SIEMENS

SINUMERIK Operate

SinuTrain
Easy milling with ShopMill

Training Documents

09/2011

6FC5095-0AB50-1BP1

Legal information

Warning notice system

This manual contains notices you have to observe in order to ensure your personal safety, as well as to prevent damage to property. The notices referring to your personal safety are highlighted in the manual by a safety alert symbol, notices referring only to property damage have no safety alert symbol. These notices shown below are graded according to the degree of danger.

⚠ DANGER

indicates that death or severe personal injury **will** result if proper precautions are not taken.

⚠ WARNING

indicates that death or severe personal injury **may** result if proper precautions are not taken.

⚠ CAUTION

with a safety alert symbol, indicates that minor personal injury can result if proper precautions are not taken.

CAUTION

without a safety alert symbol, indicates that property damage can result if proper precautions are not taken.

NOTICE

indicates that an unintended result or situation can occur if the relevant information is not taken into account.

If more than one degree of danger is present, the warning notice representing the highest degree of danger will be used. A notice warning of injury to persons with a safety alert symbol may also include a warning relating to property damage.

Qualified Personnel

The product/system described in this documentation may be operated only by **personnel qualified** for the specific task in accordance with the relevant documentation, in particular its warning notices and safety instructions. Qualified personnel are those who, based on their training and experience, are capable of identifying risks and avoiding potential hazards when working with these products/systems.

Proper use of Siemens products

Note the following:

⚠ WARNING

Siemens products may only be used for the applications described in the catalog and in the relevant technical documentation. If products and components from other manufacturers are used, these must be recommended or approved by Siemens. Proper transport, storage, installation, assembly, commissioning, operation and maintenance are required to ensure that the products operate safely and without any problems. The permissible ambient conditions must be complied with. The information in the relevant documentation must be observed.

Trademarks

All names identified by ® are registered trademarks of Siemens AG. The remaining trademarks in this publication may be trademarks whose use by third parties for their own purposes could violate the rights of the owner.

Disclaimer of Liability

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.

Siemens AG
Industry Sector
Postfach 48 48
90026 NÜRNBERG
GERMANY

Order number: 6FC5095-0AB50-1BP1
℗ 09/2011

Table of contents

Introduction 1

Faster from the drawing to the workpiece - but how?

The technological development of machine tools is highly dynamic. Particularly with the creation of NC programs, the range has extended from pure CAM system programming to programming directly at the CNC machine. Special, productive programming methods are available for each area. With ShopMill, SIEMENS therefore offers a programming method specially tailored to the shop floor which allows quick and practical programming of machining steps from the manufacture of single parts up to small batches. In conjunction with SINUMERIK Operate, the new operator interface for the control system, intuitive and effective working in the workshop is possible even for series production.

The solution is: Creating a process plan instead of programming

The creation of a process plan with intuitive and operator-friendly handling sequences, allows the ShopMill user to create the NC program directly from the drawing. Even changes and different variants of a workpiece can be quickly programmed due to the clear structure.

Even the most complicated contours and workpieces are simple to manufacture with ShopMill thanks to the integrated, powerful tools for creating traversing paths. For this reason:

Consequently: Simpler and easier from the drawing to the workpiece - with ShopMill

Although ShopMill is easy to learn, these ShopMill Training Documents allow you to enter this world even faster. Before, however, it comes to the actual work with ShopMill, important basics will be discussed in the first sections:

- First, we will show you the advantages of ShopMill.
- Then we show you the basics of the operation with SINUMERIK Operate.
- And next, the geometrical and technological fundamentals will be introduced to the beginner.
- A short introduction to tool management will be given in a further section.

The theory is followed by practical exercises with ShopMill:

- Five examples have been chosen to explain the possibilities for machining with ShopMill, whereby the degree of difficulty is increased continuously. At the beginning, all key actions are specified; later you will be prompted to proceed without help.
- Then you will learn how to machine in the AUTOMATIC mode using ShopMill.
- If you wish you may test yourself finally to find out how fit you are in ShopMill.

Please note that the technology data used here are only examples due to the wide variety of situations in the workshop.

Just as ShopMill was created with the help of skilled workers, these Training Documents were also elaborated by practical users. In this sense, we wish you much pleasure and success in your work with ShopMill.

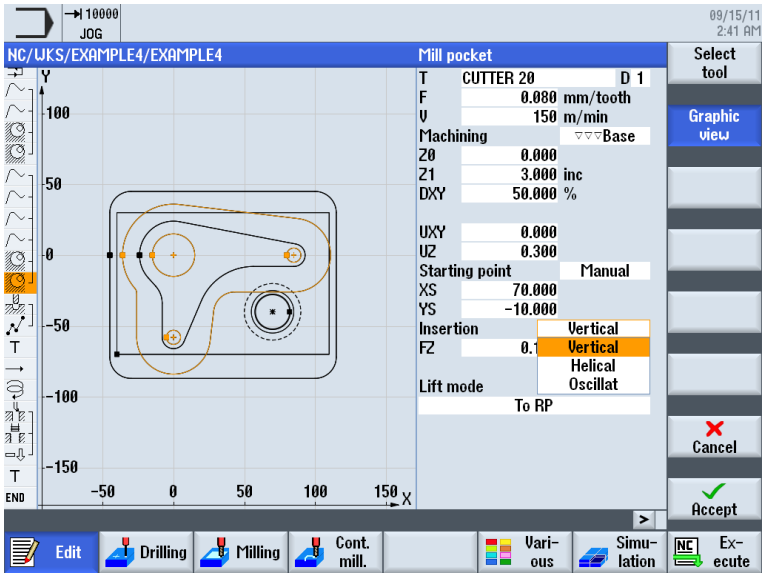
Advantages of ShopMill

2

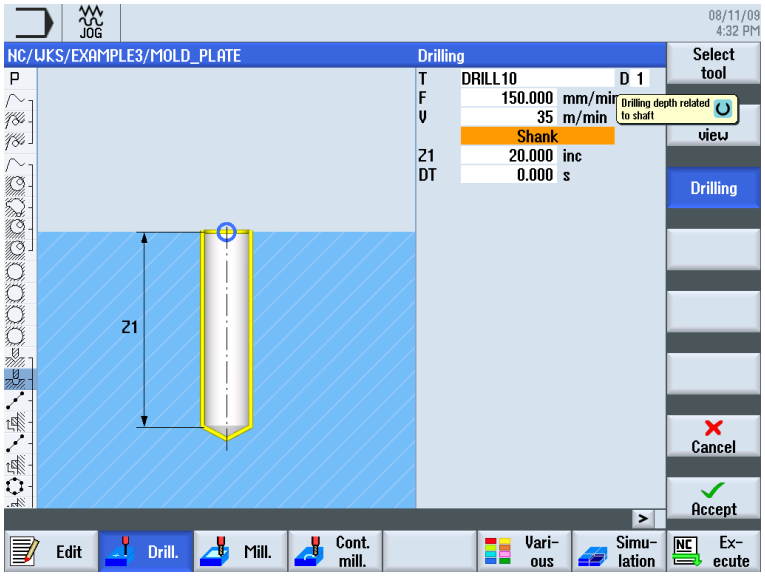
This chapter shows you the special advantages of ShopMill.

2.1 You will save time for training....

- ShopMill does not use any foreign-language terms you would otherwise have to learn. All required inputs are requested in plain text.



- When working with ShopMill, you are assisted optimally by colored help displays.



- You can also integrate DIN/ISO commands into the **Process Plan** of ShopMill. You may also program in DIN/ISO 66025 and use DIN cycles.

```
G  N25 G17 G54 G64 G90 G94
T  N30 T=EM16
G  N35 G0 X85 Y22.5
G  N40 G0 Z2 S500 M3 M8
G  N45 G0 Z-10
G  N50 G1 X-85 F200
G  N55 G0 Y-22.5
G  N60 G1 X85
G  N65 G0 Z100 M5 M9
```

- You may switch between the individual work step and the workpiece graphic at any time when creating a process plan.

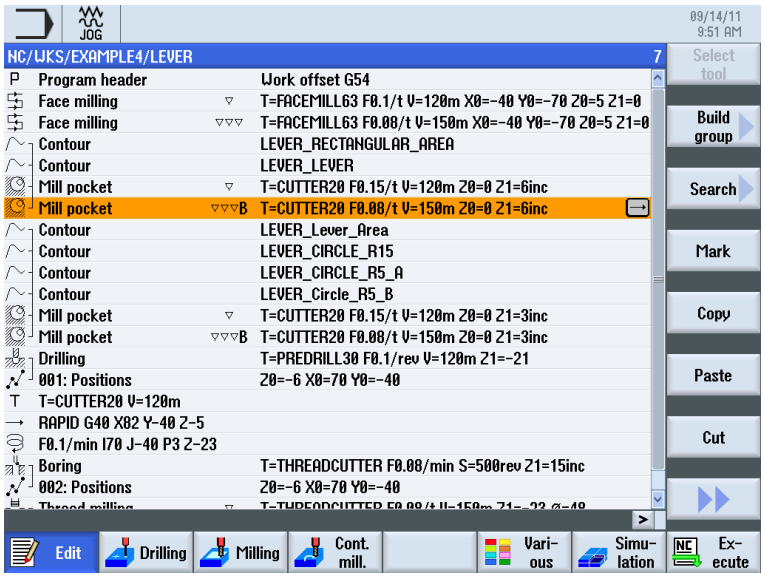


Figure 2-1 Work step in a process plan

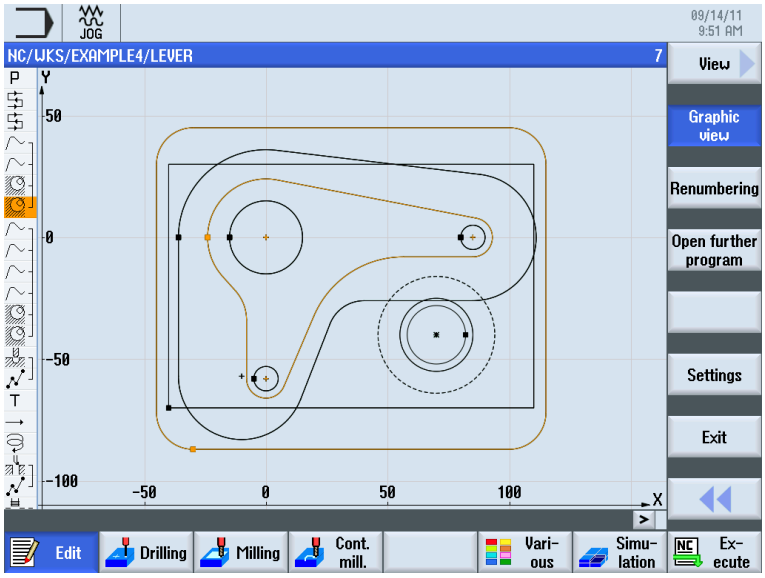
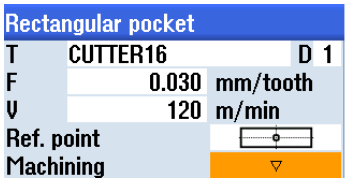
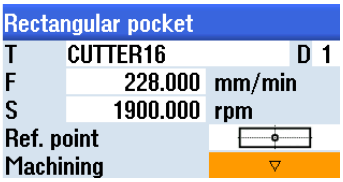


Figure 2-2 Graphic view

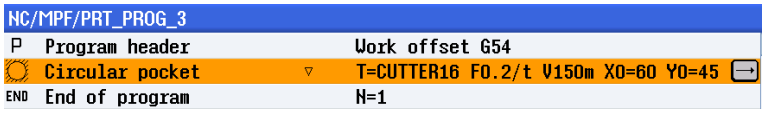
2.2 You will save time for programming...

- ShopMill already assists you in entering the technological values: You will only have to enter the handbook values **feed/tooth** and **cutting rate** – speed and feedrate are calculated by ShopMill automatically.

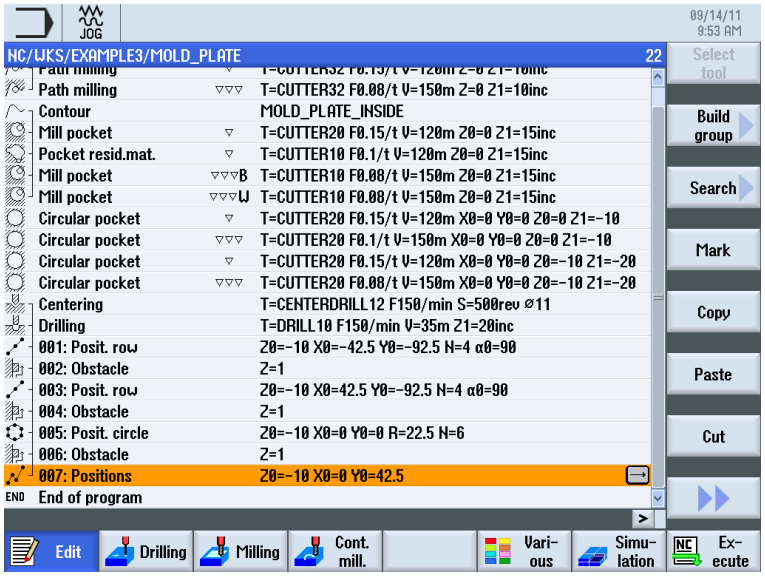




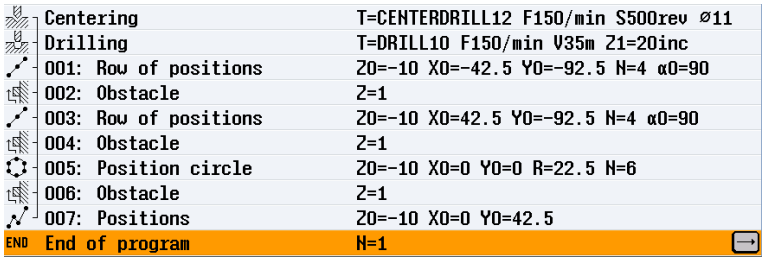
- ShopMill enables you to describe a complete machining sequence using only one work step, and the required positioning motions (in this case, from the tool change point to the workpiece and reverse) are created automatically.



- All work steps are represented by ShopMill in a compact and clear fashion in the **Graphical Process Plan**. This provides you a complete overview and thus better editing possibilities even if comprehensive manufacturing sequences are to be performed.



- In drilling, for example, several machining operations can be connected together so that they need not be called repeatedly.



- The integrated contour calculator can process all standard dimensions (Cartesian, polar); it is nevertheless very easy to handle and understand - thanks to colloquial input and graphic support.

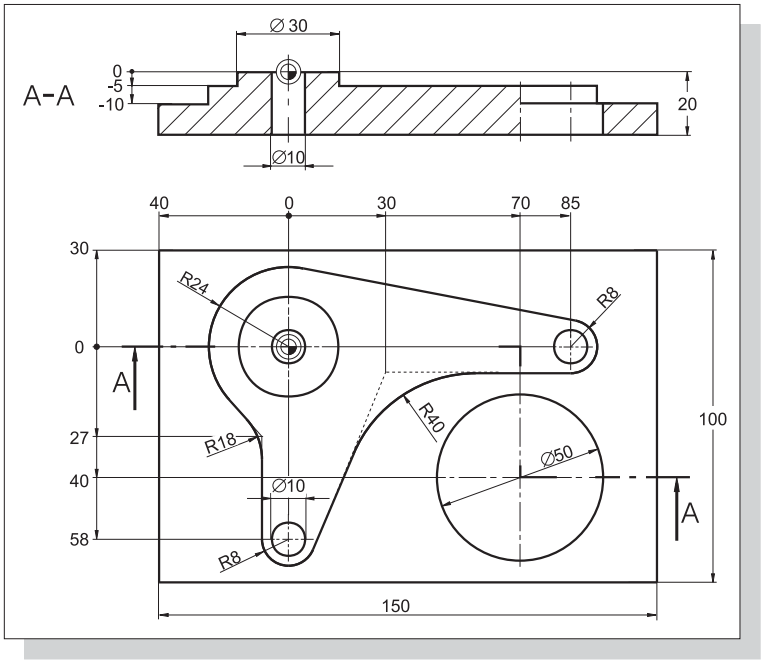


Figure 2-3 Technical drawing

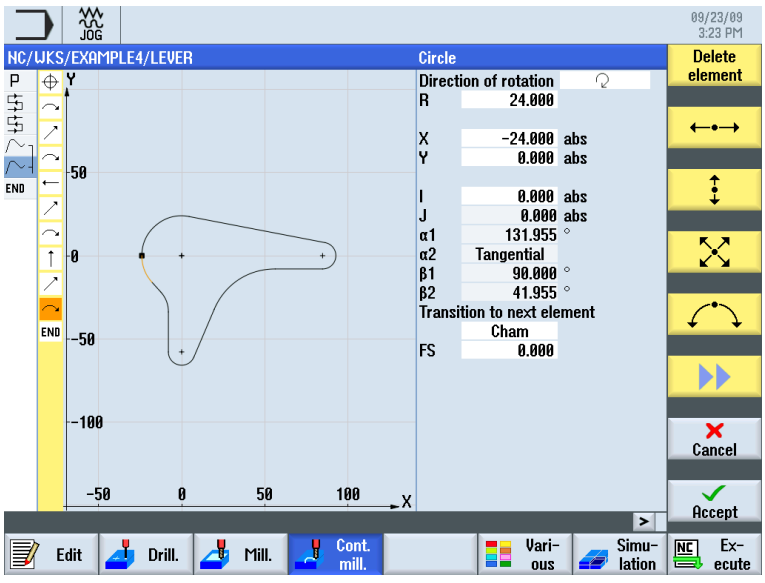


Figure 2-4 Screenform

- You may switch between the graphic view and parameter screenform with help display at any time.

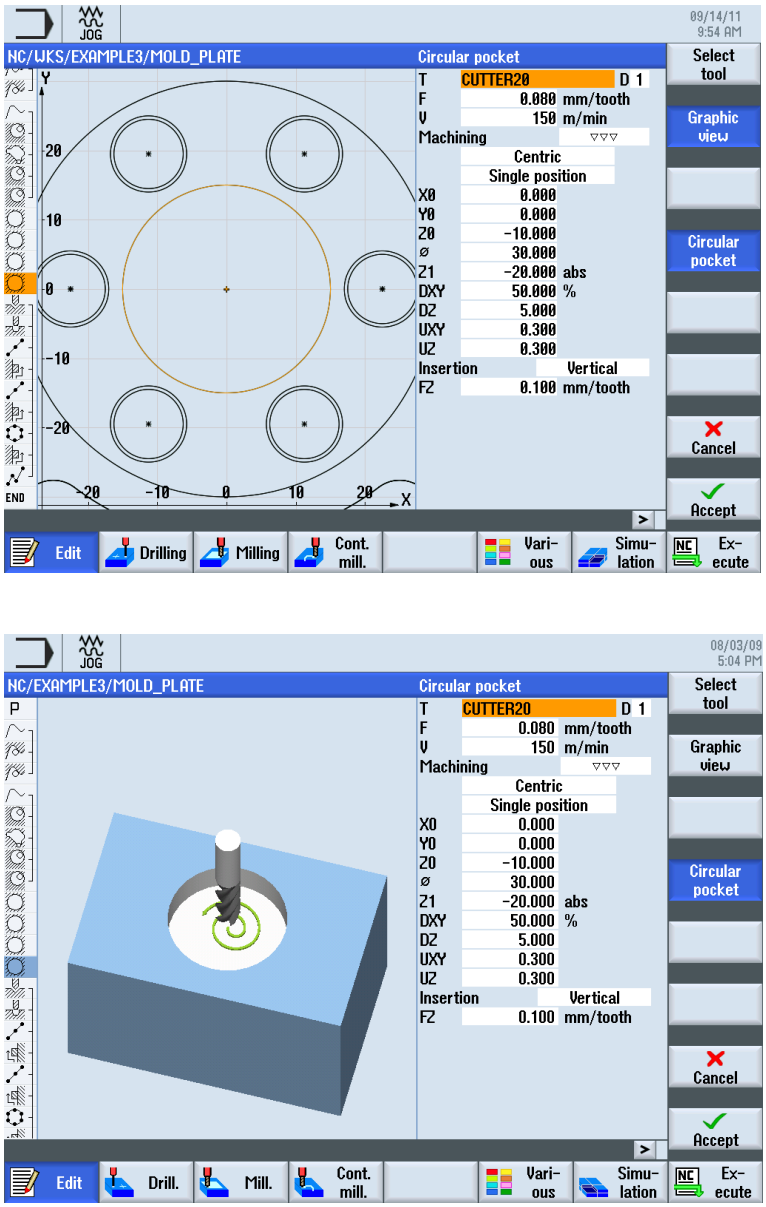
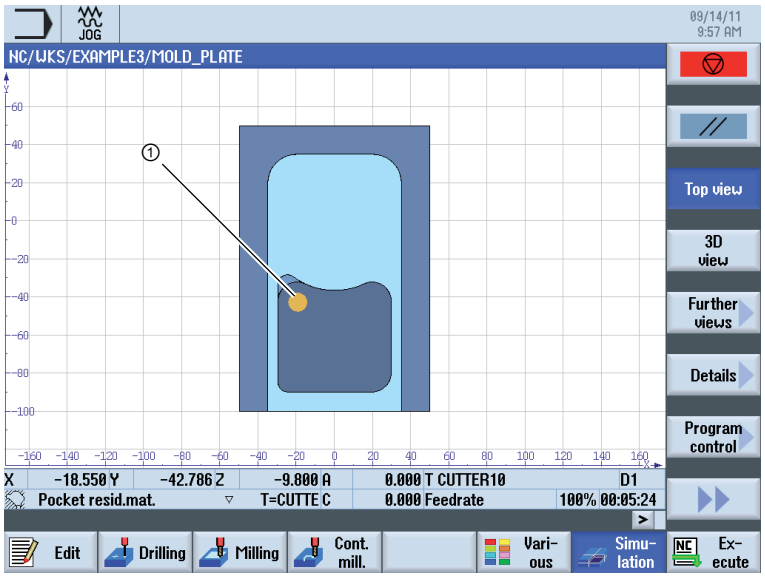


Figure 2-5 Parameter screen with help display

- Process plan and manufacturing do not exclude each other. With ShopMill, you can create a new process plan parallel to manufacturing.

2.3 You will save time for manufacturing...

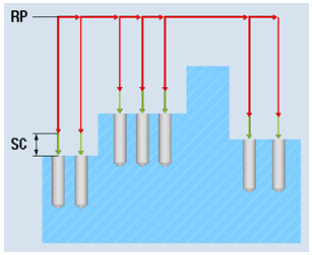
- You need not take into account the pocket radii when selecting the milling cutter for removing the contour pockets from the solid: Any residual material ① is detected and removed automatically using a smaller milling cutter.



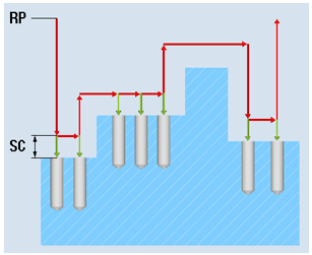
- There are no unnecessary infeed motions between retraction and machining planes when positioning the tool. This is made possible by the settings **Retract to retraction plane (RP)** and **Optimized retraction**.

  The **Optimized retraction** setting is to be made by a skilled worker in the program header. He must take into account obstacles, such as clamping elements.

Retraction to retraction plane (RP)



Retraction to retraction planes = time saved in manufacturing



- You can optimize your machining sequence with a minimum of work - thanks to the compact structure of the process plan (in this case, by saving of a tool change, for example).

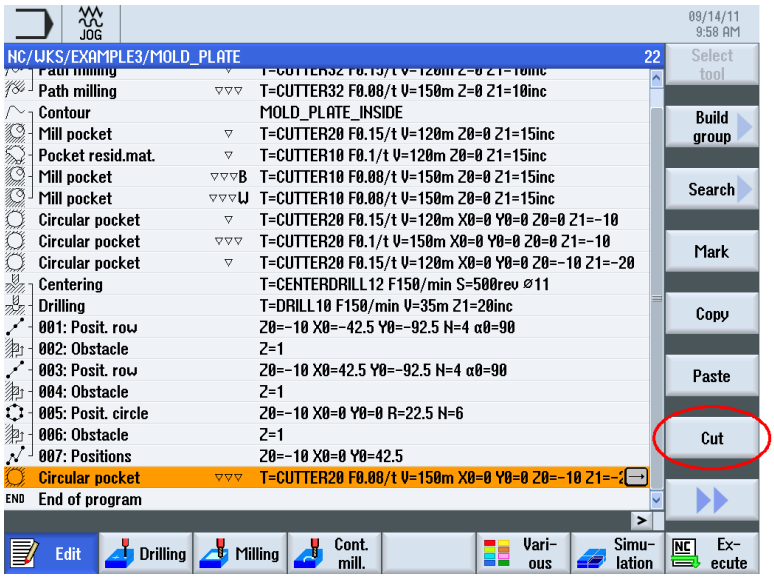


Figure 2-6 Original machining sequence

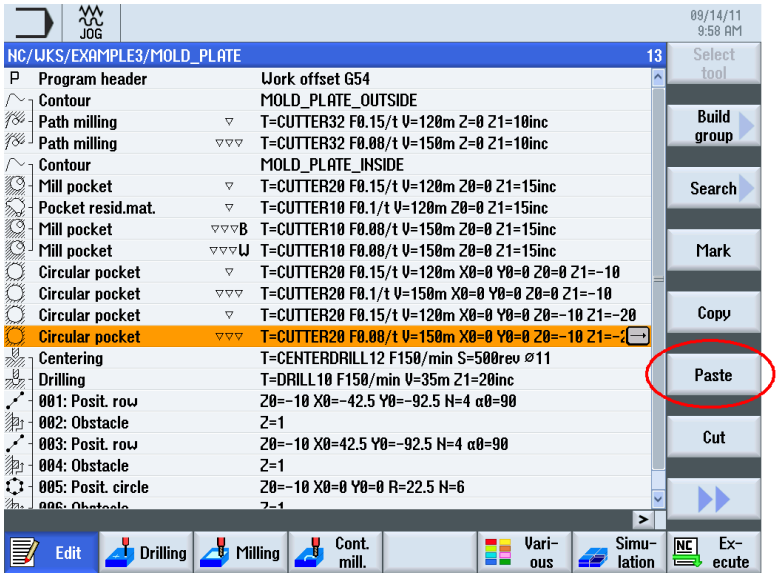


Figure 2-7 Optimized machining sequence with **cutting** and **pasting** a work step

- With ShopMill, you can achieve extremely high feedrates with optimum repeat accuracy based on consistent digital technology (SINAMICS drives, ..., SINUMERIK control systems).

To make everything function smoothly... 3

In this chapter you will learn the basics of the ShopMill operation with the help of examples.

3.1 The use of ShopMill

A powerful software is one thing, but it must also be easy to operate. Regardless of whether you work with the SINUMERIK 840D sl or SINUMERIK 828D shown here - you are always assisted by the clearly laid-out machine operator panel. The operator panel consists of 3 parts - the flat operator panel ①, the CNC full keyboard ② and the machine control panel (MCP) ③.

The most important keys of the CNC full keyboard for navigation in ShopMill are listed in the following:

Key	Function
	**<HELP>** Calls the context-sensitive online help for the selected window.
	**<SELECT>** Selects a listed value.
	**Cursor keys** The cursor is moved using the 4 cursor keys. Use the <Cursor to the right> key to open a directory or program (e.g. a cycle) in the editor.
	**<PAGE UP>** Scrolling upwards in a menu screen.
	**<PAGE DOWN>** Scrolling downwards in a menu screen.
	**<END>** Moves the cursor to the last input field in a menu screen or a table.
	**<DEL>** • Edit mode: Deletes the first character to the right. • Navigation mode: Deletes all characters.
	**<BACKSPACE>** • Edit mode: Deletes a character selected to the left of the cursor. • Navigation mode: Deletes all of the selected characters to the left of the cursor.
	**<INSERT>** • Pressing the <INSERT> key opens the "Editing" mode, and when you press the key again, you will leave the "Editing" mode and you can call the "Navigation" mode.
	**<INPUT>** • Completes input of a value in the input field. • Opens a directory or program.

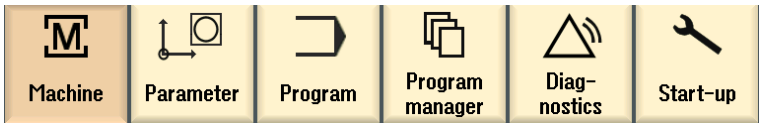
The actual function selection in ShopMill is performed using the keys located around the screen. Most of them are assigned directly to the individual menu items. Since the contents of the menus change depending on the situation, the term 'softkeys' is used.

All main functions can be called using the horizontal softkeys.

All subfunctions of ShopMill can be called using the vertical softkeys.

The main menu can be called with this key at any time - irrespective of in which operating area you are at the moment.

Main menu



3.2 The contents of the main menu

3.2.1 Machine

Machine - Manual

Select the "Machine" softkey.

Press the "JOG" key.

In this mode, the machine is set up; the tool is traversed in the MANUAL mode. It is also possible to gauge tools and to set workpiece zeros.

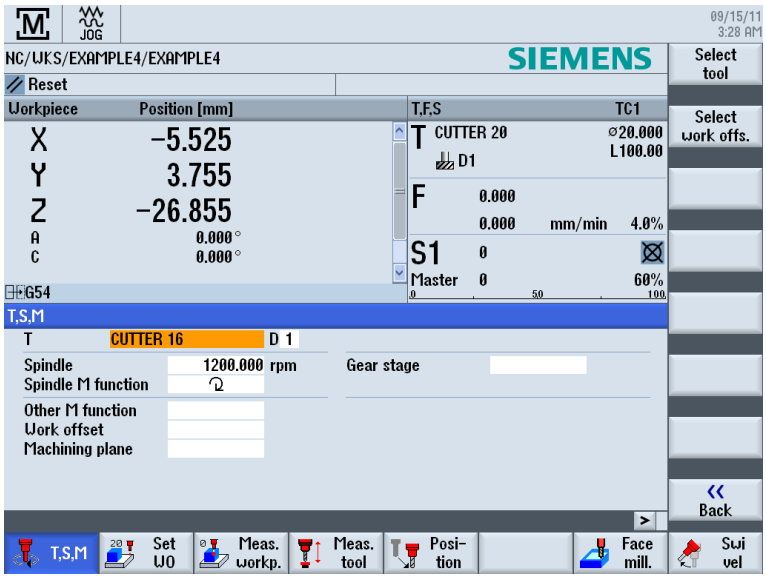


Figure 3-1 Call of a tool and input of technological values

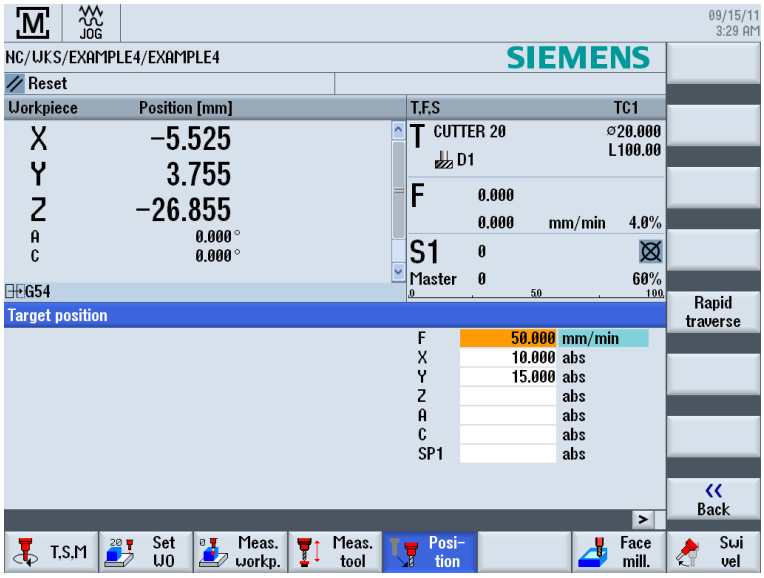


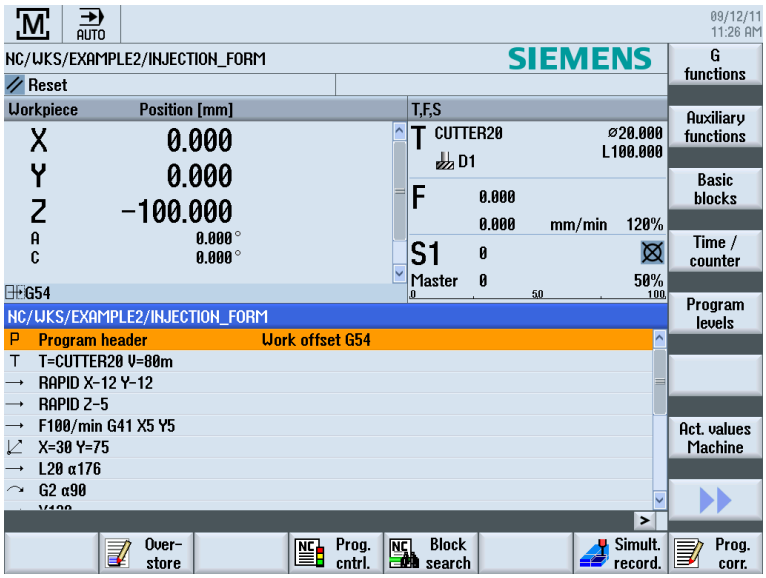
Figure 3-2 Input of a target position

Machine - AUTO

Select the "Machine" softkey.

Press the AUTO key.

During manufacturing, the current work step is displayed. It is possible to switch to a running simulation using the relevant key ("Drawing"). When executing a process plan, you may insert work steps and/or create a new process plan.



3.2.2 Parameters

Parameter lists

This key can be used to edit data for the tool management and for programs.

Tool lists

No cutting without tools.

The tools can be managed in a tool list.

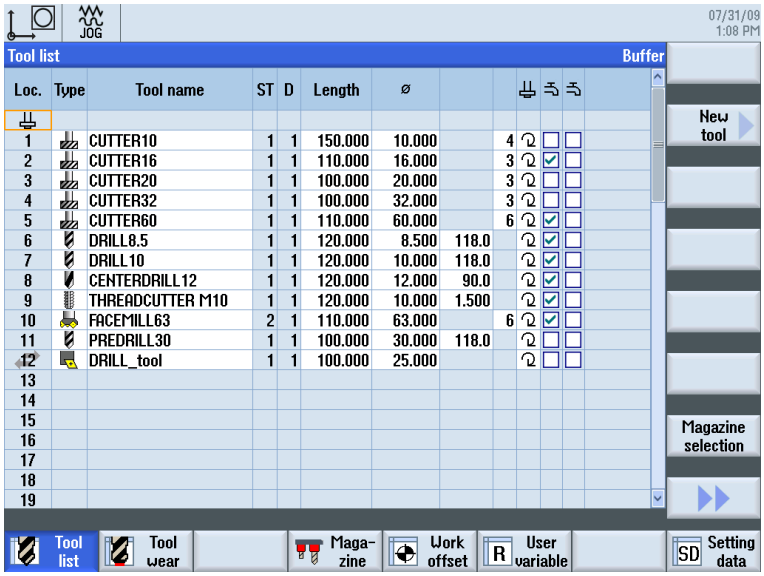


Figure 3-3 Tool list

Magazine

Tools can be organized into a magazine.

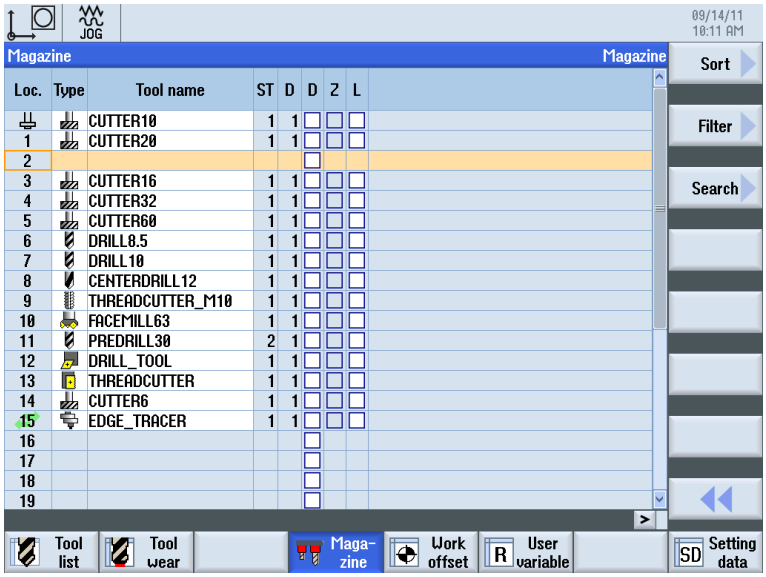


Figure 3-4 Magazine

Work offsets

Zero points are saved in a clearly laid-out zero-point table.

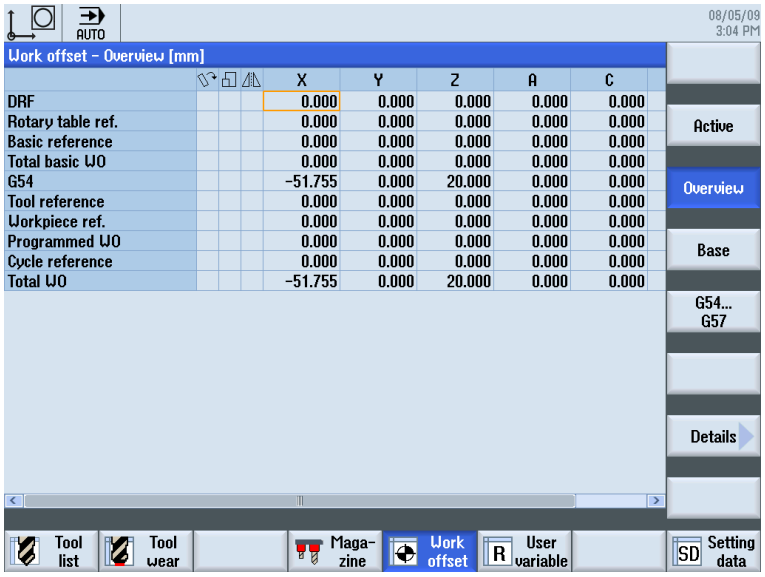


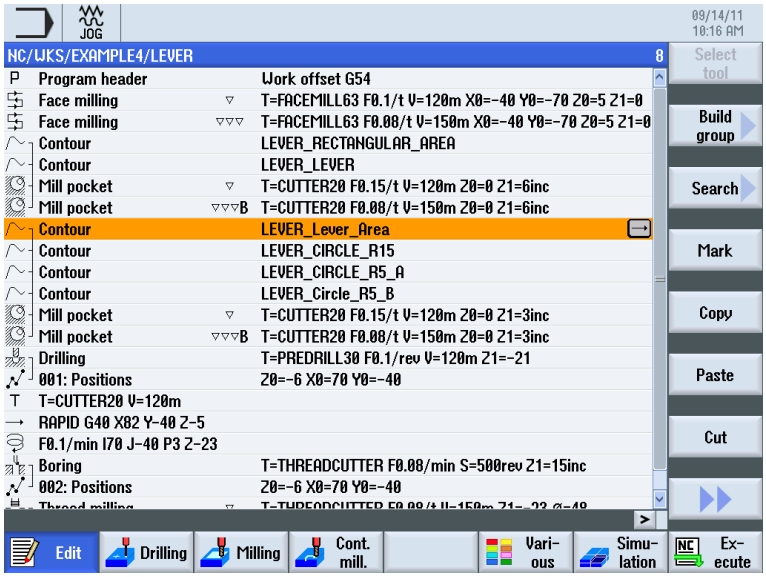
Figure 3-5 Work offsets

3.2.3 Program

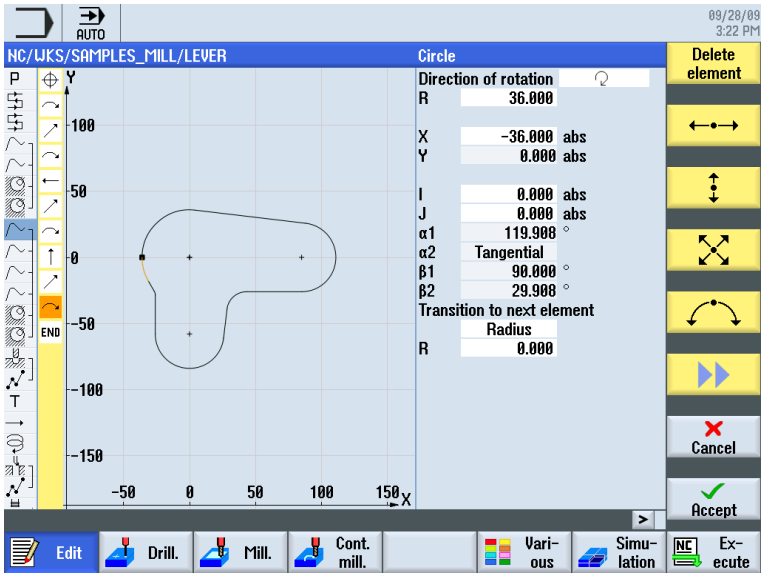
Editing programs

This key can be used to edit programs.

If you have created a **ShopMill program** in the Program Manager, you can now create the process plan with the complete machining sequence for the appropriate workpiece. Prerequisites for the optimum order of sequence are the experience and knowledge of the skilled worker.

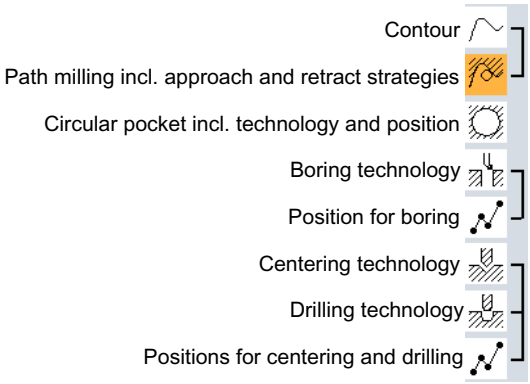


The contour to be machined is entered graphically as a machining step.



Geometry and technology constitute a unit in programming. The subsequent technological machining operations are applied to the contour.

Example for the dovetailing of geometry and technology:



This geometrical-technological interrelation is represented very clearly in the graphical display of the work steps by putting the appropriate symbols in brackets. The brackets mean linking of geometry and technology to one work step.

Simulating programs

Before machining a workpiece on the machine, it is possible to display the program execution graphically on the screen.

- To this end, select the "Simulation" and "Start" softkeys.
- To stop simulation, select the "Stop" softkey.
- To cancel simulation, use the "Reset" softkey.

The following views are available for simulation:

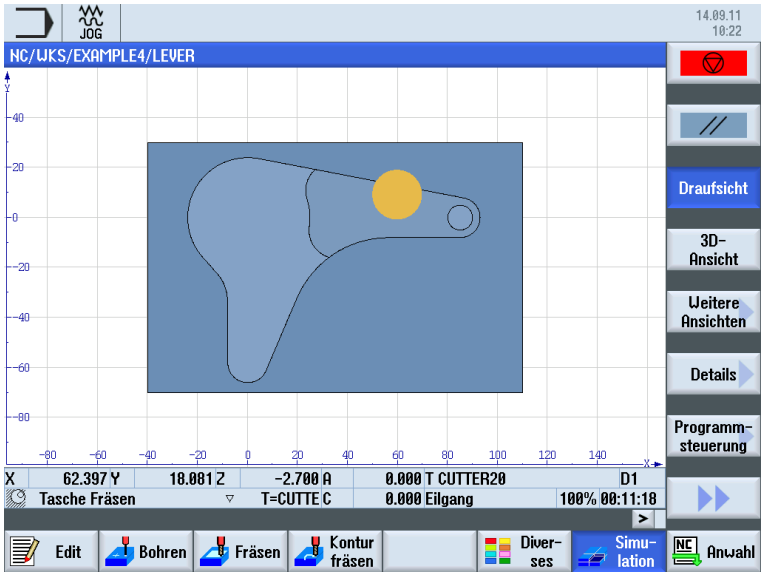


Figure 3-6 Top view

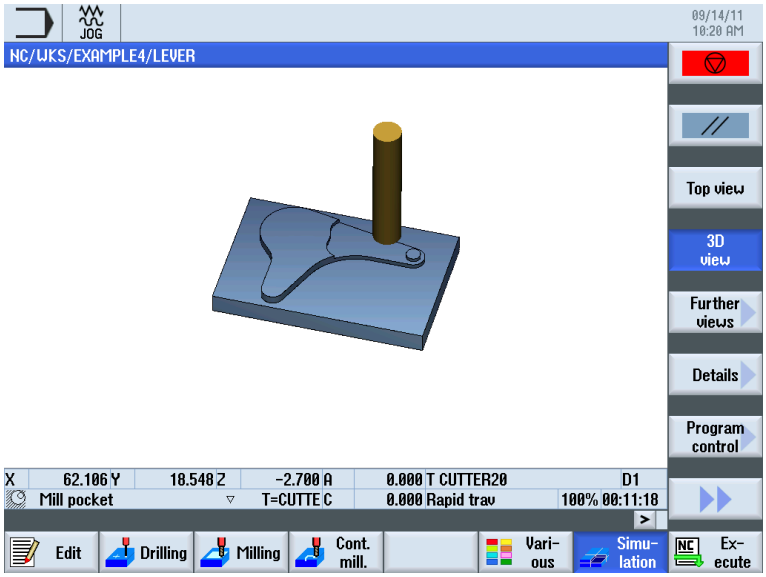


Figure 3-7 3D view

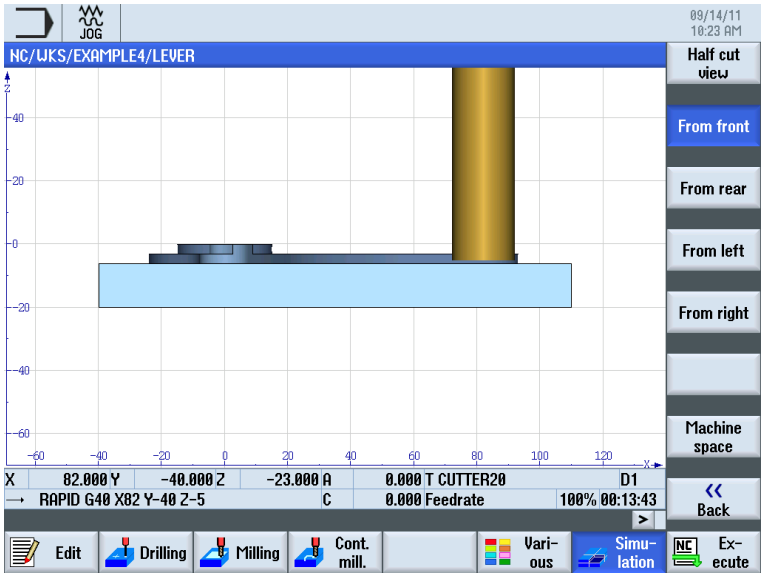
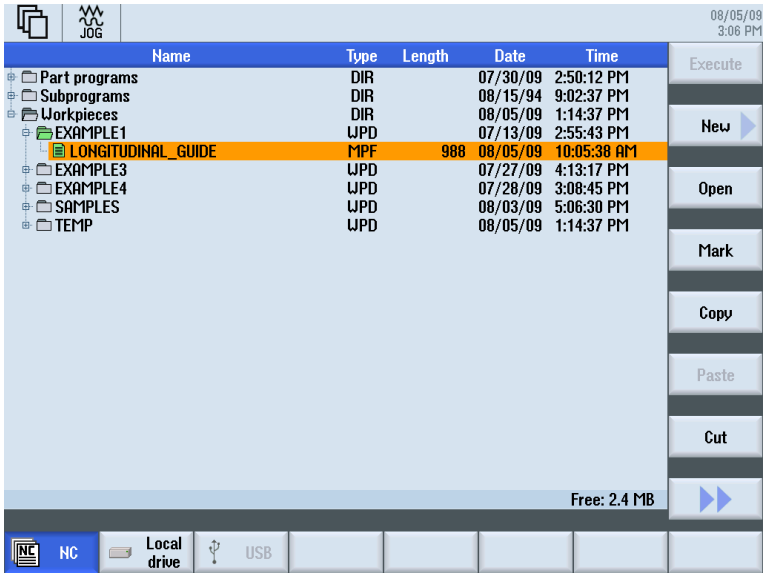


Figure 3-8 Side view

3.2.4 Program Manager

Managing programs

With the Program Manager, you can create new programs at any time. You can similarly open existing programs to execute, modify, copy or rename them. Programs no longer required can be deleted.



