SIEMENS

Training Documents

Milling made easy with ShopMill

SINUMERIK Operate

Output

10/2021

www.siemens.com/sinumerik

SIEMENS

SINUMERIK Operate

Easy milling with ShopMill

Training Documents

10/2021

6FC5095-0AB50-1BP3

Legal information

Warning notice system

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

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

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

⚠ **CAUTION**
indicates that minor personal injury can result if proper precautions are not taken.

NOTICE
indicates that property damage can result if proper precautions are not taken.

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

Qualified Personnel

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

Proper use of Siemens products

Note the following:

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

Trademarks

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

Disclaimer of Liability

We have reviewed the contents of this publication to ensure consistency with the hardware and software described. Since variance cannot be precluded entirely, we cannot guarantee full consistency. However, the information in this publication is reviewed regularly and any necessary corrections are included in subsequent editions.

Siemens AG
Digital Industries
Postfach 48 48
90026 NÜRNBERG
GERMANY

Document order number: 6FC5095-0AB50-1BP3
Ⓟ 01/2022 Subject to change

Table of contents

Introduction 1

Faster from the drawing to the workpiece - but how?

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

The solution is: Creating a process plan instead of programming

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

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

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

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

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

The theory is followed by practical exercises with ShopMill:

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

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

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

SINUMERIK ONE - Run MyVirtual Machine

With Run MyVirtual Machine, the digital twin of SINUMERIK ONE, NC programs can be programmed offline and checked without requiring a real machine. Run MyVirtual Machine provides safety, and avoids collisions occurring at the real machine.

In this ShopMill training document, Section "Entry into Run MyVirtual Machine (Page 17)" provides you with an overview about how you can use Run MyVirtual Machine and of its advantages. Further, the user interface of the project management and a machine project that has been started are explained.

In Section "Execution in Run MyVirtual Machine /3D (Page 227)", based on an example, you get to know the first steps in Run MyVirtual Machine and how you can run 3D simulation.

Note

You require the appropriate licenses for "Run MyVirtual Machine /Operate" and "Run MyVirtual Machine /3D". You can purchase the software through the Industrial Software Store (https://www.dex.siemens.com). Here, you can also apply for your individual free-of-charge test license!

Note regarding the exercises

Note

The training documentation was created based on a SINUMERIK ONE with operator interface "SINUMERIK Operate". You can choose to work through examples 1 to 5 on a SINUMERIK ONE, SINUMERIK 840D sl, SINUMERIK 828D with SINUMERIK Operate, with Sinutrain or with Run MyVirtual Machine. The basic operation of SINUMERIK Operate is identical in all cases.

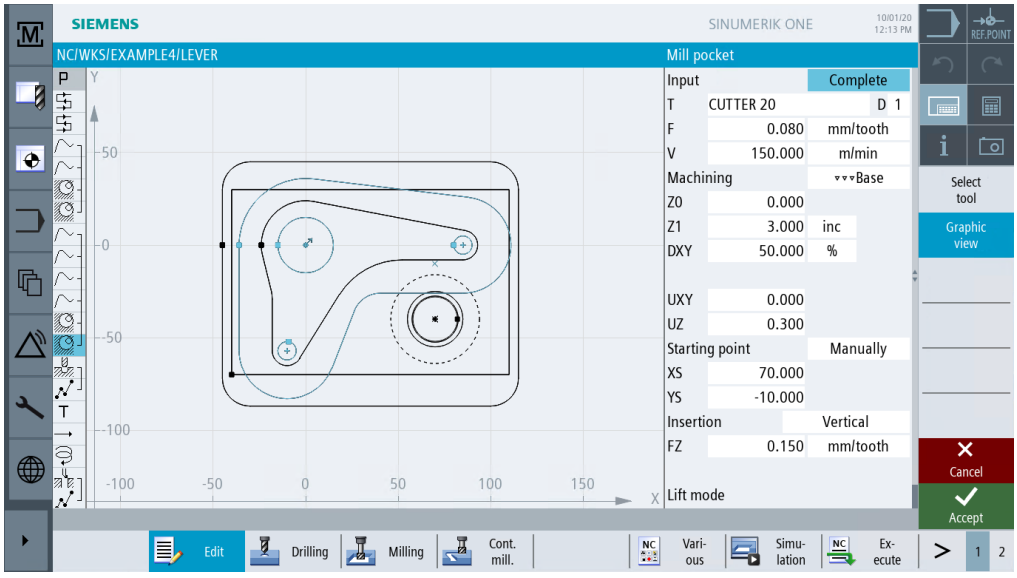
Advantages of ShopMill

2

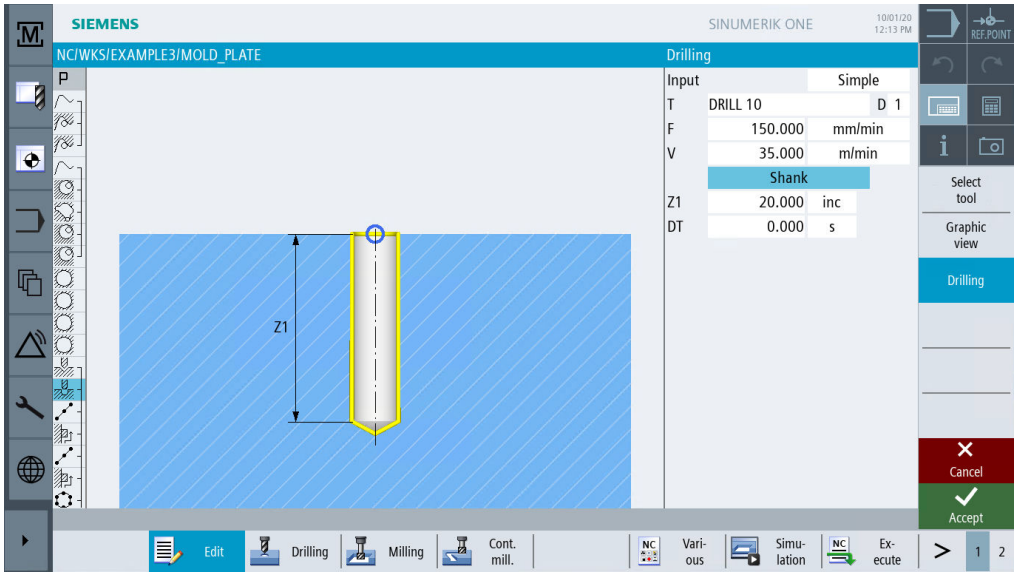
This chapter shows you the special advantages of ShopMill.

2.1 You will save time for training....

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



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



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

```
G  N25 G17 G54 G64 G90 G94¶
G  N30  T="CUTTER 16"¶
G  N35 G0 X85 Y22.5¶
G  N40 G0 Z2 S500 M3 M8¶
G  N45 G0 Z-10¶
G  N50 G1 X-85 F200¶
G  N55 G0 Y-22.5¶
G  N60 G1 X85¶
G  N65 G0 Z100 M5 M9¶
```

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

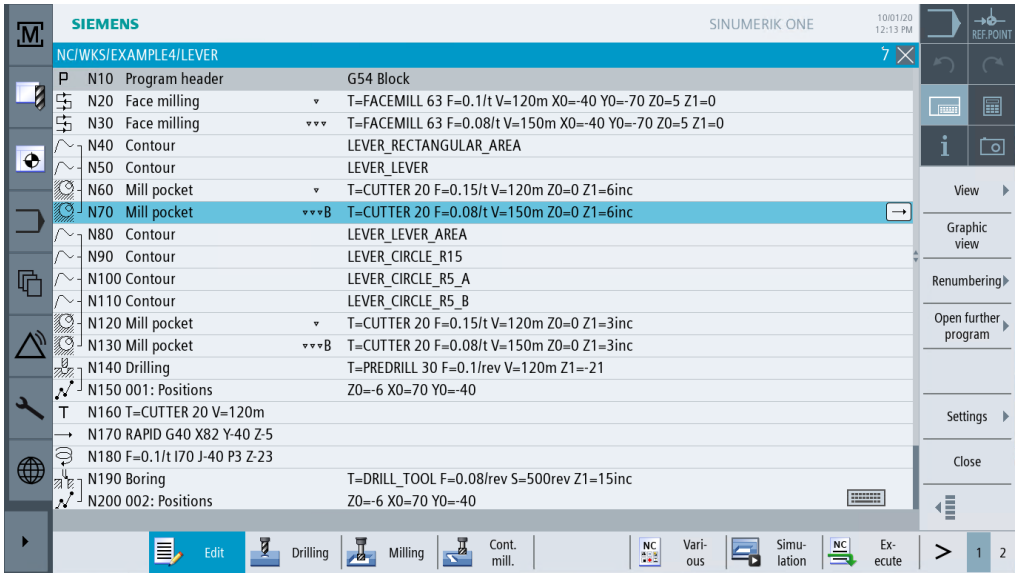


Figure 2-1 Work step in a process plan

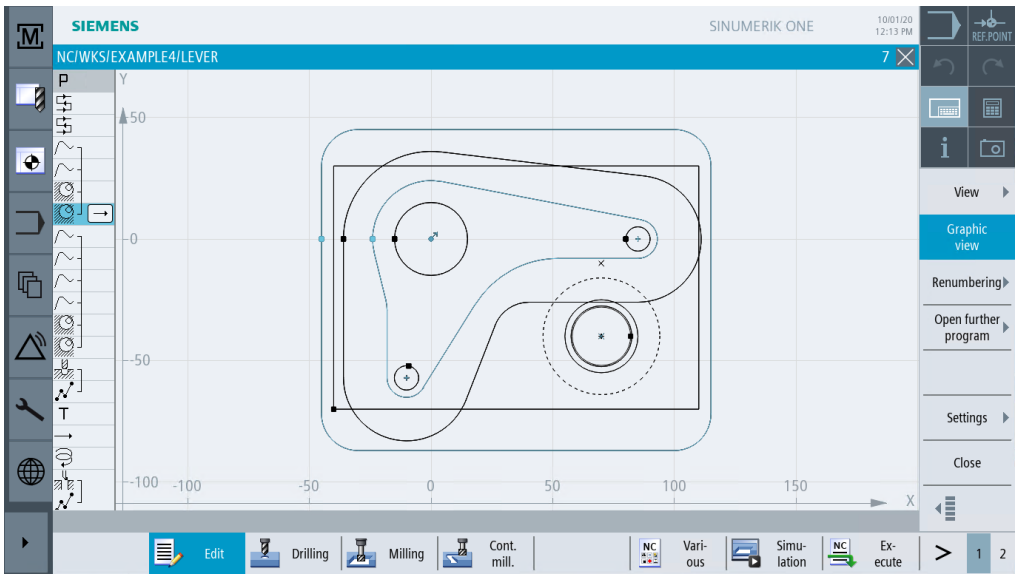


Figure 2-2 Graphic view

2.2 You will save time for programming...

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

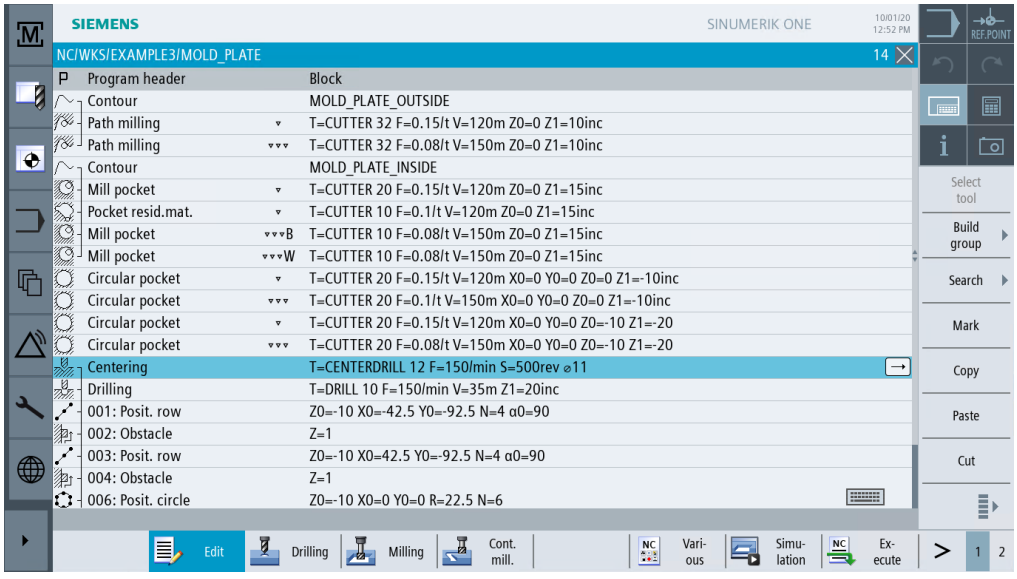
Circular pocket			
Input		Complete	
T	CUTTER 20	D	1
F	0.150	mm/tooth	
V	120.000	m/min	

Circular pocket			
Input		Complete	
T	CUTTER 20	D	1
F	859.500	mm/min	
S	1910	rpm	

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

NC/WKS/EXAMPLE3/PRT_PROG		2
Program header	G54 Block	
Circular pocket	T=CUTTER 16 F=0.2/t V=150m X0=60 Y0=45 Z0=0 Z1=20inc	
END End of program		

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



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

Centering	T=CENTERDRILL 12 F=150/min S=500rev ⌀11
Drilling	T=DRILL 10 F=150/min V=35m Z1=20inc
001: Posit. row	Z0=-10 X0=-42.5 Y0=-92.5 N=4 α0=90
002: Obstacle	Z=1
003: Posit. row	Z0=-10 X0=42.5 Y0=-92.5 N=4 α0=90
009: Obstacle	Z=1
006: Posit. circle	Z0=-10 X0=0 Y0=0 R=22.5 N=6
007: Obstacle	Z=1
008: Positions	Z0=-10 X0=0 Y0=42.5

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

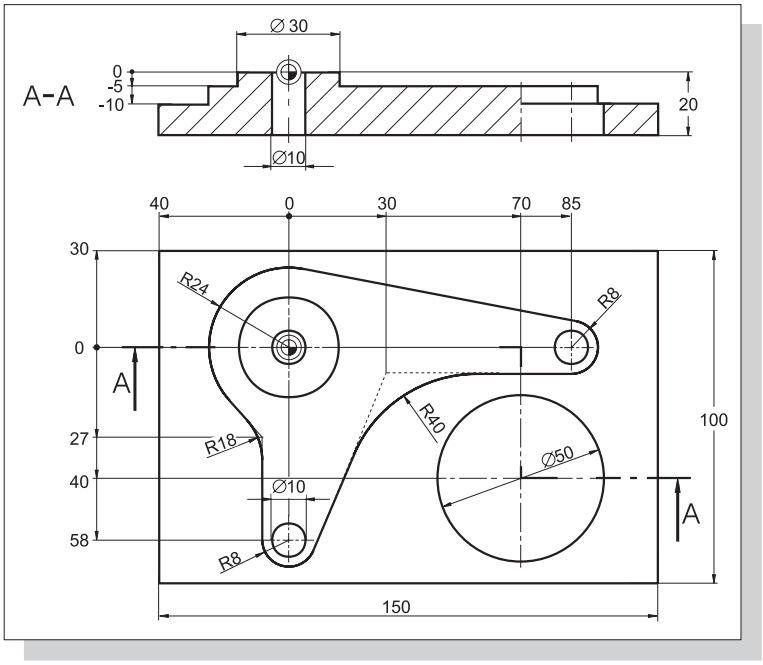


Figure 2-3 Technical drawing

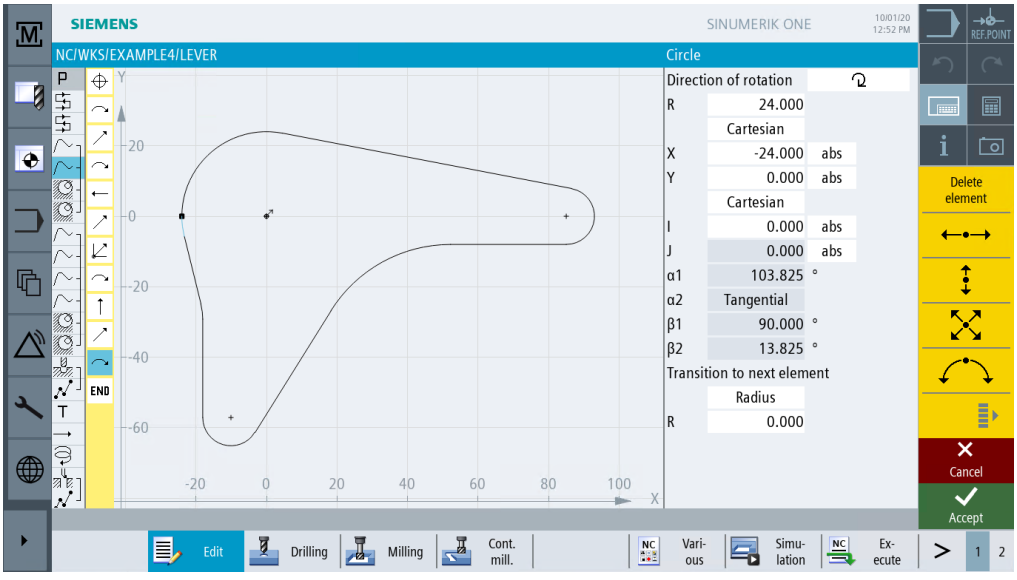
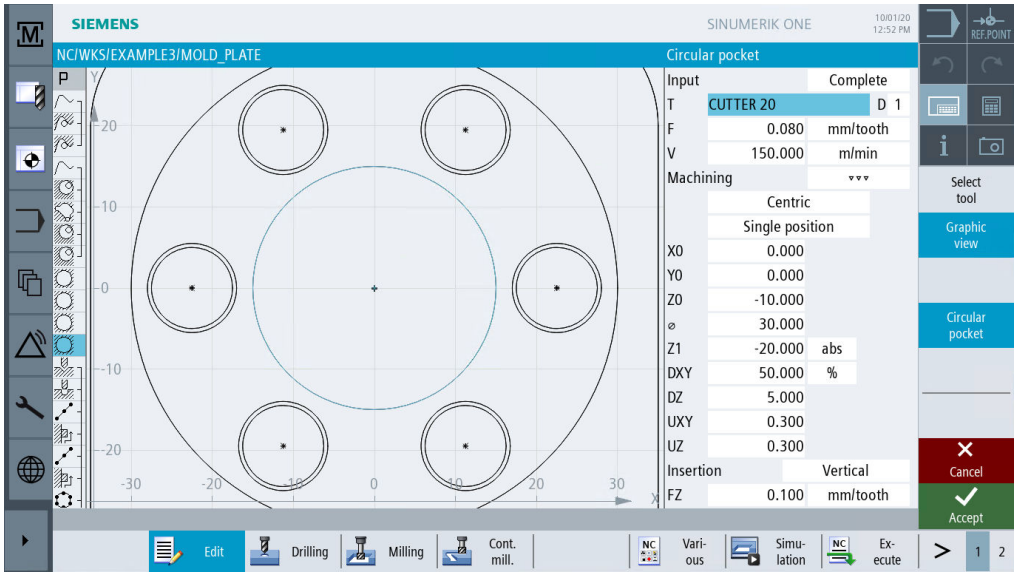


Figure 2-4 Screenform

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



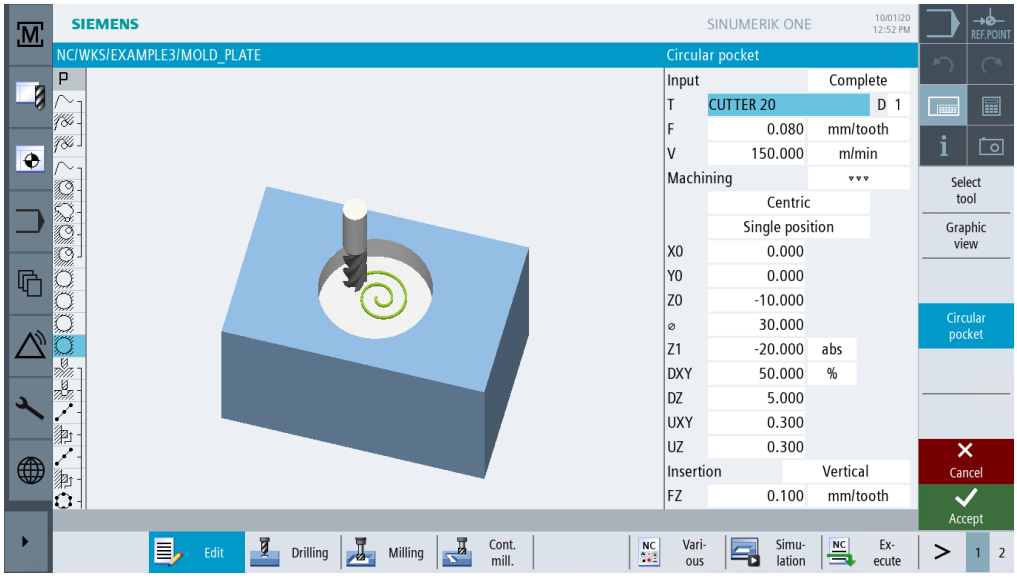
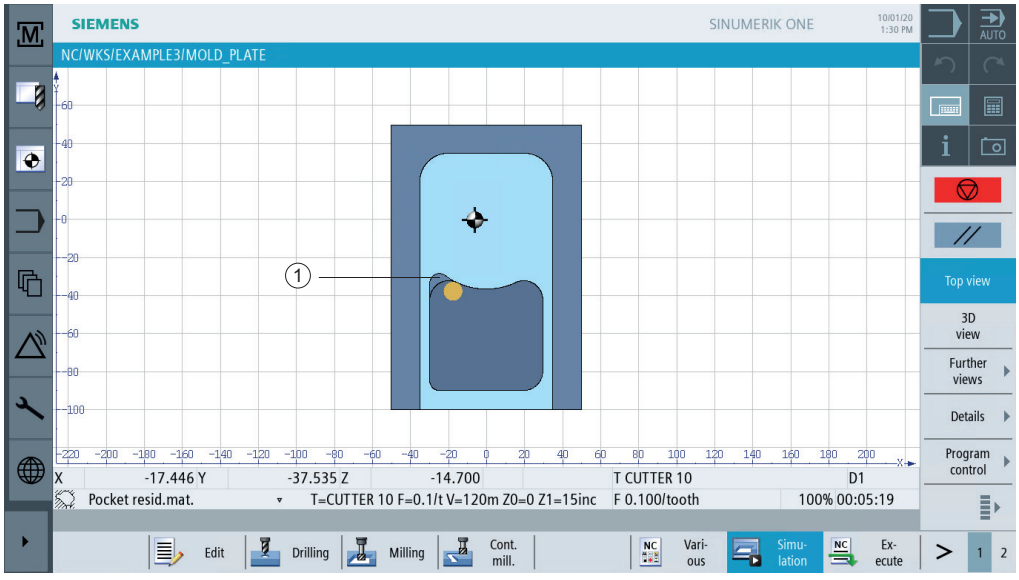


Figure 2-5 Parameter screen with help display

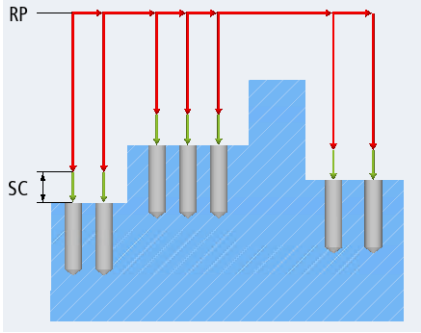
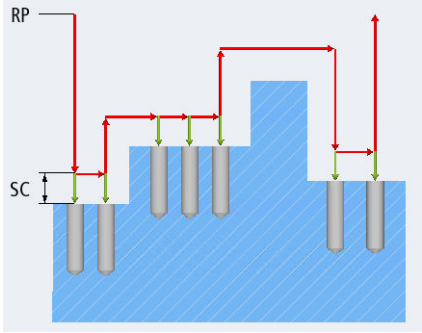
- Process plan and manufacturing do not exclude each other. With ShopMill, you can create a new process plan parallel to manufacturing.

2.3 You will save time for manufacturing...

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



- There are no unnecessary infeed motions between retraction and machining planes when positioning the tool. This is made possible by the settings **Retract to retraction plane (RP)** and **Optimized retraction**.
  The **Optimized retraction** setting is to be made by a skilled worker in the program header. He must take into account obstacles, such as clamping elements.

Retraction to retraction plane (RP)	Retraction to retraction planes = time saved in manufacturing
	

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

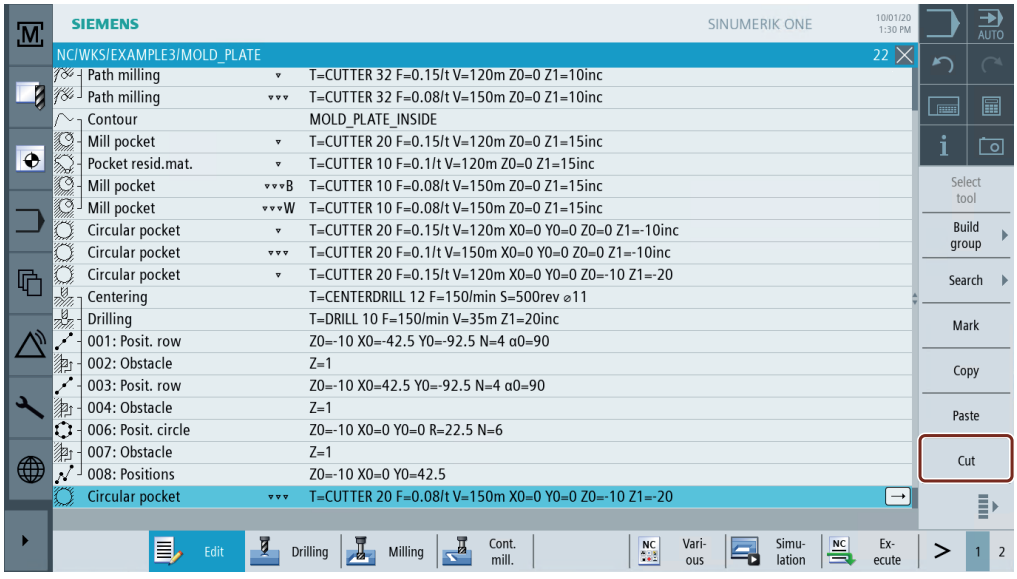


Figure 2-6 Original machining sequence

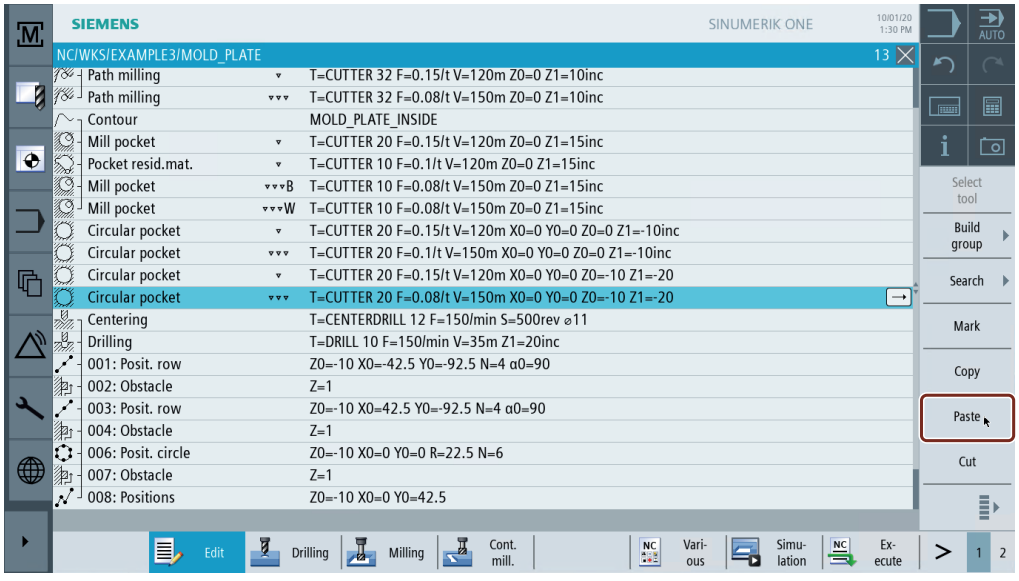


Figure 2-7 Optimized machining sequence with **cutting** and **pasting** a work step

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

Entry into Run MyVirtual Machine 3

This chapter introduces you to Run MyVirtual Machine, the digital twin of SINUMERIK ONE. In addition to the product description, the application area and the benefits, you also get to know the Run MyVirtual Machine user interface. You get to familiarize yourself with project management and the user interface of a machine project that has already been started.

In Section "Execution in Run MyVirtual Machine /3D (Page 227)" you learn how to create a machine project in Run MyVirtual Machine. You learn about the fundamentals of 3D simulation in Run MyVirtual Machine /3D, and based on a program example, you execute the first steps to start a 3D simulation.

Note

You require the appropriate licenses for "Run MyVirtual Machine /Operate" and "Run MyVirtual Machine /3D". You can purchase the software through the Industrial Software Store (https://www.dex.siemens.com). Here, you can also apply for your individual free-of-charge test license!

3.1 What is Run MyVirtual Machine?

Run MyVirtual Machine is an NC programming workstation with identical controls on the PC for machine tools using SINUMERIK ONE.

The offline programming workstation Run MyVirtual Machine simulates a machine tool controlled by SINUMERIK ONE. SINUMERIK Operate, together with a simulated machine control panel, ensures realistic operation and programming on the PC. You do not need any additional programming knowledge.

Run MyVirtual Machine enables offline CNC programming on the PC, for example in work preparation. Exactly the same scope of CNC language commands, CNC machining cycles, and ShopMill/ShopTurn work steps is available as in the real CNC. Regardless of whether CNC programs were generated via Run MyVirtual Machine itself or via CAM systems, they can be checked for freedom from errors in the best possible way.

Run MyVirtual Machine is therefore the optimal tool to increase efficiency and process reliability in CNC programming.

Furthermore, Run MyVirtual Machine permits easy learning and professional training of CNC operation and programming without a real CNC, for example in training classrooms. For this purpose, preconfigured sample machines are available for immediate use. Using SINUMERIK Operate and the original SINUMERIK CNC kernel, all operating processes and NC programming operations can be used and CNC programs executed, without any restrictions. New functions and programming options can thus be learned, tested and demonstrated in a secure environment.

To get the highest possible match with the real CNC, you can load machine projects (*.vcp) matching the respective machine. Contact your machine manufacturer for this purpose.

Since the machine projects are always assigned to a version of the SINUMERIK Virtual CNC software, different SINUMERIK Virtual CNC software versions can be stored in Run MyVirtual Machine.

This means that machines from different manufacturers can be provided with different versions of the SINUMERIK Virtual CNC software at one work station in the CNC work preparation.

Additional options:

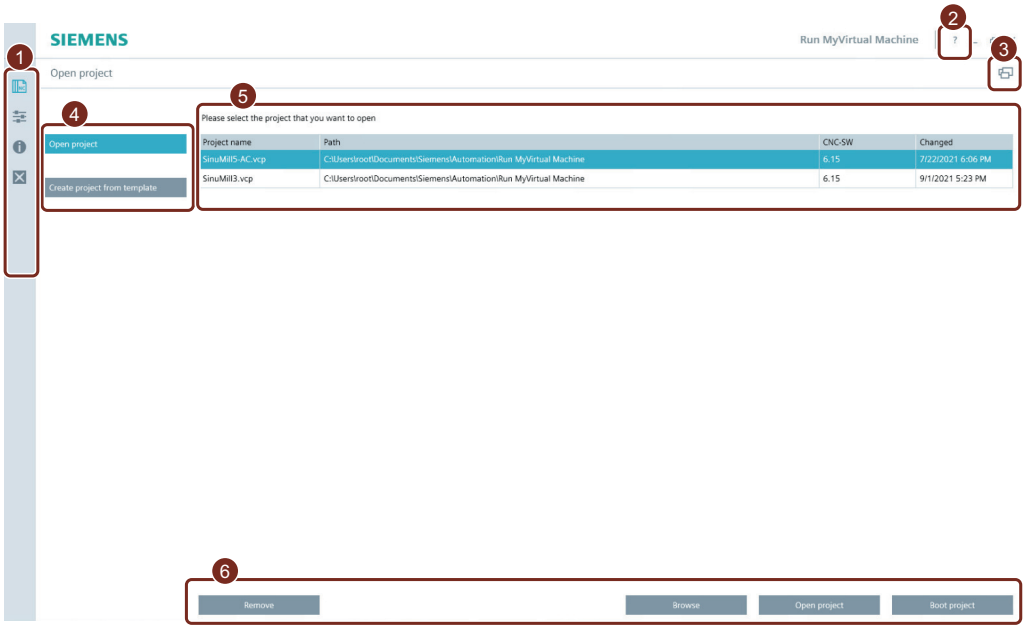
- Run MyVirtual Machine /Open is an additional option to Run MyVirtual Machine /Operate. You need it to operate an external SW application, for example, your own machine room simulation.
- Run MyVirtual Machine /3D is an additional option for Run MyVirtual Machine /Operate. This option extends Run MyVirtual Machine to include integrated 3D machining and material removal simulation. This enables you to evaluate machine movements visually and to check for freedom from collision. Using the material removal simulation, workpiece machining can be tested in advance via simulation.
  The 3D simulation is also ideal for training setup procedures and running in machines on a virtual model without any exposure to risks.

3.2 Project management in Run MyVirtual Machine

You can manage machine projects (*.vcp; **V**irtual **C**ommissioning **P**roject) based on machine templates in the project management of Run MyVirtual Machine. The machine projects are provided by the machine manufacturer, for example, or are included as project templates in Run MyVirtual Machine.

In the project management, you can open and delete projects and create a new project based on a project template.

A machine project manages all the data required to operate the machine. The machine project file contains NC, HMI, PLC and drive data specifying the version of the CNC software used.



① Basic functions

Click on the buttons to use the basic functions of Run MyVirtual Machine.

Displays of project overview

Settings
Opens the settings to switch between languages and manage the window layout.

Information
Displays the version overview

Exit
Exits Run MyVirtual Machine

② Help

Open/close Help. The Help is displayed in a separate viewlet. You can extract the viewlet and display as a separate window.

③ Display/hide viewlets

You can show/hide the viewlets/window areas using the button. Check/uncheck the checkbox in front of the respective viewlet name in the displayed list.

You can remove the individual viewlets as windows with the buttons, and dock them at any other position in Run MyVirtual Machine. For example, you can display HMI SINUMERIK Operate in a separate window.

④ Main menu

Open project
Opens existing projects from the overview

Create project from template
Creates a new project based on a template.

⑤ Project overview

Overview of the most recently opened machine projects with storage path, CNC SW version used and change date.

⑥ Buttons

Remove
Removes projects from the project overview. The machine project is only deleted from the overview, and remains on the data storage medium.

Browse
Searches the data storage medium for projects and enters them into the overview.

Open project
Opens a selected machine project in the overview.

Start project

Opens a selected machine project from the overview. The machine is automatically started.

3.3 Machine project in Run MyVirtual Machine

Using Run MyVirtual Machine corresponds to a real control system equipped with a SINUMERIK Operate user interface and machine control panel. The machine basic screen is displayed after the control system has powered up.

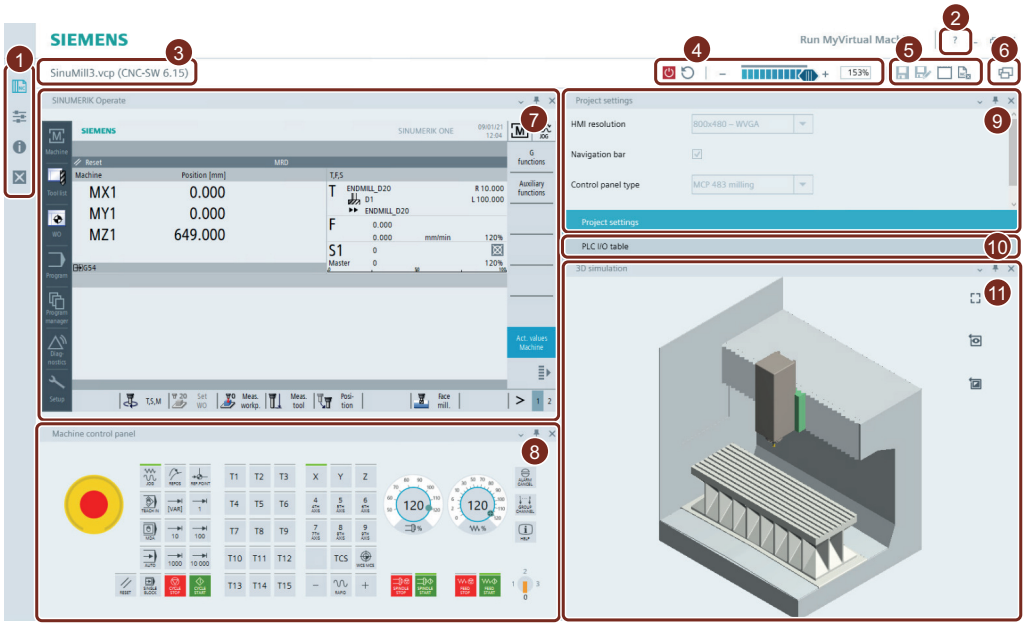


Figure 3-1 Run MyVirtual Machine with an open machine project

① Basic functions

Click on the buttons to use the basic functions of Run MyVirtual Machine.

Displays of project overview

Settings
Opens the settings to switch between languages and manage the window layout.

Information
Displays the version overview

Exit
Exits Run MyVirtual Machine

② Help

Open/close Help. The Help is displayed in a separate viewlet. You can extract the viewlet and display as a separate window.

③ Title bar

Display of the project name and version of the CNC software.

④ Simulation control

Start simulation of the machine.
The simulation control cannot be operated during the ramp-up phase.

Exit simulation of the machine.

Reset
Initiate NCK/PLC warm restart

Change simulation speed from standstill (pause – system stopped) up to the maximum speed.

- In the left position (-) 0%, the simulation is in pause mode. In this state, the "frozen" machining process can be observed.
- In the center position 100%, the simulation speed approximates the speed/clock cycle of a real machine and runs approximately in real-time.
- In the right (+) position, the simulation runs with maximum speed. The percentage display specifies how much faster the system operates compared with real-time. For example, the value 800% corresponds to approximately the 8-fold real-time speed. The maximum simulation speed is limited by several factors, including the computer power.

⑤ Management of the open machine project

- Save
  Saves the open machine project.
  Machine projects can be saved only if the machine simulation has been previously exited.
- Save as
  Saves the open machine project under a new name or in another directory.
- Memory card
  Opens Windows Explorer with the storage location of the virtual memory card.
- Close project
  Closes an open machine project. If there are unsaved changes, a note is displayed, and you can save the project before closing.

⑥ Display/hide viewlets

You can show/hide the viewlets/window areas using the button. Check/uncheck the checkbox in front of the respective viewlet name in the displayed list.

You can remove the individual viewlets as windows with the buttons, and dock them at any other position in Run MyVirtual Machine. For example, you can display HMI SINUMERIK Operate in a separate window.

⑦ HMI SINUMERIK Operate

The SINUMERIK HMI Operate viewlet contains the SINUMERIK Operate commissioning and operating software.

⑧ Virtual machine control panel

- EMERGENCY STOP
  The status of the Emergency Stop (pressed) is indicated by a pictogram below the red button. The Emergency Stop is functional only with the appropriate basic PLC program.

- Feedrate and spindle override
- Alarm, Channel, Help keys
- User assignable function keys
- Keyswitch (0-3)

⑨ Project settings

You can only change the project settings if the machine simulation has not been started.

- HMI resolution
  Select the HMI resolution of the SINUMERIK Operate. The HMI will be displayed in the selected resolution at the next startup.
- Navigation bar
  Select the checkbox if you want the side navigation bar to be displayed in the HMI. Through the navigation bar you have quick access to the machine areas of the HMI, e.g. program or tool list.
- Control panel type
  Display of the machine control panel (e.g. MPC 483 for milling or MCP 483 for turning) used in the machine project.

⑩ PLC I/O table

You read and write the PLC inputs and outputs with the integrated I/O simulation. Outputs with status LEDs and inputs with toggle switches are configured in the table lines of the expandable PLC I/O table. As a machine operator, you do not usually use a PLC I/O table.

⑪ 3D simulation

During execution of an NC program in AUTOMATIC mode, 3D simulation with collision monitoring enables you to check the machining process so that any program errors can be detected.

To make everything function smoothly...

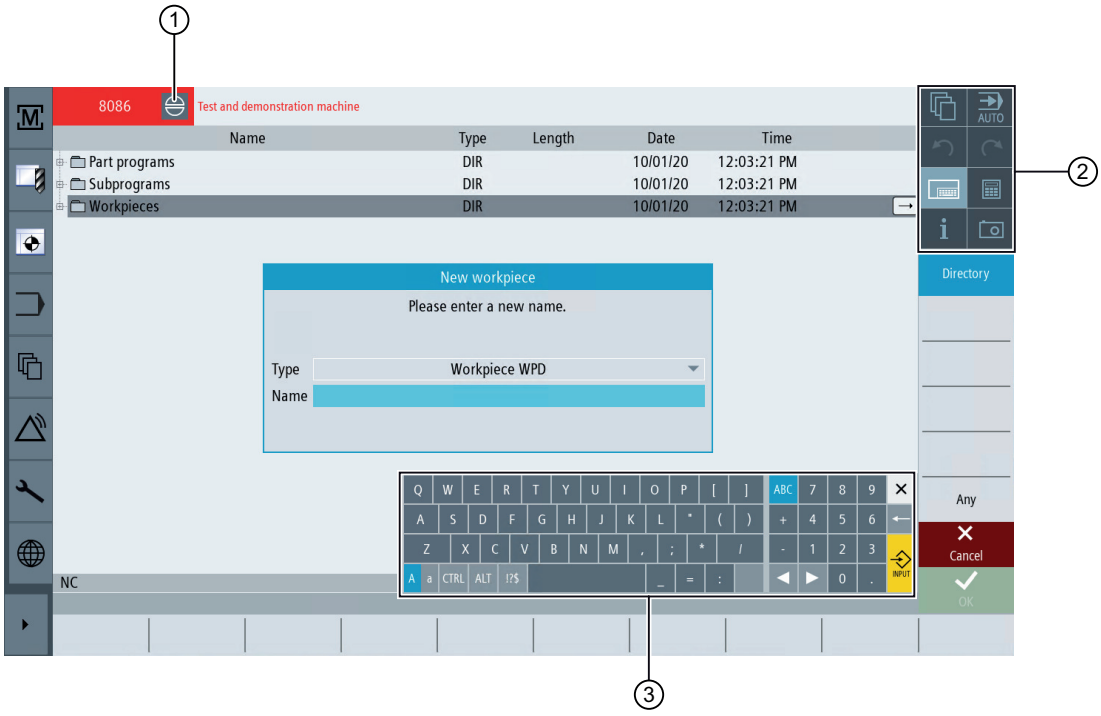
4

In this chapter, you will be familiarized with the fundamentals of multitouch operation with the "SINUMERIK Operate Generation 2" user interface and the operating areas of SINUMERIK Operate based on a number of examples.

4.1 Multitouch operation of SINUMERIK Operate

4.1.1 Screen layout

Touch and gesture operator controls for SINUMERIK Operate with the "SINUMERIK Operate Generation 2" user interface.



① Cancel alarms
② Function key block
③ Virtual keyboard

4.1.2 Function key block

Operator control	Function
	**Switch operating area** Tap the current operating area, and select the desired operating area from the operating area bar.
	**Switch operating mode** The operating mode is only displayed. To switch the operating mode, tap the operating area and select the operating area from the vertical softkey bar. The selection for the functions available for the operating mode is opened.

Operator control	Function
	Close the selection The selection for the functions available for the operating mode is closed.
	Undo Multiple changes are undone one by one. As soon as a change has been completed in an input field, this function is no longer available.
	Restoring Multiple changes are restored one by one. As soon as a change has been completed in an input field, this function is no longer available.
	Virtual keyboard Activates the virtual keyboard.
	Calculator Displays a calculator.
	Online help Opens the online help.
	Camera Generates a screenshot.

4.1.3 Further operator touch controls

Operator control	Function
	Advances to the next horizontal softkey bar. When page 2 of the menu is called, the arrow appears on the right.
	Advances to the higher-level menu.
	Advances to the next vertical softkey bar.
2130	Tapping the Cancel alarm symbol clears all queued cancel alarms.

4.1.4 Virtual keyboard

If you called the virtual keyboard using the function key block, then you have the option of adapting the key assignment using the shift keys.

① Shift key for uppercase and lowercase letters
② Shift key for letters and special characters
③ Shift key for country-specific keyboard assignment
④ Shift key for full keyboard and numerical key block

Hardware keyboard

If a real keyboard is connected, the icon of a minimized keyboard appears in place of the virtual keyboard.

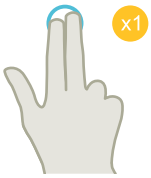
Use the icon to open the virtual keyboard again.

4.1.5 Finger gestures

Finger gestures

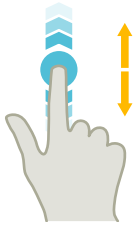
Tap

- Select window
- Select object (e.g. NC set)
- Activate entry field
  - Enter or overwrite value
  - Tap again to change the value



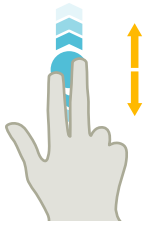
Tap with 2 fingers

- Call the shortcut menu (e.g. copy, paste)



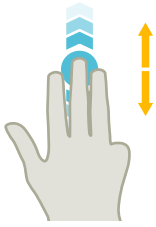
Flick vertically with one finger

- Scroll in lists (e.g. programs, tools, zero points)
- Scroll in files (e.g. NC program)



Flick vertically with two fingers

- Page-scroll in lists (e.g. ZO)
- Page-scroll in files (e.g. NC programs)



Flick vertically with three fingers

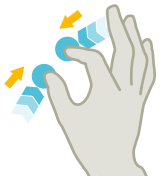
- Scroll to the start or end of lists
- Scroll to the start or end of files

Flick horizontally with one finger

- Scroll in lists with many columns

Spread

- Zoom in on graphic contents (e.g. simulation, mold making view)

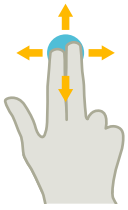


Pinch

- Zoom out from graphic contents (e.g. simulation, mold making view)

Pan with one finger

- Move graphic contents (e.g. simulation, mold making view)
- Move list contents



Pan with two fingers

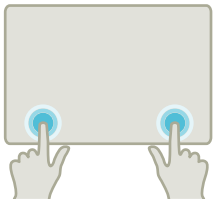
- Rotate graphic contents (e.g. simulation, mold making view)

Tap and hold

- Open input fields to change
- Activate or deactivate edit mode (e.g. current block display)

Tap and hold using 2 fingers

- Open cycles line by line to change (without input screen form)



Tap with two index fingers

- Tap with two fingers simultaneously in the lower right- and left-hand corners to open the TCU menu.
  The menu has to be opened for service purposes.

