SIEMENS

SCE Training Curriculums

Siemens Automation Cooperates with Education | 02/2016

CNC Technology Module 700-030
ShopMill Basics

Cooperates with Education
Automation
SIEMENS

Suitable SCE trainer packages to accompany these training curriculums

- **SinuTrain for SINUMERIK Operate V4.7 Basic** – free download with no time limit
  www.siemens.com/sinutrain-downloads
- **SinuTrain Classroom License for SINUMERIK Operate V4.5 –** 6 single licenses + 40 student licenses;
  Order no.: 6FC5870-1TC41-0YA0
- **SinuTrain Student Licenses for SINUMERIK Operate V4.5 –** 300 hours **–** 20 student licenses;
  Order no.: 6FC5870-1SC41-0YA0
- **SinuTrain Classroom License for SINUMERIK Operate V4.4 –** 16 single licenses + 32 student licenses; Order no.: 6FC5870-1TC40-1YA0
- **SinuTrain Student Licenses for SINUMERIK Operate V4.4 –** 300 hours **–** 32 student licenses;
  Order no.: 6FC5870-1SC40-1YA0

Note that these trainer packages will be replaced with successor packages when required.
You can find an overview of the currently available SCE packages at: siemens.com/sce/tp

In-service training courses

Get in touch with your regional SCE contact for information on regional Siemens SCE in-service training courses.
siemens.com/sce/contact

Additional information on SinuTrain

In particular: Downloads, Getting Started, Videos, Tutorials, Manuals and Programming Guides.
siemens.com/sce/sinutrain

Further information on SCE

siemens.com/sce

Notes on use

The SCE training curriculum for CNC Technology has been created for the "Siemens Automation Cooperates with Education (SCE)" program especially for educational purposes for public educational and R&D institutions. Siemens AG assumes no responsibility for the content.

This curriculum may be used only for initial education with respect to Siemens products/systems. That is, it may be copied in part or in whole and handed out to trainees for use within the framework of their education. Transmission and reproduction of this curriculum as well as communication of its content is permitted within public educational institutions for educational purposes.

Exceptions require written consent from the Siemens AG. Send all related requests to scesupportfinder.i-ia@siemens.com.

Use for industry customers is expressly prohibited. Commercial use of the curriculum is not permitted.

We would like to thank Michael Dziallas Engineering, MOSER CNC Training and all those involved for their support in creating this curriculum.

Contents

1. Objective

In this module, you will learn how you can easily go from a drawing to a finished workpiece with the help of the OPERATE machining plan sequential programming interface ShopMill.

2. Introduction

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 and productive programming methods are available for each area. With ShopMill, SIEMENS therefore offers a programming solution tailored to the workshop that allows quick programming of machining steps in line with real-world requirements, ranging from the machining of single parts up to small batches. In conjunction with SINUMERIK Operate, the new operator interface for the controller, intuitive and effective working in the workshop is possible even for series production.

Creation of a machining plan instead of programming is the solution.

The creation of a machining 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 machine with ShopMill thanks to the integrated, powerful tools for creating traversing paths. For this reason:

Easier and faster from the drawing to the workpiece – with ShopMill!

Although ShopMill is easy to learn, this ShopMill Learn-/Training Documentallows you to get started in this world even faster. Before it comes to the actual work with ShopMill, however, important basics are discussed in the first sections:

- First, we will show you the advantages of ShopMill.
- Then we show you the basics of operation with SINUMERIK Operate.
- The basics of geometry and technology for machining will be explained for beginners.
- 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 strokes are specified. Later, you will be prompted to proceed on your own.
- You will then learn how to machine in Automatic mode using ShopMill.
- Note that the technology data used here only serves as an example due to the wide variety of situations in the workshop.

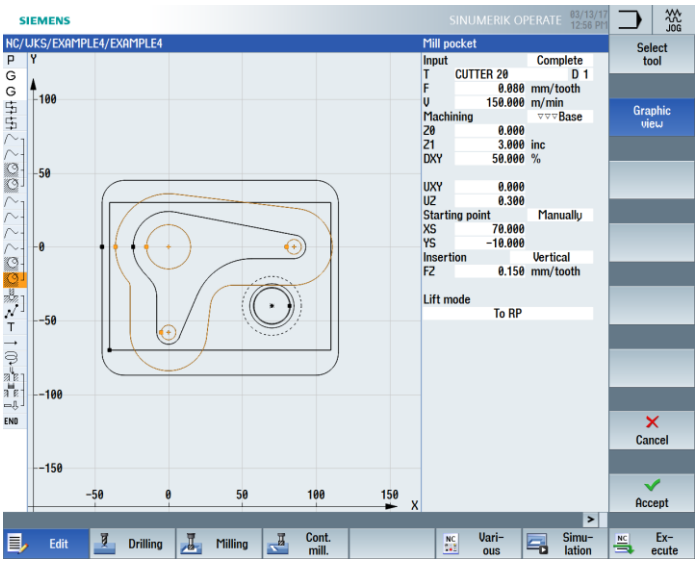
Just as ShopMill was created with the help of skilled workers, this training curriculum was also elaborated by practitioners. In this sense, we wish you much pleasure and success in your work with ShopMill.

3. Advantages of working with ShopMill

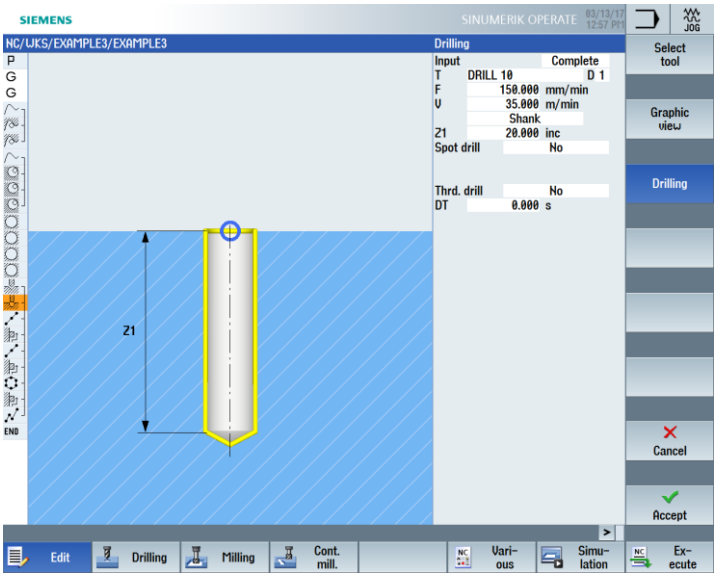
This section shows you the special advantages of working with ShopMill.

3.1 You save training time:

- ShopMill does not use any foreign-language terms you would otherwise have to learn, and all necessary inputs are prompted in plain text.



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



- You may also integrate DIN/ISO commands into the graphical machining 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 machining step and the workpiece graphic at any time when creating a machining plan.

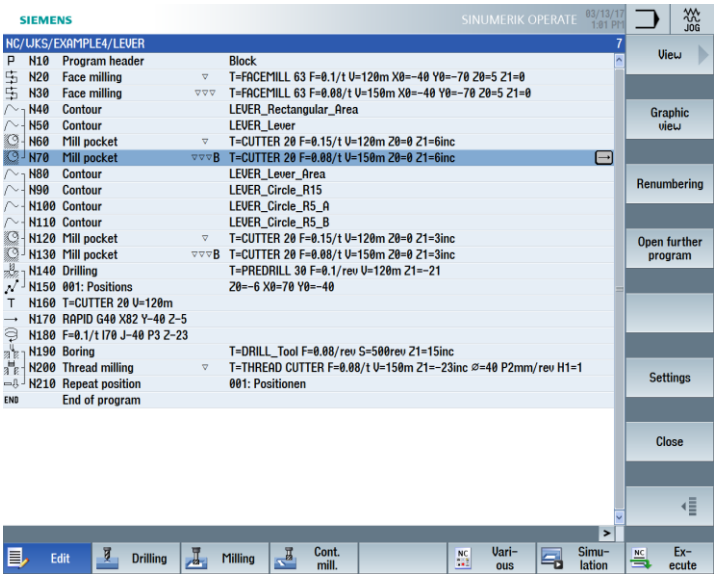


Figure 2-1 Machining step in the machining plan

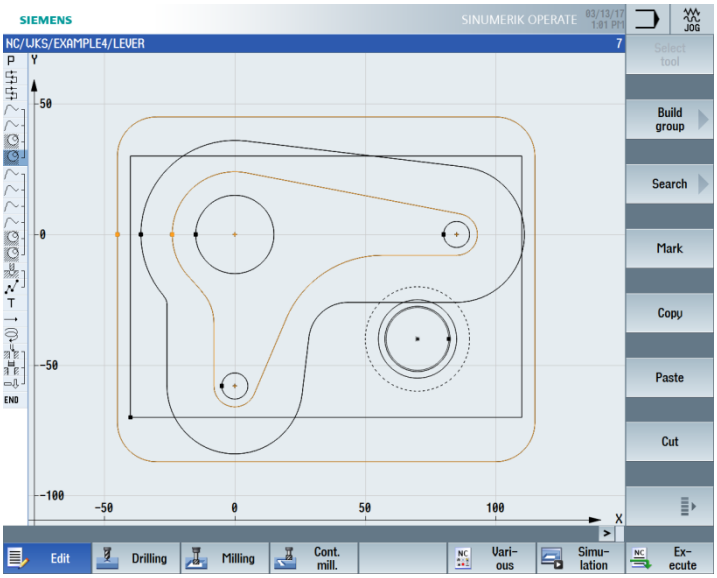
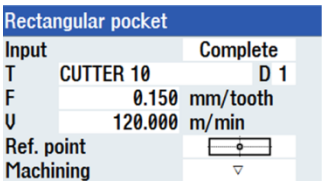
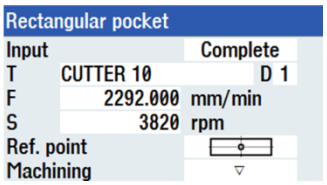


Figure 2-2 Graphic view

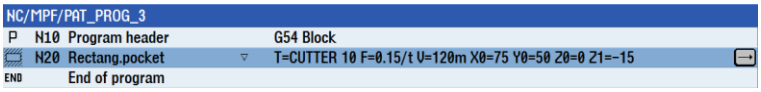
3.2 You save programming time:

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

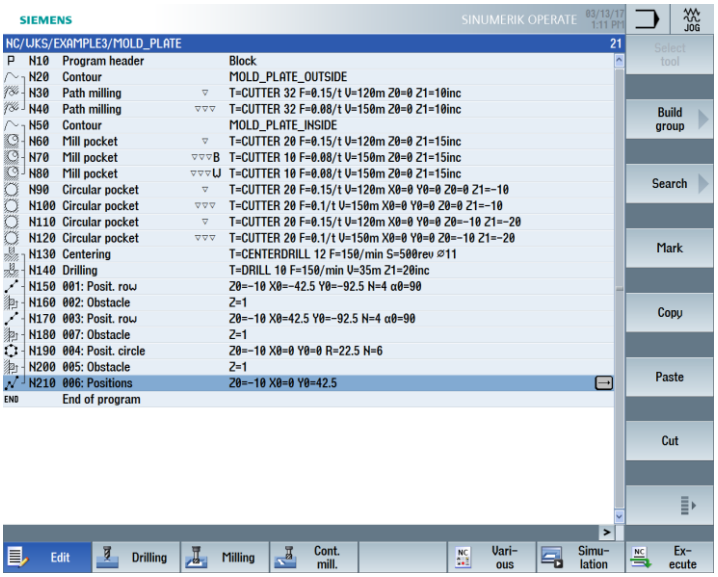




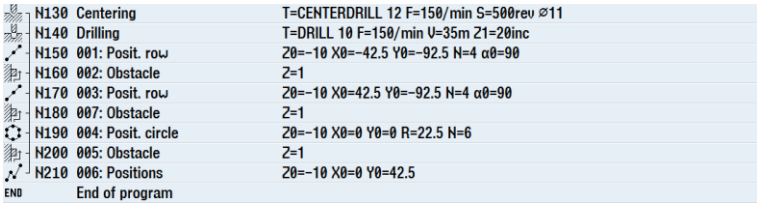
- ShopMill enables you to describe a complete machining operation with one machining step, and the required positioning motions (in this case, from the tool change point to the workpiece and reverse) are created automatically.



- All machining steps are represented by ShopMill in a compact and clear fashion in the graphic machining plan. This provides you a complete overview and thus better editing possibilities even for comprehensive machining sequences.



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



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

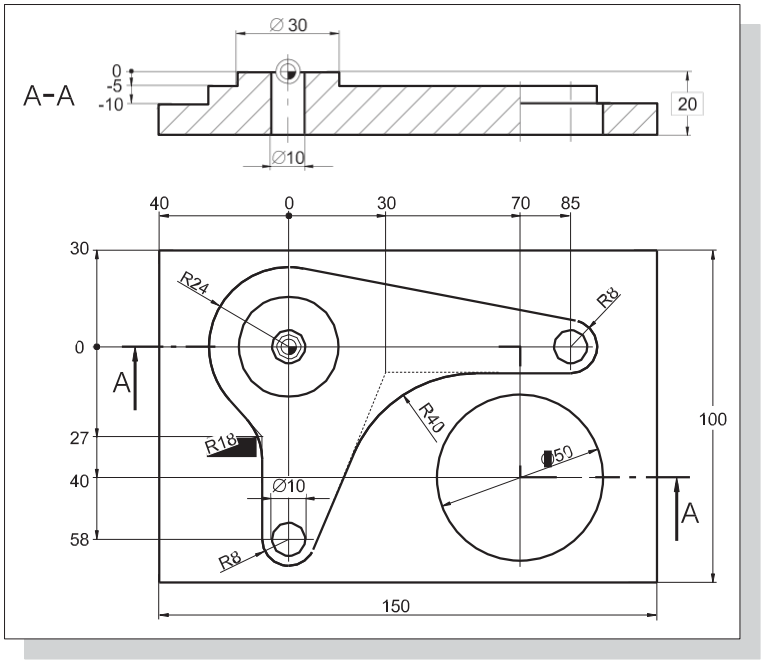


Figure 2-3 Technical drawing

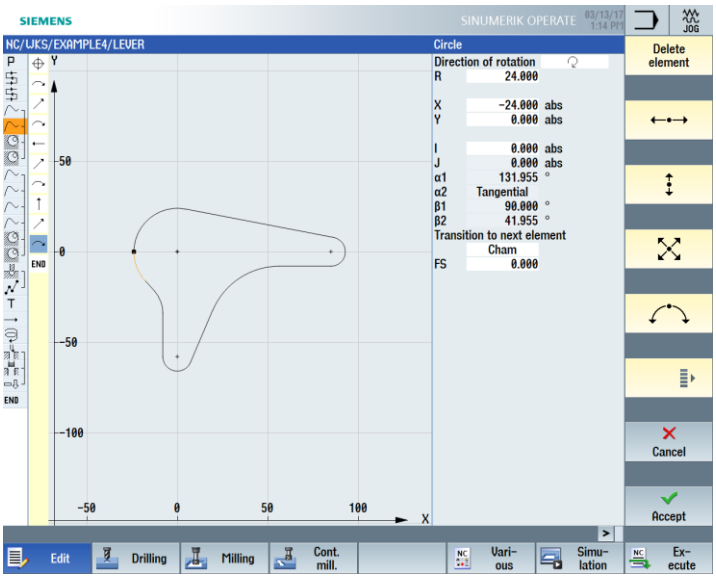


Figure 2-4 Input screen form

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

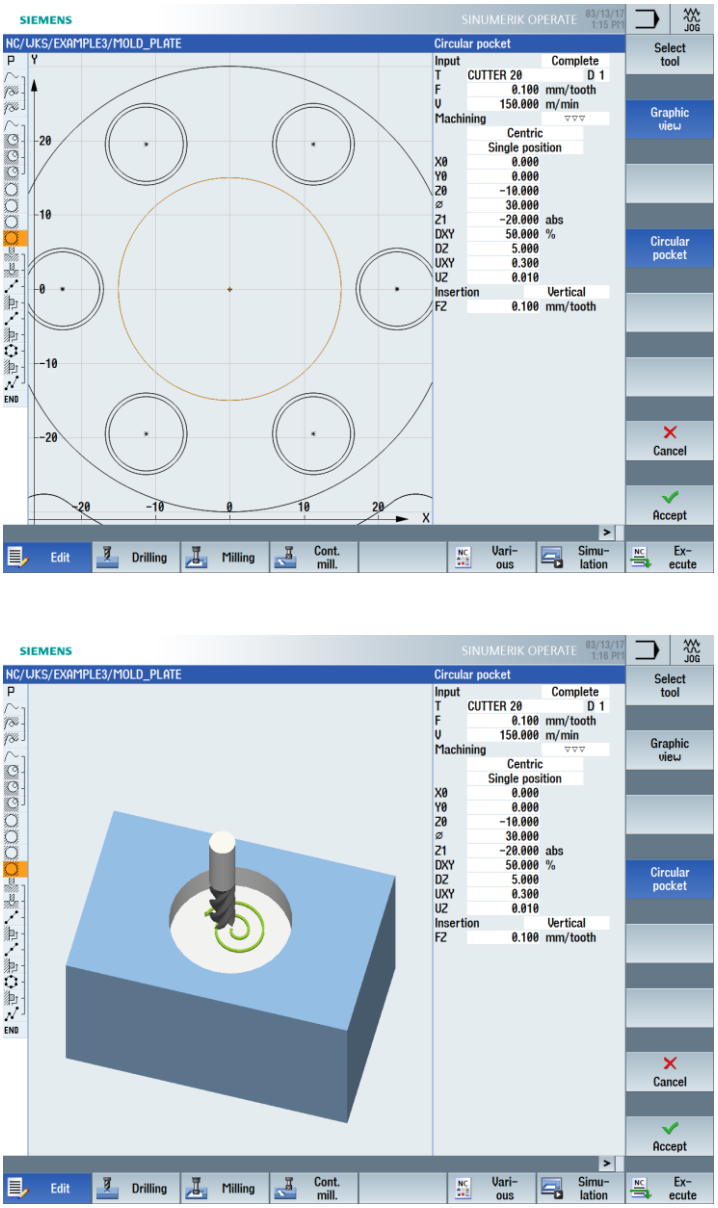
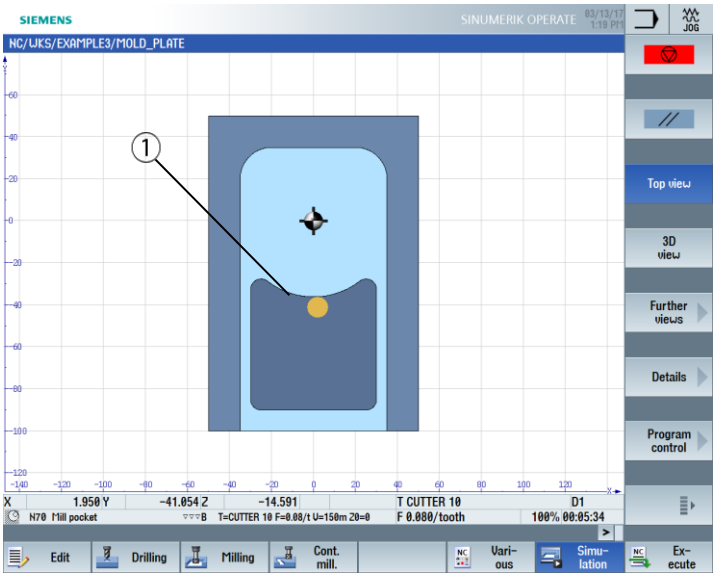


Figure 2-5 Parameter screen form with help screen

- Creating a machining plan and machining are not mutually exclusive. With ShopMill, you can create a new machining plan in parallel with machining.

3.3 You save machining time:

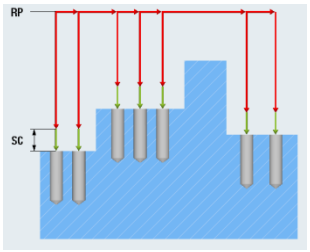
- You need not take into account the pocket radii when selecting the milling cutter for machining contour pockets: Any residual material ① is detected and removed automatically using a smaller milling cutter.



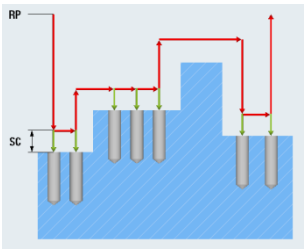
- There are no unnecessary infeed motions between the retraction plane and machining plane when positioning the tool. This is made possible by the settings "Retract to retraction plane "(RP) and "Optimized retraction".

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

Retraction to retraction plane (RP)



Retraction to retraction planes = machining time saving



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

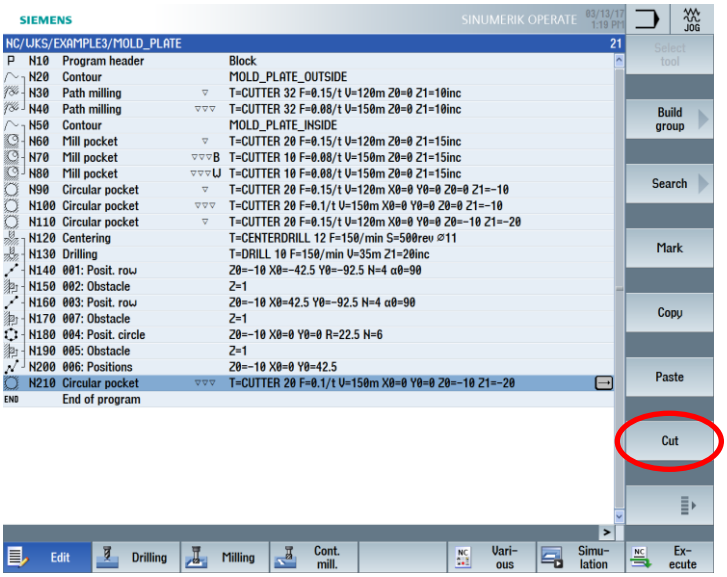


Figure 2-6 Original machining sequence

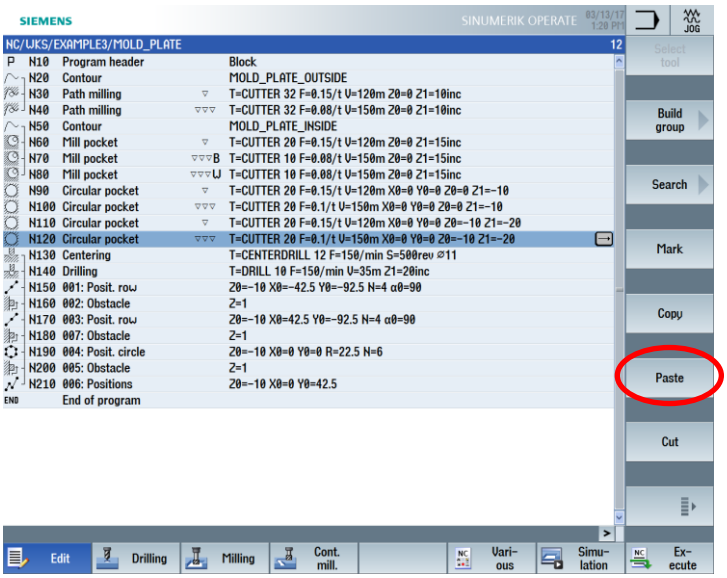


Figure 2-7 Optimized machining sequence with cutting and pasting the machining step

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

4. To ensure that everything function smoothly

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

4.1 The operation of ShopMill

Powerful software is important, but it also has to be intuitive to operate. Regardless of whether you work with SINUMERIK 840D sl or SINUMERIK 828D as shown here, you are always assisted by the clearly laid-out machine operator panel.

The operator panel consist of three 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> (also called Toggle key) • Selects a listed value.
	Cursor keys • The cursor is moved using the 4 cursor keys. • Use the <Cursor right> key shown here to open a directory or program (e.g. a cycle) in the editor in edit mode.
	<PAGE UP> • Scroll upwards in a menu screen.
	<PAGE DOWN> • Scroll downwards in a menu screen.
	<END> • Moves the cursor to the last text box 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> • Press the key to enter Edit mode. Press the key again to exit Edit mode and go to Navigation mode.
	<INPUT> • Complete input of a value in the text box. • Open a directory or program.

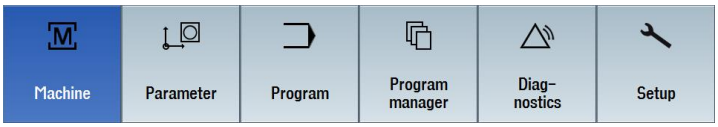
The actual function selection in ShopMill is performed using the keys located around the screen. Most of them are assigned directly to the individual menu commands. Since the contents of the menus change depending on the situation, the term "softkeys" is used.

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 opened with this key at any time – irrespective of the operating area you are in at the moment.

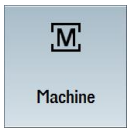
Main menu



4.2 The contents of the main menu

4.2.1 Machine

Machine – Manual



Select the "Machine" softkey.

Press the "JOG" key.

Here, the machine is set up and the tool is moved in manual operation. It is also possible to measure tools and to set workpiece zeros.

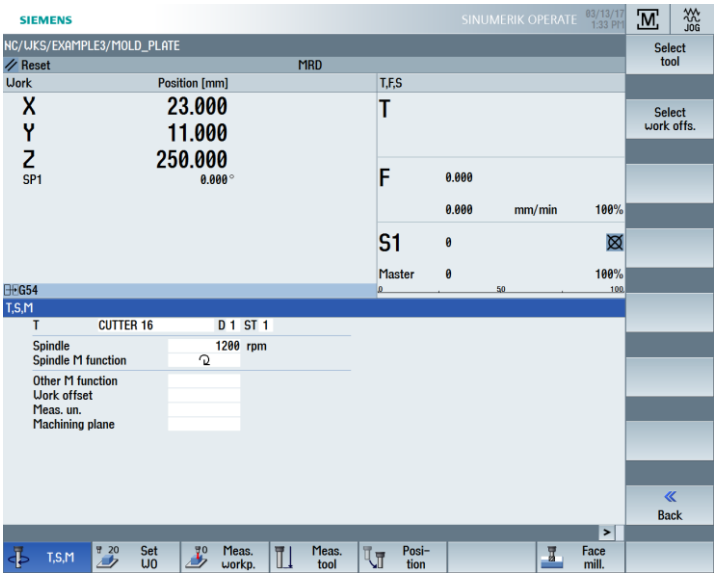
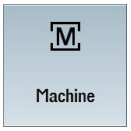


Figure 3-1 Call of a tool and input of technological values

Figure 3-2 Specification of a target position

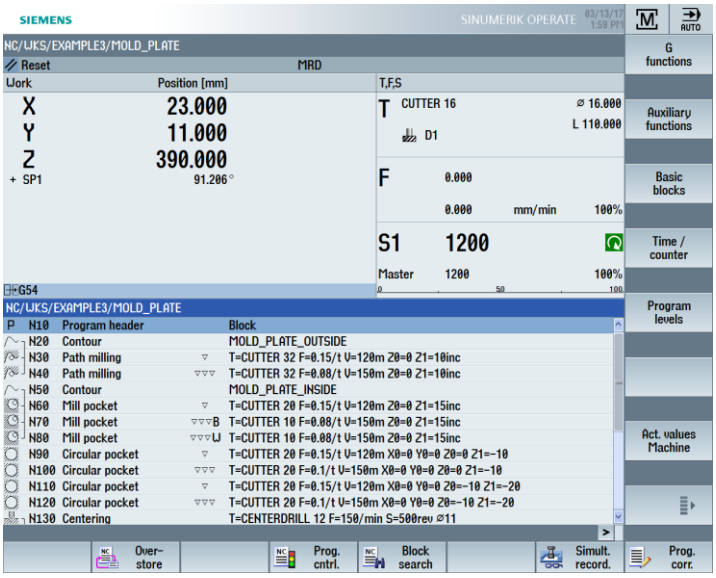
Machine – Auto



Select the "Machine" softkey.

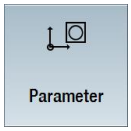
Press the "AUTO" key.

During the machining, the current machining step is displayed. It is possible to switch to a simultaneously running simulation at the press of a key ("Simult. record."). During execution of a machining plan, you may add machining steps or start a new machining plan.



4.2.2 Parameters

Parameter lists



Here, data for the tool management and for programs can be edited.

Tool lists

No cutting without tools.

These can be managed in a tool list.

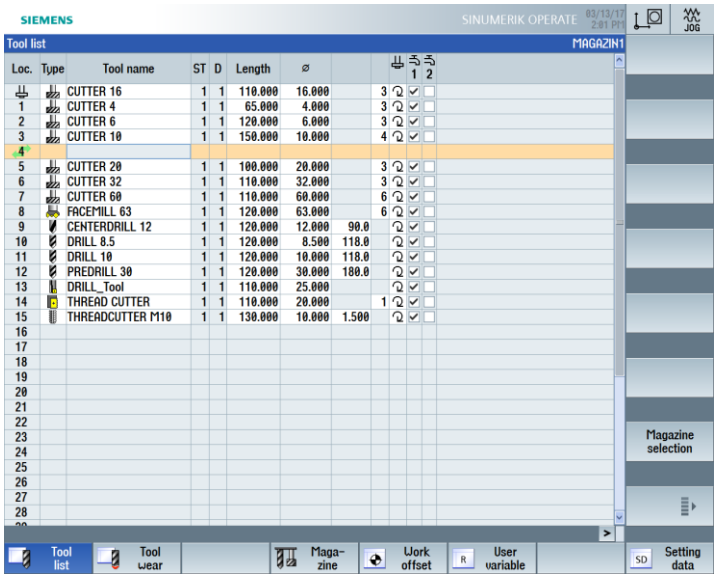


Figure 3-3 Tool list

Magazine

Tools can be organized in a magazine.

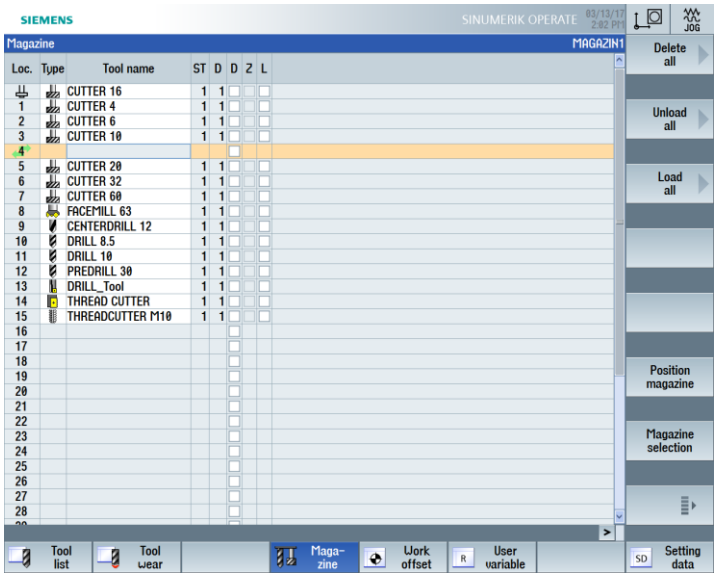


Figure 3-4 Magazine

Work offsets

Zero points are saved in a clearly laid-out work offset table.

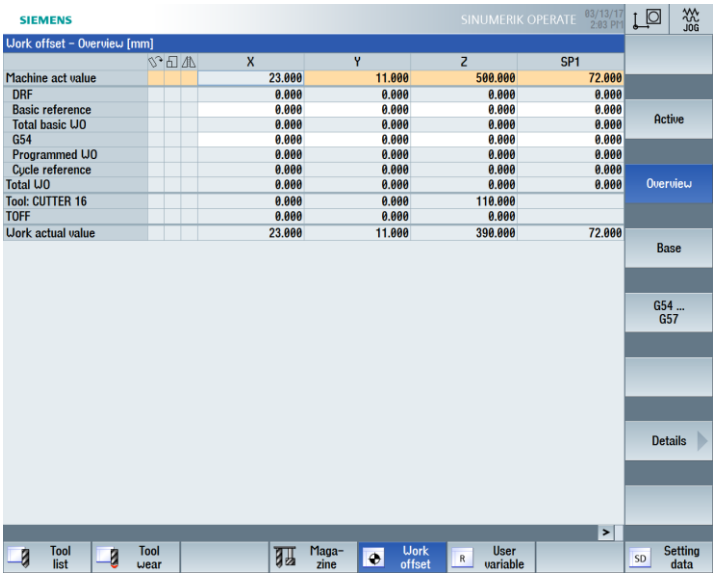


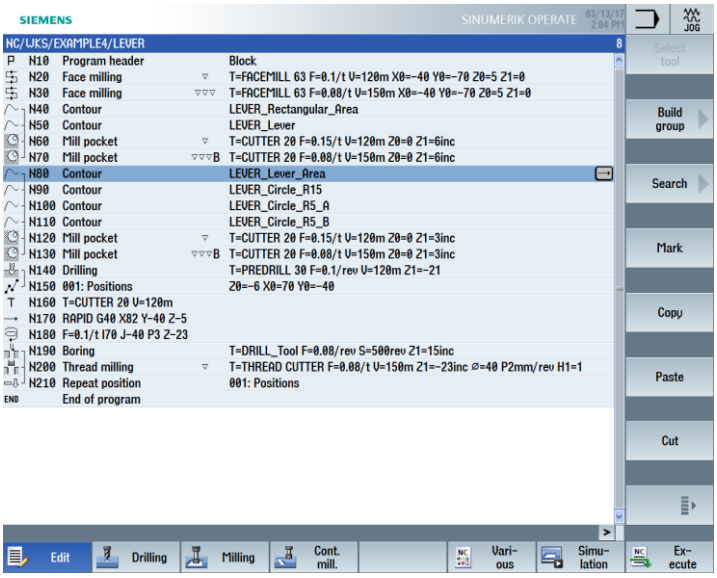
Figure 3-5 Work offsets

4.2.3 Program

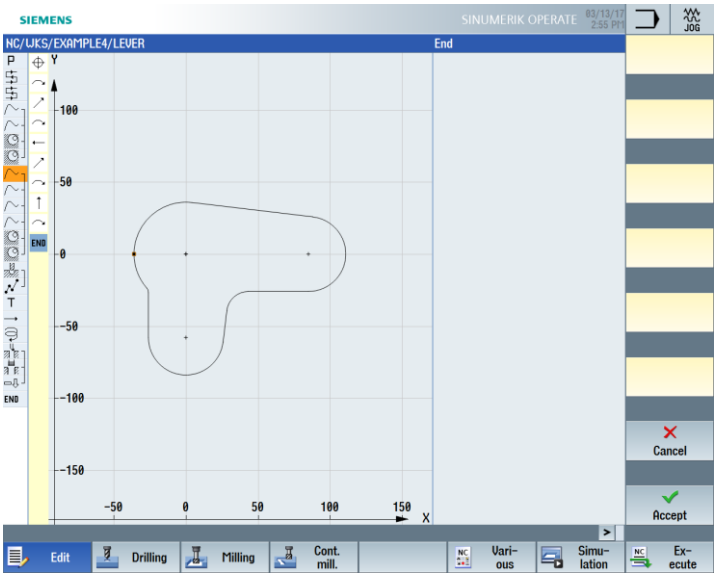
Editing programs

This key can be used to edit programs.

If you have created a ShopMill program in the Program Manager, you can now create the machining plan with the complete machining sequence for the appropriate workpiece. The practical knowledge of the skilled worker is required to create the optimal sequence.



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 of the dovetailing of geometry and technology:

Contour

Path milling incl. approach and retraction strategies

Circular pocket including technology and position

Boring technology

Position for boring

Centering technology

Drilling technology

Positions for centering and drilling

This relationship between geometry and technology is represented very clearly in the graphical display of the machining steps by a "bracketing" of the corresponding symbols. The "bracketing" signifies a linking of geometry and technology to form a machining step.

Simulating programs

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

- 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:

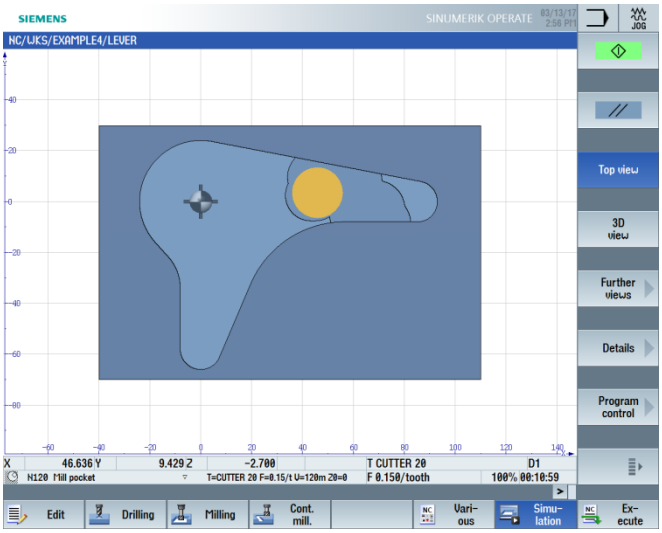


Figure 3-6 Top view

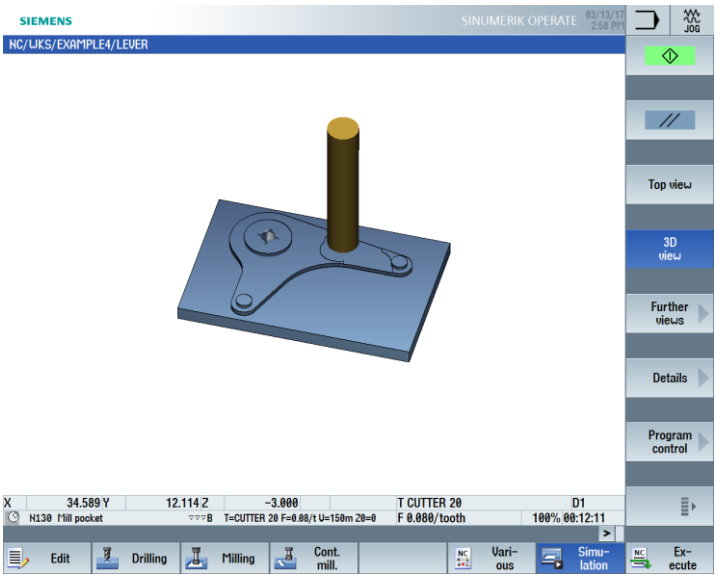


Figure 3-7 3D view

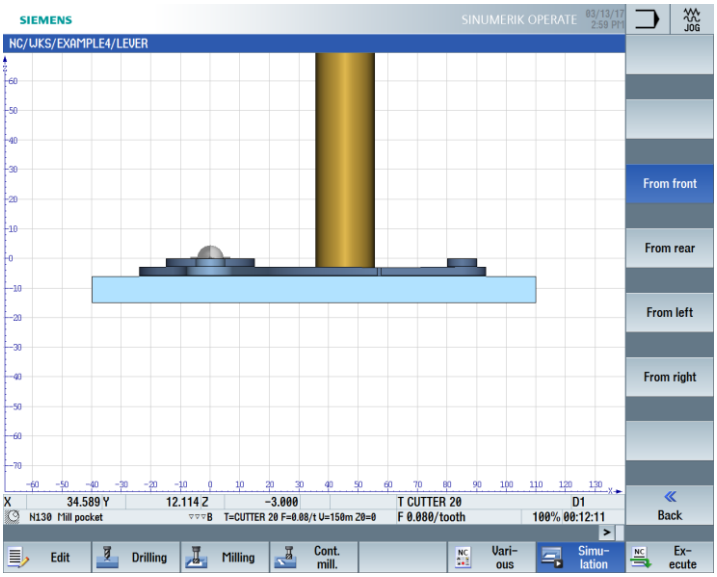
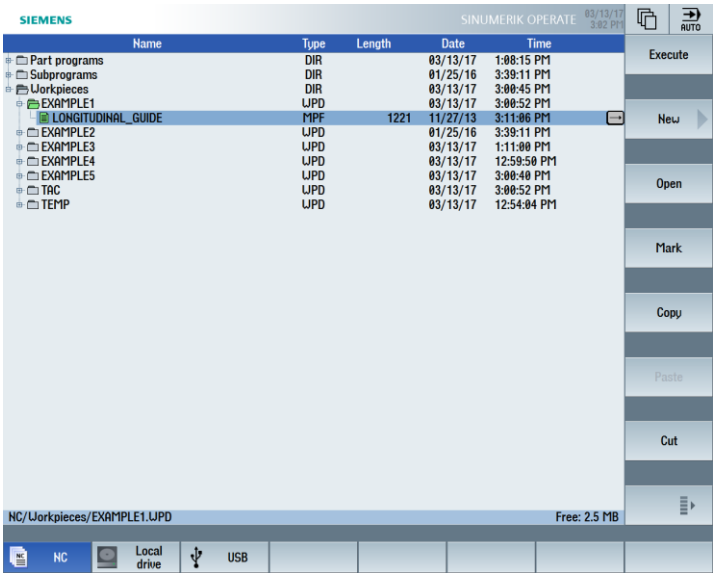


Figure 3-8 Side view

4.2.4 Program Manager

Managing programs

You can create new programs at any time using the Program Manager. You can access existing programs to execute, modify, copy or rename them. Programs that are no longer needed may be deleted.