Active programs are marked with a green symbol.

USB flash drives can be used for data exchange. For example, programs which were created on an external device can be copied and executed on the NC.

Creating a new workpiece

You can manage your programs and other files, such as tool data, zero points, magazine mapping, in a workpiece.

Creating a new program

If you create a new program, you can specify the type of programming using the following softkeys:

ShopMill	ShopMill program
G code	G code program

3.2.5 Diagnosis

Alarms and messages

Here you can see alarm lists, messages and alarm logs.

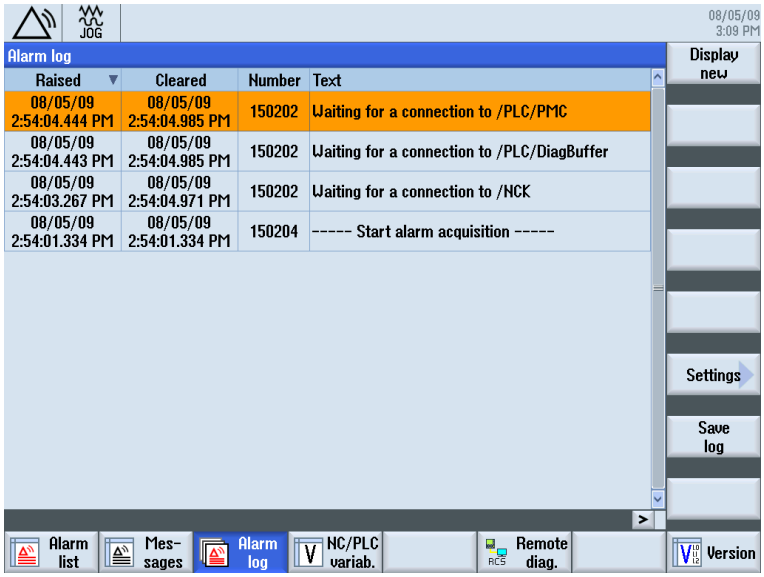


Figure 3-9 Alarm log

Basics for beginners

4

This chapter will explain the general basics of the geometry and technology for milling. No inputs for ShopMill are planned yet.

4.1 Geometrical basics

4.1.1 Tool axes and work planes

On universal milling machines, the tool can be mounted parallel to any of the three main axes. These perpendicular axes are aligned to the main guideway of the machine according to DIN 66217 or ISO 841.

The appropriate working plane results from the mounting position of the tool. Z is the tool axis in most cases.

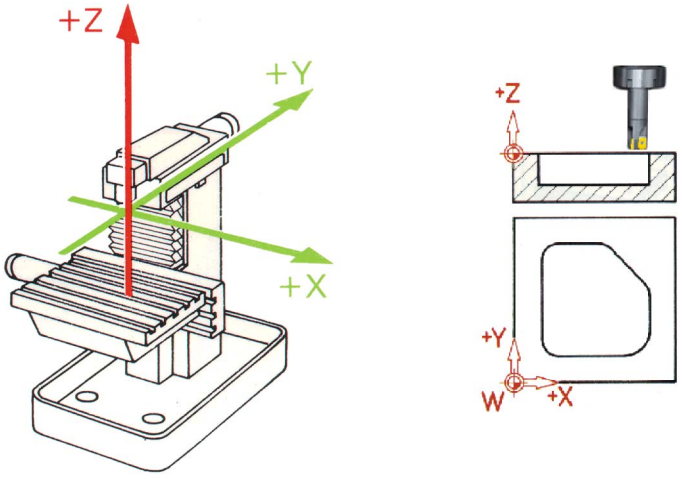


Figure 4-1 Vertical spindle

On modern machines, the tool mounting position is changed without any modification and in a few seconds by way of a universal swivel head.

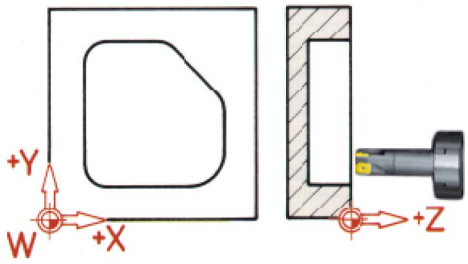


Figure 4-2 Horizontal spindle

If the coordinate system shown on the previous page is rotated accordingly, the axes and their directions in the appropriate working plane (DIN 66217) will change.

With the "Miscellaneous" and "Settings" softkeys, you can call a parameter screenform in which you can specify the working planes in the program header.

Select the "Miscellaneous" softkey.

Select the "Settings" softkey.

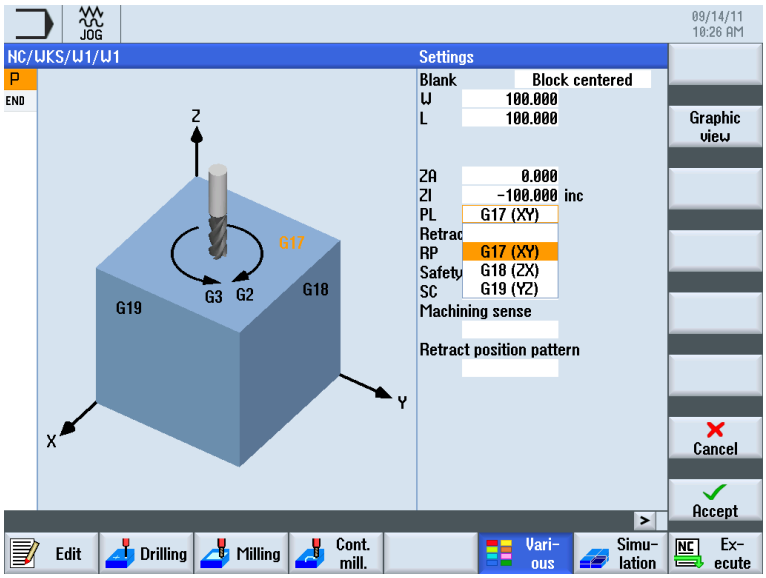
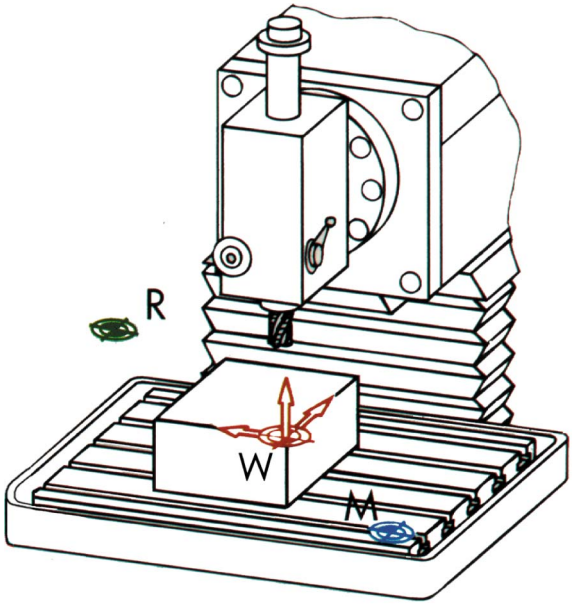


Figure 4-3 The "Working planes" parameter screenform

4.1.2 Points in the work space

Various important reference points are provided for a CNC - such as the SINUMERIK 828D with ShopMill - for orientation in the working space by way of the measuring system.



Machine zero (M)

The machine zero (M) is specified by the manufacturer and cannot be changed. It lies in the origin of the machine coordinate system.

Workpiece zero (W)

The workpiece zero (W) - also called program zero - is the origin of the workpiece coordinate system. It can be selected freely and should be located at a point from which the most dimensions start in the drawing.

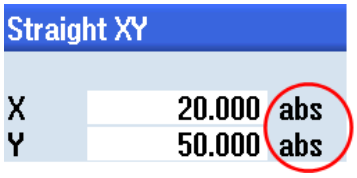
Reference point (R)

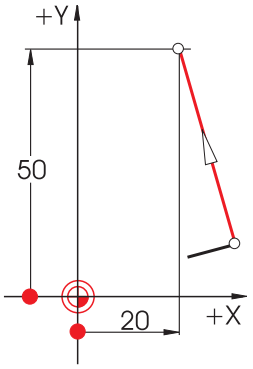
The reference point (R) is approached to set the measuring system to zero, as the machine zero cannot be approached in most cases. Thus, the control system finds the start of counting in the position measuring system.

4.1.3 Absolute and incremental dimensioning

Absolute input

The entered values refer to the workpiece zero.

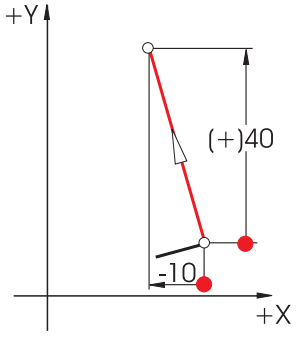




With absolute inputs, the **absolute** coordinate values of the **end point** must always be entered (the starting point is not taken into account).

Incremental input

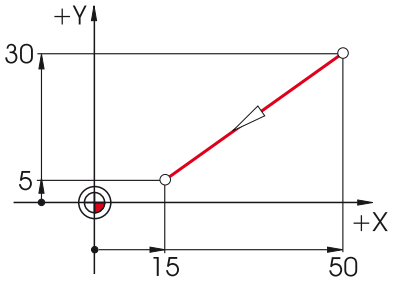
The entered values refer to the starting point.



With incremental inputs, the **difference** values between **starting point** and **end point** must always be entered, observing the **direction**.

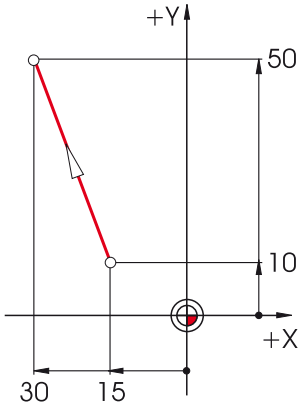
Switching between absolute and incremental input is possible at any time using the SELECT key.

A few examples for the absolute/incremental combination can be found below:



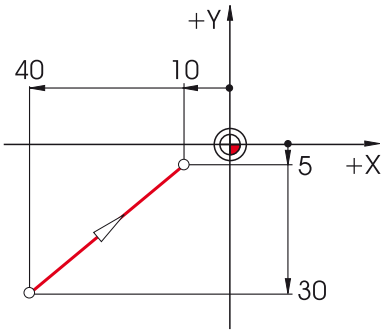
Absolute:
X15 Y5

Incremental:
X-35 Y-25



Absolute:
X-30 Y50

Incremental:
X-15 Y40



Absolute:
X-10 Y-5

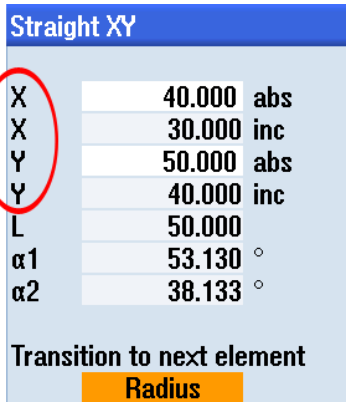
Incremental:
X30 Y25

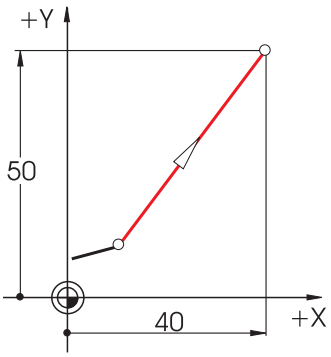
4.1.4 Linear motions

Two specifications are required to define an end point unambiguously. These specifications could be:

- Cartesian

  Input of the X and Y coordinates





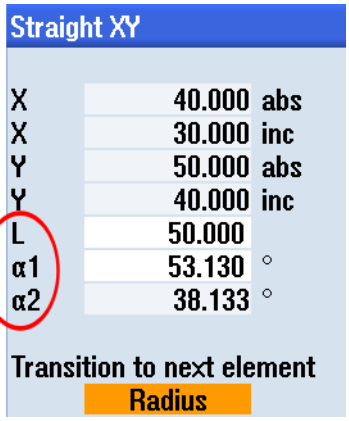
- Polar

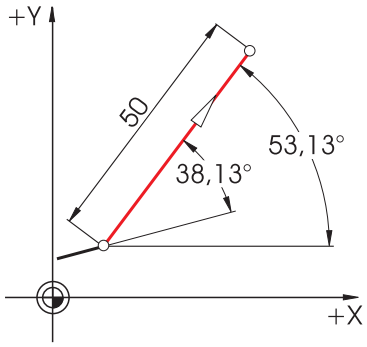
  Input of the length and an angle

  Angle 38.13° = angle with reference to the previous element

  or

  angle 53.13° = starting angle with reference to the positive X axis

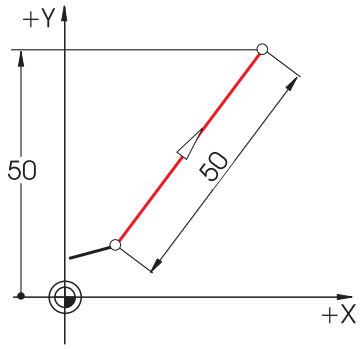




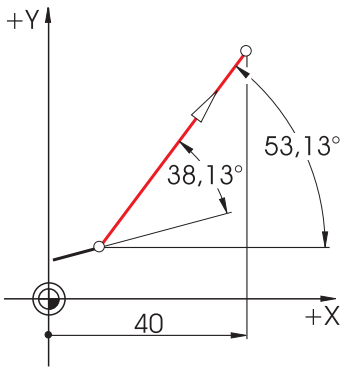
- Cartesian and polar

  It is possible to combine Cartesian and polar inputs, e.g.:

  – Input of the end point in Y and of the length



  – Input of the end point in X and of an angle (either 38.13° or 53.13°)



4.1.5 Circular motions

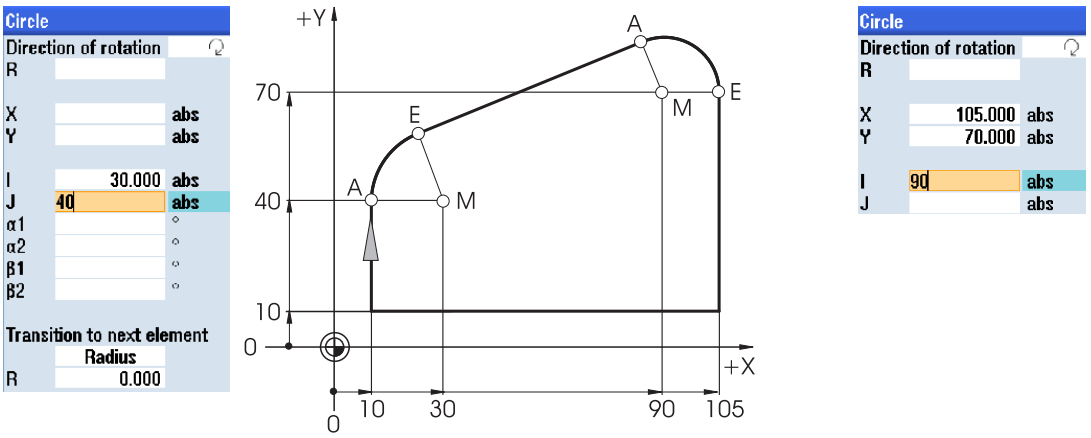
In the case of arcs, X and Y specify the end point; the circle center is specified with I and J. In ShopMill, these four values can be entered separately - either as **absolute** or **incremental** dimensions.

While X and Y are entered as absolute dimensions, the center point is specified with I and J as an incremental dimension in the most control systems. Not only the difference from the starting point **A** to the center**M** must be determined (often even in combination with mathematic calculations), but also the direction and thus the sign.

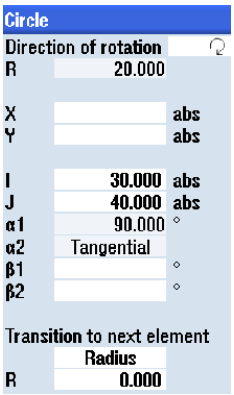
When working with ShopMill, however, you need not perform any calculations thanks to the possibility of entering the center point as an absolute dimension - even the most complicated contour can be determined easily using the graphical contour calculator.

Input of the center point (absolute)

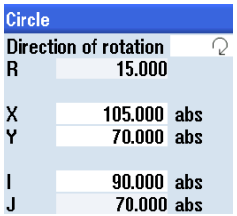
Values (here: radii) which result from data already entered are calculated by ShopMill automatically.



After the input:

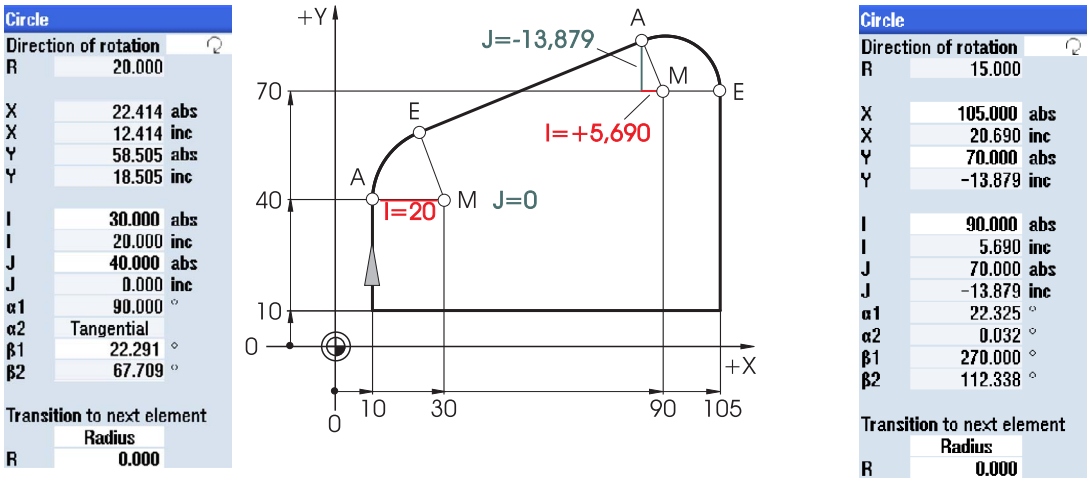


After the input:



Display of all parameters

ShopMill also allows to display **all** possible geometry values:



A further advantage of absolute center-point dimensioning: You need not recalculate the values for I and J when reversing the milling direction.

4.2 Technological basics

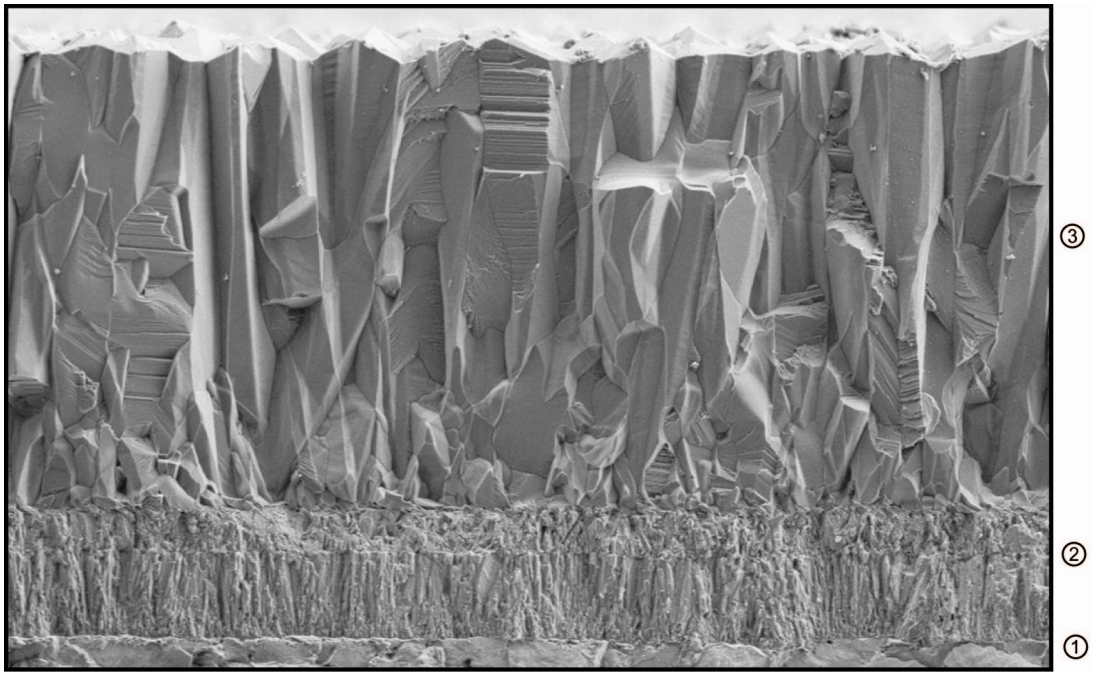
Fundamental prerequisites for optimum manufacturing are good knowledge of the tools, whereby above all the cutting materials of the tools, their possible applications and the optimum cutting data are meant. Even if tools only constitute approx. 2 ... 5 % of the total manufacturing costs of a workpiece, they influence the production costs of a component by over 50 % by way of their performance.

4.2.1 State-of-the-art milling and drilling tools

The cutting power has been improved continuously over the last few years thanks to the development of new cutting materials. In particular, the coating technology developed since the 1960s has provided a balanced strength/wear-resistance ratio. Such cutting materials have numerous other advantages: Longer service life and enhanced surface quality.

Special ceramic coatings, such as an Al_2O_3 layer, are the ideal choice for high cutting rates thanks to their heat resistance.

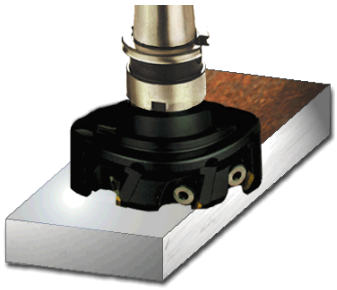
The photos which were kindly provided by the tool manufacturer SECO show a corner milling system (1st photo) with differently coated indexable inserts. The second photo shows a new kind of coating by SECO called DURATOMIC™ - in which vertically aligned Al_2O_3 crystals ③ are applied to a hard-metal substrate (HM) ① and a TiCN base layer ② .



This special coating also provides a further increase in strength and wear resistance.

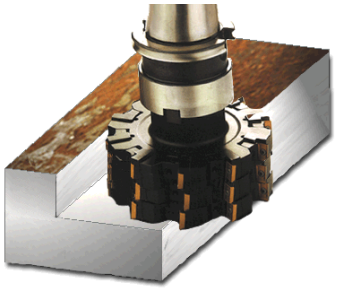
4.2.2 The tools in use

Facing cutter



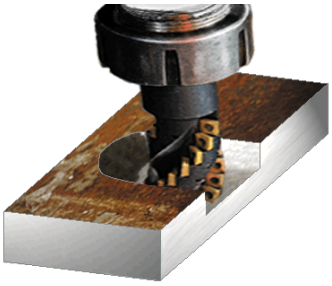
The facing cutter (also called facing head or milling head) is used to remove large amounts of material.

Shell end mill



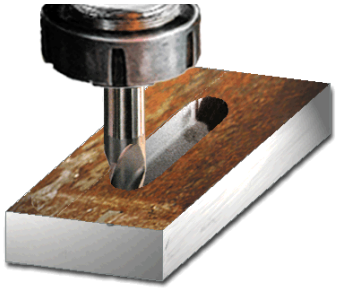
The shell end mill is used to create rectangular contour sections with vertical shoulders.

Helical shank mill



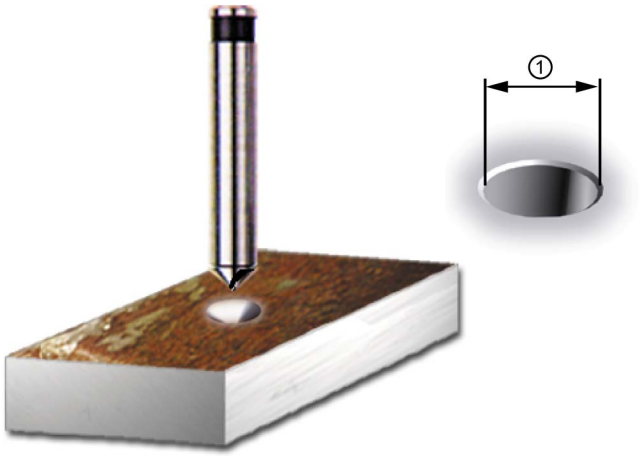
The helical shank mill is a multiple cutting-edge tool which provides especially smooth machining thanks to the spiral arrangement of the cutting edges.

Slotting end mill



The slotting end mill (also called drilling-groove cutter) cuts across the center and can therefore cut into the solid. Mostly, it possesses two or three cutting edges.

NC spotdrill



NC spotdrills are used to center and create a chamfer for the subsequent drilling. ShopMill calculates the depth automatically if you specify the outside diameter of the chamfer ①.

Twist drill

With ShopMill, you may select various drilling techniques (swarf milling, deep-hole drilling, ...). The 1/3D drill tip is taken into account in ShopMill automatically.

Solid drill

Solid drills are fitted with indexable inserts and are only available for drill holes with larger diameter. The drilling process must always be performed without interruption.

4.2.3 Cutting rate and speeds

The appropriate optimum speed of a tool depends on the cutting material of the tool and on the material of the workpiece, as well as on the tool diameter. In practice, this speed is often entered immediately, also on the basis of long-term experience. However, it is better to calculate the speed using the cutting rate taken from the relevant tables.

Example: Determination of the cutting rate

First, the optimum cutting rate is determined using either the manufacturer catalogs or a handbook.

Material of the **tool:**	Hard metal
Material of the **workpiece:**	C45
Determined value:	v_c = 80 ... 150 m/min
The mean value will be chosen:	**v_c = 115 m/min**

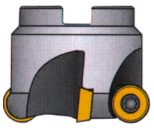
This cutting rate and the known tool diameter are used to calculate the speed **n**.

$$n = \frac{v_C \cdot 1000}{d \cdot \pi}$$

The speed for two tools is calculated in the following example:

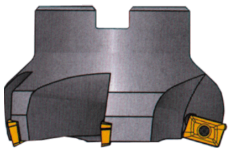
$d_1 = 40$ mm

$$n_1 = \frac{115\ \text{mm} \cdot 1000}{40\ \text{mm} \cdot \pi \cdot \text{min}}$$



$n_1 \approx 900$ rpm

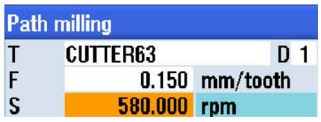
$d_2 = 63$ mm



$$n_2 = \frac{115\ \text{mm} \cdot 1000}{63\ \text{mm} \cdot \pi \cdot \text{min}}$$

$n_2 \approx 580$ rpm

In NC coding, the speed is specified with the letter **S** (from 'speed"). Therefore, the inputs are:



Note

ShopMill calculates the spindle speed automatically on the basis of the cutting rate and the tool diameter. This is useful for a cross-comparison, for example:

4.2.4 Feed per tooth and feedrates

In the previous chapter, you learned how to determine the cutting rate and the speed. The tool can only perform machining if the cutting rate or the speed is assigned a tool feedrate.

The basic value required to calculate the feedrate is the characteristic "feed per tooth". Like the cutting rate, the value for the feed per tooth is also taken from the handbook, the documents of the tool manufacturer or from the empirical knowledge.

Example: Determination of the feed per tooth

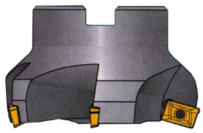
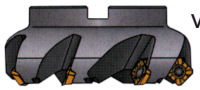
Cutting material of the **tool:**	Hard metal
Material of the **workpiece:**	C45
Determined value:	$f_z = 0.1 \ldots 0.2$ mm
The mean value will be chosen:	**$f_z = 0.15$ mm**

The feedrate v_f is calculated using the feed per tooth, the number of teeth and the known speed.

$v_f = f_z \cdot z \cdot n$

The feedrate for two tools with different number of teeth is calculated in the following example:

$d_1 = 63$ mm, $z_1 = 4$ $d_2 = 63$ mm, $z_2 = 9$

$v_{f1} = 580$ rpm $\cdot$ 0.15 mm $\cdot$ 4   $v_{f2} = 580$ rpm $\cdot$ 0.15 mm $\cdot$ 9

$v_{f1} = 348 \frac{mm}{min}$ $v_{f2} = 783 \frac{mm}{min}$

In NC coding, the feedrate is specified with **F** (from 'feed'). Therefore, the inputs are:

Path milling		
T	CUTTER63	D 1
F	340.000	mm/min
S	580.000	rpm

Path milling		
T	CUTTER63	D 1
F	780.000	mm/min
S	580.000	rpm

Note

ShopMill calculates the feedrate automatically using the feed per tooth and the number of teeth. This is useful for a cross-comparison, for example:

Well equipped

5

In this section you will learn how to create the tools required for the examples in the following sections. Furthermore, it is explained how to take into account the tool lengths and how to set the workpiece zero.

5.1 Tool management

ShopMill offers three lists for tool management:

- the tool list
- the tool wear list
- the magazine list.

5.1.1 The tool list

The tool list displays all parameters and functions required to create and set up the tools.

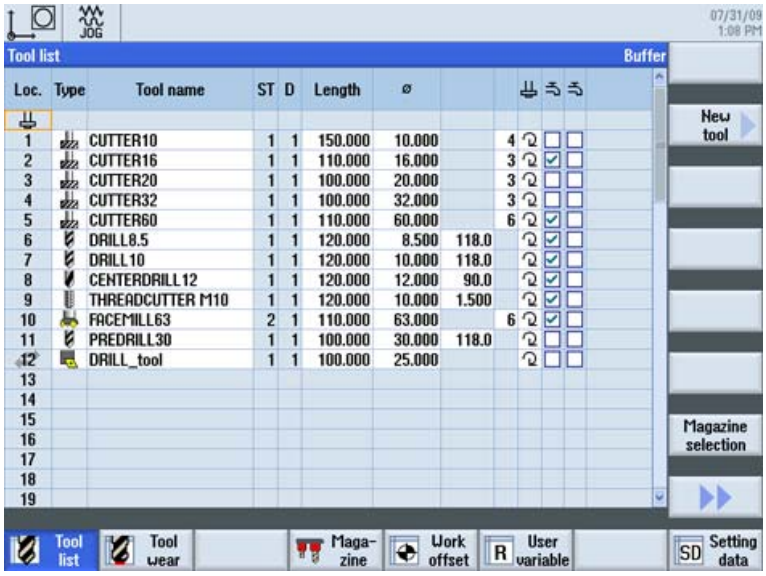


Figure 5-1 Example for tool lists

Meaning of the most important parameters in the tool list:

Location	Location number
Type	Tool type
Tool name	The tool is identified by the name and the replacement tool number. You may enter the names as text or numbers.
ST	Replacement tool number (for replacement tool strategy)
D	Cutting edge number
Length	Tool length
Diameter	Tool diameter
Point angle or lead	Point angle or lead
N	Number of teeth
	Direction of spindle rotation
	Coolants 1 and 2 (e.g. internal and external cooling)

ShopMill provides various tool types (favorites, milling cutters, drills, and special tools). Tools can be created in the tool list by means of a predefined tool catalog. The geometrical parameters (e.g. angle specifications for drills) are different for each tool type.

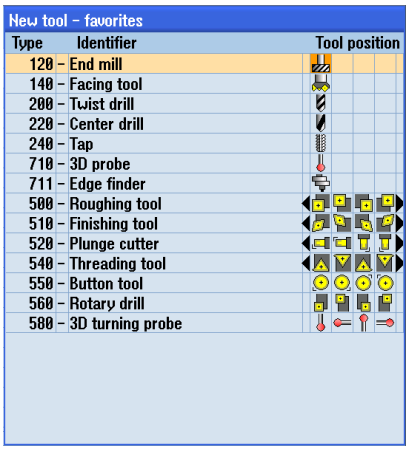


Figure 5-2 Example of Favorites list

5.1.2 The tool wear list

The wearing data for the appropriate tools are defined here.

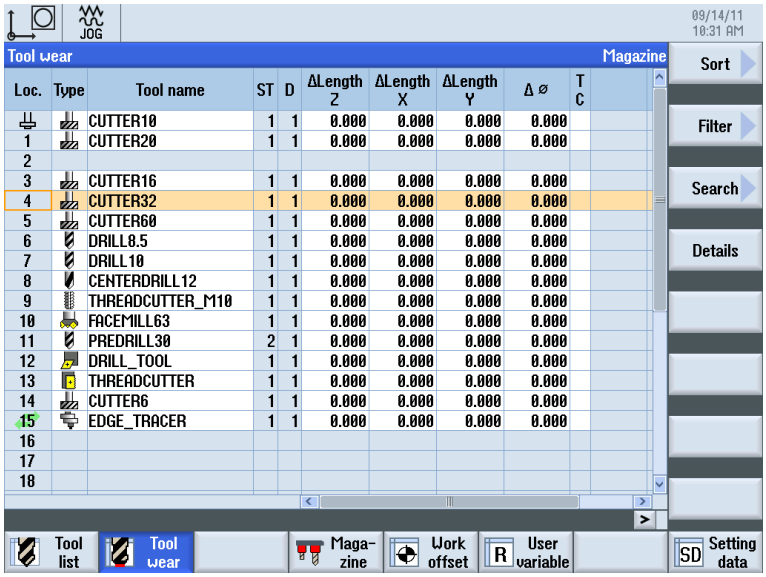


Figure 5-3 Tool wear list

The most important tool wearing parameters are:

Δ Length	Length wear
Δ Radius	Radius wear
TC	Selection of tool monitoring • by tool life (T) • by count (C) • by wear (W)
Tool life or workpiece count or wear * *Parameter depends on selection in TC	Tool life Number of workpieces Tool wear
Setpoint	Setpoint for tool life, workpiece count, or wear
Prewarning limit	Specification of the tool life, workpiece count or wear at which a warning is displayed.
G	The tool is disabled when the checkbox is selected.

5.1.3 Magazine list

All tools that are assigned to one or several tool magazines are contained in the magazine list. This list displays the condition of each tool. Individual magazine locations can be reserved or disabled for existing tools.

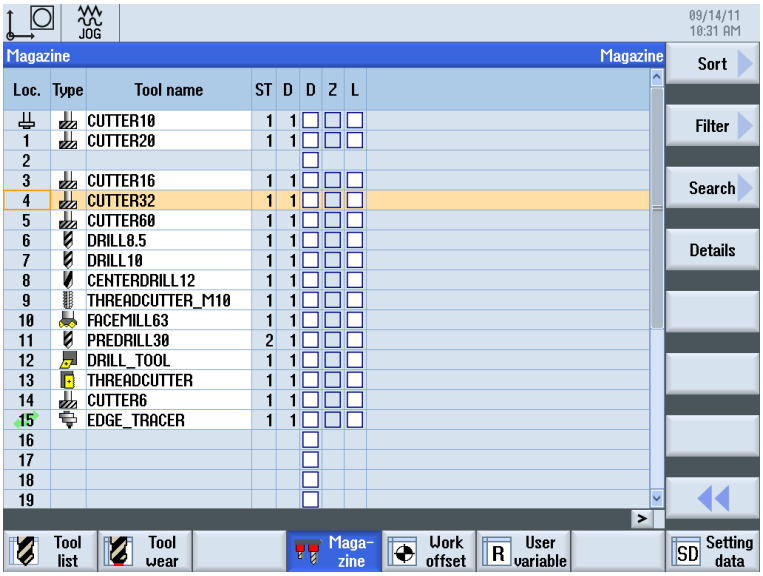


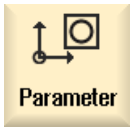
Figure 5-4 Magazine list

Meanings of the most important parameters:

G	Disabling of the magazine location
Ü	Marking of a tool as oversized. The tool occupies two half locations left, two half locations right, one half location top and one half location bottom in a magazine.
P	Fixed location coding The tool is permanently assigned to this magazine location.

5.2 Tools used

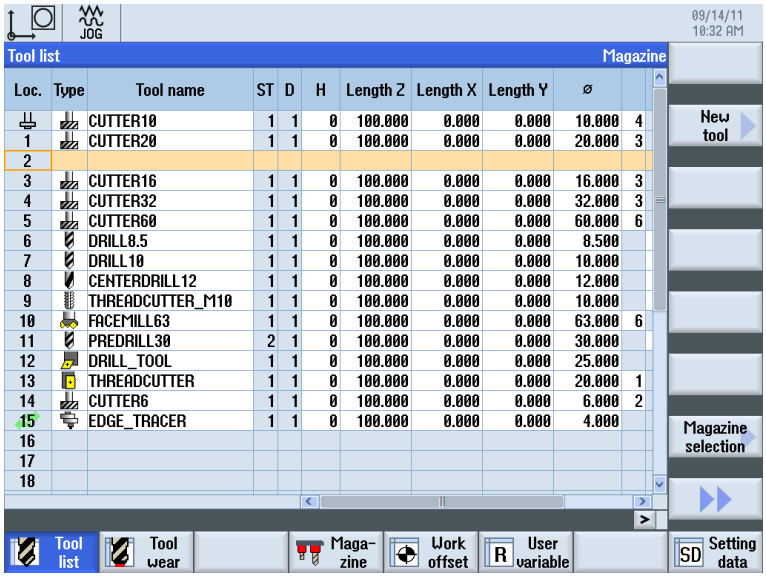
In this section you will learn how to enter tools required for the later machining in the tool list.



Select the "Parameters" area in the main menu.

Select the "Tool list" softkey.

To create a new tool, call the tool list and search for a free location.



Select the "New tool" softkey.

Select the desired tool type from the tool catalog displayed. This tool type is inserted in the tool list and you can enter the data of the tool.

Note

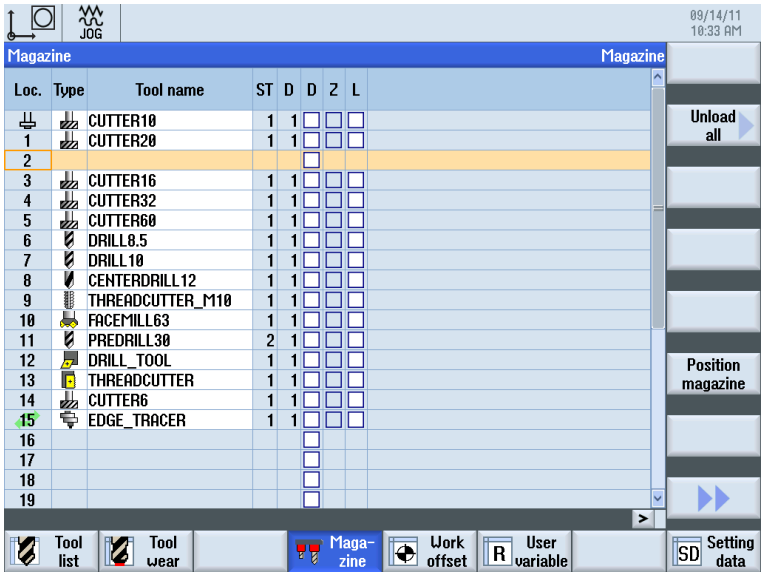
The milling cutters with the diameters 6, 10, 20 and 32 (Cutter6, 10, 20 and 32) must immerse, as they will also be used for the milling of pockets in the following examples.

5.3 Tools in the magazine

In the following you will learn how to insert the tools into the magazine.

Select a tool without location number from the tool list and select the "Load" softkey.

The following dialog offers the first free magazine location for you to change or accept directly. The magazine for the following exercises could look like the one in the screen below:



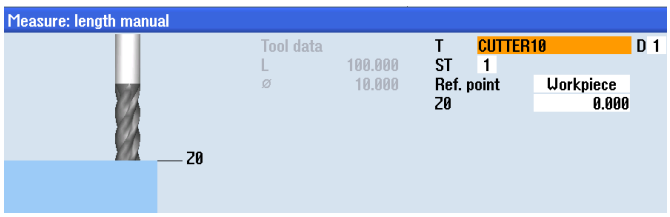
5.4 Gauging tools

In the following you will learn how to calculate tools.

Insert a tool from the tool list into the spindle using the "T,S,M" softkey.

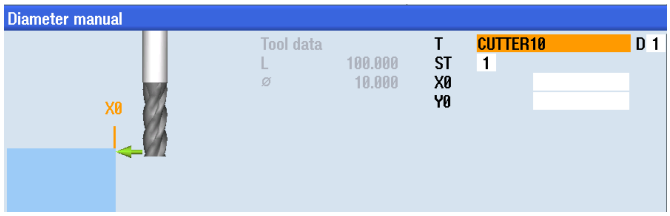
Then switch to the "Gauge tool" menu.

The tool is measured in the Z direction using the **Length, manual** function.



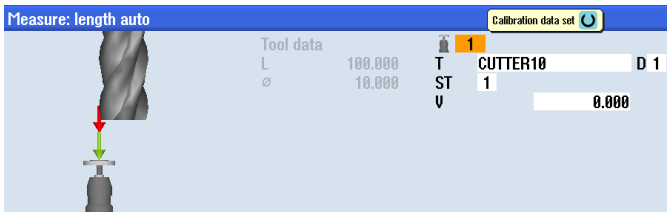
Diameter manual

The diameter of the tool is measured using the **Diameter, manual** function.



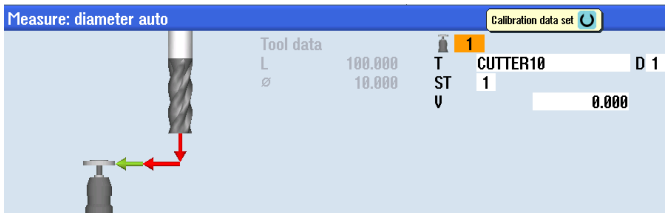
Length auto

The **Length, autom.** function can be used to measure the tool in the Z direction using a tool probe.



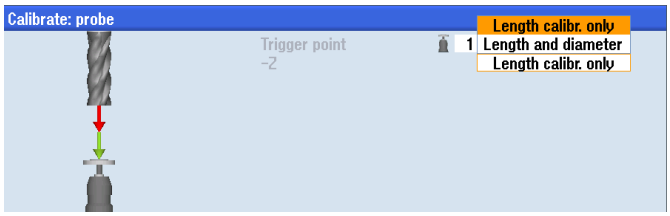
Diameter auto

The **Diameter, autom.** function is used to measure the diameter of the tool using a tool probe.



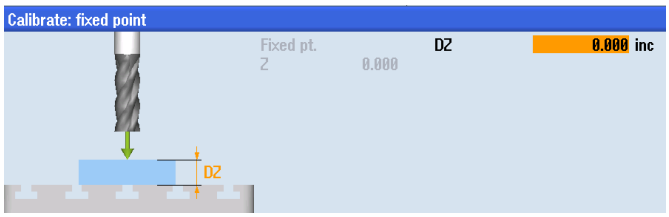
Calibrate probe

The **Calibrate probe** function is used to determine the position of the sensing probe on the machine table with reference to the machine zero.



Calibrate fixed pt.

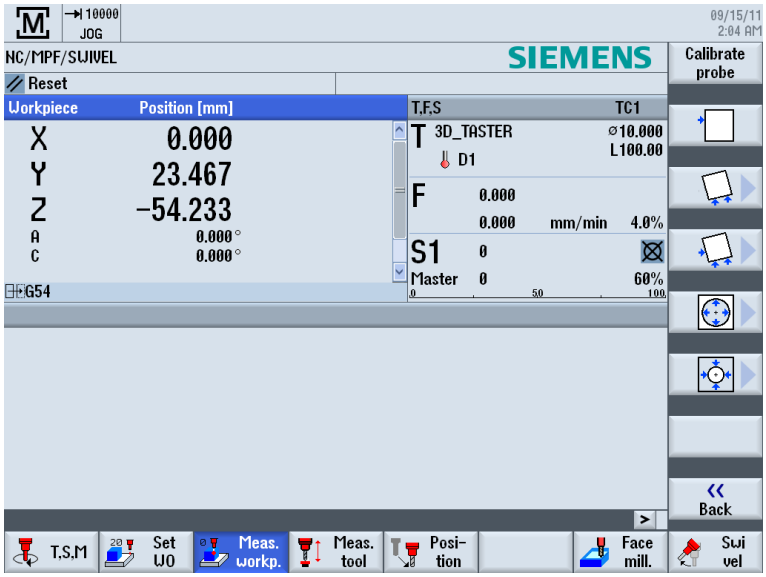
The **Fixed-point calibration** function is used to determine the fixed point used as the reference point for measuring the tool length manually.

