FRYER
MACHINE SYSTEMS

Precision Built Solutions™

FRYER / SIEMENS 828 HS-CNC CONTROL

828 TOOLROOM LATHE OPERATOR MANUAL

MADE IN USA

TABLE OF CONTENTS

READ BEFORE INSTALLING OR OPERATING

NOTE: THIS MACHINE IS AUTOMATICALLY CONTROLLED AND MAY START AT ANY TIME.

CAUTION

DO NOT WEAR JEWELRY, LOOSE CLOTHING OR LONG HAIR WHEN OPERATING

All CNC machines contain hazards from rotating parts, belts and pulleys, high voltage electricity, noise, and compressed air. When using CNC machines and their components, basic safety precautions, including all appropriate LOTO procedures must always be followed to reduce the risk of personal injury and mechanical damage.

It is the machine owner's responsibility to make sure all personnel who are involved in installation and operation of this machine is thoroughly acquainted with the procedures and safety instructions provided herein *BEFORE* they perform any actual work.

Only Fryer factory-trained service personnel should troubleshoot and repair the equipment.

Do not modify or alter this equipment in any way without first consulting Fryer Machine. Any modification or alteration of this equipment could lead to personal injury and/or mechanical damage and could void your warranty.

EMERGENCY STOP (E-STOP) BUTTON

PRESSING THE EMERGENCY STOP BUTTON WILL INSTANTLY STOP ALL MOTION OF THE MACHINE, THE SERVO MOTORS, THE TOOL CHANGER, AND THE COOLANT PUMP.

1. Keep machine and area around it clean and well lit. Never allow chips, coolant, or oil to remain on the floor. Do not leave loose objects on or around machine.

2. Use appropriate eye and ear protection while operating the machine. ANSI-approved impact safety goggles and OSHA-approved ear protection is recommended to reduce the risks of eye injury and hearing loss.

3. Keep all loose clothing, hair, and jewelry away from the machine at all times.

4. Gloves are easily caught in moving parts. Take them off before using the machine.

5. Always wear safety shoes with steel toes and oil-resistant soles.

6. Do not paint, alter, deface, or remove any warning plates from the machine. Replacement plates are available from Fryer Machine Systems.

7. Keep flammable liquids and materials away from the work area and hot chips.

8. Coolant and oils can make surfaces on the machine slippery. They can also present an electrical hazard if the machine has power on. Therefore, do not stand on any part of the machine at any time.

9. Keep vises, clamps, fixtures, or work pieces from extending beyond the back edge of the table.

10. Check for damaged parts and tools before operating the machine. Any part of a tool that is damaged should be properly repaired or replaced. Do not operate the machine if any component does not appear to be functioning correctly.

11. Improperly clamped parts machined at high speeds/feeds may be ejected and puncture the safety door. Machining oversized or marginally clamped parts is not safe.

12. To avoid tool changer damage, ensure that tools are properly aligned with the spindle drive lugs when loading tools.

13. Windows must be replaced immediately if damaged or severely scratched – contact the factory for replacement panels.

14. Do not attempt to operate the machine before all the installation instructions have been completed.

15. Be sure to review the maintenance section of this manual for instructions to keep your machine running properly.

1.01 Mechanical Safety

1. Always press Emergency Stop when the machine is not in use.

2. Never operate the machine with any cover or shield open or removed.

3. Never reach into the work area when the spindle is turning or if the machine is in automatic mode.

4. Put the machine in manual mode and be sure last programmed function has been completed before reaching inside of the work area.

5. The functions of the machine make it impossible to eliminate all pinch points. Be particularly aware of the following pinch points:

   a. Spindle and tool rotation

   b. Indexing of ATC and tools

   c. Table, saddle, and head.

6. Do not operate machine without axis motor covers or axis way covers in place.

7. Report any loose, worn, or broken parts to your supervisor. The same action should be taken if any unusual noise or machine action occurs.

8. The electric components are protected from normal moisture resulting from humidity or use of water-based coolants. **DO NOT** however, use a water hose to clean the machine or the area around it.

9. Never touch a machine control device or electrical component when your hand is wet.

10. Never clean up chips while the machine is running or is in automatic mode.

11. Do not file workpieces being rotated under power.

12. At the end of the workday the machine should be placed in "EMERGENCY STOP MODE"

13. When restarting a machine after it has been shut down always assume it has been altered. Recheck all phases of the job as though you were running the first piece.

14. Never run the spindle until hands, feet, and body are well clear of the work area.

1.02 Electrical Safety

1. **WARNING**: Electrical enclosures contain high voltage. Disconnect equipment from power source before opening cabinets.

2. Before replacing a fuse, switch off the machine.

3. Immediately turn off power if:

- Power problems develop
- In the event of electrical storms.
- Ambient temperatures exceed 105 degrees Fahrenheit (40 degrees C)

4. The electrical power must meet the specifications in this manual. Attempting to run the machine from any other source can cause severe damage and will void the warranty.

5. The electrical panel should always be closed and locked except during service.

6. When the main circuit breaker is on, there is high voltage throughout the electrical panel and some components operate at high temperatures, therefore extreme caution is required.

7. Do not reset a circuit breaker until the reason for the fault is investigated.

8. Never service the machine with the power connected.

828 LATHE QUICK REFERENCE

Tool Setup

Adding tools to the tool list
Press machine hard key if not in manual / TFS screen Press tool hard key
Press tool list soft key if needed to show the tool list

1. Select any available tool location
2. Press "New Tool" soft key (may need to toggle screen)
3. Select tool type and press "ok"
4. Type tool name, press input
5. Define tool parameters (nose radius, holder angle, insert angle, insert length, cut direction, coolant, spindle rotation direction)

Repeat steps 1-5 above for each additional tool.

Getting tool to the TFS screen/active position
Press "machine" hard key if not in manual/TFS screen Press "select tool" soft key (upper right side of screen)
Highlight/select desired tool
Press "OK" soft key (lower right side of screen) - this will take you back to TFS screen
Press Cycle start (follow screen instructions if no tool changer)

Tool length offsets

1. Press measure tool soft key from main/TFS screen (verify active tool is the one being measured)
2. Press "Length manually" soft key (right side of screen)
3. Select axis to be measured
4. Select "workpiece edge" or "main spindlechuck" and define value(zero, shim thickness, etc. for z measurement) Define diameter if measuring X(uncommon)
5. Manually touch off tool, press "set length" soft key

Program Creation / Selection

Press program hard key or Program Manager soft key

Select existing program
Arrow/select desired program, press "open" soft key to open an existing program

Create new program
Press "new" soft key
From right side of the screen, select "Shop Turn" for conversational or "Program Guide G Code" for manually writing a g-code program

Type program name into box and press "OK" - this opens the program header - follow/fill out menu to define stock
Create program using "Drilling", "Turning" menus for many basic operations, or "Contour Turning" to draw geometry and create tool paths for the created geometry using "stock removal", "grooving", or "plunge turning" functions in Contour Turning menu.

Work Offsets

Load tool into active position / TFS screen
Press machine hard key if not in manual /TFS screen
Press tool hard key
Press work offset soft key (bottom of screen) Arrow/highlight desired work offset (G54 for example)
Press "WO Select" soft key
Press details soft key to clear offset if desired

Press machine soft key or manual hard key to get back to TFS/manual screen
Press "Set WO" soft key
Touch off tool to workpiece (or make skim cut) and press "Z = 0" soft key to set part zero

----

Program Execution / Program Run

After program has been completed, simulate to verify it is doing what is desired. If this checks out, press "Execute" soft key at lower right of screen to move program into memory. This will bring you to the "auto" screen area ("Auto" hard key will also bring you here)

Check spindle over-ride and federate over-ride knobs - suggest spindle at 100%, federate at 0 or 10% if testing/running a new program

If using handwheel run, press "handwheel run" key, followed by cycle start to begin executing the program (must select and axis and turn handwheel if using handwheel run)

If not using handwheel, press cycle start to execute the program. Use the federate over-ride knob to manage program speed.
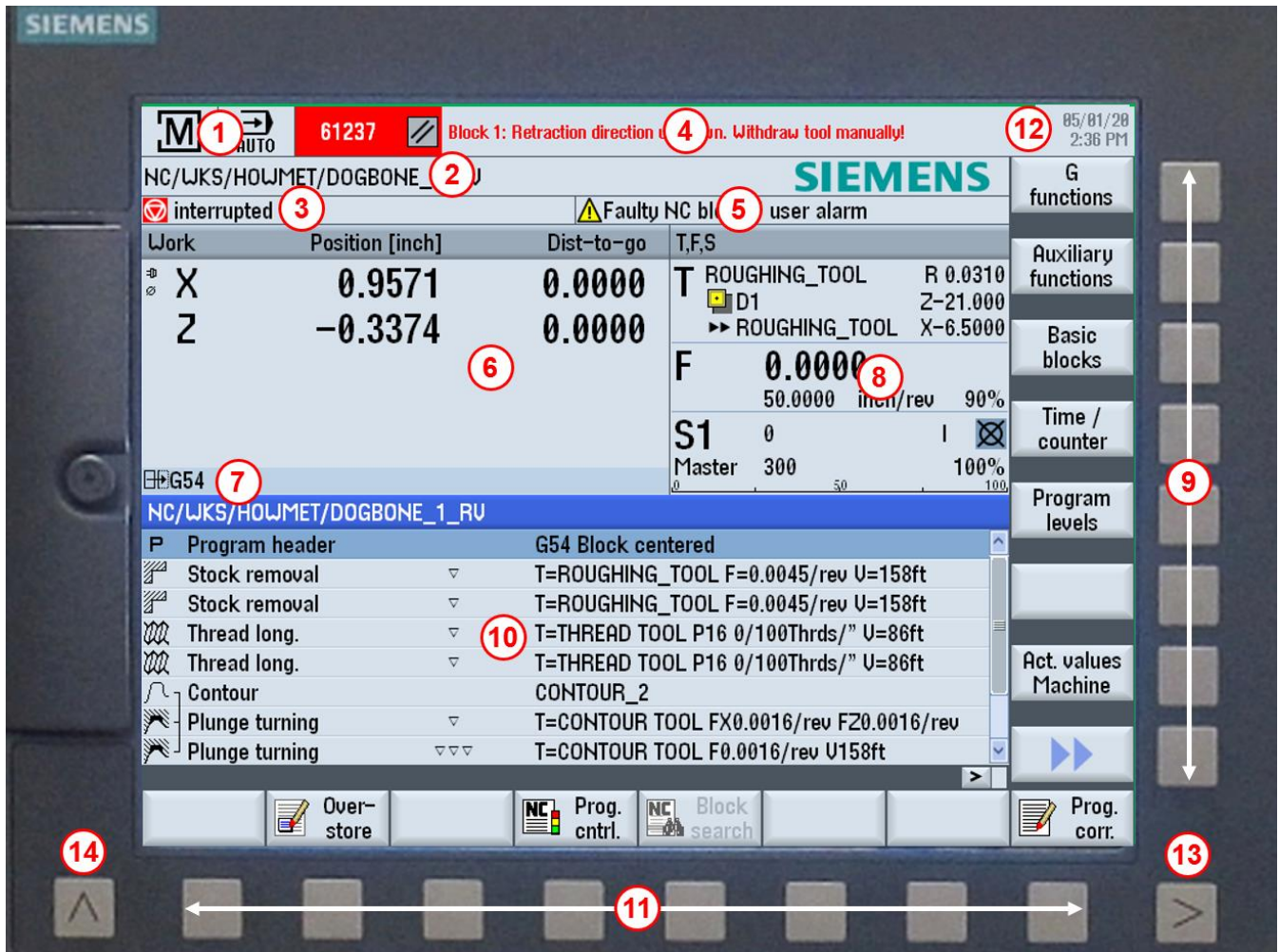
828 LATHE BASIC CONTROL INFORMATION