Active programs are marked with a green symbol.

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

Creating a new workpiece

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

Creating a new program

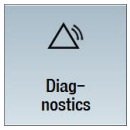
If you create a new program, you can specify the type of programming using the following softkeys:

"ShopMill Program"

"G-Code program"

4.2.5 Diagnostics

Alarms and messages



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

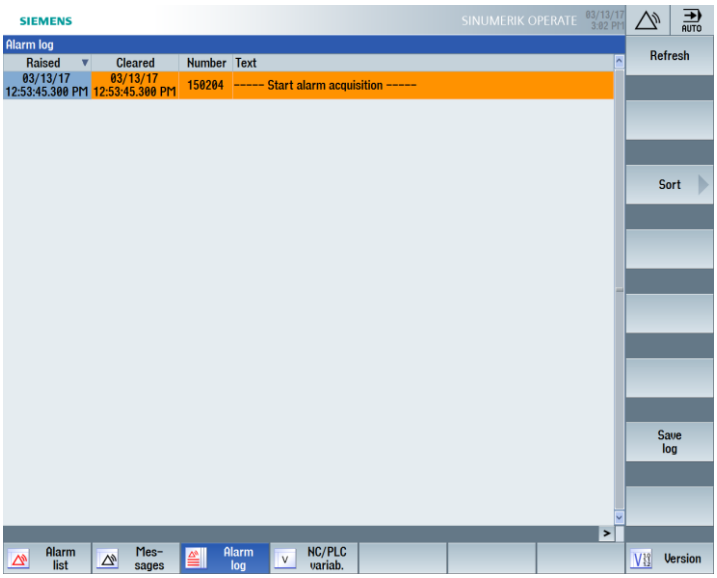


Figure 3-9 Alarm log

5. Basics for beginners

This section will explain the general basics of the geometry and technology for milling. You do not have to enter anything in ShopMill in this section.

5.1 Basics of geometry

5.1.1 Tool axes and machining 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 mounting position of the tool yields a corresponding machining plane. Z is the tool axis in most cases.

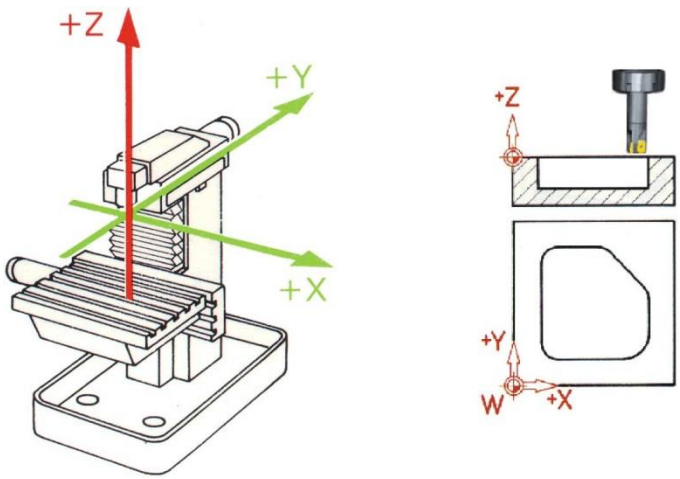


Figure 4-1 Vertical spindle

On modern machines, the tool mounting position is changed without the need for resetting measures and in a few seconds by way of a universal swivel head.

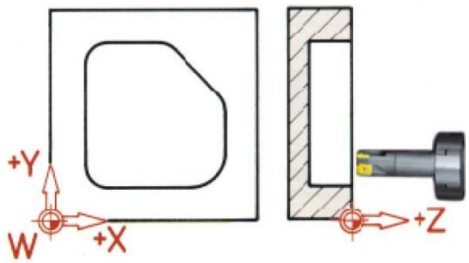


Figure 4-2 Horizontal spindle

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

With the "Various" and "Settings" softkeys, you can call a parameter screen form in which you can specify the machining planes in the program header.

Select the "Various" softkey.

Select the "Settings" softkey.

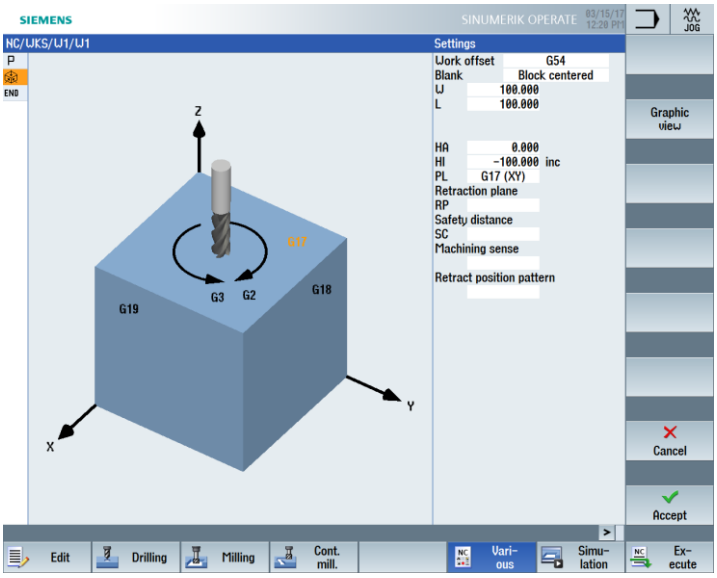
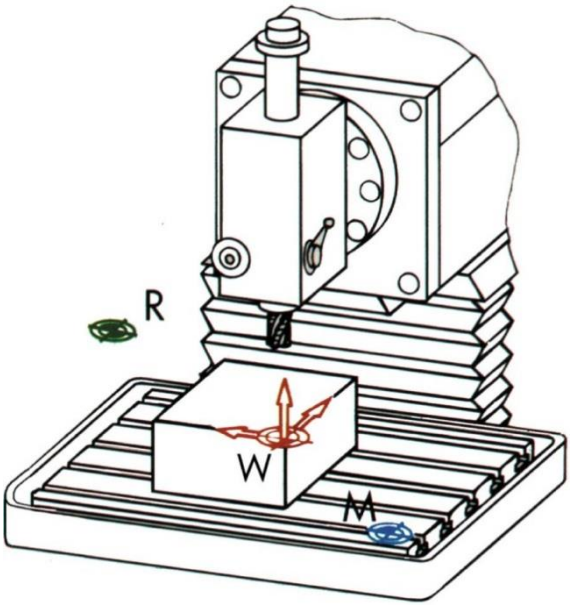


Figure 4-3 Machining planes parameter screen form

5.1.2 Points in the work area

A few important reference points are available so that a CNC – such as the SINUMERIK 828D with ShopMill – can orient itself in the existing work area by way of the measuring system.



Machine zero (M):

The machine zero (M) is specified by the manufacturer and cannot be changed. It is located at 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 freely selected and should be located at the point from which the most dimensions start in the drawing.

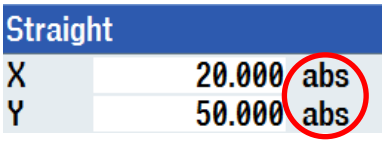
Reference point (R):

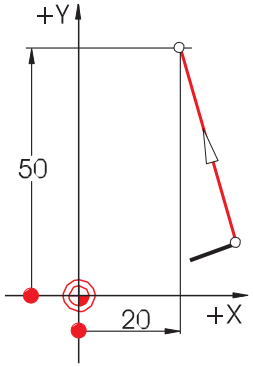
The reference point (R) is approached for setting the measuring system to zero, as the machine zero cannot be approached in most cases. This is how the controller finds its count start in the position measuring system.

5.1.3 Absolute and incremental dimensions

Absolute input

The entered values are relative to the workpiece zero.

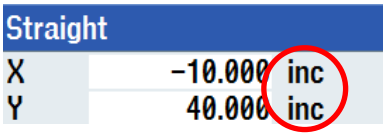


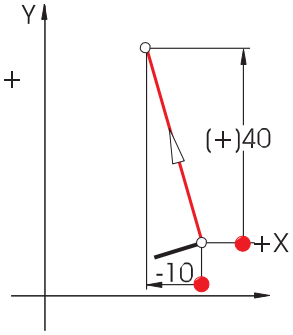


With absolute specifications, the absolute coordinate values of the end point must always be entered (the starting point is not considered).

Incremental input

The entered values are relative to the starting point.

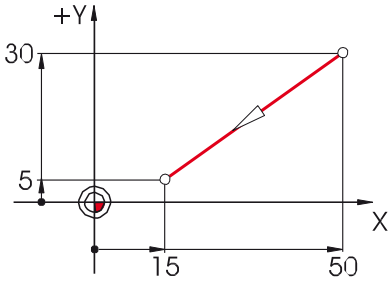




With incremental specifications, the difference values between starting point and end point must always be entered while taking the direction into account.

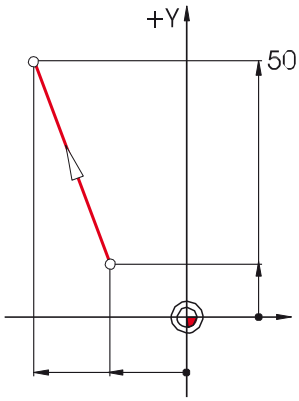
Switching between absolute and incremental input is possible at any time using the SELECT key.

A few examples combining absolute and incremental dimensions can be found below:



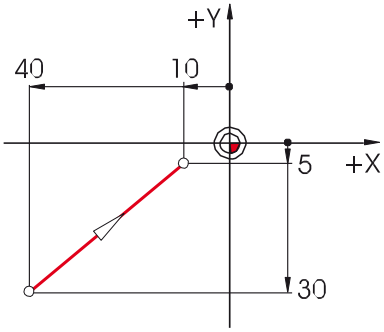
Absolute:
X15 Y5

Incremental:
X-35 Y-25



Absolute:
X-30 Y50

Incremental:
X-15 Y40



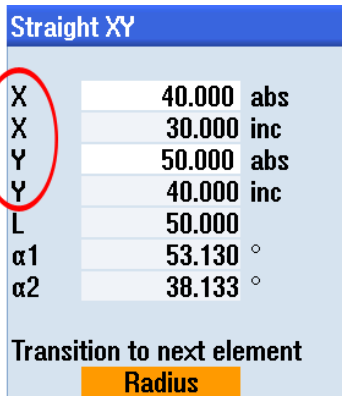
Absolute:
X-10 Y-5

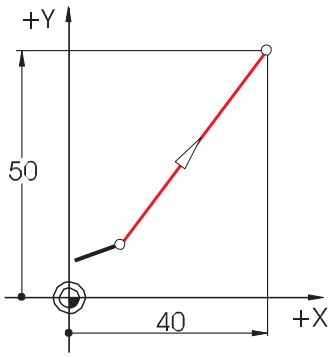
Incremental:
X30 Y25

5.1.4 Linear motions

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

- Cartesian
  - Specification of the X and Y coordinates

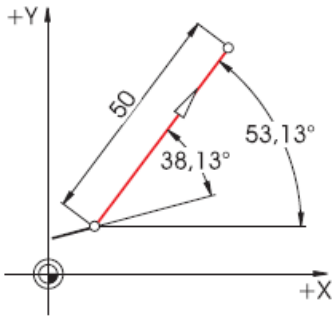




- Polar
  - Specification of the length and an angle

    Angle 38.13° = Angle relative to the previous element or angle 53.13° = starting angle relative to the positive X axis

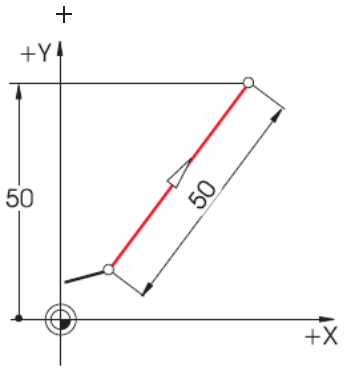
Straight XY		
X	40.000	abs
X	30.000	inc
Y	50.000	abs
Y	40.000	inc
L	50.000	
α1	53.130	°
α2	26.565	°
Transition to next element		
	Radius	
R	0.000	



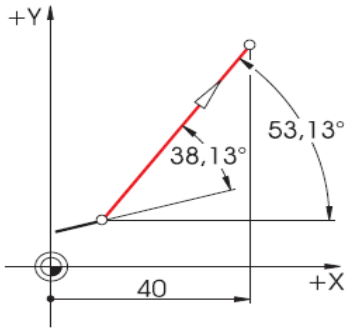
- Cartesian and polar

  Cartesian and polar inputs can be combined, e.g.

  – Specification of the end point in Y and the length



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



5.1.5 Circular motions

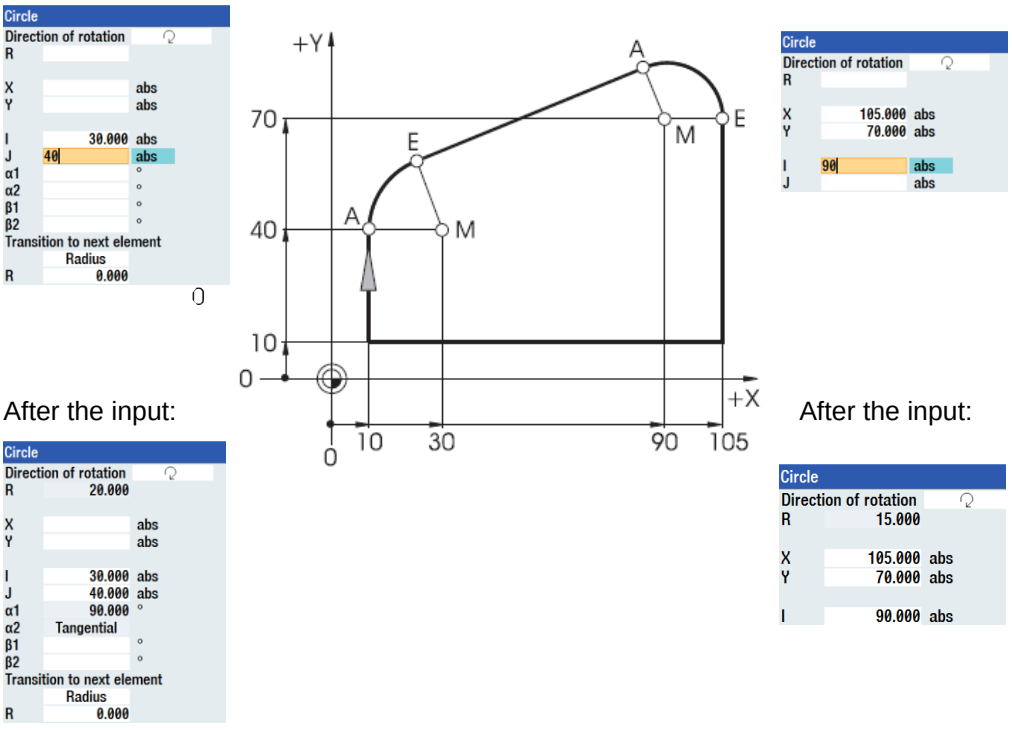
In the case of circular arcs, X and Y specify the end point; the circle center is specified with I and J. In ShopMill, these four values can be entered separately - either as absolute or incremental dimensions.

While X and Y are entered as absolute dimensions, the center point is specified with I and J as an incremental dimension in most controllers. Not only the difference from the starting point A to the center point M must be defined (often 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 defined easily using the graphical contour calculator.

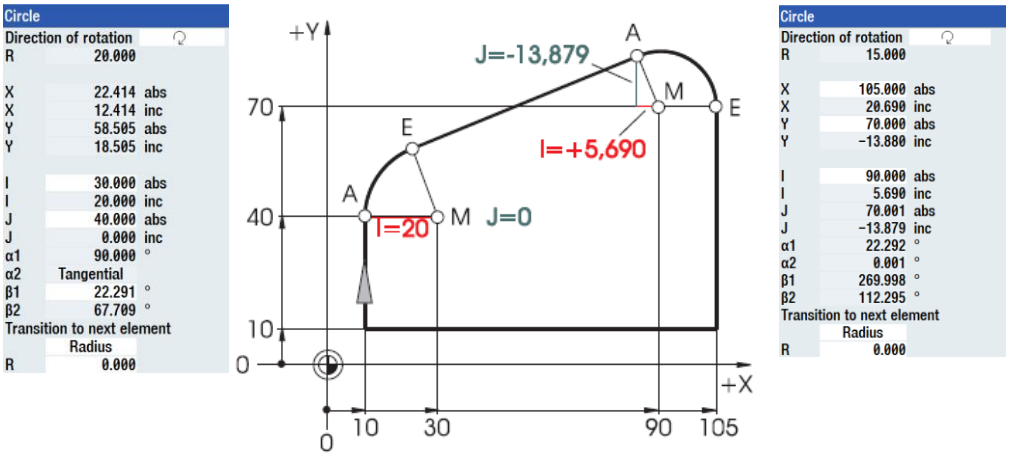
Specification of the center point (absolute)

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



Display of all parameters

With ShopMill all possible geometry values can also be displayed:



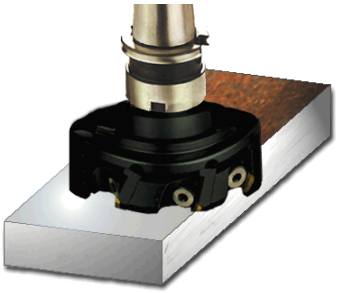
A further advantage of absolute center-point dimensioning:
You need not recalculate the values for I and J when reversing the milling direction.

5.2 Basics of technology

Well-founded knowledge of the tools is a basic requirement for optimum machining, which means cutting materials of the tools, their possible applications and the optimum cutting data are meant. Although tools themselves account for only about 2 - 5 % of the total manufacturing costs of a workpiece, they influence more than 50% of production costs of a component through their performance.

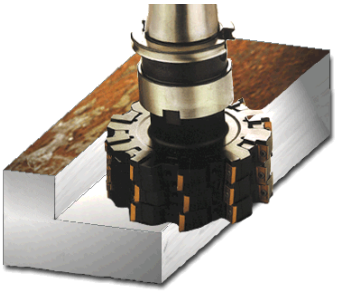
5.2.1 The tools in use

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.

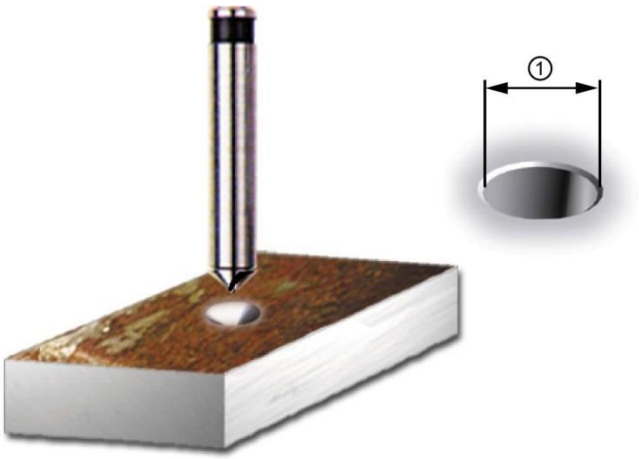
Helical shank mill

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, etc.). The drill tip is offset automatically in ShopMill.
provided that the tip angle of the drill was entered in the tool list.

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.

5.2.2 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 without a calculation, even if based on many years of 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 is selected:	v_c = 115 m/min

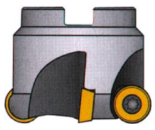
This cutting rate and the known tool diameter are used to calculate the speed n.

$$n = Vc * 1000/d*\pi$$

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

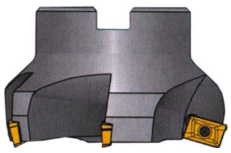
$$n = \frac{115\ mm * 1000}{40\ mm * \pi * min}$$

d_1 =40 mm



$n_1 \approx 900$ 1/min

d_2 = 63 mm



$n_2 \approx 580$ 1/min

$$n = \frac{115\ mm * 1000}{63\ mm * \pi * min}$$

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

Path milling			
T	CUTTER40		D 1
F	0.150	mm/tooth	
S	900	rpm	

Path milling			
T	CUTTER63		D 1
F	0.150	mm/tooth	
S	580	rpm	

Note:

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

5.2.3 Feed per tooth and feedrates

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

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 obtained from the handbook, the documents of the tool manufacturer or practical 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 – 0.2 mm
The mean value is selected:	f_z = 0.15 mm

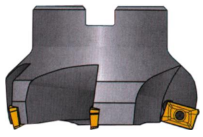
The feedrate v_f is calculated using the feed per tooth, the number of teeth and the known speed.

$V_f = fz * z * n$

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 1/min * 0.15 mm * 4



V_{f2} ≈ 580 1/min * 0.15 mm * 9

V_{f1} ≈ 348 mm/min

V_{f2} ≈ 783 mm/min

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

Path milling		
T	CUTTER63_Z4	D 1
F	340.000	mm/min
S	580	rpm

Path milling		
T	CUTTER63_Z9	D 1
F	780.000	mm/min
S	580	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:

6. Effective setup

In this section, you will learn how to create the tools required for the examples in the following, sections. Furthermore, the offset of the tool lengths and the setting of the workpiece zero is explained with examples.

6.1 Tool management

ShopMill offers three lists for tool management:

- Tool list
- Tool wear list
- Magazine list

6.1.1 Tool list

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

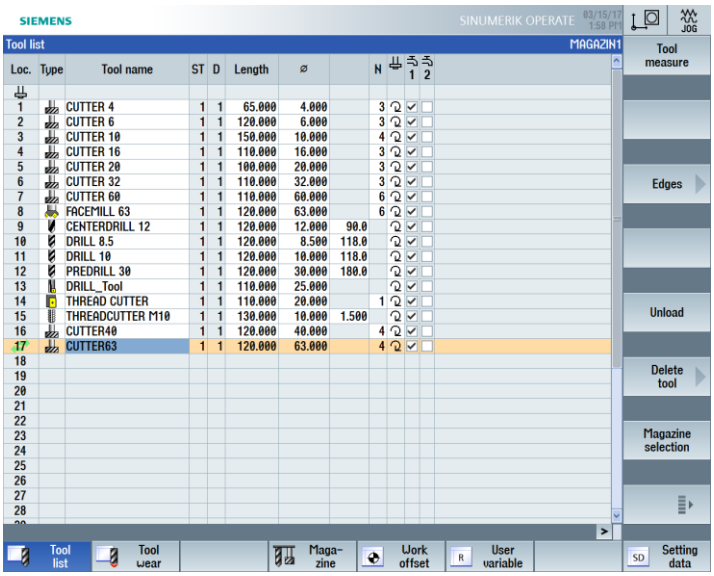


Figure 5-1 Example of tool list

Meaning of the most important parameters in the tool list:

Loc.	Location number
Type	Tool type
Tool name	The tool is identified by the name and the sister tool number. You may enter the name as a text or number.
ST	Sister tool number (for replacement tool strategy)
D	Cutting edge number
Length	Tool length
Ø	Tool diameter
Tip angle or lead	Tip 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

6.1.2 Tool wear list

The wear data for the respective tools is defined here.

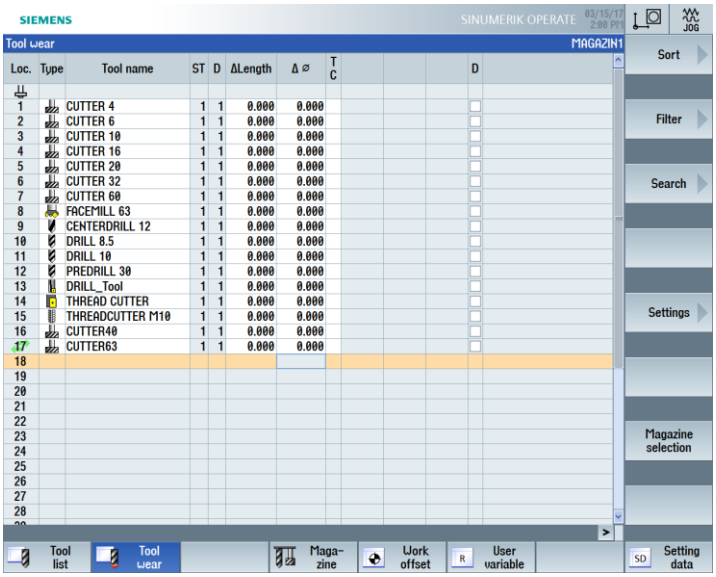


Figure 5-3 Tool wear list

The most important tool wear 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 Workpiece count 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 if the check box is selected.

6.1.3 Magazine list

All tools that are assigned to one or more tool magazines are contained in the magazine list. This list displays the status of each tool. In addition, individual magazine locations can be reserved or locked for assigned tools.

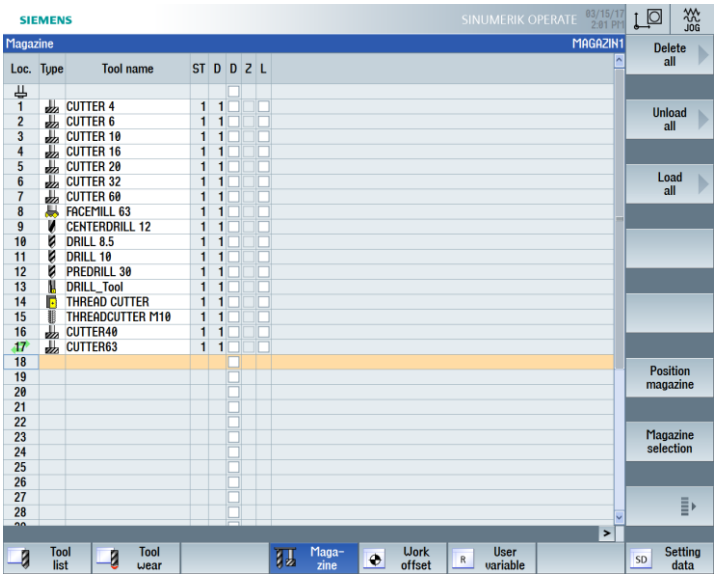


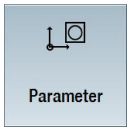
Figure 5-4 Magazine list

Meanings of the most important parameters:

D	Locking of the magazine location
Z	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.
L	Fixed location coding The tool is permanently assigned to this magazine location.

6.2 Tools used

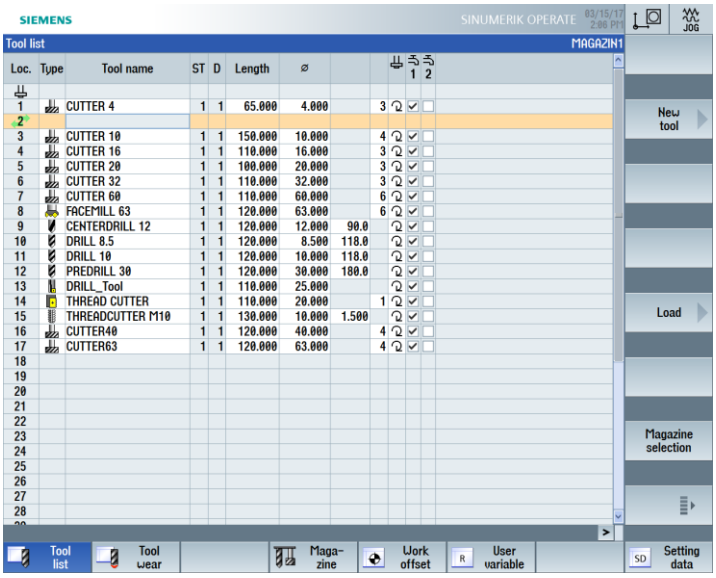
In this section, the tools that are needed for working through the examples later are entered in the tool list.



Select the "Parameter" area in the main menu.

Select the "Tool list" softkey.

To create a new tool, go to 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 is inserted in the tool list and you can enter the data of the tool.

Note:

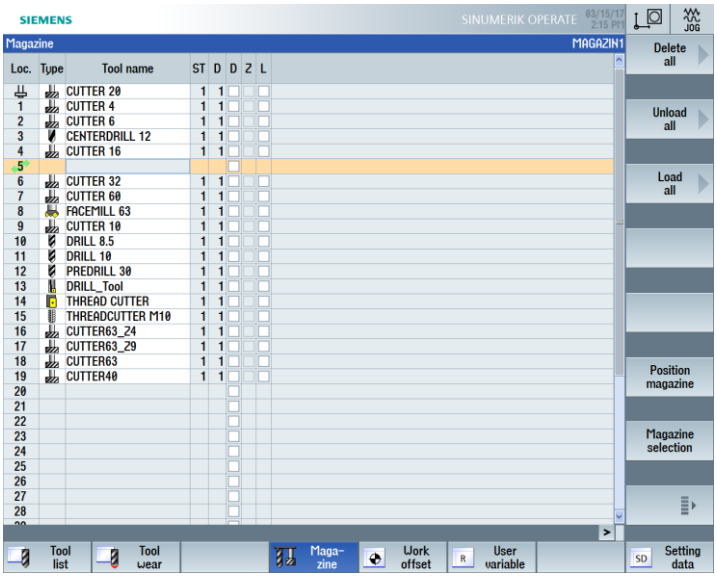
The milling cutters with diameter 6, 10, 20 and 32 (Cutter 6, 10, 20 and 32) must be insertable, as they will also be used for the milling of pockets in the following, examples.

6.3 Tools in the magazine

In the following, you will learn how to load the tools into the magazine:

Select a tool without location number in the tool list and press the "Load" key.

The following dialog offers the first free magazine location, which you can change or accept as is. For example, the magazine for the following exercises could look like this:

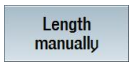


6.4 Measuring tools

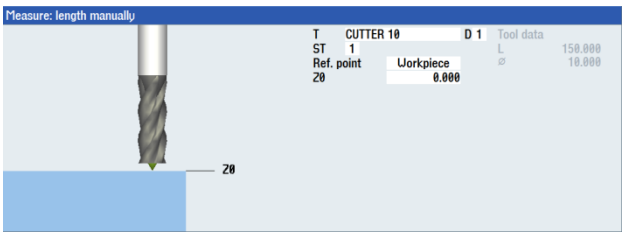
In the following, you will learn how tools are offset:

Use the "T,S,M" softkey to insert a tool from the tool list into the spindle.

Then go to the "Meas. tool" menu.

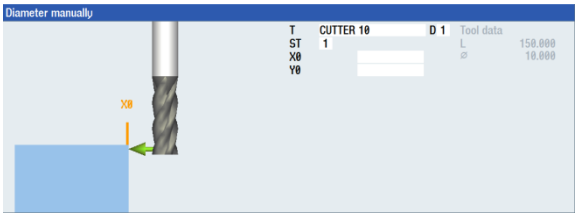


The "Length manual" function is used to measure the tool in the Z-direction.



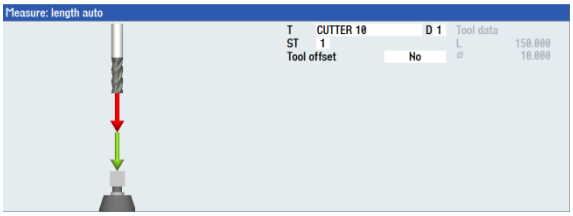
Diameter manually

The "Diameter manual" function is used to measure the diameter of the tool.



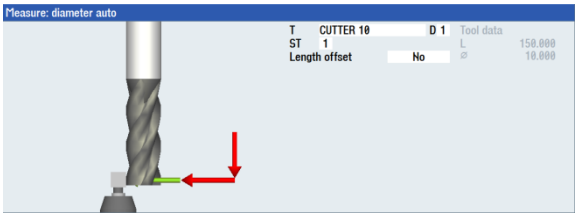
Length auto

The "Length auto" function is used to measure the tool in the Z direction using a tool probe.



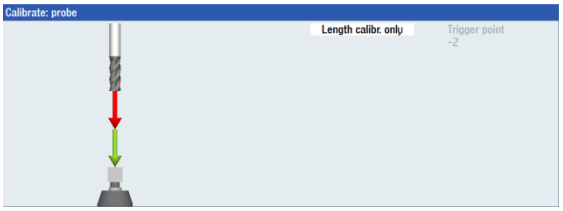
Diameter auto

The "Diameter auto" function is used to measure the diameter of the tool using a tool probe.



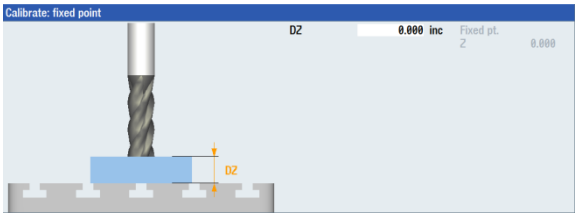
Calibrate probe

The "Calibrate probe" function is used to determine the position of the probe on the machine table relative to the machine zero.



Calibrate fixed pt.

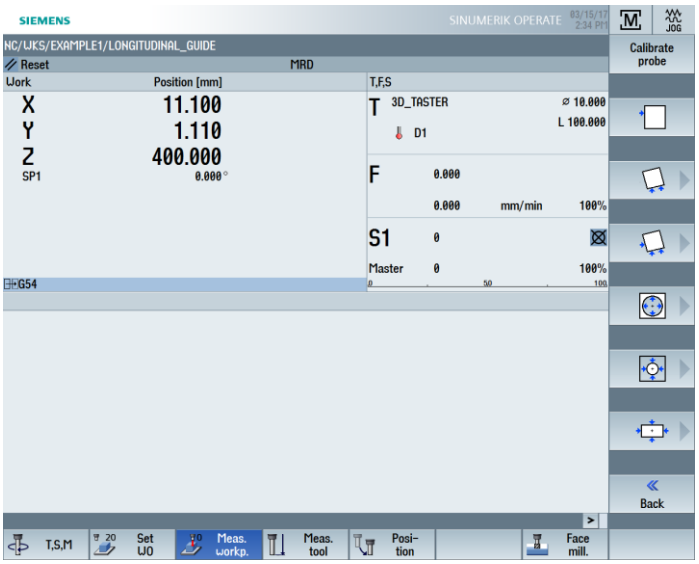
The "Calibrate fixed pt." function is used to determine the fixed point as the reference point for manual measurement of the tool length.



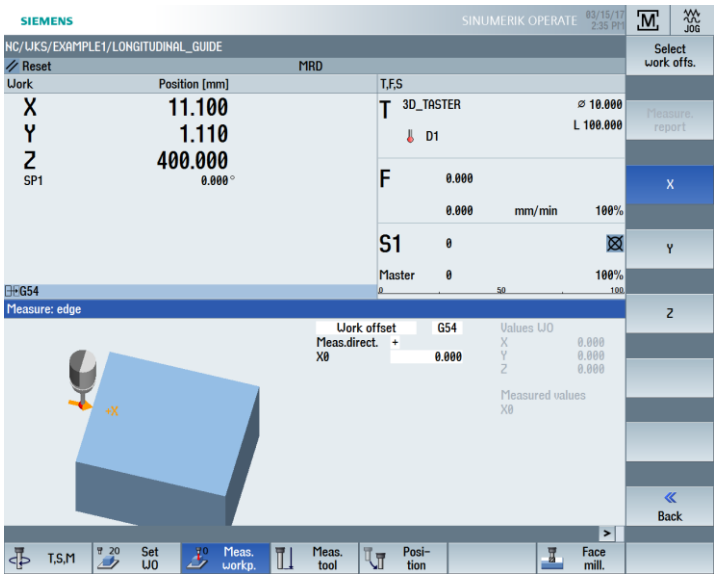
6.5 Setting the workpiece zero

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

The submenu of the "Measure workpiece" option ("Meas. workp.") offers various possibilities to set the workpiece zero.



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



1) Select the edge

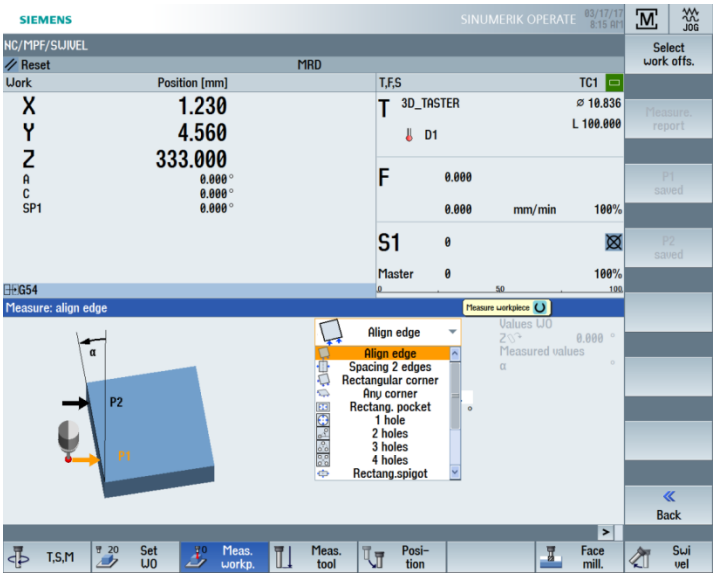
Define the measuring direction left (+) or (-). Parameter X0 can be used to specify an offset of the workpiece zero if this is not to lie on the workpiece edge.

2) Contact the workpiece edge with the probe.

3) The workpiece zero is set taking into account the edge probe diameter (5 mm).

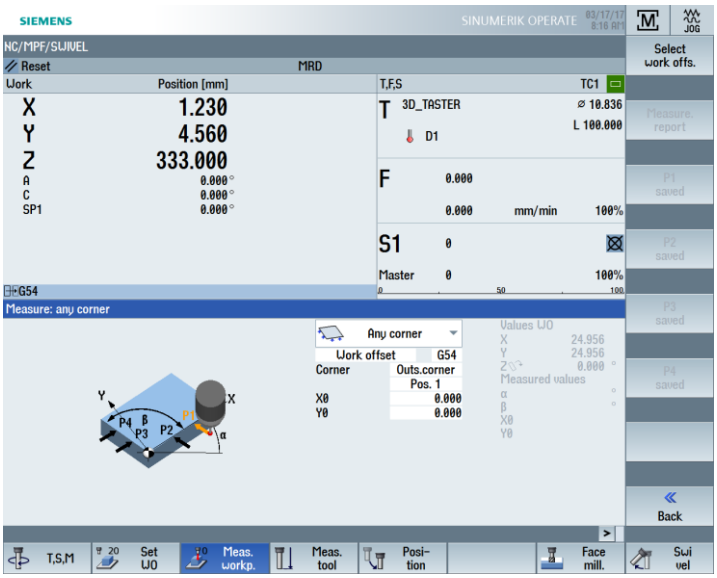
This offset operation must now be repeated for Y with the edge probe and for Z (in most cases, with the milling cutter).

Since the workpieces to be machined are not always cube-shaped or cannot always be clamped at right angles, further offset possibilities are provided:



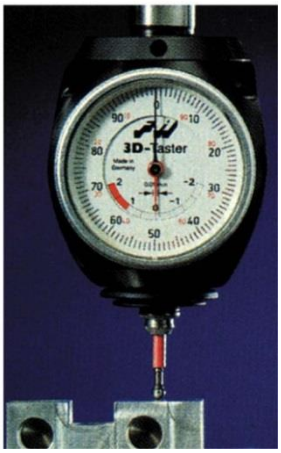
Example 1: Any corner

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

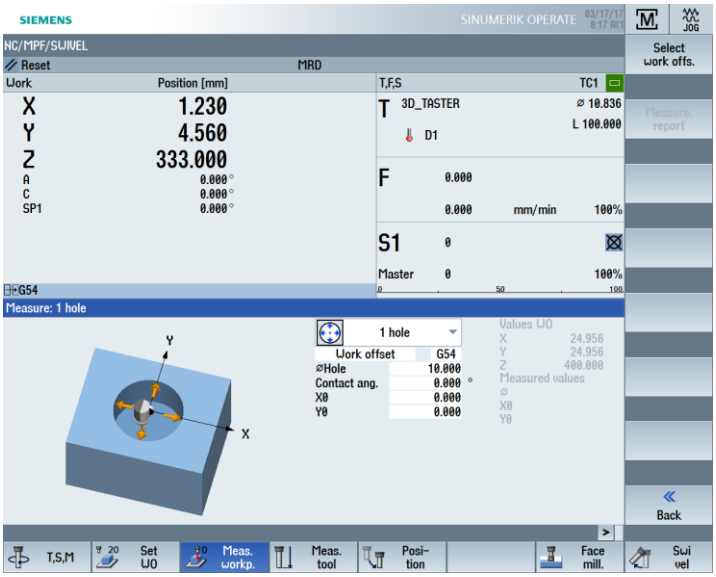


Electronic and mechanical 3D probes are available.