Note

Flicking gestures with several fingers

The gestures only function reliably if you hold your fingers sufficiently far apart. The fingers should be at least 1 cm apart.

4.2 The operating areas

4.2.1 Machine

Machine - Manual

Select the "Machine" softkey.

Switch to "JOG" mode.

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

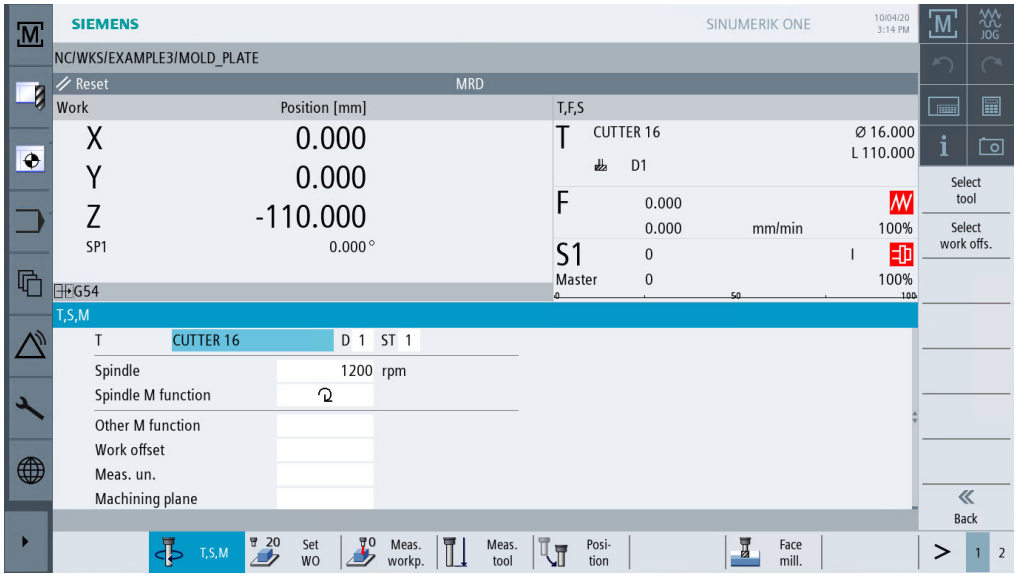


Figure 4-1 Call of a tool and input of technological values

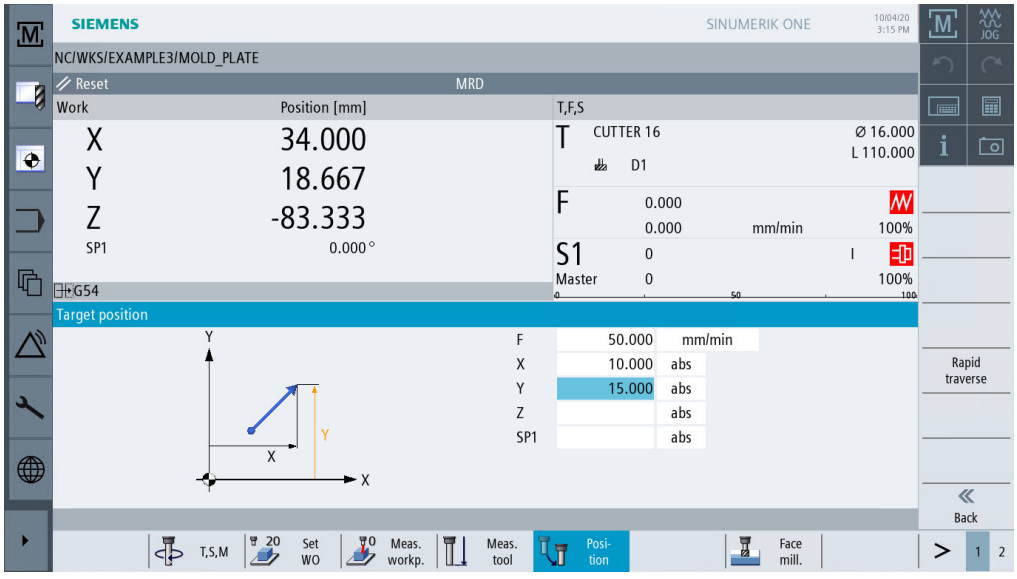


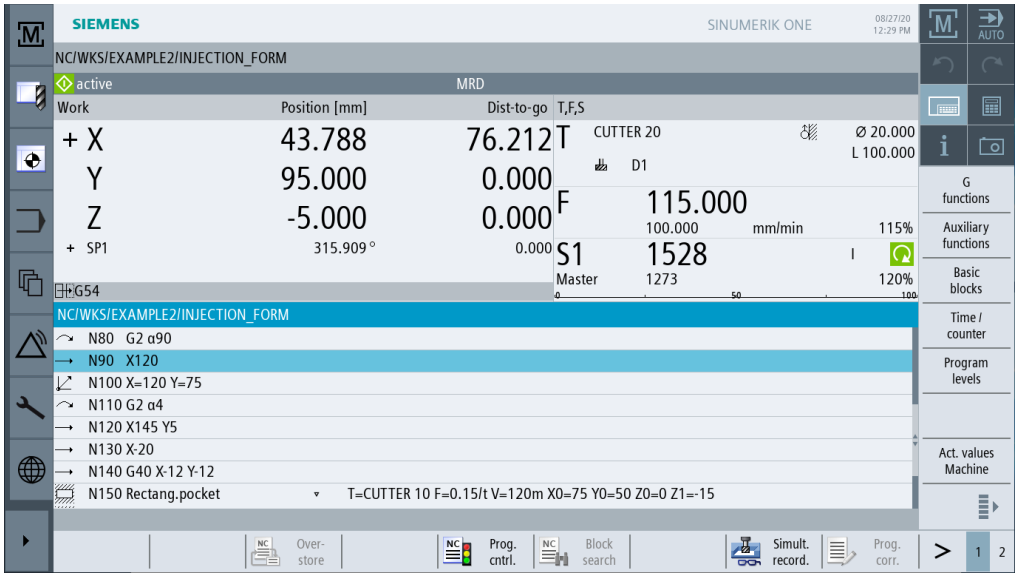
Figure 4-2 Input of a target position

Machine - AUTO

Select the "Machine" softkey.

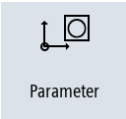
Switch to "AUTO" mode.

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



4.2.2 Parameters

Parameter lists



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

Tool lists

No cutting without tools.

The tools can be managed in a tool list.

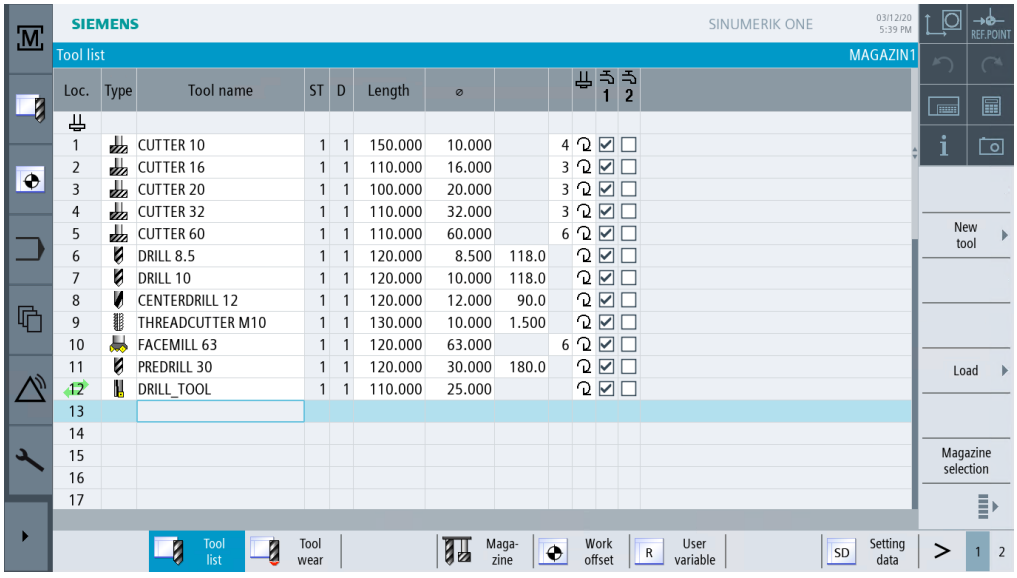


Figure 4-3 Tool list

Magazine

Tools can be organized into a magazine.

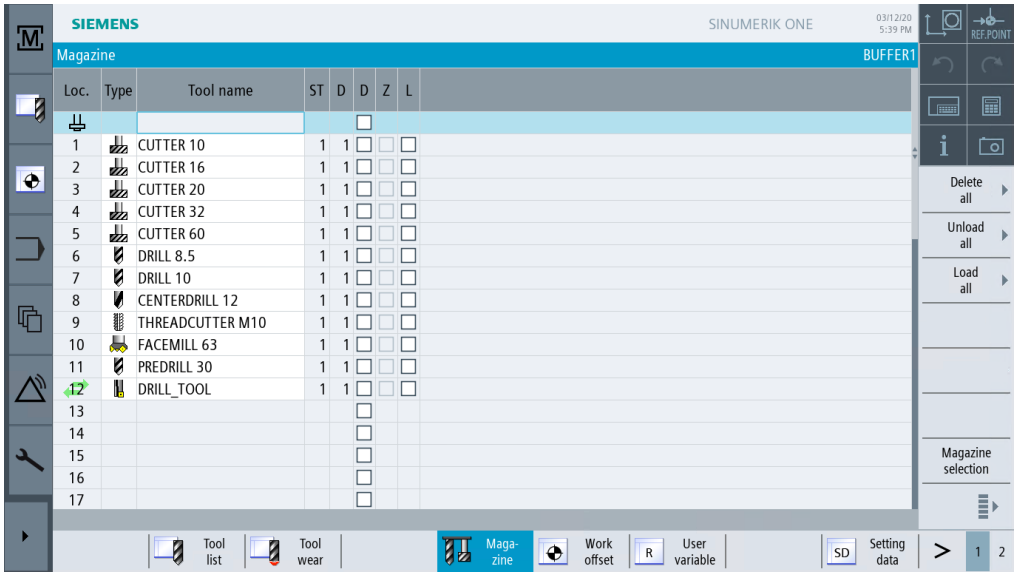


Figure 4-4 Magazine

Work offsets

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

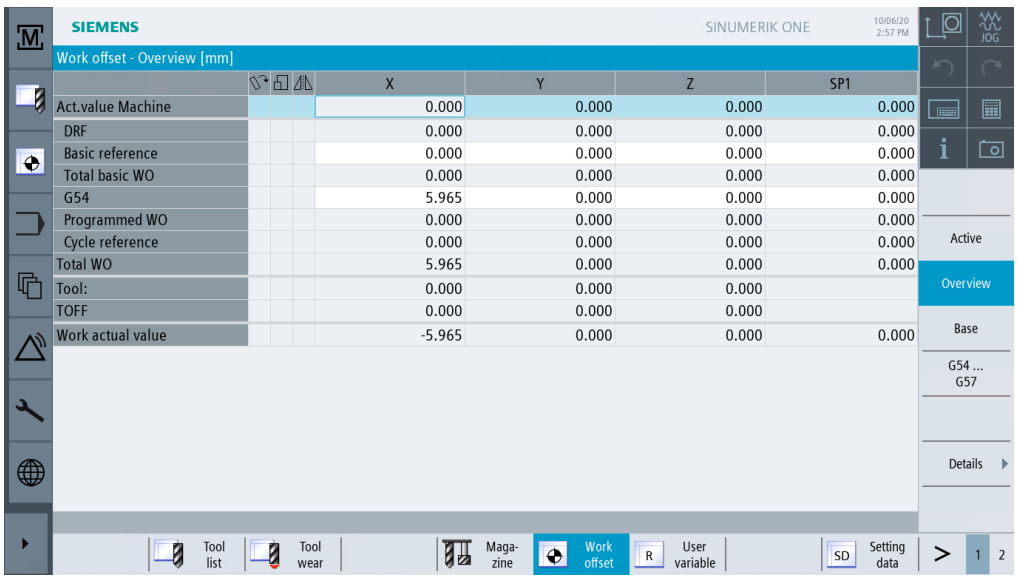


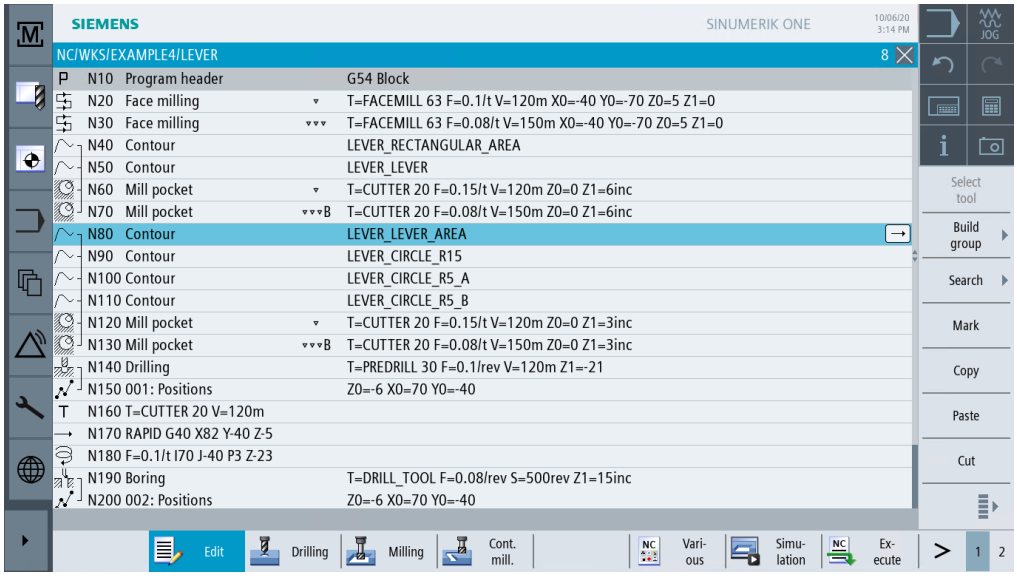
Figure 4-5 Work offsets

4.2.3 Program

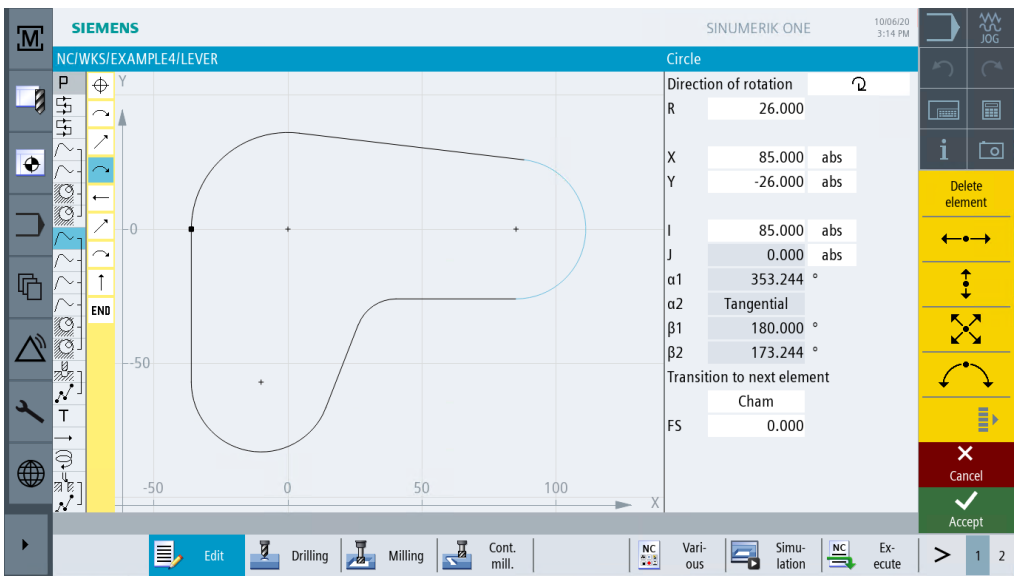
Editing programs

This key can be used to edit programs.

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

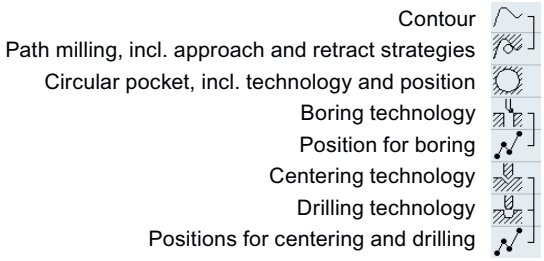


The contour to be machined is entered graphically as a machining step.



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

Example for the dovetailing of geometry and technology:



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

Simulating programs

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

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

The following views are available for simulation:

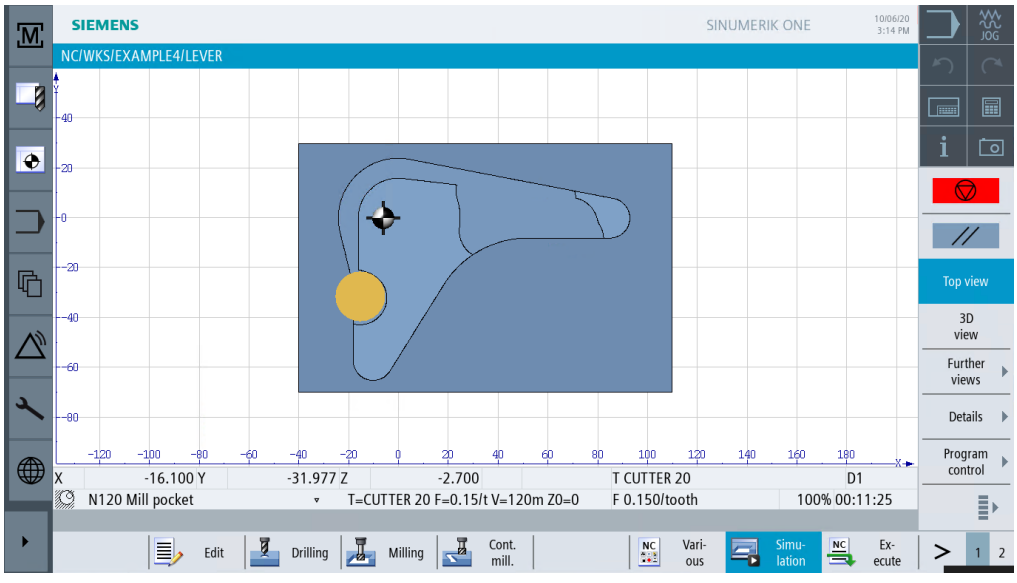


Figure 4-6 Top view

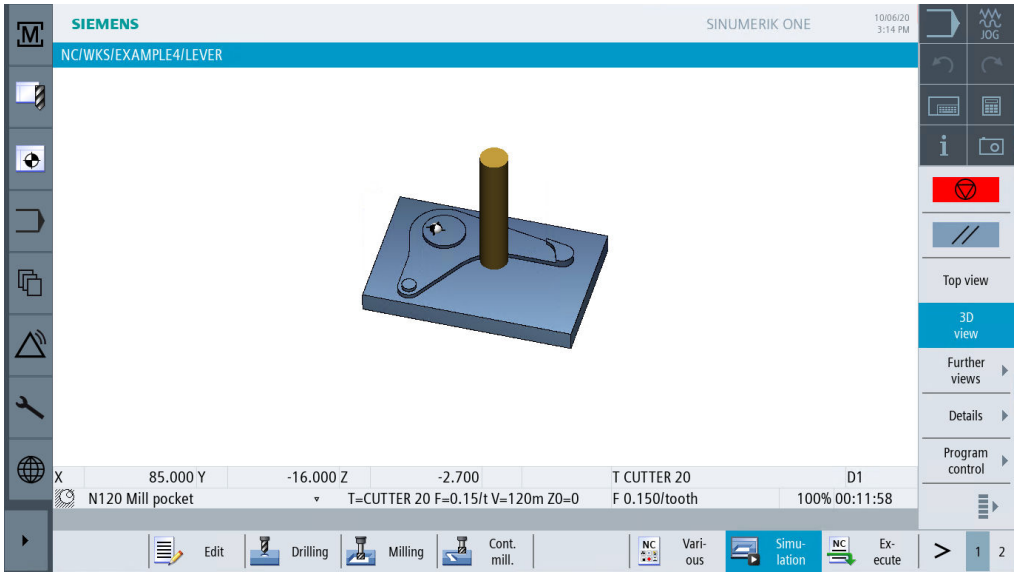


Figure 4-7 3D view

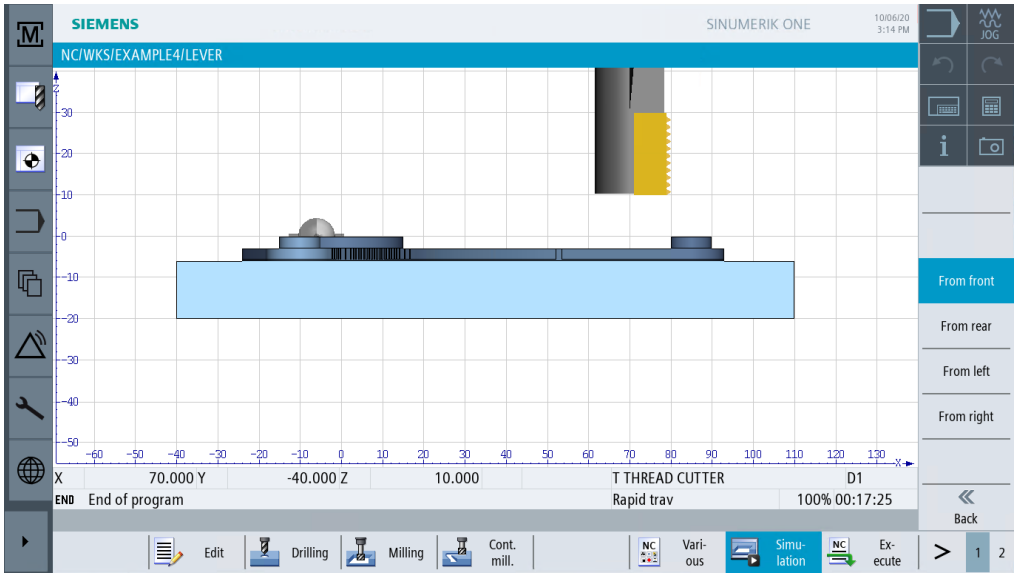
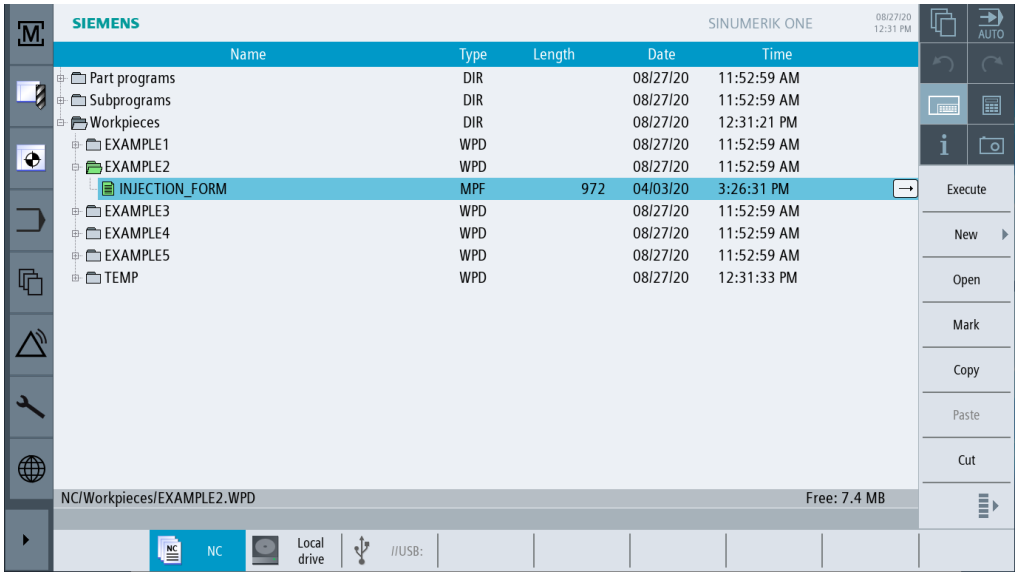


Figure 4-8 Side view

4.2.4 Program Manager

Managing programs

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



Active programs are marked with a green symbol.

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

Creating a new workpiece

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

Creating a new program

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

ShopMill program

G code program

4.2.5 Diagnosis

Alarms and messages

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

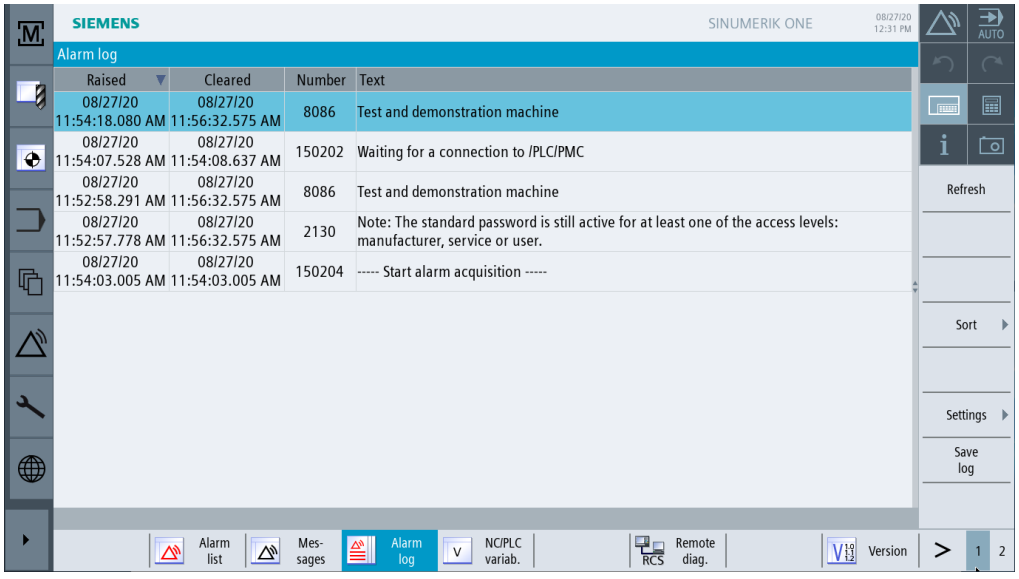


Figure 4-9 Alarm log

Geometrical basics

5

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

5.1 Tool axes and work planes

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

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

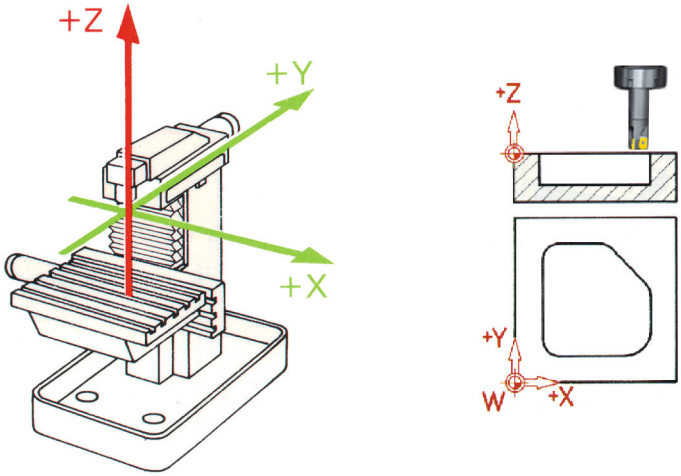


Figure 5-1 Vertical spindle

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

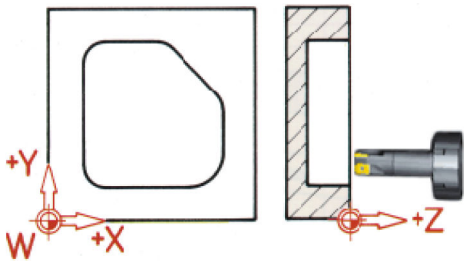


Figure 5-2 Horizontal spindle

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

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

Select the "Various" softkey.

Select the "Settings" softkey.

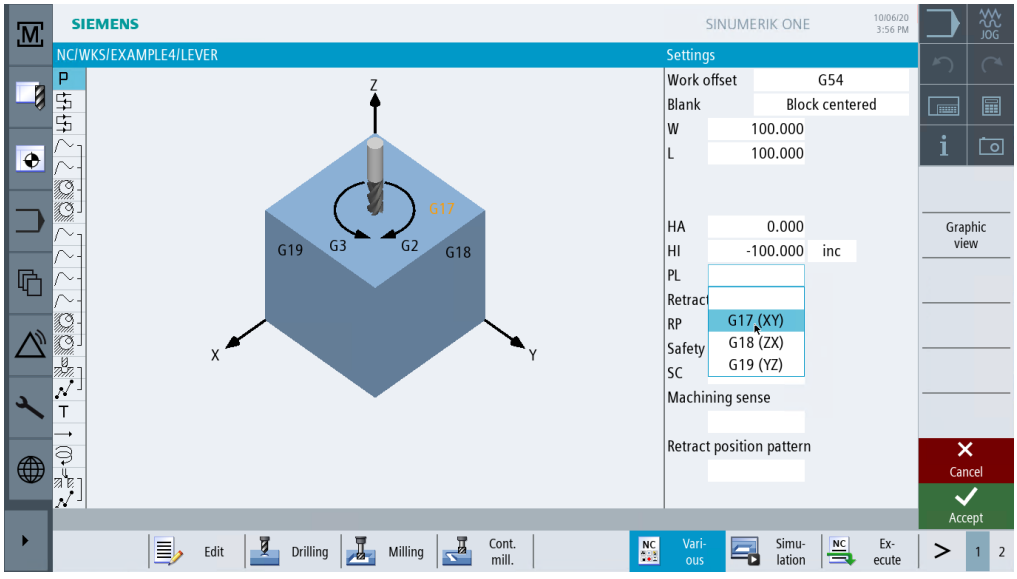
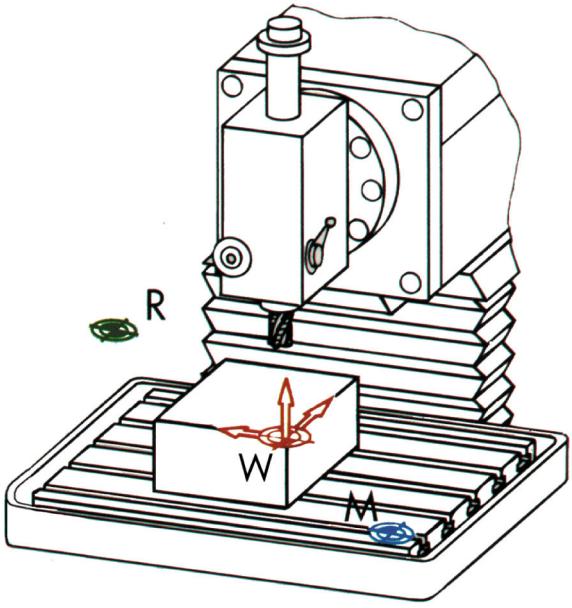


Figure 5-3 The "Working planes" parameter screenform

5.2 Points in the work space

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



Machine zero (M)

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

Workpiece zero (W)

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

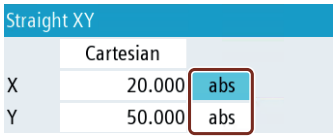
Reference point (R)

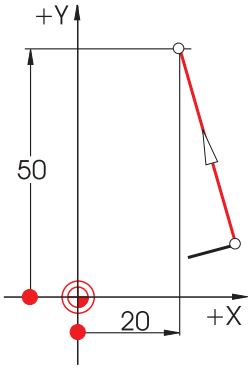
The reference point (R) is approached to set the measuring system to zero, as the machine zero cannot be approached in most cases. Thus, the control system finds the start of counting in the position measuring system.

5.3 Absolute and incremental dimensioning

Absolute input

The entered values refer to the workpiece zero.

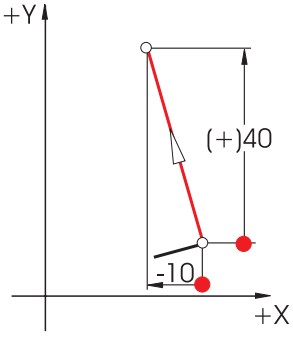




With absolute inputs, the **absolute** coordinate values of the **end point** must always be entered (the starting point is not taken into account).

Incremental input

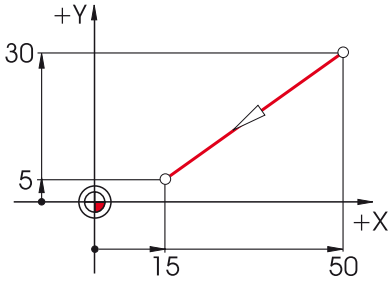
The entered values refer to the starting point.



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

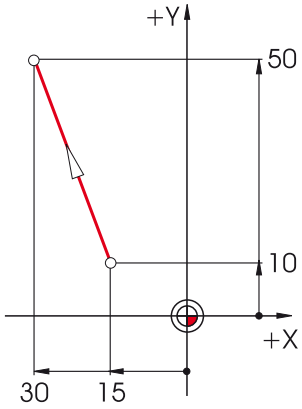
Click on the toggle field to switch between the absolute and incremental input.

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



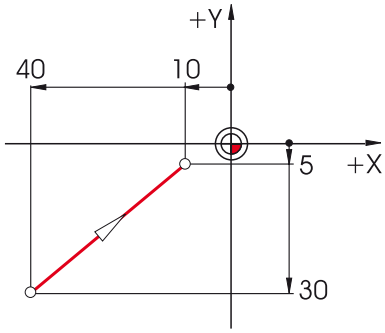
Absolute:
X15 Y5

Incremental:
X-35 Y-25



Absolute:
X-30 Y50

Incremental:
X-15 Y40



Absolute:
X-10 Y-5

Incremental:
X30 Y25

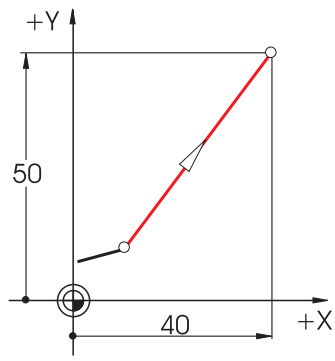
5.4 Linear motions

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

Cartesian

Input of the X and Y coordinates.

Straight XY		
	Cartesian	
X	40.000	abs
X	30.000	inc
Y	50.000	abs
Y	40.000	inc
L	50.000	
α1	53.130	°
α2	38.130	°



Polar

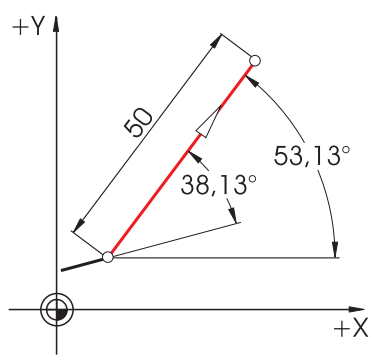
Input of the length and an angle.

Angle 38.13° = angle with reference to the previous element

or

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

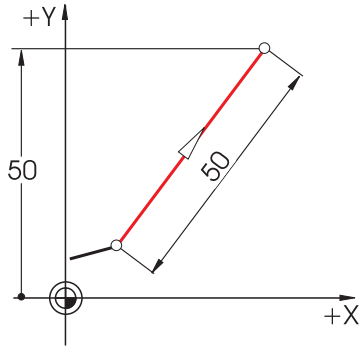
Straight XY		
	Cartesian	
X	40.000	abs
X	30.000	inc
Y	50.000	abs
Y	40.000	inc
L	50.000	
α1	53.130	°
α2	38.130	°



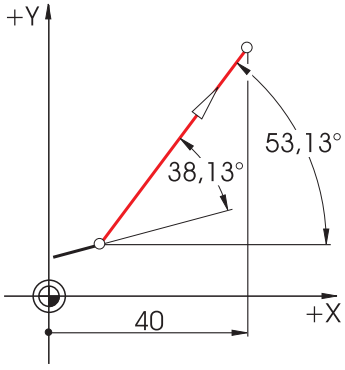
Cartesian and polar

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

- Input of the end point in Y and the length.



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



5.5 Circular motions

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

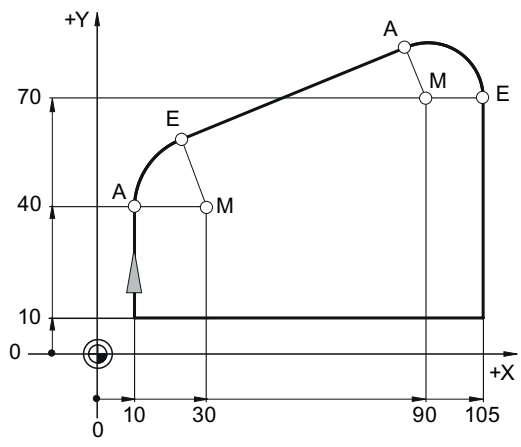
While X and Y are entered as absolute dimensions, the center point is specified with I and J as an incremental dimension in the most control systems. Not only the difference from the starting point **A** to the center **M** must be determined (often even in combination with mathematic calculations),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

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

Input of the center point (absolute)

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

Circle		
Direction of rotation		
R		
	Cartesian	
X		abs
Y		abs
	Cartesian	
I	30.000	abs
J	40	abs
α1		°
α2		°
β1		°
β2		°



Circle		
Direction of rotation		
R		
	Cartesian	
X	105.000	abs
Y	70.000	abs
	Cartesian	
I	90.000	abs
J		abs

After the input:

Circle		
Direction of rotation		
R	20.000	
	Cartesian	
X		abs
Y		abs
	Cartesian	
I	30.000	abs
J	40.000	abs
α1	90.000	°
α2	Tangential	
β1		°
β2		°

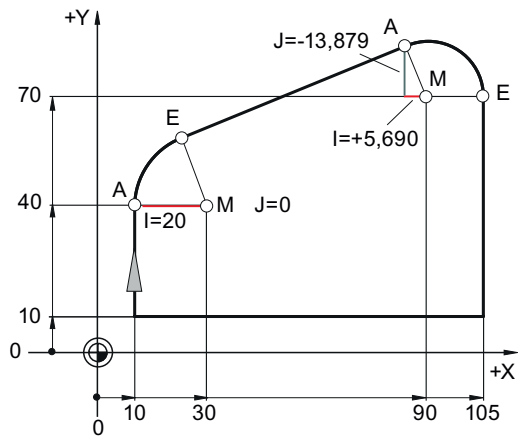
After the input:

Circle		
Direction of rotation		
R	15.000	
	Cartesian	
X	105.000	abs
Y	70.000	abs
	Cartesian	
I	90.000	abs
J	70.000	abs

Display of all parameters

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

Circle		
Direction of rotation		
R	20.000	
	Cartesian	
X	22.414	abs
X	12.414	inc
Y	58.506	abs
Y	18.506	inc
	Cartesian	
I	30.000	abs
I	20.000	inc
J	40.000	abs
J	0.000	inc
α1	90.000	°
α2	Tangential	
β1	22.290	°
β2	67.710	°



Circle		
Direction of rotation		
R	15.000	
	Cartesian	
X	105.000	abs
X	20.690	inc
Y	70.000	abs
Y	-13.879	inc
	Cartesian	
I	90.000	abs
I	5.690	inc
J	70.000	abs
J	-13.879	inc
α1	22.290	°
α2	Tangential	
β1	270.000	°
β2	112.290	°

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

Well equipped

6

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

6.1 Tool management

ShopMill offers three lists for tool management:

- Tool list (Page 52)
- Tool wear list (Page 53)
- Magazine list (Page 54)

6.1.1 Tool list

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

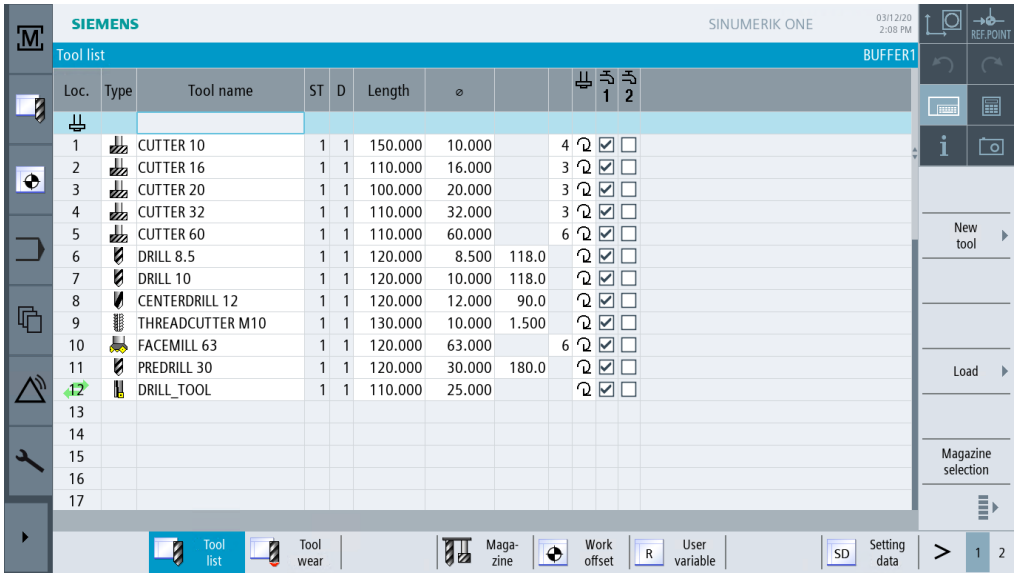


Figure 6-1 Example for tool lists

Meaning of the most important parameters in the tool list:

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

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

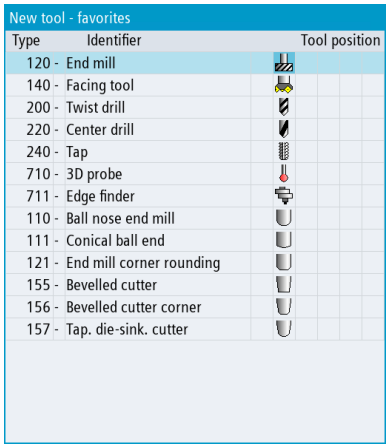


Figure 6-2 Example of Favorites list

6.1.2 Tool wear list

The wearing data for the appropriate tools are defined here.

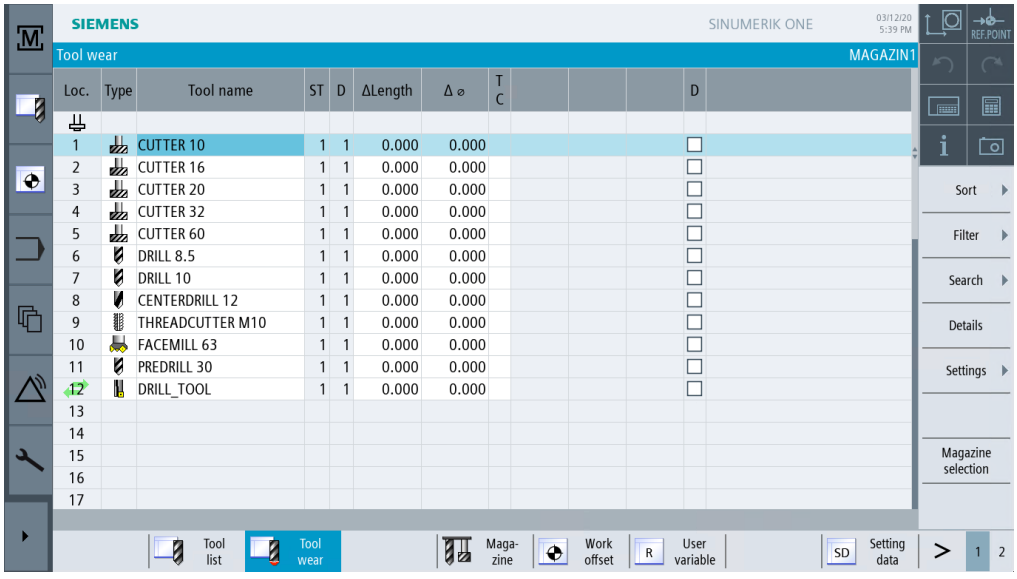


Figure 6-3 Tool wear list

The most important tool wearing parameters are:

Δ Length	Length wear
Δ Radius	Radius wear

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

6.1.3 Magazine list

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

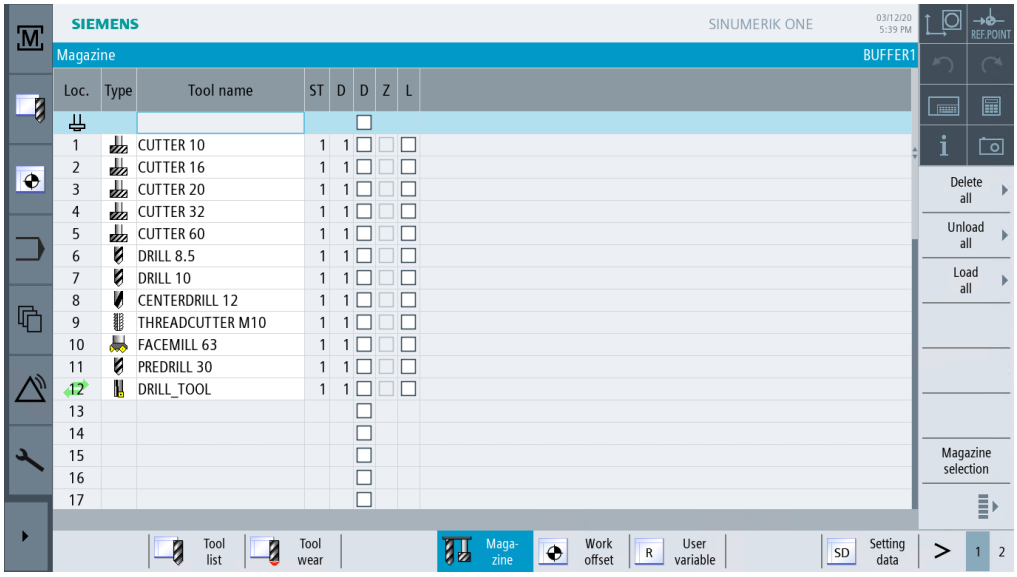


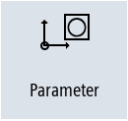
Figure 6-4 Magazine list

Meanings of the most important parameters:

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

6.2 Tools used

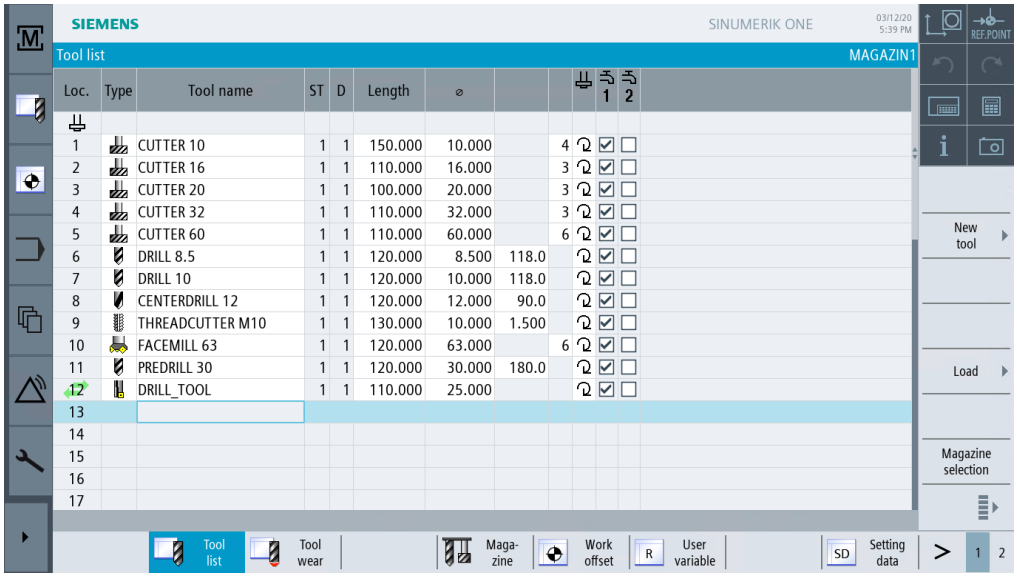
In this section you will learn how to enter tools required for the later machining in the tool list.



