as the position display screen.	
		3735		Sets the minimum clamp speed of the spindle motor.	
		3736		Sets the maximum clamp speed of the spindle motor.	
		3741		Sets the maximum spindle speed for gear 1.	
		3742		Sets the maximum spindle speed for gear 2.	
		3743		Sets the maximum spindle speed for gear 3.	
		3744		Sets the maximum spindle speed for gear 4.	
		3772		Sets the maximum speed of the spindle. When 0 is set, the speed is not clamped.	
		4133		Sets the motor model code of the serial spindle. (This setting is not required for an analog spindle.)	
		4019#7	***	The parameters for the serial spindle are: 0:Not automatically set/1:Automatically set (This setting is not required for an analog spindle.)	

FRYER MACHINE SYSTEMS INC
Roman Zahensky

Table 12.4.23.12 (c) Parameters displayed for parameter tuning (3)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
AXIS SETTING	Basic	1001#0	INM	Least command increment on linear axes: 0:Metric (millimeter machine) / 1:Inch (inch machine)	
		1005#0	ZRNx	When an automatic operation (other than G28) is executed before reference position return: 0:Alarm is issued (PS0224) / 1:No alarm is issued.	0
		1005#1	DLZx	Reference position return without dogs: 0:Disabled / 1:Enabled	
		1006#0	ROTx	Setting of linear axes or rotary axis: 0:Linear axis / 1:Rotation axis	
		1006#3	DIAx	Setting of the amount of travel: 0:Radius specification / 1: Diameter specification	
		1006#5	ZMIx	Reference position return direction: 0:Plus direction / 1:Minus direction	
		1008#0	ROAx	Rotation axis roll-over function: 0:Disabled / 1:Enabled	1
		1008#2	RRLx	With the amount of travel per revolution, relative coordinates are: 0:Not rounded / 1:Rounded	1
		1013#0	ISAx	Sets the least input increment and least command increment: 0:IS-B / 1:IS-A	
		1013#1	ISCx	Sets the least input increment and least command increment: 0:IS-B / 1:IS-C	
		1020		Program axis name	*1
		1022		Sets each axis in the basic coordinate system.	*2
		1023		Servo axis number	
		1815#1	OPTx	A separate pulse coder is: 0:Not used / 1:Used	
		1815#4	APZx	The correspondence between machine positions and absolute-position detector positions is: 0:Not established / 1:Established	
		1815#5	APCx	The position detector used is: 0:Other than an absolute-position detector / 1:Absolute-position detector	
		1825		Servo loop gain	
		1826		Effective area	
		1828		Positional deviation limit during travel	
		1829		Positional deviation limit during stop	500

*1 : For the M series: 88(X),89(Y),90(Z) sequentially starting with the first axis (no standard value for the fourth axis and up)
For the T series: 88(X),90(Z) (no standard value for the third axis and up)

*2 : For the M series: 1,2,3 sequentially starting with the first axis (no standard value for the fourth axis and up)
For the T series: 1,3 (no standard value for the third axis and up)

FRYER MACHINE SYSTEMS INC
Roman Zarensky

Table 12.4.23.12 (d) Parameters displayed for parameter tuning (4)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
AXIS SETTING	Coordinate	1240		Machine coordinate of the first reference position	
		1241		Machine coordinate of the second reference position	
		1260		Amount of travel per revolution of a rotary axis	360.000
		1320		Coordinate of the boundary of stored stroke check 1 in the positive direction	
		1321		Coordinate of the boundary of stored stroke check 1 in the negative direction	
	Feedrate	1401#6	RDR	For a rapid traverse command, dry run is: 0:Disabled / 1:Enabled	0
		1410		Dry run feedrate	
		1420		Rapid traverse rate	
		1421		Rapid traverse override F0 rate	
		1423		Jog feedrate	
		1424		Manual rapid traverse rate	
		1425		FL feedrate for reference position return	
		1428		Reference position return feedrate	
		1430		Maximum cutting feedrate	
	Accele-ra tion/ decele-ration	1610#0	CTLx	Acceleration/deceleration in cutting feed or dry run during cutting feed: 0: Exponential acceleration/deceleration is applied / 1: Linear acceleration/deceleration after interpolation is applied	
		1610#4	JGLx	Acceleration/deceleration for jog feed is: 0: Exponential acceleration/deceleration / 1: Same as acceleration/deceleration for cutting feed (The settings of bit 1 (CTBx) and bit 0 (CTLx) of parameter No. 1610 are followed.)	
		1620		Time constant for linear acceleration/deceleration for rapid traverse	
		1622		Time constant for acceleration/deceleration for cutting feed	
		1623		FL feedrate for acceleration/deceleration after interpolation for cutting feed	
		1624		Time constant for acceleration/deceleration for jog feed	
		1625		FL feedrate for exponential acceleration/deceleration for jog feed	
MISCELLANY	MISC	3030		Allowable number of digits of an M code	3
		3111#0	SVS	The servo setting screen is: 0:Not displayed / 1:Displayed	1
		3111#1	SPS	The spindle setting screen is: 0:Not displayed / 1:Displayed	1

FRYER MACHINE SYSTEMS INC
Rob Zakrevsky

12.4.24 Periodic Maintenance Screen (15/19-inch Display Unit)

Periodic maintenance screens are used for managing consumables (such as the backlight of a LCD unit and backup batteries). By setting the name of a consumable, its life time, and the method for counting down the life time, the life of the consumable can be counted down in the appropriate method set for the consumable, and the remaining life time can be displayed.
With these screens, the user can easily manage consumables that require periodic replacement.
The total number of item that can be managed is 10 in all path.

Explanation

There are four periodic maintenance screens: the status screen, the setting screen, the machine menu screen, and the NC menu screen.

Status screen : Item names, remaining times, and count statuses are displayed, and item names are set.
Setting screen : Life times, remaining times, count types (count-down method), and path number (in case of multi-path system) are set.
Machine menu screen : The names of consumables in the machine can be registered.
NC menu screen : The names of consumables in the NC are already registered.

Using periodic maintenance screens

(1) Referencing a periodic maintenance screen
Display the status screen. For the meanings of items on the status screen, see Item, “**Status screen**”.

(2) Adding the name of a new consumable item to a periodic maintenance screen or editing an existing consumable item on the screen
The name of a consumable item can be added or edited on the machine menu screen. For details, see Item, “**Machine menu screen**”.

(3) Adding or editing a life time, remaining time, and a method of counting down the remaining time for a consumable on a periodic maintenance screen
A life time and remaining time can be added or edited on the setting screen. For details, see Item, “**Setting screen**”.

(4) Newly displaying the item name and remaining time of a consumable on a periodic maintenance screen.
1 Setting an item name
Select the item name of a consumable to be displayed from the machine menu screen or NC menu screen, or enter the name using the MDI keys. For the procedure, see Item, “**Item name**” in “**Status screen**”.
2 Setting a life time, remaining time, and count type
Select the life time, remaining time, and count type of a consumable to be displayed from the setting screen. For the procedure, see Item, “**Remaining time**” in “**Status screen**”.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Procedure for displaying a periodic maintenance screen

1 Press function key [SYSTEM].
2 Press vertical soft key [NEXT PAGE] several times to display [PERIOD MAINTE].
3 Press vertical soft key [PERIOD MAINTE] to display the periodic maintenance screen.

Status screen

When vertical soft key [STATUS] is pressed, the status screen is displayed. The status screen shows the item names, count statuses, and remaining times of managed consumables.

Fig. 12.4.24 (a) Status screen (15-inch display unit)

- Item name

As the item name, set the name of a consumable to be managed by periodic maintenance. To set an item name, select a name from the machine menu screen or NC menu screen, or directly enter the name using the MDI keys.

Setting an item name from a menu screen

1 On the status screen, move the cursor to the name of the item to be set, and press horizontal soft key [ENTRY].
2 Press vertical soft key [MACHINE] or [NC] to display the machine menu screen or NC menu screen.
3 Move the cursor to an item name registered on the menu screen, and press horizontal soft key [SELECT] then [EXEC] successively.
4 The screen display returns to the status screen, and the item name selected on the menu screen is added to the status screen.

Initially, there is no item name set on the machine menu screen, so item names must be registered in advance. For the registration method, see the description of the procedure for registering item names on the machine menu screen.

Setting an item name using the MDI keys

1 Type the alphanumeric characters to be input, then press horizontal soft key [INPUT].
2 The entered item name is registered on the status screen.

When horizontal soft key [+INPUT] is pressed instead of horizontal soft key [INPUT], the entered characters can be added to an existing item name.

FRYER MACHINE INC Rob Zak

When typing 2-byte characters, type "*" before and after the character codes. The 2-byte character codes must conform to FANUC codes. (See Appendix, "FANUC 2-BYTE CHARACTER CODE TABLE".)
An item name to be registered must be up to 24 characters long if it consists of alphanumeric characters only; or it must be up to 12 characters long if it consists of 2-byte characters only.
Example: To register "LCD バックライト", enter the following:
>LCD*110E10F410CC114010B610FE*_

NOTE
1 An asterisk "*" is used as a control code, so it cannot be used in item names. In addition, characters "[", "]", "(", and ")" cannot be used in item names.
2 When an item name consisting of both alphanumeric characters and 2-byte characters is registered, the warning "DATA OUT OF RANGE" may be output.

Deleting an item name
To delete a registered item name, move the cursor to the item name, press horizontal soft key [ERASE], then press horizontal soft key [EXEC].
When an item name is deleted, its life time, remaining time, and count type are deleted at the same time.

- Remaining time

As the remaining time, the period of time left until the time for replacement is reached by count down operation is displayed.
If the percentage of the remaining time to the life time has reached to the value (%) specified by parameter No. 8911, or smaller, the remaining time is displayed in red.
Even after the life time has expired, count down operation continues.

Setting the remaining time
1 On the status screen, place the cursor on an item for which the remaining time is to be set (the item name must have been set in advance).
2 Press horizontal soft key [CHANGE].
3 The screen display changes to the setting screen.
4 Set the life time, remaining time, and count type. For the setting method and other information, see Item, "**Setting screen**".

NOTE
On the status screen, the remaining time and life time cannot be set.
These items must be set on the setting screen.

- Count status

The count status is indicated on the left side of the item number as follows:

Indication	Count status
Blank	Counting stopped
@	Counting in progress
*	Life expired

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Setting screen

On the setting screen, the life time, remaining time, and count type of a managed consumable are set.

Fig. 12.4.24 (b) Setting screen (15-inch display unit)

Display procedure

1 Press horizontal soft key [CHANGE].

- Life time

Set the life time of a consumable.

Move the cursor to an existing item, type a life time, then press horizontal soft key [INPUT] (or the INPUT key). The life time is then set, and the same value is set also as the remaining time.
At this time, "------" is indicated in the count type field.
When horizontal soft key [+INPUT] is pressed, the entered value can be added to the life time already set.
The same value as the added value is added also to the remaining time.
A value ranging from 0 to 65535 (in hours) can be set.

NOTE

1 If a setting operation is attempted when the item name is not registered, the warning "EDIT REJECTED" is issued.
2 If a value exceeding the valid range is entered, the warning "DATA IS OUT OF RANGE" is issued.
3 If horizontal soft keys [ERASE] and [TYPE] is pressed, the warning "EDIT REJECTED" is issued.

- Remaining time

The period of time left until the time for replacement is reached by count down operation is indicated.
If the percentage of the remaining time to the life time has reached the value (%) specified by parameter No. 8911, or smaller, the remaining time is displayed in red.
Even after the life time has expired, count down operation continues.
Move the cursor to the remaining time of a target registered number, type a remaining time, then press horizontal soft key [INPUT] (or the INPUT key). The remaining time is then set.

When horizontal soft key [+INPUT] is pressed, the entered value can be added to the remaining time already set.

FRYER MACHINE SALES INC
Rob Zahensky

A value ranging from 0 to (life time) can be set.
When horizontal soft key [ERASE] then horizontal soft key [EXEC] are pressed, the same value as the life time is set.

NOTE
1 If a setting operation is attempted when the item name or life time is not registered, the warning “EDIT REJECTED” is issued.
2 When a value exceeding the valid range is entered, the warning “DATA IS OUT OF RANGE” is issued.
3 If horizontal soft key [TYPE] is pressed, the warning “EDIT REJECTED” is issued.

- Count type

As the count type, select the way of counting.
Place the cursor on the count type of a target registration number, then press horizontal soft key [TYPE].
Count types are displayed as horizontal soft keys as shown below. Select one of these horizontal soft keys, then press horizontal soft key [EXEC].

Horizontal soft key	Meaning	Indication
[NO CNT]	Counting is not performed (stopped)	— — — — — —
[ALL]	Counting is performed at all times	All times
[POWER ON]	Counting is performed when power is on.	When power is on
[RUN]	Counting is performed when operation is in progress.	When operation is in progress
[CUT]	Counting is performed when cutting is being performed.	When cutting is being performed

NOTE
1 If a setting operation is attempted when the item name or life time is not registered, the warning “EDIT REJECTED” is issued.
2 Horizontal soft keys [INPUT] and [+INPUT] have no effect.
3 When counting is performed at all times, a 24-hour error is generated in leap year.
4 If horizontal soft key [ERASE] is pressed, the warning “EDIT REJECTED” is issued.

- Path number

When [RUN] or [CUT] is set to the count type in multi path, path number can be set.
Make the cursor to the count type of a target registration number, then input path number and press soft key [INPUT].
The setting range is 0-(max path number).
When the specified path is operating or cutting, counting is performed.
If 0 is set, when either of path is operating or cutting, counting is performed.

NOTE
1 If [NO CNT], [ALL] or [POWER ON] is set to count type, “--” is displayed. In this case, if the setting is operated, the warning “EDIT REJECTED” is issued.
2 If the setting is beyond the valid data range, the warning “DATA IS OUT OF RANGE” is issued.
3 If soft key [TYPE] is pressed, the warning “EDIT REJECTED” is issued.
4 If soft key [ERASE] is pressed, the warning “EDIT REJECTED” is issued. The soft key [+INPUT] is invalid.

FRYER MACHINE SYSTEMS INC
Rob Zaborensky

Registration from a program

The item name, life time, remaining time, count type and path number can be registered in the status screen and the setting screen by executing the program of the following format.

Format

G10 L60 Px [n] Aa Rr Qpt

X : Registration number
n : Item name, format:
[alphanumeric-characters*2-byte-characters*alphanumeric-characters]
a : life time
r : remaining time
p : path number
t : count type
The meaning of the value of the count type is as follow.
0 : NO COUNT
1 : ALL TIMES
2 : POWER ON
3 : RUNNING
4 : CUTTING
The range of the path number is 0 – (max path number).
EX.)
Q24 : The path number is 2, and the count type is “CUTTING”
Q103 : The path number is 10, and the count type is “RUNNING”

Machine menu screen

On the machine menu screen, the names of consumables of the machine are registered. From this screen, item names can be added to the status screen. For the method of addition to the status screen, see the description of the status screen.

Fig. 12.4.24 (c) Machine menu screen (15-inch display unit)

FRYER MACHINE SYSTEMS INC
Zaharensky

- Displaying the screen

1 When the status screen is displayed, press vertical soft key [MACHINE].

On the machine menu screen, item names can be registered using one of the following two methods:

- Registration from a program
- Registration using the MDI keys

- Registration from a program

By executing a program having the following format, an item name can be registered in the menu of the machine:

Format

G10 L61 Px [n]

X Registration number
n Item name, format:
[alphanumeric-characters*2-byte-characters*alphanumeric-characters]

- Registration using the MDI keys

An item name can be registered on the menu screen of the machine by typing the item name in the format shown below on the machine menu screen, then pressing horizontal soft key [INPUT] (or the INPUT key).

When horizontal soft key [+INPUT] is pressed, the typed characters can be added to an already registered item name.

Format

Alphanumeric-characters*2-byte-characters*alphanumeric-characters

Two-byte codes must conform to FANUC codes. (See Appendix, "FANUC 2-BYTE CHARACTER CODE TABLE".)
When typing 2-byte characters, type an asterisk "*" before and after the character codes. An item name to be registered must be up to 24 characters long if it consists of alphanumeric characters only; or it must be up to 12 characters long if it consists of 2-byte characters only.

Example:

To register "LCD バックライト", enter the following:
>LCD*110E10F410CC114010B610FE*_

NOTE

1 An asterisk "*" is used as a control code, so it cannot be used in item names. In addition, characters "[", "]", "(", and ")" cannot be used in item names.
2 When an item name consisting of both alphanumeric characters and 2-byte characters is registered, the warning “DATA IS OUT OF RANGE” may be output.
3 When a blank item name is selected on the machine screen, the warning “EDIT REJECTED” is issued.

To delete a registered item name, move the cursor to the item name, press horizontal soft key [ERASE], then press horizontal soft key [EXEC].

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NC menu screen

From this screen, an item name can be registered on the status screen. For the method of registration to the status screen, see the description of the status screen.

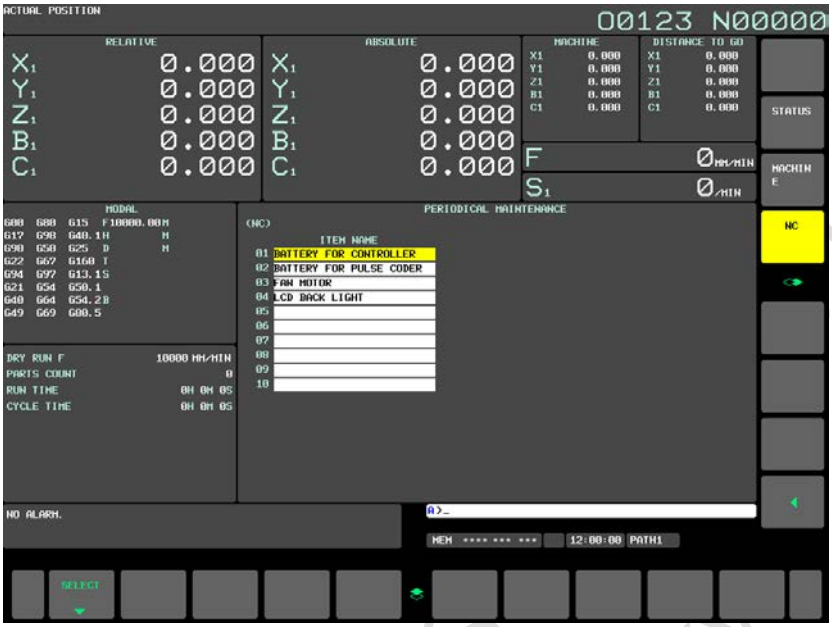


Fig. 12.4.24 (d) NC menu screen (15-inch display unit)

- Displaying the screen

1 When the status screen is displayed, press vertical soft key [NC].

> **NOTE**
> On the NC screen, the registration, deletion, and I/O of item names cannot be performed.
> When a blank item name is selected, a blank is set.

12.4.25 System Configuration Screen (15/19-inch Display Unit)

The system configuration screen provides information about the types of installed hardware and software.

Procedure for displaying the screen

Procedure

1 Press the function key [SYSTEM], then press vertical soft key [NEXT PAGE] several times until vertical soft key [SYSTEM] is displayed.

2 Press vertical soft key [SYSTEM]. The system configuration screen is displayed.

There are two types of system configuration screens: the hardware configuration screen and the software configuration screen. The screen display can switch between these screens by using [PAGE ↑] and [PAGE ↓].

When vertical soft key [SERVO INFO] or [SPINDLE INFO] is pressed, information about the connected servo system or spindles is displayed.

FRYER MACHINE SYSTEMS INC Rob Zahensky

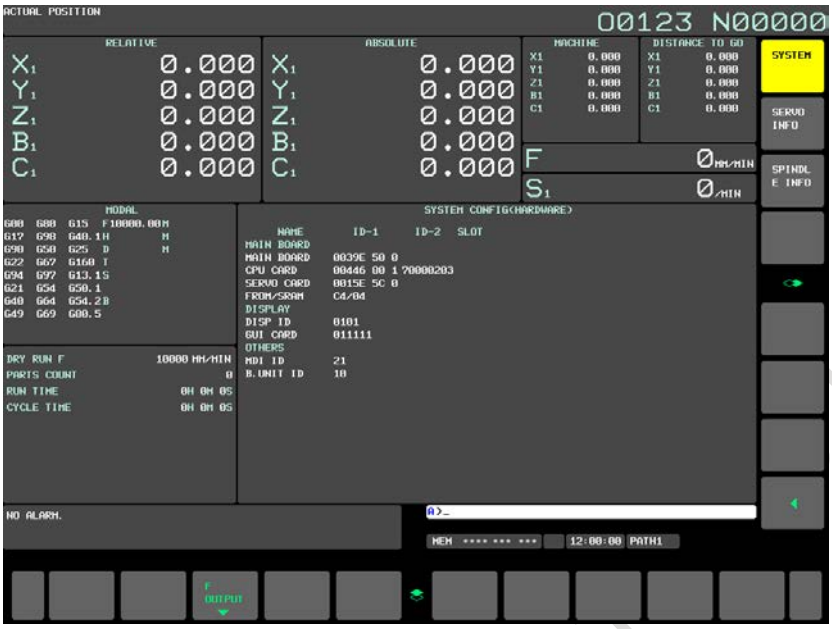


Fig. 12.4.25 (a) System configuration screen (15-inch display unit)

Hardware configuration screen

This screen shows the names and IDs of the hardware used by the NC.

Fig. 12.4.25 (b) Hardware configuration screen

Software configuration screen

This screen shows the names and series/editions of the software used by the NC.

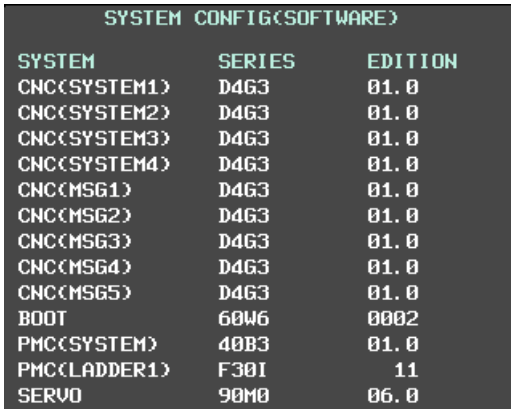


Fig. 12.4.25 (c) Software configuration screen

FRYER MA SYS INC
FRYSKY

Servo information screen

When a servo system is connected to the NC, ID information of the connected servo devices (servo motors and servo amplifiers) can be displayed on the NC.

Displaying the screen

1 When the system configuration screen is displayed, press vertical soft key [SERVO INFO].
2 The servo information screen is displayed.

```
          SERVO INFORMATION
   X1  AXIS
SERVO MOTOR SPEC A06B-2212-B002
SERVO MOTOR S/N  C151F3475
PULSECODER SPEC. A860-2051-T321
PULSECODER S/N   14121115
SV SPEC.         A06B-6240-H305
SV S/N           V14Z43833
SV SOFT EDITION  9H0015.0
PS SPEC.         A06B-6200-H008
PS S/N           V14Z39904
PS SOFT EDITION  9G0019.0
```

Fig. 12.4.25 (d) Servo information screen

Spindle information screen

When a spindle system is connected to the NC, the ID information of the connected spindle devices (spindle motors and spindle amplifiers) can be displayed on the NC.

Displaying the screen

1 When the system configuration screen is displayed, press vertical soft key [SPINDLE INFO].
2 The spindle information screen is displayed.

```
          SPINDLE INFORMATION
   S1
MAINSP MOTOR SPC A06B-1401-B105
MAINSP MOTOR S/N V00000001
SP SPEC.         A06B-6220-H002#H600
SP S/N           V14Y02875
PS SPEC.         A06B-6200-H008
PS S/N           V14Z39904
PS SOFT EDITION  9G0019.0
```

Fig. 12.4.25 (e) Spindle information screen

FRYER MACHINE SYSTEMS INC
Zahensky

12.4.26 Power Consumption Monitoring Screen (15/19-inch Display Unit)

The electric power data of servo axis and spindle axis consumption and regeneration can be displayed.

Display of power consumption monitoring screen

1 Press function key [SYSTEM].

2 Press continuous menu key [▷] several times until soft key [POWER MONIT] appears.

3 Press soft key [POWER MONIT].

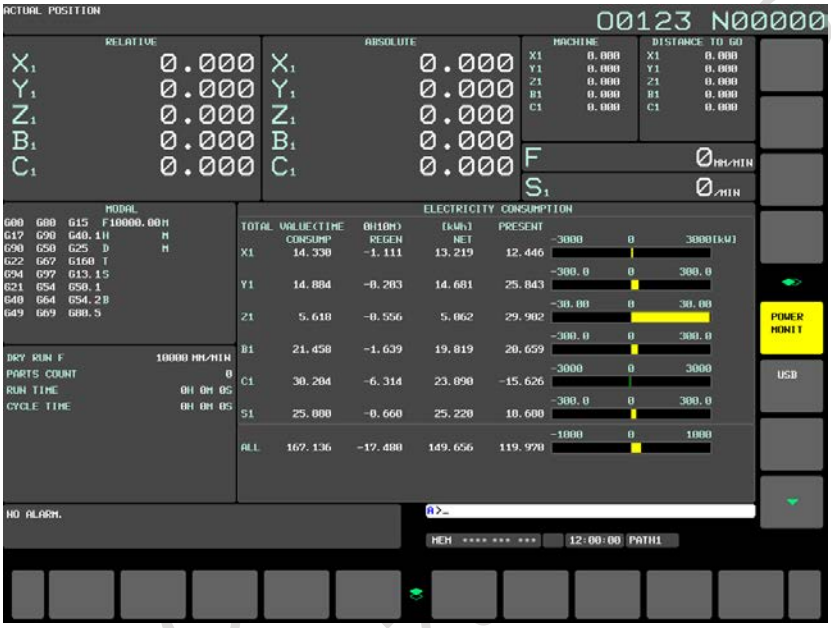


Fig. 12.4.26 (a) Power consumption monitoring screen for 15” display unit

Operation of power consumption monitoring screen

Switch of display axis

When information on all axes is not displayed, the page is switched with the MDI key [↑ PAGE] or [PAGE ↓].

Clear cumulative power consumption value

1 Press horizontal soft key [TOTAL CLEAR].
Horizontal Soft key [EXEC] and [CAN] appear.

2 Press horizontal soft key [EXEC].
Cumulative power consumption and cumulative time start from 0.
To quit clear operation, press horizontal soft key [CAN].

FRYER MACHINES INC Rob Z

Explanation

TIME

Integrating time of power consumption is displayed.

Axis name

Axis name of servo and spindle is displayed. "ALL" means the total of all servo and spindle axes.

CONSUMP

Integral power consumption is displayed.

REGEN

Integral power regeneration is displayed.

NET

Integral net amount of power consumption is displayed.
The net amount of power consumption = power consumption – power regeneration.

PRESENT

Present net power consumption is displayed. A negative value is displayed about the net power consumption while the power is regenerating like the axis is decelerating, etc.

Bar-graph of PRESENT

Present net power consumption is displayed as a bar-graph.
The scale of bar-graph can be specified in the parameter (No.11392 in case of the servo, No.11393 in case of the spindle).
The scale of the entire power consumption bar-graph can be specified in the parameter No.11371.
The bar expands from the center to right while consuming the electric power. The bar expands from the center to left while regenerating the electric power.

Unit of data

When the cumulative power consumption value can no longer be displayed in kWh, unit of data is switched to MWh automatically

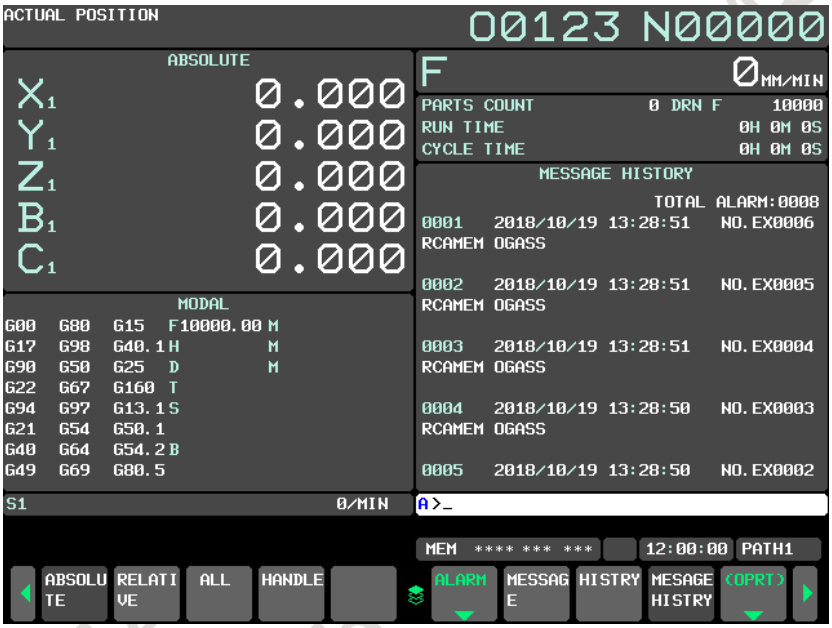
FRYER MACHINE SYSTEMS INC
Rob Zahensky

12.5 SCREENS DISPLAYED BY FUNCTION KEY

By pressing the function key , data such as alarms, alarm history, external operator message, and external operator message history data can be displayed.
For details of alarms and alarm history, see Sections, “ALARM DISPLAY” and “ALARM HISTORY DISPLAY”. Refer to the machine tool builder's manual for the external message display.

12.5.1 External Operator Message History

External operator message can be stored as history. And, stored history can be seen on the external operator message history screen.



Screen display

To display the external operator message history screen, set bit 2 (OMH) of parameter No. 3112 to 1.

Procedure

1 Press function key .
2 Press soft key [MESSAGE HISTRY]. The external operator message history screen is displayed.
3 The screen display can be changed to the previous page and the next page by using page keys and .

Erasing history data from the external operator message history screen

Procedure

1 Display the external operator message history screen.
2 Press soft key [(OPRT)].
3 Press soft key [CLEAR]. The external operator message history data is then erased.

FRYER MA Rob MS INC

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3112						OMH		

[Input type] Parameter input
[Data type] Bit

#2 OMH The external operator message history screen is:
0: Not displayed.
1: Displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3113	MS1	MS0						HMC

[Input type] Parameter input
[Data type] Bit

#0 HMC The contents of the external operator message history:
0: Cannot be erased.
1: Can be erased.

> **NOTE**
> This parameter is valid when bit 3 (SOH) of parameter No. 11354 is set to 1.

#6 MS0
#7 MS1 Set the combination of the number of characters and the number of messages to be preserved in the external operator message history.

Parameter		Maximum number of characters	Number of messages
MS0=0	MS1=0	255	8
MS0=1	MS1=0	200	10
MS0=0	MS1=1	100	18
MS0=1	MS1=1	50	32

> **NOTE**
> 1 Although up to 255 characters can be specified for each external operator message, you can use the combination of bits 6 (MS0) and 7 (MS1) of parameter No. 3113 to limit the number of characters and select the number of messages to be preserved in the external operator message history.
> 2 The settings of bits 6 (MS0) and 7 (MS1) of parameter No. 3113 take effect the next time the power is turned on. The external operator message history is erased at that time.
> 3 Even though you change the settings of bits 6 (MS0) and 7 (MS1) of parameter No. 3113, the alarm PW0000, "POWER MUST BE OFF" is not issued. You must however turn on the power again before the new settings can take effect.
> 4 If text (such as single-byte katakana or kanji characters) is entered in character code, the number of characters recorded in the external operator message history may be smaller than the maximum number of characters set by bits 6 (MS0) and 7 (MS1) of parameter No. 3113.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

	#7	#6	#5	#4	#3	#2	#1	#0
3196		HOM						

[Input type] Parameter input
[Data type] Bit

#6 HOM A history of external operator messages and macro messages #3006 is:
0: Recorded.
1: Not recorded.

	#7	#6	#5	#4	#3	#2	#1	#0
11354					SOH	SAH		

[Input type] Parameter input
[Data type] Bit

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#2 SAH When the storage capacity for history data is exceeded due to non-alarm history, alarm history will be:
0: Erased.
1: Erased, except the most recent 50 items of history data.

#3 SOH When the storage capacity for history data is exceeded due to data other than external operator message history, external operator message history will be:
0: Erased.
1: Retained.
When this parameter is set to 1, the number of retained external operator message history is determined by the settings of bits 6 (MS0) and 7 (MS1) of parameter No. 3113.

NOTE
1 The settings of bit 2 (SAH) of parameter No. 11354 and bit 3 (SOH) of parameter No. 11354 will be effective the next time the power is turned on. At this time, all history data (operation history, alarm history, and external operator message history) will be erased.
2 With the settings of bit 2 (SAH) of parameter No. 11354 and bit 3 (SOH) of parameter No. 11354, the number of history data items that can be retained varies. The number of history data items that can be recorded as follows:
SAH=0, SOH=0 . . . Approx. 8000 items
SAH=1, SOH=0 . . . Approx. 7400 items
SAH=0, SOH=1 . . . Approx. 7500 items
SAH=1, SOH=1 . . . Approx. 6900 items
(*) The numbers of items above are those if only key operation history is recorded.

FRYER MACHINE SYSTEMS INC
Rob Zahansky

12.6 SWITCHING BETWEEN MULTI-PATH DISPLAY AND SINGLE-PATH DISPLAY FUNCTION

Overview

In a multi-path system, the screen can be switched between simultaneous multi-path display and single-path display by soft key operation.
This function allows easy switching between the single-path display which contains much information and the simultaneous multi-path display which enables multiple path information to be checked at a time.

Details

The screen can be switched between the simultaneous multi-path display and single-path display by soft key operation.
In the single-path display, the screen for the path selected by the path selection signal is displayed.
In the simultaneous multi-path display, the screen for the simultaneous display path group including the path selected by the path selection signal is displayed.
This function is enabled with the following screens which can display information of multiple paths simultaneously when bit 3 (MTS) of parameter No. 11355 is 1.

- Actual position display screen
  (Absolute position display screen, relative position display screen, overall position display screen)
- Program check screen
- Tool path graphic screen
- Program edit screen

NOTE
For the setting of the simultaneous multi-path display, refer to the description of parameters Nos. 13131 and 13132 in the Parameter Manual (B-64700EN).

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Procedure (10.4-inch display unit)

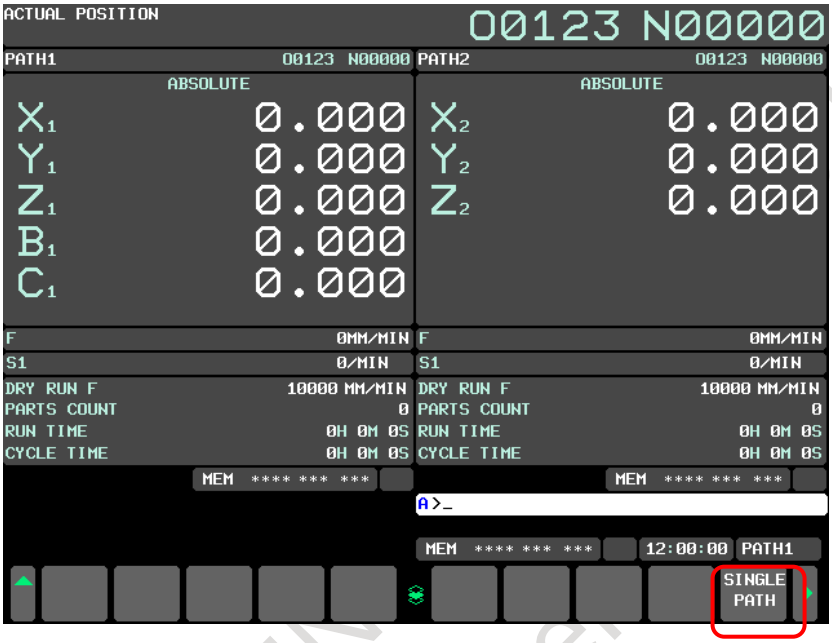
The procedure is explained using the current position display screen as an example.

1 Press function key [POS].

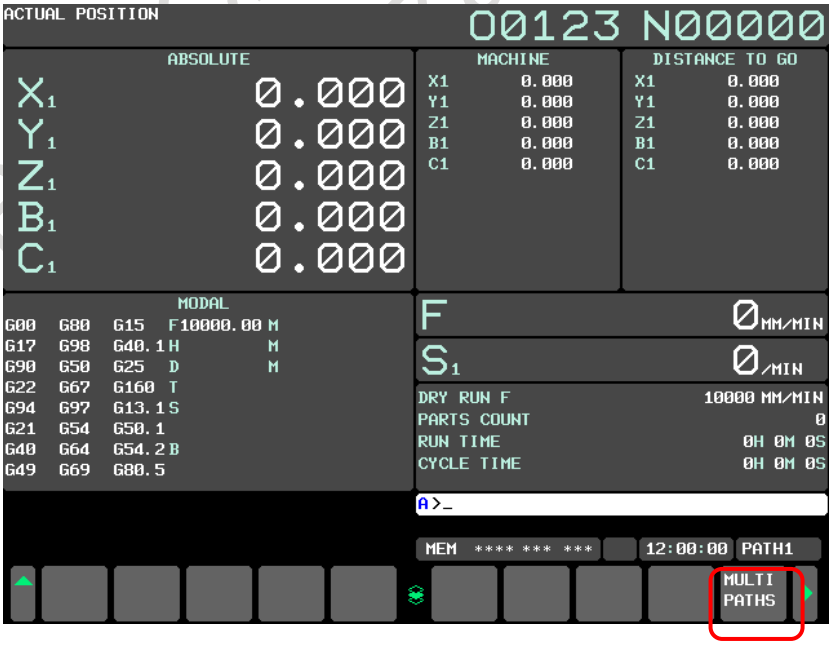
Actual position screen (multi-path simultaneous display) appears.

2 Press soft key [(OPRT)].

3 Press continuous menu key [▷] until soft key [SINGLE PATH] appears.



4 Press soft key [SINGLE PATH].
The single-path display appears as shown below:



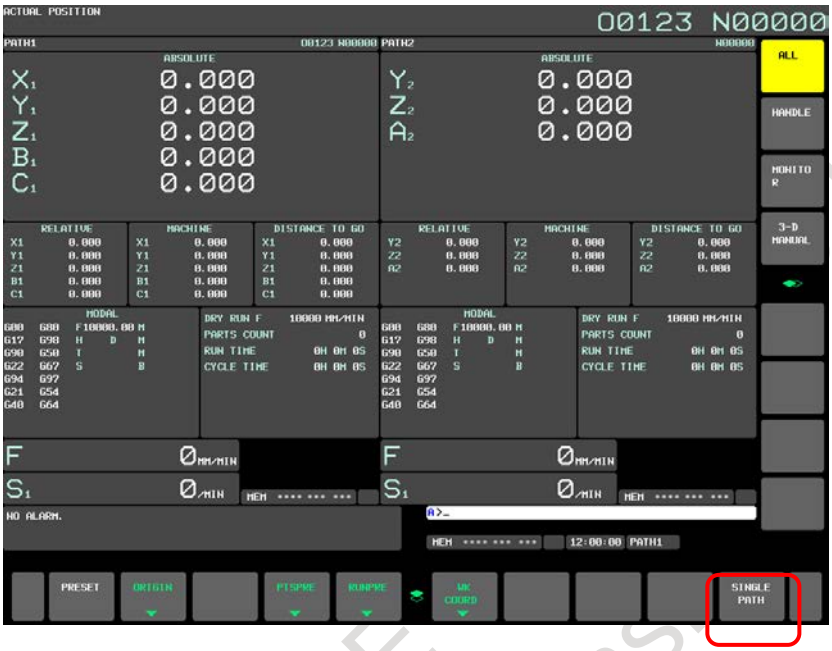
5 Press soft key [MULTI PATHS]. The multi-path display appears.

Procedure (15/19-inch display unit)

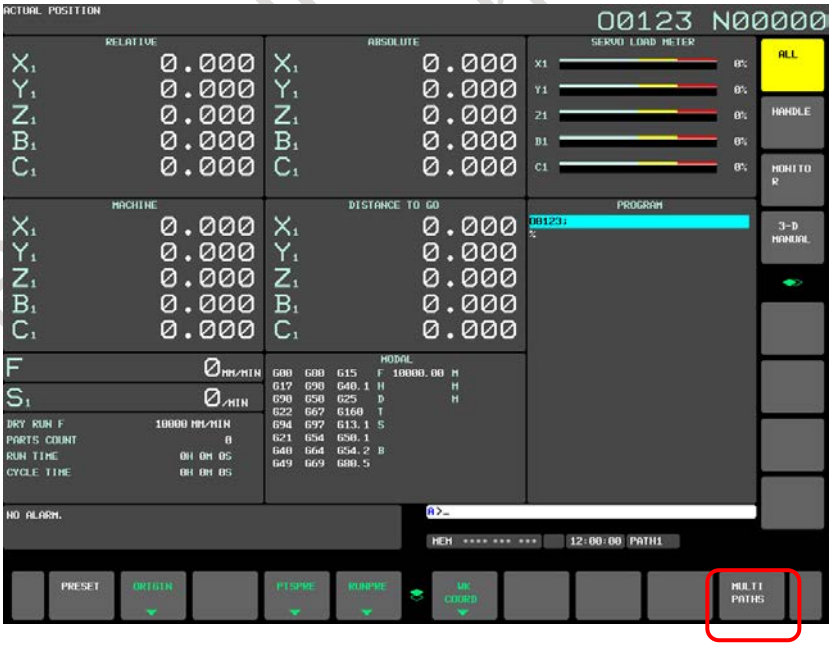
The procedure is explained using the current position display screen as an example.

1 Press function key [POS].

Actual position screen (multi-path simultaneous display) appears.



2 Press horizontal soft key [SINGLE PATH].
The single-path display appears as shown below:



3 Press horizontal soft key [MULTI PATHS]. The multi-path display appears.

FRYER LINE hensive INC

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11355					MTS			

[Input type] Parameter input
[Data type] Bit

#3 MTS The function for switching between simultaneous multi-path display and single-path display is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
11304								PGR

[Input type] Parameter input
[Data type] Bit

#0 PGR When the path select signal is changed, the screen of the multi path simultaneous display group:
0: Is not switched.
1: Is switched to the display group including the selected path.

13131	Group number for simultaneous display of multiple paths

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 4
This parameter sets a group for simultaneous display on one screen in a multi-path system.
The paths set to belong to the same group are displayed on one screen.
If the values for all paths are set to 0, the simultaneous multi-path display function is disabled.

NOTE
When specifying groups, specify group numbers not less than 1 successively.
On 10.4-inch display unit, up to three paths can be specified for simultaneous display.
On a 15-inch and 19-inch display units, up to four paths can be specified for simultaneous display.

13132	Simultaneous multi-path display order number

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of paths included in a simultaneous multi-path display group
This parameter sets the display order of a path defined to belong to a simultaneous multi-path display group.
Set the order, using numbers ranging from 1 to the number of paths included in the simultaneous multi-path display group.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

> **NOTE**
> Specify successive order numbers not less than 1 for the paths defined to belong to a group.

	#7	#6	#5	#4	#3	#2	#1	#0
6502				MPD				

[Input type] Parameter input
[Data type] Bit
#4 MPD On tool path graphic screen, operation of starting, ending and erasing is effective to:
0: all paths.
1: selected path.

12.7 PATH NAME EXPANSION DISPLAY FUNCTION

The path name expansion display function enlarges the path name displayed at the upper right of the screen, which makes easy to check the currently selected path.

Explanation

When this function is enabled, the items on the screen are displayed as follows:

- The path name displayed in the status area is enlarged and displayed at the upper right of the screen. Only the alphanumeric characters in an arbitrary path name are displayed when enlarged (see (1) in Fig. 12.7 (a)). For the display colors, see the following paragraph.
- The O and N numbers are always displayed in small characters as shown in (2) in Fig. 12.7 (a).
  The displayed program names consist of the following number of characters:
  10.4-inch display unit: 17 characters (When a program number consists of more than 17 characters, the first 16 characters and a tilde (~) are displayed.)
  15-inch display unit: 32 characters (Maximum number of characters of an arbitrary program name)
- The path name displayed in the status area is displayed as usual (see (3) in Fig. 12.7 (a)).

This function is enabled by setting bit 2 (PNE) of parameter No. 11350.
With all screens other than those for the C Language Executor and macro executor (conversational macro), this function is available.

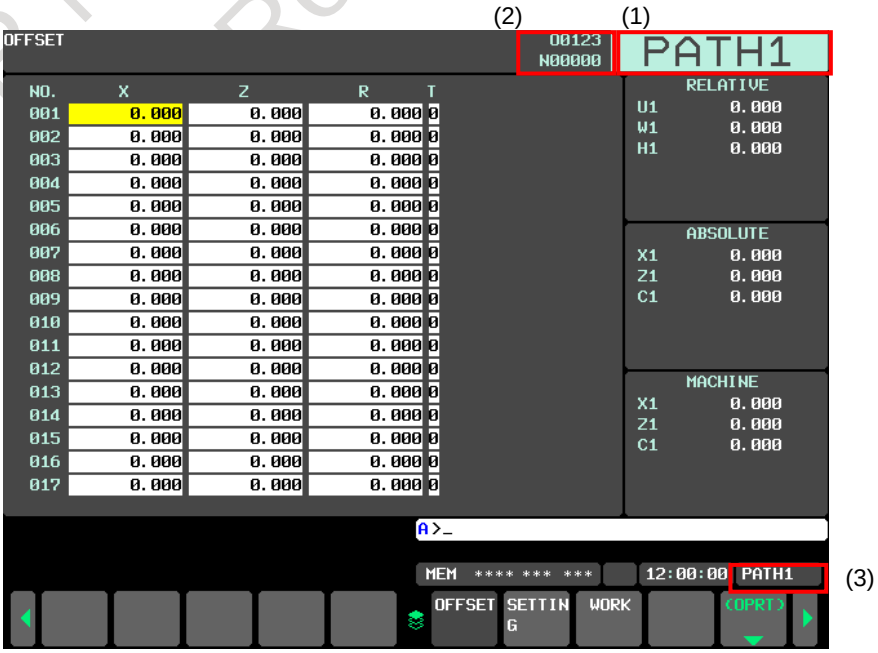


Fig. 12.7 (a) Sample screen when the path name expansion display function is enabled (offset screen)

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

Display character color

The following colors specified on the color setting screen are used for the enlarged display characters:

Character color: No. 3 color (title color)
Background color: No. 10 color (title background color)

The path name can be displayed in reverse video by setting bit 0 (PNI) of parameter No. 11352.

Table 12.7 (a) Enlarged display character colors and parameters

		Character color	Background color
Bit 0 (PNI) of parameter No. 11352	0: A normal display	No. 3 color	No. 10 color
	1: A reverse display	No. 10 color	No. 3 color

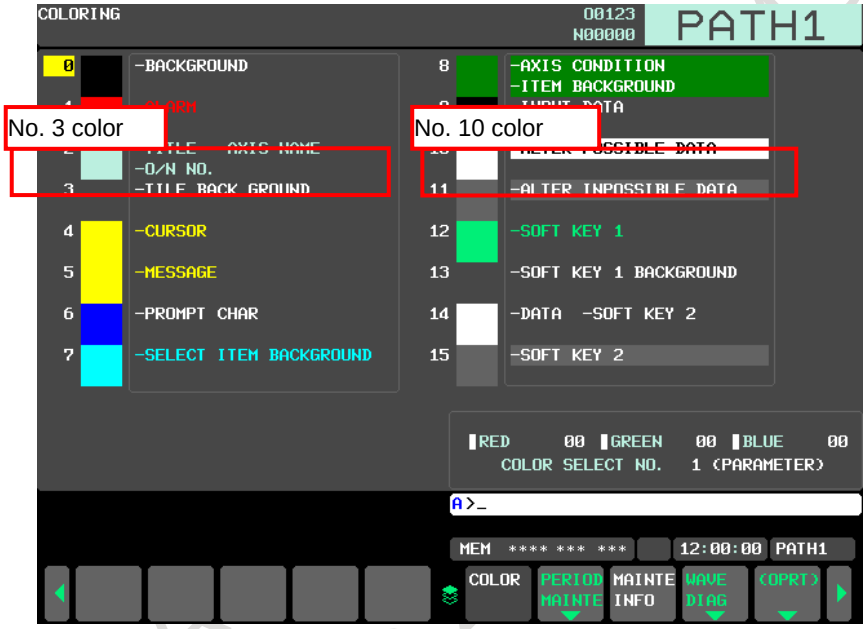


Fig. 12.7 (b) Character color numbers for the path name expansion display function

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11350						PNE		

[Input type] Parameter input
[Data type] Bit

#2 PNE Path name expansion display function is:
0: Disabled.
1: Enabled.

> **NOTE**
> 1 When this parameter is set, the power must be turned off before operation is continued.
> 2 This parameter is invalid if the number of paths is 1.
> 3 It is effective to 10.4/15/19-inch display units.

	#7	#6	#5	#4	#3	#2	#1	#0
11352								PNI

[Input type] Parameter input
[Data type] Bit path

#0 PNI The display by the path name enlarged display function is:
0: A normal display.
1: A reverse display.

NOTE
This parameter is effective to 10.4/15/19-inch display units.

3141	Path name (1st character)
3142	Path name (2nd character)
3143	Path name (3rd character)
3144	Path name (4th character)
3145	Path name (5th character)
3146	Path name (6th character)
3147	Path name (7th character)

[Input type] Parameter input
[Data type] Word path
[Valid data range] See the character-code correspondence table.
Specify a path name with codes.
Any character string consisting of alphanumeric characters, katakana characters, and special characters with a maximum length of seven characters can be displayed as a series name.

NOTE
1 For characters and codes, see Appendix, “CHARACTER CODE LIST”.
2 When 0 is set in parameter No. 3141, PATH1(,PATH2...) are displayed as path names.
3 When optional path name display is enlarged (with bit 2 (PNE) of parameter No. 11350 set to 1), only alphanumeric characters are displayed. If any other type of characters are set, spaces are displayed instead.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

12.8 SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION

Overview

Keeping the same characters displayed in the same positions on the screen for a long time will shorten the life of the LCD.
To prevent this, the CNC screen can be erased. The screen erasure function allows the user to perform a key operation to erase the screen. The automatic screen erasure function erases the screen automatically when there has been no key operation for a parameter-set period of time.

Screen erasure function

When 0 is set in parameter No. 3123, the CNC screen can be erased by pressing the CAN key and any function key (such as POS or PROG) at the same time. The CNC screen can be displayed again by pressing any function key.

Automatic screen erasure function

When there has been no key operation for a time (in minutes) set in parameter No. 3123, the CNC screen is erased automatically. The CNC screen is displayed again by pressing a key.

- Screen erasure by the automatic screen erasure function

If the following conditions are all satisfied for the time (in minutes) set in parameter No. 3123, the CNC screen is erased.
Conditions for automatically erasing the CNC screen

- Parameter No. 3123 ≠ 0
- None of the following key operations is performed.
  MDI keys
  Soft keys
  External key input
- No alarm is issued.

- Redisplay of the screen by the automatic screen erasure function

If one of the following conditions is satisfied when the CNC screen is off, the CNC screen is displayed again:
Conditions for redisplaying the CNC screen

- One of the following key operations is performed.
  MDI keys
  Soft keys
  External key input
- An alarm is issued.

- Screen erasure by using the CAN key + function key

When a non-zero value is set in parameter No. 3123, the screen is not erased with the CAN key and a function key.

- Set time

Only the time set in parameter No. 3123 for path 1 is valid.

- Alarm in another path

When an alarm is issued in any of the paths, the screen is not erased.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Parameter

3123	Time required before a screen saver is activated

[Input type] Setting input
[Data type] Byte path
[Unit of data] min
[Valid data range] 0 to 127
Set time until a screen saver activated in minutes. This parameter is effective on path 1 only.

NOTE
1 Setting 0 disables automatic screen erasure.
2 This function cannot be used together with manual screen erasure. If 1 or a larger value is set in this parameter, manual screen erasure is disabled.

12.9 LOAD METER SCREEN

Overview

The servo and spindle load meters can be displayed in place of the modal code display part and the remaining travel distance part of the current position display on the program check screen.
This function is enabled only when the 10.4-inch display unit is used.

12.9.1 Single-path Display

Screen layout

In the single-path display mode, the servo or spindle load meter is displayed in place of the remaining travel distance display of the total position display.

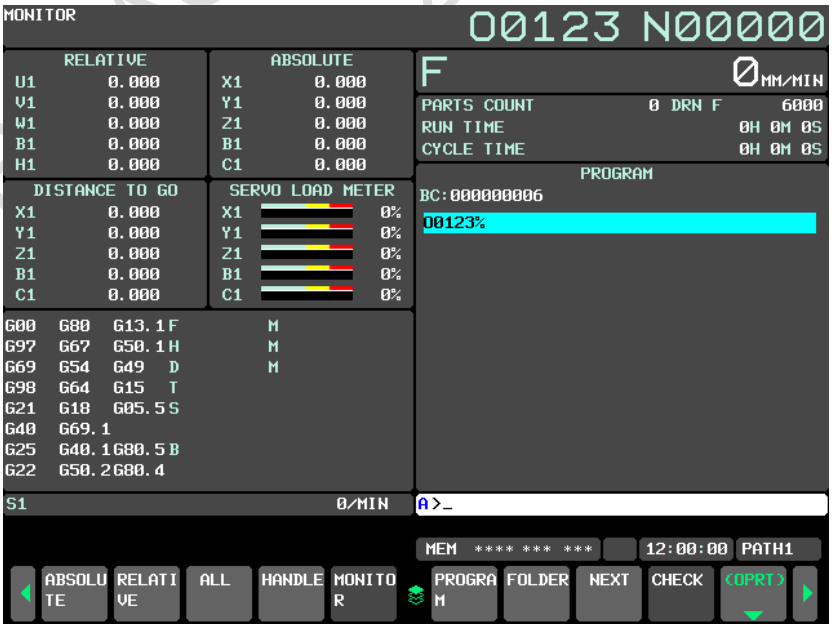


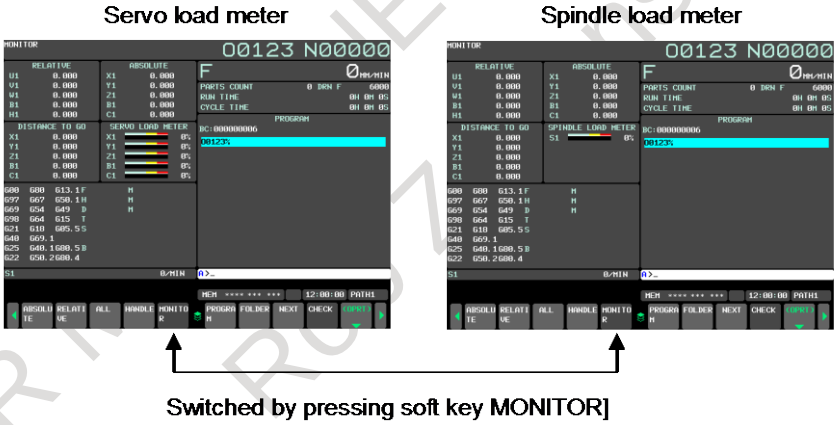
Fig. 12.9.1 (a) Servo load meter

FRYER MACHINE SYSTEMS INC
Zahensky

Fig. 12.9.1 (b) Spindle load meter

Screen switching

The servo load meter and spindle load meter are displayed by pressing soft key [MONITOR] on the left side of the screen. Initially the servo load meter is displayed. Each time soft key [MONITOR] is pressed, the servo load meter and spindle load meter alternate.



FRYER INSTITUTE SYSTEMS INC
ROJ Zelinsky

12.9.2 Two-path Display and Three-path Display

Screen layout

In the two- or three-path display mode, the modal information display part on the program check screen can be replaced by the servo or spindle load meter.

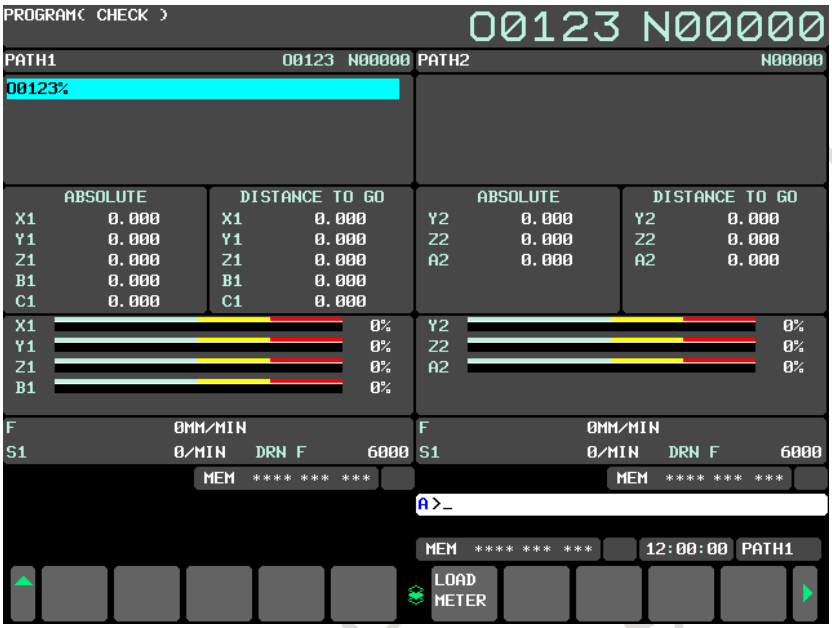


Fig. 12.9.2 (a) Servo load meter screen

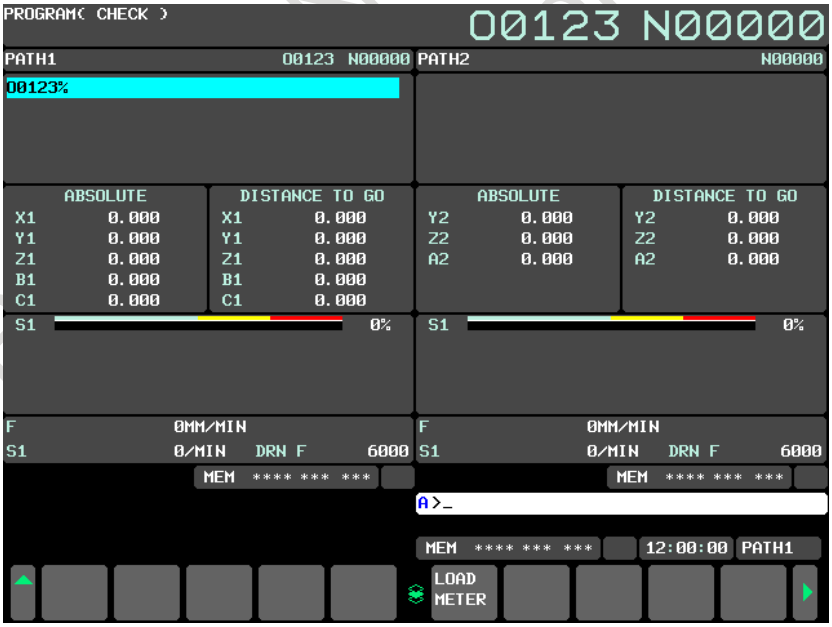


Fig. 12.9.2 (b) Spindle load meter screen

FRYER AND ASSOCIATES INC

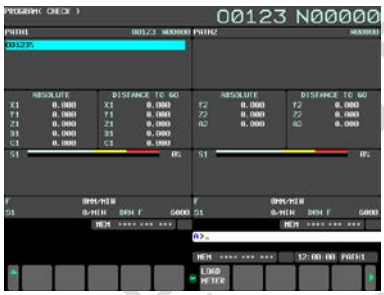
Screen switching

By pressing soft key [LOAD METER], the screen display can be switched among the modal information display, servo load meter display, and spindle load meter display.

Modal information

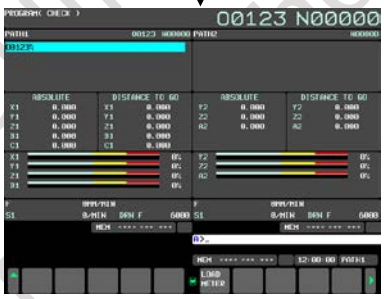
Press soft key [LOAD METER].

Spindle load meter



Press soft key [LOAD METER].

Servo load meter



Press soft key [LOAD METER].

FRYER MACHINE SYSTEMS INC
Pavlovsky

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3192	PLD							

[Input type] Parameter input
[Data type] Bit

#7 PLD When the current position is indicated for a path, and when the program check screen is displayed in a two- or three-path system, the function for displaying servo load meters and spindle load meters is :
0: Disabled.
1: Enabled.

13140	First character in spindle load meter display
13141	Second character in spindle load meter display

[Input type] Setting input
[Data type] Byte spindle
[Valid data range] These parameters set character codes to set the name of each spindle that appears in spindle load meter display. Any character string consisting of numeric characters, alphabetical characters, katakana characters, and symbols with a maximum length of two characters can be displayed as a spindle name.
If 0 is set, the following is displayed:

1st spindle	S1
2nd spindle	S2
3rd spindle	S3
4th spindle	S4

12.10 DISPLAYING THE PROGRAM NUMBER/NAME, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION

The program number, program name, sequence number, and current CNC status are always displayed on the screen except when the power is turned on, a system alarm occurs.
If data setting or the input/output operation is incorrect, the CNC does not accept the operation and displays a warning message.
This section describes the display of the program number, sequence number, and status, and warning messages displayed for incorrect data setting or input/output operation.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Screens of a 10.4-inch display unit

12.10.1 Displaying the Program Number, Program Name, and Sequence Number

The number and name of the program currently selected or currently executed and the current sequence number are indicated in the upper right part as shown below.

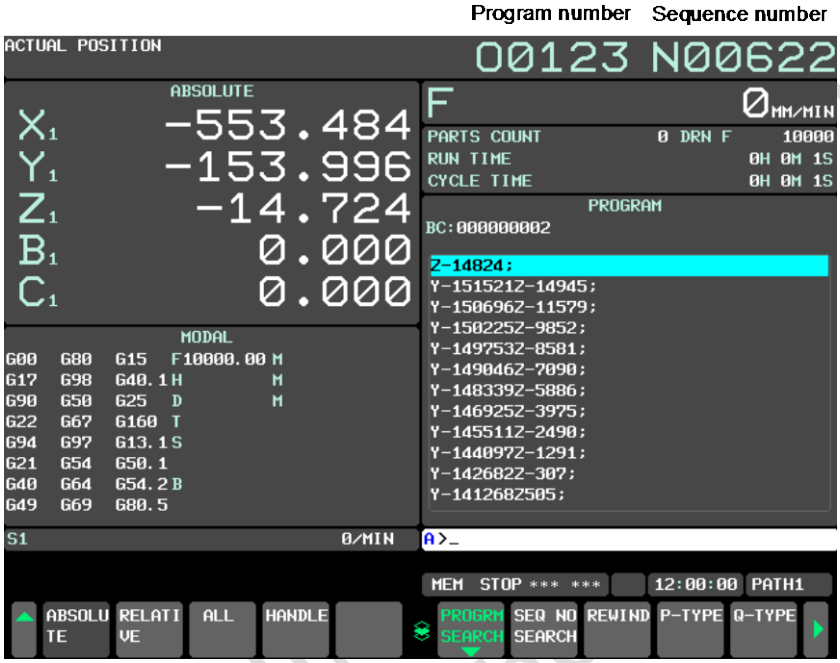


Fig. 12.10.1 (a) Program number and sequence number (10.4-inch display unit)

In the EDIT mode, the number and name of the program currently edited in the foreground are indicated.

When 10.4-inch display unit is used, the program number that is displayed in the upper right of the screen is changed according to the digit of the program number.
When the program number is 4-digit or less, “O” plus a 4-digit numeric is displayed.

Fig. 12.10.1 (b) Program number is 4-digit or less

When the program number is 5-digit or more, “O” plus a 8-digit numeric is displayed.

Fig. 12.10.1 (c) Program number is 5-digit or more

FRYER MACHINE INC Rob Zahn

12.10.2 Displaying the Status and Warning for Data Setting or Input/Output Operation

The current mode, automatic operation state, alarm state, and program editing state are displayed on the second line from the bottom on the screen allowing the operator to readily understand the operation condition of the system. If data setting or the input/output operation is incorrect, the CNC does not accept the operation and a warning message is displayed on the third line from the bottom on the screen. This prevents invalid data setting and input/output errors.

Explanation

- **Description of each display**

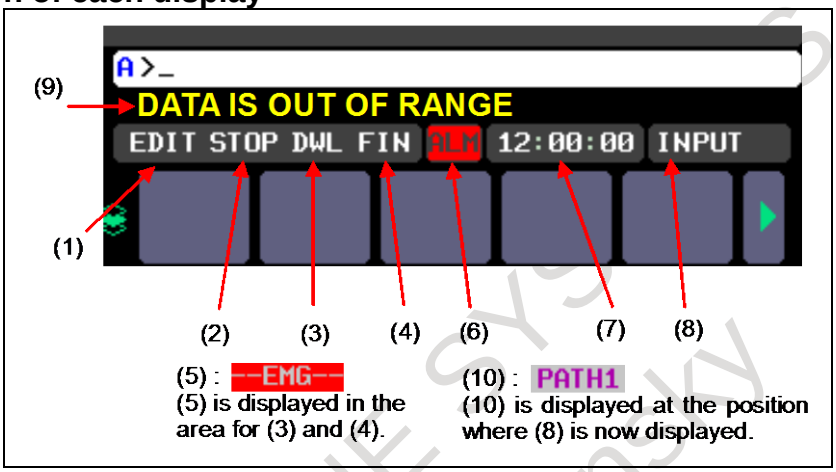


Fig. 12.10.2 (a) Positions of status indications

(1) Current mode

MDI : Manual data input, MDI operation
MEM : Automatic operation (memory operation)
RMT : Automatic operation (DNC operation, or such like)
EDIT : Memory editing
HND : Manual handle feed
JOG : Jog feed
INC : Manual incremental feed
REF : Manual reference position return
**** : Indicates a mode other than the above.

(2) Automatic operation status

**** : Reset (When the power is turned on or the state in which program execution has terminated and automatic operation has terminated.)
STOP : Automatic operation stop (The state in which one block has been executed and automatic operation is stopped.)
HOLD : Feed hold (The state in which execution of one block has been interrupted and automatic operation is stopped.)
STRT : Automatic operation start-up (The state in which the system operates automatically)
MSTR : Tool retract and recover operation state (The state in which a recover operation and repositioning operation are being performed)

(3) Axis moving status/dwell status

MTN : Indicates that the axis is moving.
DWL : Indicates the dwell state.
*** : Indicates a state other than the above.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

(4) State in which an auxiliary function is being executed

FIN : Indicates the state in which an auxiliary function is being executed. (Waiting for the complete signal from the PMC)

(5) Emergency stop or reset status

--EMG-- : Indicates emergency stop.(Blinks in reversed display.)
-RESET- : Indicates that the reset signal is being received.

(6) Alarm status

When plural alarm conditions occur at the same time, the highest priority alarm will be displayed when bit 0 (LNG) of parameter No. 8906 is set to 0.
All alarms will be displayed per 1 second in order of the priority when the parameter LNG is set to 1.

Priority high

ALM : Indicates that an alarm is issued. (Blinks in reversed display.)
APC : Indicates that the voltage of the backup battery of the absolute pulse coder has decreased. (Blinks in reversed display.)
BAT : Indicates that the voltage of the lithium battery (the backup battery of the CNC) has decreased. (Blinks in reversed display.)
FAN : Indicates that the rotation speed of the fan has decreased. (Blinks in reversed display.) Check the fan monitor screen and replace the fan motors for which the rotation speed is found to be decreased.
PMC : Indicates that a PMC alarm is issued. (Blinks in reversed display.)
MSG : Indicates that an operator message is issued. (Blinks in reversed display.)
FSB : Indicates that FSSB cable has been degraded. (Blinks in reversed display.)
LKG : Indicates that the insulation resistance value of the motor or the power line has decreased.. (Blinks in reversed display.)
ENC : Indicates that a detector has been in abnormal status. (Blinks in reversed display.)
MNT : Indicates that a life warning of periodic maintenance function has been occurred. (Blinks in reversed display.)
Space : Indicates a state other than the above.

Priority low

(7) Current time

hh : mm : ss - Hours, minutes, and seconds

(8) Program editing status

INPUT : Indicates that data is being input.
OUTPUT : Indicates that data is being output.
SEARCH : Indicates that a search is being performed.
EDIT : Indicates that another editing operation is being performed (insertion, modification, etc.)
LSK : Indicates that labels are skipped when data is input.
RSTR : Indicates that the program is being restarted
COMPARE : Indicates that a data comparison is being made.
OFST : Indicates that the tool length compensation amount measurement mode is set (for the machining center system) or that the tool length compensation amount write mode is set (for the lathe system).
WOFS : Indicates that the workpiece origin offset amount measurement mode is set.
AICC1 : Indicates that operation is being performed in the AI contour control I mode.
AICC2 : Indicates that operation is being performed in the AI contour control II mode.
MEM-CHK : Indicates that a program memory check is being made.
WSFT : Indicates that the workpiece shift amount write mode is set.

FRYER MACHINE SYSTEMS INC
Rob Zaherski

TWP : Indicates that operation is being performed in the tilted working plane indexing mode.

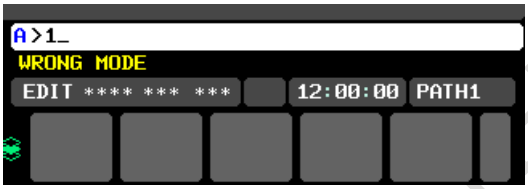
Space : Indicates that no editing operation is being performed.

(9) Warning for data setting or input/output operation

When invalid data is entered (wrong format, value out of range, etc.), when input is disabled (wrong mode, write disabled, etc.), or when input/output operation is incorrect (wrong mode, etc.), a warning message is displayed. When the RS-232C communication port is being used, “CANNOT USE I/O DEVICE” is displayed. In this case, the CNC does not accept the setting or input/output operation (retry the operation according to the message).

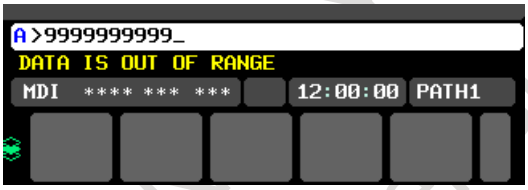
Example 1)

When a parameter is entered



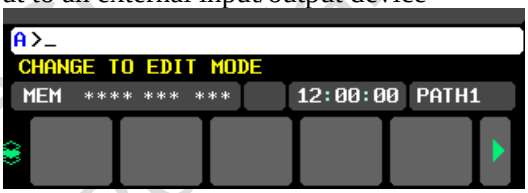
Example 2)

When a parameter is entered



Example 3)

When a parameter is output to an external input/output device



(10) Tool post name

The number of a path whose status is indicated is displayed.

PATH1 : Indicates that the status being indicated is for path 1.

Other names can be used depending on the settings of parameters Nos. 3141 to 3147. The tool post name is displayed at the position where (8) is now displayed. While the program is edited, (8) is displayed.

FRYER MACHINERY SYSTEMS INC
Rudolphe Sky

Screens of a 15/19-inch display unit

12.10.3 Displaying the Program Number, Program Name, and Sequence Number (15/19-inch Display Unit)

The number and name of the program currently selected or currently executed and the current sequence number are indicated in the upper right part as shown below.

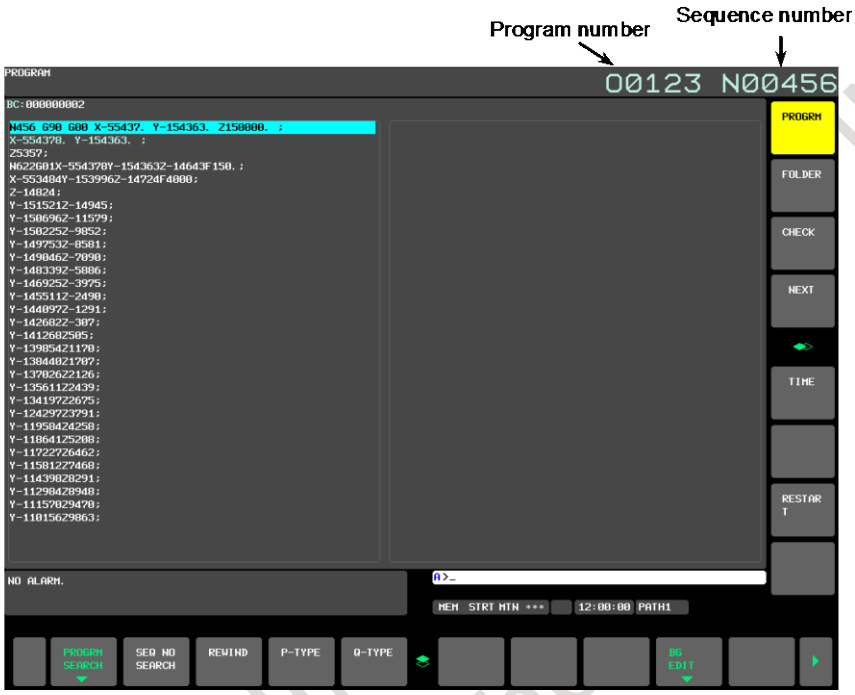


Fig. 12.10.3 (a) Program number and sequence number (15-inch display unit)

In the EDIT mode, the number and name of the program currently edited in the foreground are indicated.

FRYER MACHINE INC
Rob Zaine

12.10.4 Displaying the Program Name

When bit 4 (DPC) of parameter No. 11354 is set to 0, the program name (comment) of main program is displayed between the screen title and O number.

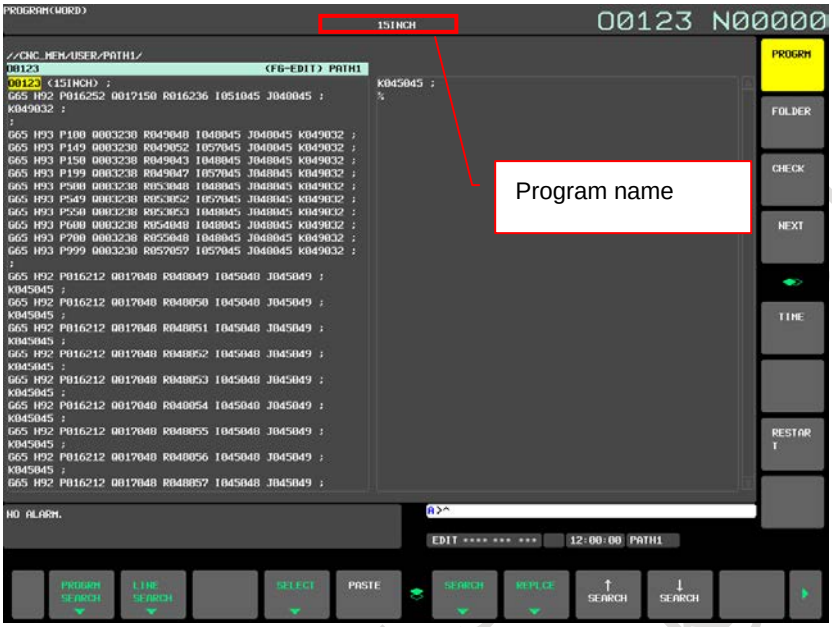


Fig. 12.10.4 (a) Program name (15inch)

A program name consisting of up to 18 characters can be displayed.
If a program name is longer than 18 characters, the first 17 characters are displayed. A tilde (~) is displayed as the 18th character, which indicates that the program name is abbreviated.

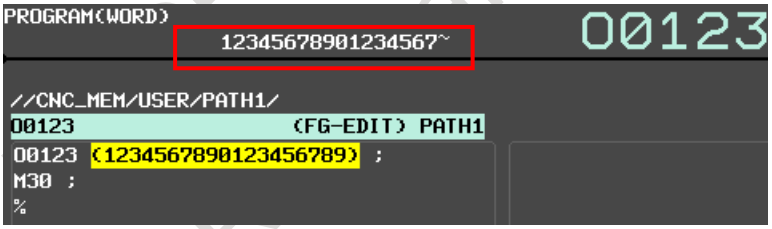


Fig. 12.10.4 (b) Program name longer than 18 characters

12.10.5 Displaying the Status and Warning for Data Setting or Input/Output Operation (15/19-inch Display Unit)

The current mode, automatic operation state, alarm state, and program editing state are displayed on the next to last line on the screen allowing the operator to readily understand the operation condition of the system. If data setting or the input/output operation is incorrect, the CNC does not accept the operation and a warning message is displayed on the next to last line of the screen. This prevents invalid data setting and input/output errors.

Explanation

- **Description of each display**

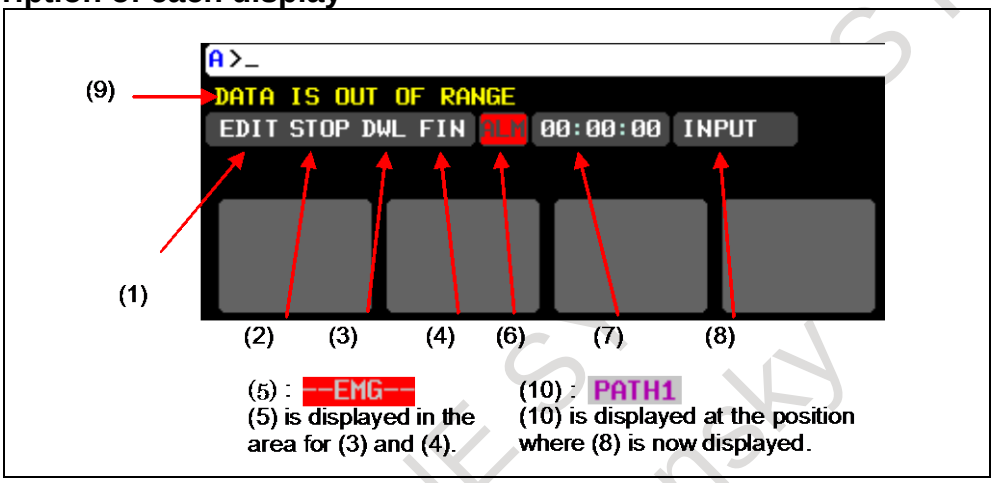


Fig. 12.10.5 (a) Positions of status indications

(1) Current mode

MDI : Manual data input, MDI operation
MEM : Automatic operation (memory operation)
RMT : Automatic operation (DNC operation, or such like)
EDIT : Memory editing
HND : Manual handle feed
JOG : Jog feed
INC : Manual incremental feed
REF : Manual reference position return
**** : Indicates a mode other than the above.

(2) Automatic operation status

**** : Reset (When the power is turned on or the state in which program execution has terminated and automatic operation has terminated.)
STOP : Automatic operation stop (The state in which one block has been executed and automatic operation is stopped.)
HOLD : Feed hold (The state in which execution of one block has been interrupted and automatic operation is stopped.)
STRT : Automatic operation start-up (The state in which the system operates automatically)
MSTR : Tool retract and recover operation state (The state in which a recover operation and repositioning operation are being performed)

(3) Axis moving status/dwell status

MTN : Indicates that the axis is moving.
DWL : Indicates the dwell state.
*** : Indicates a state other than the above.

FRYER MACHINE SALES INC
Rob Zahensky

(4) State in which an auxiliary function is being executed

FIN : Indicates the state in which an auxiliary function is being executed. (Waiting for the complete signal from the PMC)
*** : Indicates a state other than the above.

(5) Emergency stop or reset status

--EMG-- : Indicates emergency stop.(Blinks in reversed display.)
-RESET- : Indicates that the reset signal is being received.

(6) Alarm status

When plural alarm conditions occur at the same time, the highest priority alarm will be displayed when bit 0 (LNG) of parameter No. 8906 is set to 0.
All alarms will be displayed per 1 second in order of the priority when the parameter LNG is set to 1.

Priority high
ALM : Indicates that an alarm is issued. (Blinks in reversed display.)
APC : Indicates that the voltage of the backup battery of the absolute pulse coder has decreased. (Blinks in reversed display.)
BAT : Indicates that the voltage of the lithium battery (the backup battery of the CNC) has decreased. (Blinks in reversed display.)
FAN : Indicates that the rotation speed of the fan has decreased. (Blinks in reversed display.) Check the fan monitor screen and replace the fan motors for which the rotation speed is found to be decreased.
PMC : Indicates that a PMC alarm is issued. (Blinks in reversed display.)
MSG : Indicates that an operator message is issued. (Blinks in reversed display.)
FSB : Indicates that FSSB cable has been degraded. (Blinks in reversed display.)
LKG : Indicates that the insulation resistance value of the motor or the power line has decreased. (Blinks in reversed display.)
ENC : Indicates that a detector has been in abnormal status. (Blinks in reversed display.)
MNT : Indicates that a life warning of periodic maintenance function has been occurred. (Blinks in reversed display.)
Space : Indicates a state other than the above.
Priority low

(7) Current time

hh : mm : ss - Hours, minutes, and seconds

(8) Program editing status

INPUT : Indicates that data is being input.
OUTPUT : Indicates that data is being output.
SEARCH : Indicates that a search is being performed.
EDIT : Indicates that another editing operation is being performed (insertion, modification, etc.)
LSK : Indicates that labels are skipped when data is input.
RSTR : Indicates that the program is being restarted
COMPARE : Indicates that a data comparison is being made.
OFST : Indicates that the tool length compensation amount measurement mode is set (for the machining center system) or that the tool length compensation amount write mode is set (for the lathe system).
WOFS : Indicates that the workpiece origin offset amount measurement mode is set.
AICC1 : Indicates that operation is being performed in the AI contour control I mode.
AICC2 : Indicates that operation is being performed in the AI contour control II mode.
MEM-CHK : Indicates that a program memory check is being made.