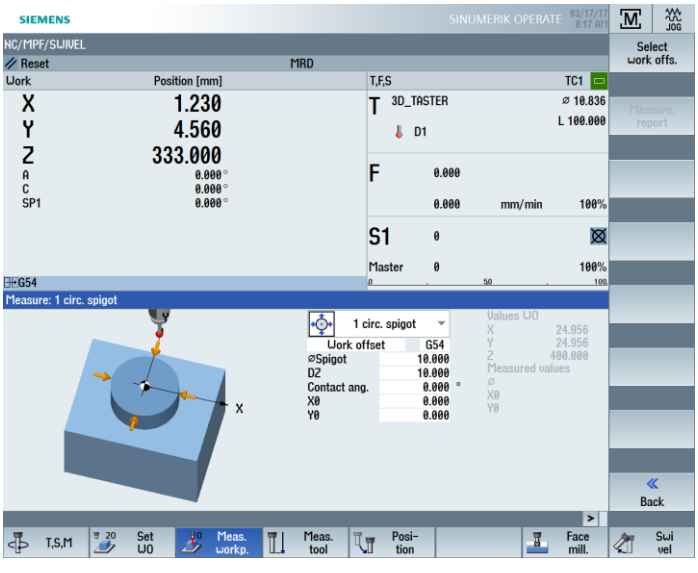
The signals of electronic probes can be processed directly by the controller.



Example 2: Offset of 1 hole



Example 3: Offset of 1 circular spigot



Calibrate probe

If an electronic 3D probe from the tool magazine is inserted into the spindle, clamping tolerances are involved. 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

7. Example 1: Longitudinal guide

7.1 Overview

Learning objectives

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

- Manage and create programs
- Call tools and perform a cutter radius compensation
- Enter traversing paths
- Create drill holes and handle position repetitions

Task

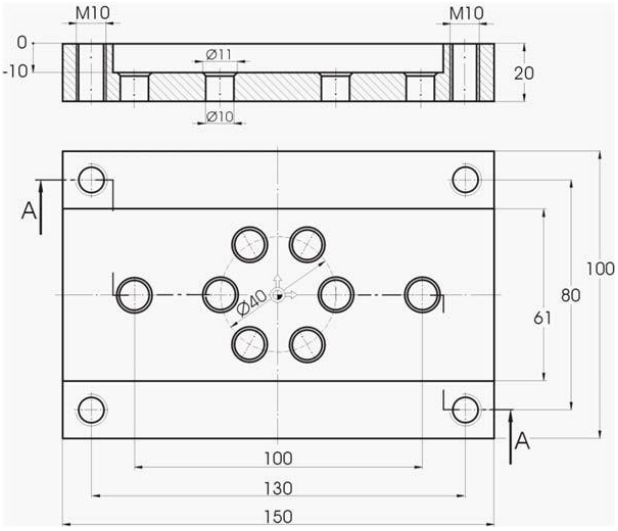


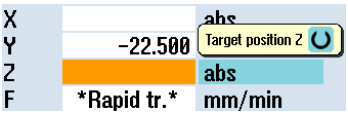
Figure 6-1 Workshop drawing – Example 1

Figure 6-2 Workpiece – Example 1

Note:

ShopMill always saves the last setting you selected with the toggle key. You must therefore ensure that the settings of all units, texts and symbols in the text boxes as well as all toggle fields are the same as in the dialog boxes of the examples.

An available toggle option is always indicated in the help text (see figure below).



7.2 Program management and creating programs

Operating sequences

When the controller starts up, you are in the main screen.

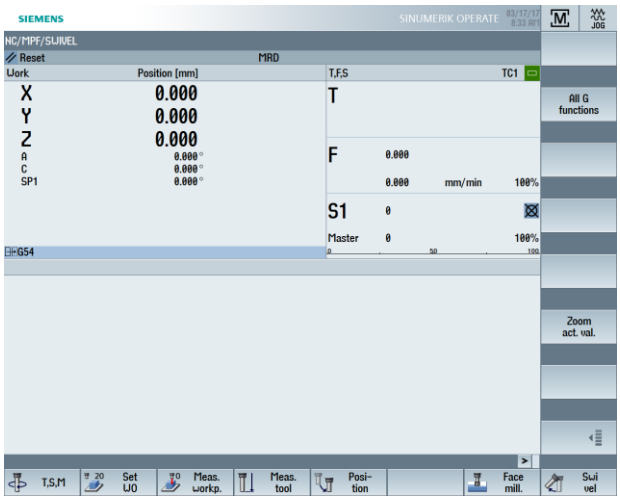


Figure 6-3 Main screen

Open the main menu using the MENU SELECT key. In the main menu, you can open various areas of ShopMill.

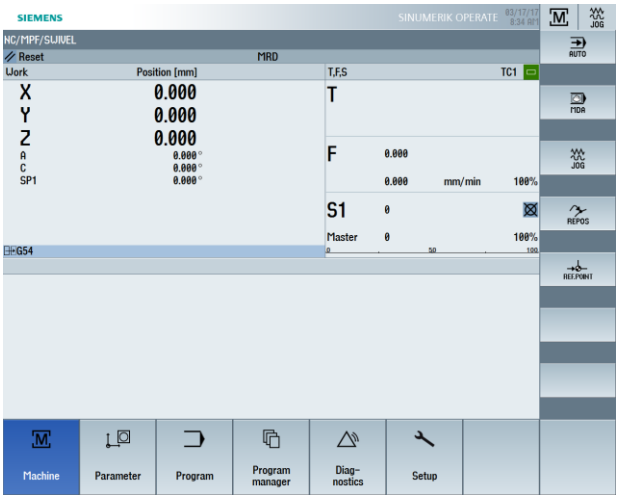


Figure 6-4 Main menu

Select the "Program Manager" softkey. The "Program Manager" is displayed.

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

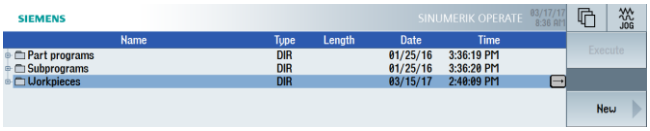


Figure 6-5 Program Manager

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.

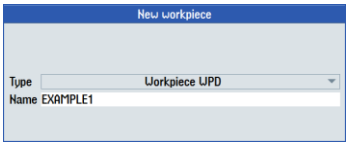


Figure 6-6 Creating a workpiece

Confirm your entry. The following dialog box opens.

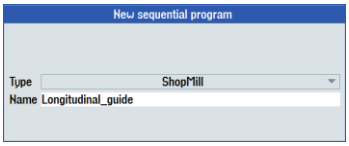


Figure 6-7 Creating a sequential program

You can select the input format with the "ShopMill" and "programGuide G code" softkeys.

You specify the program type using the ShopMill softkey.

Enter the name of the machining plan, in this case "Longitudinal_guide".

Accept your input.

Once applied, the following screen form is displayed for entering the workpiece data.

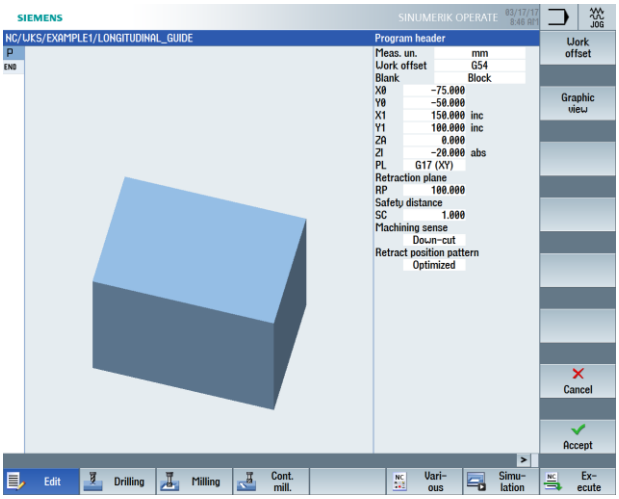


Figure 6-8 Program header – Help screen

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

Enter the following details:

Field	Value	Selection via toggle key	Notes
Unit of measurement	mm	X	
Work offset	G54	X	
Blank	Block	X	
X0	-75		Since the workpiece zero is in the center of 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 distance	1		
Machined direction of rotation	Down-cut	X	
Retract position pattern	Optimized	X	See below Retraction position pattern

Accept the entered values. Once applied, the program header is displayed.

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

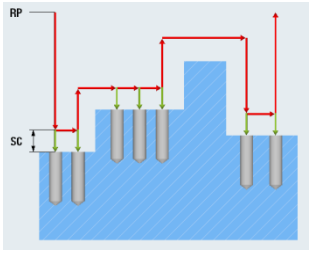
The program has now been created as the basis for further machining steps. It has a name (in the blue bar), a program header (pictogram "P") and a program end (pictogram "END"). The individual machining steps and contours are stored in the program one beneath the other. The program is subsequently executed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

You can open the program header again at any time to make changes or check the values.

Retract position pattern

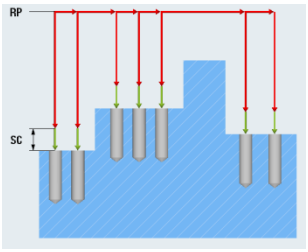
The position pattern can be set to "Optimized" (= time-optimized traversing paths) or "To retraction plane".

Optimized retraction



The tool traverses over the workpiece at the safety distance dependent on the contour.

To retraction plane (standard)



The tool moves back to the retraction plane and
is fed in to the new position.

Softkeys

Graphic view

Use the "Graphic view" softkey to change to the online graphic of the workpiece. See figure below.

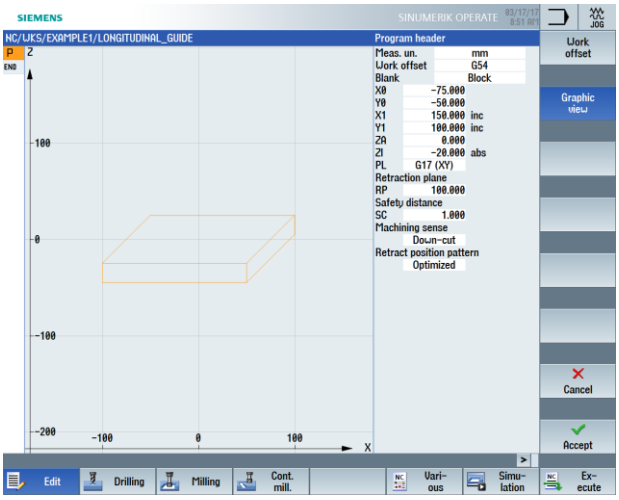


Figure 6-10 Program header – Graphic view

Graphic view

Use the "Graphic view" softkey to change back to the help screen.

7.3 Opening a tool and setting the cutter radius compensation

Operating sequences

Follow the steps below to call the required tool: Expand the horizontal softkey menu with the ETC key.

Select the "Straight Circle" softkey. Select the "Tool" softkey. Open the tool list.

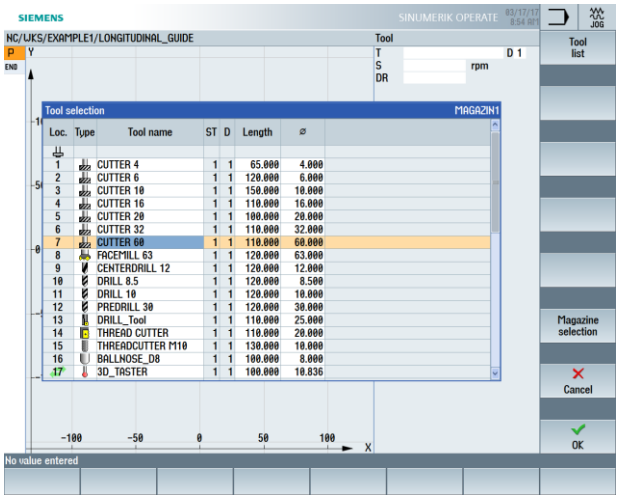


Figure 6-11 Tool list

Use the cursor key to select the "CUTTER60" tool.

Apply the tool to the program. After the tool has been applied, specify the cutting rate 80 m/min (if necessary, change the unit using the toggle key).

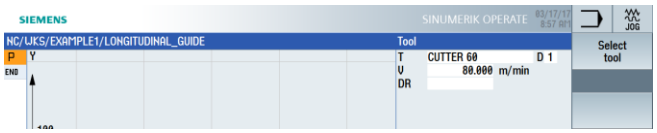


Figure 6-12 Tool – Cutting rate

Accept the entered value.

7.4 Entering the traversing path

Operating sequences

Now enter the traversing paths: Select the "Straight" softkey.

Select the "Rapid traverse" softkey.

Enter the following values in the screen form:

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

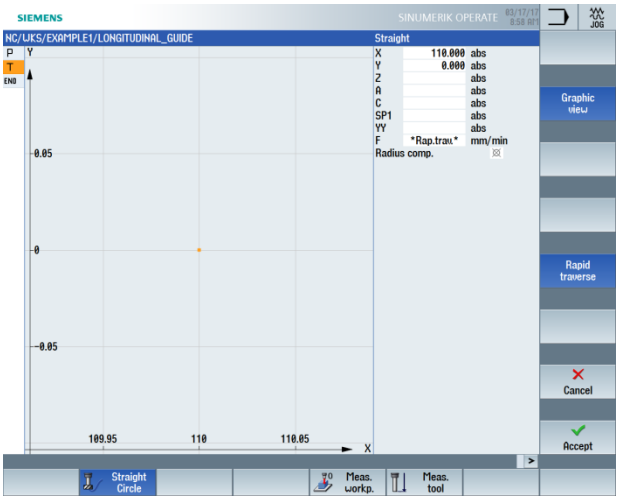


Figure 6-13 Entering the traversing path – Radius compensation

Accept the entered values.

Select the "Straight" softkey.

Select the "Rapid traverse" softkey.

Enter the following values in the screen form:

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

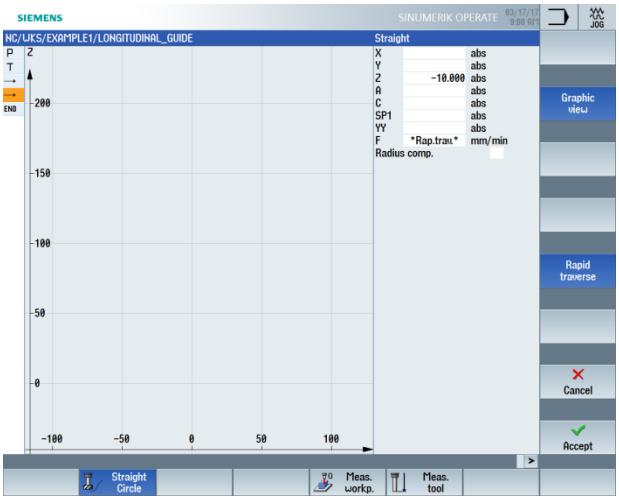


Figure 6-14 Entering the traversing path – Tool positioned in Z

Accept the entered values.

Select the "Straight" softkey.

Enter the following values in the screen form:

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

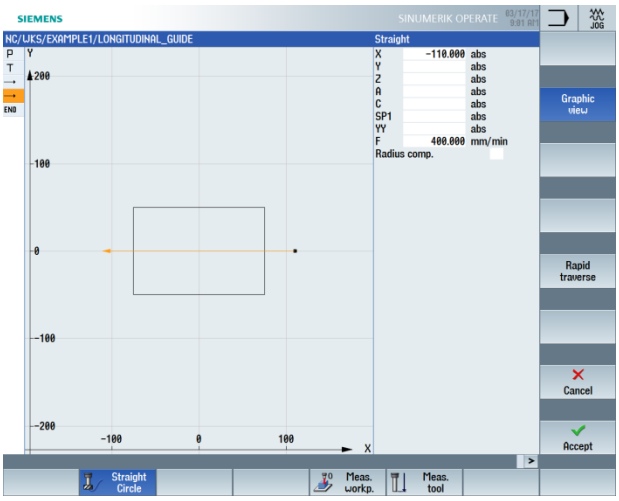


Figure 6-15 Entering the traversing path – First, machining path

Accept the entered values. Once applied, the list of machining steps looks like this:

Figure 6-16 Entering the traversing path – List of machining steps

Select the "Tool" softkey and perform the following machining steps without help.

Load the next tool "CUTTER16". After the tool has been applied, specify the cutting rate 100 m/min.

Create the traversing path according to the following list of machining steps:

Figure 6-17 Entering the traversing path – List of machining steps

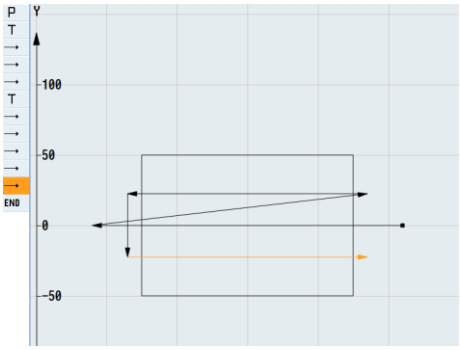


Figure 6-18 Entering the traversing path – Complete

Start the simulation.

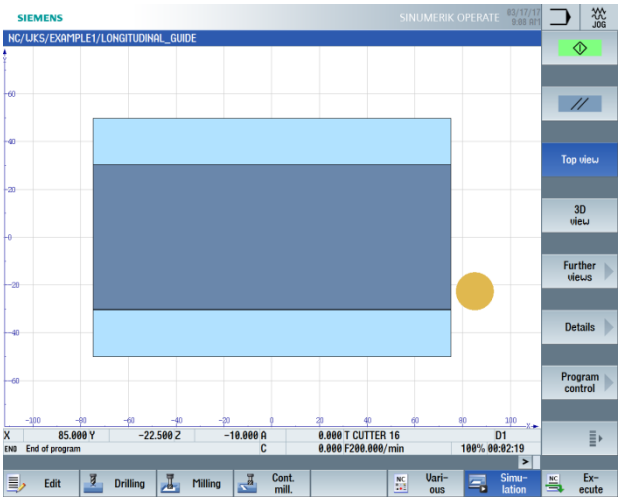
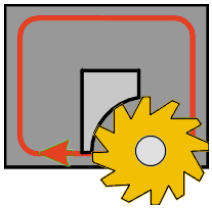
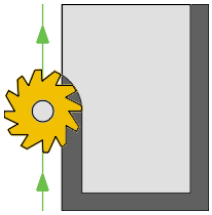
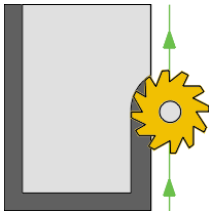


Figure 6-19 Simulation of traversing path

You can end the simulation by selecting either the "Simulation" softkey again or any other horizontal softkey.

Radius compensation

Selection	Result
	 Radius compensation is switched off. The milling cutter traverses with its center point along the created contour.
	The existing compensation setting 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.

7.5 Creating drill holes and position repetitions

Operating sequences

Now enter the values for the drill holes and position repetitions. In so doing so, you must center, through-drill and tap the 12 drill holes.

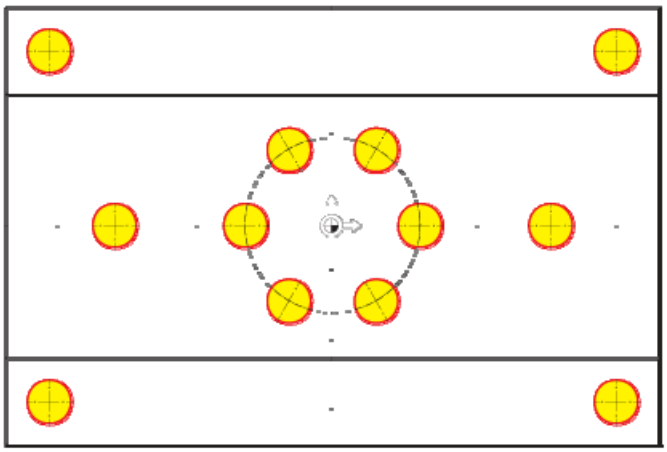


Figure 6-20 Drilling positions

Select the "Drill." softkey.

Select the "Centering" softkey.

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

Apply the tool to the program. After the tool has been applied, 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 have a 0.5 mm chamfer, you may specify a diameter of 11 mm here.

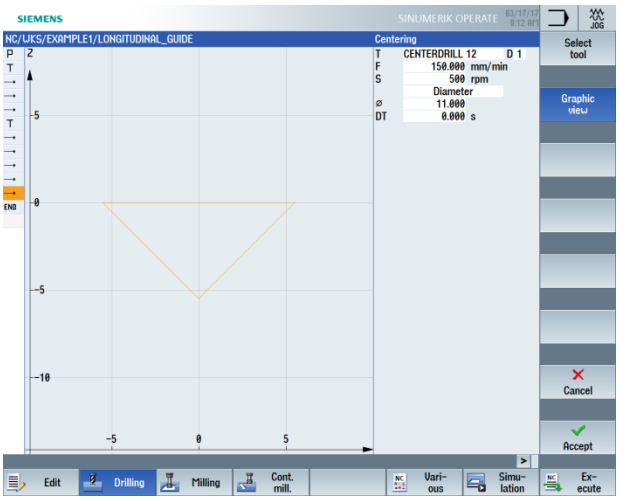


Figure 6-21 Centering

Accept the entered values.

The following steps are used to enter the drilling positions and link these positions with the cutting data.

Select the "Positions" softkey.

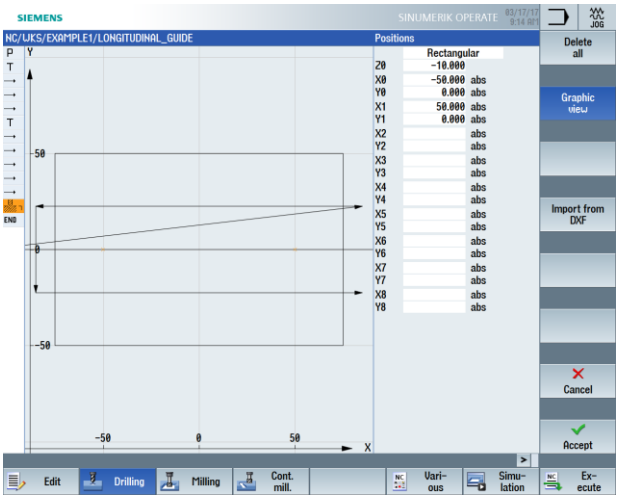


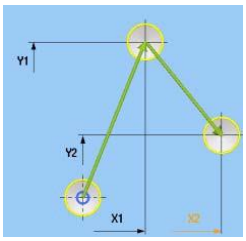
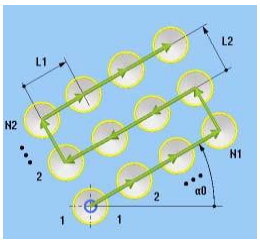
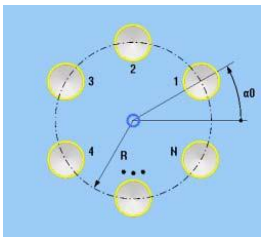
Figure 6-22 Positions – Individual drill holes

Enter the following values for the two individual drill holes:

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

Note:

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

Positions	Position pattern	Position circle
		

Help screens – Positions

Accept the entered values.

Select the "Positions" softkey.

Select the "Position circle" softkey.

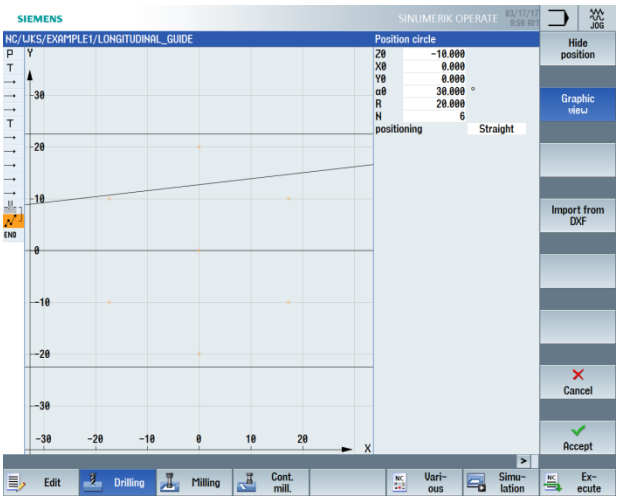
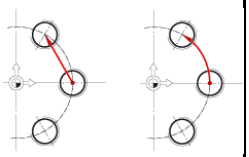


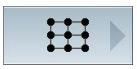
Figure 6-23 Position circle

Enter the following details:

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	X	Use the "Positioning" field to define how the drill holes are approached within the drill hole pattern. If the drill holes lie in a circumferential slot, for example, do not use "Positioning - Straight" as this would cause a contour violation.  Along a straight line, Along a circle

Accept the entered values.

Select the "Positions" softkey.



Select the "Position pattern" softkey.

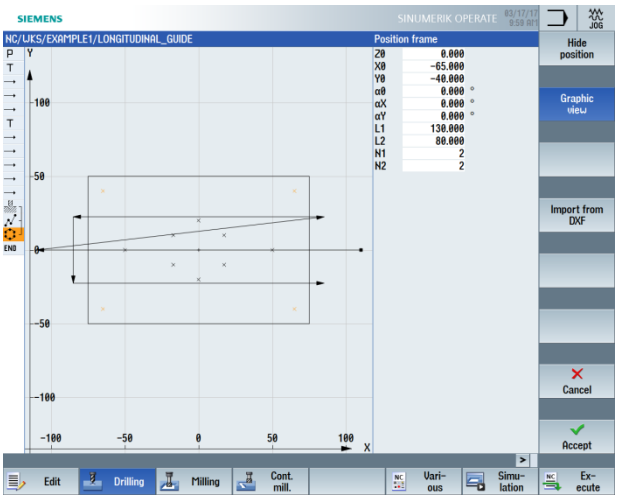
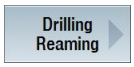


Figure 6-24 Positions – Grid

Enter the following details:

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

Accept the entered values.



Select the "Drilling Reaming" softkey.

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

Apply the tool to the program. After the tool has been applied, 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 as an incremental dimension. That is, the 1/3 D drill tip is taken into account automatically.
Z1	20 inc	X	
DT	0 sec	X	Drilling will be carried out without dwell time.

Note:

The machining steps "Centering", "Drilling" and "Tapping" are linked with each other automatically.

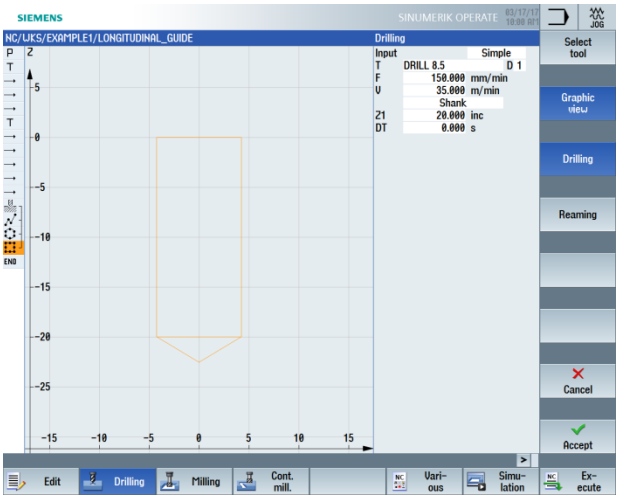
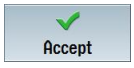


Figure 6-25 Drilling



Accept the entered values.

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.

Accept

Apply the tool to the program. After the tool has been applied, enter the following values:

Field	Value	Selection via toggle key	Notes
L	1.5 mm/rev	X	
S	60 rpm	X	
SR	60 rpm	X	
Z1	22 inc	X	The cutting depth must be entered as an incremental dimension.

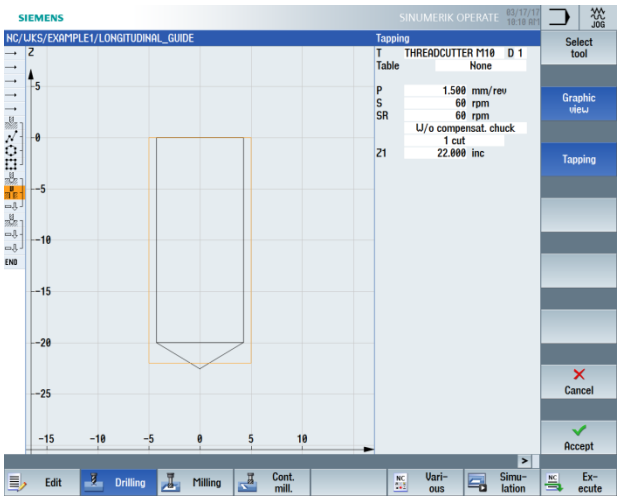


Figure 6-26 Thread

Accept

Accept the entered values.

Position repetit.

Select the "Position repetit." softkey.

The drilling positions are numbered consecutively during creation. The respective number stands directly after the block number of the respective position pattern. Enter hole matrix for position 3.

Figure 6-27 Repeating a position

Accept

Accept the entered values. Once applied, the values, you can see the linking of the machining steps in the machining step editor.

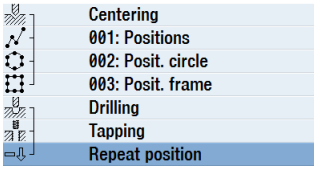


Figure 6-28 Linking of machining steps

Drilling Reaming

Select the "Drilling Reaming" softkey.

Select tool

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

Accept

Apply the tool to the program. After the tool has been applied, 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	

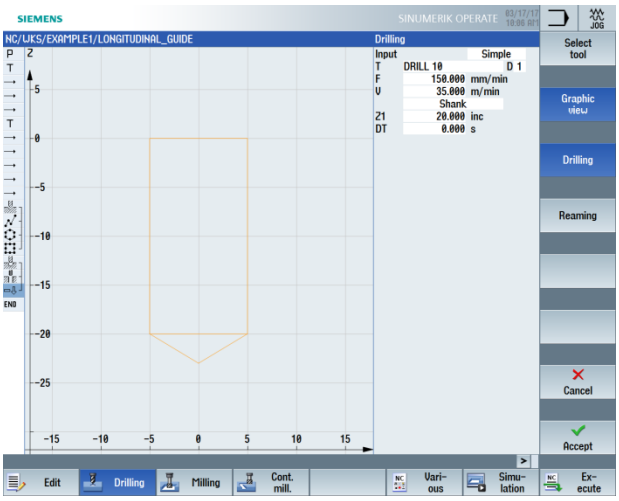
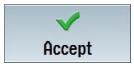


Figure 6-29 10 mm drill holes



Accept the entered values.

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

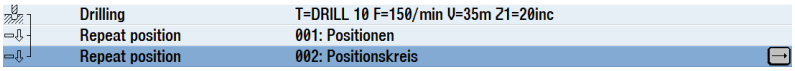


Figure 6-30 Repetition of positions 001 and 002 in the machining step editor.

Start the simulation for checking.

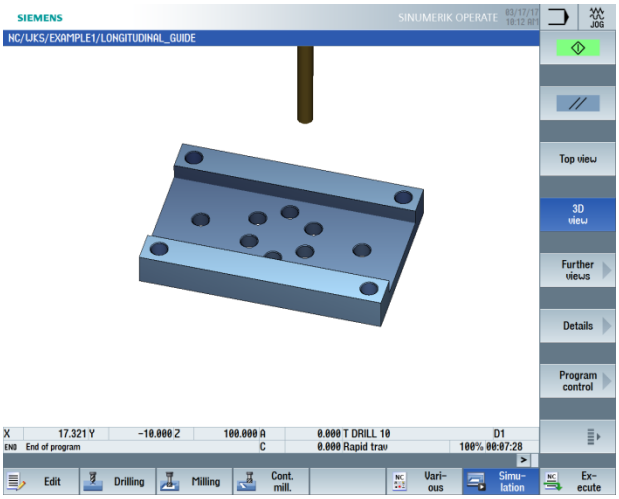


Figure 6-31 3D simulation

8. Example 2: Injection mold

8.1 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

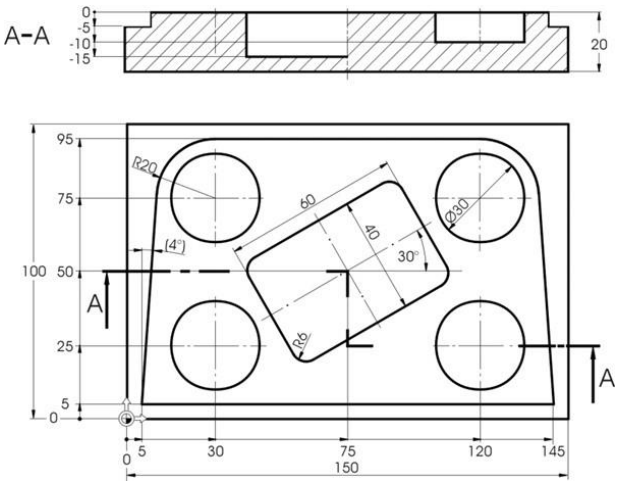


Figure 7-1 Workshop drawing – Example 2

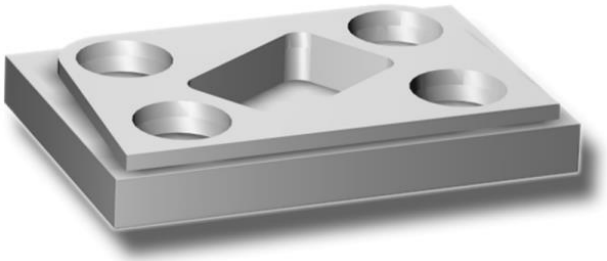


Figure 7-2 Workpiece – Example 2:

Preparations

Perform the following steps on your own:

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

Note:

Pay attention to the new zero position!

4. Load the 20 mm milling cutter (V 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 at X5 and 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 machining plan should look like this:

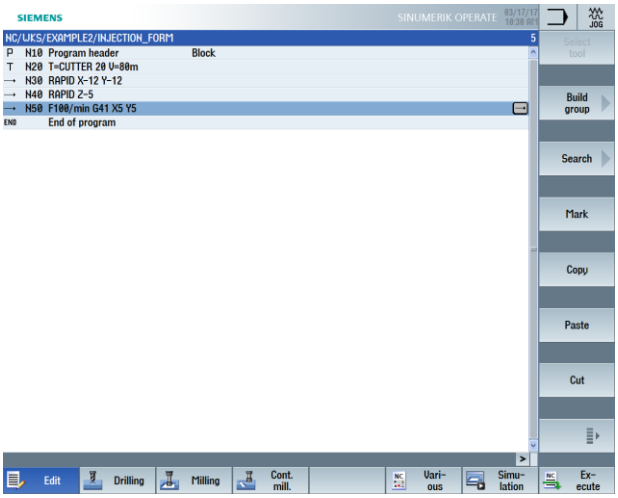


Figure 7-3 Machining step program

8.2 Straight lines and circular paths using 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 a polar reference point.

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

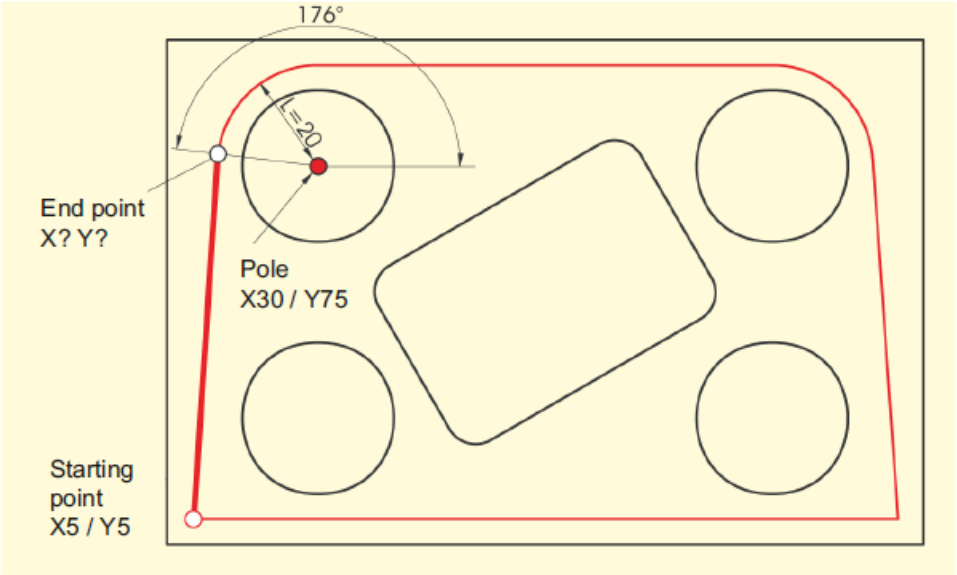


Figure 7-4 Determination of end point and polar angle

Follow the steps below to enter the contour:

Select the "Polar" softkey.

Select the "Pole" softkey.

Enter the following values in the screen form:

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

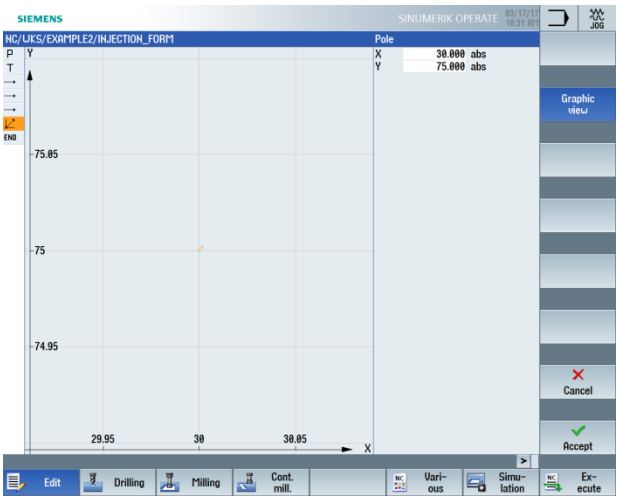
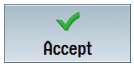
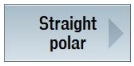


Figure 7-5 Entering the pole



Accept the entered values.



Select the "Straight polar" softkey.

Enter the following values in the screen form:

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 in either the counterclockwise (176°) or clockwise direction (-184°).

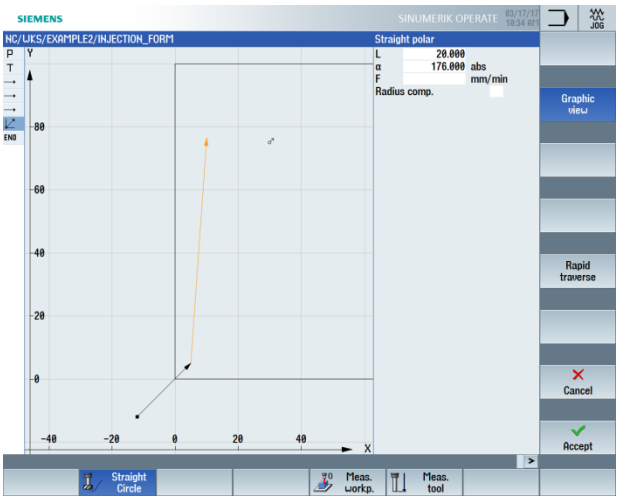
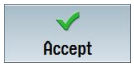
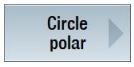


Figure 7-6 Entering the straight line using polar coordinates



Accept the entered values.



Select the "Circle polar" softkey.

A circular path can also be specified using polar coordinates.

Enter the following values in the screen form:

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 figure below)

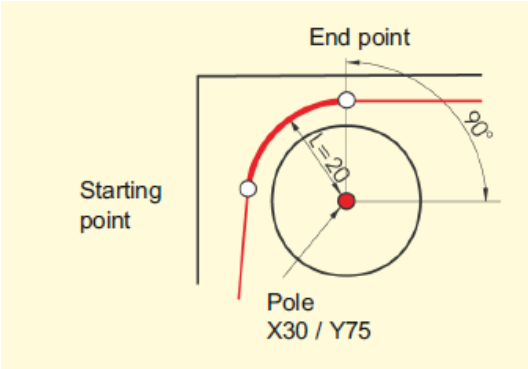


Figure 7-7 Pole starting/end points

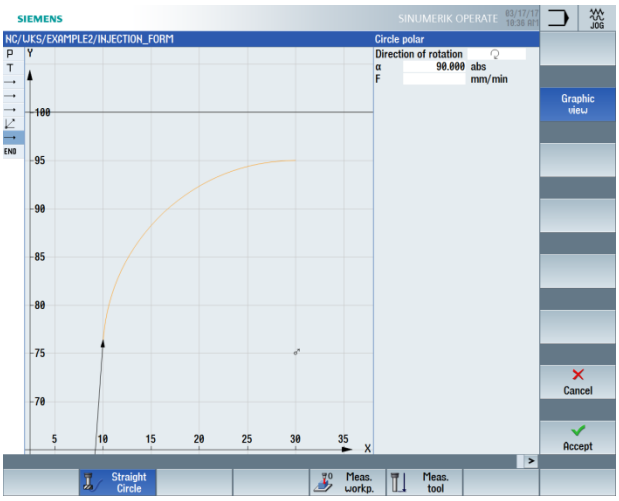
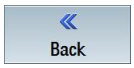


Figure 7-8 Entering the circular path

Accept the entered values.



Select the "Back" softkey.

Select the "Straight" softkey.