Select the "Parameters" area in the main menu.

Select the "Tool list" softkey.

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



Select the "New tool" softkey.

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

Note

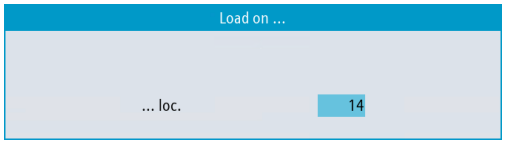
The milling cutters with the diameters 6, 10, 20 and 32 (Cutter6, 10, 20 and 32) must immerse, as they will also be used for the milling of pockets in the following examples.

6.3 Tools in the magazine

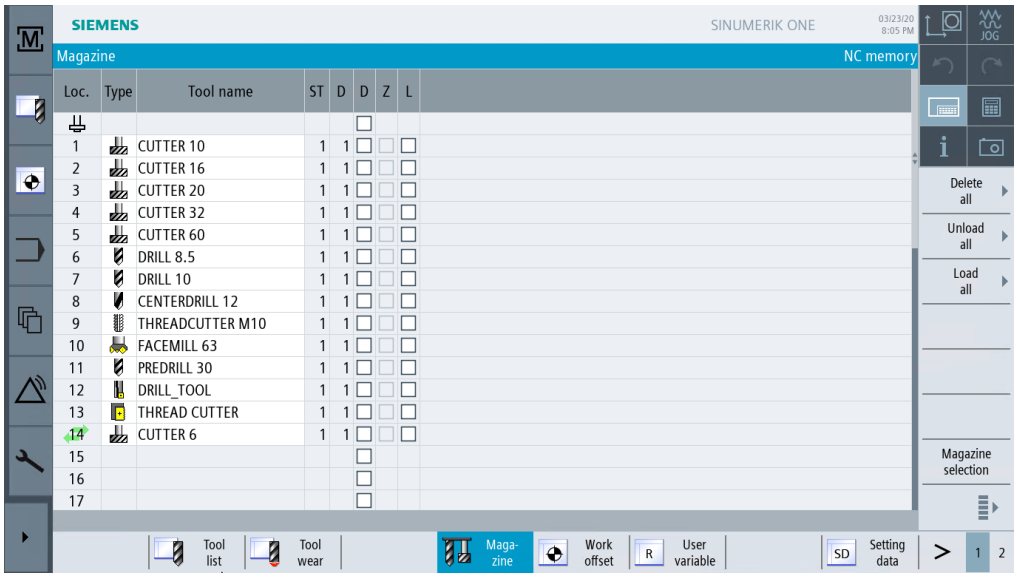
In the following you will learn how to insert the tools into the magazine.

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

The following dialog offers the first free magazine location for you to change or accept directly.



The magazine for the following exercises could look like the one in the screen below:



6.4 Gauging tools

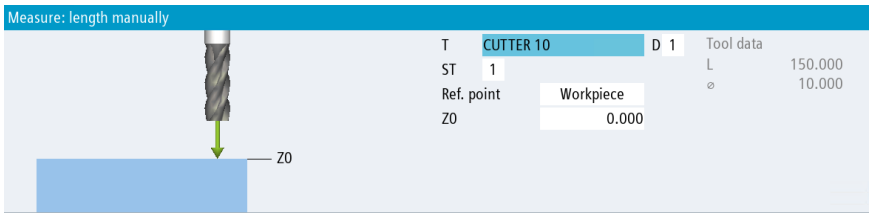
In the following you will learn how to calculate tools.

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

Then switch to the "Gauge tool" menu.

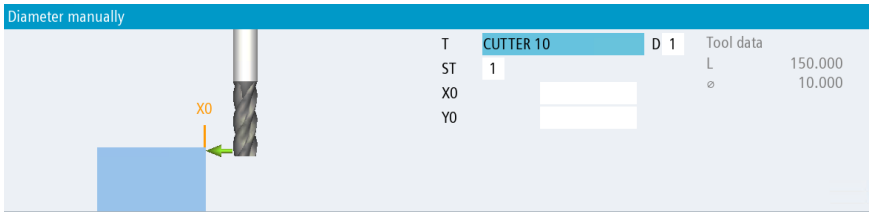
Length manually

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



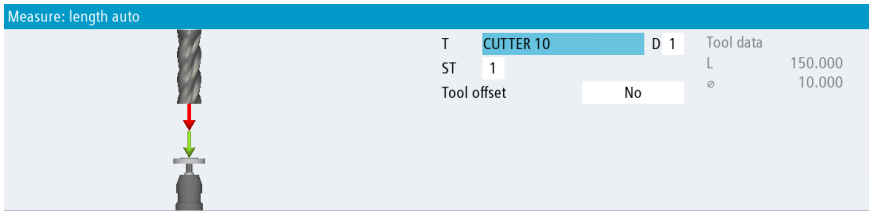
Radius manually

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



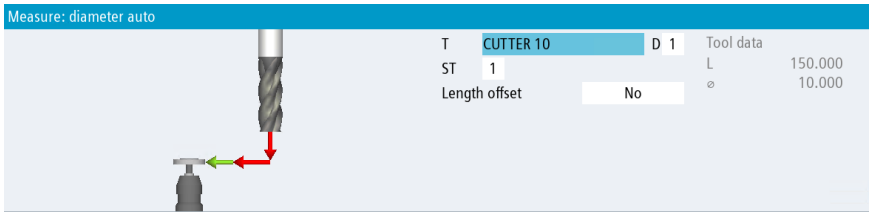
Length auto

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



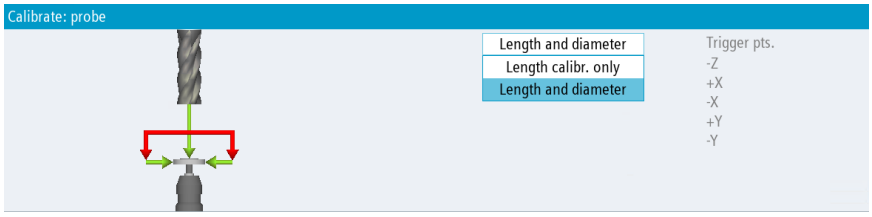
Radius auto

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



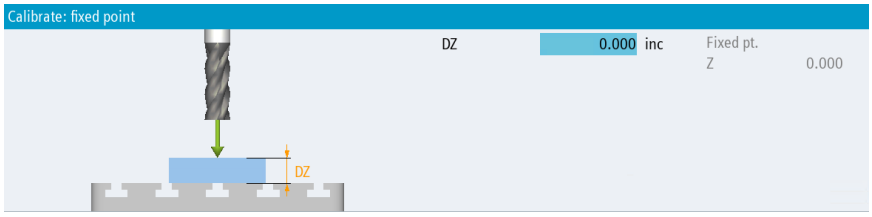
Calibrate probe

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



Calibrate fixed pt.

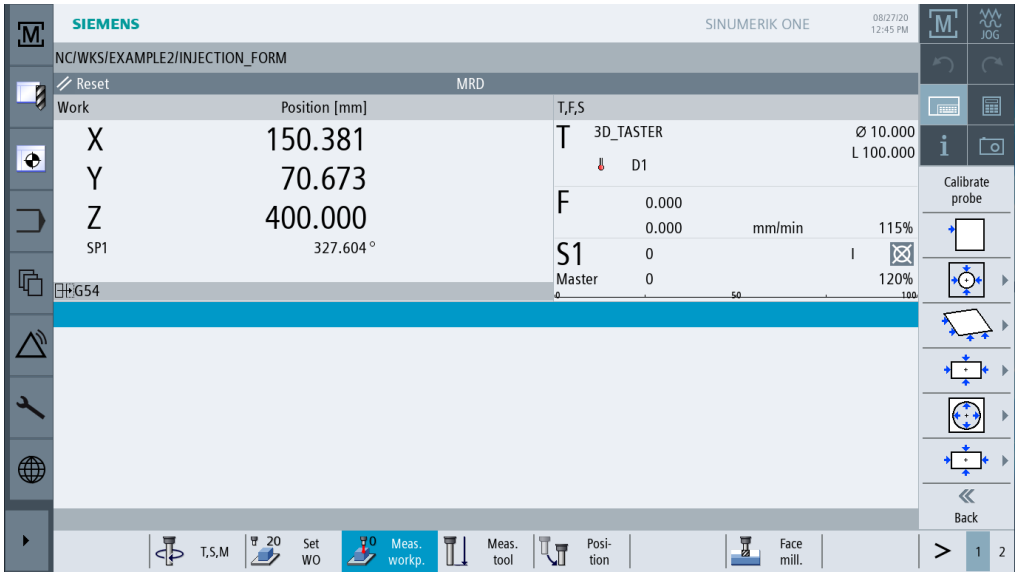
The **Fixed-point calibration** function is used to determine the fixed point used as the reference point for measuring the tool length manually.



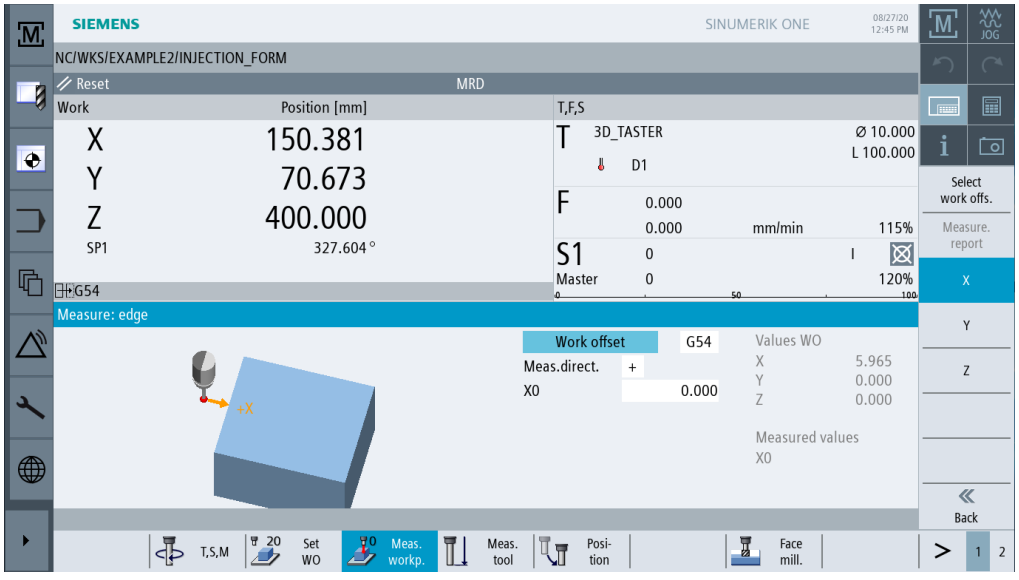
6.5 Setting the workpiece zero

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

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



The zero point of a workpiece edge will be set in the following example using a 3D probe.



Step 1: Selecting the edge

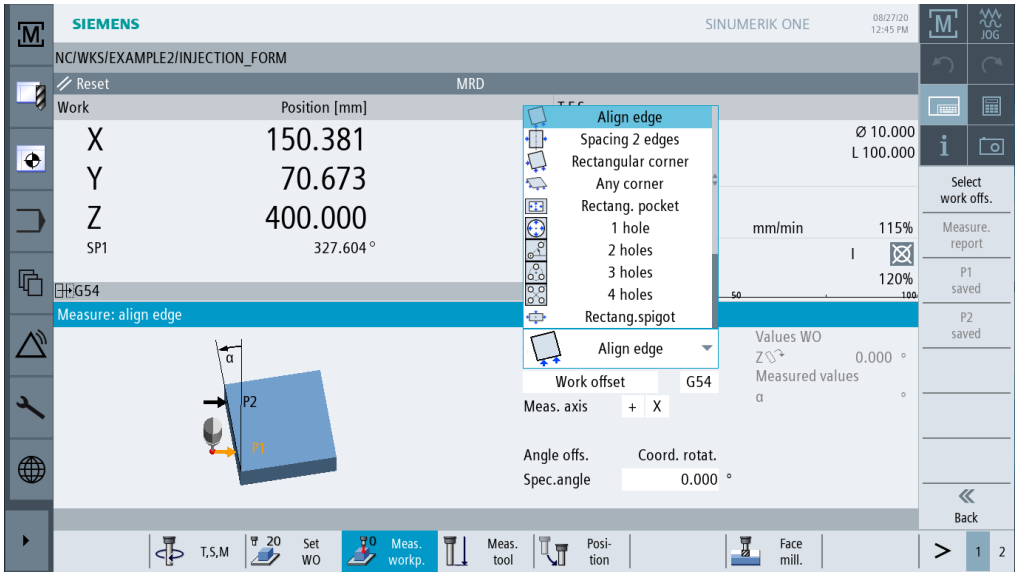
Define the sampling direction for the probe: LH (+) or (-). The X0 parameter can be used to specify an offset for the workpiece zero if the zero is not to lie on the edge of the workpiece.

Step 2: Sampling the workpiece edge

Set WO

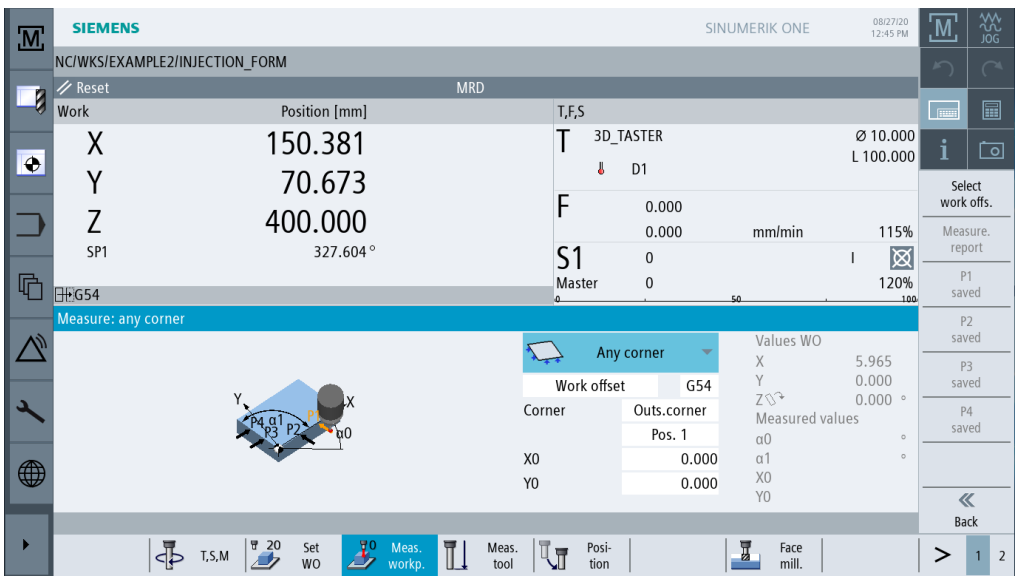
Step 3: Set the workpiece zero taking into account the edge probe diameter (5 mm). Now this process of calculation must be repeated for Y using the edge probe and for Z (in most cases, with the milling cutter).

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



Example 1: Any corner

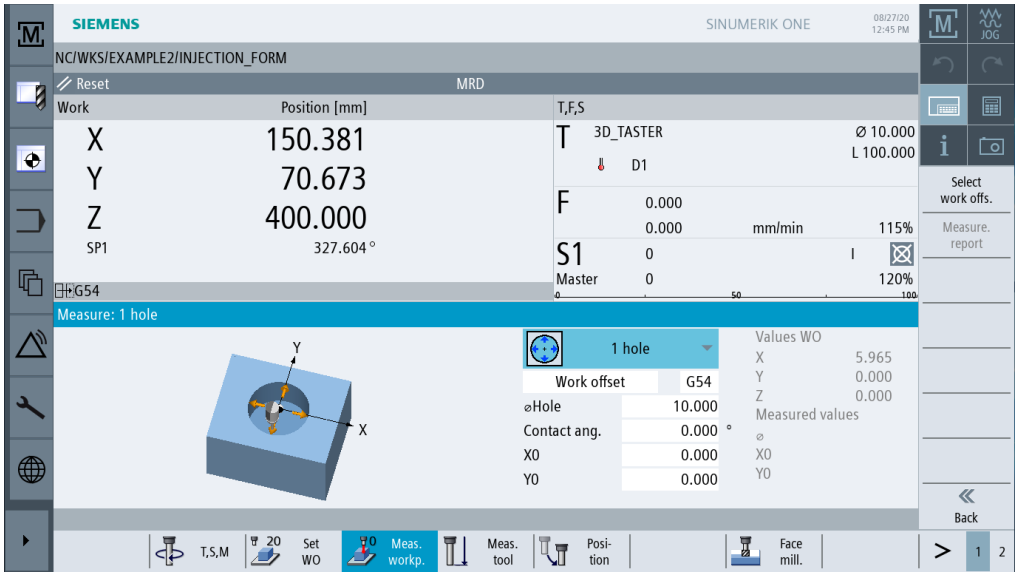
If the workpiece is positioned as shown here, the position/corner of the workpiece can be determined by approaching four points.



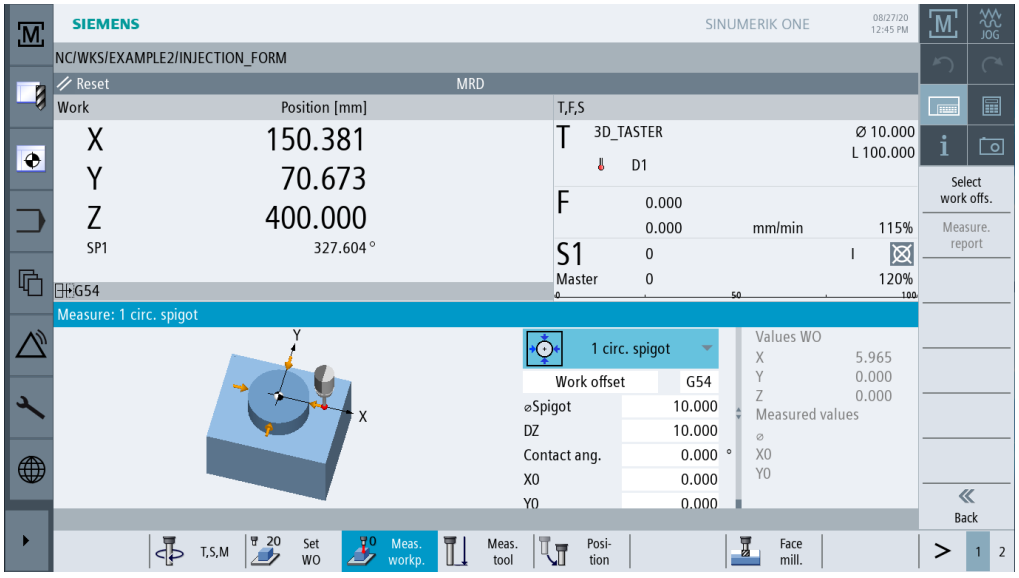
3D probes are offered in both the electronic and mechanical variants. The signals issued by electronic probes can be processed by the control system directly.

3D probe, electronic	3D probe, mechanical
	

Example 2: Calculating a drill hole



Example 3: Calculating a circular spigot



Calibrate probe

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

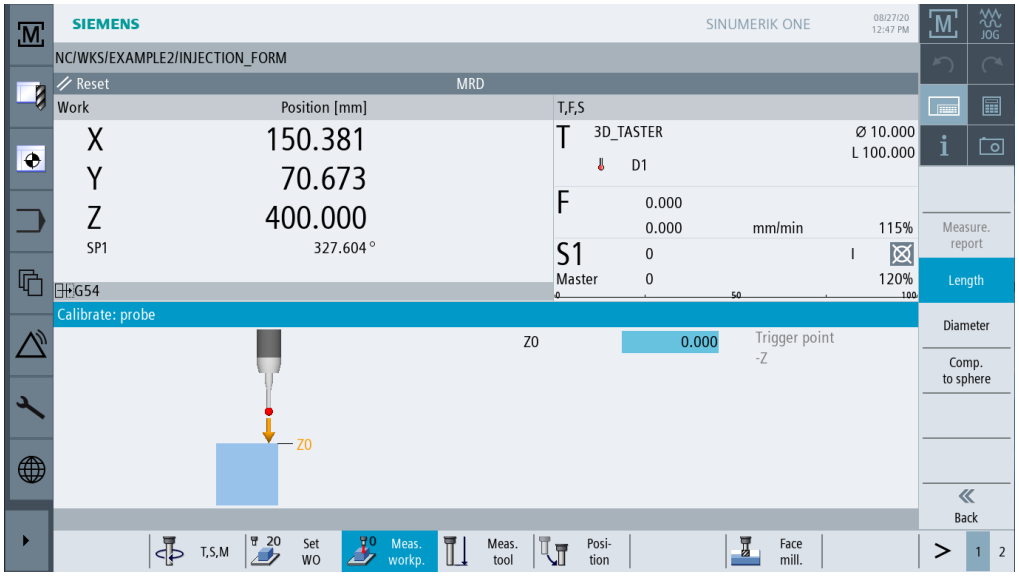


Figure 6-5 Calibrating the probe for the length

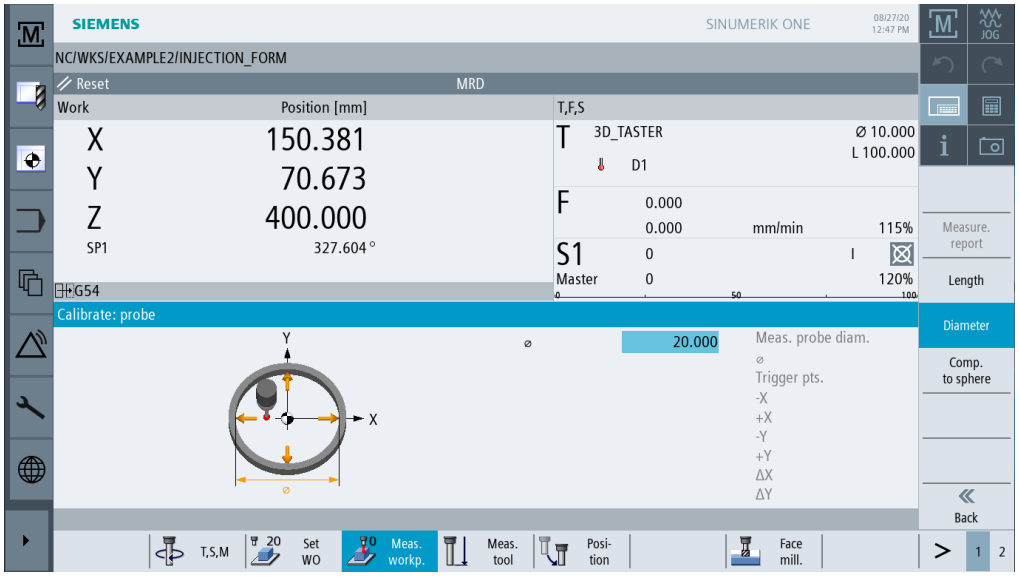


Figure 6-6 Calibrating the probe for the radius

Example 1: Longitudinal guide

7

7.1 Overview

Learning objectives

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

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

Task

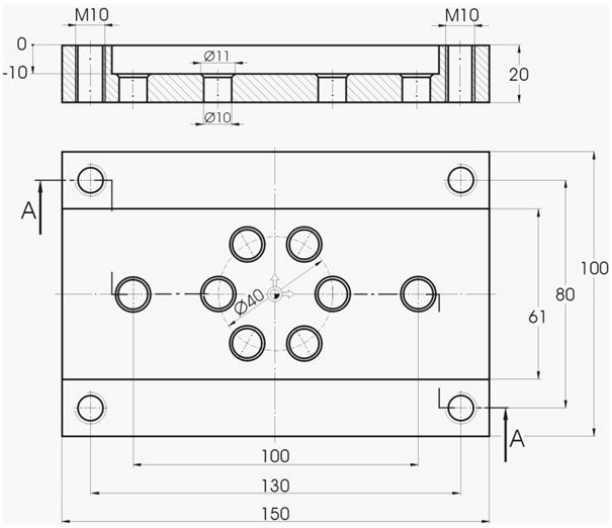


Figure 7-1 Workshop drawing - Example 1:

Figure 7-2 Workpiece - Example 1:

Note

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

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

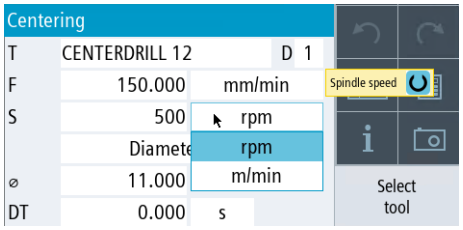


Figure 7-3 Toggle field with help text

7.2 Program management / creating programs

Operating sequences

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

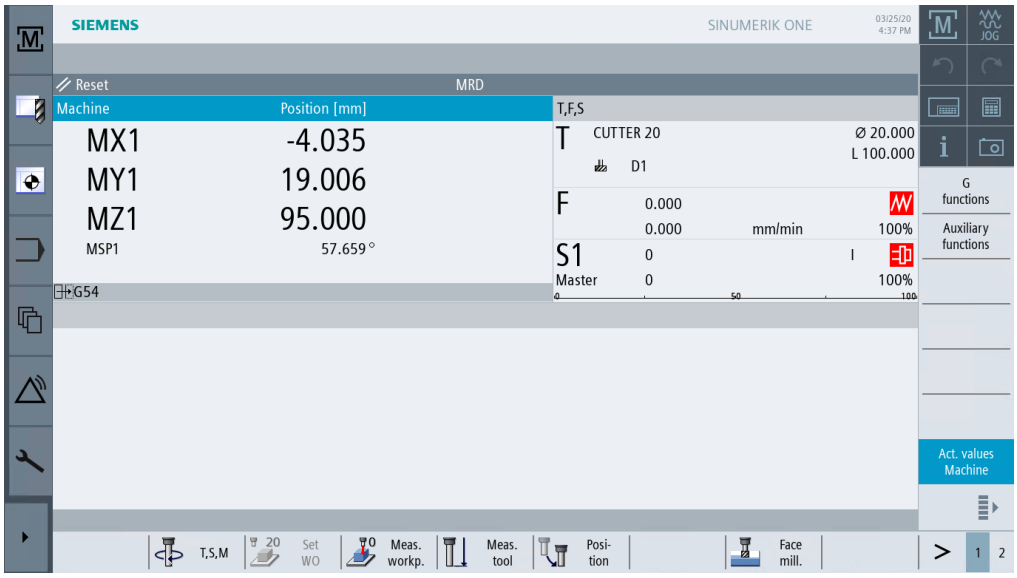


Figure 7-4 Main screen

Open the basic menu. In the main menu, you may call various areas of ShopMill.

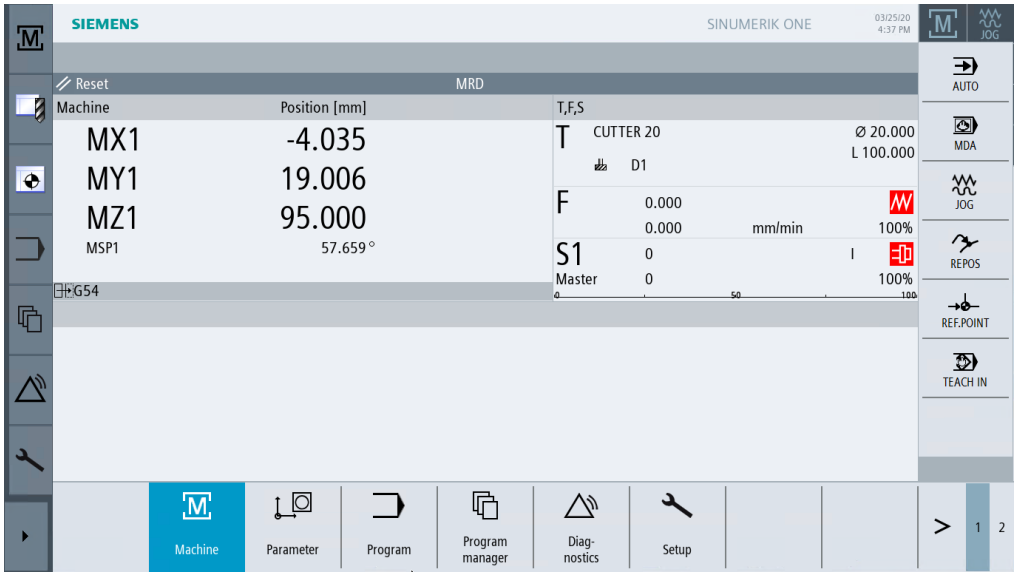


Figure 7-5 Main menu

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

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

Use the cursor key or right-click to select the 'Workpieces' directory.

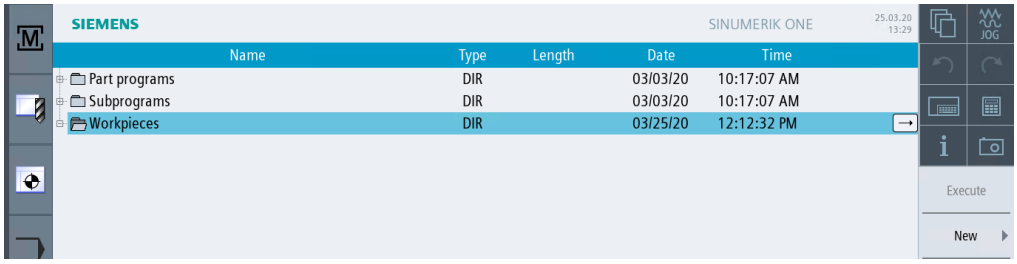


Figure 7-6 Program Manager

Open the "Workpieces" directory.

Enter the name 'EXAMPLE1' for the workpiece.

Figure 7-7 Creating a workpiece

Confirm your input. The following dialog box is opened:

Figure 7-8 Creating a step sequence program

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

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

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

Press "Accept" to apply your input.

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

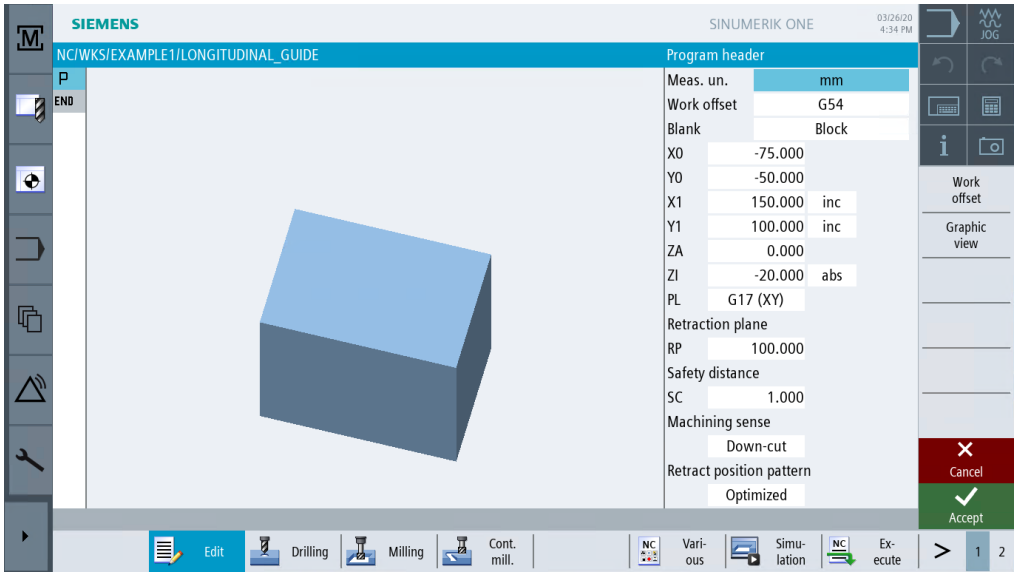


Figure 7-9 Program header - Help display

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

Enter the following values:

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

Press "Accept" to apply the values entered. After confirming, the program header is displayed.

Figure 7-10 Program header, example 1 - Work step editor

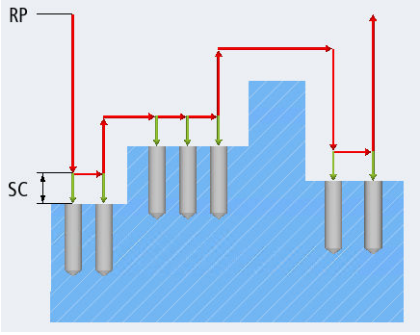
Now the program has been created as the basis for further machining steps. It has a name, a program header (pictogram "P") and a program end (pictogram "END"). The individual machining steps and contours are stored in the program one beneath the other. The later machining is performed from top to bottom.

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

Retraction position pattern

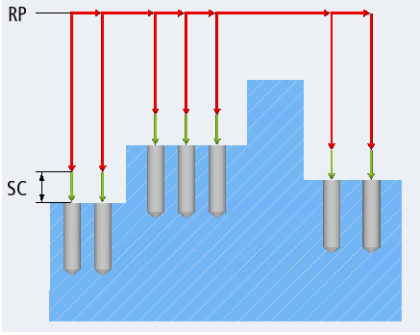
The position pattern can be set to "Optimized" (= time-optimized traversing distances) or "Retraction plane".

Optimized retraction



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

To retraction plane (standard)



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

Softkeys

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

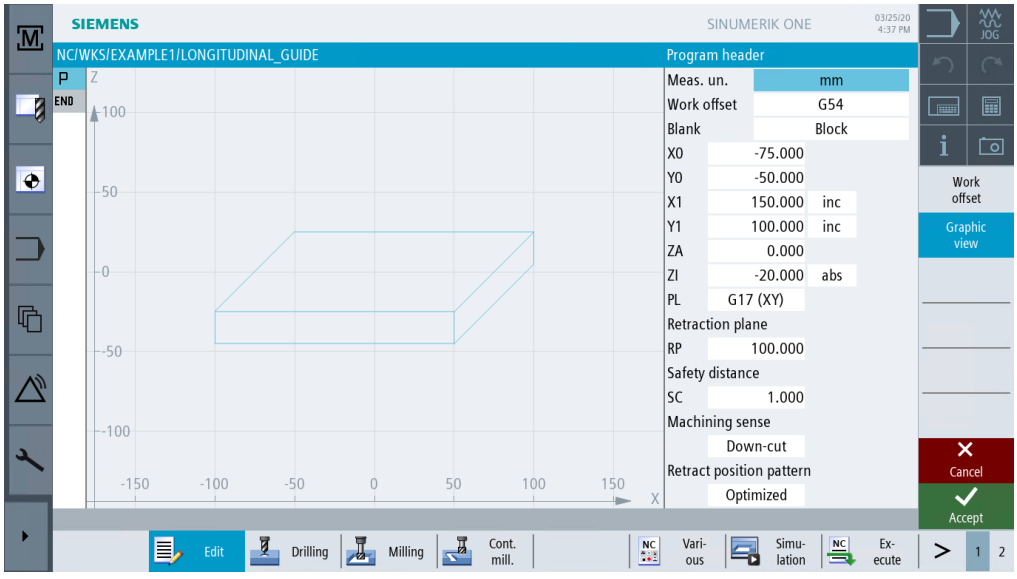


Figure 7-11 Program header - graphical view

Use this softkey to switch back to the help display.

7.3 Calling a tool and specifying cutter radius compensation

Operating sequences

To call the required tool, proceed as follows:

Use this key to extend the horizontal softkey menu.

Select the **Straight line Circle** softkey.

Select the **Tool** softkey.

Open the tool list. Select the CUTTER 60 tool.

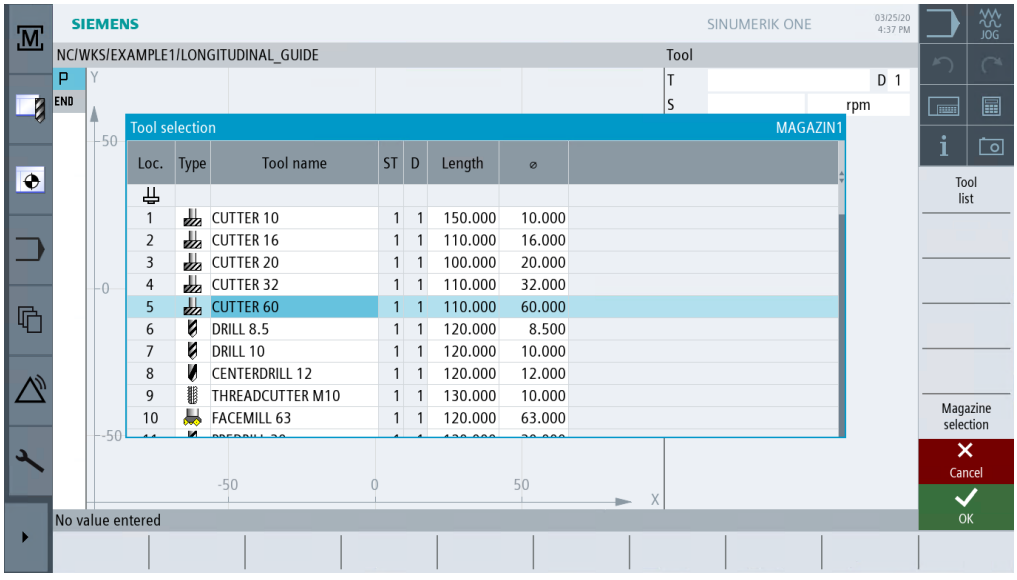


Figure 7-12 Tool list

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

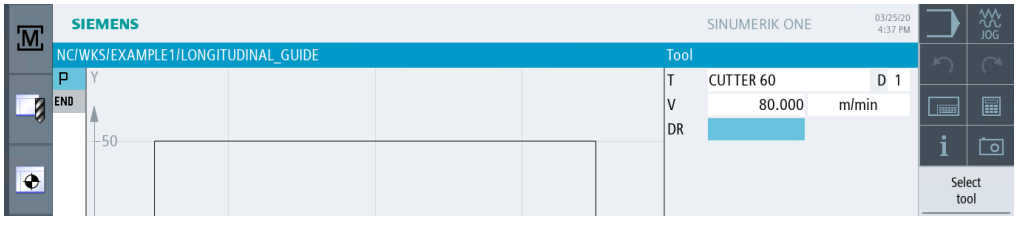


Figure 7-13 Tool cutting rate

Press "Accept" to apply the value entered.

7.4 Specifying the distance to be traversed

Operating sequences

Now enter the distances to be traversed:

Select the "Straight line" softkey.

Select the "Rapid traverse" softkey.

Enter the following values in the interactive screenform:

Field	Value	Toggle field	Notes
X	110 abs	X	
Y	0 abs	X	
Radius compensation	off	X	See below *Radius compensation*

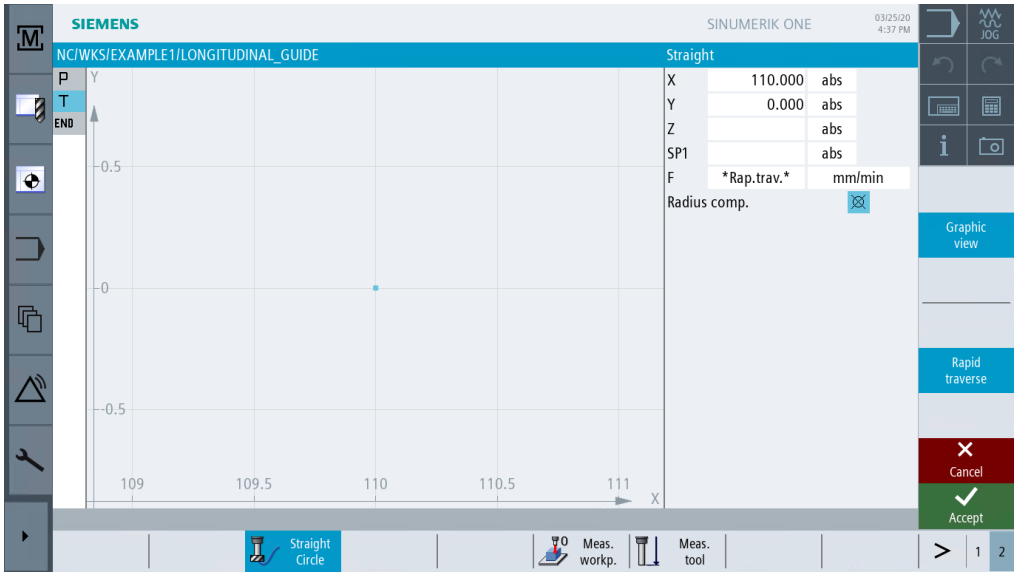


Figure 7-14 Specifying the distance to be traversed - Radius compensation

Press "Accept" to apply the values entered.

Select the "Straight line" softkey.

Select the "Rapid traverse" softkey.

Enter the following values in the interactive screenform:

Field	Value	Toggle field	Notes
Z	-10 abs	X	
Radius compensation	Empty field	X	See below *Radius compensation*

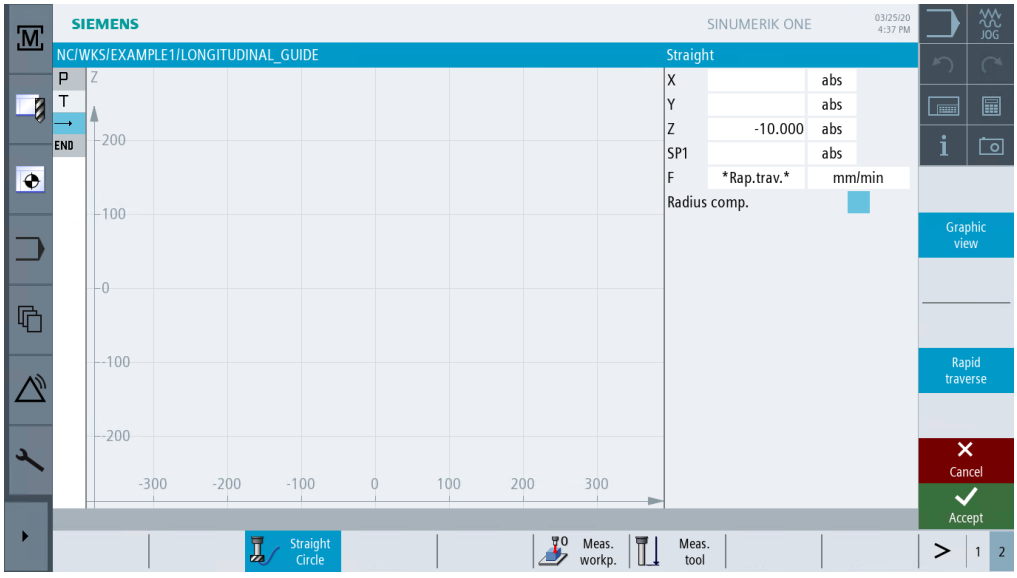


Figure 7-15 Specifying the distance to be traversed - Tool positioned in Z

Press "Accept" to apply the values entered.

Select the "Straight line" softkey.

Enter the following values in the interactive screenform:

Field	Value	Toggle field	Notes
X	- 110 abs	X	
F	400 mm/min	X	
Radius compensation	Empty field	X	See below *Radius compensation*

Figure 7-16 Specifying the distance to be traversed - First machining path

Press "Accept" to apply the values entered. After acceptance, the list of work steps looks like this:

T	T=CUTTER 60 V=80m
→	RAPID G40 X110 Y0
→	RAPID Z-10
→	F400/min X-110

Figure 7-17 Specifying the distance to be traversed - List of work steps

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

Load the next tool CUTTER 16. After accepting the tool, specify the cutting rate 100 m/min.

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

T	T=CUTTER 16 V=100m
→	RAPID X85 Y22.5
→	RAPID Z-10
→	F200/min X-85
→	RAPID Y-22.5
→	F200/min X85

Figure 7-18 Specifying the distance to be traversed - Work step list

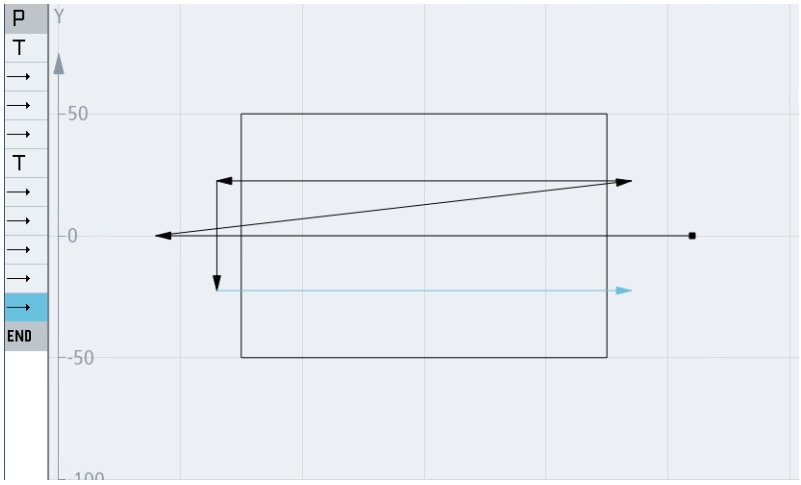


Figure 7-19 Specifying the distance to be traversed - Complete

Start simulation.