- **A - Screen**
- **B - Alpha Keypad**
- **C - Numeric Keypad**
- **D – Select & Cursor Keys**
- **E – Input / Enter Key**
- **F - USB, CF-card & Ethernet Access**
- **G – HELP Key**
- **H – Mode Select Keys**
- **I – Shortcut Hot Keys**
- **J – Auxiliary Keys**
- **K – Cycle Start /Feed Hold**
- **L - Jog Keys**
- **M - Coolant Control**
- **N - Spindle Control**
- **O – Manual Spindle Override**
- **P – Manual Feedrate Override**
- **Q - Emergency (E- Stop)**
- **R – Protection Key Switch**
- **S – Z Axis Handwheel**
- **T – Fine / Coarse Feed Control**
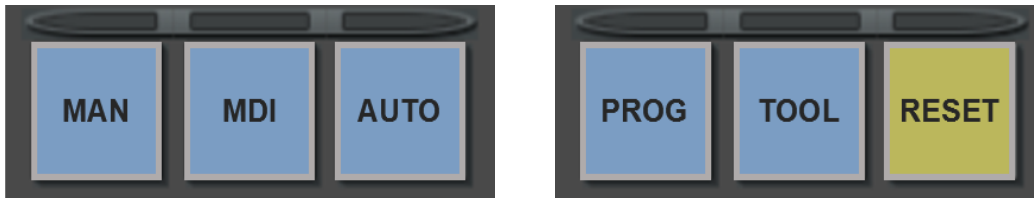- **U – X Axis Handwheel**
- **V - Joystick**

BASIC SCREEN / CONTROL INFORMATION

1. **Operation Sector**
2. **Program Path & Name**
3. **Program Status**
4. **Alarm & Message Line**
5. **Operation Messages**
6. **Axes Position**
7. **Active Work Offset**
8. **Display of:**
   T = Active Tool
   F = Present Feedrate
   S = Actual Spindle RPM
   Spindle Load %

9. **Vertical Softkeys**
10. **Working Window**
11. **Horizontal Softkeys**
12. **Date & Time**
13. **Extend Menu Key**
14. **Recall Key**

OPERATION MODE SELECT KEYS

- **MAN** - Manual Operating Screen; functions available for manual operation of the machine.
- **MDI** - Manual Data Input Screen; enter G-code commands for manual machine operations.
- **AUTO** - Run Program Screen; programed machining operations are controlled from here

- **PROG** - Program Manager Screen; programs are written and stored here

- **TOOL** - Tool List Screen; data pertaining to tools and work offsets are accessed

- **RESET** - Resets the currently active program and some alarms. Note: Using the reset key will reset the program to its beginning.
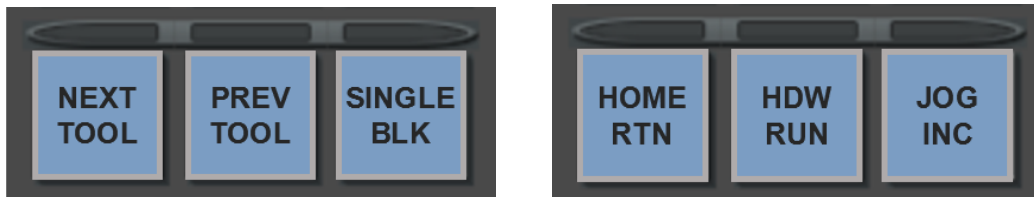
MACHINE MODE HOT KEYS

- **NEXT TOOL / PREVIOUS TOOL** - Selects the next or previous tool in the carousel and will skip blank stations. For lathes with a turret only.

- **SINGLE BLOCK** - Program will run one block at a time. Cycles like tapping will complete the in and out motion before stopping.

- **HOME RTN** - Machine is equipped with absolute encoders and always remembers its last position before shutting down. When pressing this key, a message of press **CYCLE START** to continue comes up. All axes move to their home position and stop. Machine position will now display Zero for all axes.

- **HDW RUN** (option) - Allows the machine to follow the program by turning the Z axis handwheels. When you reverse the motion of the handwheel it will back up to the beginning of the last program block.

- **JOG INCR** -   Changes the resolution of motion when pressing the axis jog keys. Toggling the key will show in the upper left corner of the screen the amount of movement per keystroke. 1=.0001", 10=.001" 100=.010"

AUXILIARY KEYS

- **TEACH** (option) - Allows you to store and save positions in G code format when moving the axes. This can be saved as program or run in **MDI** mode

- **REPOS** - Allows you to go back to the point you stopped your program without having to run all the previous movements again.

- **LIGHT** - Turns work light on and off. Light will come back on automatically when servos are turned on and reset is selected.


- **CYCLE START** – Starts the execution of the CNC in Auto Mode or operation in Manual Mode.

- **FEED HOLD** – Pauses feedrate during running of a program. Resumes after selecting CYCLE START key.
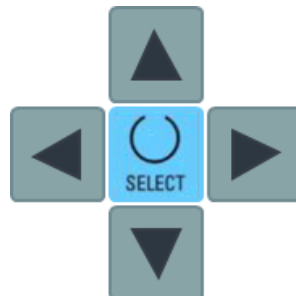
- **CURSOR** Keys - Navigates through the various fields or lines on the screen.

- **SELECT** - Makes changes to many different operations on the control. It allows you to change axis moves from absolute to incremental, spindle direction, how the tool enters the part etc.
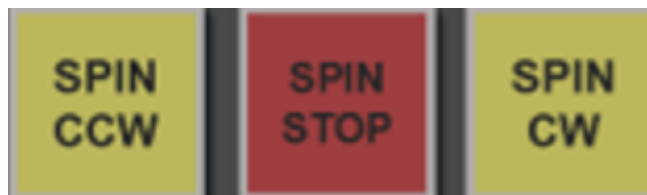

- **SPIN CCW / STOP / SPIN CW** – Control the direction after the spindle has been activated. **SPIN STOP** will stop the spindle, pressing either direction button will start it again.
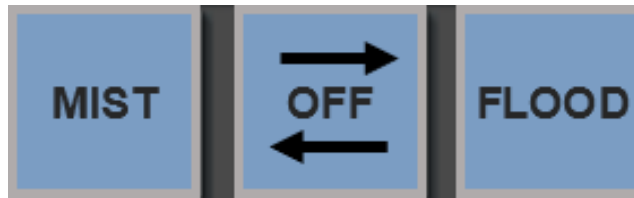
- **COOLANT** Keys – Manually turn coolant on or off. Mist is an available option.

- **HELP** - When you open a cycle and are not sure what it is asking for, select the Help key. The left side of the screen will display an explanation of every line in that cycle. You can also access the entire programming manual from this key.
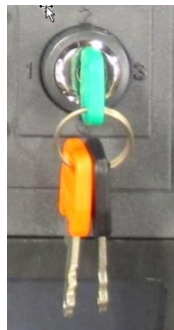

- **PROTECTION KEY SWITCH** - Reading and writing of user data are protected using the key switch. Provides a multilevel safety concept for controlling access to programs, data, and functions at various protection levels.

ADDITIONAL KEYS ON THE OPERATOR PANEL

- **MACHINE** - Opens the operating area Manual Operating screen. Corresponds to the **MAN** Operation Mode Select Key.

- **PROGRAM** - Opens the last program that was edited.

- **OFFSET** - Opens the Parameter screen for Tool list, Tool wear, Magazine, Work offset, User variable, and Setting data. Corresponds to the **TOOL** Operation Mode Select Key.
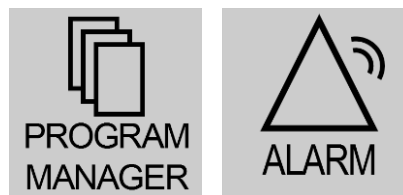
- **PROGRAM MANAGER** - Opens the operating area Program Directory screen. Corresponds to the **PROG** Operation Mode Select Key.

- **ALARM** - Opens the Alarm List screen. Return to the Manual Screen by pushing the **MAN** Operation Mode Select Key.

- **BACKSPACE** – When editing in a field block it deletes a character selected to the left of the cursor, when navigating in a program it deletes all the selected characters to the left of the cursor.

- **DEL** - When editing in a field block it deletes a character selected to the right of the cursor, when navigating in a program it deletes all characters.

- **INSERT** - Opens an editing window in the insert mode. Pressing the key again, exits the window and the entries are undone. When used in conversational programming it opens a selection box and shows the available options.
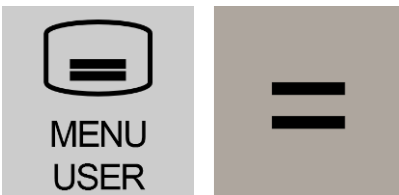
- **MENU SELECT** – Opens a screen with yellow horizontal keys to access various operating areas. Access to Diagnostics and Machine / Control data is found here.

- **EQUALS** - Opens a calculator that allows user to calculate values for entry fields. Choose between a simple standard calculator and the extended view with mathematical functions.

The following keys can be used in combinations are available for operation of the control and the machine tool.

Detailed information can be found in the **SIEMENS 828 TURNING OPERATING MANUAL**
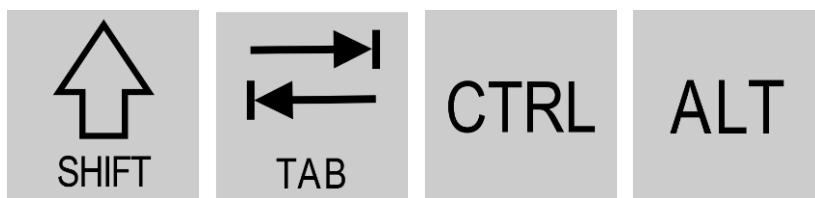
- **SHIFT** - If the Shift-key is held depressed, the upper character on keys with double usage will be entered.

- **TAB** - In the program editor, indents the cursor by one character. In the program manager, moves the cursor to the next entry to the right.

- **CTRL** – Used in combination with other keys navigate through programs.

- **ALT** - Used in combination with other keys navigate through programs.

- **NEXT WINDOW** - Toggles between the windows. Selects the first entry in selection lists and in selection fields and moves the cursor to the beginning of a text.

- **PAGE UP** - Scrolls up by one page in a window.

- **PAGE DOWN** - Scrolls down by one page in a window.