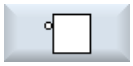


5.5 Setting the workpiece zero

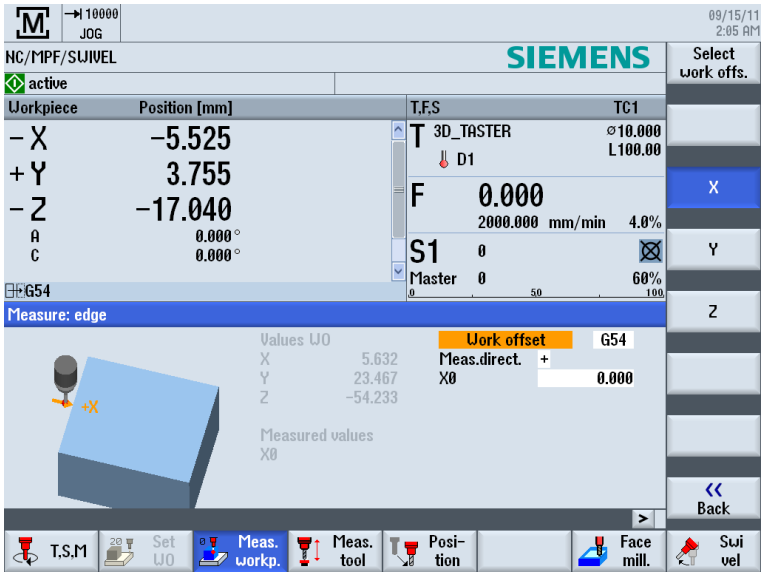
To set the workpiece zero, switch to the **Machine - Manual** mode in the main menu.

The submenu of the **Workp. zero** option offers various possibilities to set the workpiece zero.





The zero point of a workpiece edge will be set in the following example using an edge probe.



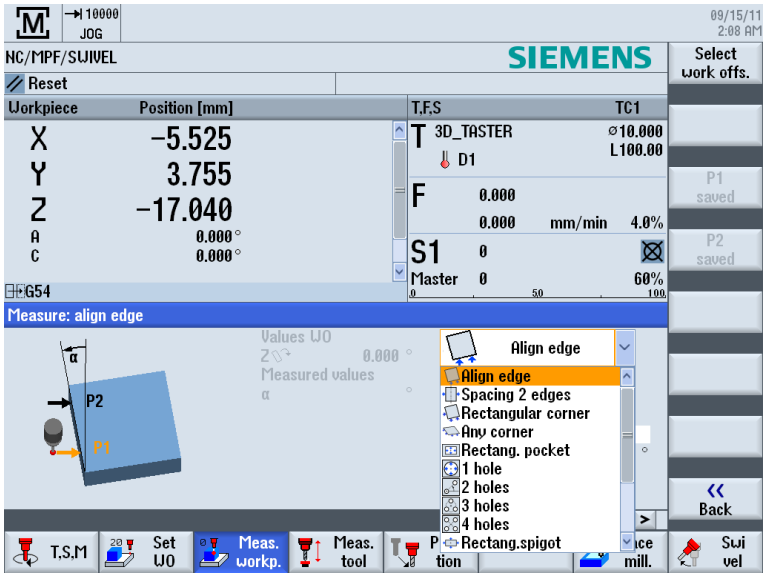
1) Selecting the edge

Define the sampling direction for the probe: LH (+) or (-). The X0 parameter can be used to specify an offset for the workpiece zero if the zero is not to lie on the edge of the workpiece.

2) Sampling the workpiece edge

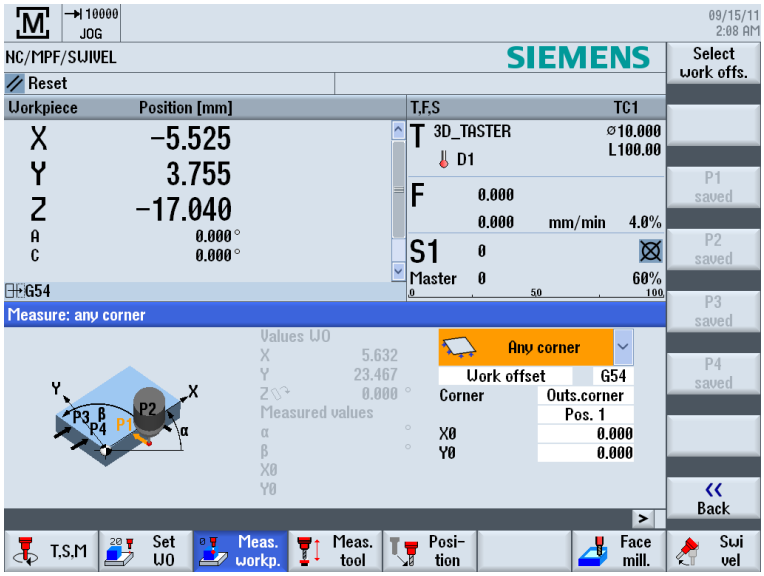
3) Set the workpiece zero taking into account the edge probe diameter (5 mm). Now this process of calculation must be repeated for Y using the edge probe and for Z (in most cases, with the milling cutter).

Since the workpieces to be machined are not always present in the form of a cuboid or can be clamped at right angles, further calculation possibilities are provided:



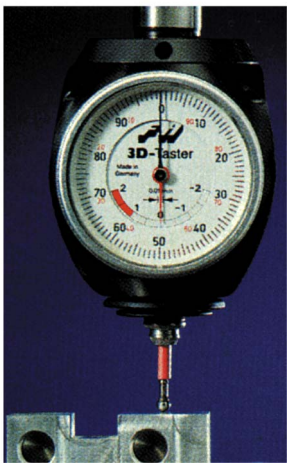
Example 1: Any corner

If the workpiece is positioned as shown here, the position/corner of the workpiece can be determined by approaching four points.

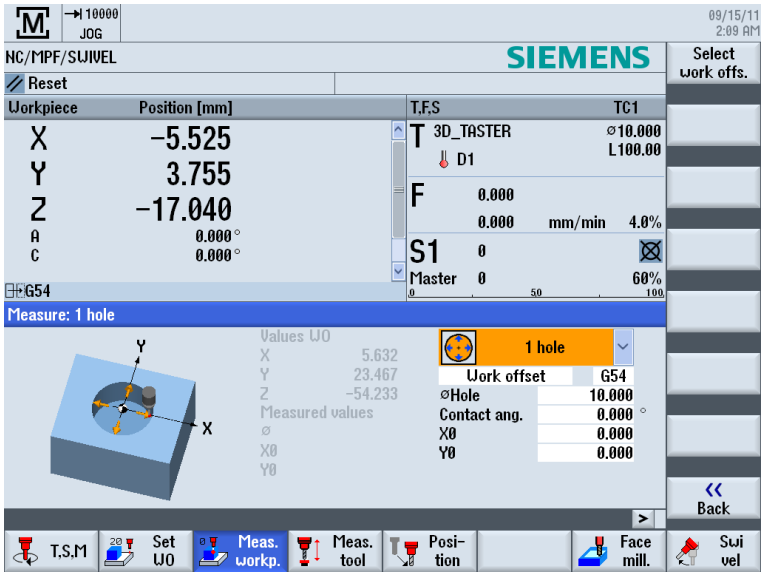


3D probes are offered in both the electronic and mechanical variants.

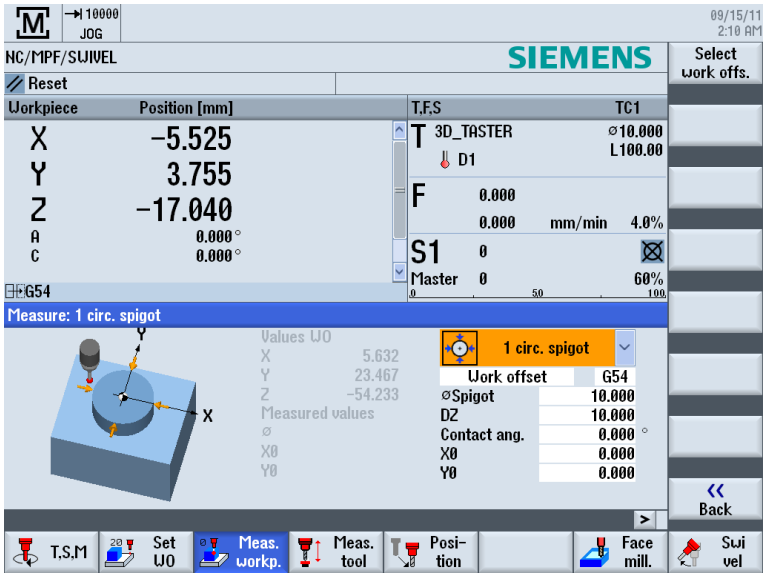
The signals issued by electronic probes can be processed by the control system directly.



Example 2: Calculating a drill hole



Example 3: Calculating a circular spigot



Calibrate probe

If an electronic 3D probe from the tool magazine is inserted into the spindle, clamping tolerances will occur. This would lead to incorrect results in further measurements. This can be avoided by calibration of the 3D probe at any reference surface or in any reference drill hole using the **Calibrate probe** cycle.

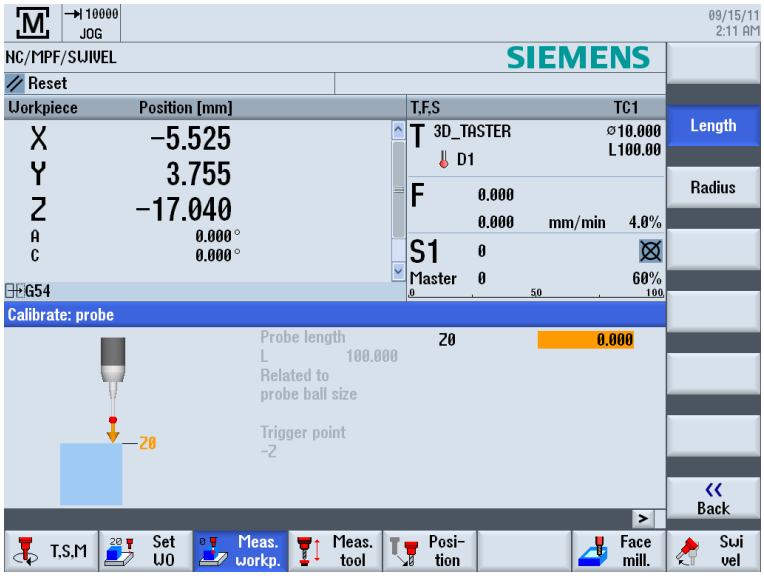


Figure 5-5 Calibrating the probe for the length

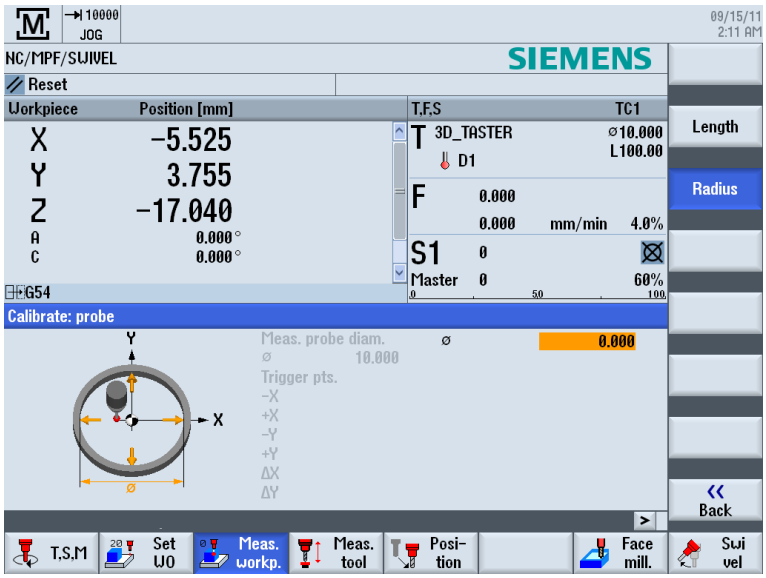


Figure 5-6 Calibrating the probe for the radius

Example 1: Longitudinal guide

6

6.1 Overview

Learning objectives

This section will explain the first steps to create a workpiece in detail. You will learn how to...

- create and manage programs;
- call tools and perform a cutter radius compensation;
- enter traversing paths;
- create drill holes and handle position repetitions.

Task

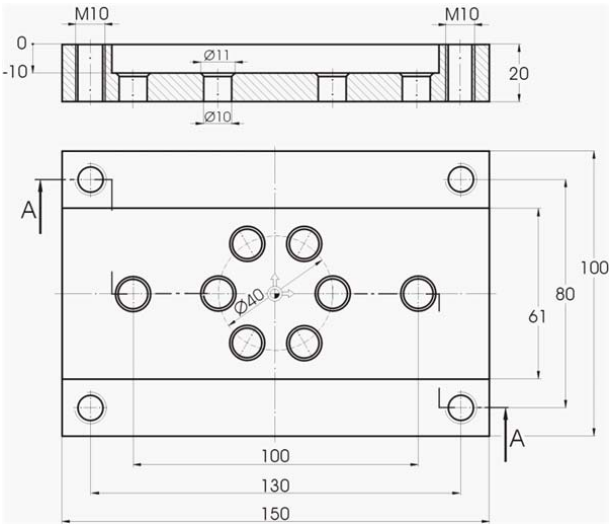


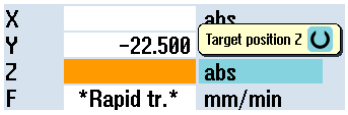
Figure 6-1 Workshop drawing - Example 1:

Figure 6-2 Workpiece - Example 1:

Note

ShopMill always saves the last setting selected with the toggle key. Therefore, make sure that all units, texts and symbols are specified as in the dialog boxes shown here in the relevant input fields and all toggle fields.

Whenever it is possible to switch this is indicated in the help text (see screenshot below).



6.2 Program management / creating programs

Operating sequences

After power-up of the control system, you are in the main menu.

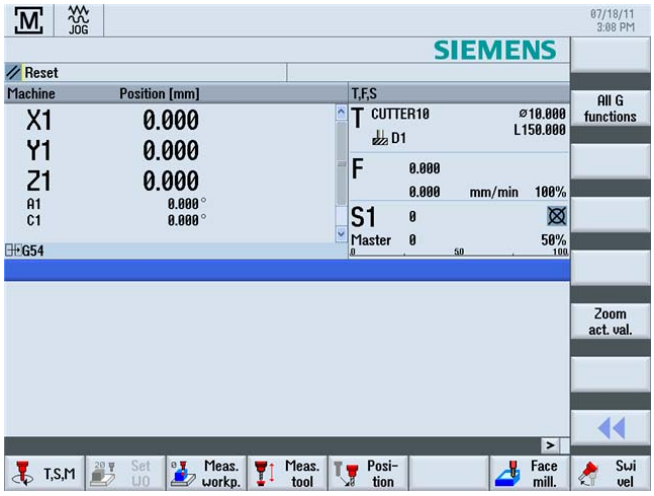


Figure 6-3 Main screen

Open the main menu using the **MENU SELECT** key. In the main menu, you may call various areas of ShopMill.

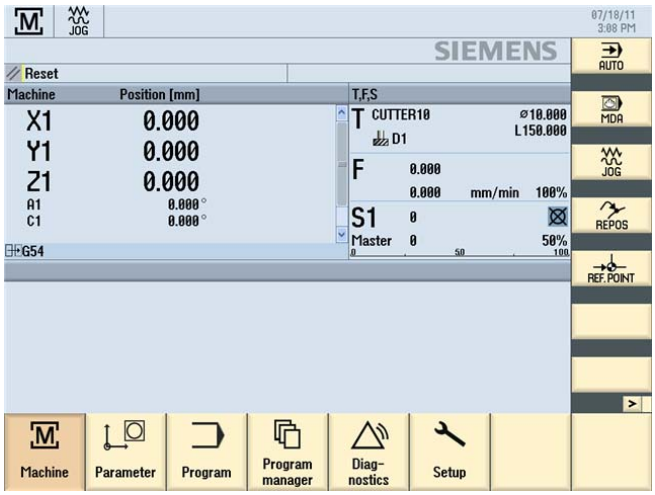


Figure 6-4 Main menu

Select the **Program Manager** softkey. The Program Manager is opened.

In the Program Manager, you can manage process plans and contours (e.g. "New", "Open", "Copy", ...).

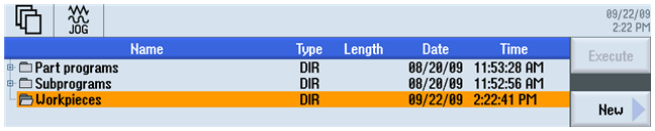


Figure 6-5 Program Manager

The Program Manager displays a list of the existing ShopMill directories. Use the cursor key to select the 'Workpieces' directory.

Open the "Workpieces" directory.

Enter the name 'EXAMPLE1' for the workpiece.

Figure 6-6 Creating a workpiece

Confirm your input. The following dialog box is opened:

Figure 6-7 Creating a step sequence program

Select the input format using the **ShopMill** and **ProgramGUIDE G code** softkeys.

Via the **ShopMill** softkey, you specify the program type.

Specify the name of the process plan, in this case 'Longitudinal_guide'.

"Apply" your input.

After confirming, the following interactive screenform is displayed to enter the workpiece data.

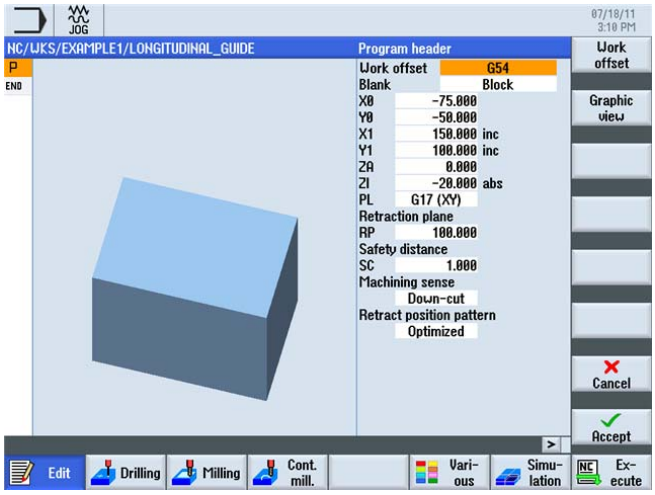


Figure 6-8 Program header - Help display

Enter the workpiece data and general program specifications in the program header.

Enter the following values:

Field	Value	Selection via toggle key	Notes
Unit of measurement	mm	X	
Work offset	G54	X	
Blank	Cuboid	X	
X0	-75		Since the workpiece zero lies centrally on the workpiece surface, the coordinates of the left workpiece corner have negative values.
Y0	-50		
X1	150 inc	X (for selection of inc/abs)	
Y1	100 inc	X (for selection of inc/abs)	
ZA	0		
ZI	-20 abs	X (for selection of inc/abs)	
PL	G17 (XY)	X	
Retraction plane	100		
Safety clearance	1		
Machining direction	Synchronous	X	
Retraction position pattern	Optimized	X	See below *Retraction position pattern*

"Apply" the set values. After confirming, the program header is displayed.

Figure 6-9 Program header, example 1 - Work step editor

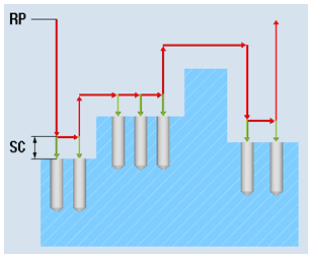
Now the program has been created as the basis for further machining steps. It has a name (in the blue bar), a program header (pictogram "P") and a program end (pictogram "END"). The individual machining steps and contours are stored in the program one beneath the other. The later machining is performed from top to bottom.

You may call the program header again at any time to make changes or check the values.

Retraction position pattern

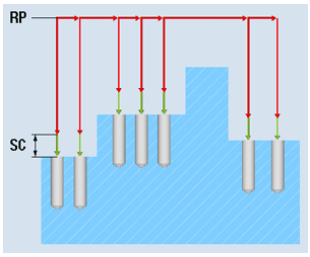
The position pattern can be set to "Optimized" (= time-optimized traversing distances) or "Retraction plane".

Optimized retraction



The tool traverses over the workpiece at the safety clearance in accordance with the specified contour.

To retraction plane (standard)



The tool traverses back to the retraction plane and performs infeed to the new position.

Softkeys

Graphic view

Use this softkey to switch to the online graphic of the workpiece (see screenform below).

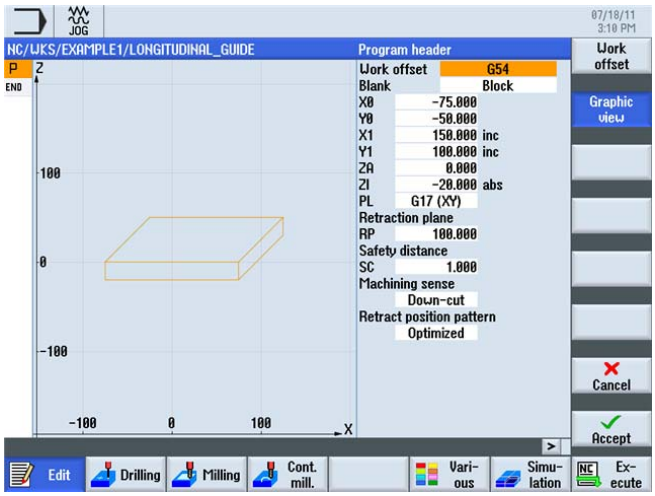


Figure 6-10 Program header - graphical view

Graphic view

Use this softkey to switch back to the help display.

6.3 Calling a tool and specifying cutter radius compensation

Operating sequences

To call the required tool, proceed as follows:

Use this key to extend the horizontal softkey menu.

Select the **Straight line Circle** softkey.

Select the **Tool** softkey.

Open the tool list.

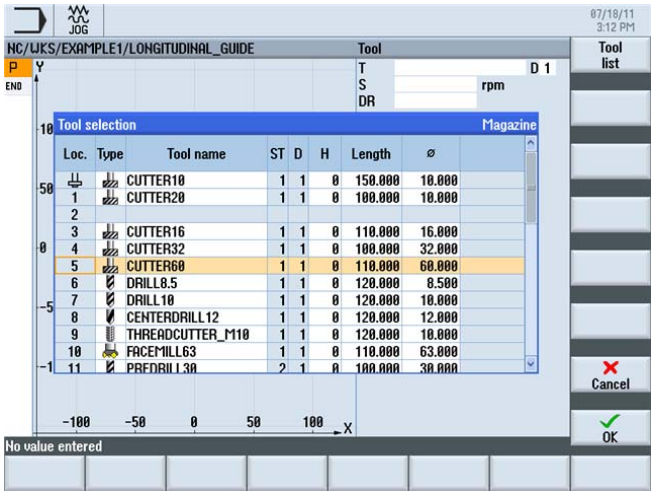


Figure 6-11 Tool list

Use the cursor key to select the CUTTER60 tool.

Accept the tool into your program. After accepting the tool, specify the cutting rate 80 m/min (if necessary, change the unit using the toggle key).

Figure 6-12 Tool cutting rate

"Apply" the set value.

6.4 Specifying the distance to be traversed

Operating sequences

Now enter the distances to be traversed:

Select the "Straight line" softkey.

Select the "Rapid traverse" softkey.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
X	110 abs	X	
Y	0 abs	X	
Radius compensation	off	X	See below *Radius compensation*

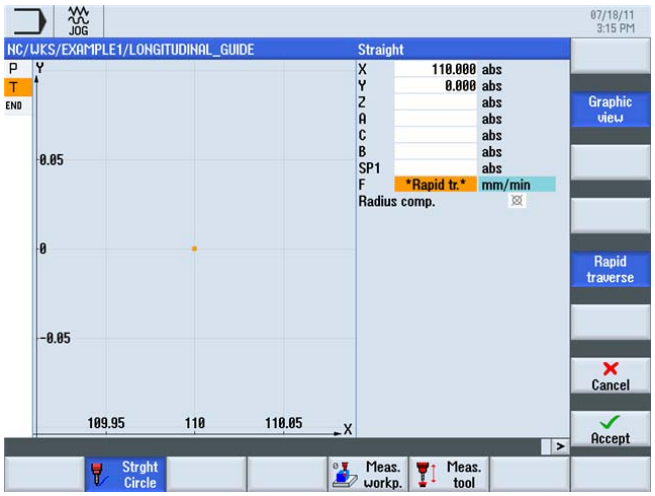


Figure 6-13 Specifying the distance to be traversed - Radius compensation

"Apply" the set values.

Select the "Straight line" softkey.

Select the "Rapid traverse" softkey.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z	-10 abs	X	
Radius compensation	Empty field	X	See below *Radius compensation*

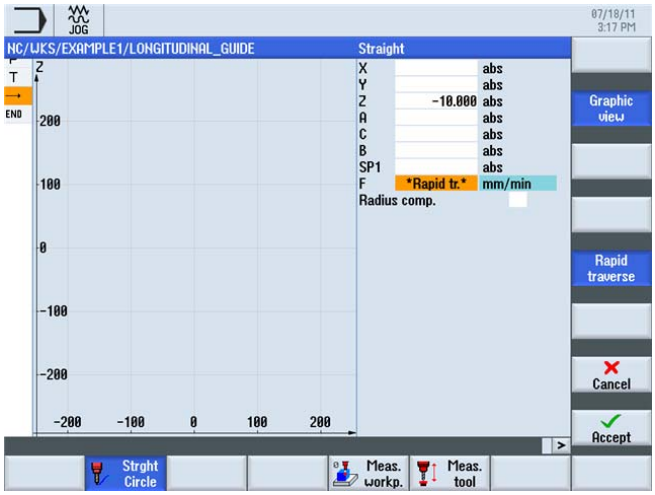


Figure 6-14 Specifying the distance to be traversed - Tool positioned in Z

"Apply" the set values.

Select the "Straight line" softkey.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
X	- 110 abs	X	
F	400 mm/min	X	
Radius compensation	Empty field	X	See below *Radius compensation*

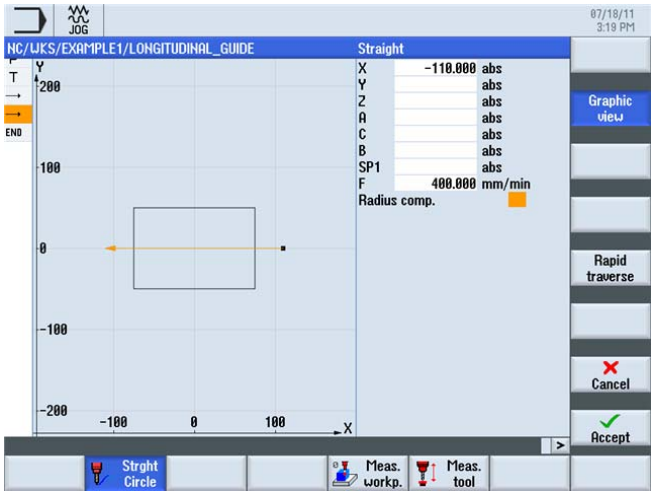


Figure 6-15 Specifying the distance to be traversed - First machining path

"Apply" the set values. After acceptance, the list of work steps looks like this:

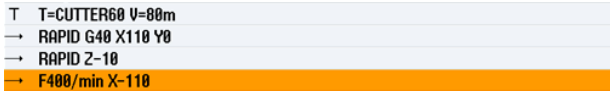


Figure 6-16 Specifying the distance to be traversed - List of work steps

Select the "Tools" softkey and perform the following work steps without help.

Load the next tool CUTTER16. After accepting the tool, specify the cutting rate 100 m/min.

Create the distance to be traversed according to the following list of work steps.

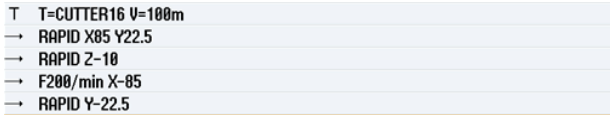


Figure 6-17 Specifying the distance to be traversed - Work step list

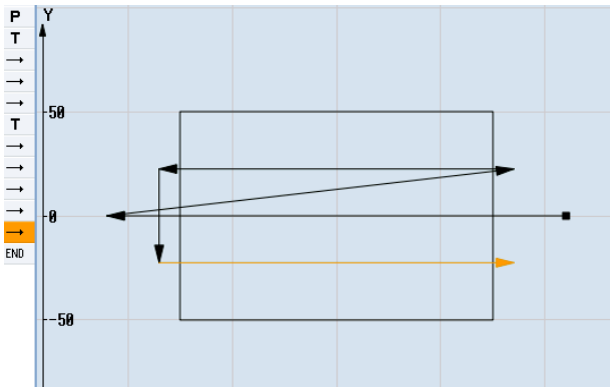


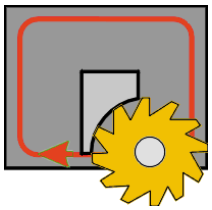
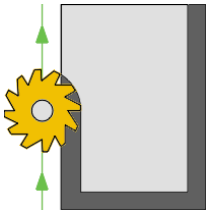
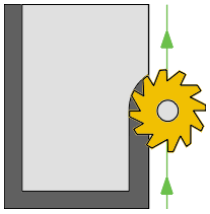
Figure 6-18 Specifying the distance to be traversed - Complete

Start simulation.

Figure 6-19 Simulation of the distance to be traversed

You may end simulation by selecting either the **Simulation** softkey again or any other softkey.

Radius compensation

Selection	Result
	 Radius compensation is disabled. The milling cutter traverses with its center point along the created contour.
	The existing settings for the radius compensation is maintained.
	 The compensation is performed to the left of the contour in the milling direction.
	 The compensation is performed to the right of the contour in the milling direction.

6.5 Creating drill holes and position repetitions

Operating sequences

Now enter the values for the drill holes and position repetitions. You will have to center, through-drill and make threads for 12 drill holes.

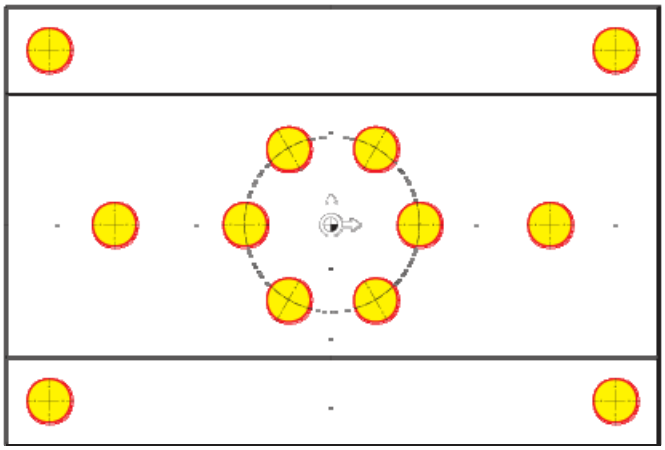


Figure 6-20 Drilling positions

Select the **Drilling** softkey.

Select the **Center** softkey.

Open the tool list. Use the cursor key to select the CENTERDRILL12 tool.

Accept the tool into your program. After accepting the tool, enter the following values:

Field	Value	Selection via toggle key	Notes
F	150 mm/min	X	
S	500 rpm	X	
Diameter/tip	Diameter	X	Centering can be entered with reference either to the diameter or to the depth (tip). Since the drill holes possess a 0.5 mm chamfer, you may specify a diameter of 11 mm.

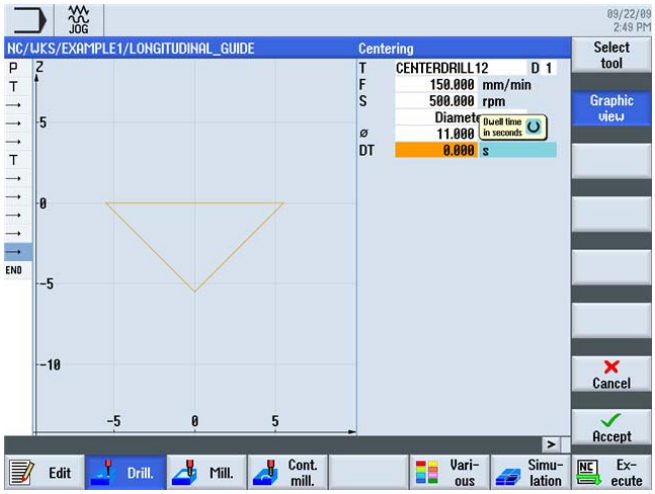


Figure 6-21 Centering

"Apply" the set values.

Proceed as follows to specify and link the drilling positions with the cutting data:

Select the **Positions** softkey.

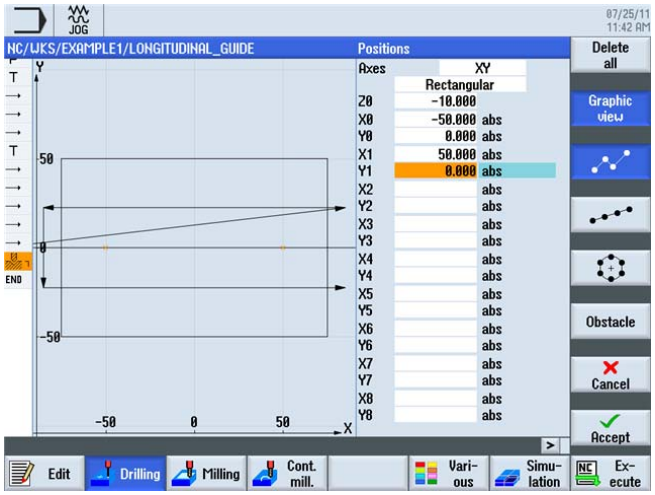


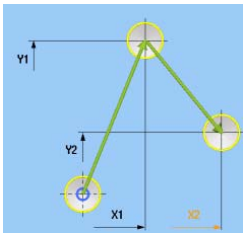
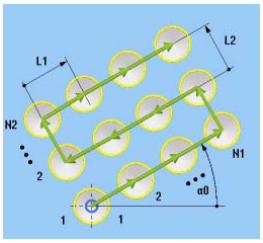
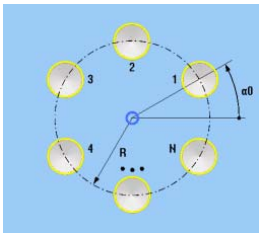
Figure 6-22 Positions - Individual drill holes

Enter the following values for the two individual drill holes:

Field	Value	Selection via toggle key	Notes
Z0	-10		The starting depth is -10mm.
X0	-50		
Y0	0		
X1	50 abs	X	
Y1	0 abs	X	

Note

If you deselect the **Graphic view** softkey, detailed help displays are displayed (see table below).

Positions	Position pattern	Position circle
		

Help displays - Positions

"Apply" the set values.

Select the **Positions** softkey.

Select the **Position circle** softkey.

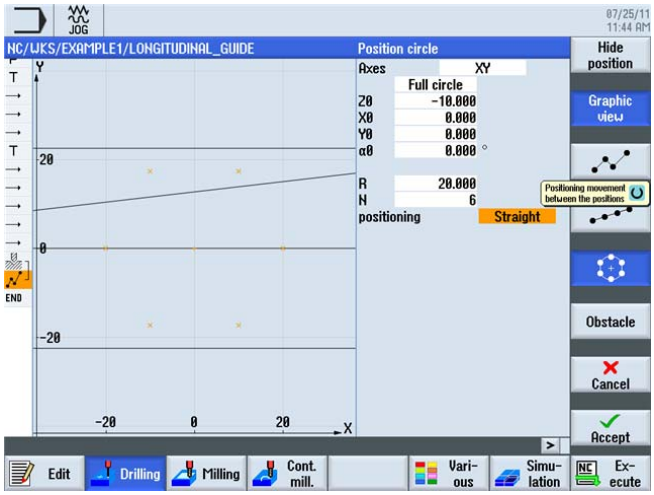
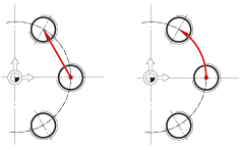


Figure 6-23 Position circle

Enter the following values:

Field	Value	Selection via toggle key	Notes
Pattern	Full circle	X	
Z0	-10		
X0	0		
Y0	0		
α1	0		
R	20		
N	6		
Positioning	Straight line	X	Use the "Positioning" field to define how to approach the drill holes within the drill pattern. If the drill holes lie in a circumferential groove, for example, do not use "Positioning - Straight line"; otherwise, a contour violation would result.  Along a straight line, along a circle

"Apply" the set values.

Select the **Positions** softkey.

Select the **Position pattern** softkey.

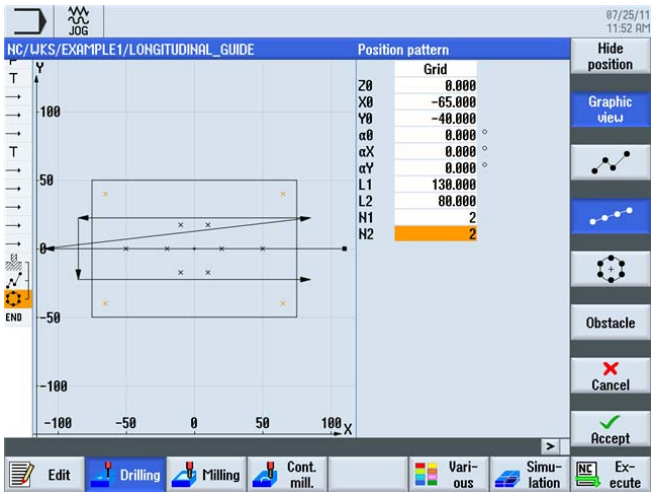


Figure 6-24 Positions - Matrix

Enter the following values:

Field	Value	Selection via toggle key	Notes
Pattern	Matrix	X	
Z0	0		
X0	-65		
Y0	-40		
α0	0		
L1	130		
L2	80		
N1	2		
N2	2		

"Apply" the set values.

Select the **Drilling Reaming** softkey.

Open the tool list. Use the cursor key to select the DRILL8.5 tool.

To program

Accept the tool into your program. After accepting the tool, enter the following values:

Field	Value	Selection via toggle key	Notes
F	150 mm/min	X	
V	35 m/min	X	
Shank/tip	Shank	X	Specify the depth with reference to the shank incrementally. In other words: The 1/3 D drill tip is taken into account automatically.
Z1	20 inc	X	
DT	0 s	X	Drilling is performed without a dwell time.

Note

The work steps 'Centering', 'Drilling' and 'Tapping' are linked with each other automatically.

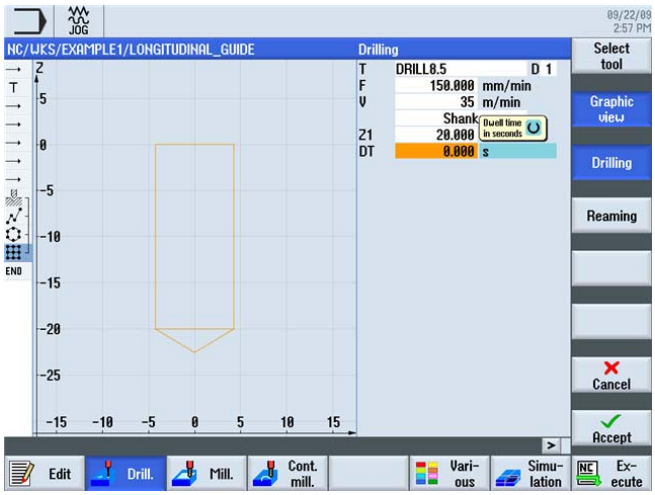


Figure 6-25 Drilling

Accept

"Apply" the set values.

Thread

Select the **Thread** softkey.

Tapping

Select the **Tapping** softkey.

Select tool

Open the tool list. Use the cursor key to select the THREADCUTTER M10 tool.

To program

Accept the tool into your program. After accepting the tool, enter the following values:

Field	Value	Selection via toggle key	Notes
P	1.5 mm/rev	X	
S	60 rpm	X	
SR	60 rpm	X	
Z1	22 inc	X	The cutting depth must be entered incrementally.

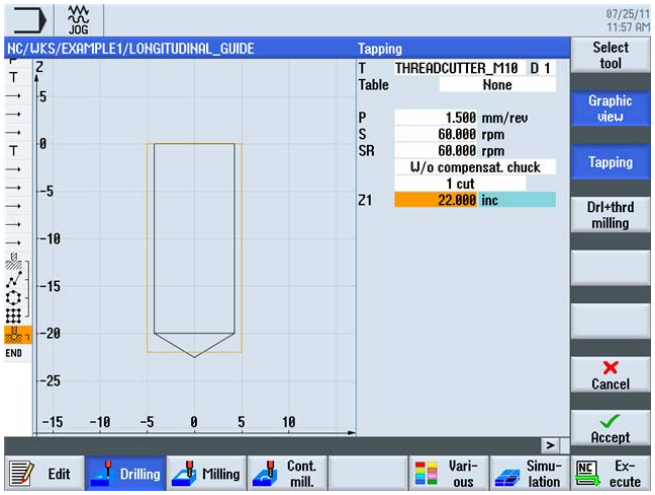


Figure 6-26 Thread

Accept

"Apply" the set values.

Position repetit.

Select the **Repeat position** softkey.

The drilling positions are numbered consecutively during creation. The appropriate number is to be found directly after the block number of the corresponding position pattern. Specify "Hole matrix" for position 3.

Figure 6-27 Repeating a position

Accept

"Apply" the set values. After accepting the values, you will see the linking of the work steps in the work step editor.

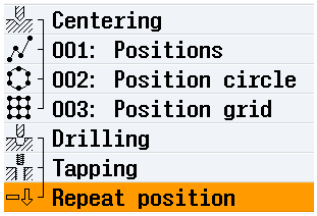


Figure 6-28 Linking of work steps

Drilling Reaming

Select the **Drilling Reaming** softkey.

Select tool

Open the tool list. Use the cursor key to select the DRILL10 tool.

To program

Accept the tool into your program. After accepting the tool, enter the following values:

Field	Value	Selection via toggle key	Notes
F	150 mm/min	X	
V	35 m/min	X	
Shank/tip	Shank	X	
Z1	20 inc	X	
DT	0	X	

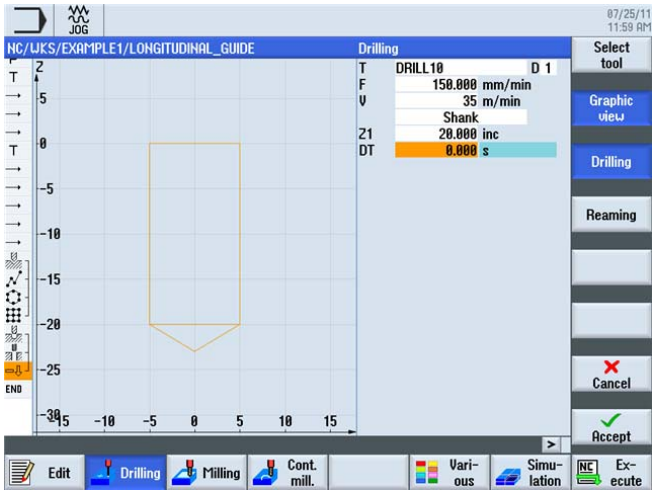


Figure 6-29 10mm drill holes

"Apply" the set values.

Last, repeat the positions 001 and 002 for the 10mm drill.

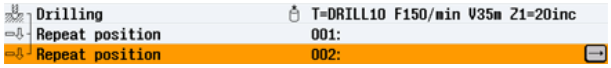


Figure 6-30 Repeat the positions 001 and 002 in the work step editor.

Call the simulation for checking.

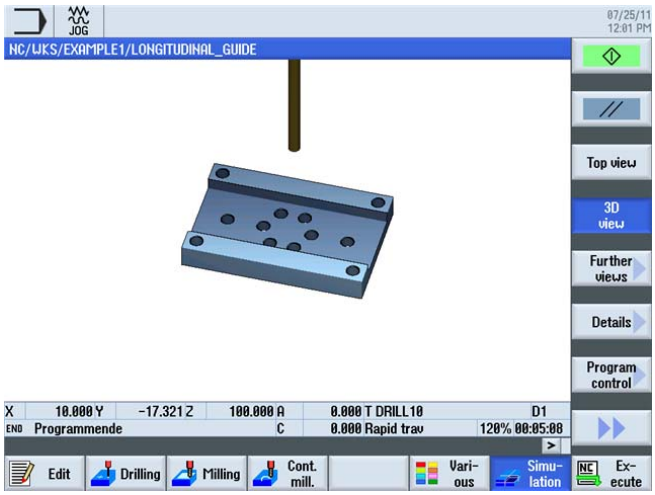


Figure 6-31 3D simulation

Example 2: Injection mold

7

7.1 Overview

Learning objectives

In this section you will learn the following new functions. You will learn how to...

- specify straight lines and circular paths using polar coordinates;
- create rectangular pockets;
- apply circular pockets to position patterns.

Task

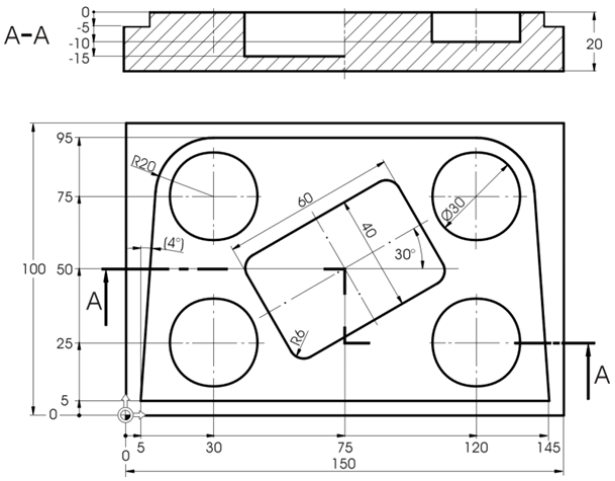


Figure 7-1 Workshop drawing - Example 2:

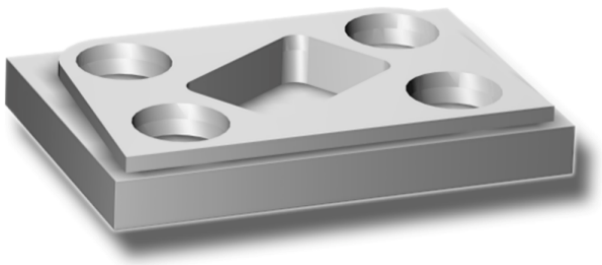


Figure 7-2 Workpiece - Example 2:

Preparation

Perform the following steps without help:

1. Create a new workpiece with the name 'EXAMPLE2'.
2. Create a new step sequence program with the name 'INJECTION_FORM' .
3. Specify the blank dimensions (for the procedure, see example 1).