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

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
X	120	X	

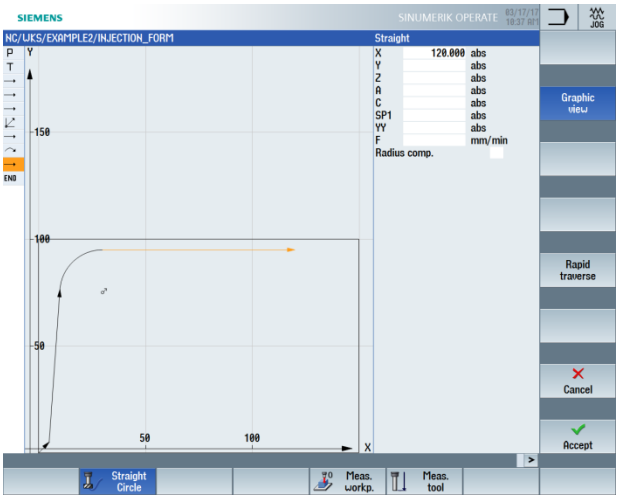


Figure 7-9 Entering the straight line

Accept the entered values.

Select the "Polar" softkey.

Select the "Pole" softkey.

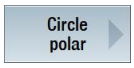
Since the end point of the next circular path is also not known, you must work with polar coordinates again here.

Enter the following values in the screen form:

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 Entering the pole for circular path

Accept the entered values.



Select the "Circle polar" softkey.

Enter the following values in the screen form:

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

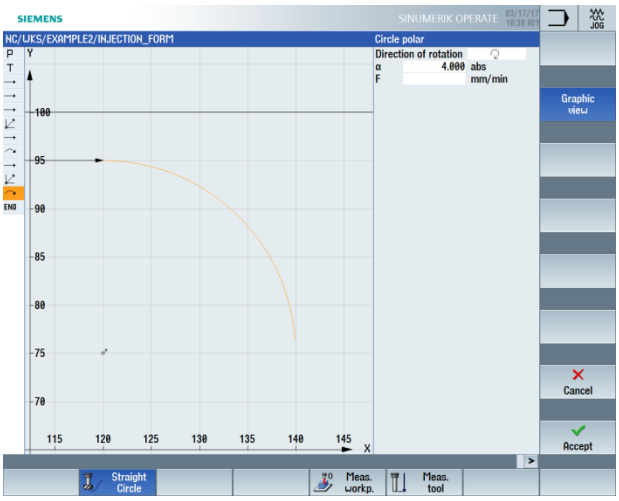
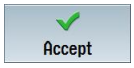
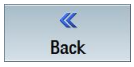


Figure 7-11 Entering the circular path using polar coordinates



Accept the entered values.



Select the "Back" softkey.

Select the "Straight" softkey.

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

Enter the following values in the screen form:

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

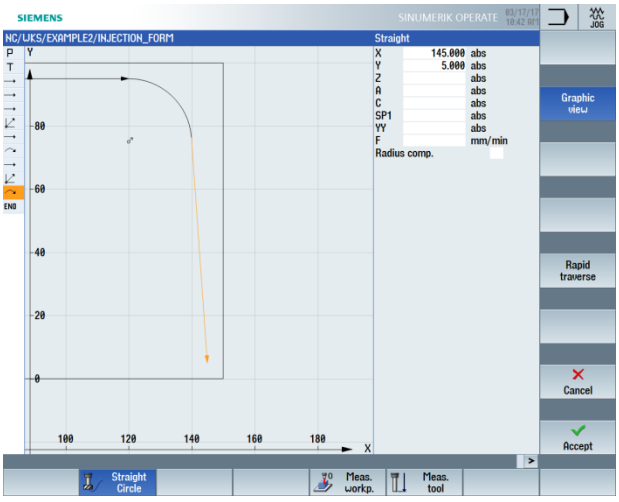


Figure 7-12 Entering the straight line

Accept the entered values.

Select the "Straight" softkey.

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

Enter the following values in the screen form:

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

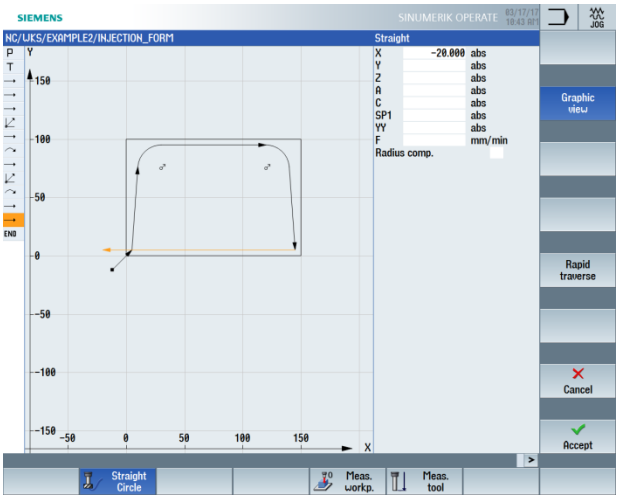
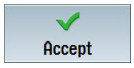


Figure 7-13 Entering the straight line



Accept the entered values.

Select the "Straight" softkey.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
X	-12 abs	X	
Y	-12 abs	X	
Radius compensation	Off	X	In the last traversing path, the tool traverses at the entered safety distance, and the radius compensation is switched off for this.

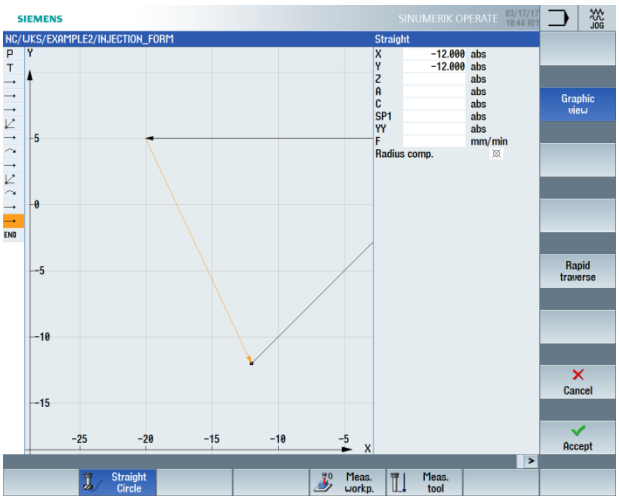
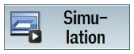


Figure 7-14 Entering the straight line - Safety distance

Accept the entered values.



The following simulation shows the machining sequence you need to check before machining the workpiece.

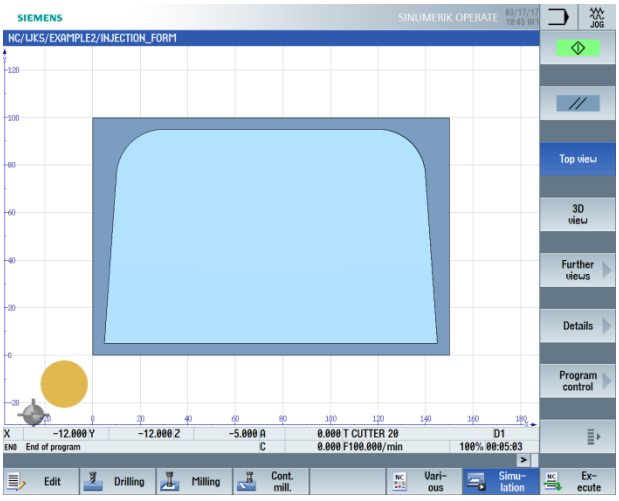


Figure 7-15 Simulation - Top view

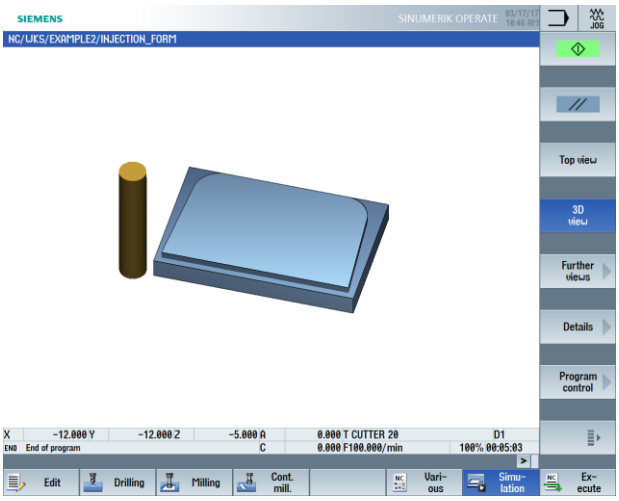


Figure 7-16 3D simulation

8.3 Rectangular pocket

Operating sequences

Follow the steps below to enter the rectangular pocket:

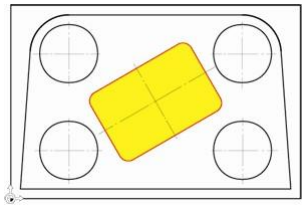


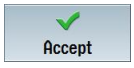
Figure 7-17 Rectangular pocket – Example 2

Select the "Mill." softkey.

Select the "Pocket" softkey.

Select the "Rectang. pocket" softkey.

Open the tool list and select "CUTTER10".



Apply the tool to the program.

After the tool has been applied, 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 is set to "Single position".
X0	75		Specify the geometrical data for the rectangular pocket in these fields: Position, width, length, …
Y0	50		
Z0	0		
W	40		
L	60		
R	6		
α0	30		
Z1	-15 abs	X	
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 cutter diameter or directly in mm. The maximum infeed in the plane is specified in % here.
DZ	2.5		
UXY	0.3		
UZ	0.3		
Insertion	Helical	X	Select "helical insertion" if not already set.
EP	2 mm/rev	X	
ER	2		

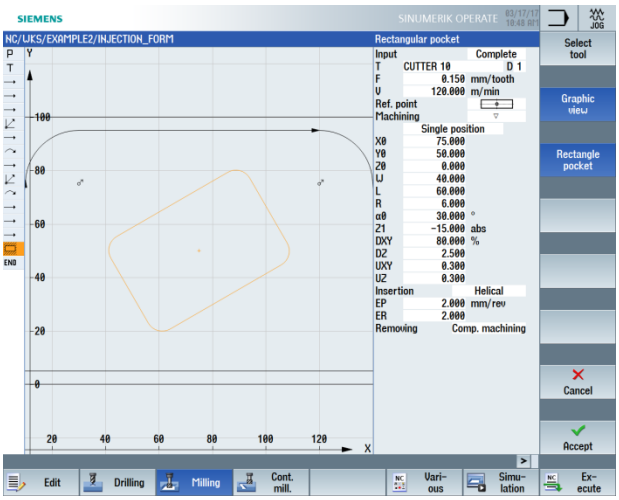
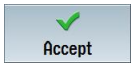


Figure 7-18 Roughing a rectangular pocket



Accept the entered values.

Select the "Pocket" softkey.

Enter the following values in the screen form:

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

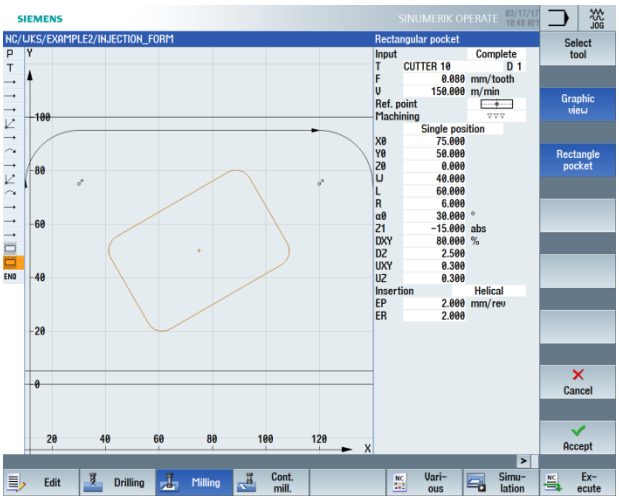
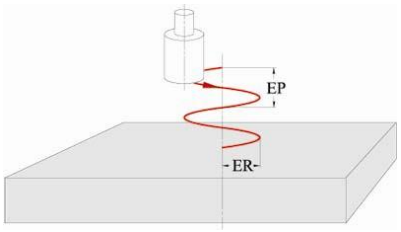
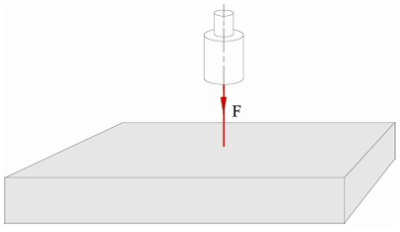
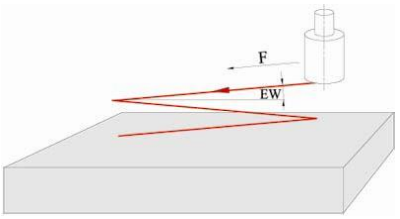


Figure 7-19 Finishing a rectangular pocket

Accept the entered values.

Insertion

Helical insertion	Perpendicular insertion	Oscillating insertion
		

8.4 Circular pockets on a position pattern

Operating sequences

Follow the steps below to enter the circular pockets:

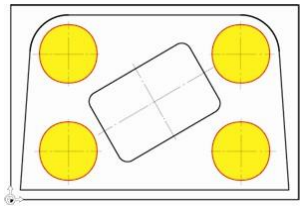
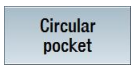


Figure 7-20 Circular pockets – Example 2

Select the "Mill." softkey.

Select the "Pocket" softkey.



Select the "Circular pocket" softkey.

Open the tool list and select "CUTTER10".

Apply the tool to the program.

After the tool has been applied, 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 create pockets on a position pattern.
Ø	30	X	
Z1	-10 abs	X	
DXY	80%	X	Specify the maximum infeed in the plane in %.
DZ	5		
UXY	0.3		
UZ	0.3		

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

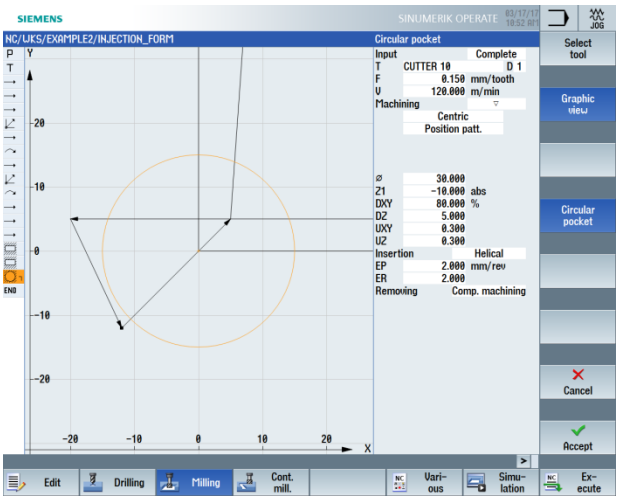
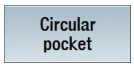


Figure 7-21 Roughing a circular pocket

Accept the entered values.

Select the "Pocket" softkey.



Select the "Circular pocket" softkey.

Enter the following details:

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

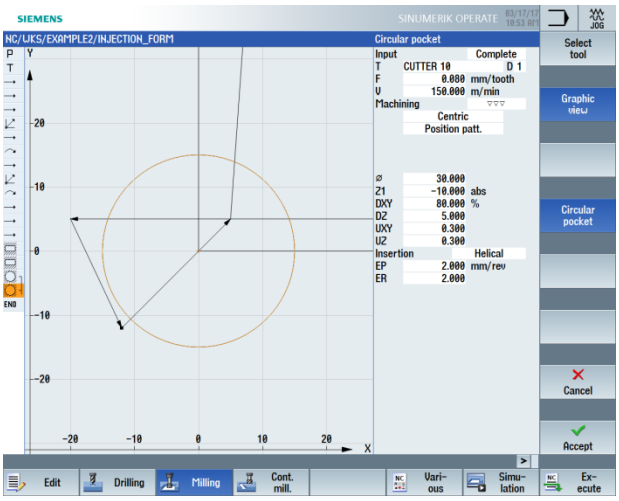


Figure 7-22 Finishing a circular pocket

Accept the entered values.

Select the "Drill." softkey.

Select the "Positions" softkey.

Select the "Position grid" softkey.

Enter the following details:

Field	Value	Selection via toggle key	Notes
Pattern	Grid	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		

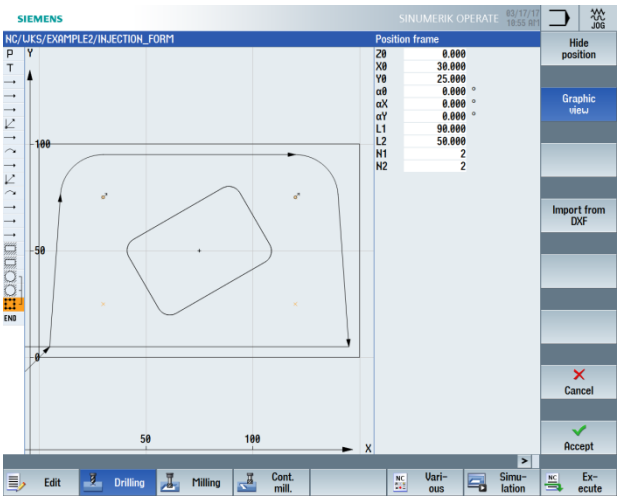


Figure 7-23 Positions of the circular pockets

Accept the entered values.

Start the simulation.

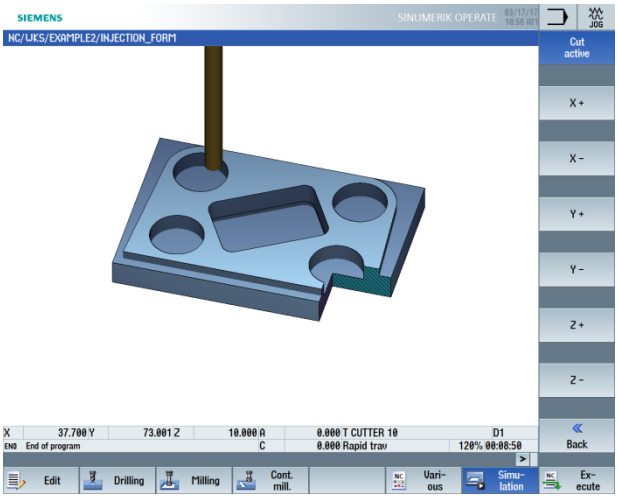


Figure 7-24 Simulation – Cut active

9. Example 3: Mold plate

9.1 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 operations to several planes
- Take obstacles into account

Task

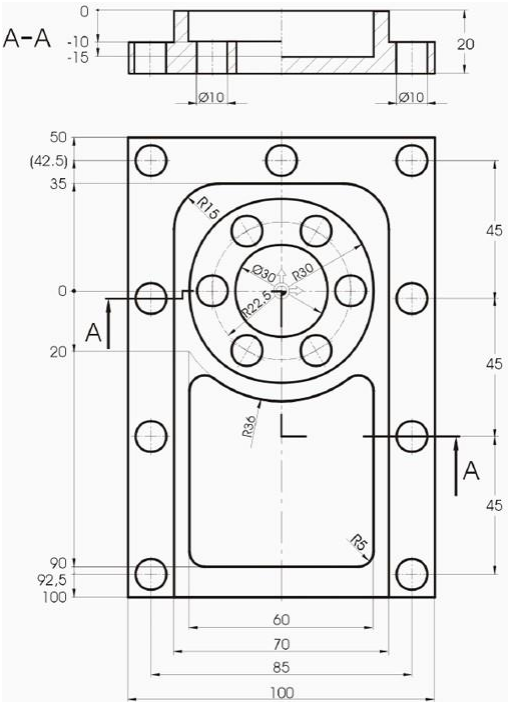


Figure 8-1 Workshop drawing – Example 3

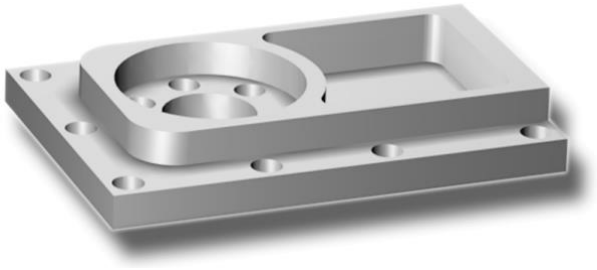


Figure 8-2 Workpiece – Example 3:

Preparations

Perform the following steps on your own:

1. Create a new workpiece with the name "Example3".
2. Create a new machining plan with the name "MOLD_PLATE".
3. Enter the blank dimensions (for the procedure, see example 1).

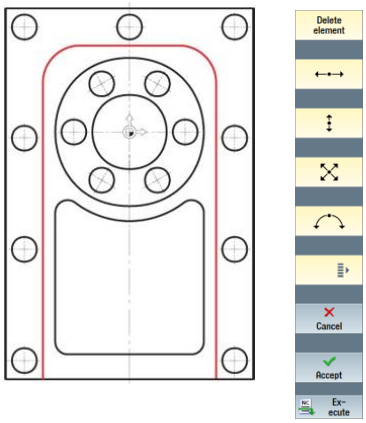
Note:

Pay attention to the new zero position!

9.2 Path milling of open contours

Contour calculator

The integrated ShopMill contour calculator allows you to enter even the most complicated contours easily.



With the graphic contour calculator, you can enter the contours faster and more easily than with conventional programming – and without any math.

Operating sequences

Follow the steps below to enter the contour: Select the "Contour milling" softkey.

Select the "New contour" softkey. Enter the name "MOLD_PLATE_Outside" for the contour.

Every contour is assigned its own name to make programs easier to read.

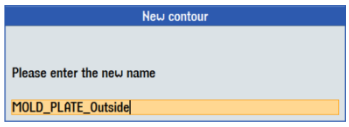


Figure 8-3 Creating the "MOLD_PLATE_Outside" contour

Accept your input.

Enter the following values for the starting point of the contour definition in the screen form:

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

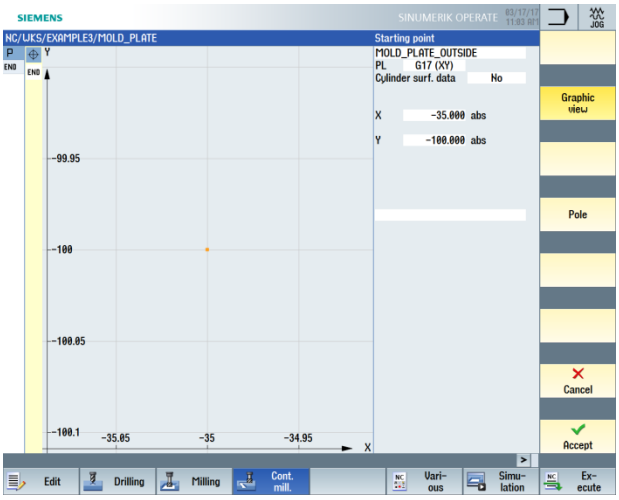
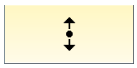


Figure 8-4 Entering the starting point

Note:

You only describe the workpiece contour here. The approach path and retraction path are not defined until later.

Accept the entered values.



Enter the following values for the straight line in the screen form:

Field	Value	Selection via toggle key	Notes
Y	35 abs	X	The first contour element is a vertical straight-line segment 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		

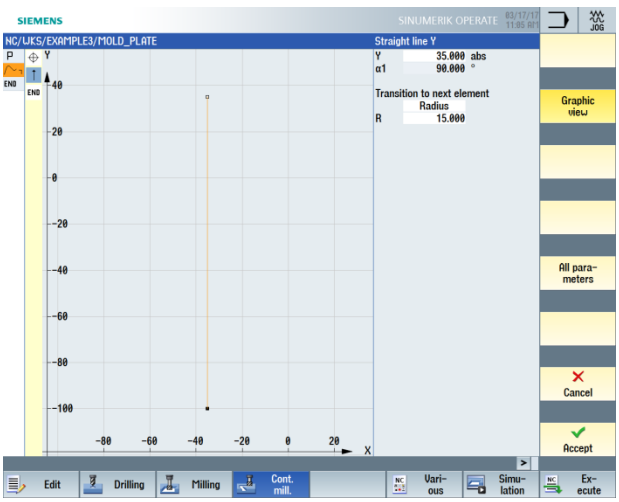
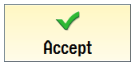
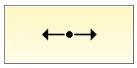


Figure 8-5 Entering the vertical straight-line segment for the contour



Accept the entered values.



Enter the following values for the horizontal straight line in the screen form:

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

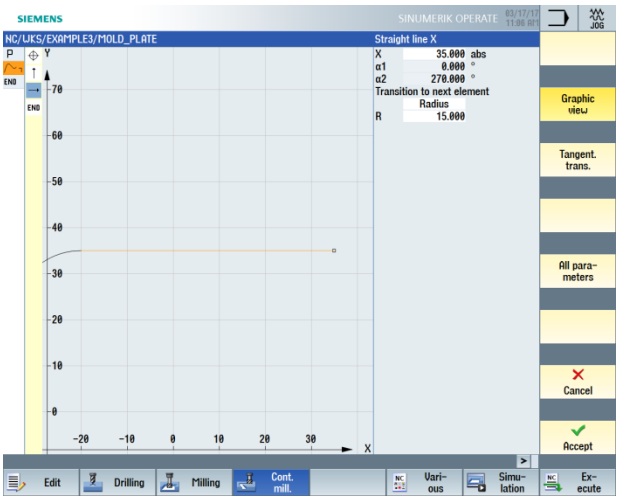


Figure 8-6 Entering the horizontal straight-line segment for the contour

Accept the entered values.

Enter the following values for the vertical straight line in the screen form:

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

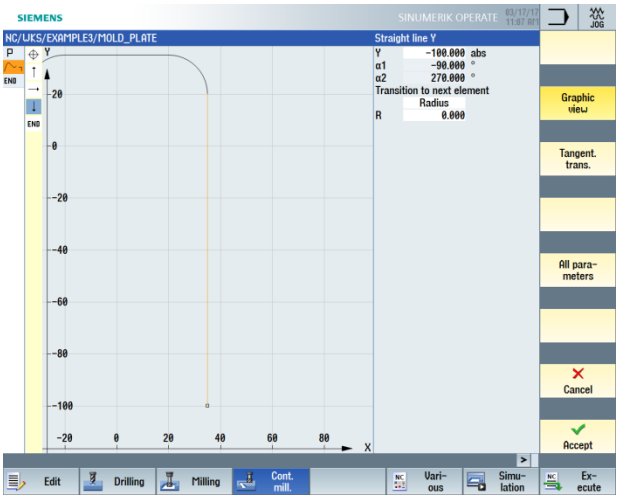
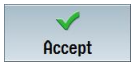


Figure 8-7 Entering the vertical straight-line segment for the contour

Accept the entered contour.

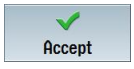


Accept the contour to apply it to your machining plan.

To be able to machine the created contour, you must now create the following machining steps. Proceed as follows:

Select the "Path milling" softkey.

Open the tool list and select "CUTTER32".



Apply the tool to the program.

Enter the following values for roughing in the screen form:

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 backwards, opposite the construction 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". The advantage of this is that in all cases only the actual depth of the pocket can be entered without a sign. This makes input easier for you, in particular with nested pockets.
DZ	5		
UZ	0.3		
UXY	0.3		
Approach	Straight	X	The approach can be in a quarter circle or semicircle, perpendicular or on a straight line. Here, it is appropriate to approach the contour tangentially on a straight line.
L1	5		For approach length L1, you do not have to take the cutter radius into account. It is offset automatically by ShopMill.
FZ	0.1 mm/tooth	X	
Retract	Straight	X	
L2	5		
Lift mode	To retraction plane	X	

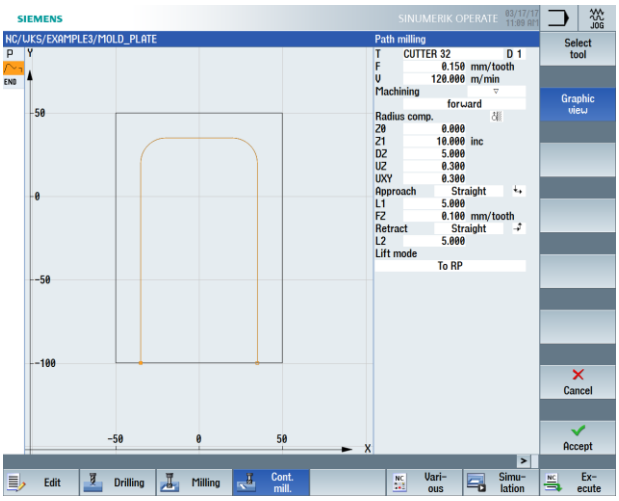


Figure 8-8 Roughing the contour

Accept the entered values.

Enter the following values for finishing in the screen form:

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

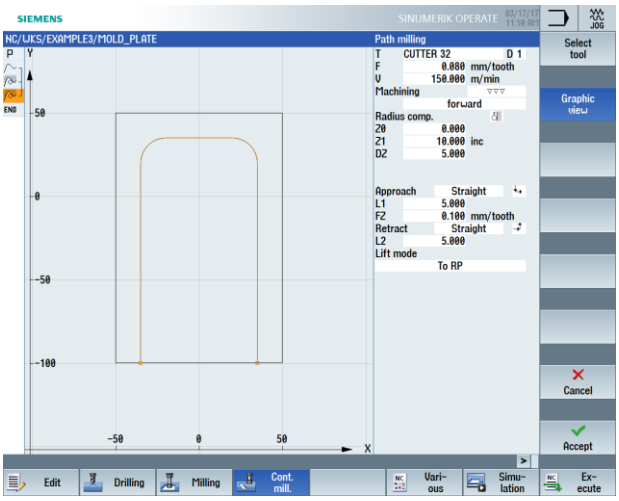


Figure 8-9 Finishing the contour

Accept the entered values.

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

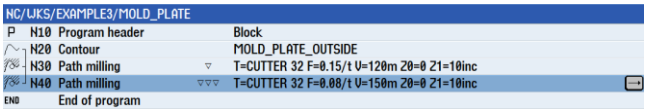


Figure 8-10 Linking of the machining steps in the machining plan

The following simulation shows the machining sequence you need to check before machining the workpiece.

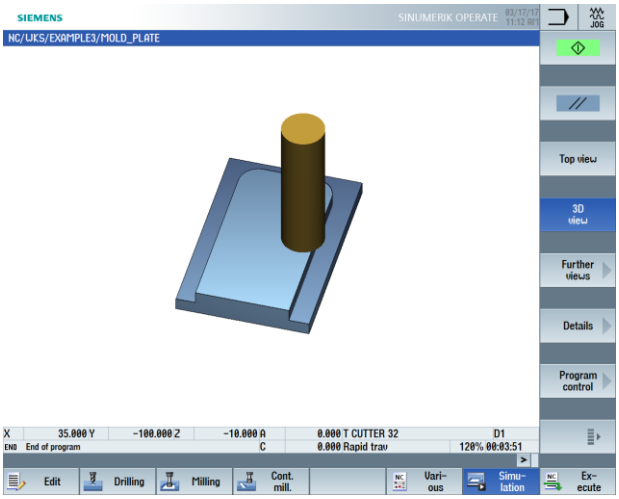


Figure 8-11 Simulation – Outside contour

9.3 Solid machining, residual material and finishing of contour pockets

Operating sequences

Follow the steps below to enter the pocket contour. The pocket is then removed from the solid by machining and finished.

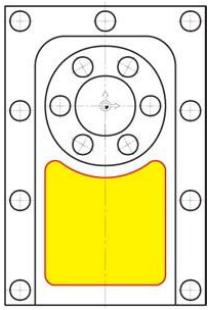


Figure 8-12 Pocket contour

Select the "Cont. mill." softkey.

Select the "New contour" softkey. Enter the name "MOLD_PLATE_Inside" for the contour.

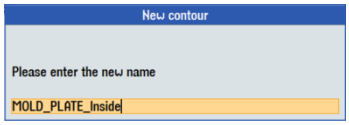


Figure 8-13 Creating the "MOLD_PLATE_Inside" contour

Accept your input.

Enter the following values for the starting point in the screen form:

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

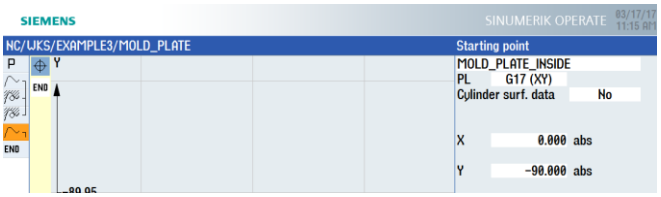


Figure 8-14 Entering the starting point

Accept the entered values.

Enter the following values for the horizontal straight line in the screen form:

Field	Value	Selection via toggle key	Notes
X	25 abs	X	For practice, specify the first arc not as a rounding but rather as a separate element. Therefore, construct the straight line only up to X25.

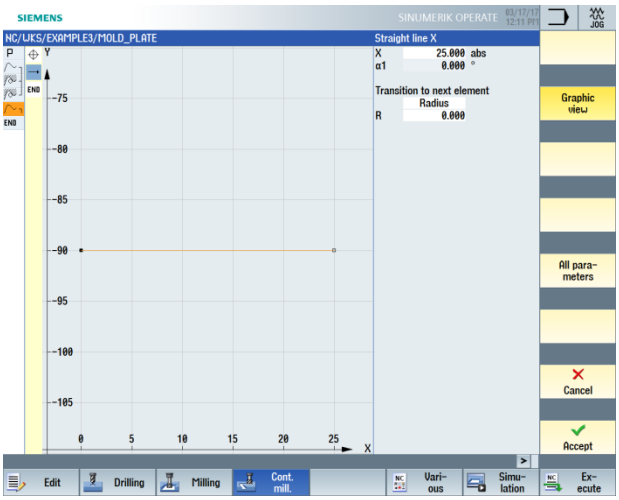


Figure 8-15 Entering the horizontal straight-line segment for the contour

Accept the entered values.

Enter the following values for the arc in the screen form:

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

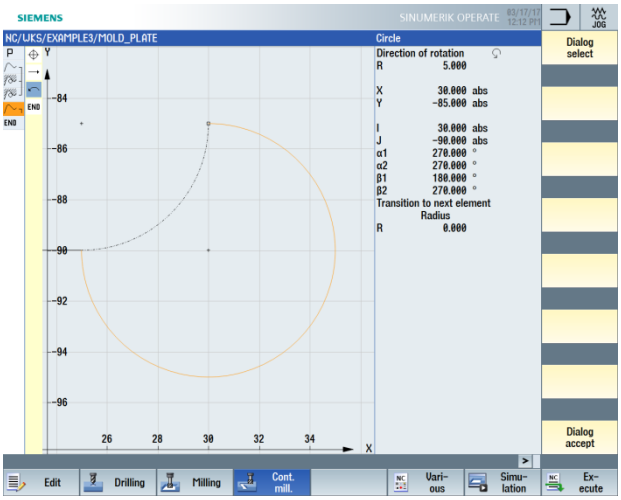


Figure 8-16 Arc for the contour (bottom right)

Dialog select

Two construction solutions result after entering the Y end point. Select the desired solution using the "Dialog select" softkey. The selected solution then turns orange, and the alternative solution is displayed as a black dotted line.

Dialog accept

Accept your selection. The geometry processor automatically detects that the programmed arc is connected tangentially to the straight line. The appearance of the "Tangent. trans." softkey changes (i.e. is pressed).

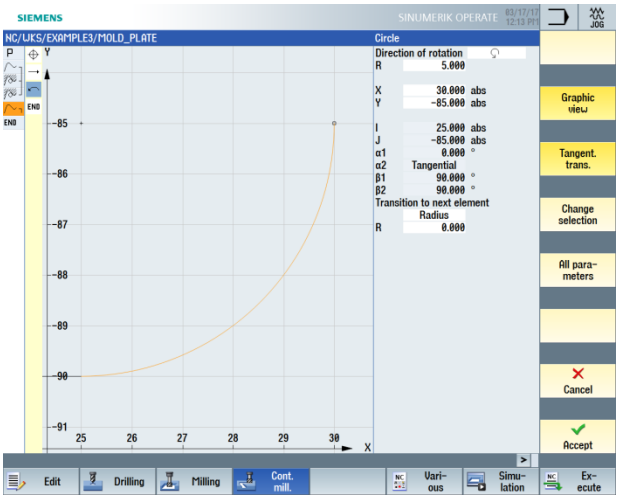


Figure 8-17 Arc for the contour – After selection

Accept

Accept the entered values.

Enter the following values for the vertical straight line in the screen form:

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

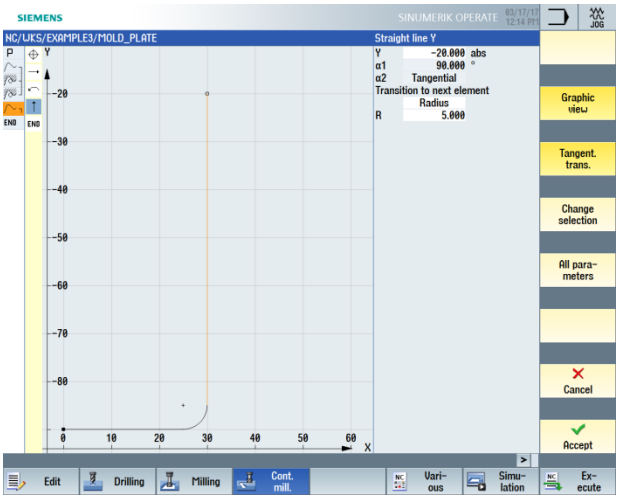


Figure 8-18 Entering the vertical straight-line segment for the contour

Accept the entered values.

Enter the following values for the arc in the screen form:

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	

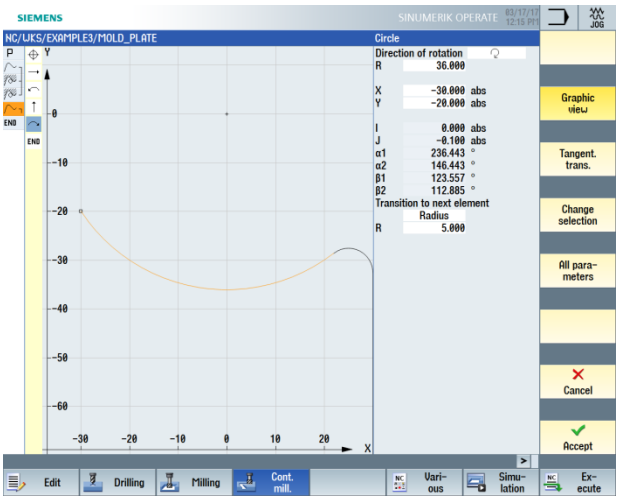
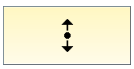


Figure 8-19 Entering the arc for the contour

Accept the entered values.



Enter the following values for the vertical straight line in the screen form:

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

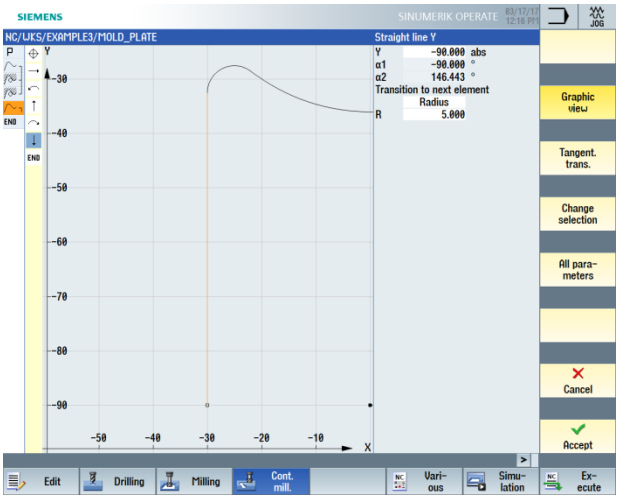
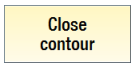


Figure 8-20 Entering the vertical straight-line segment for the contour

Accept the entered values.



Close the contour. The pocket contour is now fully described.

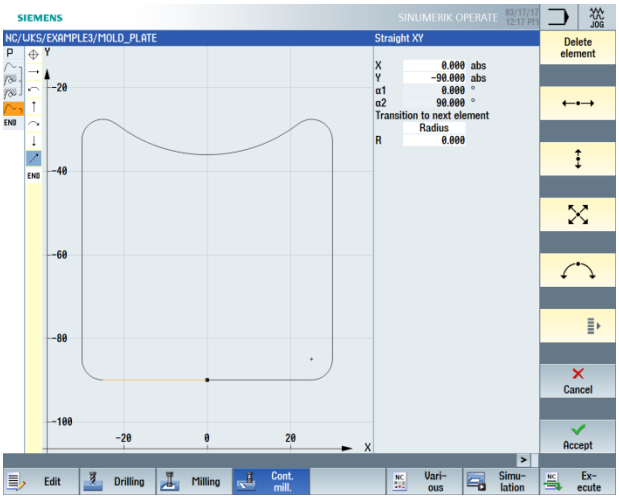


Figure 8-21 Closing the contour

Accept the contour to apply it to your machining plan.

Select the "Pocket" softkey.

Open the tool list and select "CUTTER20".

Apply the tool to the program.

Note:

The machining direction of the pocket has already been defined in the program header. The "Down-cut" setting was selected in this case.