MAN - MANUAL OPERATING SCREEN

MAN
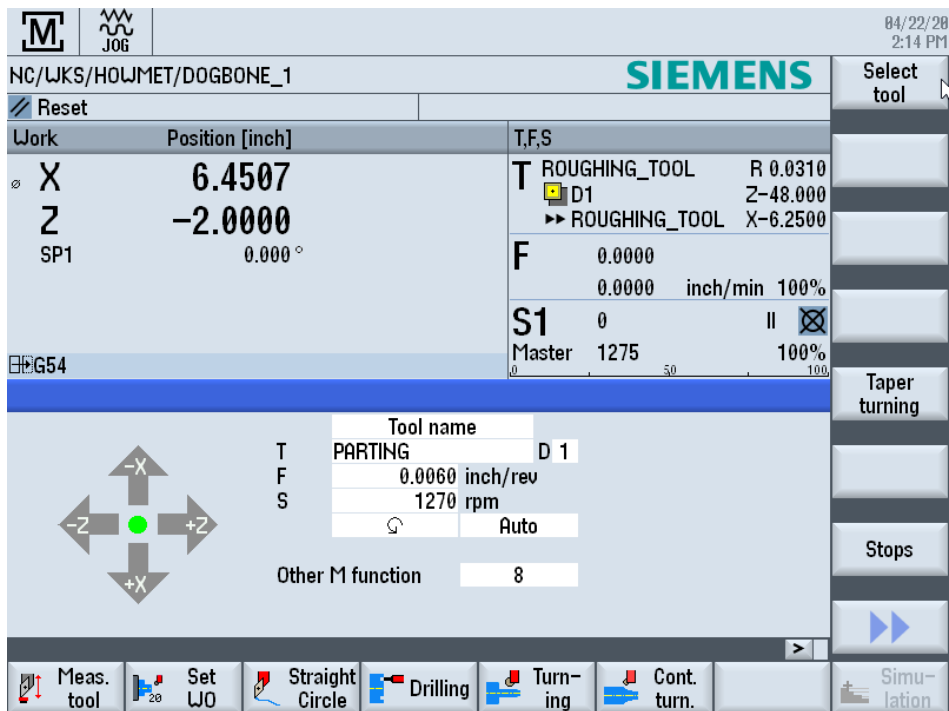
The following functions are available by selecting the designated field and typing in the information or toggling the **SELECT** key for options, select the **INPUT** key and select **CYCLE START** key to execute:

- Perform a tool change (**T**) with direct access from the tool table,
- Designate feedrate (**F**) - **IPR** or **IPM**
- Define the spindle speed (**S**) **RPM** or **CSS** and direction, **CW**, **CCW** or **STOP**
- Specify the gear stage - **Auto** or **I**, **II** (**III**, **IV** for ET30 & 40 lathes)
- o On ET30 & 40 machines, the gears may need to be manually shifted if not equipped with the optional Automatic Gear Shift option.
- Enter M function (**M**) if needed

Other Functions Available:

- **TAPER TURNING** – Move X & Z simultaneously at any angle
- **MEAS. TOOL** – Measure tool (X & Z lengths)
- **SET WO** – Set workpiece coordinates

DO ONE CYCLES
(See section on DO ONE CYCLE OPERATIONS for descriptions)

- **STRAIGHT / CIRCLE** - Straight or Circular Turning
- **DRILLING** – Drilling, Tapping
- **TURNING** – Stock Removal, Groove, Undercut, Thread and Cutoff
- **CONTOUR TURNING** - Contour, Stock Removal, Grooving, Plunge Turning
- **SYNCH THREAD** – Thread Repair

MOVING THE MACHINE
USING THE HANDWHEELS

In the **MANUAL** screen, enter a feed rate in the **F** field and select the **INPUT** key

1. **Z-AXIS** electronic handwheel

2. **X-AXIS** electronic handwheel

The handwheel feedrate is adjustable with either the Course/Fine Switch or the Feedrate Override Knob.

3. **COARSE / FINE SWITCH** - Allows you to easily switch between fast or slow movement of the handles or joystick.

4. **4-POSITION JOYSTICK** - Simple joystick feed control allows positioning of the axes with a steady feedrate. The feedrate is adjustable with either the course/fine switch or the feedrate override knob.

MOVING THE MACHINE THROUGH THE CONTROL
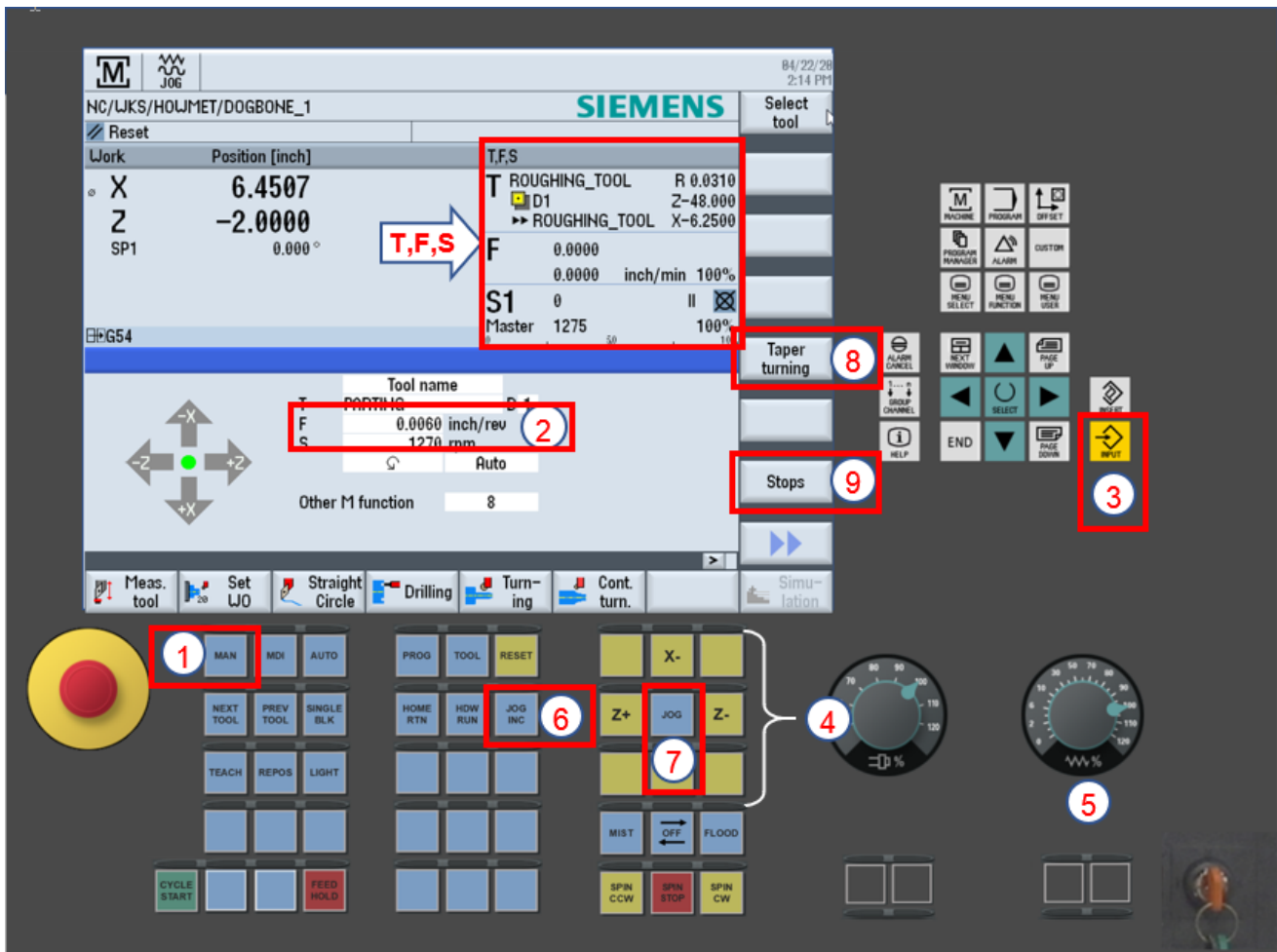
1. In the **MANUAL** screen

2. Enter a feed rate in the **F** field

3. Select the **INPUT** key

4. Select any of the **AXIS** keys (+/- determines movement direction).

5. Feed rate can be overridden with the **FEED OVERRIDE** knob, actual feedrate will be displayed in the **T, F, S** Screen

6. Selecting the **JOG INC**, will allow movements incrementally each time the **AXIS** key is Selected; (1=.0001", 10=.0010", 100=.010")

7. Selecting the **JOG** key will cancel the incremental movement.

8. Selecting the **TAPER TURNING** key allows movement to cut tapers or radius machining.

9. Select the **STOPS** key – Electronically allows you to set a stop position for any axis, preventing axis movement beyond the stop position.

T, F, S DISPLAY SCREEN


```
T,F,S

T  ROUGHING_TOOL           R 0.0310
    ◨ D1                    Z-48.000
    ►► ROUGHING_TOOL        X-6.2500

F        0.0050
         0.0050    inch/rev  100%

S1       1270              ‖   ↻

Master   1270                  100%
 0                50             100
```
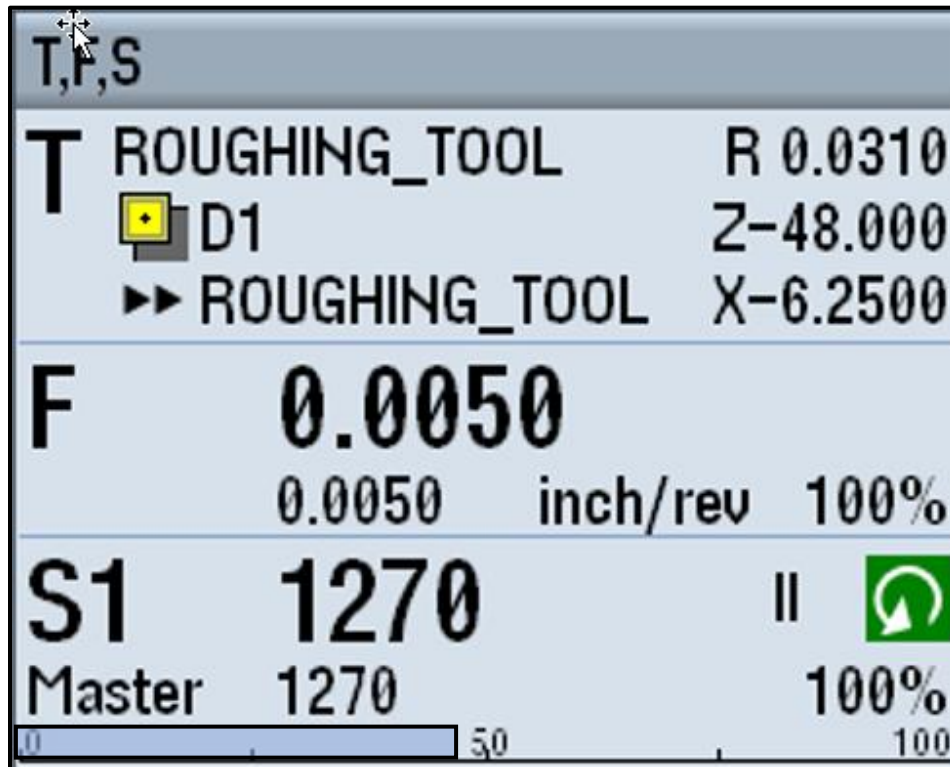
When the machine is running, this information is displayed.

- **T**: (Tool) Name of the active tool, edge number, radius of the tip and the tool offset data to the right.

- **F**: (Feed) Display of the active feed rate for the current operation in bold during machining, the programmed feed rate is under and the feed override in % is to the right.

- **S**: (Spindle) Display of the active spindle speed for the current operation in bold during machining, the programmed spindle speed is under and the speed override in % is to the right. Also displays current gear selection.

- Spindle load factor in % is displayed by a blue bar across the bottom.