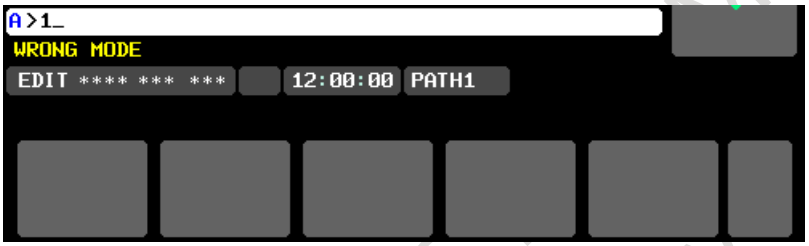
FRYER MACHINE SYSTEMS INC
Rob Zahrosky

WSFT : Indicates that the workpiece shift amount write mode is set.
TWP : Indicates that operation is being performed in the tilted working plane indexing mode.
TOLCON : Indicates that operation is being performed in Tolerance control mode.
Space : Indicates that no editing operation is being performed.

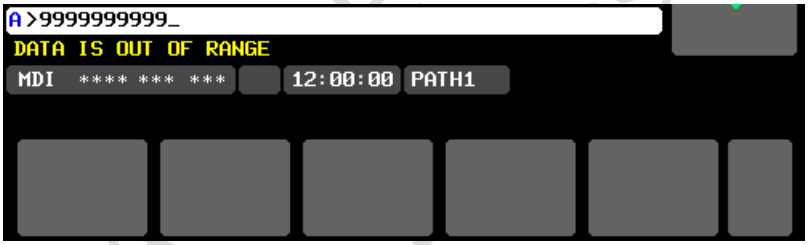
(9) Warning for data setting or input/output operation

When invalid data is entered (wrong format, value out of range, etc.), when input is disabled (wrong mode, write disabled, etc.), or when input/output operation is incorrect (wrong mode, etc.), a warning message is displayed. When the RS-232C communication port is being used, "CANNOT USE I/O DEVICE" is displayed. In this case, the CNC does not accept the setting or input/output operation (retry the operation according to the message).

Example 1)
When a parameter is entered



Example 2)
When a parameter is entered



Example 3)
When a parameter is output to an external input/output device

(10) Tool post name

The number of a path whose status is indicated is displayed.

PATH1 : Indicates that the status being indicated is for path 1.
Other names can be used depending on the settings of parameters Nos. 3141 to 3147. The tool post name is displayed at the position where (8) is now displayed. While the program is edited, (8) is displayed.

12.11 CALCULATION KEY FUNCTION

12.11.1 Overview

Calculating formula input to key-in buffer is calculated by pressing calculation key [SHIFT] + [CALC ALTER].

12.11.2 Operation

- Calculation of one calculating formula

(1) Calculating formula input to key-in buffer.

```
A>100.0+10.0_
```

(2) Calculation key [SHIFT] + [CALC ALTER] is pressed.

(3) Calculation result becomes the content of the key-in buffer.

```
A>110.0_
```

- Calculation of two or more calculating formulas

Two or more calculating formulas can be input by delimiting calculating formula with EOB, and each calculating formula is calculated.
Moreover, the same operation as EOB is performed when delimiting calculating formula with an alphabet or special characters other than the function name.
(Example)
100.0/2;200.0/2;300.0/2_ → 50.0;100.0;150.0_
X350.0*3Y120.0/5;X20.0*3.5;_ → X1050.0Y24.0;X70.0;_

12.11.3 Details

- Type of numerical value

There are two about the integer type and the real number type.
(1) Integer type: Only the numerical value from 0 to 9.
(Example) 10735
(2) Real number type: Numerical value that contains decimal point.
(Example) 10.23

- Calculation result

(1) Calculation result of the integer type and the real number type becomes a real number type.
(2) Calculation result is converted into the character string within the following ranges. In case of real number type, a value that is smaller than minimum value is rounded off.

Integer type:	-999999999 to 999999999
Real number type:	-999999999.0 to -0.000000001
	0.0
	0.000000001 to 999999999.0

- Calculating formula that can be used

(1) Arithmetic operator +, -, *, /
(2) Parentheses [,]
(3) Function

FRYER MACHINE SYSTEMS INC
Rob Zahensky

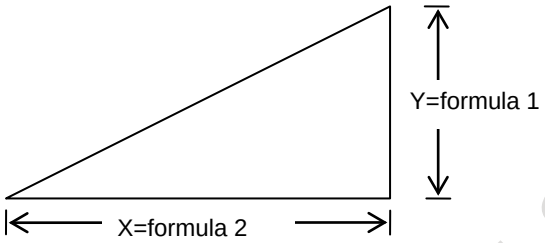
Table 12.12.3 Function that can be used by calculation function

Function name	Type	Explanation
SIN	Real number type	Sine（in degrees）
COS	Real number type	Cosine（in degrees）
TAN	Real number type	Tangent（in degrees）
ASIN	Real number type	Arc sine
ACOS	Real number type	Arc cosine
ATAN	Real number type	Arc tangent（two arguments）
SQRT	Real number type	Square root
ABS	Real number type	Absolute value
LN	Real number type	Natural logarithm
EXP	Real number type	Exponent using base e (2.718...)

Arc tangent

ATAN is described as follows.

ATAN[formula 1]/[formula 2]



(Example)

ATAN[0]/[1] → 0.0

ATAN[1]/[0] → 90.0

ATAN[1]/[1] → 45.0

ATAN[-1]/[-1] → 1350.0

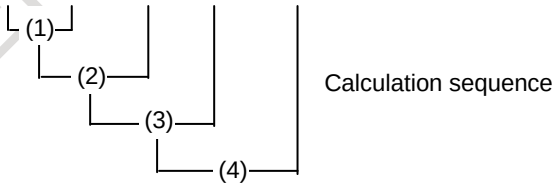
- Priority of calculation

(1) Function ＞ Multiplication type calculation ＞ addition type calculation

(2) The priority of calculation can be changed by parentheses.

(Example)

SIN [[[10 + 15] * 0.1 + 0.5] * 10]



The result of (4) becomes the argument of SIN.

- Warning

The following warning messages are displayed, when an error occurs by operating calculation key function.

Warning message	Cause
FORMAT ERROR	The format of the calculation formula is not correct.
DATA IS OUT OF RANGE	The calculation result is out of range.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Limitation

Calculation function cannot be used on the following screen.

- Conversational macro
- C Language Executor
- FANUC PICTURE
- MANUAL GUIDE *i*

FRYER MACHINE SYSTEMS INC
Rob Zahensky

13 GRAPHIC FUNCTION

13.1 GRAPHIC DISPLAY

The graphic display functions enable drawing of the tool path of the program currently used for machining.
These functions are intended to display the movement of the tool during automatic operation or during manual operation.
This enables the operator to check the progress of machining and the current tool position.
These functions include the following:

- The current tool position in the workpiece coordinate system is displayed.
- Graphic coordinates can be set freely.
- Rapid traverse and cutting feed can be drawn using a different color for each.
- The values of F, S, and T in the program during drawing are displayed.
- Graphic enlargement or reduction is possible.

Graphic display procedure

Explanation

Press the function key GRAPH then press the [GRAPH] soft key to display the tool path graphic screen.

- **Tool path graphic screen**

The tool path graphic screen consists of three major areas:

- Graphic area for drawing a tool path
- Area for displaying machining information such as tool position information
- Area for displaying a graphic coordinate system

Fig. 13.1 (a) Tool path graphic screen (M series)

FRYER MACHINE SYSTEMS INC
Zahensky

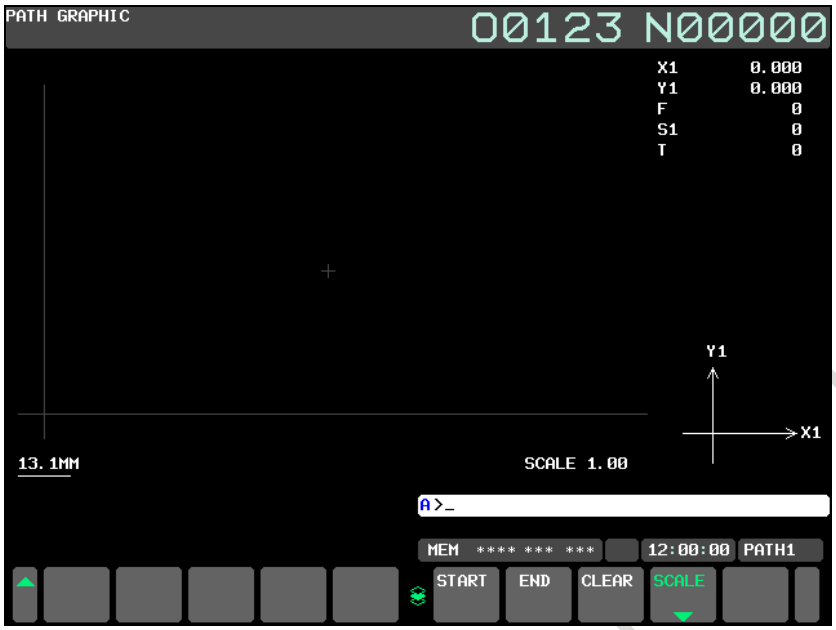


Fig. 13.1 (b) Tool path graphic screen (T series)

- Tool path

In a graphic coordinate system set by the graphic parameters described later, a tool path in the workpiece coordinate system is drawn.
Even when the tool position changes discontinuously for a cause such as the setting of the origin and the switching of the workpiece coordinate system, drawing is performed assuming that the tool has moved.
A tool path is continuously drawn even when the screen display is changed to another screen.

- Machining information

On the right side of the screen, the positions (along only three axes used for drawing) in the workpiece coordinate system, feedrate (F), spindle speed (S), and tool number (T) are displayed.

> **NOTE**
> Up to three graphic axes are used with the M series, and up to two graphic axes are used with the T series.

- Graphic coordinate system

On the lower-right portion of the screen, the coordinate axes and axis names of the graphic coordinate system are displayed.

Graphic parameter screen

Explanation

Press the function key [GRAPH] then press the [PARAM] soft key to display the tool path graphic screen.
On the graphic parameter screen, make settings necessary for drawing a tool path.
The graphic parameter screen consists of three pages.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

M

- **Graphic parameter screen page 1**

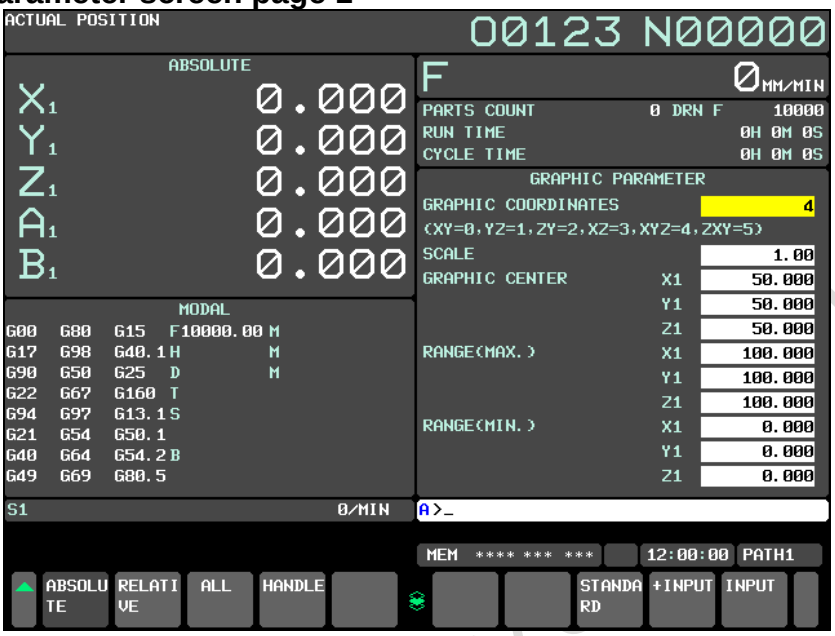


Fig. 13.1 (c) Graphic parameter screen page 1

On graphic parameter screen page 1, a graphic coordinate system, graphic range, and so forth are set.
In the setting of a graphic coordinate system, the coordinate axes and axis names of the set coordinate system are graphically displayed. When a 3-dimensional coordinate system is displayed, a rotation angle is also displayed.
A graphic range is set using one of two methods. One method sets a graphic scale and graphic center coordinates. The other sets the maximum values and minimum values of a graphic range.

- **Graphic parameter screen page 2**

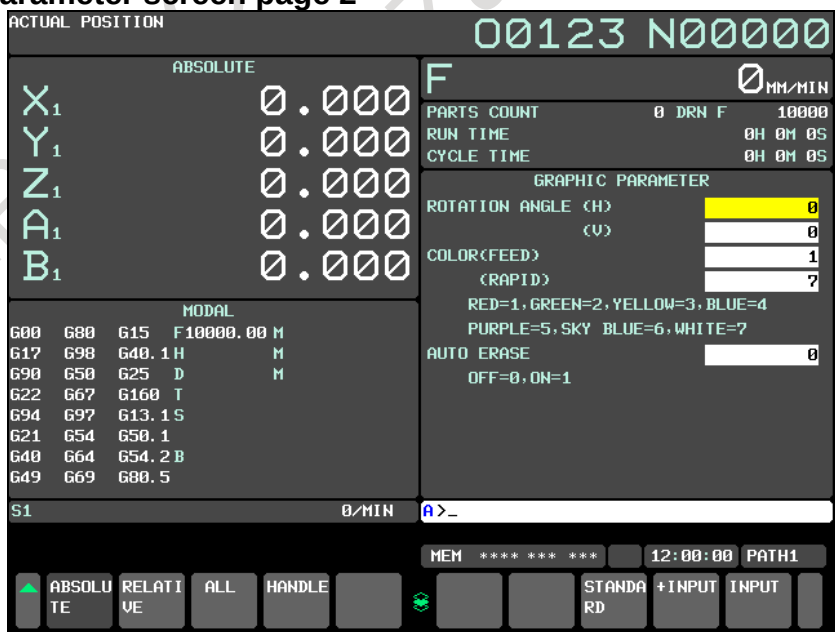


Fig. 13.1 (d) Graphic parameter screen page 2

On graphic parameter screen page 2, graphic colors, rotation angles, and whether to perform automatic erase operation are set.

- **Graphic parameter screen page 3**

Fig. 13.1 (e) Graphic parameter screen page 3

On graphic parameter screen page 3, coordinate axes to be used for drawing are set.

T

- **Graphic parameter screen page 1**

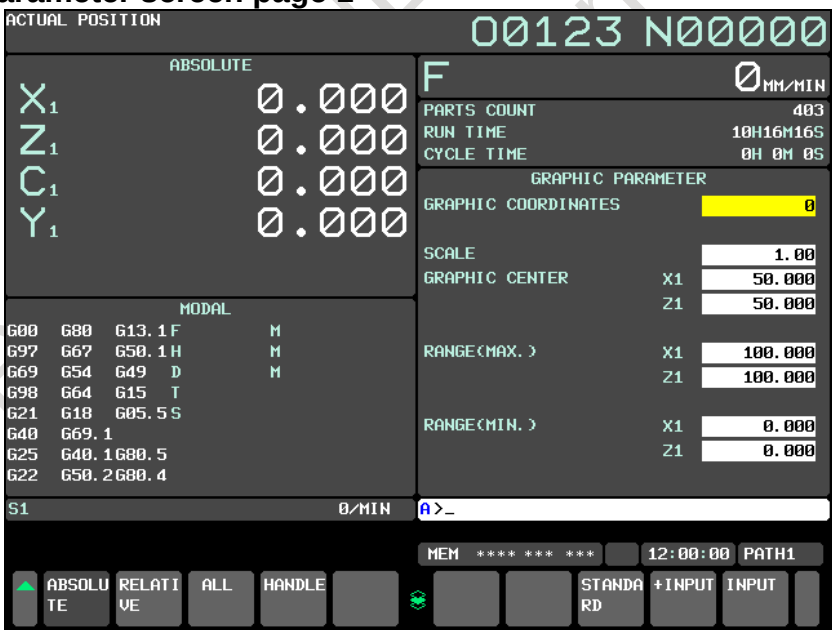


Fig. 13.1 (f) Graphic parameter screen page 1

On graphic parameter screen page 1, a graphic coordinate system, graphic range, and so forth are set.
In the setting of a graphic coordinate system, the coordinate axes and axis names of the set coordinate system are graphically displayed. When a 3-dimensional coordinate system is displayed, a rotation angle is also displayed.
A graphic range is set using one of two methods. One method sets a graphic scale and graphic center coordinates. The other sets the maximum values and minimum values of a graphic range.

FRYER E SYS INC
nsky

- **Graphic parameter screen page 2**

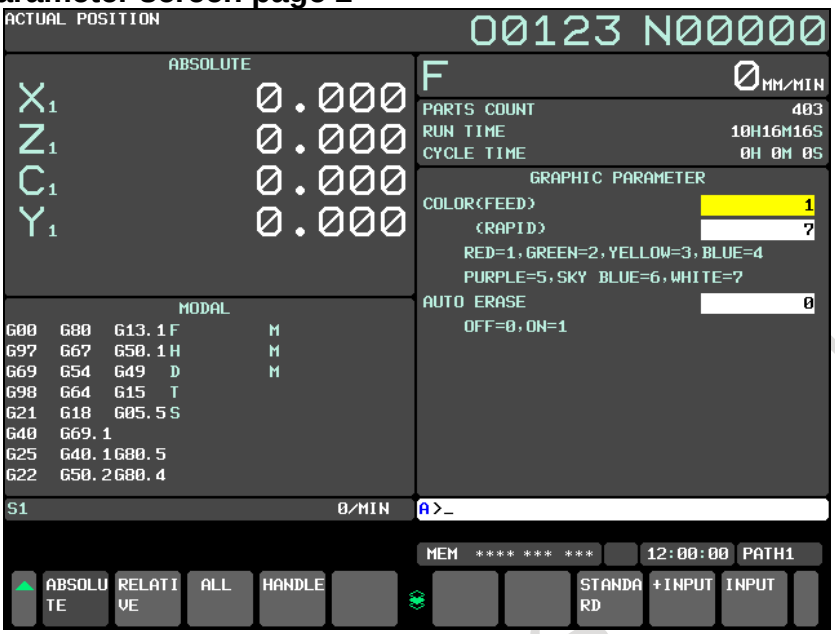


Fig. 13.1 (g) Graphic parameter screen page 2 (T series)

On graphic parameter screen page 2, graphic colors and whether to perform automatic erase operation are set.

- **Graphic parameter screen page 3**

Fig. 13.1 (h) Graphic parameter screen page 3

On graphic parameter screen page 3, coordinate axes to be used for drawing are set.

FRYER SYS INC

Graphic parameter setting

Explanation

For tool path drawing, a graphic coordinate system, tool path graphic colors, and graphic range need to be set on the graphic parameter screen.
The graphic parameters to be set on the graphic parameter screen are described below.
When a value is set for a parameter, the parameter value becomes immediately effective. If a tool path is already drawn, the tool path is cleared when new parameter values are set.

- Graphic coordinate system

Select a desired graphic coordinate system for tool path drawing then set the corresponding number.

M

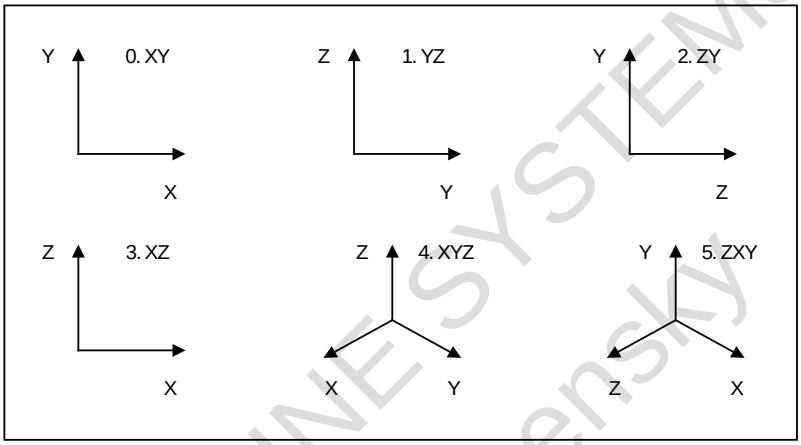


Fig. 13.1 (i) Graphic coordinate system (M series)

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

T

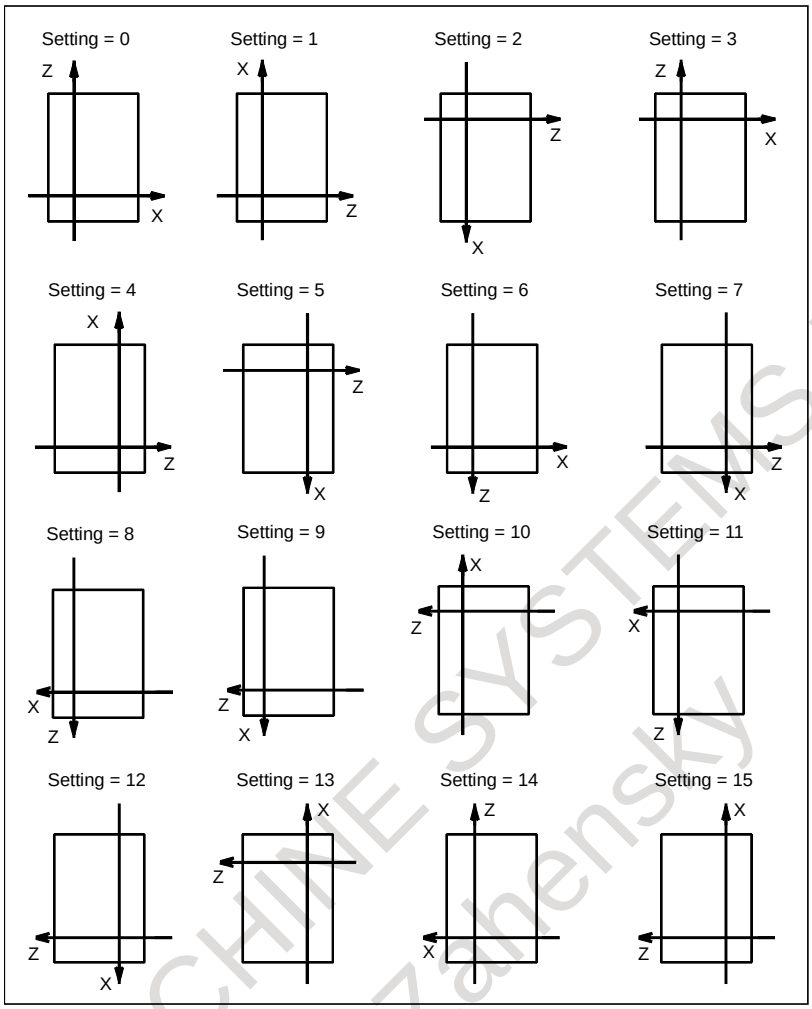


Fig. 13.1 (j) Graphic coordinate system (T series)

M

- Horizontal rotation angle

When a 3-dimensional graphic coordinate system such as 4.XYZ or 5.ZXY is selected, the coordinate system can be rotated with the horizontal plane used as the rotation plane. Set a rotation angle from -360° to +360°.

The initial rotation angle can be set with parameter No. 24831, and the actual rotation angle is equal to the "initial rotation angle + specified rotation angle".

In Fig. 13.1 (k) below, the graphic coordinate system XYZ is converted to X”Y”Z” by the following settings:

Initial rotation angle: 180°
Horizontal rotation angle: 30°

FRYER MACHINE SYSTEMS INC
Rob Zahensky

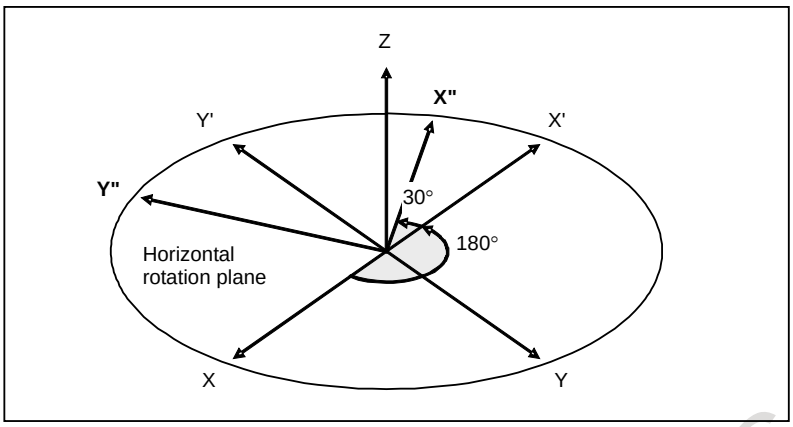


Fig. 13.1 (k) Coordinate system rotation in horizontal direction

- Vertical rotation angle

When a 3-dimensional coordinate system such as 4.XYZ or 5.ZXY is selected, the coordinate system can be rotated with an axis on the horizontal plane specified as a vertical rotation axis. Set a rotation angle from -360° to +360°.

A vertical rotation axis can be set with the angle with the horizontal axis of the screen on the horizontal plane. This angle can be set with parameter No. 24832.

In Fig. 13.1 (l) below, the graphic coordinate system XYZ is converted to X'Y'Z' by the following settings:

Rotation angle of the vertical rotation axis: 65°
Vertical rotation angle: 20°

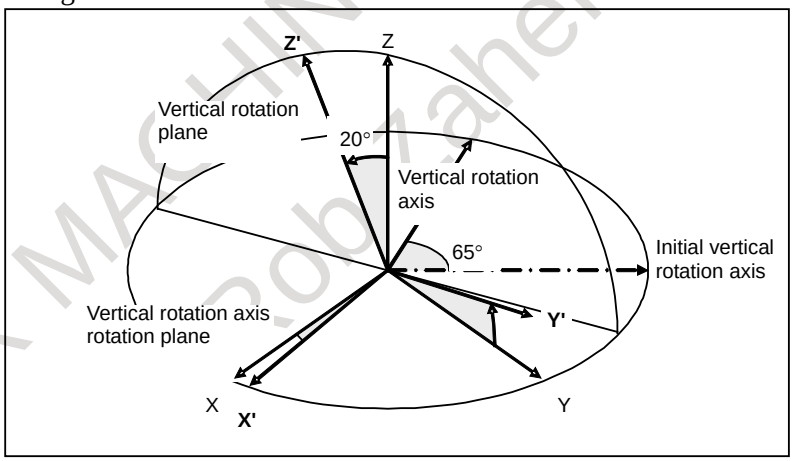


Fig. 13.1 (l) Coordinate system rotation in vertical direction

- Graphic color

Set a graphic color number for a tool path for each of cutting feed and rapid traverse.

1: Red, 2: Green, 3: Yellow, 4: Blue, 5: Purple, 6: Sky blue, 7: White

- Graphic range setting

Set a graphic range so that a tool path can be drawn in the tool path graphic area. Two methods are available:

1. Method that sets graphic center coordinates and a scale
2. Method that sets the maximum values and minimum values of a graphic range

Whether method 1 or method 2 is used is determined by which parameters were set most recently. A set graphic range is preserved even if the power is turned off.

FRYER MACHINE SYSTEMS INC
Bob Zahensky

1. Method that sets graphic center coordinates and a scale

Set the coordinates of the center of a tool path graphic area in the workpiece coordinate system. Then, set a scale used to contain the graphic range in the tool path graphic area.
Set a scale value from 0.01 to 100 (magnification).
As a smaller scale is set, a tool path can be drawn in a wider range. As a larger scale is set, a tool path is drawn by enlarging the tool path around the graphic center coordinates.

2. Method that sets the maximum values and minimum values of a graphic range

Set the maximum coordinates and minimum coordinates of a desired graphic range in the workpiece coordinate system. Drawing is performed so that the specified entire range is contained in the tool path graphic area.

From set maximum values and minimum values, graphic center coordinates and a sale are automatically calculated to update the graphic center coordinates and scale on the graphic parameter screen.
When a scale is automatically determined, the scale is clamped to within the range 0.01 to 100. Moreover, a maximum value must be greater than the corresponding minimum value.

NOTE

When the maximum values and minimum values of a graphic range are set, the graphic center coordinates and scale are automatically updated. However, when the graphic center coordinates and scale are changed, the maximum values and minimum values of the graphic range are not automatically updated.

- Automatic erasure

Before drawing is started, the previous drawing can be erased automatically.
1: Immediately before drawing is started, the previous drawing is erased automatically.
0: The previous drawing is not erased automatically.

- Graphic axis number

Set which controlled axis to be assigned to which graphic axis.
For each controlled axis, set one of the following graphic axis numbers:

First graphic axis:	1
Second graphic axis:	2
Third graphic axis:	3
Axis not used for drawing:	0

NOTE

1 When 0 is set for all controlled axes, it is assumed that 1, 2, and 3 are set sequentially for the first to the third controlled axes.
2 With the T series, a tool path is drawn along the first and second graphic axes. No tool path is drawn along the third graphic axis.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Operation for graphic parameter setting

Operation

- Moving the cursor

The cursor can be moved to a desired parameter by the page key [PAGE ↑] or [PAGE ↓] and the cursor key [↑], [↓], [←], or [→].

With the cursor keys, however, you cannot move from page 1 or 2 to page 3.

- Input of settings (absolute input)

Method 1

(1) Key in a value to be set.
(2) Press the [INPUT] soft key.

Method 2

(1) Key in a value to be set.
(2) Press the [INPUT] key.

- Input of settings (incremental input)

Method 1

(1) Key in a value to be incremented to or decremented from the current setting.
(2) Press the [+INPUT] soft key.

Procedure for tool path drawing

Procedure

- Start of drawing

(1) Display the tool path graphic screen.
(2) Press the [START] soft key.
The state that enables the movement of the tool in automatic operation or manual operation to be drawn is set.
Afterwards, tool path drawing continues even if another screen is displayed.
(3) Start automatic operation or manual operation.

- End of drawing

(1) Display the tool path graphic screen.
(2) Press the [END] soft key.
Tool path drawing ends.

- Erasure of the drawing

(1) Press the [END] soft key to end drawing.
(2) Press the [CLEAR] soft key. The tool path drawn so far is erased.

NOTE

1 Set the machine lock state to perform drawing only without moving the tool.
2 When the feedrate is high, the tool path may not be drawn correctly. In such a case, decrease the feedrate by performing, for example, a dry run.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Enlarged/reduced display

On the tool path graphic screen, you can move the center position of the tool path drawing or enlarge the tool path drawing while viewing the drawn tool path.
If any of these operations is executed, the tool path already drawn is cleared.

- Procedure for changing the graphic range by setting a graphic center and magnification

The center position of drawing can be moved. At the same time, the scale can also be changed. So, the tool path can be enlarged or reduced at a desired new center position.

(1) Press the [SCALE] soft key then the [CENTER] soft key.
A yellow cursor appears at the center of the screen, and the soft key display is changed.

(2) Move the yellow cursor to a new graphic center position by using the cursor key ⬆, ⬇, ⬅, or ➡.

(3) When changing the scale, key in a value from 0.01 to 100 (magnification) then press the [INPUT] soft key. An input value is displayed at "SCALE" in the lower-right corner of the screen.
When you press the [+INPUT] soft key, the current magnification is incremented by an input value.

(4) Press the [EXEC] soft key to end the operation.
After this step, the setting for graphic movement is effective to enable drawing with the new setting.

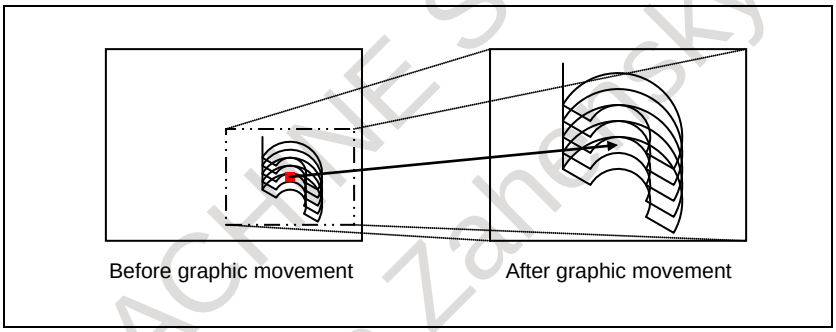


Fig. 13.1 (m) Graphic movement (magnification = 2.00)

- Procedure for changing the graphic range with a rectangle

A tool path can be drawn by enlarging a specified rectangular area.

(1) Press the [SCALE] soft key then the [RECTANGLE] soft key.
Two cursors, one in red and the other in yellow, appear at the center of the screen, and the soft key display is changed.

(2) Move the yellow cursor by using the cursor key ⬆, ⬇, ⬅, or ➡. The cursor to be moved can be switched by pressing the [CURSOR UP/DOWN] soft key.
Move the two cursors to the diagonal points of a new rectangular graphic range. A tool path is drawn next time so that the drawn tool path is contained in this rectangular range.

(3) Press the [EXEC] soft key to end the operation.
After this step, the setting of graphic enlargement is effective to enable drawing with the new setting.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

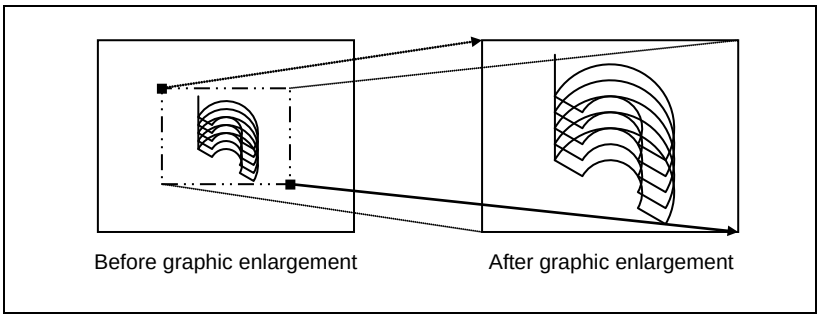


Fig. 13.1 (n) Graphic enlargement

NOTE
1 To stop an enlargement/reduction operation, press the [CANCEL] soft key.
2 Even if you perform an enlargement/reduction operation, the tool path already drawn on the screen is neither moved nor enlarged. The setting for enlargement/reduction becomes effective starting with tool path drawing after you press the [EXEC] soft key.

Limitation

In case of the setting in which coordinate system excluding the amount of travel based on cutter compensation and tool nose radius compensation is displayed (Bit 7 (DAC) of parameter No.3104 is set to 1), compensated tool path by cutter compensation and tool nose radius compensation can't be drawn.
Set bit 7 (DAC) of parameter No.3104 to 0 to display coordinate system not excluding the amount of travel based on cutter compensation and tool nose radius compensation.

- Use of this function with other functions

When virtual MDI key is enabled, this function cannot be used.

13.2 DYNAMIC GRAPHIC DISPLAY

Overview

The dynamic graphic display function has two features:

- Path Drawing
  The path of coordinates specified in a program is drawn on the screen.
  By displaying a travel path on the screen, the path can be checked easily before performing machining actually.

- Animation
  The figure of a workpiece to be machined by a programmed tool movement is drawn.
  By drawing the 3-dimensional figure of a workpiece to be machined in an animation-like way, the intermediate machining process and final figure can be grasped easily.

When machining is performed according to a program, this function can draw a tool path with another program.
This function performs drawing much faster than the graphic display function based on automatic operation, so that a program check can be made more swiftly.
This function is distinguished from automatic program operation as indicated by the following terminology:

Automatic operation	Operation performed for actual machining
Background operation	Virtual operation performed for drawing

FRYER MACHINE SYSTEMS, INC
Rob Zahensky

13.2.1 Path Drawing

Overview

The following tool path drawing screens are used to make various settings and execute drawing:

- GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen
  This screen is used to set data needed for tool path drawing.
- PATH GRAPHIC screen
  This screen is used for tool path drawing.
- PATH GRAPHIC (TOOL POSITION) screen
  This screen enables the position of the tool during operation to be checked on the drawn tool path.

13.2.1.1 GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen

This screen is used to set graphic parameters needed for tool path drawing.
Data set using this screen is made valid by displaying the PATH GRAPHIC screen or executing drawing.
If a tool path is already drawn, the path is erased.
Graphic parameter data once set is preserved even if the power is turned off.

Graphic Parameter (Dynamic Graphic) Screen Procedure

Procedure

1 Press the function key GRAPH (or CSTM GRPH when a small MDI unit is used) to display the GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen.

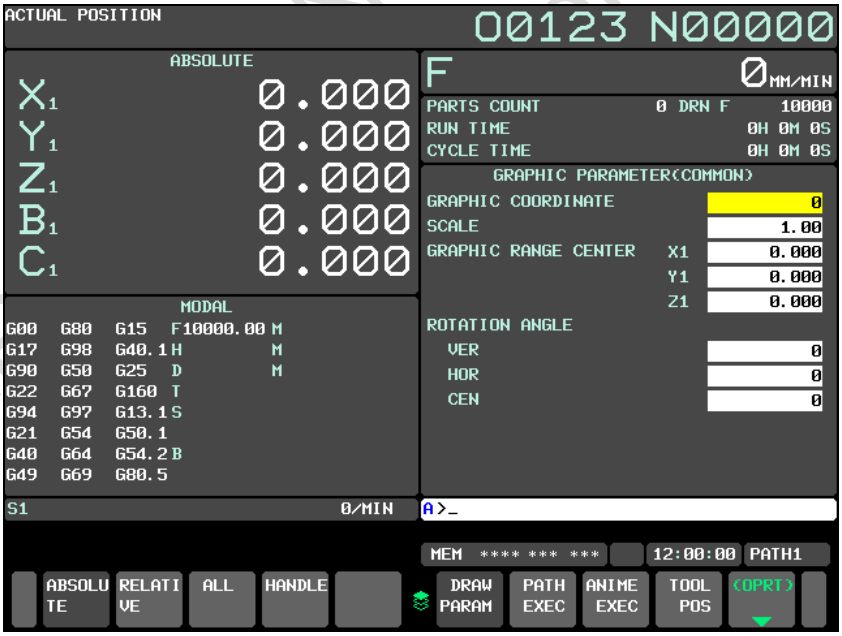


Fig. 13.2.1.1 (a) GRAPHIC PARAMETER screen (first page)

FRYER NE SYSTEMS INC
nsky

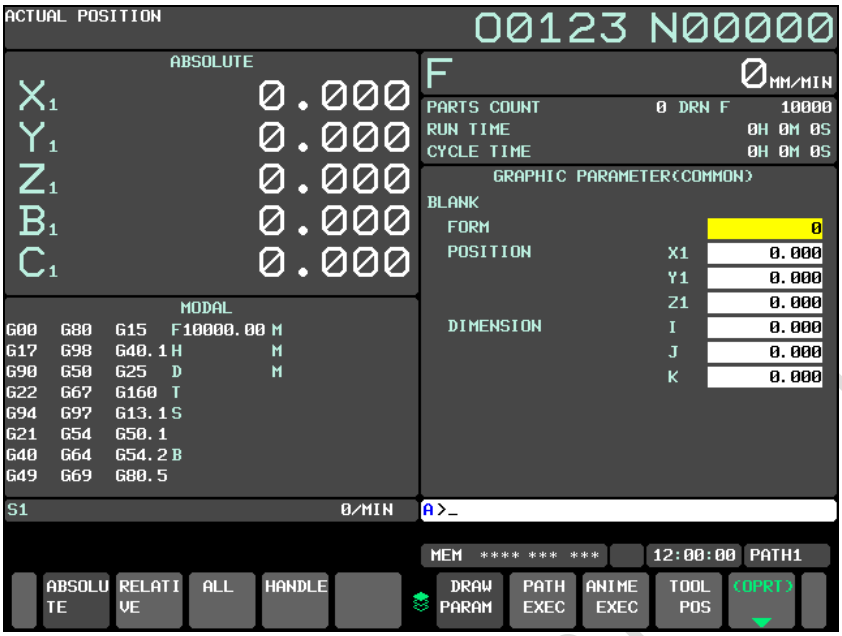


Fig. 13.2.1.1 (b) GRAPHIC PARAMETER screen (second page)

2 Two screens are used for the GRAPHIC PARAMETER screen.
Use the MDI page keys to switch between the screens for display of a desired setting item.

3 Move the cursor to a desired setting item with the MDI cursor keys then key in a desired value with the numeric keys.

4 For direct input of the value set in step 3, press the INPUT key or the [INPUT] soft key.
For incremental input of the value set in step 3, press the [+INPUT] soft key.

For each setting item, see the explanation.

Explanation

The setting items on the GRAPHIC PARAMETER screen are described below.

- Graphic coordinate system

Select a graphic coordinate system for drawing from the following and set its number.

FRYER MACHINE SYSTEMS INC
Rob Zalensky

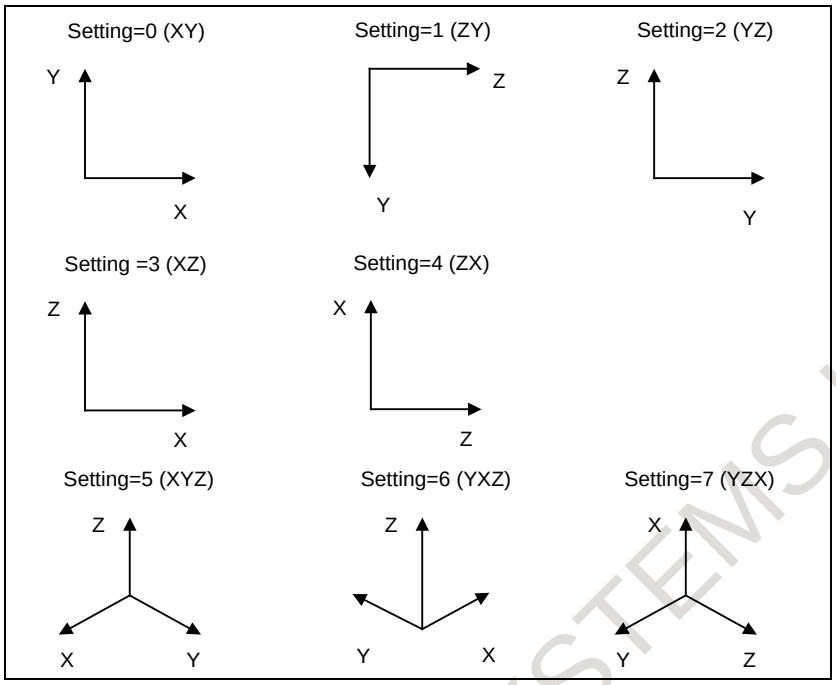


Fig. 13.2.1.1 (c) Graphic coordinate system

- Scale

Set the scale factor for drawing in the range of 0.01 to 100.00 (times).
With a small scale factor, it is possible to draw within a wide range.
With a large scale factor, it is possible to draw in the vicinity of the graphic center being enlarged.

- Graphic range center

To specify the center coordinates of a graphic range, set a coordinate on each axis in the workpiece coordinate system of the program executed for drawing.

NOTE

Set a coordinate on each axis in the machine coordinate system in case of bit 3 (BGM) of parameter No.11329=1.

- Blank figure

With a drawing program, set the figure, position, and dimensions of a blank to be machined.
The graphic range where the blank figure is included in the drawing area is automatically decided by the value set here.

NOTE

1 With a lathe system, no rectangular parallelepiped can be set.
2 With a lathe system, the Y-axis, one of the basic three axes, may not exist. In such a case, the Y-axis for the reference position of a blank need not be set.
3 When a figure or dimension is set, a scale value and screen center coordinates are automatically set as the initial graphic range values so that the blank figure is located within the graphic range on the screen. Until a modification is made to the blank figure, dimension or the setting of the graphic coordinate system, the graphic range is determined by the set scale and screen center coordinates.

Figure

Select a type of blank figure from the following (Table 13.2.1.1 (a)) and set the corresponding value:

FRYER MACHINE SYSTEMS INC
Rob Zahenskv

Table 13.2.1.1 (a)

Setting	Figure
0	Column or cylinder (parallel with the Z-axis)
1	Rectangular parallelepiped

Position

Set the reference position of a blank with coordinates (X,Y,Z) in the workpiece coordinate system.

> **NOTE**
> Set the reference position of a blank with coordinates (X,Y,Z) in the machine coordinate system in case of bit 3 (BGM) of parameter No.11329=1.

Dimensions

Set the dimensions of each type of blank figure as indicated below (Table 13.2.1.1 (b)).

Table 13.2.1.1 (b)

Type of blank figure	Dimension I	Dimension J	Dimension K
Rectangular parallelepiped	Length in X-axis direction	Length in Y-axis direction	Length in Z-axis direction
Column	Diameter of column	0	Length of column
Cylinder	Diameter of outer circle of cylinder	Diameter of inner circle of cylinder	Length of cylinder

The relationship between the position and dimensions of a blank is shown below.

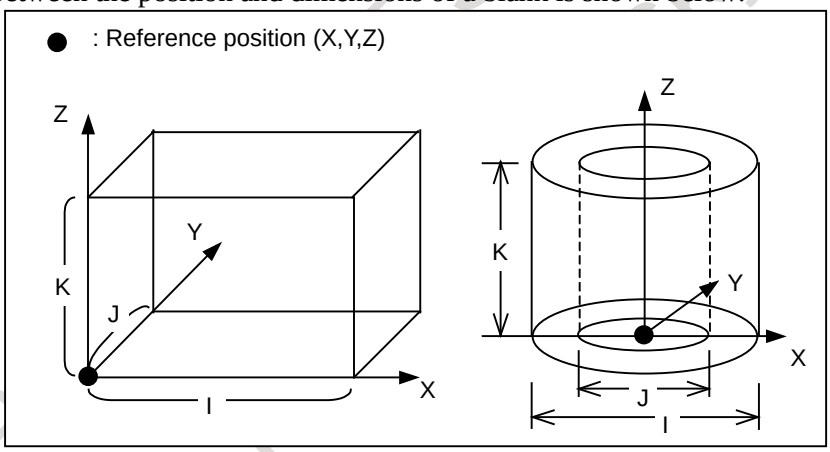


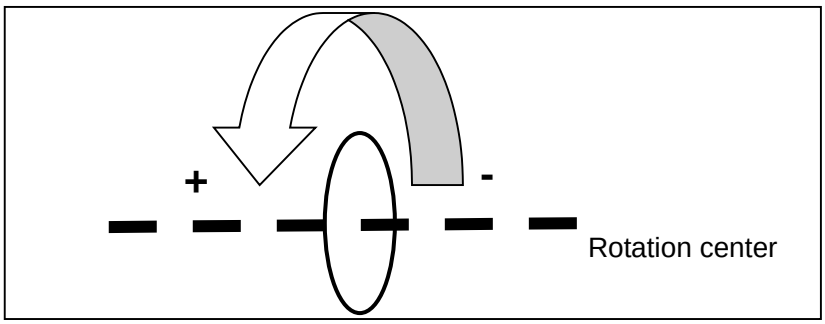
Fig. 13.2.1.1 (d)

- Rotation angle

Set a rotating angle of the graphic coordinate system that centers on the graphic range center. The rotating angle is a range of -360°-+360°.
Set a rotating angle as a reference position (position of the rotating angle 0°) on the indicated direction of each graphic coordinate system.

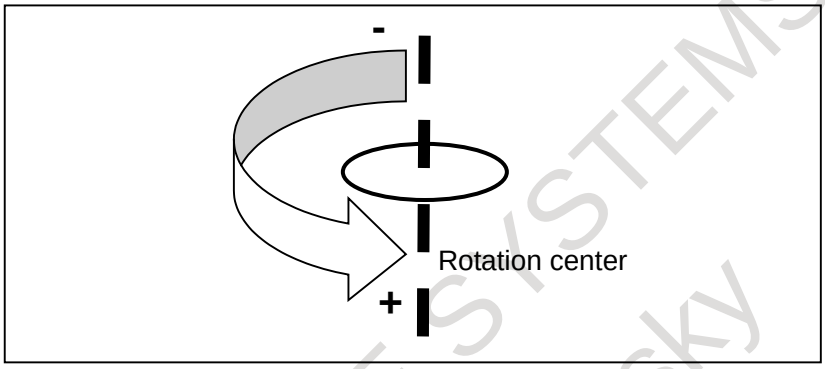
Vertical plane rotation angle

Set a rotating angle at the horizontal direction center in front of the screen.
The rotation direction is as follows.



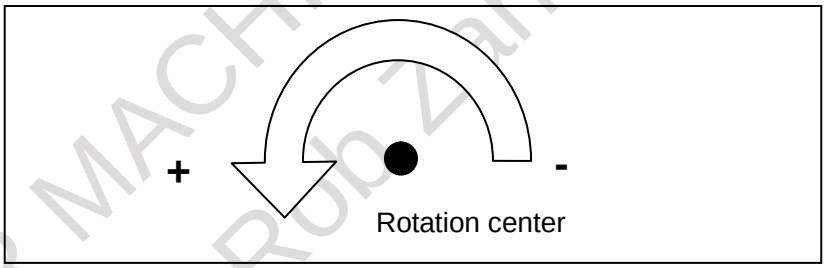
Horizontal plane rotation angle

Set a rotating angle at the vertical direction center in front of the screen.
The rotation direction is as follows.



Screen center rotation angle

Set a rotating angle at the vertical direction center of the screen plane.
The rotation direction is as follows.



- Graphic color

Set colors to be used for tool path drawing.
The colors that can be set are indicated below together with their setting values:

Graphic color	White	Red	Green	Yellow	Blue	Purple	Light blue	White
Setting value	0	1	2	3	4	5	6	7

Path

Set colors to be used for a drawn tool path.

Tool position

Set a graphic cursor color to be used for PATH GRAPHIC (TOOL POSITION) screen.

- Auto erase

When drawing is started with the [START] soft key from a state where drawing is not being executed or is not stopped temporarily, the previously drawn path can be erased.

Setting 0: The previously drawn path is not erased.
1: The previously drawn path is erased.

FOXER MACHINE SYSTEMS INC
Rob Zahensky

- **Tool offset (Path)**

For tool path drawing, whether to enable or disable the tool offset function (tool length compensation, cutter/tool-noise radius compensation) can be selected.

Setting 0: The tool offset function is enabled for drawing.
1: The tool offset function is disabled for drawing.

13.2.1.2 PATH GRAPHIC screen

The PATH GRAPHIC screen is used to draw a tool path. The following operations can be performed:

- Starting/ending tool path drawing
- Rewind of a drawing target program
- Erasing a drawn tool path

The screen is composed of the following items:

(1) Drawing area
(2) Status display of Background operation
(3) Program name and sequence number for drawing execution
(4) Current coordinates (absolute coordinates)
(5) Feedrate and M/S/T/D code instruction information
(6) Graphic coordinate system

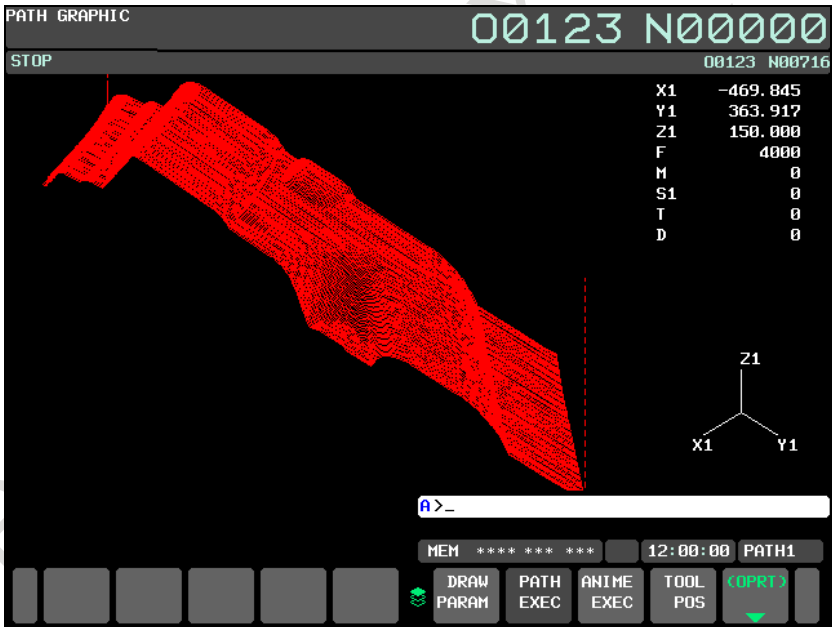


Fig. 13.2.1.2 (a) PATH GRAPHIC screen

FRYER SYSTEMS INC

Path Graphic Screen Procedure

Procedure

1 Press the function key (or when a small MDI unit is used) to display the GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen.

2 Press the [PATH EXEC] soft key. The PATH GRAPHIC screen is displayed.

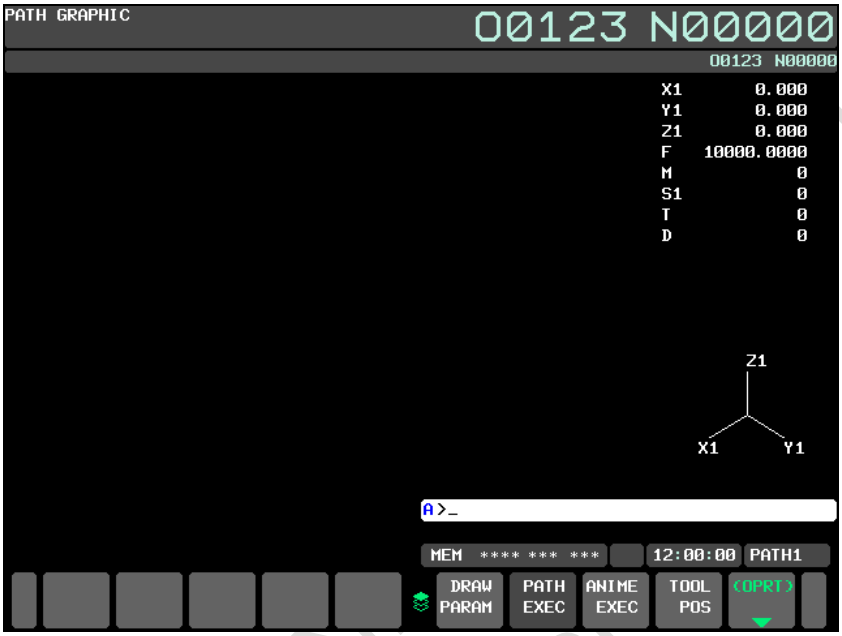


Fig. 13.2.1.2 (b) PATH GRAPHIC screen

3 Press the [(OPRT)] soft key. The soft keys for tool path drawing are displayed.

Fig. 13.2.1.2 (c) PATH GRAPHIC screen (operation)

4 Press the continuous menu key to display the soft keys for enlarging/reducing/moving the graphic range.

Fig. 13.2.1.2 (d) PATH GRAPHIC screen (enlarging/reducing/moving the graphic range)

5 Press the [COORDINATES] soft key to display the soft keys for changing the graphic coordinate system.

Fig. 13.2.1.2 (e) PATH GRAPHIC screen (changing the graphic coordinate system)

6 Press the [ROTATION] soft key to display the soft keys for rotating the graphic coordinate system.

Fig. 13.2.1.2 (f) PATH GRAPHIC screen (rotating the graphic coordinate system)

For the operation of each soft key, see the explanation.

Explanation

- Graphic program selection

Press the [PRGRAM SELECT] soft key to select a drawing target program.
Pressing the [PRGRAM SELECT] soft key displays the program list screen.
On the program list screen, move the cursor to a desired program then press the [DRAW SELECT] soft key.

Fig. 13.2.1.2 (g) Program list screen ([DRAW SELECT] soft key)

Pressing the [DRAW SELECT] soft key selects a drawing target program and switches the screen display to the PATH GRAPHIC screen.

The file whose name is prefixed by a "#" mark on the program list screen is selected as the drawing target program. And the mark of "+" is displayed in the file that has been selected as a program for drawing in other path systems.

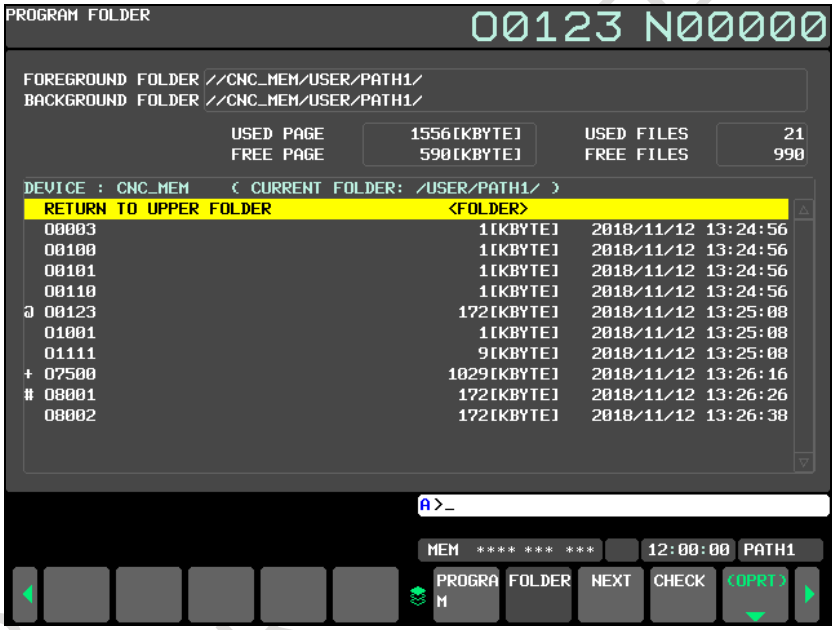


Fig. 13.2.1.2 (h) Program list screen (drawing target program selection state)

> **NOTE**
> The file that can be selected as the drawing target program is only a file that can be selected as the main program.

- Rewind of a drawing target program

If the execution of drawing of a selected program has ended or is stopped halfway, press the [REWIND] soft key to restart drawing from the beginning of the program.

- Starting / Stopping drawing

To draw the tool path of a program selected for drawing, press one of the following soft keys displayed by step 3 mentioned above:

- [START] soft key
  This soft key starts drawing from the head of the program.
- [1BLOCK] soft key
  This soft key executes the program for drawing and stops temporarily on a block-by-block basis as with ordinary single block stop operation.

If a program is executed with a soft key above, the soft key display is changed to the following:

Fig. 13.2.1.2 (i) Soft keys displayed during drawing execution

The operations of these soft keys are as follows:

- [STOP] soft key
  This soft key terminates the execution of the drawing target program to stop drawing.
- [PAUSE] soft key
  This soft key temporarily stops the execution of the drawing target program to stop drawing temporarily.
- [1BLOCK] soft key
  This soft key executes the program for drawing and stops drawing temporarily on a block-by-block basis as with ordinary single block stop operation.
- [RESTART] soft key
  If the [RESTART] soft key is pressed in the stop state set by the [PAUSE] soft key or the [1BLOCK] soft key, the execution of drawing can be restarted from the block where drawing is stopped.

NOTE

When the new path was drawn by the operation of starting drawing without erasing of the old path before the operation, it is impossible to redraw the old path by each operation of enlarging/reducing/moving the graphic range and changing/rotating the graphic coordinate system.

The state of drawing is indicated as follows:

DRAWING: Indicates that drawing is being executed.

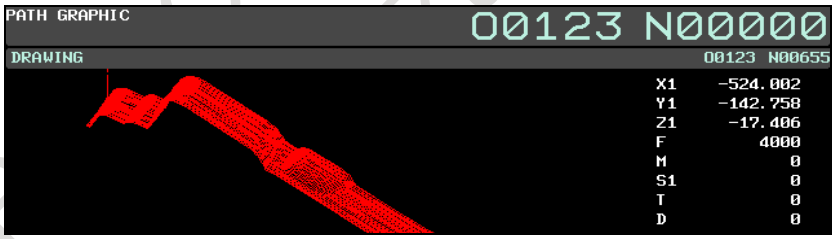


Fig. 13.2.1.2 (j) State indication during drawing

STOP: Indicates that drawing is temporarily stopped.

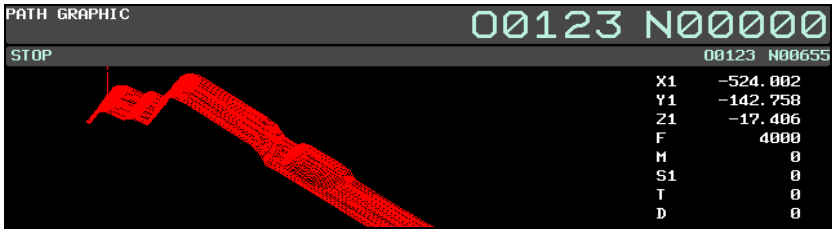


Fig. 13.2.1.2 (k) State indication during temporary stop

ALM: Indicates that occurring of alarm in the Background operation.

FRYER CHINE SYSTEMS INC
Zahensky

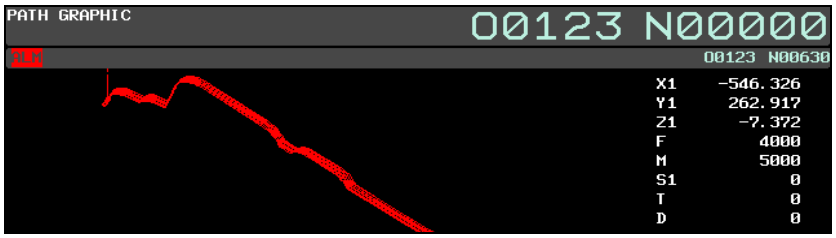


Fig. 13.2.1.2 (l) State indication during occurring of alarm

- End of drawing

When M02 or M30 is executed, the program executed for drawing terminates drawing.
Upon program termination, the soft key display returns to the soft keys (Fig. 13.2.1.2 (c)) displayed before drawing is started.

- Erasing a drawn tool path

Press the [ERASE] soft key to erase a drawn tool path.

> **NOTE**
> 1 If the screen display is switched or the path is switched during tool path drawing, the background operation is stopped to end drawing.
> 2 A tool path once drawn is erased when the screen display is switched or the path is switched.

- Enlarging/reducing the graphic range

The following soft keys displayed by step 4 are used:

- [LARGE] soft key
  This soft key increases the scale to enlarge the graphic range.
- [SMALL] soft key
  This soft key decreases the scale to reduce the graphic range.
- [AUTO] soft key
  When the blank figure specified in a graphic parameter is input, this soft key automatically scales the graphic range so that the blank figure is contained within the graphic range.
  If no blank figure is input, this soft key has no effect.

> **NOTE**
> 1 Set the unit of scale for one enlargement/reduction operation in parameter No. 14713.
> 2 An enlargement/reduction scale used here is set in the graphic parameter for scale.

- Moving the graphic range

The following soft keys displayed by step 4 are used:

- [←MOVE] soft key
  This soft key moves the graphic range to left.
- [MOVE→] soft key
  This soft key moves the graphic range to right.
- [↑MOVE] soft key
  This soft key moves the graphic range upward.
- [↓MOVE] soft key
  This soft key moves the graphic range downward.
- [CENTER] soft key
  This soft key returns the graphic range to the original position.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE
1 Set the travel increment made by one horizontal move operation in parameter No. 14714.
2 Set the travel increment made by one vertical move operation in parameter No. 14715.
3 The graphic range modified here is not set in the graphic parameter for graphic range center.

- Changing the graphic coordinate system

The following soft keys displayed by step 5 are used.
A graphic coordinate system selected here is the same one as set in the graphic parameter for the graphic coordinate system.

- [XY] soft key
  This soft key selects the graphic coordinate system of XY (with a setting of 0).
- [ZY] soft key
  This soft key selects the graphic coordinate system of ZY (with a setting of 1).
- [YZ] soft key
  This soft key selects the graphic coordinate system of YZ (with a setting of 2).
- [XZ] soft key
  This soft key selects the graphic coordinate system of XZ (with a setting of 3).
- [ZX] soft key
  This soft key selects the graphic coordinate system of ZX (with a setting of 4).
- [XYZ] soft key
  This soft key selects the graphic coordinate system of XYZ (with a setting of 5).
- [YXZ] soft key
  This soft key selects the graphic coordinate system of YXZ (with a setting of 6).
- [YZX] soft key
  This soft key selects the graphic coordinate system of YZX (with a setting of 7).
- [OK] soft key
  This soft key changes the current graphic coordinate system to the graphic coordinate system selected by one of the soft keys above.
- [CANCEL] soft key
  This soft key cancels the graphic coordinate system selected by one of the soft keys above to return to the original graphic coordinate system.

NOTE
The graphic coordinate system selected here is set in the graphic parameter for the graphic coordinate system.

- Rotating the graphic coordinate system

The following soft keys displayed by step 6 are used.

- [↑] soft key
  This soft key rotates the graphic coordinate system upward.
- [↓] soft key
  This soft key rotates the graphic coordinate system downward.
- [←] soft key
  This soft key rotates the graphic coordinate system to left.
- [→] soft key
  This soft key rotates the graphic coordinate system to right.
- [CW] soft key
  This soft key rotates the graphic coordinate system clockwise.
- [CCW] soft key
  This soft key rotates the graphic coordinate system counterclockwise.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- [OK] soft key
  This soft key changes the rotation angle of the current graphic coordinate system to the one set by one of the soft keys above.
- [CANCEL] soft key
  This soft key cancels the rotation of the graphic coordinate system performed by one of the soft keys above to return to the original graphic coordinate system.

NOTE
1 Set the travel increment made by one rotation operation in parameter No. 14716.
2 The rotation angle of the graphic coordinate system set here is not set in the graphic parameter for rotation angle.

13.2.1.3 PATH GRAPHIC (TOOL POSITION) screen

The tool position during operation can be checked by displaying the cursor for indicating the tool position during operation on the tool path drawn on the PATH GRAPHIC (EXECUTION) screen.

The screen is composed of the following items:
(1) Drawing area
(2) Current coordinates (absolute coordinates)
(3) Feedrate and M/S/T/D code instruction information
(4) Graphic coordinate system
(5) Cursor to show tool position

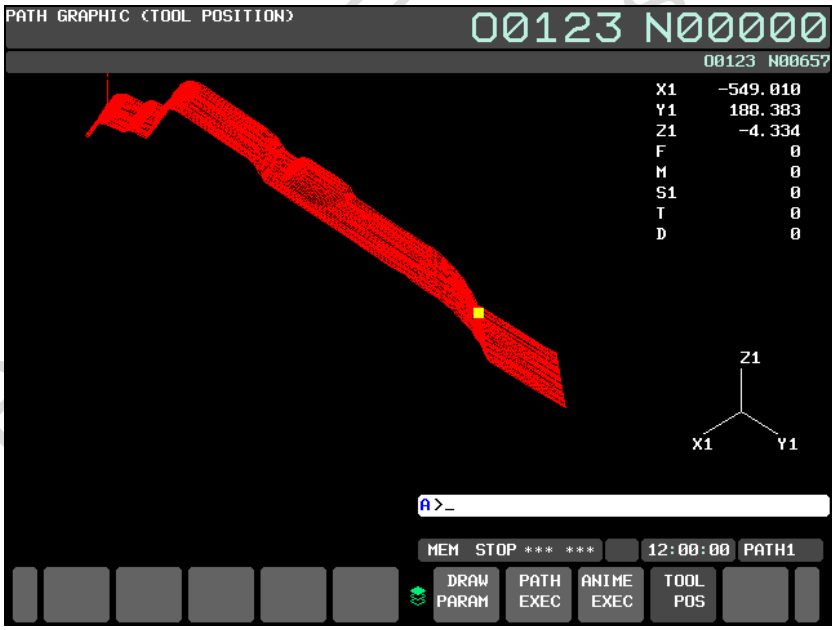


Fig. 13.2.1.3 (a) PATH GRAPHIC (TOOL POSITION) screen

FRYER SYSTEMS INC

Path Graphic (Tool Position) Screen Procedure

Procedure

1 Press the function key (or when a small MDI unit is used) to display the PATH GRAPHIC (PARAMETER) screen.

2 Press the [TOOL POS] soft key.
The screen display changes to the PATH GRAPHIC (TOOL POSITION) screen and the cursor indicating the tool position is displayed.

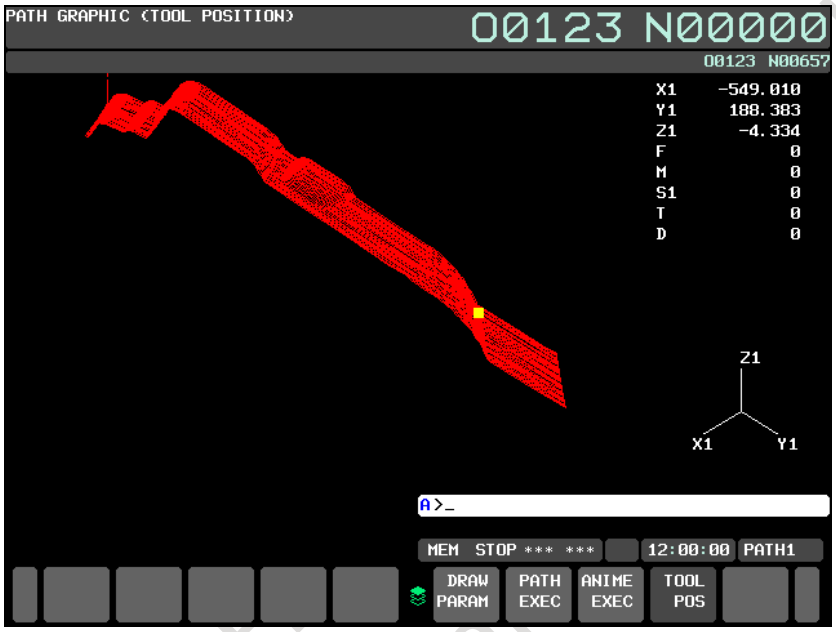


Fig. 13.2.1.3 (b) PATH GRAPHIC (TOOL POSITION) screen

For the method of checking the current tool position, see the explanation.
Pressing a soft key other than the [TOOL POS] soft key displays the corresponding screen.

Explanation

Use the following procedure to check the tool position during operation on the PATH GRAPHIC (TOOL POSITION) screen:

(1) Select a drawing target program for operation.
(2) Draw the tool path of the selected program on the PATH GRAPHIC screen.
(3) After the end of drawing, switch the screen display to the PATH GRAPHIC (TOOL POSITION) screen.
(4) Start automatic operation with the program executed for tool path drawing.

After automatic operation is started, the blinking cursor moves along the drawn tool path as the tool moves. The cursor blinking rate is fast when the tool is moving, and the cursor blinking rate is slow when the tool is stopped.

The following items displayed on the screen are provided for the program under automatic operation:

- Current coordinates
- Feedrate and M/S/T/D code specification information

FRYER MACHINE INC Rob Zak

NOTE

1 A tool path drawn by setting the tool offset parameter to 1 (to disable the tool offset function) is different from the actual tool path. In this case, the cursor indicating the tool position may not move along the drawn tool path.

2 The tool path cannot be drawn correctly if a command not supporting drawing or a command that specifies operation different from operation performed during drawing is specified or if an operation dependent on the operation state or setting on the machine side is performed. So, the tool path in actual operation may differ from a drawn tool path. In this case, the cursor indicating the tool position does not move along the drawn tool path.

3 When the graphic parameter of the graphic coordinate system, scale, graphic range center, blank figure / position / dimensions and rotation angle is changed, the drawn tool path drawn is erased.
Therefore, please draw the tool path again on the PATH GRAPHIC screen to display the tool position by setting the changed graphic parameter.

13.2.2 Animation

Overview

For animation drawing, make necessary settings and perform operations for drawing execution on the following screens:

- GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen
  On this screen, data required to execute animation drawing can be set.
- ANIMATION GRAPHIC screen
  This screen is used for animation drawing.
  On this screen, the graphic range can be reduced/enlarged, and the graphic coordinate system can be rotated.

NOTE

To execute animation drawing, set directions of the axes of the machine (parameter No.14706).

13.2.2.1 GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen

This screen is used to set graphic parameters needed for animation drawing.
Data set using this screen is made valid by displaying the ANIMATION GRAPHIC screen or executing drawing.
Graphic parameter data once set is preserved even if the power is turned off.

Graphic Parameter (Dynamic Graphic) Screen Procedure

Procedure

1 Press the function key

(or

when a small MDI unit is used) to display the GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

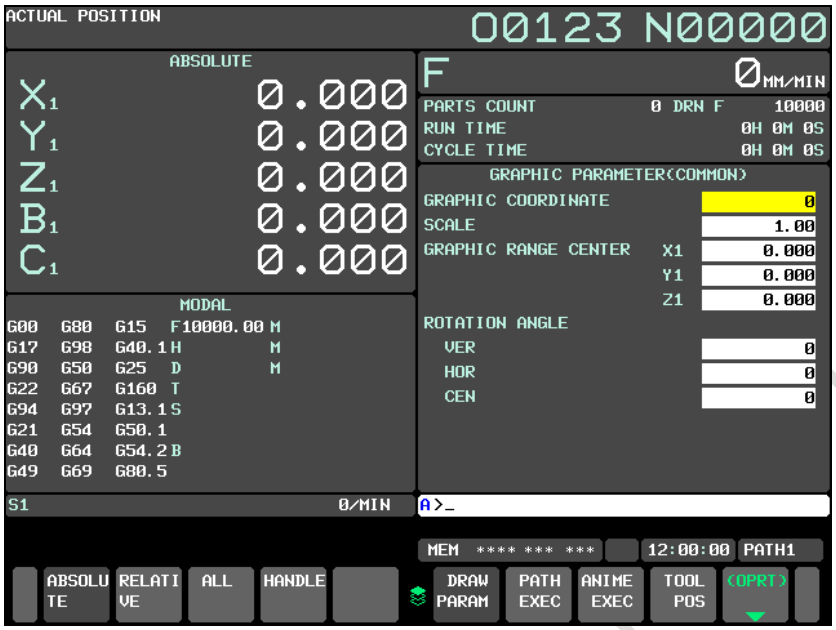


Fig. 13.2.2.1 (a) GRAPHIC PARAMETER screen (first page)

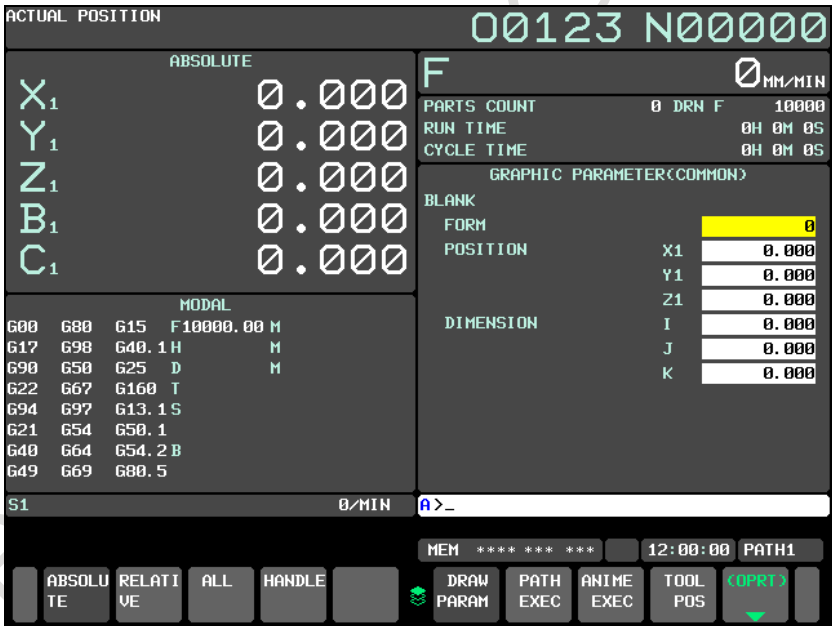


Fig. 13.2.2.1 (b) GRAPHIC PARAMETER screen (second page)

2 Two screens are used for the GRAPHIC PARAMETER screen.
Use the MDI page keys to switch between the screens for display of a desired setting item.

3 Move the cursor to a desired setting item with the MDI cursor keys then key in a desired value with the numeric keys.

4 For direct input of the value set in step 3, press the INPUT key or the [INPUT] soft key.
For incremental input of the value set in step 3, press the [+INPUT] soft key.

For each setting item, see the explanation.

Explanation

The setting items on the GRAPHIC PARAMETER screen are described below.
However, the graphic parameters listed below are shared for tool path drawing. So, refer to the explanation of the GRAPHIC PARAMETER screen for tool path drawing.

- Graphic coordinate system
- Scale
- Graphic range center
- Rotation angles (vertical plane, horizontal plane, screen center)

- Blank figure

With a drawing program, set the figure, position, and dimensions of a blank to be machined.

NOTE

1 With a lathe system, no rectangular parallelepiped can be set.
2 With a lathe system, the Y-axis, one of the basic three axes, may not exist. In such a case, the Y-axis for the reference position of a blank need not be set.
3 When a figure or dimension is set, a scale value and screen center coordinates are automatically set as the initial graphic range values so that the blank figure is located within the graphic range on the screen. Until a modification is made to the blank figure, dimension or the setting of the graphic coordinate system, the graphic range is determined by the set scale and screen center coordinates.

Figure

Select a type of blank figure from the following and set the corresponding value:

Setting	Figure
0	Column or cylinder (parallel with the Z-axis)
1	Rectangular parallelepiped

Position

Set the reference position of a blank with coordinates (X,Y,Z) in the workpiece coordinate system.

NOTE

Set the reference position of a blank with coordinates (X,Y,Z) in the machine coordinate system in case of bit 3 (BGM) of parameter No.11329=1.

Dimensions

Set the dimensions of each type of blank figure as indicated below.

Type of blank figure	Dimension I	Dimension J	Dimension K
Rectangular parallelepiped	Length in X-axis direction	Length in Y-axis direction	Length in Z-axis direction
Column	Diameter of column	0	Length of column
Cylinder	Diameter of outer circle of cylinder	Diameter of inner circle of cylinder	Length of cylinder

The relationship between the position and dimensions of a blank is shown below (Fig. 13.2.2.1 (c)).

FRYER MACHINE SYSTEMS INC
Bob Zaremsky

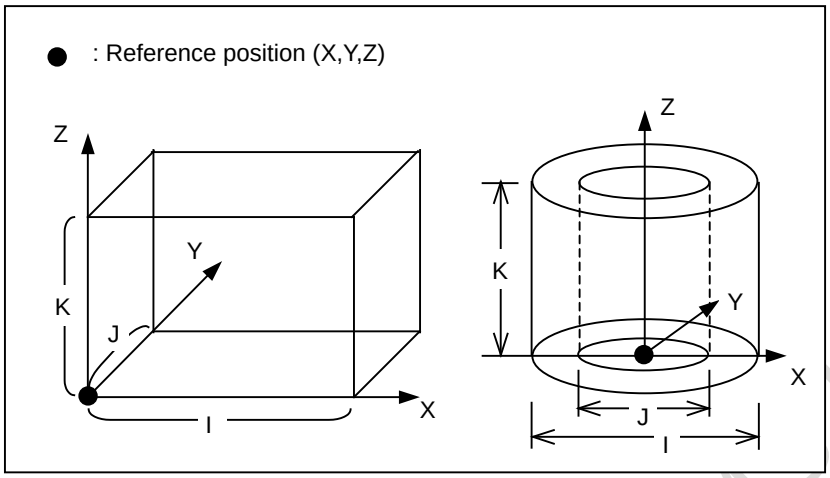


Fig. 13.2.2.1 (c)

- Graphic color

Set colors to be used for animation drawing.
The colors that can be set are indicated below together with their setting values:

Graphic color	White	Red	Green	Yellow	Blue	Purple	Light blue	White
Setting value	0	1	2	3	4	5	6	7

Blank

Set the color of a blank to be drawn in animation drawing.

Tool

Set the color of a tool to be drawn in animation drawing.

- Tool length offset (Anime)

For animation drawing, whether to enable or disable the tool length offset can be selected.
Setting 0: The tool length offset is disabled for drawing.
1: The tool length offset is enabled for drawing.

NOTE
In animation drawing, the cutter/tool-noise radius compensation is always enabled for drawing.

13.2.2.2 ANIMATION GRAPHIC screen

The ANIMATION GRAPHIC screen is used to draw a animation. The following operations can be performed:

- Starting/ending animation drawing
- Rewind of a drawing target program
- Initializing a blank
- Enlarging/reducing/moving the graphic range
- Rotating the graphic coordinate system

The screen configuration is as follows:

(1) Drawing area
(2) Status display of Background operation
(3) Program name and sequence number for drawing execution
(4) Current coordinates (absolute coordinates)
(5) Feedrate and M/S/T/D code instruction information

FRYER MACHINE SYSTEMS INC
Rob Zahensky

(6) Graphic coordinate system
(7) Material position, Material dimension, Scale, Graphic range center of graphic Parameter
(8) Tool name

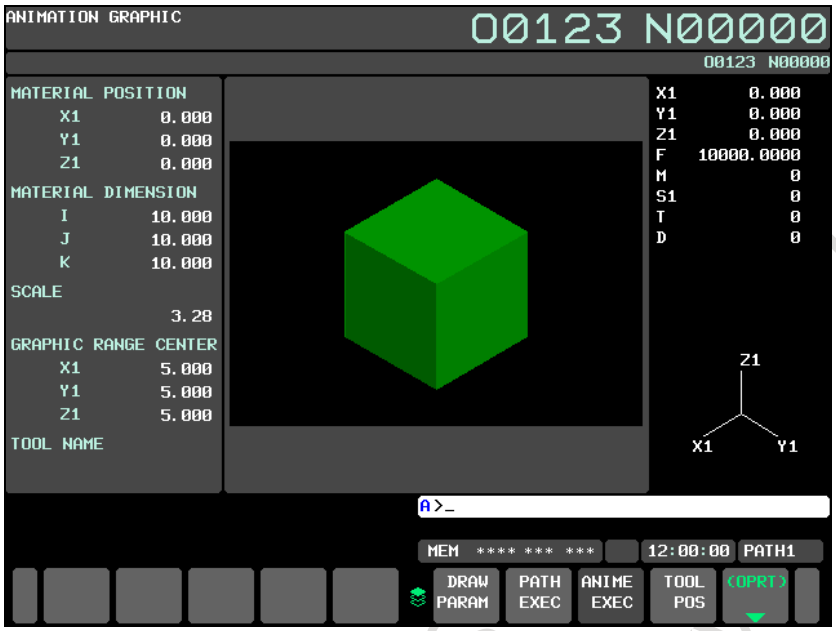


Fig. 13.2.2.2 (a) ANIMATION GRAPHIC screen

Animation Graphic Screen Procedure

Procedure

1 Press the function key (or when a small MDI unit is used) to display the GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen.