Enter the following values for roughing in the screen form:

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 enter the machining depth as an incremental dimension, you must enter a positive value for the depth.
DXY	50%	X	
DZ	5		
UXY	0.3		
UZ	0.3		
Starting point	automatically	X	If you select the "automatically" setting for the starting point (insertion position), the starting point is specified by ShopMill.
Insertion	Helical	X	Set insertion to Helical with 2 mm/revolution pitch and 2.00 mm radius.
EP	2 mm/rev	X	
ER	2		
Lift mode	To retraction plane	X	

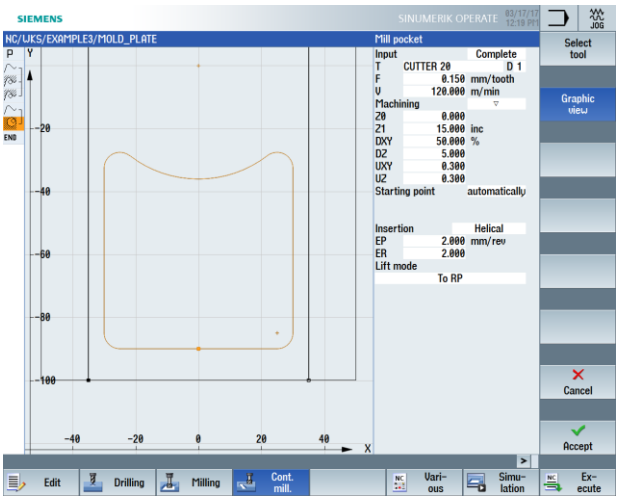
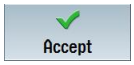


Figure 8-22 Roughing a pocket



Accept the entered values.

Select the "Pocket res. mat." softkey. Because the 20 mm cutter cannot machine R5 radii, material will remain in the corners. Use the "Pocket reside. mat." function to rough-machine the areas not yet machined with pinpoint accuracy.

Open the tool list and select "CUTTER10".

Apply the tool to the program.

Enter the following values in the screen form:

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		

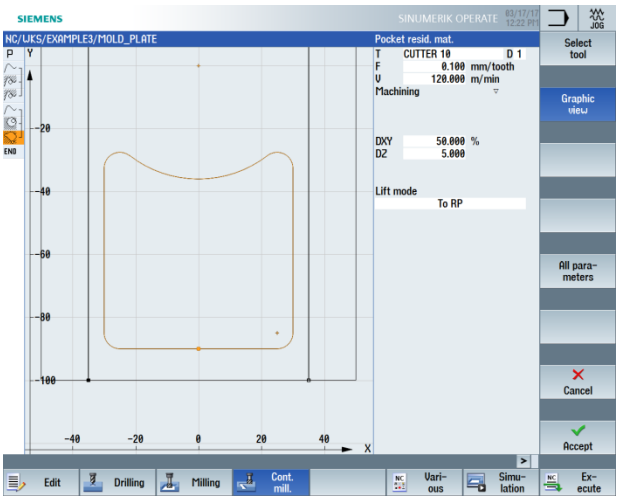


Figure 8-23 Machining residual material of the pocket

Accept the entered values.

Select the "Pocket" softkey.

Open the tool list and select "CUTTER10".

Apply the tool to the program.

Enter the following values for the re-machining of the pocket in the screen form:

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

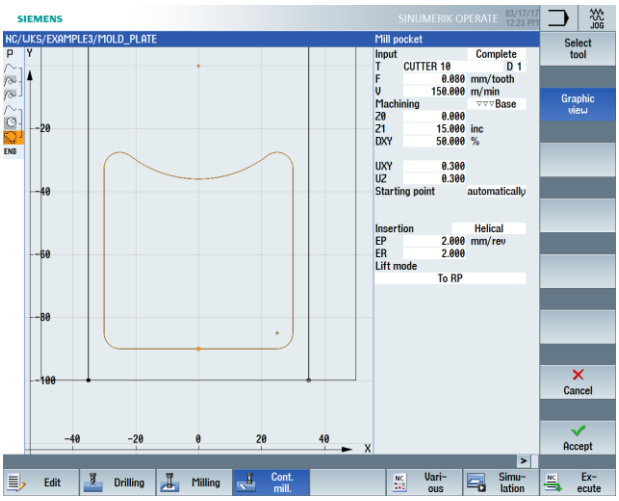


Figure 8-24 Finishing a pocket

Accept the entered values.

Select the "Pocket" softkey.

Enter the following value for removing the residual material from the contour In the screen form:

Field	Value	Selection via toggle key	Notes
Machining	Wall	X	

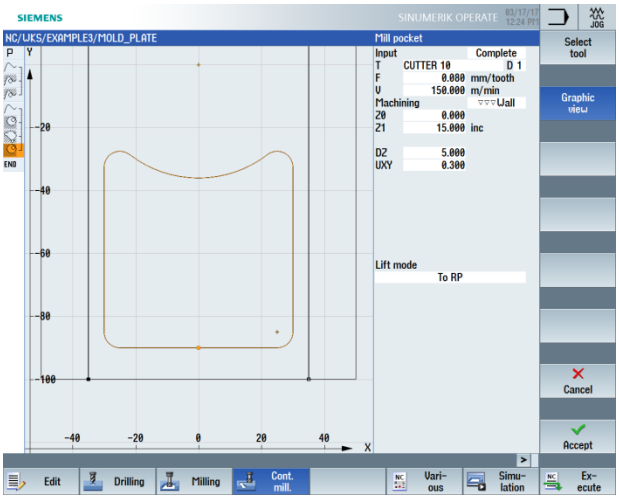


Figure 8-25 Finishing the wall

Accept the entered values.

9.4 Machining on several planes

Operating sequences

Mill the 60 mm circular pocket as in the "INJECTION_FORM" example in two machining steps.

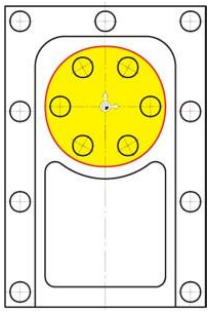


Figure 8-26 Circular pocket

1. In the first machining step, the pocket is machined by roughing with the 20 mm milling cutter to -9.7 mm.

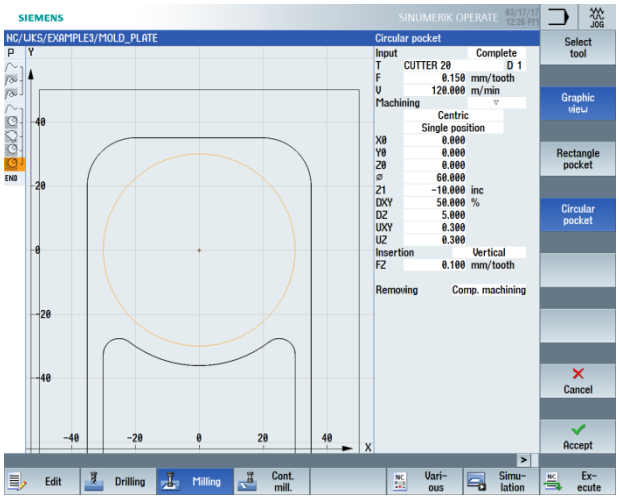


Figure 8-27 Roughing the circular pocket

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

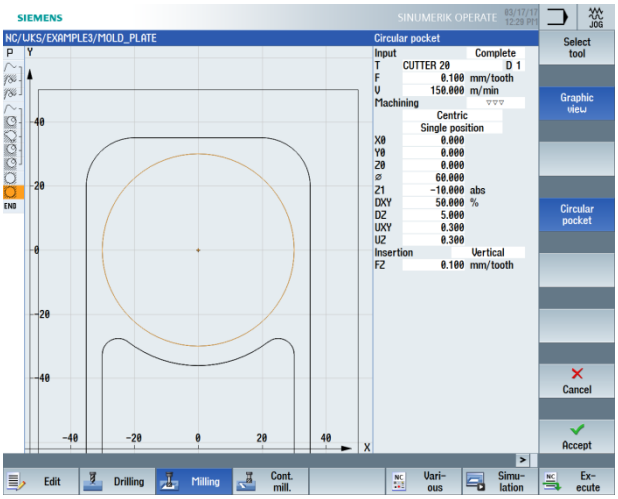


Figure 8-28 Finishing the circular pocket

Use the following steps to specify the machining of the inside circular pocket: The circular pocket is then machined to a depth of -20 mm.

Note:

The starting depth is now no longer at 0 mm, but at -10 mm!

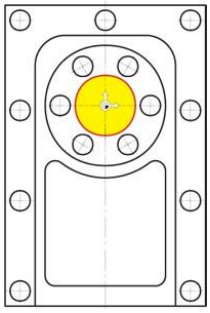
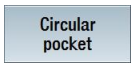


Figure 8-29 Inside circular pocket

Select the "Mill." softkey.

Select the "Pocket" softkey.



Enter the following values for machining of the circular pocket in the screen form:

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		
Ø	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	

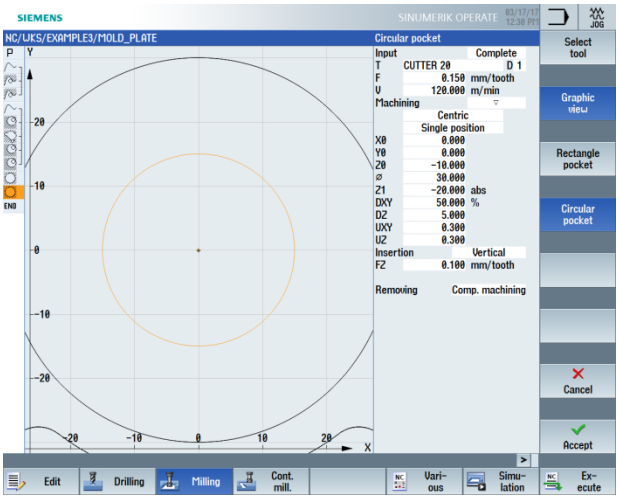


Figure 8-30 Roughing the inside circular pocket

Accept the entered values.

Select the "Mill." softkey.

Select the "Pocket" softkey.

Circular pocket

Enter the following values for machining of the circular pocket in the screen form:

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

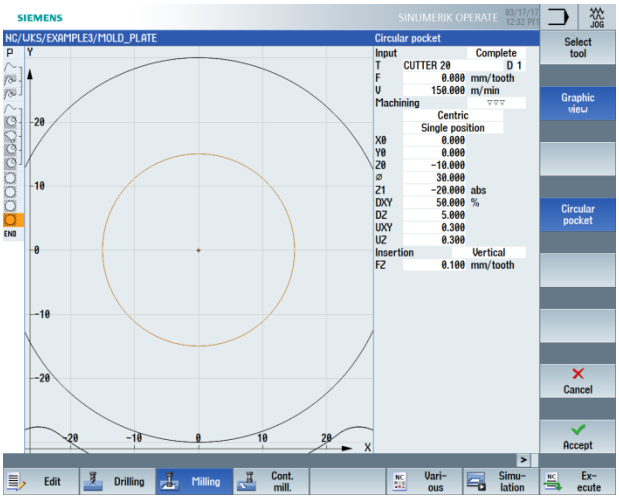


Figure 8-31 Finishing the inside circular pocket

Accept

Accept the entered values.

Simu-lation

Start the simulation.

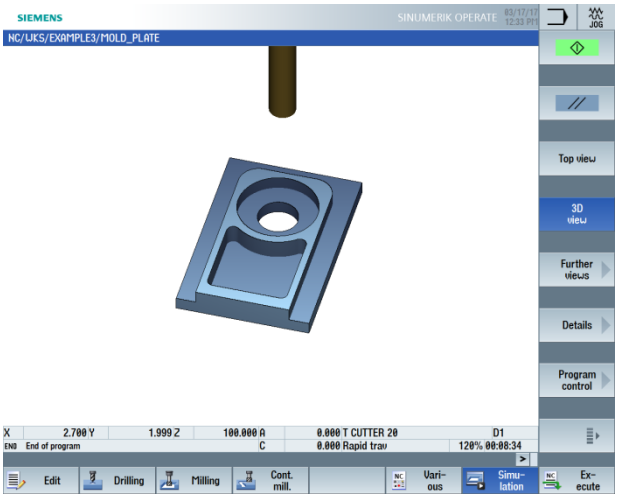


Figure 8-32 Simulation in 3D view

9.5 Taking obstacles into account

Operating sequences

The linking of various drill patterns, as you learned in example 1, can also be used for this workpiece as well. Here, however, it is necessary to bypass one or more obstacles – depending on the sequence of machining. Between the drill holes, the tool traverses to the safety distance or to the machining plane – depending on your setting.

First, create the centering and drilling machining steps as done in example 1.

1. Centering

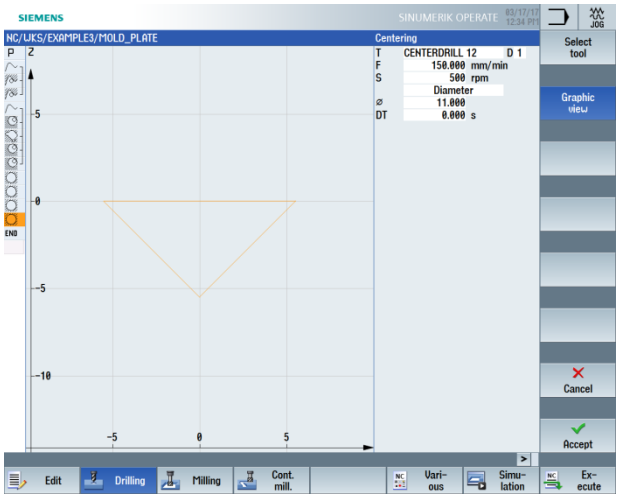


Figure 8-33 Centering machining step

2. Drilling

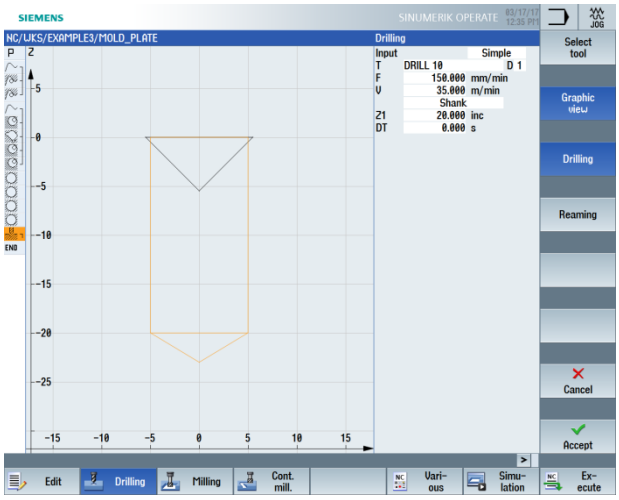


Figure 8-34 Drilling machining step

Follow the steps below to enter the associated 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 screen form:

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		

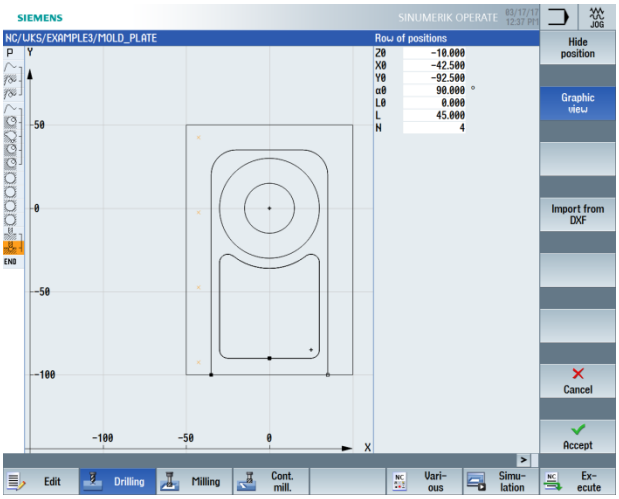
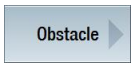


Figure 8-35 Entering the line of holes

Accept the entered values.

Select the "Positions" softkey.



Use the "Obstacle" function to specify a traversing path of 1 mm, since the right line of holes is to be drilled next from bottom to top next for practice purposes.
You only have to enter the obstacle if you have switched the "Retraction position pattern" toggle field in the program header to "Optimized" beforehand.

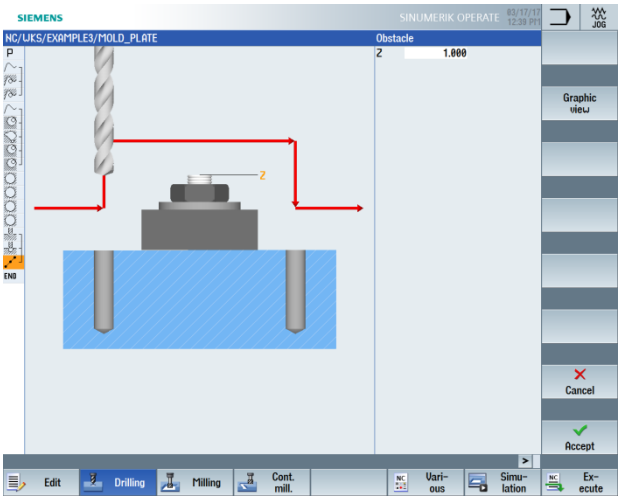


Figure 8-36 Entering the obstacle

Accept the entered values.

Select the "Positions" softkey.

Enter the following values for the second line of holes in the screen form:

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		

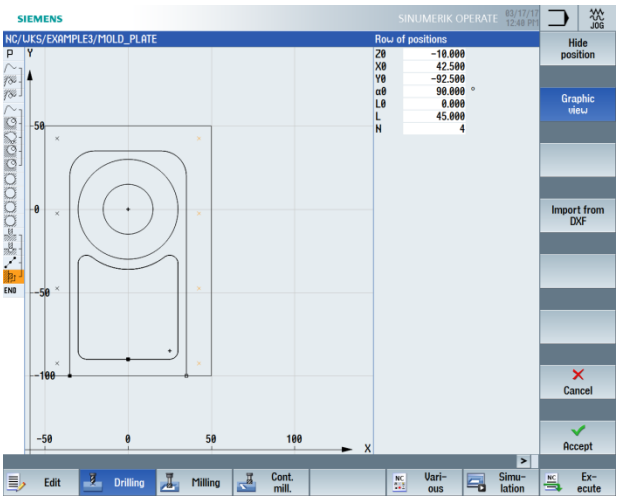
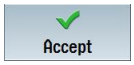


Figure 8-37 Entering the line of holes



Accept the entered values.

Select the "Positions" softkey.

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

Accept the entered value.

Select the "Positions" softkey.

Enter the following values for the six drill holes in the full circle in the screen form:

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	X	

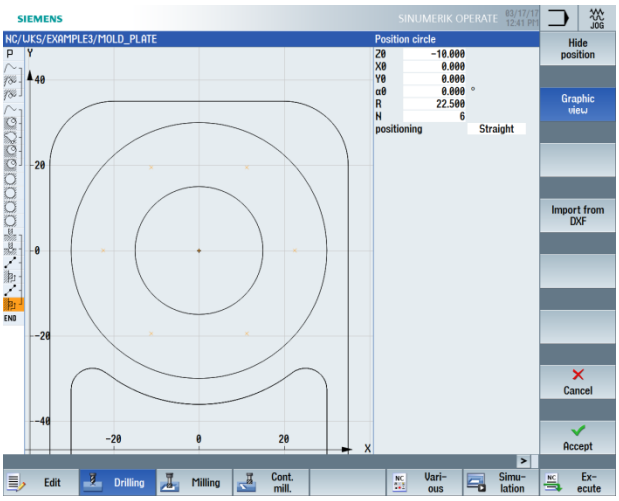
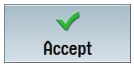


Figure 8-38 Entering the drill holes in the full circle



Accept the entered values.

Select the "Positions" softkey.

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

Accept the entered value.

Select the "Positions" softkey.

Enter the following values for the last drilling positions in the screen form:

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		

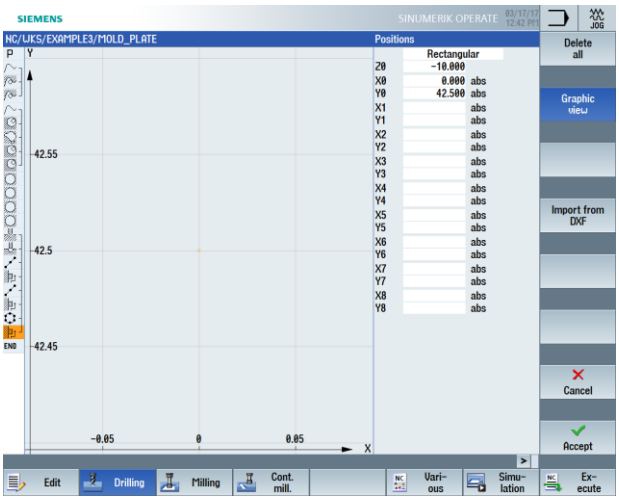


Figure 8-39 Entering the drilling positions

Accept the entered 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 with only one obstacle.

Try out different strategies and decide which is the best for you.

Start the simulation.

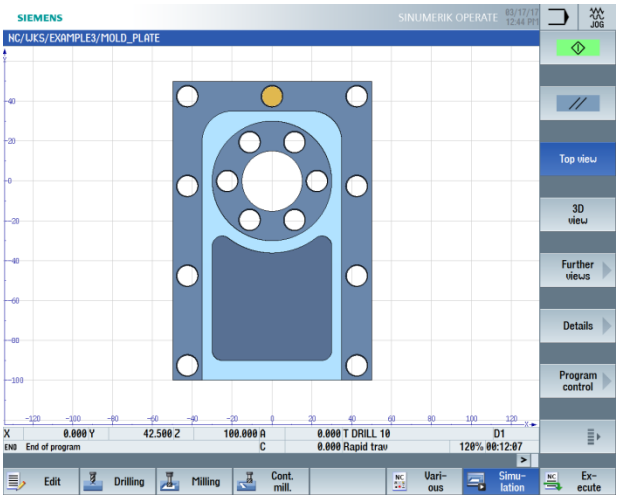


Figure 8-40 Simulation - Top view

10. Example 4: Lever

10.1 Overview

Learning objectives

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

- Perform face milling
- Create borders (auxiliary pockets) for solid machining around islands
- Create and copy circular islands
- Work with the machining step editor to machine islands
- Perform deep-hole drilling, helix milling, boring and thread milling
- Program contours using polar coordinates (version 6.4 and higher)

Task

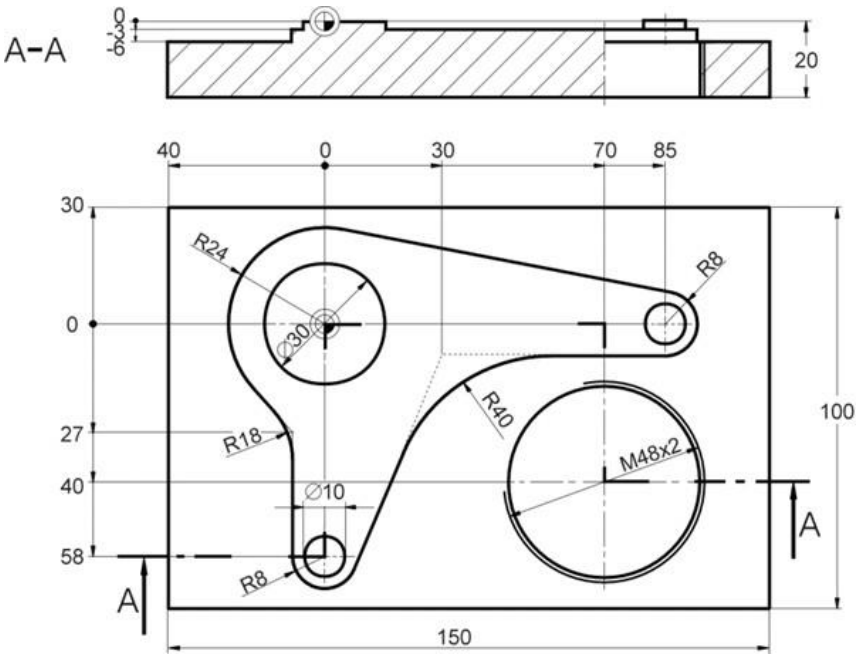


Figure 9-1 Workshop drawing – Example 4

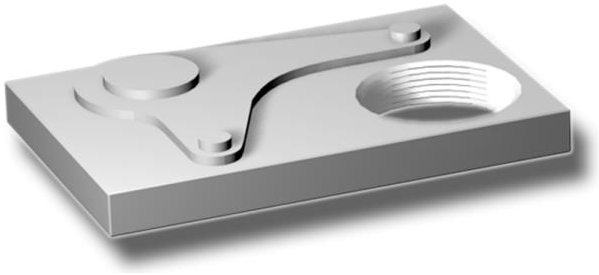


Figure 9-2 Workpiece – Example 4

Preparations

Perform the following steps on your own:

1. Create a new workpiece with the name "Example4".
2. Create a new machining plan with the name "LEVER".
3. Enter the blank dimensions (for the procedure, see example 1).

Note:

Note that the blank is to be 25 mm thick and you must therefore set ZA to 5 mm.

After entering the data, the program header should look like this:

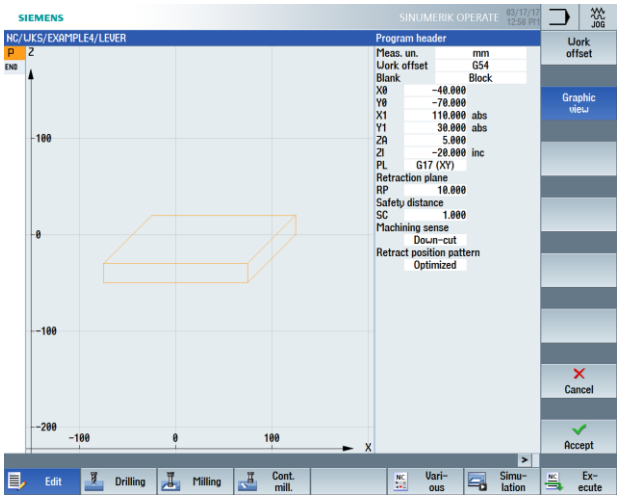


Figure 9-3 Workpiece dimensions in the program header

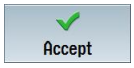
10.2 Face milling

Operating sequences

Select the "Mill." softkey.

Select the "Face milling" softkey.

Open the tool list and select "FACEMILL63".



Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.1 mm/tooth	X	
V	120 m/min	X	
Machining	Roughing	X	
Direction	Changing	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		

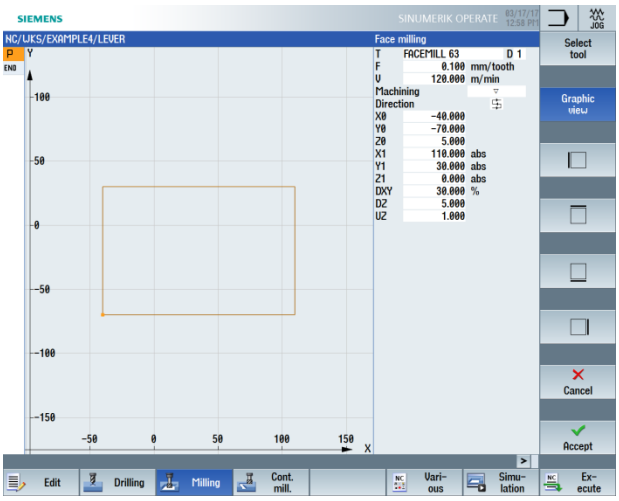
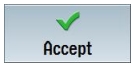


Figure 9-4 Roughing the area



Accept the entered values.

Select the "Face milling" softkey.

Enter the following values for finishing in the screen form:

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

Note:

The finishing allowance must have the same value for both roughing and finishing, because it refers to the subsequent finishing machining for the roughing step, and it refers to the thickness of the material still to be removed for the finishing step.

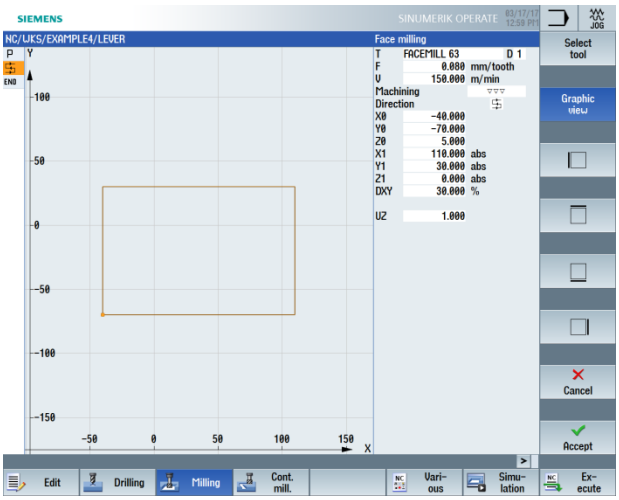
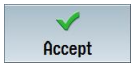


Figure 9-5 Finishing the area



Accept the entered values.

10.3 Creating the border for the lever island

Operating sequences

Note:

Just like pockets, islands are described as a contour in the graphic contour calculator. They only become islands through linking in the machining plan. The first contour in the machining plan always describes the pocket. One or more subsequent contours are interpreted as islands.

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

Select the "Cont. mill." softkey.

New contour

Create a new contour with the name "LEVER_Rectangular_Area".

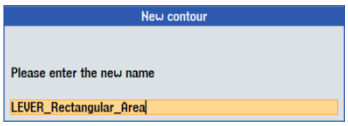


Figure 9-6 Creating the contour

Create the following contour on your own.
Round the corners with R15. Be sure to select the values in such a way that the workpiece corners are covered by the pocket.

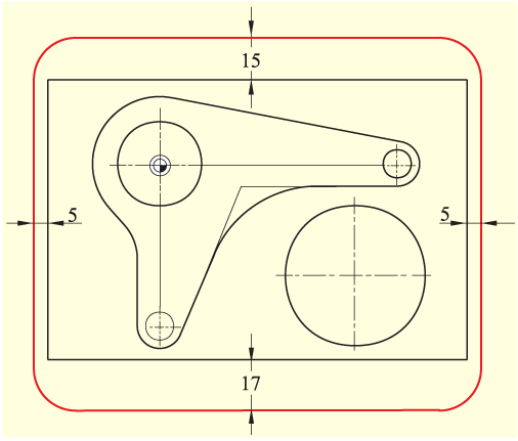


Figure 9-7 Border for the lever island

Compare your contour with the figure below.

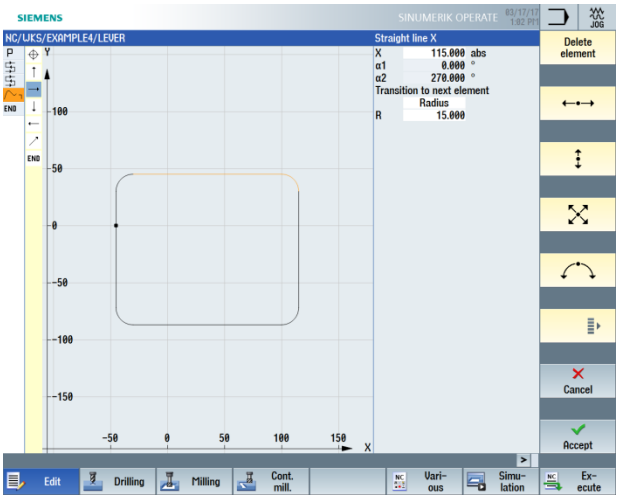


Figure 9-8 Completely constructed contour

10.4 Machining the lever

Operating sequences

Follow the steps below to enter the contour:

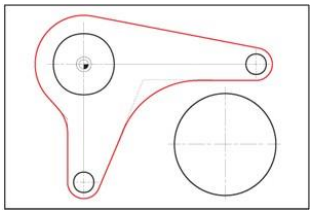


Figure 9-9 Lever contour

Select the "Cont. mill." softkey.

Create a new contour with the name 'LEVER_Lever'.

Figure 9-10 Creating the contour

Once applied, enter the following values for the starting point of the contour line in the screen form.

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

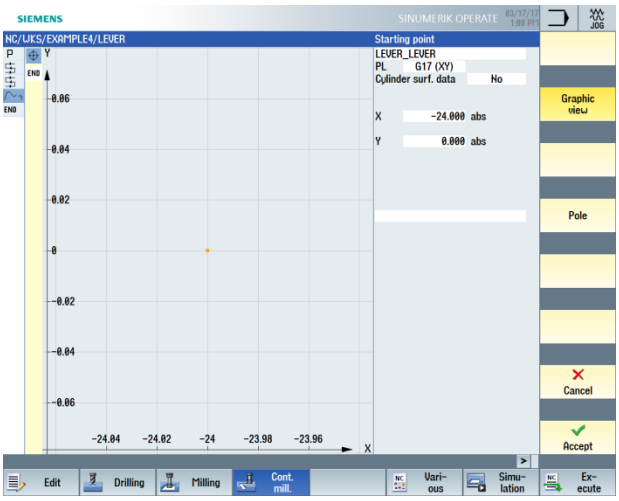


Figure 9-11 Creating the starting point

Accept the entered values.

Enter the following values for the first arc in the screen form:

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

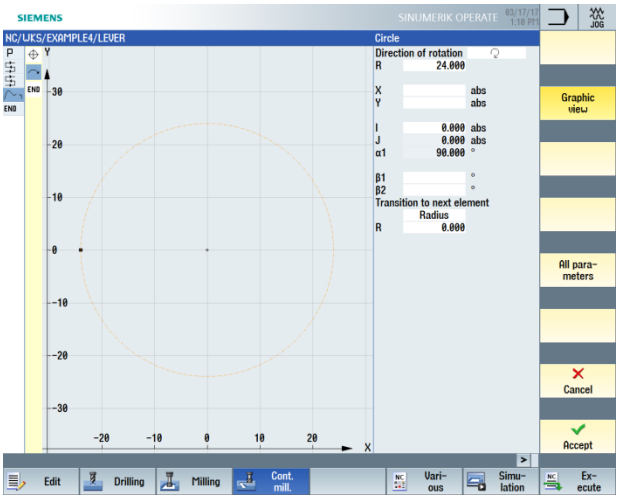


Figure 9-12 Arc for the contour

Accept the entered values.

Create the inclined straight line tangential to the preceding element.

Select the "Tangent trans." softkey.

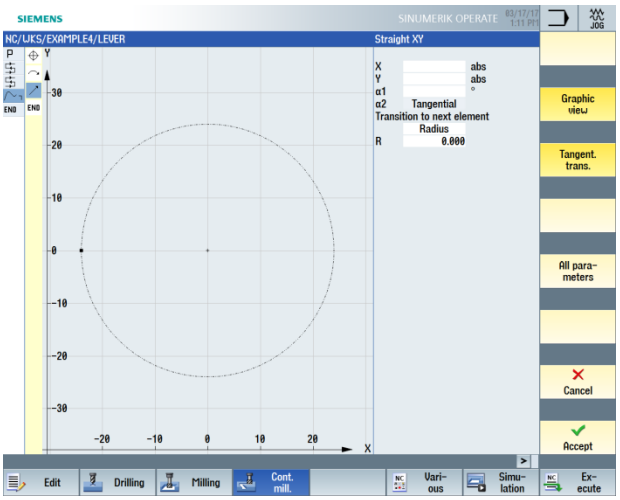


Figure 9-13 Inclined straight line for the contour

Accept your input.

Enter the tangential circular arc.

Select the "Tangent trans." softkey.

Enter the following values for the circular arc in the screen form:

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	

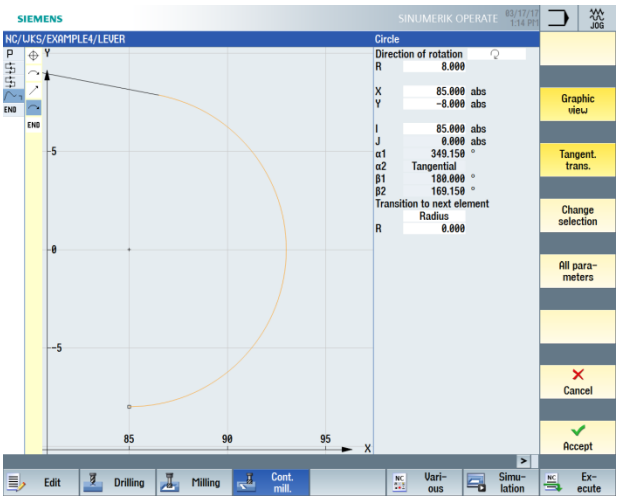
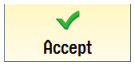


Figure 9-14 Arc for the contour

Accept the contour suggestion.



Accept the entered values.

Enter the following values for the horizontal straight-line segment up to end point X30 in the screen form:

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

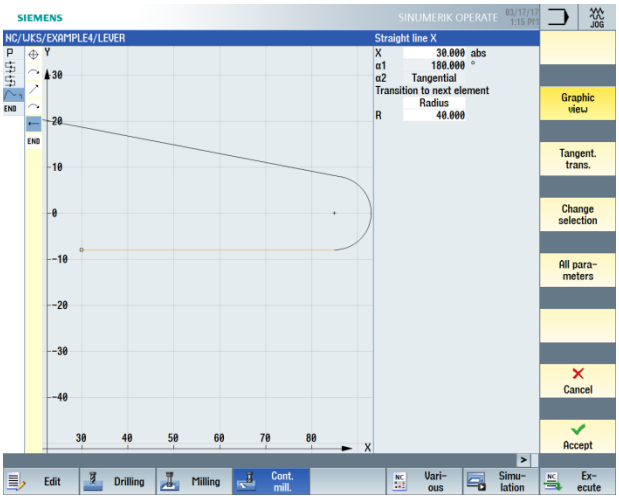


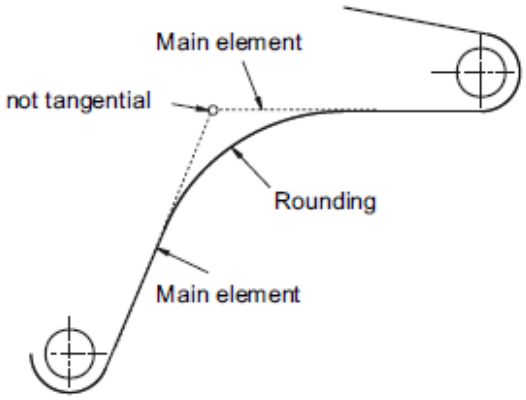
Figure 9-15 Horizontal straight-line segment for the contour

Accept the entered values.

Read the note below for the subsequent inclined straight-line segment:

Note:

The tangential transition is always referenced to the main element only. This means that the straight line does not connect tangentially in this case (see figure below).



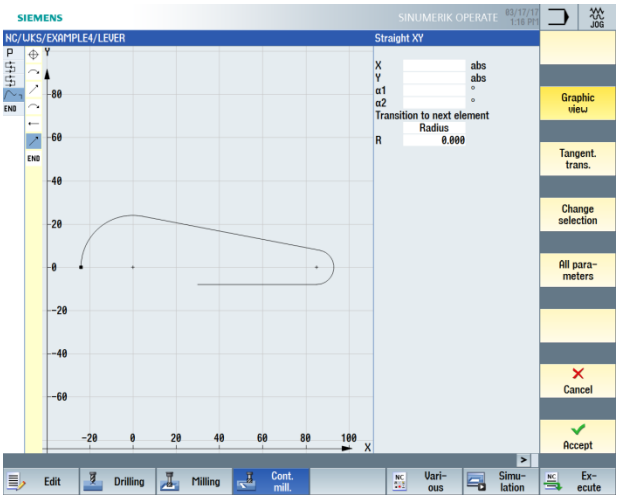
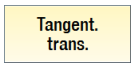


Figure 9-16 Inclined straight line for the contour

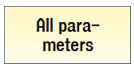
Accept

Accept your input.

Enter the tangential circular arc.



Select the "Tangent trans." softkey.



Select the "All parameters" softkey.

The "All parameters" function gives you detailed information on the arc.

This can be used, for example, to check the entered values
(e.g.: Does the arc end vertically?).

Enter the following values for the circular arc in the screen form:

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

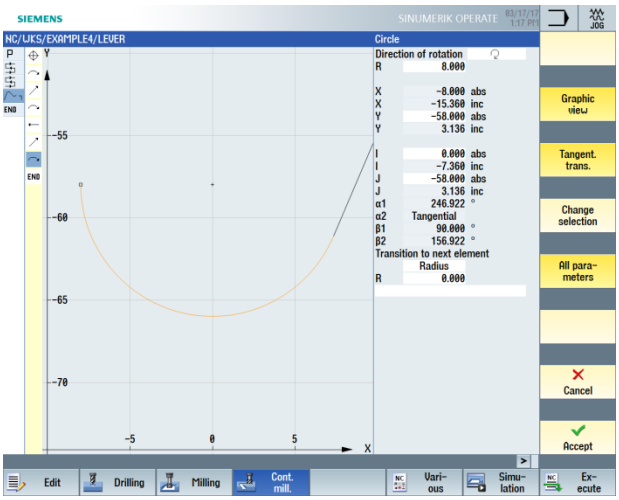
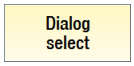
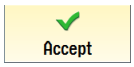


Figure 9-17 Arc for the contour

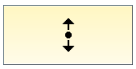


Select the desired contour suggestion.