MAN

MANUAL SCREEN INPUT OPERATIONS
(T, F, S, Other M Function)

	Tool name	
T	PARTING	D 1
F	0.0060	inch/rev
S	1270	rpm
	↻	Auto

Other M function	8

"T" Field

Select tool
- To enter a tool in the **T** (tool name) field select the **SELECT TOOL** key on the right-hand side of the screen

OK
- In the tool library move the cursor arrow until you find the tool you want then select the **OK** key on the bottom right hand side of the screen

CYCLE START
- Now select the **CYCLE START** to activate the tool loading program either for manual insertion or automatic

- To unload the tool from the spindle, type in **0** in the **T** field and press **CYCLE START**

"F" Field

INPUT
- To change the feed rate (**IPM** or **IPR**), enter the value in the **F** (Feed) field, select **INPUT**. This will change the feed rate in any manual jogging of an axis.

"S" Field

SELECT
- To start the spindle, enter an **RPM** or in the S (spindle) field

- In the next field below change the spindle direction by selecting the **SELECT** to toggle between **CW** or CCW; then select the **CYCLE START**

CYCLE START
- In next field to the right, make a software gear change by selecting the **SELECT** to toggle between **AUTO**, **I** or **II**; then select the **CYCLE START**

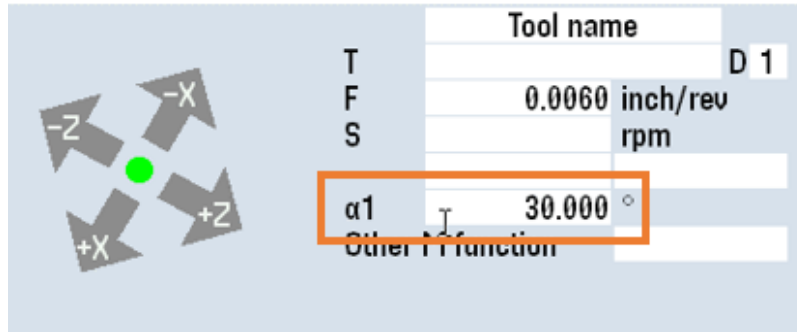**"Other M Function" Field**

CYCLE START
- Enter an M-Code number in the **OTHER M FUNCTION** field; you do not need to put an "M" in front of it, just the number, and select the **CYCLE START**

TAPER TURNING key

Taper turning

- Select this key, the screen will change to allow a taper angle to be entered in the Angle Degree Field and select **INPUT** key.

INPUT

Taper turning

- By moving the X or Z axis with the Remote Handwheel or Axis keys, the movement will allow both Axes to move simultaneously at the selected angle.
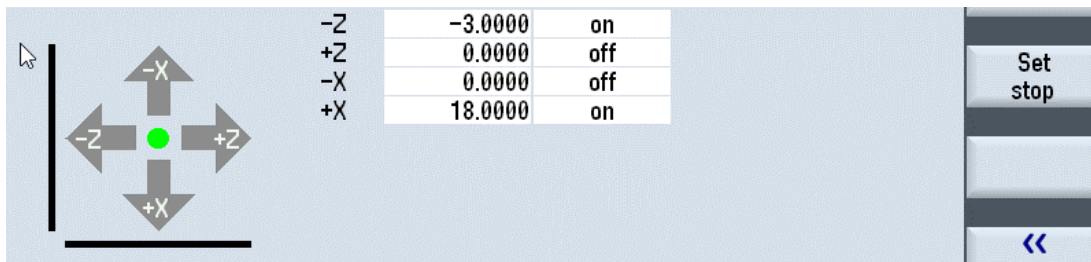
- Select the **TAPER TURNING** key again to turn off.

STOPS Key

- Move X or Z axis to any location while in the **MANUAL** screen

Stops

- Select **STOPS** key, the screen shows which axis (+/-) limit(s) can be set.

Set stop

- Highlight the required axis and either enter the desired limits or automatically set by selecting the **SET STOP** key on the right side of the screen.

SELECT

- Toggle the **OFF / ON** with **SELECT** key to activate or deactivate the stop. Black bars will appear next to the activated axis.

- Select the **BACK** key to return to the **MANUAL** screen.

TOOL LIST SCREEN

Tool operating data can be managed in the tool list. Column headers:

- **LOC** – Position in turret, if equipped.

- **TYPE** – Turning tool, drill, etc.

- **TOOL NAME** – Unique description of tool.

- **D** – Cutting edge number (max. of 9).

- **H** – Designation tool length when using ISO programming.

- **LENGTH X / LENGTH Z** – Geometry data length.

- **RADIUS / DIAMETER** – Cutter tip radius or drill & boring bar diameter.

- **CUTTING EDGE GRAPHIC** – Shows tool edge angle & direction.

- **ANGLE** – Toolholder, drill tip or tap pitch.

- **TIP LENGTH** – Approximate depth of tool edge cut.

- **SPINDLE DIRECTION** – CW or CCW. Drills normally opposite direction from turning tools, unless a live tool, if equipped.

- **COOLANT 1 / 2** – Automatically activates when checked.

Functions available:

- **TOOL MEASURE** – Measure tool lengths X & Z, manually.

- **NEW TOOL** – Opens screen to identify a new and associated data.

- **EDGES** – Add additional cutting edges descriptions for tools.

- **LOAD / UNLOAD** – Allows changing tool positions in a turret, if available.

- **DELETE** – Clears all data on selected tool.

- **TOOL WEAR** - Enter length & diameter wear. Monitor the tools' working times by workpiece count, tool life or wear.

Loc.	MT LO.	Type	Tool name	D	ΔLength X	ΔLength Z	Δ ⌀	T C		
1		⬛	5/16 DRILL 3/8-16	1	0.0000	0.0000	0.0000			
2		⚠	3/8-16 TAP	1	0.0000	0.0000	0.0000			
3		⬤	3D_PROBE	1	0.0000	0.0000	0.0000			
4										
5										
6		⬛	BORING BAR 5/8	1	0.0000	0.0000	0.0000			
7		⬛	BORING BAR 3/4	1	0.0000	0.0000	0.0000			

Tool wear — turret — 04/27/20 2:29 PM — Sort / Filter / Search / Details

- **WORK OFFSET** - This view lists all offsets in Machine Coordinate System (**MCS**) & Workpiece Coordinate System (**WCS**), rotations, scaling and mirroring.

- Under each work offset is a Fine adjustment row that lets user make small corrections to a position without changing the initial offset.

Work offset – G54 … G599 [inch] — 04/27/20 2:12 PM

				X	Z	SP1	
G54				0.0000	64.5100	0.000	
	Fine			0.0000	0.0000	0.000	
G55				0.0000	0.0000	0.000	
	Fine			0.0000	0.0000	0.000	
G56				0.0000	0.0000	0.000	
	Fine			0.0000	0.0000	0.000	
G57				0.0000	0.0000	0.000	
	Fine			0.0000	0.0000	0.000	

Workpiece zero point / Active / Overview / Base

- **R VARIABLES** - Channel-specific variables that you can use within G-code programs or **SHOPTURN** programs.

R variables — 05/05/20 2:31 PM

R 0	0	R 20	0	R 40	0.75
R 1	0	R 21	0	R 41	1.06066017177982
R 2	0	R 22	0	R 42	0
R 3	0	R 23	0	R 43	0
R 4	0	R 24	0	R 44	0
R 5	0	R 25	0	R 45	0
R 6	0	R 26	0	R 46	0

R variables / Global GUD

CREATE A TOOL IN THE LIBRARY

TOOL

Tool list										turret
Loc.	MT LO.	Type	Tool name	D	H	Length X	Length Z	Radius		
1		⬛	5/16 DRILL 3/8-16	1	0	-16.8319	0.0000	0.3125		118.0
2		⚠	3/8-16 TAP	1	0	-16.8319	-40.0000	0.3750		16.000
3		⬤	3D_PROBE	1	0	-16.8319	-101.167	0.2224		
4										
5										
6		⬛	BORING BAR 5/8	1	0	-34.4516	-105.730	0.0150 ←		95.0 8
7		⬛	BORING BAR 3/4	1	0	-34.6683	-105.758	0.0150 ←		95.0 8
8		⬛	ROUGHING_TOOL	1	0	-26.2200	-38.0000	0.0312 ←		93.0 5
9		⬛	FINISHING_TOOL	1	0	-26.2200	-38.0000	0.0320 ←		93.0 5
10										
11		⬛	COUNTERSINK 60D	1	0	-16.8319	-107.871	1.2500		60.0
12		⬛	3/8 ENDMILL	1	0	-16.8319	0.0000	0.3750		180.0
		⬛	THREAD CUTTER	1	0	0.0000	-37.3070	0.0000	1	

New tool ▶

Load ▶

04/27/2 2:28 PM

Tool list

- The **TOOL LIST** is opened.

- Select the position where the new tool should be stored using the cursor arrow keys.

- Select an empty magazine location or a position below the magazine numbers.

New tool ▶

- Press the **NEW TOOL** key.

- The **NEW TOOL – FAVORITES** window opens.

or

- To create a tool that is not in the "**Favorites**" list, press the key "**Cutters 100-199**", "**Drill 200-299**", "**Turntools 500-599**" or "**Spec.tool 700-900**", and their respective window will open.

✓ OK

- Select the tool by moving the cursor arrow keys to the selected tool and select the **OK** key. This window will close.

New tool – favorites		
Type	Identifier	Tool position
	Tool from file	
500 –	Roughing tool	
510 –	Finishing tool	
520 –	Plunge cutter	
540 –	Threading tool	
550 –	Button tool	
560 –	Rotary drill	
580 –	3D turning probe	
730 –	Stop	
120 –	End mill	
140 –	Facing tool	
150 –	Side mill	
200 –	Twist drill	
240 –	Tap	
	Multitool	

INPUT TOOL DATA

Type	Tool name	D	H	Length X	Length Z	⌀		Tip angle			⏚	🔃1	🔃2
▣	ROUGHER	1	0	−21.0000	−45.0000	0.0315	←	95.0	80	0.4	↺	✔	☐
▣	FINISHING_TOOL	1	0	−21.0000	−45.0000	0.0157	←	93.0	55	0.4	↺	✔	☐
▣	ROUGHING_TOOL	1	0	−21.0000	−45.0000	0.0012	←	95.0	80	0.4	↺	☐	☐
▣	CNMG432_OD	1	0	−21.0000	−45.0000	0.0012	←	95.0	80	0.4	↺	☐	☐
▰	.562 DRILL	1	0	−20.7190	−41.5000	0.5620		180.0			↻	☐	☐
▰	9.5 SPOT DRILL	1	0	−20.8125	−41.5000	0.3750		82.0			↻	☐	☐
▦	3/8−16 TAP	1	0	−21.0000	−45.0000	0.3750		16.000			↻	☐	☐
▣	1.5IN BORE BAR	1	0	−21.0000	−45.0000	0.0157	←	95.0	80	0.4	↺	✔	☐

- Name the tool with a unique description, i.e., "ROUGHING TOOL"

- Length X & Z data is entered when measuring the tool, leave with 0.0000 at this time.

- Enter Radius of the tip, direction of cut and toolholder angle for turning tools.

- Enter data for Tip width (Grooving & Parting) / Tip angle (Drills) / Pitch (Taps)

- Select the direction of rotation with **SELECT** key. Note that drills have the spindle arrow CCW for a static drill but CW for a live tool drill, if equipped with a live tooling turret.

- Check the coolant field to automatically turn on in the program, with the **SELECT** key.

- After all data is entered, select either the **TOOL MEASURE k**ey to continue with measuring the tool or select the **MAN** key to return to the **MANUAL** screen.

TOOL

MEASURE TOOL LENGTHS

Meas. tool

- Select the **MEAS(URE) TOOL** key, the measurement options will appear on the right side. Select either X or Z to measure. Make sure the tool shown is the one needing to be measured.