2 Press the [ANIME EXEC] soft key. The ANIMATION GRAPHIC screen is displayed.

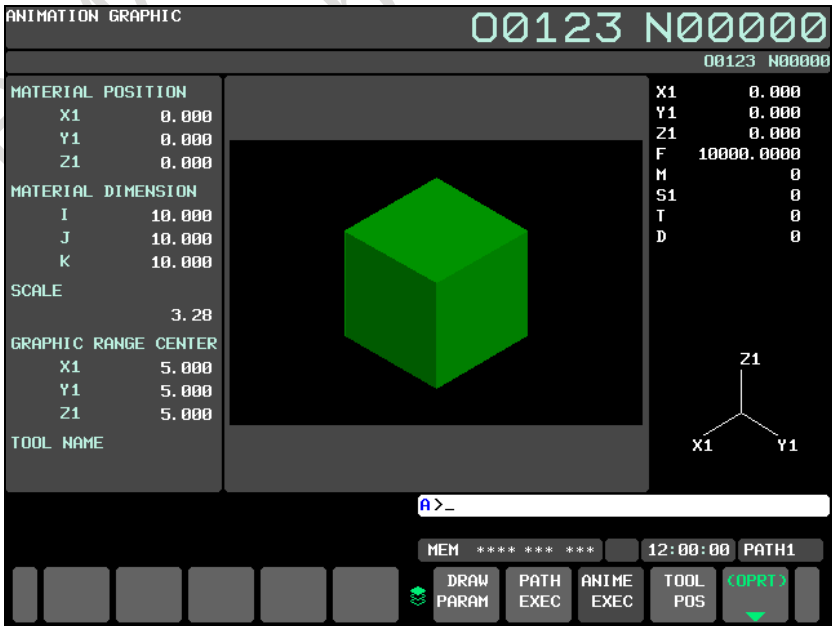


Fig. 13.2.2.2 (b) ANIMATION GRAPHIC screen

FRYER MACHINE S INC
b Zahensky

3 Press the [(OPRT)] soft key. The soft keys for tool path drawing are displayed.

Fig. 13.2.2.2 (c) ANIMATION GRAPHIC screen (operation)

4 Press the continuous menu key ▷ to display the soft keys for enlarging/reducing/moving the graphic range.

Fig. 13.2.2.2 (d) ANIMATION GRAPHIC screen (enlarging/reducing/moving the graphic range)

5 Press the [COORDINATES] soft key to display the soft keys for changing the graphic coordinate system.

Fig. 13.2.2.2 (e) ANIMATION GRAPHIC screen (changing the graphic coordinate system)

6 Press the [ROTATION] soft key to display the soft keys for rotating the graphic coordinate system.

Fig. 13.2.2.2 (f) ANIMATION GRAPHIC screen (rotating the graphic coordinate system)

For the operation of each soft key, see the explanation.

Explanation

The operations listed below are the same operations as for the tool path drawing screen. See the explanation of the tool path drawing screen.

- Graphic program selection
- Rewind of a drawing target program
- Starting / Stopping of drawing
- End of drawing

- Initializing a blank

Press the [INIT.] soft key to initialize and return a drawn machining profile to the original blank figure.

NOTE

1 Blank initialization can also be performed by any of the following operations:
   - Start of drawing
   - Changing of the graphic coordinate system and graphic range by performing enlargement/reduction/movement/rotation operations
   - Change of screen

2 A machining profile once drawn for a blank is erased when the screen or the path is switched.

- Enlarging/reducing the graphic range

The following soft keys displayed by step 4 are used:

- [LARGE] soft key
  This soft key increases the scale to enlarge the graphic range.
- [SMALL] soft key
  This soft key decreases the scale to reduce the graphic range.

FRYER MACHINE SYSTEMS INC Rob Zahensky

- [AUTO] soft key
  When the blank figure specified in a graphic parameter is input, this soft key automatically scales the graphic range so that the blank figure is contained within the graphic range.
  If no blank figure is input, this soft key has no effect.

NOTE
1 Set the unit of scale for one enlargement/reduction operation in parameter No. 14713.
2 An enlargement/reduction scale used here is set in the graphic parameter for scale.

- Moving the graphic range

The following soft keys displayed by step 4 are used:
- [←MOVE] soft key
  This soft key moves the graphic range to left.
- [MOVE→] soft key
  This soft key moves the graphic range to right.
- [↑MOVE] soft key
  This soft key moves the graphic range upward.
- [↓MOVE] soft key
  This soft key moves the graphic range downward.
- [CENTER] soft key
  This soft key returns the graphic range to the original position.

NOTE
1 Set the travel increment made by one horizontal move operation in parameter No. 14714.
2 Set the travel increment made by one vertical move operation in parameter No. 14715.
3 The graphic range modified here is not set in the graphic parameter for graphic range center.

- Changing the graphic coordinate system

The following soft keys displayed by step 5 are used.
A graphic coordinate system selected here is the same one as set in the graphic parameter for the graphic coordinate system.
- [XY] soft key
  This soft key selects the graphic coordinate system of XY (with a setting of 0).
- [ZY] soft key
  This soft key selects the graphic coordinate system of ZY (with a setting of 1).
- [YZ] soft key
  This soft key selects the graphic coordinate system of YZ (with a setting of 2).
- [XZ] soft key
  This soft key selects the graphic coordinate system of XZ (with a setting of 3).
- [ZX] soft key
  This soft key selects the graphic coordinate system of ZX (with a setting of 4).
- [XYZ] soft key
  This soft key selects the graphic coordinate system of XYZ (with a setting of 5).
- [YXZ] soft key
  This soft key selects the graphic coordinate system of YXZ (with a setting of 6).
- [YZX] soft key
  This soft key selects the graphic coordinate system of YZX (with a setting of 7).

FRIEP MACHINE SYSTEMS INC
Rob Zahensky

- [OK] soft key
  This soft key changes the current graphic coordinate system to the graphic coordinate system selected by one of the soft keys above.
- [CANCEL] soft key
  This soft key cancels the graphic coordinate system selected by one of the soft keys above to return to the original graphic coordinate system.

NOTE
The graphic coordinate system selected here is set in the graphic parameter for the graphic coordinate system.

- Rotating the graphic coordinate system

The following soft keys displayed by step 6 are used.

- [↑] soft key
  This soft key rotates the graphic coordinate system upward.
- [↓] soft key
  This soft key rotates the graphic coordinate system downward.
- [←] soft key
  This soft key rotates the graphic coordinate system to left.
- [→] soft key
  This soft key rotates the graphic coordinate system to right.
- [CW] soft key
  This soft key rotates the graphic coordinate system clockwise.
- [CCW] soft key
  This soft key rotates the graphic coordinate system counterclockwise.
- [OK] soft key
  This soft key changes the rotation angle of the current graphic coordinate system to the one set by one of the soft keys above.
- [CANCEL] soft key
  This soft key cancels the rotation of the graphic coordinate system performed by one of the soft keys above to return to the original graphic coordinate system.

NOTE
1 Set the travel increment made by one rotation operation in parameter No. 14716.
2 The rotation angle of the graphic coordinate system set here is not set in the graphic parameter for rotation angle.

- Drawing of a tool

In animation drawing, not only a blank figure but also a tool figure is drawn in three dimensions.

NOTE
The following function is necessary to draw in the tool figure.
- Tool geometry size data 100/300 pairs

Figure data

For tool figure drawing, the following data is used:

- Tool geometry size data 100/300 pairs
- Tool compensation data (tool-nose radius compensation value, virtual tool tip direction, cutter compensation value)
- Parameters

The table below indicates the drawable types of tools and figure data of each type of tool.

FRYER MACHINE SYSTEMS INC
Bob Zaharsky

Table 13.2.2.2 (a) List of drawable tools (for turning)

Tool name	Tool geometry size data	Tool compensation	Parameter No.	
General-purpose tool	Setting Cutting edge angle Tool angle		Tip position Cutting edge length Holder length Holder width Holder length 2 Holder width 2	No.27350#0 No.27351 No.27352 No.27353 No.27354 No.27355
Threading tool	Setting Tool angle		Tip position Cutting edge width Holder length Holder width	No.27356#0 No.27357 No.27358 No.27359
Grooving tool	Setting Tip width Tip length		Tip position Holder length Holder width	No.27360#0 No.27361 No.27362
Round-nose tool	Setting Tip length	Tool-nose radius Virtual tool tip	Tip position Holder length Holder width	No.27363#0 No.27364 No.27365
Point nose straight tool	Setting Cutting edge angle Tool angle		Tip position Cutting edge length Holder length Holder width Holder length 2 Holder width 2	No.27366#0 No.27367 No.27368 No.27369 No.27370 No.27371
Versatile tool	Setting Tip width Tip length		Tip position Holder length Holder width	No.27384#0 No.27385 No.27386

Table 13.2.2.2 (b) List of drawable tools (for milling)

Tool name	Tool geometry size data	Cutter compensation	Parameter No.	
Drill	Setting Tool angle	Diameter	Tip length	No.27372
counter sink tool	Setting Cutter diameter	Small diameter	Tool angle Tip length Cutter length Shank length Shank diameter	No.27375 No.27376 No.27377 No.27378 No.27379
Flat end mill	Setting	Diameter	Tip length	No.27373
Ball end mill	Setting	Diameter	Tip length	No.27380
Tap	Setting	Diameter	Tip length	No.27374
Reamer	Setting	Diameter	Tip length	No.27381
Boring tool	Setting	Diameter	Tip length	No.27382
Face mill	Setting	Diameter	Tip length	No.27383

NOTE

In lathes system, the round-nose tool and milling tools can be drawn only when the cutter compensation/tool-nose radius compensation function is provided.

FRYER MACHINE SYSTEMS INC
Ron Zahensky

Timing of drawing

If a code indicated below is specified during drawing execution, a tool figure is drawn according to the tool geometry size data that has the same number as the code.

- With a lathe system
  T code command
  If three numbers, namely, a tool number, tool geometry offset number, and tool wear offset number, are specified in a T code, the tool geometry size data that has the same number as the tool geometry offset number is obtained.

NOTE
When the tool management function is provided, a tool offset is specified with a D code. So, the tool geometry size data that has the offset number indicated by the D code is obtained.

- With a machining center system
  D code command

If the tool geometry size data corresponding to a specified number does not exist or the tool geometry size data is not set correctly, tool drawing is disabled with the warning "ILLEGAL SETTING OF TOOL FIGURE DATA".

13.2.3 Programmable Data Input (G10) for Blank Figure Drawing Parameters

Overview

Each of the drawing parameters for a blank figure, a position, and dimensions that are for the automatic scaling of the drawing areas of tool path drawing and animation drawing with the dynamic graphic display function can be set with an NC program, using the programmable data input command (G10 command).

Format

G10 L90 P_ IP_ I_ J_ K_ ;

P_ : Blank figure
IP_ : Addresses of the basic three axes and the reference position of a blank
I_,J_,K_ : Dimensions of a blank

NOTE
1 This command is enabled only during the execution of drawing. It is ignored during normal automatic operation.
2 This command is a one-shot G code.
3 This command must be specified in a single block.

Explanation

- Blank figure (P_)

Specify the type of a blank figure with either of the following settings for shapes.

Setting	Figure
0	Column or cylinder (parallel with the Z-axis)
1	Rectangular parallelepiped

The specified value is set in parameter No. 11343.

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

- Reference position of a blank (IP_)

Specify the reference position of a blank with coordinates (X,Y,Z) in the workpiece coordinate system.

> **NOTE**
> Specify the reference position of a blank with coordinates (X,Y,Z) in the machine coordinate system in case of bit 3 (BGM) of parameter No.11329=1.

The specified value is set in parameter No. 11344.

- Dimensions of a blank (I_,J_,K_)

For the shape of each blank, specify the dimensions of the blank as follows:

Blank figure	Address I	Address J	Address K
Rectangular parallelepiped	Length in X-axis direction	Length in Y-axis direction	Length in Z-axis direction
Column	Diameter of column	0	Length of column
Cylinder	Diameter of outer circle of cylinder	Diameter of inner circle of cylinder	Length of cylinder

The specified value are set in parameter No.11345 (address I), parameter No.11346 (address J), and parameter No.11347 (address K).

- Relationship between the position and dimensions of a blank

The relationship between the position and dimensions of a blank is shown in the figure below (Fig. 13.2.3).

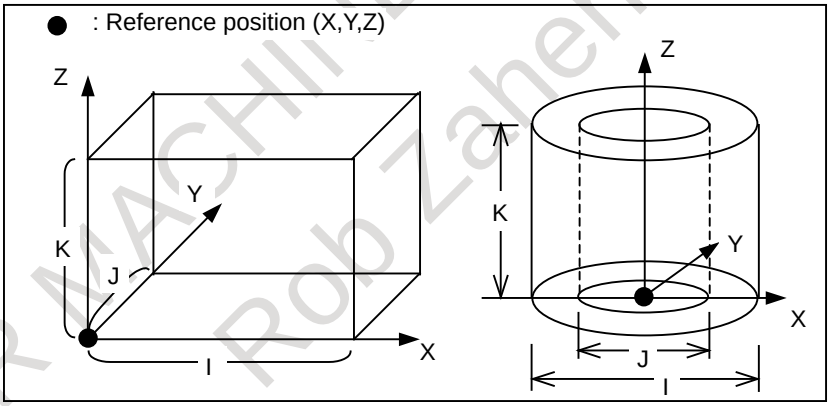


Fig. 13.2.3

- Operation to be performed when this command is issued

When this command is executed in the background operation, the specified values are set in the drawing parameters for the blank figure, reference position, and dimensions that correspond to the specified arguments, and the drawing area is automatically set again with the values set.
This causes this command and subsequent commands to perform drawing in the new drawing area.

> **NOTE**
> Before this command, drawing is performed as described below.
> - The tool path is erased.
> - In animation drawing, the blank figure is initialized.
> - The drawing start position and the tool figure become ineffective.
>
> Thus, issue this command at the start of a drawing program.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

13.2.4 Warning Messages

Warning message	Content
START REJECTED	This program cannot be drawn.
NO PROGRAM SELECTED	No drawing target program is selected.
UNAVAILABLE COMMAND IS IN DRAWING	An NC statement/macro statement that can not execute drawing is instructed.
ILLEGAL SETTING OF GRAPHIC PARAMETERS	Graphic parameters are not set correctly.
ILLEGAL SETTING OF TOOL FIGURE DATA	(a) Tool figure data are not set correctly (b) Tool figure data don't exist (c) Parameter No.14706 is not set

13.2.5 Note

NOTE

1 The coordinates used in drawing are absolute coordinates.
Therefore, even if the coordinate system is changed while it is drawing, it draws in the coordinate system when having begun to draw.
However, if bit 3 (BGM) of parameter No.11329 is set to 1, it draws by the machine coordinates value.

2 The drawing target axes are the basic three axes.
The basic three axes are the X-axis, Y-axis, and Z-axis on the basic coordinate system set with parameter No.1022.

3 In a drawing program having a sequence of contiguous small blocks, drawing requires longer processing time, possibly causing machining time to become longer than actual machining time.

4 When drawing is executed, data is treated as described below.

(1) Parameters
The same parameters as for automatic operation are used. However, parameters cannot be rewritten with a command such as the G10 command. If an attempt is made to rewrite a parameter with programmable parameter input (G10L52), the command is ignored.
However, by setting bit 7 (GST) of parameter No.11329, drawing can be temporarily stopped with the warning "UNAVAILABLE COMMAND IS IN DRAWING".

(2) Tool compensation value/workpiece origin offset/extended workpiece origin offset/macro variable, etc.
For each of drawing execution and automatic operation, separate data is used. At the start of drawing, data for drawing is produced by copying data for automatic operation. In subsequent operations, the data for drawing and the data for automatic operation are treated independently of each other. So, even if data is rewritten with a command such as G10, drawing execution and automatic operation do not affect each other. Note, however, that data rewritten in drawing execution is not reflected in the data for automatic operation but is deleted.

(3) Tool life management data, tool number offset data
These data items are not used for drawing execution. So, if an attempt is made to rewrite data with G10, the command is ignored.
However, by setting bit 7 (GST) of parameter No.11329, drawing can be temporarily stopped with the warning "UNAVAILABLE COMMAND IS IN DRAWING".

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

13.2.6 Restrictions

- Simultaneous drawing with multiple paths

This function does not support simultaneous tool path drawing by program execution with multiple paths. For example, the tool paths of programs executed simultaneously with multiple paths on a 2-path or 3-path lathe or a combined machine tool cannot be drawn. This function supports only drawing executed singly with a single path.

- Simultaneous multi-path display

The screens of this function do not support simultaneous multi-path display based on the setting of parameter No. 13131 and No. 13132.

- Functions that operate differently in drawing execution and automatic operation

The operations of the following functions in drawing execution differ from the operations in automatic operation:

1. Operations that differ, depending on the custom macro
   1) Interface signal
      #1000 to #1035 are assumed to be 0 at all times.
   2) Message output
      A message to be output with #3006 is not displayed but is ignored.
   3) clock
      #3001 and #3002 are ignored. So, note that drawing is not continued, for example, if the following is specified:
      #3001=0 ;
      WHILE [#3001 LE 100] DO1 ;
      END1 ;
   4) Mirror image
      #3007 is assumed to be 0 at all times.
   5) Program-restarted state
      #3008 is assumed to be 0 at all times.
   6) External output command
      BPRNT, DPRNT, POPEN, and PCLOS are ignored.

2 Functions that perform partly different operations
   1) When G28 (automatic reference position return) is specified, up to the intermediate point is drawn.
   2) When G29 (automatic return from the reference position) is specified, drawing is performed from the intermediate point.
   3) When G27 (reference position return check) is specified, no reference position return check is made.
   4) No stored stroke limit check is made.
   5) When G31 (skip function), or G31.1, G31.2, or G31.3 (multistage skip function) is specified, drawing is performed up to the specified position, regardless of the skip signal.
   6) When G60 (single direction positioning, only Machining center system) is specified, drawing is performed directly up to the specified position at all times even if the positioning direction is opposite.

3. Functions that perform different operations
   If the following functions are specified, the operations described below result:

G07.1 (cylindrical interpolation)
   Linear interpolation is performed only with linear axes.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Functions that do not support drawing

The following functions are ignored in drawing execution:

1) G04 (dwell)
2) G20, G21 (inch/metric switch)
3) Auxiliary function (M, S, T, B)
4) G22, G23 (stored stroke limit check on/off)
5) G10.6 (tool retract data setting)
6) G25/G26 (spindle variation detection on/off)
7) G10 (programmable data input)

NOTE

If G10 (programmable data input) is specified, drawing can be temporarily stopped with the warning "UNAVAILABLE COMMAND IS IN DRAWING" by setting bit 7 (GST) of parameter No.11329.

The following functions are warning in drawing execution:

1) M198 (external subprogram call)
2) P-CODE variables after #30000 used in execution macros

- Functions that support drawing

The operations performed with the following program commands can be drawn:

1) G00 (Positioning)
2) G01 (Linear interpolation)
3) G02/G03 (Circular interpolation)
   However, helical interpolation with the instruction axis other than basic three axes cannot be drawn.
4) G17/G18/G19 (Plane selection)
5) G33/G34 (Threading)
   However, it is drawn as linear interpolation.
6) G35/G36 (Circular threading, only Lathes system)
   However, it is drawn as circular interpolation.
7) G40/G41/G42 (Cutter compensation/cancel)
8) G52 (Local coordinate system)
9) G53 (Machine coordinate system selection)
10) G54 to G59 (Workpiece coordinate system selection)
11) G54.1 (Extended workpiece coordinate system selection)
12) G65 (Macro call)
13) G68/G69 (Coordinate system rotation, 3-dimensional coordinate system conversion/cancel)
14) G90/G91 (Absolute/incremental command)
15) G92 (Workpiece coordinate system change)
16) G92.1 (Workpiece coordinate system preset)
17) G94/G95 (Feed per minute/Feed per rotation)
18) G96/G97 (Constant surface speed control/cancel)
19) M98 (Subprogram call)
20) G90/G92/G94 (Canned cycle, only Lathes system)
21) G71/G72/G73/G74/G75/G76/G70 (Multiple repetitive cycle, only Lathes system)
22) G83/G84G85/G87/G87/G88/G80 (Canned cycle for drilling, only Lathes system)
23) G73/G74/G76/G81/G82/G83/G84/G85/G86/G87/G88/G89/G80
    (Canned cycle for drilling, only Machining center system)

NOTE

1 When a lathe system is used, refer to the G codes above as codes of G code system B.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

NOTE

2 It is possible to draw with the G68 (Coordinate system rotation, 3-dimensional coordinate system conversion) instruction only in the tool path drawing.
And, the display of coordinates when instructing in G68 is a coordinate value on the workpiece coordinate system.

3 In threading cycle of canned cycle and multiple repetitive cycle, movement of thread chamfering is not drawn.

4 In animation drawing, shape by the movement of back boring cycle command is different from actual shape.

5 In animation drawing, movement to shift amount at the bottom of a hole in the fine boring cycle and back boring cycle command is not drawn by setting bit 1 (ABC) of parameter No.11349=0.

- Operations that do not support drawing

The following operations cannot be drawn:

1) Backward movement based on the retrace function
2) Forward movement/backward movement/resumed forward movement based on the manual handle retrace function
3) Movement made with skip signal input for a skip command or multistage skip command
4) Movement on an axis based on PMC axis control
5) Movement on an axis based on real-time custom macro specification
6) Operation based on manual interrupt, manual handle interrupt, etc.
7) Operation based on synchronous control, composite control, and superimposed control
8) Operation based on mirror image of facing tool posts
9) Operation based on the 5-axis machining function
10) Operation based on the high-speed, high-precision function (AI contour control I and AI contour control II)
11) Operation based on the offset memory switching function
12) Operation based on Cs contour control
13) Operation based on Spindle positioning.

T

- Operation of the polar coordinate interpolation command

- Paths that can be drawn in polar coordinate interpolation mode
  In polar coordinate interpolation mode, the paths due to the move commands of the following axes, which make up the polar coordinate interpolation plane, are drawn.

Plane	Linear axis	Rotary axis (hypothetical axis)
X-C	X axis of basic 3 axes	C axis (Center of rotation is Z axis of basic 3 axes)

  In the above configuration, set the axis number of the rotation axis to be drawn in parameter No. 14717.
  If a linear axis (parameter No. 5460) and a rotation axis (parameter No. 5461) other than those of the above configuration are set, drawing is not performed correctly.

- Drawing due to the move commands of axes other than those of the polar coordinate interpolation plane in polar coordinate interpolation mode
  In polar coordinate interpolation mode, the paths due to the move commands of the axes making up the above-mentioned polar coordinate interpolation plane and of the axes perpendicular to that plane (those of the basic three axes) are drawn.
  The paths due to the move commands of axes not relating to polar coordinate interpolation cannot be drawn.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- Cancellation of polar coordinate interpolation mode in drawing operation
  The cancellation of polar coordinate interpolation mode in drawing operation is performed with the following:
  - Execution of the G13.1 command in drawing operation
  - Termination of drawing operation

- Origin point of the polar coordinate interpolation plane in drawing
  If bit 3 (BGM) of parameter No. 11329 is 0, the coordinates used for drawing are absolute ones, so that even if the coordinate system is changed with the workpiece coordinate system or local coordinate system command, the origin point of the coordinate system at the time drawing is started is the origin point of the polar coordinate interpolation plane.

- Display of the current position on the drawing screen
  As the current position in polar coordinate interpolation mode, the specified position in the Cartesian coordinate system is displayed.

NOTE
If the polar coordinate interpolation function is enabled and a non-zero axis number is set for the rotation axis (parameter No. 14717), the coordinates of the axis number set for the rotation axis are displayed as the coordinates of the rotation axis.

- Rotation center for a blank figure during animation drawing
  The origin point of the Cartesian coordinate system of the polar coordinate interpolation plane is the origin of the workpiece coordinate system or the local coordinate system at the time polar coordinate interpolation mode (G12.1) is specified. Thus, as the reference point position of the blank to be set as the drawing parameter blank (position), set a position by considering that rotation is performed with the origin point of that coordinate system as the center.
  For example, if a cylindrical shape is set as the drawing parameter blank (figure), set the reference point position of the blank so that the Z-axis of the above-mentioned coordinate system is at the rotation center of the cylinder.

- Drawing on the tool path graphic (tool position) screen
  If the tool position is drawn on the tool path graphic (tool position) screen through automatic operation, the tool position due to the movement of a rotation axis in polar coordinate interpolation mode cannot be drawn.
  In polar coordinate interpolation mode, only movements along linear axes are drawn.

- Operation of the P-CODE workpiece number search function in the background operation

When drawing is started with the setting of bit 3 (WNS) of parameter No. 11349, the execution macro specified in system variable #8610 is executed before the drawing program.

- Calling conditions and operations
  a) Bit 6 (PWSR) of compilation parameter No. 9002 is set to 1.
  b) The program number of an execution macro is set in system variable #8610 before the start of drawing.

  If the conditions above are met, pressing soft key [START] or [1BLOCK] to execute drawing causes the following operations to be performed:
  (1) The execution macro specified with system variable #8610 is called.
  If bit 4 (P98) of compilation parameter No. 9163 is 0, the execution macro is called with an operation equivalent to a simple call (G65), and if P98 is 1, it is called with an operation equivalent to a subprogram call (M98).

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

(2) When the execution macro terminates, the drawing program is executed.

NOTE

1 This function calls an execution macro if the conditions described in a) and b) are met, regardless of the program selected. Thus, if using this function to perform drawing with an execution macro, modify the program of the execution macro so that macro statements, auxiliary functions, and other commands in the execution macro that are not necessary for drawing are not executed, by referring to the system variable below.

System variable	Value	Execution state
#3010	1	Normal state (if drawing is not executed)
	-1	Drawing execution

2 System variable #8610, referenced during the execution of drawing, is the same as that used during automatic operation. If, therefore, the value of the variable is changed to execute drawing, it is necessary to return the variable to the original value before automatic operation is performed.
Also, automatic operation is performed with the variable value at the time drawing is started and, therefore, caution is required so that the value set is not changed during automatic operation.

- Nesting and local variables
  Nesting is accumulated in an execution macro. An execution macro is called with an operation equivalent to a simple call (G65) or a subprogram call (M98) and, therefore, other execution macros can be called from the called execution macro with all calling methods.
  In the case of an operation equivalent to a simple call (G65), the level of local variables is changed to that of the drawing program. That is, the drawing program do not inherit the local variables used in the execution macro.
  In the case of an operation equivalent to a subprogram call (M98), the local variables at the same level as the drawing program are used. That is, the drawing program inherits the local variables used in the execution macro.

- Drawing start position

In tool path drawing, if G92, G52, or G92.1 (machining center system) or G50, G52, or G50.3 (lathe system) is specified at the start of a drawing target program, the position specified with the G code is the drawing start position. If none of these G codes is specified, the end point of the first moving command is the drawing start position.
In animation drawing, the current position when the tool figure is displayed is the drawing start position.

- Use of this function with other functions

When this function is specified, the following functions cannot be used:
- Graphic display
- MANUAL GUIDE *i*

- VGA display based on the C Language Executor

If VGA display based on the C Language Executor is used, the VGA window is erased when the screen display is switched to a screen of this function.
So, when using the VGA window, determine the screen of this function by screen number and close the VGA window before switching the screen display.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

- Use of the CNC screen display function

The following restriction exists when the screen of dynamic graphic display function is displayed by the CNC screen display function.

1) The drawing screen is not displayed in the CNC screen display function by way of via embedded Ethernet.
   Please use the CNC screen display function by way of the FAST Ethernet board.
2) The drawing screen is not displayed in the CNC screen dual display function.
3) Please do not start / shut down the CNC screen display function when you display the drawing screen. Please start / shut down the CNC screen display function after it switches to other screens.

- Vertical soft keys on 10.4-inch display unit

On the screens of this function, the vertical soft keys are not displayed.
To use the vertical soft keys, switch the screen display to another screen.

- Use of the Screen hard copy function

If the screen is copied while executing drawing by screen hard copy function, the drawing stops temporarily. The drawing can be restarted by pushing [RESTART] soft key if necessary after the copy operation is finished.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

14 VIRTUAL MDI KEY FUNCTION

14.1 VIRTUAL MDI KEY

Overview

This function is used to perform program editing and changing of various data using the keyboard displayed on the LCD with a touch panel.

Screen on which a CNC screen is displayed in the upper left 1/4 area

Explanation

A CNC screen equivalent to that displayed on 8.4-inch display unit is displayed in the upper left area of the screen, and the keyboard is displayed in the remaining area.

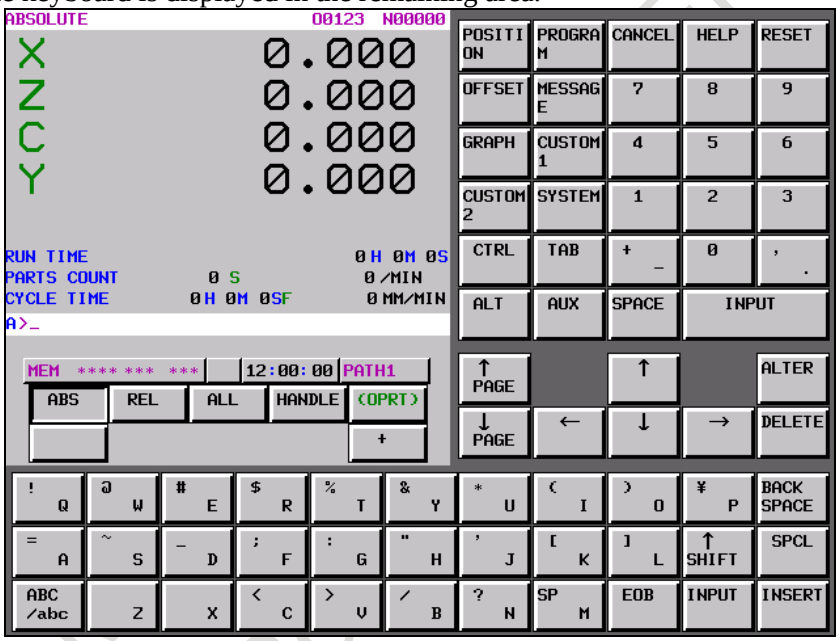


Fig. 14.1 (a) Screen on which a CNC screen is displayed in the upper left area

Operation

- Input key

The display "INPUT" on the virtual MDI keyboard is equivalent to the input key.

- Cancel key

The displays "BACK SPACE" and "CAN" on the virtual MDI keyboard are equivalent to the cancel key.

- Shift key

The display "SHIFT" ↑ on the virtual MDI keyboard is equivalent to the shift key.
Pressing the "SHIFT" key once places the system in the shift state, and pressing the "SHIFT" key again or another key releases the system from the state.
Pressing a key in the shift state causes the character indicated in the upper left corner of the key to be entered.

- Simultaneous pressing of two keys

The operation to be performed for pressing two key simultaneously, such as the "CAN" and "RESET" keys to erase alarm PS100, is as follows:

(1) Press the "SPCL" key. The "SPCL" key is held down, and places the system in SPCL mode.

(2) Press the keys to be pressed simultaneously, one at a time.
(3) Press the "INPUT" key. Of the keys pressed in SPCL mode, the last two are assumed to have been pressed simultaneously. The system is released from SPCL mode, and the SPCL key, which has been held down, returns to the normal state.
Example: "SPCL" → "CAN" → "RESET" → "INPUT"

NOTE
1 In SPCL mode, other keys are disabled until the "SPCL" or "INPUT" key is pressed.
2 Pressing the "SPCL" key in "SPCL" mode causes all keys pressed in SPCL mode to be disabled. The system is released from SPCL mode, and the SPCL key, which has been held down, returns to the normal state.

Screen on which a CNC screen is displayed in the entire area

Explanation

A CNC screen equivalent to that displayed on 8.4-inch display unit is displayed in the entire area. Function keys and the ON/OFF button for virtual keys are displayed at the bottom of the screen.

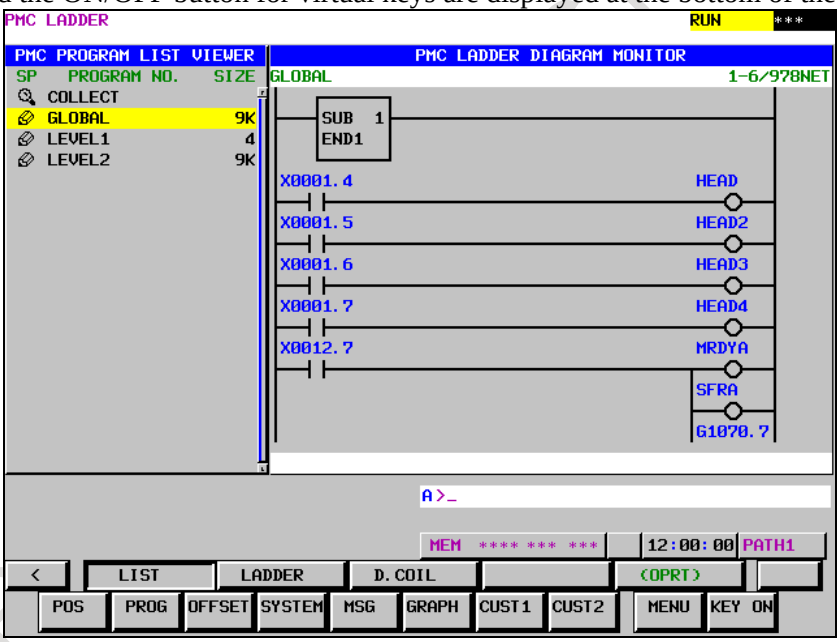


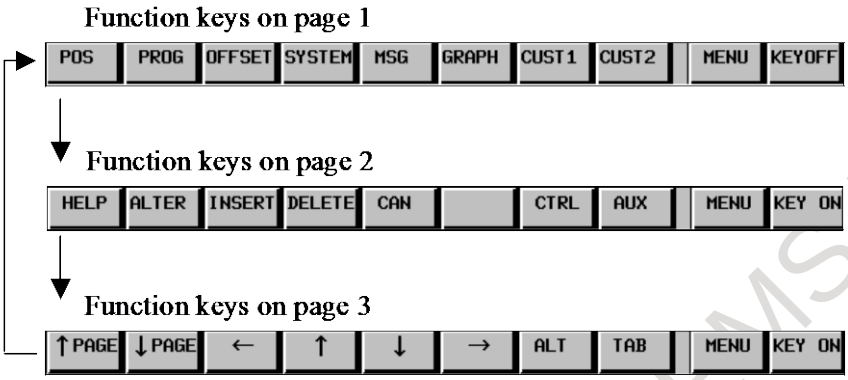
Fig. 14.1 (b) Screen on which a CNC screen is displayed in the entire area

FRYER SYSTEMS INC

Operation

- Function key page switching

Pressing "MENU" located near the lower right corner of the screen switches the screen to page 1, page 2, page 3, and back to page 1 in this order.



- Display of virtual keys

Pressing "KEY ON" located at the lower right corner of the screen displays virtual MDI keys. The character string on the key top changes to "KEYOFF".
Pressing "KEYOFF" hides virtual MDI keys.

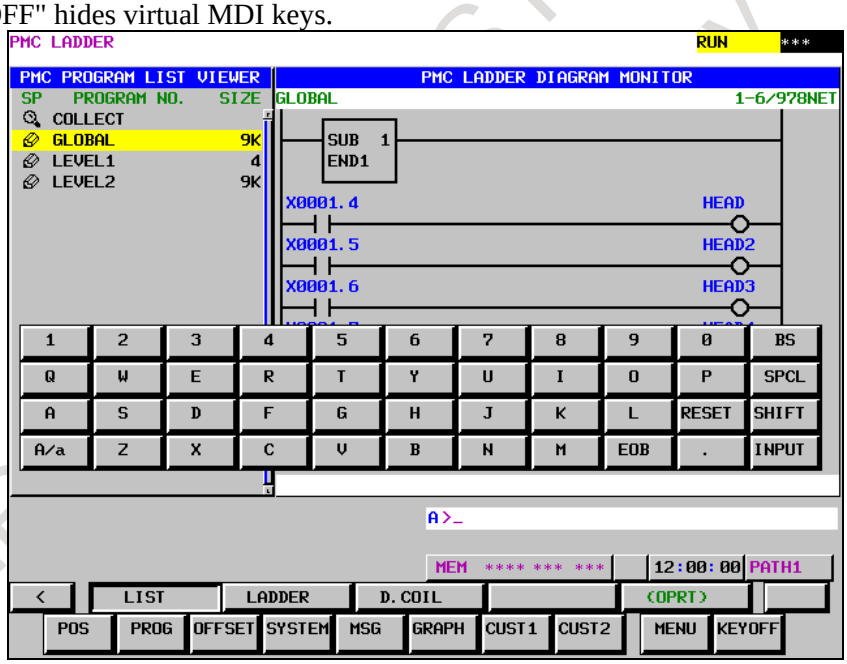


Fig. 14.1 (c) State in which virtual keys are ON

- Input key

The display "INPUT" on the virtual MDI keyboard is equivalent to the input key.

- Cancel key

The displays "BS" and "CAN" on the virtual MDI keyboard are equivalent to the cancel key.

- Shift key

The display "SHIFT" on the virtual MDI keyboard is equivalent to the shift key.
The characters on the key tops change to those that can be entered in the shift state.

FRYER SYSTEMS INC

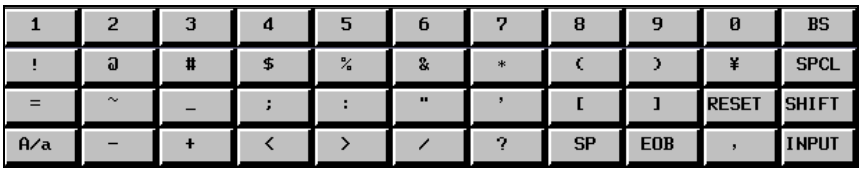


Fig. 14.1 (d) Key tops in the shift state

- Simultaneous pressing of two keys

The operation to be performed for pressing two key simultaneously, such as the "CAN" and "RESET" keys to erase alarm PS100, is as follows:

(1) Press the "SPCL" key. The "SPCL" key is held down, and places the system in SPCL mode.
(2) Press the keys to be pressed simultaneously, one at a time.
(3) Press the "INPUT" key.
Example: "SPCL" → "CAN" → "RESET" → "INPUT"

NOTE
1 In SPCL mode, other keys are disabled until the "SPCL" or "INPUT" key is pressed.
2 Pressing the "SPCL" key in "SPCL" mode causes all keys pressed in SPCL mode to be disabled. The system is released from SPCL mode, and the SPCL key, which has been held down, returns to the normal state.

- Operation history

The operation of virtual MDI key performed by the operator is recorded. The recorded contents are the same as the contents when operating a normal MDI unit.
When the original key "SPCL", "MENU", "KEY ON", "KEY OFF" of virtual MDI key is pressed, the following character string is displayed on the operation history screen and the operation history data (OPRT_HIS.TXT).

Table 14.1 Operation history of original key

Original key	Operation history screen	Operation history data
"SPCL"	{SPCL}	{SPCL}
”MENU”	{MENU}	{MENU}
”KEY ON”, ”KEY OFF”	{KEY}	{KEY ON/OFF}

Moreover, in SPCL mode, not two keys just before pressing "INPUT" key but all operated keys are recorded.
For example)
When the keys of "SPCL", "A", "CAN", "RESET", and "INPUT" are operated in this order, the operation of "A" is not input actually but is recorded.

14.1.1 Limitations

- Display of VGA windows on the C Language Executor

This function uses one VGA window, so that the number of VGA windows that can be used on the C Language Executor is reduced by one.

- Display unit

Display unit on which this function can be used are 10.4 display unit with a touch panel.

FRYER MACHINE SYSTEMS INC
Roman Zahensky

FRYER MACHINE SYSTEMS INC
Rob Zahensky

IV. MAINTENANCE

FRYER MACHINE SYSTEMS INC
Rob Zahensky

FRYER MACHINE SYSTEMS INC
Rob Zahensky

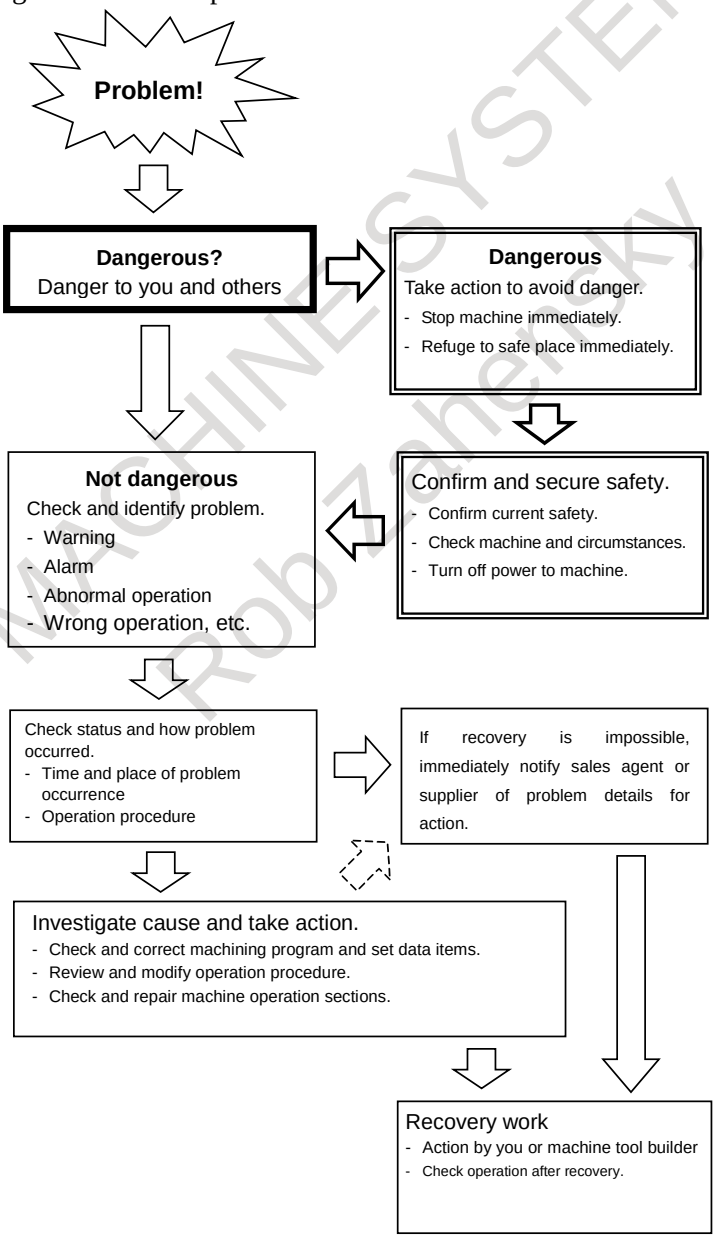
1 ROUTINE MAINTENANCE

This chapter describes routine maintenance work that the operator can perform when using the CNC.

> ⚠ **WARNING**
> Only those persons who have been educated for maintenance and safety may perform maintenance work not described in this chapter.

1.1 ACTION TO BE TAKEN WHEN A PROBLEM OCCURRED

If an unexpected operation occurs or an alarm or warning is output when the CNC and machine are used, the problem needs to be solved quickly. For this purpose, the status of the problem must be identified correctly, and a proper action must be taken.
The procedure for taking an action for a problem is shown below.



For details of investigation and action on problems arising from the CNC, refer to "TROUBLESHOOTING PROCEDURE" in the Maintenance Manual (B-64695EN) issued by FANUC.

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

1.2 BACKING UP VARIOUS DATA ITEMS

With the CNC, various data items such as offset data and system parameters are stored in the SRAM of the control unit and are protected by a backup battery. However, an accident can erase the data. By storing the data at another location (outside the CNC), the data, when lost, can be restored.
So, when the machine is started up or data is updated, for example, the data should be backed up (stored outside the CNC).

- Data backup operation

The data items listed below should be backed up. For the method of data output operation, see the chapter of "DATA INPUT/OUTPUT" in this manual.

<1> System parameters
→ See Subsection, “Inputting and Outputting Parameters”.
<2> Machining programs
→ See Subsection, “Inputting and Outputting a Program”.
<3> Tool offset data
→ See Subsection, “Inputting and Outputting Offset Data”.
<4> PMC data
→ See PMC PROGRAMMING MANUAL (B-64513EN).
<5> Pitch error compensation data (when the pitch error compensation function is selected.)
→ See Subsection, “Inputting and Outputting Pitch Error Compensation Data”.
<6> Custom macro variables (when the custom macro function is selected.)
→ See Subsection, “Inputting and Outputting Custom Macro Common Variables”.
<7> Workpiece coordinate system setting data (when the workpiece coordinate system function is selected.)
→ See Subsection, “Inputting and Outputting Workpiece Coordinates System Data”.
<8> Tool management data (when the tool management function is selected.)
→ See Subsection, “Inputting and Outputting Tool Management Data”.

It is recommended that recording media (such as floppy disks and memory cards) daily used with the machine be used to store data. Stored data should be managed properly so that the data can be restored quickly if a problem occurs.

- Data restoration work

In order to restore lost data to the state of the stored data, input the data backed up according to the previous item into the CNC. For the method of data input operation, see the chapter of "DATA INPUT/OUTPUT" in this manual.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

⚠ WARNING

1 Before recovery of the following data items, consult with the machine tool builder of the machine used:
- System parameters
- PMC data
- Macro programs and custom macro variables
- Pitch error compensation values

2 After inputting stored data, do not start an operation immediately. Instead, check that the data is input correctly and that settings are made to meet a desired operation.
If an operation is executed without making this check, the machine and workpiece can be damaged and personal injury can occur due to an unexpected machine movement. Use sufficient care.

NOTE

The method of recovery described in this section is intended just to restore the state of the backed up data, and does not guarantee recovery of the state that was present when the data was lost.

1.3 METHOD OF REPLACING BATTERY

This chapter describes how to replace the control unit backup battery and absolute Pulsecoder battery.

Battery for memory backup

Such as Offset data and system parameters are stored in SRAM in the control unit. The power to the SRAM is backed up by a lithium battery mounted on the front panel of the control unit. Therefore, the above data is not lost even if the main power is turned off.
When the battery voltage falls, alarm message "BAT" blinks on the LCD screen and the battery alarm signal is output to the PMC. When this alarm is displayed, replace the battery as soon as possible. In general, the battery can be replaced within one or two weeks of the alarm first being issued. This, however, depends on the system configuration.
If the battery voltage subsequently drops further, backup of memory can no longer be provided. Turning on the power to the control unit in this state causes system alarm to be issued because the contents of memory are lost. Replace the battery, clear the entire memory, then reenter the data.
For this reason, FANUC recommends that the battery be replaced once per year regardless of whether a battery alarm is issued.
The type of battery to be replaced differs depending on the control unit. In some control units, a dedicated lithium battery to be purchased from FANUC is used. In other control units, alkaline dry cells (size D) are used in an external battery case.

NOTE

A lithium battery is installed as standard at the factory. This battery can provide backup for the memory contents for about a year.
When discarding a battery, observe the applicable ordinances or other rules of your local government. Also, cover the terminals of the battery with vinyl tape or the like to prevent a short-circuit.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

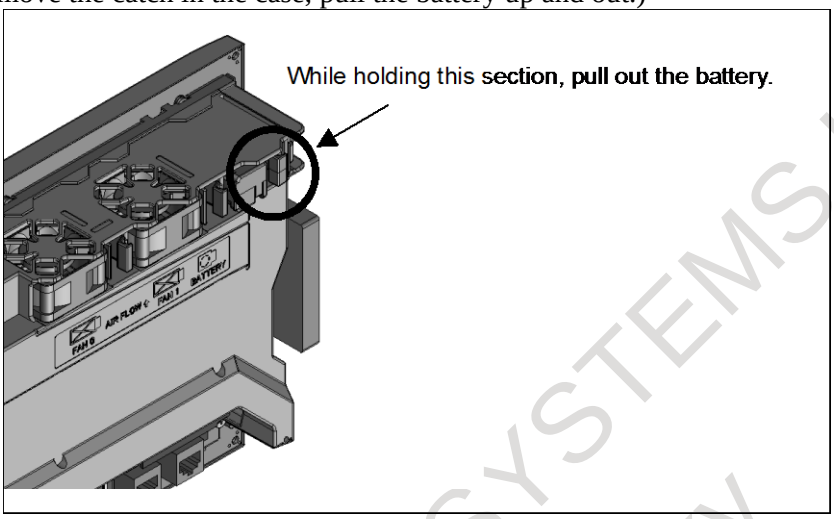
1.3.1 Replacing Battery for Control Unit

When using a lithium battery (for LCD-mounted type control unit)

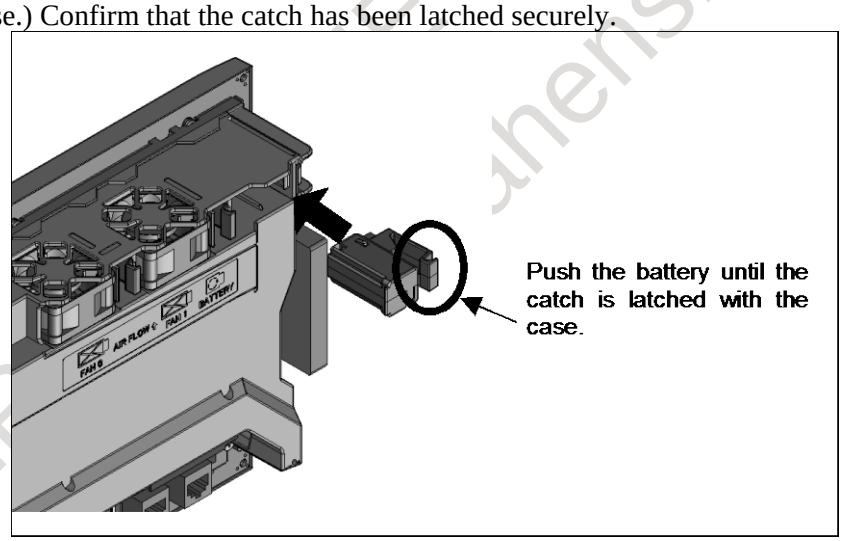
Prepare a new lithium battery (ordering code: A02B-0323-K102).

<1> Turn the power to the machine (control unit) on. After about 30 seconds, turn the power off.

<2> Pull out the lithium battery on the back of the control unit. (While holding the latch of the lithium battery to remove the catch in the case, pull the battery up and out.)



<3> Mount a new lithium battery you get ready beforehand. (Push the battery until the catch is latched with the case.) Confirm that the catch has been latched securely.



⚠WARNING

Using other than the recommended lithium battery may result in the battery exploding. Replace the battery only with the specified lithium battery (A02B-0323-K102).

FRYER SYSTEMS INC

> ⚠ **CAUTION**
> Steps <1> to <3> should be completed within 30 minutes.
> Do not leave the control unit without a battery for any longer than the specified period. Otherwise, the contents of SRAM may be lost.
> If steps <1> to <3> may not be completed within 30 minutes, save all contents of the SRAM to the memory card or USB memory beforehand. Thus, if the contents of the SRAM are lost, the contents can be restored easily.
> For the methods of saving all contents and restoring them, refer to the maintenance manual (B-64695EN).

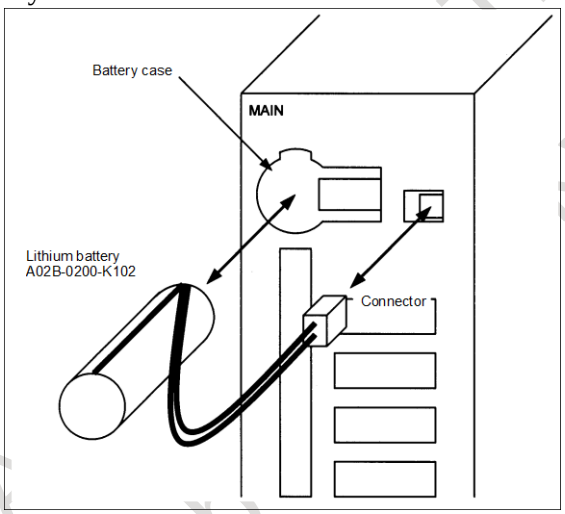
When using a lithium battery (for stand-alone type control unit)

Prepare a new lithium battery (ordering code: A02B-0200-K102).

<1> Turn the power to the machine (control unit) on. After about 30 seconds, turn the power off.

<2> Remove the lithium battery on the upper section of the control unit.
First, unplug the connector by yanking the battery cable, then take the battery out of its case.
The battery case is located in the upper section of the face plate of the main board.

<3> Insert a new lithium battery and reconnect the connector.



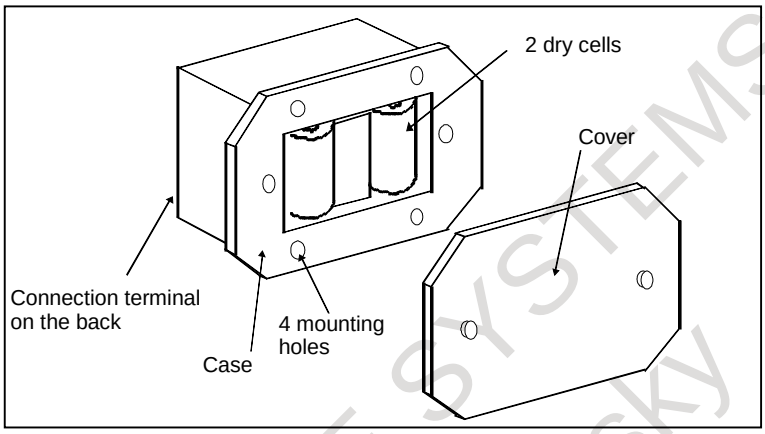
> ⚠ **WARNING**
> Using other than the recommended battery may result in the battery exploding.
> Replace the battery only with the specified battery (A02B-0200-K102).

> ⚠ **CAUTION**
> Steps <1> to <3> should be completed within 30 minutes.
> Do not leave the control unit without a battery for any longer than the specified period. Otherwise, the contents of SRAM may be lost.
> If steps <1> to <3> may not be completed within 30 minutes, save all contents of the SRAM to the memory card or USB memory beforehand. Thus, if the contents of the SRAM are lost, the contents can be restored easily.
> For the methods of saving all contents and restoring them, refer to the maintenance manual (B-64695EN).

When using commercial D-size alkaline dry cells

<1> Prepare two new alkaline dry cells (size D).
<2> Turn on the power to the machine (control unit).
<3> Remove the battery case cover.
<4> Replace the batteries, paying careful attention to their orientation.
<5> Replace the battery case cover.

> ⚠ **CAUTION**
> To replace the battery when the power is off, follow the same procedure as that for the replacement of a lithium battery, described above.



1.3.2 Battery in the PANEL *i* (3 VDC)

A lithium battery is used to back up BIOS data in the PANEL *i*. This battery is factory-set in the PANEL *i*. This battery has sufficient capacity to retain BIOS data for one year.
When the battery voltage becomes low, the following message appears on the self test screen displayed at turn-on, and the self test pauses.

CMOS Battery Failure

If this message appears, replace the battery as soon as possible (within one week). FANUC recommends that the battery be replaced once per year regardless of whether a battery alarm is issued.

- Replacing the battery

(1) To guard against the possible loss or destruction of BIOS parameters, write down the BIOS parameter values.
(2) Obtain a new lithium battery (A02B-0200-K102).
(3) After power has been supplied for at least five seconds, turn off the power to PANEL *i*. Remove the intelligent terminal from the panel so that replacement work can be done from the rear of the intelligent terminal.
(4) Detach the connector of the lithium battery, and remove the battery from the battery holder.
(5) Attach the connector, and place the battery in the battery holder.
(6) Install PANEL *i* again.
(7) Turn on the power, and check that the BIOS parameters are maintained (BIOS setup is not activated forcibly).

Between removing an old battery and inserting new battery, no more than five minutes must be allowed to elapse.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

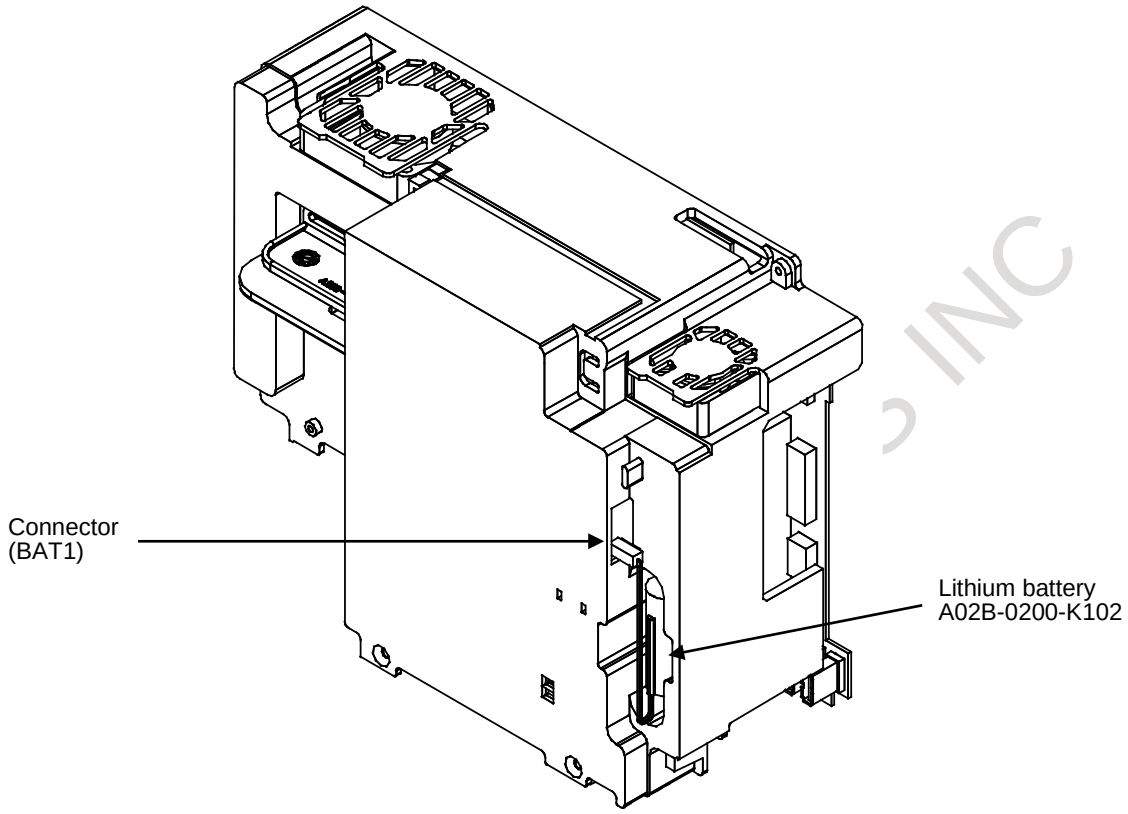


Fig. 1.3.2 Lithium battery connection for PANEL *i*

1.3.3 Replacing Battery for Absolute Pulsecoders

1.3.3.1 Overview

- When the voltage of the batteries for absolute Pulsecoders becomes low, alarm 307 or 306 occurs, with the following indication in the CNC state display at the bottom of the CNC screen.
  Alarm 307 (alarm indicating the voltage of the battery becomes low) :
  The indication "APC" blinks in reversed display.
  Alarm 306 (battery zero alarm) :
  The indication "ALM" blinks in reversed display.
- When alarm 307 (alarm indicating the voltage of the battery becomes low) occurs, replace the battery as soon as possible. In general, the battery should be replaced within one or two weeks, however, this depends on the number of Pulsecoders used.
- When alarm 306 (battery zero alarm) occurs, Pulsecoders are reset to the initial state, in which absolute positions are not held. Alarm 300 (reference position return request alarm) also occurs, indicating that reference position return is required.
- In general, replace the batteries periodically within the service life listed below.
  - A06B-6050-K061 or D-size alkaline dry cells (LR20): Two years (for each six-axis configuration)
  - A06B-6073-K001 : Two years (for each three-axis configuration)
  - A06B-6114-K504 : One year (for each three-axis configuration)

FRYER MACHINE Rob Zahen INC

NOTE

The above values indicate the estimated service life of batteries used with FANUC absolute Pulsecoders. The actual battery service life depends on the machine configuration based on, for example, detector types. For details, contact the machine tool builder.

1.3.3.2 Replacing batteries

To prevent absolute position information in absolute Pulsecoders from being lost, turn on the machine power before replacing the battery. The replacement procedure is described below.

<1> Ensure that the power to the servo amplifier is turned on.
<2> Ensure that the machine is in the emergency stop state (the motor is inactive).
<3> Ensure that the DC link charge LED of the servo amplifier is off.
<4> Detach the old batteries and attach new ones.

The replacement of the batteries in a separate battery case and the replacement of the battery built into the servo amplifier are described below in detail.

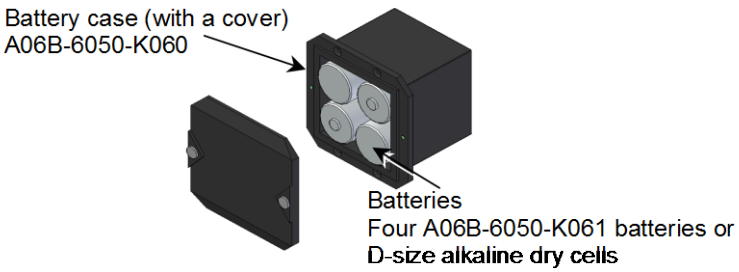
⚠ WARNING

- The absolute Pulsecoder of each of the αi/αi S series servo motors and the βi S series servo motors (βi S0.4 to βi S22) has a built-in backup capacitor. Therefore, even when the power to the servo amplifier is off and the batteries are replaced, reference position return is not required if the replacement completes within less than 10 minutes. Turn the power on and replace the batteries if the replacement will take 10 minutes or more.
- To prevent electric shock, be careful not to touch metal parts in the power magnetics cabinet when replacing the batteries.
- Because the servo amplifier uses a large-capacitance electrolytic capacitor internally, the servo amplifier remains charged for a while even after the power is turned off. Before touching the servo amplifier for maintenance or other purposes, ensure your safety by measuring the residual voltage in the DC link with a tester and confirming that the charge indication LED (red) is off.
- Be sure to replace the batteries with specified ones. Pay attention to the battery polarity. If a wrong type of battery is used or a battery is installed with incorrect polarity, the battery may overheat, blow out, or catch fire, or the absolute position information in the absolute Pulsecoders may be lost.
- Ensure that the battery connector is inserted in the correct position.

1.3.3.3 Replacing the batteries in a separate battery case

Use the following procedure to replace the batteries in the battery case.

<1> Loosen the screws on the battery case and detach the cover.
<2> Replace the batteries in the case (pay attention to the polarity).
<3> Attach the cover to the battery case.



FRYER MACHINE SYSTEMS INC
Rob Zarensky

> ⚠ **CAUTION**
> - Four D-size alkaline dry cells (LR20) that are commercially available can be used as batteries. A set of four A06B-6050-K061 batteries is optionally available from FANUC.
> - Replace all the four batteries with new ones. If old and new batteries are mixed, the absolute position information in the absolute Pulsecoders may be lost.

1.3.3.4 Replacing the battery built into the servo amplifier

Use the following procedure to replace the special lithium battery.

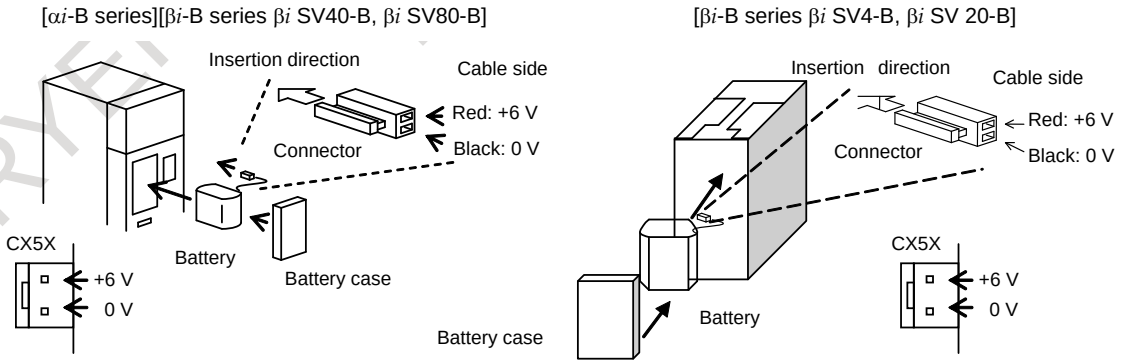
<1> Detach the battery cover.
<2> Replace the special lithium battery.
<3> Attach the battery cover.

> ⚠ **CAUTION**
> - Purchase the battery from FANUC because it is not commercially available. It is therefore recommended that you have a backup battery.
> - When the built-in battery is used, do not connect BATL (B3) of connector CXA2A/CXA2B. Also, do not connect two or more batteries to the same BATL (B3) line. These connections are dangerous because battery output voltages may be short-circuited, causing the batteries to overheat.
> - Install the battery in the servo amplifier in a direction that allows slack in the cable. If the battery cable is under tension, a bad connection may occur.
> - If the +6 V pin and 0 V pin are short-circuited, the battery may overheat, blow out, or catch fire, or the absolute position information in the absolute Pulsecoders may be lost.
> - When inserting the connector, align it to the connector pins.

[Connecting the battery]

The battery for the β*i*SV4-B and β*i*SV20-B series amplifiers is mounted in the battery case on the underside of each of the amplifiers.

The battery for the other β*i*-B series amplifiers and the α*i*-B series amplifiers is mounted at the front of each of the amplifiers.



FRYER MACHINE SYSTEMS INC
Bob Zaharsky

[Battery sets and outlines]

Battery ordering drawing number	Manufacturer model number	Applicable servo amplifier	Battery case ordering drawing number	Outline
A06B-6114-K504 (Note)	BR-2/3AGCT4A (Panasonic)	αi-B series 60/90 mm width	A06B-6114-K505	
		αi-B series 150/300 mm width	A06B-6114-K506	
		βi-B series βi SV-B (two-axis model)	A06B-6114-K505	
A06B-6093-K001	BR-AGCF2W (Panasonic)	βi-B series βiSV4-B, βiSV20-B	A06B-6093-K002	
		βi-B series βiSV40-B, βiSV80-B	A06B-6093-K002	

NOTE
When an old type BR-CCF2TH battery is used, order a battery case that accommodates battery A06B-6114-K504.

Used batteries

Old batteries should be disposed as "INDUSTRIAL WASTES" according to the regulations of the country or autonomy where your machine has been installed.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

APPENDIX

FRYER MACHINE SYSTEMS INC
Rob Zahensky

FRYER MACHINE SYSTEMS INC
Rob Zahensky

A PROGRAM CODE LIST

Character name	ISO code		EIA code		Custom macro		Usable as file name
	Character	Code (hexadecimal)	Character	Code (hexadecimal)	Without custom macro	With custom macro	
Number 0	0	30	0	20			*
Number 1	1	B1	1	01			*
Number 2	2	B2	2	02			*
Number 3	3	33	3	13			*
Number 4	4	B4	4	04			*
Number 5	5	35	5	15			*
Number 6	6	36	6	16			*
Number 7	7	B7	7	07			*
Number 8	8	B8	8	08			*
Number 9	9	39	9	19			*
Address A	A	41	a	61			*
Address B	B	42	b	62			*
Address C	C	C3	c	73			*
Address D	D	44	d	64			*
Address E	E	C5	e	75			*
Address F	F	C6	f	76			*
Address G	G	47	g	67			*
Address H	H	48	h	68			*
Address I	I	C9	i	79			*
Address J	J	CA	j	51			*
Address K	K	4B	k	52			*
Address L	L	CC	l	43			*
Address M	M	4D	m	54			*
Address N	N	4E	n	45			*
Address O	O	CF	o	46			*
Address P	P	50	p	57			*
Address Q	Q	D1	q	58			*
Address R	R	D2	r	49			*
Address S	S	53	s	32			*
Address T	T	D4	t	23			*
Address U	U	55	u	34			*
Address V	V	56	v	25			*
Address W	W	D7	w	26			*
Address X	X	D8	x	37			*
Address Y	Y	59	y	38			*
Address Z	Z	5A	z	29			*
Delete	DEL	FF	Del	7F	×	×	
Back space	BS	88	BS	2A	×	×	
Tabulator	HT	09	Tab	2E	×	×	
End of block	LF or NL	0A	CR or EOB	80			
Carriage return	CR	8D			×	×	
Space	SP	A0	SP	10	□	□	
Absolute rewind stop	%	A5	ER	0B			
Control out (start of comment)	(	28	(2-4-5)	1A			
Control in (end of comment)	)	A9	(2-4-7)	4A			

FIXER MACHINE SYSTEMS INC
Rob Zahensky

Character name	ISO code		EIA code		Custom macro		Usable as file name
	Character	Code (hexadecimal)	Character	Code (hexadecimal)	Without custom macro	With custom macro	
Plus sign	+	2B	+	70			*
Minus sign	-	2D	-	40			*
Colon (address O)	:	3A					
Optional block skip	/	AF	/	31			
Period (decimal point)	.	2E	.	6B			*
Sharp	#	A3	Parameter (No.6012)				
Dollar sign	$	24			□	□	
Ampersand	&	A6	&	0E			
Apostrophe	‘	27			□	□	
Asterisk	*	AA	Parameter (No.6010)				
Comma	,	AC	,	3B			
Semicolon	;	BB			□	□	
Left angle bracket	<	3C					
Equal sign	=	BD	Parameter (No.6011)				
Right angle bracket	>	BE					
Question mark	?	3F			△	○	
Commercial at mark	@	C0			△	△	
Quotation mark	“	22			□	□	
Left square bracket	[	DB	Parameter (No.6013)		△		
Right square bracket	]	DD	Parameter (No.6014)		△		
Underscore	_	6F	Parameter (No.6018)		△	△	*
Lowercase letter a	a	E1			△	△	*
Lowercase letter b	b	E2			△	△	*
Lowercase letter c	c	63			△	△	*
Lowercase letter d	d	E4			△	△	*
Lowercase letter e	e	65			△	△	*
Lowercase letter f	f	66			△	△	*
Lowercase letter g	g	E7			△	△	*
Lowercase letter h	h	E8			△	△	*
Lowercase letter i	i	69			△	△	*
Lowercase letter j	j	6A			△	△	*
Lowercase letter k	k	EB			△	△	*
Lowercase letter l	l	6C			△	△	*
Lowercase letter m	m	ED			△	△	*
Lowercase letter n	n	EE			△	△	*
Lowercase letter o	o	6F			△	△	*
Lowercase letter p	p	F0			△	△	*
Lowercase letter q	q	71			△	△	*
Lowercase letter r	r	72			△	△	*
Lowercase letter s	s	F3			△	△	*
Lowercase letter t	t	74			△	△	*
Lowercase letter u	u	F5			△	△	*
Lowercase letter v	v	F6			△	△	*
Lowercase letter w	w	77			△	△	*
Lowercase letter x	x	78			△	△	*

FANUC MACHINE SYSTEMS INC
Rob Zaremsky

Character name	ISO code		EIA code		Custom macro		Usable as file name
	Character	Code (hexadecimal)	Character	Code (hexadecimal)	Without custom macro	With custom macro	
Lowercase letter y	y	F9			△	△	*
Lowercase letter z	z	FA			△	△	*

NOTE

1 The symbols used in the "Custom macro" column have the following meanings.
(Space) : The character will be registered in memory and has a specific meaning. It is used incorrectly in a statement other than a comment, an alarm occurs.
× : The character will not be registered in memory and will be ignored.
△ : The character will be registered in memory, but will be ignored during program execution. The character that has "*" in its "Usable as file name" field and is used as a file name is not ignored. Lowercase alphabetic characters in an NC program follow the setting of bit 0 (ESL) of parameter No. 3459.
○ : The character will be registered in memory. If it is used in a statement other than a comment, an alarm occurs.
□ : If it is used in a statement other than a comment, the character will not be registered in memory. If it is used in a comment, it will be registered in memory.

2 The symbol used in the column "Usable as file name" has the following meaning:
* : Capable of being coded between "<" and ">" as a file name.

3 Codes not in this table are ignored if their parity is correct.

4 Codes with incorrect parity cause the TH alarm. But they are ignored without generating the TH alarm when they are in the comment section.

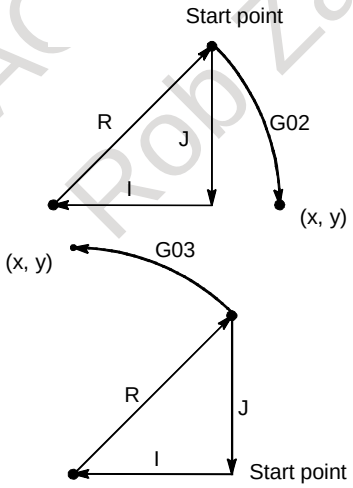
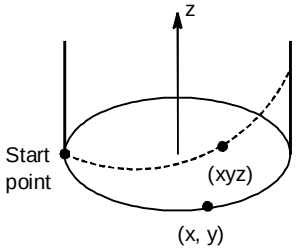
FRYER MACHINE SYSTEMS INC
Rob Zaremsky

B LIST OF FUNCTIONS AND PROGRAM FORMAT

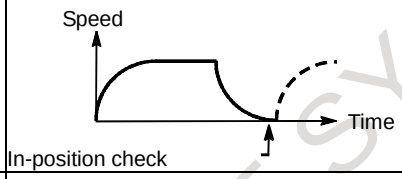
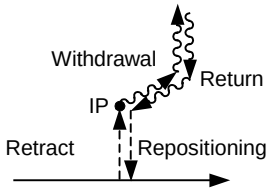
With some functions, the format used for specification on the machining center system differs from the format used for specification on the lathe system. Moreover, some functions are used for only one of the control types for the machining center system and lathe system.
Some functions cannot be added as options depending on the model.
For details of command formats, see the relevant sections or subsections.

In the list, the following symbols are used:

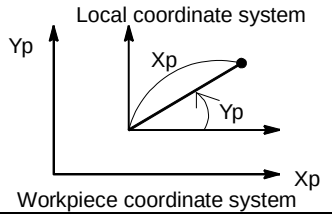
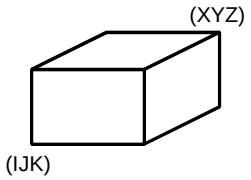
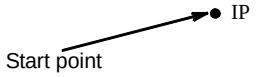
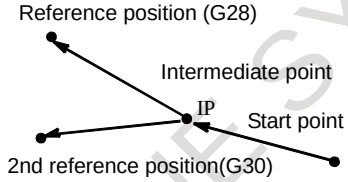
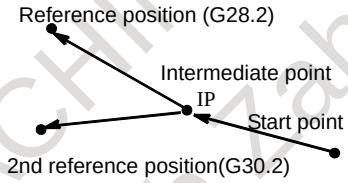
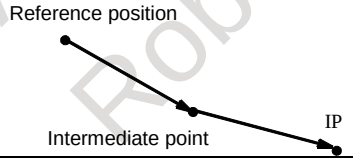
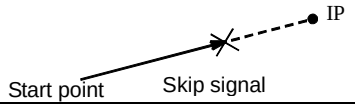
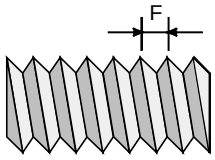
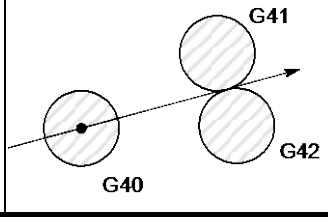
- For machining center
  x : 1st basic axis (X),
  y : 2nd basic axis (Y),
  z : 3rd basic axis (Z)
- For lathe
  x : 1st basic axis (X),
  z : 2nd basic axis (Z),
  coded using G code system A
- IP_ : presents a combination of arbitrary axis addresses using X,Y,Z,A,B and C (such as X_ Y_ Z_).