Note

Observe the new zero position.

4. Switch to the 20mm milling cutter (F = 80 m/min).
5. Position the tool to the point X-12/ X-12/ Z-5 at rapid traverse.
6. Define the starting point of the contour on X5; Y5. The starting point is approached along a straight line (F 100 mm/min, cutter radius compensation left). After you have entered the traversing blocks, your process plan should look like this:

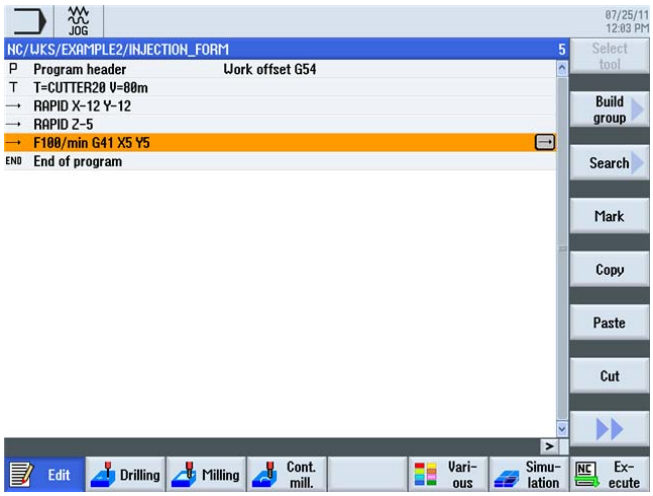


Figure 7-3 Machining step program

7.2 Straight lines and circular paths by way of polar coordinates

Operating sequences

Before you start entering the contour, observe the following note:

Note

You can describe the end point of a traversing block not only by way of its X and Y coordinates, but if necessary also via its polar reference point.

X and Y are not known in our example. However, you can determine the point indirectly: It lies 20 mm away from the center of the circular pocket which highlights the pole here. The polar angle 176° results from the calculation 180° - 4° (see workshop drawing).

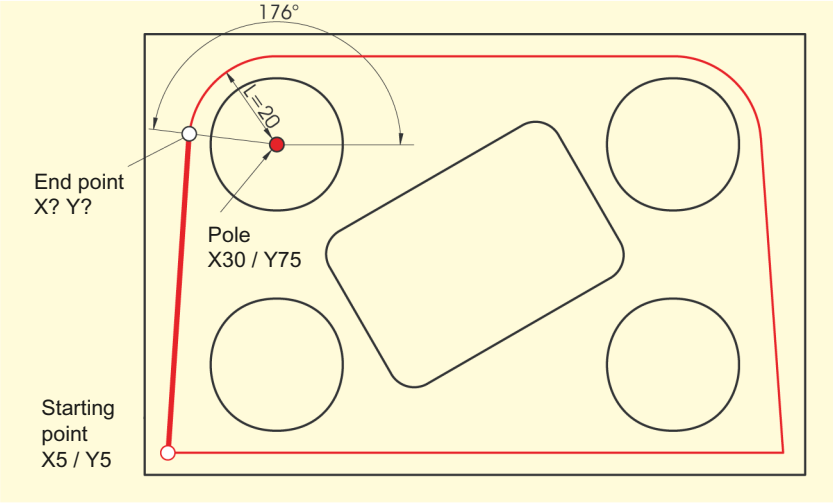


Figure 7-4 Determination of end point and polar angle

Proceed as follows to enter the contour:

Select the **Polar** softkey.

Select the **Pole** softkey.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
X	30 abs	X	
Y	75 abs	X	

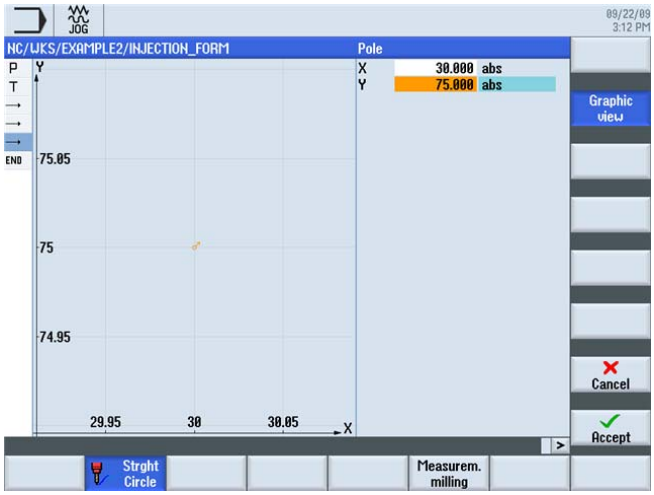


Figure 7-5 Specifying the pole

"Apply" the set values.

Select the **Straight line / polar** softkey.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
L	20		The length L specifies the distance of the end point of the straight line from the pole.
α	176		The polar angle specifies how far the length L must be rotated around the pole to reach the end point of the straight line. You may specify the polar angle either in the counterclockwise (176°) or also in the clockwise direction (-184°).

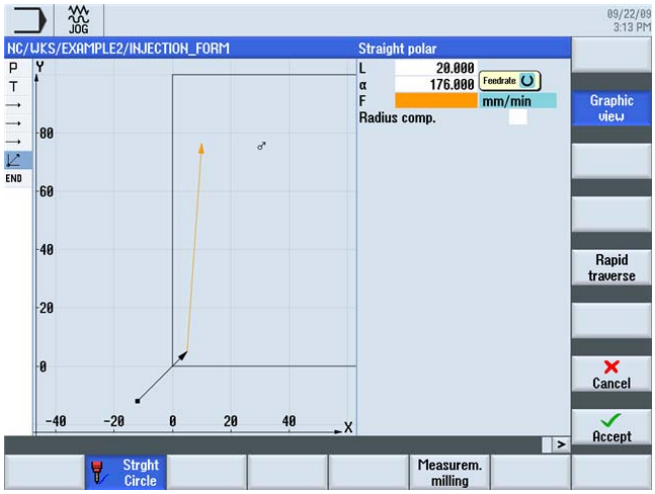


Figure 7-6 Specifying the straight line using polar coordinates

"Apply" the set values.

Select the **Circle / polar** softkey.

A circular path can also be specified using polar coordinates.

Enter the following value in the interactive screenform:

Field	Value	Selection via toggle key	Notes
α	90 abs		Since the pole applies both for the circular path and for the straight line, it need not be entered once more. The polar angle is 90° in this case. (See illustration below)

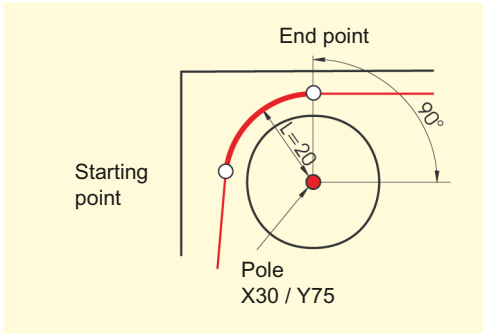


Figure 7-7 Pole starting / end points

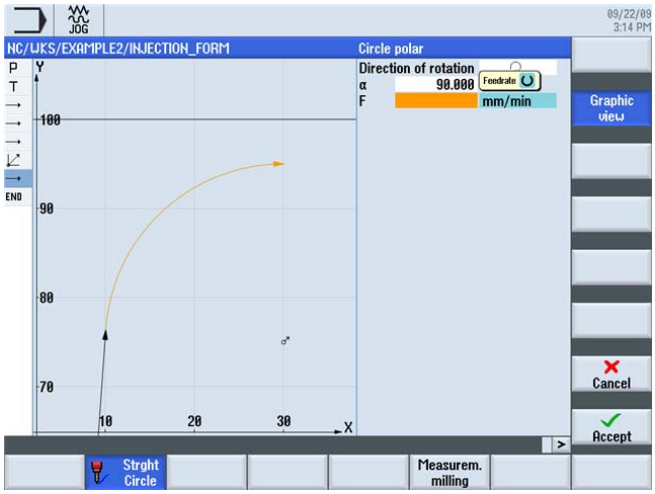


Figure 7-8 Specifying the circular path

"Apply" the set values.

Select the **Back** softkey.

Select the **Straight line** softkey.

Since the end point of the straight line is known unambiguously, you may here use the **Straight line** function.

Enter the following value in the interactive screenform:

Field	Value	Selection via toggle key	Notes
X	120	X	

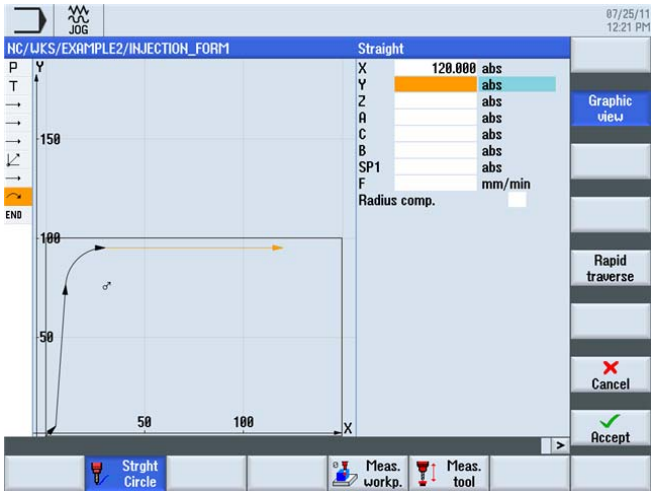


Figure 7-9 Specifying a straight line

"Apply" the set values.

Select the **Polar** softkey.

Select the **Pole** softkey.

Since the end point of the next circular path is not known, you must here work with polar coordinates again.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
X	120 abs	X	The pole of the circular path is known from the drawing.
Y	75 abs	X	

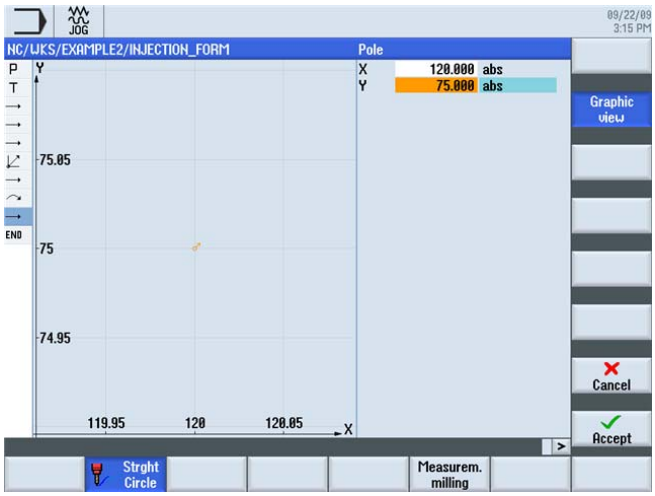


Figure 7-10 Specifying the pole for the circular path

"Apply" the set values.

Select the **Circle / polar** softkey.

Enter the following value in the interactive screenform:

Field	Value	Selection via toggle key	Notes
α	4		The polar angle is also known because of the symmetry.

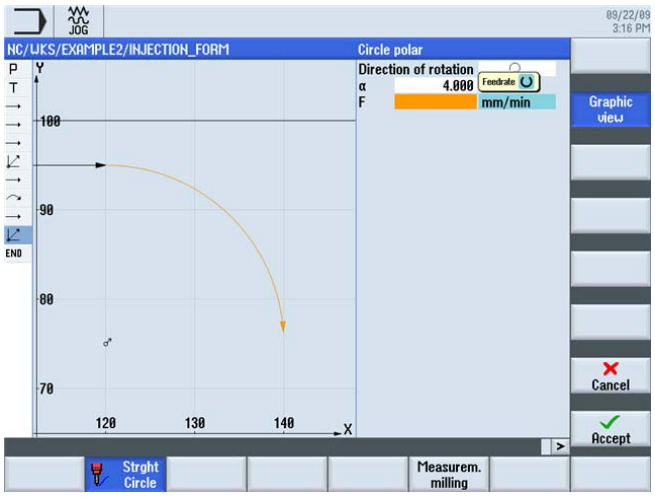


Figure 7-11 Specifying the circular path using polar coordinates

"Apply" the set values.

Select the **Back** softkey.

Select the **Straight line** softkey.

The end point of the straight line is known so that you can enter it directly.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
X	145 abs		
Y	5 abs		

Figure 7-12 Specifying a straight line

"Apply" the set values.

Select the **Straight line** softkey.

The whole contour has been milled once with the last straight line.

Enter the following value in the interactive screenform:

Field	Value	Selection via toggle key	Notes
X	-20 abs	X	

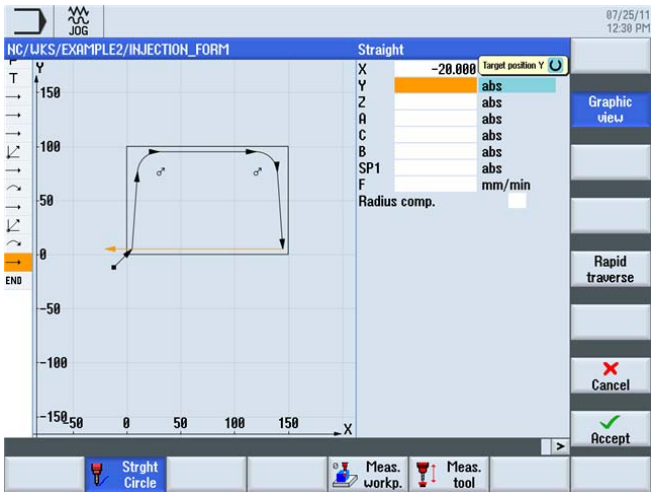


Figure 7-13 Specifying a straight line

"Apply" the set values.

Select the **Straight line** softkey.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
X	-12 abs	X	
Y	-12 abs	X	
Radius compensation	off	X	The last motion is traversing to the safety clearance, disabling the radius compensation.

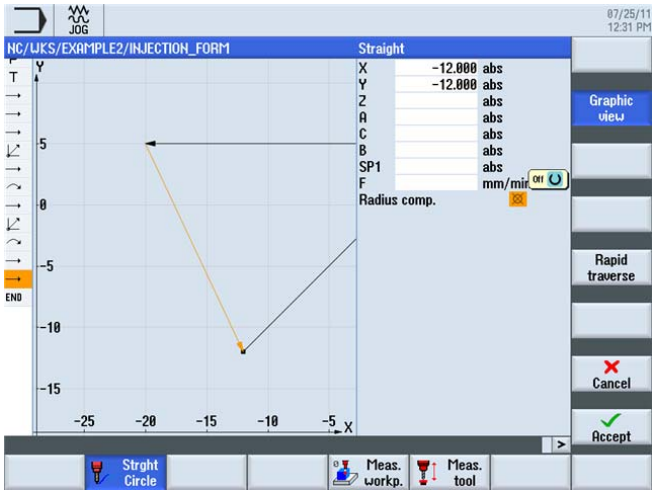


Figure 7-14 Specifying a straight line - Safety clearance

"Apply" the set values.

The following simulation shows the manufacturing sequence for you to check before manufacturing the workpiece.

Figure 7-15 Simulation - Top view

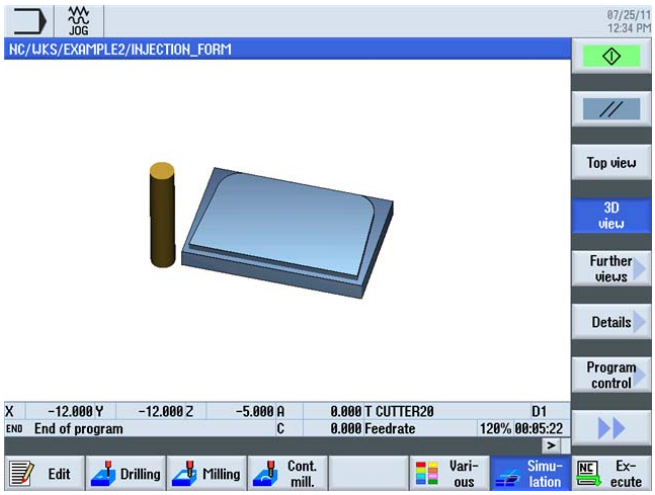


Figure 7-16 3D simulation

7.3 Rectangular pocket

Operating sequences

Proceed as follows to enter the rectangular pocket:

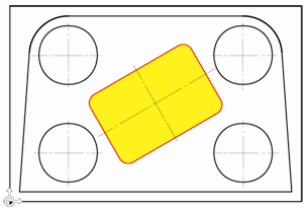


Figure 7-17 Rectangular pocket - Example 2

Select the **Milling** softkey.

Select the **Pocket** softkey.

Select the **Rectangular pocket** softkey.

Open the tool list and select CUTTER10.

Accept the tool into your program.

After accepting the tool, enter the following values:

Field	Value	Selection via toggle key	Notes
F	0.15 mm/tooth	X	
V	120 m/min	X	
Reference point	Center	X	
Machining	Roughing	X	Ensure that the toggle field stands on *Single position*.
X0	75		Specify the geometrical data for the rectangular pocket in these fields: Position, width and length, ...
Y0	50		
Z0	0		
W	40		
L	60		
R	6		
α0	30		
Z1	-15 abs	X	

Field	Value	Selection via toggle key	Notes
DXY	80%	X	The max. infeed in the plane (DXY) specifies at which width the material is removed. This can be specified either as a percentage of the milling diameter or directly in mm. The maximum infeed in the lane is specified in % here.
DZ	2.5		
UXY	0.3		
UZ	0.3		
Insertion	Helical	X	Select "helical insertion" if not yet set (see below *Insertion*).
EP	2 mm/rev	X	
ER	2		

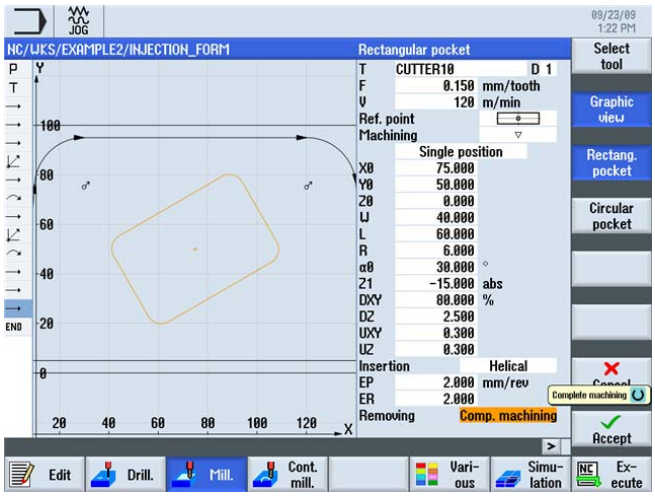


Figure 7-18 Roughing a rectangular pocket

"Apply" the set values.

Select the **Pocket** softkey.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.08 mm/tooth	X	
V	150 m/min	X	
Machining	Finishing	X	Margin and base are finished using these settings. Alternatively, you may also only finish the margin or chamfer the pocket.

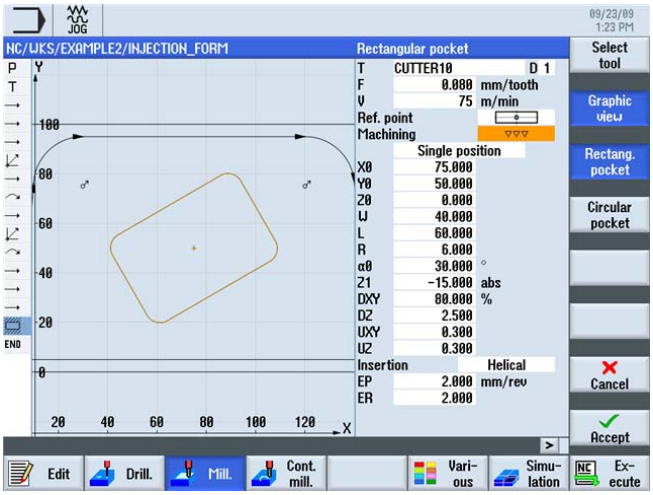
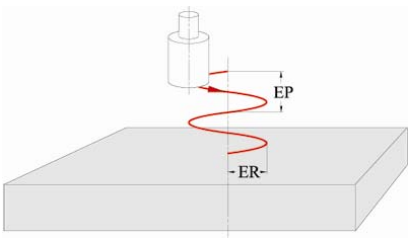
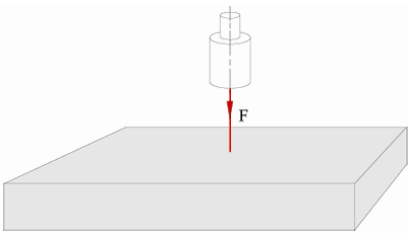
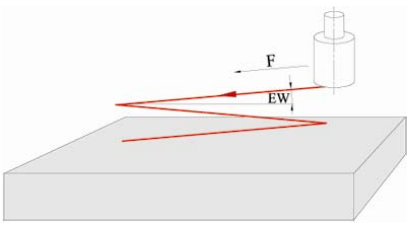


Figure 7-19 Finishing a rectangular pocket

"Apply" the set values.

Insertion

Helical insertion	Vertical insertion	Oscillating insertion
		
EP = insertion pitch ER = insertion radius		EW = insertion angle

7.4 Circular pockets on a position pattern

Operating sequences

Proceed as follows to enter the rectangular pockets:

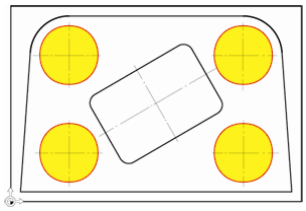
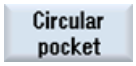


Figure 7-20 Rectangular pocket - Example 2

Select the **Milling** softkey.

Select the **Pocket** softkey.



Select the **Circular pocket** softkey.

Open the tool list and select CUTTER10.

Accept the tool into your program.

After accepting the tool, enter the following values:

Field	Value	Selection via toggle key	Notes
F	0.15 mm/tooth	X	
V	120 m/min	X	
Machining	Roughing	X	
	Position pattern	X	Similar to drilling, you can also apply a position pattern to pockets.
∅	30	X	
Z1	-10 abs	X	
DXY	80 %	X	Specify the maximum infeed in the plane in %.
DZ	5		
UXY	0.3		

Field	Value	Selection via toggle key	Notes
UZ	0.3		
Insertion	Helical	X	
EP	2 mm/rev	X	
ER	2		
Solid machining	Complete machining	X	

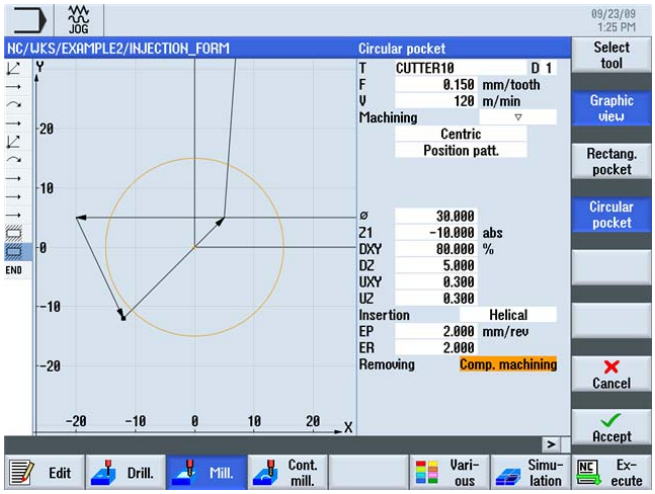
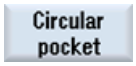


Figure 7-21 Roughing a circular pocket

"Apply" the set values.

Select the **Pocket** softkey.



Select the **Circular pocket** softkey.

Enter the following values:

Field	Value	Selection via toggle key	Notes
F	0.08 mm/tooth	X	
V	150 m/min	X	
Machining	Finishing	X	

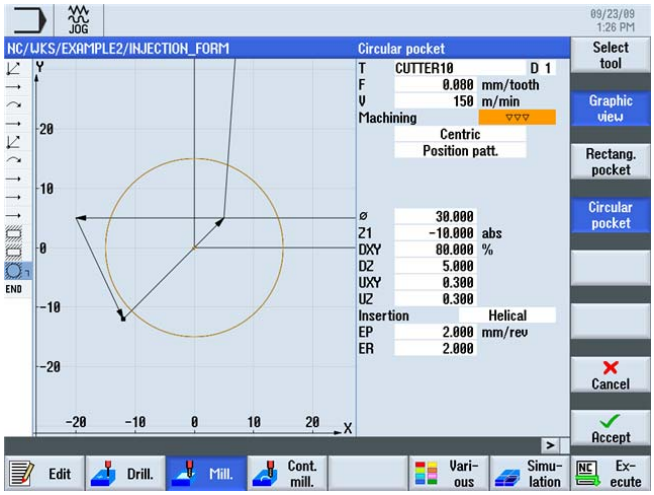


Figure 7-22 Finishing a circular pocket

"Apply" the set values.

Select the **Drilling** softkey.

Select the **Positions** softkey.

Select the **Position pattern** softkey.

Enter the following values:

Field	Value	Selection via toggle key	Notes
Pattern	Matrix	X	Position patterns are described in the **Drilling** menu with the **Positions** submenu (independent of the machining method).
X0	30 abs		
Y0	25 abs		
α0	0		
L1	90		
L2	50		
N1	2		
N2	2		

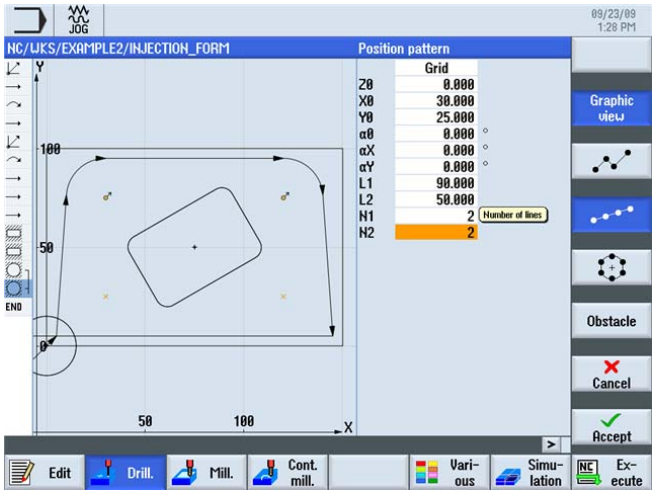


Figure 7-23 Positions of the circular pockets

"Apply" the set values.

Start simulation.

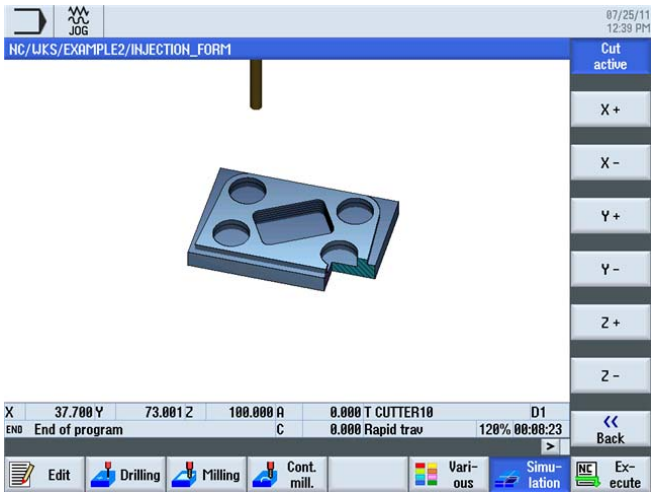


Figure 7-24 Simulation - Cut active

Example 3: Mold plate

8

8.1 Overview

Learning objectives

In this section you will learn the following new functions, in particular the contour calculator. You will learn how to...

- mill open contours;
- remove contour pockets from the solid, machine residual material and finish;
- apply machining methods on several planes;
- take into account obstacles.

Task

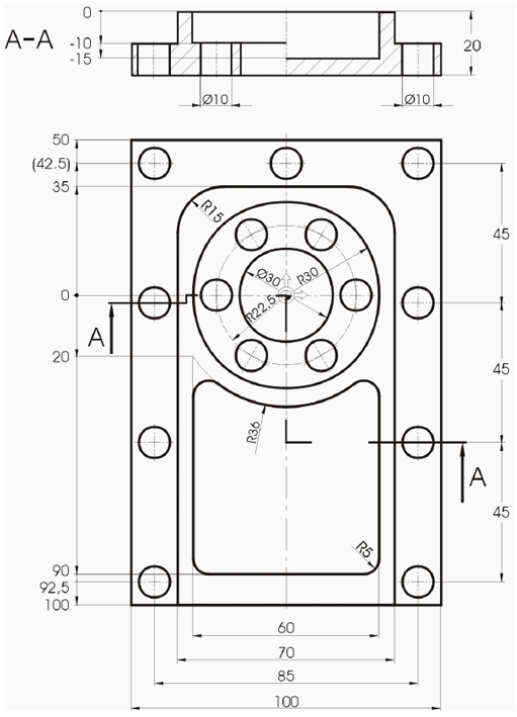


Figure 8-1 Workshop drawing - Example 3:

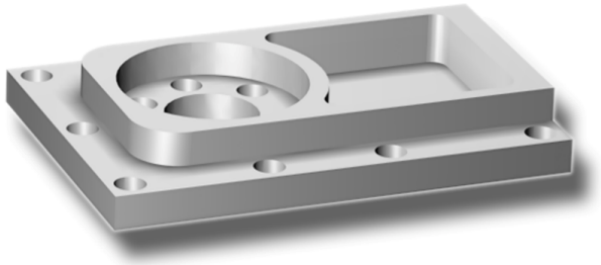


Figure 8-2 Workpiece - Example 3:

Preparation

Perform the following steps without help:

1. Create a new workpiece with the name 'Example3'.
2. Create a new process plan with the name 'MOLD_PLATE' .
3. Specify the blank dimensions (for the procedure, see example 1).

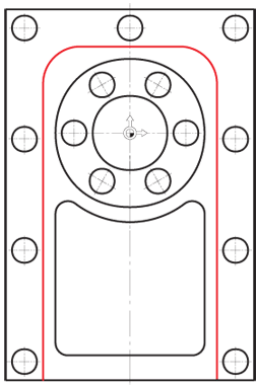
Note

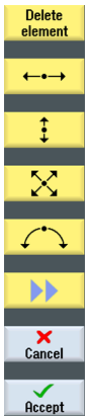
Observe the new zero position.

8.2 Path milling of open contours

Contour calculator

With the contour calculator integrated into ShopMill for entering complex contours, you can enter even the most complicated contours easily.





With the graphical contour calculator, you can enter the contours faster and more easily than with conventional programming - and even without any mathematical knowledge.

Operating sequences

Proceed as follows to enter the contour:

Select the **Contour milling** softkey.

Select the **New contour** softkey. Type the name MOLD_PLATE_Outside for the contour.

Each contour is assigned its own name. This provides for better legibility of the programs.

Figure 8-3 Creating the 'MOLD_PLATE_Outside' contour

"Apply" your input.

In the interactive screenform, enter the following values for the starting point of the contour line:

Field	Value	Selection via toggle key	Notes
X	-35		The starting point for construction is also the starting point for later machining of the contour.
Y	-100		

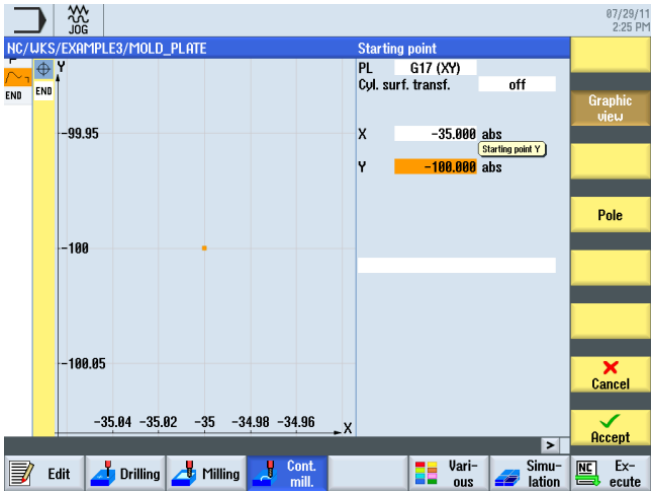


Figure 8-4 Specifying the starting point

Note

Here you only describe the workpiece contour; the approach and retraction travels will only be defined later.

"Apply" the set values.

Enter the following values for the straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Y	35 abs	X	The first contour element is a vertical straight line with the end point at Y=20. You can specify the subsequent circle contour very easily in this dialog - as a transition element to the next straight line. Therefore, the theoretical end point of the straight line lies at Y=35.
Transition to next element	Radius	X	
R	15		

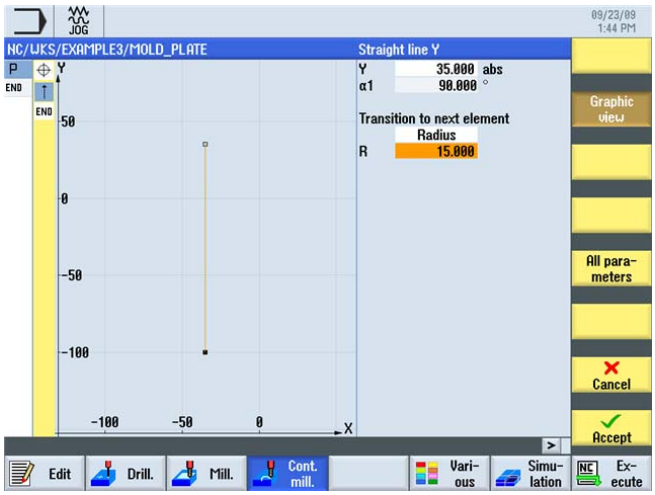


Figure 8-5 Specifying the vertical contour straight line

"Apply" the set values.

Enter the following values for the horizontal straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
X	35 abs	X	
R	15		The radius is entered as a rounding again.

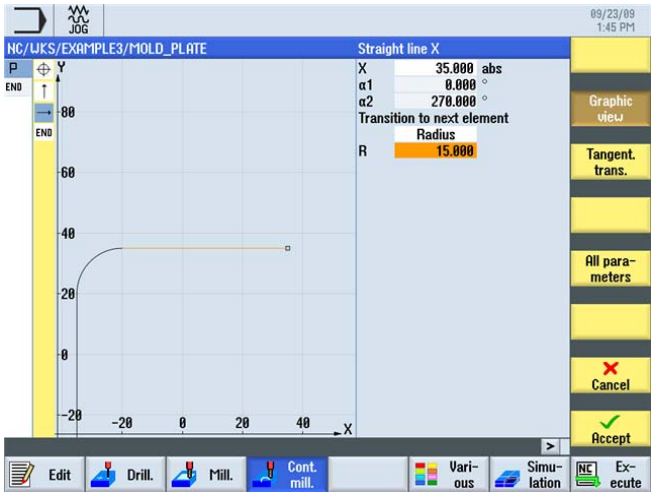


Figure 8-6 Specifying the horizontal contour straight line

"Apply" the set values.

Enter the following values for the vertical straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Y	-100 abs	X	

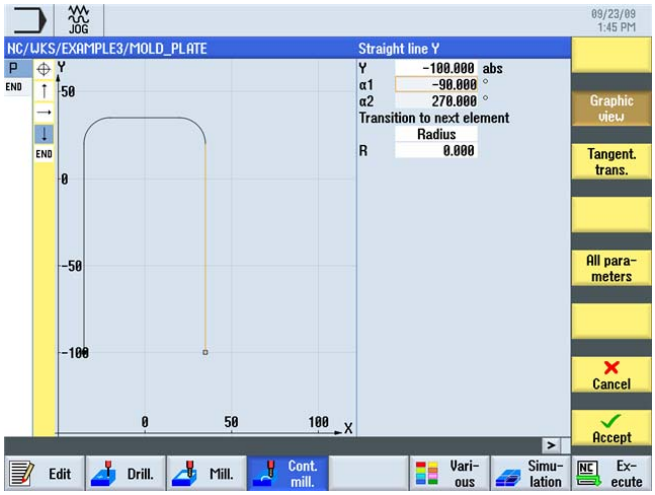


Figure 8-7 Specifying the vertical contour straight line

"Apply" the set values.

Accept the contour into your process plan.

To be able to machine the created contour, you must now create the following work steps. To this end, proceed as follows:

Select the **Path milling** softkey.

Open the tool list and select CUTTER32.

Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.15 mm/tooth	X	
V	120 m/min	X	
Machining	Roughing forward	X X	With ShopMill V6.4 and higher, you may also mill reverse against the engineering direction.
Radius compensation	Left	X	The tool is to traverse to the left of the contour.
Z0	0		
Z1	10 inc	X	Switch the depth *Z1* to "inc". This provides the advantage that in all cases only the actual depth of the pocket can be entered. This makes input easier for you, in particular with nested pockets.
DZ	5		
UZ	0.3		
UXY	0.3		
Approach	Straight line	X	Approaching can be performed either along a quarter, semicircle, vertically or along a straight line. In this case, it is reasonable to approach the contour tangentially along a straight line.
L1	5		The cutter radius need not be taken into account in the approach length *L1*; it is calculated by ShopMill automatically.
FZ	0.1 mm/tooth	X	
Retraction	Straight line	X	
L2	5		
Lift mode	To retraction plane	X	

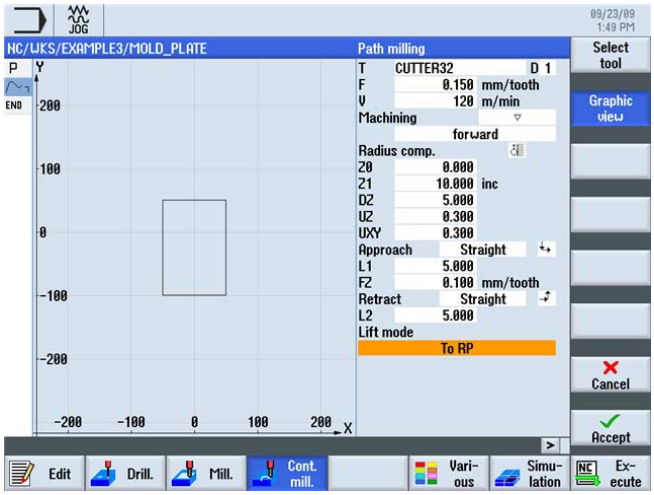


Figure 8-8 Roughing the contour

Accept "Apply" the set values.

Path milling Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.08 mm/tooth	X	
V	150 m/min	X	
Machining	Finishing		

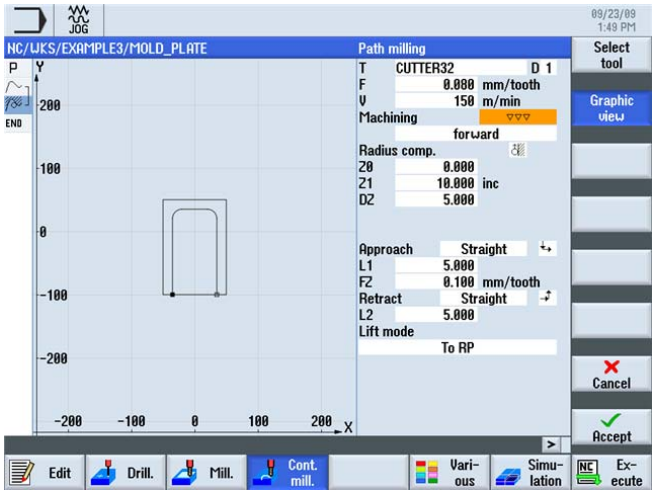


Figure 8-9 Finishing the contour

Accept "Apply" the set values.

The two machining steps are linked in the work step editor.

Figure 8-10 Linking of the work steps in the process plan

The following simulation shows the manufacturing sequence for you to check before manufacturing the workpiece.

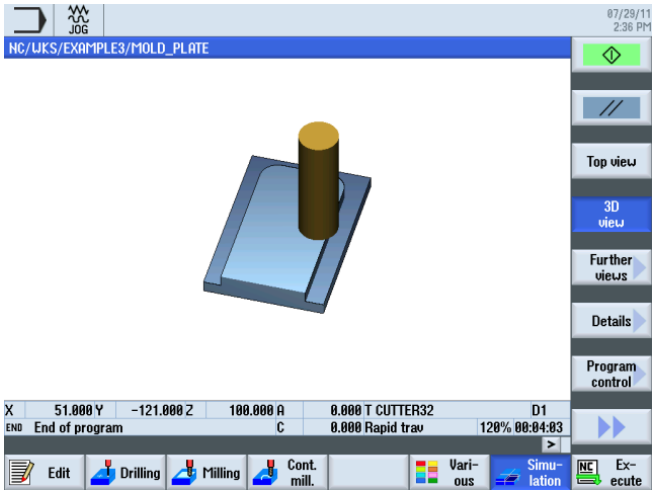


Figure 8-11 Simulation - External contour

8.3 Solid machining and residual material; finishing of contour pockets

Operating sequences

Proceed as follows to enter the pocket contour: Remove the pocket from the solid and finish.

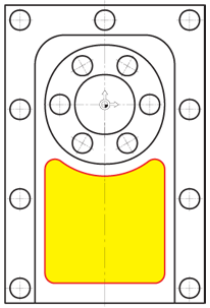


Figure 8-12 Pocket contour

Select the **Contour milling** softkey.

Select the **New contour** softkey. Type the name 'MOLD_PLATE_Inside' for the contour.

Figure 8-13 Creating the 'MOLD_PLATE_Inside' contour

"Apply" your input.

Enter the following values for the starting point in the interactive screenform:

Field	Value	Selection via toggle key	Notes
X	0 abs		
Y	-90 abs		

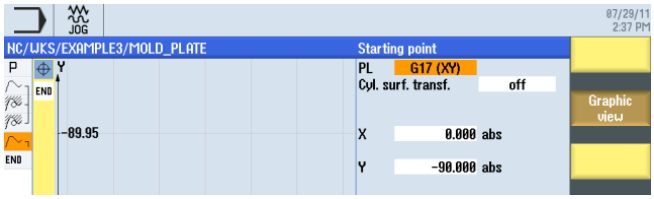


Figure 8-14 Specifying the starting point

"Apply" the set values.

Enter the following values for the horizontal straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
X	25 abs	X	Within the framework of this exercise, do not specify the arc as a rounding, but as a separate element. Therefore, design the straight line only up to X25.

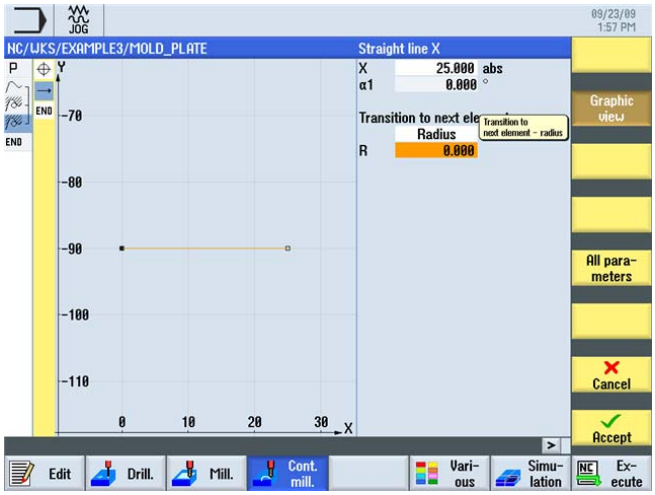


Figure 8-15 Specifying the horizontal contour straight line

"Apply" the set values.

Enter the following values for the arc in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Direction of rotation	Left	X	
R	5		
X	30 abs	X	
Y	-85 abs	X	

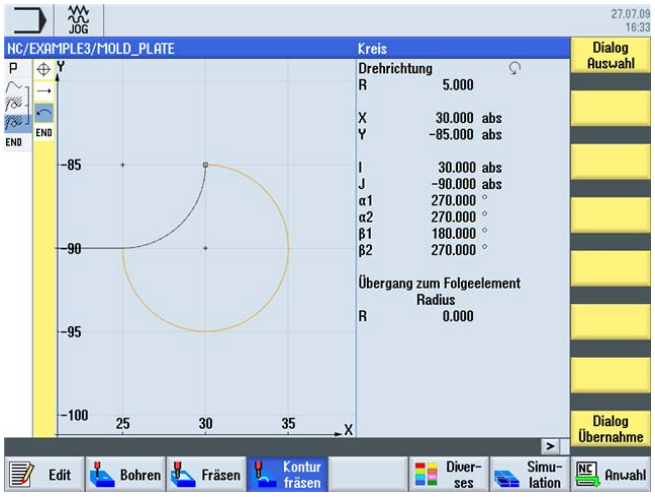


Figure 8-16 Arc contour (bottom right)

Dialog select

Two design solutions result after entering the Y end point. Select the desired solution using the **Select dialog** softkey. Subsequently, the selected solutions turns to orange, and the alternative solution is displayed with black points.

Dialog accept

Accept your selection. The geometry processor automatically detects that the programmed arc is connected tangentially to the straight line. The **Tangent to prec.elem.** is displayed inversely (i.e. held down).

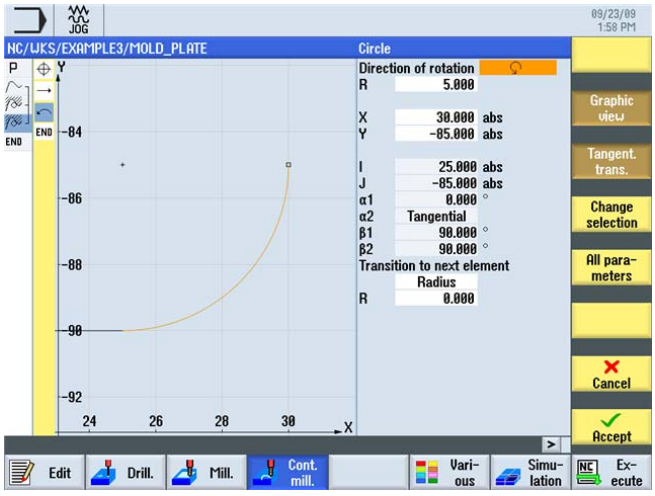


Figure 8-17 Arc contour - after selection

"Apply" the set values.