Set length

- When measuring the X length, enter the measured diameter of the part in the spindle. Move the tool until the part is touched and push the **SELECT LENGTH** key.

Measure: length manually

T	3-UNMG-332	D 1	Tool data	
X0		1.0490	X	−15.6760
			Z	−109.2378
			R	0.0320

Set length

- When measuring the Z length, the preferred method would be to select the **Main Spindle Chuck** for reference and push the **SET LENGTH** key.

Measure: length manually

T	ROUGHING_TOOL	D 1	Tool data	
Ref. point	Main spindle chuck		X	−6.5000
			Z	−21.0000
			R	0.0310

.

Set length

- The other option would be the **Workpiece Edge**. Move the tool until the part is touched and push the **SET LENGTH** key.

Measure: length manually

T	ROUGHING_TOOL	D 1	Tool data	
Ref. point	Workpiece edge		X	−21.0000
Z0		0.0000	Z	−45.0000
			R	0.0012

- Select the **BACK** key on the right-hand side to get back to the **MANUAL** screen.

SET WORK OFFSETS

- In the **MANUAL** screen, push **SELECT TOOL** key and highlight a Tool or Edge Finder in the tool library.

- Select the **OK** key.

- Select **CYCLE START** to load the tool.

- Move the tool until it touches the workpiece face for Z.

- Select **SET WO** key to open the upper right screen where Z is highlighted. Select the Z=0 key to set to 0.0000 or key in using the numerical keypad any number, for example: 0.020" for a facing dimension.

- Select the **RETURN** key to get back to the **MANUAL** screen.

DO ONE CYCLE OPERATIONS

Straight Circle	Drilling	Turn-ing	Cont. turn.

Synch. thread

More detailed structuring of the machining operations can be found in the Programming Section the **SIEMENS 828 TURNING OPERATING MANUAL**

- In the **MANUAL OPERATING SCREEN**, you can execute **DO ONE** operation without writing a program.

- This is done by selecting one of the keys located across the bottom of the screen.

- Fill in the various parameters for tool, feed, speed, and dimensions and hit **OK** to run the cycle in manual mode.

- **STRAIGHT-CIRCLE** lets you do simple line and arc moves by filling in end points and feed rates.

- **DRILLING** allows drilling on center or tapping on center of the face of the part.

- **TURNING** gives you cycles for Stock Removal, Groove turning, Undercut machining, Thread turning and Cutoff (Parting) operations. All operations except Cutoff can be done externally and internally.

- **CONTOUR TURNING** lets you create simple or complex tool paths and then connect them to a machining cycle like Stock Removal, Grooving or Plunge Turning. All operations can be done externally and internally.

- **SYNCH THREAD** (Thread Repair) permits the ability to re-machine a thread by synchronizing the spindle to the existing thread pitch.

STRAIGHT / CIRCLE Key

- This gives you options of where you want to move your Axis, either Individually or multiple axes and at your required feed rate or rapid movement.

- You have options of making linear movements, arcing movements, or circular movements.

- Select **GRAPHIC VIEW** to open a small window showing what is programmed.

- You must have the necessary tool in the spindle and **RPM or FT/MIN** and correct **GEAR** and set in the starting position prior to doing this **DO ONE CYCLE**.

- When all the information is entered, select the **CYCLE START** to display in the **AUTO** screen to start the operation.

- Activate the spindle before pushing the **CYCLE START** key. The **FEED OVERRIDE** knob can be used to adjust the programed feed rate.

DRILLING Key

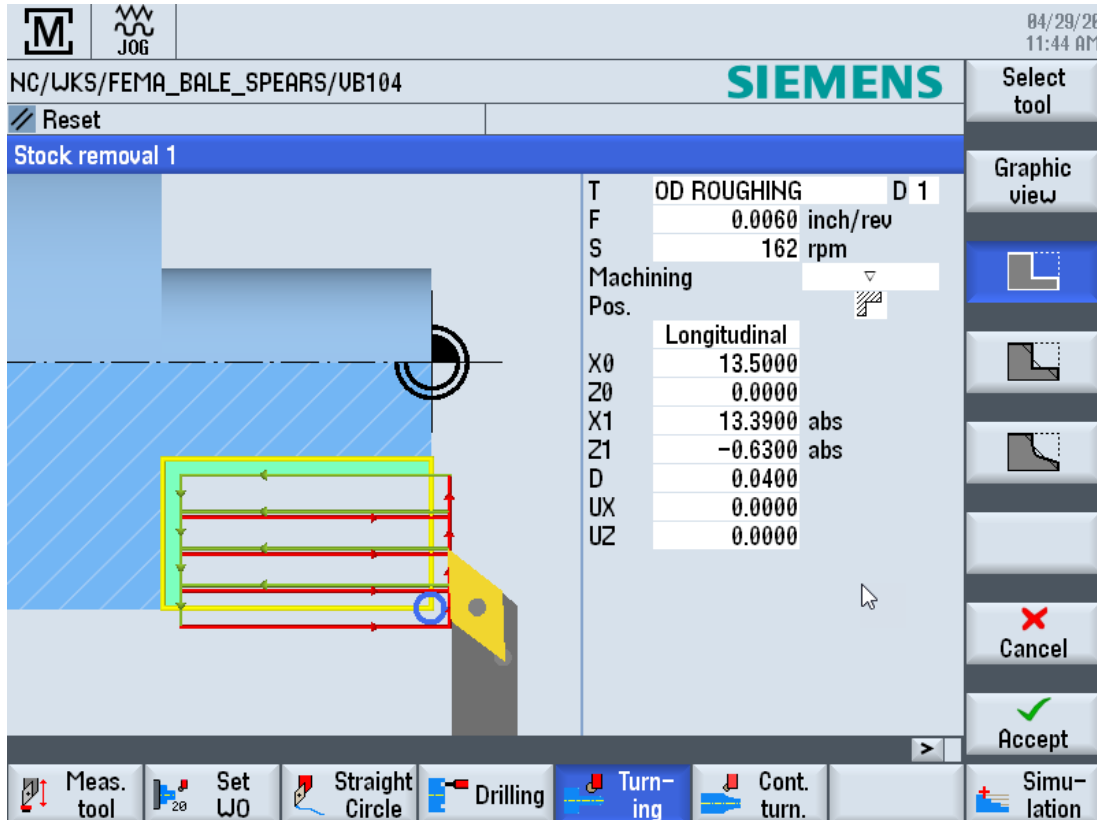- This gives you options for **DRILLING** or **THREADING** operations.
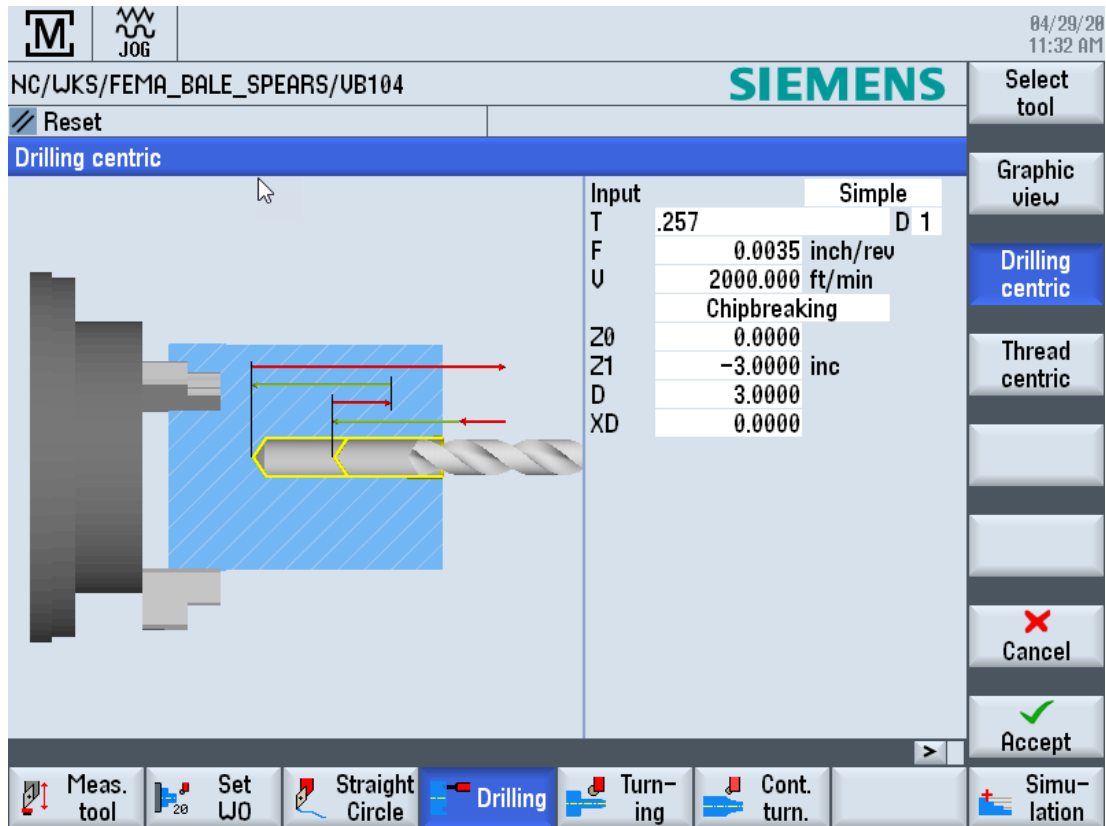
- Select **GRAPHIC VIEW** to open a window showing what is programmed.

- Prior to starting, set your Z work offset.

- Move the tool to the position where the machining operation will start in Z and select the **SET WO** key to set your Z work offsets. Then back away from that.

- When all the information is entered, select the **ACCEPT** key to display in the **AUTO** screen.

- Activate the spindle before pushing the CYCLE START key. The **FEED OVERRIDE** knob can be used to adjust the programed feed rate.

TURNING Key

- This gives you options for **STOCK REMOVAL**, **GROOVE TURNING**, **UNDERCUT MACHINING**, **THREAD TURNING** and **CUTOFF** (Parting) operations.

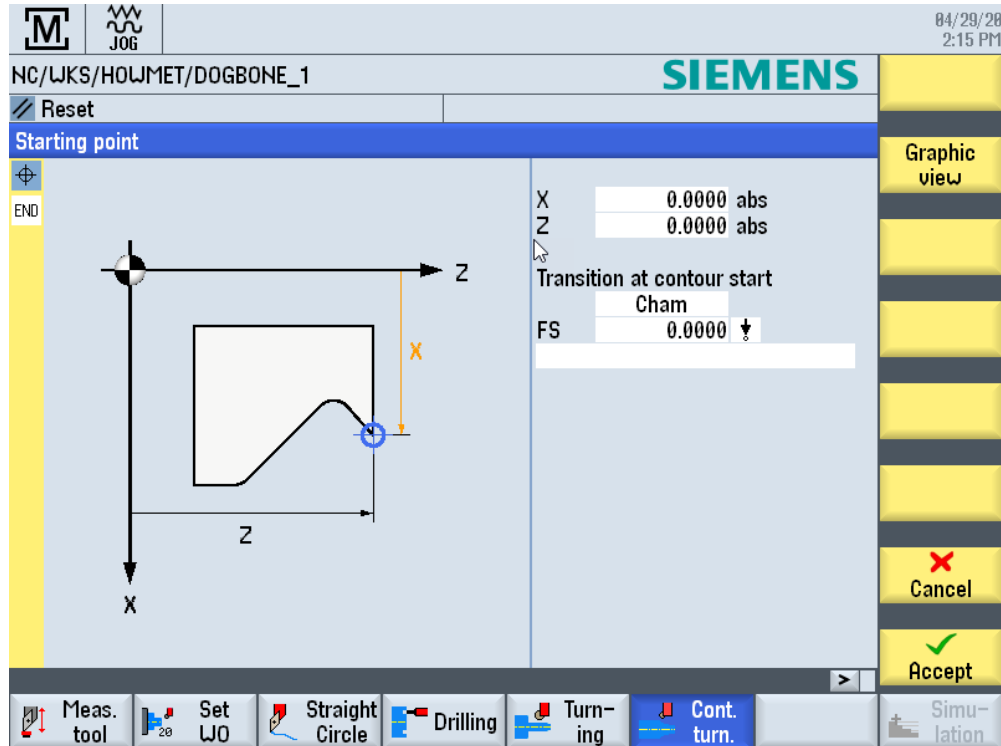- All operations except **CUTOFF** (Parting) can be done externally or internally.