Functions	Illustration	Program format
Positioning (G00)		G00 IP_ ;
Linear interpolation (G01)		G01 IP_ F_;
Circular interpolation (G02, G03)		**• For machining center** G17 {G02 / G03} X_ Y_ {R_ / I_ J_} F_ ; G18 {G02 / G03} X_ Z_ {R_ / I_ K_} F_ ; G19 {G02 / G03} Y_ Z_ {R_ / J_ K_} F_ ; **• For lathe** {G02 / G03} X_ Z_ {R_ / I_ K_} F_ ;
Helical interpolation (G02, G03)	 In case of G03 on X-Y plane	G17 {G02 / G03} X_ Y_ {R_ / I_ J_} α_ F_ ; G18 {G02 / G03} X_ Z_ {R_ / I_ K_} α_ F_ ; G19 {G02 / G03} Y_ Z_ {R_ / J_ K_} α_ F_ ; α: Arbitrary address except the circular interpolation axis

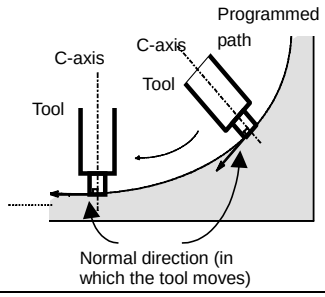
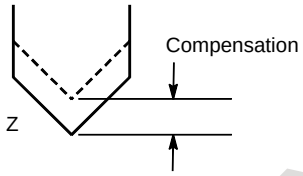
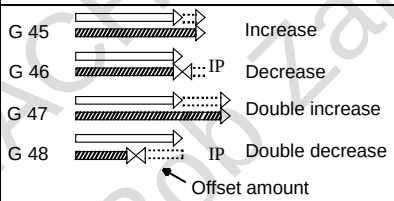
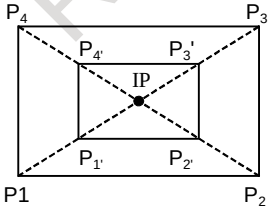
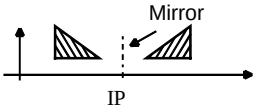
FRYER MACHINE SYSTEMS INC Rob Zahensky

Functions	Illustration	Program format
Dwell (G04)		G04 {X_ / P_} ;
G code preventing buffering (G04.1)		G04.1 (P_) ; P : The operation mode of G04.1. P1: Compatible operation to preventing buffering by command of only G31. P2: Compatible operation to preventing buffering by command of only G53.
AI contour control (G05)		G05 P10000 ; AI contour control start G05 P0 ; AI contour control end
AI contour control (G05.1)		G05.1 Q1 ; AI contour control mode on G05.1 Q0 ; AI contour control mode off
Smooth tolerance control (M Series) (G05.1)		G05.1 Q3 IP0 ; Smooth tolerance control mode on G05.1 Q0 ; Smooth tolerance control mode off
Cylindrical interpolation (G07.1)		G07 IP_ r_; Cylindrical interpolation mode r : Cylinder radius G07 IP 0; Cylindrical interpolation mode cancel
Exact stop (G09)		G09 {G01 / G02 / G03} IP_ ;
Programmable data input (G10)		• **For machining center** Tool compensation memory A G10 L01 P_ R_ ; Tool compensation memory C G10 L10 P_ R_ ; (Geometry offset amount/H) G10 L11 P_ R_ ; (Wear offset amount/H) G10 L12 P_ R_ ; (Geometry offset amount/D) G10 L13 P_ R_ ; (Wear offset amount/D) • **For lathe** Geometry offset amount G10 P_ X_ Z_ R_ Q_ ; P = 10000 + Geometry offset number Wear offset amount G10 P_ X_ Z_ C_ Q_ ; P = Wear offset number
Tool retract and recover (G10.6)		G10.6 IP_ ; Specify the amount of retraction G10.6 (as a single block containing no other commands) ; Cancel the amount of retraction
Polar coordinate interpolation(T series) (G12.1, G13.1)		G12.1 ; Polar coordinate interpolation mode on G13.1 ; Polar coordinate interpolation cancel

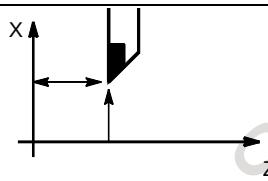
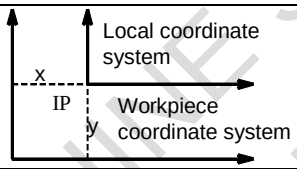
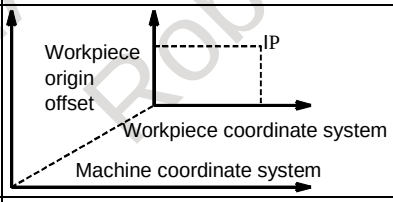
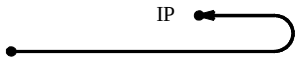
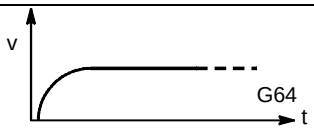
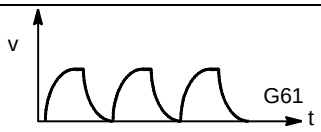
FRYER MACHINE SYSTEMS INC
Rob Zaharesky

Functions	Illustration	Program format
Polar coordinate command (G15, G16)		G17 G16 Xp_ Yp_ . . . ; G18 G16 Zp_ Xp_ . . . ; G19 G16 Yp_ Zp_ . . . ; G15 ; Cancel
Plane selection (G17, G18, G19)		G17 ; Xp Yp-plane selection G18 ; Zp Xp-plane selection G19 ; Yp Zp-plane selection
Inch/metric conversion (G20, G21)		Inch input G20 ; Metric input G21 ;
Stored stroke check (G22, 23)		G22 X_ Y_ Z_ I_ J_ K_ ; G23 ; Cancel
Reference position return check (G27)		G27 IP_ ;
Reference position return (G28) 2nd Reference position return (G30)		G28 IP_ ; G30 IP_ ;
In-position check disable reference position return (G28.2) In-position check disable 2nd reference position return (G30.2)		G28.2 IP_; G30.2 IP_;
Movement from reference position (G29)		G29 IP_ ;
Skip function (G31)		G31 IP_ F_ ;
Threading (G33)		• **For machining center** G33 IP_ F_ ; F : Lead
Threading (G32)		• **For lathe** Equal lead threading G32 P_ F_ ;
Tool radius/tool nose radius compensation, 3-dimensional cutter compensation (G38, G39, G40 to G42)		• **For machining center** $\left\{\begin{matrix}G17\\G18\\G19\end{matrix}\right\}$ $\left\{\begin{matrix}G41\\G42\end{matrix}\right\}$ D_ ; D : Tool compensation number G40 : Cancel

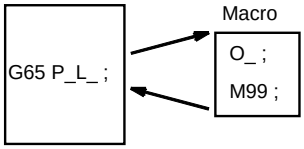
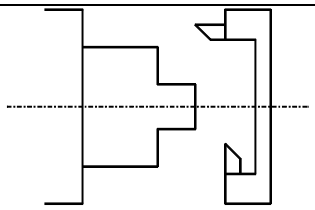
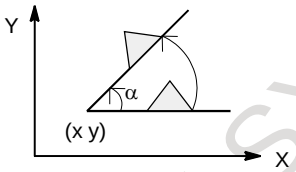
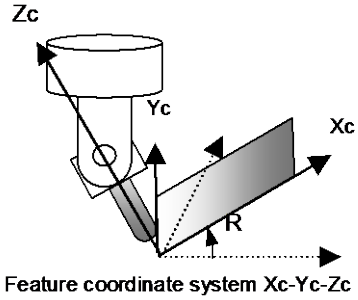
FRYER MACHINE SYSTEMS INC
Rob Zahensky

Functions	Illustration	Program format
Tool radius/tool nose radius compensation (G40 to G42) (G38, G39)		• **For lathe only** {G41, G42} IP_ ; G40 : Cancel
Normal direction control (G40.1, G41.1, G42.1)		G41.1 ; Normal direction control on : right G42.1 ; Normal direction control on : left G40.1 ; Normal direction control cancel
Tool length compensation (G43, G44, G49)		{G43, G44} Z_ H_ ; {G43, G44} H_ ; H : Tool compensation number G49 : Cancel
Tool offset (G43.7)		• **For lathe** G43.7 D_ ; D : Tool compensation number • **For machining center** G43.7 H_ ; H : Tool compensation number
Tool offset (G45 to G48)		• **For machining center** {G45, G46, G47, G48} IP_ D_ ; D : Tool offset number
Scaling(M series) (G50, G51)		• **For machining center** G51 X_ Y_ Z_ {P_, I_ J_ K_} ; P, I, J, K : Scaling magnification X, Y, Z : Control position of scaling G50 : Cancel • **For lathe only** Enabled when G code system B/C is used
Programmable mirror image (G50.1, G51.1)		G51.1 IP_ ; G50.1 ; . . . Cancel

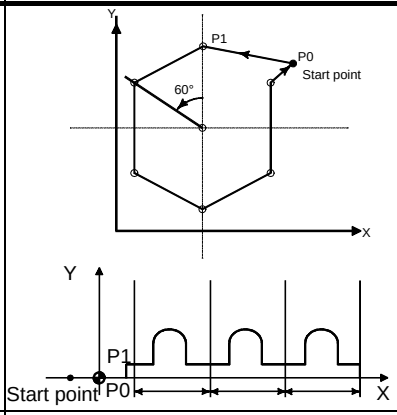
FRYER MACHINE SYSTEMS INC
Rob Zaharsky

Functions	Illustration	Program format
Synchronous, composite, and superimposed control by program command (G50.4, G51.4, G50.5, G51.5, G50.6, G51.6)		G51.4 P_Q_(L_) ; Start synchronous control (L_ can be omitted.) G50.4 Q_ ; Cancel synchronous control. P : Number to identify synchronous master axis Q : Number to identify synchronous slave axis L : Parking start command G51.5 P_Q_ ; Start composite control G50.5 P_Q_ ; Cancel composite control P : Number to identify composite axis 1 Q : Number to identify composite axis 2 G51.6 P_Q_ ; Start superimposed control G50.6 Q_ ; Cancel superimposed control P : Number to identify superimposed master axis Q : Number to identify superimposed slave axis
Coordinate system setting Maximum spindle speed clamp (G50)		• **For lathe only** G50 IP_ ; (Coordinate system setting) G50 S_ ; (Maximum spindle speed clamp)
Local coordinate system setting (G52)		G52 IP_ ;
Command in machine coordinate system (G53)		G53 IP_ ;
Tool axis direction control (G53.1)		G53.1 ; Tool axis direction control
Selection of workpiece coordinate system (G54 to G59)		{G54 ⋮ G59} IP_ ;
Single direction positioning (M series) (G60)		G60 IP_ ;
Cutting mode (G64) Exact stop mode (G61) Tapping mode (G63)		G64_ ; Cutting mode G61_ ; Exact stop mode G63_ ; Tapping mode
Automatic corner override (M series) (G62)		G62_ ; Automatic corner override

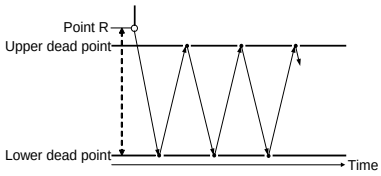
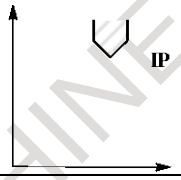
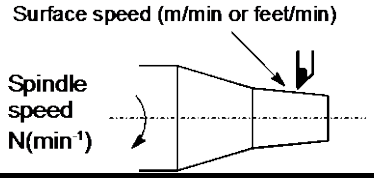
PARKER MACHINE SYSTEMS INC
Rob Zahensky

Functions	Illustration	Program format
Custom macro (G65, G66, G66.1, G67)		• **One-shot call** G65 P_ L_ <Argument assignment> ; P : Program number L : Number of repetition • **Modal call** G66 P_ L_ <Argument assignment> ; Call after the move command G66.1 P_ L_ <Argument assignment> ; Each block call G67 ; Cancel
Mirror image for double turret (G68, G69)		• **For lathe only** G68 : Mirror image for double turret G69 : Mirror image cancel
Coordinate system rotation, 3-dimensional coordinate conversion (G68, G69) (G68.1, G69.1)	 In case of X-Y plane	• **For machining center** G68 {G17 X_ Y_ / G18 Z_ X_ / G19 Y_ Z_} R α ; G69 ; Cancel • **For lathe** G68.1 {G17 X_ Y_ / G18 Z_ X_ / G19 Y_ Z_} R α ; G69.1 ; Cancel
Tilted working plane indexing with guidance (G68.2, G68.4)		G68.2/G68.4 P_ X_ Y_ Z_ I_ J_ K_ ; G68.2 : Tilted working plane indexing G68.4 : Tilted working plane indexing (incremental multiplexed command) G69 ; Tilted working plane indexing cancel Without P : Euler's angle P1 : Roll, pitch, yaw P2 : Three points P3 : Two vectors P4 : Projection angle X, Y, Z : Feature coordinate system origin I, J, K : Euler angles for determining the orientation of the feature coordinate system
Tilted working plane indexing by tool axis direction (G68.3)		G68.3 X_ Y_ Z_ R_ ; G69 ; Cancel X,Y, Z : Feature coordinate system origin R : Feature coordinate system rotation angle about the Z-axis

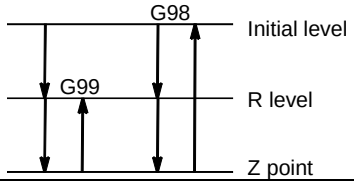
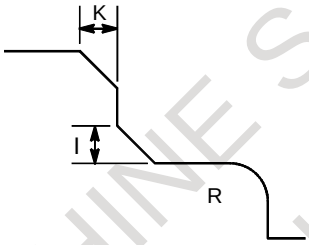
FRYER MACHINE SYSTEMS INC
Rob Zahensky

Functions	Illustration	Program format
Figure copy (G72.1, G72.2)		**• Rotational copy** {(G17) (G18) (G19)} G72.1 P_ L_ {X_ Y_ Z_ X_ Y_ Z_} R_ ; **• Linear copy** {(G17) (G18) (G19)} G72.2 P_ L_ {I_ J_ K_ I_ J_ K_} ;
Canned cycle for drilling (G73, G74, G80 to G89)		**• For machining center** G80 ; Cancel G73 G74 G76 G81 : G89 } X_Y_Z_P_Q_R_F_K_ ;
Canned cycle (G71 to G76) (G90, G92, G94)		**• For lathe only** N_ G70P_ Q_ ; G71U_ R_ ; G71P_ Q_ U_ W_ F_ S_ T_ ; G72W_ R_ ; G72P_ Q_ U_ W_ F_ S_ T_ ; G73U_ W_ R_ ; G73P_ Q_ U_ W_ F_ S_ T_ ; G74R_ ; G74X(u)_ Z(w)_ P_ Q_ R_ F_ ; G75R_ ; G75X(u)_ Z(w)_ P_ Q_ R_ F_ ; G76P_ Q_ R_ ; G76X(u)_ Z(w)_ P_ Q_ R_ F_ ; {G90 G92} X_ Z_ I_ F_ ; G94X_ Z_ K_ F_ ;
Canned grinding cycle (for grinding machine) (G71 to G75, G77 to G79)		**• For machining center** G75 I_ J_ K_ α_ R_ F_ P_ L_ ; G77 I_ J_ K_ α_ R_ F_ P_ L_ ; G78 I_ (J_) K_ α_ F_ P_ L_ ; G79 I_ J_ K_ α_ R_ F_ P_ L_ ; α : Arbitrary axis address of the grinding axis **• For lathe** G71 A_ B_ W_ U_ I_ K_ H_ ; G72 P_ A_ B_ W_ U_ I_ K_ H_ ; G73 A_ (B_) W_ U_ K_ H_ ; G74 P_ A_ (B_) W_ U_ K_ H_ ;

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

Functions	Illustration	Program format
High precision oscillation function (G81.1, G80)		G81.1 Z_Q_R_F_ ; Starts oscillation Z : Upper dead point Q : Distance between the upper dead point and lower dead point R : Distance from the upper dead point to point R F : Feedrate during oscillation base feedrate G80 ; Cancels oscillation
Absolute/incremental programming (G90/G91)		• **For machining center** G90_ ; Absolute programming G91_ ; Incremental programming : G90_ G91_ ; Programming in both modes • **For lathe** X_ Z_ C_ : Absolute programming U_ W_ H_ : Incremental programming Distinguished by an address specified in combined use with a G function such as G00 and G01.
Maximum incremental command value check (G91.1)		G91.1 IP_ ; IP_ ; Maximum incremental value Set 0 to cancel maximum incremental value check.
Change of workpiece coordinate system (G92) Maximum spindle speed clamp (G92)		• **For machining center** G92 IP_ ; Change of workpiece coordinate system G92 S_ ; Constant surface speed control : Maximum spindle speed clamp
Workpiece coordinate system preset (G92.1)		• **For machining center** G92.1 IP 0 ;
Inverse time feed(M series) (G93)		G93 ; Inverse time setting mode
Feed per minute, Feed per revolution (G94, G95)	mm/min inch/min mm/rev inch/rev	• **For machining center** G94 F_ ; Feed per minute G95 F_ ; Feed per revolution
(G98, G99)		• **For lathe** G98 F_ ; Feed per minute G99 F_ ; Feed per revolution
Constant surface speed control (G96, G97)		G96 S_ ; Constant surface speed control on (surface speed specification) G97 S_ ; Constant surface speed control off (spindle speed specification)

FRYER MACHINE SYSTEMS INC Rob Zaherski

Functions	Illustration	Program format
Speed display function of a milling tool with servo motor (G96.1,G96.2,G96.3,G96.4)		G96.1 P_R_; The next block starts operating upon completion of spindle indexing (the SV speed control mode is off). G96.2 P_R_; The next block starts operating without waiting for spindle indexing to complete. G96.3 P_; The next block starts operating after confirming the completion of spindle indexing (the SV speed control mode is off). G96.4 P_; The SV speed control mode is on.
Initial point return / R point return (G98, G99)		• **For machining center** G98_ ; G99_ ;
Optional chamfering/corner R		• **For machining center** ,C_ : Chamfering ,R_ : Corner R
Chamfering/corner R		• **For lathe only** X_ { C±K / R_ } P_; Z_ { C±K / R_ } P_;
In-feed control (for grinding machine) (M series) (G160, G161)		• **For machining center** G161 R_ ; Figure program (G01, G02, G03) G160 ;

FRYER MACHINE SYSTEMS INC
Rob Zahensky

C RANGE OF COMMAND VALUE

Linear axis

- In case of millimeter input, feed screw is millimeter

	Increment system		
	IS-A	IS-B	IS-C
Least input increment (mm)	0.01	0.001	0.0001
Least command increment (mm)	0.01	0.001	0.0001
Max. programmable dimension (mm)	±999,999.99	±999,999.999	±99,999.9999
Max. rapid traverse (mm/min)[*1]	999,000	999,000	100,000
Feedrate range (mm/min)[*1]	0.01 to 999,000	0.001 to 999,000	0.0001 to 100,000
Incremental feed (mm/step)	0.01 0.1 1.0 10.0	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1
Tool compensation amount (mm)[*2]	0 to ±9,999.99	0 to ±9,999.999	0 to ±9,999.9999
Backlash compensation amount (pulses)[*3]	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)[*4]	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999

- In case of inch input, feed screw is millimeter

	Increment system		
	IS-A	IS-B	IS-C
Least input increment (inch)	0.001	0.0001	0.00001
Least command increment (inch)	0.001	0.0001	0.00001
Max. programmable dimension (inch)	±39,370.078	±39,370.0787	±3,937.00787
Max. rapid traverse (mm/min)[*1]	999,000	999,000	100,000
Feedrate range (inch/min)[*1]	0.001 to 96,000	0.0001 to 9,600	0.00001 to 4,000
Incremental feed (inch/step)	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1	0.00001 0.0001 0.001 0.01
Tool compensation amount (inch)[*2]	0 to ±999.999	0 to ±999.9999	0 to ±999.99999
Backlash compensation amount (pulses)[*3]	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)[*4]	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999

- In case of inch input, feed screw is inch

	Increment system		
	IS-A	IS-B	IS-C
Least input increment (inch)	0.001	0.0001	0.00001
Least command increment (inch)	0.001	0.0001	0.00001
Max. programmable dimension (inch)	±99,999.999	±99,999.9999	±9,999.99999
Max. rapid traverse (inch/min)[*1]	96,000	9,600	4,000
Feedrate range (inch/min)[*1]	0.001 to 96,000	0.0001 to 9,600	0.00001 to 4,000
Incremental feed (inch/step)	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1	0.00001 0.0001 0.001 0.01
Tool compensation amount (inch)[*4]	0 to ±999.999	0 to ±999.9999	0 to ±999.99999
Backlash compensation amount (pulses)[*3]	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)[*4]	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999

FRYER MACHINE SYSTEMS INC
Rob Zalansky

- In case of millimeter input, feed screw is inch

	Increment system		
	IS-A	**IS-B**	**IS-C**
Least input increment (mm)	0.01	0.001	0.0001
Least command increment (mm)	0.01	0.001	0.0001
Max. programmable dimension (mm)	±999,999.99	±999,999.999	±99,999.9999
Max. rapid traverse (inch/min)*1	96,000	9,600	4,000
Feedrate range (mm/min)*1	0.01 to 999,000	0.001 to 999,000	0.0001 to 100,000
Incremental feed (mm/step)	0.01 0.1 1.0 10.0	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1
Tool compensation amount (mm)*2	0 to ±9,999.99	0 to ±9,999.999	0 to ±9,999.9999
Backlash compensation amount (pulses)*3	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)*4	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999

- Rotary axis

	Increment system		
	IS-A	**IS-B**	**IS-C**
Least input increment (deg)	0.01	0.001	0.0001
Least command increment (deg)	0.01	0.001	0.0001
Max. programmable dimension (deg)	±999,999.99	±999,999.999	±99,999.9999
Max. rapid traverse (deg/min)*1	999,000	999,000	100,000
Feedrate range (deg/min)*1	0.01 to 999,000	0.001 to 999,000	0.0001 to 100,000
Incremental feed (deg/step)	0.01 0.1 1.0 10.0	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1
Tool compensation amount (deg)*2	0 to ±9,999.99	0 to ±9,999.999	0 to ±9,999.9999
Backlash compensation amount (pulses)*3	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)*4	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999

NOTE

*1 The feedrate range shown above are limitations depending on CNC interpolation capacity. As a whole system, limitations depending on servo system must also be considered.

*2 If the mode of input is switched between inch input and metric input, the maximum compensation value that can be set at inch input time is (maximum compensation value) × 1/25.4. If a value exceeding this value is specified at inch input time, the compensation value is not converted to a metric value correctly when the mode of input is switched to metric input.

*3 The unit is the detection unit.

*4 Depends on the increment system of the axis at in address X.

FRYER MACHINE SYSTEMS INC
Rob Zalensky

D NOMOGRAPHS

D.1 INCORRECT THREADED LENGTH

The leads of a thread are generally incorrect in δ_1 and δ_2, as shown in Fig. D.1 (a), due to automatic acceleration and deceleration.

Thus distance allowances must be made to the extent of δ_1 and δ_2 in the program.

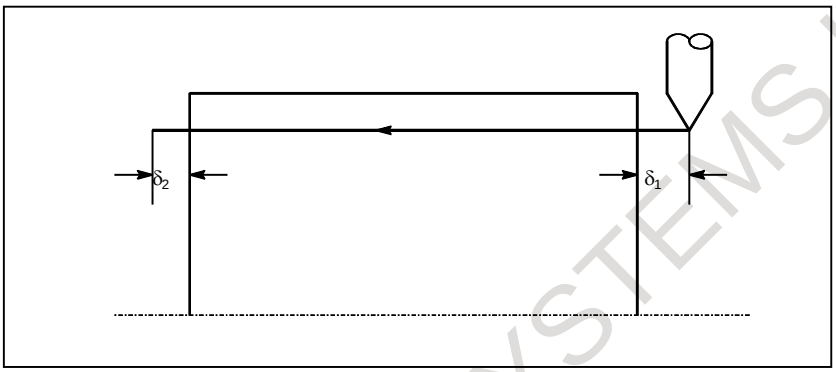


Fig. D.1 (a) Incorrect thread position

Explanation

- How to determine δ_2

$\delta_2 = T_1 V$ (mm). . . . (1)

$V = \frac{1}{60} RL$

T_1 : Time constant of servo system (sec)
V : Threading speed (mm/sec)
R : Spindle speed (min^{-1})
L : Thread feed (mm)
Time constant T_1 (sec) of the servo system: Usually 0.033 s.

- How to determine δ_1

$\delta_1 = \{t - T_1 + T_1 \exp(-\frac{t}{T_1})\} V$ . . . . . (2)

$a = \exp(-\frac{t}{T_1})$ . . . . . (3)

T_1 : Time constant of servo system (sec)
V : Threading speed (mm/sec)
Time constant T_1 (sec) of the servo system: Usually 0.033 s.

The lead at the beginning of thread cutting is shorter than the specified lead L, and the allowable lead error is ΔL. Then as follows.

$$a = \frac{\Delta L}{L}$$

When the value of “a” is determined, the time lapse until the thread accuracy is attained from formula (3). The time “t” is substituted in (2) to determine δ_1: Constants V and T_1 are determined in the same way as for δ_2. Since the calculation of δ_1 is rather complex, a nomograph as shown in Fig. E.1(b) is provided on the following pages.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

- How to use nomograph

First specify the class and the lead of a thread. The thread accuracy, a, will be obtained at (1), and depending on the time constant of cutting feed acceleration/ deceleration, the δ_1 value when V = 10mm/sec will be obtained at (2). Then, depending on the speed of thread cutting, δ_1 for speed other than 10mm/sec can be obtained at (3).

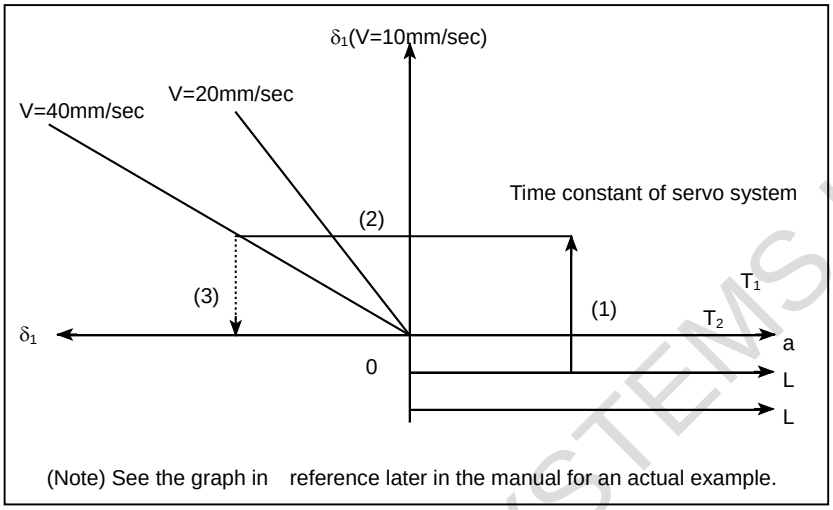


Fig. D.1 (b) Nomograph

> **NOTE**
> The equations for δ_1, and δ_2 are for when the acceleration/ deceleration time constant for cutting feed is 0.

D.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH

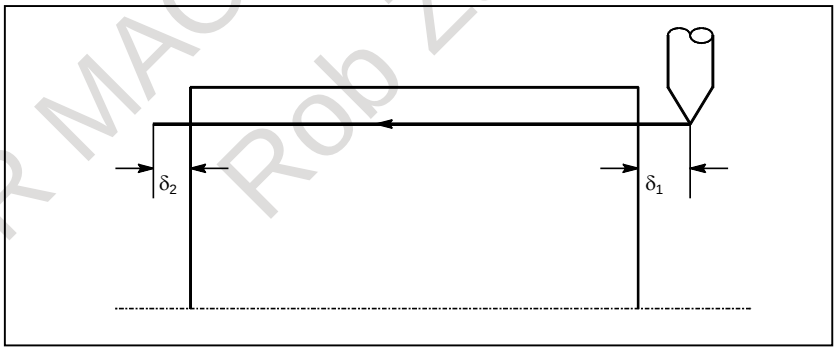


Fig. D.2 (a) Incorrect threaded portion

Explanation

- How to determine δ_2

$$\delta_2 = \frac{LR}{1800^*}(\text{mm})$$

R : Spindle speed (min^{-1})

L : Thread lead (mm)

* When time constant T_1 of the servo system is 0.033 s.

FRYER MACHINE SYSTEMS INC
Rob Zarensky

- How to determine δ_1

$$\delta_1 = \frac{LR}{1800*}(-1 - \ln a) \text{ (mm)}$$

$$= \delta_2(-1 - \ln a) \text{(mm)}$$

R : Spindle speed (min^{-1})

L : Thread lead (mm)

* When time constant T_1 of the servo system is 0.033 s.

Following a is a permitted value of thread.

a	-1-lna
0.005	4.298
0.01	3.605
0.015	3.200
0.02	2.912

Example

R=350min^{-1}

L=1mm

a=0.01

then

$$\delta_2 = \frac{350 \times 1}{1800} = 0.194\text{(mm)}$$

$$\delta_1 = \delta_2 \times 3.605 = 0.701\text{(mm)}$$

Reference

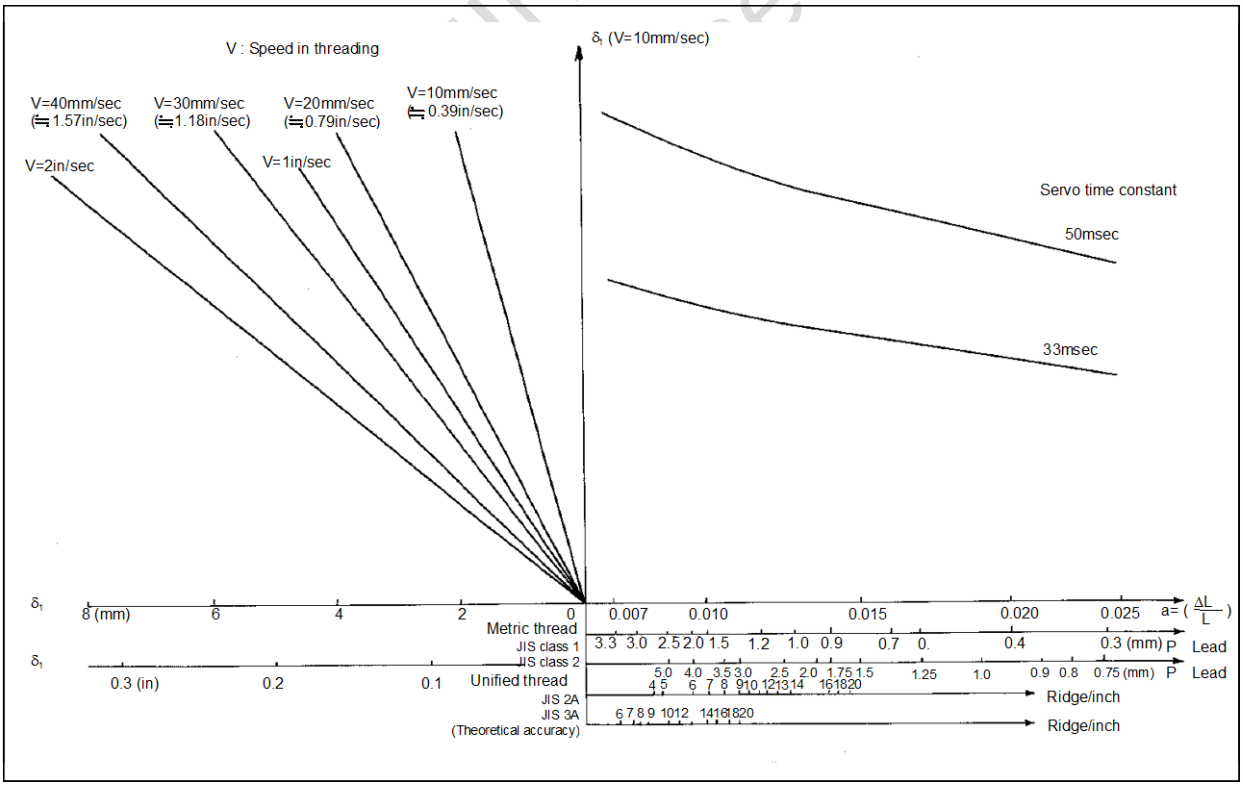


Fig. D.2 (b) Nomograph for obtaining approach distance δ_1

FANUC LINE SYSTEMS INC
Kensky

D.3 TOOL PATH AT CORNER

When servo system delay (by exponential acceleration/deceleration at cutting or caused by the positioning system when a servo motor is used) is accompanied by cornering, a slight deviation is produced between the tool path (tool center path) and the programmed path as shown in Fig. D.3 (a).

Time constant T_1 of the exponential acceleration/deceleration is fixed to 0.

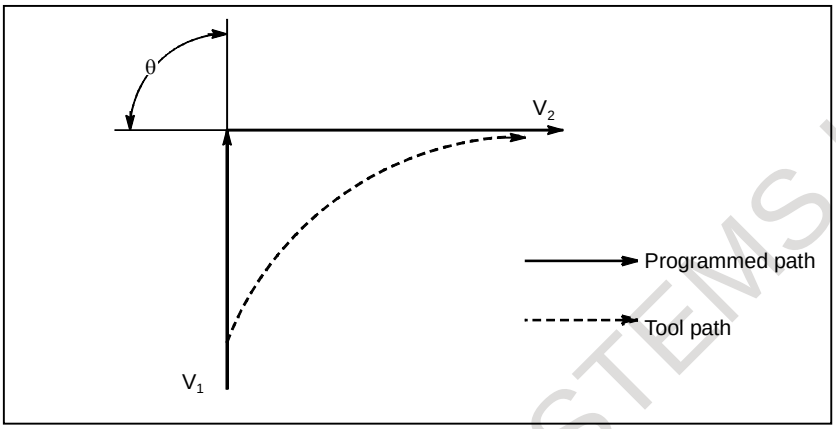


Fig. D.3 (a) Slight deviation between the tool path and the programmed path

This tool path is determined by the following parameters:

- Feedrate (V_1, V_2)
- Corner angle (θ)
- Exponential acceleration / deceleration time constant (T_1) at cutting ($T_1 = 0$)
- Presence or absence of buffer register.

The above parameters are used to theoretically analyze the tool path and above tool path is drawn with the parameter which is set as an example.

When actually programming, the above items must be considered and programming must be performed carefully so that the shape of the workpiece is within the desired precision.

In other words, when the shape of the workpiece is not within the theoretical precision, the commands of the next block must not be read until the specified feedrate becomes zero. The dwell function is then used to stop the machine for the appropriate period.

Explanation

- Analysis

The tool path shown in Fig. D.3 (b) is analyzed based on the following conditions:

- Feedrate is constant at both blocks before and after cornering.
- The controller has a buffer register. (The error differs with the reading speed of the tape reader, number of characters of the next block, etc.)

FRYER MACHINE SYSTEMS INC
Rob Zaherisky

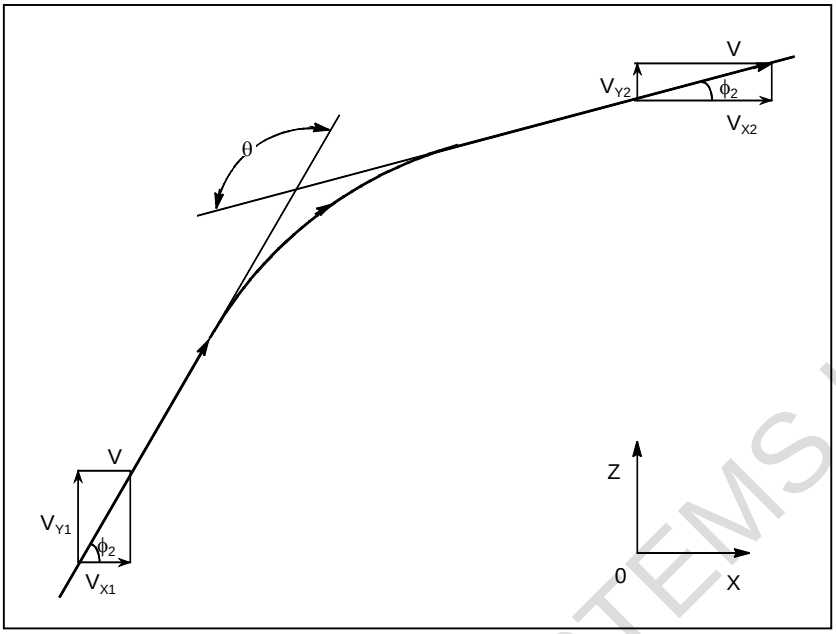


Fig. D.3 (b) Example of tool path

- Description of conditions and symbols

$V_{X1} = V\cos\phi_1$
$V_{Y1} = V\sin\phi_1$
$V_{X2} = V\cos\phi_2$
$V_{Y2} = V\sin\phi 2$

V : Feedrate at both blocks before and after cornering
V_{X1} : X-axis component of feedrate of preceding block
V_{Y1} : Y-axis component of feedrate of preceding block
V_{X2} : X-axis component of feedrate of following block
V_{Y2} : Y-axis component of feedrate of following block
θ : Corner angle
ϕ_1 : Angle formed by specified path direction of preceding block and X-axis
ϕ_2 : Angle formed by specified path direction of following block and X-axis

- Initial value calculation

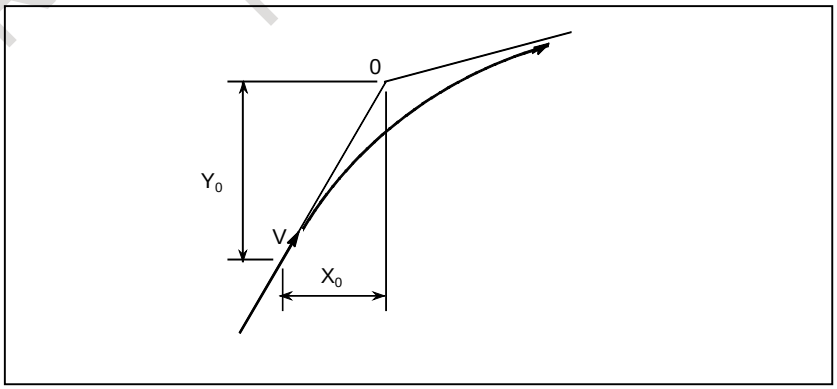


Fig. D.3 (c) Initial value

The initial value when cornering begins, that is, the X and Y coordinates at the end of command distribution by the controller, is determined by the feedrate and the positioning system time constant of the servo motor.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

$$X_0 = V_{X1}(T_1 + T_2)$$

$$Y_0 = V_{Y1}(T_1 + T_2)$$

T_1: Exponential acceleration/deceleration time constant. (T=0)
T_2: Time constant of positioning system (Inverse of position loop gain)

- Analysis of corner tool path

The equations below represent the feedrate for the corner section in X-axis direction and Y-axis direction.

$$V_X{}^{(t)} = (V_{X2} - V_{X1})[1 - \frac{V_{X1}}{T_1 - T_2}\{T_1 \exp(-\frac{t}{T_1}) - T_2 \exp(-\frac{t}{T_2})\} + V_{X1}]$$

$$= V_{X2}[1 - \frac{V_{X1}}{T_1 - T_2}\{T_1 \exp(-\frac{t}{T_1}) - T_2 \exp(-\frac{t}{T_2})\}]$$

$$V_Y{}^{(t)} = \frac{V_{Y1} - V_{Y2}}{T_1 - T_2}\{T_1 \exp(-\frac{t}{T_1}) - T_2 \exp(-\frac{t}{T_2})\} + V_{Y2}$$

Therefore, the coordinates of the tool path at time t are calculated from the following equations:

$$X(t) = \int_0^1 V_X(t)dt - X_0$$

$$= \frac{V_{X2} - V_{X1}}{T_1 - T_2}\{T_1{}^2 \exp(-\frac{t}{T_1}) - T_2{}^2 \exp(-\frac{t}{T_2})\} - V_{X2}(T_1 + T_2 - t)$$

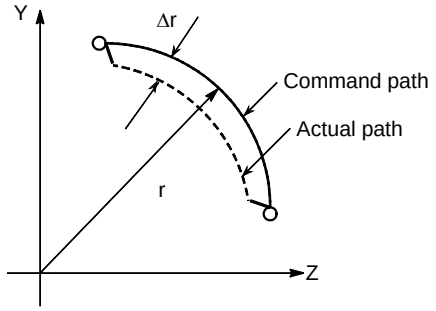
$$Y(t) = \int_0^1 V_Y(t)dt - Y_0$$

$$= \frac{V_{Y2} - V_{Y1}}{T_1 - T_2}\{T_1{}^2 \exp(-\frac{t}{T_1}) - T_2{}^2 \exp(-\frac{t}{T_2})\} - V_{Y2}(T_1 + T_2 - t)$$

FRYER MACHINE SYSTEMS INC
Rob Zahensky

D.4 RADIUS DIRECTION ERROR AT CIRCLE CUTTING

When a servo motor is used, the positioning system causes an error between input commands and output results. Since the tool advances along the specified segment, an error is not produced in linear interpolation. In circular interpolation, however, radial errors may be produced, sepecially for circular cutting at high speeds.
This error can be obtained as follows:



$$\Delta r = \frac{1}{2}(T_1^2 + T_2^2(1-\alpha^2))\frac{V^2}{r} \quad \ldots\ldots (1)$$

Δr: Maximum radius error (mm)
v : Feedrate (mm/sec)
r : Circle radius (mm)
T_1: Exponential acceleration/deceleration time constantat cutting (sec) (T=0)
T_2: Time constant of servo motor sec)
(Inverse of positon loop gain)
α : Feed forward coefficient (%)

In the case of bell-shaped acceleration/deceleration and linear acceleration/deceleration after cutting feed interpolation, an approximation of this radius error can be obtained with the following expression:

Linear acceleration/deceleration after cutting feed interpolation

$$\Delta r = (\frac{1}{24}T_1^2 + \frac{1}{2}T_2^2(1-\alpha^2))\frac{V^2}{r}$$

Bell-shaped acceleration/deceleration after cutting feed interpolation

$$\Delta r = (\frac{1}{48}T_1^2 + \frac{1}{2}T_2^2(1-\alpha^2))\frac{V^2}{r}$$

Thus, the radius error in the case of linear acceleration/deceleration after interpolation is smaller than in case of exponential acceleration/deceleration by a factor of 12, excluding any error caused by a servo motor time constant. The radius error in the case of bell-shaped acceleration/deceleration after interpolation is a half of linear acceleration/deceleration after interpolation.

Since the machining radius r (mm) and allowable error Δr (mm) of the workpiece is given in actual machining, the allowable limit feedrate v (mm /sec) is determined by equation (1).
Since the acceleration/deceleration time constant at cutting which is set by this equipment varies with the machine tool, refer to the manual issued by the machine tool builder.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

E SETTINGS AT POWER-ON, IN THE CLEAR STATE, OR IN THE RESET STATE

Either the clear state or reset state is entered during a reset is set by bit 6 (CLR) of parameter No. 3402 (0: reset state/1: clear state).
The symbols in the figure below have the following meanings.

○ : The setting remains unchanged or the operation continues.
× : The setting is deleted or the operation stops.

Item		Power-on	Clear state	Reset state
Settings	Offset	○	○	○
	Set data	○	○	○
	Parameter	○	○	○
Various data items	Program in memory	○	○	○
	Information of the block read in advance during automatic operation	×	×	×
	N (sequence number)	× (Note 7)	× (Notes 1, 2, 7)	○ (Note 1, 2)
	One-shot G code	×	×	×
	Modal G code	Initial value (Note 3) G20 and G21 are set to the state at the previous power-off.	Initial value (Note 3) G20 and G21 are not changed (Note 2).	○ (Note 2 and 6)
	F	0	0	○ (Note 2)
	S, T, M	×	○ (Note 2)	○ (Note 2)
	K (count specification)	×	×	×
Coordinate	Workpiece coordinates	0 (Note 4)	○	○
Operation during execution	Movement	×	×	×
	Dwell	×	×	×
	M, S, T code output	×	×	×
	Tool position compensation	×	See "Tool Position Compensation at a Reset".	See "Tool Position Compensation at a Reset".
	Tool length compensation	×	Depending on the setting of bit 6 (LVK) of parameter No. 5003	Depending on the setting of bit 6 (LVK) of parameter No. 5003
	Tool radius/tool nose radius compensation	×	×	×
	Storing a called subprogram	×	× (Note 5)	×

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

Item		Power-on	Clear state	Reset state
Output signals	CNC alarm signal AL	"0"(when no alarm cause is present)	“0” (when no alarm cause is present)	“0” (when no alarm cause is present)
	Reference position return completion signal ZPx	×	○ (or × for emergency stop)	○(or × for emergency stop)
	S, T, B codes	×	○	○
	M code	×	×	×
	M, S, T, B strobe signals	×	×	×
	Spindle rotation signal (S analog signal)	×	○	○
	CNC ready signal MA	“1”	○	○
	Servo ready signal SA	“1” (when a servo alarm does not occur)	“1” (when a servo alarm does not occur)	“1” (when a servo alarm does not occur)
	Cycle start signal STL	×	×	×
	Feed hold signal SPL	×	×	×

Tool position compensation at a reset

○ : Cancelled.

× : Not canceled.

Compensation method		Bit 3 (LVC) of parameter No. 5006 and bit 7 (TGC) of parameter No. 5003			
		LVC=0 TGC=0	LVC=1 TGC=0	LVC=0 TGC=1	LVC=1 TGC=1
Tool movement	Wear compensation	×	○ (Axis movement)	×	○ (Axis movement)
	Geometry compensation				
Coordinate shift	Wear compensation	×	○	×	○
	Geometry compensation	×	×	○	○

FRYER MACHINE SYSTEMS INC
Rob Zaharisky

NOTE

1 When the beginning position is found, the main program number is displayed.

2 If a reset is made during execution of a block, the states of the modal G code and modal address (such as N, F, T, or M) specified in the block are not reflected. The modal information specified in the previous and preceding blocks is kept. However, modal address of S code depends on bit 4 (MDS) of parameter No.13113.

Bit 4 (MDS) of parameter No.13113
0 : Modal information (S code) in an executing block is displayed.
1 : Modal information (S code) in a previous block is displayed.

3 The initial value of the modal G code depends on the following parameters. For details, refer to the parameter manual (B-64700EN).
- Bit 0 (G01) of parameter No. 3402
- Bit 1 (G18) of parameter No. 3402
- Bit 2 (G19) of parameter No. 3402
- Bit 3 (G91) of parameter No. 3402
- Bit 7 (G23) of parameter No. 3402
- Parameters No. 3406 to 3409

4 For detection of an absolute position, the value is generated from machine coordinates and a workpiece origin offset.

5 When a reset is made during execution of a subprogram, a return to the main program is performed. Execution at a midpoint in the subprogram is impossible.

6 When one of the following two settings, which hold the modal G code in group 1 by a reset is set:
- Reset state (bit 6 of parameter No. 3402 is 0)
- Clear state (bit 6 of parameter No. 3402 is 1) and the modal G code in group 1 is held at a time of a reset (bit 1 of parameter No. 3406 is 1),

if a reset is made during execution of one of the following canned cycles, which cause cycle operation, the mode of the modal G code in group 1 is changed to the G01 mode.
- G90 : Outer surface/inner surface turning cycle
- G92 : Threading cycle
- G94 : Edge turning cycle

7 When bit 0 (SEK) of parameter No. 11353 is set to 1, sequence number N can be held even at power-on or in the clear state.

FRYER MACHINE SYSTEMS INC
Rob Zalensky

F CHARACTER-TO-CODES CORRESPONDENCE TABLE

F.1 CHARACTER-TO-CODES CORRESPONDENCE TABLE

Character	Code	Comment	Character	Code	Comment
A	065		6	054	
B	066		7	055	
C	067		8	056	
D	068		9	057	
E	069			032	Space
F	070		!	033	Exclamation mark
G	071		“	034	Quotation mark
H	072		#	035	Sharp
I	073		$	036	Dollar sign
J	074		%	037	Percent
K	075		&	038	Ampersand
L	076		‘	039	Apostrophe
M	077		(	040	Left parenthesis
N	078		)	041	Right parenthesis
O	079		*	042	Asterisk
P	080		+	043	Plus sign
Q	081		,	044	Comma
R	082		-	045	Minus sign
S	083		.	046	Period
T	084		/	047	Slash
U	085		:	058	Colon
V	086		;	059	Semicolon
W	087		<	060	Left angle bracket
X	088		=	061	Equal sign
Y	089		>	062	Right angle bracket
Z	090		?	063	Question mark
0	048		@	064	At mark
1	049		[	091	Left square bracket
2	050		]	093	Right square bracket
3	051		^	094	
4	052		_	095	Underscore
5	053				

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

F.2 FANUC DOUBLE-BYTE CHARACTER CODE TABLE

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
0200	ぁ	あ	ぃ	い	ぅ	う	ぇ	え	ぉ	お	か	が	き	ぎ	く	ぐ
0220	け	げ	こ	ご	さ	ざ	し	じ	す	ず	せ	ぜ	そ	ぞ	た	だ
0240	ち	ぢ	っ	つ	づ	て	で	と	ど	な	に	ぬ	ね	の	は	ば
0260	ぱ	ひ	び	ぴ	ふ	ぶ	ぷ	へ	べ	ぺ	ほ	ぼ	ぽ	ま	み	む
0280	め	も	ゃ	や	ゅ	ゆ	ょ	よ	ら	り	る	れ	ろ	ゃ	わ	素
02A0	材	を	ん	種	類	棒	穴	成	形	質	寸	法	外	径	長	端
02C0	面	最	小	内	大	加	工	切	削	倣	正	途	中	荒	具	番
02E0	号	仕	上	込	点	方	向	速	度	送	量	開	始	深	主	軸
0300	回	転	数	位	置	決	直	線	時	円	反	現	在	指	令	値
0320	領	域	診	断	操	作	手	引	機	械	残	移	動	次	早	電
0340	源	投	入	間	分	秒	自	運	負	荷	実	使	用	寿	命	新
0360	規	除	隅	取	単	補	能	独	終	了	記	角	溝	刃	幅	広
0380	設	定	一	覧	表	部	炭	合	金	鋼	超	硬	先	付	摩	耗
03A0	仮	想	副	行	挿	消	去	山	高	準	備	完	後	弧	助	択
03C0	無	視	器	原	登	録	再	処	理	描	画	過	容	編	集	未
03E0	対	相	座	標	示	名	歯	変	呼	推	馬	力	系	選	達	閉

FRYER MACH
Rob Za

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
0400	禁	復	帰	書	個	桁	稼	由	両	半	逃	底	逆	下	空	四
0420	触	平	代	辺	格	子	周	心	本	群	停	止	巾	微	状	路
0440	範	囲	倍	率	注	側	特	殊	距	離	連	続	増	隔	件	初
0460	期	条	経	握	圧	扱	陰	隠	右	押	横	黄	億	屋	化	何
0480	絵	階	概	該	巻	換	気	起	軌	技	疑	供	共	境	強	教
04A0	掘	繰	係	傾	型	検	権	研	肩	見	験	元	弦	減	孔	巧
04C0	控	更	校	構	根	左	差	雑	参	散	産	算	治	耳	式	失
04E0	修	十	従	勝	商	少	尚	昇	植	色	食	伸	信	侵	振	浸
0500	真	暗	以	意	異	影	鋭	越	価	可	科	果	箇	課	各	拡
0520	核	学	掛	漢	簡	観	関	含	却	客	休	急	業	曲	均	筋
0540	継	計	軽	言	限	互	降	採	済	細	姿	思	写	射	斜	者
0560	車	借	縦	重	出	述	術	渉	照	省	章	証	象	身	進	人
0580	図	違	印	沿	遠	央	奥	往	応	会	解	改	割	活	願	基
05A0	奇	寄	岐	既	近	区	矩	駆	偶	旧	求	球	究	級	欠	結
05C0	口	語	誤	交	厚	項	刻	告	黒	財	策	糸	試	資	事	持
05E0	似	釈	弱	受	収	純	順	所	序	剰	場	常	飾	水	錐	据
0600	制	整	製	前	全	然	則	属	即	他	多	存	谷	探	短	徴
0620	鎖	調	頂	鉄	添	頭	同	導	道	熱	年	濃	箱	発	抜	伴
0640	必	百	複	物	文	聞	併	忘	末	密	有	余	与	裏	立	略
0660	青	席	石	積	赤	接	折	粗	創	双	捜	太	打	体	待	態
0680	替	段	知	地	致	遅	追	通	伝	得	読	凸	凹	突	鈍	敗
06A0	杯	背	配	品	不	布	並	頁	別	片	返	勉	弁	保	明	滅
06C0	木	目	歪	揺	様	溶	要	抑	良	輪	和	話	枠	節	説	絶
06E0	千	専	浅	旋	総	走	退	台	第	題	卓	室	着	柱	鋳	丁
0700	低	訂	肉	日	白	薄	比	皮	被	非	美	普	伏	歩	包	門
0720	問	絡	列	万	利	訳	礼	乱	放	枚	約	練	油	劣	例	郭
0740	戻	冷	垂	緑	紫	許	測	精	効	家	装	管	粉	等	守	般
0760	納	義	貫	安	固	毎	板	予	張	任	破	損	御	足	温	給
0780	界	混	丸	汎	護	己	程	抗	詳	鳥	適	論	額	縁	排	性
07A0	生	績	監	締	砥	θ	当	的	脂	料	落	確	認	報	〕〔	■■
07C0			判	搬			称	樹								
07E0							島	壁								

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
0800	阿	哀	愛	挨	逢	悪	旭	宛	案	闇	鞍	伊	依	偉	委	威
0820	慰	易	為	維	緯	胃	衣	遺	医	井	育	一	稲	員	因	飲
0840	院	羽	雨	渦	唄	浦	瓜	噂	雲	営	映	栄	永	泳	洩	英
0860	衛	液	益	駅	園	延	援	演	炎	煙	鉛	塩	汚	王	岡	沖
0880	憶	臆	牡	乙	恩	穏	音	佳	夏	暇	架	歌	河	火	花	華
08A0	菓	貨	我	牙	芽	賀	雅	介	壊	廻	快	怪	海	灰	皆	貝
08C0	劾	害	慨	街	垣	殻	獲	覚	較	革	楽	笠	括	滑	株	刈
08E0	乾	寒	刊	勧	喚	官	寛	干	幹	感	慣	敢	歓	汗	環	甘
0900	看	緩	缶	肝	還	鑑	閑	陥	韓	館	岸	眼	岩	顔	企	危
0920	喜	希	幾	揮	机	旗	棄	毅	祈	季	稀	徽	輝	騎	擬	犠
0940	議	菊	詰	脚	丘	久	及	吸	宮	弓	救	泣	牛	居	巨	拒
0960	挙	虚	魚	亨	享	京	競	協	叫	挟	橋	況	狭	胸	興	郷
0980	鏡	響	驚	凝	局	極	玉	勤	錦	琴	銀	九	句	苦	馳	屑
09A0	屈	熊	君	訓	軍	郡	刑	兄	契	掲	敬	景	茎	警	芸	迎
09C0	劇	激	隙	潔	血	月	倹	健	兼	券	剣	圏	堅	建	憲	拳
09E0	犬	献	絹	県	謙	軒	鍵	険	幻	古	庫	戸	故	湖	狐	誇
0A00	顧	五	午	侯	候	光	公	勾	喉	好	孝	幸	康	弘	拘	攻
0A20	江	港	甲	稿	絞	綱	考	肯	衡	講	購	郊	鉱	香	剛	克
0A40	国	穀	酷	腰	骨	此	頃	今	困	婚	査	砂	債	妻	彩	才
0A60	災	砕	祭	菜	裁	載	際	剤	罪	坂	阪	咲	昨	索	錯	桜
0A80	冊	刷	察	撮	擦	札	殺	皿	三	撤	讃	賛	酸	伺	刺	史
0AA0	士	姉	市	師	支	枝	死	私	紙	詞	詩	字	寺	磁	辞	七
0AC0	湿	芝	縞	捨	煮	社	謝	尺	若	酒	首	授	需	秋	習	臭
0AE0	舟	週	住	柔	宿	祝	縮	熟	春	瞬	盾	巡	暑	女	傷	唱
0B00	奨	将	床	承	招	昭	焼	焦	笑	紹	衝	賞	障	乗	城	情
0B20	浄	蒸	錠	職	唇	寝	審	森	申	神	芯	親	辛	針	震	尽
0B40	陣	須	酢	吹	粋	遂	杉	裾	澄	世	是	勢	征	政	星	晴
0B60	清	盛	聖	声	西	誓	請	静	税	昔	析	籍	責	跡	雪	舌
0B80	占	宣	尖	川	戦	扇	栓	泉	洗	染	潜	船	銑	鮮	善	組
0BA0	訴	倉	層	掃	巣	争	窓	草	騒	像	臓	蔵	贈	造	促	息
0BC0	束	俗	卒	其	揃	尊	村	詑	堕	妥	耐	帯	怠	滞	袋	貸
0BE0	隊	滝	宅	拓	濯	託	濁	奪	脱	棚	誰	嘆	担	淡	団	弾

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
0C00	暖	男	談	池	築	畜	竹	筑	秩	茶	昼	虫	駐	貯	帳	庁
0C20	彫	挑	朝	町	脹	腸	跳	沈	珍	賃	墜	痛	塚	爪	吊	釣
0C40	庭	廷	提	釘	泥	摘	滴	笛	典	天	展	店	貼	殿	田	吐
0C60	塗	徒	都	砥	努	土	怒	倒	冬	凍	刀	島	東	湯	灯	答
0C80	筒	統	到	藤	討	踏	透	働	堂	胴	銅	峠	徳	毒	届	曇
0CA0	謎	鍋	縄	南	軟	難	二	匂	乳	尿	念	燃	粘	悩	脳	農
0CC0	把	波	派	廃	拝	肺	買	売	博	拍	泊	舶	麦	肌	畑	八
0CE0	罰	版	犯	班	繁	販	飯	盤	否	彼	悲	扉	批	疲	秘	肥
0D00	費	避	飛	尾	鼻	菱	筆	俵	氷	票	評	病	浜	貧	敏	夫
0D20	婦	富	怖	浮	父	符	腐	武	舞	封	風	服	福	腹	払	沸
0D40	噴	憤	奮	紛	丙	兵	幣	柄	米	壁	癖	偏	便	捕	募	墓
0D60	母	簿	宝	崩	捧	泡	胞	芳	訪	豊	飽	亡	傍	剖	妨	帽
0D80	忙	房	暴	望	紡	肪	膨	防	北	僕	撲	釦	没	翻	磨	魔
0DA0	幕	膜	迄	満	味	魅	脈	妙	民	務	夢	矛	迷	鳴	免	綿
0DC0	模	茂	毛	盲	網	黙	紋	冶	夜	野	矢	役	薬	躍	諭	輸
0DE0	優	友	遊	郵	融	誉	預	幼	揚	曜	洋	葉	陽	養	浴	翼
0E00	螺	来	頼	欄	陸	律	流	留	粒	旅	療	稜	林	臨	隣	涙
0E20	累	励	鈴	暦	歴	烈	裂	労	漏	老	六	脇	惑	詫	湾	腕
0E40	斡	椅	萎	宇	嘘	閲	宴	欧	懐	拐	涯	穫	闇	潟	渇	冠
0E60	患	汽	貴	鬼	偽	戯	欺	喫	窮	糾	拠	漁	恐	狂	脅	仰
0E80	緊	愚	遇	靴	啓	慶	憩	携	撃	傑	嫌	懸	厳	雇	娯	洪
0EA0	紅	耕	航	貢	挫	催	栽	崎	柵	拶	傘	志	施	旨	至	誌
0EC0	識	狩	趣	就	秀	衆	襲	蹴	充	渋	緒	署	諸	叙	掌	訟
0EE0	鐘	壌	織	紳	酔	瀬	誠	繊	漸	繕	塑	礎	阻	奏	族	惰
0F00	戴	諾	叩	旦	誕	恥	仲	宙	忠	抽	兆	懲	抵	敵	撤	党
0F20	盗	糖	陶	闘	督	馴	覇	媒	爆	縛	髪	閥	泌	匹	府	敷
0F40	仏	慕	縫	乏	霧	盟	勇	誘	踊	裸	雷	卵	里	隆	慮	虜
0F60	寮	塁	隷	霊	恋	浪	郎	功	坑	々	〆	〼	⊾	⊾	⊿	⊿
0F80	`	a	b	c	d	e	f	g	h	i	j	k	l	m	n	o
0FA0	Б	Г	Д	Ж	З	И	Й	К	Л	М	П	У	Ф	Ц	Ч	Ш
0FC0	À	Á	Ã	Â	Æ	Ç	È	É	Ê	Ë	Ì	Í	Î	Ï	Ò	Ó
0FE0	Å	Ä	Ö	Ü	Ñ	¿	ç	è	é	ê	ë	ì	í	î	ï	ñ

	p	q	r	s	t	u	v	w	x	y	z	{	:	}	~	
0F80																
0FA0	Щ	Ъ	Ы	Ь	Э	Ю	Я		?	⊥	⊿	⊿				
0FC0	Ô	Õ	Œ	Ø	Ù	Ú	Û	Ÿ	ß	à	á	â	ã	ä	å	æ
0FE0	ò	ó	ô	õ	ö	œ	ø	ù	ú	û	ü	ÿ	¡			

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
1000	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P
1020	Q	R	S	T	U	V	W	X	Y	Z	a	b	c	d	e	f
1040	g	h	i	j	k	l	m	n	o	p	q	r	s	t	u	v
1060	w	x	y	z	//				γ	ε	μ	π	φ	ω	Δ	Σ
1080	Ω								ガ	ギ	グ	ゲ	ゴ	ザ	ジ	ズ	ゼ	ゾ	ダ	ヂ	ヅ	デ	ド
10A0	バ	ビ	ブ	ベ	ボ	パ	ピ	プ	ペ	ポ	ヴ	ヵ	ヶ		ァ	ア	ィ	イ	ゥ	ウ	ェ	エ
10C0	ォ	オ	カ	ガ	キ	ギ	ク	グ	ケ	ゲ	コ	ゴ	サ	ザ	シ	ジ
10E0	ス	ズ	セ	ゼ	ソ	ゾ	タ	ダ	チ	ヂ	ッ	ツ	ヅ	テ	デ	ト
1100	ド	ナ	ニ	ヌ	ネ	ノ	ハ	バ	パ	ヒ	ビ	ピ	フ	ブ	プ	ヘ
1120	ベ	ペ	ホ	ボ	ポ	マ	ミ	ム	メ	モ	ャ	ヤ	ュ	ユ	ョ	ヨ
1140	ラ	リ	ル	レ	ロ	ヮ	ワ	ヲ	ン	ヴ	ヵ	ヶ				
1160	―	、	。	,	.	・	:	;	?	!	゛	゜	´	`	¨	^
1180	‾	＿	ヽ	ヾ	ゝ	ゞ	〃	仝	々	〆	○	ー	―	-	／	＼
11A0	～	‖	｜	…	‥	‘	’	“	”	(	)	〔	〕	[	]	{
11C0	}	〈	〉	《	》	「	」	『	』	【	】	＋	－	±	×	÷
11E0	＝	≠	＜	＞	≦	≧	∞	∴	♂	♀	°	′	″	℃	￥	$
1200	¢	£	%	♯	&	＊	@	§	☆	★	○	●	◎	◇	◆	□
1220	■	△	▲	▽	▼	※	〒	→	←	↑	↓	＝				
1240	1/1	2/2	3/3	4/4	5/5	6/6			mm	cm	km	cm²	m²	km²	cm³	m³
1260	mg	kg	cc	dℓ	ℓ	kℓ	ms	μs	ns	HP	ps	Hz	㈱	©		
1280	亜	芦	尉	壱	逸	芋	姻	韻	詠	疫	悦	謁	猿	殴	翁	虞
12A0	卸	嫁	禍	彦	蚊	餓	悔	塊	戒	嚇	岳	樫	喝	褐	轄	且
12C0	勘	堪	棺	款	憾	艦	頑	忌	紀	飢	棋	宜	儀	吉	虐	朽
12E0	凶	峡	恭	矯	暁	斤	桐	菌	謹	襟	吟	隈	勲	薫	恵	渓
1300	蛍	鶏	鯨	遣	賢	繭	顕	玄	孤	枯	鼓	呉	悟	碁	后	恒
1320	皇	慌	酵	拷	豪	獄	昆	恨	紺	魂	墾	懇	佐	唆	詐	宰
1340	斎	歳	搾	桟	蚕	惨	暫	氏	司	祉	肢	嗣	飼	雌	賜	諮
1360	児	侍	滋	慈	璽	疾	執	漆	舎	赦	遮	邪	蛇	勺	酌	爵
1380	寂	朱	珠	儒	囚	州	宗	拾	愁	酬	醜	汁	銃	獣	叔	淑
13A0	粛	塾	俊	旬	准	殉	循	潤	遵	庶	如	徐	升	召	匠	肖
13C0	償	抄	昌	晶	松	沼	宵	症	祥	硝	粧	詔	彰	礁	丈	冗
13E0	畳	嬢	譲	醸	殖	嘱	辱	臣	娠	慎	薪	仁	迅	甚	尋	炊

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
1400	帥	衰	睡	穂	錘	随	髄	枢	崇	菅	畝	姓	斉	牲	逝	婿
1420	脆	夕	斥	隻	惜	拙	窃	摂	仙	践	銭	遷	薦	禅	祖	租
1440	措	疎	壮	荘	桑	曹	喪	葬	僧	遭	槽	燥	藻	霜	憎	賊
1460	孫	駄	舵	胎	泰	逮	瀧	沢	但	丹	胆	鍛	壇	痴	稚	畜
1480	逐	窒	嫡	衷	著	弔	眺	潮	聴	勅	朕	陳	鎮	津	漬	坪
14A0	呈	弟	邸	亭	貞	帝	艇	遜	偵	提	迭	哲	徹	澱	斗	渡
14C0	奴	桃	悼	棟	痘	唐	塔	搭	謄	豆	騰	洞	童	匿	篤	屯
14E0	豚	尼	弐	妊	忍	寧	婆	俳	輩	梅	培	陪	賠	伯	迫	漠
1500	鉢	伐	帆	畔	煩	頒	藩	晩	蛮	妃	披	卑	碑	罷	姫	漂
1520	苗	猫	賓	頻	瓶	扶	赴	膚	賦	附	譜	侮	覆	雰	墳	陛
1540	塀	弊	遍	舗	暮	邦	奉	峰	抱	俸	砲	褒	坊	某	冒	貿
1560	謀	朴	牧	墨	堀	奔	凡	盆	麻	妹	埋	又	抹	慢	漫	岬
1580	眠	娘	銘	妄	猛	匁	厄	愉	癒	唯	幽	悠	猶	裕	雄	憂
15A0	羊	庸	窯	擁	謡	翌	羅	酪	濫	吏	痢	履	柳	竜	硫	涼
15C0	猟	陵	僚	糧	厘	倫	零	齢	麗	廉	錬	炉	露	朗	廊	楼
15E0	賄	國	揺	條	櫻	澤	濾	碌	緞	鐵	靱	靖	槻	浩	郁	

FRYER MACHINE
Rob Zahe

G ALARM LIST

(1) Alarms on program and operation (PS alarm)
(2) Background edit alarms (BG alarm)
(3) Communication alarms (SR alarm)

Alarm numbers are common to all these alarm types.
Depending on the state, an alarm is displayed as in the following examples:
PS"alarm number" Example: PS0003
BG"alarm number" Example: BG0085
SR"alarm number" Example: SR0001

Number	Message	Description
0001	TH ERROR	A TH error was detected during reading from an input device. The read code that caused the TH error and how many statements it is from the block can be verified in the diagnostics screen.
0002	TV ERROR	An error was detected during the single–block TV error. The TV check can be suppressed by setting bit 0 (TVC) of parameter No. 0000 to “0”.
0003	TOO MANY DIGIT	Data entered with more digits than permitted in the NC instruction word. The number of permissible digits varies according to the function and the word.
0004	INVALID BREAK POINT OF WORDS	NC word(s) address + numerical value not in word format. This alarm is also generated when a custom macro does not contain a reserved word, or does not conform to the syntax.
0005	NO DATA AFTER ADDRESS	NC word(s) address + numerical value not in word format. This alarm is also generated when a custom macro does not contain a reserved word, or does not conform to the syntax.
0006	ILLEGAL USE OF MINUS SIGN	A minus sign (–) was specified at an NC instruction word or system variable where no minus signal may be specified.
0007	ILLEGAL USE OF DECIMAL POINT	A decimal point (.) was specified at an address where no decimal point may be specified, or two decimal points were specified.
0009	IMPROPER NC-ADDRESS	An illegal address was specified, or parameter 1020 is not set.
0010	IMPROPER G-CODE	(1) An unusable G code is specified. (2) The groove cutting by continuous circle motion option parameter is not effective. (3) The groove cutting by continuous circle motion enable signal is "0".
0011	FEED ZERO (COMMAND)	(1) The cutting feedrate instructed by an F code has been set to 0. (2) This alarm is also generated if the F code instructed for the S code is set extremely small in a rigid tapping instruction as the tool cannot cut at the programmed lead. (3) During groove cutting by continuous circle motion, correct Q or F value is not specified or the acceleration clamp value for continuous circle motion in parameter No. 3490 is invalid.

FRYER MACHINE SYSTEMS INC
Roman Ansky

Number	Message	Description
0015	TOO MANY SIMULTANEOUS AXES	A move command was specified for more axes than can be controlled by simultaneous axis control. Either add on the simultaneous axis control extension option, or divide the number of programmed move axes into two blocks.
0020	OVER TOLERANCE OF RADIUS	An arc was specified for which the difference in the radius at the start and end points exceeds the value set in parameter No. 3410. Check arc center codes I, J and K in the program. The tool path when parameter No. 3410 is set to a large value is spiral.
0021	ILLEGAL PLANE SELECT	(1) In the selected plane (by G17, G18 or G19 command), the basic axis(X,Y, or Z) and its parallel axis are commanded at the same time. Modify the program. (2) In the following functions, an axis not included in the selected plane was commanded, or the selected plane was not correct. Modify the program : - circular interpolation - involute interpolation - multiple repetitive cycle (if parameter DSA(No.5109#0) is set to 1.) etc. (3) In the helical interpolation or the helical interpolation B, the number of axes that can be commanded simultaneously is exceeded. Modify the program. Or add the helical interpolation option or the helical interpolation B option.
0022	R OR I,J,K COMMAND NOT FOUND	The command for circular interpolation lacks arc radius R or coordinate I, J, or K of the distance between the start point to the center of the arc.
0025	CIRCLE CUT IN RAPID (F0)	F0 (rapid traverse in inverse feed or feed specified by an F code with 1–digit number) was specified during circular interpolation (G02, G03) or involute interpolation (G02.2, G03.2).
0027	NO AXES COMMANDED IN G43/G44	No axis is specified in G43 and G44 blocks for the tool length offset type C. Offset is not canceled but another axis is offset for the tool length offset type C. Multiple axes were specified for the same block when the tool length compensation type is C.
0029	ILLEGAL OFFSET VALUE	Illegal offset No.
0030	ILLEGAL OFFSET NUMBER	An illegal offset No. was specified. This alarm is also generated when the tool shape offset No. exceeds the maximum number of tool offset sets in the case of tool offset memory B. In cutting point command, this alarm is also generated when the corner R offset No. ([H code + 2] or [H code + 2 * (parameter No. 11419)]) exceeds the maximum number of tool offset sets in the case of tool offset memory A and B.
0031	ILLEGAL P COMMAND IN G10	The relevant data input or option could not be found for the L No. of G10. No data setting address such as P or R was specified. An address command not concerned with data setting was specified. An address varies with the L No. The sign or decimal point of the specified address is in error, or the specified address is out of range.

FRYER MACHINE SYSTEMS INC
Rob Zabenski

Number	Message	Description
0032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.
0033	NO INTERSECTION AT CUTTER COMPENSATION	The intersection cannot be obtained by the intersection calculation in tool radius/tool nose radius compensation. Modify the program.
0034	ONLY G00/G01 ALLOWED IN STUP/EXT BLK	In tool radius/tool nose radius compensation, a startup or cancellation is performed in the following mode. Modify the program. G02/G03 : Circular interpolation or helical interpolation G35/G36 : Circular threading G02.2/G03.2 : Involute interpolation
0035	CAN NOT COMMANDED G31	- G31 cannot be specified. This alarm is generated when a G code (such as for tool radius/tool nose radius compensation) of group 07 is not canceled. - A torque limit skip was not specified in a torque limit skip command (G31P98 or P99). Specify the torque limit skip in the PMC window or the like. Or, specify the torque limit override by address Q.
0037	CAN NOT CHANGE PLANE IN G41/G42	The compensation plane G17/G18/G19 was changed during cutter or tool-nose radius compensation. Modify the program.
0041	INTERFERENCE IN CUTTER COMPENSATION	In tool radius/tool nose radius compensation, excessive cutting may occur. Modify the program.
0042	G45/G48 NOT ALLOWED IN CRC	Tool offset (G45 to G48) is commanded in tool radius compensation or 3-dimensional cutter compensation. Modify the program.
0044	G27-G30 NOT ALLOWED IN FIXED CYC	The command which relates to the reference position return is commanded in canned cycle mode for drilling. Modify the program.
0045	ADDRESS Q NOT FOUND (G73/G83)	In a high-speed peck drilling cycle (G73) or peck drilling cycle (G83), the amount of each-time cutting is not specified by address Q, or Q0 is specified. Modify the program.
0046	ILLEGAL REFERENCE RETURN COMMAND	A command for a return to the second, third or fourth reference position is error. (The address P command is in error.) Although an option for a return to the third or fourth reference position was not set, 3 or 4 was specified in address P.
0047	ILLEGAL AXIS SELECT	Two or more parallel axes (in parallel with a basic axis) have been specified upon start-up of 3-dimensional tool compensation or 3-dimensional coordinate conversion.
0048	BASIC 3 AXIS NOT FOUND	Start-up of 3-dimensional tool compensation or 3-dimensional coordinate conversion has been attempted, but the three basic axes used when Xp, Yp, or Zp is omitted are not set in parameter No. 1022.
0049	ILLEGAL COMMAND(G68,G69)	When 3-dimensional coordinate conversion (G68 or G69) was specified, the tool compensation was not canceled. Or, programs of 3-dimensional coordinate conversion (G68, G69) and tool compensation (G43, G44 or G49) were not nested. Or, the 3-dimensional coordinate conversion was specified during the tool length compensation and another tool length compensation was specified.
0050	CHF/CNR NOT ALLOWED IN THRD BLK	Chamfering or corner R is commanded in the thread cutting block. Modify the program.
0051	MISSING MOVE AFTER CNR/CHF	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	Description
0052	CODE IS NOT G01 AFTER CHF/CNR	The block next to the chamfering or corner R block is not G01 (or vertical line). Modify the program.
0053	TOO MANY ADDRESS COMMANDS	In the chamfering and corner R commands, two or more of I, J, K and R are specified.
0054	NO TAPER ALLOWED AFTER CHF/CNR	A block in which chamfering in the specified angle or the corner R was specified includes a taper command. Modify the program.
0055	MISSING MOVE VALUE IN CHF/CNR	In chamfering or corner R block, the move distance is less than chamfer or corner R amount. Modify the program.
0056	NO END POINT & ANGLE IN CHF/CNR	In direct dimension drawing programming, both an end point and an angle were specified in the block next to the block in which only an angle was specified (Aa). Modify the program.
0057	NO SOLUTION OF BLOCK END	Block end point is not calculated correctly in direct dimension drawing programming. Modify the program.
0058	END POINT NOT FOUND	Block end point is not found in direct dimension drawing programming. Modify the program.
0060	SEQUENCE NUMBER NOT FOUND	[External data input/output] The specified number could not be found for program number and sequence number searches. Although input/output of a pot number of tool data or offset input was requested, no tool number was input after power on. The tool data corresponding to the entered tool number could not be found. [External workpiece number search] The program corresponding to the specified workpiece number could not be found. [Program restart] In the program restart sequence number specification, the specified sequence number could not be found. Check the restart block.
0061	P OR Q COMMAND IS NOT IN THE MULTIPLE REPETIVE CYCLES BLOCK	Address P or Q is not specified in multiple repetitive cycle (G70, G71, G72, or G73) command.
0062	THE CUTTING AMOUNT IS ILLEGAL IN THE ROUGH CUTTING CYCLE	A zero or a negative value was specified in a multiple repetitive canned rough-cutting cycle (G71 or G72) as the depth of cut.
0063	THE BLOCK OF A SPECIFIED SEQUENCE NUMBER IS NOT FOUND	The sequence number specified by addresses P and Q in multiple repetitive cycle (G70, G71, G72, or G73) command cannot be searched.
0064	THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(FIRST AXES)	In a shape program for the multiple repetitive canned rough-cutting cycle (G71 or G72), the command for the first plane axis was not a monotonous increase or decrease.
0065	G00/G01 IS NOT IN THE FIRST BLOCK OF SHAPE PROGRAM	In the first block of the shape program specified by P of the multiple repetitive canned cycle (G70, G71, G72, or G73), G00 or G01 was not specified.
0066	UNAVAILABLE COMMAND IS IN THE MULTIPLE REPETIVE CYCLES BLOCK	An unavailable command was found in a multiple repetitive canned cycle (G70, G71, G72, or G73) command block.
0067	THE MULTIPLE REPETIVE CYCLES IS NOT IN THE PART PROGRAM STORAGE	A multiple repetitive canned cycle (G70, G71, G72, or G73) command is not registered in a tape memory area.
0069	LAST BLOCK OF SHAPE PROGRAM IS AN ILLEGAL COMMAND	In a shape program in the multiple repetitive canned cycle (G70, G71, G72, or G73), a command for the chamfering or corner R in the last block is terminated in the middle.
0070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
0071	DATA NOT FOUND	- The address to be searched was not found. - The program with specified program number was not found in program number search. - In the program restart block number specification, the specified block number could not be found. Check the data.
0072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 63 (basic), 125 (option), 200 (option), 400 (option) or 1000 (option). Delete unnecessary programs and execute program registration again.
0073	PROGRAM NUMBER ALREADY IN USE	The commanded program number has already been used. Change the program number or delete unnecessary programs and execute program registration again.
0074	ILLEGAL PROGRAM NUMBER	The program number is other than 1 to 9999. Modify the program number.
0075	PROTECT	An attempt was made to register a program whose number was protected. In program matching, the password for the encoded program was not correct. An attempt was made to select a program being edited in the background as the main program. An attempt was made to call a program being edited in the background as a subprogram.
0076	PROGRAM NOT FOUND	The specified program is not found in the subprogram call, macro call or graphic copy. The M, G, T or S codes are called by a P instruction other than that in an M98, G65, G66, G66.1 or interrupt type custom macro, and a program is called by a No. 2 auxiliary function code. This alarm is also generated when a program is not found by these calls.
0077	TOO MANY SUB,MACRO NESTING	The total number of subprogram and macro calls exceeds the permissible range. Another subprogram call was executed during an external memory subprogram call.
0078	SEQUENCE NUMBER NOT FOUND	The specified sequence No. was not found during sequence number search. The sequence No. specified as the jump destination in GOTO— and M99P— was not found.
0079	PROGRAM NOT MATCH	The program in memory does not match the program stored on tape. Multiple programs cannot be matched continuously when bit 6 of parameter No. 3201 is set to “1”. Set bit 6 of parameter No. 3201 to “0” before executing a match.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	Description
0080	G37 MEASURING POSITION REACHED SIGNAL IS NOT PROPERLY INPUT	- For machining center series When the tool length measurement function (G37) is performed, a measuring position reached signal goes 1 in front of the area determined by the ε value specified in parameter No.6254. Alternatively, the signal does not go 1. - For lathe When the automatic tool compensation function (G36, G37) is used, a measuring position reached signals (XAE1, XAE2) does not go 1 within the range determined by the ε value specified in parameters Nos.6254 and 6255.
0081	G37 OFFSET NO. UNASSIGNED	- For machining center series The tool length measurement function (G37) is specified without specifying an H code. Correct the program. - For lathe The automatic tool compensation function (G36, G37) is specified without specifying an T code. Correct the program.
0082	G37 SPECIFIED WITH H CODE	- For machining center series The tool length measurement function (G37) is specified together with an H code in the same block. Correct the program. - For lathe The automatic tool compensation function (G36, G37) is specified together with an T code in the same block. Correct the program.
0083	G37 IMPROPER AXIS COMMAND	- For machining center series An error has been found in axis specification of the tool length measurement function (G37). Alternatively, a move command is specified as an incremental command. Correct the program. - For lathe An error has been found in axis specification of the automatic tool compensation function (G36, G37). Alternatively, a command is specified as an incremental command. Correct the program.
0085	OVERRUN ERROR	The next character was received from the I/O device connected to RS-232C interface 1 before it could read a previously received character. During a read by RS-232C interface 1, overrun, parity or framing error has been occurred. Number of bits of input data is mismatch, baud rate setting or I/O device specification number is wrong.
0086	DR OFF	During I/O process by RS-232C interface 1, the data set ready input signal of the I/O device (DR) was OFF. Possible causes are an I/O device not turn on, a broken cable, and a defective printed circuit board.
0087	BUFFER OVERFLOW	During a read by RS-232C interface 1, although a read stop command was issued, more than 10 characters were input. The I/O device or printed circuit board was defective.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
0090	REFERENCE RETURN INCOMPLETE	(1) The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return. (2) An attempt was made to set the zero position for the absolute position detector by return to the reference position when it was impossible to set the zero point. Rotate the motor manually at least one turn, and set the zero position of the absolute position detector after turning the CNC and servo amplifier off and then on again.
0091	MANUAL REFERENCE POSITION RETURN IS NOT PERFORMED IN FEED HOLD	Manual return to the reference position cannot be performed when automatic operation is halted. Perform the manual return to the reference position when automatic operation is stopped or reset.
0092	ZERO RETURN CHECK (G27) ERROR	The axis specified in G27 has not returned to zero. Reprogram so that the axis returns to zero.
0094	P TYPE NOT ALLOWED (COORD CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.) Perform the correct operation according to the OPERATOR'S MANUAL.
0095	P TYPE NOT ALLOWED (EXT OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the external workpiece offset amount changed.) Perform the correct operation according to the OPERATOR'S MANUAL.
0096	P TYPE NOT ALLOWED (WRK OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece offset amount changed.) Perform the correct operation according to the OPERATOR'S MANUAL.
0097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be directed when the program is restarted. (After power ON, after emergency stop or alarms 0094 to 0097 reset, no automatic operation is performed.) Perform automatic operation.
0098	G28 FOUND IN SEQUENCE RETURN	A command of the program restart was specified without the reference position return operation after power ON or emergency stop, and G28 was found during search. Perform the reference position return.
0099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI.
0109	FORMAT ERROR IN G08	A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.
0110	OVERFLOW :INTEGER	An integer went out of range during arithmetic calculations.
0111	OVERFLOW :FLOATING	A decimal point (floating point number format data) went out of range during arithmetic calculations.
0112	ZERO DIVIDE	An attempt was made to divide by zero in a custom macro.
0114	ILLEGAL EXPRESSION FORMAT	The format used in an expression in a custom macro statement is in error. The parameter tape format is in error.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
0115	VARIABLE NO. OUT OF RANGE	A number that cannot be used for a local variable, common variable, or system variable in a custom macro is specified. In the EGB axis skip function or skip function for flexible synchronization control (G31.8), a non-existent custom macro variable number is specified. Or, the number of custom macro variables used to store skip positions is not sufficient. Alternatively, the header data in high-speed cycle machining is improper. This alarm is issued in the following cases. (1) The header corresponding to the specified call machining cycle number is absent. (2) The value of cycle connection information falls outside the allowable range (0 to 999). (3) The number of data items in the header falls outside the allowable range (1 to 65535). (4) The storage start data variable number of executable data falls outside the allowable ranges (#20000 to #85535/#200000 to #986431/#2000000 to #3999999). (5) The storage end data variable number of executable data falls outside the allowable ranges (#85535/#986431/#3999999). (6) The storage start data variable number of executable data is the same as the variable number used by the header.
0116	WRITE PROTECTED VARIABLE	An attempt was made in a custom macro to use on the left side of an expression a variable that can only be used on the right side of an expression.
0118	TOO MANY BRACKET NESTING	Too many brackets “[ ]” were nested in a custom macro. The nesting level including function brackets is 5.
0119	ARGUMENT VALUE OUT OF RANGE	The value of an argument in a custom macro function is out of range.
0122	TOO MANY MACRO NESTING	Too many macro calls were nested in a custom macro.
0123	ILLEGAL MODE FOR GOTO/WHILE/DO	A GOTO statement or WHILE–DO statement was found in the main program in the MDI or DNC mode.
0124	MISSING END STATEMENT	The END instruction corresponding to the DO instruction was missing in a custom macro.
0125	MACRO STATEMENT FORMAT ERROR	The format used in a macro statement in a custom macro is in error.
0126	ILLEGAL LOOP NUMBER	DO and END Nos. in a custom macro are in error, or exceed the permissible range (valid range: 1 to 3).
0127	DUPLICATE NC,MACRO STATEMENT	An NC statement and macro statement were specified in the same block.
0128	ILLEGAL MACRO SEQUENCE NUMBER	The specified sequence No. could not be found for sequence number search. The sequence No. specified as the jump destination in GOTO-- and M99P-- could not be found.
0129	USE 'G' AS ARGUMENT	Illegal address is commanded in a custom macro call. Confirm whether the address is valid in the macro call. When using the axis name expansion, confirm the setting of the parameter No.11647.
0130	NC AND PMC AXIS ARE CONFLICTED	(1) The NC command and the PMC axis control command were conflicted. Modify the program or ladder. (2) Auxiliary function for PMC axis control was commanded for the group used with real-time custom macro auxiliary function.
0136	SPOS AXIS - OTHER AXIS SAME TIME	The spindle positioning axis and another axis are specified in the same block.