Enter the following values for the straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Y	-20 abs	X	Enter the end point of the straight line. The transition to R36 is rounded with R5.
Transition to next element	Radius 5	X	

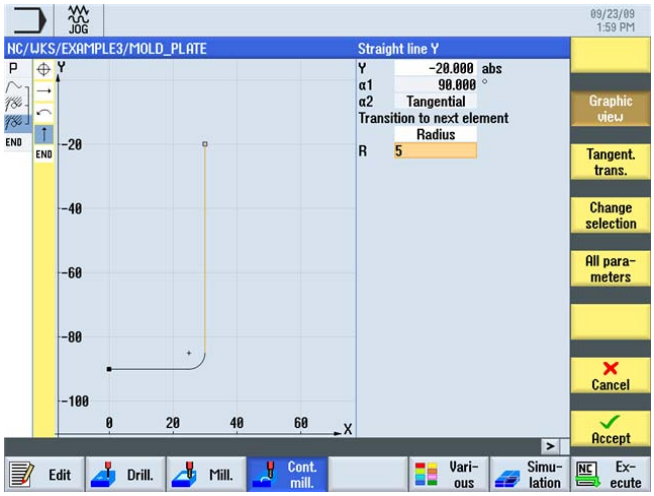


Figure 8-18 Specifying the vertical contour straight line

"Apply" the set values.

Enter the following values for the arc in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Direction of rotation	Right	X	
R	36		
X	-30 abs	X	
Y	-20 abs	X	
Transition to next element	Radius 5	X	

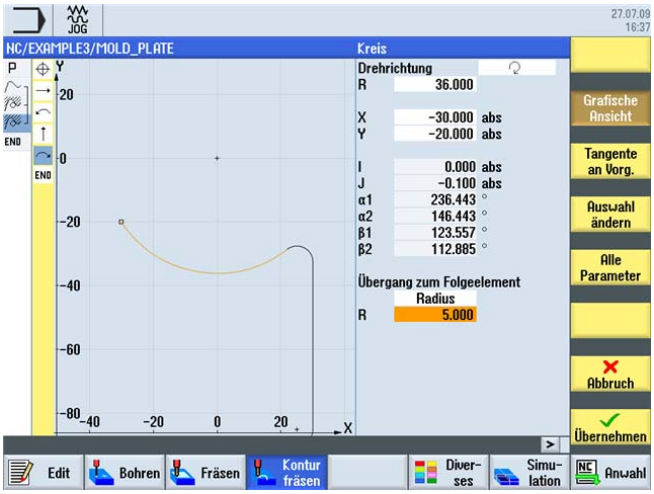


Figure 8-19 Specifying the contour arc

"Apply" the set values.

Enter the following values for the straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Y	-90 abs	X	
Transition to next element	Radius 5	X	Specify the radius R5 as a rounding.

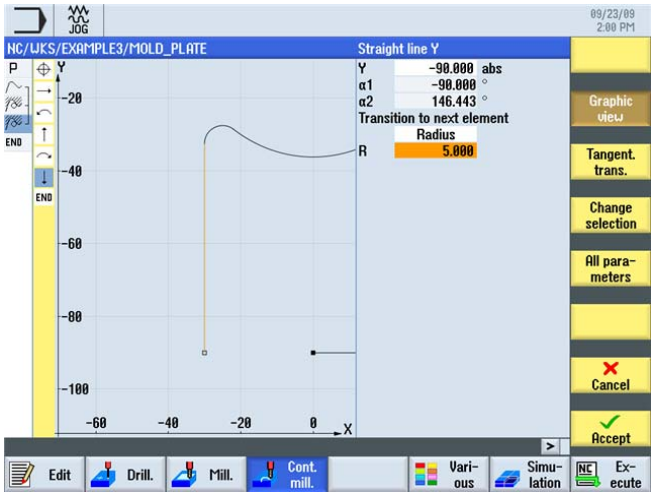


Figure 8-20 Specifying the vertical contour straight line

"Apply" the set values.

Close the contour. Thus, the pocket contour is described completely.

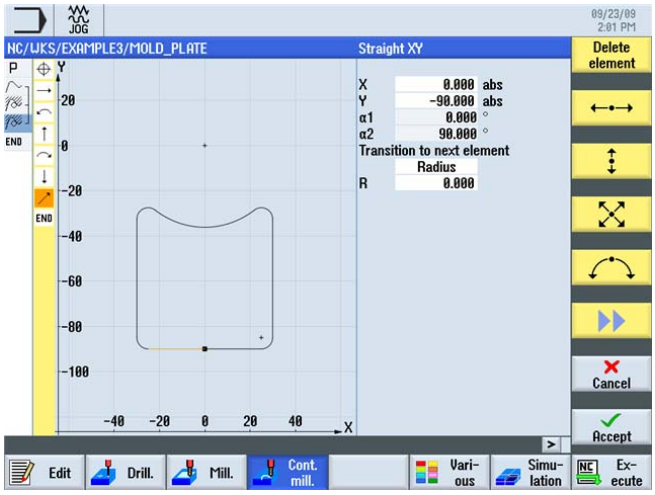


Figure 8-21 Closing the contour

Accept the contour into your process plan.

Select the **Pocket** softkey.

Open the tool list and select CUTTER20.

Accept the tool into your program.

Note

The manufacturing direction of the pocket has already been defined in the program header. The "Synchronous" setting was selected in this case.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.15 mm/tooth	X	
V	120 m/min	X	
Machining	Roughing	X	
Z0	0		
Z1	15 inc	X	If you specify the machining depth with *incremental* , you must specify a positive value for the depth.
DXY	50%	X	
DZ	5		
UXY	0.3		
UZ	0.3		
Starting point	Automatic	X	If you select the *Autom* setting for the starting point (insertion), the starting point is specified by ShopMill.
Insertion	Helical	X	Set insertion to *Helical* with 2mm for both pitch and radius.
EP	2 mm/rev	X	
ER	2		
Lift mode	To retraction plane	X	

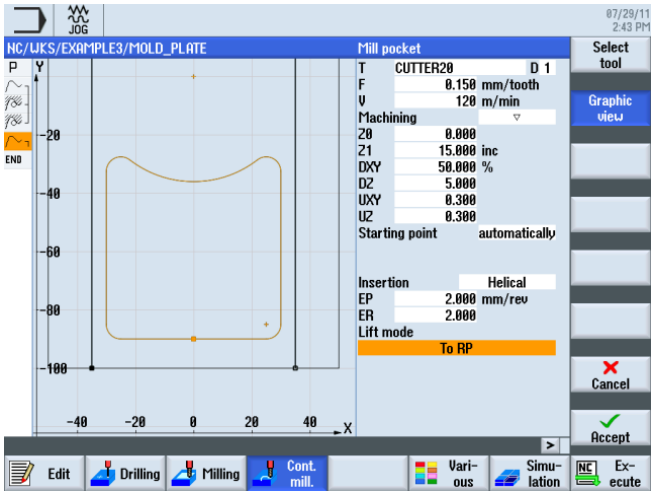


Figure 8-22 Roughing a pocket

"Apply" the set values.

Select the **Pocket Resid.mat.** softkey. As the 20mm cutter cannot machine R5 radii, material will remain in the corners. Use the **Pocket Resid. mat.** to remove areas not yet machined by roughing with pinpoint accuracy.

Open the tool list and select CUTTER10.

Accept the tool into your program.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.1 mm/tooth	X	
V	120 m/min	X	
Machining	Roughing	X	
DXY	50%		The maximum infeed in the plane must be 50 %.
DZ	5		

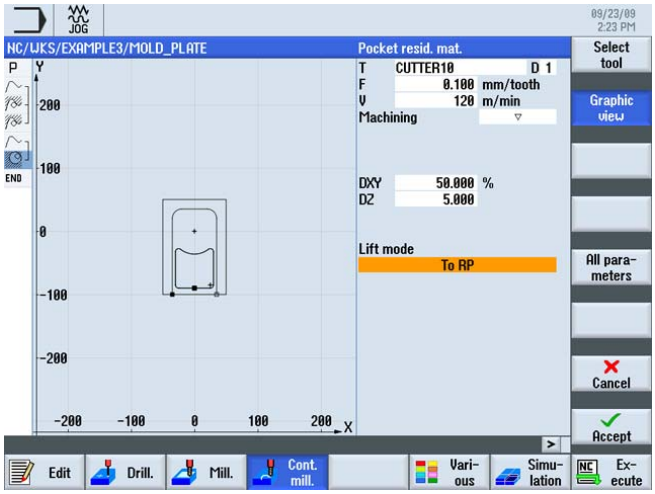


Figure 8-23 Machining residual material of the pocket

"Apply" the set values.

Select the **Pocket** softkey.

Open the tool list and select CUTTER10.

Accept the tool into your program.

Enter the following values for the reworking the pocket in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.08 mm/tooth	X	
V	150 m/min	X	
Machining	Base	X	
UXY			The allowance which you have previously entered for roughing must remain set for the values in the "Finishing allowance fields in the plane (UXY)" and "Finishing allowance in the depth (UZ)" fields. This value is important for automatic calculation of the distances to be traversed.
UZ			

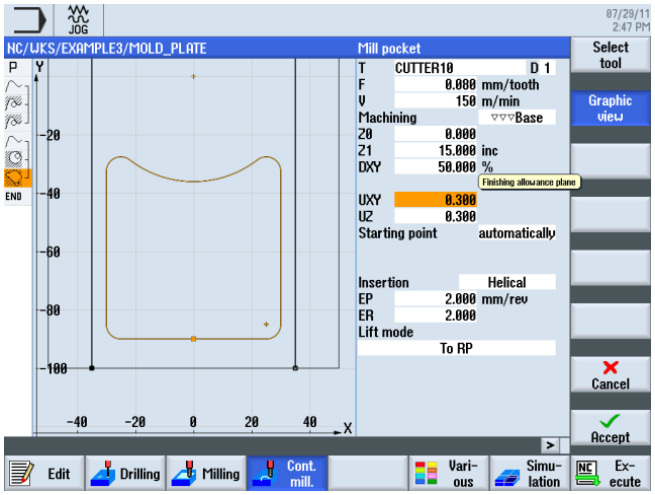


Figure 8-24 Finishing a pocket

"Apply" the set values.

Select the **Pocket** softkey.

In the interactive screenform, specify the following value for removing the residual material of the contour:

Field	Value	Selection via toggle key	Notes
Machining	Edge	X	

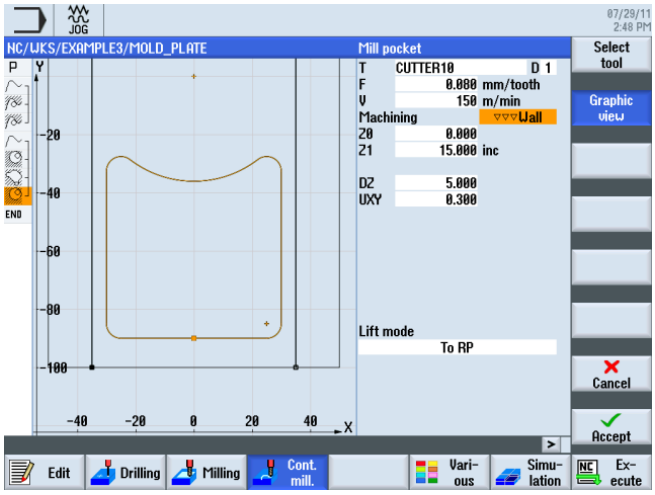


Figure 8-25 Finishing the edge

"Apply" the set values.

8.4 Machining on several planes

Operating sequences

Mill a 60mm circular pocket in two work steps as described in the example 'INJECTION_FORM' .

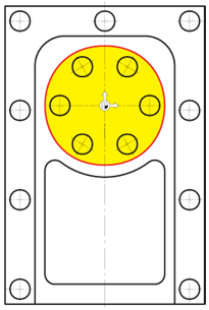


Figure 8-26 Circular pocket

1. In the first work step, the pocket is machined by roughing up to -9.7 mm using a 20 mm cutter.

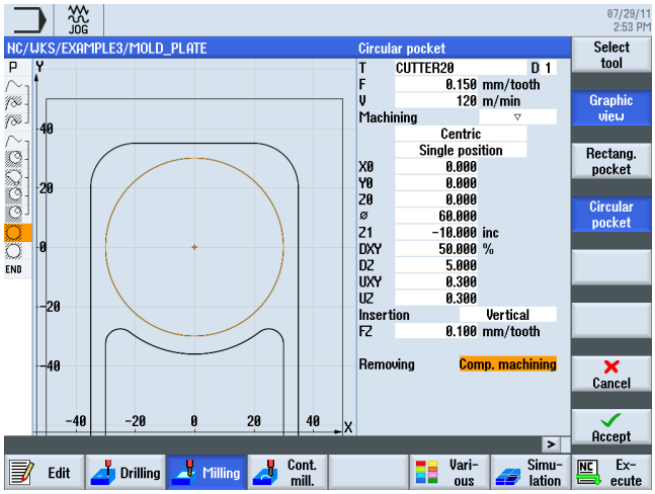


Figure 8-27 Roughing a circular pocket

2. In the second work step, the pocket is finished using the same tool.

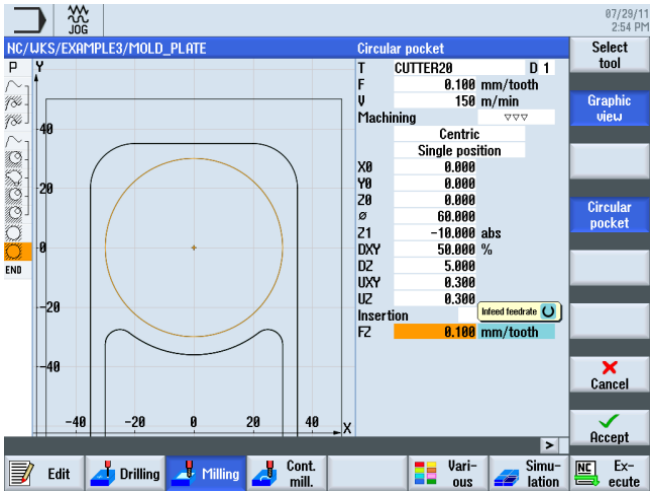


Figure 8-28 Finishing a circular pocket

To specify how the inside circular pocket is machined, proceed as follows: Machine the circular pocket down to a depth of -20 mm.

Note

Now the starting depth is no longer at 0 mm, but at -10 mm.

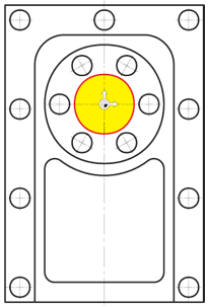


Figure 8-29 Inside circular pocket

Select the **Milling** softkey.

Select the **Pocket** softkey.

Circular pocket

Enter the following values for machining of the circular pocket in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.15 mm/tooth	X	
V	120 m/min	X	
Machining	Roughing	X	
X0	0		
Y0	0		
Z0	-10		
∅	30		
Z1	-20 abs	X	
DXY	50%	X	
DZ	5		
UXY	0.3		
UZ	0.3		
Insertion	Vertical	X	
FZ	0.1 mm/tooth	X	

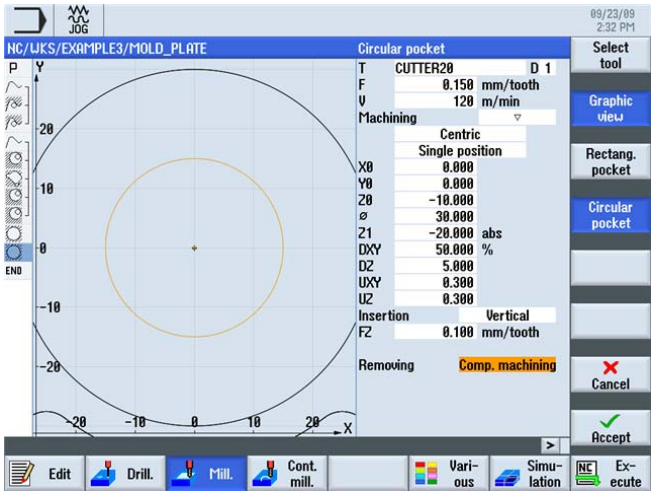


Figure 8-30 Roughing the inside circular pocket

Accept

"Apply" the set values.

Mill.

Select the **Milling** softkey.

Pocket

Select the **Pocket** softkey.

Circular pocket

Enter the following values for machining of the circular pocket in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.08 mm/tooth	X	
V	150 m/min	X	

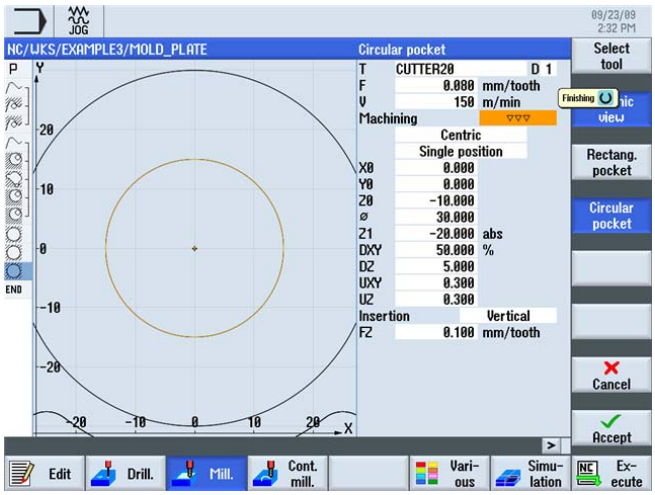


Figure 8-31 Finishing the inside circular pocket

Accept

"Apply" the set values.

Simu- lation

Start the simulation.

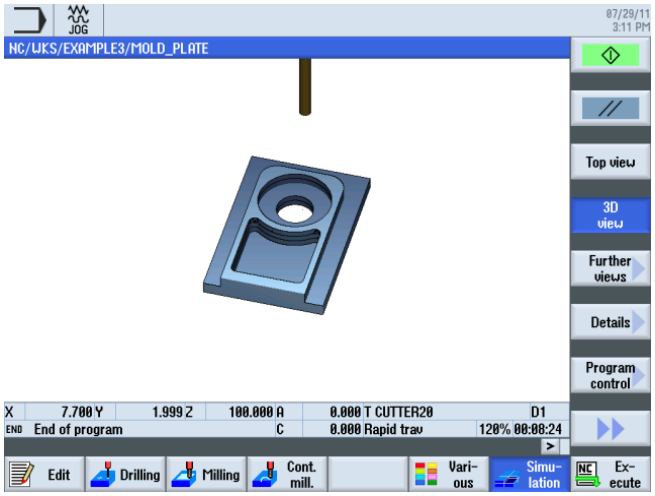


Figure 8-32 Simulation in 3D display

8.5 Taking into account obstacles

Operating sequences

As you have already seen in example 1, different drill patterns can also be interlinked in the case of this workpiece. However, you should take into account that one or several obstacles must be bypassed - depending on the sequence of machining. Traverse either to *safety clearance* or to the *machining plane* between the drill holes - depending on the settings you have made.

First create the work steps 'Centering' and 'Drilling' as done in example 1.

1. Centering

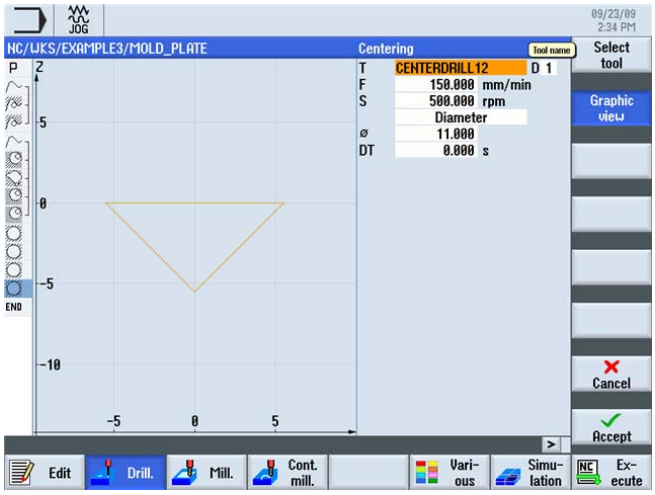


Figure 8-33 Work step 'Centering'

2. Drilling

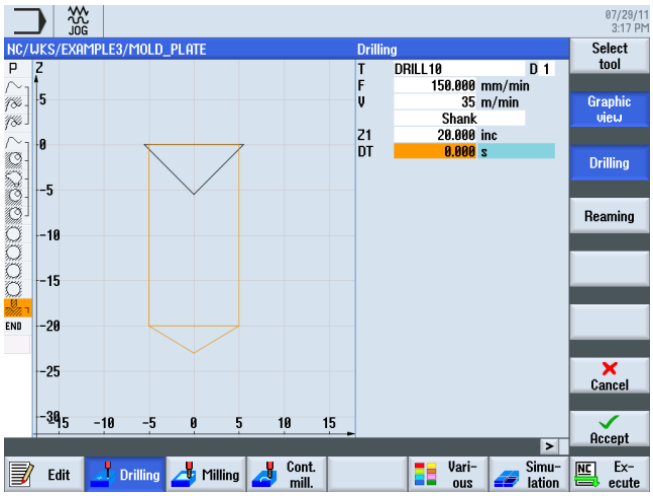


Figure 8-34 Work step 'Drilling'

Proceed as follows to enter the relevant drilling positions:

Select the **Positions** softkey.

First create the left line of holes in the sequence from bottom to top.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Pattern	Line	X	
Z0	-10		
X0	-42.5		
Y0	-92.5		
α0	90		
L0	0		
L	45		
N	4		

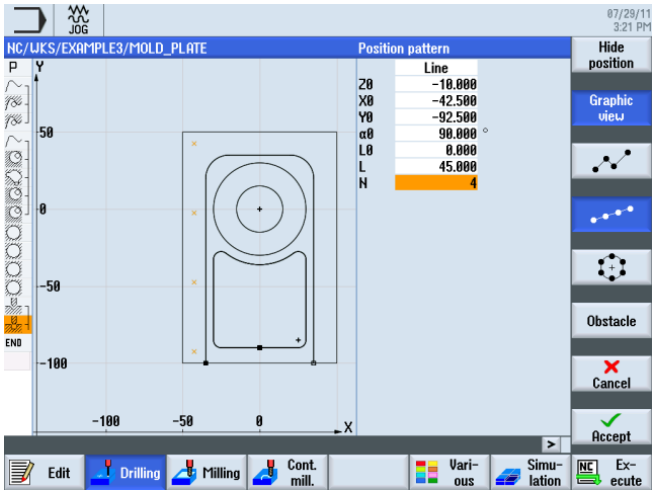


Figure 8-35 Specifying the line of holes

"Apply" the set values.

Select the **Positions** softkey.

Use the "Obstacle" function to specify a distance of 1 mm to be traversed, as the right-hand line of holes must also be drilled from bottom to top for the purposes of an exercise. The obstacle only needs to be entered if you have first switched the "Retraction position pattern" toggle field to "Optimized".

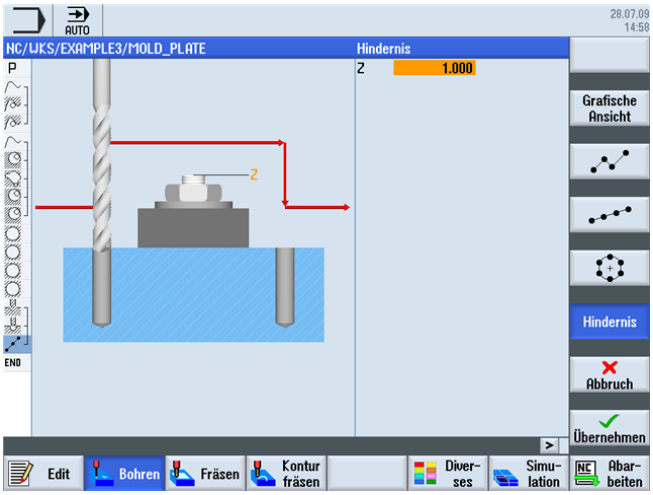
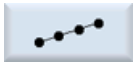


Figure 8-36 Specifying an obstacle

"Apply" the set values.

Select the **Positions** softkey.



Enter the following values for the second line of holes in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Pattern	Line	X	
Z0	-10		
X0	42.5		
Y0	-92.5		
α0	90		
L0	0		
L	45		
N	4		

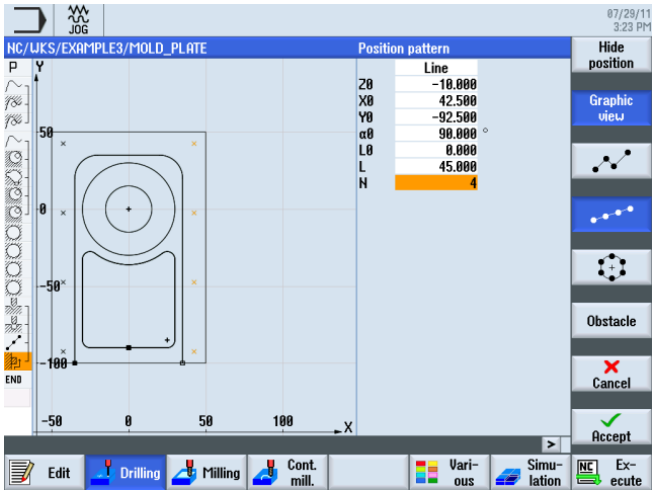


Figure 8-37 Specifying the line of holes

"Apply" the set values.

Select the **Positions** softkey.

To get to the next drill pattern - the circle of holes -, another obstacle must be bypassed.
Enter Z=1.

"Apply" the set value.

Select the **Positions** softkey.

In the interactive screenform, enter the following values for the 6 drill holes in the full circle:

Field	Value	Selection via toggle key	Notes
Pattern	Full circle	X	
Z0	-10		
X0	0		
Y0	0		
α0	0		
R	22.5		
N	6		
Positioning	Straight line	X	

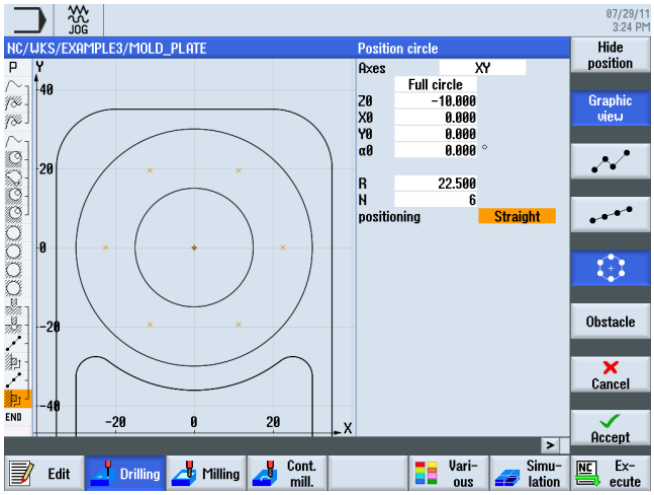


Figure 8-38 Specifying the drill holes in the full circle

"Apply" the set values.

Select the **Positions** softkey.

To make the last drill hole, another obstacle must be bypassed. Enter Z=1.

"Apply" the set value.

Select the **Positions** softkey.

Enter the following values for the last drilling positions in the interactive screenform:

Note

If necessary delete any existing positions using the DEL key.

Field	Value	Selection via toggle key	Notes
Pattern	Rectangular	X	
Z0	-10		
X0	0		
Y0	42.5		

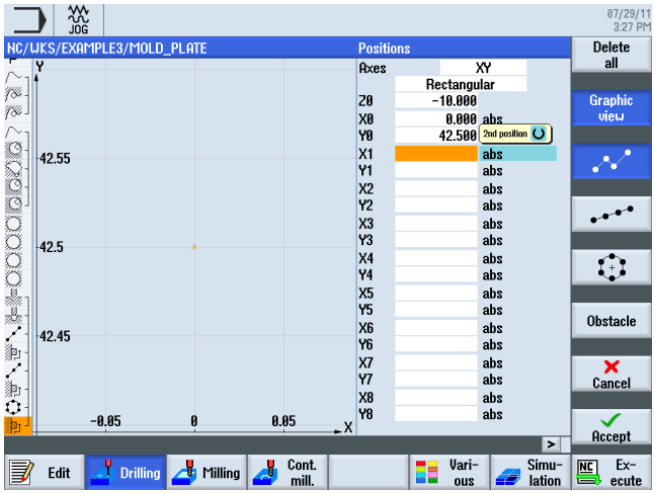


Figure 8-39 Specifying the drilling positions

"Apply" the set values.

Note

This programming example is intended to familiarize you with the "Obstacle" function. There are naturally more elegant methods of programming drilling positions, including only one obstacle.
Try out different strategies and decide which is the best for you.

Start simulation.

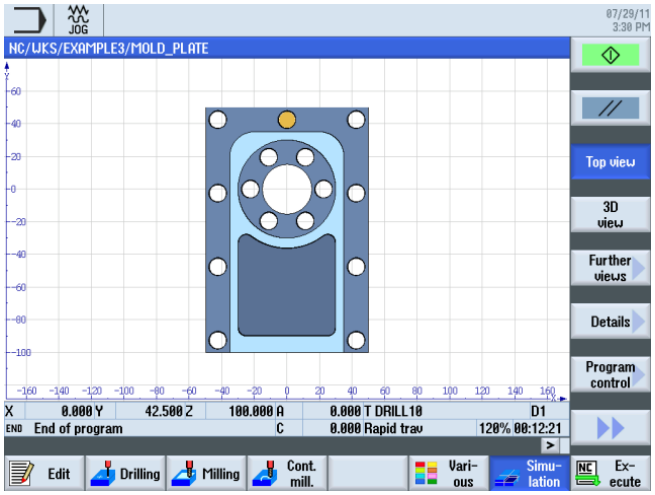


Figure 8-40 Simulation - Top view

Example 4: Lever

9

9.1 Overview

Learning objectives

In this section you will learn the following new functions. You will learn how to...

- perform face milling;
- create edges (auxiliary pockets) for removing material from the solid around islands;
- create and copy circular islands;
- work with the work step editor and machine islands;
- perform deep-hole drilling, helix milling, boring and thread milling;
- program contours using polar coordinates (version 6.4 and higher).

Task

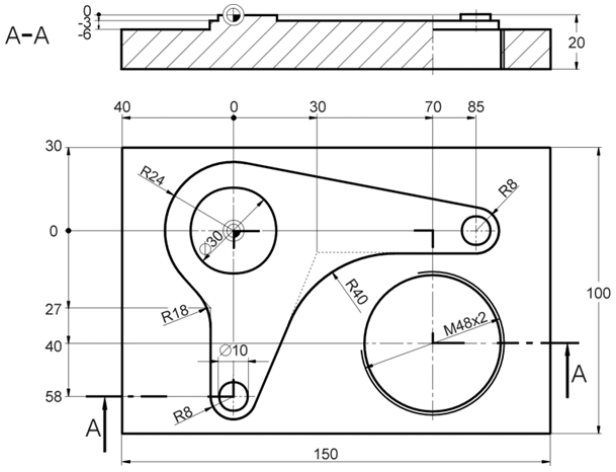


Figure 9-1 Workshop drawing - Example 4:

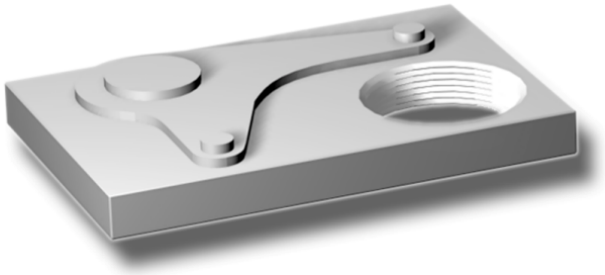


Figure 9-2 Workpiece - Example 4:

Preparation

Perform the following steps without help:

1. Create a new workpiece with the name 'Example4'.
2. Create a new process plan with the name 'LEVER' .
3. Specify the blank dimensions (for the procedure, see example 1).

Note

The thickness of the blank will be 25 mm; therefore, it is imperative to set ZA to 5 mm.

After input of the data, the program header should look as in the screen below.

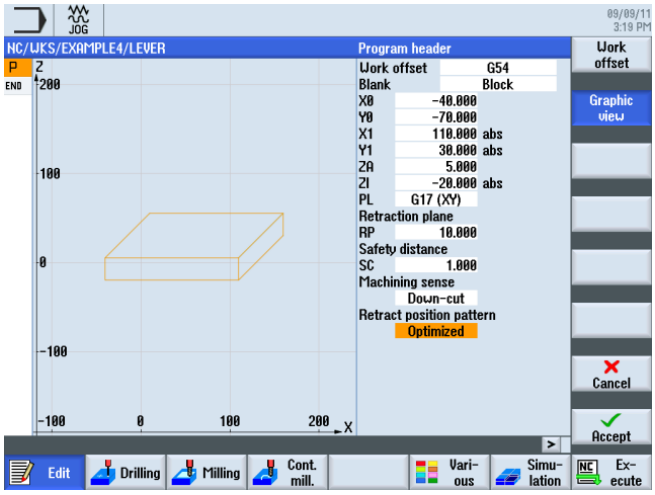


Figure 9-3 Workpiece dimensions as in the program header

9.2 Face milling

Operating sequences

Select the **Milling** softkey.

Select the **Face milling** softkey.

Open the tool list and select FACEMILL63.

Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.1 mm/tooth	X	
V	120 m/min	X	
Machining	Roughing	X	
Direction	Alternating	X	
X0	-40		
Y0	-70		
Z0	5		
X1	110 abs	X	
Y1	30 abs	X	
Z1	0 abs	X	
DXY	30 %	X	
DZ	5		
UZ	1		

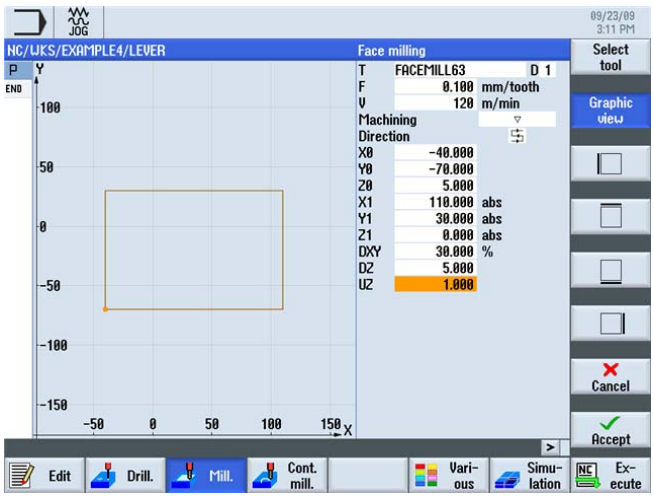


Figure 9-4 Roughing a surface

"Apply" the set values.

Select the **Face milling** softkey.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.08 mm/tooth	X	
V	150 m/min	X	
Machining	Finishing	X	

Note

The values for the finishing allowance must be identical for both roughing and finishing, as this value specifies the allowance for the subsequent machining by finishing, and then when finishing, the thickness of the material to be removed is meant.

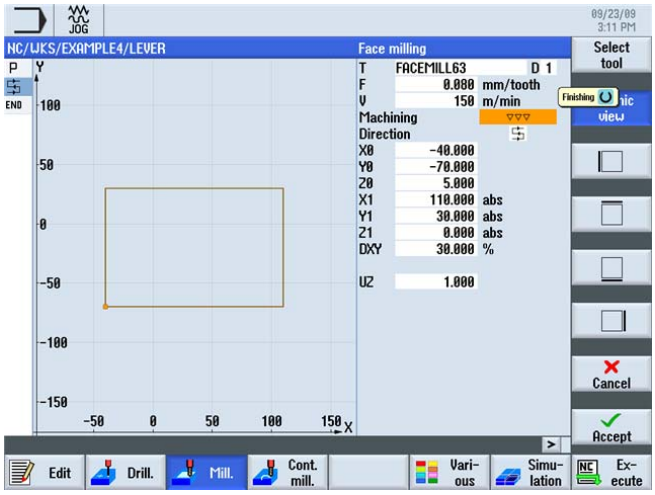


Figure 9-5 Finishing a surface

"Apply" the set values.

9.3 Creating an edge for the lever island

Operating sequences

Note

Islands are described - like pockets - as a contour in the graphical contour calculator. They only become islands through linking in the process plan. The first contour in the process plan always describes the pocket. One or several of the subsequent contours are interpreted as islands.

Since no pocket exists in the case of the 'LEVER' example, you must create a fictitious auxiliary pocket around the external contour. This serves as a necessary outside boundary for the traversing paths and thus forms a frame in which the tool motions take place.

Select the **Contour milling** softkey.

Create a new contour with the name 'LEVER_Rectangular_Area'.

Figure 9-6 Creating the contour

Create the following contour without help. Round the corners with R15. Select such values that the workpiece corners are covered by the pocket.

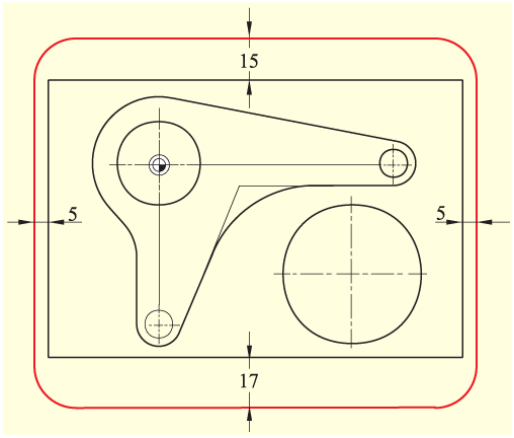


Figure 9-7 Edge for the lever island

Compare your contour with the screen below.

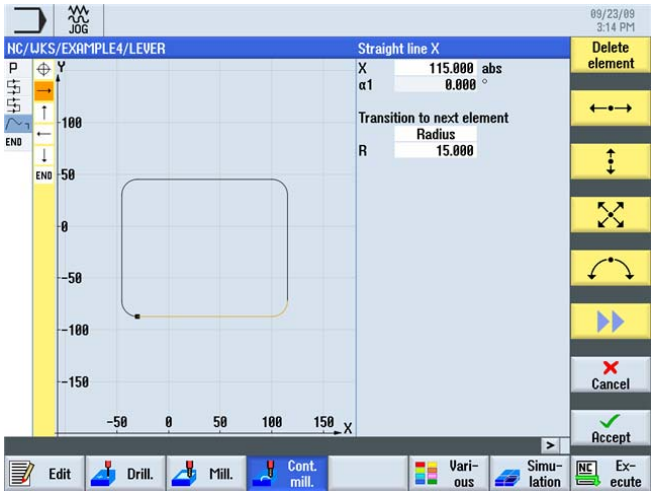


Figure 9-8 Readily designed contour

9.4 Machining the lever

Operating sequences

Proceed as follows to enter the contour:

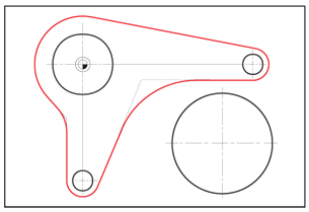


Figure 9-9 Lever contour

Select the **Contour milling** softkey.

Create a new contour with the name 'LEVER_Lever'.

Figure 9-10 Creating the contour

In the interactive screenform, enter the following values for the starting point of the contour line:

Field	Value	Selection via toggle key	Notes
X	-24 abs		
Y	0 abs		

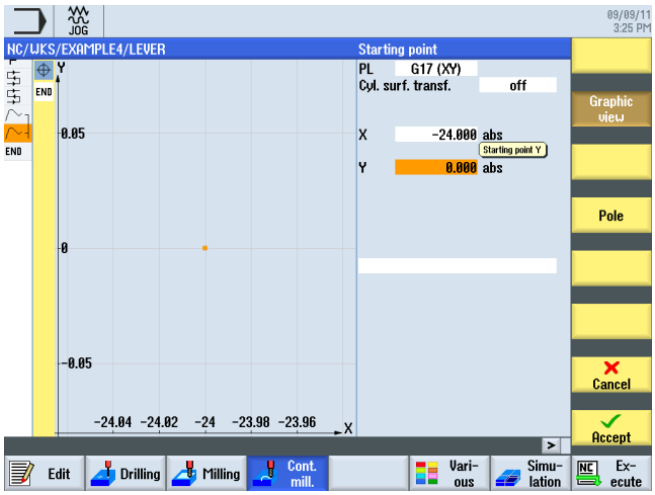


Figure 9-11 Specifying the starting point

"Apply" the set values.

Enter the following values for the first arc in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Direction of rotation	Clockwise	X	
R	24		Radius and center point are known.
I	0		

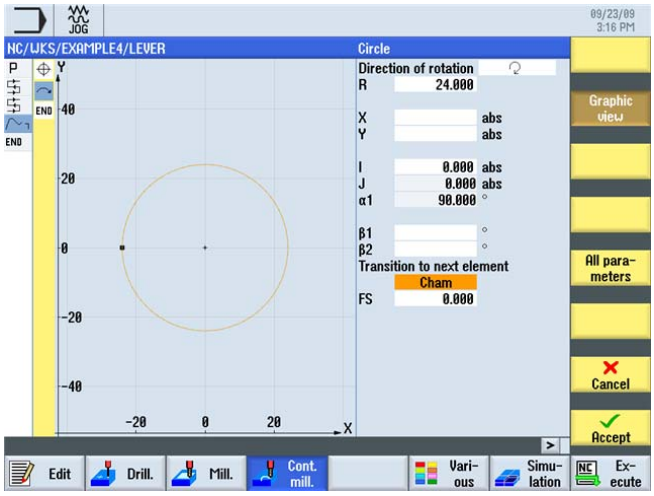


Figure 9-12 Arc contour

"Apply" the set values.

Create the inclined straight line connected tangentially to the preceding element.

Select the **Tangent to prec.elem.** softkey.

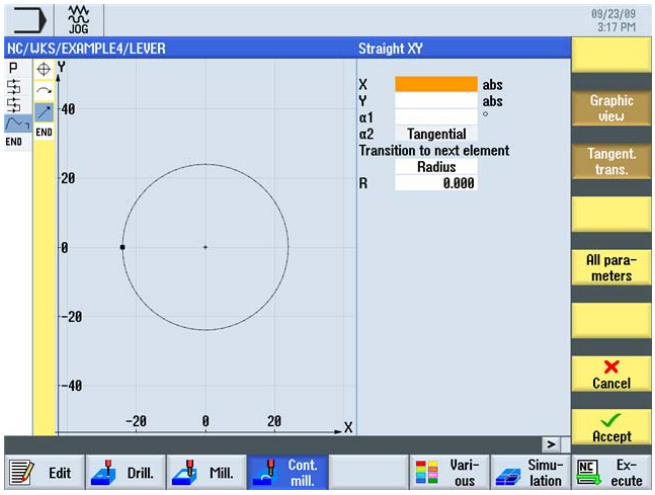
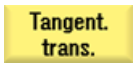


Figure 9-13 Inclined straight line contour

Press ENTER to confirm your input.

Enter the arc connected tangentially.



Select the **Tangent to prec.elem.** softkey.

Enter the following values for the arc in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Direction of rotation	Right	X	
R	8		Radius, center point and end point are known.
X	85 abs	X	
Y	-8 abs	X	
I	85 abs	X	

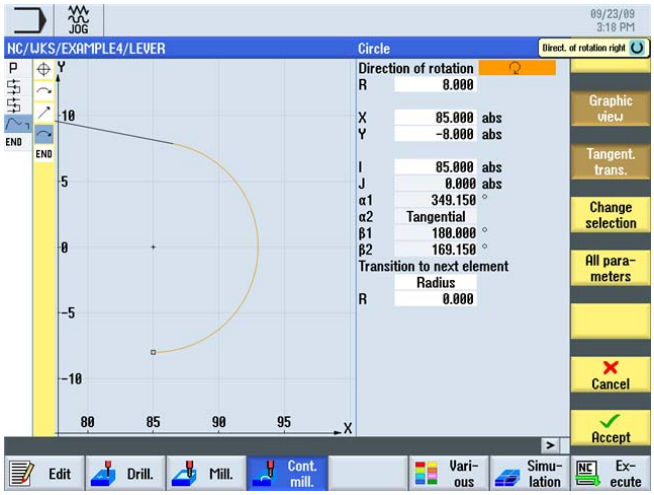


Figure 9-14 Arc contour

Press ENTER to confirm the suggested contour.

"Apply" the set values.

Enter the following values for the horizontal straight line up to end point X30 in the interactive screenform:

Field	Value	Selection via toggle key	Notes
X	30 abs	X	
R	40		Enter 40 mm for the radius to the subsequent element.

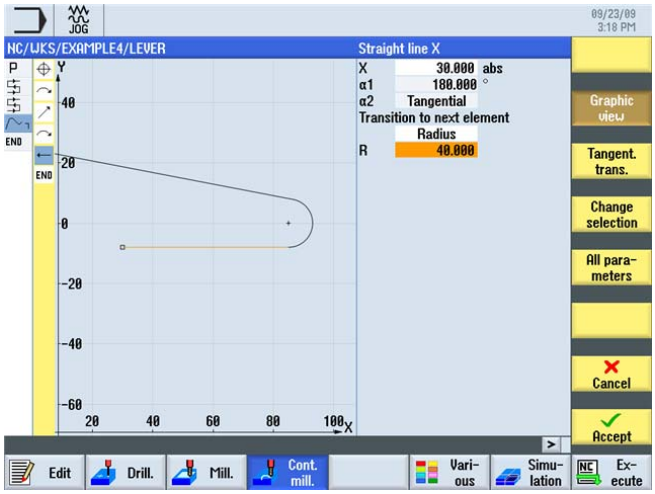


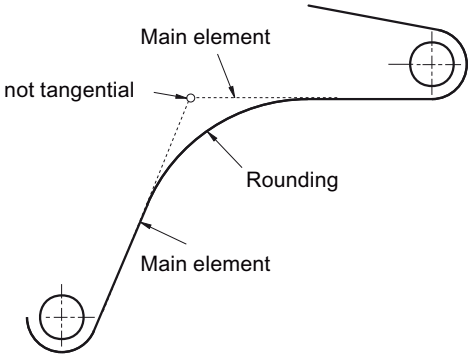
Figure 9-15 Horizontal straight line contour

"Apply" the set values.