Accept the contour suggestion.



Accept your input.



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

Select the "Tangent trans." softkey.

Enter the following values in the screen form:

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

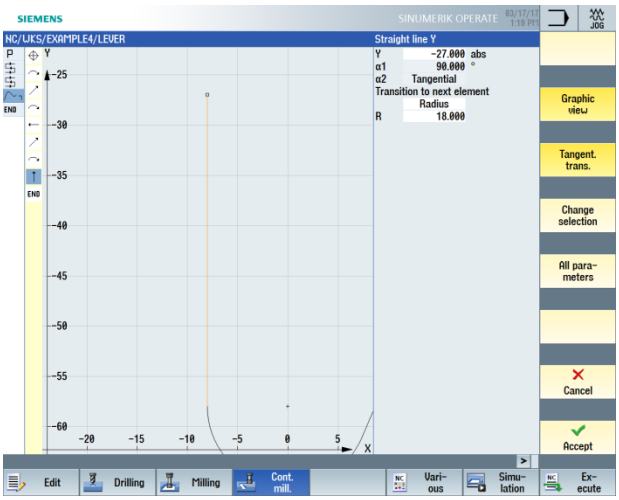


Figure 9-18 Vertical straight-line segment for the contour

Accept the entered values.

Specify the inclined straight line.

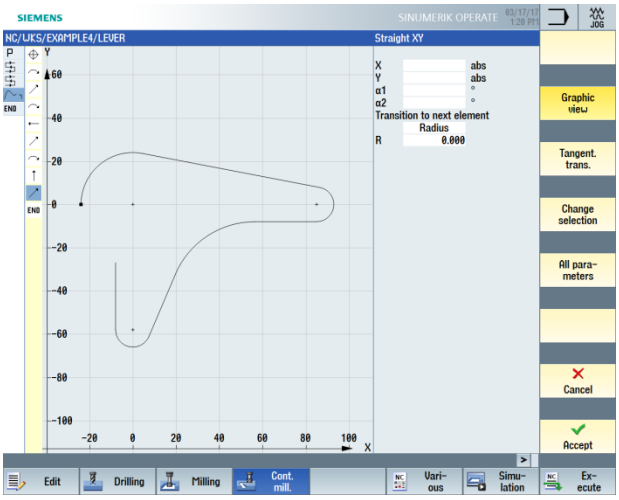
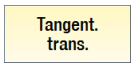


Figure 9-19 Inclined straight line for the contour

Accept your input.

Close the contour to the starting point with an arc.



Select the "Tangent trans." softkey.

Enter the following values for the starting point of the contour definition in the screen form:

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

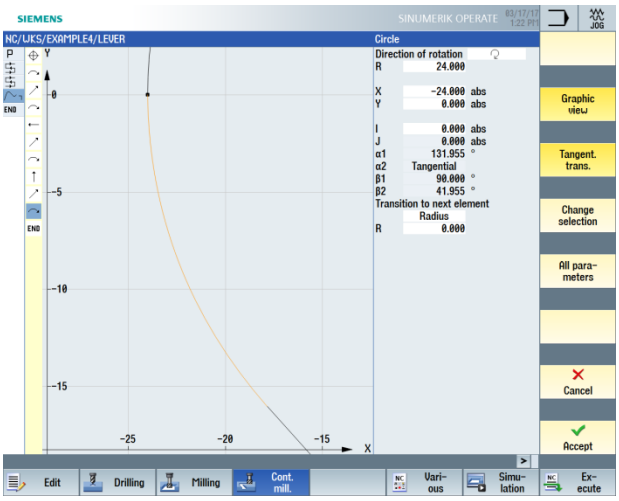
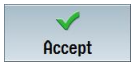


Figure 9-20 Arc for the contour

Accept the entered values.



Accept the contour.

Follow the steps below to rough and finish the pocket taking into account the lever contour:

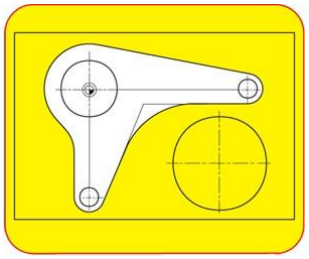


Figure 9-21 Roughing and finishing around the lever

Select the "Pocket" softkey.

Open the tool list and select the "CUTTER20" milling cutter.

Apply the tool to the program.

Enter the following values for roughing in the screen form:

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	automatically	X	
Insertion	Vertical	X	
FZ	0.15 mm/tooth	X	
Lift mode	To RP	X	

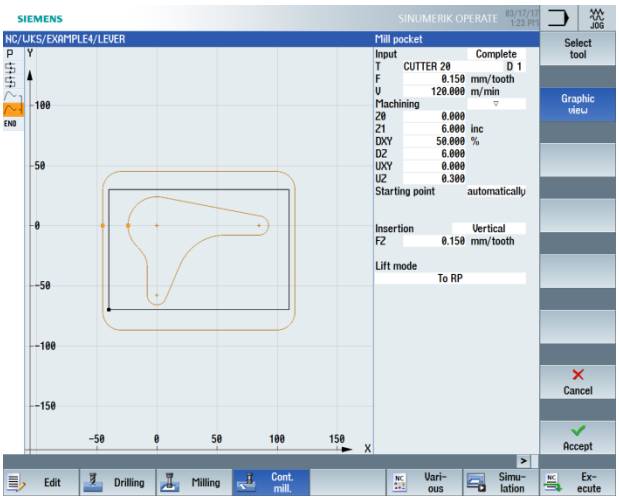


Figure 9-22 Roughing the contour

Accept the entered values.

Select the "Pocket" softkey.

Enter the following values for finishing in the screen form:

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

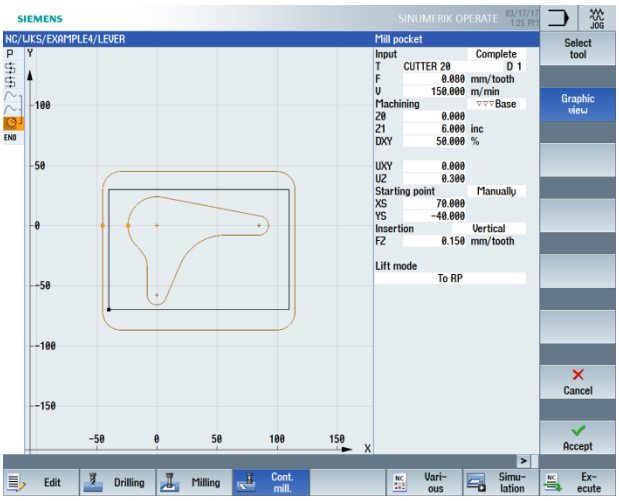
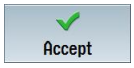


Figure 9-23 Finishing the base



Accept the entered values.

10.5 Creating the border for the circular island

Operating sequences

Create the border as the traversing path boundary for milling on your own
Mill to a depth of -3.

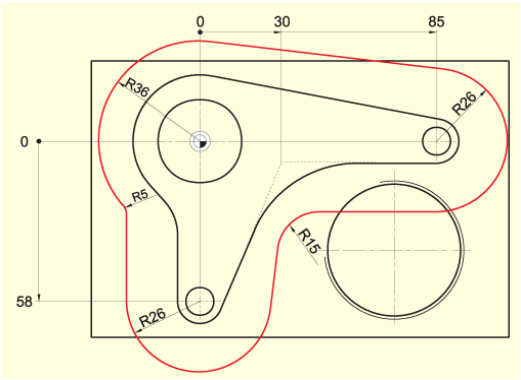


Figure 9-24 Contour of border for the circular islands

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 freely selected.

Select the "Cont. mill." softkey.

Create a new contour with the name "LEVER_Lever_Area".

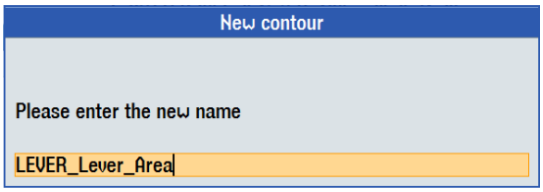


Figure 9-25 Creating the contour

Construct the boundary of the traversing paths as described above such that the 20 mm milling cutter fits through everywhere between the boundary and the islands. Enter this boundary contour in the same way as the lever contour.

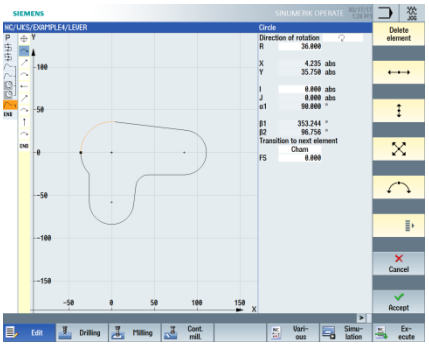


Figure 9-26 Arc contour section, left

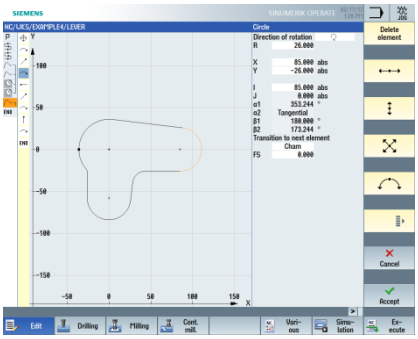


Figure 9-27 Arc contour section, right

10.6 Creating the 30 mm circular island

Operating sequences

Follow the steps below to create the 30 mm circular island shown in the figure.

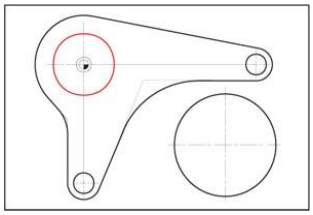


Figure 9-28 30 mm circular island

Select the "Cont. mill." softkey.

Create a new contour with the name "LEVER_Circle_R15".

Figure 9-29 Creating the contour

Create the circular contour on your own (see figure below). The starting point of the circle construction is at X-15 and Y0.

Note:

Make sure that some values are specified with incremental dimensions!

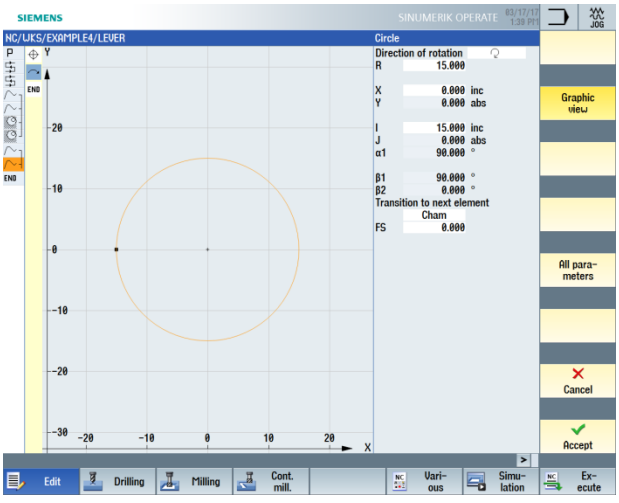


Figure 9-30 Circular island contour

10.7 Creating a 10 mm circular island

Operating sequences

Follow the steps below to create the 10 mm circular island shown in the figure:

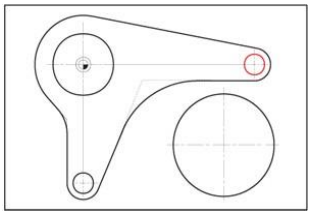


Figure 9-31 10 mm circular island

Select the "Cont. mill." softkey.

Create a new contour with the name "LEVER_Circle_R5_A".

Figure 9-32 Creating the contour

Create the circular contour on your own (see figure below). The starting point of the circle construction is at X80 and Y0.

Note:

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

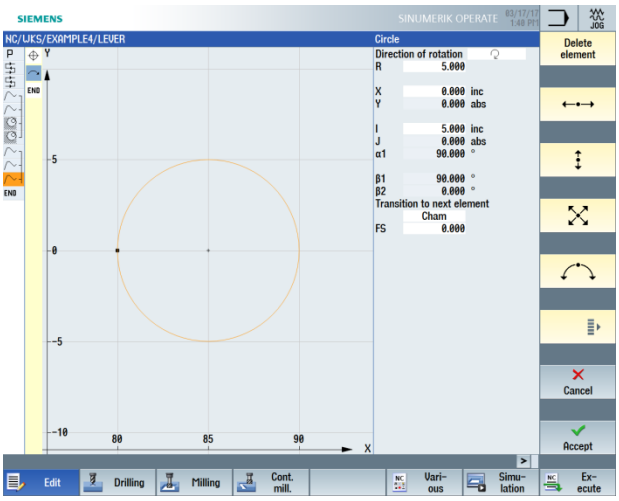


Figure 9-33 10 mm circular island contour

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

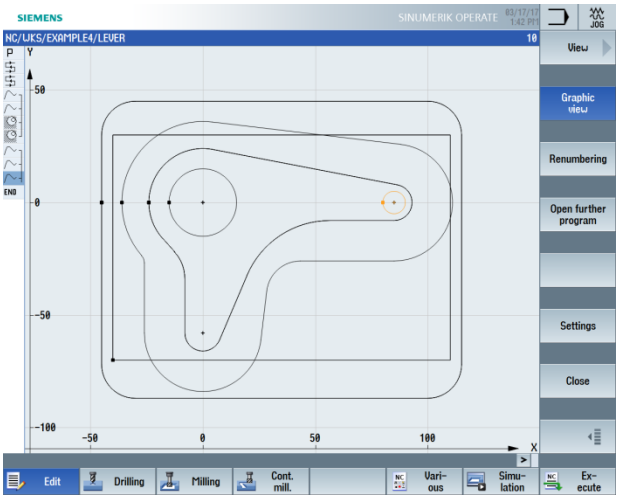


Figure 9-34 Broken-line graphic

10.8 Copying the 10 mm circular island

Operating sequences

Follow the steps below to copy the circular island created in the previous step:

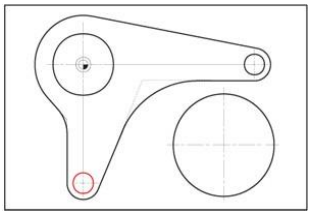


Figure 9-35 10 mm circular island

Navigate to the "LEVER_Circle_R5_A" contour and copy it.

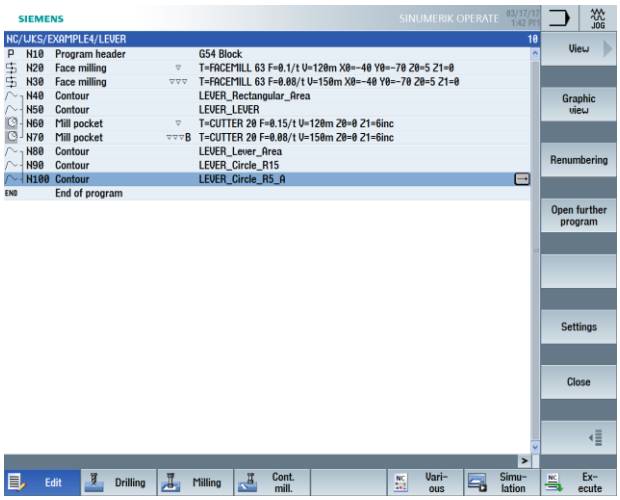


Figure 9-36 Copying the contour

Paste the copied contour and name it "LEVER_Circle_R5_B".

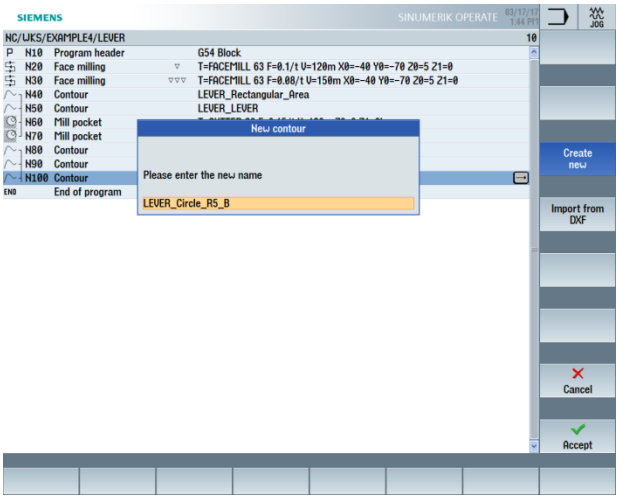


Figure 9-37 Entering the name for the copied contour

Accept your input.

Once applied, your machining plan should look like this:

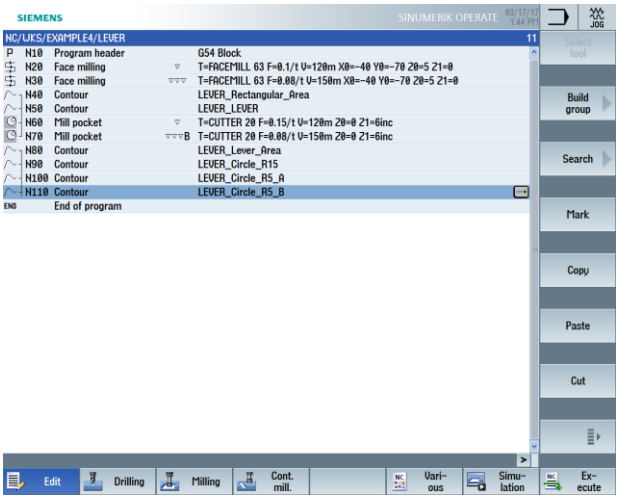


Figure 9-38 Pasted contour in the machining step editor

Now you only have to change the starting point, as you have specified the contour using incremental dimensions.

Open the contour. You can now use this key in the open contour to open the selected geometry element for changing.

Enter the following values for the starting point of the contour definition in the screen form:

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

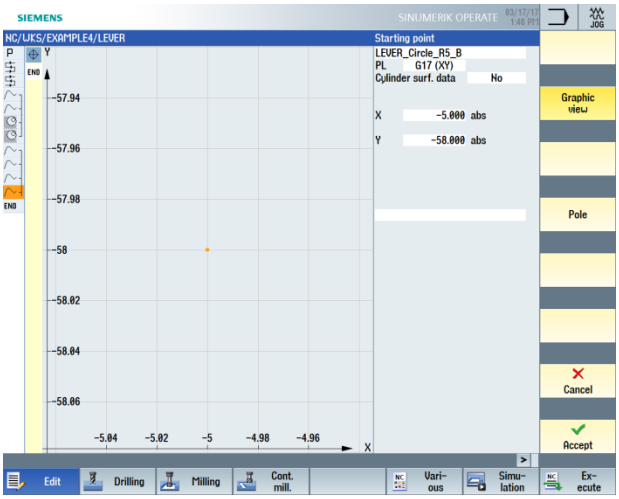


Figure 9-39 Changing the starting point

Accept the entered values.

10.9 Machining the circular island with the help of the editor

Operating sequences

Follow the steps below to machine the three circular islands: In doing so, you will learn additional functions of the machining step editor that will help you to reuse and manage parts of the machining plan (see Functions of the machining step editor).

The following contour serves as the traversing path boundary for machining the islands.

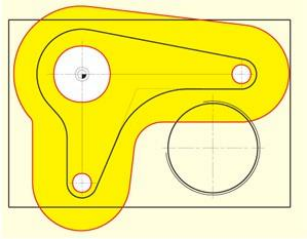


Figure 9-40 Traversing path boundary

Your machining plan will look as follows:

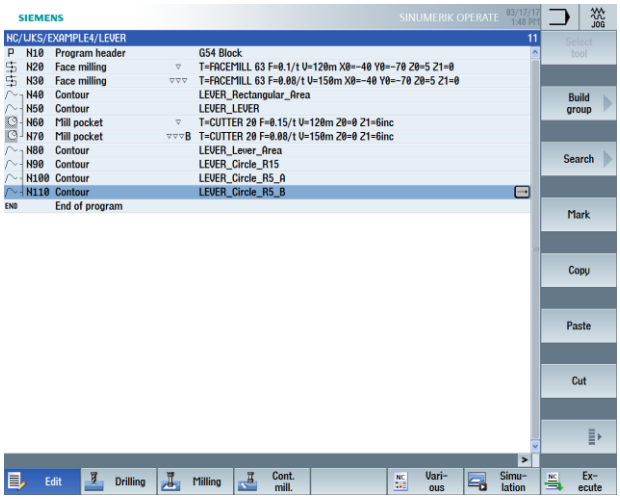


Figure 9-41 Machining plan

Mark the two machining steps for roughing and finishing the pocket.

Copy

Copy the marked machining steps.

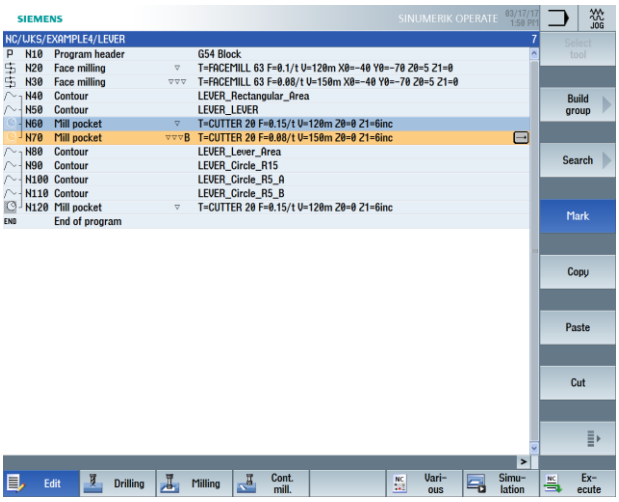


Figure 9-42 Marked machining steps

Paste

Paste the machining steps below the contours. This links the solid machining technologies with the contours.

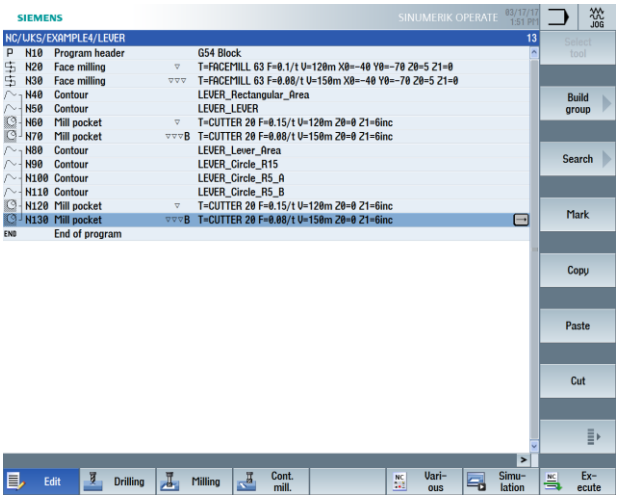


Figure 9-43 Pasted machining steps

The two solid machining technologies – roughing and finishing – still have to be adapted to the new machining depth:

Open the machining step for roughing.

Enter the following values for roughing in the screen form:

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

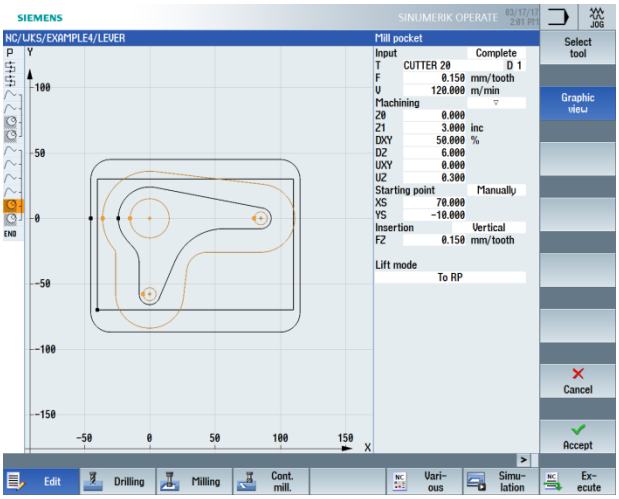


Figure 9-44 Adaptations for roughing

Accept

Accept the entered values.

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

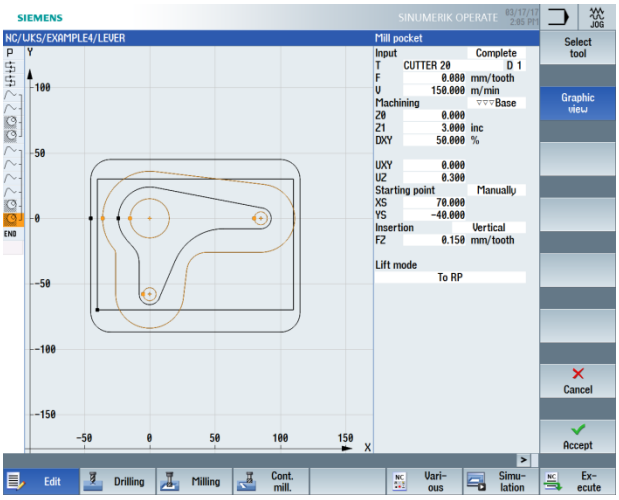


Figure 9-45 Adaptations for finishing

Accept

Accept the entered values.

Graphic view

This will show which geometries belong to the finishing technology (machining plan graphic).

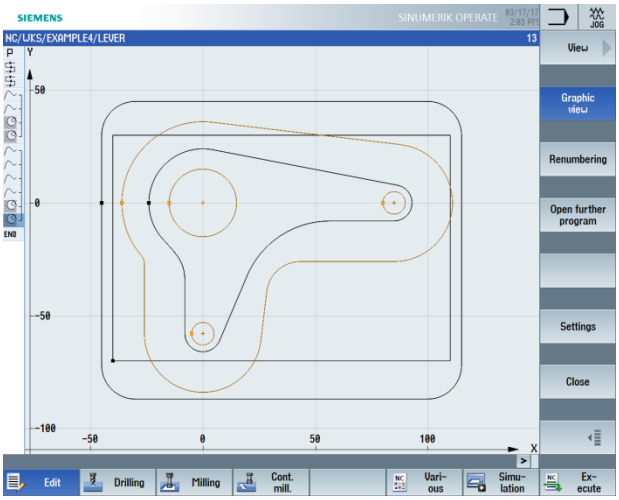


Figure 9-46 Broken-line graphic

Simu-lation

Check your intermediate result by way of simulation.

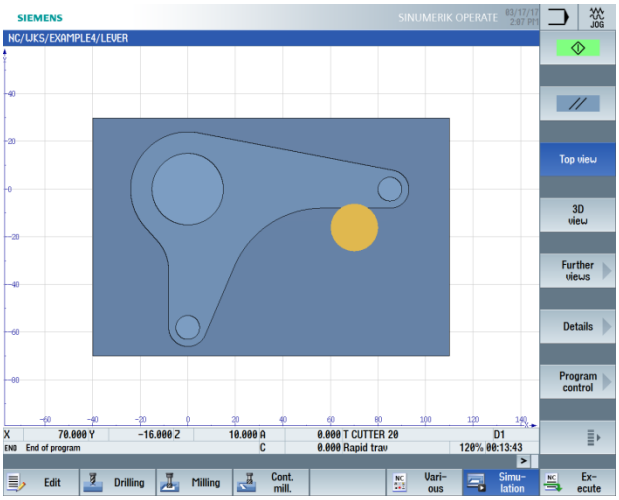
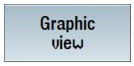


Figure 9-47 Simulation - Top view

Functions of the machining step editor

Below is an overview of the functions of the machining step editor:



Use the "Graphic view" softkey to change to the broken-line graphic.

You can use the "Search" softkey to search for texts in the program.

You can use the "Mark" softkey to select more than one machining step for further editing (e.g. copying or cutting).

You can use the "Copy" softkey to copy machining steps to the clipboard.

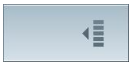
You can use the "Paste" softkey to paste machining steps from the clipboard to the machining plan. The machining step(s) are always inserted after the currently marked machining step.

You can use the "Cut" softkey to copy machining steps to the clipboard and simultaneously delete them from their point of origin. This softkey can also be used as a deletion key.

Use this softkey to go to the expanded menu.

Use the "Renumbering" softkey to renumber the machining steps.

Use the "Settings" softkey to open the Settings dialog. Here, you set whether numbering is to be automatic or whether the block end, among other things is to be displayed as a symbol.



Use this softkey to return to the previous menu.

10.10 Deep hole drilling

Operating sequences

Follow the steps below to predrill:

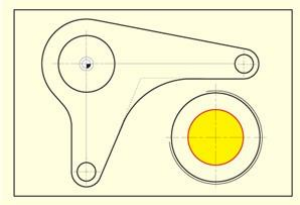
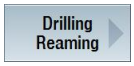


Figure 9-48 Deep hole drilling

Select the "Drill." softkey.



Select the "Drilling Reaming" softkey.

Open the tool list and select "PREDRILL30".

Apply the tool to the program.

Enter the following values for deep hole drilling in the screen form:

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	

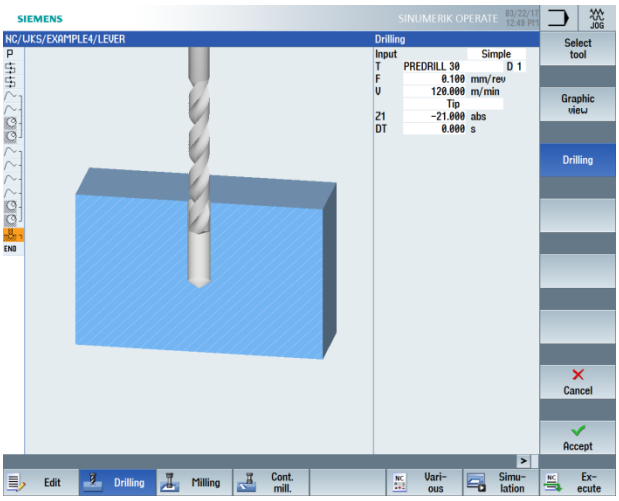
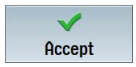


Figure 9-49 Entering the drill hole



Accept the entered values.

Enter the following values for the drilling position in the screen form:

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

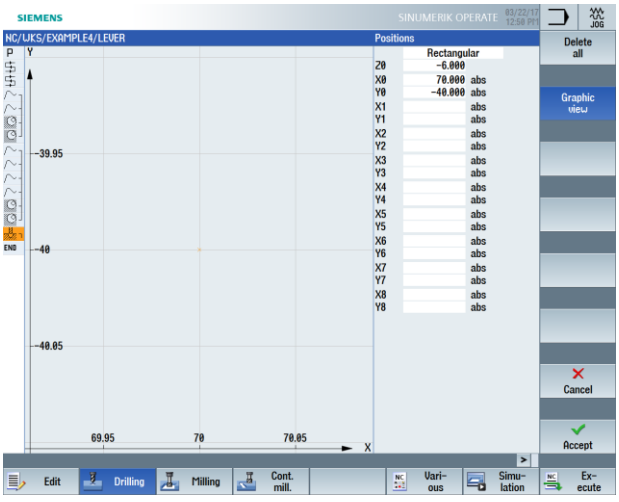


Figure 9-50 Entering the position

Accept the entered values.

10.11 Milling a helix

Operating sequences

Follow the steps below to remove the residual material of the circular ring remaining after the drilling by way of a helical motion:

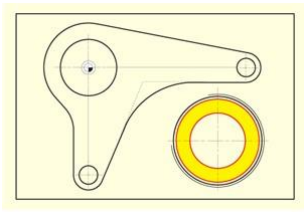


Figure 9-51 Milling a helix

Select the "Strght Circle" softkey.

Open the tool list and select "CUTTER20".

Apply the tool to the program.

Enter the following values in the screen form:

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

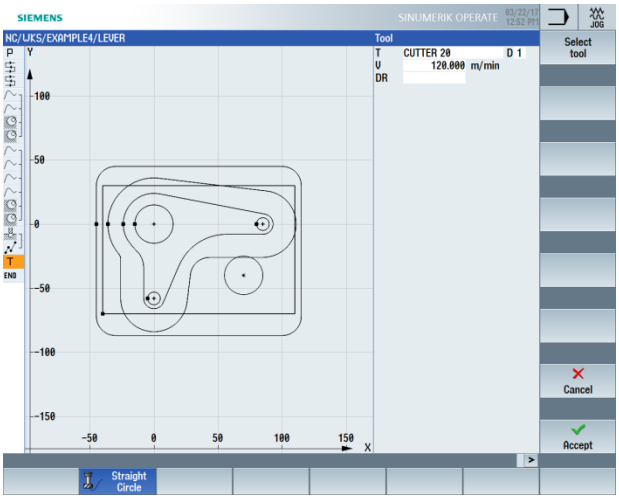


Figure 9-52 Milling a helix

Accept your input.

Select the "Straight" softkey.

Select the "Rapid traverse" softkey.

Enter the following values for the starting point of the contour definition in the screen form:

Note:

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

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

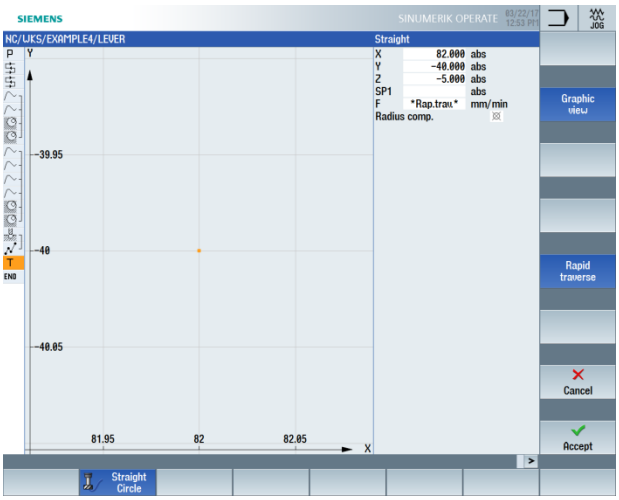


Figure 9-53 Positioning

Accept the entered values.

Select the "Helix" softkey. Enter the following values for the helix in the screen form:

Field	Value	Selection via toggle key	Notes
I	70	X	
J	-40	X	
L	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 prevent any residual material from remaining (even though the end depth is already reached after 5 revolutions).

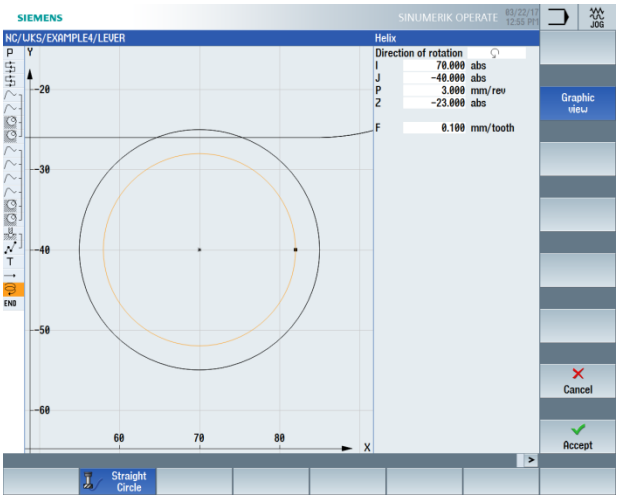
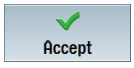


Figure 9-54 Entering the helix



Accept the entered values.

10.12 Boring

Operating sequences

Follow the steps below to machine the circular pocket to the required dimensions using a boring tool:

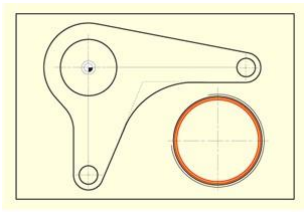


Figure 9-55 Boring a circular pocket

Select the "Drilling" softkey.

Select the "Boring" softkey.

Open the tool list and select "DRILL_tool".

Apply the tool to the program.

Enter the following values for the machining in the screen form:

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	Lift	X	The "Lift" option withdraws the tool from the contour before it retracts from the drill hole. This option may only be used with one-edged tools.
D	0.5		

Note:

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

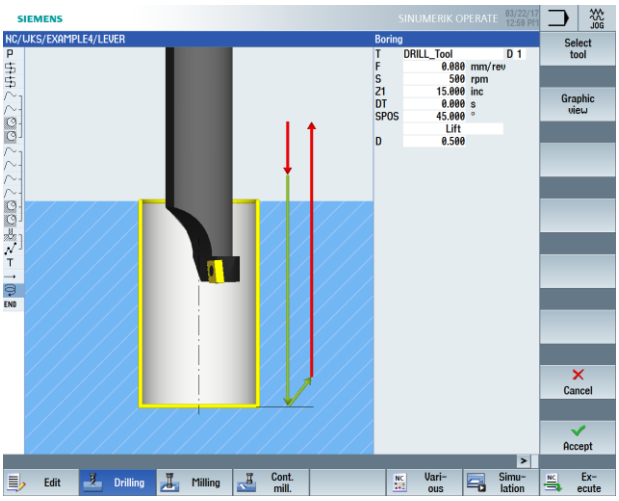


Figure 9-56 Boring

Accept the entered 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 can also use the Repeat position function here.

Enter the following values for the position in the screen form:

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

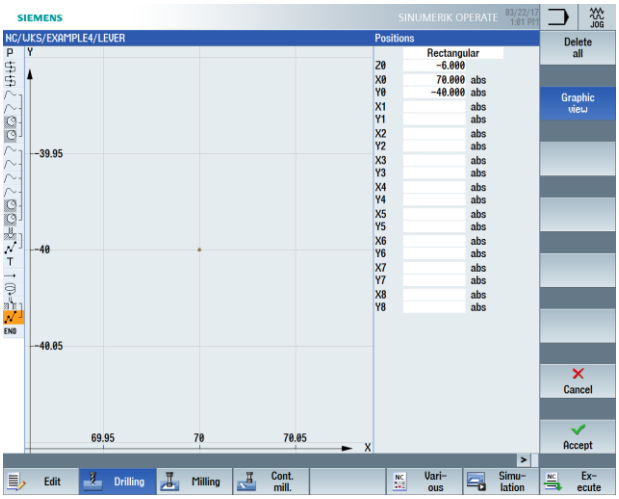


Figure 9-57 Positioning

Accept the entered values.

10.13 Thread milling

Operating sequences

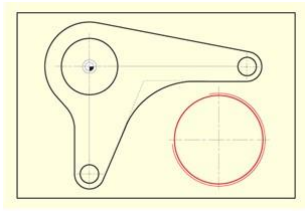


Figure 9-58 Thread milling

Select the "Milling" softkey.

Select the "Thread milling" softkey.

Open the tool list and select "THREADCUTTER".

Apply the tool to the program.

Mill the thread from top to bottom. The "THREADCUTTER" is used for this (F 0.08 mm/tooth, v 150 m/min and pitch 2 mm). A right thread is to be milled to Z-23 (absolute dimension). 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 screens are very useful for the input. Compare your entries with the figure below:

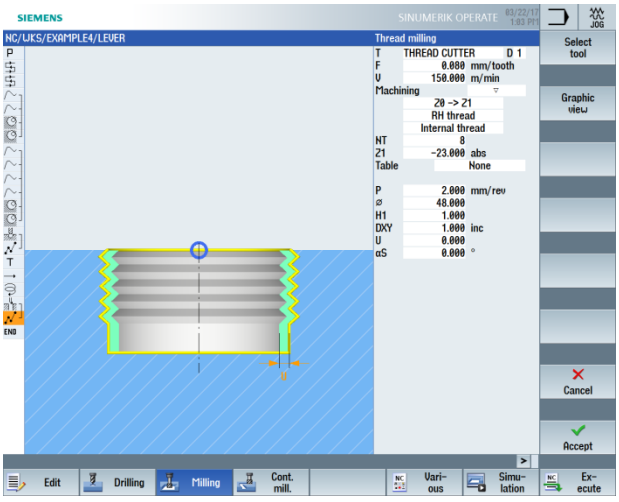
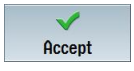


Figure 9-59 Thread milling



Accept the entered values.

Positions

Specify the position for the thread.

Enter the following values in the screen form:

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

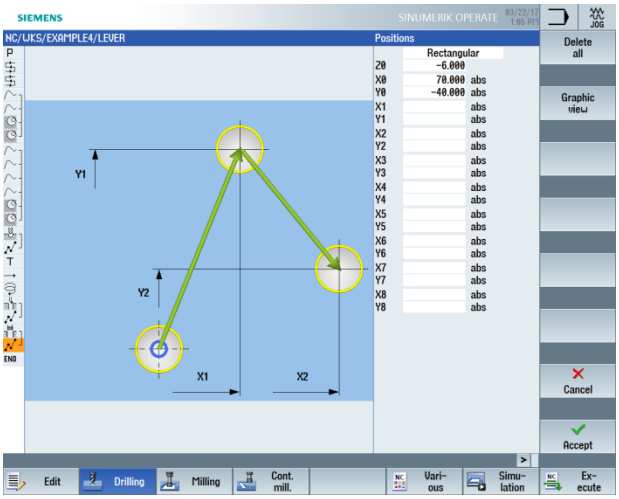


Figure 9-60 Entering the position

Accept

Accept the entered values.