- Select **GRAPHIC VIEW** to open a window showing what is programmed.

- Prior to starting, set your Z work offset.

- Move the tool to the position where the machining operation will start in Z and select the **SET WO** key to set your Z work offsets. Then back away from that.

- When all the information is entered, select the **ACCEPT** key to display in the **AUTO** screen.

- Activate the spindle before pushing the **CYCLE START** key. The **FEED OVERRIDE** knob can be used to adjust the programed feed rate.

CONTOUR TURN Key

- This gives you options to create simple or complex tool paths and then connect them to a machining cycle like **STOCK REMOVAL**, **GROOVING** or **PLUNGE TURNING**.

- All operations can be done externally or internally.

- Select **GRAPHIC VIEW** to open a window showing what is programmed.

Prior to starting, set your Z work offset.

- Move the tool to the position where the machining operation will start in Z and select the **SET WO** key to set your Z work offsets. Then back away from that.

- Select **NEW CONTOUR** key and the Starting Point window opens to set the first point in the contour. If doing a repeated contour, the **LAST CONTOUR** key can be selected if there is no interruption in operations.

- When the contour is completed select the **ACCEPT** key to display in the **AUTO** screen where optional machining keys are displayed on the right, i.e., **STOCK REMOVAL**.

- Select the machining option key needed and fill in the required data and select the **ACCEPT** key. The contour and machining operation is ready to start.

- Activate the spindle before pushing the **CYCLE START** key. The **FEED OVERRIDE** knob can be used to adjust the programed feed rate.
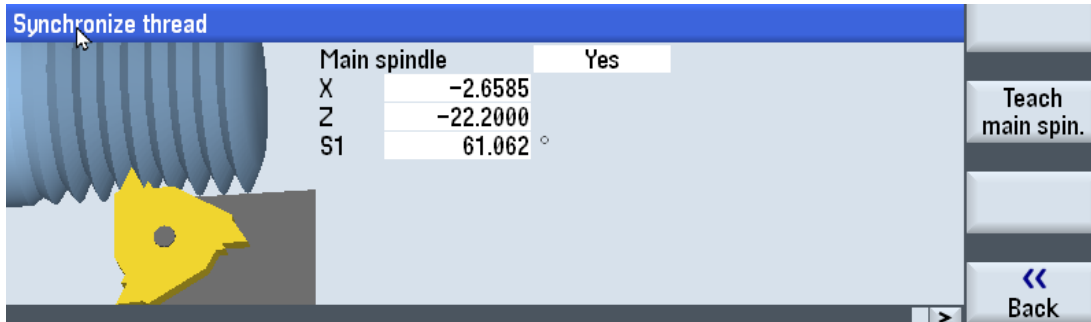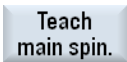
SYNCH THREAD Key (Thread Repair)

> - Select the **MENU FORWARD** key.

[Synch. thread]

- Select the **SYNCH THREAD** key.

Synchronize thread

Main spindle	Yes
X	-2.6585
Z	-22.2000
S1	61.062 °

Teach main spin.

« Back

- Using the handwheels, locate the thread cutting tool tip into the thread turn as shown in the help screen.

[Teach main spin.]

- Select the **TEACH MAIN SPIN** key to set the location of the thread.

- Move the tool tip away in the X axis and then move the tool tip away from the part in the Z axis.

[« Back]

- Select the **BACK** key.

>

- Select the **MENU FORWARD** key to return to the **MANUAL** screen.
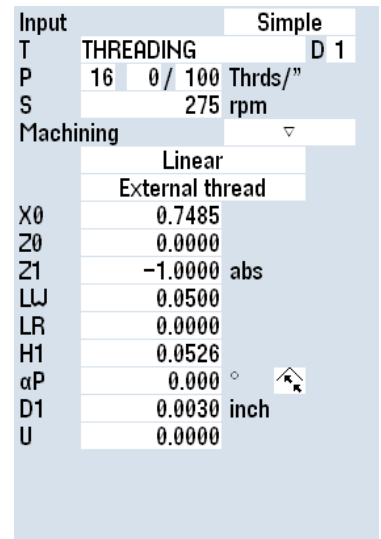
[Turn-ing]

- Select the **TURNING** key.

[Thread]

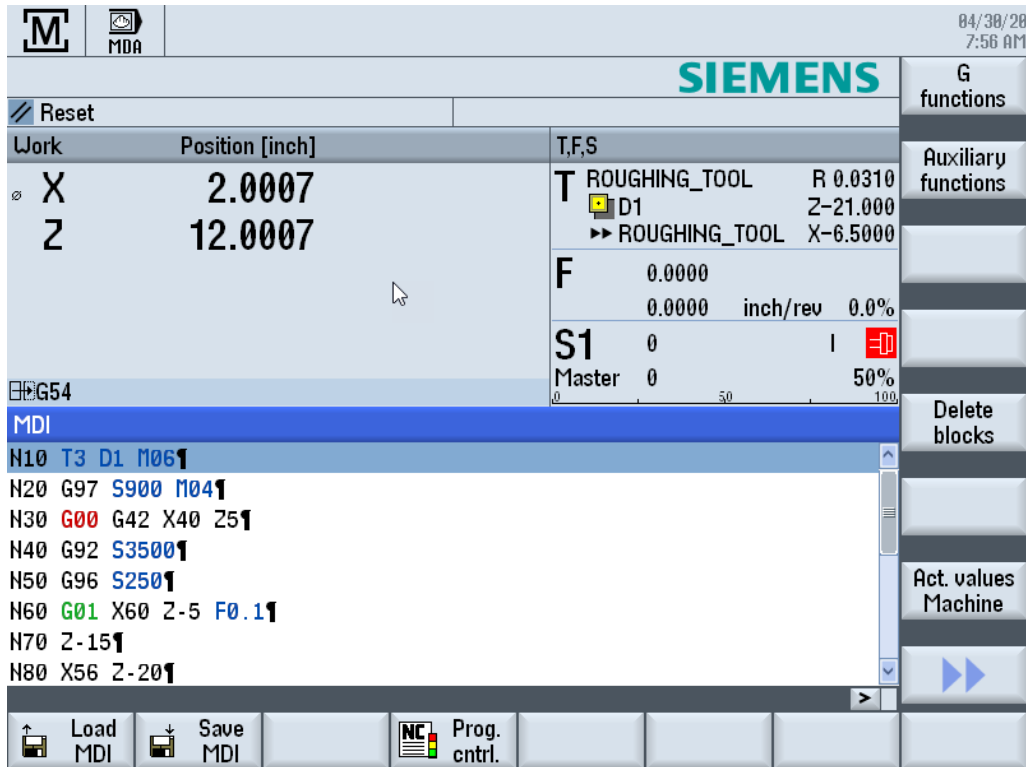- Select the **THREAD** key on the right side of the screen.

- When all the information is entered, select the **ACCEPT** key to display in the **AUTO** screen.

- Activate the spindle before pushing the **CYCLE START** key. **NOTE:** The **FEED OVERRIDE** knob will **NOT** adjust the programed feed rate in this cycle. Use the **SPINDLE OVERRIDE** knob to control, if needed.

Input		Simple
T	THREADING	D 1
P	16 0 / 100	Thrds/"
S	275	rpm
Machining		▽
	Linear	
	External thread	
X0	0.7485	
Z0	0.0000	
Z1	-1.0000	abs
LW	0.0500	
LR	0.0000	
H1	0.0526	
αP	0.000 °	
D1	0.0030	inch
U	0.0000	

MDI - MANUAL DATA INPUT SCREEN

MDI

- Enter G-code commands or standard cycles block-by-block and immediately execute them for setting up the machine.

- Program is executed with **CYCLE START** key.

- **LOAD MDI** – Saved MDI or standard programs with standard cycles can be loaded from the program manager.

- **SAVE MDI** - Programs generated or modified in the MDI working window can be saved in the program manager, and later loaded, or edited.

- **DELETE BLOCKS** - If not wanting to save, easily deleted.

- **PROGRAM CONTROL** – Allows different ways of testing & running the program.

PROG – PROGRAM MANAGER SCREEN

More detailed structuring of the machining operations can be found in the Programming Section the **SIEMENS 828 TURNING OPERATING MANUAL**


.

- **SHOPTURN** or **PROGRAM GUIDE G-CODE** programs can be created in the **WORKPIECES** directory or in the **PART PROGRAMS** directory.

- **WORKPIECES** – Uses subdirectories (**WPD**) to contain program subfiles (.**MPF**) to store programs.

Functions on this screen-

- **NC** – Shows the directories and programs.

- **USB** – Data can be loaded and unloaded to the control. Direct program execution from a USB Flash Drive is not recommended.

- **EXECUTE** – Activates selected .MPF program to the **AUTO** screen.

- **NEW** – To start a new program; will need to designate either a **SHOPTURN** or **PROGRAM GUIDE G-CODE** program and enter a name.

- **OPEN** – Toggle the cursor arrow keys to a .MPF extension program to open for editing.

- **MARK** – Select this key and scroll down to select any other lines below.

- **COPY** - Copies any highlighted lines.

- **PASTE** – When another item is highlighted after selecting the **COPY** key, the information will be pasted.

- **CUT** – In a program, this will delete a line in a program.