Observe the following note for the next inclined straight line:

Note

The tangential transition is always only referred to the main element, i.e. in this case, the straight is not connected tangentially (see screenform below).



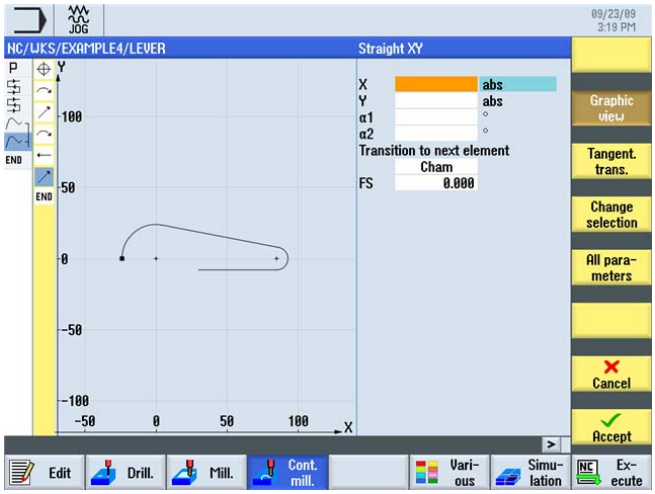
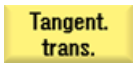


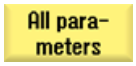
Figure 9-16 Inclined straight line contour

Press ENTER to confirm your input.

Enter the arc connected tangentially.



Select the **Tangent to prec.elem.** softkey.



Select the **All parameters** softkey.

Use the **All parameters** function to display detailed information on the arc. This can serve to check the entered values, for example (e.g.: Does the arc end vertically...?).

Enter the following values for the arc in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Direction of rotation	Right	X	
R	8		
Y	-58 abs		
I	0 abs		
J	-58 abs		

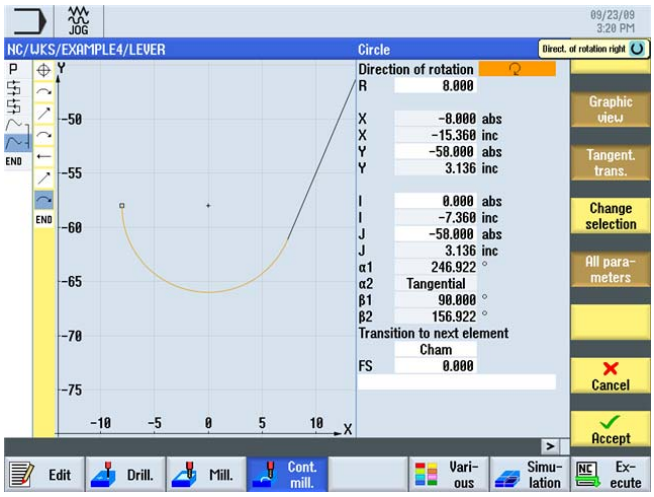


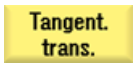
Figure 9-17 Arc contour

Select the desired contour suggestion.

Press ENTER to confirm the suggested contour.

Press ENTER to confirm your input.

Specify the vertical straight line (automatically tangential) up to the end point Y-27.



Select the **Tangent to prec.elem.** softkey.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Y	-27 abs	X	
R	18	X	Round the transition to the subsequent straight line using R18.

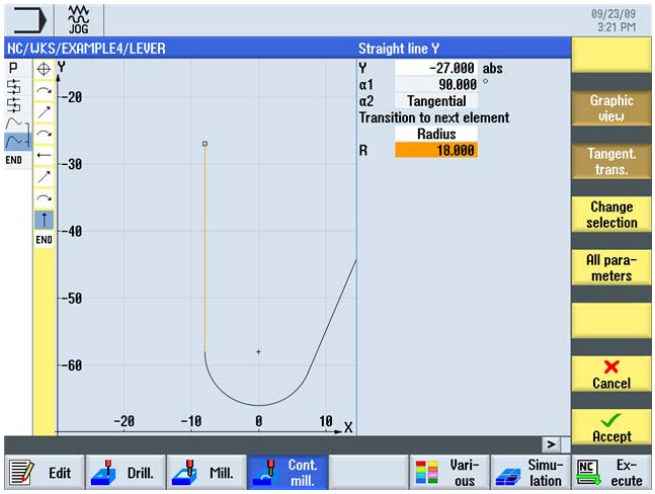


Figure 9-18 Vertical straight line contour

"Apply" the set values.

Specify the inclined straight line.

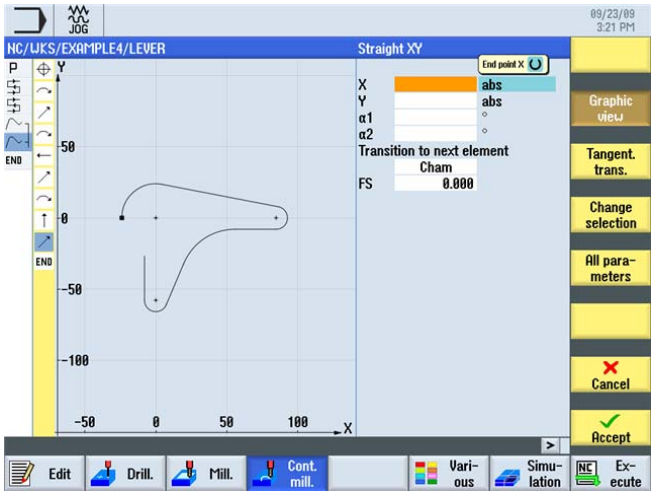
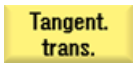


Figure 9-19 Inclined straight line contour

Press ENTER to confirm your input.

Close the contour to the starting point with an arc.



Select the **Tangent to prec.elem.** softkey.

In the interactive screenform, enter the following values for the starting point of the contour line:

Field	Value	Selection via toggle key	Notes
R	24		
X	-24	X	
Y	0	X	
I	0	X	

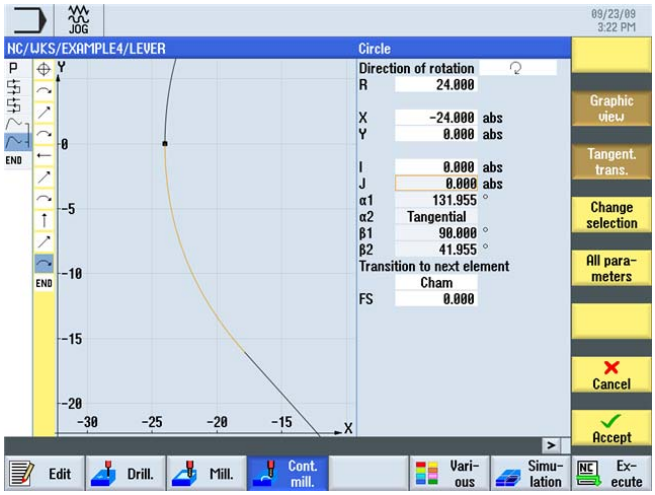


Figure 9-20 Arc contour

"Apply" the set values.

Accept the contour.

Proceed as follows to rough and finish the pocket taking into account the lever contour:

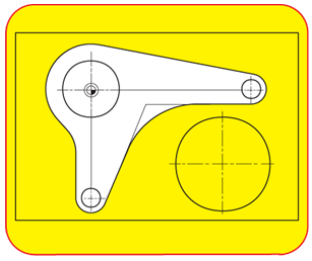


Figure 9-21 Roughing and finishing around the lever

Pocket

Select the **Pocket** softkey.

Select tool

Open the tool list and select CUTTER20.

To program

Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.15 mm/tooth	X	
V	120 m/min	X	
Machining	Roughing	X	
Z0	0		
Z1	6 inc	X	
DXY	50%	X	Specify the maximum infeed in the plane in %.
DZ	6		
UXY	0		
UZ	0.3		
Starting point	Automatic	X	
Insertion	Vertical	X	
FZ	0.15 mm/tooth	X	
Lift mode	To RP	X	

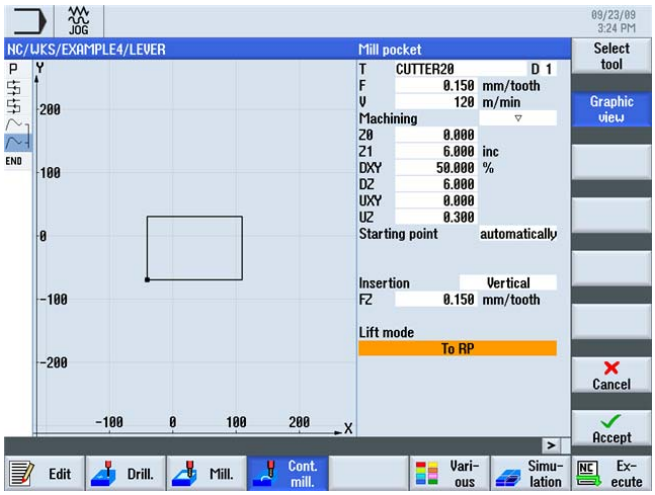


Figure 9-22 Roughing the contour

Accept

"Apply" the set values.

Select the **Pocket** softkey.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.08 mm/tooth	X	
V	150 m/min	X	
Machining	Finishing the base	X	
Z0	0		
Z1	6 inc	X	
DXY	50%	X	Specify the maximum infeed in the plane in %.
UXY	0		
UZ	0.3		
Starting point	Manual	X	
XS	70		
YS	-40		
Insertion	Vertical	X	
Lift mode	To RP	X	

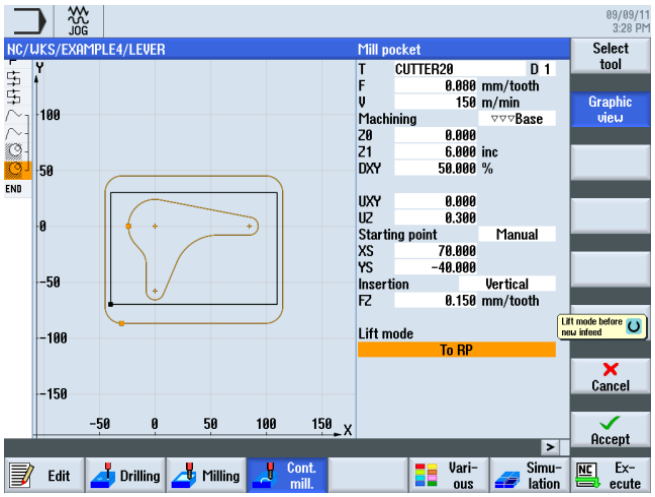


Figure 9-23 Finishing the base

"Apply" the set values.

9.5 Creating an edge for the circular island

Operating sequences

Create an edge for boundary when milling without help. Mill down to a depth of -3.

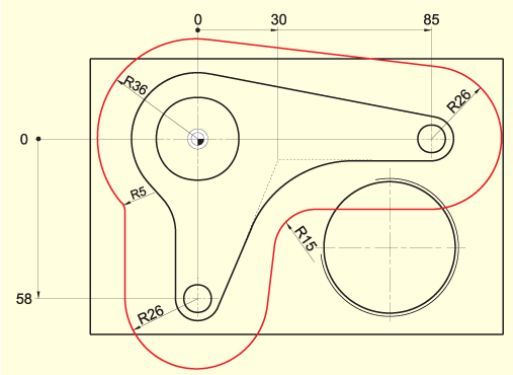


Figure 9-24 Edge contour for the circular islands

Note

The values R36 and R26 result from the corresponding island radius + cutter diameter (here: 20 mm + 1 mm allowance).

The radii R5 and R15 are selected freely.

Select the **Contour milling** softkey.

Create a new contour with the name 'LEVER_Lever_Area'.

Figure 9-25 Creating the contour

Design the boundary for the traversing paths around the workpiece contour as described above such that the 20mm milling cutter fits everywhere between the boundary and the islands. Enter this boundary contour in the same way as the lever contour.

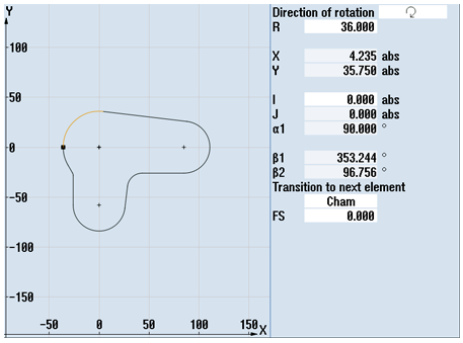


Figure 9-26 Arc contour section, left

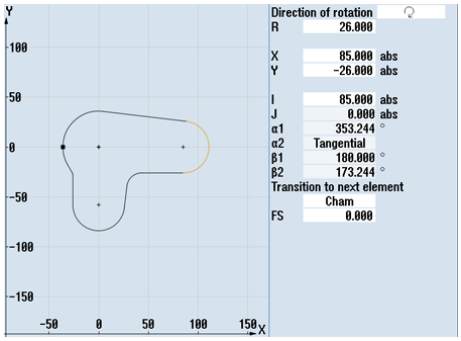


Figure 9-27 Arc contour section, right

9.6 Creating a 30mm circular island

Operating sequences

To create the 30mm circular island, I will proceed as follows:

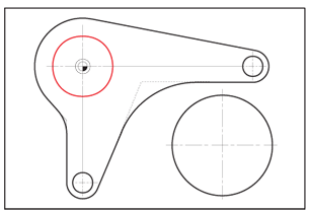


Figure 9-28 30mm circular island

Select the **Contour milling** softkey.

Create a new contour with the name 'LEVER_Circle_R15'.

Figure 9-29 Creating the contour

Create a circular contour without help (see illustration below). The starting point of the circular construction lies at X-15; Y0.

Note

Ensure that various values must be dimensioned incrementally.

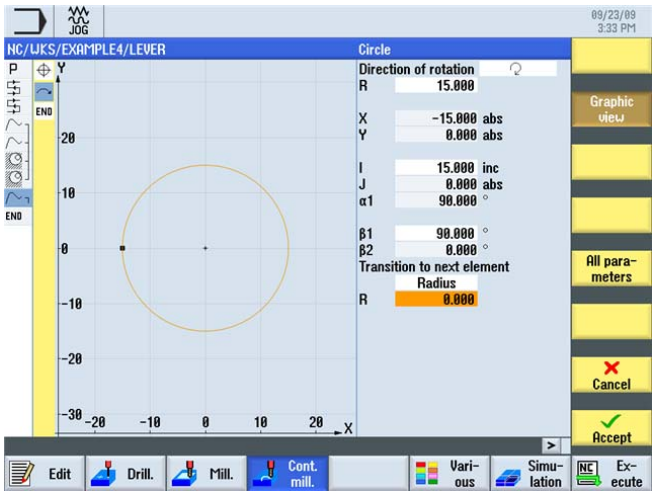


Figure 9-30 Circular island contour

9.7 Creating a 10mm circular island

Operating sequences

To create the 10mm circular island, proceed as follows:

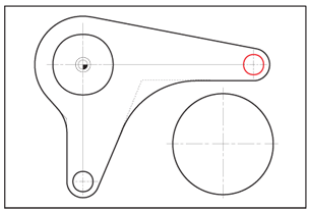


Figure 9-31 10mm circular island

Select the **Contour milling** softkey.

Create a new contour with the name 'LEVER_Circle_R5_A'.

Figure 9-32 Creating the contour

Create a circular contour without help (see illustration below). The starting point of the circular construction lies at X80; Y0.

Note

Since this circular island will be copied in the next step, you must specify the contour incrementally so that you only need to change the starting point when copying.

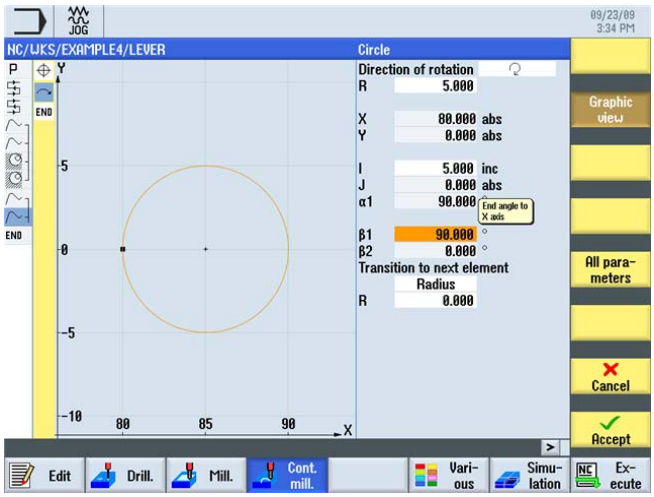


Figure 9-33 10mm circular island contour

After entering the circle, the broken-line graphics looks like this.

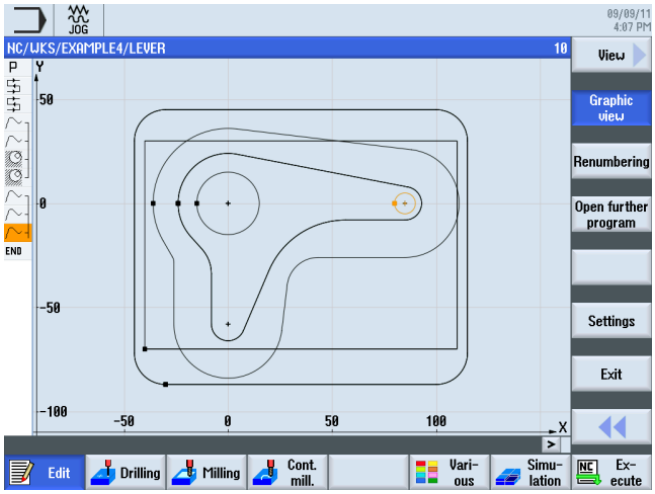


Figure 9-34 Broken-line graphics

9.8 Copying the 10mm circular island

Operating sequences

To copy the circular island created in the previous step, proceed as follows:

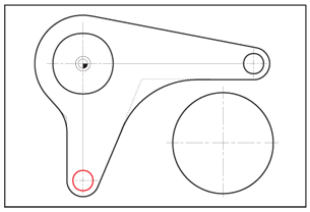


Figure 9-35 10mm circular island

Copy

Navigate and copy the 'LEVER_Circle_R5_A' contour.

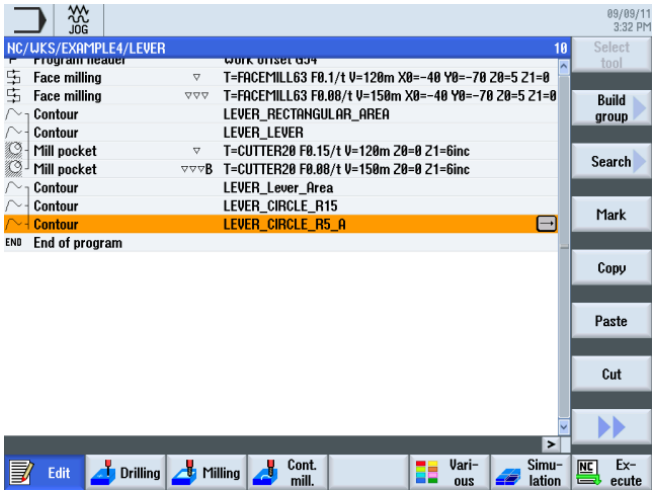


Figure 9-36 Copying the contour

Paste

Paste the copied contour and give it the name 'LEVER_Circle_R5_B'.

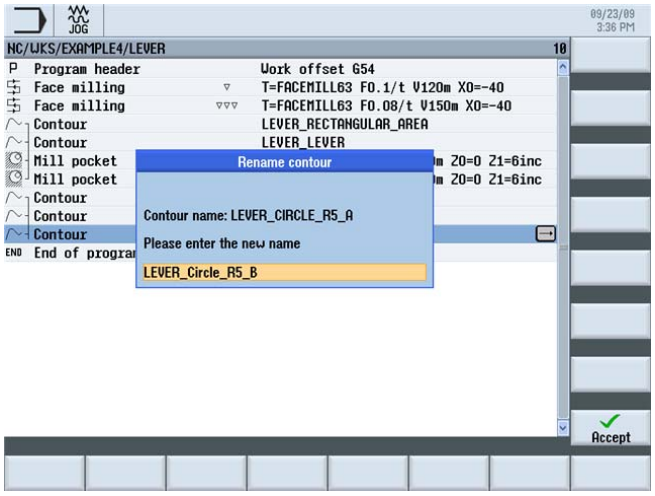


Figure 9-37 Specifying the name for the copied contour

Accept

Press ENTER to confirm your input.

After accepting the values, your process plan should look like this:

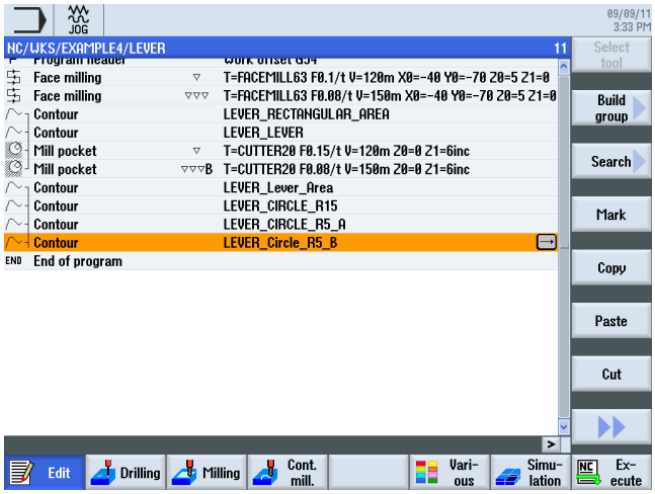


Figure 9-38 Pasted contour in the work step editor

Now you must only change the starting point, as you have specified the contour incrementally.

Open the contour. This key can then also be used in the open contour to open the selected geometry element for changing.

In the interactive screenform, enter the following values for the starting point of the contour line:

Field	Value	Selection via toggle key	Notes
X	-5		
Y	-58		

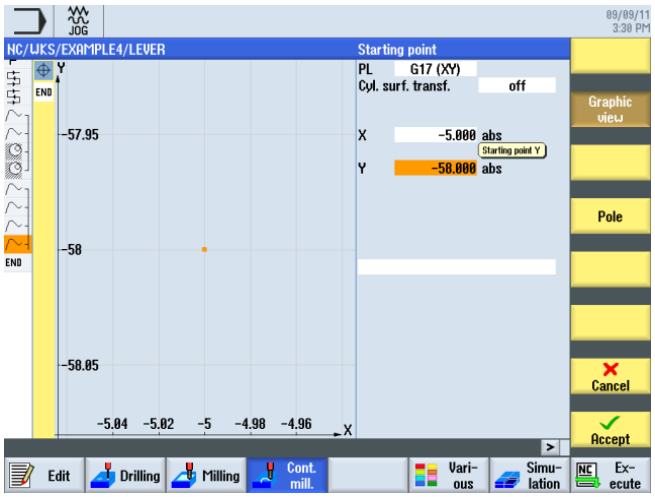


Figure 9-39 Changing the starting point

"Apply" the set values.

9.9 Machining the circular island using the editor

Operating sequences

To machine the three circular islands, proceed as follows: When machining the circular islands, you will learn further functions of the work step editor, helping you understand how to use parts of the process plans several times and how to manage the process plan (see *Functions of the work step editor*).

The following contour serves the boundary of the traversing path during manufacturing.

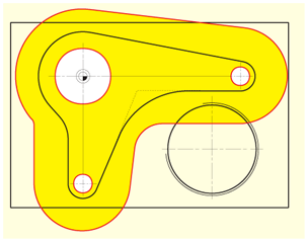


Figure 9-40 Boundary of the traversing path

Your process plan will look as follows:

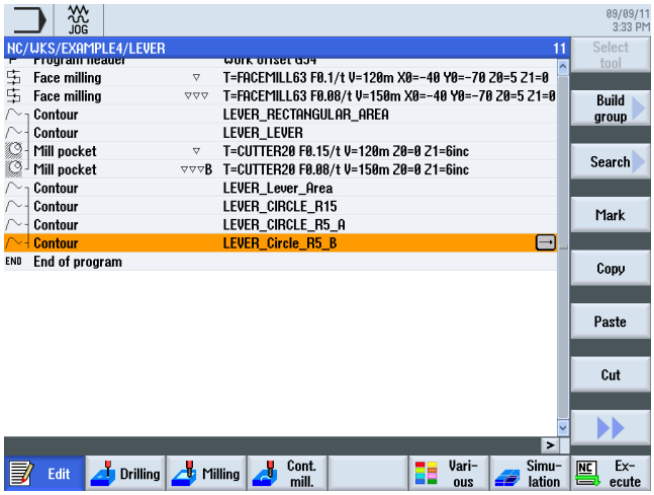


Figure 9-41 Process plan

Highlight the two work steps for roughing and finishing of the pocket.

Copy

Copy the highlighted work steps.

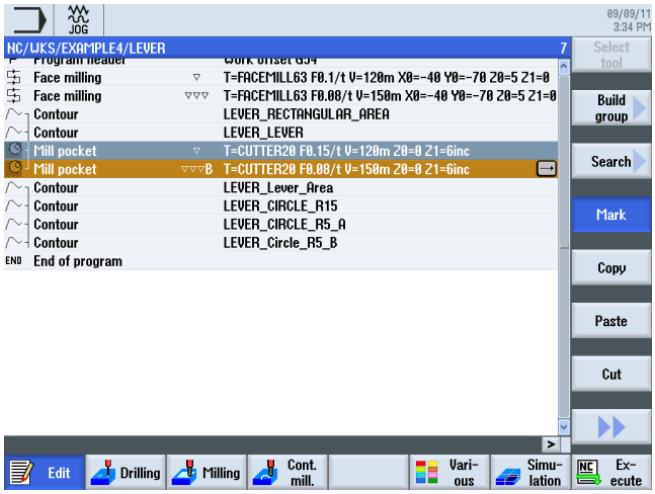


Figure 9-42 Highlighted machining steps

Paste

Paste the work steps beneath the contours. This will link the technologies for removing material from the solid with the contours.

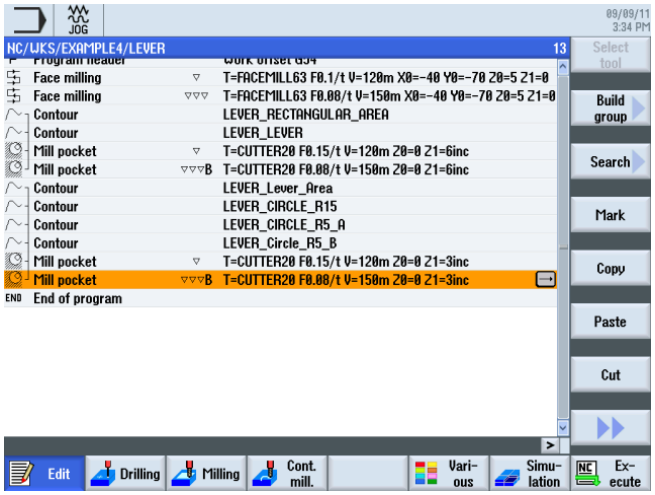


Figure 9-43 Pasted machining steps

Now you must only adapt the technologies solid 'roughing' and 'finishing' for removing material from the solid to the new machining depth:

Open the work step for roughing.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z1	3 inc	X	
Starting point	Manual	X	
XS	70		
YS	-10		

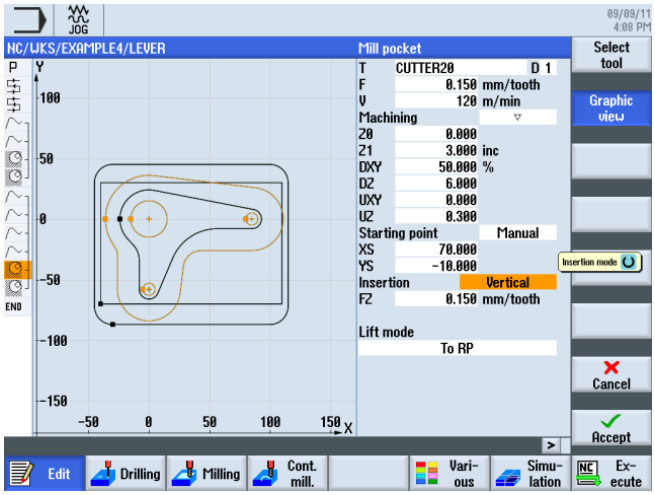


Figure 9-44 Adaptations for roughing

"Apply" the set values.

Open the work step for finishing. Change the values similar to roughing.

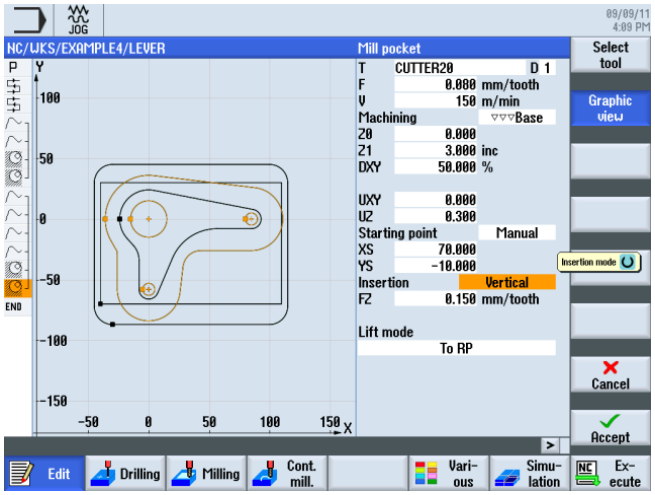


Figure 9-45 Adaptations for finishing

"Apply" the set values.

The screenform above shows which geometries are used in finishing (process plan graphic).

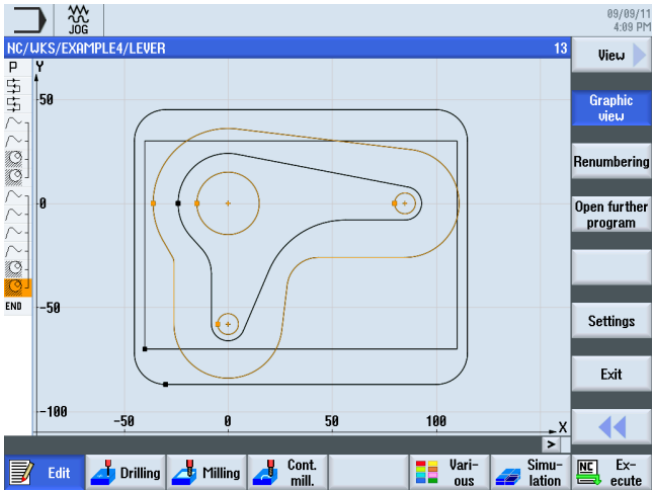


Figure 9-46 Broken-line graphics

Check your intermediate result by way of simulation.

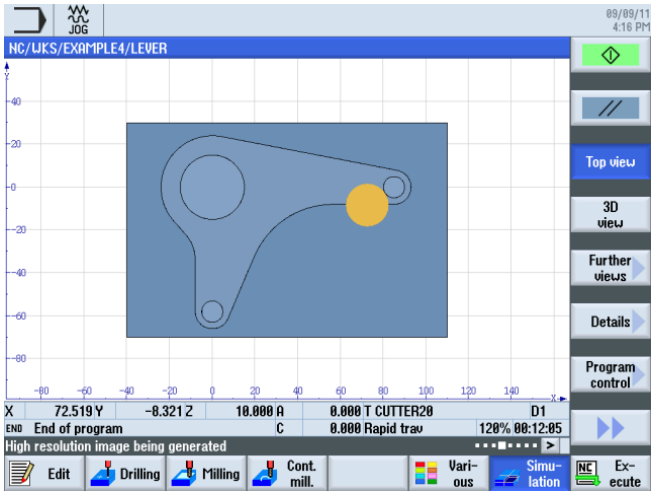


Figure 9-47 Simulation - Top view

Functions of the work step editor

The following information provides you with an overview of the functions of the work step editor.

Use this softkey to switch to the broken-line graphics.

Use this softkey to search for texts in the program.

Use this softkey to select several work steps for further processing (e.g., "Copy" or "Cut").

Use this softkey to copy work steps to the clipboard.

Use this softkey to paste work steps from the clipboard to the process plan. The copied step is always inserted after the currently highlighted step.

Use this softkey to copy work steps to the clipboard; at the same time, it is deleted at its origin. This softkey can also be used for "pure" deletion.

Use this softkey to switch to the extended menu.

Use this softkey to renumber the work steps.

Use this softkey to open the "Settings" dialog. Here you can specify, e.g. automatic numbering or whether you wish the end of the block to be represented as a symbol.

Use this softkey to return to the previous menu.

9.10 Deep hole drilling

Operating sequences

Proceed as follows for rough-boring:

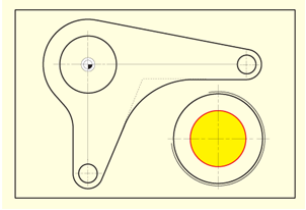


Figure 9-48 Deep hole drilling

Select the **Drilling** softkey.

Select the **Drilling Reaming** softkey.

Open the tool list and select PREDRILL30.

Accept the tool into your program.

Enter the following values for deep hole drilling in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.1 mm/rev	X	
V	120 m/min	X	
Depth reference	Tip	X	
Z1	-21 abs	X	
DT	0 s	X	

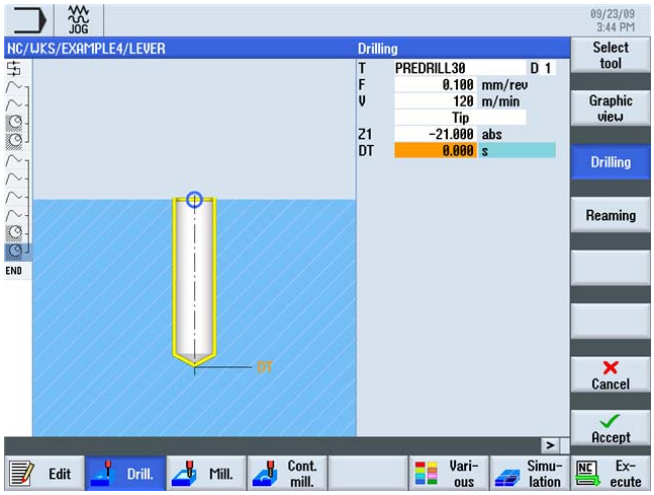


Figure 9-49 Specifying the drill hole

"Apply" the set values.

Positions

Enter the following values for the drilling position in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Positions	Rectangular	X	
Z0	-6		
X0	70		
Y0	-40		

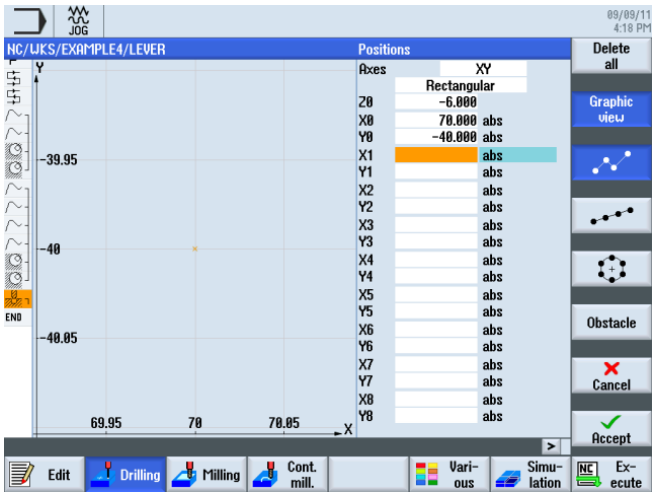


Figure 9-50 Entering the position

Accept

"Apply" the set values.

9.11 Milling a helix

Operating sequences

Proceed as follows to remove the residual material of the circular ring remaining after the drilling by way of a helical motion ("helix"):

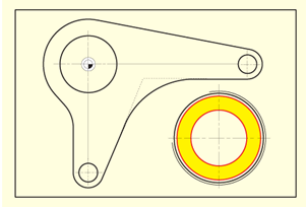


Figure 9-51 Milling a helix

Select the **Straight line Circle** softkey.

Open the tool list and select CUTTER20 .

Accept the tool into your program. Enter the following value in the interactive screenform:

Field	Value	Selection via toggle key	Notes
V	120 m/min	X	

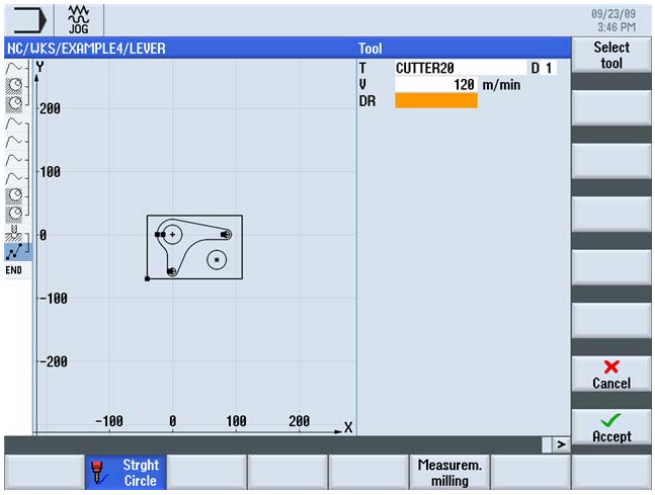


Figure 9-52 Milling a helix

Press ENTER to confirm your input.

Select the **Straight line** softkey.

Select the **Rapid traverse** softkey.

In the interactive screenform, enter the following values for the starting point of the contour line:

Note

Since milling is performed without cutter radius compensation here, you must position the cutter with its circumference to the tap hole diameter (here: 45.84 mm) minus finishing allowance.

Field	Value	Selection via toggle key	Notes
X	82	X	
Y	-40	X	
Z	-5	X	
Radius compensation	off	X	

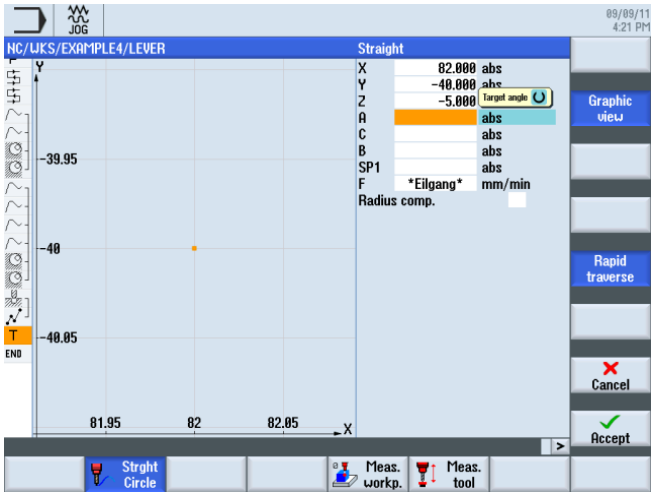


Figure 9-53 Positioning

"Apply" the set values.

Select the **Helix** softkey. Enter the following values for the helix in the interactive screenform:

Field	Value	Selection via toggle key	Notes
I	70	X	
J	-40	X	
P	3 mm/rev		The pitch of the helix is 3.
Z	-23 abs	X	
F	0.1 mm/tooth	X	

Note

Since the tool traverses along an inclined path, 6 revolutions are created here to avoid that no residual material remains (even though the final depth is already reached after 5 revolutions).

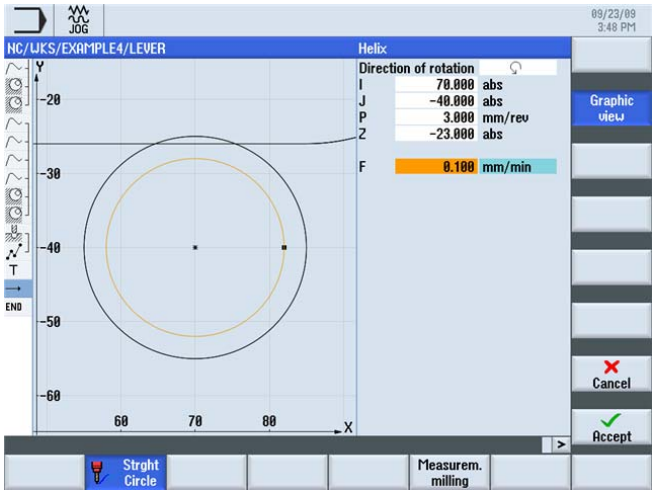


Figure 9-54 Specifying the helix

"Apply" the set values.

9.12 Boring

Operating sequences

Proceed as follows to machine the circular pocket to the required dimensions using a boring tool:

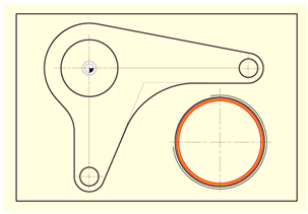


Figure 9-55 Boring a circular pocket

Select the **Drilling** softkey.

Select the **Boring** softkey.

Open the tool list and select the boring tool DRILL_tool.

Accept the tool into your program.

Enter the following values for the processing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.08 mm/rev	X	
S	500 rpm	X	
Z1	15 inc	X	
DT	0 s	X	
SPOS	45		
Lift mode	Lifting	X	The Lift option withdraws the tool from the contour before it retracts from the drill hole. This option may only be used with single-edge tools.
D	0.5		

Note

The angular position of the tool during lifting is specified by the machine manufacturer.

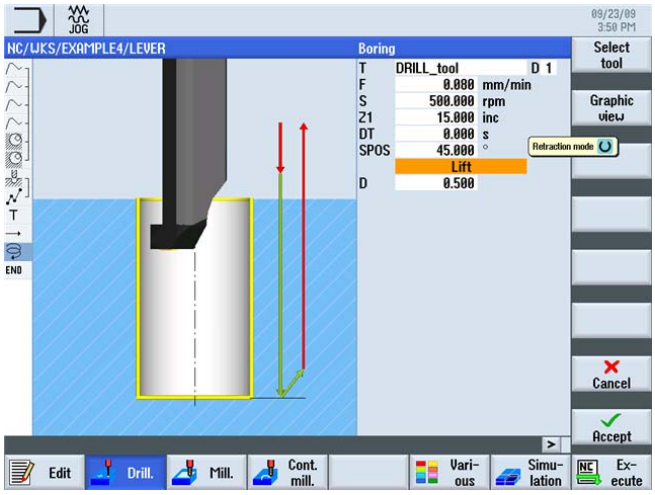


Figure 9-56 Boring

"Apply" the set values.

Position the tool to the drill hole center. The dimension 45.84 mm is specified by the set tool diameter. Instead of entering the position, you should also use the *Repeat position* function here.

Enter the following values for the position in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z0	-6		
X0	70		
Y0	-40		

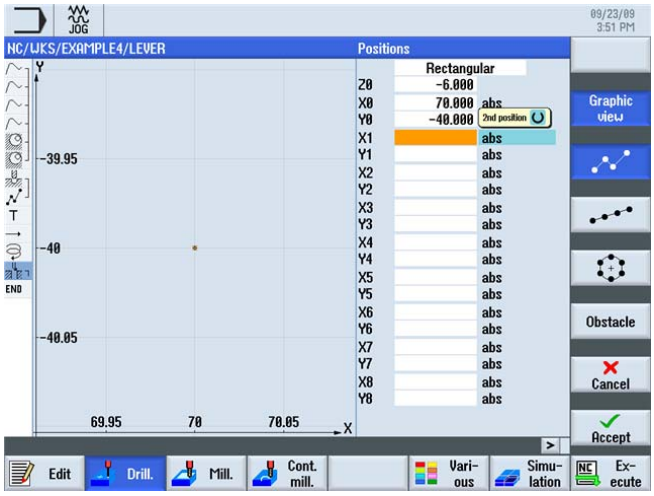


Figure 9-57 Positioning

"Apply" the set values.

9.13 Thread milling

Operating sequences

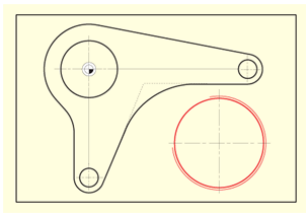


Figure 9-58 Thread milling

Select the **Milling** softkey.

Select the **Thread milling** softkey.

Select tool

Open the tool list and select THREADCUTTER .

To program

Accept the tool into your program.

Mill the thread from the top to the bottom. To this end, use the THREADCUTTER (F = 0.08 mm/tooth, v = 150 m/min and a pitch of 2 mm). A rectangular thread is to be milled absolutely to Z-23. Due to the overtravel of 3 mm, the thread is always milled cleanly down to the workpiece lower edge even if the lowest tooth is slightly worn.

The help displays are very useful for your inputs.

Compare your inputs with the screen below.

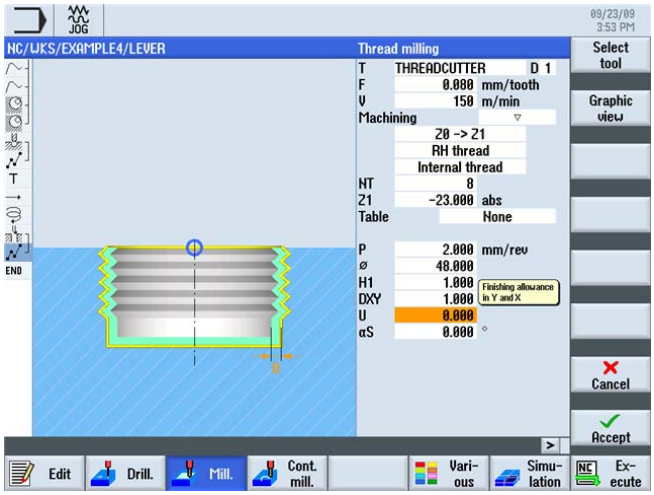


Figure 9-59 Thread milling

Accept

"Apply" the set values.