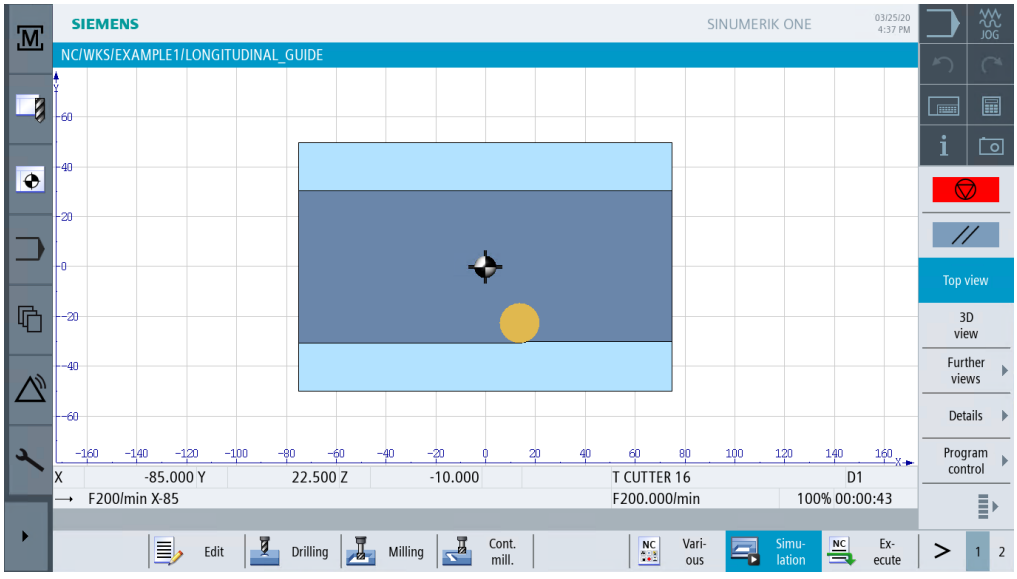
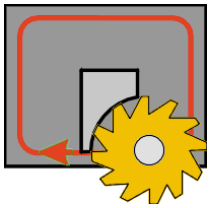
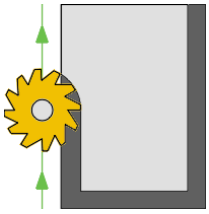
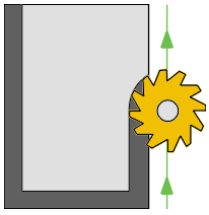


Figure 7-20 Simulation of the distance to be traversed

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

Radius compensation

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

7.5 Creating drill holes and position repetitions

Operating sequences

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

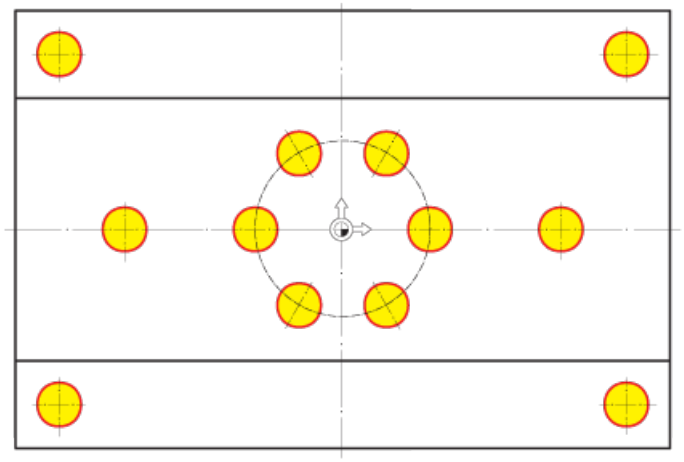


Figure 7-21 Drilling positions

Select the **Drilling** softkey.

Select the **Center** softkey.

Open the tool list. Use the cursor key to select the CENTERDRILL 12 tool.

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

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

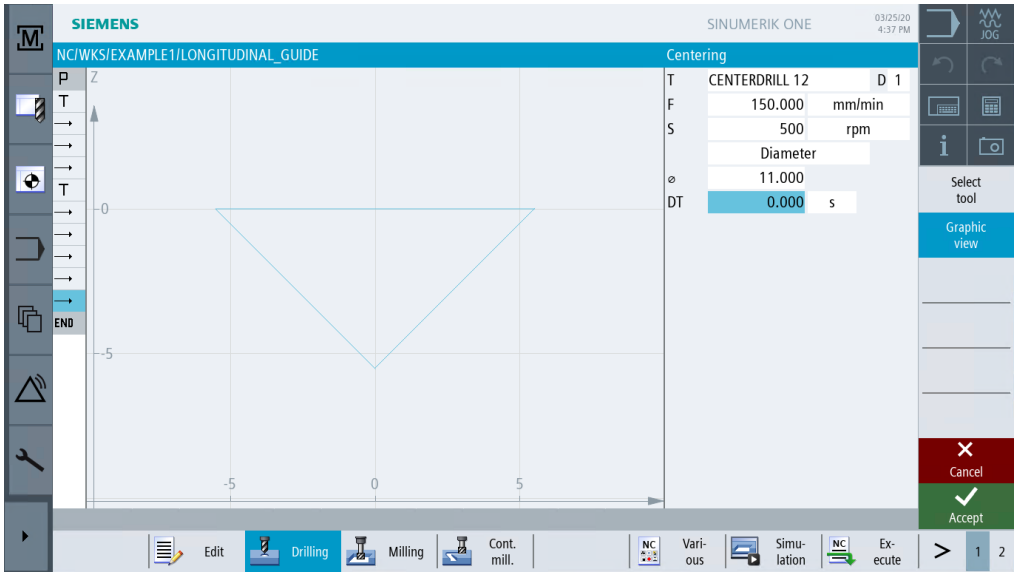


Figure 7-22 Centering

Press "Accept" to apply the values entered.

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

Select the **Positions** softkey.

Select the **Positions** softkey for the position pattern.

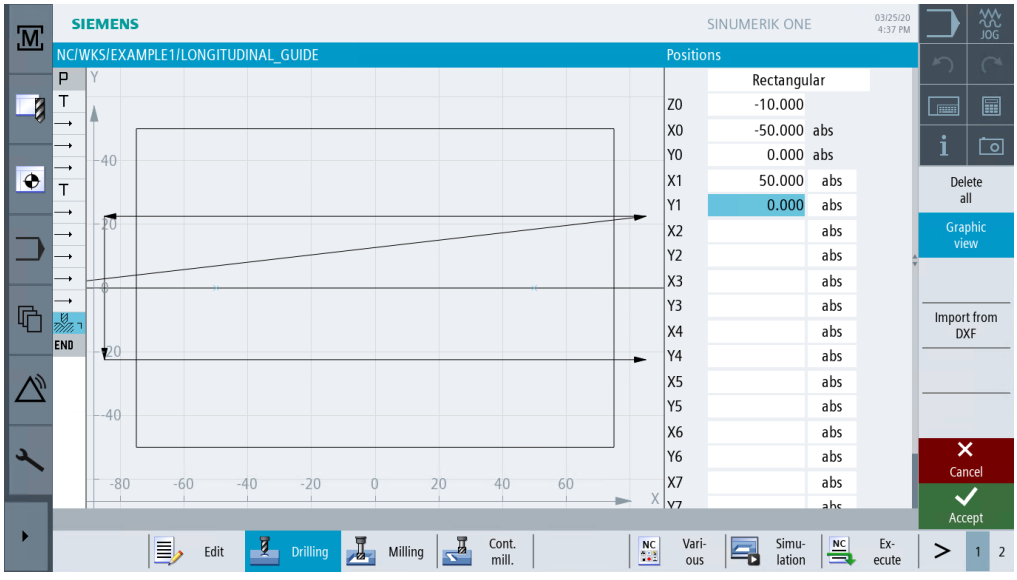


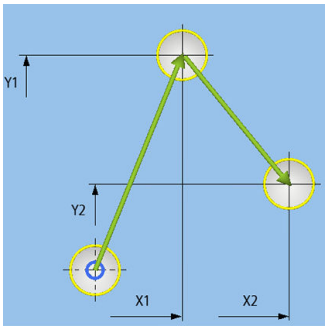
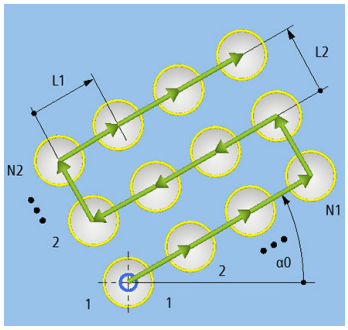
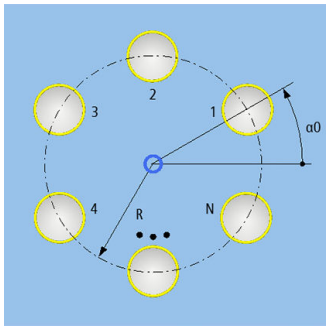
Figure 7-23 Positions - Individual drill holes

Enter the following values for the two individual drill holes:

Field	Value	Toggle field	Notes
Coordinate system	Rectangular	X	
Z0	-10		The starting depth is -10mm.
X0	-50		
Y0	0		
X1	50 abs	X	
Y1	0 abs	X	

Note

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

Positions	Position grid	Position circle
		

Help displays - Positions

Press "Accept" to apply the values entered.

Select the **Positions** softkey.

Select the **Position circle** softkey.

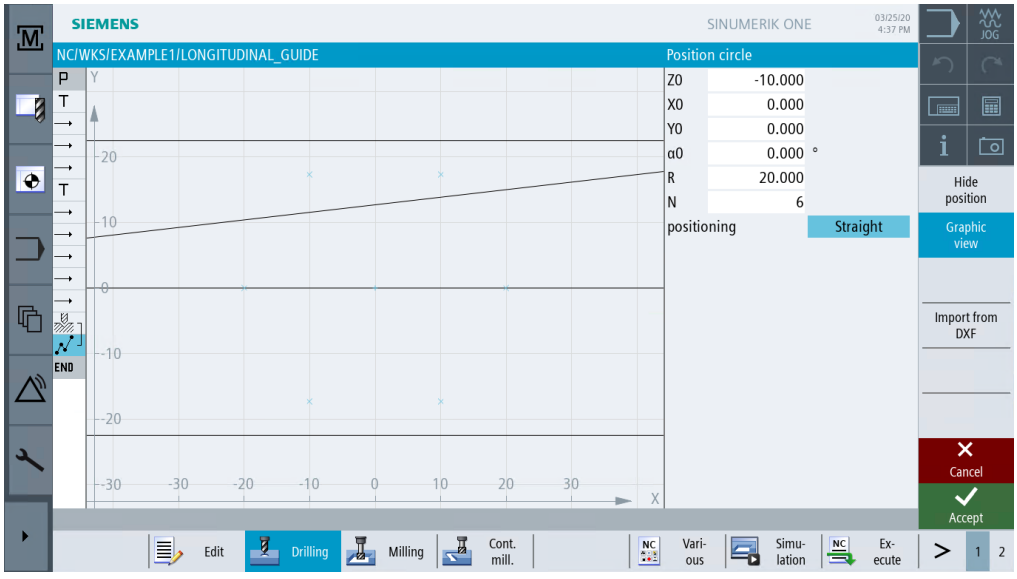
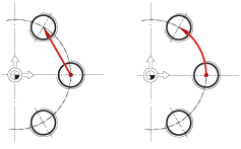


Figure 7-24 Position circle

Enter the following values:

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

Press "Accept" to apply the values entered.

Select the **Positions** softkey.

Select the **position grid** softkey.

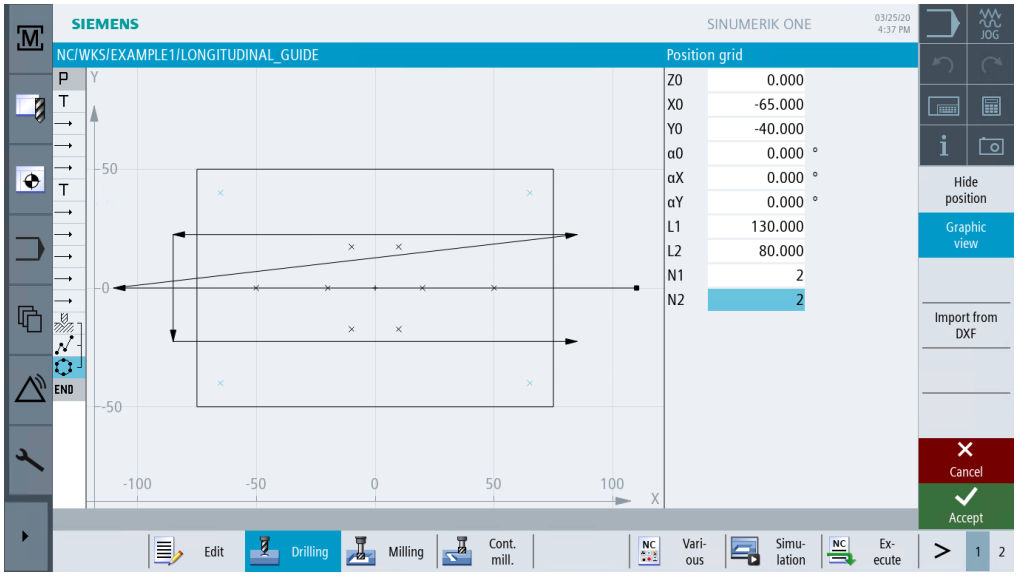


Figure 7-25 Position grid

Enter the following values:

Field	Value	Toggle field	Notes
Z0	0		
X0	-65		
Y0	-40		
α0	0		
L1	130		
L2	80		
N1	2		
N2	2		

Press "Accept" to apply the values entered.

Select the **Drilling Reaming** softkey.

Open the tool list. Use the cursor key to select the DRILL 8.5 tool.

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

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

Note

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

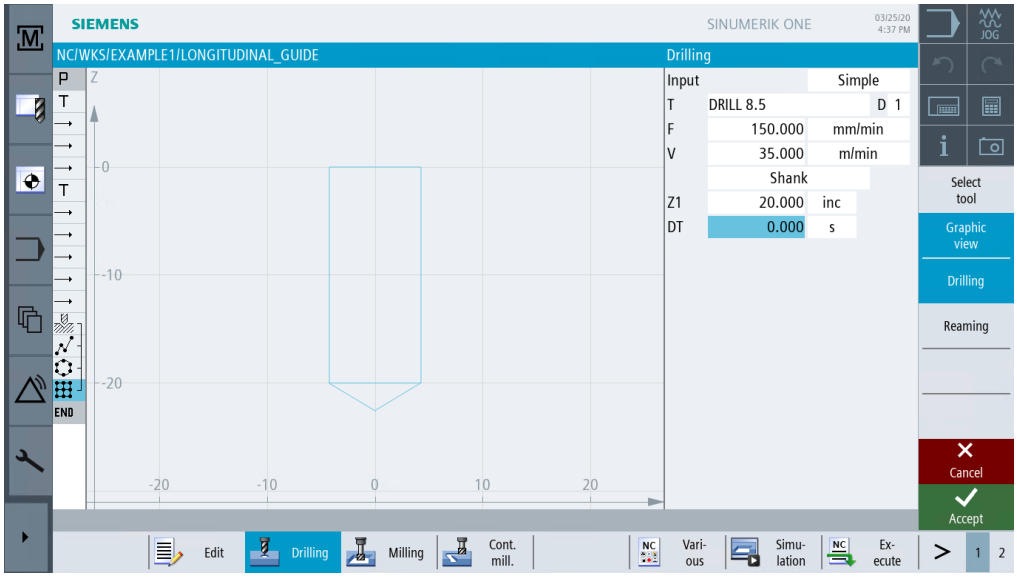


Figure 7-26 Drilling

Press "Accept" to apply the values entered.

Select the **Thread** softkey.

Select the **Tapping** softkey.

Select tool

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

OK

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

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

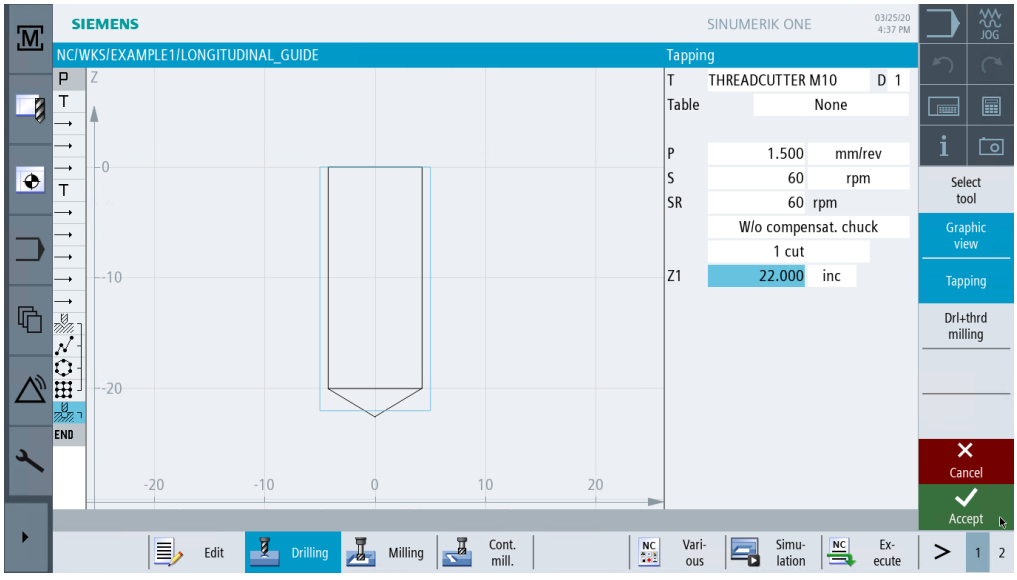


Figure 7-27 Thread

Accept

Press "Accept" to apply the values entered.

Position repetit.

Select the **Repeat position** softkey.

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

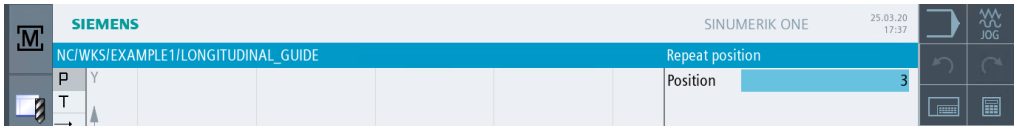


Figure 7-28 Repeating a position

Press "Accept" to apply the values entered. After accepting the values, you will see the linking of the work steps in the work step editor.

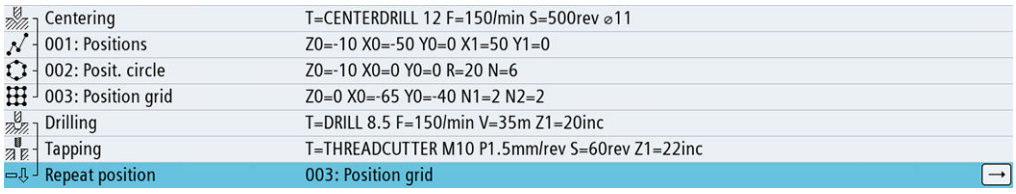


Figure 7-29 Linking of work steps

Select the **Drilling Reaming** softkey.

Open the tool list. Use the cursor key to select the DRILL 10 tool.

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

Field	Value	Toggle field	Notes
F	150 mm/min	X	
V	35 m/min	X	
Shank/tip	Shank	X	
Z1	20 inc	X	
DT	0	X	

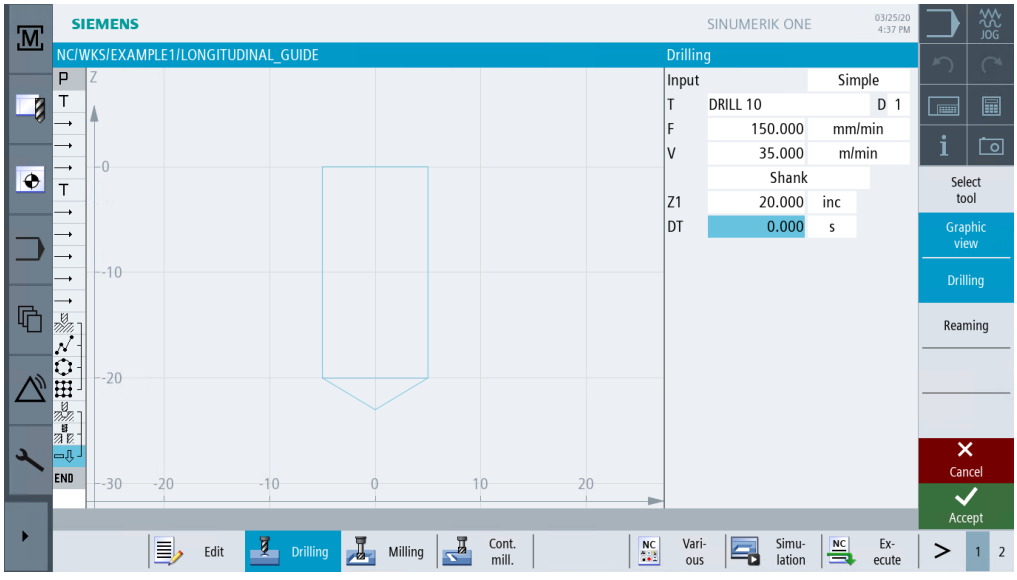


Figure 7-30 10mm drill holes

Press "Accept" to apply the values entered.

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

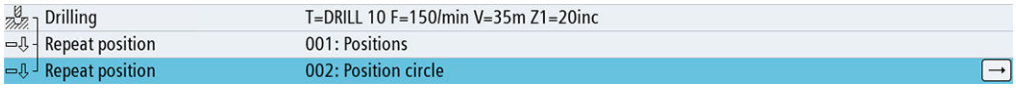


Figure 7-31 Repeat the positions 001 and 002 in the work step editor.

Call the simulation for checking.

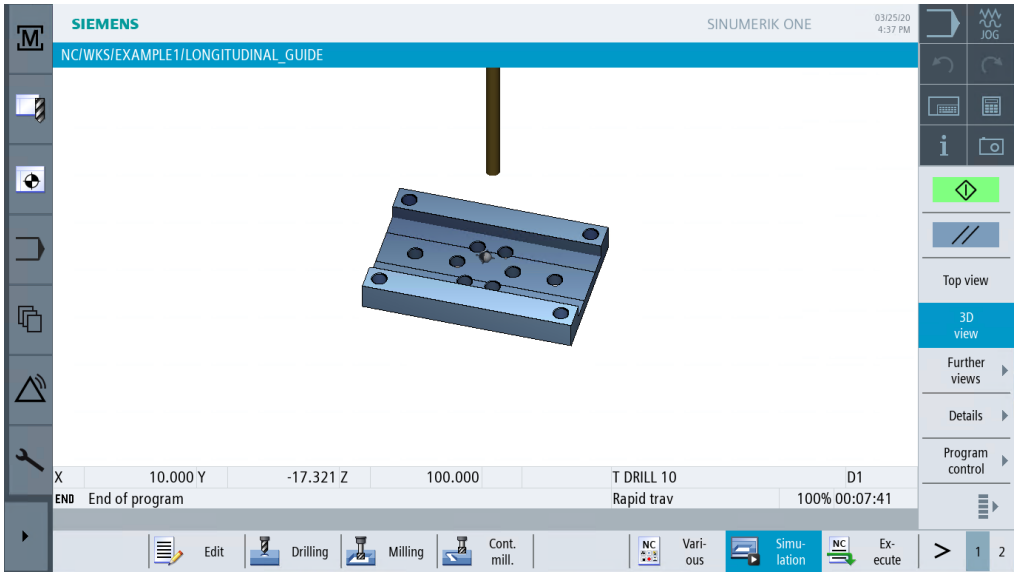


Figure 7-32 3D simulation

Example 2: Injection mold

8

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

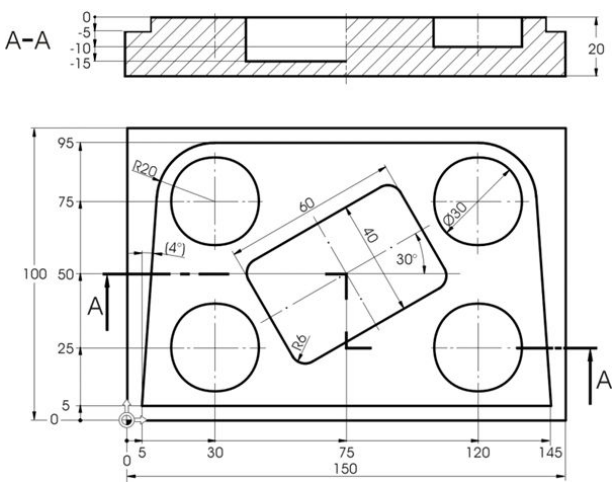


Figure 8-1 Workshop drawing - Example 2:

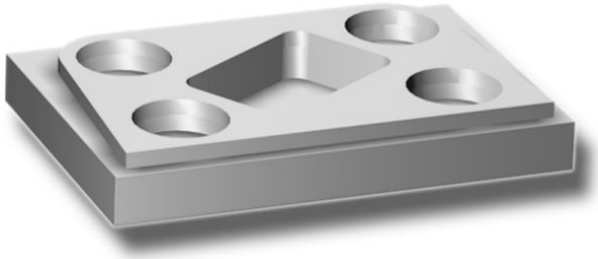


Figure 8-2 Workpiece - Example 2:

Preparation

Perform the following steps without help:

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

Note

Observe the new zero position.

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

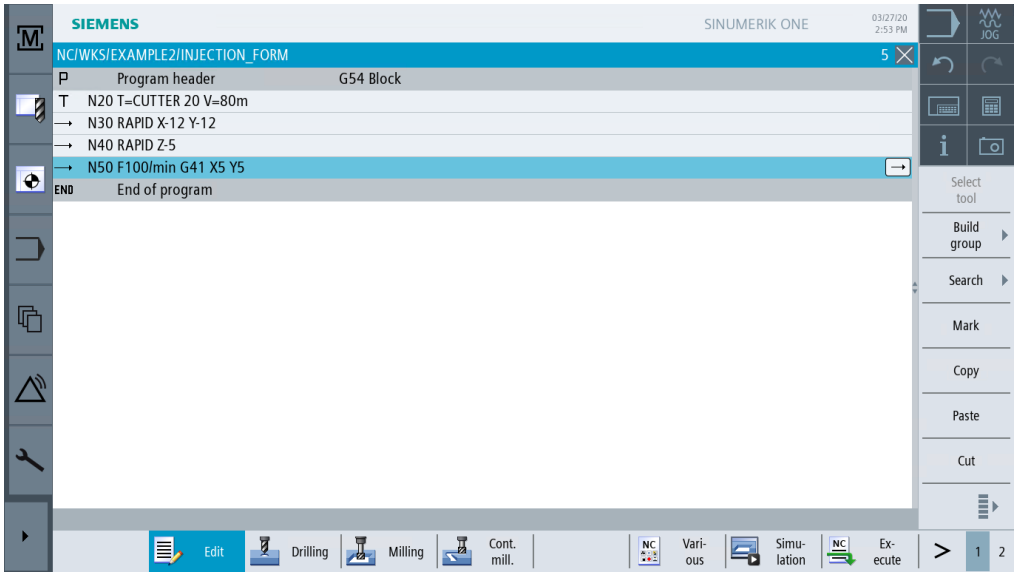


Figure 8-3 Machining step program

8.2 Straight lines and circular paths by way of polar coordinates

Operating sequences

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

Note

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

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

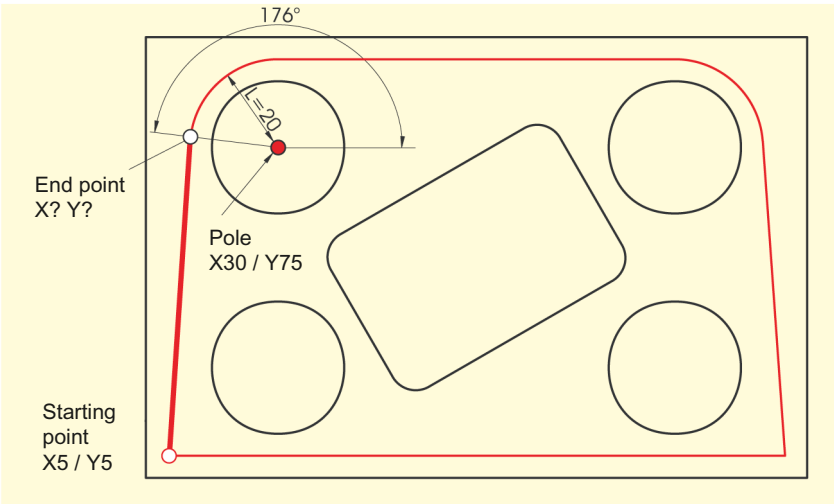


Figure 8-4 Determination of end point and polar angle

Proceed as follows to enter the contour:

Select the **Polar** softkey.

Select the **Pole** softkey.

Enter the following values in the interactive screenform:

Field	Value	Toggle field	Notes
X	30 abs	X	
Y	75 abs	X	

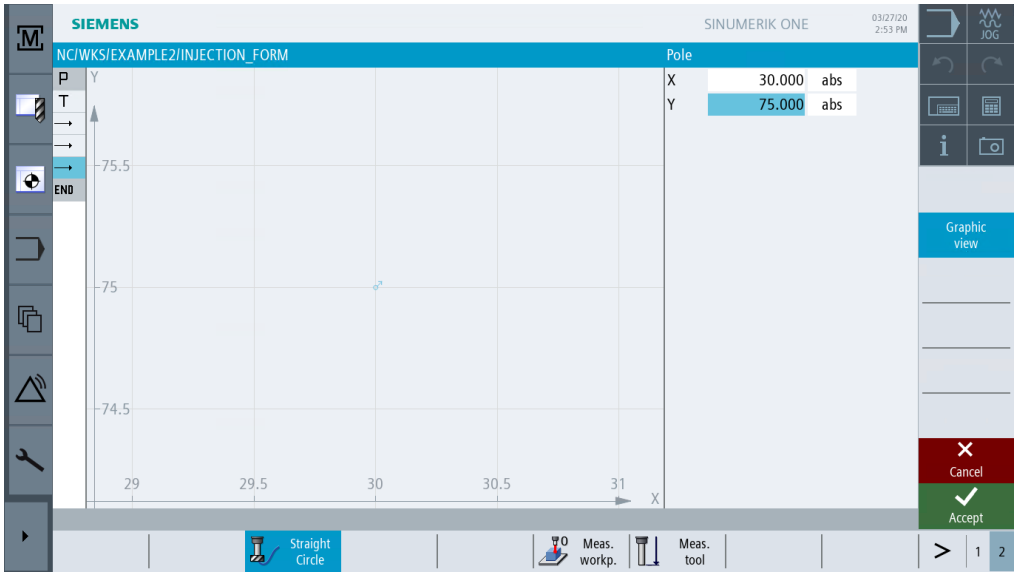


Figure 8-5 Specifying the pole

Press "Accept" to apply the values entered.

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

Enter the following values in the interactive screenform:

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

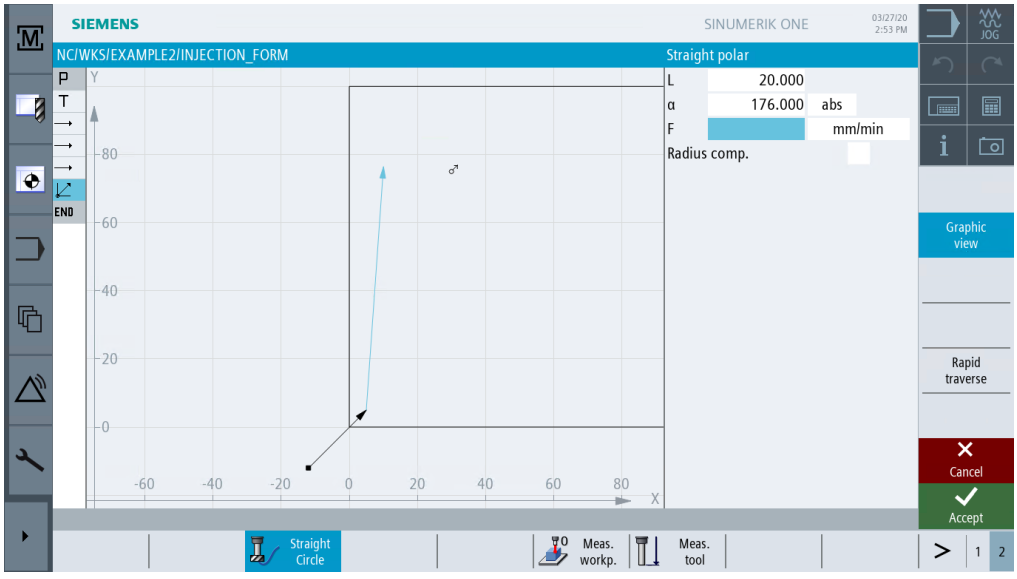


Figure 8-6 Specifying the straight line using polar coordinates

Press "Accept" to apply the values entered.

Select the **Circle / polar** softkey.

A circular path can also be specified using polar coordinates.

Enter the following value in the interactive screenform:

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

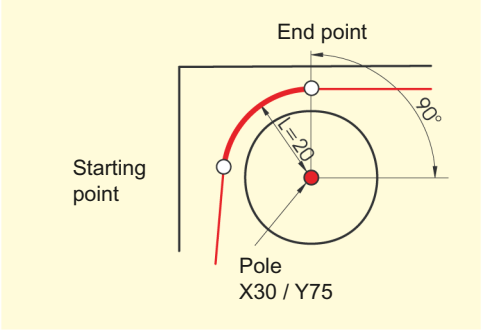


Figure 8-7 Pole starting / end points

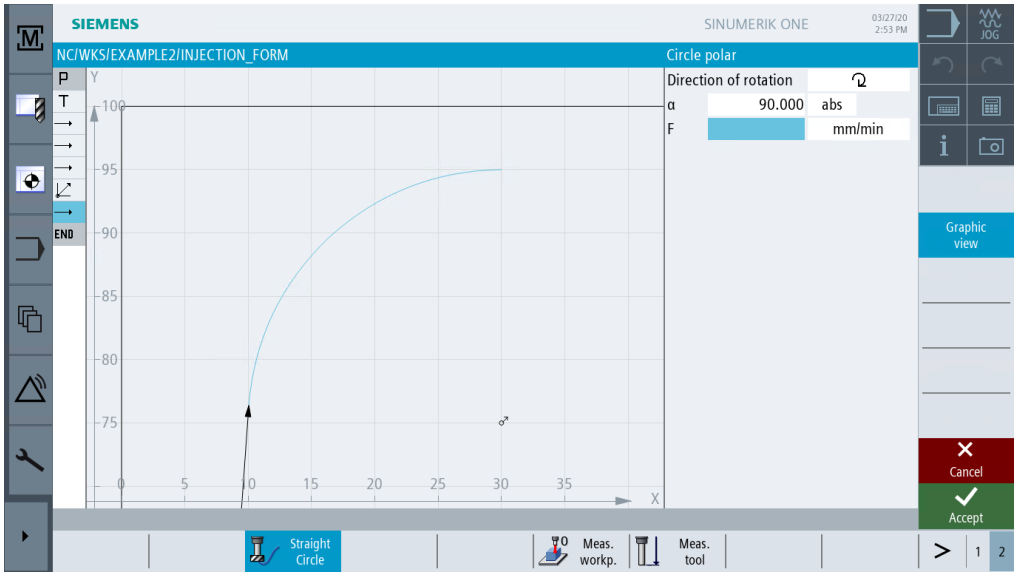


Figure 8-8 Specifying the circular path

Press "Accept" to apply the values entered.

Select the **Back** softkey.

Select the **Straight line** softkey.

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

Enter the following value in the interactive screenform:

Field	Value	Toggle field	Notes
X	120 abs	X	

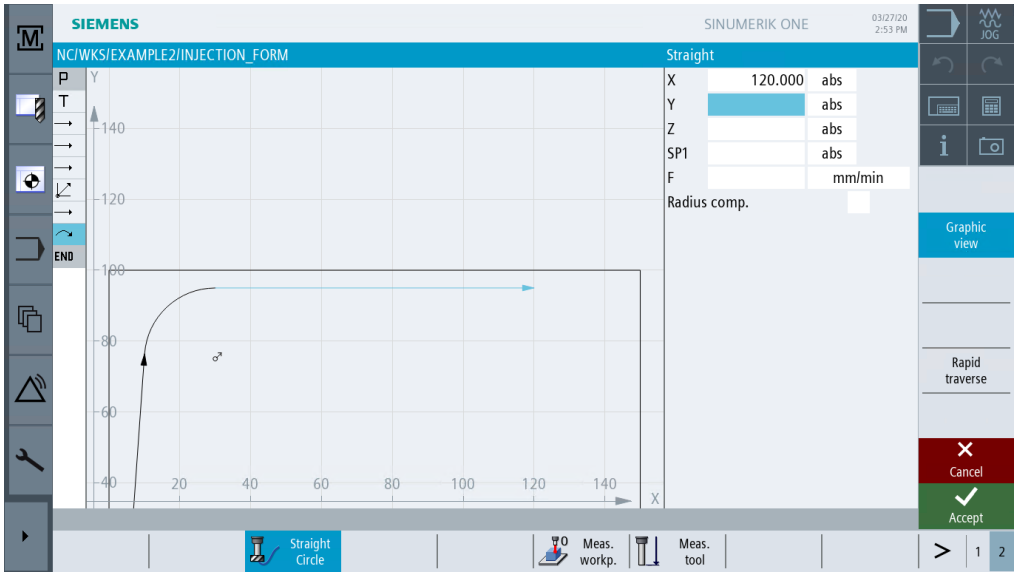


Figure 8-9 Specifying a straight line

Press "Accept" to apply the values entered.

Select the **Polar** softkey.

Select the **Pole** softkey.

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

Enter the following values in the interactive screenform:

Field	Value	Toggle field	Notes
X	120 abs	X	The pole of the circular path is known from the drawing.
Y	75 abs	X	

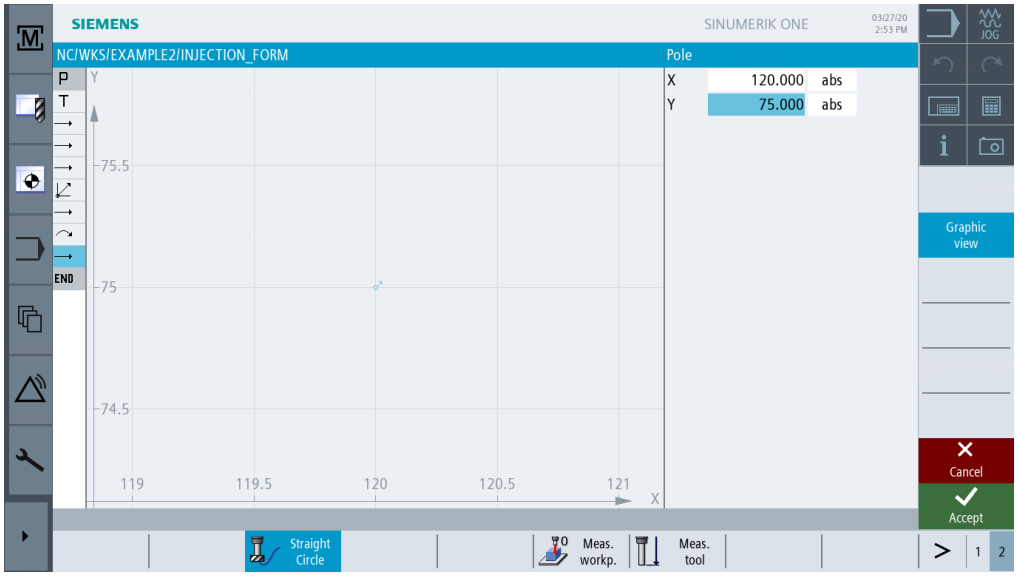


Figure 8-10 Specifying the pole for the circular path

Press "Accept" to apply the values entered.

Select the **Circle / polar** softkey.

Enter the following value in the interactive screenform:

Field	Value	Toggle field	Notes
α	4 abs	X	The polar angle is also known because of the symmetry.

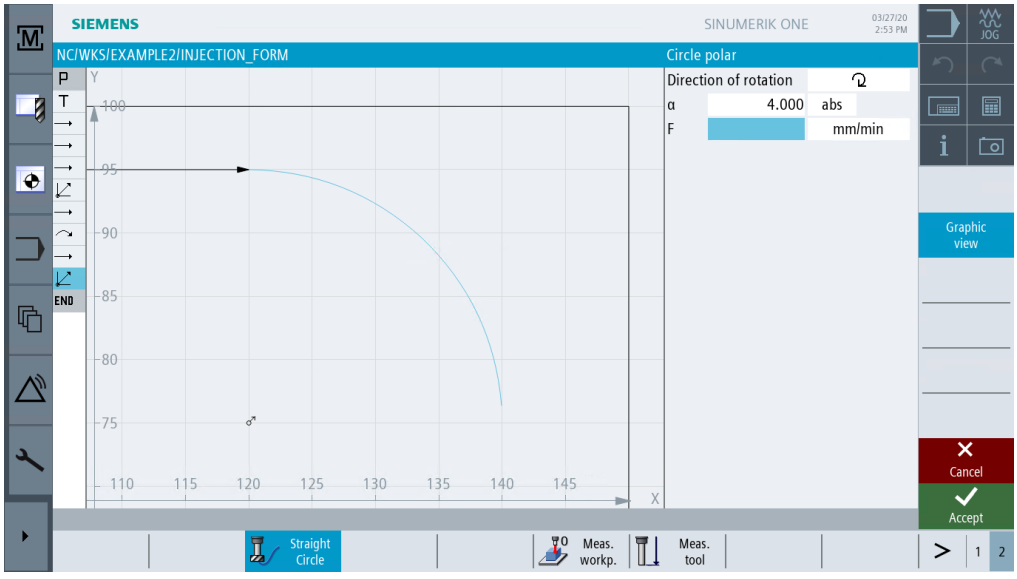


Figure 8-11 Specifying the circular path using polar coordinates

Press "Accept" to apply the values entered.

Select the **Back** softkey.

Select the **Straight line** softkey.

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

Enter the following values in the interactive screenform:

Field	Value	Toggle field	Notes
X	145 abs	X	
Y	5 abs	X	

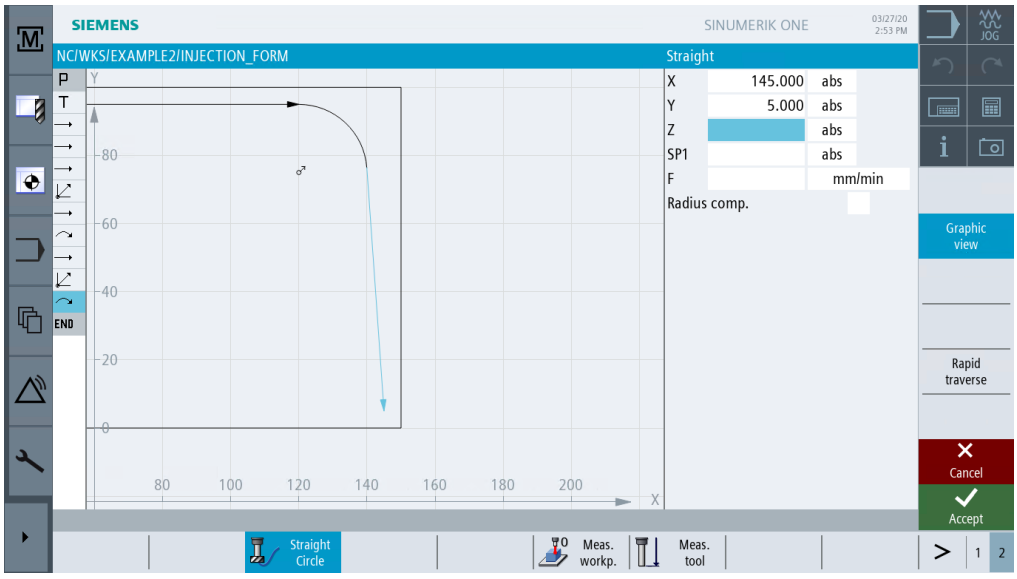


Figure 8-12 Specifying a straight line

Press "Accept" to apply the values entered.

Select the **Straight line** softkey.

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

Enter the following value in the interactive screenform:

Field	Value	Toggle field	Notes
X	-20 abs	X	

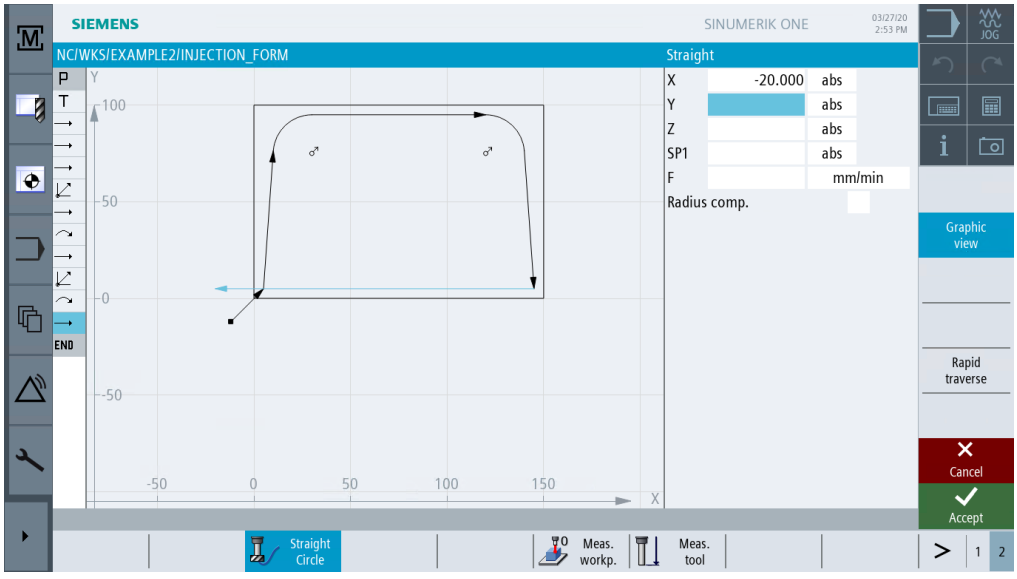


Figure 8-13 Specifying a straight line

Press "Accept" to apply the values entered.

Select the **Straight line** softkey.

Enter the following values in the interactive screenform:

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

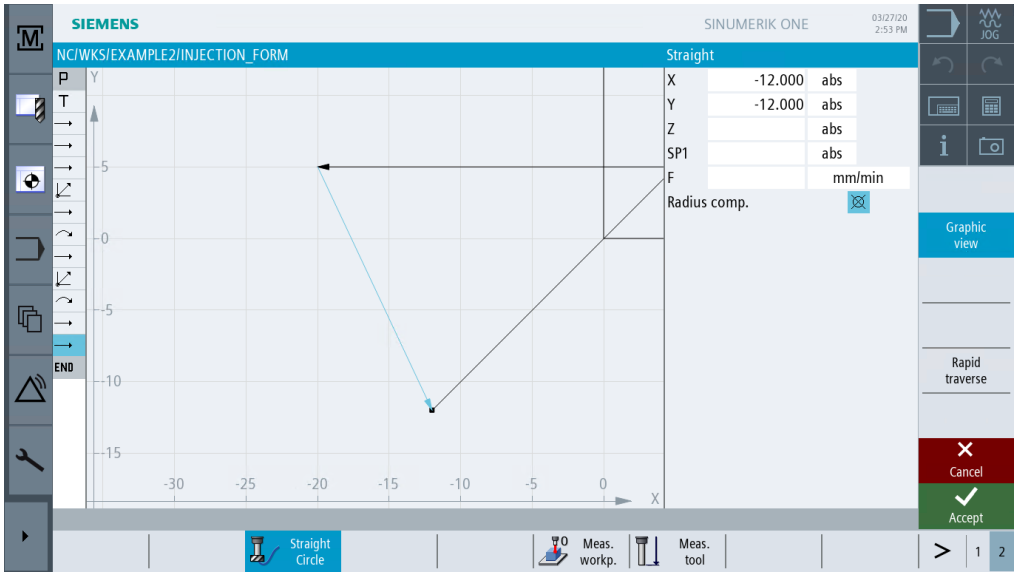


Figure 8-14 Specifying a straight line - Safety clearance

Press "Accept" to apply the values entered.