10.14 Programming contours using polar coordinates

Programming with polar coordinates

Contour elements in workpiece drawings often reference a pole point. In this case, you do not know the Cartesian coordinates (X/Y), but instead 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" is no longer perpendicular to the zero point at X0, but is rotated clockwise by 10°.

In this example, you will learn how to program this position graphically without using the pocket calculator or any auxiliary constructions.

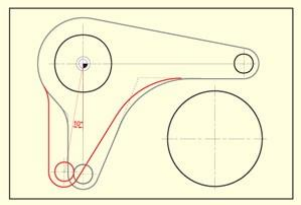


Figure 9-61 Programming the lever using polar coordinates

Operating sequences

First, move the cursor to the arc, whose center point has to be re-dimensioned (see figure below).

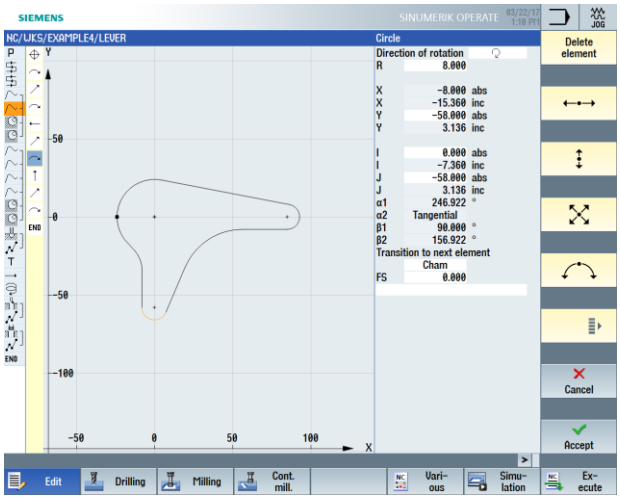


Figure 9-62 Cursor on arc

Expand the menu.

Pole

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

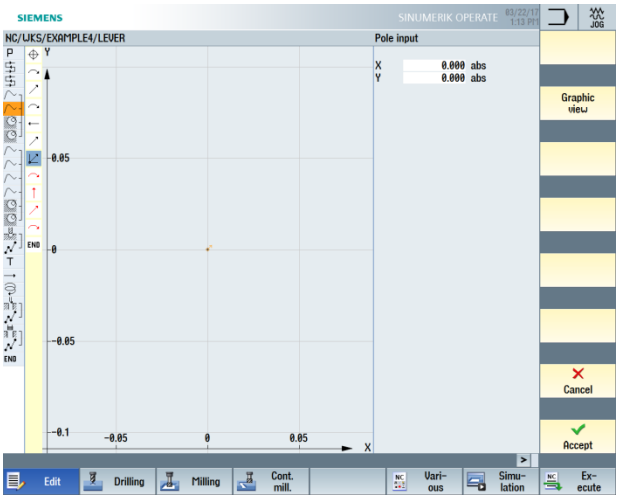


Figure 9-63 Entering the pole

Accept

Accept your input.

Next, adjust the values of the arc:

1. In the dialog window of the arc, delete the values that are no longer valid: Y-58, I0 and J-58.

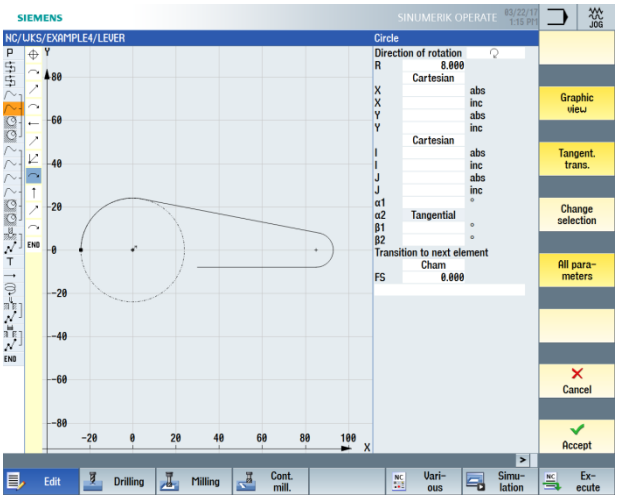


Figure 9-64 Deleting the values

2. Toggle the coordinates for input of the center point from "Cartesian" to "Polar". Enter the distance to the pole and the polar angle (see figure below).

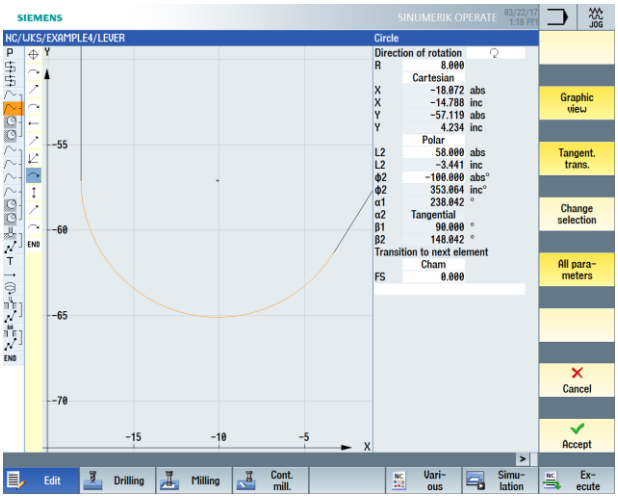


Figure 9-65 Entering the distance to the pole and polar angle

Accept your input.

Accept the change.

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

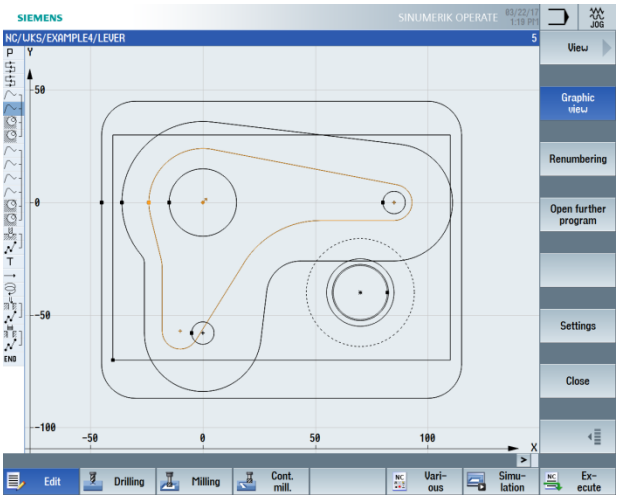


Figure 9-66 Broken-line graphic after shifting

Change these two contours on your own. In doing so, note the following information.

Notes:

For the auxiliary pocket, you may proceed a bit "rougher" and approximate the polar dimensions of the center point of arc R26 with Cartesian dimensioning (X-10/Y-57). Then, the contour can be closed 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.

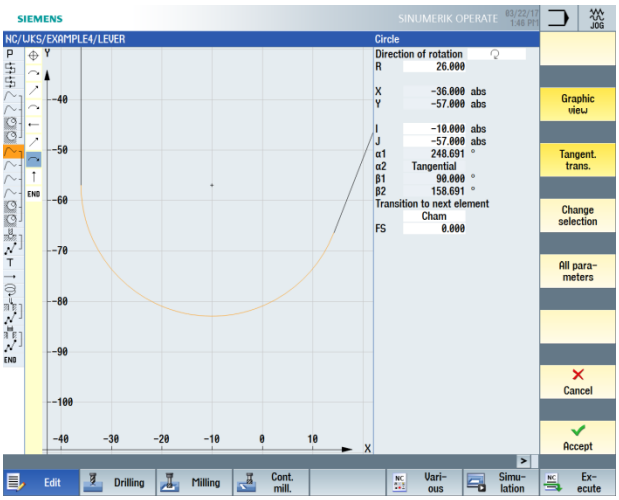


Figure 9-67 Adapting the border

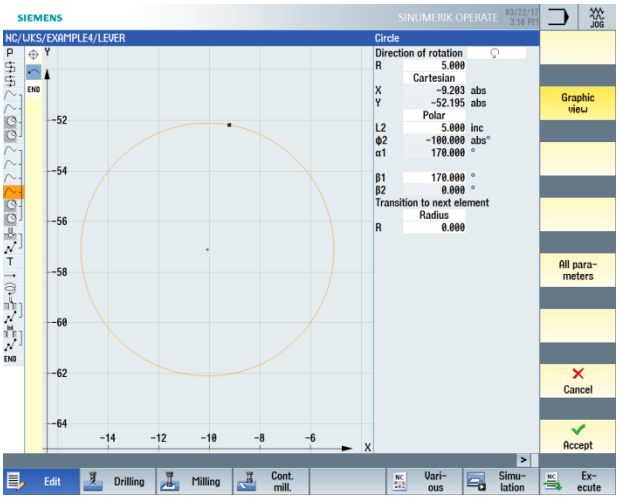


Figure 9-68 Adapting the circular island

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

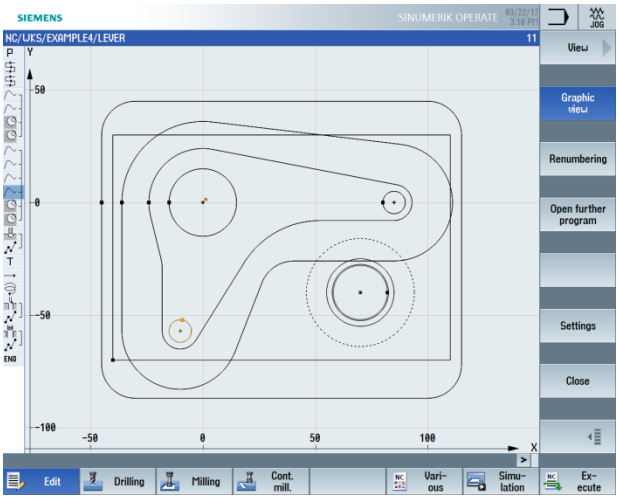


Figure 9-69 Broken-line graphic

11. Example 5: Flange

11.1 Overview

Learning objectives

In this section, you will learn how to

- Create a subprogram
- Mirror machining steps
- Chamfer any contours
- Create longitudinal and circular slots

Task

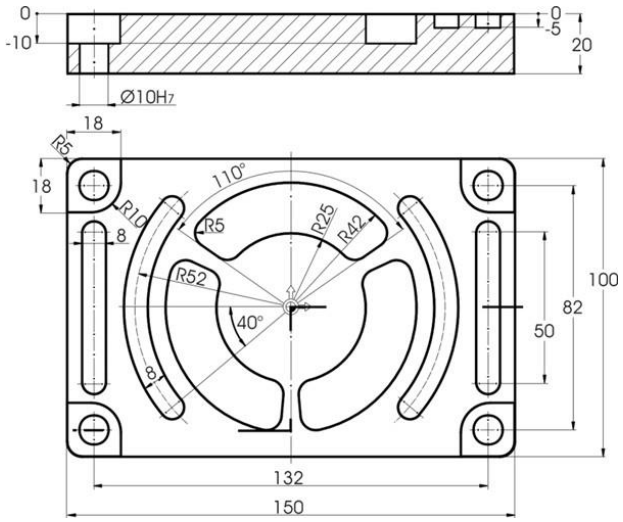
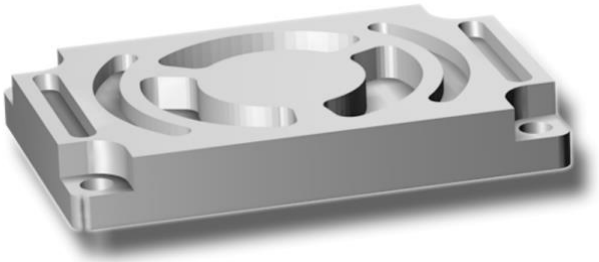


Figure 10-1 Workshop drawing – Example 5



Note:

All machining steps have been explained and nearly all softkeys / keys to be selected / pressed have been shown in the previous examples. In this 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.

11.2 Creating a subprogram

Operating sequences

The creation and functioning of subprograms is explained taking the example of the workpiece "CORNER_MACHINING".

Following the steps below you can machine the four corners with the help of a subprogram and the Mirror function.

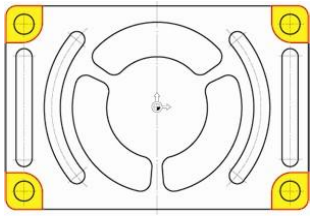


Figure 10-3 Contour of the four corners

Create a new sequential program with the name "CORNER_MACHINING". You will later link this program as a subprogram.

Figure 10-4 Creating a subprogram

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

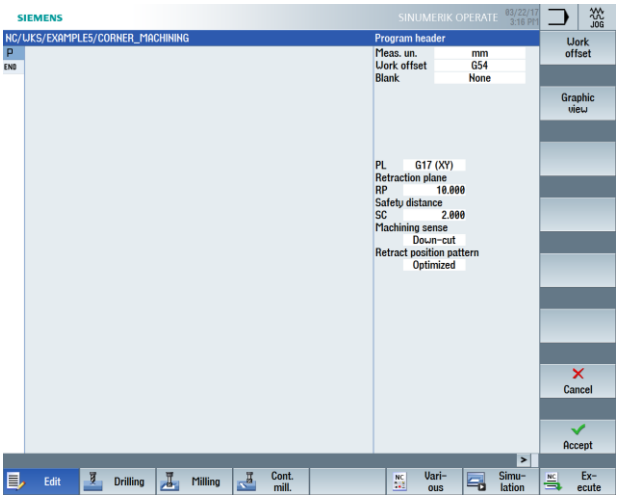


Figure 10-5 Entering the program header for the subprogram

Accept the entered values.

Select the "Cont. mill." 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 constructed.

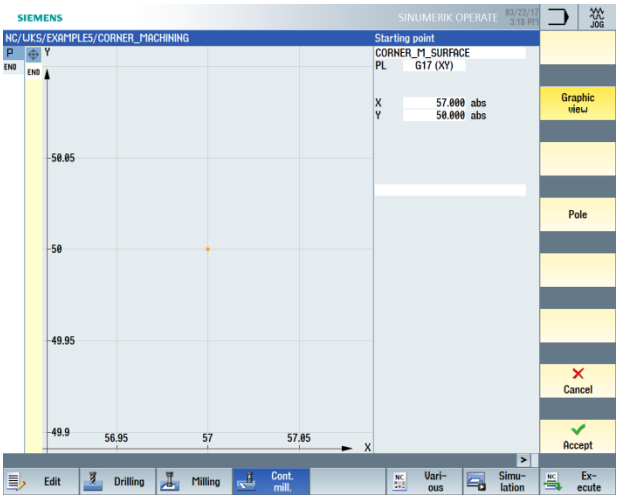


Figure 10-7 Entering the starting point

Accept the entered values.

Create the contour. After entering the two contour elements, your screen should look like this: Accept the contour to apply it to your machining plan.

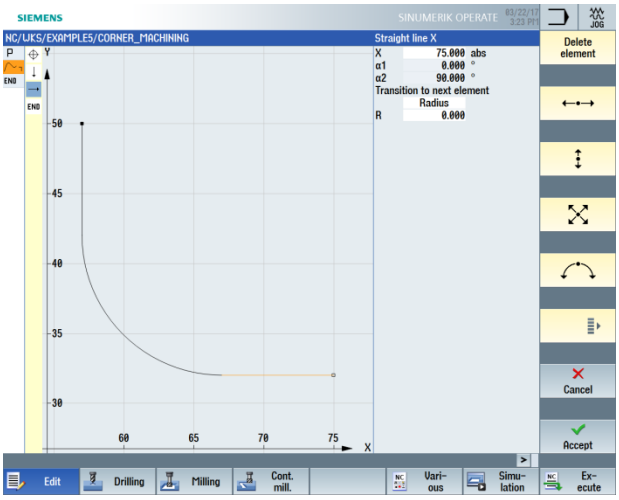


Figure 10-8 Subprogram for top right corner contour

Path milling

The contour is to be roughed with the 20 mm milling cutter (F = 0.15 mm/tooth and V = 120 m/min).

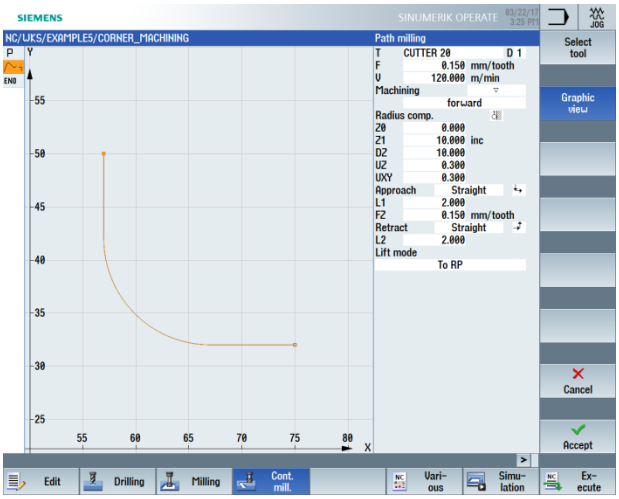


Figure 10-9 Roughing the contour

The approach and retraction distances are traversed in a straight line. The length values are the distances between the cutter edge and the workpiece.

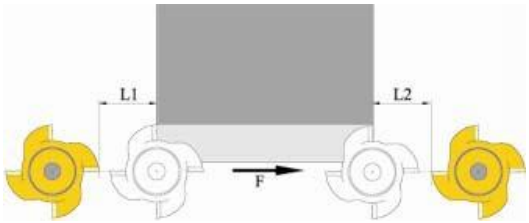


Figure 10-10 Approach and retraction distances in a straight line

Accept

Accept the entered values.

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

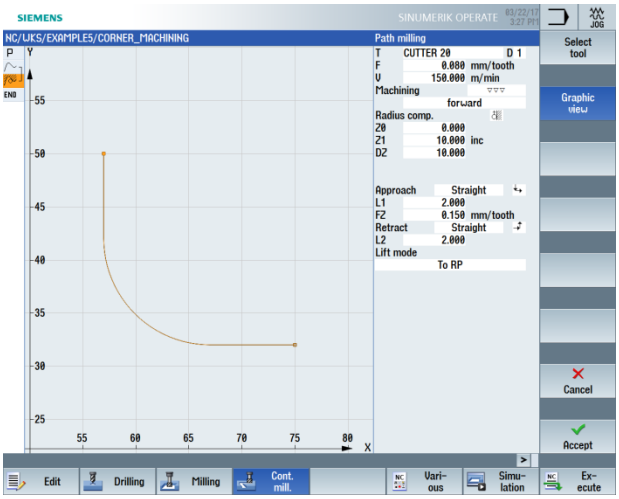
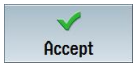


Figure 10-11 Finishing the contour



Accept the entered values.

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

Select the "Cont. mill." softkey.

Create a new contour with the name "CORNER_M_ARC".

Figure 10-12 Creating the contour

Specify the starting point.

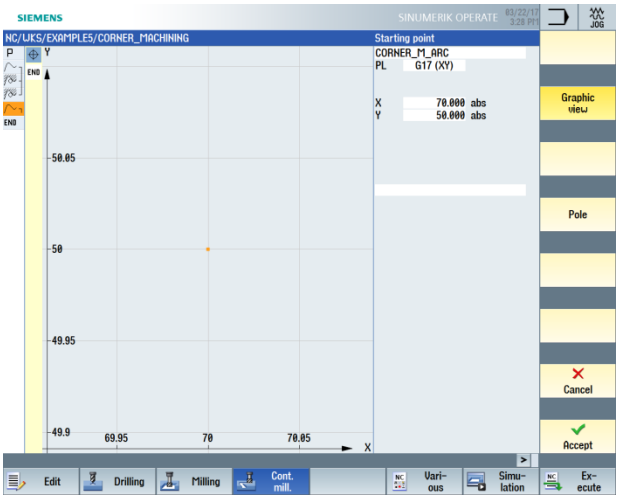


Figure 10-13 Entering the starting point

Accept the entered values.

Specify the contour and the associated machining steps:

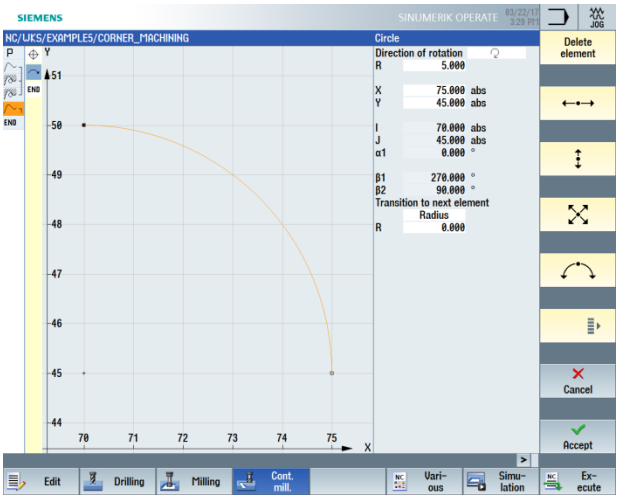


Figure 10-14 Entering the geometry

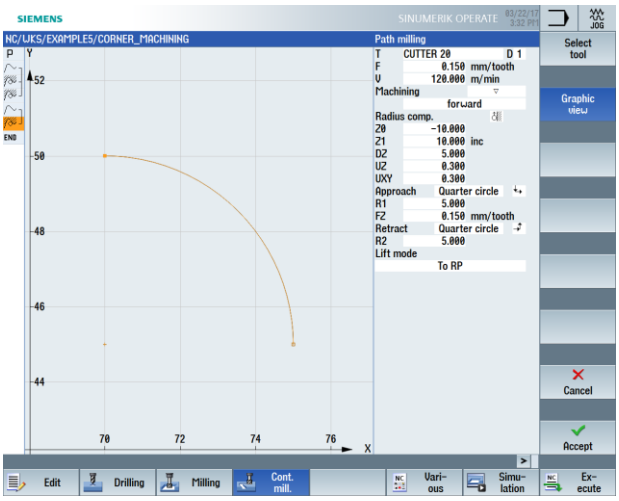


Figure 10-15 Roughing the contour

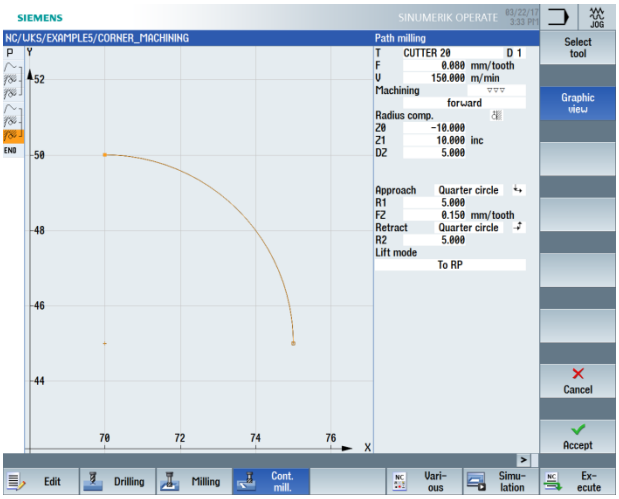


Figure 10-16 Finishing the contour

NC/UKS/EXAMPLE5/CORNER_MACHINING		
P	Program header	G54 Blank: None
	Contour	CORNER_M_SURFACE
	Path milling ▽	T=CUTTER 20 F=0.15/t V=120m Z0=0 Z1=10inc
	Path milling ▽▽▽	T=CUTTER 20 F=0.08/t V=150m Z0=0 Z1=10inc
	Contour	CORNER_M_ARC
	Path milling ▽	T=CUTTER 20 F=0.15/t V=120m Z0=-10 Z1=10inc
	Path milling ▽▽▽	T=CUTTER 20 F=0.08/t V=150m Z0=-10 Z1=10inc
END	End of program	

Figure 10-17 Complete subprogram in the machining step editor

11.3 Mirroring of machining steps

Task

Now that you have finished the subprogram, create the main program. By using the "Mirroring" function in the "Transformations" menu, you can use the subprogram for all four workpiece corners.

Mirroring can be performed in two different ways:

- new:

  Mirroring is performed from the location at which the 1st machining took place.

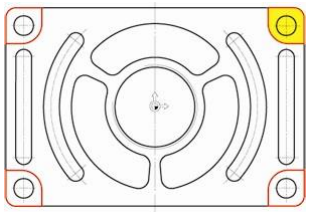
- additive:

  Mirroring is performed from the last machined location

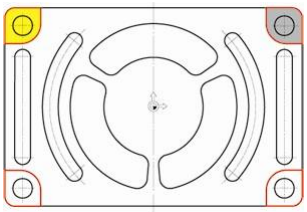
The sequence of machining is then shown schematically with setting "New":

1st machining (see subprogram)

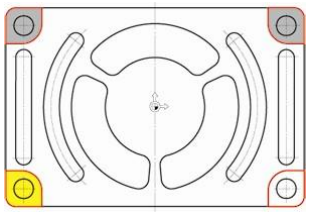
(the X values are mirrored here)



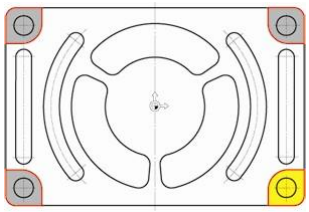
2nd machining: Mirroring of the X axis



3rd machining: Mirroring of the X and Y axes (the X and Y values are mirrored here)



4th machining: Mirroring of 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.

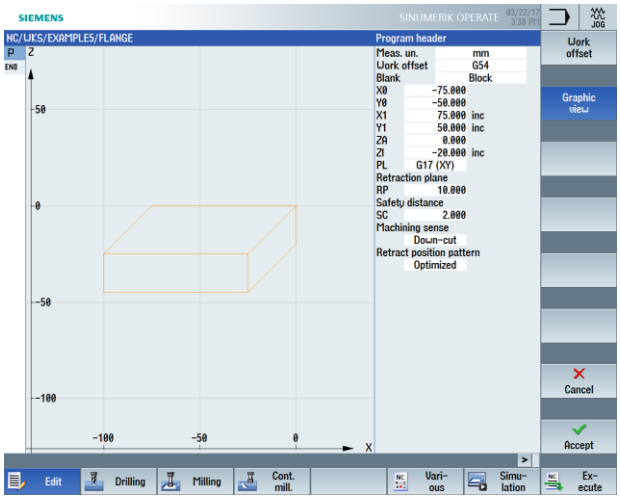
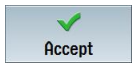
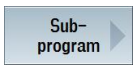


Figure 10-19 Entering the program header of the main program



Accept the entered values.

Select the "Various" softkey.



Insert the subprogram into the main program.

Note:

If you have created the subprogram in the same directory as the main program, the "Path/Workpiece" text box may remain empty.

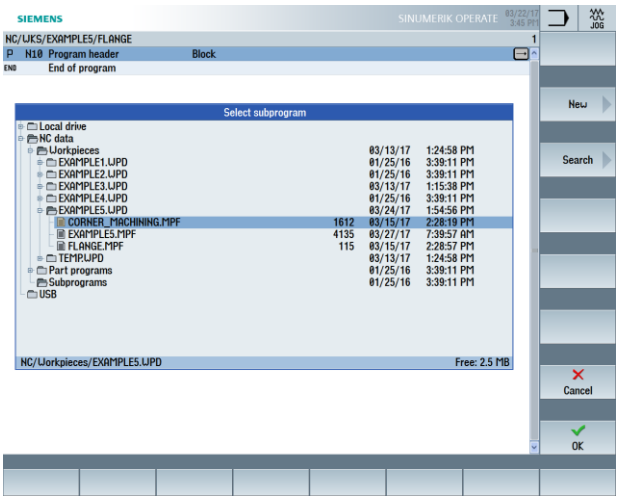
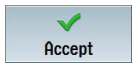


Figure 10-20 Inserting the subprogram



Accept your input. Once applied, your machining step program looks like this:

Figure 10-21 Subprogram inserted in the main program

The axes can be shifted, rotated, etc., by selecting the "Transformations" softkey.

Preparation for the 2nd machining: Mirroring of the X values.

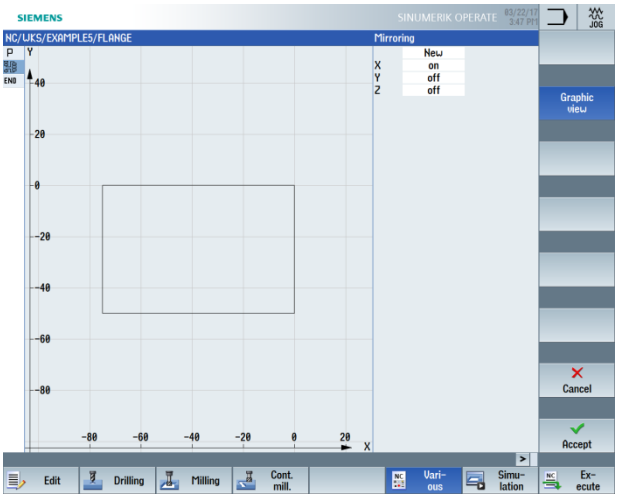


Figure 10-22 Mirroring

Accept your input.

To mirror the remaining machining operations, proceed as follows:

Copy the subprogram after the "Mirroring" machining step.
The 2nd machining occurs.

You must then repeat the "Mirroring" and "Subprogram call" operations for the two other corners.

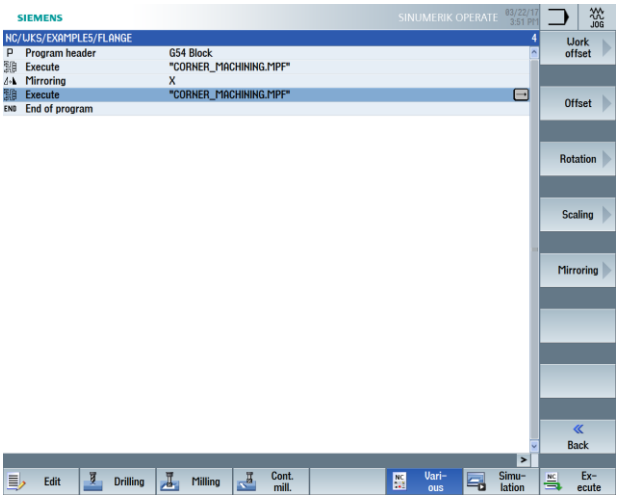


Figure 10-23 Copying the subprogram

The help screen illustrating this procedure will help you. After you have entered all four machining operations, you must disable the mirroring in all three axes.

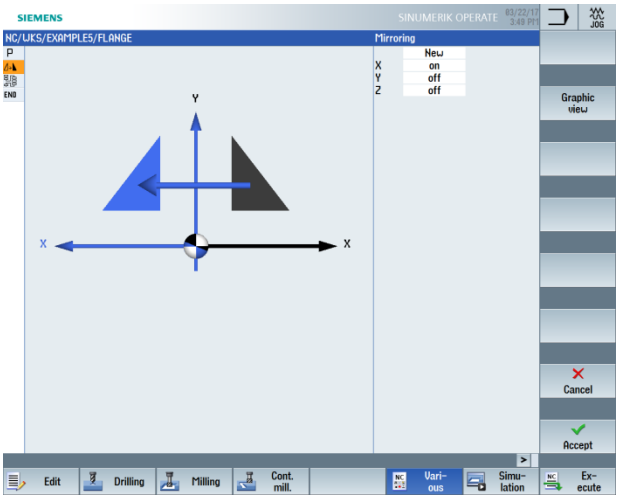


Figure 10-24 Mirroring help screen

Your machining step program will look like this:

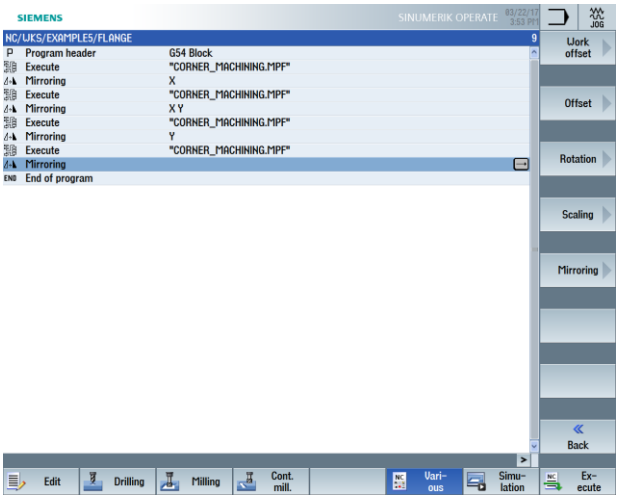


Figure 10-25 Complete mirroring in the machining step editor

Check your work by now using the simulation.

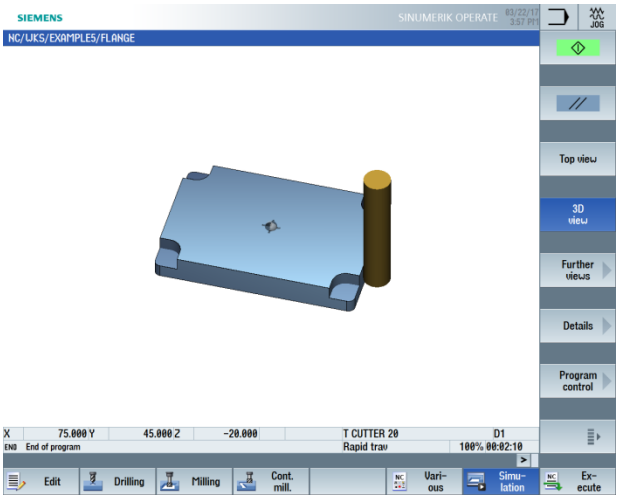


Figure 10-26 Simulation in 3D view

11.4 Holes

Operating sequences

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

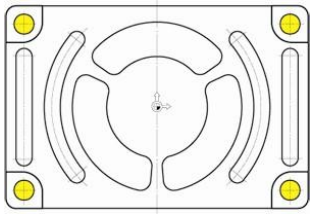


Figure 10-27 Holes

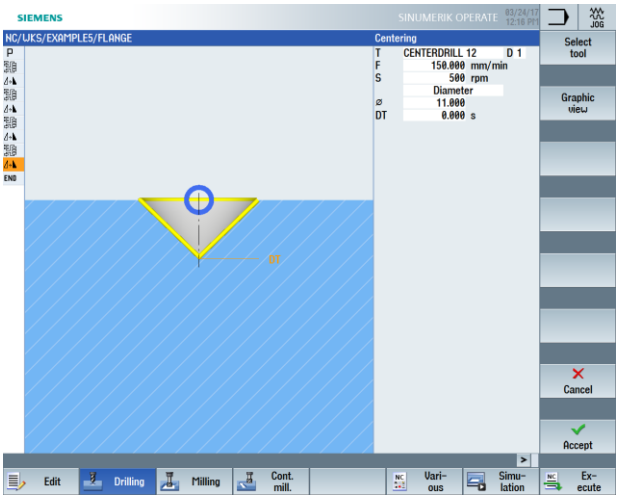


Figure 10-28 Centering

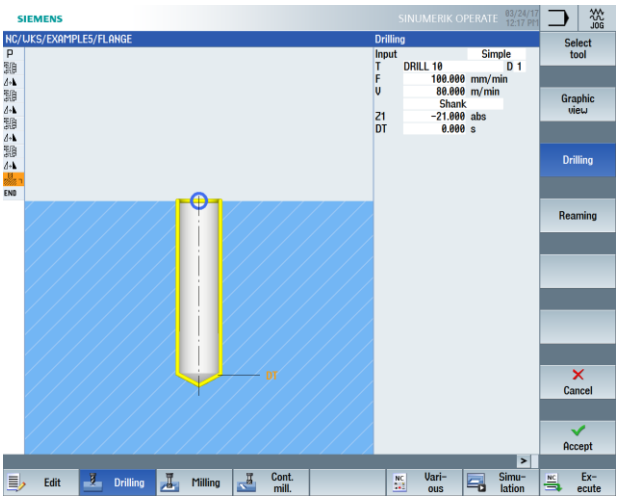


Figure 10-29 Drilling

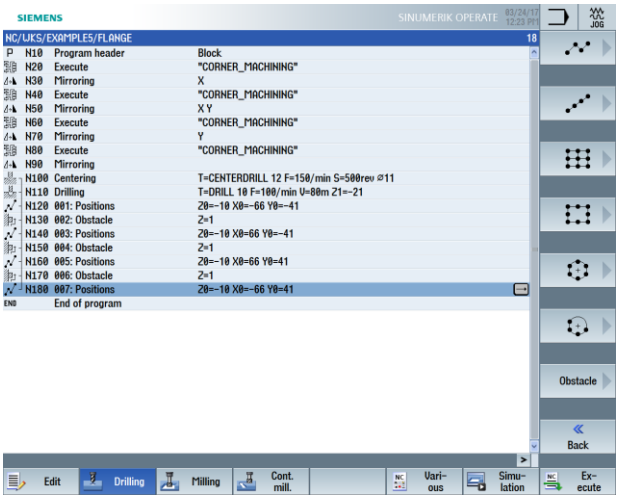


Figure 10-30 Entering the positions of the obstacles

11.5 Rotation of pockets

Operating sequences

Follow the steps below to program the contour and the machining for the pocket highlighted in yellow.

By rotating the coordinate system, the other two pockets are then created.

Select the "Cont. mill." softkey.

Create a new contour with the name "FLANGE_NODULE".

Figure 10-31 Creating a new contour

Specify the starting point.

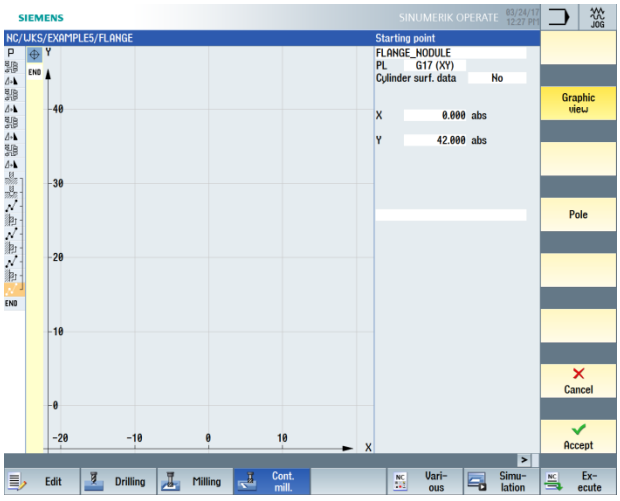


Figure 10-32 Entering the starting point

Accept the entered values.

Select the "Arc" softkey.

All para-
meters

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. Construct in the counterclockwise direction to ensure that the pocket can also be finished by synchronized milling.

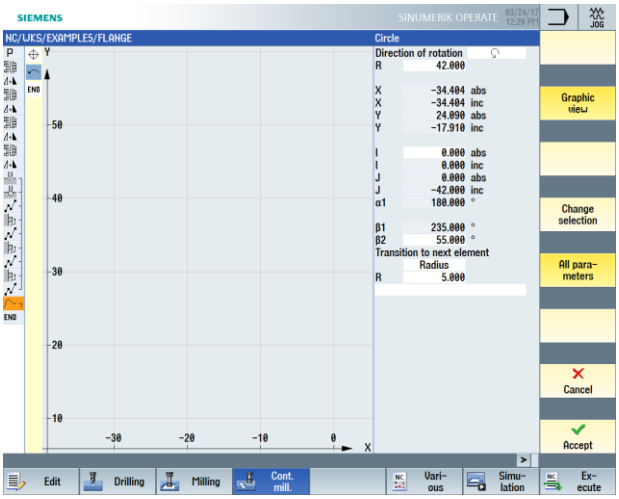
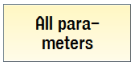


Figure 10-33 Entering the arc

Accept

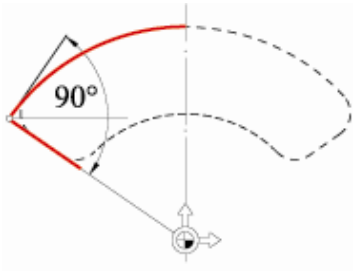
Accept the entered values.

Select the "Diagonal" softkey.



Select the "All parameters" softkey.

Create the diagonal straight-line segment.



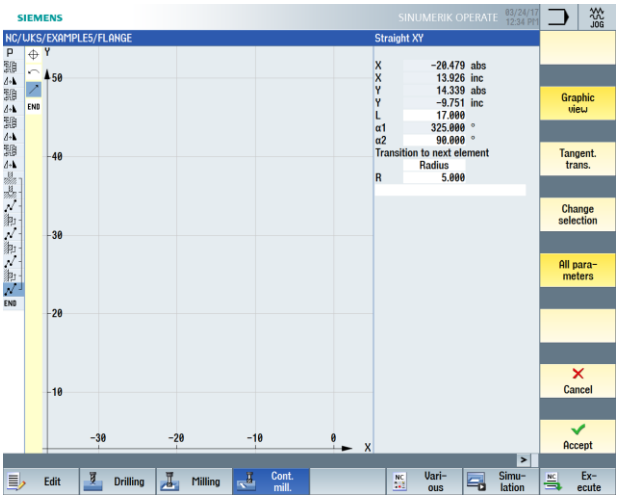
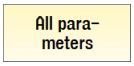


Figure 10-34 Entering the diagonal

Accept the entered values.