Positions

Specify the position for the thread.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z0	-6		
X0	70		
Y0	-40		

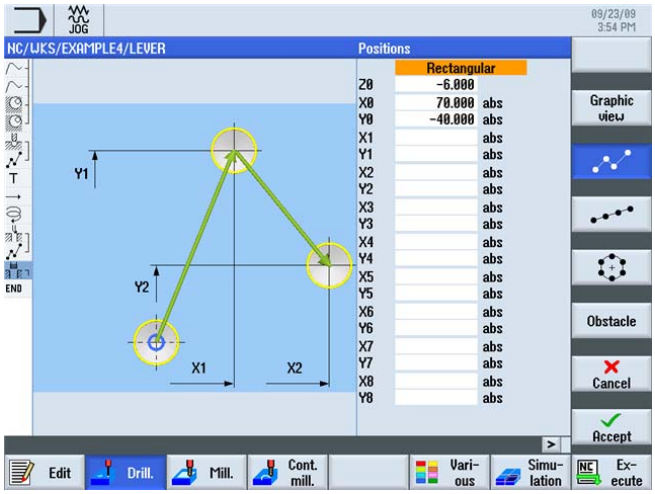


Figure 9-60 Entering the position

"Apply" the set values.

9.14 Programming a contour using polar coordinates

Programming with polar coordinates

Contour elements in workpiece drawings often refer to a pole. In this case, you do not know the Cartesian coordinates (X/Y), but the polar coordinates, i.e. the distance and the angle to this pole.

Now we will slightly modify the lever as a further exercise: The lower "lever arm" no longer lies vertically to zero at X0, but is rotated CW by 10°.

In this example you will learn how this is programmed graphically without using the pocket calculator or any auxiliary constructions.

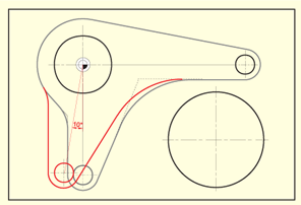


Figure 9-61 Programming the lever using polar coordinates

Operating sequences

Move the cursor first to the arc to redimension its center (see screenform below).

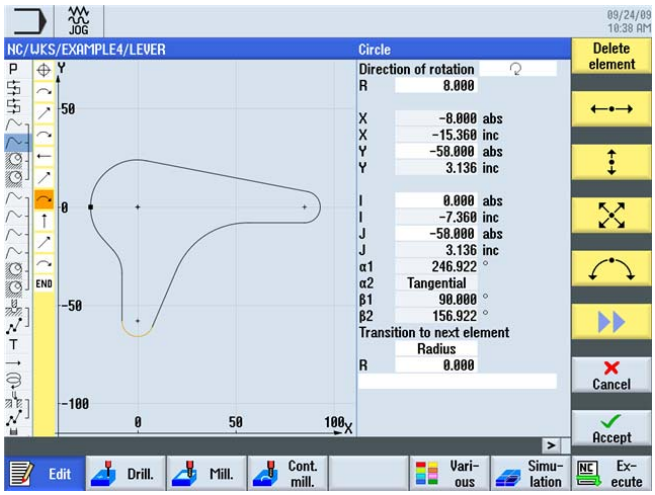


Figure 9-62 Positioning the cursor on the arc

Extend the menu.

Position the cursor on the element in front of the arc and paste the pole at this point. Apply the pole to the zero point.

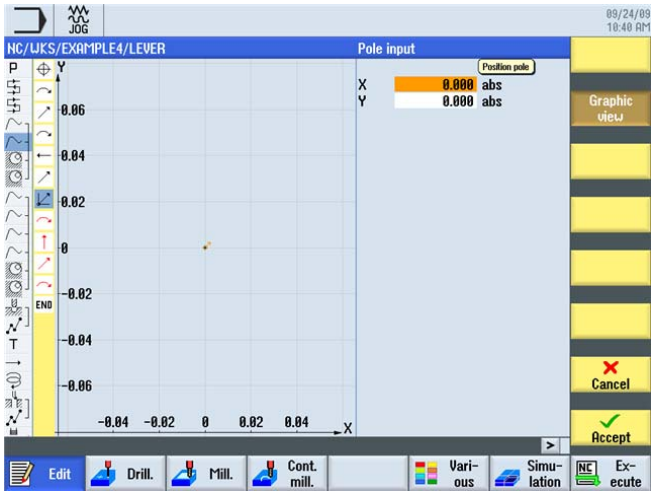


Figure 9-63 Specifying the pole

Press ENTER to confirm your input.

Subsequently, change the values matching the arc:

1. In the arc dialog box, delete the values Y-58, I0 and J-58 which are no longer valid.

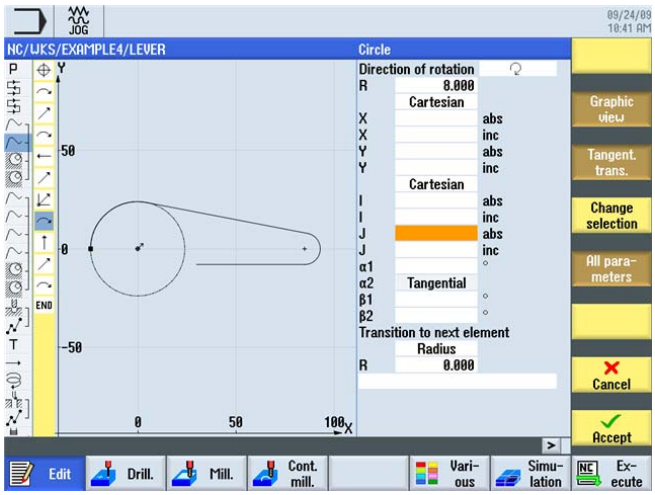


Figure 9-64 Deleting the values

2. To be able to specify the center point, switch the coordinates from "Cartesian" to "Polar". Enter the distance to the pole and specify the polar angle (see screenform below).

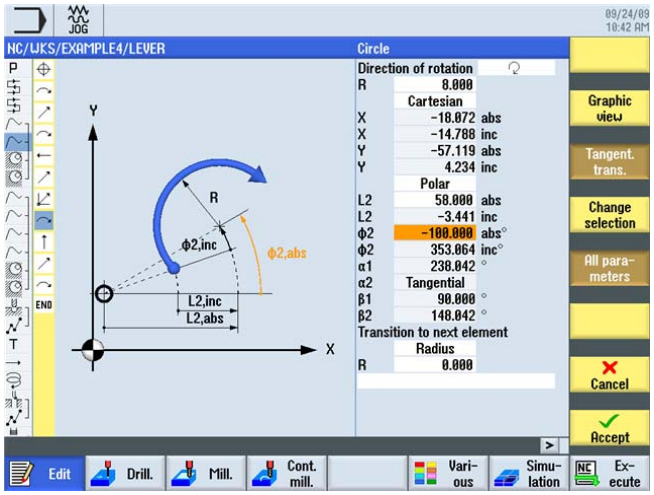


Figure 9-65 Entering the distance to the pole and specifying the polar angle

Press ENTER to confirm your input.

Apply the change.

The broken-line graphics shows that the auxiliary pocket LEVER_Lever_Area and the circular island LEVER_Circle_R5_B must also be adapted in the same way.

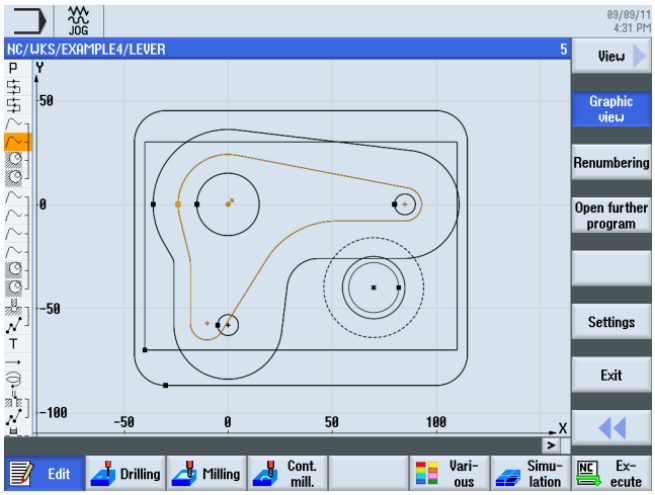


Figure 9-66 Broken-line graphics after shifting

Change these two contours without help. Note the following:

Note

As far as the auxiliary pocket is concerned, you may naturally proceed a bit "rougher" and approach the center of the arc R26 dimensioned with polar coordinates to Cartesian dimensioning (X-10/Y-57). Then, the contour can be continued directly with a vertical line.

The starting point for the circular island is already dimensioned with polar coordinates. Only the center point of the full-circle arc must be changed.

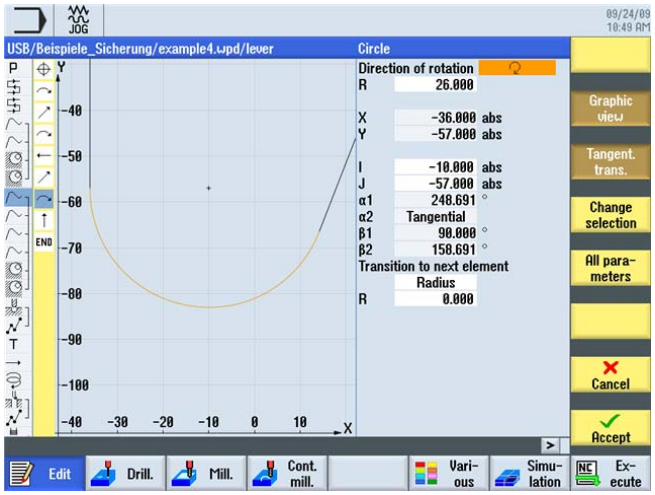


Figure 9-67 Adapting the edge

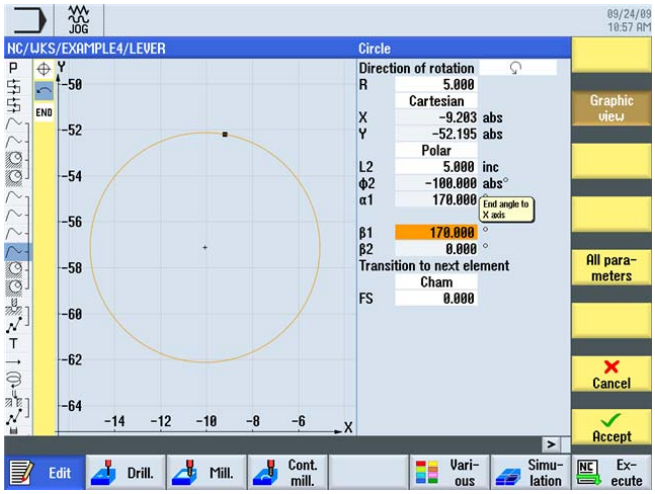


Figure 9-68 Adapting the circular island

After successful adaptation, your broken-line graphics looks like this:

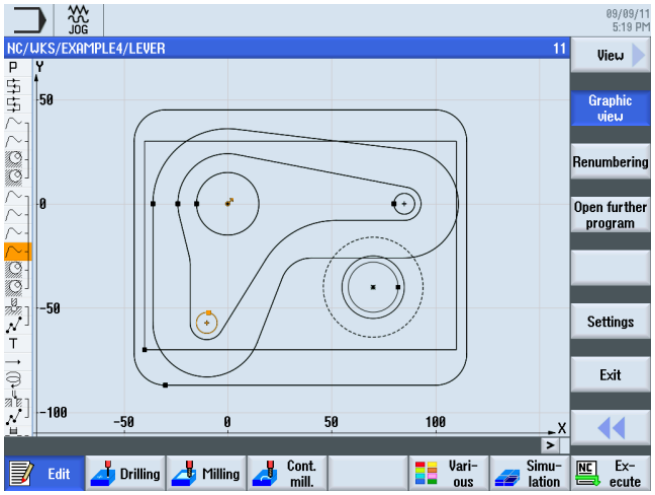


Figure 9-69 Broken-line graphics

Example 5: Flange

10

10.1 Overview

Learning objectives

In this chapter you will learn how to ...

- create a subroutine;
- mirror work steps;
- chamfer any contours, and
- create longitudinal and circular grooves.

Task

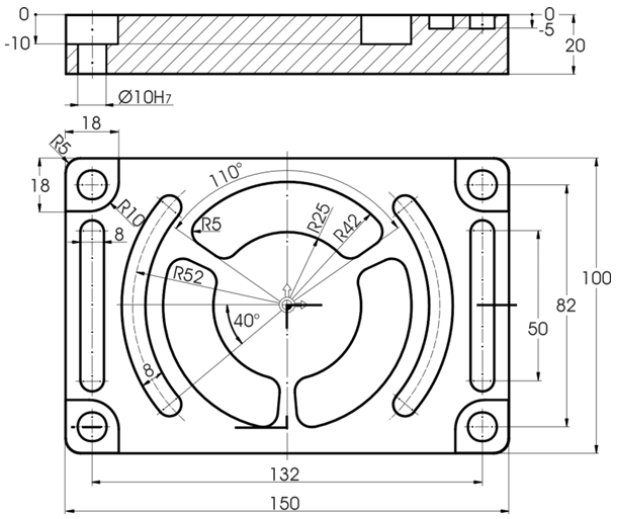


Figure 10-1 Workshop drawing - Example 5:

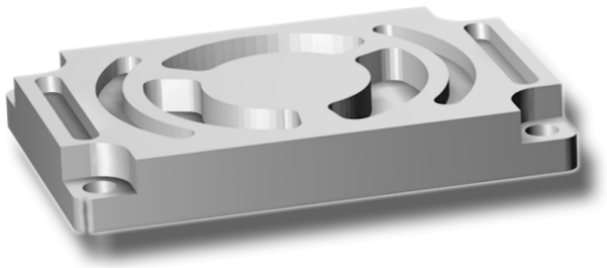


Figure 10-2 Workpiece - Example 5:

Note

All work steps were explained in the previous examples and nearly all softkeys / keys to be selected / pressed were indicated. In the following example, the whole sequence of inputs will no longer be specified, but instead only essential information and the most important softkeys and keys to be pressed.

10.2 Creating a subroutine

Operating sequences

The creation and functioning of subroutines will be explained taking the example of the workpiece CORNER_MACHINING.

The four corners will be machined by way of the following steps using a subroutine and the "Mirror" function.

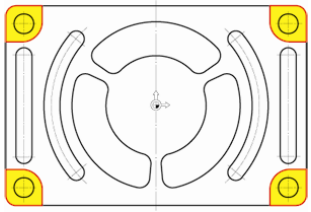


Figure 10-3 Contour of the four corners

Create a new step sequence program with the name CORNER_MACHINING. Later you will embed this program as a subroutine.

Figure 10-4 Creating a subroutine

Enter the following data for the program header. The blank dimensions will be specified later centrally in the main program.

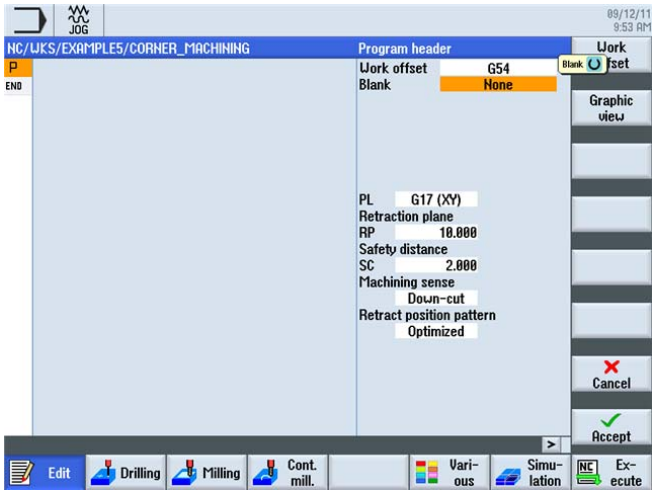


Figure 10-5 Entering the data for the subroutine program header

"Apply" the set values.

Select the **Contour milling** softkey.

Create a new contour with the name CORNER_M_SURFACE .

Figure 10-6 Creating the contour

Specify the starting point. The top right corner, for example, will be designed.

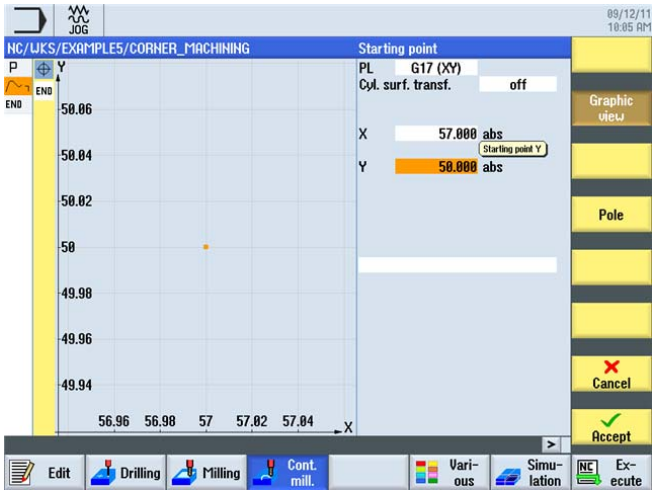


Figure 10-7 Specifying the starting point

"Apply" the set values.

Create the contour. After entering the two contour elements, your screen should look like this: Accept the contour into your process plan.

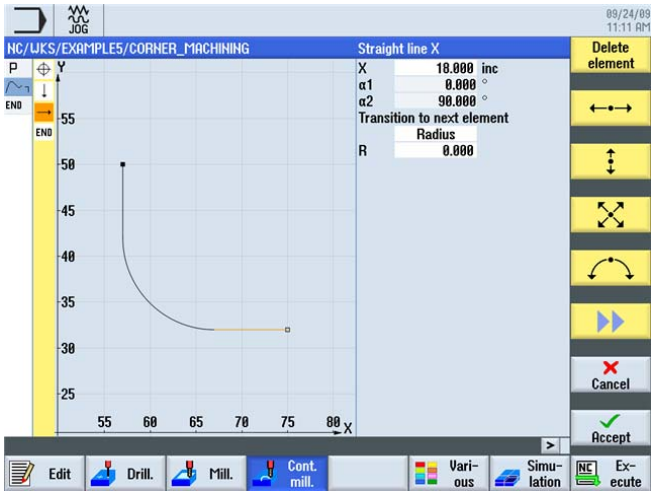


Figure 10-8 Contour subroutine, top right corner

Path milling

The contour is to be roughed using an R20 milling cutter (F = 0.15 mm/tooth and v = 120 m/min).

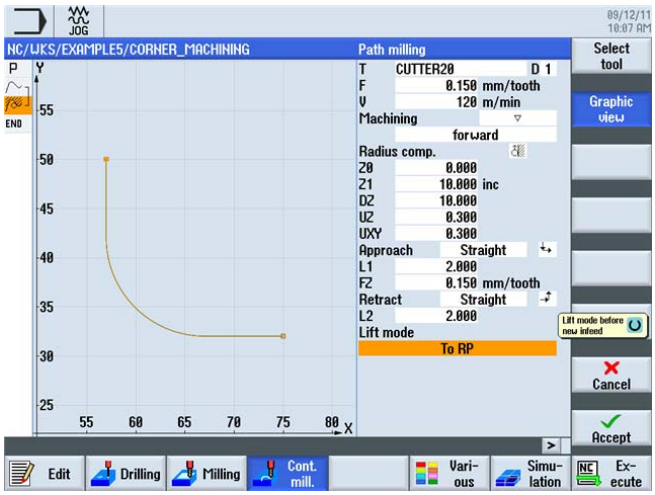


Figure 10-9 Roughing the contour

The approach and retract paths are approached along a straight line. The length values are the distances between the cutter edge and the workpiece.

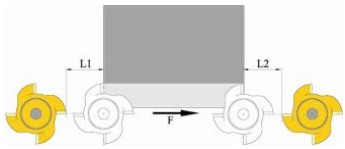


Figure 10-10 Approach and retract paths along a straight line

Accept

"Apply" the set values.

Path milling

The contour is to be finished using the same milling cutter (F = 0.08 mm/tooth and v = 150 m/min).

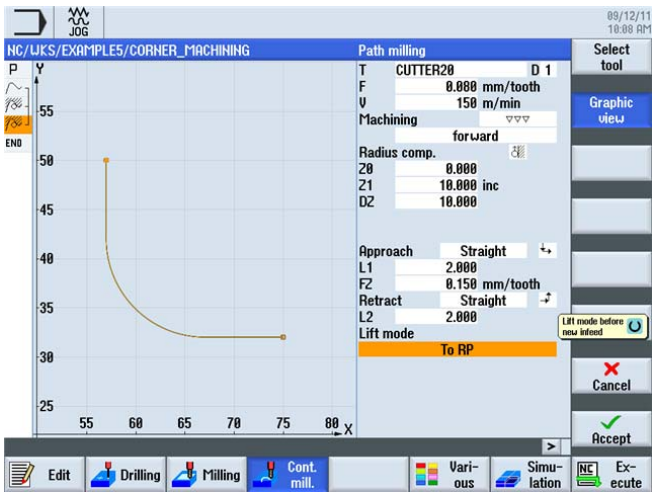


Figure 10-11 Finishing the contour

"Apply" the set values.

In the next few steps, the corner of the blank cuboid is to be rounded using R5:

Select the **Contour milling** softkey.

Create a new contour with the name CORNER_M_ARC .

Figure 10-12 Creating the contour

Specify the starting point.

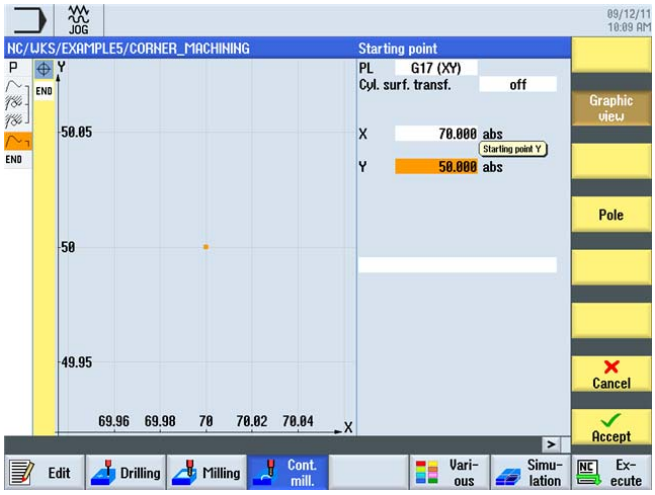


Figure 10-13 Specifying the starting point

"Apply" the set values.

Subsequently, specify the contour and the relevant work steps:

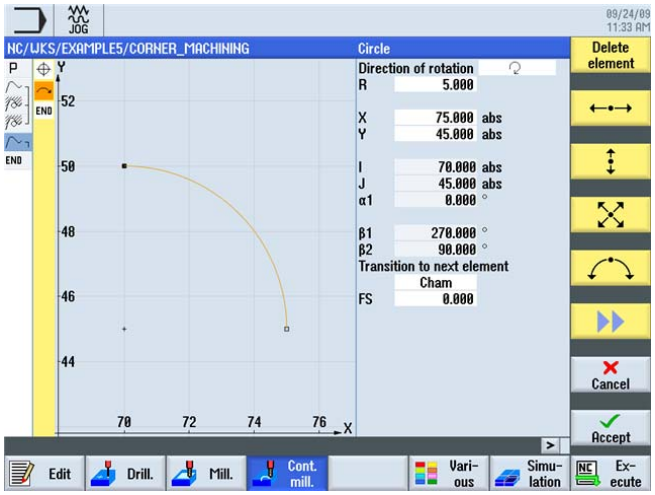


Figure 10-14 Specifying the geometry

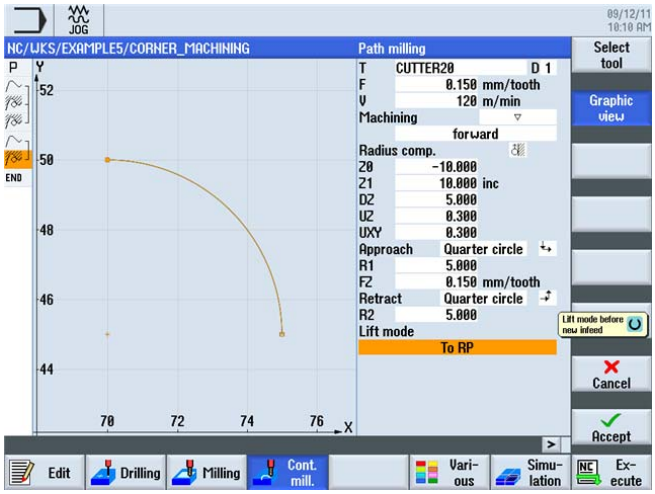


Figure 10-15 Roughing the contour

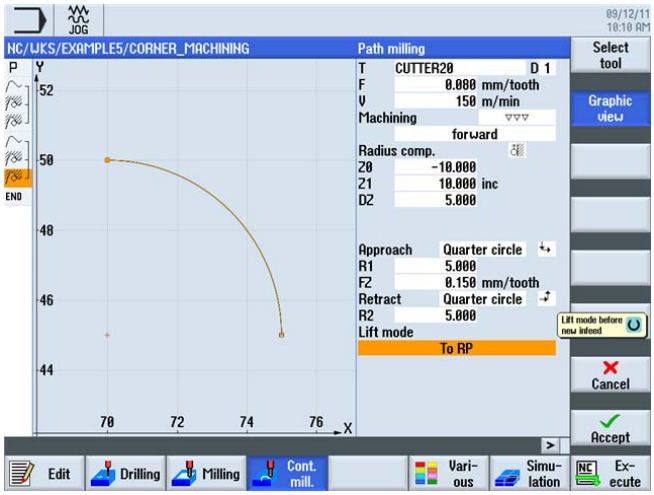


Figure 10-16 Finishing the contour

Figure 10-17 Complete subroutine in the work step editor

10.3 Mirroring work steps

Task

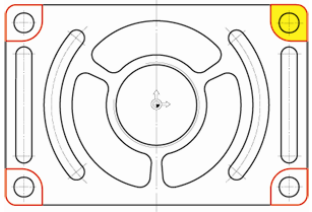
When you have finished the subroutine, create the main program. You may use the subroutine for all workpiece corners using the "Mirror" function in the "Transformation" menu.

Mirroring can be performed in two different ways:

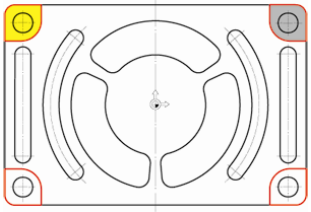
- New:
  Mirroring is performed from the point at which the 1st processing operation was performed.
- Additive:
  Mirroring is performed from the point last machined.

The sequence of processing with the setting *New* is shown below in the form of drawings:

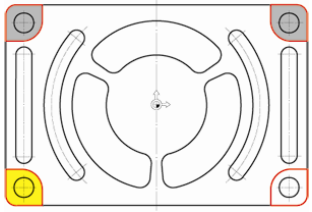
1. Processing (see subroutine)



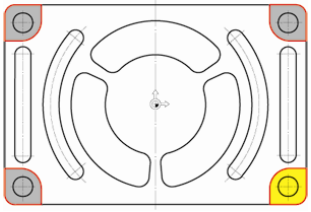
2. Processing: Mirroring the X axis
(the X values are mirrored here)



3 . Processing: Mirroring the X and Y axes
(the X and Y values are mirrored here)



4. Processing: Mirroring the Y axis
(the Y values are mirrored here)



Operating sequences

Create the main program with the name FLANGE .

Figure 10-18 Creating the main program

Enter the program header.

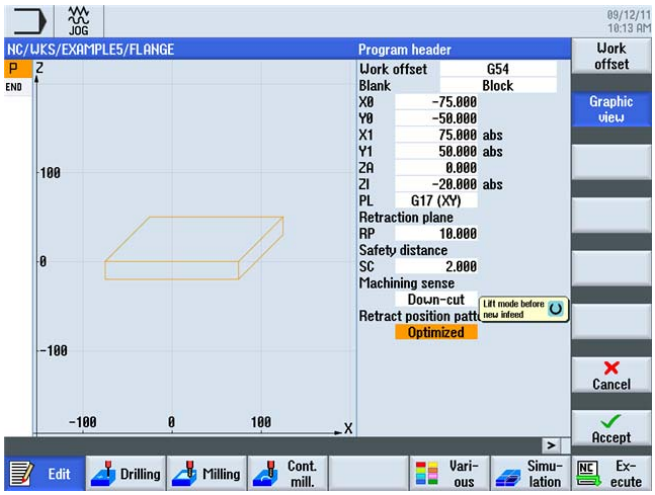


Figure 10-19 Specifying the program header of the main program

"Apply" the set values.

Select the **Miscellaneous** softkey.

Paste the subroutine into the main program.

Note

If you have created the subroutine in the same directory as the main program, the "Path/Workpiece" input field may remain empty.

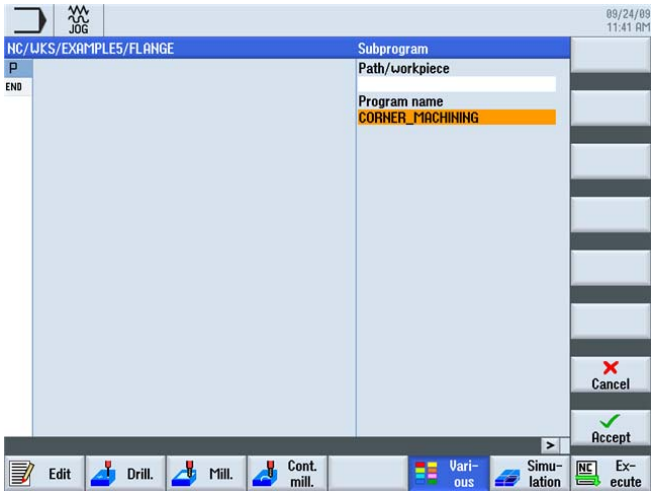


Figure 10-20 Pasting the subroutine

Press ENTER to confirm your input. After acceptance, your work step program looks like this:

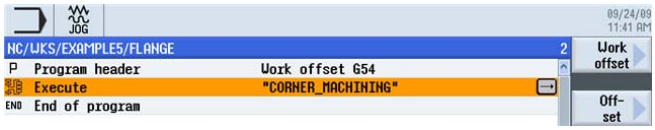


Figure 10-21 Subroutine pasted into the main program

The **Transformation** softkey can be used to shift, rotate, etc. the axes.

Preparing the 2nd processing: Mirror the X values.

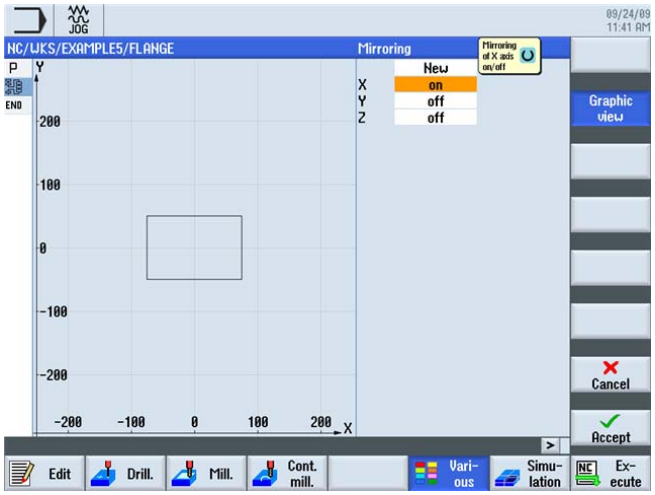


Figure 10-22 Mirroring

Press ENTER to confirm your input.

To mirror the remaining processing operations, proceed as follows:

Copy the subroutine after the "Mirror" work step. The 2nd processing follows.

Then you must repeat the processes *Mirror* and *Subroutine call* for the two other corners.

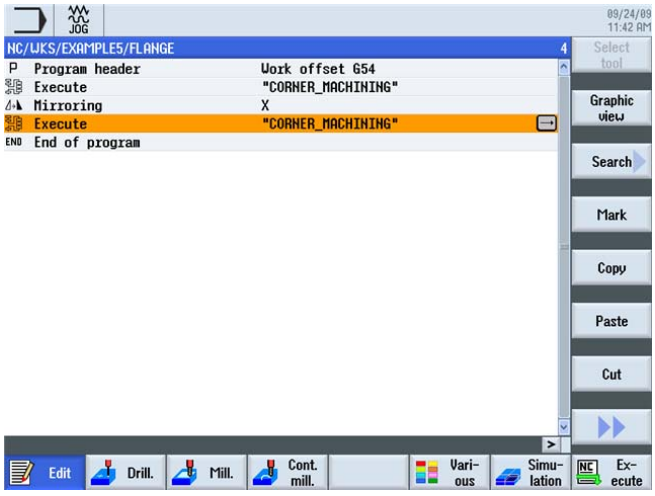


Figure 10-23 Copying the subroutine

The help display illustrating this procedure will help you. After you have entered all 4 processing operations, disable mirroring for all three axes.

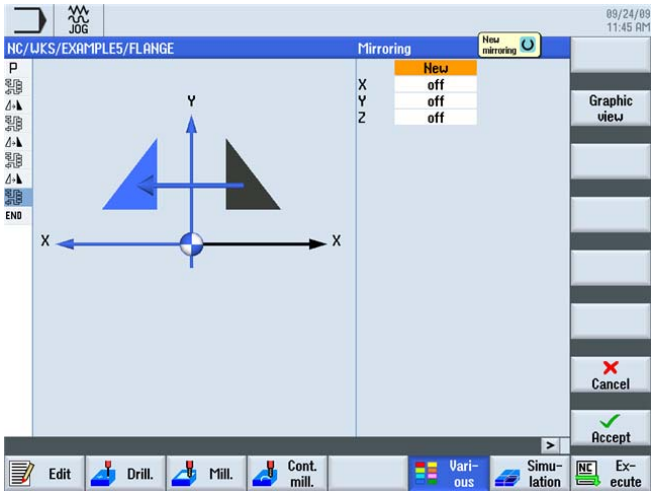


Figure 10-24 Mirroring help display

Your process plan will look as follows:

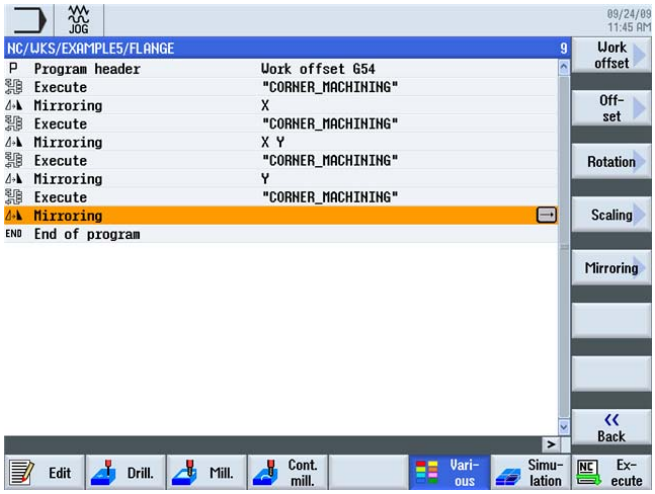


Figure 10-25 Complete mirroring in the work step editor

Check your work by now using the simulation.

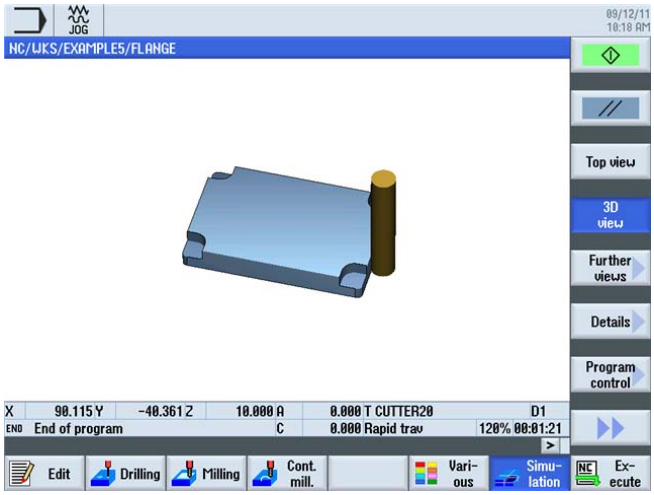


Figure 10-26 Simulation in 3D display

10.4 Holes

Operating sequences

With the next few work steps, you will create four drill holes in the corners. Since an obstacle lies between the individual drill holes, it must be specified between the positions.

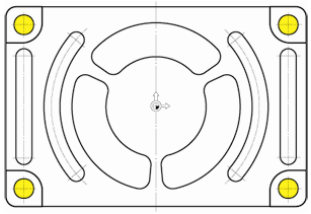


Figure 10-27 Holes

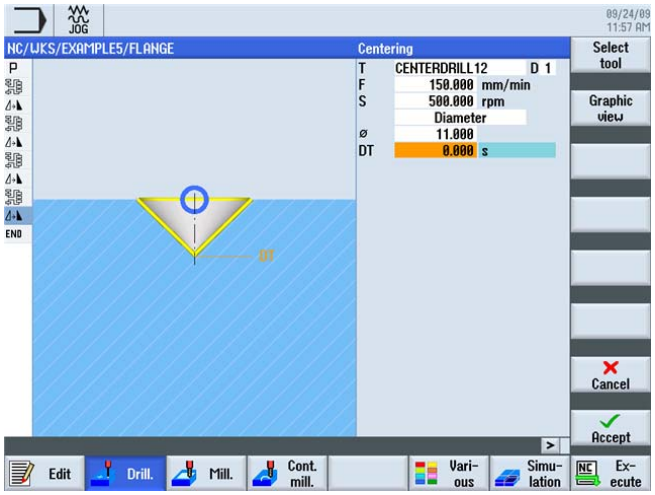


Figure 10-28 Centering

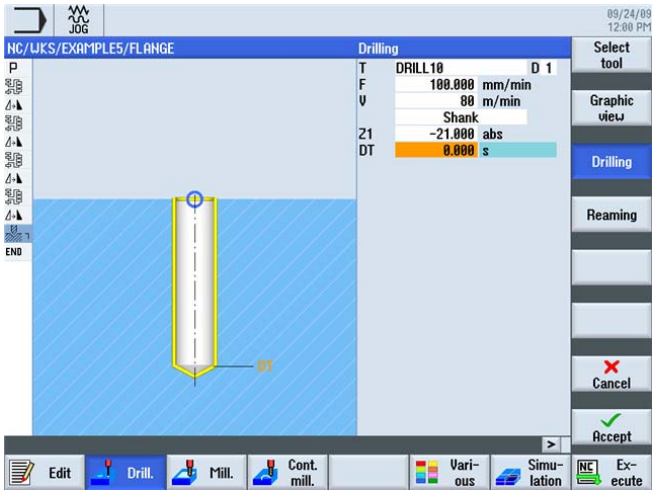


Figure 10-29 Drilling

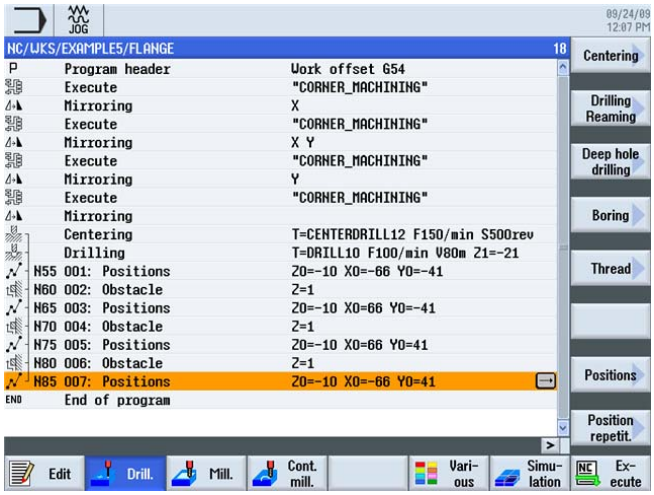


Figure 10-30 Specifying the positions of the obstacles

10.5 Rotation of pockets

Operating sequences

To program the contour and the processing for the pocket highlighted yellow, proceed as described in the following.
By rotating the coordinate system, subsequently the other two pockets are created.

Select the **Contour milling** softkey.

Create a new contour with the name 'FLANGE_NODULE' .

Figure 10-31 Creating a new contour

Specify the starting point.

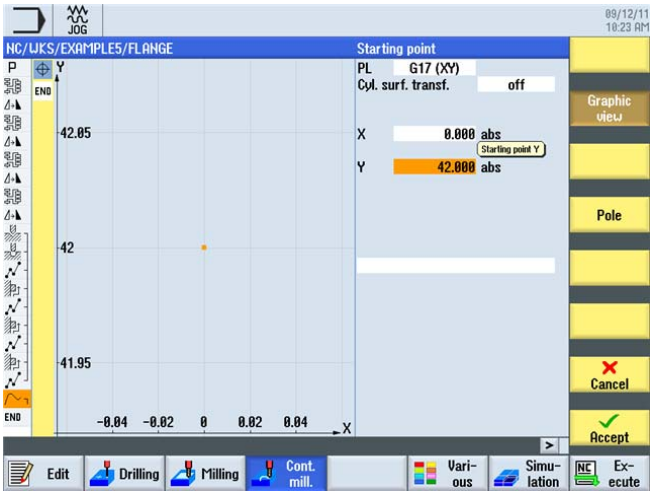
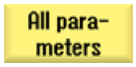


Figure 10-32 Specifying the starting point

"Apply" the set values.

Select the **Arc** softkey.



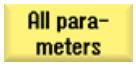
Select the **All parameters** softkey.

The arc R42 is described unambiguously, e.g. via the radius, the center point in X and the runout angle. Design in the counterclockwise direction to ensure that the pocket can also be finished by synchronized milling.

Figure 10-33 Specifying the arc

"Apply" the set values.

Select the **Diagonal** softkey.



Select the **All parameters** softkey.

Create the diagonal straight line.

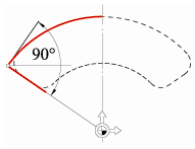
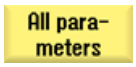


Figure 10-34 Specifying the diagonal

"Apply" the set values.

Select the **Arc** softkey.



Select the **All parameters** softkey.

Create the 2nd arc.

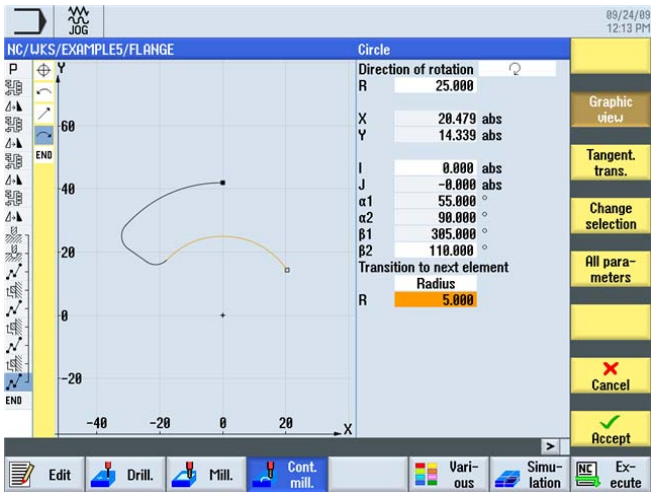
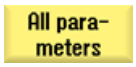


Figure 10-35 Specifying the arc

"Apply" the set values.

Select the **Diagonal** softkey.



Select the **All parameters** softkey.

Create the 2nd diagonal straight line.

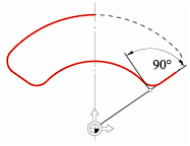


Figure 10-36 Specifying the diagonal

"Apply" the set values.

Select the **Arc** softkey.

Create the final arc.

Figure 10-37 Specifying the final arc

"Apply" the set values.

Accept the contour pocket into your process plan.

Create the following work steps without help:

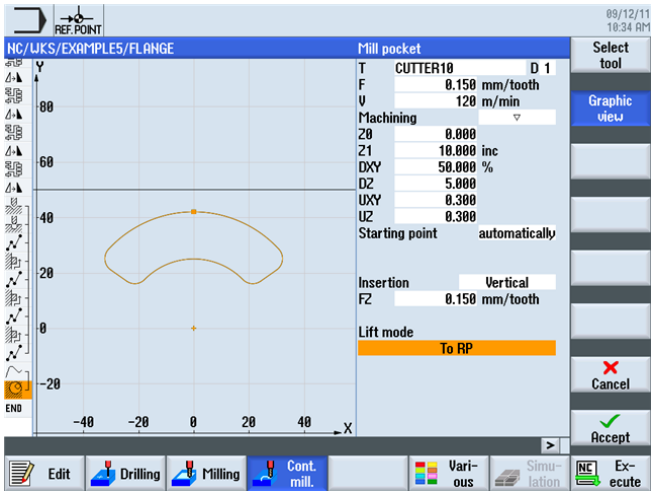


Figure 10-38 Roughing pockets

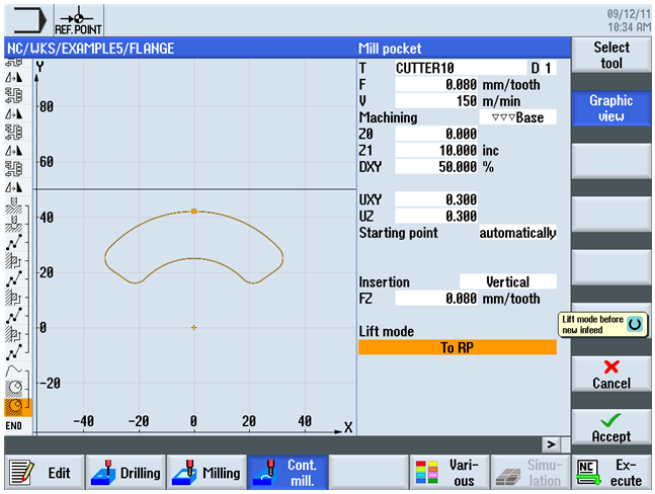


Figure 10-39 Finishing the pocket base

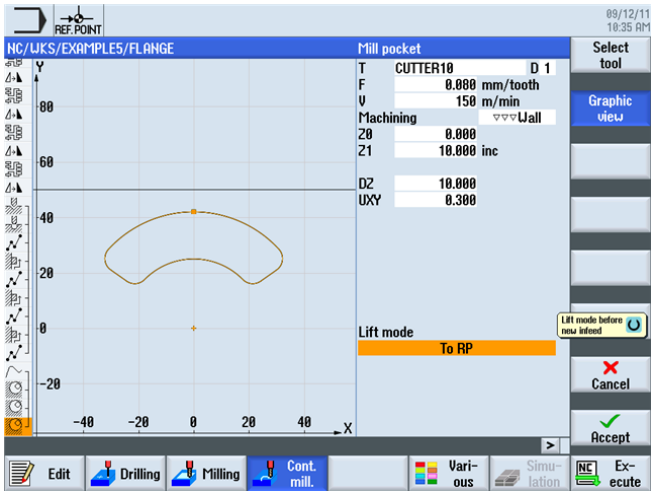


Figure 10-40 Finishing the pocket edge

To copy the created work step sequence for the machining of the three pockets, proceed as follows:

Mark

Now highlight the complete work step sequence describing machining of the pocket in the work step editor.