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

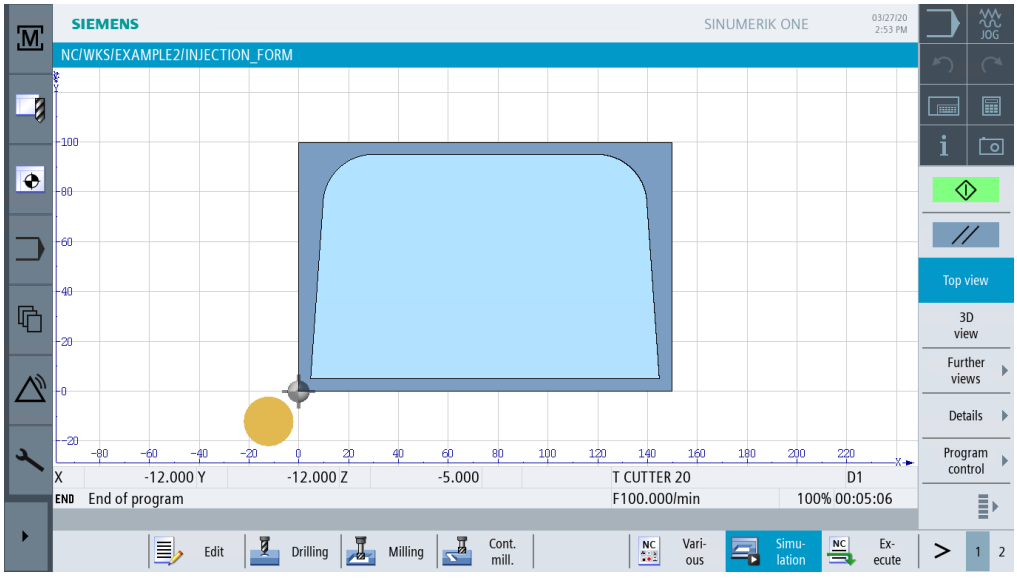


Figure 8-15 Simulation - Top view

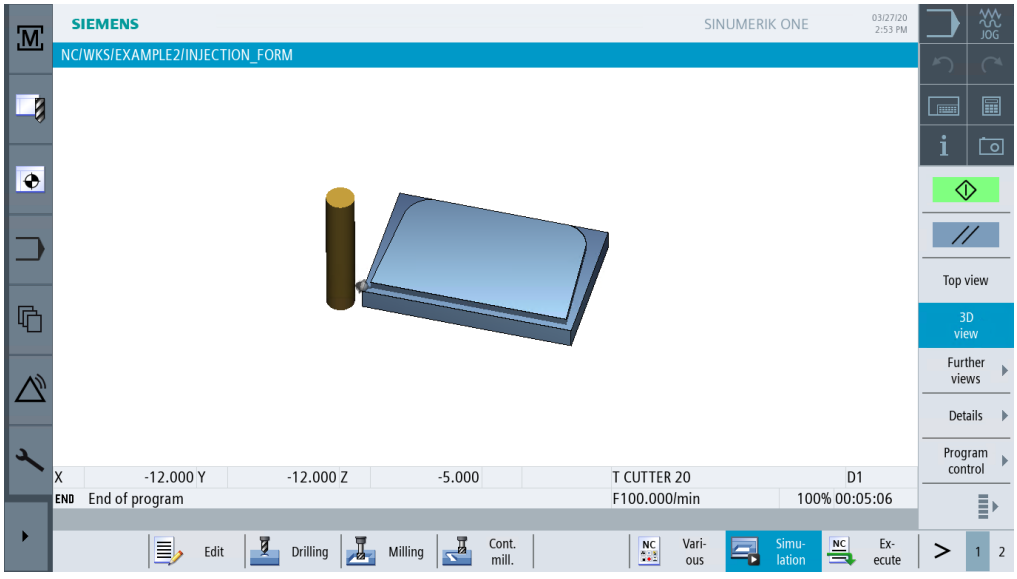


Figure 8-16 Simulation - 3D view

8.3 Rectangular pocket

Operating sequences

Proceed as follows to enter the rectangular pocket:

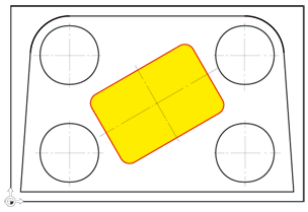


Figure 8-17 Rectangular pocket - Example 2

Select the **Milling** softkey.

Select the **Pocket** softkey.

Select the **Rectangular pocket** softkey.

Open the tool list and select CUTTER 10.

Accept the tool into your program.

After accepting the tool, enter the following values:

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

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

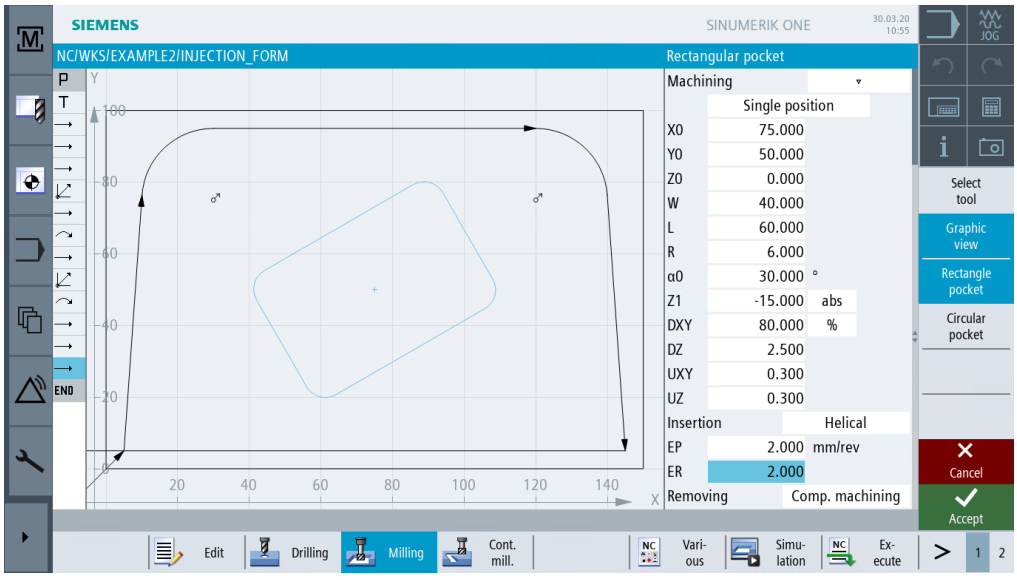


Figure 8-18 Roughing a rectangular pocket

Press "Accept" to apply the values entered.

Select the **Pocket** softkey.

Enter the following values in the interactive screenform:

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

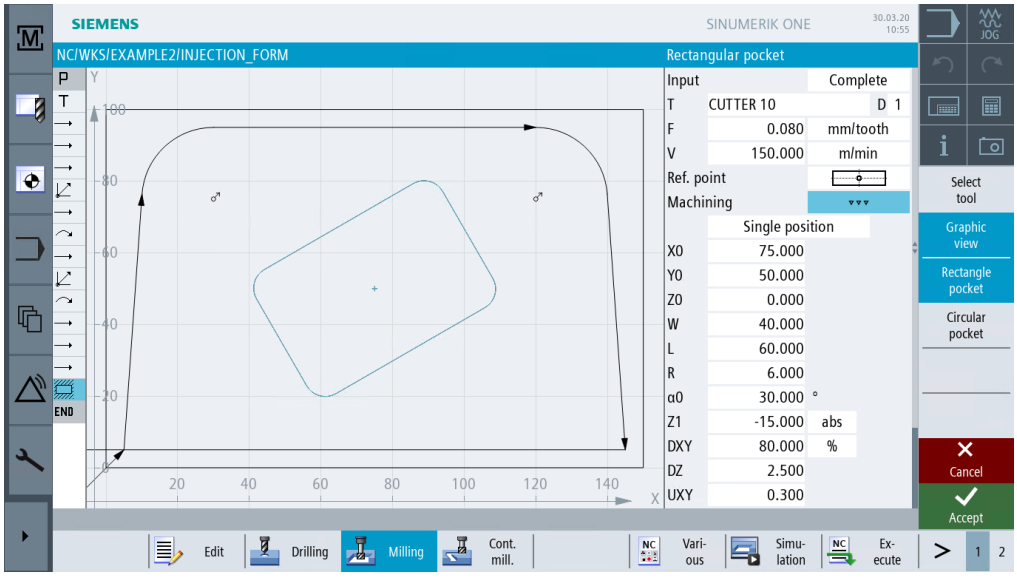
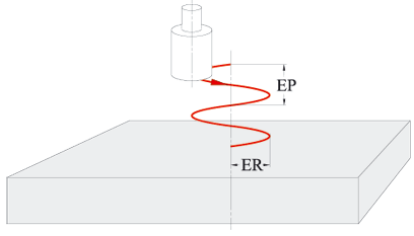
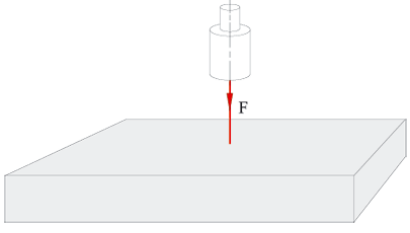
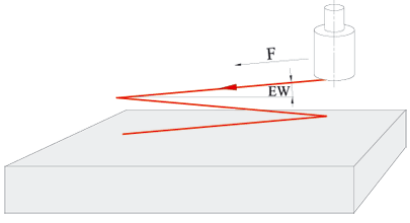


Figure 8-19 Finishing a rectangular pocket

Press "Accept" to apply the values entered.

Insertion

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

8.4 Circular pockets on a position pattern

Operating sequences

Proceed as follows to enter the rectangular pockets:

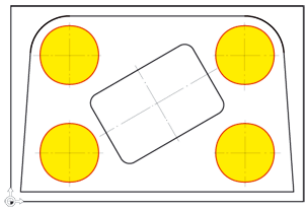


Figure 8-20 Rectangular pocket - Example 2

Select the **Milling** softkey.

Select the **Pocket** softkey.

Select the **Circular pocket** softkey.

Open the tool list and select CUTTER 10.

Accept the tool into your program.

After accepting the tool, enter the following values:

Field	Value	Toggle field	Notes
F	0.15 mm/tooth	X	
V	120 m/min	X	
Machining	Roughing	X	
	Position pattern	X	Similar to drilling, you can also apply a position pattern to pockets.
∅	30		
Z1	-10 abs	X	
DXY	80%	X	Specify the maximum infeed in the plane in %.
DZ	5		
UXY	0.3		
UZ	0.3		
Insertion	Helical	X	
EP	2 mm/rev	X	

Field	Value	Toggle field	Notes
ER	2		
Solid machining	Complete machining	X	

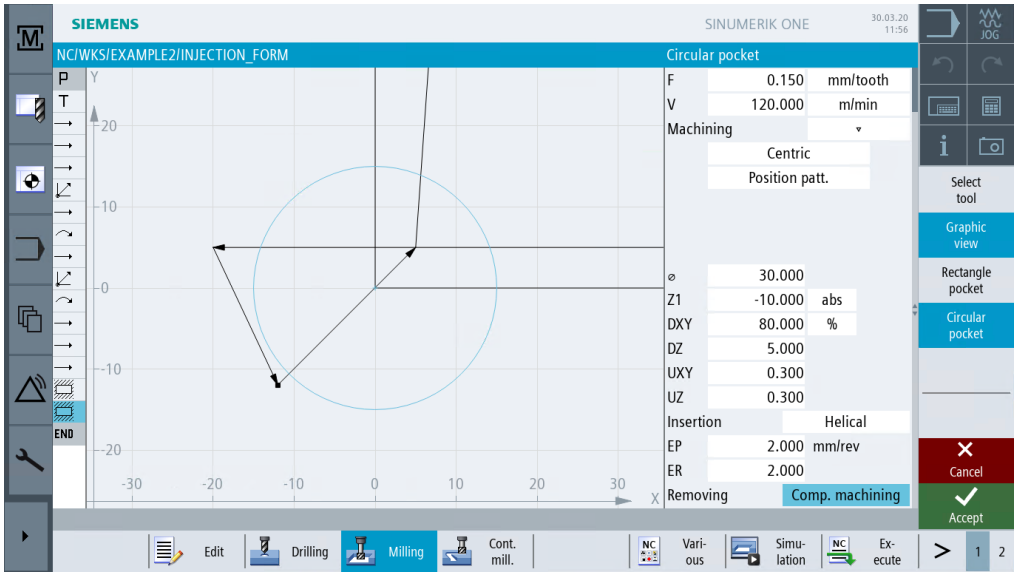


Figure 8-21 Roughing a circular pocket

Press "Accept" to apply the values entered.

Select the **Pocket** softkey.

Select the **Circular pocket** softkey.

Enter the following values:

Field	Value	Toggle field	Notes
F	0.08 mm/tooth	X	
V	150 m/min	X	
Machining	Finishing	X	

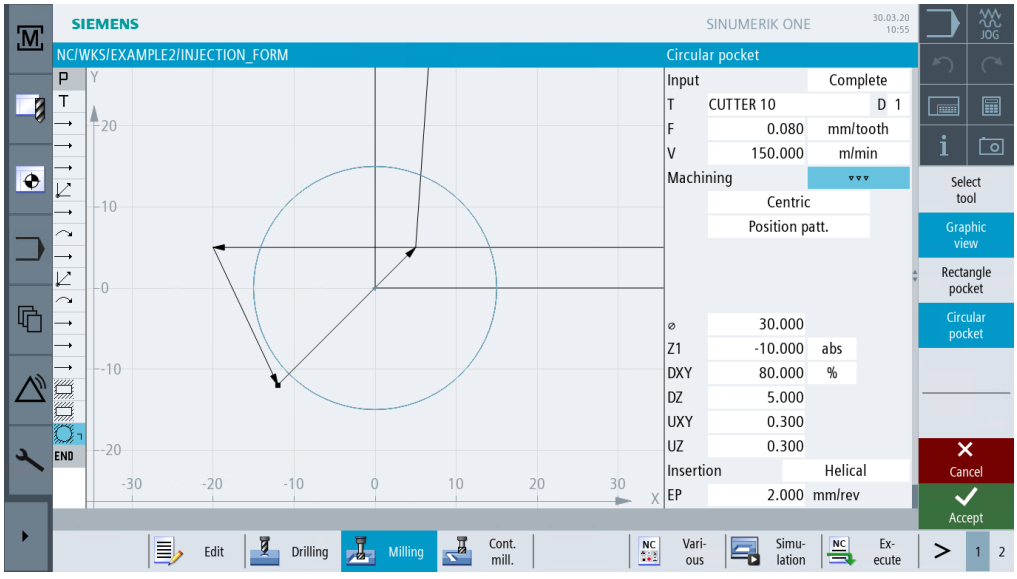
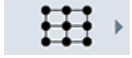


Figure 8-22 Finishing a circular pocket

Press "Accept" to apply the values entered.

Select the **Drilling** softkey.

Select the **Positions** softkey.



Select the **position grid** softkey.

Note

Position patterns are described in the **Drilling** menu with the **Positions** submenu (independent of the machining method).

Enter the following values:

Field	Value	Toggle field	Notes
X0	30		
Y0	25		
α0	0		
L1	90		
L2	50		
N1	2		
N2	2		

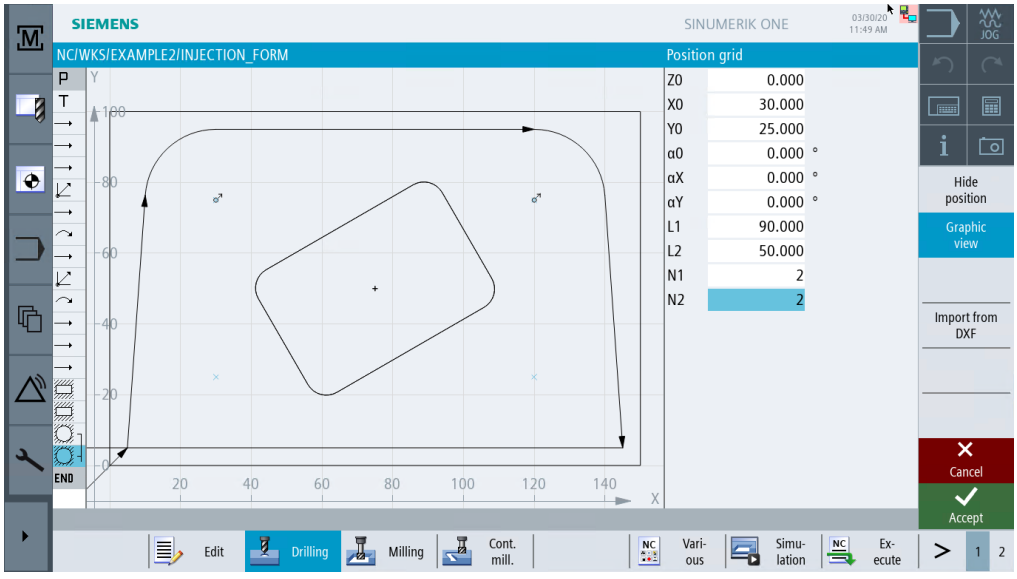


Figure 8-23 Positions of the circular pockets

Press "Accept" to apply the values entered.

Start simulation.

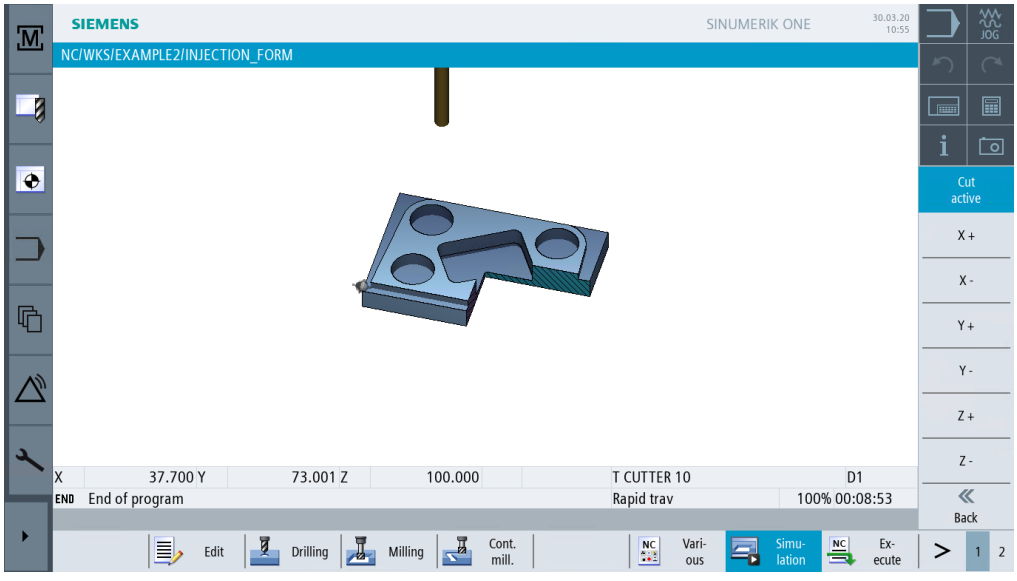


Figure 8-24 Simulation - Cut active

Example 3: Mold plate

9

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 methods on several planes;
- take into account obstacles.

Task

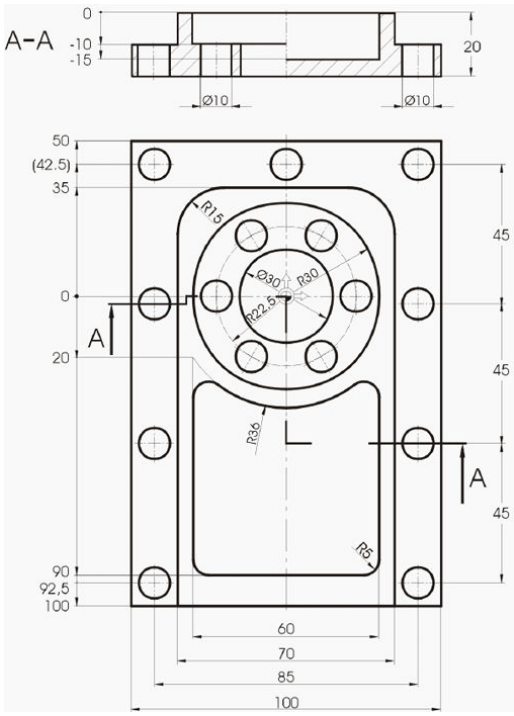


Figure 9-1 Workshop drawing - Example 3:

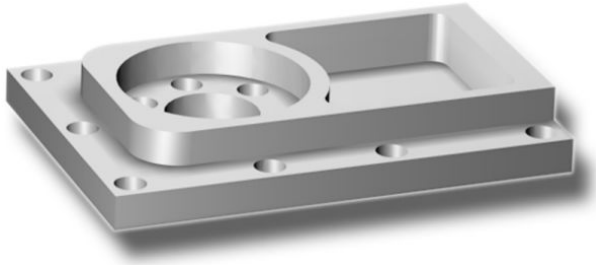


Figure 9-2 Workpiece - Example 3:

Preparation

Perform the following steps without help:

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

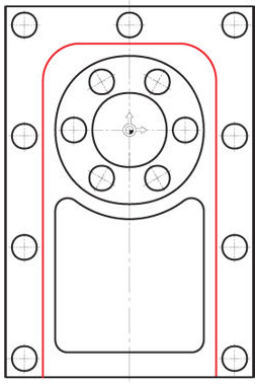
Note

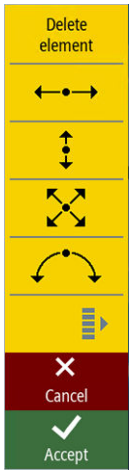
Observe the new zero position.

9.2 Path milling of open contours

Contour calculator

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





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

Operating sequences

Proceed as follows to enter the contour:

Select the **Contour milling** softkey.

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

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

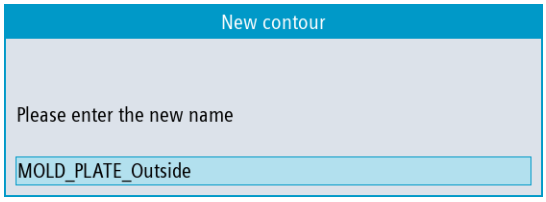


Figure 9-3 Creating the 'MOLD_PLATE_Outside' contour

Press "Accept" to apply your input.

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

Field	Value	Toggle field	Notes
X	-35		The starting point for construction is also the starting point for later machining of the contour.
Y	-100		

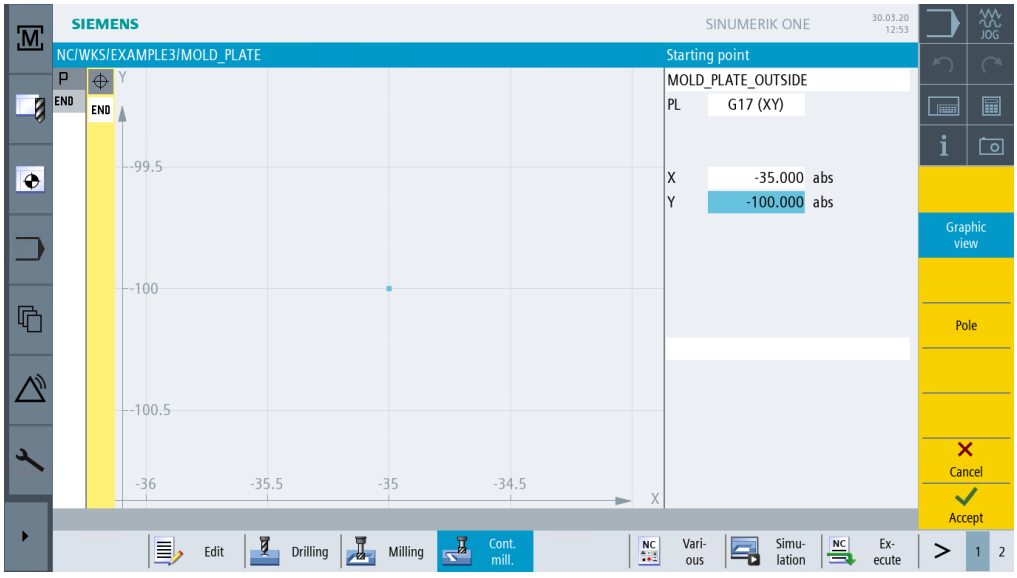


Figure 9-4 Specifying the starting point

Note

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

Press "Accept" to apply the values entered.

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

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

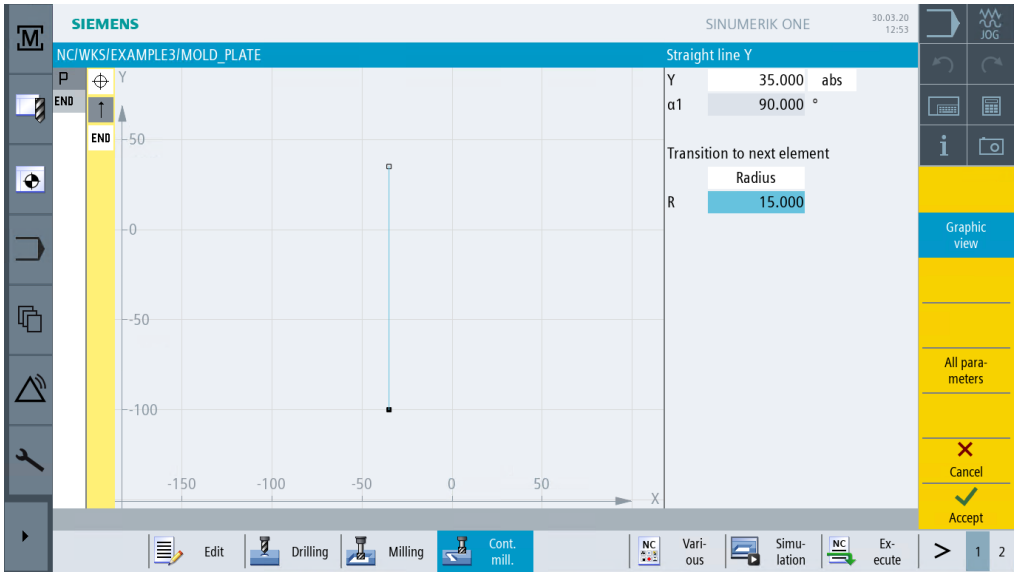


Figure 9-5 Specifying the vertical contour straight line

Press "Accept" to apply the values entered.

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

Field	Value	Toggle field	Notes
X	35 abs	X	
Transition to next element	Radius	X	
R	15		The radius is entered as a rounding again.

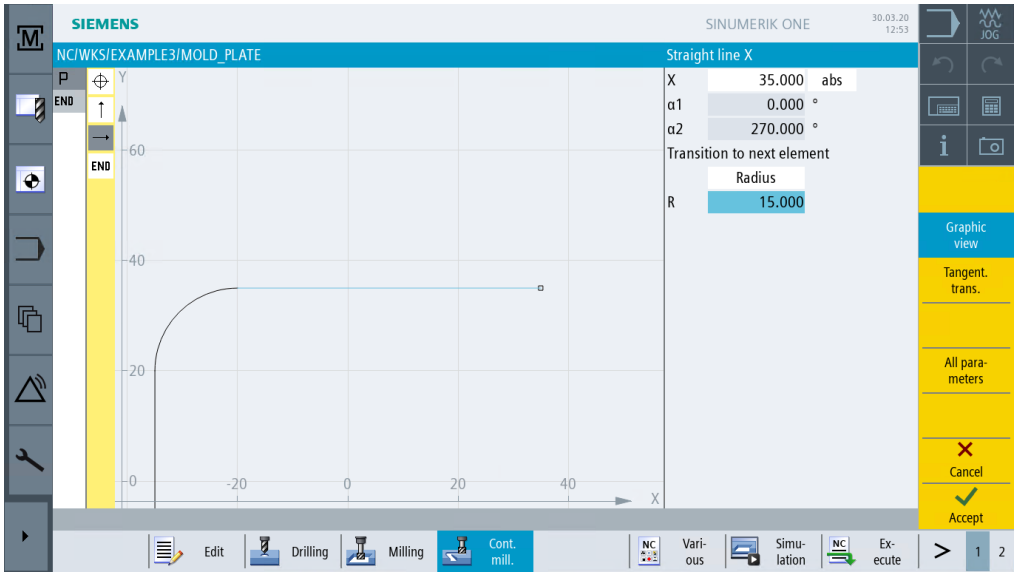


Figure 9-6 Specifying the horizontal contour straight line

Press "Accept" to apply the values entered.

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

Field	Value	Toggle field	Notes
Y	-100 abs	X	

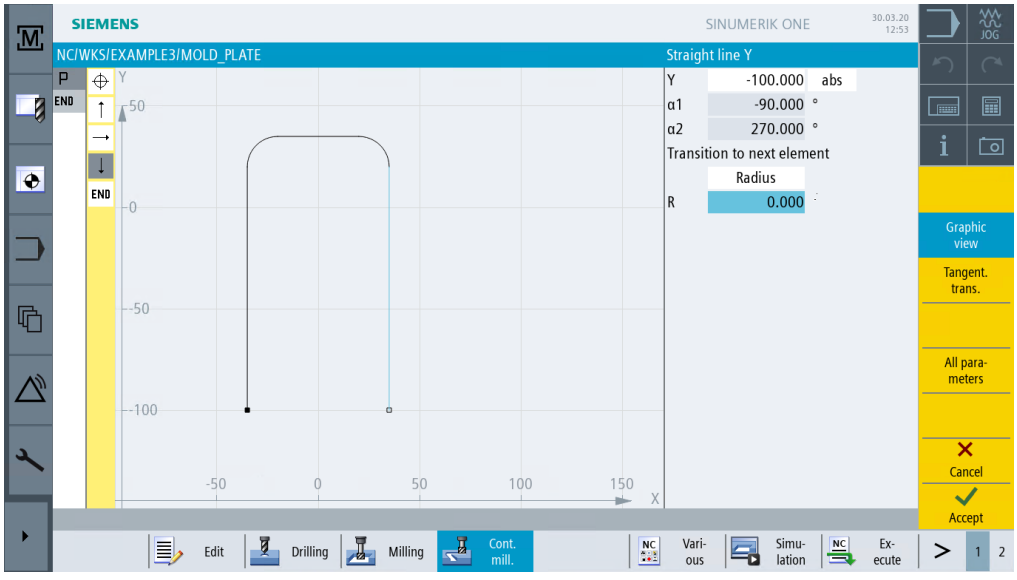


Figure 9-7 Specifying the vertical contour straight line

"Apply" the set values.

Accept the contour into your process plan.

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

Select the **Path milling** softkey.

Open the tool list and select CUTTER 32.

Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

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

Field	Value	Toggle field	Notes
FZ	0.1 mm/tooth	X	
Retraction	Straight line	X	
L2	5		
Lift mode	To retraction plane	X	

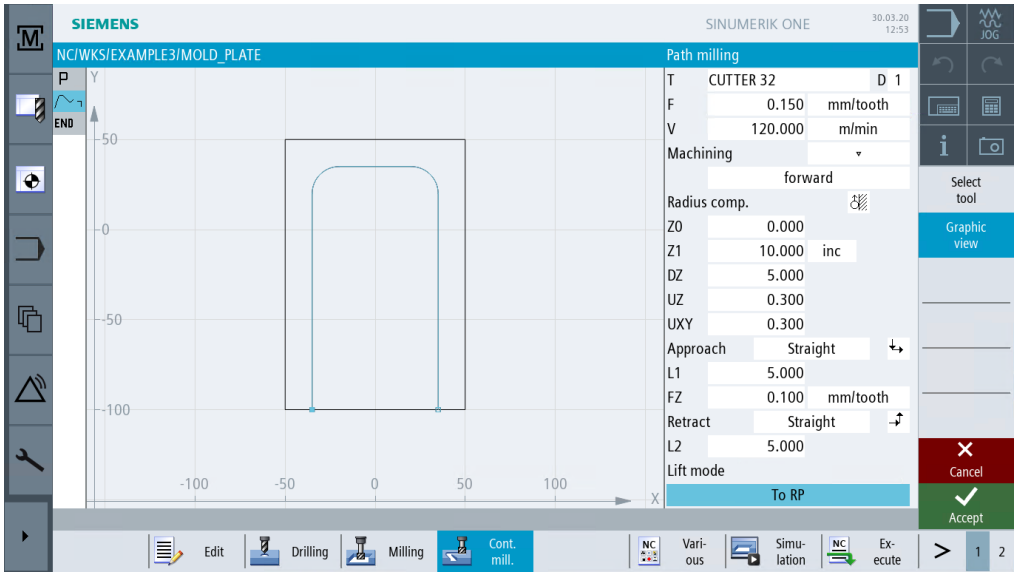


Figure 9-8 Roughing the contour

Press "Accept" to apply the values entered.

Enter the following values for finishing in the interactive screenform:

Field	Value	Toggle field	Notes
F	0.08 mm/tooth	X	
V	150 m/min	X	
Machining	Finishing	X	

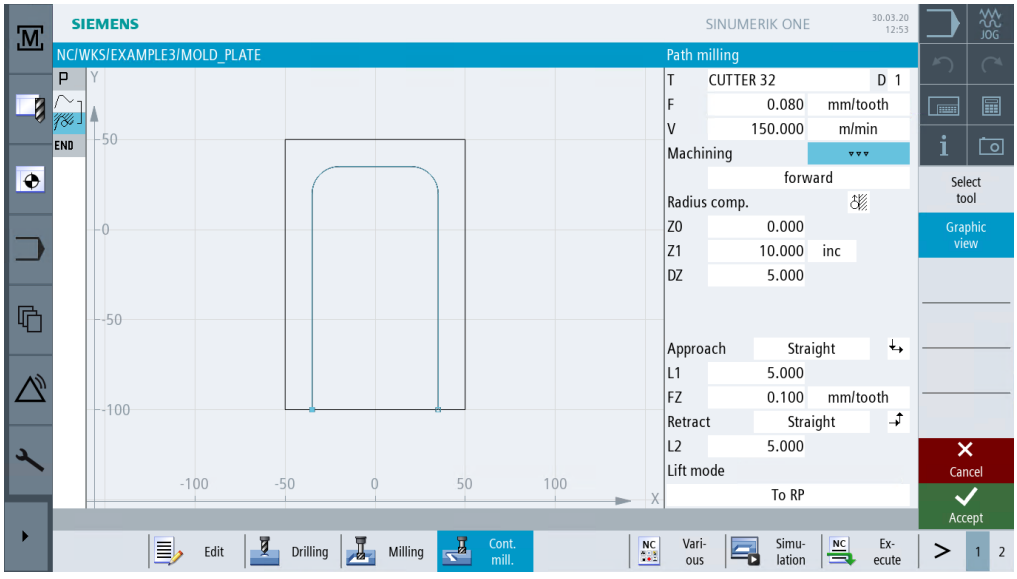


Figure 9-9 Finishing the contour

Press "Accept" to apply the values entered.

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

P	Program header	G54 Block
	Contour	MOLD_PLATE_OUTSIDE
	Path milling	T=CUTTER 32 F=0.15/t V=120m Z0=0 Z1=10inc
	Path milling	T=CUTTER 32 F=0.08/t V=150m Z0=0 Z1=10inc
END	End of program	

Figure 9-10 Linking of the work steps in the process plan

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

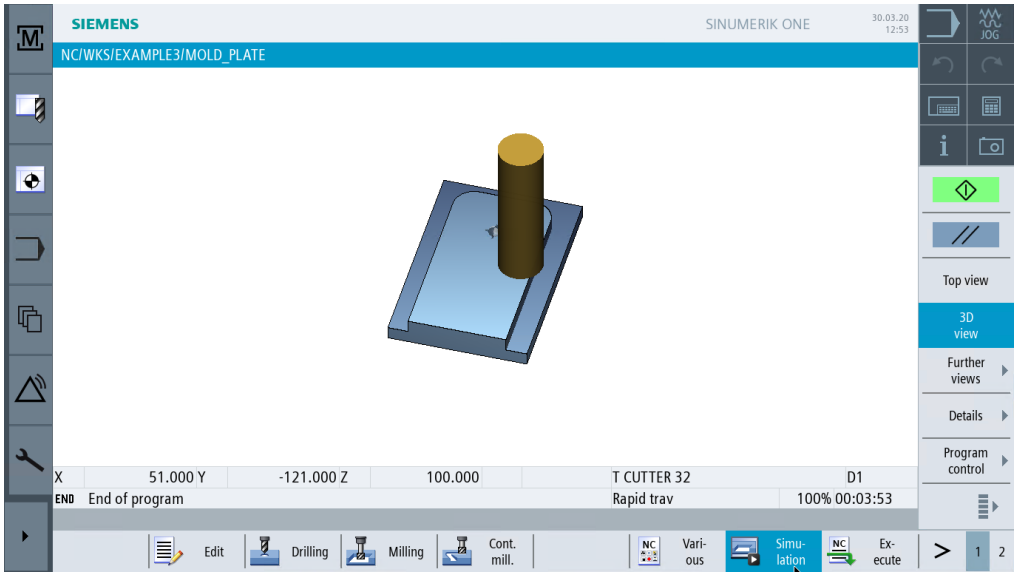


Figure 9-11 Simulation - External contour

9.3 Solid machining and residual material; finishing of contour pockets

Operating sequences

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

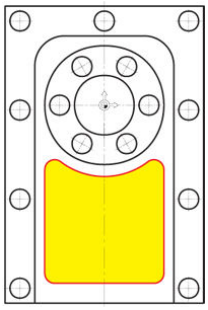


Figure 9-12 Pocket contour

Select the **Contour milling** softkey.

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

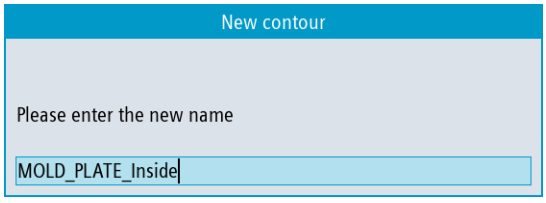


Figure 9-13 Creating the 'MOLD_PLATE_Inside' contour

Press "Accept" to apply your input.

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

Field	Value	Toggle field	Notes
X	0		
Y	-90		

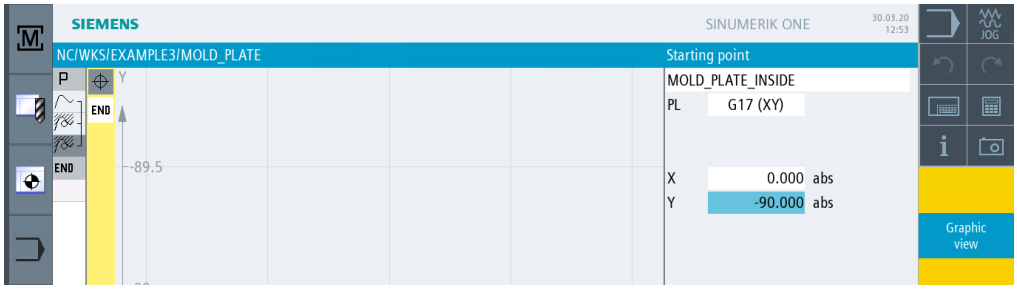


Figure 9-14 Specifying the starting point

Press "Accept" to apply the values entered.

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

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

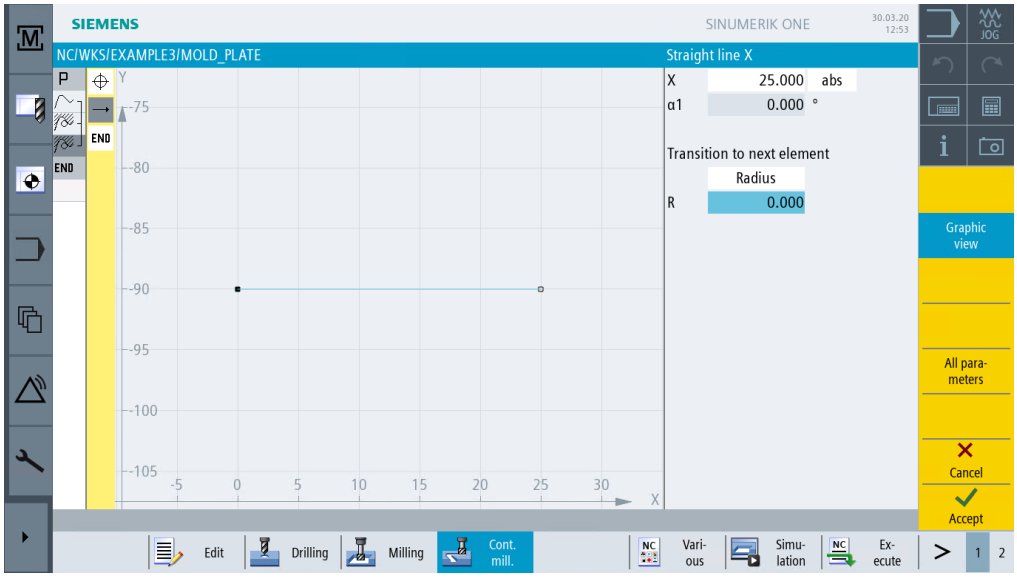


Figure 9-15 Specifying the horizontal contour straight line

Press "Accept" to apply the values entered.

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

Field	Value	Toggle field	Notes
Direction of rotation	Left	X	
R	5		
X	30 abs	X	
Y	-85 abs	X	

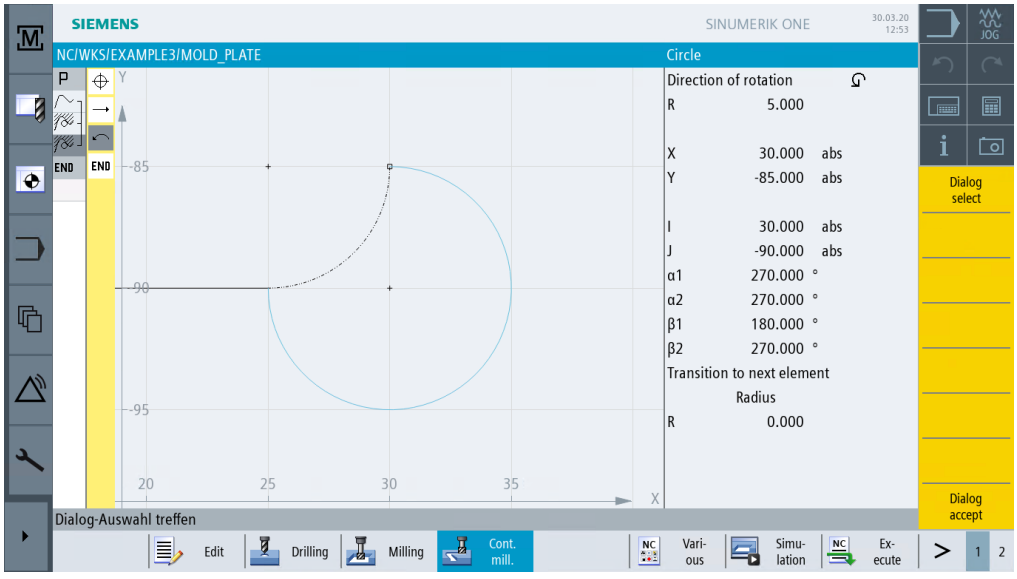


Figure 9-16 Arc contour (bottom right)

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

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

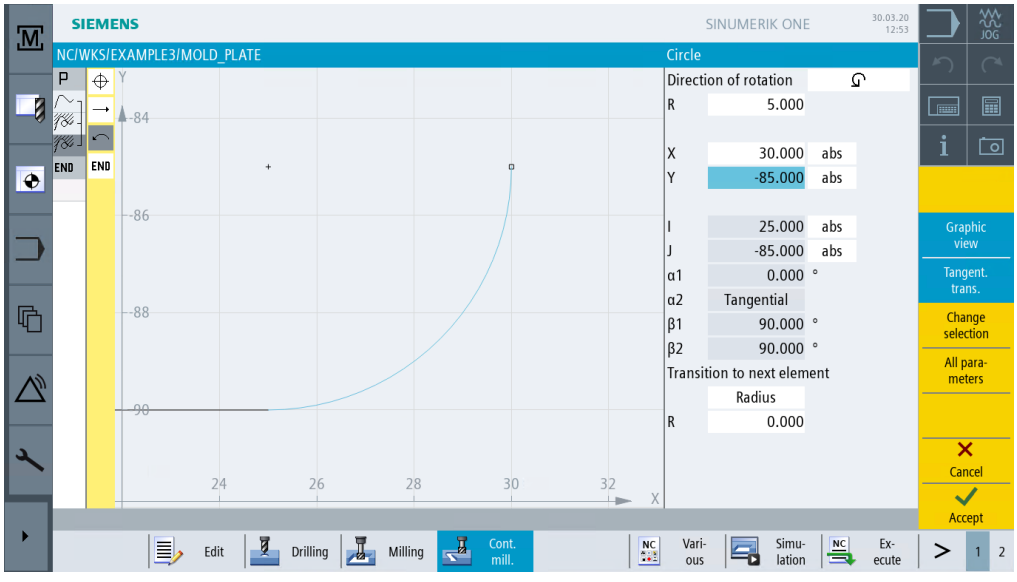


Figure 9-17 Arc contour - after selection

Press "Accept" to apply the values entered.

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

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

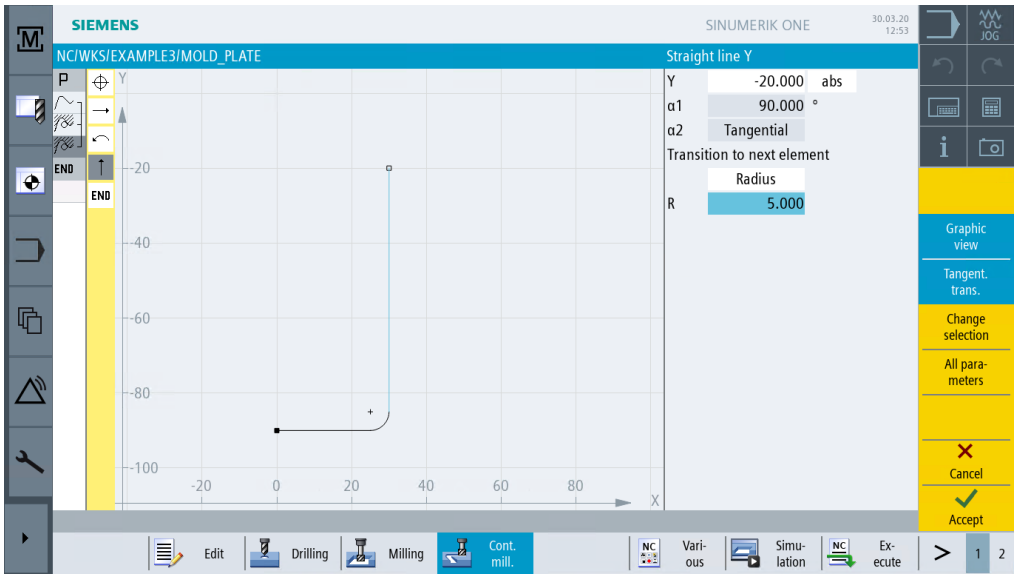


Figure 9-18 Specifying the vertical contour straight line

Press "Accept" to apply the values entered.