FRYER MACHINERY SYSTEMS INC
Rob Zahensky

Number	Message	Description
0137	M-CODE & MOVE CMD IN SAME BLK.	The spindle positioning axis and another axis are specified in the same block.
0139	CANNOT CHANGE PMC CONTROL AXIS	The PMC axis was selected for the axis for which the PMC axis is being controlled.
0140	PROGRAM NUMBER ALREADY IN USE	In the background, an attempt was made to select or delete the program being selected in the foreground. Perform the correct operation for the background edition.
0141	CAN NOT COMMAND G51 IN 3-D OFFSET	G51 (Scaling ON) is commanded in the 3-dimensional tool compensation mode. Modify the program.
0142	ILLEGAL SCALE RATE	The scaling rate is 0 times or 10000 times or more. Modify the setting of the scaling rate. (G51P_ … or G51I_J_K_ … or parameter No. 5411 or 5421)
0143	COMMAND DATA OVERFLOW	An overflow occurred in the storage length of the CNC internal data. This alarm is also generated when the result of internal calculation of scaling, coordinate rotation and cylindrical interpolation overflows the data storage. It also is generated during input of the manual intervention amount.
0144	ILLEGAL PLANE SELECTED	The coordinate rotation plane and arc or tool radius·tool nose radius compensation plane must be the same. Modify the program.
0145	ILLEGAL USE OF G12.1/G13.1	The axis No. of plane selection parameter No. 5460 (linear axis) and No. 5461(axis of rotation) in the polar coordinate interpolation mode is out of range (1 to number of controlled axes).
0146	ILLEGAL USE OF G-CODE	The modal G code group contains an illegal G code in the polar coordinate interpolation mode or when a mode was canceled. Only the following G codes are allowed: G40, G50, G69.1 An illegal G code was specified while in the polar coordinate interpolation mode. The following C codes are not allowed: G27, G28, G30, G30.1, G31 to G31.4, G37 to G37.3, G52, G92, G53, G17 to G19, G81 to G89, G68 In the 01 group, G codes other than G01, G02, G03, G02.2 and G03.2 cannot be specified.
0149	FORMAT ERROR IN G10L3	In registration (G10L3 to G11) of tool life management data, an address other than Q1, Q2, P1, and P2 or an unusable address was specified.
0150	ILLEGAL LIFE GROUP NUMBER	The tool group number exceeded the maximum allowable value. The tool group number (P after specification of G10 L3;) or the group number given by the tool life management T code in a machining program.
0151	GROUP NOT FOUND AT LIFE DATA	The tool group specified in a machining program is not set in tool life management data.
0152	OVER MAXIMUM TOOL NUMBER	The number of tools registered in one group exceeded the maximum allowable registration tool number.
0153	T-CODE NOT FOUND	In registration of tool life data, a block in which the T code needs to be specified does not include the T code. Alternatively, in tool exchange method D, M06 is specified solely. Modify the program.

FRYER MACHINE SYSTEMS INC
Robertamensky

Number	Message	Description
0154	NOT USING TOOL IN LIFE GROUP	- For the tool management command H99 or D99 was specified when no tool management data number is assigned to the spindle position. Modify the program. - For the tool life management command The H99 command, D99 command, or the H/D code set by parameters Nos. 13265 and 13266 was specified when no tool belonging to a group is used.
0155	ILLEGAL T-CODE COMMAND	In the machining program, the T code that is present in the block containing M06 does not correspond to the group currently being used. Modify the program.
0156	P/L COMMAND NOT FOUND	The P and L commands are not specified in the beginning of a program for setting a tool group. Modify the program.
0157	TOO MANY TOOL GROUPS	In registration of tool life management data, the group setting command block counts of P (group number) and L (tool life) exceeded the maximum group count.
0158	TOOL LIFE VALUE OUT OF RANGE	The life value that is being set is too large. Change the setting.
0159	ILLEGAL TOOL LIFE DATA	Tool life management data is corrupted for some reason. Register the tool data in the tool group or the tool data in the group again by G10L3; or MDI input.
0160	MISMATCH WAITING M-CODE	A waiting M-code is in error. (1) When different M codes are specified for path 1 and path 2 as waiting M codes without a P command. (2) When the waiting M codes are not identical even though the P commands are identical (3) When the waiting M codes are identical and the P commands are not identical (This occurs when a P command is specified with binary value.) (4) When the number lists in the P commands contain a different number even though the waiting M codes are identical (This occurs when a P command is specified by combining path numbers.) (5) When a waiting M code without a P command (2-path waiting) and a waiting M code with a P command (3-or-more-path waiting) were specified at the same time (6) When a waiting M code without a P command was specified for 3 or more paths. (7) When the waiting function by specifying start point and a waiting M code without a P command (2-path waiting) were specified at the same time.
0161	ILLEGAL P OF WAITING M-CODE	P in a waiting M-code is incorrect. (1) When address P is negative (2) When a P value inappropriate for the system configuration was specified (3) When a waiting M code without a P command (2-path waiting) was specified in the system having 3 or more paths. (4) When P7 and L0 were specified at the same time.
0163	ILLEGAL COMMAND IN G68/G69	G68 and G69 are not independently commanded in balance cut. An illegal value is commanded in a balance cut combination (address P).
0169	ILLEGAL TOOL GEOMETRY DATA	Incorrect tool figure data in interference check. Set correct data, or select correct tool figure data.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
0175	ILLEGAL G07.1 AXIS	An axis which cannot perform cylindrical interpolation was specified. More than one axis was specified in a G07.1 block. An attempt was made to cancel cylindrical interpolation for an axis that was not in the cylindrical interpolation mode. For the cylindrical interpolation axis, set not “0” but one of 5, 6 or 7 (parallel axis specification) to parameter No. 1022 to instruct the arc with axis of rotation (ROT parameter No. 1006#1 is set to “1” and parameter No. 1260 is set) ON.
0176	ILLEGAL G-CODE USE(G07.1 MODE)	A G code was specified that cannot be specified in the cylindrical interpolation mode. This alarm also is generated when an 01 group G code was in the G00 mode or code G00 was instructed. Cancel the cylindrical interpolation mode before instructing code G00.
0177	CHECK SUM ERROR (G05)	A checksum error occurred.
0178	ILLEGAL COMMAND G05	This alarm is issued in the following cases. (1) The settings of bits 4 to 6 of parameter No.7501 are invalid. (2) G05 was specified in any of the following mode. - Hypothetical axis interpolation (G07) - Cylindrical interpolation (G07.1) - Polar coordinate interpolation (G12.1) - Polar coordinates command (G16) - Spindle speed fluctuation detection (G26) - Tool radius · tool nose radius compensation (G41/G42) - Normal direction control (G41.1/G42.1) - Scaling (G51) - Programmable mirror image (G51.1) - Coordinate system rotation (G68) - Canned cycle for drilling (Except for G80) - Constant surface speed control (G96) - Interruption type custom macro (M96) (3) G05 was specified while the Servo invalidation function and the Pole position detection function are effective at the same time.
0179	PARAM. (NO.7510) SETTING ERROR	The number of controlled axes set by the parameter No. 7510 exceeds the maximum number. Modify the parameter setting value. The distribution of high-speed cycle machining or high-speed binary program operation stopped.
0190	ILLEGAL AXIS SELECTED (G96)	An illegal value was specified in P in a G96 block or parameter No. 3770. When spindle selection by address P or extended spindle name of multi-spindle control is enabled, selection of axis as the calculation reference in constant surface speed control “G96 P_;” cannot be commanded.
0194	SPINDLE COMMAND IN SYNCHRO-MODE	A Cs contour control mode, spindle positioning command, or rigid tapping mode was specified during the spindle synchronous control mode or spindle command synchronous control mode.
0197	C-AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cs-axis when the Cs contour control switching signal was off.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

Number	Message	Description
0200	ILLEGAL S CODE COMMAND	(1) In the rigid tap, an S value was out of range or was not specified. The parameters Nos. 5241 to 5243 setting is an S value which can be specified for the rigid tap. Correct the parameters or modify the program. (2) Rigid tapping is commanded during constant surface speed control. Command rigid tapping after canceling constant surface speed control.
0201	FEEDRATE NOT FOUND IN RIGID TAP	The command F code for a cutting feedrate is a zero. If the value of F command is much smaller than that of the S command, when a rigid tap command is specified, this alarm is generated. This is because cutting is not possible by the lead specified by the program.
0202	POSITION LSI OVERFLOW	In the rigid tap, spindle distribution value is too large. Decrease spindle speed.
0203	PROGRAM MISS AT RIGID TAPPING	(1) In the rigid tap, position for a rigid M code (M29) or an S command is incorrect. Modify the program. (2) FSSB high-speed rigid tapping cannot be used on the rigid tapping with one servo motor and multiple spindle motors by spindle command synchronous control. (3) In FSSB high-speed rigid tapping, parameter setting is wrong (bits 1 and 2 of parameter No.2429, and bits 1 and 2 of parameter No.4549). (4) In FSSB high-speed rigid tapping, parameter No.24204 is wrong. Set the same index number of the spindle to three axes of 3-dimensional rigid tapping.
0204	ILLEGAL AXIS OPERATION	In the rigid tap, an axis movement is specified between the rigid M code (M29) block and G84 (or G74) block. Modify the program.
0205	RIGID MODE DI SIGNAL OFF	(1) Although a rigid M code (M29) is specified in rigid tapping, the rigid mode DI signal (G061.0) is not ON during execution of the G84 (or G74) block. Check the PMC ladder diagram to find the reason why the DI signal is not turned on. (2) In rigid tapping, no spindle has selected commands in the path of interest. Alternatively, spindles in two or more paths have selected commands in the same path. Check the spindle command select signal. (3) A spindle specified for rigid tapping is already being used in another path. (4) In a machine provided with the multi-spindle control, no spindle has been selected for attempted rigid tapping. (5) Rigid tapping command is issued to a spindle control axis with servo motor in a path other than the command issuing path.
0206	CAN NOT CHANGE PLANE (RIGID TAP)	Plane changeover was instructed in the rigid mode. Modify the program.
0207	RIGID DATA MISMATCH	The specified distance was too short or too long in rigid tapping.
0210	CAN NOT COMMAND M198/M99	(1) The execution of an M198 or M99 command was attempted during scheduled operation. Alternatively, the execution of an M198 command was attempted during DNC operation. Modify the program. (2) The execution of an M99 command was attempted by an interrupt macro during pocket machining in a multiple repetitive canned cycle.

FRYER MACHINE SYSTEMS INC
Rob Zaborsky

Number	Message	Description
0213	ILLEGAL COMMAND IN SYNCHRO-MODE	In axis synchronous control, the following errors occurred during the synchronous operation. (1) The program issued the move command to the slave axis. (2) The manual operation (jog feed or incremental feed) is performed to the slave axis. (3) The program issued the automatic reference position return command without specifying the manual reference position return after the power was turned on.
0214	ILLEGAL COMMAND IN SYNCHRO-MODE	Coordinate system is set or tool compensation of the shift type is executed in the synchronous control. Correct the program.
0217	DUPLICATE G51.2(COMMANDS)	G51.2 is further commanded in the G51.2 mode. Modify the program.
0218	NOT FOUND P/Q COMMAND	P or Q is not commanded in the G51.2 block, or the command value is out of the range. Modify the program. For a polygon turning between spindles, more information as to why this alarm occurred is indicated in diagnosis data No. 0471.
0219	COMMAND G51.2/G50.2 INDEPENDENTLY	G51.2 and G50.2 were specified in the same block for other commands. Modify the program in another block.
0220	ILLEGAL COMMAND IN SYNCHR-MODE	In the synchronous operation, movement is commanded by the NC program or PMC axis control interface for the synchronous axis. Modify the program or check the PMC ladder.
0221	ILLEGAL COMMAND IN SYNCHR-MODE	Polygon machining synchronous operation and axis control or balance cutting are executed at a time. Modify the program.
0224	ZERO RETURN NOT FINISHED	A reference return has not been performed before the start of automatic operation. (Only when bit 0 (ZRNx) of parameter No. 1005 is 0) Perform a reference position return.
0230	R CODE NOT FOUND	Cut depth R is not specified in the block including G161. Alternatively, the value specified for R is negative. Modify the program.
0231	ILLEGAL FORMAT IN G10 L52	Errors occurred in the specified format at the programmable-parameter input.
0232	TOO MANY HELICAL AXIS COMMAND	Three or more axes were specified as helical axes in the helical interpolation mode. Five or more axes were specified as helical axes in the helical interpolation B mode.
0233	DEVICE BUSY	When an attempt was made to use a unit such as that connected via the RS-232C interface, other users were using it.
0241	ILLEGAL FORMAT IN G02.2/G03.2	The end point of an involute curve on the currently selected plane, or the center coordinate instruction I, J or K of the corresponding basic circle, or basic circle radius R was not specified.
0242	ILLEGAL COMMAND IN G02.2/G03.2	An illegal value was specified in the involute curve. The coordinate instruction I, J or K of the basic circle on the currently selected plane or the basic circle radius R is 0, or the start and end points are not inside the basic circle.
0243	OVER TOLERANCE OF END POINT	The end point is not positioned on the involute curve that passes through the start point, and this error exceeds the permissible error limit (parameter No. 5610).
0245	T-CODE NOT ALLOWED IN THIS BLOCK	One of the G codes, G50, G10, G04, G28, G28.2, G29, G30, G30.2, G30.1, and G53, which cannot be specified in the same block as a T code, was specified with a T code.
0247	THE MISTAKE IS FOUND IN THE OUTPUT CODE OF DATA.	When an encrypted program is output, EIA is set for the output code. Specify ISO.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
0250	TOOL CHANGE ILLEGAL Z AXIS COMMAND	A Z-axis move command was performed in the same block for M06 command.
0251	TOOL CHANGE ILLEGAL T COMMAND	An unusable T code was specified in M06Txx.
0253	G05 CAN NOT BE COMMANDED	A binary operation was specified during advanced preview control mode.
0300	ILLEGAL COMMAND IN SCALING	An illegal G code was specified during scaling. Modify the program. For the T system, one of the following functions is specified during scaling, this alarm is generated. - finishing cycle (G70 or G72) - outer surface rough-cutting cycle (G71 or G73) - end side rough-cutting cycle (G72 or G74) - closed loop cutting cycle (G73 or G75) - end side cutting-off cycle (G74 or G76) - outer surface or inner surface cutting-off cycle (G75 or G77) - multiple repetitive threading cycle (G76 or G78) - face drill cycle (G83 or G83) - face tap cycle (G84 or G84) - face boring cycle (G85 or G85) - side drill cycle (G87 or G87) - side tap cycle (G88 or G88) - side boring cycle (G89 or G89) - outer surface turning cycle or inner surface boring cycle (G77 or G20) - threading cycle (G78 or G21) - end side turning cycle (G79 or G24) (Specify G codes for systems B and C in that order.)
0301	RESETTING OF REFERENCE RETURN IS INHIBITED	Although bit 0 (IDGx) of parameter No. 1012 was set to 1 to inhibit the reference position from being set again for a return to the reference position without a dog, an attempt was made to perform a manual return to the reference position.
0302	SETTING THE REFERENCE POSITION WITHOUT DOG IS NOT PERFORMED	The reference position could not be set for a return to the reference position without a dog. Possible causes are: - The axis was not moved in the direction of a return to the reference position for jog feeding. - The axis was moved in the direction opposite to the direction of a manual return to the reference position. - Since the one-rotation signal sent from the position detector is not detected, the grid for manual reference position return is not established. (Bit 6 of diagnostic data No. 0201 must be 1.)
0303	REFERENCE POSITION RETURN IS NOT PERFORMED	When the setting of a reference position at any position was possible in Cs contour control (bit 0 (CRF) of parameter No. 3700 = 1), a G00 command was issued for the Cs contour axis without a return to the reference position after the serial spindle was switched to Cs contour control mode. Perform a reference position return with a G28 command before issuing a G00 command.
0304	G28 IS COMMANDED WITHOUT ZERO RETURN	Although a reference position was not set, an automatic return to the reference position (G28) was commanded.
0305	INTERMEDIATE POSITION IS NOT ASSIGNED	Although a G28 (automatic return to the reference position), G30 (return to the second, third, or fourth reference position), or G30/1 (return to the floating reference position) command was not issued after power-up, G29 (return from the reference position) was commanded.

FRYER MACHINE SYSTEMS INC
Rob Zapersky

Number	Message	Description
0306	MISMATCH AXIS WITH CNR/CHF	The correspondence between the moving axis and the I, J, or K command is incorrect in a block in which chamfering is specified.
0307	CAN NOT START REFERENCE RETURN WITH MECHANICAL STOPPER SETTING	An attempt was made to set a butt-type reference position for an axis for which to use the function to set a reference position without a dog.
0308	G72.1 NESTING ERROR	G72.1 was specified again during G72.1 rotation copying.
0309	G72.2 NESTING ERROR	G72.2 was specified again during G72.2 parallel copying.
0310	FILE NOT FOUND	The specified file could not be found during a subprogram or macro call.
0311	CALLED BY FILE NAME FORMAT ERROR	An invalid format was specified to call a subprogram or macro using a file name.
0312	ILLEGAL COMMAND IN DIRECT DRAWING DIMENSIONS PROGRAMMING	Direct input of drawing dimensions was commanded in an invalid format. An attempt was made to specify an invalid G code during direct input of drawing dimensions. Two or more blocks not to be moved exist in consecutive commands that specify direct input of drawing dimensions. Although non-use of commas (,) (bit 4 of parameter No. 3405 = 1) was specified for direct input of drawing dimensions, a comma was specified.
0313	ILLEGAL LEAD COMMAND	The variable-lead threading increment specified in address K exceeds the specified maximum value in variable-lead threading. Or, a negative lead value was specified.
0314	ILLEGAL SETTING OF POLYGONAL AXIS	An axis was specified invalidly in polygon turning. For polygon turning: A tool rotation axis is not specified. (Parameter No. 7610) For polygon turning between spindles: Valid spindles are not specified. (Parameters Nos. 7640 to 7643) - A spindle other than the serial spindle. - A spindle is not connected. For concurrent use of polygon turning and polygon turning with two spindles: - In the polygon turning mode, the value of parameter No. 7605 (selecting the type of polygon turning) was changed. - An attempt is made to use a spindle used for polygon turning also for polygon turning with two spindles.
0315	ILLEGAL NOSE ANGLE COMMAND IS IN THE THREAD CUTTING CYCLE	An invalid tool tip angle is specified in a multiple repetitive canned threading cycle (G76).
0316	ILLEGAL CUTTING AMOUNT IS IN THE THREAD CUTTING CYCLE	A minimum depth of cut higher than the thread height is specified in a multiple repetitive canned threading cycle (G76).
0317	ILLEGAL THREAD COMMAND IS IN THE THREAD CUTTING CYCLE	A zero or a negative value is specified in a multiple repetitive canned threading cycle (G76) as the thread height or the depth of cut.
0318	ILLEGAL RELIEF AMOUNT IS IN THE DRILLING CYCLE	Although an escape directions is set in a multiple repetitive canned cutting-off cycle (G74 or G75), a negative value is specified for Δd.
0319	THE END POINT COMMAND IS ILLEGAL IN THE DRILLING CYCLE	Although the Δi or Δk travel distance is set to 0 in a multiple repetitive canned cutting-off cycle (G74 or G75), a value other than 0 us specified for a U or W.
0320	ILLEGAL MOVEMENT AMOUNT/CUTTING AMOUNT IS IN THE DRILLING CYCLE	A negative value is specified in a multiple repetitive canned cutting-off cycle (G74 or G75) as Δi or Δk (travel distance/the depth of cut).

FRYER MACHINE SYSTEMS INC
Rob Zaharnsky

Number	Message	Description
0321	ILLEGAL REPEATED TIME IS IN THE PATTERN REPEATING CYCLE	A zero or a negative value is specified in a multiple repetitive canned closed loop cycle (G73) as a repeated time.
0322	FINISHING SHAPE WHICH OVER OF STARTING POINT	An invalid shape which is over the cycle starting point is specified in a shape program for a multiple repetitive canned rough-cutting cycle (G71 or G72).
0323	THE FIRST BLOCK OF SHAPE PROGRAM IS A COMMAND OF TYPE II	Type II is specified in the first block of the shape program specified by P in a multiple repetitive canned rough-cutting cycle (G71 or G72). Z (W) command is for G71. X (U) command is for G72.
0324	THE INTERRUPTION TYPE MACRO WAS DONE IN THE MULTIPLE REPETIVE CYCLES	An interruption type macro was issued during the multiple repetitive canned cycle (G70, G71, G72, or G73).
0325	UNAVAILABLE COMMAND IS IN SHAPE PROGRAM	(1) An usable command was issued in a shape program for a multiple repetitive cycle (G70, G71, G72, or G73). (2) In the multiple repetitive cycle (G70), when the tool nose radius compensation can be commanded in the target figure program (parameter NT2(No.5106#3)=1), G41 or G42 is commanded at the target figure program other than the first block.
0326	LAST BLOCK OF SHAPE PROGRAM IS A DIRECT DRAWING DIMENSIONS	In a shape program in the multiple repetitive canned cycle (G70, G71, G72, or G73), a command for direct input of drawing dimensions in the last block is terminated in the middle.
0327	MODAL THAT MULTIPLE REPETIVE CYCLES CANNOT BE DONE	A multiple repetitive canned cycle (G70, G71, G72, or G73) was commanded in a modal state in which a multiple repetitive canned cycle could not be commanded.
0328	ILLEGAL WORK POSITION IS IN THE TOOL NOSE RADIUS COMPENSATION	The specification for the blank side for a tool nose radius compensation (G41 or G42) is incorrect in a multiple repetitive canned cycle (G71 or G72).
0329	THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(SECOND AXES)	In a shape program for the multiple repetitive canned rough-cutting cycle (G71 or G72), the command of the second plane axis was not a monotonous increase or decrease.
0330	ILLEGAL AXIS COMMAND IS IN THE TURNING CANNED CYCLE	An axis other than the plane is specified n a canned cycle(G90, G92, or G94).
0331	ILLEGAL AXIS NUMBER IN AX[]	An illegal value is specified for an AX[] axis number.
0332	ILLEGAL AXIS ADDRESS IN AXNUM[]	An illegal value is specified for an AXNUM[] axis address.
0333	TOO MANY SPINDLE COMMANDS	Multiple spindle commands could be found in the same block in using an expansion spindle name. Only one spindle could be commanded in the same block.
0334	INPUT VALUE OUT OF EFFECTIVE RANGE	An offset data which was out of the effective range was specified. (malfunction prevention function)
0335	PLURAL M CODE	Multiple M codes are commanded simultaneously in a block for a wait function with peripheral devices by an M code.
0336	TOOL COMPENSATION COMMANDED MORE TWO AXES	For a tool length compensation C, an attempt was made to command the offset to other axes without canceling the offset. Or, for a tool length compensation C, multiple axes are specified in G43 or G44 block.
0337	EXCESS MAXIMUM INCREMENTAL VALUE	The command value exceeded the maximum amount of incremental. (malfunction prevention function)
0345	TOOL CHANGE ILLEGAL Z AXIS POS	A tool change position on the Z-axis is incorrect.
0346	TOOL CHANGE ILLEGAL TOOL NUM	A tool change position is not set.
0347	TOOL CHANGE ILLEGAL COMMAND IN SAME BLK.	Tool changing is commanded twice or more in the same block.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
0348	TOOL CHANGE Z AXIS POS NOT ESTABLISHED	A tool change spindle on the Z-axis is not set.
0349	TOOL CHANGE SPINDLE NOT STOP	A tool change spindle stop is not stopped.
0350	PARAMETER OF THE INDEX OF THE SYNCHRONOUS CONTROL AXIS SET ERROR.	An illegal synchronization control axis number (parameter No. 8180) is set.
0351	BECAUSE THE AXIS IS MOVING, THE SYNC CONTROL IS CAN'T BE USED.	While the axis being subject to synchronization control was moving, an attempt was made to start or cancel the synchronization control by a synchronization control axis selection signal.
0352	SYNCHRONOUS CONTROL AXIS COMPOSITION ERROR.	This error occurred when: (1) An attempt was made to perform synchronization control for the axis during a synchronization, composition, or superposition. (2) An attempt was made to synchronize a further great-grandchild for a parent-child-grandchild relation. (3) An attempt was made to operate synchronization control although a parent-child-grandchild relation was not set.
0353	THE INSTRUCTION WAS DONE FOR THE AXIS WHICH WAS NOT ABLE TO MOVE.	This error occurred when: - For synchronization 1) A move command was issued to the axis for which bit 7 (NUMx) of parameter No. 8163 is set to 1. 2) A move command was issued to the slave axis. - For composition 1) A move command was issued to the axis for which bit 7 (NUMx) of parameter No. 8163 is set to 1. 2) A move command was issued to the axis for which bit 7 (MUMx) of parameter No. 8162 is set to 1.
0354	THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN SYNC MODE	This error occurred when G28 was specified to the master axis being parking during synchronization control, but an axis reference position is not set for the slave axis.
0355	PARAMETER OF THE INDEX OF THE COMPOSITE CONTROL AXIS SET ERROR.	An illegal composite control axis number (parameter No. 8183) is specified.
0356	BECAUSE THE AXIS IS MOVING, THE COMP CONTROL IS CAN'T BE USED.	While the axis being subject to composite control was moving, an attempt was made to start or cancel the composite control by a composite control axis selection signal.
0357	COMPOSITE CONTROL AXIS COMPOSITION ERROR.	This error occurred when an attempt was made to perform composite control for the axis during a synchronization, composition, or superposition.
0359	THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN COMP MODE	This error occurred when G28 was specified to the composite axis during composite control, but a reference position is not set to the other part of the composition.
0360	PARAMETER OF THE INDEX OF THE SUPERPOS CONTROL AXIS SET ERROR.	An illegal superposition control axis number (parameter No. 8186) is specified.
0361	BECAUSE THE AXIS IS MOVING, THE SUPERPOS CONTROL IS CAN'T BE USED.	While the axis being subject to superposition control was moving, an attempt was made to start or cancel the superposition control by a superposition control axis selection signal.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
0362	SUPERPOSITION CONTROL AXIS COMPOSITION ERROR.	This error occurred when: (1) An attempt was made to perform superposition control for the axis during a synchronization, composition, or superposition. (2) An attempt was made to synchronize a further great-grandchild for a parent-child-grandchild relation.
0363	THE G28 WAS INSTRUCTED IN TO THE SUPERPOS CONTROL SLAVE AXIS.	This error occurred when G28 was specified to the superposition control slave axis during superposition control.
0364	THE G53 WAS INSTRUCTED IN TO THE SUPERPOS CONTROL SLAVE AXIS.	This error occurred when G53 was specified to the slave axis being moved during superposition control.
0365	TOO MANY MAXIMUM SV/SP AXIS NUMBER PER PATH	The maximum control axis number or maximum control spindle number which could be used within a path was exceeded.
0366	IMPROPER G-CODE IN TURRET METHOD	When the turret change tools method was selected (bit 3 (TCT) of parameter No. 5040 = 0), G43, G43.1, G43.4, G43.5, or G43.7 was commanded.
0367	3-D CONV. WAS COMMANDED IN SYNC MODE AS THE PARAMETER PKUx(NO.8162#2) IS 0.	A 3-dimensional coordinate conversion was commanded during synchronization control when the bit 2 (PKUx) of parameter No. 8162 was 0.
0368	OFFSET REMAIN AT OFFSET COMMAND	- When the ATC change tools method was selected (bit 3 (TCT) of parameter No. 5040 = 1) during G43, G43.1, G43.4, or G43.5 mode, G43.7 was commanded. Or, G43, G43.1, G43.4, or G43.5 was commanded during G43.7 mode. - After the setting of bit 3 (TCT) of parameter No. 5040 was changed in a state in which tool offset remained, the tool offset command was specified.
0369	G31 FORMAT ERROR	- No axis is specified or two or more axes are specified in the torque limit switch instruction (G31P98/P99). - The specified torque Q value in the torque limit switch instruction is out of range. The torque Q range is 1 to 254. - The high-speed continuous skip option is not present.
0370	G31P/G04Q ERROR	(1) The specified address P value for G31 is out of range. The address P range is 1 to 4 in a multistage skip function. (2) The specified address Q value for G04 is out of range. The address Q range is 1 to 4 in a multistage skip function. (3) P1-4 for G31, or Q1-4 for G04 was commanded without a multistage skip function option. (4) <T series > The specified value of address P of G72 or G74 falls outside the range. Address P ranges from 1 to 4 in the multistage skip function. P1-4 was specified in G72 or G74 even though the multistage skip function option is not present.
0371	ILLEGAL FORMAT IN G10 OR L50	In a command format for a programmable parameter input, an attempt was made to change the parameter for an encryption (No. 3220), key (No. 3221), or protection range (No.3222 or No.3223) as a "the encryption function for the key and program." Modify the program.
0373	ILLEGAL HIGH-SPEED SKIP SIGNAL SELECTED	In the skip commands (G31, G31P1 to G31P4) and dwell commands (G04, G04Q1 to G04Q4), the same high-speed signal is selected in different paths.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	Description
0374	ILLEGAL REGISTRATION OF TOOL MANAGER(G10)	G10L75 or G10L76 data was registered during the following data registration: - From the PMC window. - From the FOCAS2. - By G10L75 or G10L76 in another system. Command G10L75 or G10L76 again after the above operation is completed.
0376	SERIAL DCL: ILLEGAL PARAMETER	(1) When bit 1 (OPTx) of parameter No. 1815 is set to “1”, bit 3 of parameter No.2002 is set to “0” (2) The absolute-position detection function is enabled. (Bit 5 (APCx) of parameter No. 1815 is set to “1”.)
0377	ILLEGAL MODE FOR GOTO/WHILE/DO	The GOTO statement, WHILE statement, and DO statement cannot be executed in a program created in the MDI mode. When a program including those statements is to be executed, register the program in the program memory then call the program for execution.
0387	ILLEGAL RTM DI/DO VAR	There is no DI/DO variable that has a specified signal address (alphabet, number).
0389	ILLEGAL RTM SIGNAL BIT	Bits other than bits 0 to 7 cannot be specified with a DI/DO signal.
0390	ILLEGAL MACRO VAR	A macro variable which was not supported by the real time custom macro function was used.
0391	RTM BRANCH OVER	The number of branches supported with real time custom macros was exceeded.
0392	TOO MANY SENTENCE CONTROL	Many reserved words (ZONCE, ZEDGE, ZWHILE, ZDO, ZEND, G65, M99) for RTM control were used in a real time macro command.
0393	NO SENTENCE CONTROL	In a real time macro command, there is no data to be assigned.
0394	ILLEGAL SENTENCE CONTROL	The matching of reserved words (ZONCE, ZEDGE, ZWHILE, ZDO, ZEND, ZCANCEL, G65, M99) for RTM control is incorrect.
0395	ILLEGAL NC WORD CONTROL	Control code G65 or M99 for calling a subprogram or returning from a subprogram is not coded correctly.
0396	ILLEGAL RTM SENTENCE CONTROL	In other than a real time macro command, a reserved word (ZONCE, ZEDGE, ZWHILE, ZDO, ZEND, ZCANCEL) for RTM control is used.
0397	RTM BUFFER OVER	There is no buffer available for real time macro commands. Too many blocks read in advance are buffered as triggers used by real time macro commands.
0398	'ID OVER IN BUFFER	In blocks read in advance, there are too many real time macro commands with the same ID.
0399	'ID EXECUTION IN SAME TIME	An attempt was made to execute real time macro commands with the same ID by using the same NC statement as a trigger.
0400	ONESHOT CMDOVER	Too many one-shot real time macro commands are specified.
0401	EXEC CMD NUM OVER IN SAME TIME	The number of real time macro commands that can be executed simultaneously was exceeded
0402	ILLEGAL TOKEN FOR RTM	A token, variable, or function that is not supported by the real time custom macro function was detected.
0403	ACCESS TO RTM PROTECT VAR	An attempt was made to access a protected variable.
0404	RTM ERROR	An error related to a real time macro command occurred.
0406	CODE AREA SHORTAGE	The storage size of the real time macro area is insufficient.
0407	DOULE SLASH IN RTM MODE	In the compile mode, an attempt was made to set the compile mode again.
0408	G90 IS NOT PERMITTED	The absolute command cannot be specified.

FRYER MACHINE SYSTEMS INC
Roh Zahansky

Number	Message	Description
0409	ILLEGAL AXIS NO	An invalid axis number is specified.
0410	MIDDLE POINT IS NOT ZERO	An intermediate point other than 0 is specified with G28.
0411	SIMULTANEOUSLY AXES OVER	The maximum number of axes that can be controlled simultaneously was exceeded.
0412	ILLEGAL G CODE	An unusable G code was used.
0413	ILLEGAL ADDRESS	An unusable address was used.
0415	GROUP IS IN USE	The group to which the specified axis belongs is already in used.
0416	UNABLE TO USE THE AXIS	The specified axis cannot be used.
0417	AXIS IS UNABLE TO MOVE	Axis control of the real-time custom macro was executed in the following state. Resolve the alarm condition or abnormal load condition. When the alarm signal EIALg is "1", input the reset signal ECLRg and set it to "0". (1) Alarm state of target axis of PMC axis control (alarm signal EIALg is "1"). (2) Emergency stop. (3) Alarm state of servo, spindle, overtravel, turning off the power (SV, SP, OT, PW alarm). (4) Target axis is unexpected disturbance torque detection by unexpected disturbance torque detection function.
0418	ILLEGAL FEED SETTING	An incorrect feedrate is set.
0419	ILLEGAL DISTANCE SETTING	A travel distance beyond the specifiable range is specified.
0420	CONSTANT NUMBER P	A subprogram is specified not by using a constant.
0421	ILLEGAL ARGUMENT G54	With G65, an invalid argument, L, is used.
0422	ILLEGAL ARGUMENT G54	With G65, an invalid argument is used.
0423	NO PMC AXIS CONTROL OPTION	The option for PMC axis control is missing.
0424	MULTIPLE AXES IN ONE GROUP	Multiple axes are using one group.
0426	M FUNCTION IS UNABLE TO USE	(1) Auxiliary function was commanded with one-shot RTM command. (2) Available group number is not set for ID of modal RTM command.
0427	ILLEGAL M CODE	(1) M code of M00, M01, M02, M30, M98, M198, M65536 or higher was commanded. (2) Multiple M commands in a single block was commanded.
0428	AUX FUNCTION IS IN USE	Real-time custom macro or PMC axis control is using auxiliary function.
0429	ILLEGAL COMMAND IN G10.6	When retract was started in a threading block, a retract command had been issued for the long axis direction of threading.
0430	TOOL LIFE PAIRS ZERO	Tool life management group number parameter No.6813 is 0.
0431	ILLEGAL T/R DATA OF TOOL LIFE	The arbitrary group number (T) or remaining amount setting (R) is invalid.
0432	UNAVAILABLE POSTURE IN TPC	A command specifying an impossible tool posture was issued in tool posture control. Review the machine configuration and command. In circular interpolation or helical interpolation under tool posture control, a command to reverse the tool posture with respect to the interpolation plane was specified. Review the machine configuration and command.
0436	ILLEGAL PARAMETER IN WSC	An incorrect parameter was specified in compensation of workpiece placement error. - he basic three axes are not specified in parameter No.1022.

FRYER MACHINE SYSTEMS INC
Rob Zahorsky

Number	Message	Description
0437	ILLEGAL COMMAND IN WSC	An invalid command related to compensation of workpiece placement error was specified. - An illegal G code was specified in the workpiece placement error compensation mode. - There is an error in the modal setting used when the compensation of workpiece placement error is started. - G54.4 was not specified solely. - There is not the P command in the block including the G54.4 command. Alternatively, the value following P is out of the range. - Compensation of workpiece placement error was specified redundantly.
0438	ILLEGAL PARAMETER IN TOOL DIRC CMP	If, on a 5-axis machine, either of the two cases below applies, a parameter is illegal. (1) Workpiece setting error compensation is performed. (2) Tool center point retention type tool axis direction control (G53.6) is performed. - Look-ahead acc./dec. before interpolation is disabled. Set parameter No. 1660. - Rapid traverse Acc./Dec. before interpolation is disabled. Set bit 1 (LRP) of parameter No. 1401, bit 5 (FRP) of parameter No. 19501, and parameter No. 1671. - The parameters (No.19680 to No.19714) for configuring the machine are incorrect. - The axis set by parameters Nos. 19681 and 19686 is not a rotation axis. - The basic three axes are not set in parameter No. 1022. - In tool length compensation during workpiece setting error compensation, bit 6 (TOS) of parameter No. 5006 is 0 and bit 2 (TOP) of parameter No. 11400 is 0. Set either parameter to 1. - In case that tilted working plane indexing and tool center point control are used at the same time, bit 0 (RCM) of parameter No. 11200 is set to 0. Set to 1.
0439	ILLEGAL COMMAND IN TOOL DIRC CMP	When compensation of workpiece placement error was performed in a 5-axis cutting machine (compensation in the tool direction (bit 0 (RCM) of parameter No. 11200 is 1)), an illegal command was issued. - An unspecifiable G code was specified. - There is an error in the modal setting used during startup. - An axis not related to 5-axis machining was specified. - The absolute coordinates of a rotation axis could not be obtained in the startup block of compensation of workpiece placement error, tool center point control, or cutting point command.
0441	DUPLICATE PATH TABLE	The same Path Table numbers exist. Example) <AXIS_TABLE_1234_X1> and <TIME_TABLE_1234_X1> exist. <AXIS_TABLE_0001_M> and <TIME_TABLE_0001_M> exist.

FRYER MACHINE SYSTEMS INC
Rob Zanowsky

Number	Message	Description
0442	PATH TABLE COMMAND EXCES ERR	(1) At the start of the Path Table Operation, the difference between the actual axis position and the start command at the Path Table exceeds the parameter No.11101. (2) At the start of the Path Table Operation, the difference between the actual spindle speed and the start command of spindle speed at the Path Table exceeds the parameter No.11102.
0443	PTRDY SIGNAL IS OFF	Even though the Path Table Ready signal PTRDY <Fn519.6> is "0", the Path Table Operation is started. Retry Path Table Operation after Path Table conversion.
0444	ILLEGAL PATH TABLE M-CODE	M/P/Q code for starting the Path Table operation is not correct.
0445	ILLEGAL AXIS OPERATION	The positioning command was issued in the speed control mode. Check the SV speed control mode signal.
0446	ILLEGAL COMMAND IN G96.1/G96.2/G96.3/G96.4	G96.1, G96.2, G96.3, and G96.4 are specified in the block that includes other commands. Modify the program.
0447	ILLEGAL SETTING DATA	The live tool axis is incorrectly set. Check the parameter for the spindle control function with servo motor.
0451	ILLEGAL AUXILIARY FUNCTION TABLE COMMAND	When M code is output, the ladder of the PMC does not execute the completion processing of the previous M code.
0452	ILLEGAL PATH TABLE OPERATION	In Path Table Operation, the following problems occurred. - Skip command is not correct. - The connection of the Path Table is not correct. - Path Table Operation is not correct for other reason. etc. The detail alarm number is read by using the cnc_rdptexedistalm function. The detail alarm number can be read by using the C Language Executor or FOCAS2 cnc_ rdptexedistalm function. For details of cnc_rdptexedistalm function, refer to “CNC/PMC window library” in “C Language Executor Programming Manual (B-63944EN-3).
0455	ILLEGAL COMMAND IN GRINDING	In grinding canned cycles: (1) <M series> The signs of the I, J, and K commands do not match. (2) <M series/T series > The amount of travel of the grinding axis is not specified.
0456	ILLEGAL PARAMETER IN GRINDING	Parameters related to grinding canned cycles are incorrectly set. Probable causes are given below. (1) <M series/T series> The axis number of the grinding axis is incorrectly set (parameters Nos. 5176 to 5179). (2) <M series> The axis number of the dressing axis is incorrectly set (parameters Nos. 5180 to 5183). (3) <M series/T series> The axis numbers of the cut axis, grinding axis, and dressing axis (only for the M series) overlap.
0459	ALL PARALLEL AXES IN PARKING	All the axes specified during automatic operation are parking.
0460	ILLEGAL TORCH AXIS NUMBER	The axis number set in parameter No. 5490 (torch control axis) exceeds the number of control axes.
0461	ILLEGAL SETTING OF ROTATE AXIS FOR TORCH	The parameter setting (bit 0 (ROTx) of parameter No. 1006 = 1) of the rotation axis is not applied to the torch turning axis.
0492	3DCHK FIG. ILLEGAL: [Target name]	The figure data of [Target name] specified for built-in 3D interference check is invalid.
0493	3DCHK AXIS ILLEGAL: [Target name]	The move axis data of [Target name] specified for built-in 3D interference check is invalid.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
0494	3DCHK FUNCTION INVALID	Built-in 3D interference check function is disabled by bit 0 (ICE) of parameter No. 10930.
0495	3DCHK TOO MANY FIGURE	The total number of shapes included in all interference check targets except the tool exceeds 23.
0496	ILLEGAL P,Q COMMAND IN G22.2	As for G22.2 command, parameter P or Q is out of range or not specified. Correct G22.2 command.
0497	CANNOT MAKE TOOL FIGURE BY TOOL MANAGEMENT	According to bit 2 (ICT) of parameter No.10930, though the tool figure should be automatically made by using the tool management function, the option for the tool management function is not available.
0501	THE COMMANDED M-CODE CAN NOT BE EXECUTED	The M code specified in parameter No. 11631 to 11646 was specified in other than an execution macro, macro interrupt, macro call using a G or M code, or subprogram call using a T, S, or second auxiliary function code.
0502	ILLEGAL G-CODE	(1) A G code unavailable in the inter-path flexible synchronization mode was specified. (2) A G code unavailable in the advanced superimposition state was specified.
0503	ILLEGAL MODAL IN SUPERIMPOSED MODE	Modal state of a G code that cannot be superimposed.
0507	ILLEGAL PARAMETER(NO.7526)	The address of the R signal for the high-speed cycle machining operation information output function is invalid. (1) The specified R signal address is invalid. (2) The start address is not a multiple of 4 (0, 4, 8, ...). (3) A 36-byte area is not allocated.
0508	G code to need G90(PAC)	In parallel axis control, a G code requiring an absolute command (G90) in the block immediately before was specified.
0509	TOOL OFFSET COMMAND IS NOT AVAILABLE	- Tool offset (for the lathe system) was specified in the thread cutting block. - Tool offset was specified (lathe system) in the scaling mode, coordinate system rotation mode, or programmable mirror image mode.
0511	CS HI-SPEED SWITCHING FORMAT ERROR	The format of Cs contour control high speed switching is invalid.
0512	IMPOSSIBLE COMMAND FOR CS HI-SPEED SWITCHING	The following commands cannot be specified in Cs contour control high speed switching: - Move command not for high-speed cycle machining - Synchronous/composite control, superimposed control - Spindle command synchronous control - Simple spindle electronic gear box - Manual reference position return
0513	CS HI-SPEED SWITCHING SETTING ERROR	The setting for Cs contour control high speed switching is invalid. Possible causes are: - An M code value for Cs contour control high speed switching is used for multiple Cs contour control axes. - FIN is returned for the M code for high-speed switching of Cs contour control when the Cs contour control high speed switching completion signal CSMCx does not become “1”. - The spindle software does not support the spindle control switching function for high-speed cycle machining.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
0514	ILLEGAL COMMAND IN FLEXIBLE PATH AXIS ASSIGNMENT	(1) An assignment command in flexible path axis assignment was issued for an axis yet to be removed. (2) The P, Q, R, I, J, K, or L value specified by G52.1, G52.2, or G52.3 is invalid. (3) The value of the parameter No. 11560 is duplicated. (4) An attempt was made to execute a removal command (G52.1) for an axis already removed. (5) An attempt was made to exchange axes having different settings of bit 1 (FAN) of parameter No. 11562. (6) An attempt was made to perform flexible path axis assignment without canceling the offset.

0515	ILLEGAL FORMAT IN SMOOTH TCP(G43.4L1)	An illegal command was specified in Rotation axes compensation (G43.4L1, G43.5L1, G43.8L1, G43.9L1).
		- An illegal command was specified in a Rotation axes control start block.
		  - An invalid value was specified with address "L". A value other than 0 and 1 was specified with address “L”.
		  - G10.8 was specified at the same time.

0516	ILLEGAL PARAMETER IN SMOOTH TCP(G43.4L1)	A parameter related to in Rotation axes compensation (G43.4L1, G43.5L1, G43.8L1, G43.9L1) is illegal.
0517	SETTING ERROR AMOUNT IS OUT OF RANGE	An attempt was made to start workpiece setting error compensation when a rotation direction setting error was outside the range set in the corresponding parameter No. 11753 to 11758.
0518	CODE OF CONTROL-IN DOES NOT EXIST	When program is read, the program dose not have the control-in “)” which is the comment terminator.

0520	ILLEGAL FORMAT IN G10.8L1	An illegal command was specified to change a tolerance of in Rotation axes compensation (G43.4L1, G43.5L1, G43.8L1, G43.9L1).
		- A negative value was specified as a tolerance.
		  - Specify positive values as addresses "α" and "β".
		- An invalid P value was specified.
		  - Specify either 0 or 1 as address "P".
		- Address P is specified together with addresses "α" and "β".
		  - Specify only either address "P" or "addresses "α" and "β".
		- An invalid address was specified.
		  - In G10.8L1, only L, P, α, β, O, N, and M can be specified.
		- Another G code was specified at the same time.
		  - Specify G10.8L1 alone.
		- G10.8 was specified in Rotation axes compensation mode, but address "L" is not specified or the value of address "L" is not 1.
		  - In Rotation axes compensation mode, only G10.8L1 can be specified.

FRYER MACHINE SYSTEMS INC
Rob Zaremski

Number	Message	Description

0521	ILLEGAL USAGE OF G10.8L1	Modal information used when specifying G10.8L1 contains an error.
		- The system is not in Rotation axes compensation mode (G43.4L1, G43.5L1, G43.8L1, G43.9L1).
		  - G10.8L1 can be specified in Rotation axes compensation mode only.
		- The system is in Rotation axes compensation mode (G43.4L1, G43.5L1, G43.8L1, G43.9L1), but the command is not linear interpolation (G01).
		  - G10.8L1 can be specified only during linear interpolation (G01).

0522	ILLEGAL CMD. IN RATE FEED (G93.2)	During AI contour control mode or the interpolation except linear interpolation, circular interpolation is commanded in the rate feed.
0523	ILL. COMMAND IN HIGH-SPEED S-TCP	An illegal command was specified during Smooth control (G43.4P3, G43.5P3, G43.8P3, G43.9P3). - Smooth control was specified without option of High-speed Smooth TCP. - The manual intervention is performed. - In the block where "the suppress motion of quick program restart" is done, an all related axis (5-axis) of smooth control is not commanded.
0524	PARAMETER CHANGE ERROR	In the hypothetical liner axis control, parameter No. 13281 or No. 13284 had been changed. However the hypothetical coordinate setting was not executed because of following cause. - Automatic operation is feed hold state.
0525	SUB PROGRAM/MACRO CALLING	The mode was changed and the cycle start was executed while executing subprogram call or macro call. Reset and cycle start is executed again.
0526	ILLEGAL PARAMETER IN HLAC	The setting of the parameter of hypothetical liner axis control function is illegal. Check the following parameters and set to be 2 hypothetical axes, a real rotary axis and a real liner axis in the hypothetical plane. - No.1006#0 - No.1023 - No.11802#4 - No.13280
0527	ILLEGAL DATA IN PITCH ERROR	In the effective pitch error compensation points that is decided by considering the setting of the parameters, the difference between two consecutive data exceeds the range of -128 to +127. Correct the pitch error compensation data or change the parameter.

0533	ILL. PARAM IN HIGH-SPEED S-TCP	A parameter related to Smooth control (G43.4P3, G43.5P3, G43.8P3, G43.9P3) is illegal.
		- When setting the following, Smooth control was commanded.
		  - the rotation axis is a hypothetical axis Parameter IA1,IA2(No.19696#0,1)
		  - the reference angle Parameter RA,RB (No.19698,19699)

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
0535	TL OFS CONVERSION SETTING ERROR	In the following cases, this alarm will occur when G44.1 is commanded. (1) The rotation axis is not set to the swivel head axis. (No. 19640#0) (2) The same rotation axis is set to both the tool nose rotation axis and the swivel head axis. (3) The axis direction of the tool nose rotation axis is not X-axis or Z-axis. (4) The axis direction of the swivel head axis is not Y-axis. (5) The number of axis for converted offset is short. (No. 1022)
0536	ILLEGAL COMMAND IN G44.1	An illegal command is specified in the tool offset conversion function. (1) An offset number is not commanded in G44.1 block. (2) Even if the imaginary tool nose direction is commanded in G44.1 block, the commanded value is not in range of 1 to 8. (3) The offset number is specified by D code that is different from the offset number specified by H code in G44.1 mode. (Machining center system only.) (4) Not allowed G code is specified at G44.1 block. (5) The rotation of the tool nose rotation axis or the swivel head axis is specified in G44.1 block. (6) The rotation of the tool nose rotation axis or the swivel head axis is specified in the block in which basic three axes movement are commanded. (7) In G44.1 mode, even if the rotation axis that is used in calculation of the tool offset conversion function is moved, G44.1 did not be commanded again before offset axis is moved. (8) G44.1 is commanded in the following mode. - Tool center point control. - Workpiece setting error compensation.
0538	OFFSET IS NOT CANCELED	In the multiple repetitive cycle (G70), when the tool nose radius compensation can be commanded in the target figure program (parameter NT2 (No.5106#3)=1), G40 is not commanded at the end block in the target figure program.
0539	MAX SP SPEED CLAMP COMMAND ERROR	Clamp of maximum spindle speed is illegal. Extended spindle name cannot be commanded for clamp of maximum spindle speed.
0545	PRESSURE AND POSITION CONTROL COMMAND IS ILLEGAL	The G-codes related to pressure and position control (G14.1, G14.2, G14.4) is not commanded correctly. Possible causes are: - The format of the command is illegal. - Pressure and position control mode ON/OFF command cannot be specified in the current modal G code states. - An unspecifiable axis is specified.
0546	PRESSURE AND POSITION CONTROL MODE OFF	The pressure and position control command (G14.2) was commanded in pressure and position control mode off.
0547	IMPROPER G-CODE (PRESSURE AND POSITION CONTROL)	G-codes not available in pressure and position control mode are commanded.

FRYER MACHINE SYSTEMS INC
Rob Zaharasky

Number	Message	Description
0552	SP-MODE CHANGE COMMAND ERROR	Spindle control mode changing command is illegal. Causes of the alarm are as follows. - Spindle control mode changing is commanded for disabling spindle. - Multiple M commands are commanded in spindle orientation ON/OFF command or Cs contour control ON/OFF command. - Illegal G code (G53, G54) is commanded at spindle control mode changing block.
0553	SP-MODE CHANGE IMPOSSIBLE	Spindle control mode changing command cannot execute. Causes of the alarm are as follows. - Spindle control mode changing by program command is commanded during spindle orientation or Cs contour control by DI signal. - During spindle control mode changing, spindle control mode changing is commanded from the other path. - Spindle control mode changing is commanded when spindle control mode off signal MDOFF1 to MDOFF4<Gn586.4 to Gn586.7> is "1".
0554	S-CODE IS NOT COMMANDED	M03/M04 is commanded alone without S code.
0555	P2ACC MODE ON ERROR	(1) Specified format of Cross rail axis control is incorrect. (2) Parameter setting of Cross rail axis control is incorrect. (3) Modal G-codes are not allowed to command Cross rail axis control. (4) Reference position of the master axis or the slave axis is not established.
0556	P2ACC ILLEGAL COMMAND	(1) G-codes that are not allowed in Cross rail axis control mode are commanded. (2) Command to the master axis or the slave axis is executed in Cross rail axis control mode on. (3) Command to the hypothetical axis is executed in Cross rail axis control mode off.
0564	ILLEGAL COMMAND OF SMART RTAP	The command for Smart rigid tapping is illegal. The reason of this alarm is in the followings. - The moving distance of feed axis is larger than 4096 times of screw pitch. - The used spindle software doesn't support Smart rigid tapping.
0601	ILLEGAL AXIS COMMAND FOR SERVO MOTOR SPINDLE	The move command was executed to the servo axes for spindle use of the spindle control with servo motor. Modify the program.
1001	AXIS CONTROL MODE ILLEGAL	Axis control mode is illegal.
1013	ILLEGAL POS. OF PROGRAM NO.	Address O or N is specified in an illegal location (e.g. after a macro statement).
1014	ILLEGAL FORMAT OF PROGRAM NO.	Address O or N is not followed by a number.
1016	EOB NOT FOUND	EOB (End of Block) code is missing at the end of a program input in the MDI mode.
1059	COMMAND IN BUFFERING MODE	The manual intervention compensation request signal MIGET became "1" when a advanced block was found during automatic operation. To input the manual intervention compensation during automatic operation, a sequence for manipulating the manual intervention compensation request signal MIGET is required in an M code instruction without buffering.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
1077	PROGRAM IN USE	An attempt was made in the foreground to execute a program being edited in the background. The currently edited program cannot be executed, so end editing and restart program execution.
1079	PROGRAM FILE NOT FOUND	The program of the specified file No. is not registered in an external device. (external device subprogram call)
1080	DUPLICATE DEVICE SUB PROGRAM CALL	Another external device subprogram call was made from a subprogram after the subprogram called by the external device subprogram call.
1081	EXT DEVICE SUB PROGRAM CALL MODE ERROR	The external device subprogram call is not possible in this mode.
1090	PROGRAM FORMAT ERROR	A lowercase alphabetic character is found in other than an NC program statement comment section, program name, or folder name.
1091	DUPLICATE SUB-CALL WORD	More than one subprogram call instruction was specified in the same block.
1092	DUPLICATE MACRO-CALL WORD	More than one macro call instruction was specified in the same block.
1093	DUPLICATE NC-WORD & M99	An address other than O, N, P or L was specified in the same block as M99 during the macro modal call state.
1095	TOO MANY TYPE-2 ARGUMENT	More than ten sets of I, J and K arguments were specified in the type–II arguments (A, B, C, I, J, K, I, J, K, ...) for custom macros.
1096	ILLEGAL VARIABLE NAME	An illegal variable name was specified. A code that cannot be specified as a variable name was specified. [#_OFSxx] does not match the tool offset memory option configuration.
1097	TOO LONG VARIABLE NAME	The specified variable name is too long.
1098	NO VARIABLE NAME	The specified variable name cannot be used as it is not registered.
1099	ILLLEGAL SUFFIX [ ]	A suffix was not specified to a variable name that required a suffix enclosed by [ ]. A suffix was specified to a variable name that did not require a suffix enclosed by [ ]. The value enclosed by the specified [ ] was out of range.
1100	CANCEL WITHOUT MODAL CALL	Call mode cancel (G67) was specified even though macro continuous–state call mode (G66) was not in effect.
1101	ILLEGAL CNC STATEMENT IRT.	An interrupt was made in a state where a custom macro interrupt containing a move instruction could not be executed.
1115	READ PROTECTED VARIABLE	An attempt was made in a custom macro to use on the right side of an expression a variable that can only be used on the left side of an expression.
1120	ILLEGAL ARGUMENT FORMAT	The specified argument in the argument function (ATAN, POW) is in error.
1124	MISSING DO STATEMENT	The DO instruction corresponding to the END instruction was missing in a custom macro.
1128	SEQUENCE NUMBER OUT OF RANGE	The jump destination sequence No. in a custom macro statement GOTO instruction was out of range (valid range: 1 to 99999999).
1129	MISSING IF STATEMENT	There is no “IF” corresponding to "ELSE" and "ENDIF".
1130	TOO MANY IF STATEMENT NESTING	The maximum value of the nesting of IF statement was exceeded. Up to 10 levels are allowed.
1131	MISSING OPEN BRACKET	The number of left brackets ([) is less than the number of right brackets (]) in a custom macro statement.
1132	MISSING CLOSE BRACKET	The number of right brackets (]) is less than the number of left brackets ([) in a custom macro statement.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	Description
1133	MISSING '='	An equal sign (=) is missing in the arithmetic calculation instruction in a custom macro statement.
1135	SPECIFIED LABEL IS NOT FOUND	LABEL specified with GOTO[label] of the custom macro is not found in the program.
1137	IF STATEMENT FORMAT ERROR	The format used in the IF statement in a custom macro is in error.
1138	WHILE STATEMENT FORMAT ERROR	The format used in the WHILE statement in a custom macro is in error.
1139	SETVN STATEMENT FORMAT ERROR	The format used in the SETVN statement in a custom macro is in error.
1141	ILLEGAL CHARACTER IN VAR. NAME	The SETVN statement in a custom macro contacts a character that cannot be used in a variable name.
1142	TOO LONG V-NAME (SETVN)	The variable name used in a SETVN statement in a custom macro exceeds 8 characters.
1143	BPRNT/DPRNT STATEMENT FORMAT ERROR	The format used in the BPRINT statement or DPRINT statement is in error.
1144	G10 FORMAT ERROR	The G10 L No. contains no relevant data input or corresponding option. Data setting address P or R is not specified. An address not relating to the data setting is specified. Which address to specify varies according to the L No. The sign, decimal point or range of the specified address are in error.
1145	G10.1 TIME OUT	The response to a G10.1 instruction was not received from the PMC within the specified time limit.
1146	G10.1 FORMAT ERROR	The G10.1 instruction format is in error.
1148	SWITCH STATEMENT ERROR	SWITCH statement of custom macro contains an error. The cause may be one of the following: 1. Format error - Not “SWITCH[<expression>]” - The next of “SWITCH[<expression>]” is not “EOB”, “CASE” or “DEFAULT”. 2. The value of <expression> is out-of-range (-2147483648 to 2147483647). <Empty> is regarded as out-of-range value.
1149	CASE STATEMENT ERROR	CASE statement of custom macro contains an error. The cause may be one of the following: 1. CASE statement is commanded with SWITCH statement not executed. 2. Format error - Not “CASE[<expression>]” - The next of “CASE[<expression>]” is not “GOTO”. However, when the value saved by SWITCH statement is not corresponding to <expression>, the alarm does not occur. 3. The value of <expression> is out-of-range (-2147483648 to 2147483647). <Empty> is regarded as out-of-range value.
1150	DEFAULT STATEMENT ERROR	DEFAULT statement of custom macro contains an error. The cause may be one of the following: 1. DEFAULT statement is commanded with SWITCH statement not executed. 2. Format eeror - Not “DEFAULT[ ]” - The next of “DEFAULT[ ]” in not “EOB” or “GOTO”.
1151	LABEL FORMAT ERROR	The error of the format is contained in LBL[label].

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

Number	Message	Description
1152	G31.9/G31.8 FORMAT ERROR	The format of the G31.9(continuous high-speed skip function) or G31.8(EGB skip function / skip function for flexible synchronization control) block is erroneous in the following cases: - The axis was not specified in the G31.9 or G31.8 block. - Multiple axes were specified in the G31.9 or G31.8 block. - The P code was specified in the G31.9 or G31.8 block. - In flexible synchronization control mode, G31.8 was commanded on the axis other than the slave axis. - The Q in G31.8 block was specified out of range in flexible synchronization control mode.
1153	CANNOT USE G31.9	G31.9 cannot be specified in this modal state. This alarm is also generated when G31.9 is specified when a group 07 G code (e.g. tool radius compensation) is not canceled.
1160	COMMAND DATA OVERFLOW	An overflow occurred in the position data within the CNC. This alarm is also generated if the target position of a command exceeds the maximum stroke as a result of calculation such as coordinate conversion, offset, or introduction of a manual intervention amount.
1196	ILLEGAL DRILLING AXIS SELECTED	An illegal axis was specified for drilling in a canned cycle for drilling. If the zero point of the drilling axis is not specified or parallel axes are specified in a block containing a G code in a canned cycle, simultaneously specify the parallel axes for the drilling axis.
1200	PULSCODER INVALID ZERO RETURN	The grid position could not be calculated during grid reference position return using the grid system as the one–revolution signal was not received before leaving the deceleration dog. This alarm is also generated when the tool does not reach a feedrate that exceeds the servo error amount preset to parameter No. 1841 before the deceleration limit switch is left (deceleration signal *DEC returns to “1”).
1202	NO F COMMAND AT G93	F codes in the inverse time specification mode (G93) are not handled as modal, and must be specified in individual blocks.
1223	ILLEGAL SPINDLE SELECT	1) An attempt was made to execute an instruction that uses the spindle although the spindle to be controlled has not been set correctly. 2) Interpolation type rigid tapping was specified in a path in which the Cs contour control function is not enabled.
1282	ILLEGAL COMMAND IN 3-D OFFSET	An illegal G code was specified in the 3-dimensional tool compensation mode.
1283	ILLEGAL IJK IN 3-D OFFSET	I, J, and K commands are specified without the decimal point in three–dimensional tool compensation mode.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	Description
1298	ILLEGAL INCH/METRIC CONVERSION	(1) When the bit 1 (CIM) of parameter No. 11222 set to 0, and the workpiece coordinate system has been shifted from the machine coordinate system by the following, the inch/metric conversion has been performed. - Manual intervention performed with the manual absolute signal being off - Machine locked - Handle interrupt - Mirror image - Workpiece coordinate system shift caused by local coordinate system setting (G52) or workpiece coordinate system setting (G92) (2) Inch/metric conversion was executed in a modal state where it can not command. Check the modal state. (3) Inch/metric conversion can not be commanded with programmable parameter input or # 3005 (setting data). It commands G20/G21 or G70/G71. (4) During automatic operation, inch / metric conversion by MDI input or PMC window was done. Stop automatic operation and command.
1300	ILLEGAL ADDRESS	The axis No. address was specified even though the parameter is not an axis–type while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. Axis No. cannot be specified in pitch error compensation data.
1301	MISSING ADDRESS	The axis No. was not specified even though the parameter is an axis–type while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. Or, data No. address N, or setting data address P or R are not specified.
1302	ILLEGAL DATA NUMBER	A non–existent data No. was found while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. An invalid address R value is specified in a pattern program for each machining purpose on the high–speed high–precision setting screen. This alarm is also generated when illegal word values are found.
1303	ILLEGAL AXIS NUMBER	An axis No. address exceeding the maximum number of controlled axes was found while loading parameters from a tape or by entry of the G10 parameter.
1304	TOO MANY DIGIT	Data with too many digits was found while loading parameters or pitch error compensation data from a tape.
1305	DATA OUT OF RANGE	Out–of–range data was found while loading parameters or pitch error compensation data from a tape. The values of the data setting addresses corresponding to L Nos. during data input by G10 was out of range. This alarm is also generated when NC programming words contain out–of–range values.
1306	MISSING AXIS NUMBER	A parameter which requires an axis to be specified was found without an axis No. (address A) while loading parameters from a tape.
1307	ILLEGAL USE OF MINUS SIGN	Data with an illegal sign was found while loading parameters or pitch error compensation data from a tape, or by entry of the G10 parameter. A sign was specified to an address that does not support the use of signs.

FRYER MACHINE SYSTEMS INC
Rob Zaherosky

Number	Message	Description
1308	MISSING DATA	An address not followed by a numeric value was found while loading parameters or pitch error compensation data from a tape.
1332	DATA WRITE LOCK ERROR	Could not load data while loading parameters, pitch error compensation data and work coordinate data from tape.
1333	DATA WRITE ERROR	Could not write data while loading data from tape.
1370	PARAMETER SETTING ERROR (DM3H-1)	Out–of–range data was set during setting of the three–dimensional handle feed parameter.
1371	PARAMETER SETTING ERROR (DM3H-2)	An illegal axis of rotation was set during setting of the three–dimensional handle feed parameter.
1372	PARAMETER SETTING ERROR (DM3H-3)	An illegal master axis was set during setting of the three–dimensional handle feed parameter.
1470	G40.1 –G42.1 PARAMETER MISS	A parameter setting related to normal direction control is illegal. The axis number of a normal direction controlled axis is set in parameter No. 5480, but that axis number is in the range of the number of controlled axes. The axis set as a normal direction controlled axis is not set as a rotation axis (bit 0 (ROTx) of parameter No. 1006) = 1 and No.1022=0).
1471	ILLEGAL COMMAND IN G40.1 –G42.1	An invalid G code was specified in the normal direction control mode.
1508	DUPLICATE M-CODE (INDEX TABLE REVERSING)	A function to which the same code as this M code is set exists. (index table indexing)
1509	DUPLICATE M-CODE (SPOS AXIS ORIENTATION)	A function to which the same code as this M code is set exists. (spindle positioning, orientation)
1510	DUPLICATE M-CODE (SPOS AXIS POSITIONING)	A function to which the same code as this M code is set exists. (spindle positioning, positioning)
1511	DUPLICATE M-CODE (SPOS AXIS RELEASE)	A function to which the same code as this M code is set exists. (spindle positioning, mode cancel)
1537	ADDRESS F UNDERFLOW (OVERRIDE)	The speed obtained by applying override to the F instruction is too slow.
1538	ADDRESS F OVERFLOW (OVERRIDE)	The speed obtained by applying override to the F instruction is too fast.
1543	ILLEGAL GEAR SETTING	The gear ratio between the spindle and position coder, or the set position coder number of pulses is illegal in the spindle positioning function.
1561	ILLEGAL INDEXING ANGLE	The specified angle of rotation is not an integer multiple of the minimum indexing angle.
1564	INDEX TABLE AXIS – OTHER AXIS SAME TIME	The index table indexing axis and another axis have been specified in the same block.
1567	INDEX TABLE AXIS DUPLICATE AXIS COMMAND	Index table indexing was specified during axis movement or on an axis for which the index table indexing sequence was not completed.
1580	ENCODE ALARM (PSWD&KEY)	When an attempt was made to input a program, the specified password did not match the password on the tape and the password on tape was not equal to 0. When an attempt was made to output an encrypted tape, the password was not in the range 1 to 99999999. The password parameter is No. 3220.

FRYER MACHINE SYSTEMS INC
Rob Kaczansky

Number	Message	Description
1581	ENCODE ALARM (PARAMETER)	When an attempt was made to output an encrypted tape, the output code parameter was set to EIA. Set bit 1 (ISO) of parameter No. 0000 to 1. An incorrect instruction was specified for program encryption or protection. This alarm is generated if an attempt is made to perform program editing, deletion, or range-specified output in the protected range in the lock state. Or, a program outside the protected range is specified in rage specification output in the unlock state. The protected range is defined from the program No. preset by parameter No. 3222 up to the program No. preset to parameter No. 3223. When both parameters are set to 0, the protected range becomes O9000 to O9999.
1590	TH ERROR	A TH error was detected during reading from an input device. The read code that caused the TH error and how many statements it is from the block can be verified in the diagnostics screen.
1591	TV ERROR	An error was detected during the single–block TV error. The TV check can be suppressed by setting bit 0 (TVC) of parameter No. 0000 to 0.
1593	EGB PARAMETER SETTING ERROR	Error in setting a parameter related to the EGB (1) The setting of bit 0 (SYN) of parameter No. 2011, is not correct. (2) The slave axis specified with G81 is not set as a rotation axis. (bit 0 (ROT) of parameter No. 1006) (3) Number of pulses per rotation (Parameter No. 7772, 7773, 7782, or 7783 is not set.) (4) For a hobbing-machine-compatible command, parameter No. 7710 is not specified. (5) The slave axis specified with G81 for the spindle EGB is set as the slave axis for the simple spindle EGB. (6) No signal-based EGB synchronization ratio (parameters Nos. 7784 and 7785) has been set.
1594	EGB FORMAT ERROR	Error in the format of the block of an EGB command (1) T (number of teeth) is not specified in the G81 block. (2) In the G81 block, the data specified for one of T, L, P, and Q is out of its valid range. (3) In the G81 block, only one of P and Q is specified. (4) In the G81.5 block, there is no command for the master or slave axis. (5) In the G81.5 block, data out of the specified range is specified for the master or slave axis.
1595	ILL-COMMAND IN EGB MODE	During synchronization with the EGB, a command that must not be issued is issued. (1) Slave axis command using G27, G28, G29, G30,G30.1, G33, G53, etc. (2) Inch/metric conversion command using G20, G21, etc. (3) Synchronization start command using G81 or G81.5 when bit 3 (ECN) of parameter No. 7731 is 0 (4) For the slave axis of the spindle EGB, the Cs contour control mode is not selected.
1596	EGB OVERFLOW	An overflow occurred in the calculation of the synchronization coefficient.

FRYER MACHINE SYSTEMS INC
Rob Zahersky

Number	Message	Description
1597	EGB AUTO PHASE FORMAT ERROR	Format error in the G80 or G81 block in EGB automatic phase synchronization (1) R is outside the permissible range. (2) In the spindle EGB, reference position return is not performed for the master axis before G81R2 is specified.
1598	EGB AUTO PHASE PARAMETER SETTING ERROR	Error in the setting of a parameter related to EGB automatic phase synchronization (1) The acceleration/deceleration parameter is not correct. (2) The automatic phase synchronization parameter is not correct.
1618	ILLEGAL P-DATA(WHEEL WEAR COMPENSATION)	There is an error in P-data in compensation selection of grinding wheel wear compensation. Alternatively, the P command is not present.
1619	ILLEGAL AXIS(WHEEL WEAR COMPENSATION)	The compensation axis was switched in the grinding wheel wear compensation mode or compensation vector hold mode. Alternatively, parameters Nos. 5071 and 5072, which determine the axis to be subjected to grinding wheel wear compensation, are incorrectly set.
1805	ILLEGAL COMMAND	[I/O Device] An attempt was made to specify an illegal command during I/O processing on an I/O device. [G30 Zero Return] The P address Nos. for instructing No. 2 to No. 4 zero return are each out of the range 2 to 4. [Single Rotation Dwell] The specified spindle rotation is “0” when single rotation dwell is specified. [3-dimensional tool compensation] A G code that cannot be specified was specified in the 3-dimensional tool compensation mode. Scaling instruction G51, skip cutting G31 and automatic tool length measurement G37 were specified.
1806	DEVICE TYPE MISS MATCH	An operation not possible on the I/O device that is currently selected in the setting was specified. This alarm is also generated when file rewind is instructed even though the I/O device is not a FANUC Cassette.
1807	PARAMETER SETTING ERROR	An I/O interface option that has not yet been added on was specified. The external I/O device and baud rate, stop bit and protocol selection settings are erroneous.
1808	DEVICE DOUBLE OPENED	An attempt was made to open a device that is being accessed.

FRYER MACHINE SYSTEMS INC
Rob Zaherlsky

Number	Message	Description
1820	ILLEGAL DI SIGNAL STATE	(1) An each axis workpiece coordinate system preset signal was turned “1” in the state in which all axes on the path including the axis on which to perform preset with the each axis workpiece coordinate system were not stopped or in which a command was in execution. (2) When an M code for performing preset with an each axis workpiece coordinate system preset signal was specified, the each axis workpiece coordinate system preset signal was not turned “1”. (3) The auxiliary function lock is enabled. (4) When bit 6 (PGS) of parameter No. 3001 was set to 0 (M, S, T, and B codes are not output in the high speed program check mode), an M code for turning “1” an each axis workpiece coordinate system preset signal in the high speed program check mode was specified.
1823	FRAMING ERROR(1)	The stop bit of the character received from the I/O device connected to RS-232C interface 1 was not detected.
1830	DR OFF(2)	The data set ready input signal DR of the I/O device connected to RS-232C interface 2 turned OFF. The I/O device may not be turned on, or the cable may be broken or the printed-circuit board may be defective.
1832	OVERRUN ERROR(2)	The next character was received from the I/O device connected to RS-232C interface 2 before it could read a previously received character. The number of bits of input data may not match, or baud rate setting or I/O device specification number may be incorrect.
1833	FRAMING ERROR(2)	The stop bit of the character received from the I/O device connected to RS-232C interface 2 was not detected.
1834	BUFFER OVERFLOW(2)	The NC received more than 10 characters of data from the I/O device connected to RS-232C interface 2 even though the NC sent a stop code (DC3) during data reception. The I/O device or the printed-circuit board may be defective.
1889	ILLEGAL COMMAND IN G54.3	An illegal command was issued in G54.3 block. (1) An attempt was made to command G54.3 in a mode in which it cannot be accepted. (2) The command was not issued in a single block.
1892	ILLEGAL PARAMETER IN G43.3	A parameter related to nutating rotary head tool length compensation is incorrect.
1893	ILLEGAL PARAMETER IN G44.9	A parameter related to spindle unit compensation is incorrect.
1898	ILLEGAL PARAMETER IN G54.2	Illegal parameters Nos. 7580 to 7588 were specified for fixture offset.
1912	V-DEVICE DRIVER ERROR (OPEN)	An error occurred during device driver control.
1919	FATAL ERROR(USB MEMORY)	A fatal error occurred in the USB file system. To restore the file system, turn the power off.
1924	UNEXPECTED ERROR(USB MEMORY)	An unexpected error occurred in the USB file system.
1925	ILLEGAL PATH/FILE(USB MEMORY)	An invalid path or file name was specified.
1926	ACCESS DENIED(USB MEMORY)	The USB memory could not be accessed.
1927	DEVICE IN FORMATTING(USB MEMORY)	The USB memory is being formatted.
1928	DEVICE NOT FOUND(USB MEMORY)	No USB memory is inserted. Check the connection.
1932	DEVICE IS FULL(USB MEMORY)	The capacity of the USB memory is insufficient.
1937	RECOGNITION ERROR(USB MEMORY)	The format of the USB memory is invalid. Format the USB memory in FAT or FAT32 format. If the alarm is still issued, replace the USB memory.

FLYER MACHINE SYSTEMS INC
Rob Zaremski

Number	Message	Description
1938	END OF FILE FOUND(USB MEMORY)	The end of file was detected before EOR(%) was read. The file may be damaged.
1939	UNDEFINED ERROR(USB MEMORY)	An undefined error occurred.
1951	DEVICE IS BUSY(USB MEMORY)	The USB memory is busy.
1952	TOO MANY FILES(USB MEMORY)	The maximum number of files that can be opened concurrently is exceeded.
1953	REMOVED IN ACCESSING(USB MEMORY)	The USB memory was removed while being accessed.
1954	PATH/FILE EXIST(USB MEMORY)	The specified path or file already exists.
1955	PATH/FILE NOT FOUND(USB MEMORY)	The specified path or file is not found.
1956	DEVICE OVERCURRENT(USB MEMORY)	Overcurrent was detected in the USB memory. Replace the USB memory.
1957	PARITY ERROR(USB MEMORY)	A parity error occurred in the USB memory. Turn the power to the CNC off.
1960	ACCESS ERROR (MEMORY CARD)	Illegal memory card accessing This alarm is also generated during reading when reading is executed up to the end of the file without detection of the EOR code.
1961	NOT READY (MEMORY CARD)	The memory card is not ready.
1962	CARD FULL (MEMORY CARD)	The memory card has run out of space.
1963	CARD PROTECTED (MEMORY CARD)	The memory card is write–protected.
1964	NOT MOUNTED (MEMORY CARD)	The memory card could not be mounted.
1965	DIRECTORY FULL (MEMORY CARD)	The file could not be generated in the root directory for the memory card.
1966	FILE NOT FOUND (MEMORY CARD)	The specified file could not be found on the memory card.
1967	FILE PROTECTED (MEMORY CARD)	The memory card is write–protected.
1968	ILLEGAL FILE NAME (MEMORY CARD)	Illegal memory card file name
1969	ILLEGAL FORMAT (MEMORY CARD)	Check the file name.
1970	ILLEGAL CARD (MEMORY CARD)	This memory card cannot be handled.
1971	ERASE ERROR (MEMORY CARD)	An error occurred during memory card erase.
1972	BATTERY LOW (MEMORY CARD)	The memory card battery is low.
1973	FILE ALREADY EXIST	A file having the same name already exists on the memory card.
1990	SPL:ILLEGAL AXIS COMMAND	The axis specified by the smooth interpolation (G5.1Q2) is illegal.
1993	SPL:CAN'T MAKE VECTOR	The end point and the 2 previous point are the same in generation of the 3–dimensional tool compensation vector by the end point for smooth interpolation.
2002	NO KNOT COMMAND (NURBS)	Knot has not been specified, or a block not related to NURBS interpolation was specified in the NURBS interpolation mode.
2003	ILLEGAL AXIS COMMAND (NURBS)	An axis not specified as a control point was specified in the No. 1 block.
2004	ILLEGAL KNOT	There is an insufficient number of knot individual blocks.
2005	ILLEGAL CANCEL (NURBS)	The NURBS interpolation mode was turned OFF even though NURBS interpolation was not completed.
2006	ILLEGAL MODE (NURBS)	A mode that cannot be paired with the NURBS interpolation mode was specified.
2007	ILLEGAL MULTI-KNOT	Nested knots for each level can be specified for the start and end points.
2010	ILL. COMMAND IN TOLERANCE CON.	Illegal command is specified in tolerance control mode. Check the program.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	Description
2011	ILL. PARAMETER IN TOLERANCE CON.	Illegal parameter is set, which is related to tolerance control. Check the parameter settings.
2012	ILL. OPERATION IN TOLERANCE CON.	Illegal operation is done in tolerance control mode.
2032	EMBEDDED ETHERNET/DATA SERVER ERROR	An error was returned in the built-in Ethernet/data server function. For details, see the error message screen of the built-in Ethernet or data server.
2038	WRONG COMMUNICATION COMBINATION	The combination of the hardware and the software about communication function is wrong. The detail information is indicated to diagnosis information 4400 and 4401.
2052	#500-#549P-CODE MACRO COMMON SELECT(CANNOT USE SETVN)	The variable name cannot be entered. The SETVN command cannot be used with the P-CODE macro common variables #500 to #549.
2053	P-CODE VARIABLE NUMBER IS OUTSIDE OF RANGE	An attempt was made to enter a P-CODE-only variable not existing in the system.
2054	EXTENDED P-CODE VARIABLE NUMBER IS OUTSIDE OF RANGE	An attempt was made to enter an extended P-CODE-only variable not existing in the system.
2060	ILLEGAL PARAMETER IN G43.4/G43.5	The parameter for the pivot tool length compensation is incorrect.
2061	ILLEGAL COMMAND IN G43.4/G43.5	An illegal command was specified in tool center point control. - A rotation axis command was specified in tool center point control (type 2) mode. - With a table rotary type or mixed-type machine, a I, J, or K command was specified in the tool center point control (type 2) command (G43.5) block. - A command that does not move the tool center point (only a rotation axis is moved) was specified for the workpiece in the G02 mode. - G43.4 or G43.5 was specified in the tool center point control mode. - When the workpiece coordinate system is set as the programming coordinate system (bit 5 (WKP) of parameter No. 19696 is 1), G02 or G03 was specified while the rotation axis was not perpendicular to the plane.
2070	G02.1/ G03.1 FORMAT ERROR	- The format is invalid. - The specified arc exceeds the interpolation enable range.
2085	G04.1 FORMAT ERROR	G04.1 block is not specified singly.
2090	INTERRUPTED BLOCK NOT FOUND	The coordinates of the end point of the interrupted block in temporary execution and the coordinates of the memorized end point of interrupted block are different. The followings are assumed as a cause. - The number of blocks counted during temporary execution and the number of blocks counted during a normal operation are different. - Because number of the blocks or travel distance were changed by editing the program before interruption block, the coordinates of the end point of the interruption block were changed. - Amount of the offset was changed. - Coordinate system was changed.
4010	ILLEGAL REAL VALUE OF OBUF :	The real value for a output buffer is in error.
5006	TOO MANY WORD IN ONE BLOCK	The number of words in a block exceeds the maximum. The maximum is 26 words. However, this figure varies according to NC options. Divide the instruction word into two blocks.

FRYER MACHINE SYSTEMS INC
Rob Zakrzecky

Number	Message	Description
5007	TOO LARGE DISTANCE	Due to compensation, point of intersection calculation, interpolation or similar reasons, a movement distance that exceeds the maximum permissible distance was specified. Check the programmed coordinates or compensation amounts.
5009	PARAMETER ZERO (DRY RUN)	The dry run feedrate parameter No. 1410 or maximum cutting feedrate parameter for each axis has been set to 0. The parameter for the maximum cutting feedrate for each axis is No. 1432 if look-ahead acceleration/deceleration before interpolation is enabled and No. 1430 otherwise. Functions that cause look-ahead acceleration/deceleration before interpolation include AI contour control, tool center point control, and workpiece setting error compensation.
5010	END OF RECORD	The EOR (End of Record) code is specified in the middle of a block. This alarm is also generated when the percentage at the end of the NC program is input.
5011	PARAMETER ZERO (CUT MAX)	The maximum cutting feedrate parameter No. 1430 has been set to 0. The parameter is No. 1432 if look-ahead acceleration/deceleration before interpolation is enabled and No. 1430 otherwise. Functions that cause look-ahead acceleration/deceleration before interpolation include AI contour control, tool center point control, and workpiece setting error compensation.
5016	ILLEGAL COMBINATION OF M CODES	M codes which belonged to the same group were specified in a block. Alternatively, an M code which must be specified without other M codes in the block was specified in a block with other M codes.
5018	POLYGON SPINDLE SPEED ERROR	In G51.2 mode, the speed of the spindle or polygon synchronous axis either exceeds the clamp value or is too small. The specified rotation speed ratio thus cannot be maintained. For polygon turning between spindles: More information as to why this alarm occurred is indicated in diagnosis data No. 0471.
5020	PARAMETER OF RESTART ERROR	An invalid value is set in parameter No. 7310, which specifies the axis order in which the tool is moved along axes to the machining restart position in dry run. A value ranging from 1 to the number of controlled axes may be set in this parameter.
5043	TOO MANY G68 NESTING	3-dimensional coordinate conversion has been specified three or more times. To perform another coordinate conversion, perform cancellation, then specify the coordinate conversion.
5044	G68 FORMAT ERROR	Errors for 3-dimensional coordinate conversion command are: (1) No I, J, or K command was issued in 3-dimensional coordinate conversion command block. (without coordinate rotation option) (2) All of I, J, or K command were 0 in 3-dimensional coordinate conversion command block. (3) No rotation angle R was not commanded in 3-dimensional coordinate conversion command block.