Select the next page key for additional functions.
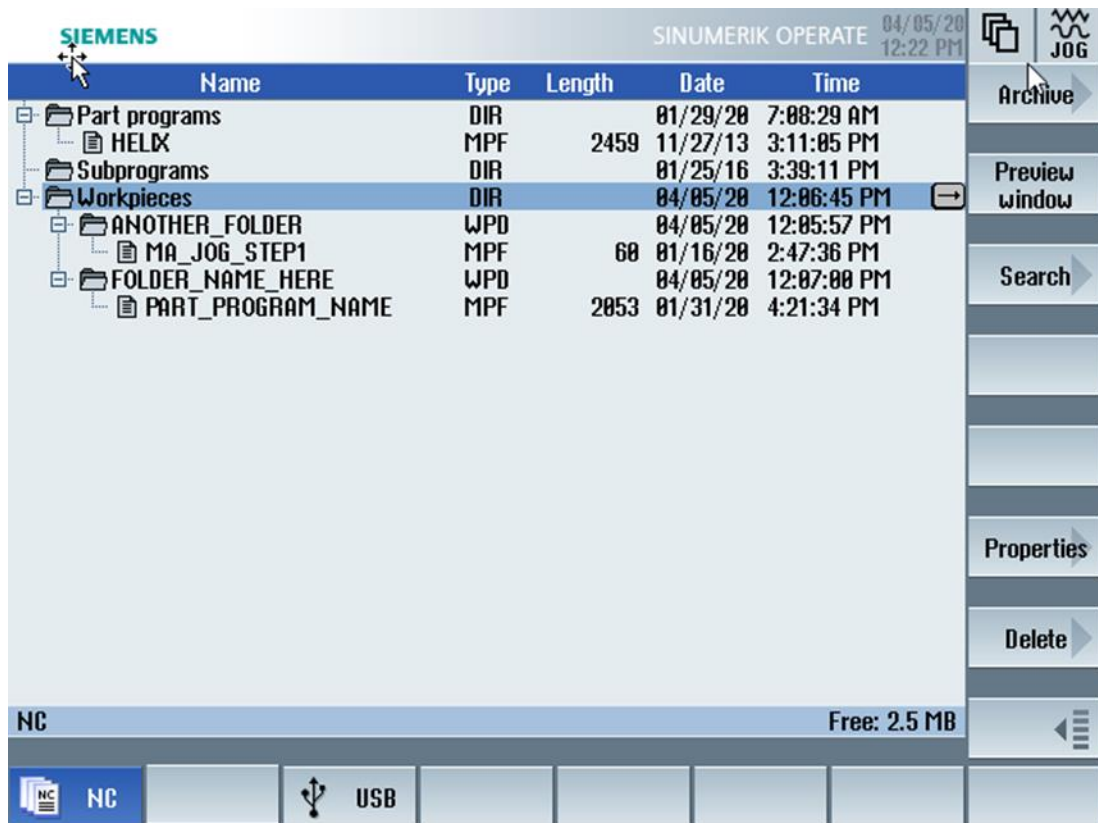
- **ARCHIVE** – You have the option of archiving individual files from the **NC** memory and the local drive.

- **PREVIEW WINDOW** – Opens any .MPF program to view. Shows in G-Code.

- **SEARCH** – Find information in the Program Manager for certain directories and files.

- **PROPERTIES** - Access rights for execution, writing, and reading.

CREATING A SHOPTURN PROGRAM

More detailed structuring of the machining operations can be found in the Programming Section the **SIEMENS 828 TURNING OPERATING MANUAL**

- Select either **PART PROGRAMS** or **WORKPIECES** directories to designate where the program will reside.
  - **NOTE:** Names can contain up to 28 characters.   Use any letters, digits. Use an underscore symbol "_" instead of the space key, no other symbols are accepted.
- Highlight if using the **PART PROGRAM** directory select the **NEW** key.
- Select the **PROGRAM GUIDE G-CODE** key and enter the unique program name in the screen then select the **OK** key.

programGUIDE G code

New G code program	
Type	Main program MPF
Name	PROGRAM_NAME

- The program header screen will open.

-or-

- Highlight if using the **WORK PIECES**, a subdirectory (**WPD**) will need to be entered in the blank field and then select the **OK** key.
- When an existing **WPD** folder is highlighted and the **NEW** key is selected, a new program will be created in that directory.

| New workpiece	
Type	Workpiece WPD
Name	FOLDER_NAME

ShopMill

- Select the **SHOPTURN** key and enter the unique program name in the screen then select the **OK** key.

New sequential program	
Type	ShopTurn
Name	SLEEVE_BUSHING

- The program header screen will open.

FILLING OUT THE PROGRAM HEADER

More detailed structuring of the machining operations can be found in the Programming Section the **SIEMENS 828 TURNING OPERATING MANUAL**
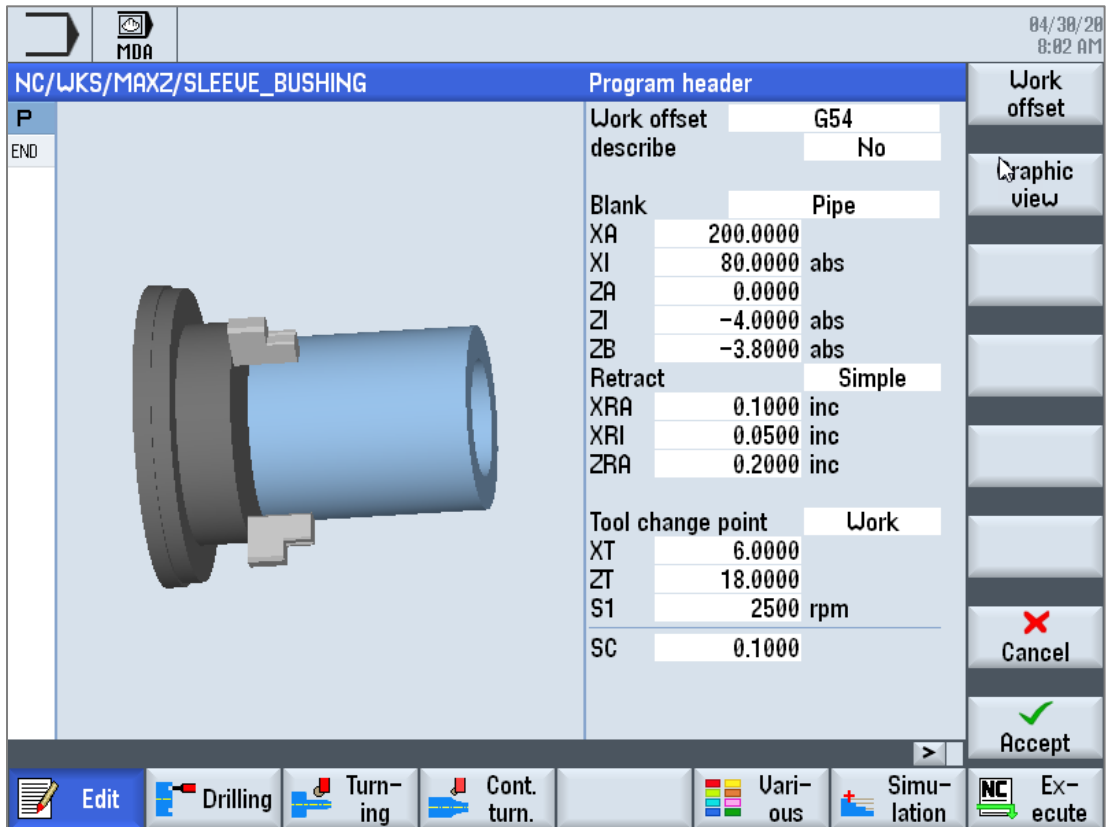
- Select a work offset and enter the type and dimensions of the blank and the parameters, tool axis, retraction plane, safety clearance and machining direction.

- Press the **ACCEPT** softkey.

- Program header and end of program are created as program blocks.

- Conversational programming can now be entered.

PROG

CREATING A PROGRAM GUIDE
G-CODE PROGRAM

More detailed structuring of the machining operations can be found in the
Programming Section the **SIEMENS G CODE PROGRAMMING MANUAL -TURNING**

- Select either **PART PROGRAMS** or **WORKPIECES** directory to
  designate where the program will reside.

- **NOTE:** Names can contain up to 28 characters.   Use any letters, digits.
  Use an underscore symbol "_" instead of the space key, no other
  symbols are accepted.

- Highlight if using the **PART PROGRAM** directory select the **NEW** key.

- Select the **PROGRAM GUIDE G-CODE** key and enter the unique
  program name in the screen then select the **OK** key.

**programGUIDE
G code**

| New G code program		
Type	Main program MPF	⌄
Name	SLEEVE_OUTER	

- The blank program editor screen will open.

-or-

- Highlight if using the **WORK PIECES**, a subdirectory (**WPD**) will need to be
  entered in the blank field and then select the **OK** key.

New workpiece		
Type	Workpiece WPD	⌄
Name	WIDGET_PARTS	

- When an existing **WPD** is highlighted and the **NEW** key is selected, a
  new program will be created in that directory.

- Select the **PROGRAM GUIDE G-CODE** key and enter the unique
  program name in the screen then select the **OK** key.
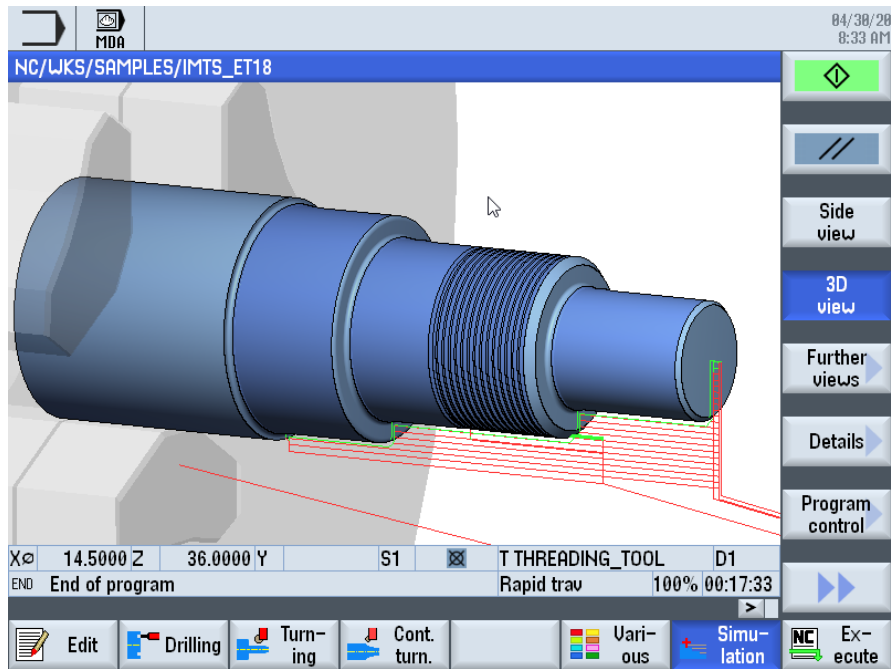
**programGUIDE
G code**

New G code program		
Type	Main program MPF	▼
Name	TEST	

- The blank program editor screen will open.