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

Field	Value	Toggle field	Notes
Direction of rotation	Clockwise	X	
R	36		
X	-30 abs	X	
Y	-20 abs	X	
Transition to next element	Radius	X	Specify the radius R5 as a rounding.
R	5		

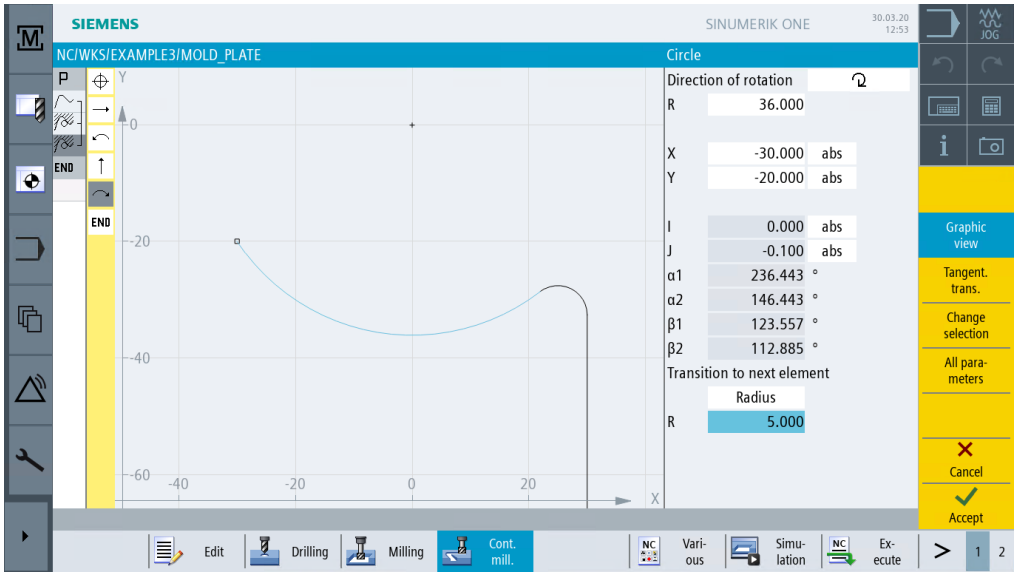


Figure 9-19 Specifying the contour arc

Press "Accept" to apply the values entered.

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

Field	Value	Toggle field	Notes
Y	-90 abs	X	
Transition to next element	Radius	X	Specify the radius R5 as a rounding.
R	5		

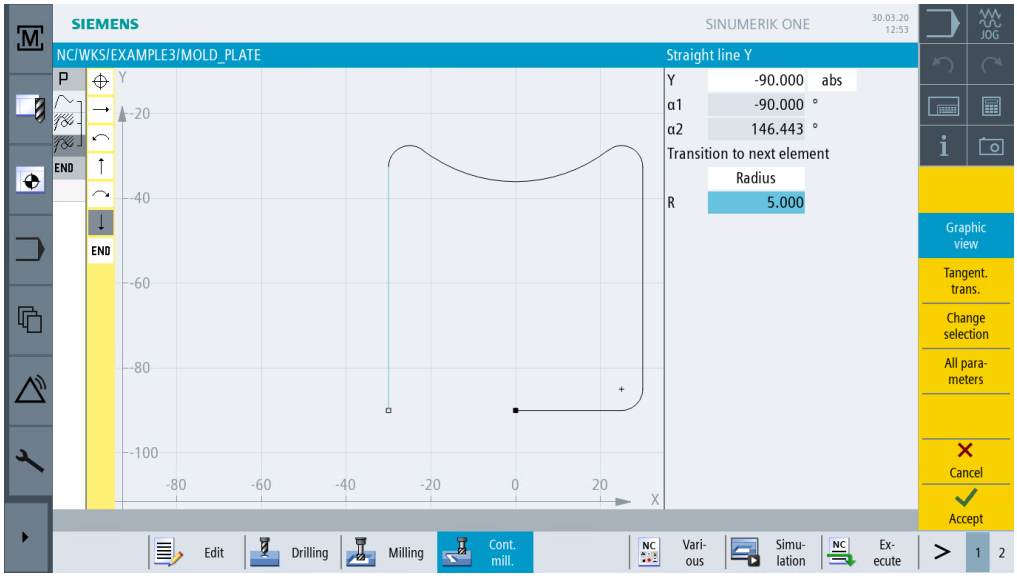


Figure 9-20 Specifying the vertical contour straight line

Press "Accept" to apply the values entered.

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

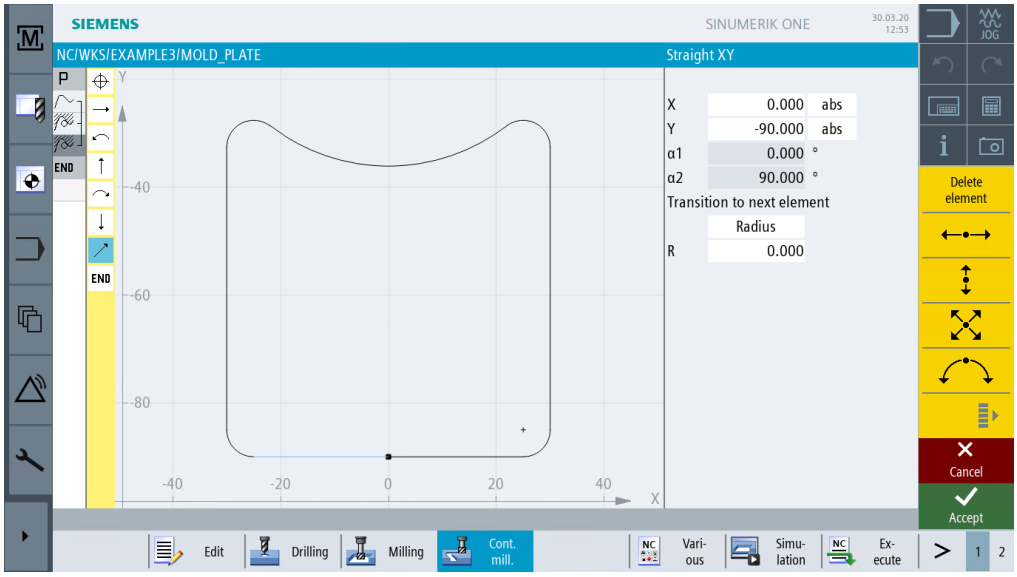


Figure 9-21 Closing the contour

Accept the contour into your process plan.

Select the **Pocket** softkey.

Open the tool list and select CUTTER 20.

Accept the tool into your program.

Note

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

Enter the following values for roughing in the interactive screenform:

Field	Value	Toggle field	Notes
F	0.15 mm/tooth	X	
V	120 m/min	X	
Machining	Roughing	X	
Z0	0		

Field	Value	Toggle field	Notes
Z1	15 inc	X	If you specify the machining depth with *incremental* , you must specify a positive value for the depth.
DXY	50%	X	
DZ	5		
UXY	0.3		
UZ	0.3		
Starting point	Automatic	X	If you select the *Autom* setting for the starting point (insertion), the starting point is specified by ShopMill.
Insertion	Helical	X	Set insertion to *Helical* with 2mm for both pitch and radius.
EP	2 mm/rev	X	
ER	2		
Lift mode	To retraction plane	X	

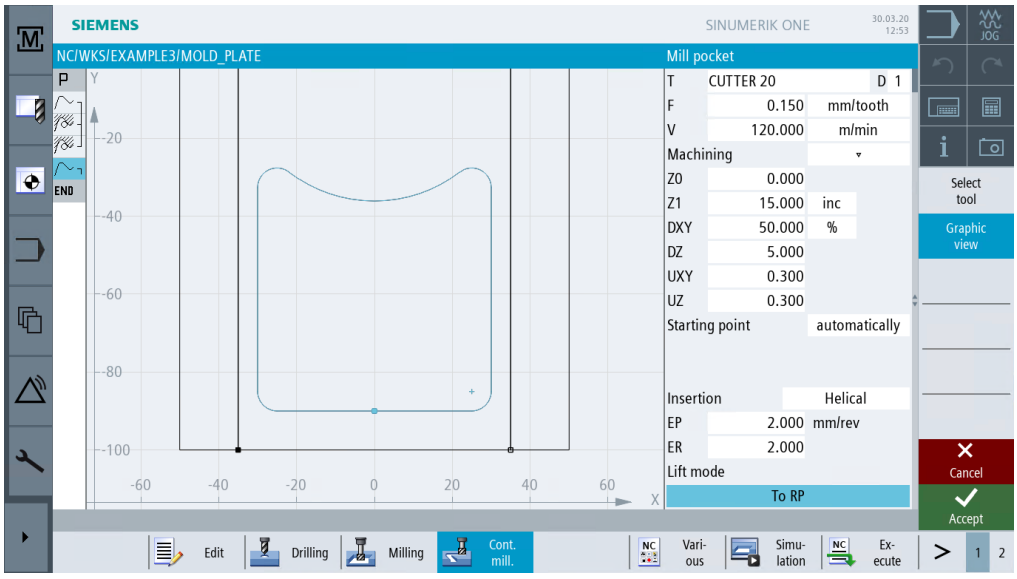


Figure 9-22 Roughing a pocket

Press "Accept" to apply the values entered.

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

Select tool

Open the tool list and select CUTTER 10.

OK

Accept the tool into your program.

Enter the following values in the interactive screenform:

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

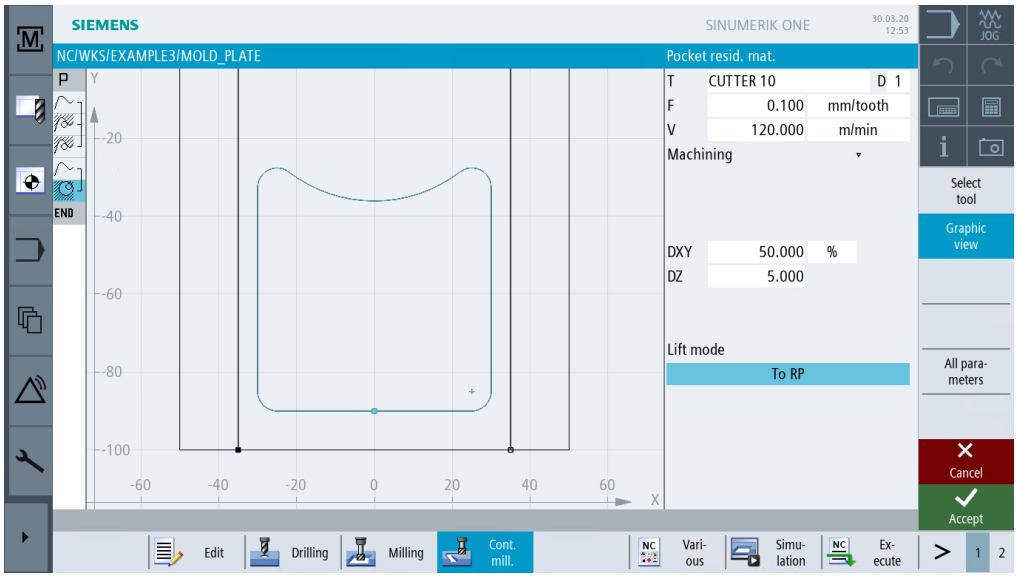


Figure 9-23 Machining residual material of the pocket

Accept

Press "Accept" to apply the values entered.

Pocket

Select the **Pocket** softkey.

Select tool

Open the tool list and select CUTTER 10.

OK

Accept the tool into your program.

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

Field	Value	Toggle field	Notes
F	0.08 mm/tooth	X	
V	150 m/min	X	

Field	Value	Toggle field	Notes
Machining	Base	X	
UXY			The allowance which you have previously entered for roughing must remain set for the values in the "Finishing allowance fields in the plane (UXY)" and "Finishing allowance in the depth (UZ)" fields. This value is important for automatic calculation of the distances to be traversed.
UZ			

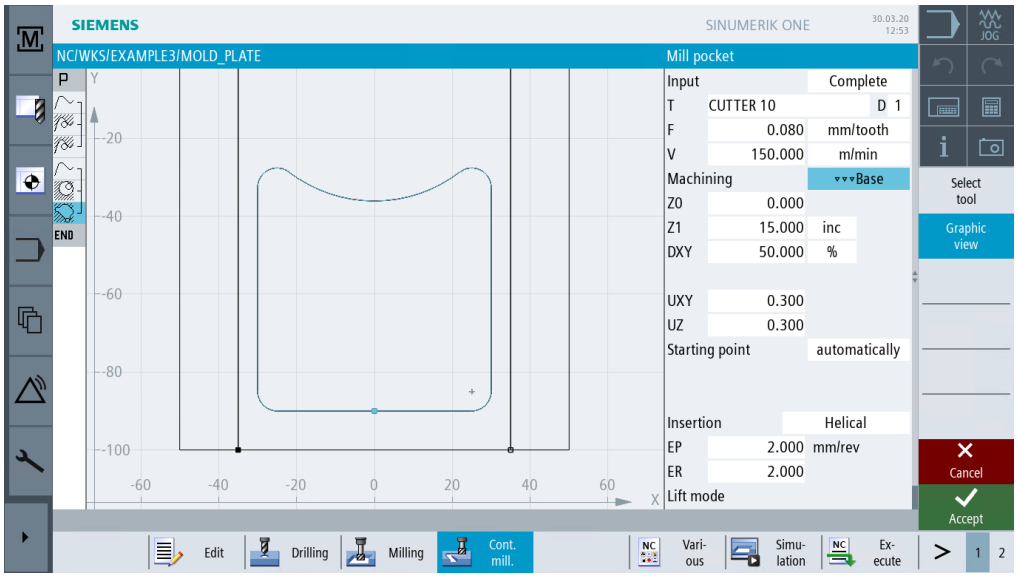


Figure 9-24 Finishing a pocket

Press "Accept" to apply the values entered.

Select the **Pocket** softkey.

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

Field	Value	Toggle field	Notes
Machining	Edge	X	

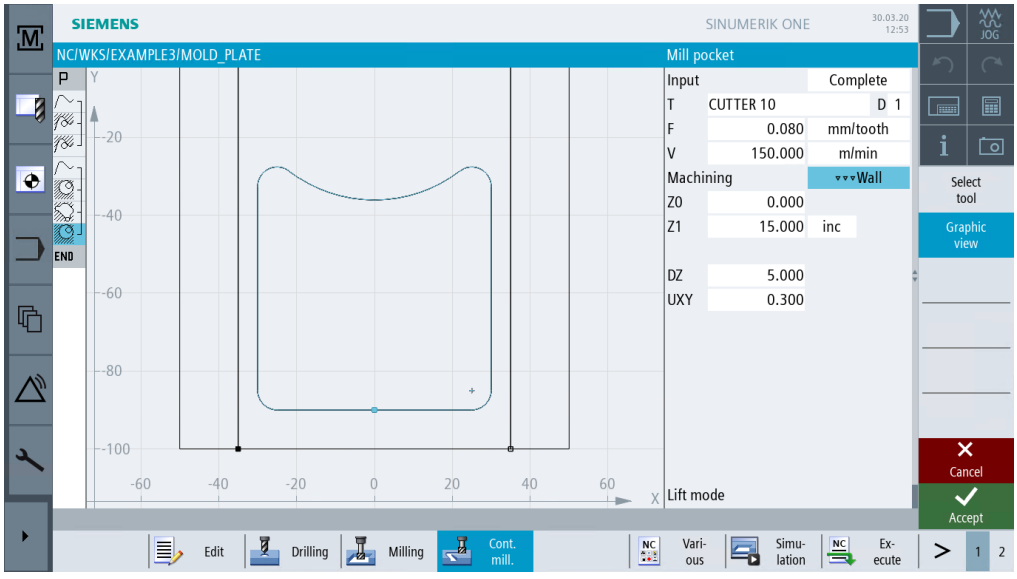


Figure 9-25 Finishing the edge

Press "Accept" to apply the values entered.

9.4 Machining on several planes

Operating sequences

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

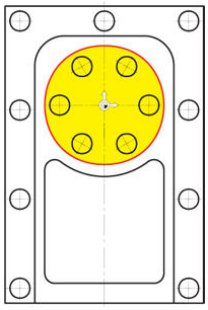


Figure 9-26 Circular pocket

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

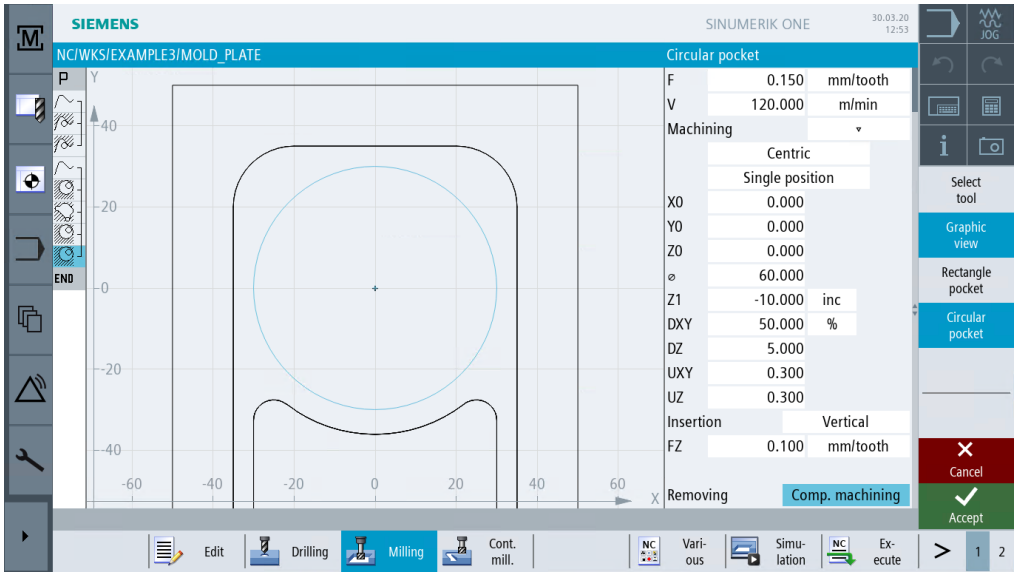


Figure 9-27 Roughing a circular pocket

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

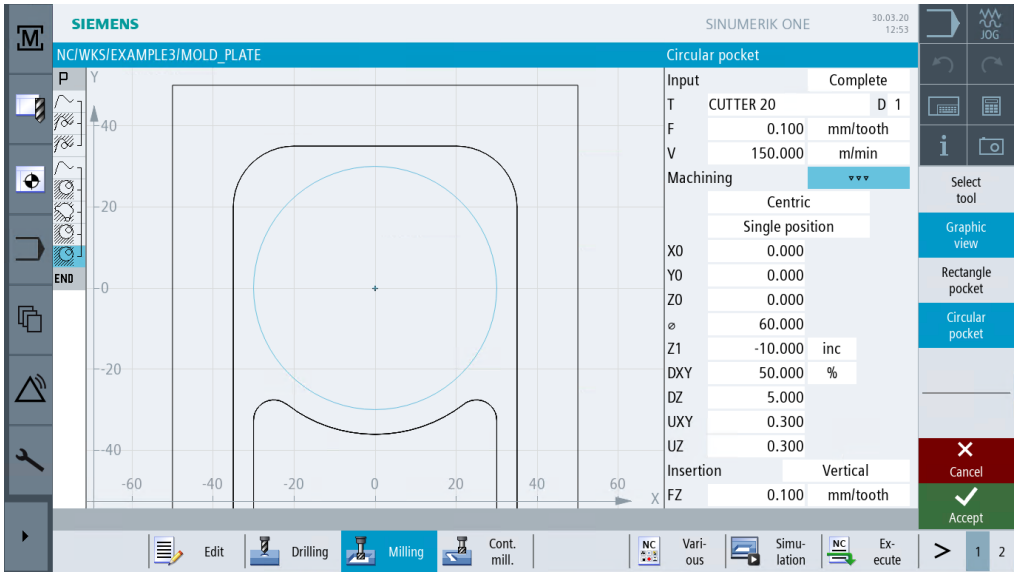


Figure 9-28 Finishing a circular pocket

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

Note

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

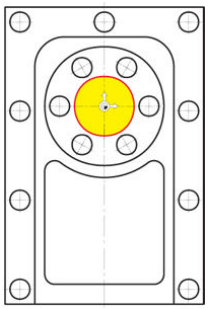


Figure 9-29 Inside circular pocket

Select the **Milling** softkey.

Select the **Pocket** softkey.

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

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

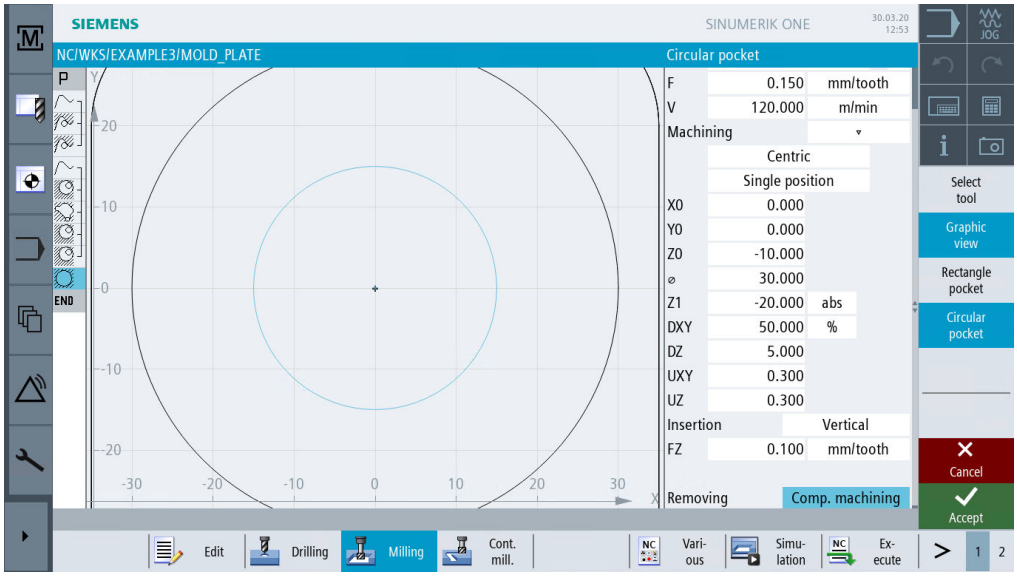


Figure 9-30 Roughing the inside circular pocket

Press "Accept" to apply the values entered.

Select the **Milling** softkey.

Select the **Pocket** softkey.

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

Field	Value	Toggle field	Notes
F	0.08 mm/tooth	X	
V	150 m/min	X	

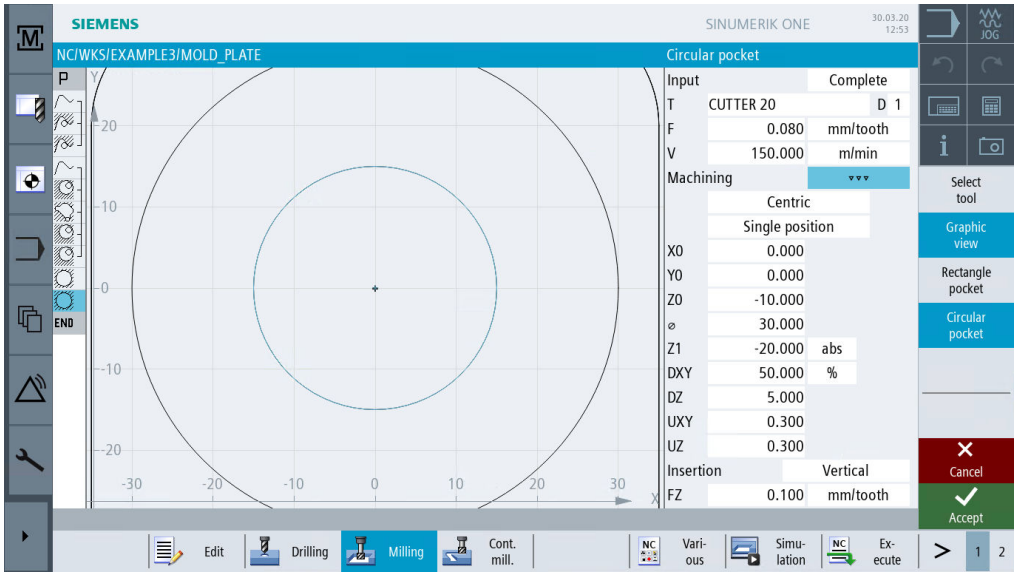


Figure 9-31 Finishing the inside circular pocket

Press "Accept" to apply the values entered.

Start simulation.

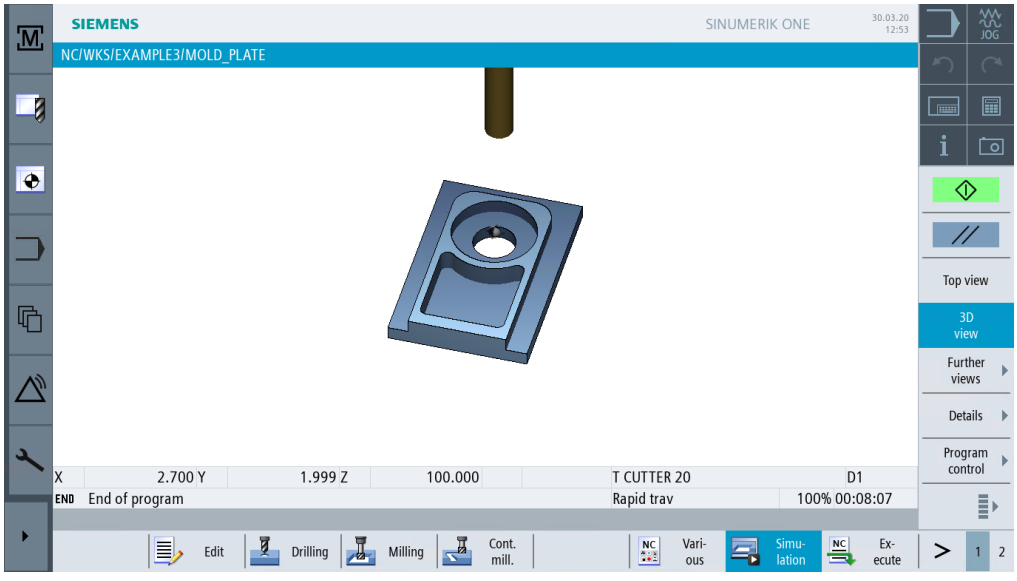


Figure 9-32 Simulation in 3D display

9.5 Taking into account obstacles

Operating sequences

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

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

1. Centering

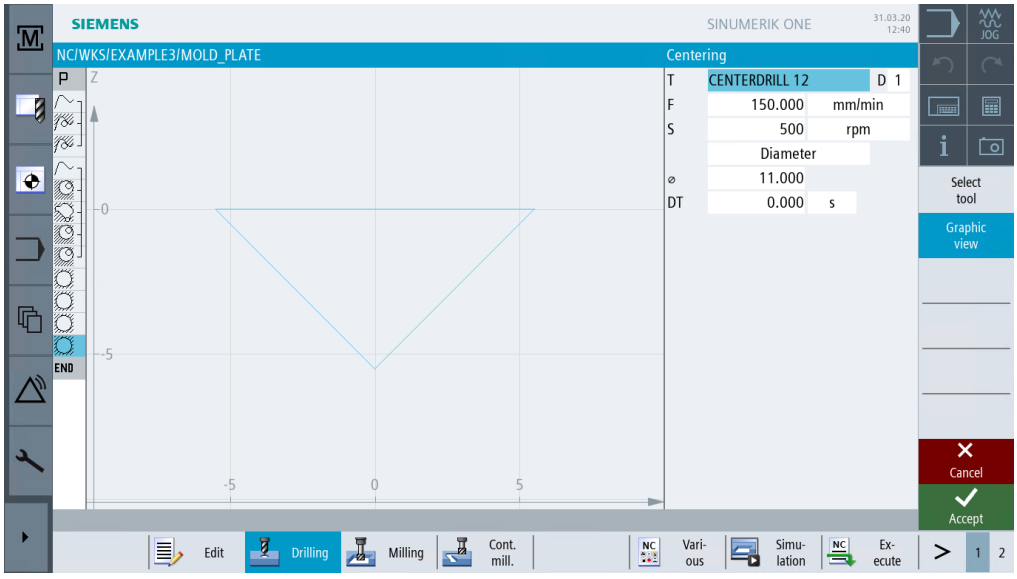


Figure 9-33 Work step 'Centering'

2. Drilling

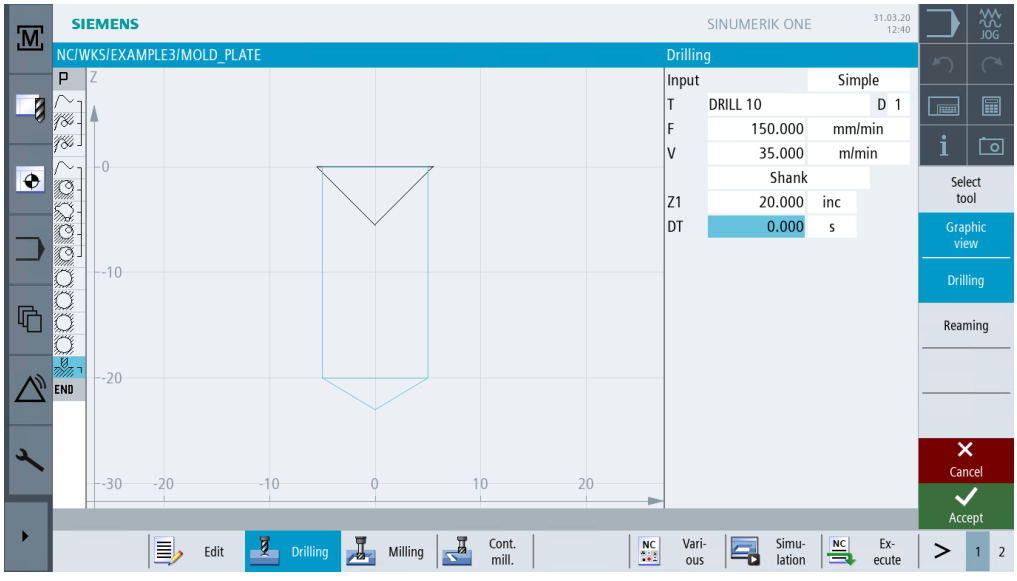


Figure 9-34 Work step 'Drilling'

Proceed as follows to enter the relevant drilling positions:

Select the **Positions** softkey.

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

Enter the following values in the interactive screenform:

Field	Value	Toggle field	Notes
Z0	-10		
X0	-42.5		
Y0	-92.5		
α0	90		
L0	0		
L	45		
N	4		

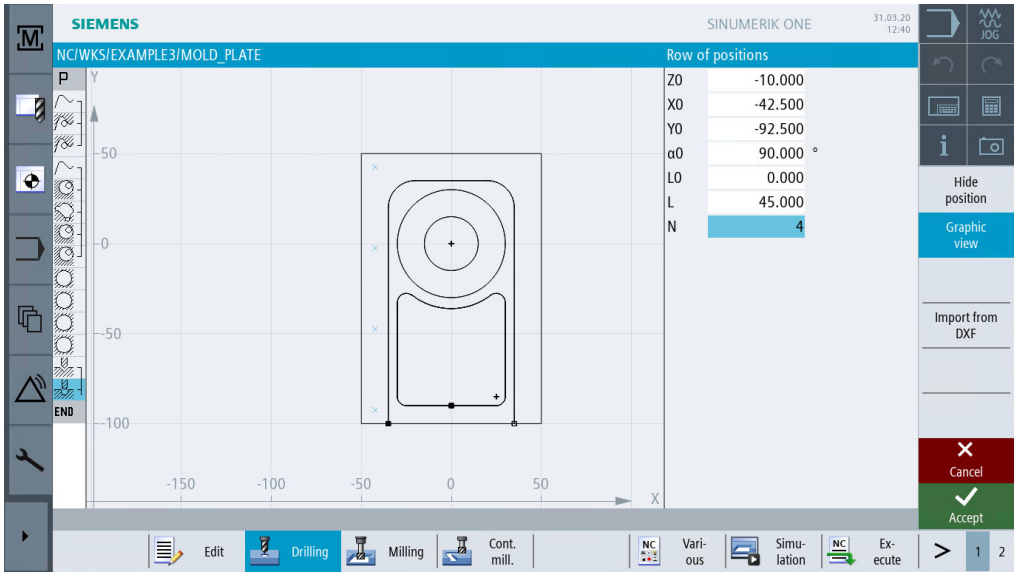


Figure 9-35 Specifying the line of holes

Press "Accept" to apply the values entered.

Select the **Positions** softkey.

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

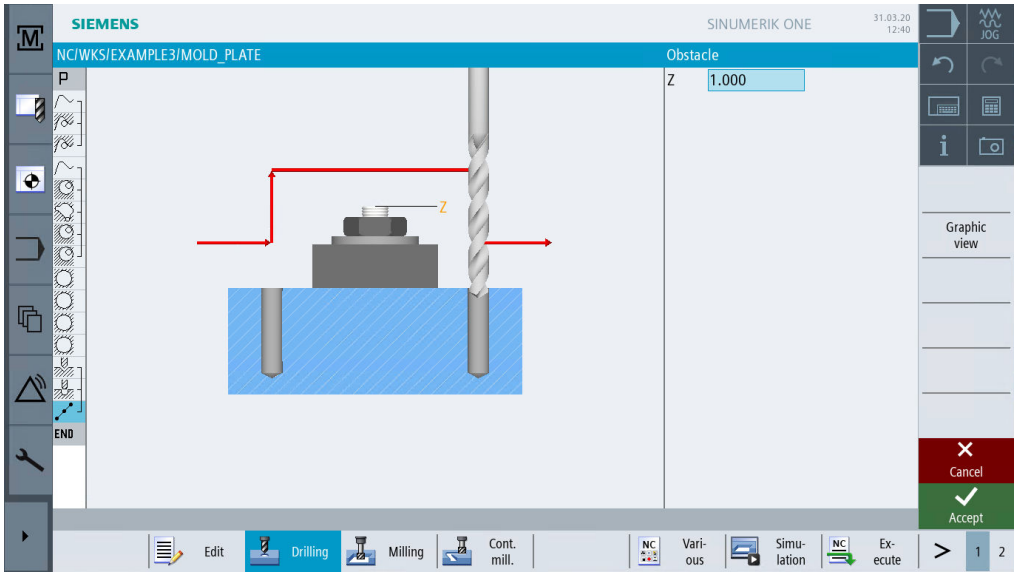
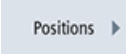


Figure 9-36 Specifying an obstacle

Press "Accept" to apply the values entered.



Select the **Positions** softkey.

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

Field	Value	Toggle field	Notes
Z0	-10		
X0	42.5		
Y0	-92.5		
α0	90		
L0	0		
L	45		
N	4		

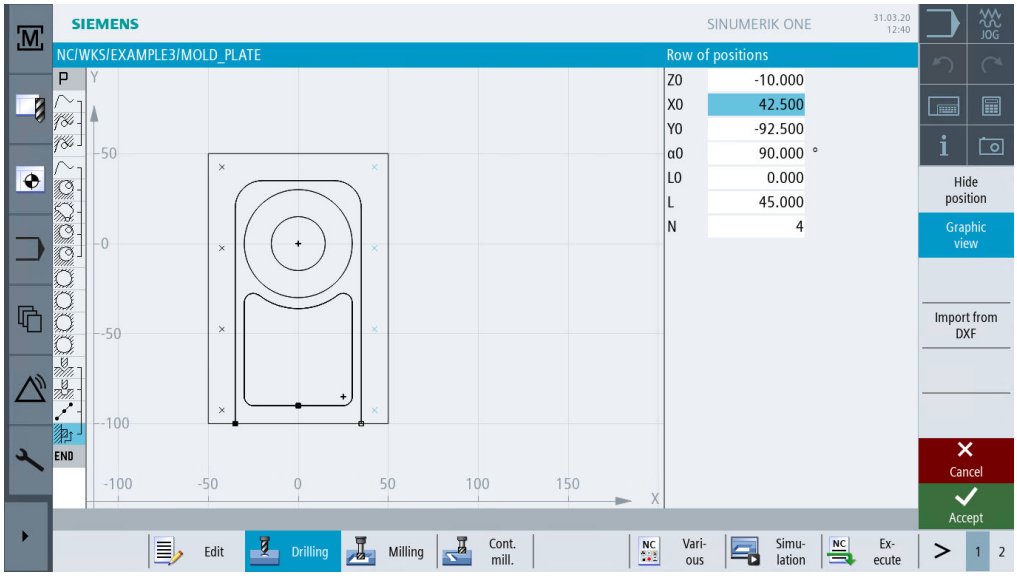


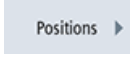
Figure 9-37 Specifying the line of holes

Press "Accept" to apply the values entered.

Select the **Positions** softkey.

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

Press "Accept" to apply the value entered.



Select the **Positions** softkey.

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

Field	Value	Toggle field	Notes
Z0	-10		
X0	0		
Y0	0		
α0	0		
R	22.5		
N	6		
Positioning	Straight line	X	

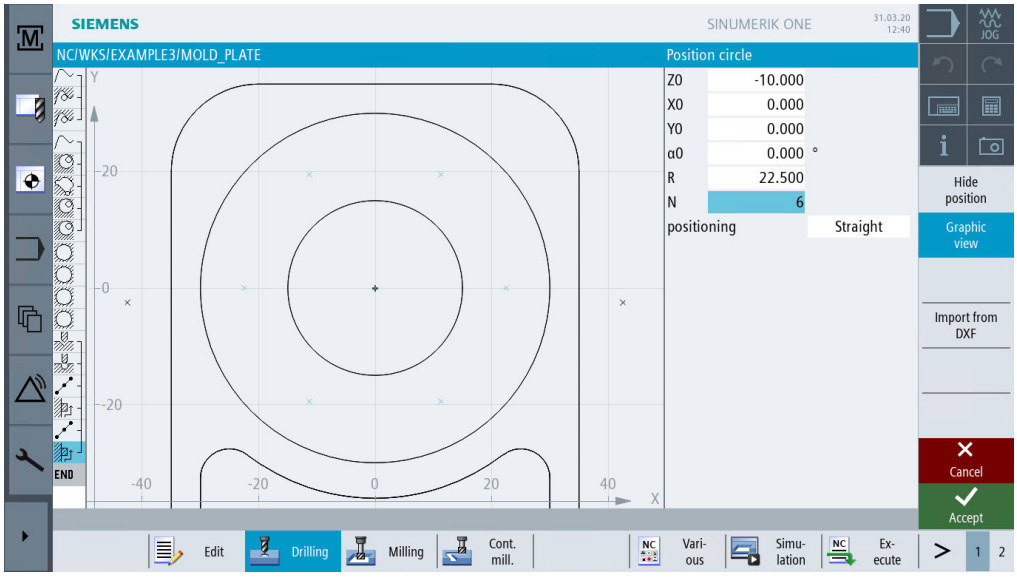


Figure 9-38 Specifying the drill holes in the full circle

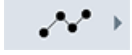
Press "Accept" to apply the values entered.

Select the **Positions** softkey.

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

Press "Accept" to apply the value entered.

Select the **Positions** softkey.



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

Note

If necessary delete any existing positions using the DEL key.

Field	Value	Toggle field	Notes
Pattern	Rectangular	X	
Z0	-10		
X0	0		
Y0	42.5		

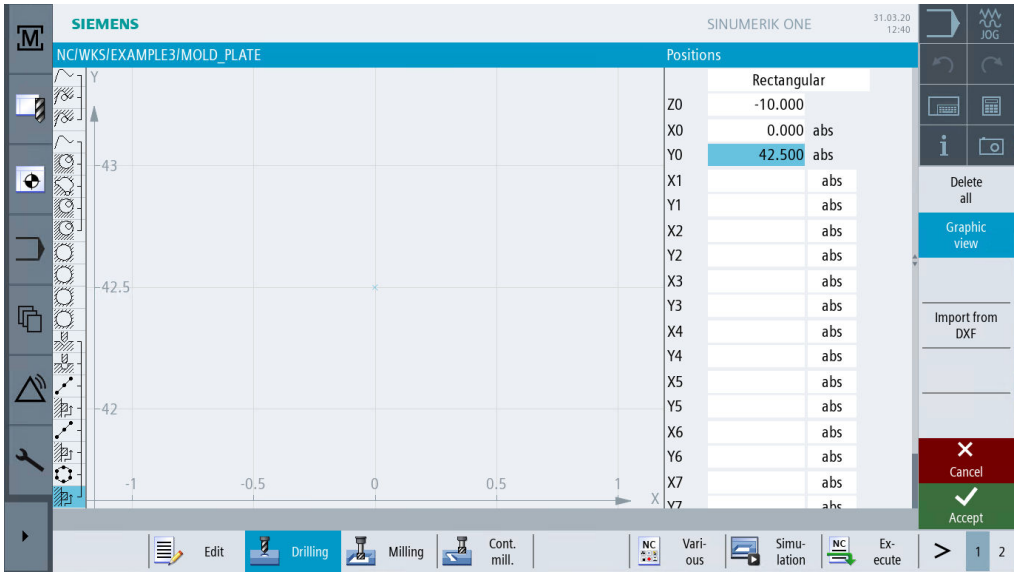


Figure 9-39 Specifying the drilling positions

Press "Accept" to apply the values entered.

Note

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

Start simulation.

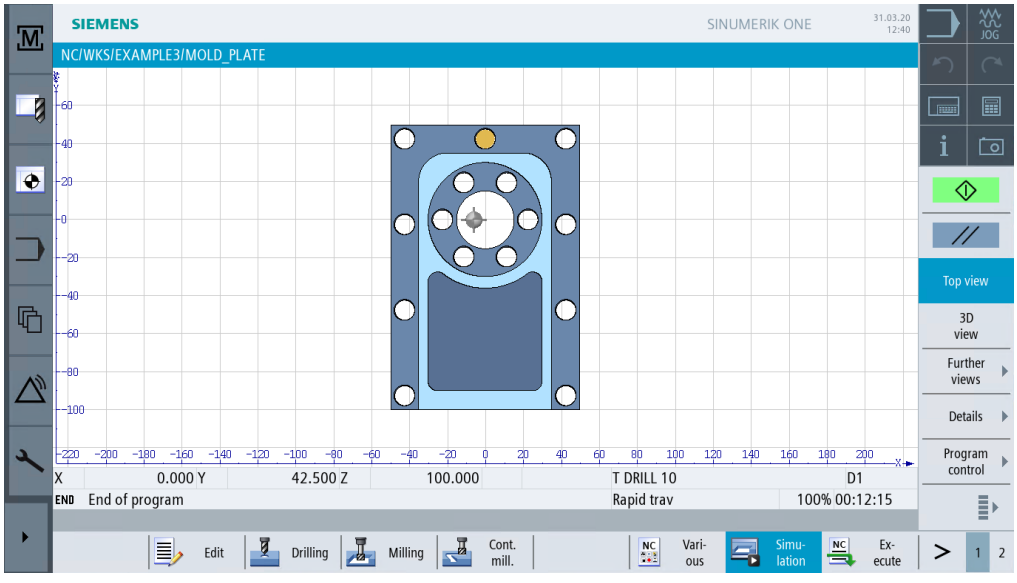


Figure 9-40 Simulation - Top view

Example 4: Lever

10

10.1 Overview

Learning objectives

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

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

Task

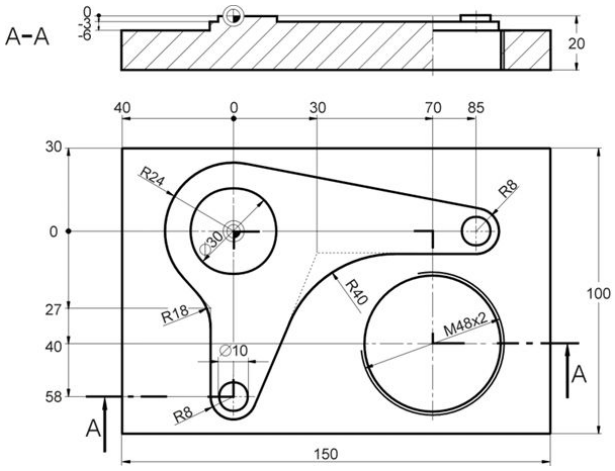


Figure 10-1 Workshop drawing - Example 4:

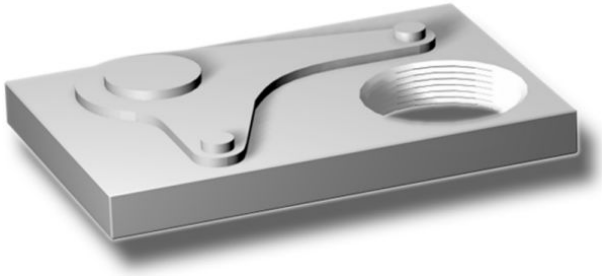


Figure 10-2 Workpiece - Example 4:

Preparation

Perform the following steps without help:

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

Note

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

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

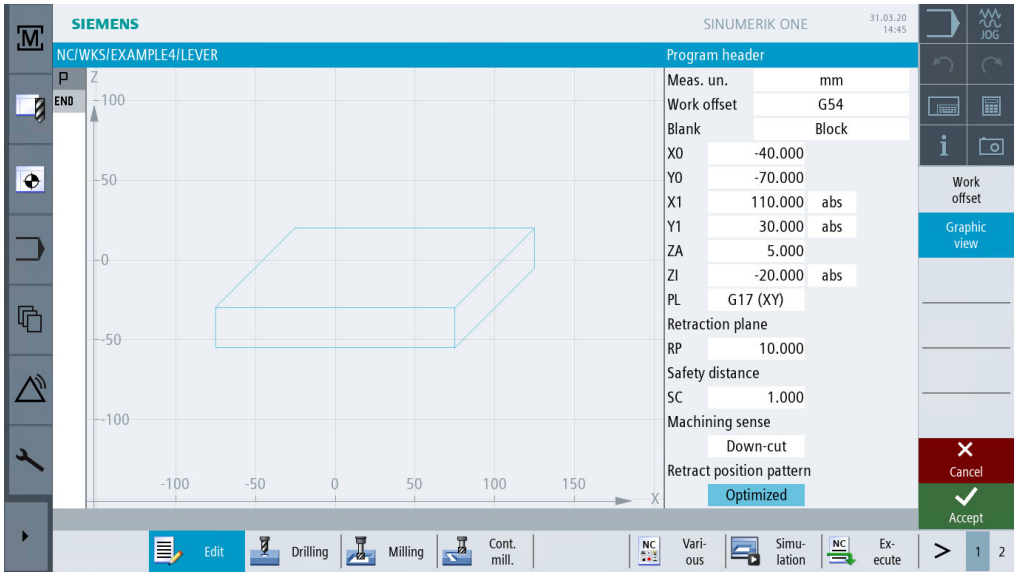


Figure 10-3 Workpiece dimensions as in the program header

10.2 Face milling

Operating sequences

Select the **Milling** softkey.