FRYER MACHINE SYSTEMS INC
Rob Zaremski

Number	Message	Description
5050	ILL-COMMAND IN G81.1 MODE	- During chopping/oscillation, a move command has been issued for the chopping/oscillation axis. - Chopping/oscillation is specified for Cs contour control axis. - Machine lock for the axis while oscillation motion is executed. - Oscillation is specified for the slave axis of Superimposed control.
5058	G35/G36 FORMAT ERROR	A command for switching the major axis has been specified for circular threading. Alternatively, a command for setting the length of the major axis to 0 has been specified for circular threading.
5060	ILLEGAL PARAMETER IN G02.3/G03.3	The axis parameter setting to perform an exponential interpolation is in error. Parameter No. 5641: A liner axis number for performing an exponential interpolation Parameter No. 5642: A rotation axis number for performing an exponential interpolation The settable value is 1 to the number of control axes, but it must not be duplicated.
5061	ILLEGAL FORMAT IN G02.3/G03.3	The exponential interpolation command (G02.3/G03.3) has a format error. The command range for address I or J is -89.0 to -1.0 or +1.0 to +89.0. No I or J is specified or out-of -range value is specified. No address R, or 0 is specified.
5062	ILLEGAL COMMAND IN G02.3/G03.3	The value specified in an exponential interpolation command (G02.3/G03.3) is illegal. A value that does not allow exponential interpolation is specified. (For example, the value for In is 0 or negative.)
5065	DIFFERRENT AXIS UNIT(PMC AXIS)	Axes having different increment systems have been specified in the same DI/DO group for PMC axis control. Modify the setting of parameter No. 8010.
5068	FORMAT ERROR IN G31P90	No travel axis was specified. Two or more travel axes were specified.
5073	NO DECIMAL POINT	No decimal point has been specified for an address requiring a decimal point.
5074	ADDRESS DUPLICATION ERROR	The same address has been specified two or more times in a single block. Alternatively, two or more G codes in the same group have been specified in a single block.
5085	SMOOTH IPL ERROR 1	A block for specifying smooth interpolation contains a syntax error.
5110	IMPROPER G-CODE (AICC MODE)	An unspecifiable G code was specified in the AI contour control mode.
5115	ILLEGAL ORDER (NURBS)	There is an error in the specification of the rank.
5116	ILLEGAL KNOT VALUE (NURBS)	Monotone increasing of knots is not observed.
5117	ILLEGAL 1ST CONTROL POINT (NURBS)	The first control point is incorrect. Or, it does not provide a continuity from the previous block.
5118	ILLEGAL RESTART (NURBS)	After manual intervention with manual absolute mode set to on, NURBS interpolation was restarted.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
5122	ILLEGAL COMMAND IN SPIRAL	A spiral interpolation or conical interpolation command has an error. Specifically, this error is caused by one of the following: (1) L = 0 is specified. (2) Q = 0 is specified. (3) R/, R/, C is specified. (4) Zero is specified as height increment. (5) Zero is specified as height difference. (6) Three or more axes are specified as the height axes. (7) A height increment is specified when there are two height axes. (8) Q is specified when radius difference = 0. (9) Q < 0 is specified when radius difference > 0. (10) Q > 0 is specified when radius difference < 0. (11) A height increment is specified when no height axis is specified.
5123	OVER TOLERANCE OF END POINT IN SPIRAL	The difference between a specified end point and the calculated end point exceeds the allowable range (parameter No. 3471).
5124	CAN NOT COMMAND SPIRAL	A spiral interpolation or conical interpolation was specified in any of the following modes: (1) Scaling (2) Polar coordinate interpolation (3) In tool radius·tool nose radius compensation mode, the center is set as the end point.
5130	NC AND SUPERIMPOSE AXIS CONFLICT	In the PMC superposition axis control, the NC command and The PMC axis control command were conflicted. Modify the program and the ladder.
5131	NC COMMAND IS NOT COMPATIBLE	The PMC axis control and 3-dimensional coordinate conversion or a polar coordinate interpolation were specified simultaneously.
5132	CANNOT CHANGE SUPERIMPOSED AXIS	The superposition axis was selected for the axis for which the PMC superposition axis is being controlled.
5155	NOT RESTART PROGRAM BY G05	When learning control/preview repetitive control was enabled, an attempt was made to use feed hold or interlock to stop high-speed cycle machining/high-speed binary operation. Neither feed hold nor interlock can be used in such a case.
5195	DIRECTION CAN NOT BE JUDGED	Measurement is invalid in the tool compensation measurement value direct input B function. [For 1-contact input] (1) The recorded pulse direction is not constant. - The machine is at a stop in the offset write mode. - The servo power is off. - Pulse directions are diverse. (2) The tool is moving along the two axes (X-axis and Z-axis). [For the movement direction discrimination specification] (1) The recorded pulse direction is not constant. - The machine is at a stop in the offset write mode. - The servo power is off. - Pulse directions are diverse. (2) The tool is moving along the two axes (X-axis and Z-axis). (3) The direction indicated by the tool compensation write signal does not match the movement direction of the axis.
5196	ILLEGAL AXIS OPERATION	During HPCC or during the execution of a 5-axis-related function, an unavailable function was used.
5199	ILLEGAL FINE TORQUE SENSING PARAMETER	The setting of a fine torque sensing parameter is invalid. - An invalid controlled axis number is set as the target axis.

FRYER MACHINERY SYSTEMS INC
Rob Zanchetty

Number	Message	Description
5219	CAN NOT RETURN	Manual intervention and return cannot be performed during execution of 3-dimensional coordinate system conversion, tilted working plane indexing, tool center point control, or work setting error compensation.
5220	REFERENCE POINT ADJUSTMENT MODE	In case of distance coded linear scale I/F, the reference point auto setting bit 2 (DATx) of parameter No. 1819 is set to 1. Move the machine to reference position by manual operation and execute manual reference return.
5242	ILLEGAL AXIS NUMBER	A master axis number or a slave axis number was not set correctly when the flexible synchronization control mode was turned from off to on during automatic operation. In inter-path flexible synchronization control, this alarm is issued in either of the following cases. (The alarm is issued at the start of inter-path flexible synchronization control.) (1) The axis number of the master or slave axis is incorrect. (2) The master and slave axis settings make a loop.
5243	DATA OUTRANGE	A gear ratio was not set correctly when the flexible synchronization control mode was turned from off to on during automatic operation.

5244	TOO MANY DI ON	- When an attempt was made to change the flexible synchronization control status, the select signal was not turned on or off after the execution of the M code.
		- An attempt was made to turn flexible synchronization control on or off without stopping the tool along all axes. (Except when automatic phase synchronization for flexible synchronization control is used)
		- Flexible synchronization control was turned off in any of the following function modes:
		  - Tool center point control
		  - Tilted working plane indexing
		  - 3-dimensional cutter compensation
		  - Workpiece setting error compensation

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
5245	OTHER AXIS ARE COMMANDED	- For a flexible synchronization control group for which a PMC axis was a master axis, an attempt was made to turn on the synchronization mode during time other than automatic operation. - An attempt was made to turn on a synchronization group for which an PMC axis was a master axis when there existed a flexible synchronization control group for which a non-PMC, normal axis was a master axis. - The master and slave axes as synchronization axes overlap the EGB dummy axis. - The master and slave axes as synchronization axes overlap the chopping axis. - The master and slave axes as synchronization axes overlap the axis related to angular axis control. - The master and slave axes as synchronization axes overlap the axis related to composite control. - The master and slave axes as synchronization axes overlap the axis related to superposition control. - The slave axis as a synchronization axis overlaps the axis related to synchronization control. - The reference position return mode is turned on (was turned on). - A servo alarm occurred in a path in inter-path flexible synchronization control. - An emergency stop was applied in another path in inter-path flexible synchronization control. - When an attempt was made to execute flexible synchronization between different paths during automatic operation, the inter-path flexible synchronization mode was not enabled.
5255	G12.4/G13.4 FORMAT ERROR	The specified P, I, and K are incorrect or I is less than K.
5256	G12.4/G13.4 EXECUTION ERROR	(1) In groove cutting by continuous circle motion mode, a command other than G01, G02, G03, G04, G90, G91, and auxiliary functions is specified. (2) In a mode that cannot be used, the groove cutting by continuous circle motion command is specified.
5257	G41/G42 NOT ALLOWED IN MDI MODE	Tool radius/tool nose radius compensation was specified in MDI mode. (Depending on the setting of the bit 4 (MCR) of parameter No. 5008)
5303	TOUCH PANEL ERROR	The touch panel is not connected correctly, or the touch panel cannot be initialized when the power is turned on. Correct the cause then turn on the power again.
5305	ILLEGAL SPINDLE NUMBER	In a spindle select function by address P for a multiple spindle control, (1) Address P is not specified. (2) Parameter No. 3781 is not specified to the spindle to be selected. (3) An illegal G code which cannot be commanded with an S_P_; command is specified. (4) A multi spindle cannot be used because the bit 1 (EMS) of parameter No. 3702 is 1. (5) The spindle amplifier number of each spindle is not set in parameter No. 3717. (6) A prohibited command for a spindle was issued (parameter No. 11090). (7) An invalid value is set in parameter No. 11090.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	Description
5312	ILLEGAL COMMAND IN G10 L75/76/77	One of formats in G10L75, G10L76, or G10L77 to G11 commands is in error, or the command value is out of data range. Modify the program.
5316	TOOL TYPE NUMBER NOT FOUND	A tool with the specified tool-type number could not be found. Modify the program or register the tool.
5317	ALL TOOL LIFE IS OVER	The lives of all tools with the specified tool-type number have expired. Replace the tool.
5320	DIA./RAD.MODE CAN'T BE SWITCHED	In any of the following states, diameter/radius specification was switched: (1) When a buffered program is being executed (2) When a movement is being made on the axis (3) Diameter/radius setting switching function is specified to the axis for which bit 0 (TMDx) of parameter No. 10730 is set to 1.
5324	REFERENCE RETURN INCOMPLETE	Manual reference position return cannot be performed during tool center point control, three-dimensional coordinate conversion, the tilted working plane indexing, or workpiece setting error compensation.
5329	M98 AND NC COMMAND IN SAME BLOCK	A subprogram call which is not a single block was commanded during canned cycle mode.
5330	G50.9 FORMAT ERROR	- There is not coordinates value specification in G50.9 block. - There is not M code, B code command in the G50.9 block. - G50.9 is commanded in canned cycle mode. - 3 blocks of G50.9 block are commanded consecutively.
5331	ILLEGAL COMMANDED POSITION	It didn't reach a commanded absolute coordinate value. The movement command or absolute coordinates at G50.9 block are wrong.
5339	ILLEGAL FORMAT COMMAND IS EXECUTED IN SYNC/MIX/OVL CONTROL.	(1) The value of P, Q, or L specified by G51.4/G50.4/G51.5/G50.5/G51.6/G50.6 is invalid. (2) A duplicate value is specified by parameter No. 12600.
5346	RETURN TO REFERENCE POINT	The coordinate establishment of the Cs contour control axis is not made. Perform a manual reference position return. (1) When Cs coordinate establishment is made for the Cs-axis for which the Cs-axis reference position status signal CSPENx is 0 (2) When positional information is not sent from the spindle amplifier (3) When the servo off state is entered during the start of Cs-axis coordinate establishment (4) When the Cs-axis is subjected to synchronous control or superposition control (5) When the emergency stop state is entered during coordinate establishment (6) When an attempt is made to release composite control for the Cs axis being subjected to coordinate establishment (7) When an attempt is made to start synchronous, composite, or superposition control for the Cs axis being subjected to coordinate establishment.
5355	S CODE IS NOT COMMANDED AT G96	S code is not commanded at G96. Command S code at G96 block.
5356	IMPROPER G-CODE	Illegal G code was commanded in the hypothetical axis command mode or in the real axis command mode.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

Number	Message	Description
5357	ILLEGAL AXIS SELECT	The real axis is commanded in the hypothetical axis commanded mode. The hypothetical axis is commanded in the real axis command mode.
5359	MODE CHANGE ERROR	This alarm is generated in the following cases. (1) Mode switching was executed without using non-buffering M code in automatic operation. (2) Mode was switched to the hypothetical axis command mode in case that any axis in the hypothetical plane loses the reference position. (3) During the hypothetical axis command mode, any axis in the hypothetical plane loses the reference position. (4) Mode was switched in case the parameter LRP (No.1401#1) is set to 0. (5) Mode switching was executed during the following mode. - Cutter radius compensation - Tool length compensation - Tool offset - Scaling - Programmable mirror image - Coordinate system rotation - Canned cycle (6) Mode switching was executed when real axis in the hypothetical plane is moving.
5360	TOOL INTERFERENCE CHECK ERROR	This alarm is issued when interference with another tool is caused by a data modification based on G10 data input or file inputting or when an attempt is made to modify the tool figure data of a tool registered in the cartridge.
5361	ILLEGAL MAGAZINE DATA	Tools stored in the cartridge are interfering with each other. Reregister the tools in the cartridge, or modify the tool management data or tool figure data. If this alarm is issued, no tool interference check is made when tools are registered in the cartridge management table. Moreover, empty pot search operation does not operate normally. If this alarm is issued, the power must be turned off before operation is continued.
5362	CONVERT INCH/MM AT REF-POS	(1) When the bit 1 (INAx) of parameter No. 14000 set to 1, the inch-metric switch command is executed at a position other than the machine coordinate system origin (position = zero). (2) When the bit 3 (IMRx) or bit 2 (IRFx) of parameter No. 14000 set to 1, the inch-metric switch command is executed at the other than reference position (parameter No. 1240). Perform an inch/metric conversion after returning to the reference position. (3) When bit 0 (NIM) of parameter No.11222 set to 1, the inch/metric conversion by programmable parameter input or #3005 (setting data) was executed at a position other than the machine coordinate system origin (position = zero).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	Description
5364	ILLEGAL COMMAND IN PROGRAM CHECK	(1) An unspecifiable G code was specified in the high-speed program check mode. (2) The angular axis control option or customer's board option is enabled. (3) One of the following operations was performed. - Chopping in the high-speed program check mode - Starting the high-speed program check mode during chopping - High-speed cycle machining in the high-speed program check mode - Reference position return of an axis for which the reference position is not established, in the high-speed program check mode (4) Switching of PMC axis selection signal EAX* was performed. (5) G10 was specified for bit 3 (PGR) of parameter No. 3454 in the high-speed program check mode. (6) G10 was specified for bit 6 (PGS) of parameter No. 3001 in the high-speed program check mode.
5365	CANNOT CHANGE OF PROGRAM CHECK MODE	Switching of high-speed program check input signal PGCK was performed during execution of the program.
5372	IMPROPER MODAL G-CODE (G53.2)	In a block in which G53.2 is specified, a G code in group 01 other than G00 and G01 is specified. Or, G53.2 is specified when the modal G code in group 01 is in a state other than the G00 and G01 states.
5373	ARGUMENT CONVERSION ERROR	For outputting a target MDI program for program restart, a macro call argument cannot be converted to a 9-digit number.
5374	FSC MODE MISMATCH IN RESTART	The current flexible synchronization mode differs from the flexible synchronization mode specified in a programmed command in the program restart block. Check the restart block.
5375	FSC MODE CAN NOT CHANGED	The flexible synchronization mode was changed during the execution of program restart. Check the restart block.
5376	FSC SLAVE AXIS CAN NOT COMMANDED	In the flexible synchronization mode, a command was specified for the slave axis.
5377	INVALID COMMAND AFTER FSC OFF	After the flexible synchronization mode was canceled, an incremental command was specified before an absolute command for the axis specified as the slave axis.
5378	INVALID RESTART BLOCK	The block specified as the restart block after the flexible synchronization mode was canceled was not a block after an absolute command for the axis specified as the slave axis. Check the restart block.
5379	WRITE PROTECTED TO SLAVE AXIS	A parameter cannot be set directly for a slave axis of axis synchronous control.
5381	INVALID COMMAND IN FSC MODE	An attempt was made to issue the following commands: - When the reference position for the master axis under flexible synchronization control has not been established, G28 command for the master axis. - G27/G28/G29/G30/G30.1/G53 command for a slave axis.

FRYER MACHINE SYSTEMS INC Rob Zahensky

Number	Message	Description
5384	RETRACT FOR RIGID CANNOT BE CMD	In retraction for 3-dimensional rigid tapping by the G30 command, different program coordinate system are used at the stop of rigid tapping and in retraction for rigid tapping. The program coordinate system, is the coordinate system for the program after conversion is determined by the rotation center, the direction of rotation, and the rotation angle that was commanded by the arguments of the G68 or G68.2 command. In other words, the arguments of G68 or G68.2 command are different at the stop of rigid tapping and in retraction for rigid tapping. Modify the program.
5389	REVERSE MOTION CANNOT START	Reverse motion function for restart cannot start for the following reasons. It is necessary to recover a tool by a manual operation. - The interruption was not done during the program performing (The program is normally ended by M02/M30). - The interruption was done while rigid tapping or thread cutting was performed.
5391	CAN NOT USE G92	Workpiece coordinate system setting G92 (or G50 for the lathe system G-code system A) cannot be specified. (1) After tool length compensation was changed by tool length compensation shift type, G92 was specified when no absolute command is present. (2) G92 was specified in the block in which G49 is present.
5406	G41.3/G40 FORMAT ERROR	(1) The G41.3 or G40 block contains a move command. (2) The G41.3 block contains a G or M code that suppresses buffering.
5407	ILLEGAL COMMAND IN G41.3	(1) In the G41.3 mode, a G code of group 01 other than G00 and G01 is specified. (2) In the G41.3 mode, an offset command (a G code of group 07) is specified. (3) The block next to G41.3 (startup) specifies no movement.
5408	G41.3 ILLEGAL START_UP	(1) In a mode of group 01 other than G00 and G01, G41.3 (startup) is specified. (2) The included angle between the tool vector and move vector is 0 or 180 degrees at the time of startup.
5420	ILLEGAL PARAMETER IN G43.4/G43.5	A parameter related to tool center point control is illegal. - Look-ahead acceleration/deceleration before interpolation is disabled. Set parameter No. 1660. - Rapid traverse acceleration/deceleration before interpolation is disabled. Set bit 1 (LRP) of parameter No. 1401, bit 5 (FRP) of parameter No. 19501, parameter No. 1671, and parameter No. 1672. - The AI contour control I or AI contour control II option is absent. Set bit 2 (AAI) of parameter No. 11260 to 0.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	Description
5421	ILLEGAL COMMAND IN G43.4/G43.5	An illegal command was specified in tool center point control. - A rotation axis command was specified in tool center point control (type 2) mode. - With a table rotary type or mixed-type machine, a I,J,K command was specified in the tool center point control (type 2) command (G43.5) block. - A command that does not move the tool center point (only a rotation axis is moved) was specified for the workpiece in the G02/G03 mode. - When the workpiece coordinate system is set as the programming coordinate system (bit 5 (WKP) of parameter No. 19696 is 1), G02 or G03 was specified while the rotation axis was not perpendicular to the plane. - A G code not specifiable during the tool center point control mode was specified. - The modal code used to specify tool center point control is incorrect. - If, in tool center point control mode, any of the following conditions is met, an axis not related to tool center point control (non 5-axis machining control axis) is specified: (1) The option, the expansion of axis move command in tool center point control, is not provided. (2) The number of non 5-axis machining control axes exceeds the maximum number of axes that can be specified. (3) NURBS interpolation is performed. - When bit 0 (RCM) of parameter No. 11200 is set to 0 to disable tool direction compensation, tool center point control is specified during the workpiece setting error compensation/tilted working plane indexing mode. - When tool posture control is enabled under tool center point control (type 2), a command is specified to set a tool posture near a singular point. (This alarm may be suppressed with bit 3 (NPC) of parameter No. 19696.) Check the machine configuration and specification. - When tool posture control is enabled under tool center point control (type 1), a rotary axis angular displacement that disables tool posture control is specified. Check the machine configuration and specification. - During tool center point control (type 2) or tool posture control, NURBS interpolation is specified. Check the specification.

FRYER MACHINE SYSTEMS INC
Rob Zelinsky

Number	Message	Description
5421	ILLEGAL COMMAND IN G43.4/G43.5	- In a state in which the shift of a mirror image remains, tool center point control, tool posture control, or the cutting point command is specified. - It is intervened with an MDI command during a single block stop in high-speed smooth TCP mode. - With a table rotary type or composite type machine, one of the following conditions is satisfied. (1) Manual intervention is performed on a rotation axis during tool center point control. (2) Tool center point control is specified in the state in which the value of manual intervention is not zero. - When the tool management function is available, H/D command and P command are specified at the same time in the tool center point control start block (G43.4/G43.5). - The ",L3" command of block for starting tool center point control (type 2) is illegal command. - Illegal address P - Illegal address Q on roll-pitch-yaw command "P1"
5422	EXCESS VELOCITY IN G43.4/G43.5	An attempt was made to make a movement at an axis feedrate exceeding the maximum cutting feedrate by tool center point control.
5424	ILLEGAL TOOL DIRECTION	The rotation axis position for specifying the tool axis direction is not ±90° × n (n = 0, 1, 2,・・・).
5425	ILLEGAL OFFSET VALUE	The offset number is incorrect.
5430	ILLEGAL COMMAND IN 3-D CIR	In a modal state in which 3-dimensional circular interpolation cannot be specified, a 3-dimensional circular interpolation (G02.4/G03.4) is specified. Alternatively, in 3-dimensional circular interpolation mode, a code that cannot be specified is specified.
5432	G02.4/G03.4 FORMAT ERROR	A 3-dimensional circular interpolation command (G02.4/G03.4) is incorrect.
5433	MANUAL INTERVENTION IN G02.4/G03.4 (ABS ON)	In 3-dimensional circular interpolation mode (G02.4/G03.4), manual intervention was made when the manual absolute switch was on.
5435	PARAMETER OUT OF RANGE (TLAC)	Illegal parameter setting. (Set value is out of range.) Check parameters Nos. 19655, 19656, 19657, and 1022.
5436	ILLEGAL PARAMETER SETTING OF ROTARY AXIS(TLAC)	Illegal parameter setting. (axis of rotation setting)
5437	ILLEGAL PARAMETER SETTING OF MASTER ROTARY AXIS(TLAC)	Illegal parameter setting. (master axis of rotation setting)
5445	CAN NOT COMMAND MOTION IN G39	Corner circular interpolation (G39) of tool radius/tool nose radius compensation is not specified alone but is specified with a move command.
5446	NO AVOIDANCE AT G41/G42	Because there is no interference avoidance vector, the interference check avoidance function of tool radius/tool nose radius compensation does not work.
5447	DANGEROUS AVOIDANCE AT G41/G42	The interference check avoidance function of tool radius/tool nose radius compensation operation will lead to danger.
5448	INTERFERENCE TO AVD. AT G41/G42	In the interference check avoidance function of tool radius/tool nose radius compensation, a further interference occurs for an already created interference avoidance vector.
5456	TOO MANY G68.2 NESTING	Tilted working plane indexing G68.2 was specified more than once. To perform another coordinate conversion, perform cancellation, then specify the coordinate conversion.

FRYER MACHINE SYSTEMS INC
Rob Ashelsky

Number	Message	Description
5457	G68.2 FORMAT ERROR	A G68.2 format error occurred.
5458	ILLEGAL USE OF G53.1/G53.6	- G53.1/G53.6 was specified preceding G68.2. - G53.1/G53.6 needs to be specified solely. - There is no angle solution for the rotation axis that controls the tool direction in the +Z-axis direction of the feature coordinate system. - No tool is specified with a G53.6 command.

5459	MACHINE PARAMETER INCORRECT	- The parameters Nos.19665 to 19667, 19680 to 19714, and 12321 for configuring the machine are incorrect.
		- The axis specified with parameter No. 19681 or No. 19686 is not a rotary axis.
		- In parameter No. 1022, the basic three axes are not set.
		- A rotary axis end point found by the NC with tool center point control type 2, 3-dimensional cutter compensation type 2, cutting point command type 2, or the tilted working plane indexing is not within the range set by parameter No. 19741 to No. 19744.
		- No rotary axis end point was found with tool center point control type 2, 3-dimensional cutter compensation type 2, or cutting point command type 2. Check the machine configuration and specification.
		- On a machine whose rotary axis is a virtual axis, tool center point control type 2, 3-dimensional cutter compensation type 2, or cutting point command type 2 was specified.
		- When programming is performed in the workpiece coordinate system, tool center point control type 2, 3-dimensional cutter compensation type 2, or cutting point command type 2 is specified.
		- 5-axis machining function which can not be used in the machine which has two linear axes (bit 6 (HAL) of parameter No.11269 = 1) is executed.
		- When tool axis direction control (G53.1/G53.6) is specified on the following condition that parameter is set incorrectly if bit 7 (AIR) of pa PW0037rameter No.11221 is set 1.
		  - In case that the movement range of rotation axis is set to the roll-over axis
		  - In case that roll-over function is set to rotary axis (B type)
		- If a parameter for setting a 5-axis machining function, the reference rotation axis position, is to be set in the machine coordinate system (bit 7 (SPM) of parameter No.19754 = 1), an inclined rotary axis is used.

FRYER MACHINE SYSTEMS INC
Rob Zaharsky

Number	Message	Description
5460	ILLEGAL USE OF 3-DIMENSIONAL CUTTER COMPENSATION	- In the 3-dimensional cutter compensation mode (except the tool side offset function for a tool rotation type machine), a move command other than G00/G01 is specified. - With a table rotation type machine, when bit 1 (PTD) of parameter No. 19746 is set to 1, a plane selection is made with an axis other than the basic three axes at the start of 3-dimensional cutter compensation. - When bit 1 (SPG) of parameter No. 19607 is set to 1, there is a discrepancy between the machine type set in parameter No. 19680 and a G code specifying 3-dimensional cutter compensation (G41.2, G42.2, G41.4, G42.4, G41.5, or G42.5). - With a machine that is not of the tool rotation type, G41.3 is specified. - When bit 5 (WKP) of parameter No. 19696 is set to 0, and bit 4 (TBP) of parameter No. 19746 is set to 0, 3-dimensional cutter compensation and tool center point control are used at the same time. - A rotation axis command is specified in the 3-dimensional cutter compensation mode (type 2). - With a table rotation type or mixed type machine, IJK is specified in a block that specifies 3-dimensional cutter compensation (type 2) (G41.6/G42.6). - An illegal G code is specified in the 3-dimensional cutter compensation mode. - When 3-dimensional cutter compensation is specified, the modal state is illegal. - When the table coordinate system is set as the programming coordinate system, table rotate and 3-dimensional cutter compensation are specified after the start of tool center point control. - There is a difference in type1/type 2 specification between 3-dimensional cutter compensation and tool center point control. - When 3-dimensional cutter compensation and tool center point control are used at the same time, one of these functions that is specified earlier than the other is canceled earlier.
5461	ILLEGAL USE OF G41.2/G42.2/G41.5/G42.5	A move command other than G00 or G01 was performed during 3-dimensional cutter compensation in a mixed-type machine.
5462	ILLEGAL COMMAND (G68.2/G69)	(1) The modal setting used when G68.2 or G69 is specified is incorrect. (2) An unspecifiable G code was specified in the G68.2 mode. (3) The offset vector of tool radius/tool nose radius compensation is not canceled when G68.2 or G69 is specified.
5463	ILLEGAL PARAMETER IN 3-DIMENSIONAL CUTTER COMPENSATION	A parameter related to 3-dimensional cutter compensation is illegal. - Look-ahead acceleration/deceleration before interpolation is disabled. Set parameter No. 1660. - Rapid traverse acceleration/deceleration before interpolation is disabled. Set bit 1 (LRP) of parameter No. 1401, bit 5 (FRP) of parameter No. 19501, and parameter No. No.1671.

FRYER MACHINE SYSTEMS INC
Rob Zohorsky

Number	Message	Description
5464	ILLEGAL COMMAND IN G43.8/G43.9	An illegal value is specified with the cutting point command. - A negative value is set to the parameter (No.11419). - In case of tool compensation memory A and B of machining center system, D code is specified in G43.8/G43.9 block. - In case of tool offset memory C and bit 3 (ON1) of parameter No. 11269 is 1 in machining center system, D code is specified in G43.8/G43.9 block. - All of (I, J, K) following ",L2" are specified as 0. - The angle between the tool length direction and the perpendicular to the cutting surface exceeds the setting value of parameter No.11418. - The tool radius offset is less than the corner-R offset. - On a tool rotation type machine, manual intervention is performed on the rotation axis. - In the case that scaling is not valid (bit 0 (SCLx) of parameter No.5401 is 1 and bit 6 (XSC) of parameter No.5400 is 0), scaling is specified. - An illegal G code was specified in cutting point command mode.
5557	NO MAX SP SPEED CLAMP COMMAND	No command for maximum spindle speed clamping (M series: G92S_; T series: G50S_;) was executed before the constant surface speed control command (G96S_) was started. Execute a command for maximum spindle speed clamping.
5559	ILL. AXIS OPERATION (COORD SYS)	When the parameter MSC(No.11501#2)=1, and the workpiece coordinate system is shifted from the machine coordinate system by the movement command in the machine lock state, the axis movement was commanded. Please operate "reference position return" or "workpiece coordinate system preset" etc.
5560	ILLEGAL DEPTH OF CUT	On the peck rigid tapping, the depth of cut (Q) is smaller than the cutting start distance (d).
PS5561	CANCEL CANNED GRINDING CYCLE	Multiple canned cycle is specified without canceling canned grinding cycle. Please specify a G code of 01 group for canceling canned grinding cycle.
5577	ILLEGAL AXIS SELECTED (G05)	The number of controlled axes set to 1 by bit 0 (HAS) of parameter No. 7506 is 0 or greater than the maximum number of controlled axes of High-speed cycle machining/High-speed binary operation.
5580	CANNOT GET SYSTEM VARIABLE	System variable (#180801 to #180850) cannot be read when parameter No.8311 is incorrect value. Confirm the setting of parameter No.8311.

(4) Parameter writing alarm (SW alarm)

Number	Message	Description
SW0100	PARAMETER ENABLE SWITCH ON	The parameter setting is enabled (bit (PWE) of parameter No.8900 is set to “1”). To set the parameter, turn this parameter ON. Otherwise, set to OFF.

FRYER MACHINE SYSTEMS INC
Rob Zanetti

(5) Servo alarms (SV alarm)

Number	Message	Description
SV0001	SYNC ALIGNMENT ERROR	In feed axis synchronization control, the amount of compensation for synchronization exceeded the parameter No. 8325 setting value. This alarm occurs for a master or slave axis.
SV0002	SYNC EXCESS ERROR ALARM 2	In feed axis synchronization control, the amount of synchronization error exceeded the parameter No. 8332 setting value. When the synchronization is not completed after power-up, the determination is made by the parameter value (No. 8332) multiplied by the parameter No. 8330 multiplier. This alarm occurs only for a slave axis only.
SV0003	SYNCHRONOUS/COMPOSITE/SUPER IMPOSED CONTROL MODE CAN'T BE CONTINUED	Since as axis in synchronization, composition, or superposition mode caused a servo alarm, the mode could not be continued, If one of the axes in a mode causes a servo alarm, all axes relating to the axis enter the servo-off state. This alarm is generated to enable the cause of the servo-off state to be checked.
SV0004	EXCESS ERROR (G31)	The amount of positional deviation during torque limit skip command operation exceeded the limit value of the parameter No.6287.
SV0005	SYNC EXCESS ERROR (MCN)	In feed axis synchronization control, for synchronization, the difference value of the machine coordinate between a master and slave axes exceeded the parameter No. 8314 setting value. This alarm occurs for a master or slave axis.
SV0006	ILLEGAL TANDEM AXIS	For the slave axis under tandem control, absolute position detection is set (parameter bit 5 (APC) of parameter No. 1815 = 1).
SV0007	SV ALM ANOTHER PATH(MULTI AMP.)	When a multi-axis amplifier was used in a multi-path system across paths, a servo alarm occurred on an axis belonging to another path. When a system with two or more paths and multiple servo axes between paths are controlled by a multi-axis amplifier, if a servo alarm occurs on an axis belonging to another path of the same amplifier, the MCC of the amplifier goes down and SV0401 V-READY OFF occurs on an axis belonging to the local path in the same amplifier. Since SV0401 is caused by a servo alarm occurred on an axis in another path, SV0007 is caused together to clearly indicate the fact. The axis belonging to another path in the same amplifier resolves the cause of the servo alarm.
SV0010	SV OVERHEAT	Amplifier internal overheat
SV0011	SV MOTOR OVER CURRENT(SOFT)	The digital servo software detected an abnormal specified value. Possible causes include an unconnected power cable, cable disconnection (open phase), and short-circuit.
SV0012	SV DRIVE OFF CIRCUIT FAILURE	The two drive off inputs are not in the same status or a drive off circuit error occurred.
SV0013	SV CPU BUS FAILURE	An error was found in CPU bus data in the amplifier.
SV0014	SV CPU WATCH DOG	An error occurred in CPU operation in the amplifier.
SV0015	SV LOW VOLT DRIVER	The driver power supply voltage has dropped in the amplifier. Possible causes include improper insertion of the control PC board and amplifier failure.

FRYER MACHINE SYSTEMS INC
Rob Kamensky

Number	Message	Description
SV0016	SV CURRENT DETECT ERROR	An error was found in motor current detection data in the amplifier. Possible causes include improper insertion of the control PC board and amplifier failure.
SV0017	SV INTERNAL SERIAL BUS FAILURE	An error occurred in serial bus communication in the amplifier. Possible causes include improper insertion of the control PC board and amplifier failure.
SV0018	SV ROM DATA FAILURE	An error was found in ROM data in the amplifier.
SV0019	SV MOTOR OVER CURRENT (GND FAULT)	A ground fault occurred in the motor, power cable, or amplifier.
SV0020	PS GROUND FAULT	A ground fault occurred in the motor, power cable, or amplifier.
SV0024	PS SOFT THERMAL	A load higher than the rating was applied.
SV0031	PS ILLEGAL PARAMETER	An invalid value is set for a PS control parameter.
SV0032	PS CONTROL AXIS ERROR 1	An invalid value is set for parameter No. 2557. Set parameter APS (No.11549#0) to 1 and execute automatic setting.
SV0033	PS CONTROL AXIS ERROR 2	An invalid value is set for parameter No. 2557. Set parameter APS (No.11549#0) to 1 and execute automatic setting.
SV0034	PS HARDWARE ERROR	A PS hardware error was detected.
SV0040	PS EXTERNAL INPUT COMPONENT ERROR	There is something trouble in control parameter for PS, cable, PS itself, or the external component such as input filter or transducer.
SV0041	PS PFB-R ERROR	A PFB-R error is detected.
SV0042	PS PFB-C ERROR	A PFB-C error is detected.
SV0043	PS SUB MODULE ERROR	The wiring or the combination of components is illegal.
SV0044	MISMATCHED FUNCTION CODE	CNC, SV, SP or PS software has been update. Turn the power off, then restart.
SV0046	DIDN'T ARRIVE AT SPECIFIED VALUE (PRESSURE AND POSITION CONTROL)	Pressure or position feedback did not arrived at specified value at the end of blocks commanded pressure arrival confirming mode (G14.2P1) or position arrival confirming mode (G14.2P2).
SV0048	SMART RIGID TAPPING STOP	The excitation of servo motors is turned off because smart rigid tapping is stopped. This alarm is accompanied by spindle alarm.
SV0049	SYNC EXCESS ERROR (POS DEV)	In axis synchronous control during pressure and position control mode, the difference in the amount of positional deviation between the master and slave axes exceeded the parameter No. 10416 setting value. This alarm occurs for the master or slave axis.
SV0050	CAN NOT CHANGE AXIS SYNCHRONOUS	In axis synchronous control, the synchronous operation and the normal operation were changed illegally. This alarm occurs for the master or slave axis.
SV0301	APC ALARM: COMMUNICATION ERROR	(1) Since the absolute-position detector of the phase A/B caused a communication error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective. (2) When the servo axis is disabled, an absolute position detector cannot be used. Check the setting of bit 5 (APCx) of parameter No. 1815 and bit 4 (KSVx) of parameter No. 11802.

FRYER MACHINE SYSTEMS INC
Rob Zakrzewsky

Number	Message	Description
SV0302	APC ALARM: OVER TIME ERROR	Since the absolute-position detector of the phase A/B caused an overtime error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0303	APC ALARM: FRAMING ERROR	Since the absolute-position detector of the phase A/B caused a framing error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0304	APC ALARM: PARITY ERROR	Since the absolute-position detector of the phase A/B caused a parity error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0305	APC ALARM: PULSE ERROR	Since the absolute-position detector of the phase A/B caused a pulse error, the correct machine position could not be obtained. The absolute-position detector, or cable is thought to be defective.
SV0306	APC ALARM: OVER FLOW ERROR	Since the amount of positional deviation overflowed, the correct machine position could not be obtained. Check to see the parameter No. 2084 or No. 2085.
SV0307	APC ALARM: MOVEMENT EXCESS ERROR	Since the machine moved excessively, the correct machine position could not be obtained.
SV0360	ABNORMAL CHECKSUM(INT)	The checksum alarm occurred on the built–in Pulsecoder.
SV0361	ABNORMAL PHASE DATA(INT)	The phase data abnormal alarm occurred on the built–in Pulsecoder.
SV0363	ABNORMAL CLOCK(INT)	The clock alarm occurred on the built–in Pulsecoder.
SV0364	SOFT PHASE ALARM(INT)	A digital servo soft detected an abnormality on the built in Pulsecoder.
SV0365	BROKEN LED(INT)	The digital servo software detected abnormal data on the built–in Pulsecoder.
SV0366	PULSE MISS(INT)	A pulse error occurred on the built–in Pulsecoder.
SV0367	COUNT MISS(INT)	A count error occurred on the built–in Pulsecoder.
SV0368	SERIAL DATA ERROR(INT)	The communications data could not be received from the built–in Pulsecoder.
SV0369	DATA TRANS. ERROR(INT)	A CRC error or stop bit error occurred in the communications data from the built–in Pulsecoder.
SV0380	BROKEN LED(EXT)	Separate detector error
SV0381	ABNORMAL PHASE (EXT)	An abnormal alarm in the position data occurred on the separate detector.
SV0382	COUNT MISS(EXT)	A count error occurred on the separate detector.
SV0383	PULSE MISS(EXT)	A pulse error occurred on the separate detector.
SV0384	SOFT PHASE ALARM(EXT)	The digital servo software detected abnormal data on the separate detector.
SV0385	SERIAL DATA ERROR(EXT)	The communications data could not be received from the separate detector.
SV0386	DATA TRANS. ERROR(EXT)	A CRC error or stop bit error occurred in the communications data from the standalone detector.
SV0387	ABNORMAL ENCODER(EXT)	An abnormality occurred on a separate detector. For more information, contact the scale manufacturer.
SV0401	IMPROPER V_READY OFF	Although the ready signal (PRDY) of the position control was ON, the ready signal (VRDY) of the velocity control was OFF.

FRYER MACHINE SYSTEMS INC
Rob Kamenski

Number	Message	Description
SV0403	CARD/SOFT MISMATCH	The combination of the axis control card and the servo software is incorrect. Probable causes are given below. - The correct axis control card is not attached. - The correct servo software is not installed in flash memory.
SV0404	IMPROPER V_READY ON	Although the ready signal (PRDY) of the position control was OFF, the ready signal (VRDY) of the velocity control was ON.
SV0407	EXCESS ERROR	The difference value of the amount of positional deviation for the synchronization axis exceeded the setting value. (during synchronization control only)
SV0409	DETECT ABNORMAL TORQUE	An abnormal load was detected on the servo motor, or during Cs axis or spindle positioning. The alarm can be canceled by RESET.
SV0410	EXCESS ERROR (STOP)	The amount of positional deviation during stopping exceeded the parameter No. 1829 setting value.
SV0411	EXCESS ERROR (MOVING)	The amount of positional deviation during traveling became excessive than the parameter setting value.
SV0413	LSI OVERFLOW	The counter for the amount of positional deviation overflowed
SV0415	MOTION VALUE OVERFLOW	The velocity exceeding the travel velocity limit was commanded.
SV0417	ILL DGTL SERVO PARAMETER	A digital servo parameter setting is incorrect. When bit 4 of diagnosis data No. 0203 is 1, an illegal parameter was detected by the servo software. Identify the cause with reference to diagnosis data No. 0352. When bit 4 of diagnosis data No. 0203 is 0, the CNC software detected an illegal parameter. Probable causes are given below (see diagnosis data No. 0280). (1) The value specified in parameter No. 2020 as the motor model falls outside the specified range. (2) The motor rotation direction in parameter No. 2022 is not set to a correct value (111 or -111). (3) The speed feedback pulse count per motor rotation in parameter No. 2023 is set to a negative or other incorrect value. (4) The position feedback pulse count per motor rotation in parameter No. 2024 is set to a negative or other incorrect value.
SV0420	SYNC TORQUE EXCESS	In feed axis synchronization control, for synchronization, the difference value of torque between a master and slave axes exceeded the parameter No. 2031 setting value. This alarm occurs for a master axis.
SV0421	EXCESS ERROR(SEMI-FULL)	The difference between the feedback from the semi and full sides exceeded the setting of parameter No. 2118.
SV0422	EXCESS VELOCITY IN TORQUE	In torque control, the commanded permissible velocity was exceeded.
SV0423	EXCESS ERROR IN TORQUE	In torque control, the total permissible move value specified as a parameter was exceeded.
SV0430	SV MOTOR OVERHEAT	The servo motor has overheated.
SV0431	PS OVERLOAD	Overheat
SV0432	PS LOW VOLT. CONTROL	The control power supply voltage has dropped.
SV0433	PS LOW VOLT. DC LINK	Low DC link voltage

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	Description
SV0434	SV LOW VOLT CONTROL	Low control power voltage
SV0435	SV LOW VOLT DC LINK	Low DC link voltage
SV0436	SOFTTHERMAL(OVC)	The digital servo software detected a software thermal (OVC).
SV0437	PS OVERCURRENT	Overcurrent on input circuit section.
SV0438	SV ABNORMAL CURRENT	Motor overcurrent
SV0439	PS OVER VOLT. DC LINK	The DC link voltage is too high.
SV0440	PS EXCESS-REGENERATION2	Excessive generative discharge
SV0441	ABNORMAL CURRENT OFFSET	The digital servo software detected an abnormality in the motor current detection circuit.
SV0442	PS PRE-CHARGE FAILURE	The spare charge circuit for the DC link is abnormal.
SV0443	PS INTERNAL FAN FAILURE	Internal cooling fan failure.
SV0444	SV INTERNAL FAN FAILURE	Internal cooling fan failure.
SV0445	SOFT DISCONNECT ALARM	The digital servo software detected a disconnected Pulsecoder.
SV0446	HARD DISCONNECT ALARM	The hardware detected a disconnected built–in Pulsecoder.
SV0447	HARD DISCONNECT(EXT)	The hardware detected a disconnected separate detector.
SV0448	UNMATCHED FEEDBACK ALARM	The sign of the feedback signal from the standalone detector is opposite to that from the feedback signal from the built–on Pulsecoder.
SV0449	SV IPM ALARM	The IPM (Intelligent Power Module) detected an alarm.
SV0453	SPC SOFT DISCONNECT ALARM	Software disconnection alarm of the built-in Pulsecoder. Turn off the power to the CNC, then remove and insert the Pulsecoder cable. If this alarm is issued again, replace the Pulsecoder.
SV0454	ILLEGAL ROTOR POS DETECT	The magnetic pole detection function terminated abnormally. The magnetic pole could not be detected because the motor did not run.
SV0456	ILLEGAL CURRENT LOOP	An attempt was made to set the current loop that could not be set. The amplifier pulse module in use does not comply with HIGH SPEED HRV. Or, requirements to control are not satisfied in the system.
SV0458	CURRENT LOOP ERROR	The specified current loop differs from the actual current loop.
SV0459	HI HRV SETTING ERROR	For two axes whose servo axis numbers (parameter No. 1023) are consecutively even and odd numbers, HIGH SPEED HRV control is possible for one axis and impossible for the other.
SV0460	FSSB DISCONNECT	The FSSB connection was discontinued. Probable causes are: (1) The FSSB connection cable was disconnected or broken. (2) The amplifier was turned off . (3) In the amplifier, the low-voltage alarm occurred.
SV0462	SEND CNC DATA FAILED	The correct data could not be received on a slave side because of the FSSB communication error.
SV0463	SEND SLAVE DATA FAILED	The correct data could not be received in the servo software because of the FSSB communication error.
SV0465	READ ID DATA FAILED	A read of the ID information for the amplifier has failed at power-on.

FRYER MACHINE SYSTEMS INC
Rob Zarenski

Number	Message	Description
SV0466	MOTOR/AMP. COMBINATION	The maximum current of an amplifier is different to that of a motor. Probable causes are: (1) The connection command for an amplifier is incorrect. (2) The parameter No.2165 setting is incorrect
SV0468	HI HRV SETTING ERROR(AMP)	An attempt was made to set up HIGH SPEED HRV control for use when the controlled axis of an amplifier for which HIGH SPEED HRV control could not be used.
SV0474	EXCESS ERROR(STOP:SV)	The servo detected that the positional deviation during a stop exceeded the setting (parameters Nos. 1839 and 1842) in the n-axis.
SV0475	EXCESS ERROR(MOVE:SV)	The servo detected that the positional deviation during a travel exceeded the setting (parameters Nos. 1838 and 1841) in the n-axis.
SV0476	ILLEGAL SPEED CMD.(SV)	The servo detected that the speed command exceeded the safety speed (parameters Nos. 13821 to 13824 (during position control) or parameters Nos. 13826 to 13829 (during speed control)) during safety monitoring (when safety monitoring request signal *VLDVx is “0”) in the n-axis. Keep the safety speed.
SV0477	ILLEGAL MACHINE POS.(SV)	The servo detected that the machine position fell outside the safety area (setting by parameters Nos. 13831 to 13838) during safety monitoring (when safety monitoring request signal *VLDVx is “0”) in the n-axis. Keep the safety area. A machine position check is performed only on the axis for which the reference position has been established. The axis for which the reference position is not established is not subjected to a machine position check.
SV0478	ILLEGAL AXIS DATA(SV)	The servo detected that an error occurred during transfer of axis data in the n-axis. When an alarm occurred because the configuration of the servo amplifier was changed, set the axis number for the servo amplifier (set bit 4 of parameter No. 2212 of the corresponding axis to 1 and then 0 and turn off the power of the entire system). When using a multiaxis amplifier, this operation may not clear the alarm. In this case, repeat this operation for the axes on which the alarm persists. If an alarm occurs due to a cause other than the above, replace the servo amplifier.
SV0481	SAFETY PARAM ERROR(SV)	The servo detected that a safety parameter error occurred in the n-axis.
SV0484	SAFETY FUNCTION ERROR(SV)	A safety function error related to the servo was detected in the n-axis. - The servo or CNC detected that the safety function was not executed in the servo. - The result of a servo safety function check did not match the result of a CNC safety function check. - An error occurred during a test of the CPU of the servo. - An error occurred during a check of RAM of the servo.
SV0488	SELF TEST OVER TIME	An MCC interruption test was not complete within the set period of time (parameter No. 1946). Check the MCC contact.
SV0489	SAFETY PARAM ERROR(CNC)	The CNC detected that a safety parameter error occurred in the n-axis.

FRYER MACHINE SYSTEMS INC
Robert Lanersky

Number	Message	Description
SV0490	SAFETY FUNCTION ERROR(CNC)	A CNC safety function error occurred in the n-axis. - The servo detected that the safety function was not executed in the CNC. - The result of a servo safety function check did not match the result of a CNC safety function check.
SV0494	ILLEGAL SPEED CMD.(CNC)	The CNC detected that the speed command exceeded the safety speed (parameters Nos. 13821 to 13824 (during position control) or parameters Nos. 13826 to 13829 (during speed control)) during safety monitoring (when safety monitoring request signal *VLDVx is “0”) in the n-axis. Keep the safety speed.
SV0495	ILLEGAL MACHINE POS.(CNC)	The CNC detected that the machine position fell outside the safety area (set range by parameters Nos. 13831 to 13838) during safety monitoring (when safety monitoring request signal *VLDVx is “0”) in the n-axis. Keep the safety area. A machine position check is performed only on the axis for which the reference position has been established.
SV0496	ILLEGAL AXIS DATA(CNC)	The CNC detected that an error occurred during transfer to axis data. When an alarm occurred because the configuration of the servo amplifier was changed, set the axis number for the servo amplifier (set bit 4 of parameter No. 2212 of the corresponding axis to 1 and 0 again and turn off the power of the entire system). When using a multiaxis amplifier, this operation may not clear the alarm. In this case, repeat this operation for the axes on which the alarm persists. If an alarm occurs due to a cause other than the above, replace the servo amplifier.
SV0498	AXIS NUMBER NOT SET(CNC)	The CNC detected that the axis number of the n-axis was not set for the servo amplifier. The axis number is set automatically, so turn off the power of the entire system.
SV0600	SV DC LINK OVER CURRENT	DC link overcurrent.
SV0601	SV EXTERNAL FAN FAILURE	Radiator cooling fan failure.
SV0602	SV RADIATOR OVERHEAT	The servo amplifier radiator has overheated.
SV0603	INV. IPM ALARM(OH)	The IPM (Intelligent Power Module) detected an overheat alarm.
SV0604	AMP COMMUNICATION ERROR	The communication between Servo Amplifier (SV) and Common Power Supply (PS) is in error.
SV0605	PS EXCESS-REGENERATION1	The motor regenerative power is too much.
SV0606	PS EXTERNAL FAN FAILURE	External radiator cooling fan failure.
SV0607	PS IMPROPER INPUT POWER	An abnormality was found with the input power supply.
SV0642	PRESSURE FEEDBACK VALUE EXCEED THE MAXIMUM	Pressure feedback is over the maximum.
SV0643	PRESSURE COMMAND IS SMALLER THAN THE MINIMUM	Pressure command is under the minimum.
SV0644	INITIAL PRESSURE IS NOT PROPER	Initial pressure is not proper.
SV0645	EXCESS ERROR (CHANGING INTO POSITION CONTROL MODE)	Servo positioning deviation is over the maximum when the control is changed to position control from pressure control.
SV0646	ABNORMAL ANALOG SIGNAL(EXT)	An error occurred in the analog 1Vp-p output of the separate detector. The separate detector, cable, or separate detector interface unit may be failed.
SV0652	TEMP.ERROR	Communication between the separate detector interface unit and temperature sensor was disconnected.
SV0654	DB RELAY FAILURE	A failure occurs in the dynamic brake relay of the servo amplifier. Replace the amplifier.

FRYER MACHINE SYSTEMS INC Rob Zahensky

Number	Message	Description
SV0659	ILLEGAL SETTING OF SSM	The setting for SSM is invalid. The followings are assumed as a cause. (1) The servo amplifier does not support SSM. (2) The setting of parameters related to SSM is invalid.
SV1025	V_READY ON (INITIALIZING)	The ready signal (VRDY) of the velocity control which should be OFF is ON while the servo control is ON.
SV1026	ILLEGAL AXIS ARRANGE	The parameter for servo axis arrange is not set correctly. - Parameter No. 1023 (servo axis number of each axis) is set to a negative value or a duplicate value. - The settings for parameter No. 1023 (servo axis number of each axis) were made with a certain setting skipped among 1 to 6, 9 to 14, or 17 to 22. - A setting of a multiple of 8 or a multiple of 8 minus 1 was made.
SV1051	ILLEGAL SYNCHRONOUS AXIS	In axis synchronous control, the parameter setting for the axis for which distance coded linear scale I/F is used, is incorrect. Set the following parameters to the same values for the master axis and slave axis. - Parameter No. 1821 (mark-1 interval) - Parameter No. 1882 (mark-2 interval) - Bit 2 (DC2x) and Bit 1 (DC4x) of parameter No.1802 (the number of marks)
SV1055	ILLEGAL TANDEM AXIS	In tandem control, the setting of the parameter No. 1023 is incorrect. In tandem control, the setting of Bit 6 (TDMx) of parameter No.1817 is incorrect.
SV1067	FSSB:CONFIGURATION ERROR(SOFT)	An FSSB configuration error occurred (detected by software). The connected amplifier type is incompatible with the FSSB setting value.
SV1068	DUAL CHECK SAFETY ALARM	An alarm that turns off the MCC of the entire system occurred in the Dual Check Safety function.
SV1069	EXCESS ERROR(SERVO OFF:CNC)	The CNC detected that the positional deviation during servo-off exceeded the set value (parameter No. 1840) in the n-axis.
SV1070	EXCESS ERROR(SERVO OFF:SV)	The servo detected that the positional deviation during servo-off exceeded the set value (parameter No. 1840) in the n-axis.
SV1071	EXCESS ERROR(MOVE:CNC)	The CNC detected that the positional deviation during a travel exceeded the set value (parameters Nos. 1838 and 1841) in the n-axis.
SV1072	EXCESS ERROR(STOP:CNC)	The CNC detected that the positional deviation during a stop exceeded the set value (parameters Nos. 1839 and 1842) in the n-axis.
SV1100	S-COMP. VALUE OVERFLOW	The amount of compensation for the straightness exceeded a maximum value of 32767.
SV5134	FSSB:OPEN READY TIME OUT	In the initialization, the FSSB could not be in an open ready sate. The axis card is thought to be defective.
SV5136	FSSB:NUMBER OF AMP. IS INSUFFICIENT	The number of amplifier identified by the FSSB is insufficient than the number of control axes. Or, the setting of the number of axes or the amplifier connection is in error.
SV5137	FSSB:CONFIGURATION ERROR	An FSSB configuration error occurred. The connecting amplifier type is incompatible with the FSSB setting value.

FRYER MACHINE SYSTEMS INC
Rob Zabenisky

Number	Message	Description
SV5139	FSSB:ERROR	Servo initialization has not completed successfully. It is probable that an optical cable failed or a connection between the amplifier and another module failed.
SV5197	FSSB:OPEN TIME OUT	The FSSB could not be opened although the CNC permitted the opening of the FSSB. Check the connection between the CNC and the amplifier.
SV5311	FSSB:ILLEGAL CONNECTION	Different current loops (HRV) are set for FSSB lines. Specify the same current loop for the FSSB lines.

(6) Overtravel alarms (OT alarm)

Number	Message	Description
OT0500	+ OVERTRAVEL (SOFT 1)	Exceeded the positive side stored stroke check 1.
OT0501	- OVERTRAVEL (SOFT 1)	Exceeded the negative side stored stroke check 1.
OT0502	+ OVERTRAVEL (SOFT 2)	Exceeded the positive side stored stroke check 2. Or, in the chuck tail stock barrier, an entry to the inhibited area was made during movement in the positive direction.
OT0503	- OVERTRAVEL (SOFT 2)	Exceeded the negative side stored stroke check 2. Or, in the chuck tail stock barrier, an entry to the inhibited area was made during movement in the negative direction.
OT0504	+ OVERTRAVEL (SOFT 3)	Exceeded the positive side stored stroke check 3.
OT0505	- OVERTRAVEL (SOFT 3)	Exceeded the - side stored stroke check 3.
OT0506	+ OVERTRAVEL (HARD)	The stroke limit switch in the positive direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.
OT0507	- OVERTRAVEL (HARD)	The stroke limit switch in the negative direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.
OT0508	INTERFERENCE:+	A tool moving in the positive direction along the n axis has fouled another tool post.
OT0509	INTERFERENCE:-	A tool moving in the negative direction along the n axis has fouled another tool post.
OT0510	+ OVERTRAVEL (PRE-CHECK)	The block end point or the tool path between blocks was found in the + side stroke limit forbidden area during Stroke limit check before move. Modify the program.
OT0511	- OVERTRAVEL (PRE-CHECK)	The block end point or the tool path between blocks was found in the - side stroke limit forbidden area during Stroke limit check before move. Modify the program.
OT0514	(n) INTERFERENCE:+	An interference occurred on the plus side of the n-axis in an Interference check for rotary area.
OT0515	(n) INTERFERENCE:-	An interference occurred on the minus side of the n-axis in an Interference check for rotary area.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	Description
OT0518	(n)+: INTERFERE ([Target name 1],[Target name 2])	A check made for interference between [Target name1] and [Target name2] detected interference on the + side of axis n. n represents the name of the axis with which an alarm is issued.
OT0519	(n)-: INTERFERE ([Target name 1],[Target name 2])	A check made for interference between [Target name1] and [Target name2] detected interference on the - side of axis n. n represents the name of the axis with which an alarm is issued.
OT1710	ILLEGAL ACC. PARAMETER(OPTIMUM TORQUE ACC/DEC)	The permissible acceleration parameter for the optimum torque acceleration/deceleration is in error. A possible cause is either of the following: (1) The ratio of a negative acceleration to a positive acceleration is not more than the limit value. (2) The time to reduce to a velocity of 0 exceeded the maximum time.

(7) Memory file alarms (IO alarm)

Number	Message	Description
IO1001	FILE ACCESS ERROR	The resident–type file system could not be accessed as an error occurred in the resident–type file system.
IO1002	FILE SYSTEM ERROR	The file could not be accessed as an error occurred in the CNC file system.
IO1030	CHECK SUM ERROR	The checksum of the CNC part program storage memory is incorrect.
IO1032	MEMORY ACCESS OVER RANGE	Accessing of data occurred outside the CNC part program storage memory range.
IO1034	PROGRAM FOLDER FILE IS BROKEN	Abnormality of data was detected in the program folder file. It is necessary to initialize the program file to recover. Please refer to the paragraph of the maintenance manual "IPL monitor" for the initialization operation of the program file.
IO1035	PROGRAM MANAGEMENT FILE IS BROKEN	Abnormality of data was detected in the program management file. It is necessary to initialize the program file to recover. Please refer to the paragraph of the maintenance manual "IPL monitor" for the initialization operation of the program file.
IO1104	OVER MAXIMUM TOOL LIFE PAIRS	The maximum number of tool life management pairs is exceeded. Modify the setting of the maximum number of tool life management pairs in parameter No. 6813.

(8) Alarms requiring power to be turned off (PW alarm)

Number	Message	Description
PW0000	POWER MUST BE OFF	A parameter was set for which the power must be turned OFF then ON again.
PW0001	X-ADDRESS(*DEC) IS NOT ASSIGNED.	The X address of the PMC could not be assigned correctly. This alarm may occur in the following case: - During the setting of parameter No. 3013, the X address could not be assigned correctly for the deceleration dog (*DEC) for a return to the reference position. When there are four or more paths or there are nine or more axes for one path, the signals must be assigned by setting bit 2 (XSG) of parameter No. 3008, parameter No. 3013, and parameter No. 3014.

FIXER MACHINE SYSTEMS INC
Rob Zarnansky

Number	Message	Description
PW0002	PMC address is not correct(AXIS).	The address to assign the axis signal is incorrect. This alarm may occur in the following case: - The parameter No.3021 setting is incorrect.
PW0003	PMC address is not correct(SPINDLE).	The address to assign the spindle signal is incorrect. This alarm may occur in the following case: - The parameter No.3022 setting is incorrect.
PW0004	SETTING THE LOADER SYSTEM PATH IS NOT CORRECT.	The loader system could not be assigned correctly. The parameter No. 0984 setting is incorrect. - The number of loader systems and the number of systems specified to the loader system in the bit 0 (LCP) of parameter No. 0984 does not match. - The bit 0 (LCP) of parameter No. 0984 of the path 1 is set to 1.
PW0006	POWER MUST BE OFF (ILL-EXEC-CHK)	The malfunction prevention function detected an alarm to require the power off.
PW0007	X-ADDRESS(SKIP) IS NOT ASSIGNED	The X address of PMC could not be assigned correctly. Possible causes are: - During the set of parameter No. 3012, the skip signal of the X address was not assigned correctly. - During the set of parameter No. 3019, the address other than the skip signal of the X address was not assigned correctly. When there are four or more paths, the signals must be assigned by bit 2 (XSG) of parameter No. 3008, parameter No. 3012, and parameter No. 3019.
PW0008	CPU SELF TEST ERROR(DCS PMC)	On the DCS PMC side: - An error was detected by the CPU self diagnosis function. - An error was detected by the RAM check function.
PW0009	CPU SELF TEST ERROR(PMC)	On the PMC side: - An error was detected by the CPU self diagnosis function. - An error was detected by the RAM check function.
PW0010	SAFE I/O CROSS CHECK ERROR(DCS PMC)	On the DCS PMC side, an error was detected by the I/O cross check function in system-defined safety-related DI/DO.
PW0011	SAFE I/O CROSS CHECK ERROR(PMC)	On the PMC side, an error was detected by the I/O cross check function in system-defined safety-related DI/DO.
PW0012	USER I/O CROSS CHECK ERROR(DCS PMC)	On the DCS PMC side, an error was detected by the I/O cross check function in user-defined safety-related DI/DO.
PW0013	USER I/O CROSS CHECK ERROR(PMC)	On the PMC side, an error was detected by the I/O cross check function in user-defined safety-related DI/DO.
PW0014	CPU TEST ALARM (CNC)	An error occurred in a test of the CPU of the CNC.
PW0015	SAFETY PARAM ERROR	The CNC detected that an error occurred in a safety parameter for other than servo axes or spindle axes.
PW0016	RAM CHECK ERROR	An error was detected in a RAM check of the CNC.
PW0017	INEXECUTION OF SAFETY FUNCTIONS	The safety function was not executed normally in the CNC.
PW0018	CRC CHECK ERROR	An error was detected in a CRC check of the CNC.
PW0020	HSP PARAMETER ERROR(MPATH)	The total value specified for all paths in parameter No. 11604 exceeds 600 (1000).

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
PW0036	ILLEGAL SETTING FOR SERVO MOTOR SPINDLE	The parameter setting for the servo axes for spindle use of the spindle control with servo motor is illegal. - The servo axes number for spindle use has exceeded 4 axes. - Servo axes number for spindle use are fewer than the axes number of designation of servo axes for spindle use. - The servo axes for spindle use is not set as a spindle control with servo motor axis. - The servo axes for spindle use is not set as a rotation axis.. - The servo axis number (parameter No.1023) is a negative value.
PW0037	SV/SP COMBINATION ERROR	(1) The servo axis might be a setting of the dummy axis. Check to see the parameter No.1023, bit 0 (DMYx) of parameter No.2009, and bit 4 (KSVx) of parameter No.11802. (2) The index number of spindle axis that synchronizes servo axis might be illegal. Check to see the bit 0 (A/Ss) of parameter No.3716, parameter No.3717, and No.24204. (3) FSSB high-speed rigid tapping cannot be used with the rigid tapping by the analog spindle or the spindle control with servo motor.
PW0041	DATA TABLE(D) SETTING IS ILLEGAL	The address of data table (D) PMC calculated from parameter (No.1313), stored stroke limit range switching data selection signal (OTD0-OTD15), and the switching data area is illegal. (1) The specified first address is illegal. (2) It is not a multiple of four. (3) The range of the specified data table is illegal.
PW0050	POWER MUST BE OFF (INITIALIZED COMMUNICATION)	When the power is turned on, the hardware of the communication function was initialized. The power must be turned off to be effective. This alarm may occur in the following cases: - When the option board is added or replaced - When the software option is changed
PW0053	SP-MODE CHANGE SETTING ERROR	The setting of parameters Nos.25800 to 25806 is illegal. Causes of the alarm are as follows. - The setting value is out of range. - The same value is set when spindle selection by address P is not used. - The same value is not set to all spindles when spindle selection by address P is used. - The value except for 0 is set when spindle orientation option or Cs contour control option is not selected. - One of M code of spindle orientation ON/OFF is set. - One of M code of Cs contour control ON/OFF is set.
PW0060	PS CONTROL AXIS ERROR 3	Amplifier group number duplication. Set parameter APS (No.11549#0) to 1 and execute automatic setting.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	Description
PW1102	ILLEGAL PARAMETER (I-COMP.)	The parameter for setting slope compensation is incorrect. This alarm occurs in the following cases: - When the number of pitch error compensation points on the axis on which slope compensation is executed exceeds 128 between the most negative side and most positive side - When the size relationship between the slope compensation point Nos. is incorrect - When the slope compensation point is not located between the most negative side and most positive side of pitch error compensation - When the compensation per compensation point is too small or too large.
PW1103	ILLEGAL PARAMETER (S-COMP.128)	The parameter for setting 128 straightness compensation points or the parameter compensation data is incorrect,
PW1104	ILLEGAL PARAMETER (3-D MACHINE POSITION COMPENSATION.)	A parameter for setting 3-dimensional machine position compensation is incorrect.
PW1105	ILLEGAL PARAMETER (3D_3DR-COMP)	The compensation point number of 3-dimensional error compensation exceeds 15625 points. Parameter setting for three-dimensional rotary error compensation is wrong. One of the following may be the cause. - The designation for rotary axes is wrong. - The number of the total compensation points is larger than 7812. - The designation for machine type is wrong. - The designation for compensation axis is wrong. - The setting for compensation number of reference point is wrong. - The setting for compensation interval is wrong. - The parameter is set in more than two path systems.
PW5046	ILLEGAL PARAMETER (S-COMP.)	The setting of a parameter related to straightness compensation is invalid. Possible causes are: - A nonexistent axis number is set in a moving or compensation axis parameter. - The straightness compensation point numbers do not have correct magnitude relationships. - No straightness compensation point is found between the furthest pitch error compensation point in the negative region and that in the positive region. - The compensation per compensation point is either too large or too small.
PW5390	R-ADDRESS SETTING IS ILLEGAL	The start address or the set range of the PMC R addresses set by parameters Nos.13541 and 13542 is invalid.

(9) Spindle alarms (SP alarm)

Number	Message	Description
SP0740	RIGID TAP ALARM : EXCESS ERROR	The positional deviation of the stopped spindle has exceeded the set value of parameter No.5313 during rigid tapping.
SP0741	RIGID TAP ALARM : EXCESS ERROR	(1) The positional deviation of the moving spindle has exceeded the set value of parameter No.5311 during rigid tapping. (2) Rigid tapping synchronization error has exceeded the set value of parameter No.5214.

FRYER MACHINE SYSTEMS INC
Rob Zaherski

Number	Message	Description
SP0742	RIGID TAP ALARM : LSI OVERFLOW	An LSI overflow has occurred for the spindle during rigid tapping.
SP0752	SPINDLE MODE CHANGE ERROR	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contour control, spindle positioning, rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.
SP0754	ABNORMAL TORQUE	An abnormal load was detected in a spindle motor. The alarm can be canceled by RESET.
SP0755	SAFETY FUNCTION ERROR	The CNC CPU detected that the safely function of the n-th spindle was not executed. Alternatively, the result of a CNC safety function check did not match the result of a spindle safety function check.
SP0756	ILLEGAL AXIS DATA	The CNC CPU detected an error during transfer of spindle data in the nth spindle. When this alarm is issued because the configuration of the spindle amplifiers was changed, set the spindle number for the spindle amplifier (set bit 7 of parameter No. 4541 to 1 and then 0, and turn the power to the entire system off).
SP0757	SAFETY SPEED OVER	The CNC CPU detected that during safety monitoring (when safety monitoring request signal *VLDPs is 0), the spindle motor speed was greater than the safety speed (parameter No. 4372, 4438, 4440, or 4442) on the n-th spindle. Operate within the safety speed.
SP1202	SPINDLE SELECT ERROR	In a multi spindle control, the spindle number other than the valid spindle number was selected by a position coder select signal. An attempt was made to select the spindle number of the system having no valid spindle.
SP1210	TOOL CHANGE SP MOTION OVERFLOW	The amount of distribution to a spindle is too much. (specific to the FANUC ROBODRILL)
SP1211	TOOL CHANGE SP ORTN EXCESS ERROR	During a tool change, a too much orientation error was detected for the spindle. (specific to the FANUC ROBODRILL)
SP1212	TOOL CHANGE SP MOVE EXCESS ERROR	During a tool change, a too much moving error was detected for the spindle. (specific to the FANUC ROBODRILL)
SP1213	TOOL CHANGE SP STOP EXCESS ERROR	During a tool change, a too much stop error was detected for the spindle. (specific to the FANUC ROBODRILL)
SP1214	TOOL CHANGE SP ILLEGAL SEQUENCE	During changing tools, an abnormal spindle sequence was detected. (specific to the FANUC ROBODRILL)
SP1220	NO SPINDLE AMP.	Either the cable connected to a serial spindle amplifier is broken, or the serial spindle amplifier is not connected.
SP1221	ILLEGAL MOTOR NUMBER	The spindle No. and the motor No. are incorrectly matched.
SP1224	ILLEGAL SPINDLE-POSITION CODER GEAR RATIO	The spindle–position coder gear ratio was incorrect.
SP1227	RECEIVING ERROR (SERIAL SPINDLE)	A receive error occurred in communications between the CNC and the serial spindle amplifier.
SP1228	COMMUNICATION ERROR (SERIAL SPINDLE)	A communications error occurred between the CNC and the serial spindle amplifier.
SP1229	COMMUNICATION ERROR SERIAL SPINDLE AMP.	A communications error occurred between serial spindle amplifiers (motor Nos. 1 and 2, or motor Nos. 3–4).
SP1231	SPINDLE EXCESS ERROR (MOVING)	The position deviation during spindle rotation was greater than the value set in parameters.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
SP1232	SPINDLE EXCESS ERROR (STOP)	The position deviation during spindle stop was greater than the value set in parameters.
SP1233	POSITION CODER OVERFLOW	The error counter/speed instruction value of the position coder overflowed.
SP1234	GRID SHIFT OVERFLOW	Grid shift overflowed.
SP1240	DISCONNECT POSITION CODER	The analog spindle position coder is broken.
SP1241	D/A CONVERTER ERROR	The D/A converter for controlling analog spindles is erroneous.
SP1243	ILLEGAL SPINDLE PARAMETER SETTING(GAIN)	The setting for the spindle position gain is incorrect.
SP1244	MOTION VALUE OVERFLOW	The amount of distribution to a spindle is too much
SP1245	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC.
SP1246	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC.
SP1247	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC.
SP1252	ILLEGAL SPINDLE PARAMETER SETTING (TANDEM)	Setting of parameter No.4597 is invalid. Possible causes are : - A value that is larger than the maximum number of controlled spindle axes is set in parameter No.4597. - A negative value except for –1 is set in parameter No.4597. - Relationship of master axis and slave axis that is set in parameter No.4597 is illegal. - When the spindle amplifier that is not applicable the function is used, a value except for 0 is set in parameter No.4597.
SP1255	CAN NOT CHANGE SPINDLE MODE	In the analog spindle control, rigid tapping and spindle positioning can not be used except when the number of pulses output from the position coder of the spindle is 4096pulse/rev.
SP1256	SPINDLE PHASE-SYNC IMPOSSIBLE	Arbitrary spindle position phase synchronous control cannot be executed. Save the spindle position.
SP1257	ILLEGAL PARAMETER (No.3791#0)	Bit 0(SSE) of parameter No.3791 is illegal setting. Causes of the alarm are as follows. - The setting of parameter SSE is different because of the master and slave. Please change the following function to the same setting. - Spindle synchronous control - Spindle command synchronous control - Polygon turning with two spindles - The parameter SSE is set to 1 the spindle control software that does not support this function. Please set 0 to parameter SSE.
SP1258	ILLEGAL SPINDLE PARAMETER SETTING (SOFT)	The parameter is an illegal setting. Please check as follows. - Bit 0 (ORPUNT) of parameter No.4542 is set to 1 the spindle control software that does not support this function. Please set 0 to the bit 0 (ORPUNT) of parameter No.4542.
SP1700	SAFETY PARAM ERROR	The CNC detected that a safety parameter error occurred in the n-th spindle.
SP1969	SPINDLE CONTROL ERROR	An error occurred in the spindle control on the CNC. Report the conditions (the system configuration, the operation, the frequency of error occurrence and so on) to FANUC.

FRYER MACHINE SYSTEMS INC
Rob Ashlansky

Number	Message	Description
SP1970	SPINDLE CONTROL ERROR	An initialization of a spindle didn't complete in the spindle control on the CNC. Check a state of the connection between CNC and the spindle amplifier. If this alarm still occurs even after the check of the connection, report the conditions (the system configuration, the operation, the frequency of error occurrence and so on) to FANUC.
SP1971	SPINDLE CONTROL ERROR	An error occurred in the spindle control on the CNC. Report the conditions (the system configuration, the operation, the frequency of error occurrence and so on) to FANUC.
SP1972	SPINDLE CONTROL ERROR	An error occurred in the spindle control on the CNC. M Confirm whether or not it isn't selecting the 4th gear in the rigid tapping. If this alarm occurred in other case, report the conditions (the system configuration, the operation, the frequency of error occurrence and so on) to FANUC. T Report the conditions (the system configuration, the operation, the frequency of error occurrence and so on) to FANUC.
SP1975	ANALOG SPINDLE CONTROL ERROR	An position coder error was detected on the analog spindle.
SP1976	SERIAL SPINDLE COMMUNICATION ERROR	The amplifier No. could not be set to the serial spindle amplifier.
SP1977	SERIAL SPINDLE COMMUNICATION ERROR	An error occurred in the spindle control software. The trouble of the hardware for the serial spindle control or influences of the noise might be the cause.
SP1978	SERIAL SPINDLE COMMUNICATION ERROR	A time–out was detected during communications with the serial spindle amplifier.
SP1979	SERIAL SPINDLE COMMUNICATION ERROR	The communications sequence was no longer correct during communications with the serial spindle amplifier.
SP1980	SERIAL SPINDLE AMP. ERROR	Defective SIC–LSI on serial spindle amplifier
SP1981	SERIAL SPINDLE AMP. ERROR	An error occurred during reading of the data from SIC–LSI on the serial spindle amplifier side.
SP1982	SERIAL SPINDLE AMP. ERROR	An error occurred during reading of the data from SIC–LSI on the serial spindle amplifier side.
SP1984	SERIAL SPINDLE AMP. ERROR	An error occurred during the re-initialization of the serial spindle amplifier. The trouble of the hardware for the serial spindle control or influences of the noise might be the cause.
SP1985	SERIAL SPINDLE CONTROL ERROR	Failed to automatically set parameters
SP1986	SERIAL SPINDLE CONTROL ERROR	An error occurred during the reading of the spindle parameters. The trouble of the hardware for the serial spindle control or influences of the noise might be the cause.
SP1987	SERIAL SPINDLE CONTROL ERROR	Defective SIC–LSI on the CNC
SP1988	SPINDLE CONTROL ERROR	An error occurred in the spindle control on the CNC. Report the conditions (the system configuration, the operation, the frequency of error occurrence and so on) to FANUC.
SP1989	SPINDLE CONTROL ERROR	An error occurred during communications with the serial spindle amplifier. The trouble of the hardware for the serial spindle control or influences of the noise might be the cause.
SP1996	ILLEGAL SPINDLE PARAMETER SETTING	The spindle was assigned incorrectly. Check to see the following parameter. (No.3716 or 3717)

FRYER MACHINE SYSTEMS INC
Rob Zaremski

Number	Message	Description
SP1999	SPINDLE CONTROL ERROR	An error occurred in the spindle control on the CNC. Report the conditions (the system configuration, the operation, the frequency of error occurrence and so on) to FANUC.

(10) Alarm list (serial spindle)

When a serial spindle alarm occurs, the following number is displayed on the CNC.

NOTE

*1 Note that the meanings of the Spindle Amplifier (SP) indications differ depending on which LED, the red or yellow LED, is on. When the red LED is on, the Spindle Amplifier (SP) indicates a 2-digit alarm number. When the yellow LED is on, the Spindle Amplifier (SP) indicates an error number that designates a sequence problem (for example, when a rotation command is entered with the emergency stop state not released).
See "Error Codes (Serial Spindle)."

*2 For information about serial spindle alarms for numbers that are not listed below, refer to the following documents according to the actual spindle motor to be connected.
- FANUC AC SPINDLE MOTOR αi series MAINTENANCE MANUAL (B-65285EN)
- Technical reports etc.

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9001	MOTOR OVERHEAT	01	1 Check and correct the peripheral temperature and load status. 2 If the cooling fan stops, replace it.	The internal temperature of the motor exceeds the specified level. The motor is used in excess of the continuous rating, or the cooling component is abnormal.
SP9002	EX DEVIATION SPEED	02	1 Check and correct the cutting conditions to decrease the load. 2 Correct parameter No. 4082.	The motor speed cannot follow a specified speed. An excessive motor load torque is detected. The acceleration/deceleration time in parameter No. 4082 is insufficient.
SP9003	DC-LINK FUSE IS BROKEN	03	1 Replace the Spindle Amplifier (SP). 2 Check the motor insulation status.	The Power Supply becomes ready (00 is indicated), but the DC link voltage is too low in the Spindle Amplifier (SP). The fuse in the DC link section in SP is blown. (The power device is damaged or the motor is ground-fault.)
SP9004	PS IMPROPER INPUT POWER		Check the state of the input power supply to the Power Supply (PS).	The power supply (PS) detected a power supply failure. (Power Supply alarm 14)
SP9006	THERMAL SENSOR DISCONNECT	06	1 Check and correct the parameter. 2 Replace the feedback cable.	The temperature sensor of the motor is disconnected.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9007	OVER SPEED	07	Check for a sequence error. (For example, check whether spindle synchronization was specified when the spindle could not be turned.)	The motor speed has exceeded 115% of its rated speed. When the spindle axis was in position control mode, position error were accumulated excessively (SFR and SRV were turned off during spindle synchronization.)
SP9009	OVERHEAT MAIN CIRCUIT	09	1 Improve the heat sink cooling status. 2 If the heat sink cooling fan stops, replace the Spindle Amplifier (SP).	The temperature in the power semiconductor cooling radiator is abnormally high.
SP9010	LOW VOLT INPUT POWER	10	1 Replace the cables. 2 Replace the SP control printed circuit board.	A drop in the input power voltage in the Spindle Amplifier (SP) is detected.
SP9011	PS OVER VOLT. DC LINK		1 Check the selected Power Supply (PS). 2 Check the input power voltage and change in power during motor deceleration. If the voltage exceeds 253 VAC (for the 200-V system) or 530 VAC (for the 400-V system), improve the power supply impedance.	Overvoltage of the DC link section of the Power Supply (PS) was detected. (Power Supply alarm indication: 07) PS selection error. (The maximum output specification of the PS is exceeded.)
SP9012	OVERCURRENT POWER CIRCUIT	12	1 Check the motor insulation status. 2 Check the spindle parameters. 3 Replace the Spindle Amplifier (SP).	The motor current is abnormally high. A motor-specific parameter does not match the motor model. Poor motor insulation
SP9013	CPU DATA MEMORY FAULT	13	Replace the Spindle Amplifier (SP) control printed circuit board.	Abnormality in a Spindle Amplifier (SP) control circuit component is detected. (RAM within the Spindle Amplifier (SP) is abnormal.)
SP9014	SOFTWARE SERIES MISMATCH	14	Replace the Spindle Amplifier (SP).	The mismatch of the spindle software and a Spindle Amplifier (SP) is detected.
SP9015	SPINDLE SWITCHING FAULT	15	1 Check and correct the ladder sequence. 2 Replace the switching MCC.	The switch sequence in spindle switching/speed range switching operation is abnormal. The switching MCC contact status check signal and command do not match.
SP9016	RAM ERROR	16	Replace the Spindle Amplifier (SP) control printed circuit board.	Abnormality in a Spindle Amplifier (SP) control circuit component is detected. (RAM for external data is abnormal.)
SP9017	ID NUMBER PARITY ERROR	17	Replace the Spindle Amplifier (SP)	Abnormality in Spindle Amplifier (SP) ID data is detected.