- A G-code program can now be entered, various cycles can be opened and used.
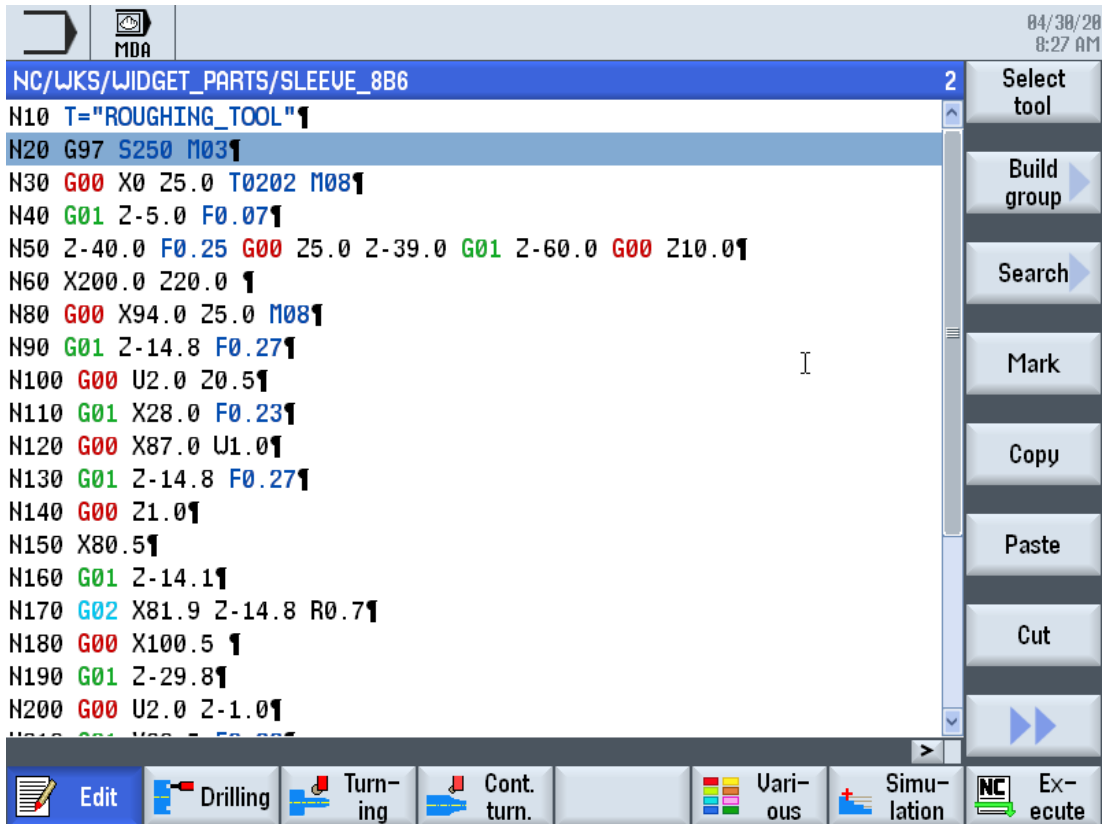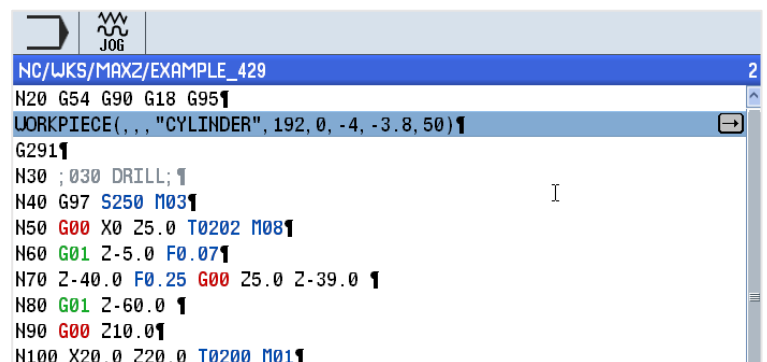
SIMULATION SCREEN

- During simulation, the current program is calculated in its entirety and the result displayed in graphic form. The result of programming is verified without moving the machine axes.

- Incorrectly programmed machining steps are detected at an early stage and incorrect machining on the workpiece prevented.

- The simulation represented on the screen uses the correct workpiece and tool measurements.

- The tool paths are displayed in color. Rapid traverse is red and the feedrate is green.

- When using a **SHOPTURN** program, the blank dimensions that are entered in the program header to define the stock material and the coordinates for machining.

- When using **PROGRAM GUIDE**, the (**WORKPIECE**) blank data is entered in the next block of code after the work offset is defined. It is accessed by selecting the **VARIOUS** key and selecting the **BLANK** key.

  For example:

N10 G54 G90 G18 G95

N20 WORKPIECE(, , , "CYLINDER", 192, 0, -4, -3.8,50)

If you select another work offset, then the coordinate system is converted, but the display of the blank is not changed.

EXECUTE A PROGRAM

- After program has been completed and verified with the simulation, select the **EDIT** key at lower left of the screen and select the **EXECUTE** key on the bottom right.

| Edit | Drilling | Turn-ing | Cont. turn. | | Vari-ous | Simu-lation | Ex-ecute |

- This will open the **AUTO** – Run Program screen.

- Select the **RESET** key to ensure the program starts from the beginning.

- The **FEED OVERRIDE** knob should be set to 0% and the **SPINDLE OVERRIDE** knob should be set to 100%.

- Select the **CYCLE START** to start the program.

- Adjusting the **FEED OVERRIDE** knob will control the programmed feedrate approaching the workpiece and during the machining process.

- The **FEED HOLD** key can be used to stop the feedrate at any time.

HANDWHEEL RUN (Option)

This feature allows you to control your program execution with the control handwheels.

- Turning the handwheel causes the program to run with you in charge of the axis feed.

- Designed to make proving-out programs easier with safety and confidence.

To activate:

- The **HANDWHEEL RUN** option can be activated anytime and anywhere while running a program.
- Select the **HDW RUN** key on the control. A message will be displayed on the top of the screen indicating it has been activated.

- Select the **CYCLE START** key.

- Turning the Z axis handwheel will move the axes through the program.

- When you stop turning, the axes will pause.

- Turn the handwheel the opposite direction and the axes move backwards though the program.

To deactivate:

- Stop turning the handwheel

- Select the **HDW RUN** key.

- Select the **CYCLE START** key to continue running the axes automatically.

AUTO – RUN PROGRAM SCREEN

More detailed structuring of the machining operations can be found in the Programming Section the **SIEMENS 828 TURNING OPERATING MANUAL**

.

Execute a program completely or only partially when **CYCLE START** key is selected.

Functions available are:

- **OVERSTORE** - Option of executing partial program information before the program is started.

- **PROGRAM CONTROL** – Allows different ways of testing & running the program.

- **BLOCK SEARCH** – Allows different methods of searching in a program

- **SIMULATION RECORD** – activate the simultaneous recording while machining the workpiece. You can select various display formats for the simultaneous recording.

- **PROGRAM CORRECTION** – Open the program to be edited, if needed

- **BASIC BLOCKS** – Displays both in test mode and when machining the workpiece on the machine. All G code commands that initiate a function on the machine are displayed on a side screen.

- **TIME / COUNTER** – Shows timing of running the current program; can be set up to keep count of workpieces.

- **PROGRAM LEVELS** - Displays the current program level during the execution of a large program with several subprograms.

KEYSWITCH INFORMATION

You can use the keyswitch to set various access rights.

The keyswitch has four settings for protection levels 4 to 7.

Machine data can be programmed to interlock access to programs, data, and functions at various protection levels.

The keyswitch has three keys of different colors that you can remove in the specified positions:
- Position 0   No key
  - Lowest access rights
  - Level of protection 7
    - End user, Semi-skilled operator

- Position 1   Key Black
  - Increasing access authorization
  - Level of protection 6
    - End user, Trained operator without programming knowledge

- Position 2   Key Green
  - Level of protection 5
    - End user, Skilled operator without programming knowledge

- Position 3   Key Red
  - Highest access rights
  - Level of protection 4
    - End user Programmer, Machine setter

When you change the key position to change the access authorization, this is immediately not visible on the operator interface.

You have to initiate an action first (e.g. close or open a directory).

Key switch   Colour coded keys

The following table shows the password/key-switch positions. and associated protection levels/users.

Protection level	Intended user:-	Protected via:-
1	Manufacturer	Password: SUNRISE
2	Service	Password: EVENING
3	User	Password: CUSTOMER
4	Programmer, Setter	Key-switch position 3
5	Qualified operator	Key-switch position 2
6	Trained operator	Key-switch position 1
7	Untrained operator	Key-switch position 0

The key switch is located to the bottom right of the MCP and has four positions. Three colour coded keys are available, each of which allow a different access level. This is from a standard Siemens machine control panel. It is possible that a manufacturer may use a different method to set the interface signal.

Access levels of the three keys:

Key colour	Switch position	Protection level
No key required	0	7
Black	0 & 1	6-7
Green	0, 1 & 2	5-7
Orange	0, 1, 2 & 3	4-7

It is the responsibility of the OEM to ensure that the key-switch signals are transferred to the PLC interface.

DB2600.DBB0 is used for the transfer of the key-switch signals.

DB2600	Key-switch signals to NC (PLC → NC)							
	Bit7	Bit6	Bit5	Bit4	Bit3	Bit2	Bit1	Bit0
DBB0	Key-switch Pos.3	Key-switch Pos.2	Key-switch Pos.1	Key-switch Pos. 0				

Converting a Fanuc G Code Program for the 828 Control for a Lathe

%   ◄─────────────────   Remove symbol

O#####   ◄─────────────────   Remove program number

WORKPIECE(,,,"CYLINDER",64,0,-4.485,-3.617,2)

G291   ◄─────────────────   Add this

(2.0 STOCK)
(OP2 TURN AND THREAD)
G20
G0 T0101   ◄──────────────────────────────────┐
G92 S750
G96 S500 M03
G00 G54 X2.1 Z0.2
G71 U.050 R.010
....
G00 X5. Z5.
M01
(GROOVING)
G00 T0606   ◄───────────────────────────┐
G20
G92S600
....
M01
(THREADING)
(1-8 2A UNC)
G0 T0303   ◄────────────────────────┐
G97 S600 M03
G00 G54 X1.2 Z0.1
....
M09
M05
M30
%   ◄─────────────────   Remove symbol

Tool name in the program is determined by the location and corresponding H number
The actual tool name will be in the control screen when activated.

Tool selection						
Loc.	MT LO.	Type	Tool name	D	H	Le
1			CNMG 432	1	1	-1
2			2	1	2	-1
3			THREADING_TOOL OD	1	3	-1
4			.500 DRILL	1	4	-1
5			BORING BAR	1	5	-1
6			606	1	6	-1
7			THREADING_TOOL ID	1	7	-1
8			FINISH BORING BAR	1	8	-1

T,F,S

T CNMG 432   R 0.0312
   D1   Z-25.000
   ►► CNMG 432   X-13.400

F   0.0000

Manual Screen

NOTE: *Also edit file extension to output .MPF format when saving.*

43

PRINTED REFERENCE MANUALS INCLUDED WITH YOUR MACHINE
ET-16, 18, 21, 25, 30 & 40 - 828 MAINTENANCE MANUAL

828 TOOLROOM LATHE OPERATOR MANUAL

REFERENCE MANUALS ON A USB INCLUDED WITH YOUR MACHINE

- **SIEMENS TURNING MADE EASY WITH SHOPTURN**

  o This training manual guides you through the basics of SHOPTURN which allows quick and practical programming of machining. Includes step by step instructions how to program sample parts.

- **SIEMENS 828 TURNING OPERATING MANUAL**

  o This operating manual covers in detail the available control elements and programming. More advanced than the TURNING MADE EASY WITH SHOPTURN manual.

- **SIEMENS G CODE PROGRAMMING MANUAL -TURNING**

  o This programming manual covers in detail utilizing ISO G-Code programming.

- **SIEMENS DXF READER MANUAL**

  o This manual steps through the setup and use of DXF files in the control. **(Option)**

- **SIEMENS OFFLINE SOFTWARE MANUAL**

  o This manual guides you through the basic of installing and using SINUTRAIN offline programming software which allows quick programming on a computer. **(Option)**

- **SIEMENS 828 ALARMS MANUAL**

  o This diagnostics manual enables the users to evaluate alarm errors and fault indications and how to respond accordingly.

Disclaimer

We have reviewed the contents of this publication to ensure consistency with the hardware and software described.

Since variance cannot be precluded entirely, we cannot guarantee full consistency. However, the information in this publication is reviewed regularly and any necessary corrections are included in subsequent editions.

FRYER MACHINE SYSTEMS

The Toolroom Company

Phone: 845-878-2500
Fax: 845-878-2525
Email: info@fryermachine.com
sales@fryermachine.com
Website: www.fryermachine.com

© 2020 Fryer Machine Systems. Inc.
70 Jon Barrett Road Patterson, New York 12563 USA