Select the **Face milling** softkey.

Open the tool list and select FACEMILL 63.

Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

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

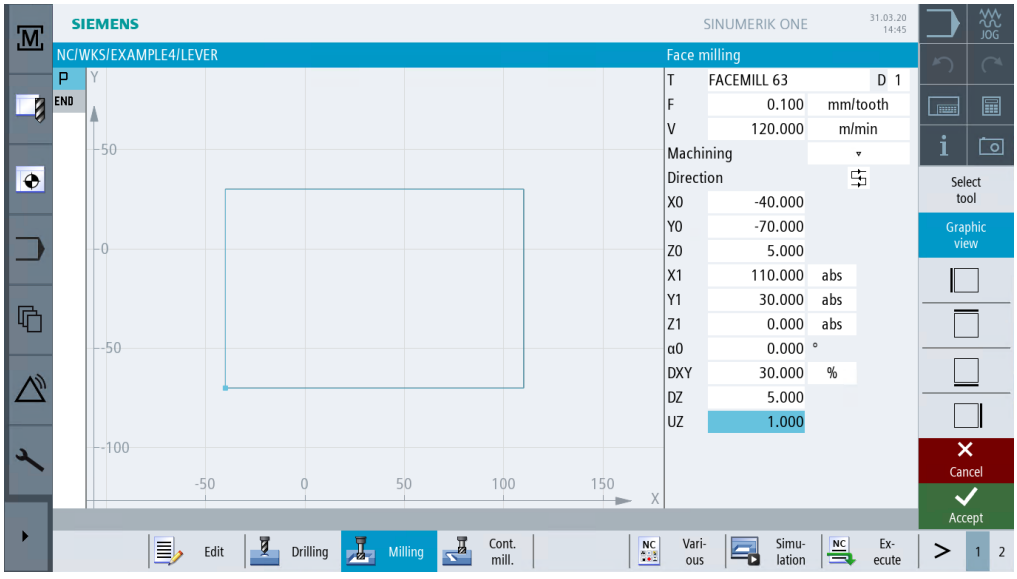


Figure 10-4 Roughing a surface

Press "Accept" to apply the values entered.

Select the **Face milling** softkey.

Enter the following values for finishing in the interactive screenform:

Field	Value	Toggle field	Notes
F	0.08 mm/tooth	X	
V	150 m/min	X	
Machining	Finishing	X	

Note

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

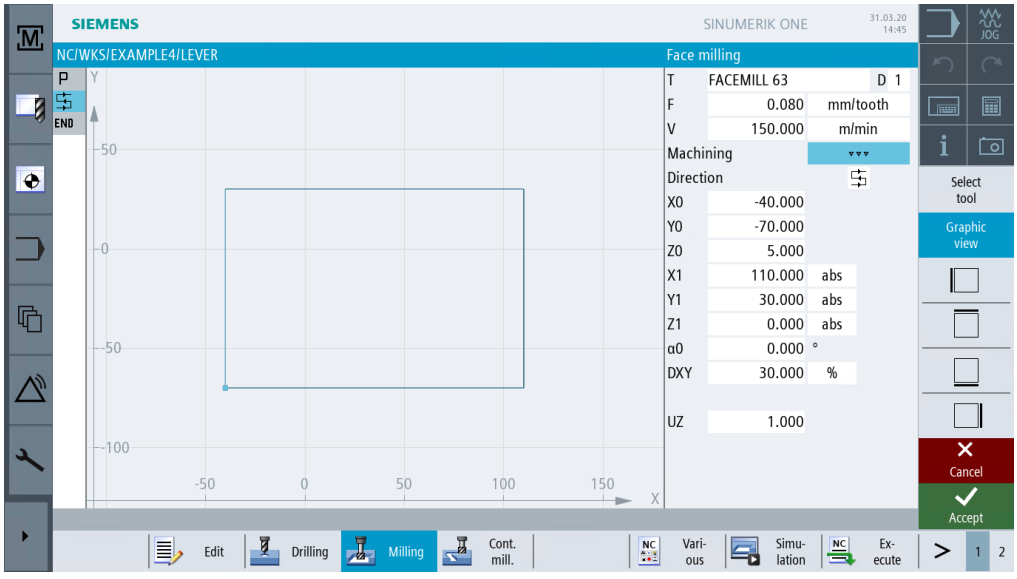


Figure 10-5 Finishing a surface

Press "Accept" to apply the values entered.

10.3 Creating an edge for the lever island

Operating sequences

Note

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

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

Select the **Contour milling** softkey.

Create a new contour with the name 'LEVER_Rectangular_Area'.

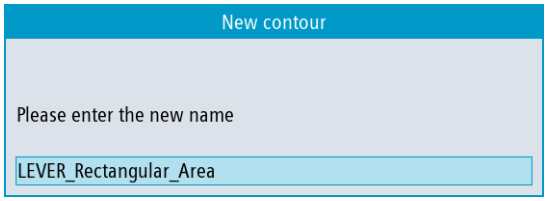


Figure 10-6 Creating the contour

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

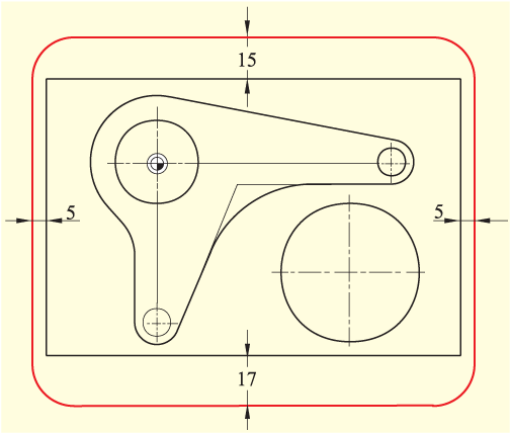


Figure 10-7 Edge for the lever island

Compare your contour with the screen below.

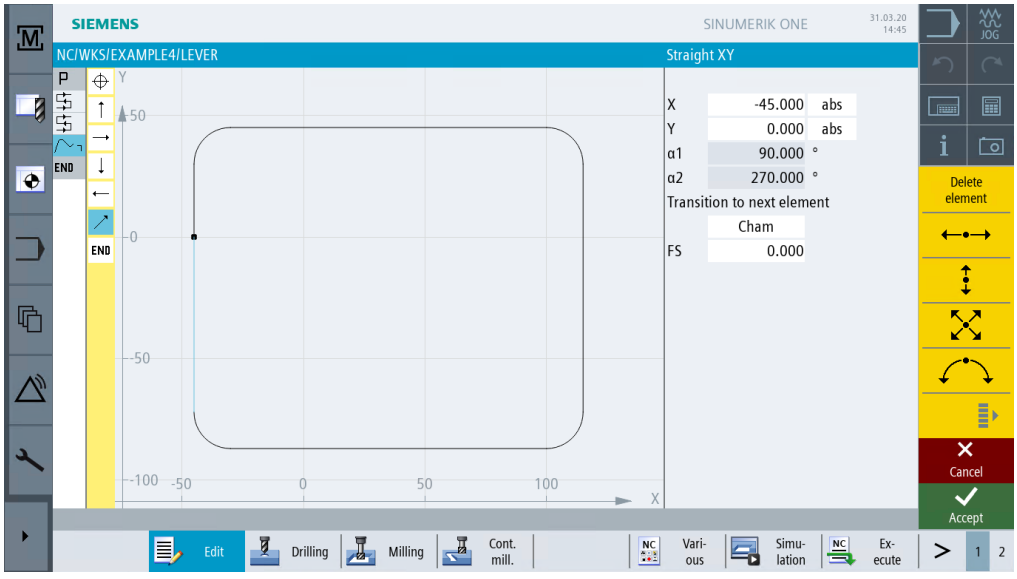


Figure 10-8 Readily designed contour

10.4 Machining the lever

Operating sequences

Proceed as follows to enter the contour:

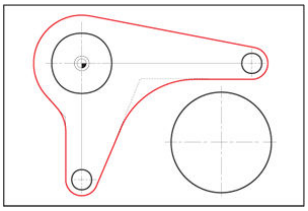


Figure 10-9 Lever contour

Select the **Contour milling** softkey.

Create a new contour with the name 'LEVER_Lever'.

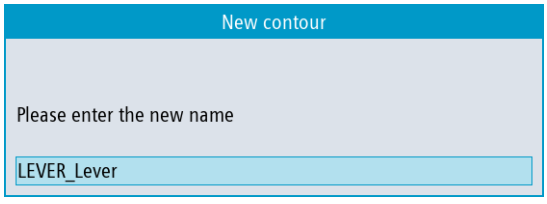


Figure 10-10 Creating the contour

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

Field	Value	Toggle field	Notes
X	-24 abs		
Y	0 abs		

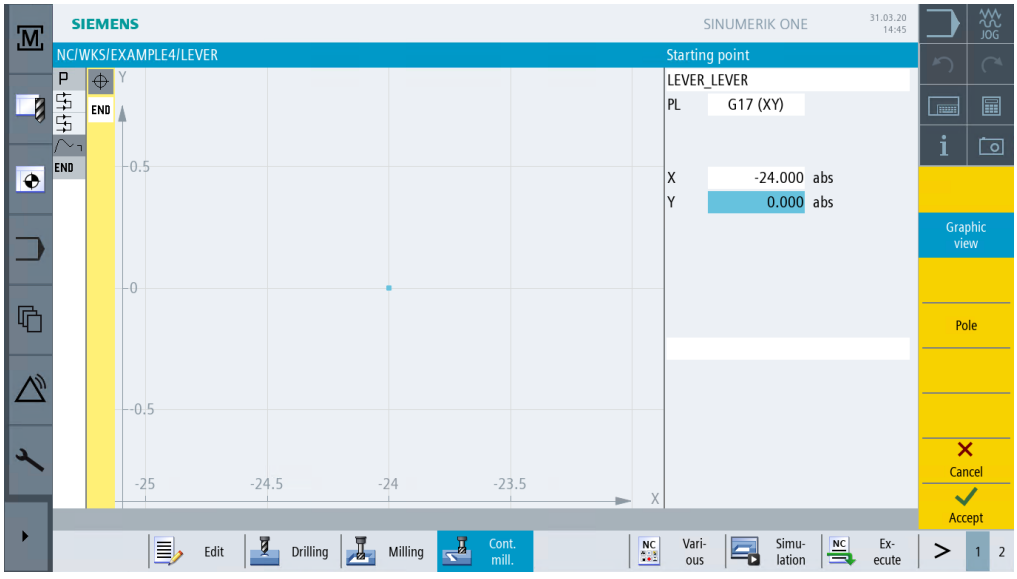


Figure 10-11 Specifying the starting point

Press "Accept" to apply the values entered.

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

Field	Value	Toggle field	Notes
Direction of rotation	Clockwise	X	
R	24		Radius and center point are known.
I	0	X	

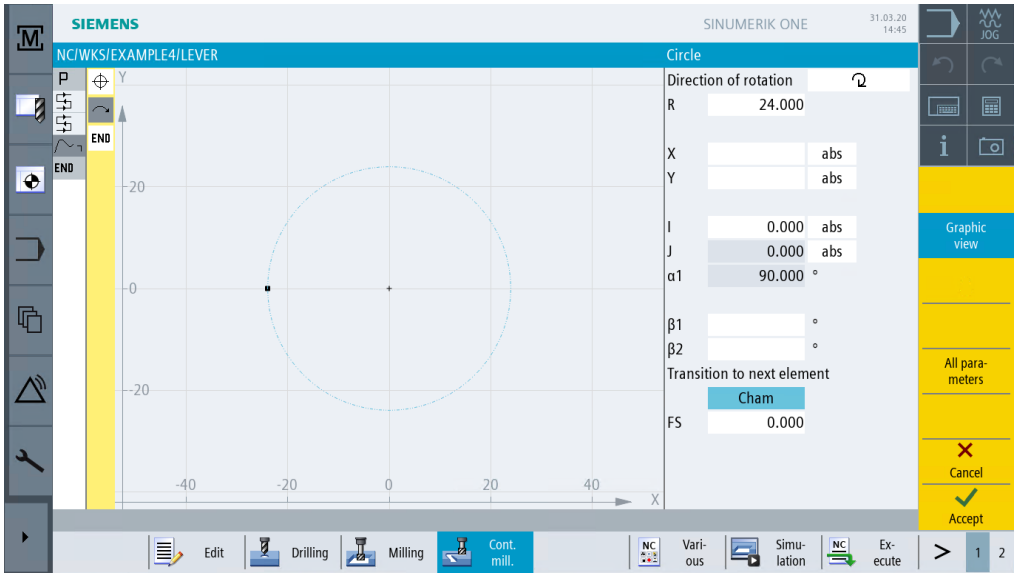


Figure 10-12 Arc contour

Press "Accept" to apply the values entered.

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

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

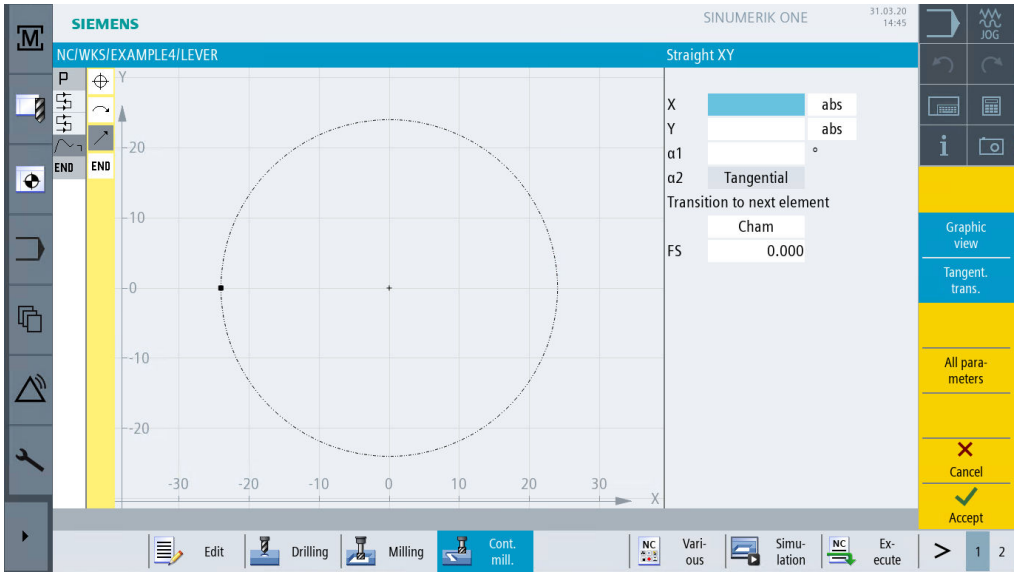


Figure 10-13 Inclined straight line contour

Press "Accept" to apply your input.

Enter the arc connected tangentially.

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

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

Field	Value	Toggle field	Notes
Direction of rotation	Clockwise	X	
R	8		Radius, center point and end point are known.
X	85 abs	X	
Y	-8 abs	X	
I	85 abs	X	

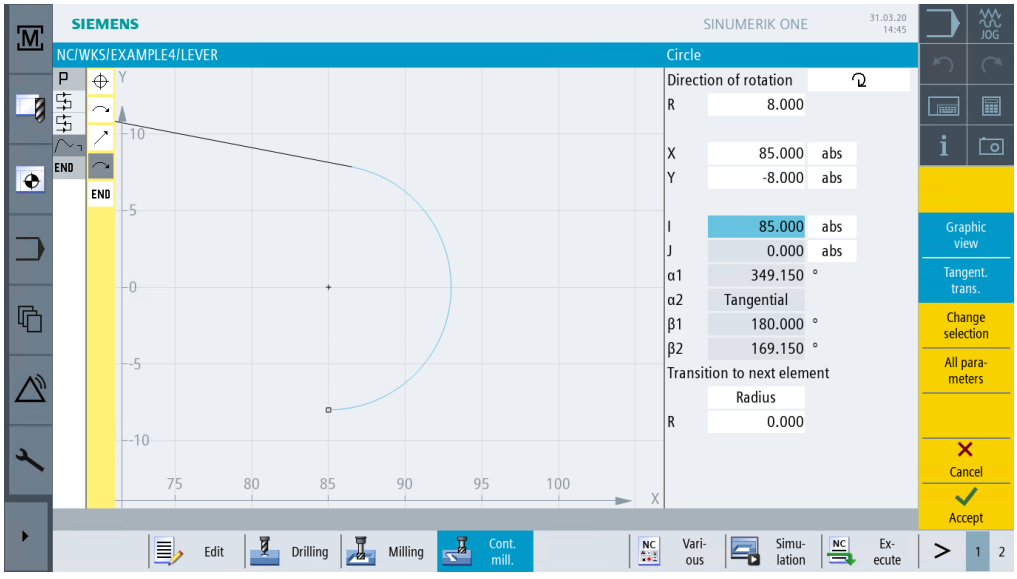


Figure 10-14 Arc contour

Press ENTER to confirm the suggested contour.

Press "Accept" to apply the values entered.

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

Field	Value	Toggle field	Notes
X	30 abs	X	
Transition to next element	Radius	X	Enter 40 mm for the radius to the subsequent element.
R	40		

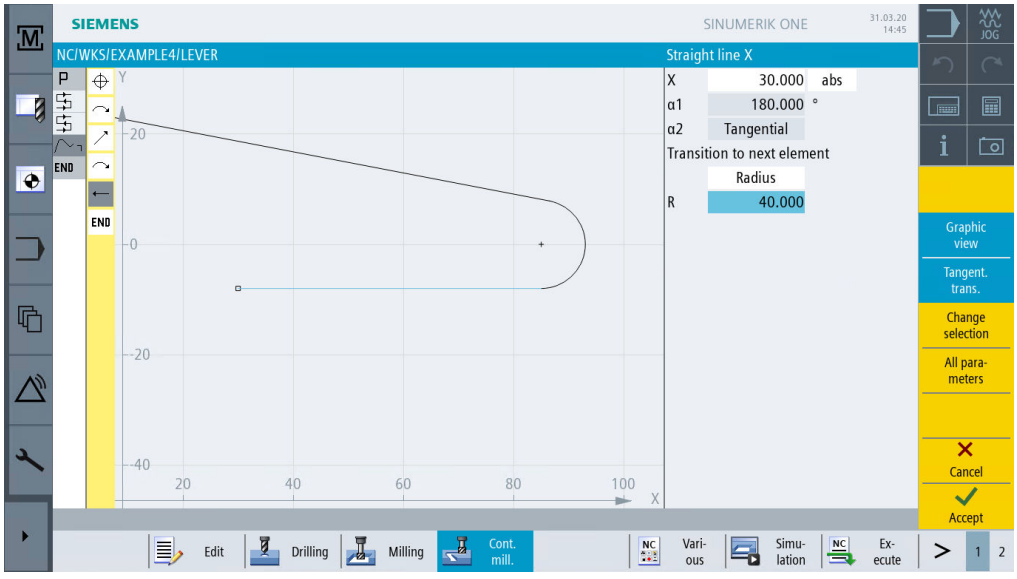


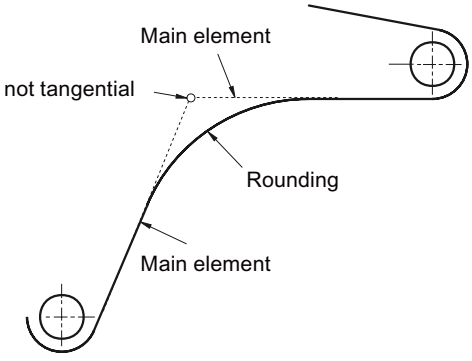
Figure 10-15 Horizontal straight line contour

Press "Accept" to apply the values entered.

Observe the following note for the next inclined straight line:

Note

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



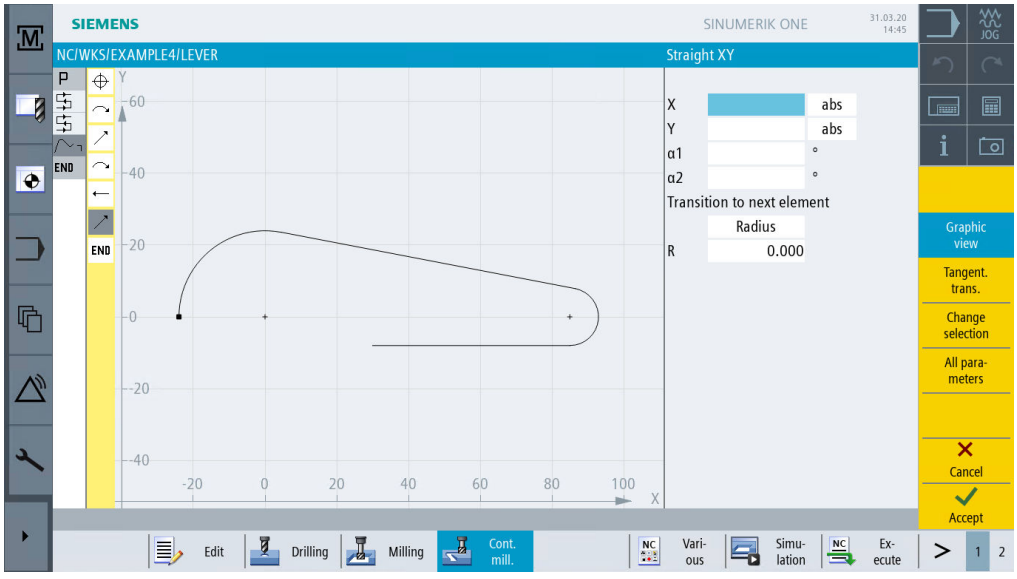


Figure 10-16 Inclined straight line contour

Press "Accept" to apply your input.

Enter the arc connected tangentially.

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

Select the **All parameters** softkey.

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

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

Field	Value	Toggle field	Notes
Direction of rotation	Clockwise	X	
R	8		
Y	-58 abs		
I	0 abs		
J	-58 abs		

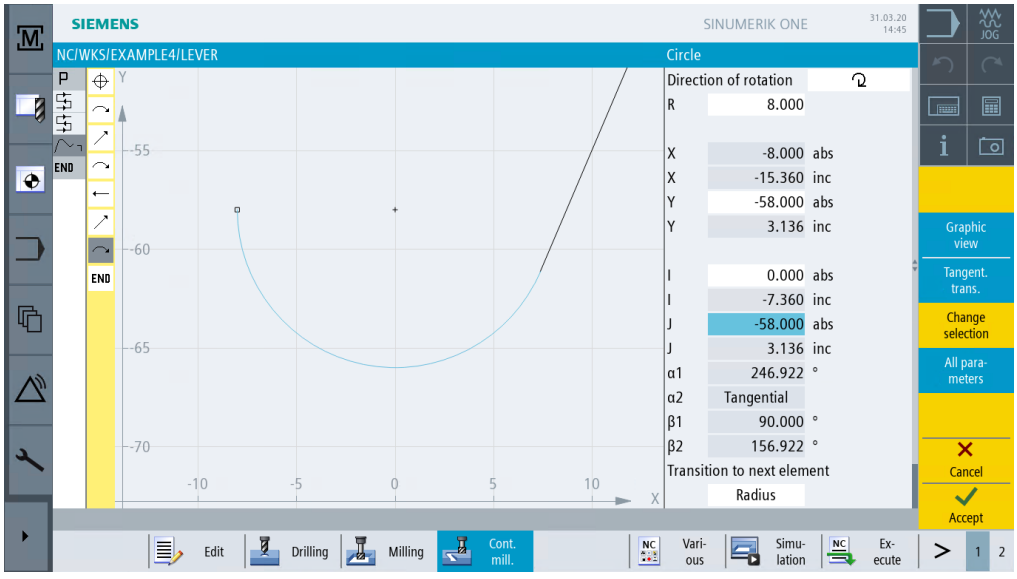


Figure 10-17 Arc contour

Select the desired contour suggestion.

Press ENTER to confirm the suggested contour.

Press "Accept" to apply your input.

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

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

Enter the following values in the interactive screenform:

Field	Value	Toggle field	Notes
Y	-27 abs	X	
Transition to next element	Radius	X	Round the transition to the subsequent straight line using R18.
R	18	X	

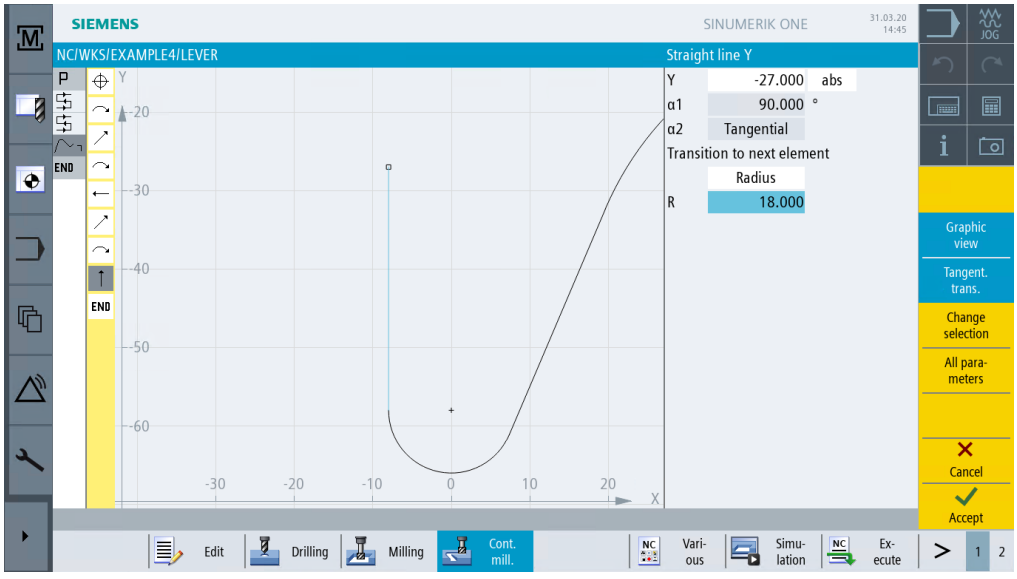


Figure 10-18 Vertical straight line contour

Press "Accept" to apply the values entered.

Specify the inclined straight line.

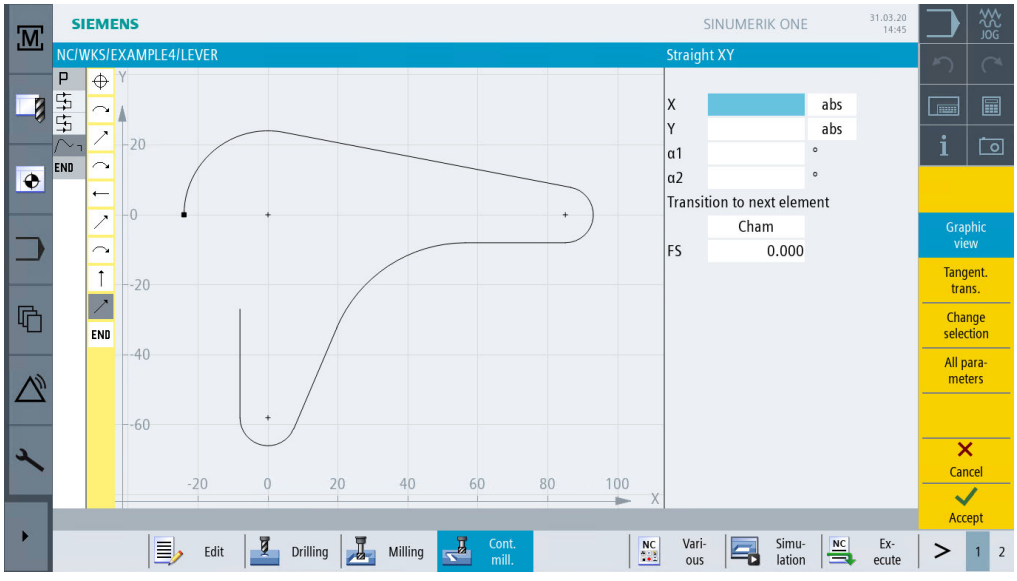


Figure 10-19 Inclined straight line contour

Press "Accept" to apply your input.

Close the contour to the starting point with an arc.

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

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

Field	Value	Toggle field	Notes
R	24		
X	-24	X	
Y	0	X	
I	0	X	

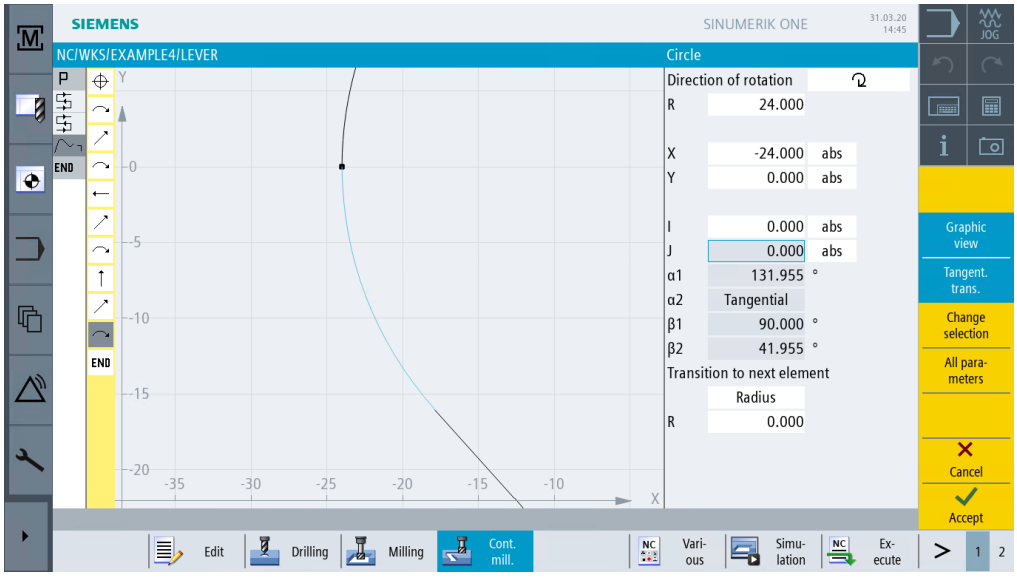


Figure 10-20 Arc contour

Press "Accept" to apply the values entered.

Accept the contour.

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

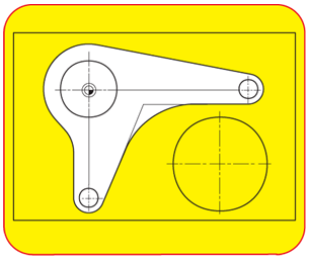


Figure 10-21 Roughing and finishing around the lever

Select the **Pocket** softkey.

Open the tool list and select CUTTER 20.

Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Toggle field	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 in-feed in the plane in %.
DZ	6		
UXY	0		
UZ	0.3		
Starting point	Automatic	X	
Insertion	Vertical	X	
FZ	0.15 mm/tooth	X	
Lift mode	To RP	X	

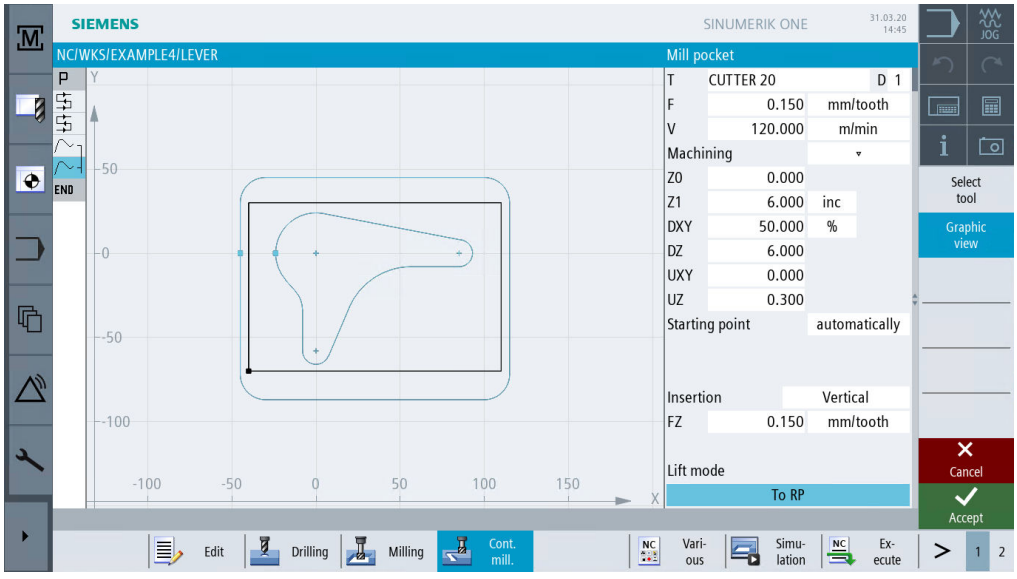


Figure 10-22 Roughing the contour

Press "Accept" to apply the values entered.

Select the **Pocket** softkey.

Enter the following values for finishing in the interactive screenform:

Field	Value	Toggle field	Notes
F	0.08 mm/tooth	X	
V	150 m/min	X	
Machining	Finishing the base	X	
Z0	0		
Z1	6 inc	X	
DXY	50%	X	Specify the maximum in-feed in the plane in %.
UXY	0		
UZ	0.3		
Starting point	Manual	X	
XS	70		
YS	-40		
Insertion	Vertical	X	
FZ	0.15 mm/tooth	X	
Lift mode	To RP	X	

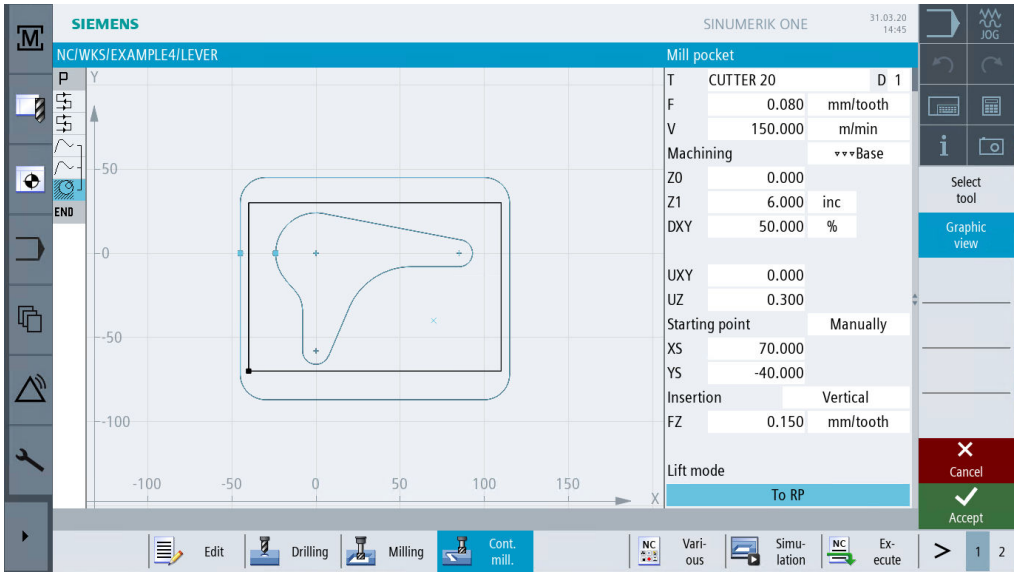


Figure 10-23 Finishing the base

Press "Accept" to apply the values entered.

10.5 Creating an edge for the circular island

Operating sequences

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

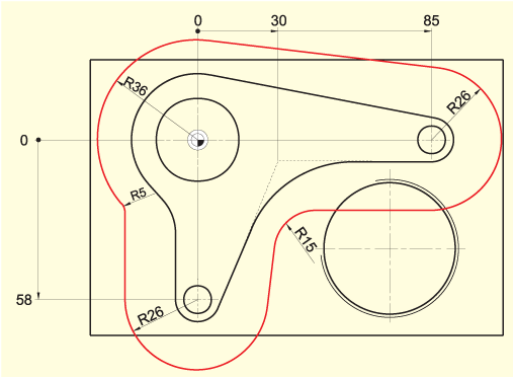


Figure 10-24 Edge contour for the circular islands

Note

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

The radii R5 and R15 are selected freely.

Select the **Contour milling** softkey.

Create a new contour with the name 'LEVER_Lever_Area'.

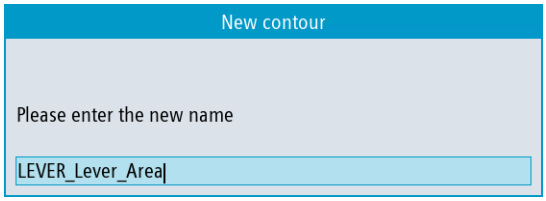


Figure 10-25 Creating the contour

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

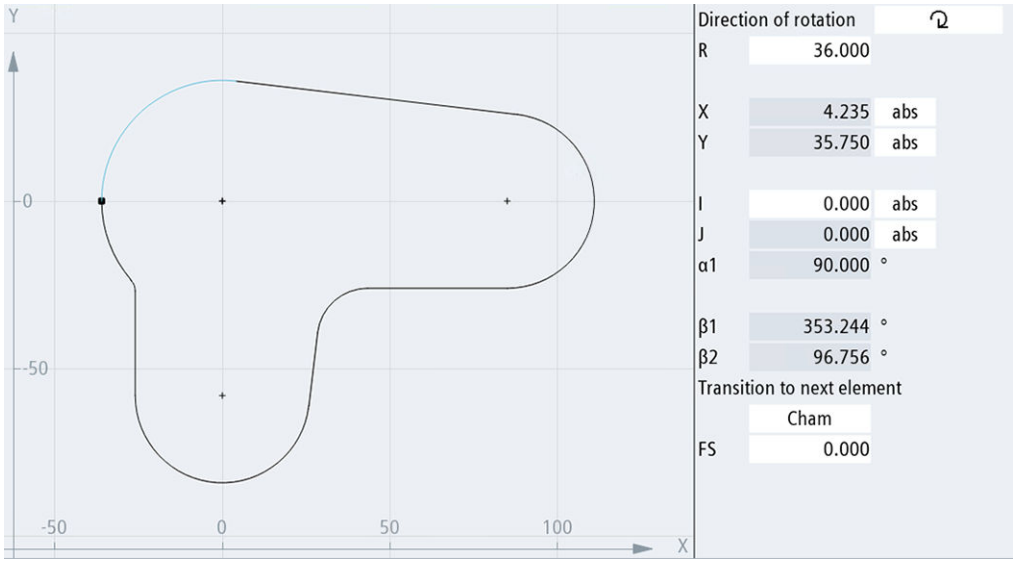


Figure 10-26 Arc contour section, left

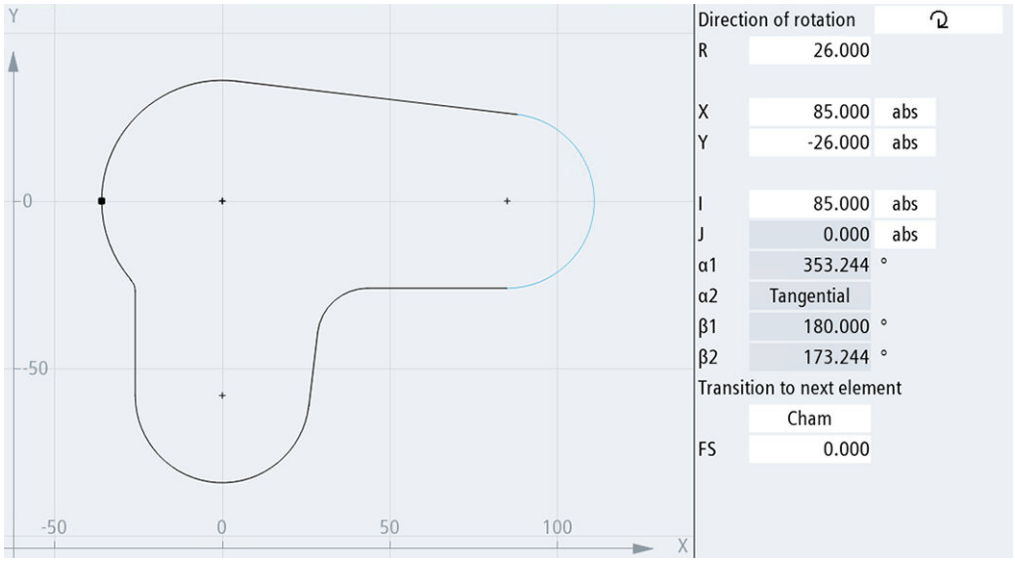


Figure 10-27 Arc contour section, right

10.6 Creating a 30mm circular island

Operating sequences

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

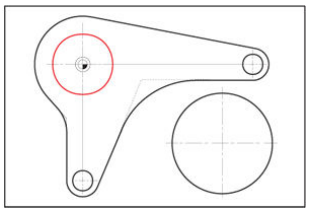


Figure 10-28 30mm circular island

Select the **Contour milling** softkey.

Create a new contour with the name 'LEVER_Circle_R15'.

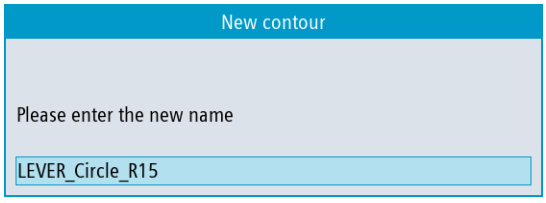


Figure 10-29 Creating the contour

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

Note

Ensure that various values must be dimensioned incrementally.

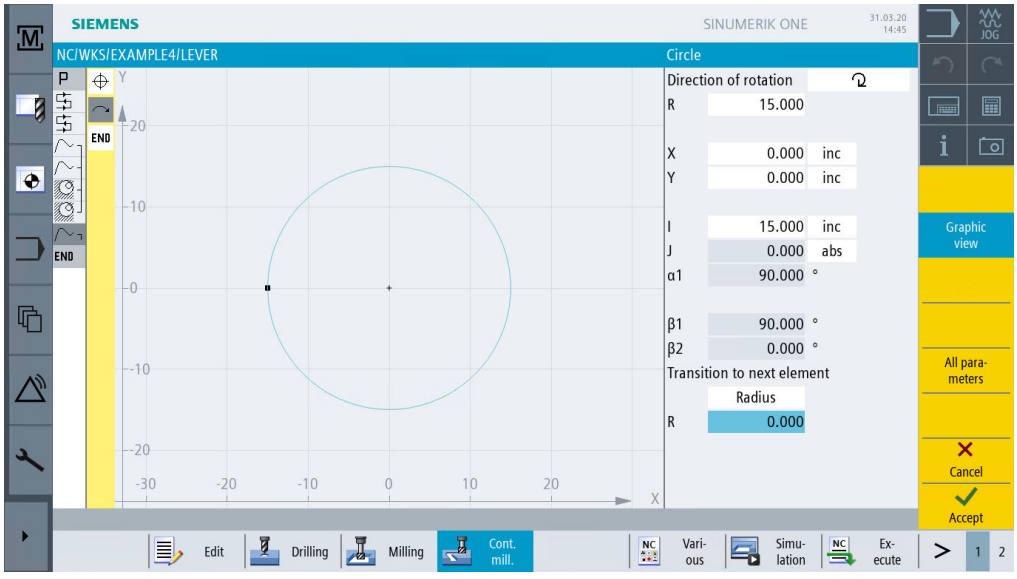


Figure 10-30 Circular island contour

10.7 Creating a 10mm circular island

Operating sequences

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

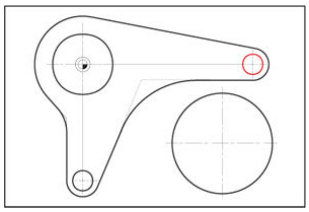


Figure 10-31 10mm circular island

Select the **Contour milling** softkey.

Create a new contour with the name 'LEVER_Circle_R5_A'.

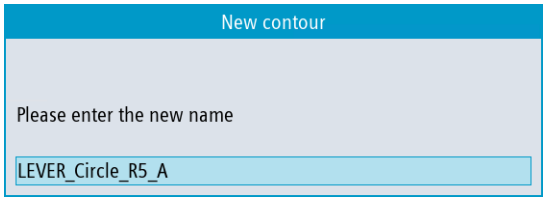


Figure 10-32 Creating the contour

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

Note

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

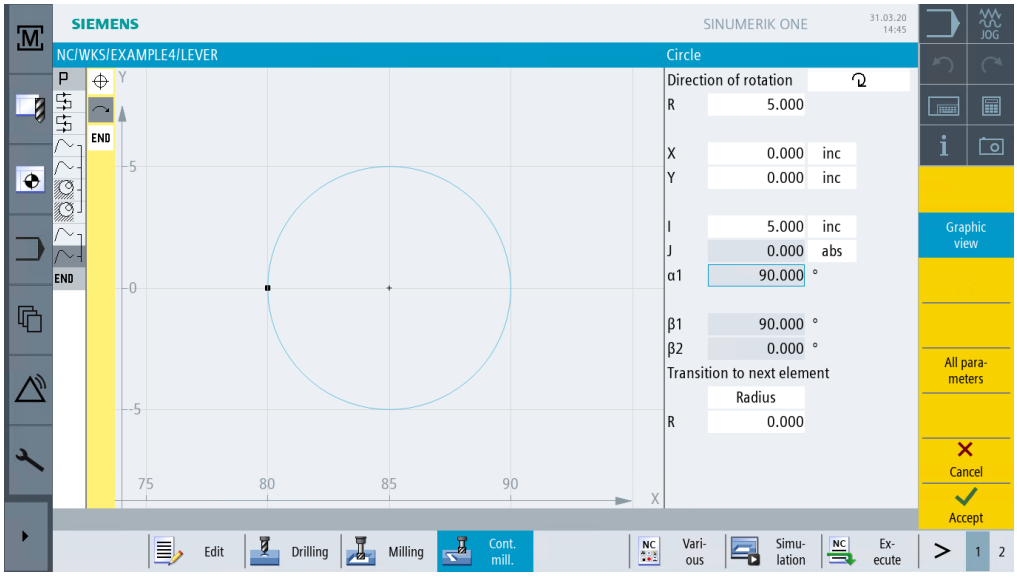


Figure 10-33 10mm circular island contour

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

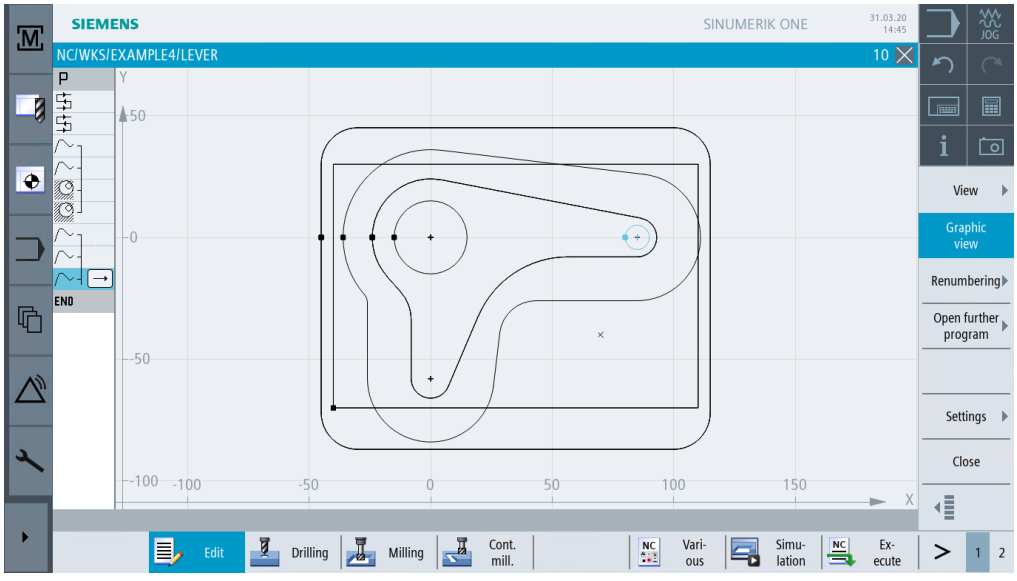


Figure 10-34 Broken-line graphics

10.8 Copying the 10mm circular island

Operating sequences

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

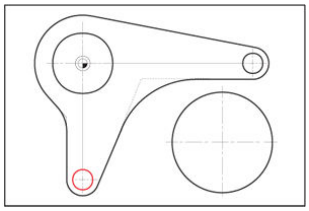


Figure 10-35 10mm circular island

Copy

Navigate and copy the 'LEVER_Circle_R5_A' contour.