FRYER MACHINE SYSTEMS INC
Rob Zaremski

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9018	SUMCHECK ERROR PROGRAM ROM	18	Replace the Spindle Amplifier (SP) control printed circuit board.	Abnormality in a Spindle Amplifier (SP) control circuit component is detected. (Program ROM data is abnormal.)
SP9019	EXCESS OFFSET CURRENT U	19	Replace the Spindle Amplifier (SP).	Abnormality in a Spindle Amplifier (SP) component is detected. (The initial value for the U phase current detection circuit is abnormal.)
SP9020	EXCESS OFFSET CURRENT V	20	Replace the Spindle Amplifier (SP).	Abnormality in a Spindle Amplifier (SP) component is detected. (The initial value of the V phase current detection circuit is abnormal.)
SP9021	POS SENSOR POLARITY ERROR	21	Check and correct the parameters. (parameters Nos. 4000#0, 4001#4)	The polarity parameter setting of the position sensor is wrong.
SP9022	SP AMP OVER CURRENT	22	1 Review operation conditions (acceleration/ deceleration and cutting) to reduce the load. 2 Check and correct the parameters.	A Spindle Amplifier (SP) overload current was detected.
SP9024	SERIAL TRANSFER ERROR	24	1 Place the CNC-to-spindle cable away from the power cable. 2 Replace the cable.	The CNC power is turned off (normal power-off or broken cable). An error is detected in communication data transferred to the CNC.
SP9027	DISCONNECT POSITION CODER	27	Replace the cable.	The spindle position coder (connector JYA3) signal is abnormal.
SP9029	OVERLOAD	29	Check and correct the load status.	Excessive load has been applied continuously for a certain period of time. (This alarm is issued also when the motor shaft has been locked in the excitation state.)
SP9030	OVERCURRENT		Check and correct the power supply voltage.	Overcurrent is detected in Power Supply (PS) main circuit input. (Power Supply alarm indication: 01) Unbalanced power supply. PS selection error (The maximum PS output specification is exceeded.)
SP9031	MOTOR LOCK	31	1 Check and correct the load status. 2 Replace the motor sensor cable (connector JYA2).	The motor cannot rotate at a specified speed. (A level not exceeding the SST level for the rotation command has existed continuously.)
SP9032	SIC-LSI RAM FAULT	32	Replace the Spindle Amplifier (SP) control printed circuit board.	Abnormality in a Spindle Amplifier (SP) control circuit component is detected. (The LSI device for serial transfer is abnormal.)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9033	PS PRE-CHARGE FAILURE		1 Check and correct the power supply voltage. 2 Replace the Power Supply (PS).	Charging of direct current power supply voltage in the power circuit section is insufficient when the magnetic contractor is turned on (such as open phase and defective charging resistor). (Power Supply alarm indication: 05)
SP9034	ILLEGAL PARAMETER	34	Correct a parameter value according to FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN). If the parameter number is unknown, connect the spindle check board, and check the indicated parameter.	Parameter data exceeding the allowable limit is set.
SP9036	OVERFLOW ERROR COUNTER	36	Check whether the position gain value is too large, and correct the value.	An error counter overflow occurred.
SP9037	ILLEGAL SETTING VELOCITY DETECTOR	37	Correct the parameter value according to FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN).	The setting of the parameter for the number of pulses in the speed detector is incorrect.
SP9041	ILLEGAL 1REV SIGN OF POSITION CODER	41	1 Check and correct the parameter. 2 Replace the cable.	1 The 1-rotation signal of the spindle position coder (connector JYA3) is abnormal. 2 Parameter setting error
SP9042	NO 1REV SIGN OF POSITION CODER	42	Replace the cable.	The 1-rotation signal of the spindle position coder (connector JYA3) is disconnected.
SP9043	DISCONNECT POSITION CODER DEF. SPEED	43	Replace the cable.	The position coder signal (connector JYA3S) for differential speed in the submodule SW is abnormal.
SP9046	ILLEGAL 1REV SIGN OF SCREW CUT	46	1 Check and correct the parameter. 2 Replace the cable. 3 Re-adjust the BZ sensor signal.	The 1-rotation signal in threading is abnormal.
SP9047	ILLEGAL SIGNAL OF POSITION CODER	47	1 Replace the cable. 2 Correct the cable layout (vicinity of the power line).	The A/B phase signal of the spindle position coder (connector JYA3) is abnormal. The relationship between the A/B phase and 1-rotation signal is incorrect (Pulse count mismatch).
SP9049	DEF. SPEED IS OVER VALUE	49	Check whether the calculated differential speed value exceeds the maximum motor speed.	In differential speed mode, the speed of the other spindle converted to the speed of the local spindle has exceeded the allowable limit (the differential speed is calculated by multiplying the speed of the other spindle by the gear ratio).

FRYER MACHINERY SYSTEMS INC
Rob Zahenskiy

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9050	SYNCRONOUS VALUE IS OVER SPEED	50	Check whether the calculated value exceeds the maximum motor speed.	In spindle synchronization, the speed command calculation value exceeded the allowable limit (the motor speed is calculated by multiplying the specified spindle speed by the gear ratio).
SP9051	PS LOW VOLT. DC LINK		1 Check and correct the power supply voltage. 2 Replace the MC.	Input voltage drop was detected. (Power Supply alarm (PS) indication: 04) (Momentary power failure or poor MCC contact)
SP9052	ITP FAULT 1	52	1 Replace the Spindle Amplifier (SP) control printed circuit board. 2 Replace the main board or additional spindle board in the CNC.	An abnormality is detected in the interface between the CNC and spindle amplifier (the ITP signal stopped).
SP9053	ITP FAULT 2	53	1 Replace the Spindle Amplifier (SP) control printed circuit board. 2 Replace the main board or additional spindle board in the CNC.	An abnormality is detected in the interface between the CNC and spindle amplifier (the ITP signal stopped).
SP9054	OVERCURRENT	54	Review the load state.	An overload current was detected.
SP9055	ILLEGAL POWER LINE	55	1 Replace the magnetic contactor. 2 Check and correct the sequence.	The power line state signal of the magnetic contactor for spindle switching/speed range switching is abnormal.
SP9056	COOLING FAN FAILURE	56	Replace the internal cooling fan.	The internal cooling fan stopped.
SP9057	PS EXCESS-REGENERATION2		1 Decrease the acceleration/deceleration duty. 2 Check the cooling condition (peripheral temperature). 3 If the cooling fan stops, replace the resistor. 4 If the resistance is abnormal, replace the resistor.	An overload was detected in the regenerative resistance. (Power Supply alarm indication: 16) Thermostat operation or short-time overload was detected. The regenerative resistor was disconnected, or an abnormal resistance was detected.
SP9058	PS OVERLOAD		1 Check the Power Supply (PS) cooling status. 2 Replace the Power Supply (PS).	The temperature of the radiator of the Power Supply (PS) has increased abnormally. (Power Supply (PS) alarm indication: 03)
SP9059	PS INTERNAL FAN FAILURE		Replace the Power Supply (PS).	The internal cooling fan for the Power Supply (PS) stopped. (Power Supply alarm indication: 02)
SP9061	SEMI-FULL ERROR EXCESS	61	Check parameter settings.	The error between the semi-closed and full-closed sides when the dual position feedback function is used is too large.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9065	SERIAL SPINDLE ALARM	65	1 Check parameter settings. 2 Check sensor connections and signals. 3 Check power line connections.	The move distance is too long when the magnetic pole is confirmed. (synchronous spindle motor)
SP9066	COM. ERROR BETWEEN SP AMPS	66	1 Replace the cable. 2 Check and correct the connection.	An error was found in communication (connector JX4) between Spindle Amplifiers (SP).
SP9067	FSC/EGB COMMAND ERROR	67	Check the sequence (reference position return command).	In the spindle EGB mode, reference position return was specified.
SP9069	SAFETY SPEED OVER	69	1 Check the speed command. 2 Check parameter settings. 3 Check the sequence.	In the state in which safety speed monitoring was enabled, the system detected that the motor speed exceeded the safety speed or detected an error during a free-run stop.
SP9070	ILLEGAL AXIS DATA	70	1 When this alarm is issued because the configuration of the spindle amplifiers (SPs) was changed, set the spindle number for the spindle amplifier (bit 7 of parameter No. 4541 to 1 and then 0, and turn the power to the entire system off). 2 Replace the spindle amplifier.	An error was detected during transfer of spindle data of a spindle.
SP9071	SAFETY PARAMETER ERROR	71	1 Input the safety parameter again. 2 Replace the Spindle Amplifier (SP) control printed-circuit board.	An error was detected in an axis parameter check.
SP9072	MISMATCH RESULT OF MOTOR SPEED CHECK	72	1 Replace the Spindle Amplifier (SP) control printed-circuit board. 2 Replace the main board or additional spindle board in the CNC.	A mismatch was detected between the safety speed check results of the Spindle Amplifier (SP) and those of the CNC.
SP9073	MOTOR SENSOR DISCONNECTED	73	1 Replace the feedback cable. 2 Check the shield. 3 Check and correct the connection. 4 Adjust the sensor.	The motor sensor feedback signal is not present. (connector JYA2)
SP9074	CPU TEST ERROR	74	Replace the Spindle Amplifier (SP) control printed-circuit board.	An error was detected in a CPU test.
SP9075	CRC ERROR	75	Replace the Spindle Amplifier (SP) control printed circuit board.	An error was detected in a ROM CRC test.

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9076	INEXECUTION OF SAFETY FUNCTIONS	76	Replace the Spindle Amplifier (SP) control printed-circuit board.	The spindle amplifier (SP) detected that the safety function was not executed.
SP9077	MISMATCH RESULT OF AXIS NUMBER CHECK	77	1 Replace the Spindle Amplifier (SP) control printed-circuit board. 2 Replace the main board or additional spindle board in the CNC.	A mismatch was detected between the axis number check results of the Spindle Amplifier (SP) and those of the CNC.
SP9078	MISMATCH RESULT OF SAFETY PARAMETER CHECK	78	1 Replace the Spindle Amplifier (SP) control printed-circuit board. 2 Replace the main board or additional spindle board in the CNC.	The result of a safety parameter check by the spindle amplifier (SP) conflicts with that of a safety parameter check by the CNC.
SP9079	INITIAL TEST ERROR	79	Replace the Spindle Amplifier (SP) control printed-circuit board.	An error was detected in a initial test operation.
SP9080	ALARM AT THE OTHER SP AMP.	80	Remove the cause of the alarm of the remote Spindle Amplifier (SP).	During inter-Spindle Amplifier (SP) communication, an alarm was generated on the remote Spindle Amplifier (SP).
SP9081	1-ROT MOTOR SENSOR ERROR	81	1 Check and correct the parameter. 2 Replace the feedback cable. 3 Adjust the sensor.	The one-rotation signal of the motor sensor cannot be correctly detected. (connector JYA2)
SP9082	NO 1-ROT MOTOR SENSOR	82	1 Replace the feedback cable. 2 Adjust the sensor.	The one-rotation signal of the motor sensor is not generated. (connector JYA2)
SP9083	MOTOR SENSOR SIGNAL ERROR	83	1 Replace the feedback cable. 2 Adjust the sensor.	An irregularity was detected in a motor sensor feedback signal. (connector JYA2)
SP9084	SPNDL SENSOR DISCONNECTED	84	1 Replace the feedback cable. 2 Check the shield. 3 Check and correct the connection. 4 Check and correct the parameter. 5 Adjust the sensor.	The spindle sensor feedback signal is not present. (connector JYA4)
SP9085	1-ROT SPNDL SENSOR ERROR	85	1 Check and correct the parameter. 2 Replace the feedback cable. 3 Adjust the sensor.	The one-rotation signal of the spindle sensor cannot be correctly detected. (connector JYA4)
SP9086	NO 1-ROT SPNDL SENSOR	86	1 Replace the feedback cable. 2 Adjust the sensor.	The one-rotation signal of the spindle sensor is not generated. (connector JYA4)
SP9087	SPNDL SENSOR SIGNAL ERROR	87	1 Replace the feedback cable. 2 Adjust the sensor.	An irregularity was detected in a spindle sensor feedback signal. (connector JYA4)
SP9088	COOLING RADI FAN FAILURE	88	Replace the Spindle Amplifier radiator cooling fan.	The radiator cooling fan stopped.

FRYER MACHINE SYSTEMS INC
Robi Zahenski

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9089	SUB MODULE SM (SSM) ERROR	89	1 Check the connection between the Spindle Amplifier (SP) and the submodule SM (SSM). 2 Replace the submodule SM(SSM). 3 Replace the Spindle Amplifier (SP) control printed-circuit board.	Submodule SM (SSM) error (synchronous spindle motor)
SP9090	UNEXPECTED ROTATION	90	1 Check magnetic pole detection operation. 2 Check whether the rotor and sensor are aligned correctly.	Unexpected rotation of the synchronous spindle motor was detected.
SP9091	POLE POSITION COUNT MISS	91	Replace the motor sensor cable.	Count error of the magnetic pole position of the synchronous spindle motor
SP9092	OVER SPEED TO VELOCITY COMMAND	92	Check the sequence (whether SFR or SRV is turned on and off in the position control mode).	The motor speed exceeds the overspeed level corresponding to the velocity command.
SP9110	AMP COMMUNICATIO N ERROR	b0	1 Replace the communication cable between Spindle Amplifier (SP) and Power Supply (PS). 2 Replace the Spindle Amplifier (SP) or Power Supply (PS) control printed circuit board.	Communication error between Spindle Amplifier (SP) and Power Supply (PS)
SP9111	PS LOW VOLT. CONTROL		Replace the Power Supply (PS) control printed circuit board.	Low converter control power supply voltage (Power Supply indication : 06)
SP9112	PS EXCESS-REGENE RATION1		1 Check the regenerative resistance. 2 Check the motor selection. 3 Replace the Power Supply (PS).	Excessive converter regenerative power (Power Supply indication : 08)
SP9113	PS EXTERNAL FAN FAILURE		Replace the external radiator cooling fan for Power Supply (PS).	Stopped the external radiator cooling fan for Power Supply (PS) (Power Supply indication = 10)
SP9114	PS CONTROL AXIS ERROR 1	b4	Set parameter No. 4657 to 0. Or set parameter APS (No.11549#0) to 1 and execute automatic setting.	Invalid parameter
SP9115	PS CONTROL AXIS ERROR 2	b5	Set parameter No. 4657 to other than 0. Or set parameter APS (No.11549#0) to 1 and execute automatic setting.	Invalid parameter

FRYER MACHINE SYSTEMS INC
Rod Zabensky

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9120	COMMUNICATION DATA ERROR	C0	1 Replace the communication cable between CNC and Spindle Amplifier (SP). 2 Replace the Spindle Amplifier (SP) control printed circuit board. 3 Replace the main board or additional spindle board in the CNC.	Communication data alarm
SP9121	COMMUNICATION DATA ERROR	C1	1 Replace the communication cable between CNC and Spindle Amplifier (SP). 2 Replace the Spindle Amplifier (SP) control printed circuit board. 3 Replace the main board or additional spindle board in the CNC.	Communication data alarm
SP9122	COMMUNICATION DATA ERROR	C2	1 Replace the communication cable between CNC and Spindle Amplifier (SP). 2 Replace the Spindle Amplifier (SP) control printed circuit board. 3 Replace the main board or additional spindle board in the CNC.	Communication data alarm
SP9123	SPINDLE SWITCH CIRCUIT ERROR	C3	Replace the submodule SW(SSW).	Submodule SW (SSW) error (spindle switching)
SP9124	LEARNING CTRL ROTATION CMD ERROR	C4	1 Correct the velocity command. 2 Correct the setting of the learning period parameter (No. 4425).	A velocity that could not be used for learning control was specified in time-based learning control.
SP9125	LEARNING CTRL COMP ORDER ERROR	C5	Correct the settings of the parameters for the maximum and minimum orders of dynamic characteristic compensation elements for learning control (Nos. 4427 and 4428).	An order outside the allowable range was specified for dynamic characteristic compensation elements.
SP9127	LEARNING CTRL PERIOD ERROR	C7	Correct the setting of the parameter for the number of divisions per period for learning control (No. 4425).	In angle-based learning control, the reference angle period is invalid.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9128	SP SYNC VELOCITY ERROR EXCESS	C8	1 Check the sequence (whether SFR or SRV is turned off or on in position control). 2 Check and correct the load status. 3 Check the parameter setting.	In spindle synchronous control, velocity error exceeds the setting.
SP9129	SP SYNC POSITION ERROR EXCESS	C9	1 Check the sequence (whether SFR or SRV is turned off or on in position control). 2 Check and correct the load status. 3 Check the parameter setting.	In spindle synchronous control, position error exceeds the setting.
SP9130	TORQUE TANDEM POLARITY ERROR	d0	Check the setting of the parameter for the rotation direction (bit 2 of No. 4353).	In tandem control, the rotation directions of the master and slave motors are invalid.
SP9131	SPINDLE TUNING FUNCTION ALARM	d1	Check the message displayed by SERVO GUIDE. Set “0” to the parameter for spindle tuning function (No. 4402#7, #6, No.4125, No.4126, No.4410~4415) and input alarm reset signal (ARSTx).	Spindle tuning function alarm
SP9132	SER.SENSOR DATA ERROR	d2	1 Replace the feedback cable. 2 Replace the sensor. 3 Replace the Spindle Amplifier (SP).	Serial data error between a serial sensor and spindle amplifier (SP)
SP9133	SER.SENSOR TRANSFER ERROR	d3	1 Replace the feedback cable. 2 Replace the sensor. 3 Replace the Spindle Amplifier (SP).	Serial data error between a serial sensor and spindle amplifier (SP)
SP9134	SER.SENSOR SOFT PHASE ERROR	d4	1 Check and correct the sensor parameter setting. 2 Take action against noise. 3 Replace the sensor.	The change in serial sensor positional data is too large.
SP9135	SAFETY SPEED ZERO ERROR(SP)	d5	Perform operation within the safety speed zero range.	The motor position exceeded the safety speed zero monitoring width.
SP9136	MISMATCH RESULT OF SAFETY SPEED ZERO CHECK(SP)	d6	Replace the Spindle Amplifier (SP) control printed circuit board.	The Spindle Amplifier (SP) speed zero determination result did not match the CNC speed zero determination result.
SP9137	SP DEVICE COMMUNICATIO N ERROR	d7	Replace the Spindle Amplifier (SP) control printed circuit board.	Device communication error in the spindle amplifier (SP)

FRYER MACHINE SYSTEMS INC
Rob Zarnensky

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9138	CURRENT LIMIT SETTING ERROR	d8	Check the parameter.	The current limit level setting is out of the specified range.
SP9139	SER.SENSOR PULSE MISS	d9	Replace the sensor.	An error occurred in the serial sensor interpolation circuit.
SP9140	SER.SENSOR COUNT MISS	E0	1 Take action against noise. 2 Replace the sensor.	The number of feedback pulses per one-rotation signal of the serial sensor is outside the specified range.
SP9141	SER.SENSOR NO 1-ROT SIGNAL	E1	1 Check and correct the sensor parameter setting. 2 Replace the sensor.	The serial sensor is not placed in the status in which the one-rotation signal has been detected.
SP9142	SER.SENSOR ABNORMAL	E2	Replace the sensor.	A serial sensor error occurred.
SP9143	CS HIGH SPEED CHANGE CMD ERROR	E3	Check the sequence.	A command was issued though the one-rotation signal had not been detected.
SP9144	CURRENT DETECT CIRCUIT ERROR	E4	Replace the Spindle Amplifier (SP).	An error was found in the current detection circuit.
SP9145	LOW VOLTAGE DRIVER	E5	Replace the Spindle Amplifier (SP).	The driver circuit voltage has dropped.
SP9146	SP:INTERNAL OVERHEAT	E6	1 Check that the temperature in the power magnetics cabinet is within the specification range. 2 If this alarm is issued immediately after power-on, replace the spindle amplifier (SP).	The internal temperature of the spindle amplifier (SP) exceeds the specified value.
SP9147	SP:GROUND FAULT	E7	Replace the motor or amplifier.	A ground fault occurred in the motor or amplifier.
SP9148	AXIS NUMBER NOT SET	E8	Turn the power to the entire system off.	The spindle number of the spindle (SP) is not set.
SP9149	EXT.CURRENT FB U-OFFSET	E9	Replace the Spindle Amplifier (SP) control printed circuit board.	An error was detected in the part of spindle amplifier (SP).
SP9150	EXT.CURRENT FB V-OFFSET	F0	Replace the Spindle Amplifier (SP) control printed circuit board.	An error was detected in the part of spindle amplifier (SP).
SP9151	FILTER MODULE ERROR	F1	Replace the filter module.	A filter module error was detected.
SP9152	EXT.CUR.FB DISCONNECT	F2	Replace the cable (JYA4).	Disconnection of the external current sensor was detected.
SP9153	SP NO FAILURE	F3	Check the parameter.	No hardware error is determined by a self check.
SP9154	PHASE OPEN	F4	Replace the motor.	A motor error was detected.
SP9155	FAILURE OF SP (OPEN)	F5	Replace the Spindle Amplifier (SP).	An error was detected in the spindle amplifier (SP).
SP9156	FAILURE OF CURRENT CTRL.	F6	Replace the Spindle Amplifier (SP) control printed circuit board.	An error was detected in the part of spindle amplifier (SP).

FRYER MACHINE SYSTEMS INC
Robert Zahensky

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9157	FAILURE OF SP (SHORT)	F7	Replace the Spindle Amplifier (SP).	An error was detected in the spindle amplifier (SP).
SP9159	MISMATCHED FUNCTION CODE	F9	Turn the power off, then restart.	CNC, SV, SP or PS software has been update.
SP9200	PS GROUND FAULT		Replace the motor or amplifier.	A ground fault occurred in the motor or amplifier. (Power Supply (PS) alarm indication: 09)
SP9204	PS SOFT THERMAL		Review the current condition.	Overload was detected in the power supply (PS). (Power Supply (PS) alarm indication: 15)
SP9211	PS ILLEGAL PARAMETER		Change the parameter to an appropriate value.	An invalid value was set in a parameter for controlling the power supply (PS). (Power Supply (PS) alarm indication: 23)
SP9212	PS HARDWARE ERROR		Replace the Power Supply (PS).	A hardware error was detected in the power supply (PS). (Power Supply (PS) alarm indication: 24)
SP9213	PS EXTERNAL INPUT COMPONENT ERROR		1. Change parameters to proper value 2. Change cables (K1, K127, K143) 3. Replace the Power Supply (PS) or the external component such as input filter or transducer	There is something trouble in control parameter for PS, cable, PS itself, or the external component such as input filter or transducer.
SP9214	PS PFB-R ERROR		Replace PFB-R	A PFB-R error is detected.
SP9215	PS PFB-C ERROR		Replace PFB-C	A PFB-C error is detected.
SP9216	PS SUB MODULE ERROR		Confirm the wiring or the combination of PS, PFB-R, and PFB-C.	The wiring or the combination of PS, PFB-R, and PFB-C is illegal

Error codes (serial spindle)

NOTE

*1 Note that the meanings of the Spindle Amplifier (SP) indications differ depending on which LED, the red or yellow LED, is on. When the yellow LED is on, an error code is indicated with a 2-digit number. An error code is indicated in the CNC diagnosis data No.0710. When the red LED is on, the Spindle Amplifier (SP) indicates the number of an alarm generated in the serial spindle.
→ See Item, “Alarm list (serial spindle).”

FRYER MACHINE SYSTEMS INC Rob Zahensky

Spindle amplifier indication (*1)	Description	Faulty section and action
01	Although neither *ESP (emergency stop signal; there are two types of signals including the input signal and Power Supply (PS) contact signal) nor MRDY (machine ready signal) is input, SFR (forward rotation command)/SRV (reverse rotation command)/ORCM (orientation command) is input.	Check the *ESP and MRDY sequence. For MRDY, pay attention to the parameter setting regarding the use of the MRDY signal (bit 0 of parameter No. 4001).
03	The parameter settings are such that a position sensor is not used (position control not performed) (bits 3, 2, 1, 0 of parameter No. 4002 = 0, 0, 0, 0), but a Cs contour control command is input. In this case, the motor is not excited.	Check the parameter settings.
04	The parameter settings are such that a position sensor is not used (position control not performed) (bits 3, 2, 1, 0 of parameter No.4002 = 0, 0, 0, 0), but a servo mode (rigid tapping, spindle positioning, etc.) or spindle synchronization command is input. In this case, the motor is not excited.	Check the parameter settings.
05	The orientation function option parameter is not specified, but ORCM (orientation command) is input.	Check the orientation function parameter settings.
06	The speed range switching control function option parameter is not specified, but low-speed characteristic winding is selected (RCH = 1).	Check the speed range switching control function parameter settings and the power line state check signal (RCH).
07	A Cs contour control command is input, but SFR (forward rotation command)/SRV (reverse rotation command) is not input.	Check the sequence.
08	A servo mode (rigid tapping, spindle positioning, etc.) control command is input, but SFR (forward rotation command)/SRV (reverse rotation command) is not input.	Check the sequence.
09	A spindle synchronization command is input, but SFR (forward rotation command)/SRV (reverse rotation command) is not input.	Check the sequence.
10	A Cs contour control command is input, but another mode (servo mode, spindle synchronization or orientation) is specified.	Do not switch to another mode during a Cs contour control command. Before moving to another mode, cancel the Cs contour control command.
11	A servo mode (rigid tapping, spindle positioning, etc.) command is input, but another mode (Cs contour control, spindle synchronization or orientation) is specified.	Do not switch to another mode during a servo mode command. Before moving to another mode, cancel the servo mode command.
12	A spindle synchronization command is input, but another mode (Cs contour control, servo mode or orientation) is specified.	Do not switch to another mode during a spindle synchronization command. Before moving to another mode, cancel the spindle synchronization command.
13	An orientation command is input, but another mode (Cs contour control, servo mode or spindle synchronization control) is specified.	Do not switch to another mode during an orientation command. Before moving to another mode, cancel the orientation command.
14	Both SFR (forward rotation command) and SRV (reverse rotation command) are input at the same time.	Issue either of them.

FRYER MACHINE SYSTEMS INC
Rob Zaherlsky

Spindle amplifier indication (*1)	Description	Faulty section and action
16	The parameter settings are such that the differential speed control function is not used (bit 5 of parameter No. 4000 = 0), but DEFMD (differential speed mode command) is input.	Check the parameter settings and differential speed mode command.
17	The speed detector parameter settings (bits 2, 1, and 0 of parameter No. 4011) are not valid. There is no corresponding speed detector.	Check the parameter settings.
18	The parameter settings are such that a position sensor is not used (position control not performed (bits 3, 2, 1, and 0 of parameter No. 4002), but position coder system orientation is issued.	Check the parameter settings and the input signal.
19	The magnetic sensor orientation command is input, but another mode (Cs contour control, servo mode or spindle synchronization control) is specified.	Do not switch to another mode during an orientation command. Before moving to another mode, cancel the orientation command.
21	The tandem operation command was input in the spindle synchronization control enable state.	Input the tandem operation command when spindle synchronization control is canceled.
22	Spindle synchronization control was specified in the tandem operation enable state.	Specify spindle synchronization control when torque tandem operation is canceled.
23	The tandem operation command is input without the required option.	Torque tandem control requires a CNC software option. Check the option.
24	If index is performed continuously in position coder method orientation, an incremental operation is performed first (INCMD = 1), then an absolute position command (INCMD = 0) is input.	Check INCMD (incremental command). If an absolute position command is to follow, be sure to perform absolute position command orientation first.
26	The parameter settings are such that both spindle switch and three-stage speed range switch are used.	Check the parameter settings and the input signal.
29	The parameter settings are such that the shortest-time orientation function is used (bit 6 of parameter No. 4018 = 0, Nos. 4320 to 4323 ≠0).	In the α*i* series spindle amplifier, the shortest-time orientation function cannot be used. The use of the optimum orientation function is recommended.
30	The magnetic pole has not been detected, but a command is input.	In the magnetic pole undetected state (EPFIXA = 0), the motor cannot be driven even when a command is input. Input a command in the magnetic pole detected state (EPFIXA = 1). When EPFSTR is set to 1, any command is ignored and this error is displayed even in the magnetic pole detected state. After the completion of magnetic pole detection, set EPFSTR to 0.
31	The hardware configuration is such that the spindle FAD function cannot be used. In this case, the motor is not activated.	Check the CNC model. With the FS30*i* series, the spindle FAD function is not used.
32	S0 is not specified in the velocity mode, but the disturbance input function is enabled (bit 7 of parameter No. 4395 is set to 1).	Specify S0 in the velocity mode before enabling the disturbance input function (bit 7 of parameter No. 4395 to 1).
33	The hardware configuration is such that the spindle EGB function cannot be used. In this case, the motor is not activated.	Check the CNC model.
34	Both the spindle FAD function and the spindle EGB function are enabled. In this case, the motor is not activated.	The two functions cannot be used at the same time. Enable either function only.
35	Spindle Amplifier (SP) ID information cannot be obtained.	Replace the spindle amplifier with one with correct ID information.

FRYER MACHINE SYSTEMS INC
Rob Zakrzewski

Spindle amplifier indication (*1)	Description	Faulty section and action
36	The submodule SM (SSM) is faulty .	For action to be taken, refer to the FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN).
37	The current loop setting (No. 4012) has been changed.	Check the setting of parameter No. 4012, and turn the power off, then on again.
38	A parameter related to communication between spindle amplifiers is specified incorrectly. Alternatively, a function unavailable with the torque tandem function is set.	Check the parameters.
39	Although SFR (forward rotation command), SRV (reverse rotation command), or ORCM (orientation command) is input, DSCN (disconnection detection disable signal) is input.	Check the sequence. Do not input DSCN (disconnection detection disable signal) during the input of a command which excites the motor.
43	A setting which does not support the αiCZ sensor (serial) is used.	Check the parameter settings.
44	The spindle amplifier does not support the control period setting.	Check the setting of parameter No. 4012.

(11) Overheat alarms (OH alarm)

Number	Message	Description
OH0700	LOCKER OVERHEAT	CNC cabinet overheat
OH0701	FAN MOTOR STOP	PCB cooling fan motor abnormality
OH0704	OVERHEAT	Spindle overheat due to detection of changes in the spindle speed - When the cutting load is large, offload the cutting conditions. - Check if the cutting tool became dull. - Check if the spindle amplifier malfunctions.

(12) Other alarms (DS alarm)

Number	Message	Description
DS0001	SYNC EXCESS ERROR (POS DEV)	In feed axis synchronization control, the difference in the amount of positional deviation between the master and slave axes exceeded the parameter No. 8323 setting value. This alarm occurs for the master or slave axis.
DS0002	SYNC EXCESS ERROR ALARM 1	In feed axis synchronization control, the difference in the amount of synchronization between the master and slave axes exceeded the parameter No. 8331 setting value. This alarm occurs only for the slave axis.
DS0003	SYNCHRONIZE ADJUST MODE	The system is in the synchronize adjust mode.
DS0004	EXCESS MAXIMUM FEEDRATE	The malfunction prevention function detected the command in which a value exceeding the maximum speed was specified.
DS0005	EXCESS MAXIMUM ACCELERATION	The malfunction prevention function detected the command in which a value exceeding the maximum acceleration was specified.
DS0006	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0007	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0008	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0009	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.

FRYER MACHINE SYSTEMS INC
Rob Zahersky

Number	Message	Description
DS0010	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0011	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0012	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0013	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0014	TOOL CHANGE DETECT MACHINE LOCK	A machine lock is turned on for the Z axis for which the tool is being changed.
DS0015	TOOL CHANGE DETECT MIRROR IMAGE	A mirror image is turned on for the Z axis for which the tool is being changed.
DS0016	SERIAL DCL:FOLLOW-UP ERROR	(1) The settings of parameters Nos.1883 and 1884 fall outside the range. (2) The current position at establishment of the origin subtracted by the distance between the reference positions (detection unit) exceeded ±2147483647. Change the current position or reference position to prevent this situation.
DS0017	SERIAL DCL:REF-POS ESTABLISH ERR	The travel amount at the FL speed at establishment of the origin exceeded the setting of parameter No. 14010.
DS0018	SERIAL DCL:MISMATCH(SSYNC CTRL)	Of the master and slave axes for feed axis synchronous control, one axis is a linear scale with the origin and the other is not a linear scale with the origin. In such a configuration, the feed axis synchronous control selection signal (SYNC<Gn138> or SYNCJ <Gn140>) needs to be set to 0 to establish the origin.
DS0019	SERIAL DCL:MISMATCH(ANGL-AXIS)	In angular axis control, one of the angular axis and the Cartesian axis is a linear scale with an origin, while the other is not a linear scale with an origin. Angular axis control cannot be used in such a configuration.
DS0020	REFERENCE RETURN INCOMPLETE	An attempt was made to perform an automatic return to the reference position on the perpendicular axis before the completion of a return to the reference position on the angular axis. However, this attempt failed because a manual return to the reference position during angular axis control or an automatic return to the reference position after power-up was not commanded. First, return to the reference position on the angular axis, then return to the reference position on the perpendicular axis.
DS0022	DUAL CHECK SAFETY IS NOT WORKED	Bit 6 (DCE) of parameter No. 1902 has set the Dual Check Safety function to be disabled.
DS0023	ILLEGAL PARAMETER (I-COMP VAL)	The setting of the inclination compensation parameter is incorrect. The compensation per compensation point is too large or too small.
DS0024	UINT SIGNAL WAS ILLEGALLY INPUT	An interruption type custom macro was called during movement to the program restart position. Please do not call the interruption type custom macro.
DS0025	G60 CANNOT BE EXECUTED	The state of a mirror image is different between the time when look-ahead of a block for unidirectional positioning was performed and the time when execution of the block was started, so unidirectional positioning cannot be performed. Modify the program.
DS0026	MISMATCH OF ANGULAR AXIS(D.C.S)	On angular axis control, one of the angular/perpendicular axes is the scale with ref-pos, and the other of them is not the scale with ref-pos. Such system is not admired.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	Description
DS0027	MISMATCH OF SYNCHRONOUS AXIS(D.C.S)	- One of the master/slave axes of axis synchronous control is the linear scale with distance-coded reference marks, and the other is not the linear scale with distance-coded reference marks. Please establish reference position with the input signal SYNCn<Gn138>, SYNCJn<Gn140> or parameter setting to 1. - The slave axis of the axis synchronous control is two or more settings. Such system is not admired.
DS0029	UNAVAILABLE ROT AXIS MOVE IN TPC	In tool posture control, the rotation axis fell outside the set operation range (parameters Nos. 19741 to 19744). Review the machine configuration and command.
DS0030	TOOL DIRC CMP IMPOSSIBLE	When tool center point control is performed in the angular surface machining command mode or workpiece placement error compensation is performed in a 5-axis cutting machine, if settings are made to make compensation in the tool direction (bit 0 (RCM) of parameter No. 11200 is 1), the position of the rotation axis could not be obtained.
DS0050	TOO MANY SIMULTANEOUS AXES	A movement was performed along more axes than can be controlled simultaneously. Check whether a command in the program is specified for more axes than can be controlled simultaneously.
DS0059	SPECIFIED NUMBER NOT FOUND	[External data I/O] The No. specified for a program No. or sequence No. search could not be found. There was an I/O request issued for a pot No. or offset (tool data), but either no tool numbers have been input since power ON or there is no data for the entered tool No. [External workpiece No. search] The program corresponding to the specified workpiece No. could not be found.
DS0060	MEMORY CARD ACCESS ERROR DETECT	The error occurred while accessing the memory card. Confirm the memory card. Whether the card is recommended and not breakdown.
DS0070	SUPERIMPOSE FOR HIGH-SPEED CYCLE CANNOT BE USED	Superimposed control for high-speed cycle machining is not enabled. Alternatively, the status is not the advanced superimposition state.
DS0071	START OR RELEASE CANNOT BE DONE	(1) To start or cancel the inter-path flexible synchronization mode, the tool must be stopped along all axes. (2) To start or cancel advanced superimposition, movement along axes must stop.
DS0072	MANUAL REFERENCE RETURN CANNOT BE DONE	Manual reference position return cannot be performed in the advanced superimposition state.
DS0087	ILLEGAL OPERATION IN P2ACC MODE	(1) Relating parameters are changed in Cross rail axis control mode on. (2) Axis control that is not allowed in Cross rail axis control mode on is used. (3) Reference position of the master axis or the slave axis becomes not established in the mode.
DS0131	TOO MANY MESSAGE	An attempt was made to display an external operator message or external alarm message, but five or more displays were required simultaneously.
DS0132	MESSAGE NUMBER NOT FOUND	An attempt to cancel an external operator message or external alarm message failed because the specified message number was not found.

FRYER MACHINE SYSTEMS INC
Rod Zaneisky

Number	Message	Description
DS0133	TOO LARGE NUMBER	A value other than 0 to 4095 was specified as the external operator message or the external alarm message number.
DS0300	APC ALARM: NEED REF RETURN	A setting to zero position for the absolute position detector (association with reference position and the counter value of the absolute position detector) is required. Perform the return to the reference position. This alarm may occur with other alarms simultaneously. In this case, other alarms must be handled first.
DS0306	APC ALARM: BATTERY VOLTAGE 0	The battery voltage of the absolute position detector has dropped to a level at which data can no longer be held. Or, the power was supplied to the Pulsecoder for the first time. The battery or cable is thought to be defective. Replace the battery with the machine turned on.
DS0307	APC ALARM: BATTERY LOW 1	The battery voltage of the absolute position detector has dropped to a level at which a replacement is required. Replace the battery with the machine turned on.
DS0308	APC ALARM: BATTERY LOW 2	The battery voltage of the absolute position detector dropped to a level at which a replacement was required in the past. (including during power off) Replace the battery with the machine turned on.
DS0309	APC ALARM: REF RETURN IMPOSSIBLE	An attempt was made to set the zero point for the absolute position detector by MDI operation when it was impossible to set the zero point. Rotate the motor manually at least one turn, and set the zero position of the absolute position detector after turning the CNC and servo amplifier off and then on again.
DS0310	NOT ON RETURN POINT	The return position recorded during retraction is not reached during recovery. The position may be displaced during recovery due to a machine lock or mirror image. Perform the operation again after making a reset.
DS0405	ZERO RETURN END NOT ON REF	The axis specified in automatic zero return was not at the correct zero point when positioning was completed. Perform zero return from a point whose distance from the zero return start position to the zero point is 2 or more revolutions of the motor. Other probable causes are: - The positional deviation after triggering the deceleration dog is less than 128. - Insufficient voltage or malfunctioning Pulsecoder.
DS0608	SV COOLING FAN FAILURE	Internal agitating fan failure.
DS0609	SV RADIATOR FAN FAILURE	Radiator cooling fan failure.
DS0610	PS INTERNAL FAN FAILURE	Internal agitating fan failure.
DS0611	PS EXTERNAL FAN FAILURE	External radiator cooling fan failure.
DS0612	PS OVERLOAD	Overheat
DS0613	PS IMPROPER INPUT POWER	Input power supply fault
DS0614	PS EXTERNAL INPUT COMPONENT ERROR	There is something trouble in control parameter for PS, cable, PS itself, or the external component such as input filter or transducer.
DS0650	POLE DETECTION REQUEST	With an absolute detection axis (bit 5 (APCx) of parameter No.1815=1), pole position detection is not completed (RPFIN1 to RPFIN8 <Fn159>="0"). With a non-absolute detection axis (bit 5 (APCx) of parameter No.1815=0), pole position detection is once completed then the state is changed to the pole position detection uncompleted state (RPFIN1 to RPFIN8 <Fn159>=0).

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Number	Message	Description
DS0651	ACC.ERROR	Communication between the separate detector interface unit and acceleration sensor was disconnected.
DS0653	SSM FAILURE	An error occurred in SSM driving of the servo amplifier. The followings are assumed as a cause. (1) The setting of parameters related to SSM is invalid. (2) SSM or connection cable of SSM is defective.
DS1105	ILLEGAL PARAMETER (3DR-COMP)	Parameter setting for three-dimensional rotary error compensation is wrong. One of the following may be the cause. - The designation for rotary axes is wrong. - The number of the total compensation points is larger than 7812. - The designation for machine type is wrong. - The designation for compensation axis is wrong. - The setting for compensation number of reference point is wrong. - The setting for compensation interval is wrong. - The parameter is set in more than two path systems.
DS1120	UNASSIGNED ADDRESS (HIGH)	The upper 4 bits (EIA4 to EIA7) of an external data I/O interface address signal are set to an undefined address (high bits).
DS1121	UNASSIGNED ADDRESS (LOW)	The lower 4 bits (EIA0 to EIA3) of an external data I/O interface address signal are set to an undefined address (low bits).
DS1124	OUTPUT REQUEST ERROR	OUTPUT REQUEST ERROR An output request was issued during external data output, or an output request was issued for an address that has no output data.
DS1128	DI.EIDLL OUT OF RANGE	The numerical value input by external data input signals ED0 to ED31 has exceeded the permissible range.
DS1130	SEARCH REQUEST NOT ACCEPTED	No requests can be accepted for a program No. or a sequence No. search as the system is not in the memory mode or the reset state.
DS1131	EXT-DATA ERROR (OTHER)	[External Data I/O] An attempt was made to input tool data for tool offset by a tool No. during loading by the G10 code.
DS1150	A/D CONVERT ALARM	A/D converter malfunction
DS1184	PARAMETER ERROR IN TORQUE	An invalid parameter was set for torque control. The torque constant parameter is set to “0”.
DS1185	OVER MAXIMUM FEED	The maximum cutting feedrate or rapid traverse feedrate was exceeded in G54.3.
DS1448	ILLEGAL PARAMETER (D.C.S.)	The setting value of parameter for distance coded linear scale I/F is satisfied the following any conditions. - The absolute-position detection function is enabled. - Either parameter 1821 (mark-1 interval) or parameter 1882 (mark-2 interval) is set to 0. - Parameters 1821 and 1882 have identical settings. - The difference between the settings made for parameters 1821 and 1882 is greater than or equal to twice either setting. - The setting value of parameters 1883 and 1884 are over the valid data range. - In case of Rotary axis (B type), the values other than 0 are set to the parameter No.1868 and No.1869 or bit 0 (RVSx) of parameter No.1815 is set to 1.
DS1449	REFERENCE MARKS ARE DIFFERENT FROM PARAMETER	In case of distance coded linear scale I/F, the actual interval of reference marks is different from parameters Nos.1821 and 1882 setting value.

FRYER MACHINE SYSTEMS INC
Rockaway NJ

Number	Message	Description
DS1450	ZERO RETURN NOT FINISHED	1st reference position return (CDxX7 to CDxX0: 17h (Hex)) was specified when the manual reference position return was not executed with the reference position return function enabled (bit 0 (ZRN) of parameter No. 1005 set to “0”).
DS1451	IMPROPER PMC AXIS COMMAND	(1) During deceleration of PMC axis control, next PMC axis control command is commanded. Please correct PMC ladder sequence to execute the next command after the PMC axis control is stopped. (2) While the controlled axis selection signals EAX1 to EAX8 <Gn136> or PMC controlled-axis selection variable (#8700) are “0”, PMC axis control command is commanded. Please correct PMC ladder sequence or macro executor program so that the signal or variable is set to 1.
DS1512	EXCESS VELOCITY	The feedrate of the linear axis during polar coordinate interpolation exceeded the maximum cutting feedrate.
DS1514	ILLEGAL MOTION IN G12.1 MODE	In a hypothetical axis direction compensation during the polar coordinate interpolation mode, an attempt is made to travel to the area in which the travel cannot be made.
DS1553	EXCESS VELOCITY IN G43.4/G43.5	The axis rate was attempt to exceed the maximum cutting feedrate and travel by the pivot tool length compensation.
DS1710	ILLEGAL ACC. PARAMETER (OPTIMUM TORQUE ACC/DEC)	There are errors in the parameters of permissible acceleration for Optimum Torque Acceleration/Deceleration. One of the following is the cause. (1) The ratio of the acceleration for deceleration to the acceleration for the acceleration is lower than the limited value. (2) The time to decelerate to 0 is larger than the maximum.
DS1711	ILLEGAL ACC. PARAMETER (RIGID TAPPING OPTIMUM ACC/DEC)	The permissible acceleration parameter for rigid tapping optimum acceleration/deceleration contains an error. The cause is one of the following: (1) The ratio of the deceleration to the acceleration is less than 1/3. (2) The time required to slow down to a speed of 0 exceeds the maximum. (3) The maximum acceleration (parameters Nos. 11421 to 11424) is 0.
DS1931	MACHINE PARAMETER INCORRECT	- One of parameters Nos. 19665 to 19667 and Nos.19680 to 19744 used to configure the machine contains an error. - 5-axis machining function which can not be used in the machine which has two linear axes (bit 6 (HAL) of parameter No. 11269 = 1) is executed. - In the machine which has two linear axes (bit 6 (HAL) of parameter No. 11269 = 1), some motion of nonexistent liner axis is generated.
DS1932	DI.THML SIGNAL ON	One of the parameters used to configure the machine is rewritten while the tool direction thermal displacement compensation function is enabled.
DS1933	NEED REF RETURN(SYNC:MIX:OVL)	The relation between a machine coordinate of an axis in synchronization, composition, or superposition control, and the absolute, or relative coordinate was displaced. Perform the manual return to the reference position.
DS2003	ILLEGAL USE FOR SERVO MOTOR SPINDLE	The servo axes for spindle use of the spindle control with servo motor was used by the following functions. - PMC axis control - Chopping

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Number	Message	Description
DS2091	CAN NOT RETURN TO BREAKPOINT	It was not possible to return to the machine position of the interruption point because of machine lock.
DS2092	REVERSE START ERROR(RMFR)	Reverse motion function for restart could not be begun. The followings are assumed as a cause. - When Reverse motion function for restart begins, CNC is not reset status. - Interruption block is in a mode that cannot begin Reverse motion function for restart. - The program has never executed before. - The program has never executed after the reverse motion is done until becoming the reverse end (RVED) by Reverse motion function for restart.
DS2096	ALARM OCCURRED IN MAIN PATH	The alarm occurred in the path that used the peripheral axis control.
DS2097	ALARM OCCURRED IN PERIPHERAL	The alarm occurred in the peripheral axis control.
DS5258	FEED HOLD IN RESTART	When reverse motion for restart is interrupted by feed hold, this alarm occurs.
DS5259	INTERRUPTED POSITION NOT FOUND	The interruption point was not found. (Coordinates of the processing interruption point by temporary execution and coordinates of the memorized processing interruption point are different.) The following is assumed as a cause. - During feed hold state or single block stop, manual intervention was executed.
DS5340	PARAMETER CHECK SUM ERROR	Since a parameter was changed, the parameter checksum did not match the reference checksum. Set the original value to the parameter or set the reference checksum again.
DS5387	CAN NOT START REFERENCE RETURN WITH MECHANICAL STOPPER SETTING	The slave axis is not within the in-position width, for example, when the distance between the mechanical stoppers for the master and slave axes is larger than the withdrawal distance specified in parameter No. 7181 or 7182 for the master axis. Adjust the position of the mechanical stoppers or the setting of parameter No. 7181 or 7182.
DS5550	AXIS IMMEDIATE STOP	The movement along an axis was stopped immediately by the axis immediate stop function.

(13) Malfunction prevention function alarms (IE alarm)

Number	Message	Description
IE0001	+ OVERTRAVEL (SOFT 1)	The malfunction prevention function detected that stored stroke check 1 on the positive side was exceeded.
IE0002	- OVERTRAVEL (SOFT 1)	The malfunction prevention function detected that stored stroke check 1 on the negative side was exceeded.
IE0003	+ OVERTRAVEL (SOFT 2)	The malfunction prevention function detected that stored stroke check 2 on the positive side was exceeded.
IE0004	- OVERTRAVEL (SOFT 2)	The malfunction prevention function detected that stored stroke check 2 on the negative side was exceeded.
IE0005	+ OVERTRAVEL (SOFT 3)	The malfunction prevention function detected that stored stroke check 3 on the positive side was exceeded.
IE0006	- OVERTRAVEL (SOFT 3)	The malfunction prevention function detected that stored stroke check 3 on the negative side was exceeded.
IE0007	EXCESS MAXIMUM REV. DATA	The malfunction prevention function detected the command in which a value exceeding the maximum speed was specified.
IE0008	ILLEGAL ACC/DEC	The malfunction prevention function detected the acceleration/deceleration error.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

H WARNING LIST

(1) Data input-related

Message	Description
DATA IS OUT OF RANGE	A value tried to be set to the amount of the work shift or the amount of the offset, etc. is out of the range that is set by parameter. Set the value within the range set by the parameter.
DATA IS OUT OF RANGE	A value tried to be set to the parameter or the offset, etc is out of the range. Set data within the range.
TOO MANY DIGITS	A number that exceed valid digits was entered to the O number or the offset value, etc. Enter a number within the valid range.
WRONG MODE	Such as selecting the main program or inputting the parameter could not execute in the current mode. Switch to the correct mode, and retry.
PARAMETER WRITE PROTECT	When the parameter write is disabled, the parameters have been written. Set the parameter write enable and write the parameters.

(2) EDIT related

Message	Description
PROTECTED	The protected program was edited. Release protecting.
BG EDIT REJECTED	The protected program was selected on the program folder screen, the INPUT key was pressed and background editing was performed. Release protecting, and select the program.
EDIT REJECTED	The program which is protected from editing tired to be edited by the background editing.
SPECIFIED PROGRAM ALREADY EXISTS	The program of the same name had already existed when the program was created or was moved. Change the name of the program or delete an existing program of the same name.
SPECIFIED PROGRAM NOT FOUND	The specified program was not found in program search. The specified program was not found in deleting a program on word editing screen. Check the program name.
TOO MANY CHARACTERS	Exceeds the number of characters that can be specified editing (create, copy, move, etc.) of program and folder in the program name and folder name was specified. Try again by changing the number of characters that can be specified.
characters not found	Search string can not be found in the program. Check the search string.
NO MORE SPACE	In all replacements, disk free space is low. Delete some programs, and increase disk space.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

(3) Program List Related

Message	Description
ABNORMAL END	The error occurred by the data server factor when [GET], [PUT], [MGET], [MPUT], [LIST-GET], [LIST-PUT], [LIST-DELETE] was done or operation was canceled for the data server host's program (file) on program folder screen. The presumed cause is as follows. Remove it and operate again. - Connect or login to the host failed. - Specified file does not exist or the specified number of files is too much. - List file does not exist or mistake is found in the format of list file. - The file transfer was rejected from the FTP server. (there was no right to access for the folder or same name folder exists, etc.)
CNC RUNNING	(a) ATA card of the data server function, present connected host or connected host of built-in Ethernet function are used by other function when the file operation of copy, move, and delete, etc. was done on the program folder screen. Operate it after the use of other function ends. (b) Tool life values, tool life counter, tool life counter type, tool number, tool length compensation number, the tool diameter compensation number or tool information was changed during automatic operation. Change it after automatic operation ends. M (c) Titled working plane indexing, cylindrical interpolation or polar coordinate interpolation was instructed in by the processing simulation of a dynamic graphic function during automatic operation. Start simulation after automatic operation ends. T (c) Titled working plane indexing, cylindrical interpolation was instructed in by the processing simulation of a dynamic graphic function during automatic operation. Start simulation after automatic operation ends.
CAN NOT COPY/MOVE	The device that was not available to be selected to the copy origin or copy destination was selected on the program folder screen. Change the device.
DISABLE TO SELECT	(a) On the program folder screen, when the copy or move method is old type (No.11374#4=1), the folder was selected as a copy or move target. Don't select it. (b) On the program folder screen, "RETURN TO UPPER FOLDER" is selected as a copy or move target. Don't select it. (c) On the USB folder screen, the file or folder whose name length is 33 or more was selected. Don't select the file or the folder, because the max length of name is 32 characters in CNC. (d) On the USB folder screen, the name of the file or folder which was omitted by "~" (tilde) was tried to be changed. The name of the file or folder which was omitted by "~" (tilde) can not be changed. (e) On the program character editing screen, %(EOR) was selected. Don't select it.

FRYER MACHINE SYSTEMS INC
Rob Zaborsky

Message	Description
CNC MODE ERROR	(a) Mode was changed from EDIT to the others before the program insertion ends on program (word/character) editing screen. Do not change the mode from EDIT to others during processing. (b) Copy or move was interrupted by the following causes on program folder screen. - Copy/move was done during automatic operation - Mode was changed from EDIT to the others or emergency stop state was canceled during processing It is possible to copy/move the file between the data server and the USB memory in the state of automatic operation stops and all paths are EDIT mode or the emergency stop. (c) Restart point was search on program restart screen in the mode other than MEM/RMT. Change the mode to MEM/RMT before searching restart point.
WRITE PROTECTED	(a) Data server program (file) which was tried to be copied, moved or deleted on the program folder screen was protected. (b) Program which was tried to be deleted or rename on program folder screen was protected. (c) Program to which the program insertion was done on program editing (word/character) screen was protected. (d) Folder tried to make a new program on program editing screen was protected. (e) In tool management function, the data which was tried to be changed on each tool data screen was protected. (Tool management data protection signal was set to 1.)
REJECT CNC	(a) Heading the program pointer was tried on program screen or program check screen during automatic operation. Heading again after the automatic operation ends. (b) Operation was not able to be completed because the specified program was in automatic operation when copy, move or delete of the program was done on program folder screen. Operate again after the automatic operation ends.
CAN'T CREATE SPECIFIED FOLDER.	Folder name that was not able to be used to create folder was used on program folder screen.
CLEAR DNC DATA	Schedule operating was attempted to be set though the DNC operating is being set. Release the setting of the DNC operating.
CLEAR SCHEDULE DATA	DNC operating was attempted to be set though the schedule operating is being set. Release the setting of the DNC operating.
CAN'T MOVE TO FOLDER	The following folders ware tried to be moved to another folder. - Folder with the main program - Folder with protected program - Folder that is edit interdiction attribute Move after releasing the above condition.
SPECIFIED FOLDER NOT EXISTS	In program folder operation, the specified folder doesn't exist. Confirm the name of the folder.
NO FILE SELECTED	In file delete operation for USB memory in program folder screen, the files are not selected. Operate it after selecting the files.
NO MORE SPACE	The program cannot be edited because program memory is full. Delete unnecessary programs or execute program condense.
PROGRAM IN FOLDER IS NOT DELETED	The folder including the program was deleted. Delete the folder after deleting the file in the folder.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Message	Description
CAN'T MOVE TO UPPER FOLDER	Though the operation to move to upper folder was done on the program folder screen, the operation cannot be completed because it is in the most upper folder.
CAN'T MOVE FROM THIS FOLDER BY SEARCH	The main program search was done. However, it is a setting that cannot display the folder of the main program. Operate it after changing the setting that can display the folder of the main program.
IMPOSSIBLE TO OUTPUT	The operation to the USB device was done. However, the USB device is in the state that cannot be used. Attach the USB device again or use a different USB device once.
SETTING A BACKGROUND FOLDER FAILED	The folder for the background was not able to be set. Operate it again after confirming the attribute of the folder, etc.
DNC FILE SET FAILED	The program of an external device of the data server, the data server host, and the floppy disk cassette was not able to be selected as DNC operation target. Set the device that can be used by the DNC operation.
SPECIFIED FILE NOT EXISTS	The operated file was not found. Check the file name.
FILE OF COPY PLACE IS SELECTED BY NC	Because the program of the copy or the moving destination has been selected as the main program, it is not possible to overwrite. Delete the program of the copy or the moving destination.
SELECTED FILE IS SELECTED BY NC	Because the specified file exists, and has been selected as the main program, it is not possible to operate it to the file. Operate it again after selecting a different program as the main program.
SPECIFIED FOLDER ALREADY EXISTS	The folder name that already existed was specified in creating and rename of folder. Change the folder name.
FOLDER IS NOT EMPTY	Contents of a folder that specified to delete are not empty. Delete file/folder in the specified folder.
SELECTED FOLDER IS SELECTED BY NC	The selected folder in NC was specified in renaming a folder and deleting a folder. Release the selection of the specified folder.
SETTING A FOREGROUND FOLDER FAILED	Setting of foreground folder was done in a mode other than EDIT mode. Switch to EDIT mode
M198 FOLDER SET FAILED	M198 operation folder was set in a mode other than EDIT mode. Switch to EDIT mode.
SETTING A MAIN PROGRAM FAILED	Selecting of main program was done in illegal conditions. - Set the unlock state in keys and program encryption. - Set the automatic operation stop state or the reset state when bit 1 (OPC) of parameter No. 3204 is 0. - Set the reset state when bit 1 (OPC) of parameter No. 3204 is 1. - The specified program is editing in background. End the background editing. - The folder is specified. Specify the program.
THE NUMBER OF REGISTRATION IS FULL	The number of registration was filled with the following operations. - create/copy of file/folder - addition of tool geometry data - addition of screen to main menu Ensure the number of registration.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Message	Description
SEARCHING FAILED	(a) The word search failed on quick program restart screen. (b) The search failed on USB memory file list screen. - Adjust the length of word to 28 characters or less. - Check the searching string.
LIMITS OVER	The number of selected program to copy/move/delete exceeds the upper bound. Adjust the number of selections of programs to ten or less.
CHANGING A WORK FOLDER FAILED IN ATA CARD	The error occurred by changing a working folder in memory card of Data Server. Confirm whether there is a specified folder. Check the log message of Data Server.
COPYING A FILE FAILED IN ATA CARD	The error occurred by copying the file in the memory card of Data Server. The presumed cause is as follows. - There is not a source file. - Destination file already exists. Check the log message of Data Server.
REMOVING THE FILE FAILED FROM ATA CARD	The error occurred by deleting the file in the memory card of Data Server. The presumed cause is as follows. - There is not a specified file. - A specified file is being used. Check the log message of Data Server.
THE SPECIFIED FILE IS NOT FOUND	The error occurred by searching the file in memory card of Data Server, in host computer of Data Server, or in host computer of Embedded Ethernet. Check the specified file name.
REMOVING THE FILE FAILED ON FTP SERVER	The error occurred by deleting the files in the host computer of Data Server or Embedded Ethernet. The presumed cause is as follows. - There is not a specified file. - A specified file is being used. Check the log message of Data Server or Embedded Ethernet.
ERROR OCCURRED ON GET OPERATION	The error occurred by GET operation of Data Server. There is a possibility of the problem on the personal computer side, too. Check the log message of Data Server.
GETTING A FILE LIST FAILED	The error occurred by displaying the file list in memory card of Data Server, in host computer of Data Server, in host computer of Embedded Ethernet, or in USB memory stick. In case of displaying the file list in host computer, there is a possibility of the problem on the personal computer side, too. Check the log message of Data Server, Embedded Ethernet, or USB memory stick.
CHANGING A WORK FOLDER OF HOST FAILED	The error occurred by changing the working folder in the host computer of Data Server or Embedded Ethernet. The presumed cause is as follows. - There is not a specified file. Check the working folder of Data Server or Embedded Ethernet.
REMOVING THE FILE FAILED FROM HOST	The error occurred by deleting the files in the host computer of Data Server or Embedded Ethernet. The presumed cause is as follows. - There is not a specified file. - A specified file is being used. - There is no access right of a specified file. Check the log message of Data Server or Embedded Ethernet.

FRYER MACHINE SYSTEMS INC
Rod Zarensky

Message	Description
MAKING A NEW FOLDER FAILED IN HOST	The error occurred by making the folder in the host computer of Data Server or Embedded Ethernet. The presumed cause is as follows. - The name of a specified folder is incorrect. - There is no access right for write operation. Check the log message of Data Server or Embedded Ethernet.
RENAMING A NAME OF HOST FAILED	The error occurred by renaming the file in the host computer of Data Server or Embedded Ethernet. The presumed cause is as follows. - There is not a specified file. - A specified file is being used. - There is no access right of a specified file. - There is a file after changing. Check the log message of Data Server or Embedded Ethernet.
ERROR OCCURRED ON LIST-DEL OPERATION	The error occurred by LIST-DEL operation of Data Server. Check the log message of Data Server.
ERROR OCCURRED ON LIST-GET OPERATION	The error occurred by LIST-GET operation of Data Server. There is a possibility of the problem on the personal computer side, too. Check the log message of Data Server.
ERROR OCCURRED ON LIST-PUT OPERATION	The error occurred by LIST-PUT operation of Data Server. There is a possibility of the problem on the personal computer side, too. Check the log message of Data Server.
ERROR OCCURRED ON MGET OPERATION	The error occurred by MGET operation of Data Server. There is a possibility of the problem on the personal computer side, too. Check the log message of Data Server.
MAKING A NEW FOLDER FAILED IN ATA CARD	The error occurred by making the folder in the memory card of Data Server. The presumed cause is as follows. - A specified folder already exists. - The name of a specified folder is incorrect. Check the log message of Data Server.
ERROR OCCURRED ON MPUT OPERATION	The error occurred by MPUT operation of Data Server. There is a possibility of the problem on the personal computer side, too. Check the log message of Data Server.
ERROR OCCURRED ON PUT OPERATION	The error occurred by PUT operation of Data Server. There is a possibility of the problem on the personal computer side, too. Check the log message of Data Server.
RENAMING A FILE NAME FAILED IN ATA CARD	The error occurred by renaming the file in the memory card of Data Server. The presumed cause is as follows. - There is not a specified file. - A file name after changing is incorrect. - A specified file is being used. - There is a file after changing. Check the log message of Data Server.
REMOVING THE FOLDER FAILED FROM ATA CARD	The error occurred by deleting the folder in the memory card of Data Server. The presumed cause is as follows. - There is not a specified folder. - The contents of a specified folder are not empty. Check the log message of Data Server.

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Message	Description
PROGRAM NOT SELECT	The program has not been selected by deleting, copying, and moving the program. Execute them after selecting the program.
DELETE FILE ?	On the USB folder screen, this is displayed to confirm a deleting of file.
DELETE FILE OR FOLDER ?	On the USB folder screen, this is displayed to confirm a deleting of file or folder.
DELETE PROGRAM (FOLDER) ?	On the program folder screen or the data server file list screen, this is displayed to confirm a deleting of program or folder. (No.11374#4=0)
DELETE FOLDER ?	On the tree display of the program folder screen or the data server file list screen, this is displayed to confirm a deleting of folder.
DELETE PROGRAM ?	On the program editing screen or the program folder screen or the data server file list screen, this is displayed to confirm a deleting of program.
CANNOT COPY TO SAME FILE	On the program folder screen, the copy or move target program is same to the copy or move destination. Set different program name.
SCHEDULE ERROR	The setting of the schedule operation is wrong. Confirm the schedule operation setting.

(4) Input and output related

Message	Description
CHANGE TO EDIT MODE	The input/output of the program was operated in a mode other than EDIT mode. Switch to EDIT mode.
CANNOT USE I/O DEVICE	The input/output of data was operated. The device is busy according to other input/output operations. Operate input/output after completing other input/output operations.
PLEASE RELEASE EMERGENCY STOP	In the state of the emergency stop, output operation was done on all data output screen. Operate all data output after releasing the emergency stop for all paths.
EXCUTING OR MODE CHANGING	(a) The previous operation hasn't finished in All I/O screen. Operate after the previous operation has finished. (b) The precision level cannot be set during the axis movement. Set it during the axis stop.
FILE ALREADY EXIST	NC data can't be output to external device because the file of the same name already exists. Change the name of output file or delete the file of the same name in external device.
FORMAT ERROR	The format of memory card or USB memory is illegal. Use the memory device in proper format.
PARAMETER SETTING ERROR	When the output was operated, the settings of the parameters were not correct in NC data output function. - Set bit 0 (MDP) of parameter No. 138 to 1 - Set parameter No. 20 to 4 (Memory card) or 17 (USB memory) - Set parameter No.20 and parameter No.21 to the same value - Set parameter No.20 to the same value when the screen is displayed and data output
USB MEMORY IS NOT MOUNTING	USB memory was not inserted in the USB slot when using. Retry after insert the USB memory.
OVER WRITE FAILED	When [REWRITE] or [ALL] is executed in data output, overwriting failed because overwrite file is protected. Retry after removing protection of object file.

FRYER MACHINE SYSTEMS INC
ROD ZAMONSKY

Message	Description
I/O DEVICE SETTING ERROR	The setting of the input/output device was changed other than USB memory while displaying USB memory on ALL/IO screen. Select the input/output device correctly.
MEMORY CARD IS NOT MOUNTING	On the memory card folder screen, Input/output or search was operated when the memory card is not attached. Attach the memory card.

(5) Execute related

Message	Description
THE MOVING AXIS HAS NOT BEEN SET.	The specified axis was not set to the movement axis though the movement axis setting of the program restart function was released. Operate the axis set to the movement axis.
START FROM MIDDLE OF PROG(START/RESET)	The cycle start tried to be executed halfway in the program. - If you want to execute the cycle start from halfway , turn the cycle start signal ST from "1" to "0", and start the program operation again. - If you want to execute the cycle start from top, rewind the program.

(6) Work coordinate system related

Message	Description
NG SETTING (U-LMT AND L-LMT ILLEGAL)	In Work coordinate system shift function, the shift value is out of the range. In Workpiece origin offset function, the offset value is out of the range. Set it according to a set unit.
NG SETTING (OFFSET NUM OVERLAP)	The offset number of Work coordinate system shift or Workpiece origin offset overlaps. Specify the number not used.

(7) Tool management related

Message	Description
PLEASE CHANGE TO EDIT MODE	In tool management function, data change operation was done on magazine management table screen, magazine property screen, pot property screen, tool management data screen, each tool data screen or tool geometry data screen in the state that was not tool management edit mode. Change the mode to tool management edit mode before changing the data by pressing softkey [EDIT].
NUMBER IS NOT REGISTERED	In tool management function, tool was not found by searching tool management number on magazine management table screen, or by searching tool type number on total life data screen.
CAN NOT FILTER	In tool management function, an item other than the edge group number was specified for extracting. Move the cursor to edge group number and then operate extracting.
NOW ATTACHING	In tool management function, though tool attachment was not completed, other operation was started. Start the operation after tool attachment is completed.
NOW DETACHING	In tool management function, though tool detachment was not completed, other operation was started. Start the operation after tool detachment is completed.

FRYER MACHINE SYSTEMS INC
Rob Zaharensky

Message	Description
TOOL MOVING	In tool management function, though tool movement was not completed, other operation was started. Start the operation after tool movement is completed.
TEMP OUT EXECUTING	In tool management function, tool attachment, temporary detachment, or changing tool management data number was operated on temporary detaching pot. Cancel the state of temporary detaching and then start the operation.
TOOL NUMBER NOT FOUND	In tool management function, specified tool number was searched, but it was not found. Check if the tool number is correct.
THE SPECIFIED TOOL HAS BEEN ATTACHED	In tool management function, the tool which was already registered in the magazine was attempted to attach. Select the tool which is not registered in magazine and attach it.
FORMAT ERROR	Tool management data values was inputted continuously (Example: 100;200) on the each tool data screen Input a tool management data which divided into one data.
CLEAR DATA	In Tool management function, the magazine number or the pot number is deleted.
INPUT TOOL TYPE NUMBER	In Tool management function, tool kind searched number is not input. Execute the search after inputting tool kind number.
EDITING	In Tool management function, the tool management data is being edited.
NO DETAIL LIFE SCREEN	In tool management function, detail life data screen cannot be displayed because there is no tool type number to which life management is effective. Check the setting of tool information bit 0 and tool life status of tool management data.
NOT TURN ABNORMAL SIGNAL OFF	In tool management function, tool attachment, detachment, or movement was operated under tool attachment/detachment/movement error signal was "1". Operate after the error signal becomes to "0".
NOT TURN FINISH SIGNAL OFF	In tool management function, tool attachment, detachment, or movement was operated under tool attachment/detachment/movement finish signal was "1". Operate after the finish signal becomes to "0".
TOOL INTERFERES	In tool management function, interference is generated by attaching or moving a tool. Check if the pot number to which the tool is attached or the destination of the tool is correct.
MAGAZINE NUMBER ERR	In tool management function, when tool registration, movement, attachment, or setting of an origin position was executed, an illegal magazine number was specified. Specify an available magazine number.
POT NUMBER ERR	In tool management function, when tool registration, movement, attachment, or setting of an origin position was executed, an illegal pot number was specified. Specify an available pot number.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Message	Description
INPUT IS REJECTED	(a) Time type value (unit of 8ms) was attempted to set to the tool of count type. Change the life count type to the type for cutting time. (b) Tool geometry data of the tool that was registered in a magazine was attempted to change. Change the tool geometry data after detaching the tool from the magazine. (c) When tool kind of tool geometry size data was not set, data of the other items was attempted to input. Input it after setting the tool kind. (d) Second geometric tool offset was attempted to input though it is in the state that cannot be input. Don't input second geometric tool offset. (e) Offset value was attempted to input though it is in the state that offset number was not set in the tool management data. Input the offset value after setting the tool offset number. (f) Tool geometry size data was attempted to input though it is in the state that offset number was not set in the tool management data. Input tool geometry size data after setting the tool offset number. (g) Tool geometry size data was attempted to input though it is in the state that illegal offset number was set in the tool management data. Input tool geometry size data after setting correct tool offset number. (h) An attempt was made to input a value to unusable tool data of tool geometry size data. Check if the tool kind is correct. (i) Nonexistent magazine or pot number was attempted to input on tool management data screen or each tool data screen. Check if the magazine and pot number is correct.
ORIGIN POSITION DUPLICATED	The tool having the same origin position was already existed. Specify other origin position.
ORIGIN POSITION ERR	Spindle position or stand by position was specified as an origin position. Specify magazine 1 to 8 as an origin position.
L-COUNT TYPE MATCH	The life count types of all tools with same tool type number match.
L-COUNT TYPE MISMATCH:	The tools with same tool type number have deferent life count types, which are number of use and cutting time. Match to either the number of use or cutting time.
ABNORMAL END(ATTACH)	In tool management function, errors occurred while attaching a tool. Check the machine.
NOT EMPTY POT	Specifying the tool destination or the attaching the tool to the pot where a tool was already registered. Specify the pot where a tool is not registered.
ABNORMAL END(MOVE)	In tool management function, errors occurred while moving a tool. Check the machine.
ABNORMAL END(DETACH)	In tool management function, errors occurred while detaching a tool. Check the machine.
THERE IS NO TOOL IN THE SPECIFIED POT	An attempt was made to detach or move at the pot where a tool was not registered. Select the pot where a tool is registered.

FRYER MACHINE SYSTEMS INC
Rob Zahansky

Message	Description
ERROR (PROT)	Tool attachment was executed with magazine management table protected by 8-level data protection function. Raise the operation level.

(8) Graphic related

Message	Description
NO MORE EXPANSION	In dynamic graphic display function, operation to increase the drawing scale was done when the scale was already maximum.
NO MORE REDUCTION	In dynamic graphic display function, operation to reduce the drawing scale was done when the scale was already minimum.
NO PROGRAM SELECTED	In dynamic graphic display function, drawing began without selecting the program. Select the program at first.

(9) Other

Message	Description
THERE IS NO SPECIFIED NAME OF AXIS	(a) On work coordinates screen, the axis specified by measured value input or counter input does not exist. Confirm the axis name. (b) On position screen, the axis specified by workpiece coordinate system preset or resetting relative coordinates does not exist. Confirm m the axis name. (c) On the manual handle interruption screen, the axis specified by clearing the amount of interruption does not exist. Confirm the axis name. (d) For approaching the program restart position per arbitrary axis, the axis specified by the movement axis setting does not exist on the program restart screen. Confirm the axis name.
DATA NOT FOUND	The signal address that did not exist was searched on the safety I/O signal history screen of dual check safety function. Confirm the searched signal address.
DATA SETTING ERROR	The combination of the waveform data kind of settings to which the trouble diagnosis parameter screen can be displayed at the same time is wrong. Set the waveform data kind of the combination that can be displayed at the same time.
FORMAT ERROR	The format of input data such as parameters and offset etc by MDI key is wrong. Input in correct format.
ILLEGAL LEVEL	In 8 level data protection setting screen, the operation is forbidden under the present operation level was operated. Operate it after the limit of the operation is released.
INCORRECT CHARACTER CODE IS CONTAINED	The character code that cannot be used is input in periodic maintenance screen. Input the allowed character code.
LACK OF DATA	The password for 8 level data protection is too short. Set the password from 3 to 8 characters.
EMPTY POT IS NOT FOUND	Empty POT for the Oversize Tool was not found. Detach an unnecessary tool, and have empty POT.
OUTPUT ERROR(PMC PARAM)	Output of the PMC parameter failed. Output it again after confirming the output device.
IT IS NOT EXECUTED	No hole machining cycle has been operated in quick program restart for a machining cycle in MANUAL GUIDE i. After performing automatic operation, press soft key [MACHIN CYCLE].

FRYER MACHINE SYSTEMS INC
Rob Jankowsky

Message	Description
WORK COORDI. SHIFT AMOUNT IS BEING SET	The cursor movement being limited by the Direct Input of Offset Value Measured B function, the cursor was moved. After completion of the measurement, move the cursor.
WRITE PROTECT	For protected data such as "8-level data protection and key signal", data was entered. By releasing the protection, enter your data.
ILLEGAL CHUCK DATA(L<=L1)	In Chuck and Tail Stock Barrier function, setting of the chuck is incorrect. Set to be L<=L1.
ILLEGAL CHUCK DATA(W<=W1)	In Chuck and Tail Stock Barrier function, setting of the chuck is incorrect. Set to be W<=W1.
AUTO SET IS PROHIBITED	FSSB automatic setting is forbidden. Check bit 0 (FMD) of parameter No.1902.
NUMBER OF ICON IN CATEGORY IS 0	The category to which any icon is not registered exists in main menu screen. Register one or more icons in any category.
REGISTRATION PROHIBITION SCREEN	The registration of the screen to main menu screen was tried. The object screen cannot be registered in main menu screen.
DIFFERENCE WITH NEAR IS RANGE OVER	In Stored pitch error compensation total value input function, the difference between two consecutive data is range over. Adjust the difference between two consecutive data within the range of -128 to +127.
NOT DELETE	(a) In Main menu function, the icon can not be deleted. (b) In paste of the program, the program can not be deleted. Check the program has not been protected."
EXECUTING AUTOMATIC OPERATION	In High-speed program management function, saving the program was executed during the operation of the program. Save the program after stopping operation.
ARE YOU SURE YOU WANT TO EXECUTE?	On the titled working plane index setting screen, this is displayed to confirm that data is inserted or overwritten.
PROGRAM HAS BEEN VERIFIED	In the program verification function, the verification is success between the program of the I/O device and the program of the CNC build-in memory.
FAILED TO SAVE PROGRAMS	In the high-speed program management function, this is displayed when program registering is failed. (without automatic operation executing) Try again later.
THERE IS NO INIT-VALUE	On the parameter setting screen and axis setting screen, this is displayed when the initialization is not executed.
ID DATA COMPLETE	On the servo information screen, the ID has already been acquired when the servo ID information reading operation is executed. This message means that ID acquisition is finished, so next operation is not necessary.
NOT READ ID	(a) The servo ID information was not correctly read in the servo information screen. Check whether servo motor and amplifier are correctly connected. (b) The spindle ID information was not correctly read in the spindle information screen. Check whether spindle motor and amplifier are correctly connected.
ABORT END	The transmission of the learning data was interrupted because softkey [STOP] is pressed while transmitting the learning data in the learning control screen.

FRYER MACHINE SYSTEMS INC
ROB ZABRECKY

Message	Description
ERROR END	The error occurred while transmitting learning data in the learning control screen. Check whether the transmission of the learning data is possible.
NORMAL END	The transmission of the learning data ended correctly in the learning control screen.
CHECK CODE CAN NOT BE REGISTERED	The check code can not be registered because registration object such as C language executor program doesn't exist in the data modification detection setting screen. Check whether the registration object exists.
OPERATION IS ENABLED	The password has been input and the operation has become possible in the data modification detection setting screen.
INPUT PASSWORD AGAIN	(a) When the password was changed, a new password was input once. (b) When the password is changed, the message is generated on the parameter screen and warning function against modification of setting. (c) When the password of the following parameters is changed, the message is generated on the parameter screen in case of bit 6 (RPW) of parameter No.11391is set to 1. - No.3210 Program protection (PSW) - No.3220 Password (PSW) - No.11311 Password for embedded macro - No.3225 Password for protecting against modifications to parameters related to Dual Check Safety function Input the same password again for confirmation.
PASSWORD CHANGED	(a) The password was changed. (b) When the password is changed, the message is generated on the parameter screen and warning function against modification of setting. (c) When the password of the following parameters is changed, the message is generated on the parameter screen in case of bit 6 (RPW) of parameter No.11391is set to 1. - No.3210 Program protection (PSW) - No.3220 Password (PSW) - No.11311 Password for embedded macro - No.3225 Password for protecting against modifications to parameters related to Dual Check Safety function.
ILLEGAL CHARACTER USED	In the warning function against modification of setting, when the password was input, the character that was not able to be used was input. Input the following characters. "A" - "Z", "a" - "z", "0" - "9"
PASSWORD TOO LONG	In the warning function against modification of setting, a too long password was input. Input the password up to 16 characters.

FRYER MACHINE SYSTEMS INC
Rob Zaremski

Message	Description
PASSWORD MISMATCH	(a) A wrong password was input. (b) When the password is input, the message is generated on the parameter screen and warning function against modification of setting. (c) When the password of the following parameters is changed, the message is generated on the parameter screen in case of bit 6 (RPW) of parameter No.11391is set to 1. - No.3210 Program protection (PSW) - No.3220 Password (PSW) - No.11311 Password for embedded macro - No.3225 Password for protecting against modifications to parameters related to Dual Check Safety function Input a correct password.
SPECIFIED PASSWORD CAN NOT BE SET	In warning function against modification of setting, the same password as an initial password of the shipment of CNC was input. Input other passwords.
SET FORMAL PASSWORD	In warning function against modification of setting, an initial password of the shipment of CNC is set. Set formal passwords other than an initial password.
MEMORY CARD IS NOT READY.	In memory card program operation/editing function, the memory card is not mounting. Mount the memory card.
THE MEMORY CARD CAN BE TAKEN OUT	In memory card program operation/editing function, the memory card can be detached.
MEMORY CARD IS USED BY OTHER FUNCTION	In memory card program operation/editing function, the device was changed or the memory card was detached while using the memory card. Detach the memory card or change the device after the use of the memory card is canceled or use ends.
MEMORY CARD IS USED BY AUTOMATIC OPERATION	In memory card program operation/editing function, the memory card is used by automatic operation. Detach the memory card after the use of the memory card is canceled or use ends.
NO EXTENSION OPTION	In memory card program operation/editing function, the number of programs registered on the memory card has exceeded 63. When the program that exceeds 63 is used, the option of the memory card program registration number enhancing is necessary.
THIS DEVICE IS NOT AVAILABLE	In memory card program operation/editing function, the program in the device of "MEM CARD" was edited. This program can not be edited on CNC.
NG SETTING (WORK COORD VAL OVERLAP)	In wrong operation prevention function, the valid setting range of the workpiece zero point offset overlaps. Set the valid setting range correctly.
THE STRING IS NOT FOUND	In the Search Word of the extended program restart function, search word does not exist. The address specified in the symbol screen or I/O diagnostic screen does not exist. Retry after confirmation of the string you want to search.
SYMBOL UNDEFINED	In the real time macro range select screen, the start number or end number is input though the address is not set. Set the address previously.
INPUT INVALID	(a) The address specified in the safety IO signal history screen does not exist. (b) Input on the module configuration screen is invalid. Retry after checking the input value.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Message	Description
COMMAND ILLEGAL USE	In the Approach for Each Arbitrary Axis in Program Restart, incorrect axis is specified. In the Suppress Motion of Quick Program Restart, incorrect axis is specified. Or, the program is not being restart state Try again after specifying the correct axis in the program restart state or in the state that automatic operation has stopped.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

I PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING

I.1 PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING

Overview

By using this PC tool, you can make the memory card program file ("FANUCPRG.BIN") which is needed for the function "Memory Card Program Operation/Editing".
The maximum size of the memory card program file is 2048 Mbytes (2 Gbytes). The "Memory Card Program Operation/Editing" function needs the memory card which has the memory card program file on the FAT16 formatted memory card.

This PC tool can be operated on a PC in the marketplace and with following OS :
- Windows(R)NT4.0 Workstation (SP5 or later)
- Windows(R)2000 Professional (SP4 or later)
- Windows(R)XP Professional (SP2 or later)
- Windows(R)Vista Ultimate
- Windows(R)7 Professional
- Windows(R)8 Pro
- Windows(R)10

And acceptable specification is as followings :
- Memory : 32Mbytes or more
- Hard disk : 10Mbytes or more free space and additional space for the memory card program file

I.1.1 Usage Notes

Before using this PC tool, please make sure there is no [temp] folder on the same place of this PC tool.
[temp] folder is created and used by this PC tool as work folder.
If [temp] folder is created, please do not access this folder.
[temp] folder and the files located in this folder will be deleted by this PC tool.

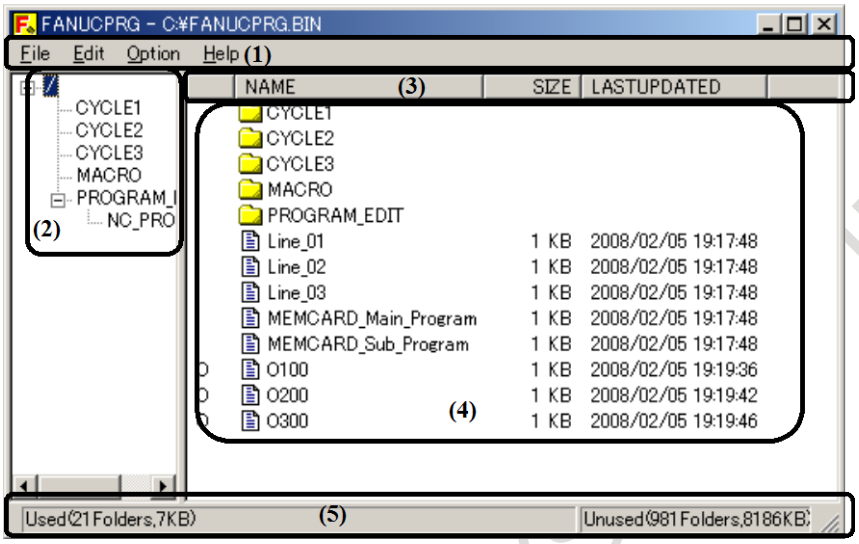
I.1.2 List of Functions of PC Tool

- Browsing the folders of the memory card program file
- Adding a program to the memory card program file by dropping the program into this PC tool from Explorer etc. (hereinafter referred to as "drop in")
- Extracting a program in the memory card program file as a text file on Windows file system by dropping out from this PC tool into Explorer etc. (hereinafter referred to as "drop out")
- Renaming a program in the memory card program file
- Deleting a program in the memory card program file
- Creating a new folder into the memory card program file
- Renaming a folder in the memory card program file
- Deleting a folder in the memory card program file
- Display of free space on the memory card program file
- Sorting list view of the memory card program file

FRYER MACHINE SYSTEMS INC
Rob Zahensky

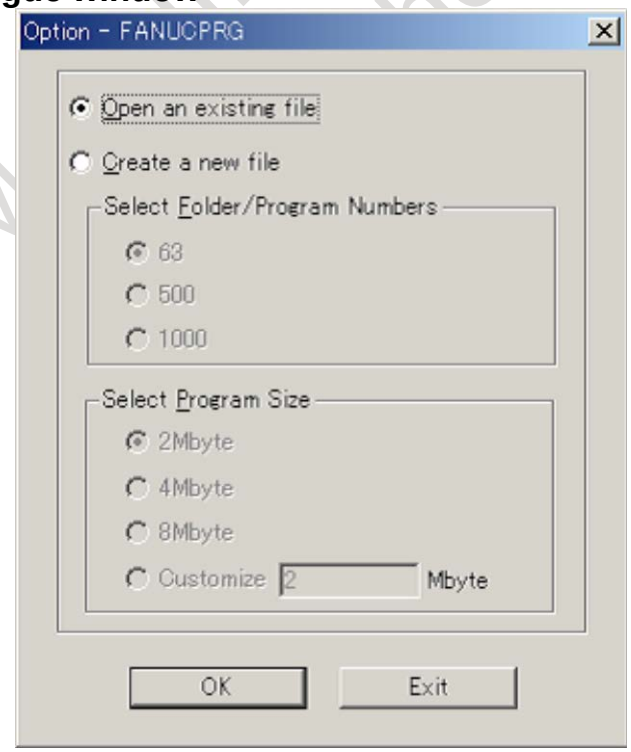
I.1.3 Explanation of Operations

- Outline of screen



(1) Menu bar : The menu of this PC tool is displayed.
(2) Tree view : Browsing the folders of the memory card program file.
(3) Column : Attributes of each file or folder in the memory card program file.
(4) List view : Contents of selected folder are displayed.
(5) Status bar : Used and Unused space in the memory card program file are displayed.