Select the "Arc" softkey.



Select the "All parameters" softkey.

Create the second arc.

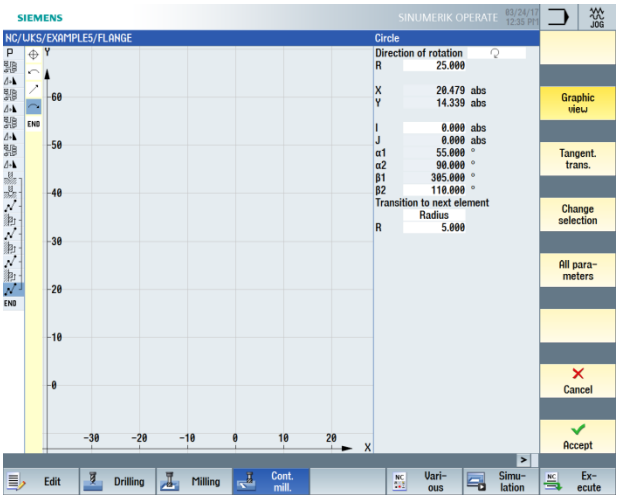
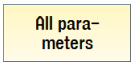


Figure 10-35 Entering the arc

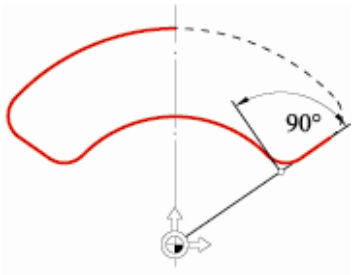
Accept the entered values.

Select the "Diagonal" softkey.



Select the "All parameters" softkey.

Create the second diagonal straight-line segment.



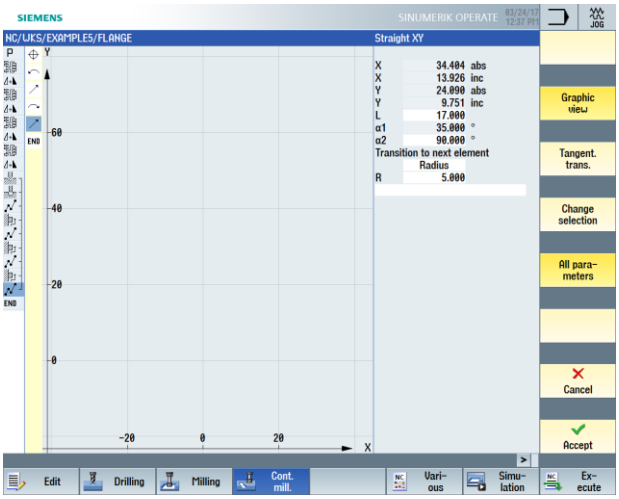


Figure 10-36 Entering the diagonal

Accept the entered values.

Select the "Arc" softkey.

Create the final arc.

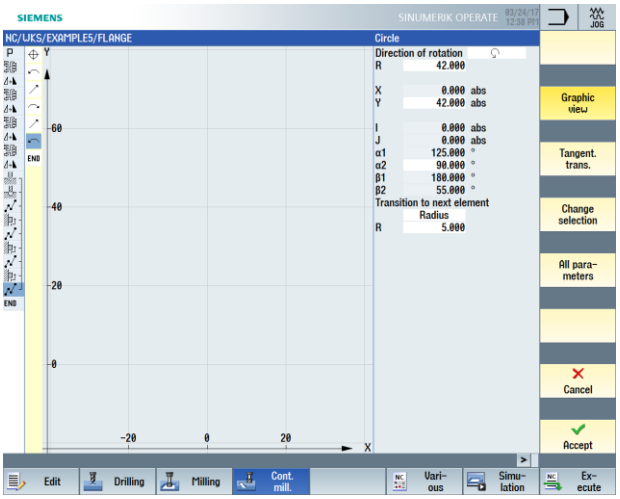


Figure 10-37 Entering the final arc

Accept the entered values.

Accept the contour pocket to apply it to your machining plan.

Create the following machining steps on your own:

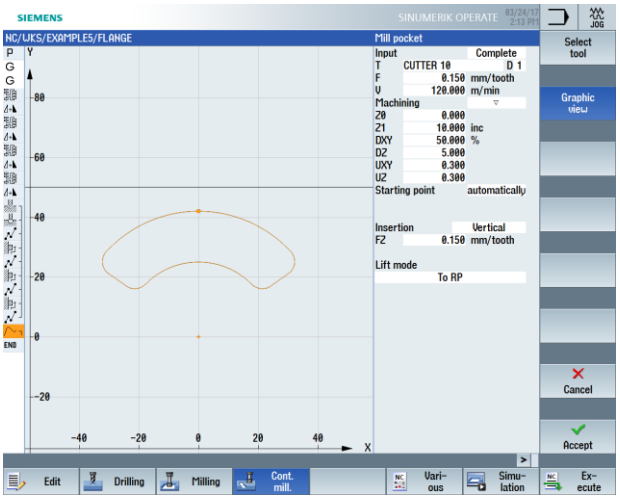


Figure 10-38 Roughing a pocket

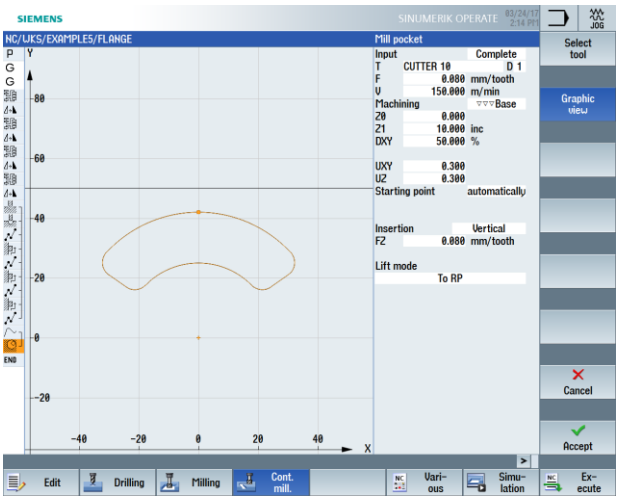


Figure 10-39 Finishing the pocket base

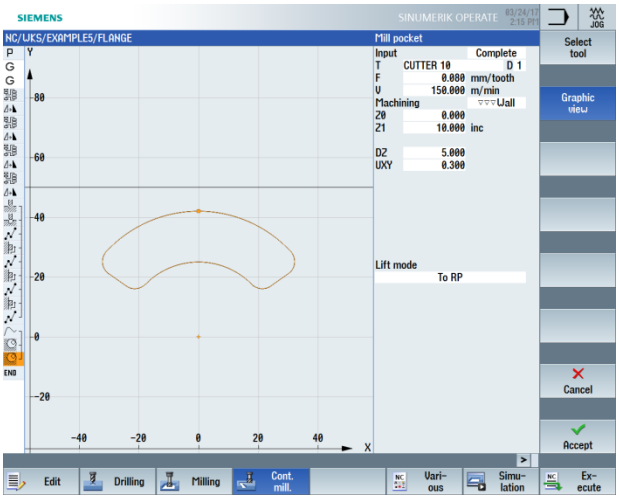


Figure 10-40 Finishing the pocket wall

Follow the steps below to copy the created machining step sequence for machining the three pockets:

Mark

Mark the complete machining step sequence describing the pocket machining in the machining step editor.

Copy

Copy the machining step sequence to the clipboard.

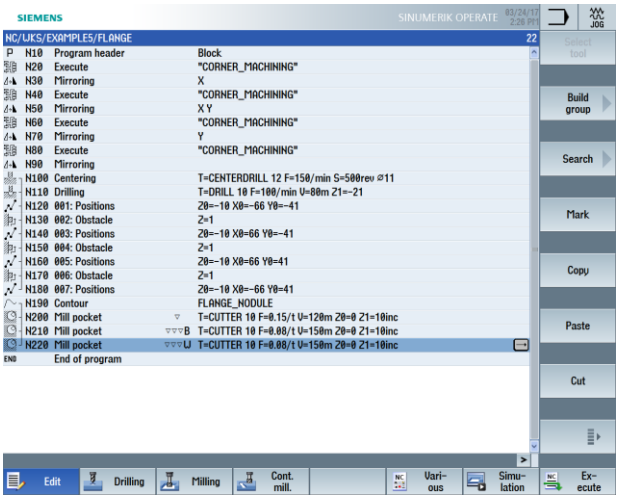


Figure 10-41 Copying the machining steps

Vari- ous

Select the "Various" softkey.

Transfor- mations

Select the "Transformations" softkey.

Rotation

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

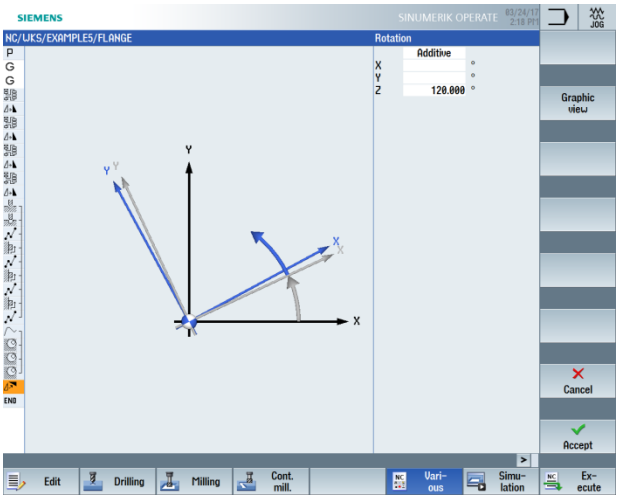


Figure 10-42 Rotation around the Z axis

Accept

Accept your input.

Paste

Paste the copied machining steps.

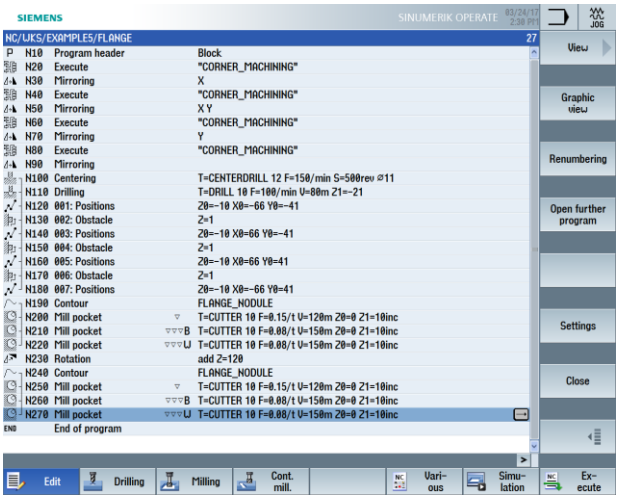


Figure 10-43 Pasting the copied machining steps

Transfor-mations

Select the "Transformations" softkey.

Rotation

Enter another rotation by 120°.

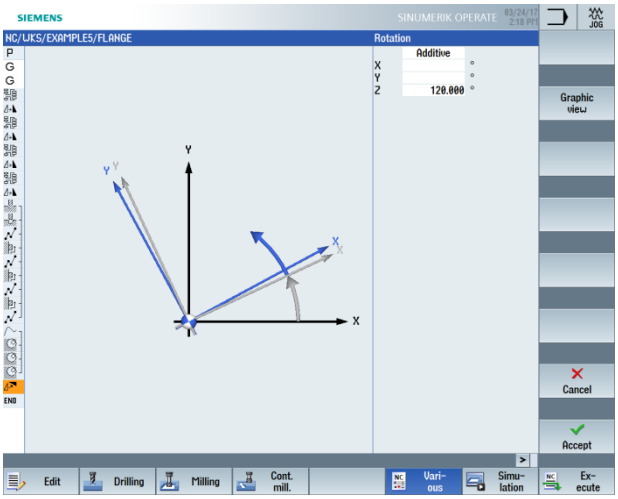


Figure 10-44 Rotation around the Z axis

Accept

Accept your input.

Paste

Paste the copied machining steps.

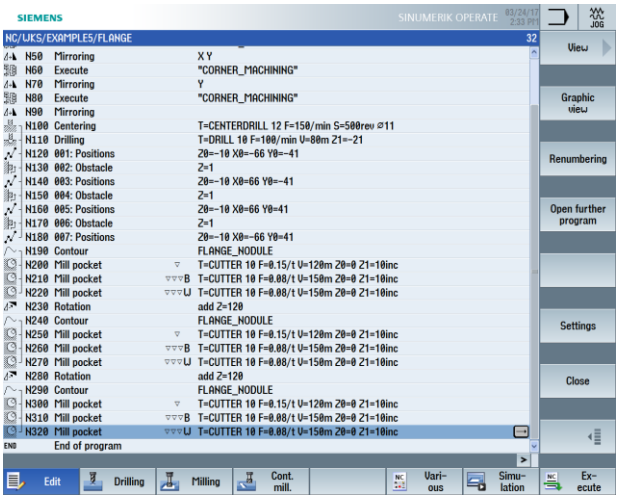


Figure 10-45 Pasting the copied machining steps

Rotation

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

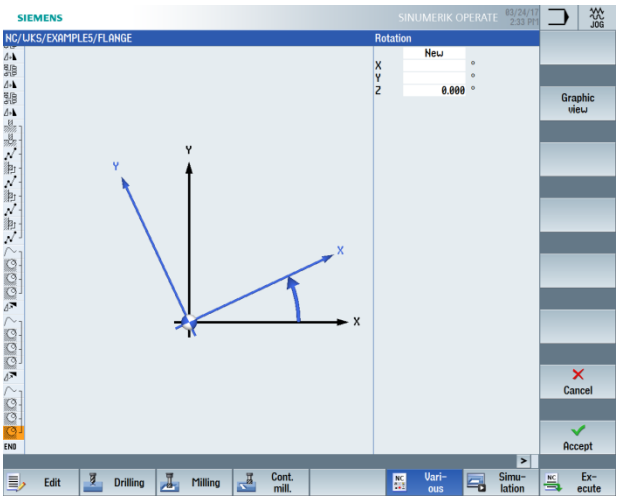


Figure 10-46 Undoing the rotation

Accept

Accept your input.

11.6 Chamfering contours

Operating sequences

Chamfer the circular pocket last machined without help.

For chamfering, you need a tool type which allows a tip angle to be entered, in the example "CENTERDRILL12".

Loc.	Type	Tool name	ST	D	Length	⌀	Tip angle		1	2
1		CUTTER 4	1	1	65.000	4.000		3	✓	
2		CUTTER 6	1	1	120.000	6.000		3	✓	
3		CENTERDRILL 12	1	1	120.000	12.000	90.0		✓	

Figure 10-47 Centering drill

Select "Chamfer" for the machining. The machining of the chamfer is programmed using the chamfer width (FS) and the insertion depth of the tool tip (ZFS).

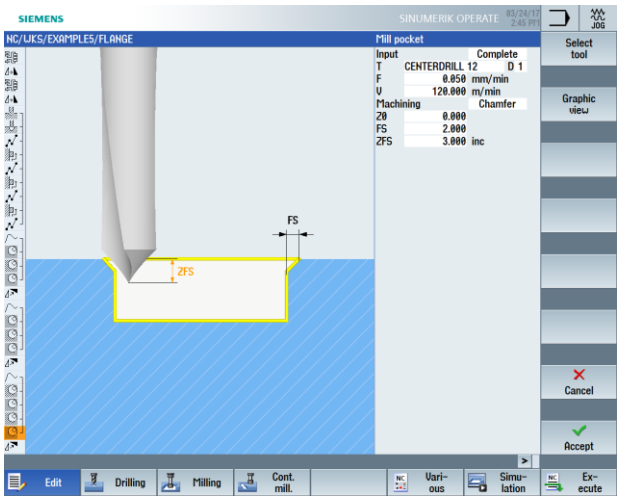


Figure 10-48 Chamfering

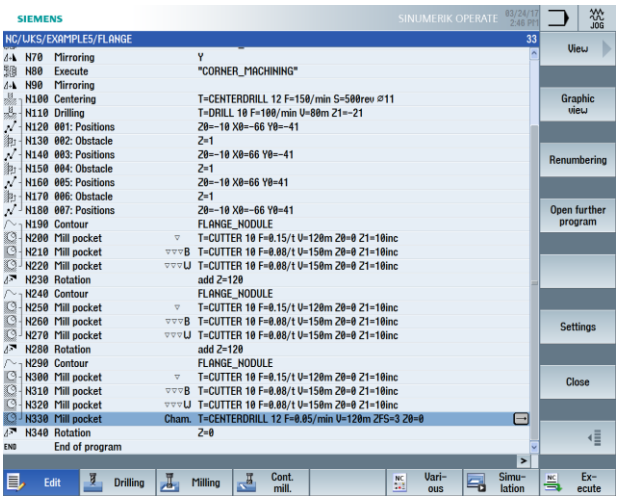


Figure 10-49 Chamfering machining step in the machining step editor

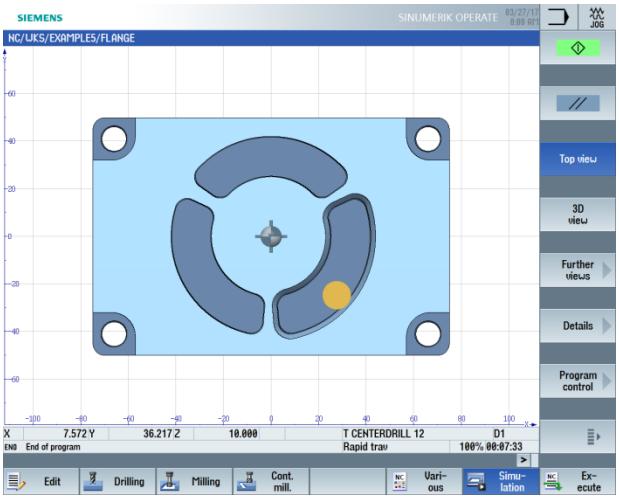


Figure 10-50 Top view of chamfered contour

11.7 Longitudinal and circumferential slots

Operating sequences

Finally, program the slots. They will be positioned correctly using Position pattern and Positioning to full circle.

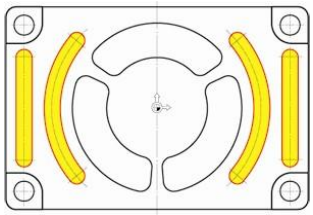


Figure 10-51 Longitudinal and circular slots

Select the "Milling" softkey.

Select the "Slot" softkey.

For roughing the longitudinal slots, use the "CUTTER6" tool (F = 0.08 mm/tooth and v = 120 m/min).

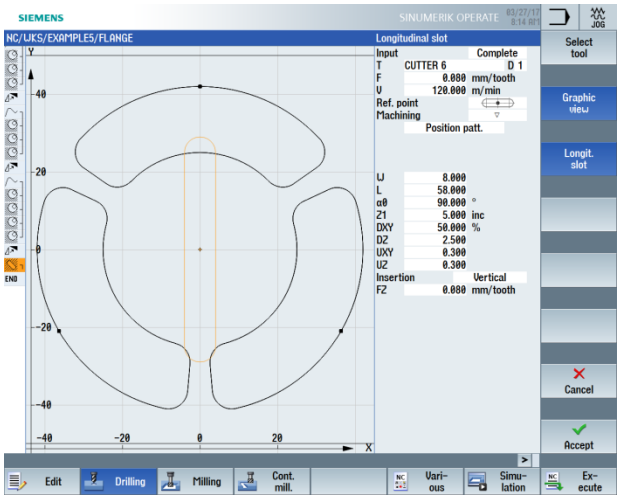


Figure 10-52 Roughing a longitudinal slot

Accept the entered values.

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

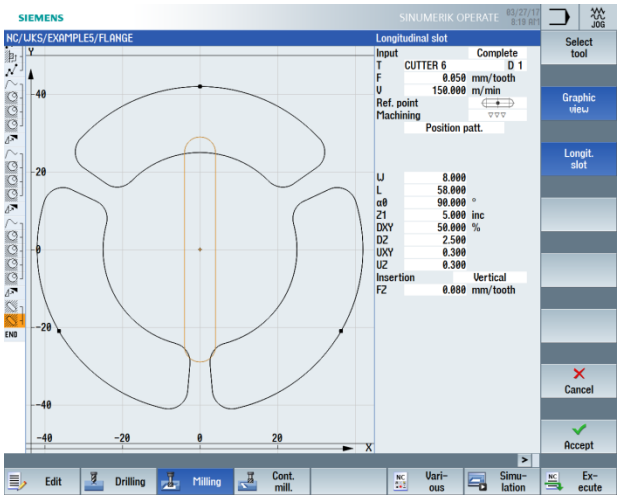


Figure 10-53 Finishing a longitudinal slot

Accept the entered values.

Select the "Drilling" softkey.

Specify the positions of the longitudinal slots below.
The reference point lies in the center of the slot.

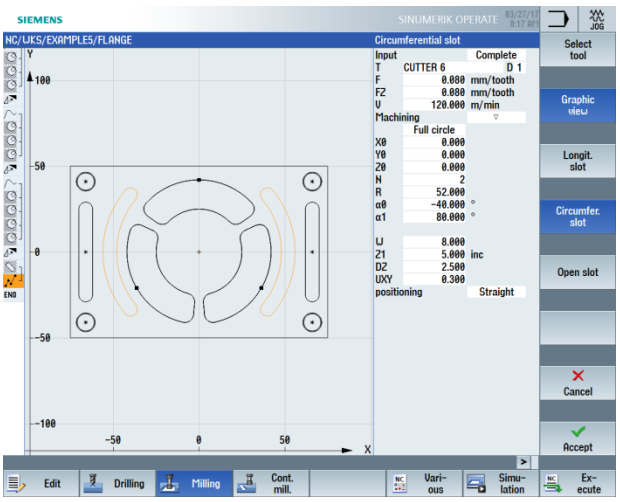


Figure 10-54 Entering the positions for the longitudinal slot

Accept the entered values.

Select the "Milling" softkey.

Select the "Slot" softkey.

Rough the circumferential slots with the "CUTTER6" tool
(F = 0.08 mm/tooth and FZ= 0.08 mm/tooth and V = 120 m/min).

The Full circle option positions the circumferential slots automatically with equal spacing. The reference point in X/Y/Z refers to the center point of the circumferential slots.

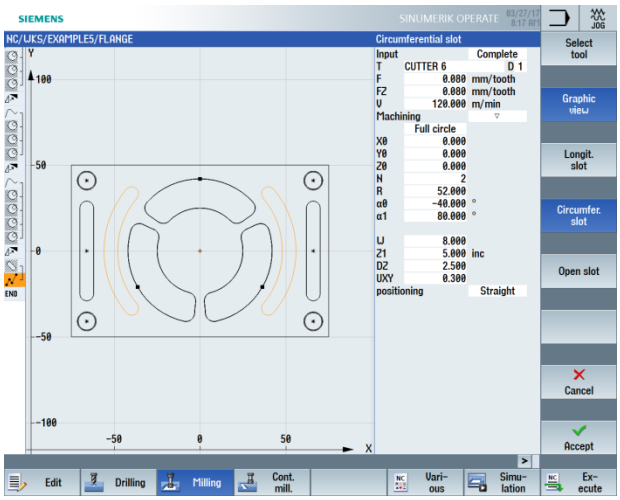


Figure 10-55 Roughing a circumferential slot

Accept the entered values.

Select the "Slot" softkey.

Circumfer. slot

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

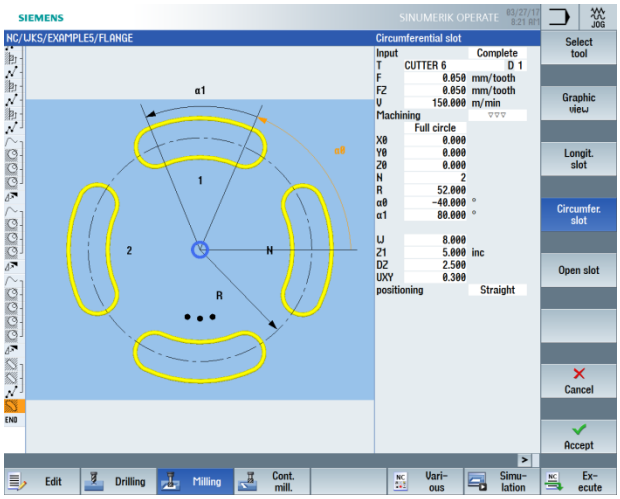


Figure 10-56 Finishing the circumferential slot

Accept

Accept the entered values.

Machining plan

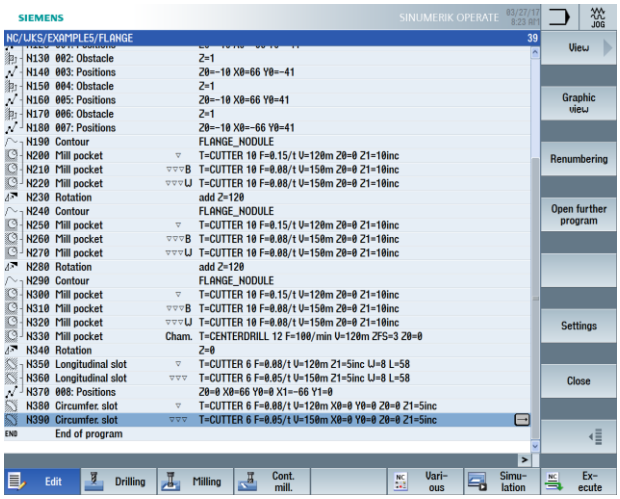


Figure 10-57 Extract from machining plan

Broken-line graphic

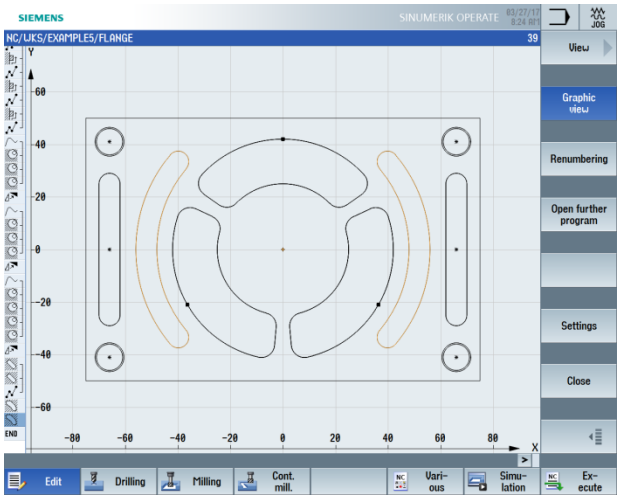


Figure 10-58 Broken-line graphic

Simulation in 3D view

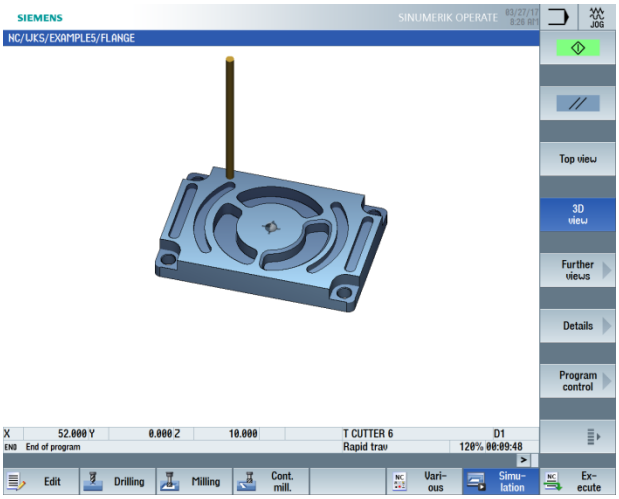


Figure 10-59 3D view

12. Machining the workpieces

Now that you have worked through the examples and acquired a well-founded knowledge of the machining plan creation process in ShopMill, the next step is to machine the workpieces.

The following steps are required for the machining:

Reference point approach

After turning on the controller and before executing the machining plans or traversing, you must approach the reference point of the machine manually. This is how ShopMill finds the count start in the position measuring system of the machine.

Since the reference point approach differs depending on machine type and manufacturer, only basic information can be given here:

1. If necessary, move the tool to a free spot in the work space from which it can traverse in all directions without collision. In doing so, ensure that the tool is then not already beyond the reference point of the corresponding axis (since the reference point approach is performed in only one direction for each axis, this point cannot be reached otherwise).
2. Perform the reference point approach exactly according to the specifications of the machine manufacturer.

Clamping the workpiece

For dimensionally-correct machining and, naturally, also for your own safety, secure chucking that is appropriate for the workpiece is required. Machine jaw vices or clamps are normally used for this.

Setting the workpiece zero

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

In the plane, the workpiece zero is set in most cases using one of the following:

- Contacting with a 3D probe
- Contacting with an edge probe

In the tool axis, the workpiece zero is set in most cases using one of the following:

- Contacting with the 3D probe
- Scratching with a tool

Note:

Observe the manufacturer's specifications for use of measuring tools and measuring cycles.

Executing the machining plan

The machine is now prepared, the workpiece is set up and the tools are measured. Now you can start the machining.

First, select the program you want to use for the machining in the Program Manager, e.g. "INJECTION_FORM".

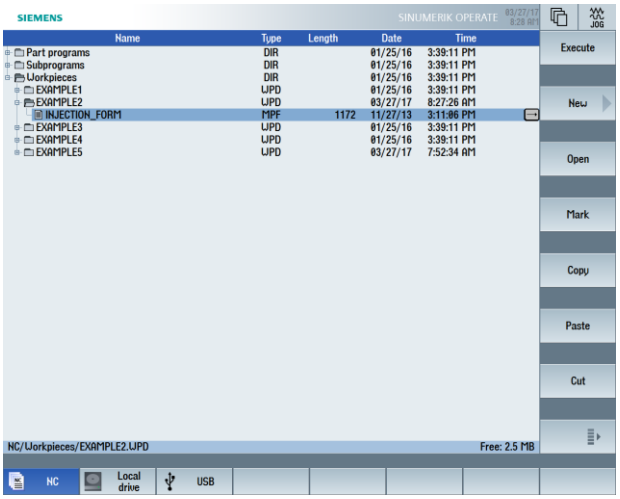


Figure 11-1 Selecting the program

Open the program.

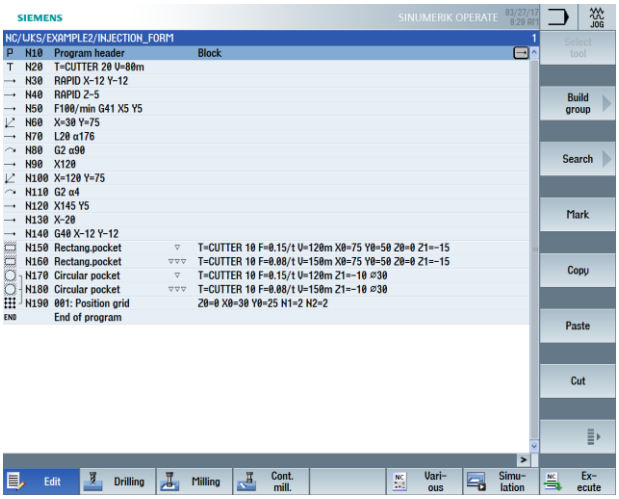


Figure 11-2 Opening the machining plan

Select the "NC Execute" softkey.

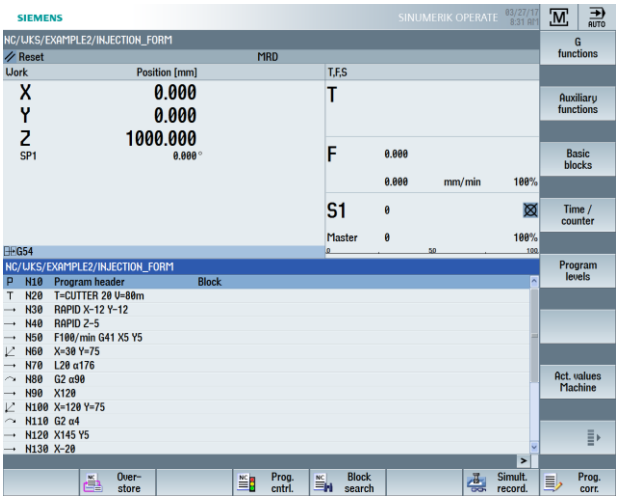
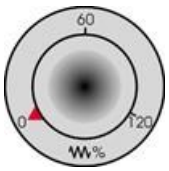


Figure 11-3 Executing

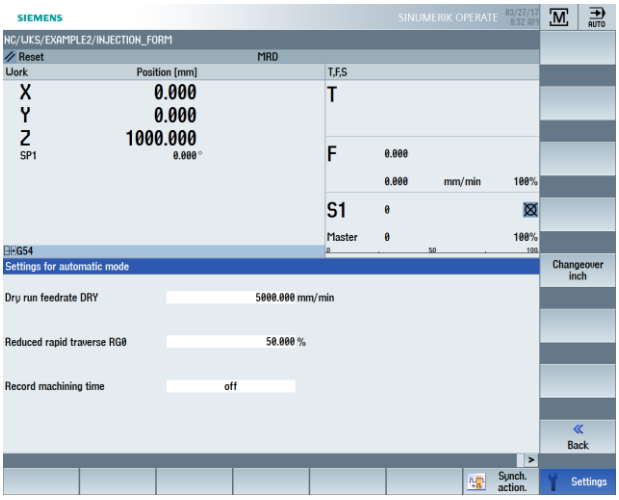


Due to the fact that the machining plan has not yet been executed with control, turn the feedrate potentiometer to zero position to ensure that you keep everything under control from the beginning.

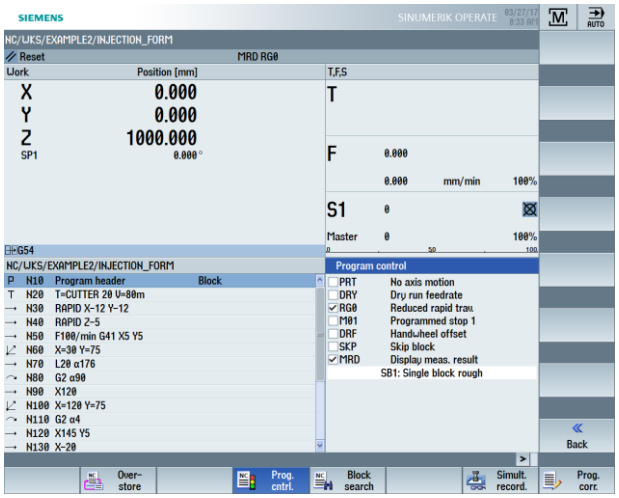
If you also want to see a simulation during the machining, select the "Simult. record." softkey before starting. Only then are all traversing motions and their effects displayed.

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

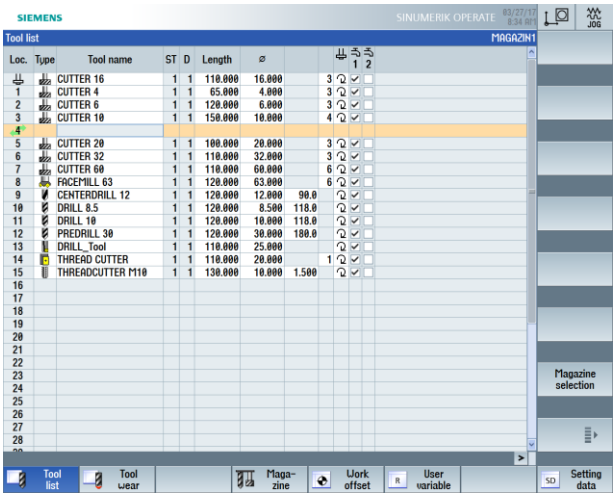
To move in at reduced rapid traverse RG0, the percentage of the maximum rapid traverse of the machine, in the second level of the horizontal softkey bar, must be set under Settings, e.g. 50%.



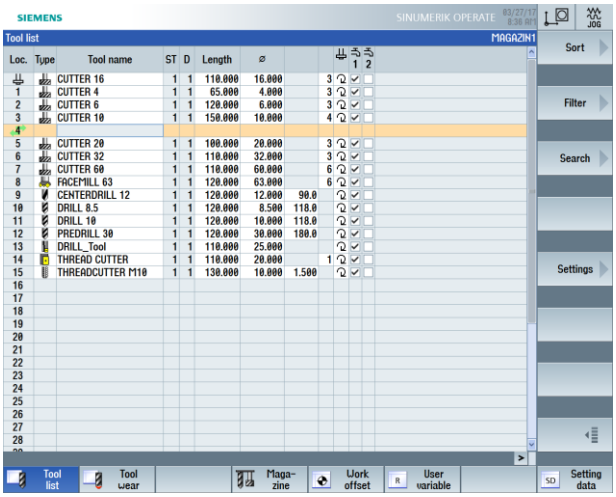
Once the percentage is set, the reduced rapid traverse must also be selected under "Program control" with the toggle key.



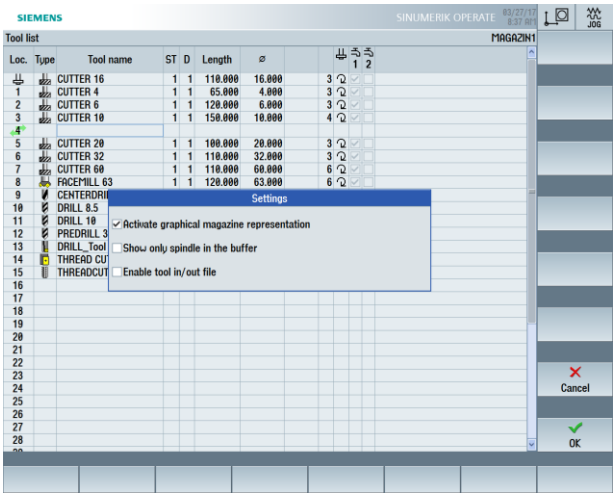
Set the graphic tool display in the tool list.

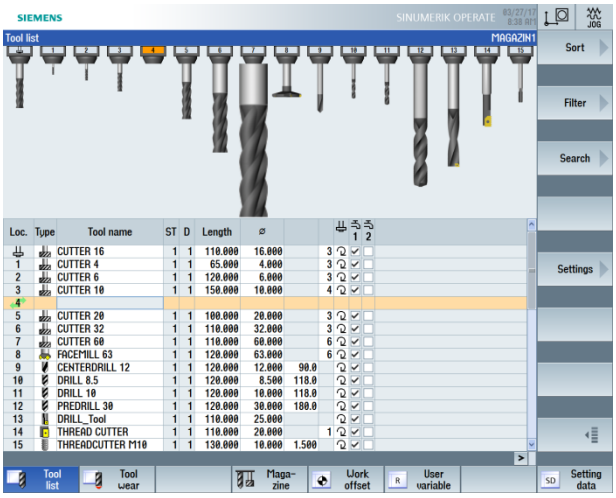


Go to the next level in the vertical softkey bar, and select the "Settings" softkey.



Activate the "Activate graphical magazine representation" in the dialog window with the toggle key.





Result of the setting:
The tools are represented above the tool list.

OPERATE keyboard shortcuts

Control key:	
CTRL + P	For screenshots – Storage location: Commissioning (password) – System data – HMI data – Logs – Screenshots
CTRL + L	Language switchover
CTRL + C	Copy
CTRL + X	Cut
CTRL + V	Paste
CTRL + Y	Redo (editor functionality)
CTRL + Z	Undo – max. five lines in the editor (editor functionality)
CTRL + A	Select all (editor functionality)
CTRL + NEXT WINDOW	Go to start of program
CTRL + END	Go to end of program
CTRL + ALT + S	Save complete archive – NCK/PLC/drives/HMI
CTRL + ALT + D	Backup log files on USB or CompactFlash card
CTRL + E	Control energy

Keyboard shortcuts, continued

CTRL + M	Maximum simulation speed
CTRL + F	Search in all screen forms Wildcards "?" and '"*" can be used in search screen forms. "?" stands for any character, "*" for any number of any characters.
Miscellaneous:	
Shift + INSERT	Commenting out of cycles and direct editing of programGUIDE cycles
Shift + END	Select up to end of block
Shift + NEXT WINDOW	Select up to start of line
ALT + NEXT WINDOW	Jump to start of line
ALT + S	Enter Asian characters
=	Calculator function
HELP	Help function
END	Jump to end of line

Keyboard shortcuts, continued

Simulation and simultaneous recording:	
▲ ▶ ▼ ◀	Move
Shift + ▲ / ▼	Rotate in 3D display
▲ ▼	Move section
CTRL + ▲ / ▼	Override +/- (simulation)
CTRL + S	Single block on/off (simulation)
Insert key:	
INSERT	It brings you into the Edit mode for text boxes or into the Selection mode of combo boxes and toggle fields. You can exit this without making any changes by pressing Insert again.
INSERT	Undo function,as long as no Input key is pressed or no data has already been transferred to the fields.
Toggle key:	
SELECT	You can directly switch between toggle fields using the Toggle key (Select) without having to open them. With Shift-Toggle you can switch through these in the reverse direction.
Cursor key:	
◀ ▶	Open/close directory Open/close program Open/close cycle