Copy

Copy the work step sequence to the clipboard.

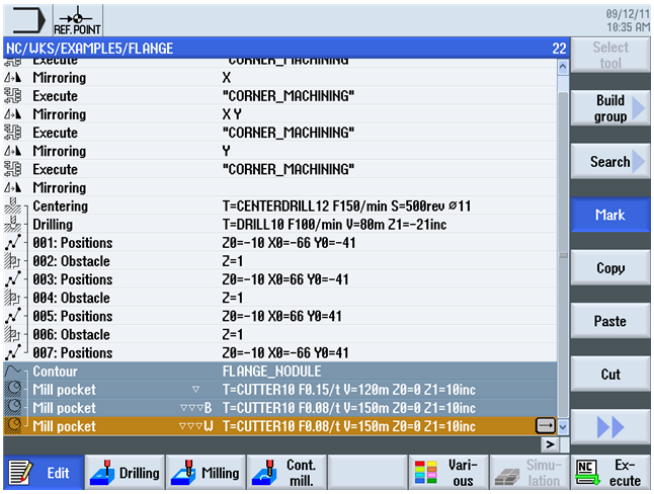


Figure 10-41 Copying the work steps

Vari- ous

Select the **Miscellaneous** softkey.

Transfor- mations

Select the **Transformations** softkey.

Rotation

The coordinate system is rotated around the Z axis by 120°.

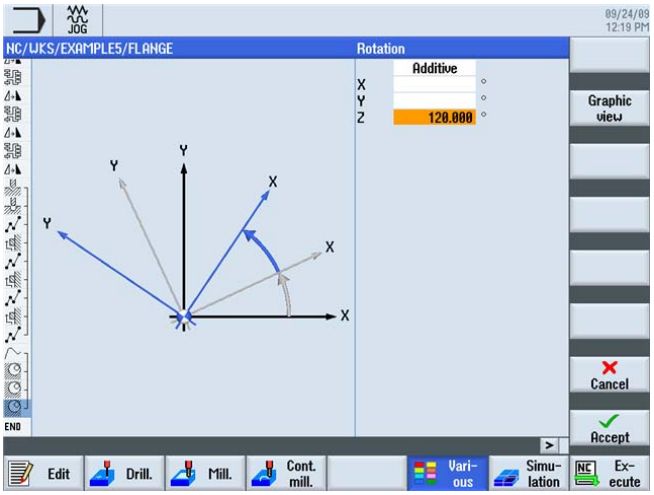


Figure 10-42 Rotation around the Z axis

Accept

Press ENTER to confirm your input.

Paste

Paste the copied work steps.

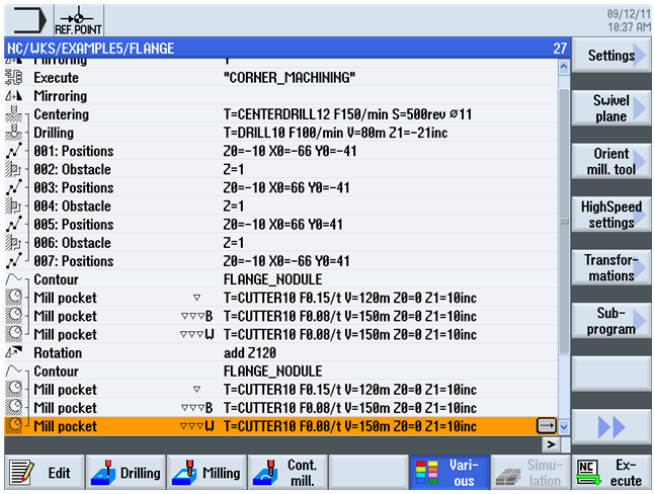


Figure 10-43 Pasting the copied work steps

Transfor-mations

Select the **Transformations** softkey.

Rotation

Enter another rotation by 120°.

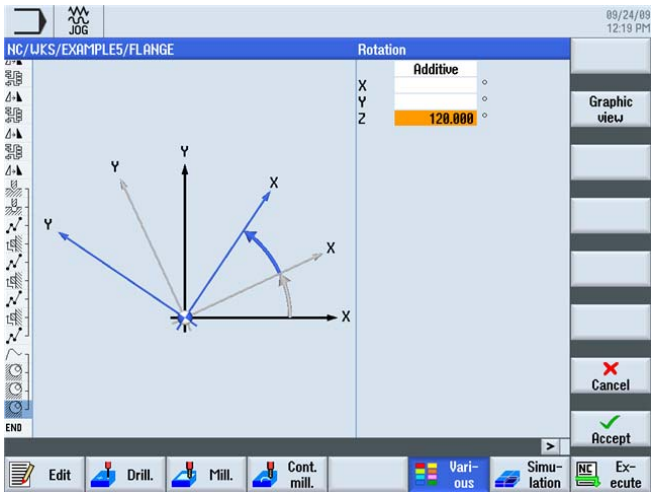


Figure 10-44 Rotation around the Z axis

Accept

Press ENTER to confirm your input.

Paste

Paste the copied work steps.

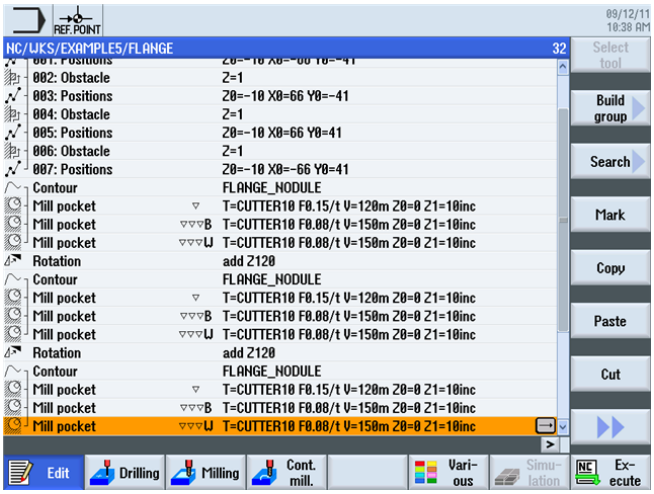


Figure 10-45 Pasting the copied work steps

Rotation

Select *New* and specify the value 0° to undo the rotation.

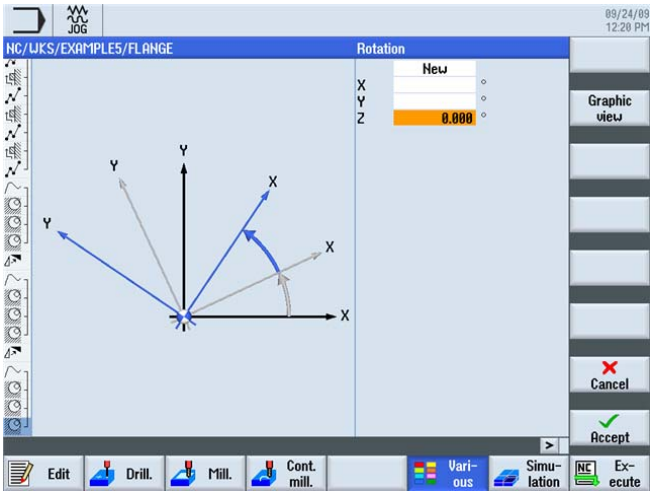


Figure 10-46 Undoing the rotation

Accept

Press ENTER to confirm your input.

10.6 Chamfering contours

Operating sequences

Chamfer the circular pocket last machined without help.

For chamfering, you will need a tool type which allows entering of an acute angle, in the example CENTERDRILL12.

Loc.	Type	Tool name	ST	D	Length	⌀	Tip angle				
1		CUTTER60	1	1	110.000	60.000		6		☑	☐
2		CUTTER16	1	1	110.000	16.000		4		☑	☐
3		CENTERDRILL12	1	1	120.000	12.000	90.0			☑	☐

Figure 10-47 Center drill

Select *Chamfering* for machining. The machining of the chamfer is programmed via the chamfer width (FS) and the insertion depth of the tool tip (ZFS).

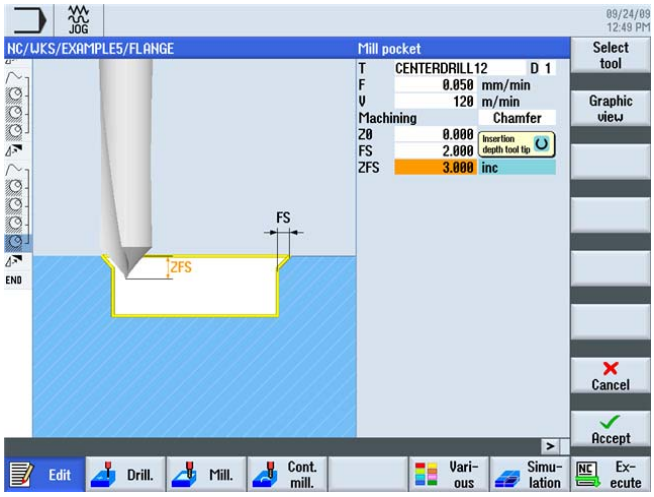


Figure 10-48 Chamfering

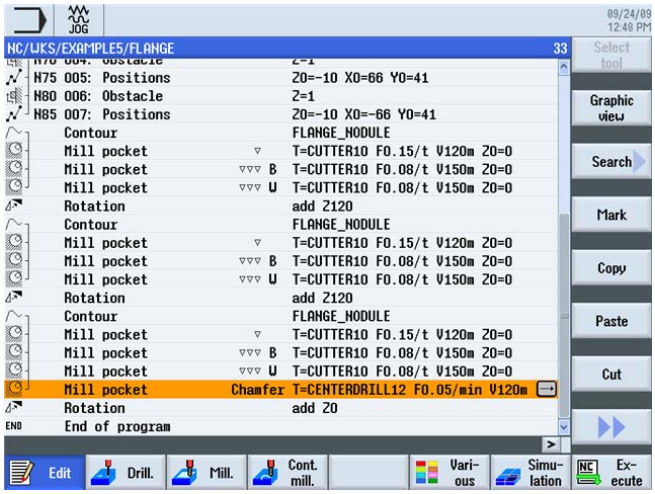


Figure 10-49 Work step "Chamfering" in the work step editor

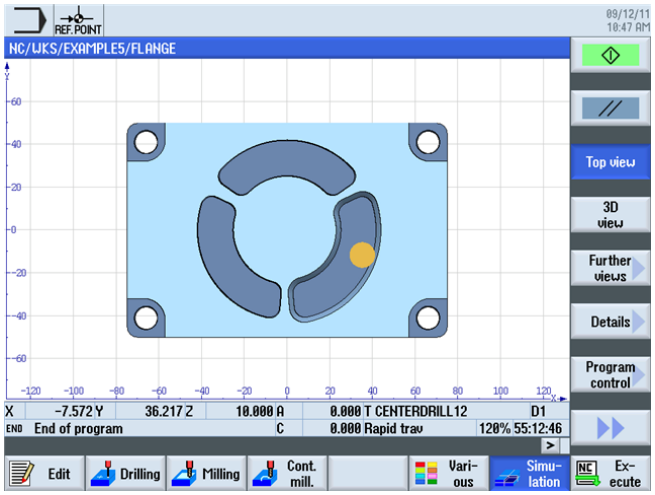


Figure 10-50 Top view on the chamfered contour

10.7 Longitudinal and circular grooves

Operating sequences

Finally, program the grooves. They will be positioned to the correct point by way of *Position pattern* and Positioning to *full circle*.

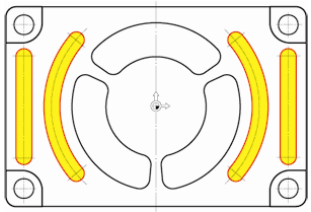
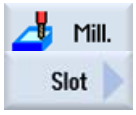


Figure 10-51 Longitudinal and circular grooves



Select the **Milling** softkey.

Select the **Groove** softkey.

Use the tool CUTTER6 (F = 0.08 mm/tooth and v = 120 m/min) for the roughing of the longitudinal grooves.

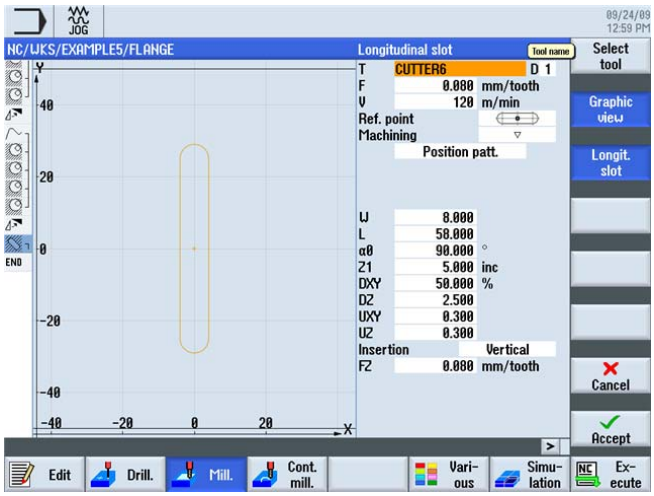


Figure 10-52 Roughing a longitudinal groove

"Apply" the set values.

Slot

Use the same tool (F = 0.05 mm/tooth and v = 150 m/min) for finishing.

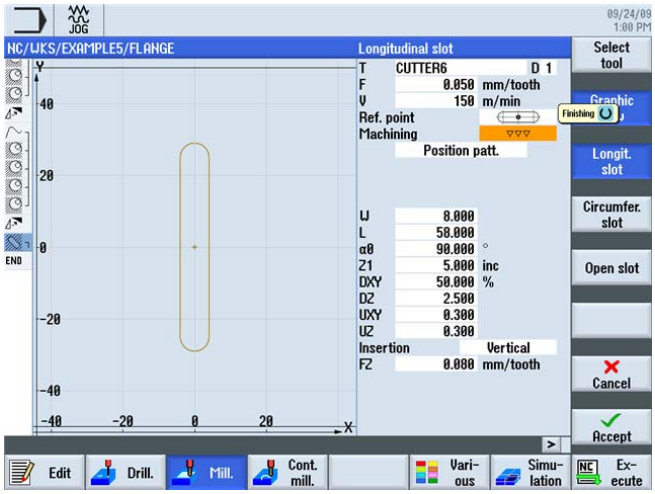


Figure 10-53 Finishing a longitudinal groove

Accept

"Apply" the set values.

Drill.

Select the **Drilling** softkey.

Positions

Subsequently, specify the positions of the longitudinal grooves. The reference point lies in the groove center.

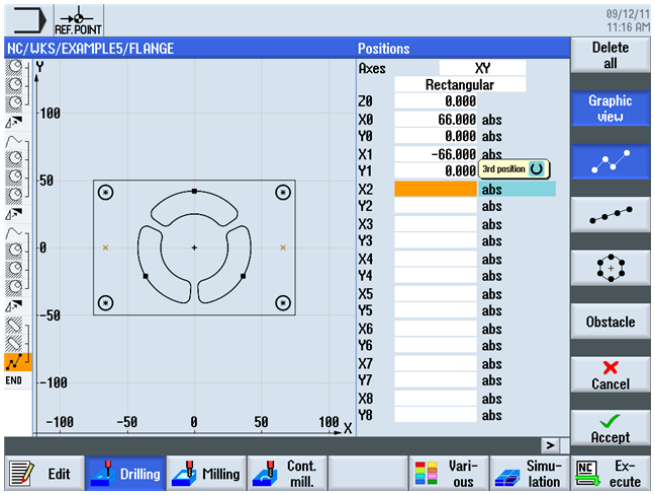


Figure 10-54 Specifying the positions for the longitudinal groove

Accept

"Apply" the set values.

Select the **Milling** softkey.

Select the **Groove** softkey.

Use the tool CUTTER6 (F = 0.08 mm/tooth,
FZ = 0.08 mm/tooth and v = 120 m/min) for the roughing of the circular grooves.

Thanks to the *Full circle* option, the circular grooves are positioned to each other automatically at the same distance
. The reference point in X/Y/Z refers to the center point of the circular grooves.

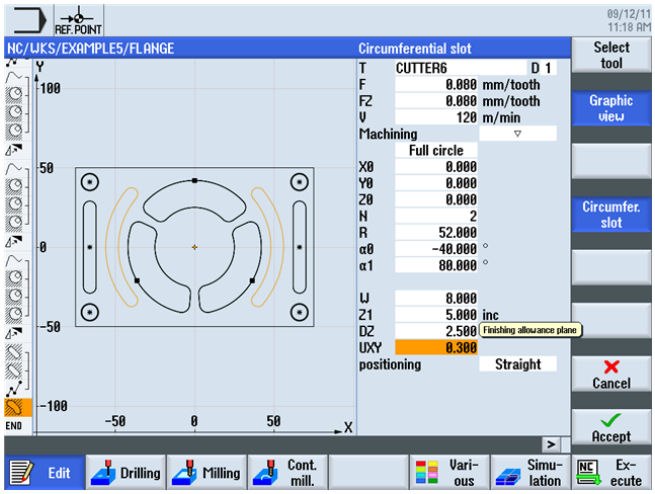


Figure 10-55 Roughing a circular groove

"Apply" the set values.

Select the **Groove** softkey.

Circumfer. slot

Use the same tool (F = 0.05 mm/tooth,
FZ = 0.05 mm/tooth and v = 150 m/min) for finishing.

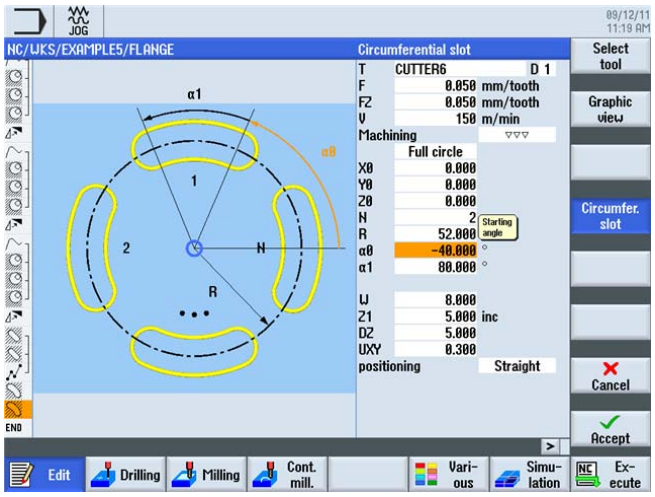


Figure 10-56 Finishing the circular groove

Accept

"Apply" the set values.

Process plan

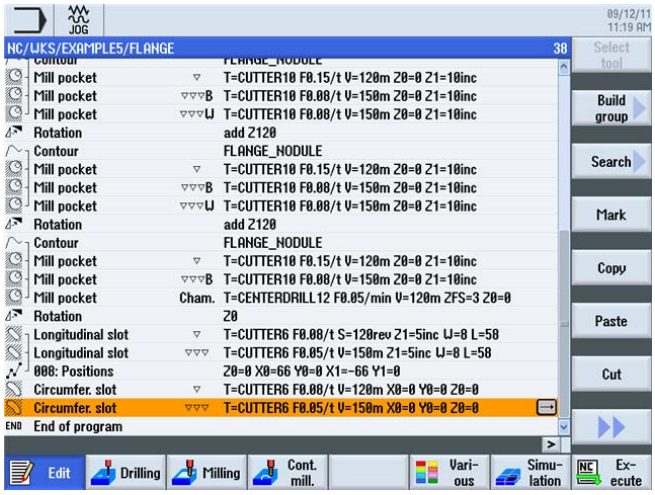


Figure 10-57 Extract from process plan

Broken-line graphics

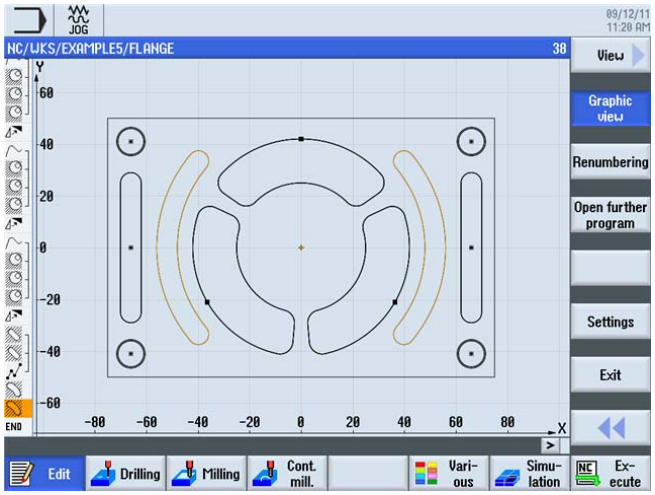


Figure 10-58 Broken-line graphics

Simulation in 3D display

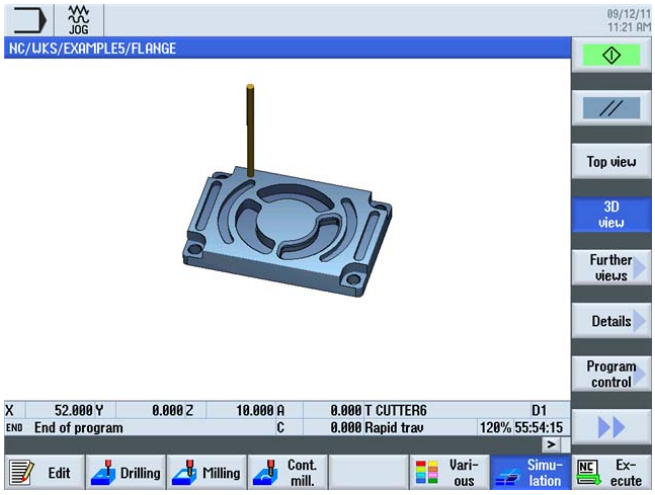


Figure 10-59 3D view

And now you can start manufacturing! 11

After you have acquired well-founded knowledge of the creation of process plans in ShopMill by working with the examples, we will now machine workpieces.

To machine a workpiece, proceed as follows:

Approaching reference point

After turning on the control system and before traversing the axes according to the process plans or traversing manually, you will have to approach the reference point of the machine. In this way, ShopMill will find the start for counting in the position measuring system of the machine.

Since approaching of the reference point is different depending on machine type and manufacturer, only a few hints can be given here for orientation:

1. If necessary traverse the tool to a free point in the work space from which traversing is possible in all directions without collision. Make sure that the tool is then not beyond the reference point of the corresponding axis (since reference point approach is only performed in one direction for each axis; otherwise, this point cannot be reached).
2. Perform the reference point approach exactly according to the specifications of the machine manufacturer.

Clamping the workpiece

To guarantee machining in accordance with the specified dimensions and, naturally, also for your own safety, it is imperative to clamp the workpiece tightly. As a rule, machine jaw vices or clamps are used.

Setting the workpiece zero

Since ShopMill cannot guess where in the work space the workpiece is located, you must determine the workpiece zero.

In the plane, the workpiece zero is set in most cases

- using either a 3D probe or
- an edge probe

through sampling.

In the tool axis, the workpiece zero is set in most cases

- using the 3D probe through sampling or
- using a tool through sampling

.

Note

When working with the measuring devices and measuring cycles, observe the manufacturer specifications.

Executing the process plan

Now the machine is prepared, the workpiece is set up, and the tools are gauged. At last we can start!

First select the program you want to execute in the Program Manager, e.g. INJECTION_FORM.

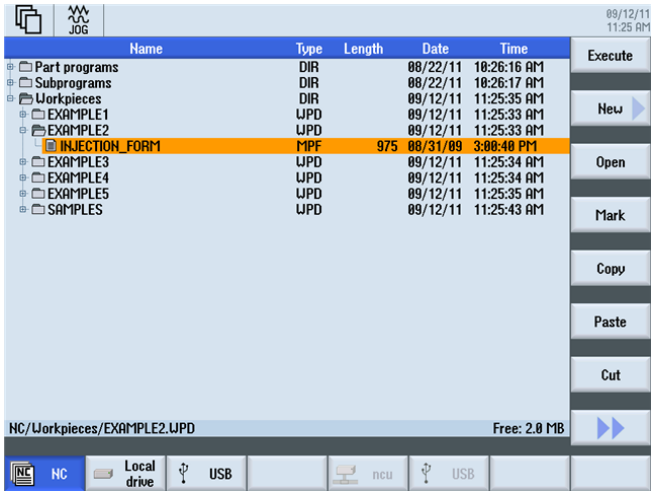


Figure 11-1 Selecting the program

Open the program.

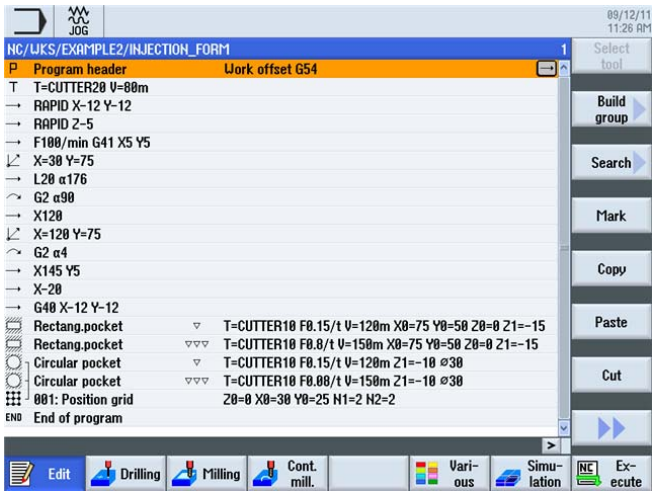
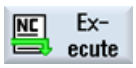


Figure 11-2 Opening the process plan



Select the **NC selection** softkey.

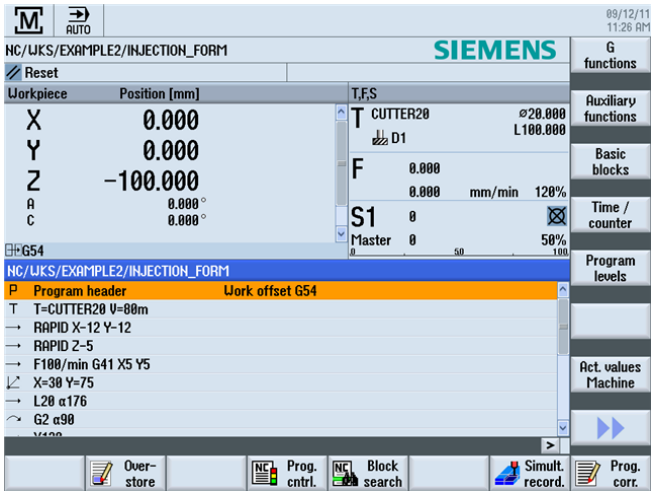
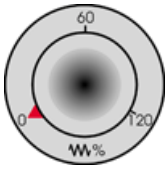


Figure 11-3 Executing



Due to the fact that the process plan has not yet been executed with control, turn the feedrate potentiometer to zero position to ensure that you keep everything under control from the beginning.

If you also want to see a simulation during machining, select the **Drawing** softkey before starting. Only then are all traversing paths and their effects are displayed.

Start machining and check the speed of the tool motions using the feedrate potentiometer.

How fit are you in ShopMill? 12

12.1 Introduction

The following four exercises are the basis for a personal test of your work with ShopMill. You will be assisted in all four exercises with an indication of one possible process plan each. The specified times are based on proceeding in accordance with this process plan. Consider the specified times as a rough guideline for your answer to the question above.

12.2 Exercise 1

Will you manage this task using ShopMill within 15 minutes?

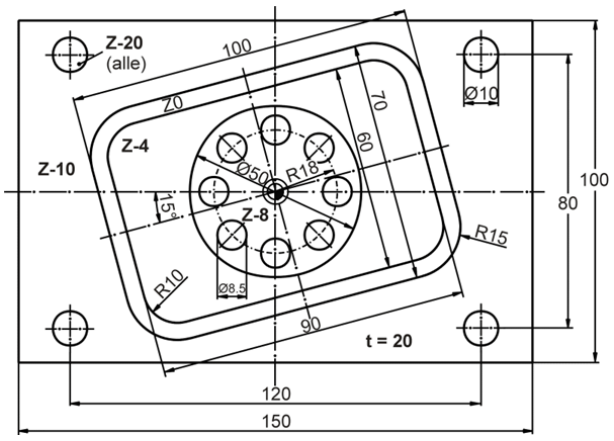


Figure 12-1 Workshop drawing DIYS1

Notes

The rotated rectangular pocket shown above has been designed in the original coordinate system. The starting point lies first on zero. Subsequently, an auxiliary straight line is programmed at an angle of 15° to the edge of the pocket. The coordinates of this end point are the starting point for the actual construction. The auxiliary straight line must be deleted. ShopMill also provides other paths to the objective, e.g. with the "Rotation" function or the "Rectangular spigot" cycle. Test how you achieve the objective fastest and with which method you achieve the shortest machining time.

Model

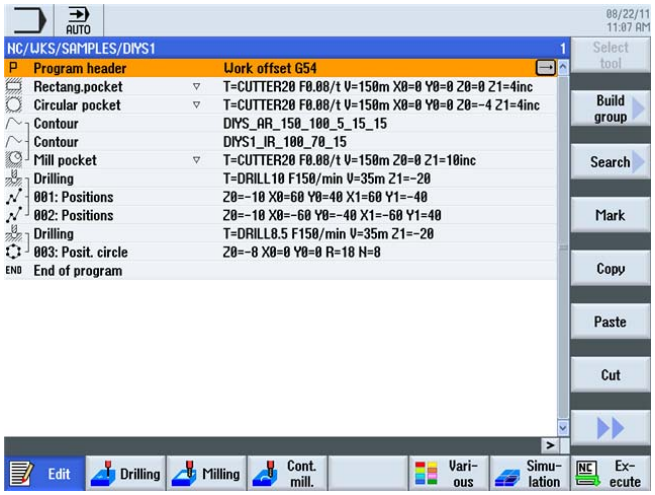


Figure 12-2 Process plan

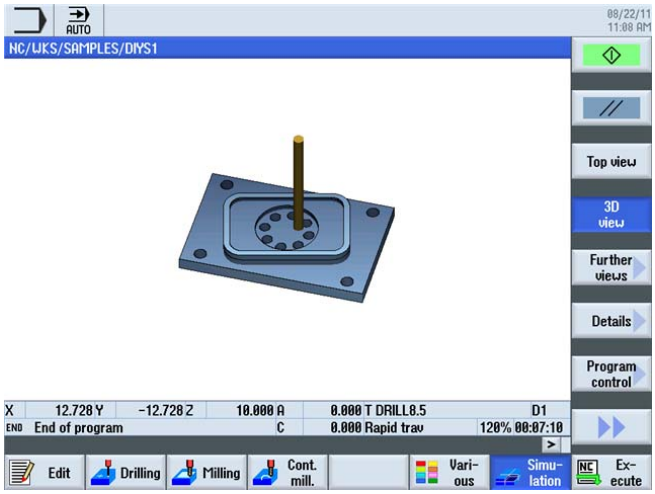


Figure 12-3 Workpiece simulation

12.3 Exercise 2

Will you manage this task using ShopMill within 20 minutes?

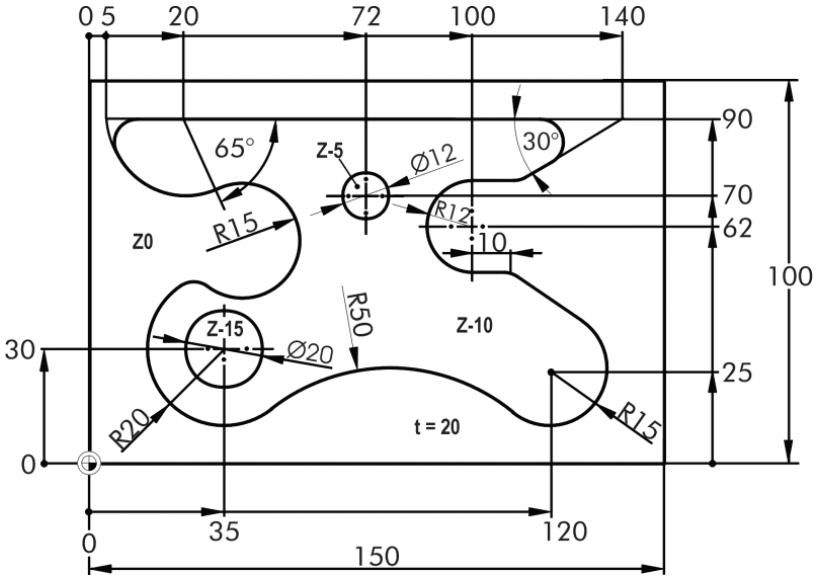


Figure 12-4 Workshop drawing COMPLEX_POCKET

Notes

Even though it looks complex: This contour is no problem with ShopMill. And automatic removal of residual material can here be applied optimally. Compare the machining times if you were to use CUTTER10 for stock removal.

Notes regarding the contour:

- Design the contour in the counterclockwise direction.
- The aperture angle of the top left arc is 115°.

Model

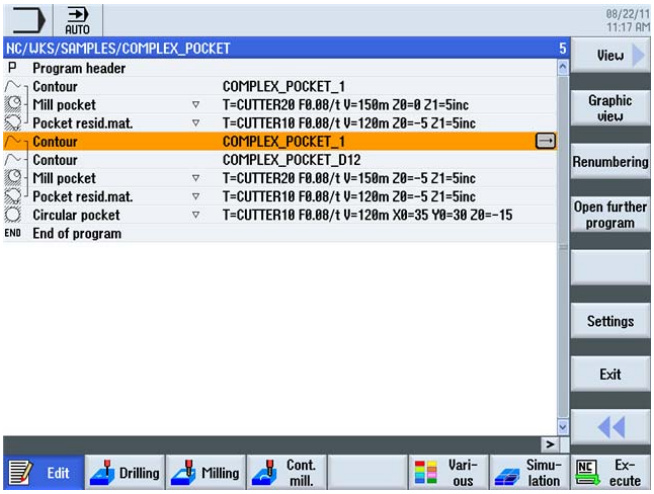


Figure 12-5 Process plan

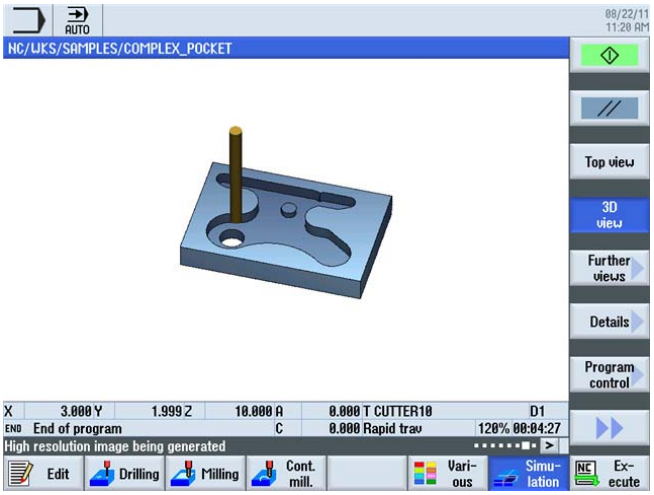


Figure 12-6 Workpiece simulation

12.4 Exercise 3

Will you manage this task using ShopMill within 30 minutes?

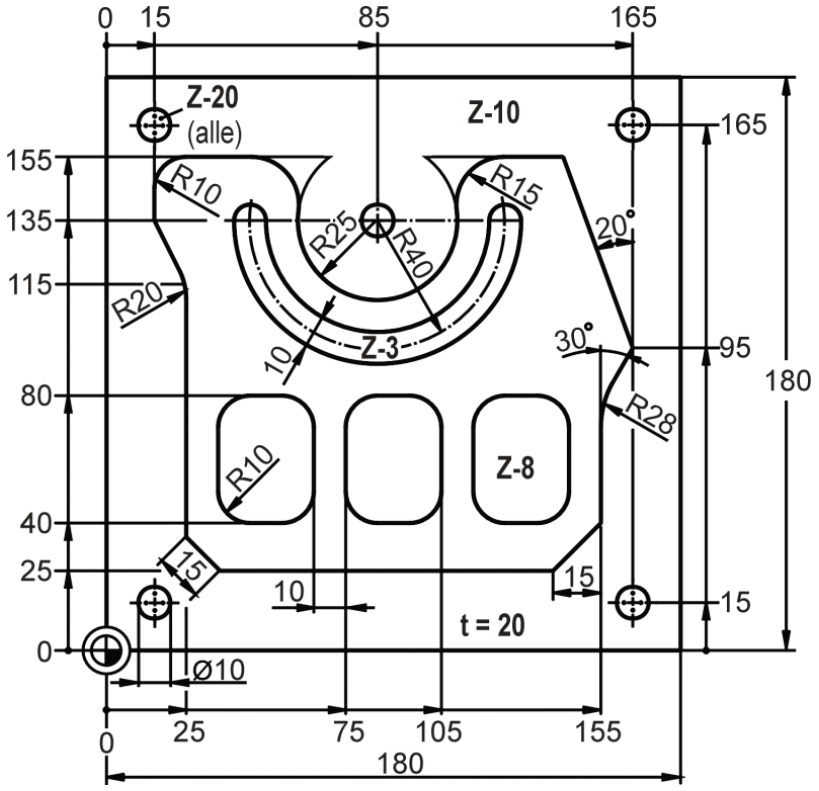


Figure 12-7 Workshop drawing PLATE

Notes

In this sample process plan, the area around the island has first been premilled roughly using the "Rectangular spigot" cycle from the "Milling" menu. The rectangle described in this cycle is approached along a circle and will reach the contour at the point described by the length and angle of rotation. The rectangle is traveled completely once and left at the same point again along a circle. Approach and retraction radii result from the geometry of the residual spigot.

Model

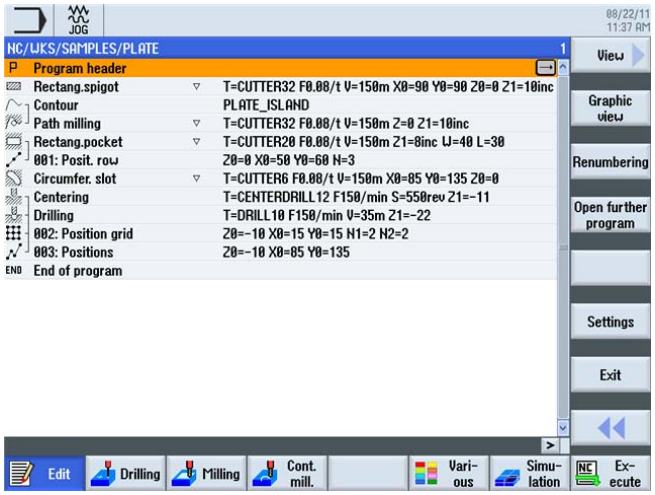


Figure 12-8 Process plan

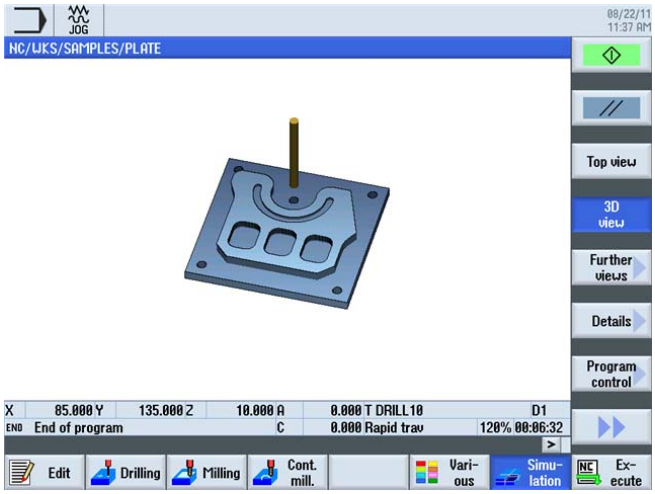


Figure 12-9 Workpiece simulation

12.5 Exercise 4

Will you manage this task using ShopMill within 30 minutes?

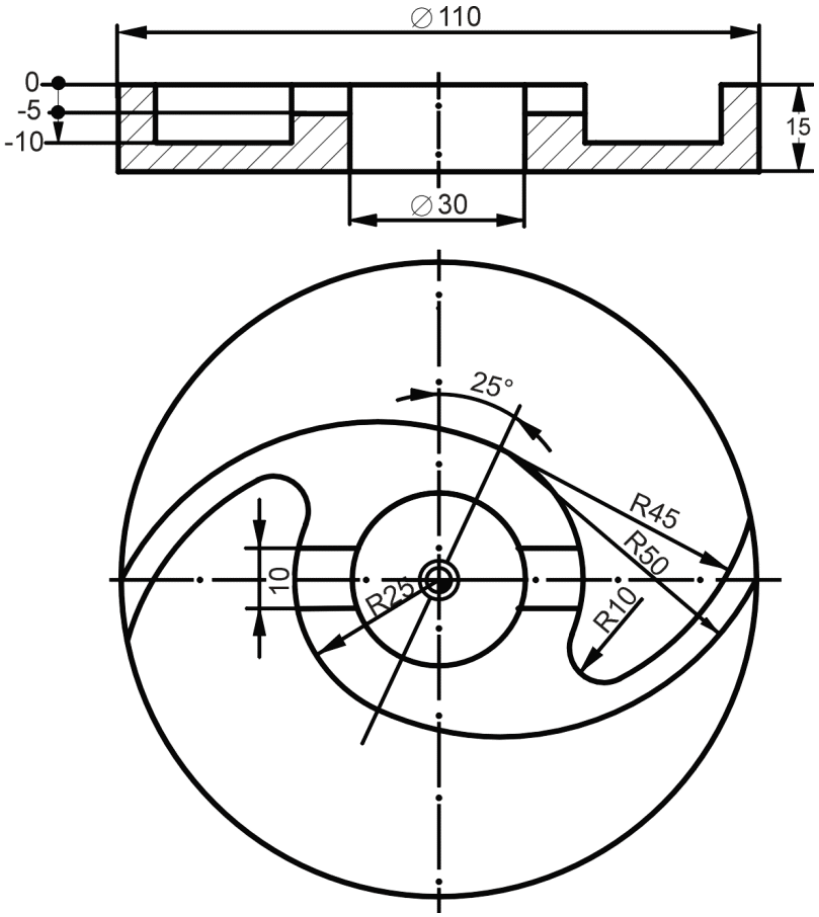


Figure 12-10 Workshop drawing WING

Notes

In this sample process plan, the circular outside contour is milled using the "Circular spigot" cycle. The functioning corresponds mainly to that of the rectangular spigot (see Sample Pattern Process Plan for Exercise 3). The common center point of the two arcs R45 and R50 (= starting point for the actual construction) is specified using polar coordinates (25 mm at 65° referred to the pole point at X0/Y0).

With software version V6.4 and higher, the "Milling" menu also provides a flexible "Engraving" cycle.

Model

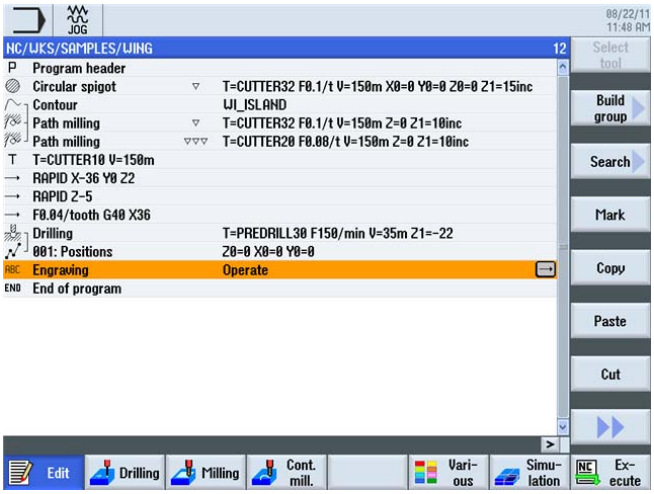


Figure 12-11 Process plan

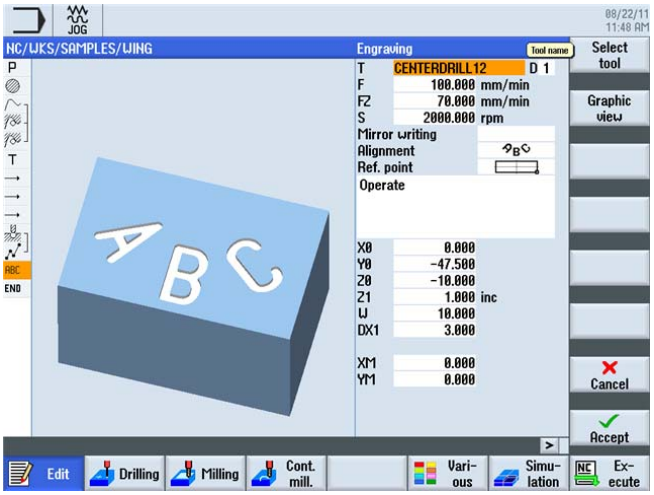


Figure 12-12 Specifying engraving

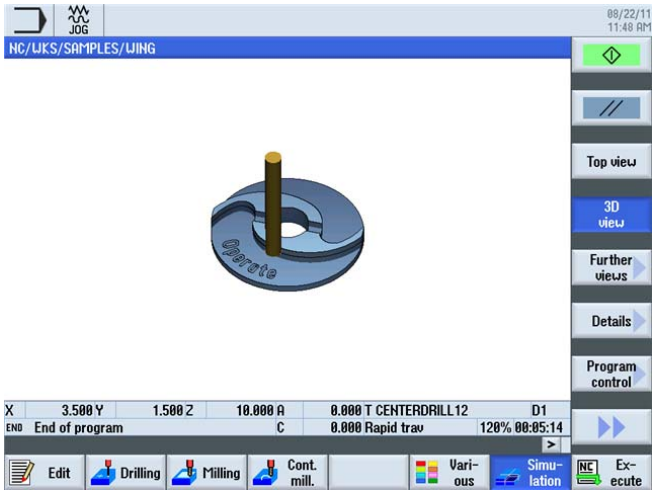


Figure 12-13 Workpiece simulation

Index

N

O

P

R

S

T

W