- Initial Option dialogue window

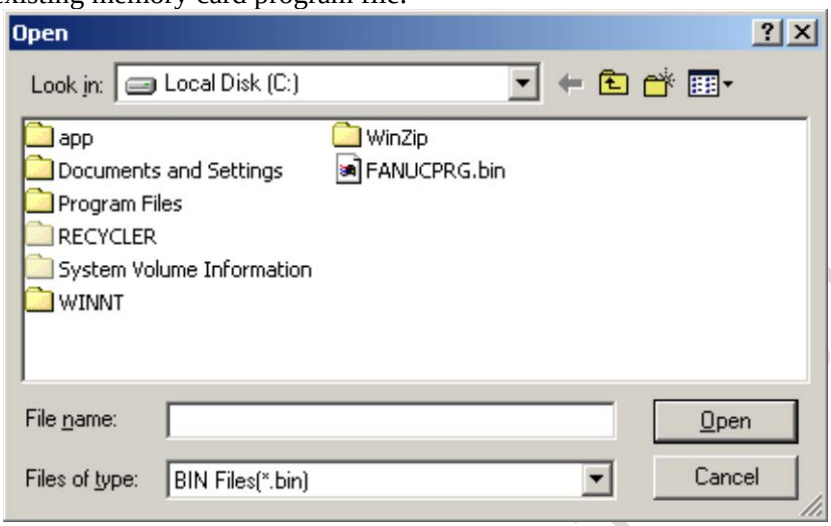


When this PC tool starts up, the Option dialogue window is displayed. Please select "Open an existing file" or "Create a new file".

FRYER MACHINE SYSTEMS INC
Chensky

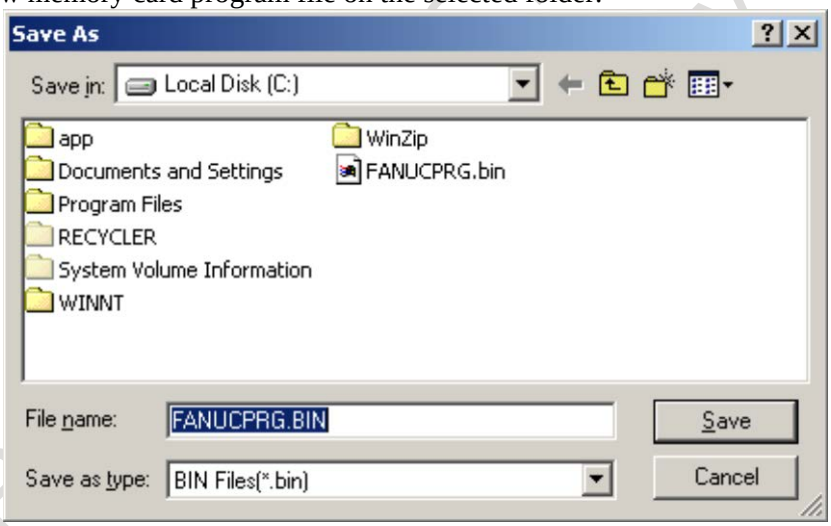
When "Open an existing file" is selected

After OK button pushed, "Open" dialogue window is displayed.
Please select the existing memory card program file.



When "Create a new file" is selected

After OK button pushed, "Save As" dialogue window is displayed.
Please create a new memory card program file on the selected folder.



When the new memory card program file is created, the following items need to be selected:

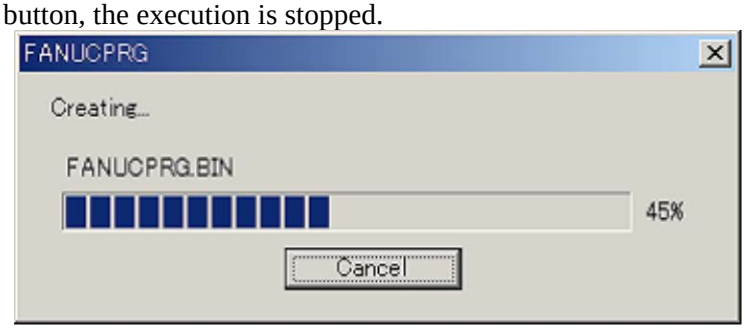
- Folder/Program Numbers
- Program Size

"Folder/Program Numbers" can be selected among 63 / 500 / 1000. The default value is 63.
"Program Size" can be selected among 2Mbyte, 4Mbyte, 8Mbyte, and Customize. The default value is 2Mbyte.

NOTE

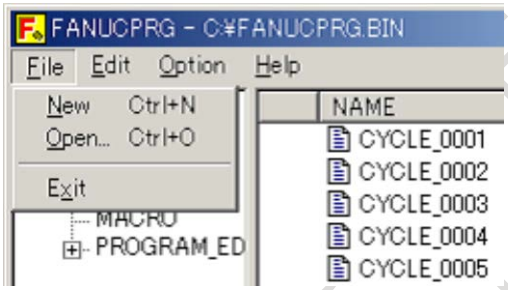
1 When "Customize" size is selected, it is available to cover the range from 2Mbyte to 2048Mbyte.
2 Though the maximum size is 2048Mbyte, a slight loss exists by system use.
3 Nonnumeric value cannot be input.
4 The number of "Folders" in the status bar includes number of program file and folder.

During creating of the memory card program file, the progress bar is being displayed. This progress bar is also displayed during executing of Drop-in and Drop-out.
If you push [Cancel] button, the execution is stopped.



- Menu

File menu

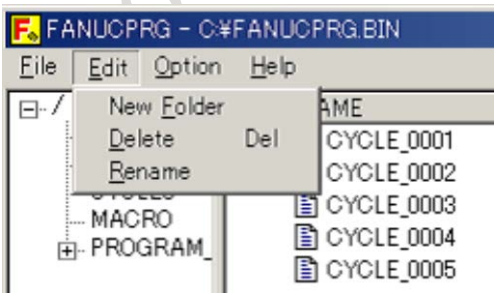


[New]
Create a new memory card program file.

[Open...]
Open the existing memory card program file.

[Exit]
Terminate this PC tool.

Edit menu



[New Folder]
Create new folder. It is available during Tree view selected.
Up to seven hierarchical levels starting from the user root folder (/USER) are permitted.
/USER/PATH1/Aaa/Bbb/Ccc/Ddd/O123
1 2 3 4 5 6 7(only files)

[Delete]
Delete program files or folders.
If you delete a folder, all folders and program files in the folder will be deleted.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

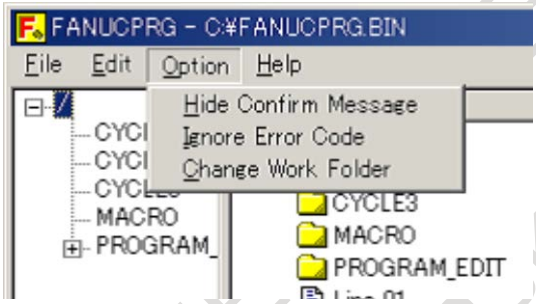
[Rename]

Rename a folder or file.

NOTE

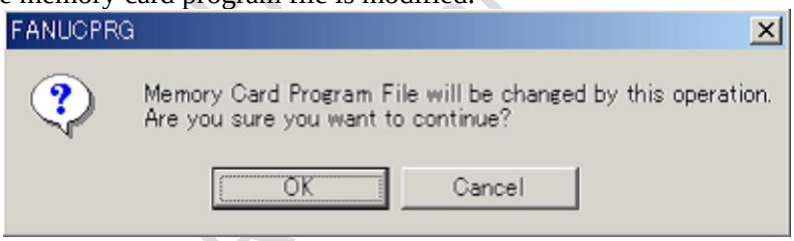
For naming folder and program file, characters which can be used are limited. Please refer to "Naming rules".

Option menu



[Hide Confirm Message]

When the following operations are executed, the following Confirm Message is popped up before the memory card program file is modified.



(1) Delete folder or program file
(2) Rename folder or program file
(3) Drop-in program file
(4) Add folder

If the [OK] button is pushed, the operation advances.
If the [Cancel] button is pushed, the operation is abandoned.

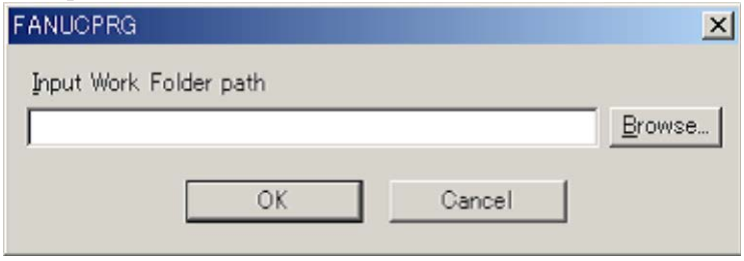
When the [Hide Confirm Message] in menu bar is checked, the Confirm Message is not displayed and the operation advances at once. The default setting is the Confirm Message displayed.

[Ignore Error Code]

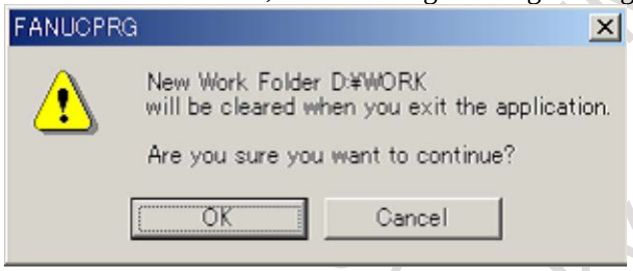
When the [Ignore Error Code] in menu bar is checked, program file which includes unusable characters can be dropped in. The unusable characters in the program file will be ignored and will not be written into the memory card program file.

FRYER FINE SYSTEMS INC

[Change Work Folder]

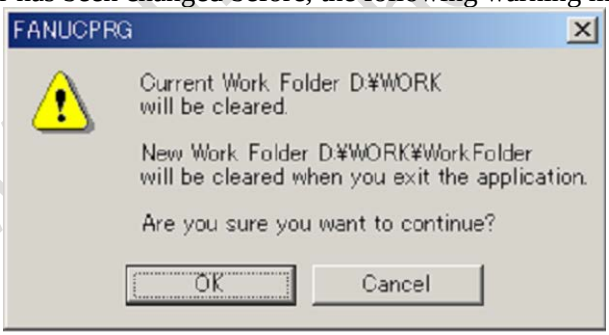


Work folder is used for temporarily keeping the dropped out files. If work folder has no enough free space, Drop-out will not be executed.
To avoid this, you can check this option and change the work folder to an area where there is enough free space. With default setting, work folder [temp] will be created on the same place of this PC tool "FANUCPRG.exe".
If you changed the default work folder, the following warning message is popped up.



If the [OK] button is pushed, the operation advances.
If the [Cancel] button is pushed, the operation is abandoned.

If the work folder has been changed before, the following warning message is popped up.



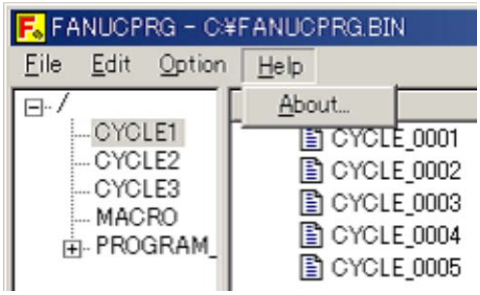
If the [OK] button is pushed, the operation advances.
If the [Cancel] button is pushed, the operation is abandoned.

⚠ CAUTION

1 At this PC tool terminated, all files in the work folder are deleted.
2 During this PC tool executing, do not access the files in the work folder. If the access is done, there is no assurance of normal operation.

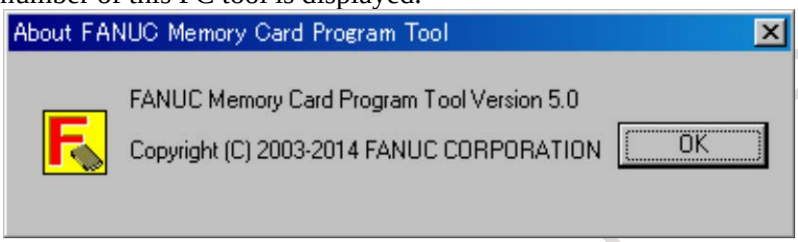
FRYER MACHINE SYSTEMS INC

Help menu



[About...]

Version number of this PC tool is displayed.



- Mouse Operation

[Drop-in and Drop-out]

- Drop-in from Explorer

  NC program can be added by dropping files including the NC files into the List view window of this PC tool from Explorer

  NC program name and update time is the same as the Dropped-in files.

  If "Oxxxxxxxx" or "<xxxx>" exists at the top of files, These "Oxxxxxxxx" and "<xxxx>" become NC program name. If not exist, each file name becomes NC program name.

Examples			
Name of external file	**Top of external file**	**Name of internal file**	**Program number**
O1234	N10G00	O1234	1234
O123N10G00	N10G00	O123N10G00	Not an O number program
test.txt	O1234N10G00	O1234	1234
test.txt	<O1234>	O1234	1234
test.txt	<O1234N10>	O1234N10	Not an O number program
O1234	<O1234N10>	O1234N10	Not an O number program
O001234	N10G00	O1234	1234
O001234N10G00	N10G00	O001234N10G00	Not an O number program
test.txt	O001234	O1234	1234
test.txt	<O001234>	O1234	1234
test.txt	<O001234N10G00>	O001234N10G00	Not an O number program

NOTE

1 For naming program file, please refer to the following chapter "Naming rules of Program file".

2 For usable characters in Program file, please refer to the following chapter "Rules of characters in Program file".

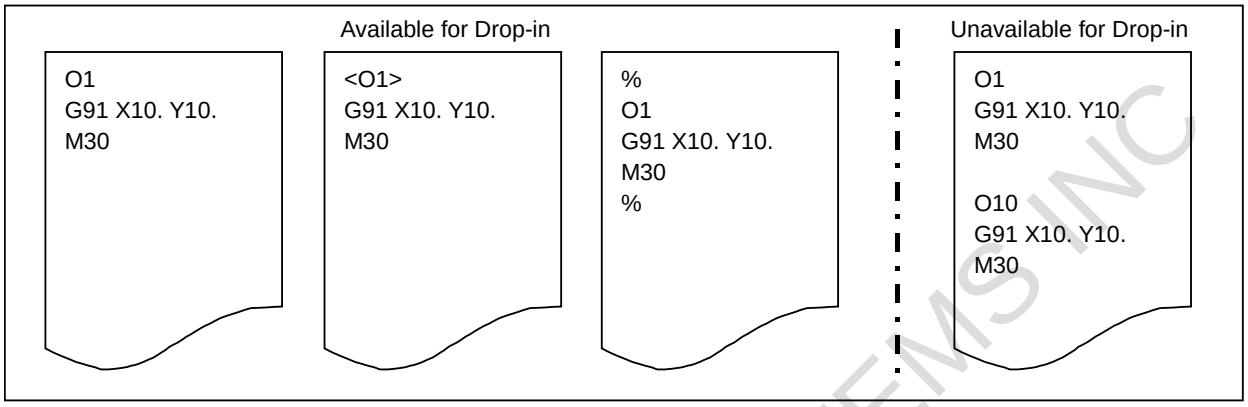
3 The updating time of program file is available from 1997 to 2037.

This PC tool can check contents of dropped-in program file according to "Rules of characters in Program file". However, this PC tool does not check grammar of NC program.

FIXER MACHINE SYSTEMS INC
Rod Zahensky

The program file can only have one NC program. Therefore, the NC program output from CNC cannot be dropped in directly.

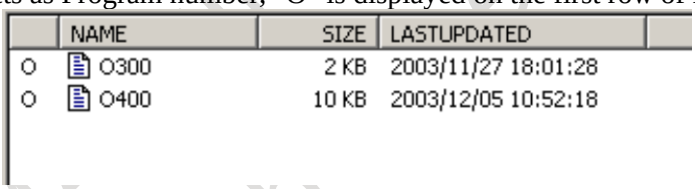
Example of Program



NOTE

1 If the same named program file exists, another same named program file cannot be dropped in.
2 If free space on the memory card program file is lacking, new program file cannot be dropped in.
3 If the NC program name is not compliant with "Naming rules of Program file", the program file can not be dropped in.

If the File name acts as Program number, "O" is displayed on the first row of list view.



- Drop-out from List view window
  Drop-out from list view of this PC tool to Explorer is available.

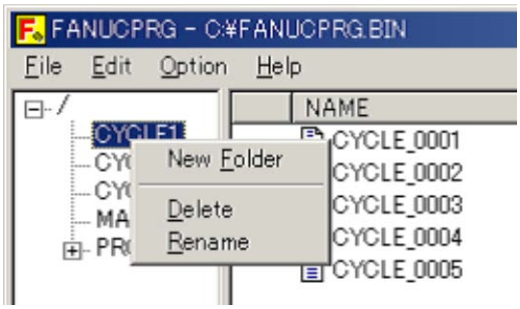
⚠ CAUTION

Do not drop out to Work folder. If dropped out to Work folder, this PC tool cannot continue to function normally.

- Pop-up menu

Pop-up menu is displayed by clicking the right mouse button.

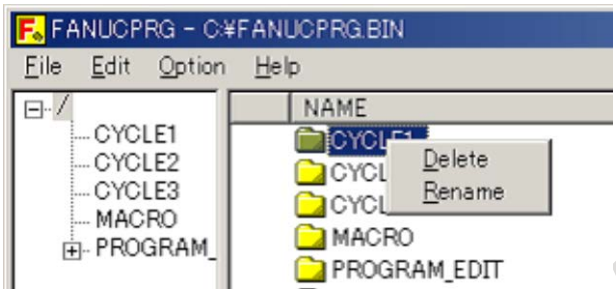
- Focus on Tree view



FRYER MACHINE SYSTEMS INC
Robert Jensky

Clicking "New Folder", a new folder is created on selected folder.
Clicking "Delete", the selected folder is deleted.
Clicking "Rename", the selected folder is renamed.
If clicking on root folder, "Delete" and "Rename" are not activated.

- Focus on List view



- Clicking "Delete", the selected folder or program file is deleted.
- Clicking "Rename", the selected folder or program file is renamed.

- Display of free space on the memory card program file ("FANUCPRG.BIN")

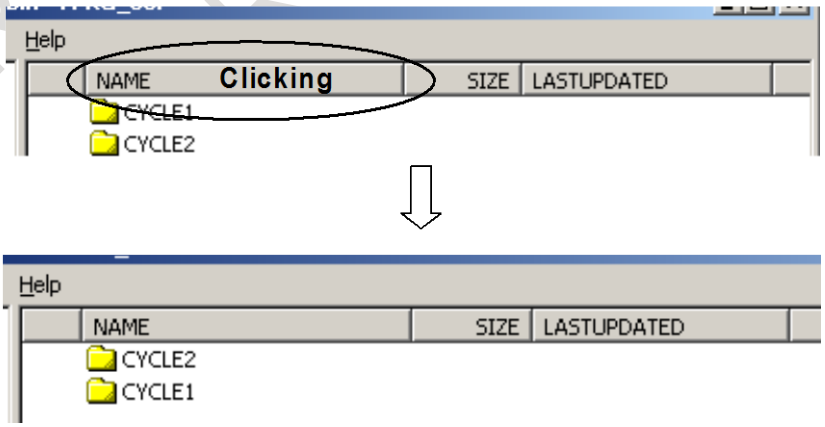
Number of used folder, size of used space, number of unused folder, and size of free space are displayed on status bar in the lower portion of the screen.

When a new memory card program file is created, two reserved folder are also created. Therefore, the number of used folder is two. However it does not show that number of unused folder is reduced.

The display of status bar is renewed at creating or deleting a folder, dropping-in from Explorer, and deleting a program file.

- Sorting list view of the memory card program file

When a column is being clicked, the list view of the memory card program file is being sorted by the column key in ascending or descending order.
The initial display is sorted by NAME key in ascending order.



FRYER MACHINE SYSTEMS INC
Rob Zahensky

I.2 NAMING RULES

Overview

Naming rules of folder and program file are described as follows.

I.2.1 Naming Rules of Program File

Here are Naming rules of Program file:

- Program file name can have a maximum of 32 characters.
- Program file name can have following characters.
  Alphabet(Upper and lower case letter), numeric character,
  "-"(minus), "+"(plus), "_"(under bar), "."(dot)
  "." and ".." can not be used since these are reserved for system use.

- The File name acts as Program number

When the File name is "O"+ 1-99999999, the file name acts as Program number.

Example)

"O123"	Program number 123
"O1"	Program number 1
"O3000"	Program number 3000
"O99999999"	Program number 99999999
"O0123"	Program number 123

The file name does not act as Program number, and can be dropped in:

"ABC"	Top of character is not upper case letter of "O".
"o123"	Top of character is not upper case letter of "O".
"O0123XY"	Characters following the "O" contains alphabet.

The file name does not act as Program number, and cannot be dropped in:

"O123456789"	Numeric characters exceed 8 digits.

NOTE

1. Program file name cannot be repeated in a Folder.
2. If program file name starts with "O" and the next eight characters are all numeric characters, the "0" (zero) after the "O" will be deleted.

I.2.2 Naming Rules of Folder

Here are Naming rules of Folder:

- Folder name can have a maximum of 32 characters.
- Folder name can have following characters.
  Alphabet(Upper and lower case letter), numeric character,
  "-"(minus), "+"(plus), "_"(under bar), "." (dot)
  "." and ".." can not be used since these are reserved for system use.

NOTE

Folder name cannot be repeated in a Folder.

FRYER MACHINE SYSTEMS INC
Rob Zaborsky

I.3 RULES OF CHARACTERS IN PROGRAM FILE

Overview

Words in parentheses "( )" in Program file are treated as comments.
The mark of comment start "(" is named "Control-out".
The mark of comment end ")" is named "Control-in".
"Control-out" and "Control-in" must make a pair. The order is 1st - "Control-out" and 2nd -"Control-in".
And nested parentheses is not available.

NOTE

1. When a program file is dropped in, space code (0x20 SPC) , tab code (0x09 HT) , carriage return code(0x0d CR) and percent code(0x25 %) is deleted. If space code is found in Control-out (characters in parentheses), it is not deleted. If "%" is found in Control-in, characters between "%" and next "LF" (0x0a) is deleted.
2. The front of program number ":" is changed to "O"(O as in Oscar) while the program file dropped in.
3. The program file can only have one NC program.

I.3.1 Usable Characters in Program File

- Usable characters in Control-in

List of ANSI(ASCII) codes of usable characters(hexadecimal form)

Code	Character	Code	Character	Code	Character	Code	Character
0a	LF	3f	?	58	X	74	t
23	#	40	@	59	Y	75	u
26	&	41	A	5a	Z	76	v
28	(	42	B	5b	[	77	w
29	)	43	C	5d	]	78	x
2a	*	44	D	5f	_	79	y
2b	+	45	E	61	a	7a	z
2c	,	46	F	62	b		
2d	-	47	G	63	c		
2e	.	48	H	64	d		
2f	/	49	I	65	e		
30	0	4a	J	66	f		
31	1	4b	K	67	g		
32	2	4c	L	68	h		
33	3	4d	M	69	i		
34	4	4e	N	6a	j		
35	5	4f	O	6b	k		
36	6	50	P	6c	l		
37	7	51	Q	6d	m		
38	8	52	R	6e	n		
39	9	53	S	6f	o		
3a	:	54	T	70	p		
3c	<	55	U	71	q		
3d	=	56	V	72	r		
3e	>	57	W	73	s		

FRYER MACHINE SYSTEMS INC Rob Zaremsky

NOTE
In the Control-in, "O", ":", and "<" can not be used at top of the line except for the 1st line.

- Usable characters in Control-out(characters in parentheses)

List of ANSI(ASCII) codes of usable characters(hexadecimal form)

Code	Character	Code	Character	Code	Character	Code	Character
0a	LF	3c	<	55	U	71	q
20	SPC	3d	=	56	V	72	r
22	“	3e	>	57	W	73	s
23	#	3f	?	58	X	74	t
24	$	40	@	59	Y	75	u
26	&	41	A	5a	Z	76	v
27	‘	42	B	5b	[	77	w
2a	*	43	C	5d	]	78	x
2b	+	44	D	5f	_	79	y
2c	,	45	E	61	a	7a	z
2d	-	46	F	62	b		
2e	.	47	G	63	c		
2f	/	48	H	64	d		
30	0	49	I	65	e		
31	1	4a	J	66	f		
32	2	4b	K	67	g		
33	3	4c	L	68	h		
34	4	4d	M	69	i		
35	5	4e	N	6a	j		
36	6	4f	O	6b	k		
37	7	50	P	6c	l		
38	8	51	Q	6d	m		
39	9	52	R	6e	n		
3a	:	53	S	6f	o		
3b	;	54	T	70	p		

I.4 ERROR MESSAGE AND NOTE

Error may occur when using this application, hereafter explains the error messages and gives relative instructions.

I.4.1 List of Error Message

When an error occurred, the error message box is displayed as follows.

Message	Remarks
Failed to open the file you specified.	If failed to open once again, the file may be broken.
Failed to read or write to the specified file.	Reading or writing program storage file failed.
There is insufficient disk space.	There is no enough free space to create a new memory card program file or to put drop out files to work folder. In the latter case, please refer to chapter "Menu" [Change Work Folder]
File name is not correct.	Please refer to the chapter "Naming rules of Program file".
Input name is already exists.	Please input another name.
File name is already exists.	Please input another name.
Input name is not correct.	Please refer to the chapter "Naming rules".

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

Message	Remarks
Please input an integer between 2 and 2048	The size of the memory card program file is available from 2Mbyte to 2048Mbyte.
An illegal character is included in the specified file.	Please refer to the chapter "Rules of characters in Program file".
Last update time of the specified file is unsupported.	The updating time of program file is available from 1997 to 2037.
The memory card program file you specified cannot be identified.	The specified file format is not the memory card program file
There is insufficient free folder.	Please delete unnecessary program files or folders.
There is insufficient free program space.	Please delete unnecessary program files.
Folder deeper than 7 cannot be created.	Attempted to create a folder beyond seven levels. Can't create folders after the eighth level.
Root folder cannot be deleted.	Attempted to delete the root folder.
Root folder cannot be renamed.	Attempted to rename the root folder.
Failed to create work folder.	Please check whether the [temp] work folder is able to create or not. In default, the [temp] work folder is created on the same place of this PC tool "FANUCPRG.exe".
Process has been cancelled.	Processing was interrupted.
The specified work folder is not found.	The Drop-out cannot be executed. Terminate this PC tool. And check the setting of Work Folder on the Option dialogue window.
Only one instance of this application can be executed.	Double activation of this PC tool cannot be executed.

I.4.2 Note

- Folder and Program Numbers

Folder and Program Numbers can be selected among 63 / 500 / 1000.
For selecting the Numbers of 500 or 1000, the option "Registered programs expan. On the memory card" is needed in CNC side.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

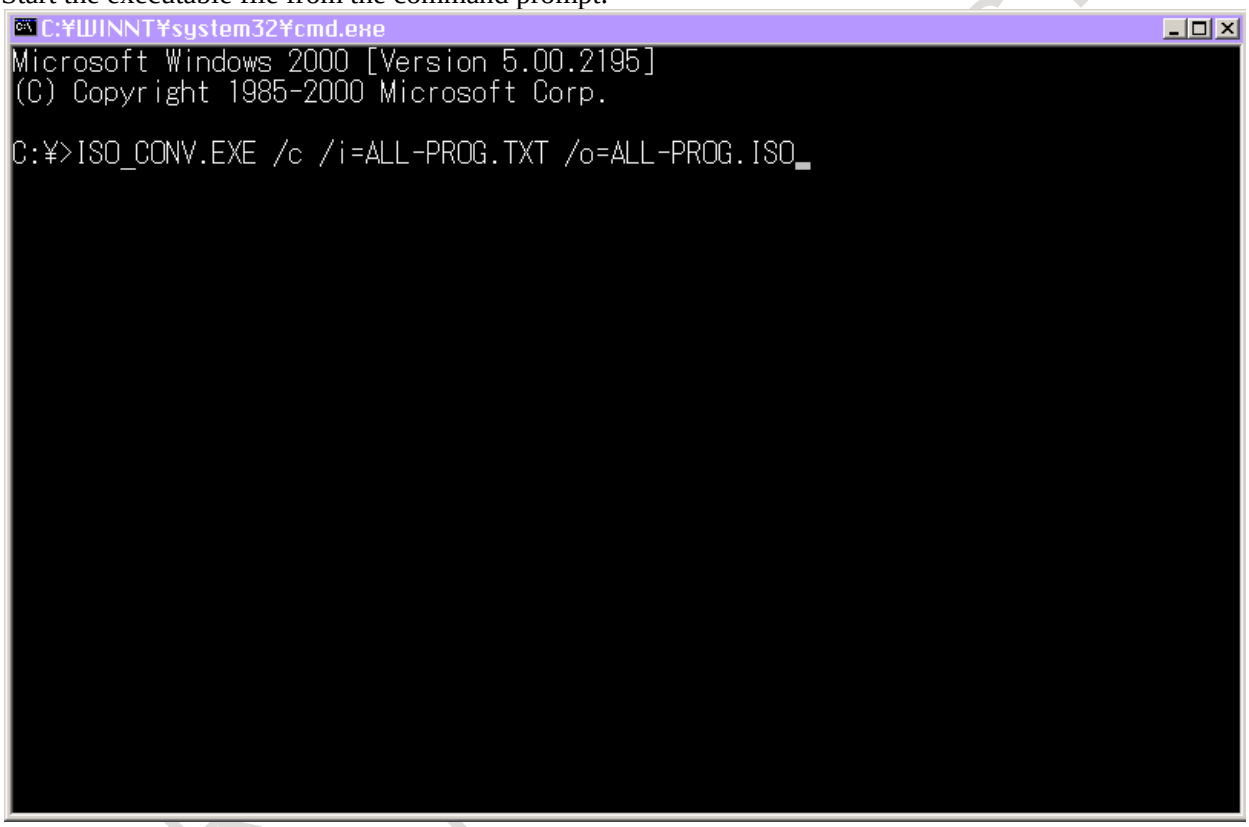
J ISO/ASCII CODE CONVERSION TOOL

Overview

FANUC ISO Converter is a tool that converts a file created or externally output with ASCII code to ISO code format.
This tool runs on Windows 2000, Windows XP, and Windows Vista.
This tool can be used in two modes, CUI and GUI.

CUI

Start the executable file from the command prompt.



Specify the following arguments when starting the tool.

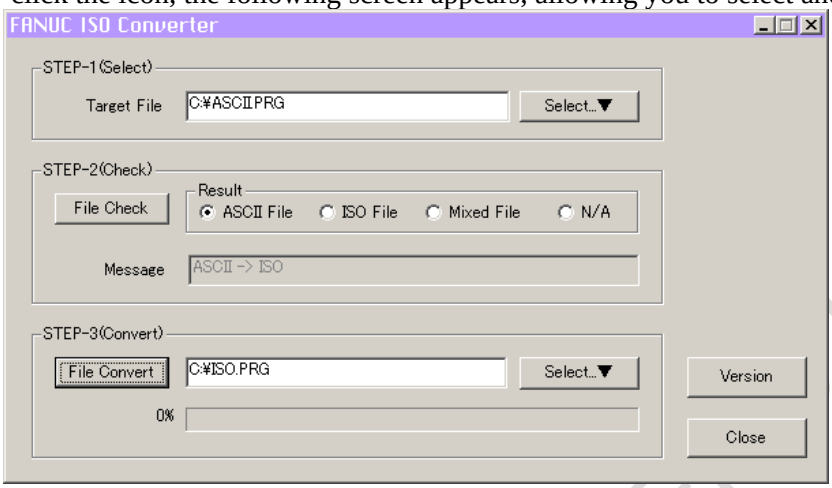
/c : Specify that the tool runs in command line mode.
When /c is omitted, the tool runs in GUI mode.
/i= : Specify the name of an input file to be converted.
/o= :Specify the name of the converted file.

Example
C:¥>ISO_CONV.EXE /c /i=ALL-PROG.TXT /o=ALL-PROG.ISO

Multiple files can be converted at a time by listing them in a batch file.

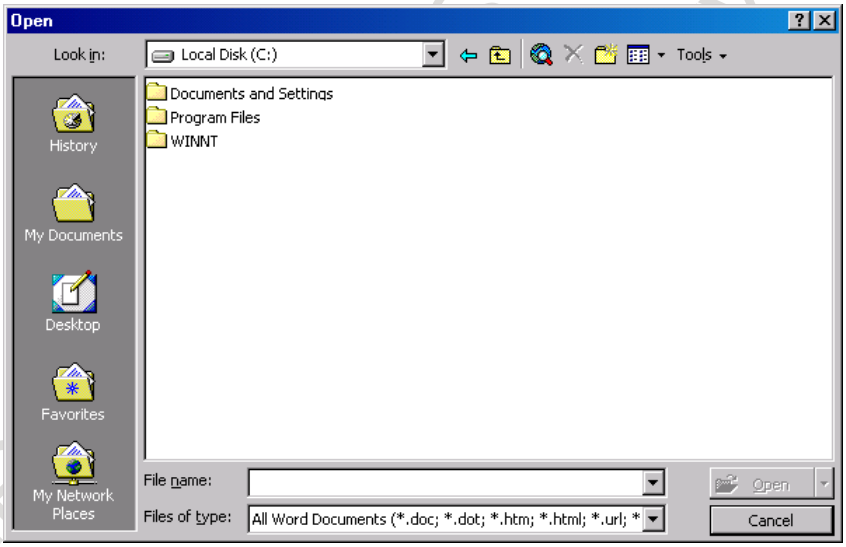
GUI

When you double-click the icon, the following screen appears, allowing you to select and convert a file.



Conversion procedure

1. Step 1
   In [Target File], specify a file you want to convert.
   When you click the [Select...▼] button, a file selection dialogue appears, allowing you to select a file.



2. Step 2
   After you specify the file name in step 1, you can determine the code of the specified file by clicking the [File Check] button. When you select the file by clicking the [Select...▼] button, the code is automatically determined.
   The file is determined as an ASCII file, ISO file, or ASCII/ISO-mixed file, or is indeterminable. In [Message], an additional message appears depending on the determination result.
   If the file is determined as an ASCII/ISO-mixed file, it cannot be converted, so you cannot proceed to the next step. The [File Convert] button is disabled and you cannot click it. If file contains only characters represented by the code values common to ASCII and ISO, the code of the file is indeterminable. This file need not be converted, so the [File Convert] button is disabled and you cannot click it.

3. Step 3
   When you specify the name of a converted file and click the [File Convert] button, the converted file is created. When the original file is an ASCII file, an ISO file is created; when the original is an ISO file, an ASCII file is created.
   You can also click the [Select...▼] button and specify the name of the converted file from a file selection dialogue.

Other

When you click the [Version] button, the software version information is displayed as shown below.

FRYER MACHINE SYSTEMS INC
Rob Zahensky

K LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE PROGRAM COMMAND

K.1 LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF G CODE

The function including address P in the argument of G code is shown below.

Function name	Machining center system	Lathe system G code system A	Lathe system G code system B	Lathe system G code system C	Reference item
Dwell	G04	G04	G04	G04	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
G code preventing buffering	G04.1	G04.1	G04.1	G04.1	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
AI contour control II(high-precision contour control/AI high-precision contour control compatible command)	G05	G05	G05	G05	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
AI contour control (advanced preview control compatible command)	G08	G08	G08	G08	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Programmable data input	G10	G10	G10	G10	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Programmable parameter input	G10	G10	G10	G10	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Programmable internal data change	G10.8	G10.8	G10.8	G10.8	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES)
Machine configuration selecting function	G10.8	G10.8	G10.8	G10.8	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Spindle speed fluctuation detection	G26	G26	G26	G26	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Reference position return	G30	G30	G30	G30	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Multi-step skip	G31	G31	G31	G31	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Torque limit skip	G31	G31	G31	G31	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Continuous high-speed skip	G31	G31	G31	G31	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Skip function for EGB axis	G31.8	G31.8	G31.8	G31.8	CONNECTION MANUAL(FUNCTION)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Function name	Machining center system	Lathe system G code system A	B	C	Reference item
Wheel wear compensation	G41	G41	G41	G41	OPERATOR'S MANUAL(For Machining Center System) II. PROGRAMMING
Scaling	G51	-	G51	G51	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Polygon turning	G51.2	G51.2	G51.2	G51.2	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
		(G251)	(G251)	(G251)	
Synchronous, Composite and Superimposed control by program command	G51.4,	G51.4,	G51.4,	G51.4,	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
	G51.5,	G51.5,	G51.5,	G51.5,	
	G51.6,	G51.6,	G51.6,	G51.6,	
	G50.5	G50.5	G50.5	G50.5	
Flexible path axis assignment	G52.1,	G52.1,	G52.1,	G52.1,	CONNECTION MANUAL(FUNCTION)
	G52.2,	G52.2,	G52.2,	G52.2,	
	G52.3	G52.3	G52.3	G52.3	
High-speed G53 function	G53	G53	G53	G53	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Workpiece coordinate system	G54	G54	G54	G54	CONNECTION MANUAL(FUNCTION)
	(G54.1)	(G54.1)	(G54.1)	(G54.1)	OPERATOR'S MANUAL(COMMON TO T/M SERIES)
Custom macro	G65,	G65,	G65,	G65,	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
	G66,	G66,	G66,	G66,	
	G66.1	G66.1	G66.1	G66.1	
Execution macro Note1)	G65,	G65,	G65,	G65,	Macro Executor PROGRAMMING MANUAL
	G66,	G66,	G66,	G66,	
	G66.1	G66.1	G66.1	G66.1	
	Note1)	Note1)	Note1)	Note1)	
Pattern Data Input	G65	G65	G65	G65	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Balanced cutting	-	G68	G68	G68	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(For Lathe System)
Tilted working plane indexing	G68.2	G68.2	G68.2	G68.2	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Tilted working plane indexing by tool axis direction	G68.3,	G68.3,	G68.3,	G68.3,	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES)
Tilted working plane indexing (incremental multi-command)	G68.4	G68.4	G68.4	G68.4	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Multiple repetitive cycles	G70.7 to G76.7	G70 to G76	G70 to G76	G72 to G78	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(For Lathe System) OPERATOR'S MANUAL(For Machining Center System)
Figure copying	G72.1,	G72.1,	G72.1,	G72.1,	OPERATOR'S MANUAL(COMMON TO T/M SERIES)
	G72.2	G72.2	G72.2	G72.2	
Canned cycle	G74, G76	-	-	-	OPERATOR'S MANUAL(For Machining Center System)

FRYER MACHINE SYSTEMS INC
Rob Zahensky

Function name	Machining center system	Lathe system G code system A	B	C	Reference item
Canned cycle for grinding	G75,	G72,	G72,	G73,	OPERATOR'S MANUAL(For Lathe System) II. PROGRAMMING
	G77,	G74	G74	G75	
	G78,				
	G79				
Canned cycle	G82 to G84,	G82 to G85,	G82 to G85,	G82 to G85,	OPERATOR'S MANUAL(For Lathe System) OPERATOR'S MANUAL(For Machining Center System)
	G87 to G89	G87 to G89,	G87 to G89,	G87 to G89,	
		G83.5,	G83.5,	G83.5,	
		G83.6,	G83.6,	G83.6,	
		G87.5,	G87.5,	G87.5,	
		G87.6	G87.6	G87.6	
Electronic gear box	G81	G81	G81	G81	CONNECTION MANUAL(FUNCTION)
Electronic gear box 2 pair	G81.5	G81.5	G81.5	G81.5	CONNECTION MANUAL(FUNCTION)
High-speed peck drilling cycle	-	G83.1	G83.1	G83.1	OPERATOR'S MANUAL(For Lathe System)
Rigid tapping cycle (FS15-T format)	G84.2	G84.2	G84.2	G84.2	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(For Lathe System) OPERATOR'S MANUAL(For Machining Center System)
Left-handed rigid tapping cycle(FS15-T format)	G84.3	-	-	-	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(For Machining Center System)
Constant surface speed control	G96	G96	G96	G96	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Spindle Indexing Function	G96.1 to G96.3	G96.1 to G96.3	G96.1 to G96.3	G96.1 to G96.3	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Spindle control with servo motor	G96.4	G96.4	G96.4	G96.4	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING

Note 1) : Arbitrary G code is optional with the following compilation parameter.
No.9013 to No.9022,No.9034,No.9045 to No.9047,No.9129 to No.9137

K.2 LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF M AND S CODE

The function including address P in the argument of M and S code is shown below.

Function name	M code format	related parameters	Reference item
Waiting M codes	M_P_	No.8110,No.8111, MWP(No.8103#1)	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Waiting M codes of high-speed Type	M_P_	No.8114,No.8115, MWP(No.8103#1)	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING

FRYER MACHINE SYSTEMS INC Ron Zaranosky

Function name	M code format	related parameters	Reference item
Waiting function by specifying start point	M_P_L_IP	STW(No.8101#1), No.8110,No.8111, MWP(No.8103#1)	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
(Custom macro) Subprogram	M98P_		OPERATOR'S MANUAL(COMMON to T/M series) II. PROGRAMMING
Program end	M99P_		
External subprogram call	M198P_	SBP(No.3404#2)	OPERATOR'S MANUAL(COMMON to T/M series) III. OPERATION
Custom macro Macro Call Using an M code (include specification of multiple definitions and special macro call)	M_P_	MAA(No.6009#2)	OPERATOR'S MANUAL(COMMON to T/M series) II. PROGRAMMING
Execution macro Subprogram call	M98P_		Macro Executor PROGRAMMING MANUAL
Subprogram call for user program	M_P_		
Execution macro Macro Call Using an M code (include specification of multiple definitions and special macro call)	M_P_		
Multi-spindle	S_P_	MPP(No.3703#3), No.3781	CONNECTION MANUAL(FUNCTION)

FRYER MACHINE SYSTEMS INC
Rob Zaherlsky

INDEX

<Number>

10.4" LCD CNC Display Panel (15"/19" LCD CNC Display Panel) 682
3-DIMENSIONAL COORDINATE SYSTEM CONVERSION 257
3-DIMENSIONAL MANUAL FEED....718

<A>

ABSOLUTE AND INCREMENTAL PROGRAMMING....137
Absolute multiple command 301
ACTION TO BE TAKEN WHEN A PROBLEM OCCURRED 1533
Actual Feedrate Display 1107
Actual Feedrate Display (15/19-inch Display Unit)...1120
Addition of Workpiece Coordinate System Pair (G54.1 or G54) 130
AI CONTOUR CONTROL FUNCTION I AND AI CONTOUR CONTROL FUNCTION II (G05.1) 567
ALARM AND SELF-DIAGNOSIS FUNCTIONS....914
Alarm Display 680,914
ALARM HISTORY DISPLAY 916
ALARM LIST....1576
All Data Output Screen 999
Altering a Word 1022
Angular Axis Control 735
Animation....1508
ANIMATION GRAPHIC screen 1511
ARITHMETIC AND LOGIC OPERATION 418
ARITHMETIC AND LOGICAL OPERATION....515
Automatic Coordinate System Setting 132
Automatic Corner Override....102
AUTOMATIC INSERTION OF SEQUENCE NUMBERS....1009
AUTOMATIC OPERATION 672,742
Automatic override for inner corners (G62)....102
AUXILIARY FUNCTION....213,535
AUXILIARY FUNCTION (M FUNCTION)....213
Auxiliary Function Lock 851
AUXILIARY FUNCTION OUTPUT BLOCK REVERSE MOVEMENT FOR MANUAL HANDLE RETRACE 871
AUXILIARY FUNCTION OUTPUT IN MOVING AXIS....220
Auxiliary Function Output in Program Restart Function784
Available Keys....1036
AXIS CONTROL COMMAND....525
AXIS CONTROL FUNCTIONS 590
Axis status display 899
Axis Synchronization Control 734
AXIS SYNCHRONOUS CONTROL 590

<B>

Background Editing 1158
Background Editing (15/19-inch Display Unit) 1206
BACKING UP VARIOUS DATA ITEMS 1534
Battery in the PANEL *i* (3 VDC) 1538
BRANCH AND REPETITION....428

<C>

CALCULATION KEY FUNCTION 1480
Call stack display 1132
Call stack display (15/19-inch display unit)....1191
Cancel Command for Real Time Custom Macro 505
Canceling Spindle Positioning 154
Change Tolerance in Smooth Tolerance+ Control Mode76
CHANGING CURRENT FOLDER....1060
CHANGING FILE ATTRIBUTES....1065
CHANGING FOLDER ATTRIBUTES....1062
Changing Workpiece Coordinate System 125
CHARACTER-TO-CODES CORRESPONDENCE TABLE....1569
Check by Running the Machine....673
Check of the Tool Path between Blocks by Stroke Limit Check before Move 886
CHECKING BY DIAGNOSTIC DISPLAY....918
CIRCULAR INTERPOLATION (G02, G03)....45
CIRCULAR THREADING (G35, G36) 79
CODES AND RESERVED WORDS USED IN CUSTOM MACROS....475
Color Setting Screen 1378
Color Setting Screen (15/19-inch Display Unit) 1420
COMMAND FOR MACHINE OPERATIONS - AUXILIARY FUNCTION....23
COMMAND PATH OF MACRO CALL OR SUBPROGRAM CALL 249
Command type selection screen....1178,1226
COMMON MEMORY BETWEEN EACH PATH 654
COMPENSATION FUNCTION 336
Condition Transition (ZEDGE Statement)....519
Conditional Branch (IF Statement) 431
Conditional Branch (ZONCE Statement)....518
Confirmation of a data update during the data setting process....897
Confirmation of incremental input....895
Confirmation of the deletion of all data 897
Confirmation of the deletion of the program....896
Confirmation of the start from a middle block....900
CONSTANT SURFACE SPEED CONTROL (G96, G97) 145
CONTINUOUS HIGH-SPEED SKIP FUNCTION 87
CONTROL ON REAL TIME MACRO COMMANDS517
CONTROLLED AXES....27
Controlled axis configuration example 632
CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION 1013
COORDINATE SYSTEM 118
Coordinate System on Part Drawing and Coordinate System Specified by CNC - Coordinate System 13

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

COORDINATE VALUE AND DIMENSION.....137
Copy.....1038
Copy and movement between different devices.....1071
Copy of program file with memory card device 1072
CREATING A FOLDER 1059
CREATING PROGRAMS.....1008
CREATING PROGRAMS IN TEACH IN MODE (PLAYBACK).....1010
CREATING PROGRAMS USING THE MDI UNIT 1008
Creation.....1039
Current block screen 1150
Current Position Display 679
CURSOR MOVEMENT LIMITATIONS ON PROGRAM EDITING 1031
CUSTOM MACRO.....358
Custom Macro Variables.....511
Customization of Tool Management Data Display 181
Cut.....1039
CUTTING FEED 93
CUTTING FEEDRATE CONTROL 100
Cutting Point Interpolation for Cylindrical Interpolation 64
CUTTING SPEED - SPINDLE FUNCTION.....21
Cylindrical Interpolation 59
CYLINDRICAL INTERPOLATION (G07.1).....59
Cylindrical Interpolation by Plane Distance Command .62

<D>

DATA INPUT/OUTPUT 922
Data range check.....901
DECIMAL POINT PROGRAMMING.....142
Default Folders.....229
Definition of the Custom Macro Screen.....559
Definition of the Pattern Menu Screen.....557
DEFINITION OF THE SCREEN 556
Deleting a Block.....1025
DELETING A FILE.....1064
DELETING A FOLDER.....1063
Deleting a Word 1023
Deleting All Programs.....1029
DELETING BLOCKS 1025
Deleting files in memory card device.....1074
Deleting Multiple Blocks 1025
Deleting One Program.....1029
DELETING PROGRAMS 1029
Description of commands compatible with those for a hobbing machine (G80, G81) 628
Details 1046,1480
Details of Functions 485
Details of the tilted working plane data setting screen..... 1182,1230
DIAMETER AND RADIUS PROGRAMMING.....144
Direct Input of Workpiece Origin Offset Value Measured 1244
Direct Input of Workpiece Origin Offset Value Measured (15/19-inch Display Unit).....1311
Disabling Life Count.....211
DISPLAY.....678
Display interpreted and original programs on the same screen 1134
Display interpreted and original programs on the same screen (15/19-inch display unit) 1194
Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount) 1113
Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount) (15/19-inch Display Unit) 1126
Display of Run Time and Parts Count 1109
Display of Run Time and Parts Count (15/19-inch Display Unit).....1122
DISPLAY OF SYSTEM VARIABLES 413
Display of updated modal information.....898
Displaying and Entering Setting Data.....1237
Displaying and Entering Setting Data (15/19-inch Display Unit).....1303
Displaying and Setting Custom Macro Common Variables 1246
Displaying and Setting Custom Macro Common Variables (15/19-inch Display Unit) 1313
DISPLAYING AND SETTING DATA.....675
Displaying and setting magazine screen 1251
Displaying and setting magazine screen (15/19-inch display unit).....1319
Displaying and Setting Parameters 1370
Displaying and Setting Parameters (15/19-inch Display Unit) 1413
Displaying and Setting Pattern Data Inputs 1291
Displaying and Setting Pattern Data Inputs (15/19-inch Display Unit).....1358
Displaying and Setting Real Time Custom Macro Data 1247
Displaying and Setting Real Time Custom Macro Data (15/19-inch Display Unit) 1314
Displaying and Setting Run Time, Parts Count, and Time 1241
Displaying and Setting Run Time, Parts Count, and Time (15/19-inch Display Unit) 1308
Displaying and setting the FSSB axis setting screen..1394
Displaying and setting the FSSB axis setting screen (15/19-inch display unit) 1436
Displaying and setting the FSSB servo amplifier setting screen 1393
Displaying and setting the FSSB servo amplifier setting screen (15/19-inch display unit) 1435
Displaying and setting the FSSB spindle amplifier setting screen.....1393
Displaying and setting the FSSB spindle amplifier setting screen (15/19-inch display unit).....1435
Displaying and setting the machining parameter tuning screen 1397
Displaying and setting the machining parameter tuning screen (15/19-inch display unit) 1439
Displaying and setting the servo setting screen.....1394
Displaying and setting the servo setting screen (15/19-inch display unit) 1436

FRYER MACHINE SYSTEMS INC
Rob Zaleresky

Displaying and setting the servo tuning screen 1396
Displaying and setting the servo tuning screen (15/19-inch display unit) 1438
Displaying and Setting the Software Operator's Panel1249
Displaying and Setting the Software Operator's Panel (15/19-inch Display Unit).....1317
Displaying and setting the spindle tuning screen 1396
Displaying and setting the spindle tuning screen (15/19-inch display unit) 1438
Displaying and Setting the Workpiece Origin Offset Value 1243
Displaying and Setting the Workpiece Origin Offset Value (15/19-inch Display Unit) 1310
Displaying and Setting Tool Life Management Data.1280
Displaying and Setting Tool Life Management Data (15/19-inch Display Unit).....1347
Displaying and setting tool management screen 1253
Displaying and setting tool management screen (15/19-inch display unit) 1321
Displaying and Switching the Display Language 1269
Displaying and Switching the Display Language (15/19-inch Display Unit).....1336
Displaying Memory Data.....1386
Displaying Memory Data (15/19-inch Display Unit).1428
Displaying Program Being Executed in Backward Movement.....870
Displaying the executed block (BC) 1129,1191
Displaying the menu screen and selecting a menu item 1388
Displaying the menu screen and selecting a menu item (15/19-inch display unit) 1429
Displaying the Program Name 1476
Displaying the Program Number, Program Name, and Sequence Number.....1471
Displaying the Program Number, Program Name, and Sequence Number (15/19-inch Display Unit) 1475
DISPLAYING THE PROGRAM NUMBER/NAME, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION 1470
Displaying the Status and Warning for Data Setting or Input/Output Operation 1472
Displaying the Status and Warning for Data Setting or Input/Output Operation (15/19-inch Display Unit) 1477
Displaying the total life of tools of the same type.....1261
Displaying the total life of tools of the same type (15/19-inch display unit) 1329
DISPLAYING VARIABLE IN MACRO ALARM OR MACRO MESSAGE.....492
DISTANCE CODED LINEAR SCALE INTERFACE732
Distance Coded Rotary Encoder 733
DNC OPERATION.....747
DNC Operation File Set 996
DRY RUN.....853
DWELL.....108
DYNAMIC GRAPHIC DISPLAY.....1494

<E>

Each tool Data Screen 190,1258
Each tool data screen (15/19-inch display unit) 1326
Eco Monitoring Screen 1299
Eco Monitoring Screen (15/19-inch Display Unit) 1366
Eco Setting Screen 1295
Eco Setting Screen (15/19-inch Display Unit) 1361
EDIT DISABLE ATTRIBUTE AND DISPLAY DISABLE ATTRIBUTE 1018
EDITING A PROGRAM 674,1137
Editing a Program (15/19-inch Display Unit) 1196
EDITING OF CUSTOM MACROS 1030
EDITING PROGRAM CHARACTERS.....1034
EDITING PROGRAMS.....1017
Electronic Gear Box.....607
Electronic Gear Box 2 Pair 626
Electronic Gear Box Automatic Phase Synchronization620
EMBEDDED ETHERNET OPERATIONS 988
EMERGENCY STOP 878
Error message.....1006
ERROR MESSAGE AND NOTE.....1690
Exact Stop (G09, G61), Cutting Mode (G64), Tapping Mode (G63) 101
EXPLANATION 550
EXPLANATION OF OPERATION 553
Explanation of Operations.....1680
EXPLANATION OF THE MDI UNIT.....686
EXTENSION OF THE MANUAL HANDLE RETRACE FUNCTION.....874
EXTERNAL I/O DEVICES.....698
External Operator Message History 1455
EXTERNAL OUTPUT COMMANDS.....476
EXTERNAL SUBPROGRAM CALL (M198).....752
EXTERNAL SUBPROGRAM CALLS USING THE DATA SERVER AVAILABLE IN MULTI-PATH SYSTEMS 757

<F>

FANUC DOUBLE-BYTE CHARACTER CODE TABLE.....1570
FEED FUNCTIONS 91
FEED-FEED FUNCTION 11
FEEDRATE INSTRUCTION ON IMAGINARY CIRCLE FOR A ROTARY AXIS.....105
FEEDRATE OVERRIDE 852
FIGURE COPYING (G72.1, G72.2) 251
File Format and Error Messages 987
Folder Attributes 228
Folder Configuration.....226
Folder for Subprogram/Macro Calls 1078
FOLDER MANAGEMENT 1075
Folder tree display.....1145
Folder tree display (15/19-inch display unit) 1204
FOLDERS.....226
Format.....1001
FTP File Transfer Function.....993
FUNCTION FOR LOADER CONTROL 665
Function Keys 690

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

FUNCTION KEYS AND SOFT KEYS.....688
Functions that are Used When Data is Set 894
Functions that are Used when the Program is Executed898
FUNCTIONS TO SIMPLIFY PROGRAMMING.....251

<G>

G CODE LIST IN THE LATHE SYSTEM 36
G CODE LIST IN THE MACHINING CENTER SYSTEM 33
G CODE PREVENTING BUFFERING 585
G53, G28, and G30 Commands in Tool Length Compensation Mode.....341
GENERAL 3,9,669
General Screen Operations.....688
General specifications of the tilted working plane indexing.....274
GOTO Statement Using Stored Sequence Numbers / LGOTO Statement Using Stored Label.....429
GRAPHIC DISPLAY 1483
GRAPHIC FUNCTION 1483
GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen.....1495,1508

<H>

Heading a Program.....1021
HELICAL INTERPOLATION (G02, G03).....49
HIGH PRECISION OSCILLATION FUNCTION 641
HIGH SPEED PROGRAM CHECK FUNCTION.....855
HIGH-SPEED CUTTING FUNCTIONS.....567
HIGH-SPEED PROGRAM MANAGEMENT 1080
HIGH-SPEED SKIP (G31) 86
Host File List Display 988
How to Indicate Command Dimensions for Moving the Tool (Absolute and Incremental Programming).....18
How to View the Position Display Change without Running the Machine 674

<I>

Improvement of Quick program restart for non-machining program 824
Improvement of Tool compensation for Tool Retract and Recover.....605
INCH/METRIC CONVERSION (G20, G21) 139
INCORRECT THREADED LENGTH.....1559
INCREMENT SYSTEM.....28
INCREMENTAL FEED 703
Incremental multiple command.....303
INDIRECT AXIS ADDRESS SPECIFICATION.....425
Input data range check 894
Input Mode.....1037
Input of Customize Data with the Decimal Point.....187
INPUT/OUTPUT FILE NAME FOR MULTI-PATH SYSTEM 1006
Input/output format of operation history signal selection data 945
Input/output format of pitch error compensation data..937
INPUT/OUTPUT ON EACH SCREEN.....925
INPUT/OUTPUT ON THE ALL IO SCREEN.....963
Inputting a program.....925
Inputting and Outputting a Program.....925
Inputting and Outputting All Tool Management Data at a Time 982
Inputting and Outputting Custom Macro Common Variables 939
Inputting and outputting of O8-digit 929
Inputting and Outputting Offset Data.....931,971
Inputting and Outputting Operation History Data.943,977
Inputting and Outputting Parameters 929,970
Inputting and Outputting Pitch Error Compensation Data 935
Inputting and Outputting Tool Life Management Data 961
Inputting and Outputting Tool Management Data 946,978
Inputting and Outputting Workpiece Coordinates System Data 941,976
Inputting and Outputting Workpiece Setting Error Compensation Value 986
Inputting custom macro common variables 939
Inputting customize data displayed as tool management data 954
Inputting decimal point position data of customize data957
Inputting magazine data 949
Inputting name data of customize data.....952
Inputting offset data 931
Inputting operation history signal selection data.....944
Inputting parameters 929
Inputting pitch error compensation data.....935
Inputting spindle waiting position name data.....956
Inputting tool geometry data 959
Inputting tool life management data.....961
Inputting tool life status name data 951
Inputting tool management data.....946
Inputting workpiece coordinate system data 941
Inputting/Outputting a Program 966
Inputting/Outputting all Programs and Folders.....968
Inputting/Outputting Custom Macro Common Variables 974
Inputting/Outputting Pitch Error Compensation Data..973
Inserting a Word 1022
INSERTING, ALTERING AND DELETING A WORD.....1019
Internal circular cutting feedrate change.....104
INTERPOLATION FUNCTIONS.....40
INTERRUPTION TYPE CUSTOM MACRO.....483
ISO/ASCII CODE CONVERSION TOOL.....1692

<J>

JOG FEED (JOG) 701

<K>

KEYS AND PROGRAM ENCRYPTION 1040

<L>

LCD mounted type CNC.....1001
LIMITATION 540,1190,1236
Limitations 1529
Line Number Display.....1037

FRYER MACHINE SYSTEMS INC
Rob Zaharisky

Line Search 1040
LINEAR INTERPOLATION (G01) 43
LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL).....737
List of Error Message.....1690
LIST OF FUNCTIONS AND PROGRAM FORMAT1548
LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF G CODE 1695
LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF M AND S CODE.....1697
LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE PROGRAM COMMAND.....1695
List of Functions of PC Tool.....1679
List of Message 1005
List of Output Data.....1004
LOAD METER SCREEN.....1466
LOCAL COORDINATE SYSTEM 134
Local variables 513

<M>

M Code Group Check Function 217
M CODE GROUPING FUNCTION.....215
M198 Folder Set.....996
MACHINE COORDINATE SYSTEM.....119
Machine Lock 850
MACHINE LOCK AND AUXILIARY FUNCTION LOCK 850
MACHINING CONDITION SELECTING FUNCTION.....583
Machining Level Selection.....1279
Machining Level Selection (15/19-inch Display Unit)1345
Machining Parameter Tuning.....1380
Machining Parameter Tuning (15/19-inch Display Unit) 1422
Machining parameter tuning (AI contour) 1380,1422
MACRO CALL.....445,523
MACRO CALL ARGUMENT FOR AXIS NAME EXPANSION 471
Macro Call Using a G Code 458
Macro Call Using a G Code (Specification of Multiple Definitions).....459
Macro Call Using a G Code with a Decimal Point (Specification of Multiple Definitions) 460
Macro Call Using an M Code 461
Macro Call Using an M Code (Specification of Multiple Definitions).....462
MACRO STATEMENTS AND NC STATEMENTS..427
MAKING A PROGRAM COMPACT.....1067
MANUAL ABSOLUTE ON AND OFF 707
MANUAL HANDLE FEED 704
MANUAL HANDLE INTERRUPTION 759
MANUAL HANDLE RETRACE.....857
MANUAL HANDLE RETRACE FUNCTION FOR MULTI-PATH.....871
Manual Interruption of 3-dimensional Coordinate System Conversion.....764
MANUAL INTERVENTION AND RETURN.....835
MANUAL LINEAR/CIRCULAR INTERPOLATION711
MANUAL OPERATION.....669,700
MANUAL REFERENCE POSITION RETURN.....700
MAXIMUM COMMAND VALUES 30
Maximum incremental value check 902
MAXIMUM STROKE.....29
MDI OPERATION 744
MEMORY OPERATION 742
METHOD OF REPLACING BATTERY 1535
MIRROR IMAGE.....765
Modal Call: Call After the Move Command (G66) 453
Modal Call: Each Block Call (G66.1).....456
Modal G codes display in 3 columns on program check screen 1149
Modal Real Time Macro Command / One-shot Real Time Macro Command 499
MUITI-PATH CONTROL FUNCTION.....650
Multidirectional branch (SWITCH-CASE Statement).440
MULTI-PATH EDITING FUNCTION 1046
Multiple Command of Tilted Working Plane Indexing301
MULTIPLE M COMMANDS IN A SINGLE BLOCK214
Multi-statement (ZDO...ZEND Statement).....521
MULTI-STEP SKIP (G31) 85

<N>

NAMES OF AXES 27
NAMING RULES.....1688
Naming Rules of Folder.....1688
Naming Rules of Program File.....1688
NC DATA OUTPUT FUNCTION 999
Next Block Display Screen 1147
Next Block Display Screen (15/19-inch Display Unit)1204
NOMOGRAPHS.....1559
NORMAL DIRECTION CONTROL (G40.1,G41.1,G42.1).....351
Note.....736,1519,1691
NOTES.....538
NOTES ON READING THIS MANUAL 6
NOTES ON VARIOUS KINDS OF DATA 6
NUMBER OF CONTROLLED AXES.....27

<O>

Operating Monitor Display 1111
Operating Monitor Display (15/19-inch Display Unit)1124
Operation 1480
Operation confirmation function setting screen 903
Operation level setting 1271
Operation level setting (15-inch display unit).....1338
Operation Procedure for a Canned Cycle for Drilling..833
OPERATIONAL DEVICES 681
OPTIMUM ACCELERATION/DECELERATION FOR RIGID TAPPING 584
OTHERS.....524
Output Directory 1001
Output operation 1000
Outputting a program 927
Outputting custom macro common variables.....940
Outputting customize data displayed as tool management data.....955

FRYER MACHINE SYSTEMS INC
Rob Zakensky

Outputting decimal point position data of customize data 958
Outputting magazine data....950
Outputting name data of customize data 953
Outputting offset data....932
Outputting operation history data....943
Outputting operation history signal section data 945
Outputting parameters....930
Outputting pitch error compensation data 937
Outputting spindle waiting position name data 957
Outputting tool geometry data....960
Outputting tool life management data 962
Outputting tool life status name data....952
Outputting tool management data 948
Outputting workpiece coordinate system data 942
Overall Position Display 1104
Overall Position Display (15/19-inch Display Unit) ..1116
OVERTRAVEL 878
OVERVIEW 91,336,550,650,1046,1480,1539
OVERWRITING FILES ON A MEMORY CARD/USB MEMORY 923

<P>

Parameter Tuning Screen 1388
Parameter Tuning Screen (15/19-inch Display Unit) .1429
Parameter tuning screen (axis setting)....1392
Parameter tuning screen (axis setting) (15/19-inch display unit)....1434
Parameter tuning screen (miscellaneous settings) 1395
Parameter tuning screen (miscellaneous settings) (15/19-inch display unit) 1437
Parameter tuning screen (spindle setting)....1395
Parameter tuning screen (spindle setting) (15/19-inch display unit)....1437
Parameter tuning screen (system setting)....1391
Parameter tuning screen (system setting) (15/19-inch display unit)....1432
PART DRAWING AND TOOL MOVEMENT 12
Part Program Storage Size / Number of Registerable Programs....235
Parts Count Display, Run Time Display 680
PASSWORD FUNCTION 1032
Password modification 1272
Password modification (15/19-inch display unit)....1339
Paste 1039
Path Drawing....1495
PATH GRAPHIC (TOOL POSITION) screen....1506
PATH GRAPHIC screen....1500
PATH NAME EXPANSION DISPLAY FUNCTION1462
PATTERN DATA INPUT 550
PC TOOL FOR MEMORY CARD PROGRAM OPERATION/ EDITING 1679
PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING 1679
Periodic Maintenance Screen 1401
Periodic Maintenance Screen (15/19-inch Display Unit) 1443
PLANE SELECTION 136
PMC Axis Control 735
POLAR COORDINATE INTERPOLATION (G12.1, G13.1) 51
POLYGON TURNING (G50.2, G51.2, G250, G251) .592
Position Display in the Relative Coordinate System..1101
Position Display in the Workpiece Coordinate System1100
POSITIONING (G00)....40
Power Consumption Monitoring Screen 1411
Power Consumption Monitoring Screen (15/19-inch Display Unit)....1453
Power Disconnection 682
POWER ON/OFF 681
Precision Level Selection.... 1277,1280,1346
Precision Level Selection (15/19-inch Display Unit).1344
Precision Specification for Macro Relational Operators438
PREPARATORY FUNCTION (G FUNCTION) 32
Procedure for Reference Position Establishment 732
PROCESSING MACRO STATEMENTS 472
PROGRAM AND FOLDER COPY/MOVE FUNCTION....1068
Program Attributes....231
Program Check Screen....1148
Program Check Screen (15/19-inch Display Unit)....1205
PROGRAM CODE LIST....1545
PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS 237
PROGRAM CONFIGURATION 24,236
Program Contents Display 1129
Program Contents Display (15/19-inch Display Unit)1190
Program Display 678
Program Folder Screen....1140
Program Folder Screen (15/19-inch Display Unit) 1199
PROGRAM MANAGEMENT 226,1052
Program Management only in the Path Folder....1076
Program Management under the Path Folder....1076
Program Name 229
PROGRAM RESTART 768
Program Screen for MDI Operation....1139
Program Screen for MDI Operation (15/19-inch Display Unit) 1198
PROGRAM SEARCH 1026
PROGRAM SECTION CONFIGURATION....239
PROGRAM VERIFICATION 1079
PROGRAMMABLE DATA INPUT (G10)....542
Programmable Data Input (G10) for Blank Figure Drawing Parameters 1517
PROGRAMMABLE MIRROR IMAGE (G50.1, G51.1) 349
PROGRAMMABLE PARAMETER INPUT (G10)....545
PROGRAMS....229
Prohibition of the absolute input by the soft key....896
Protection level setting....1273
Protection level setting (15/19-inch display unit) 1340
Protection of Data at Eight Levels 1271
Protection of Data at Eight Levels (15/19-inch Display Unit) 1338
Protection of Tool Information with the KEY Signal ..189

FRYER MACHINE SYSTEMS INC
Rob Zaremsky

<Q>

QUICK PROGRAM RESTART....788
Quick Program Restart for a Machining Cycle....809
Quick Program Restart for Multi Path System....817

<R>

RADIUS DIRECTION ERROR AT CIRCLE CUTTING....1565
RANGE OF COMMAND VALUE....1557
RAPID TRAVERSE....92
RAPID TRAVERSE OVERRIDE....852
READING AND WRITING VARIABLES FOR ANOTHER PATH....414,514
READING PARAMETERS....426
Real time macro variables (RTM variables)....510
REAL-TIME CUSTOM MACRO....497
REFERENCE POSITION....110
Reference Position (Machine-specific Position)....12
REFERENCE POSITION RETURN....110,733
REGISTERING CUSTOM MACRO PROGRAMS....475
Registering, Changing, and Deleting Tool Life Management Data....196
Related Parameters....234
RELATION WITH CONVENTIONAL FUNCTIONS232
Relation with Folders....232
Relation with Program Names....233
Relationship with Other Functions....920
Remaining Tool Number Check Function....211
RENAMING A FILE....1064
RENAMING A FOLDER....1060
Repetition (WHILE Statement)....436
Repetition (ZWHILE Statement)....520
Replacement....1037
REPLACING A WORD OR ADDRESS....1024
Replacing batteries....1540
Replacing Battery for Absolute Pulsecoders....1539
Replacing Battery for Control Unit....1536
Replacing the batteries in a separate battery case....1540
Replacing the battery built into the servo amplifier....1541
Repositioning....829
RESTRICTIONS....481,1520
Restrictions of Tilted Working Plane Indexing....332
RETRACE....838
Retract....828
Retract function....638
RETRACTION FOR RIGID TAPPING / RETRACTION FOR 3-DIMENSIONAL RIGID TAPPING....847
Return....829
Return from the Alarm Screen....919
Reversing Edit Operations (Undo Function)....1038
RIGID TAPPING BY MANUAL HANDLE....715
ROTARY AXIS ROLL-OVER....601
ROUTINE MAINTENANCE....1533
RULES OF CHARACTERS IN PROGRAM FILE....1689

<S>

SAFETY FUNCTIONS....878
SAFETY PRECAUTIONS....S-1
Sample programs....632
Saving....1039
SCALING (G50, G51)....342
SCHEDULE OPERATION....750
SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION1465
Screen for Assistance in Entering Tilted Working Plane Indexing....1172
Screen for Assistance in Entering Tilted Working Plane Indexing (15/19-inch Display Unit)....1220
SCREEN HARD COPY FUNCTION....996
SCREENS DISPLAYED BY FUNCTION KEY....1099,1128,1237,1370,1455
Search....1037
SECOND AUXILIARY FUNCTIONS (B CODES)....218
SELECTING A DEFAULT FOLDER....1063
SELECTING A DEVICE....1053
Selecting a From Cassette as a Device....1057
SELECTING A MAIN PROGRAM....1066
Selecting a Memory Card Program as a Device....1053
Selecting a USB Memory as a Device....1058
Selecting a Workpiece Coordinate System....124
Selection of a Tool Life Count Period....190
SELECTION OF TOOL USED FOR VARIOUS MACHINING - TOOL FUNCTION....22
Sequence Number Comparison and Stop....1240
Sequence Number Comparison and Stop (15/19-inch Display Unit)....1306
SEQUENCE NUMBER SEARCH....1027
Servo Parameters....1373
Servo Parameters (15/19-inch Display Unit)....1415
Servo Tuning....1374
Servo Tuning (15/19-inch Display Unit)....1416
Setting a Workpiece Coordinate System....122
Setting an M Code Group Number Using a Program....217
Setting an M Code Group Number Using the Setting Screen....215
SETTING AND DISPLAY UNITS....682
SETTING AND DISPLAYING DATA....1081
Setting and Displaying Data When the Tool Offset for Milling and Turning Function Is Enabled....1293
Setting and Displaying Data When the Tool Offset for Milling and Turning Function Is Enabled (15/19-inch Display Unit)....1360
Setting and Displaying Tool Management Data....1251
Setting and Displaying Tool Management Data (15/19-inch Display Unit)....1319
Setting of Spindle Position / Standby Position Display185
Setting Screen....903
Setting the change protection level and output protection level of a program....1276
Setting the change protection level and output protection level of a program (15/19-inch display unit)....1343
Setting the Character-codes....562
SETTINGS AT POWER-ON, IN THE CLEAR STATE, OR IN THE RESET STATE....1566

FRYER MACHINE SYSTEMS INC
Rob Zaborski

SIMPLE CALCULATION OF INCORRECT THREAD LENGTH....1560
Simple Call (G65)....446
SIMULTANEOUS EDITING OF MULTIPATH PROGRAMS....1043
SINGLE BLOCK....854
SINGLE DIRECTION POSITIONING (G60)....41
Single-path Display....1466
SKIP FUNCTION (G31)....83
Skip Function for EGB Axis....624
SKIP FUNCTION FOR FLEXIBLE SYNCHRONIZATION CONTROL....640
SKIP POSITION MACRO VARIABLE IMPROVEMENT....86
Small MDI Unit (ONG Key)....685
SMOOTH TOLERANCE+ CONTROL....74
Smoothing level selection....1279,1345
Soft Keys....691
Special Macro Call using M code....463
Specification Method....484
Specification method (G80.5, G81.5)....626
SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5-DIGIT COMMAND)....145
SPECIFYING THE SPINDLE SPEED WITH A CODE....145
Speeding up of rigid tapping....587
SPINDLE CONTROL BETWEEN EACH PATH....656
Spindle Control with Servo Motor....162
Spindle Electronic Gear Box....613
Spindle Indexing Function....163
Spindle Monitor....1377
Spindle Monitor (15/19-inch Display Unit)....1419
Spindle Orientation....152
Spindle Positioning....153
SPINDLE POSITIONING FUNCTION....152
Spindle Setting....1375
Spindle Setting (15/19-inch Display Unit)....1417
SPINDLE SPEED FLUCTUATION DETECTION....156
SPINDLE SPEED FUNCTION (S FUNCTION)....145
Spindle Tuning....1376
Spindle Tuning (15/19-inch Display Unit)....1418
Split display on the program folder screen....1141,1200
SRAM data and User files....1005
Stamping the Machining Time....1164
Stamping the Machining Time (15/19-inch Display Unit)....1212
Stand-alone type CNC (with personal computer function / Ethernet connection)....1003
Stand-alone type CNC (with personal computer function / HSSB connection)....1002
Stand-alone type CNC (without personal computer function)....1001
Standard MDI Unit (ONG Key)....683
Standard MDI Unit (QWERTY Key)....684
Start check signal....899
STORED STROKE CHECK....879
STORED STROKE LIMIT RANGE SWITCHING FUNCTION BY SIGNAL....893
STROKE LIMIT AREA CHANGING FUNCTION....892
STROKE LIMIT CHECK BEFORE MOVE....883
SUBPROGRAM (M98, M99)....245
Subprogram Call Using a Specific Address....468
Subprogram Call Using an M Code....465
Subprogram Call Using an M Code (Specification of Multiple Definitions)....466
Subprogram Calls Using a Secondary Auxiliary Function....467
Subprogram Calls Using a T Code....466
Subprogram Calls Using an S Code....467
Suppress Motion of Quick Program Restart....805
SWITCHING BETWEEN MULTI-PATH DISPLAY AND SINGLE-PATH DISPLAY FUNCTION....1458
Synchronization ratio specification range....635
SYNCHRONOUS, COMPOSITE AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND (G50.4, G51.4, G50.5, G51.5, G50.6, AND G51.6)....597
SYNCHRONOUS/COMPOSITE/SUPERIMPOSED CONTROL....657
System Configuration Screen....1408
System Configuration Screen (15/19-inch Display Unit)....1450
SYSTEM VARIABLES....364,508,512

<T>

Table Horizontal Direction Handle Feed / Table Horizontal Direction JOG Feed / Table Horizontal Direction Incremental Feed....728
Table Vertical Direction Handle Feed / Table Vertical Direction JOG Feed / Table Vertical Direction Incremental Feed....727
TANDEM CONTROL....640
TEST OPERATION....850
TESTING A PROGRAM....673
Text data....1004
Text display....1131
Tilted working plane data setting screen....1179,1227
Tilted Working Plane Indexing....270
Tilted working plane indexing based on Eulerian angle273
Tilted working plane indexing based on projection angles....288
Tilted working plane indexing based on roll-pitch-yaw279
Tilted working plane indexing based on three points...281
Tilted working plane indexing based on two vectors...285
Tilted working plane indexing by tool axis direction...291
Tilted Working Plane Indexing during manual operation....1189
Tilted Working Plane Indexing in Tool Length Compensation....328
Tool axis direction control....305
Tool Axis Direction Handle Feed / Tool Axis Direction JOG Feed / Tool Axis Direction Incremental Feed..719
Tool Axis Right-Angle Direction Handle Feed / Tool Axis Right-Angle Direction JOG Feed / Tool Axis Right-Angle Direction Incremental Feed....721

FRYER MACHINE SYSTEMS INC
Rob Zakrensky

Tool center point retention type tool axis direction control 322
TOOL FUNCTION (T FUNCTION).....165
Tool geometry data screen 1265
Tool geometry data screen (15/19-inch display unit).1332
TOOL LENGTH COMPENSATION (G43, G44, G49)336
Tool Life Count Restart M Code.....209
Tool Life Counting and Tool Selection.....207
TOOL LIFE MANAGEMENT 192
Tool life management (group editing screen).....1285
Tool life management (group editing screen) (15/19-inch display unit) 1352
Tool life management (list screen).....1281
Tool life management (list screen) (15/19-inch display unit) 1348
Tool Life Management Commands in Machining Program 201
Tool Life Management Data 194
TOOL MANAGEMENT EXTENSION FUNCTION .181
TOOL MANAGEMENT FUNCTION 166
TOOL MANAGEMENT FUNCTION FOR OVERSIZE TOOLS.....191
TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE-INTERPOLATION.....9
TOOL MOVEMENT BY PROGRAMING - AUTOMATIC OPERATION.....671
TOOL MOVEMENT RANGE - STROKE.....26
TOOL OFFSET FOR MILLING AND TURNING FUNCTION (G43.7) 356
Tool offset range setting screen 905
TOOL PATH AT CORNER 1562
TOOL RETRACT AND RECOVER.....602,825
Tool Retract and Recover for Threading.....829
TOOL SELECTION FUNCTION 165
Tool Tip Center Rotation Handle Feed / Tool Tip Center Rotation JOG Feed / Tool Tip Center Rotation Incremental Feed 725
TORQUE LIMIT SKIP.....88
Total Life Time Display for Tools of The Same Type.190
Turning on the Power.....681
Two-path Display and Three-path Display 1468
TYPES OF REAL TIME MACRO COMMANDS.....499

<U>

U-axis Control.....639
Unconditional Branch (GOTO Statement) / Unconditional Branch with label (LGOTO Statement) 428
Usable Characters in Program File.....1689
Usage Notes 1679
USB FUNCTION.....998

<V>

VARIABLE LEAD THREADING (G34).....78
VARIABLES 358,507
Variables Dedicated to Real Time Custom Macros 508
VIRTUAL MDI KEY 1526
VIRTUAL MDI KEY FUNCTION 1526

<W>

WAITING FUNCTION BY SPECIFYING START POINT.....659
WAITING FUNCTION FOR PATHS.....650
WAITING M CODES OF HIGH-SPEED TYPE 654
Warning display during a reset in program operation ..902
WARNING LIST.....1664
Warning message 1005
Warning Messages 1519
Withdrawal.....828
Word Search.....1020
WORKPIECE COORDINATE SYSTEM.....122
Workpiece Coordinate System Preset.....1106
Workpiece Coordinate System Preset (15/19-inch Display Unit).....1119
Workpiece Coordinate System Preset (G92.1).....128
Workpiece Coordinate System Shift.....132
Workpiece origin offset range setting screen 908
Workpiece shift range setting screen 912
WRONG OPERATION PREVENTION FUNCTIONS893

<Y>

Y-axis tool offset range setting screen 910

FRYER MACHINE SYSTEMS INC
Rob Zahensky

FRYER MACHINE SYSTEMS INC
Rob Zahensky

REVISION RECORD

Edition	Date	Contents
01	Feb., 2019	

FRYER MACHINE SYSTEMS INC
Rob Zahensky

FRYER MACHINE SYSTEMS INC
Rob Zahensky

B-64694EN/01

B-64694EN/01