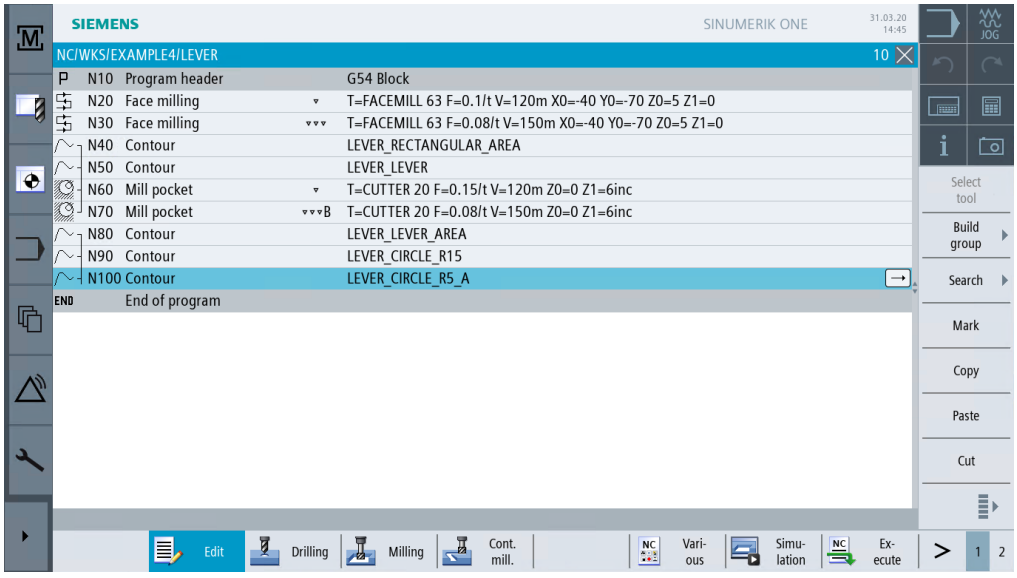


Figure 10-36 Copying the contour

Paste

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

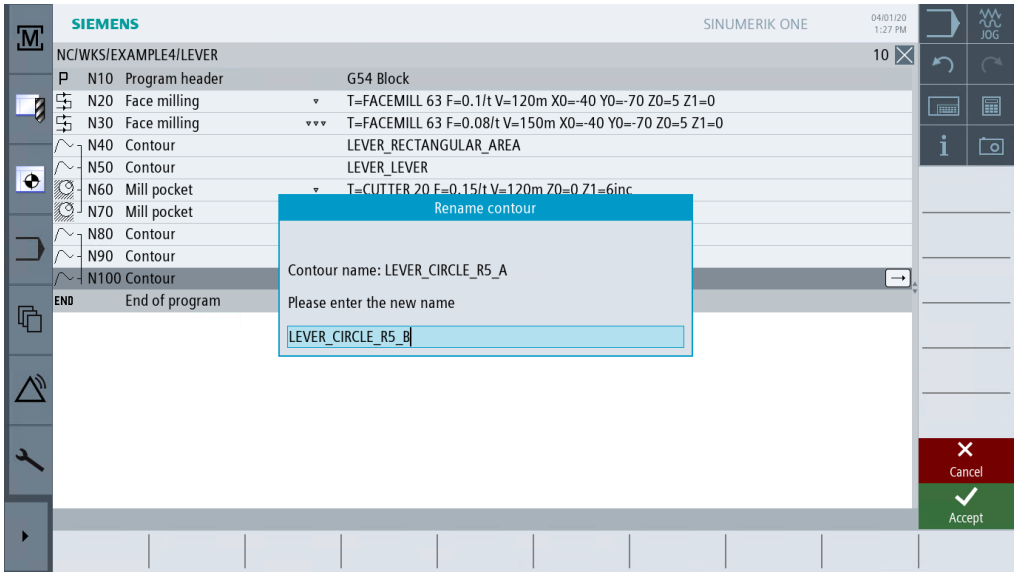


Figure 10-37 Specifying the name for the copied contour

Press "Accept" to apply your input.

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

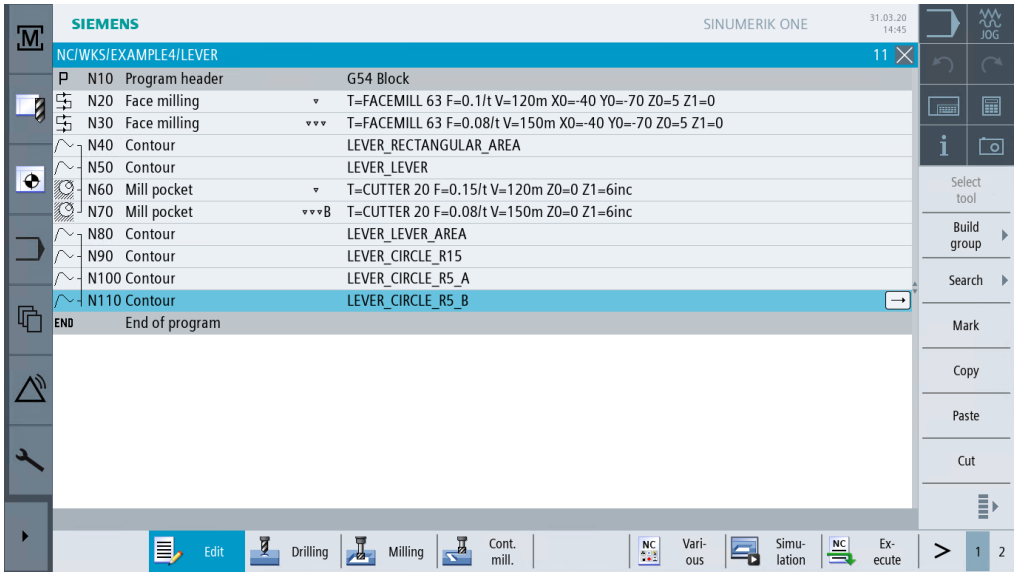


Figure 10-38 Pasted contour in the work step editor

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

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

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

Field	Value	Toggle field	Notes
X	-5		
Y	-58		

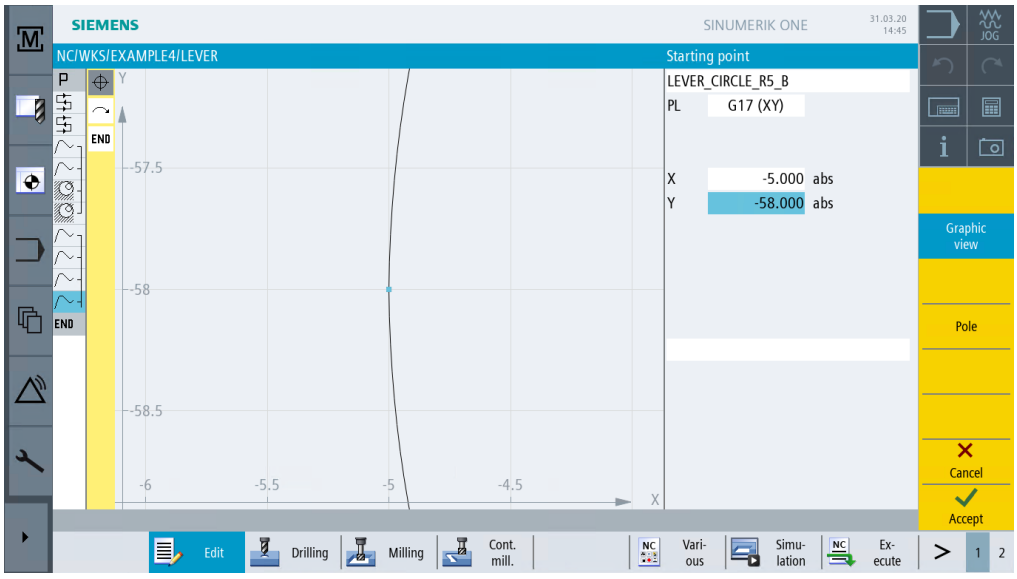


Figure 10-39 Changing the starting point

Press "Accept" to apply the values entered.

10.9 Machining the circular island using the editor

Operating sequences

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

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

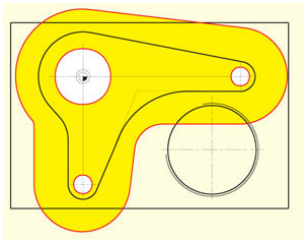


Figure 10-40 Boundary of the traversing path

Your process plan will look as follows:

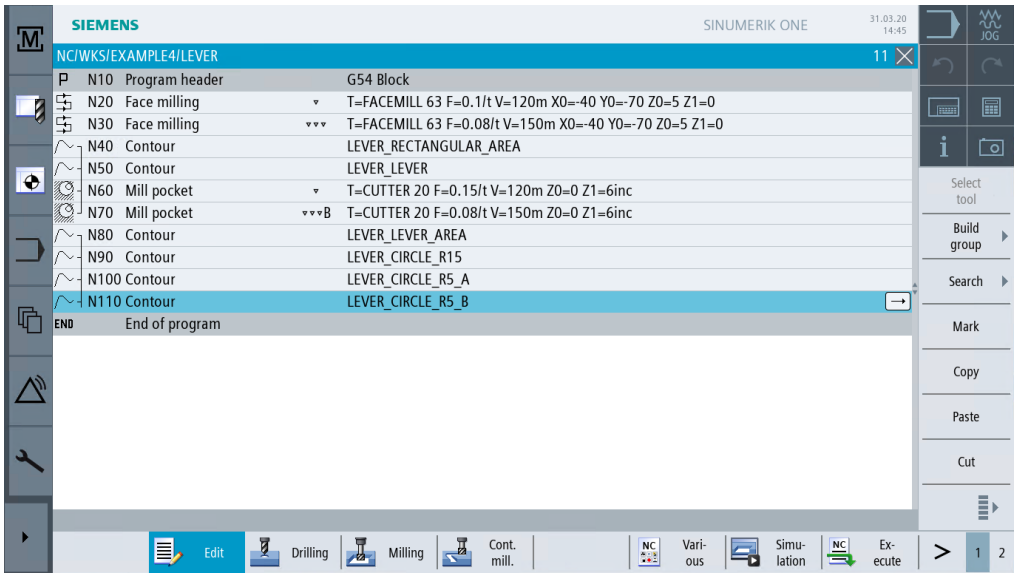


Figure 10-41 Process plan

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

Copy the highlighted work steps.

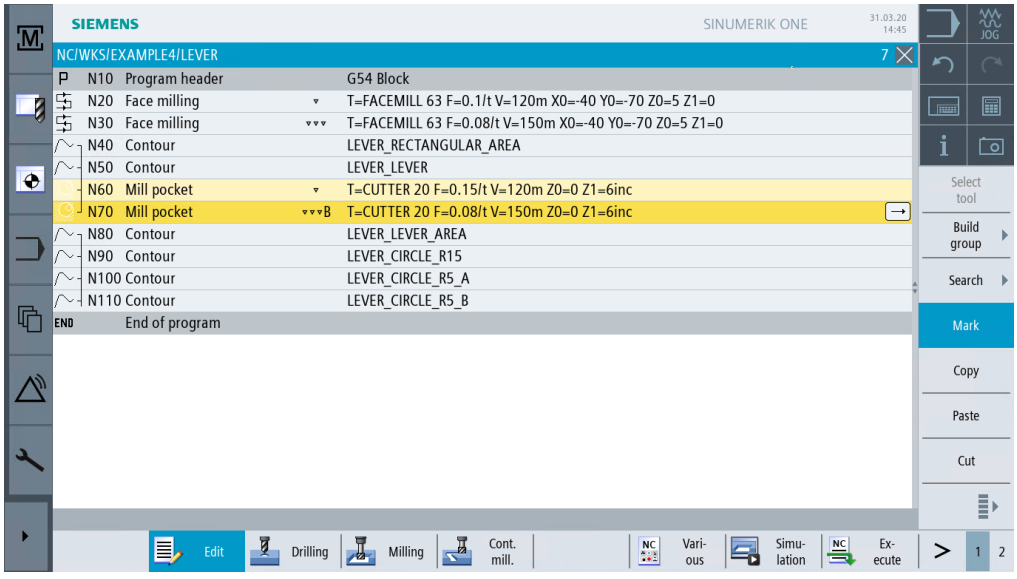


Figure 10-42 Highlighted machining steps

Paste

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

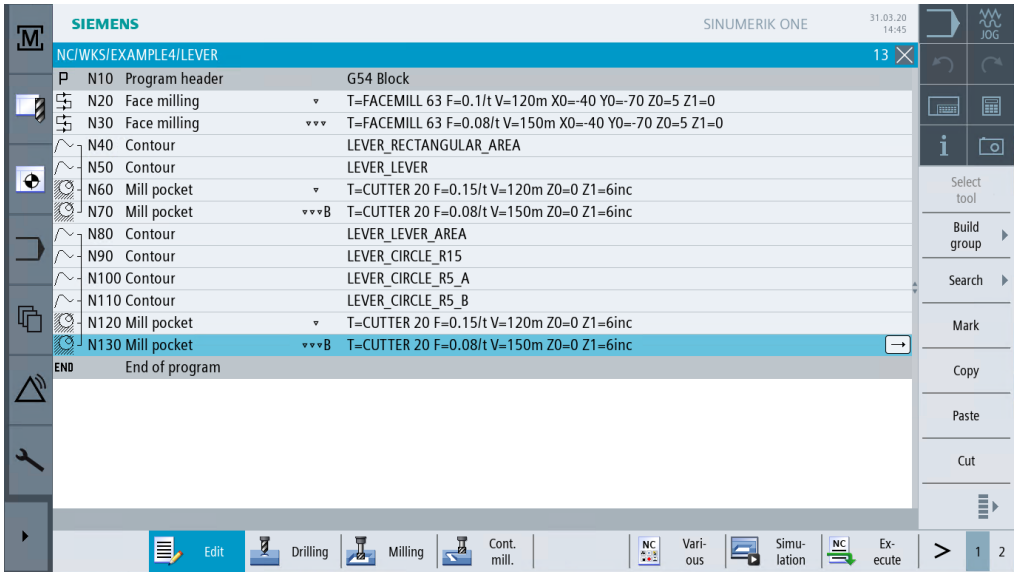


Figure 10-43 Pasted machining steps

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

Open the work step for roughing.

Enter the following values for roughing in the interactive screenform:

Field	Value	Toggle field	Notes
Z1	3 inc	X	
Starting point	Manual	X	
XS	70		
YS	-10		

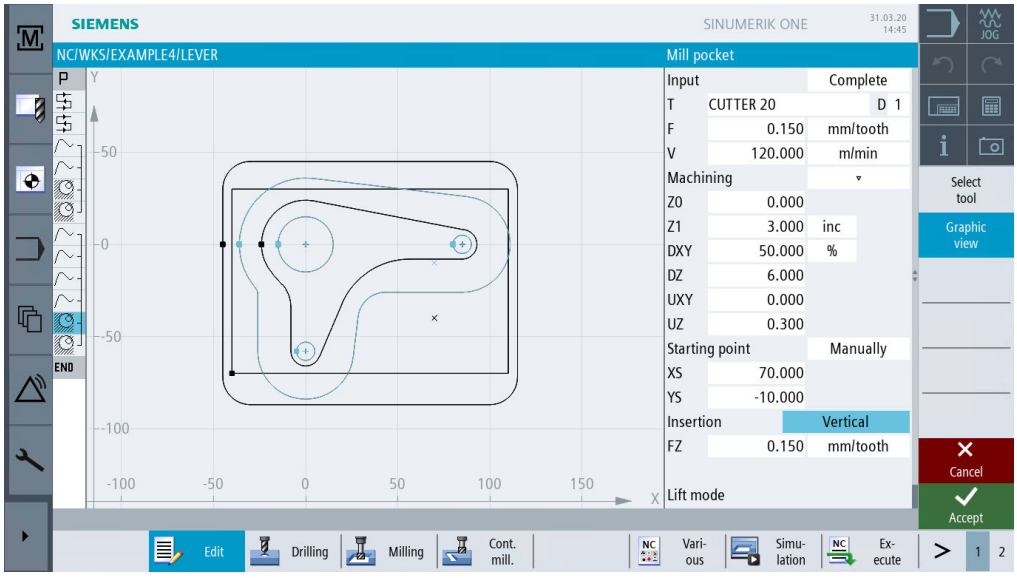


Figure 10-44 Adaptations for roughing

Press "Accept" to apply the values entered.

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

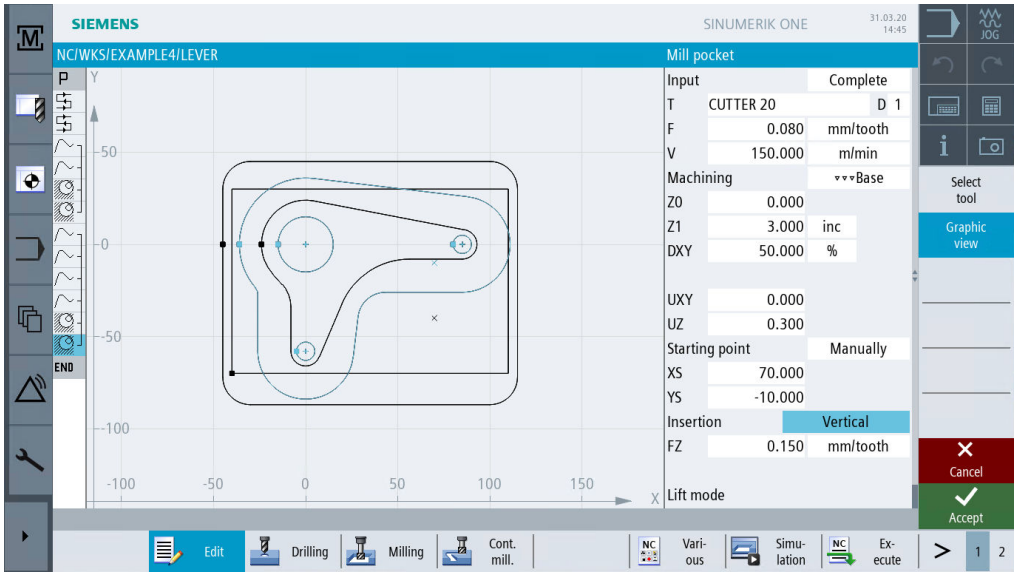


Figure 10-45 Adaptations for finishing

Press "Accept" to apply the values entered.

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

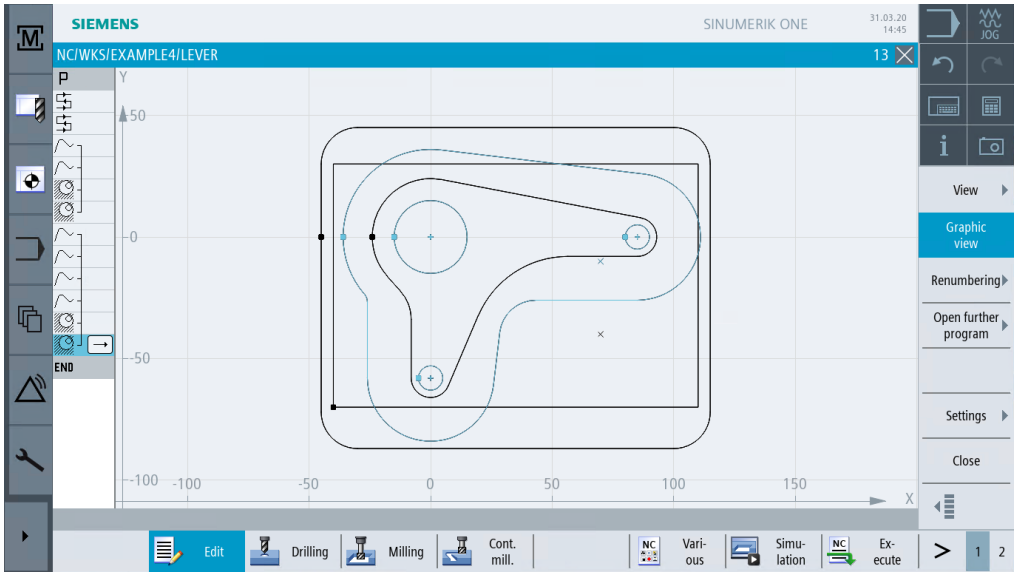


Figure 10-46 Broken-line graphics

Check your intermediate result by way of simulation.

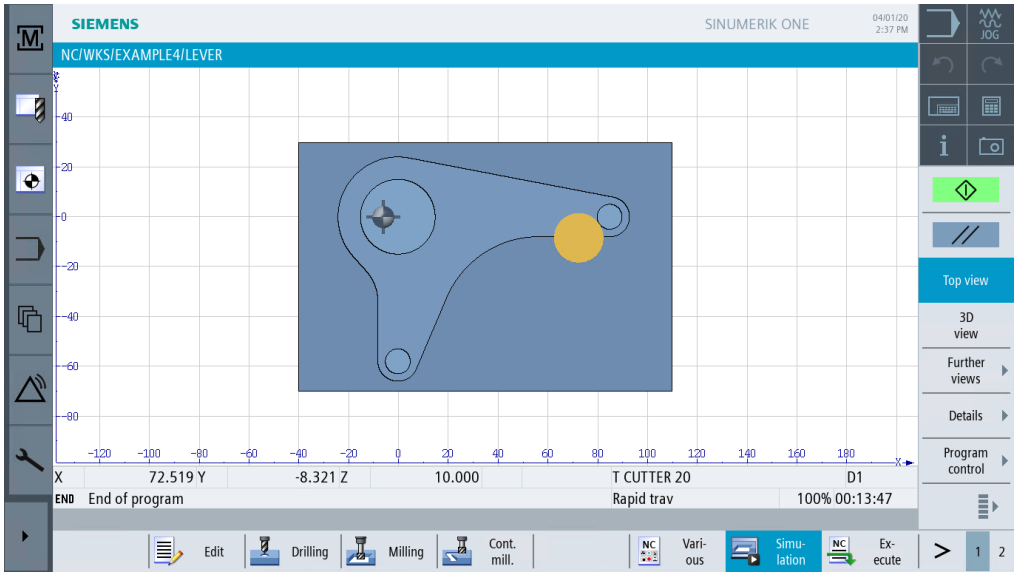


Figure 10-47 Simulation - Top view

Functions of the work step editor

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

Search	Use this softkey to search for texts in the program.
Mark	Use this softkey to select several work steps for further processing (e.g., "Copy" or "Cut").
Copy	Use this softkey to copy work steps to the clipboard.
Paste	Use this softkey to paste work steps from the clipboard to the process plan. The copied step is always inserted after the currently highlighted step.
Cut	Use this softkey to copy work steps to the clipboard; at the same time, it is deleted at its origin. This softkey can also be used for "pure" deletion.
	Use this softkey to switch to the extended menu.
Graphic view	Use this softkey to switch to the broken-line graphics.
Renumbering	Use this softkey to renumber the work steps.
Settings	Use this softkey to open the "Settings" dialog. Here you can specify, e.g. automatic numbering or whether you wish the end of the block to be represented as a symbol.
	Use this softkey to return to the previous menu.

10.10 Deep hole drilling

Operating sequences

Proceed as follows for rough-boring:

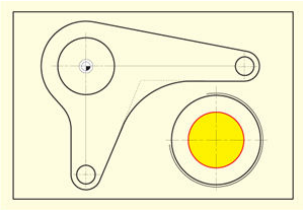


Figure 10-48 Deep hole drilling

Select the **Drilling** softkey.

Select the **Drilling Reaming** softkey.

Open the tool list and select PREDRILL 30.

Accept the tool into your program.

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

Field	Value	Toggle field	Notes
F	0.1 mm/rev	X	
V	120 m/min	X	
Depth reference	Tip	X	
Z1	-21 abs	X	
DT	0 s	X	

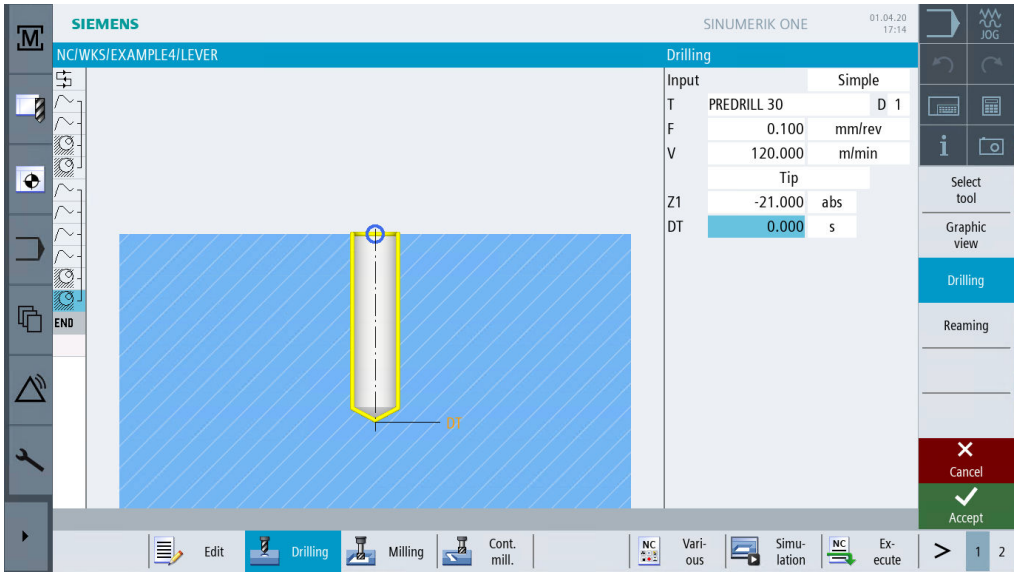


Figure 10-49 Specifying the drill hole

Press "Accept" to apply the values entered.

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

Field	Value	Toggle field	Notes
Positions	Rectangular	X	
Z0	-6		
X0	70		
Y0	-40		

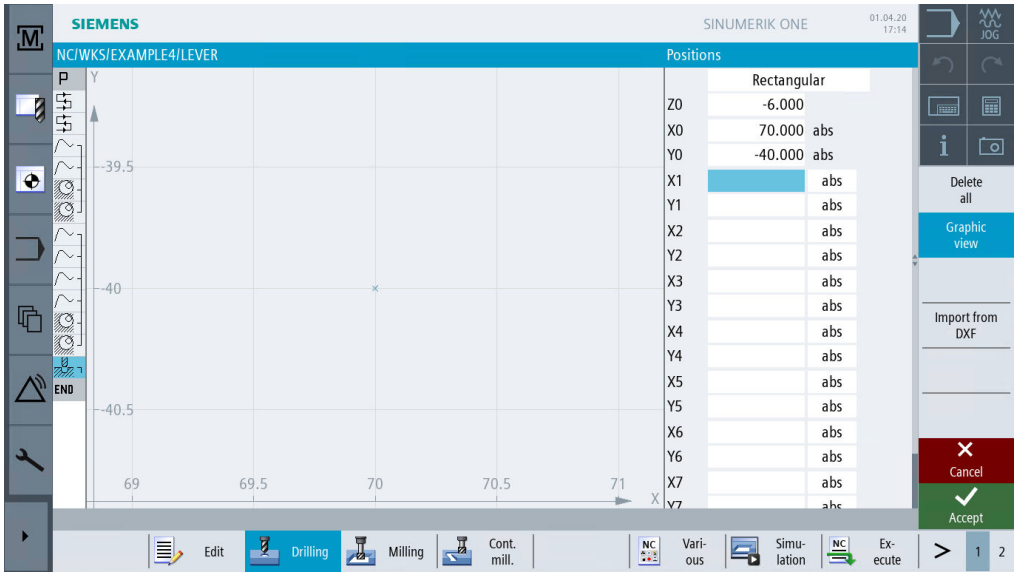


Figure 10-50 Entering the position

Press "Accept" to apply the values entered.

10.11 Milling a helix

Operating sequences

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

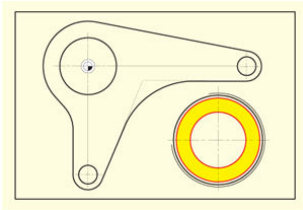


Figure 10-51 Milling a helix

Select the **Straight line Circle** softkey.

Open the tool list and select CUTTER 20 .

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

Field	Value	Toggle field	Notes
V	120 m/min	X	

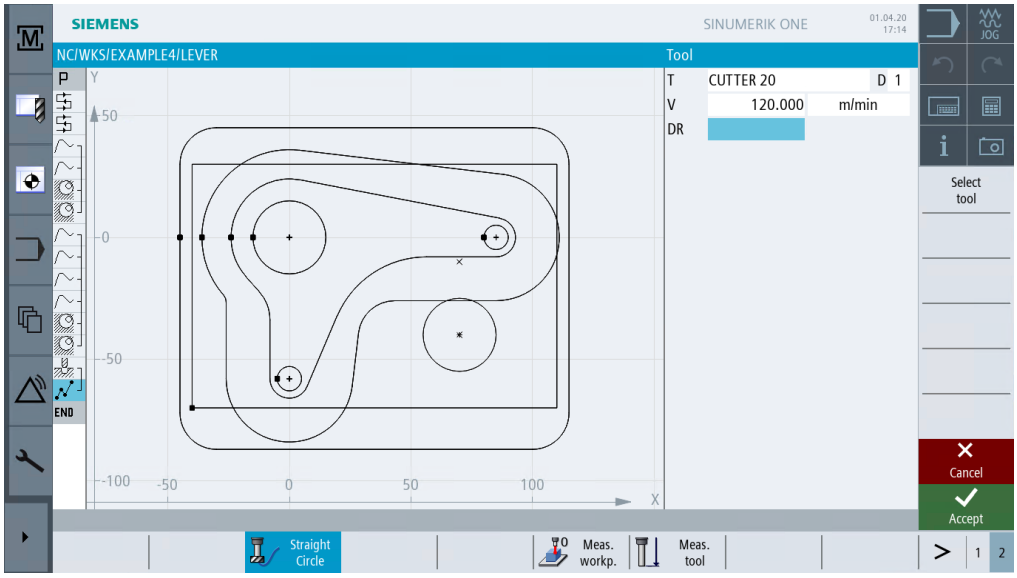


Figure 10-52 Milling a helix

Press "Accept" to apply your input.

Select the **Straight line** softkey.

Select the **Rapid traverse** softkey.

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

Note

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

Field	Value	Toggle field	Notes
X	82 abs	X	
Y	-40 abs	X	
Z	-5 abs	X	
Radius compensation	off	X	

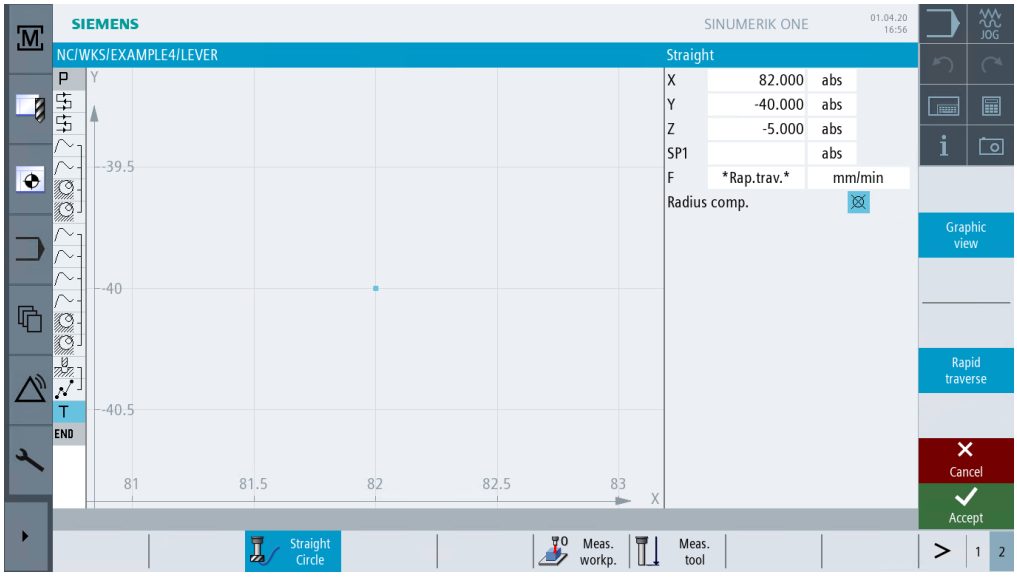


Figure 10-53 Positioning

Press "Accept" to apply the values entered.

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

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

Note

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

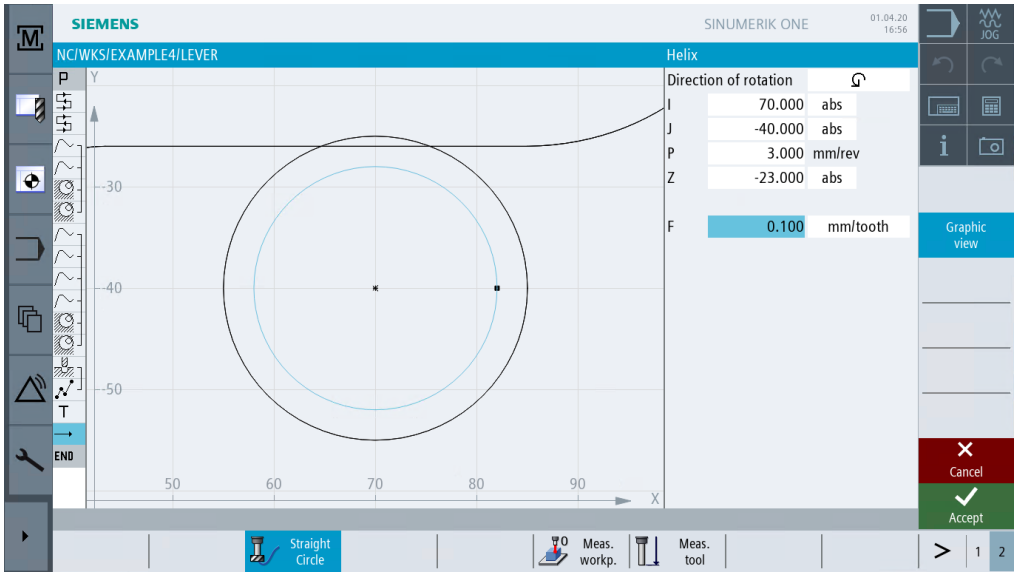


Figure 10-54 Specifying the helix

Press "Accept" to apply the values entered.

10.12 Boring

Operating sequences

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

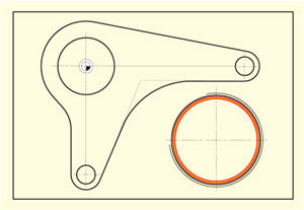
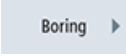


Figure 10-55 Boring a circular pocket

Select the **Drilling** softkey.



Select the **Boring** softkey.

Open the tool list and select the boring tool DRILL_TOOL.

Accept the tool into your program.

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

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

Note

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

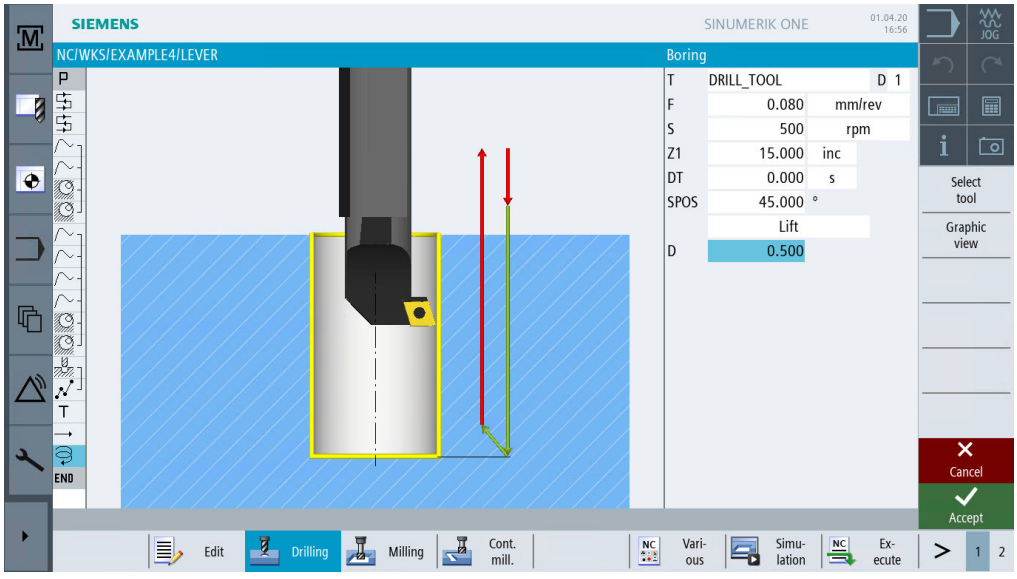


Figure 10-56 Boring

Press "Accept" to apply the values entered.

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 interactive screenform:

Field	Value	Toggle field	Notes
Z0	-6		
X0	70		
Y0	-40		

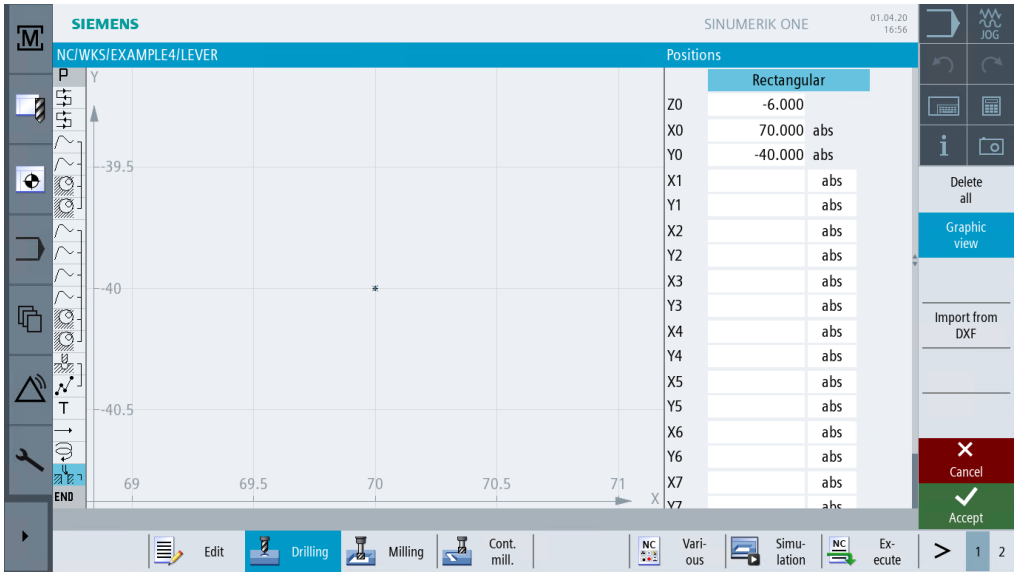


Figure 10-57 Positioning

Press "Accept" to apply the values entered.

10.13 Thread milling

Operating sequences

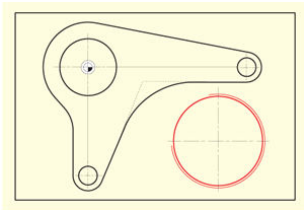


Figure 10-58 Thread milling

Select the **Milling** softkey.

Select the **Thread milling** softkey.

Open the tool list and select THREAD CUTTER .

Accept the tool into your program.

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

The help displays are very useful for your inputs.

Compare your inputs with the screen below.

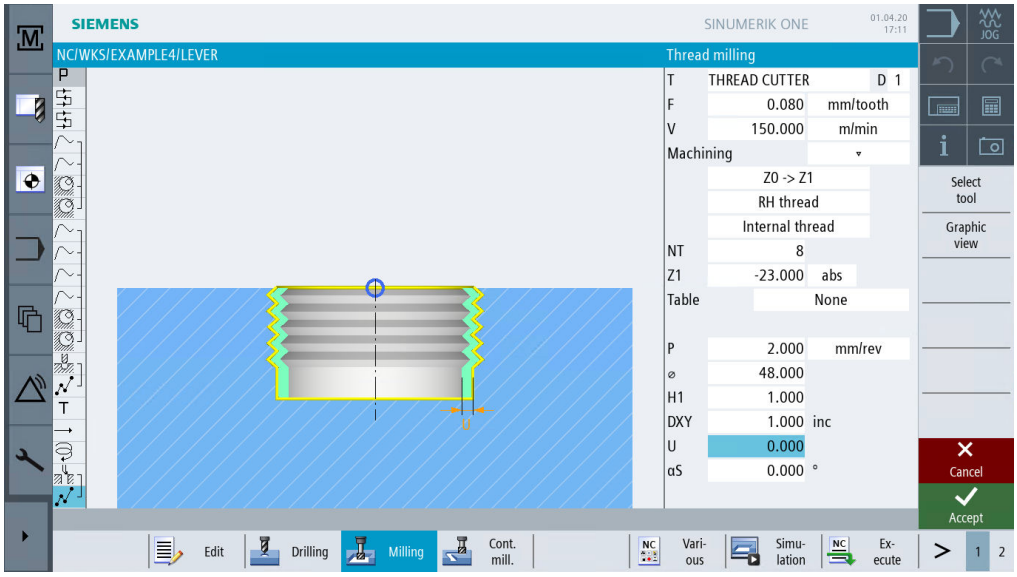


Figure 10-59 Thread milling

Press "Accept" to apply the values entered.

Specify the position for the thread.

Enter the following values in the interactive screenform:

Field	Value	Toggle field	Notes
Z0	-6		
X0	70		
Y0	-40		

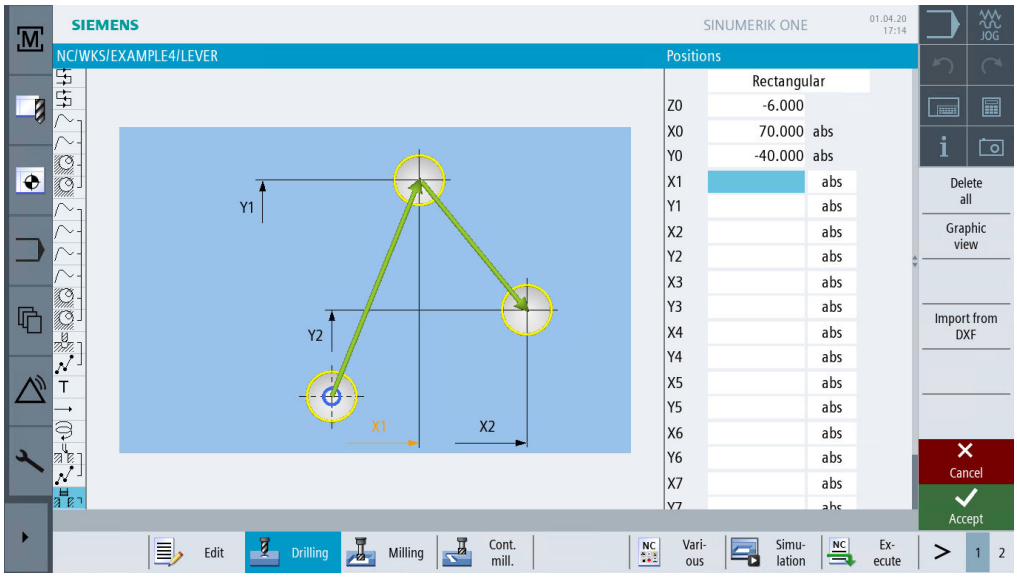


Figure 10-60 Entering the position

Press "Accept" to apply the values entered.

10.14 Programming a contour using polar coordinates

Programming with polar coordinates

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

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

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

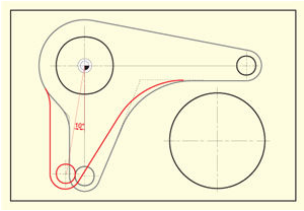


Figure 10-61 Programming the lever using polar coordinates

Operating sequences

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

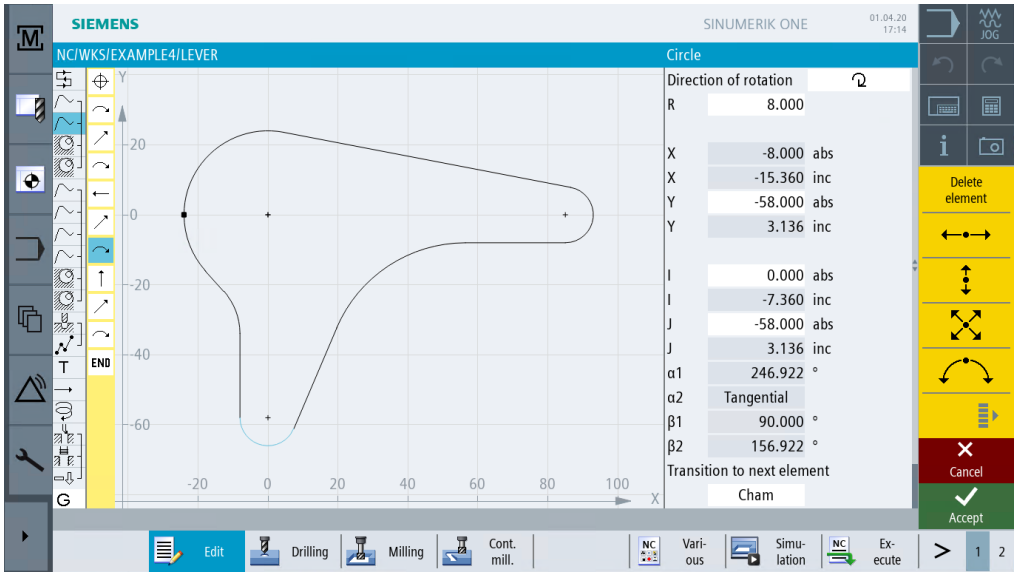


Figure 10-62 Positioning the cursor on the arc

Extend the menu.

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

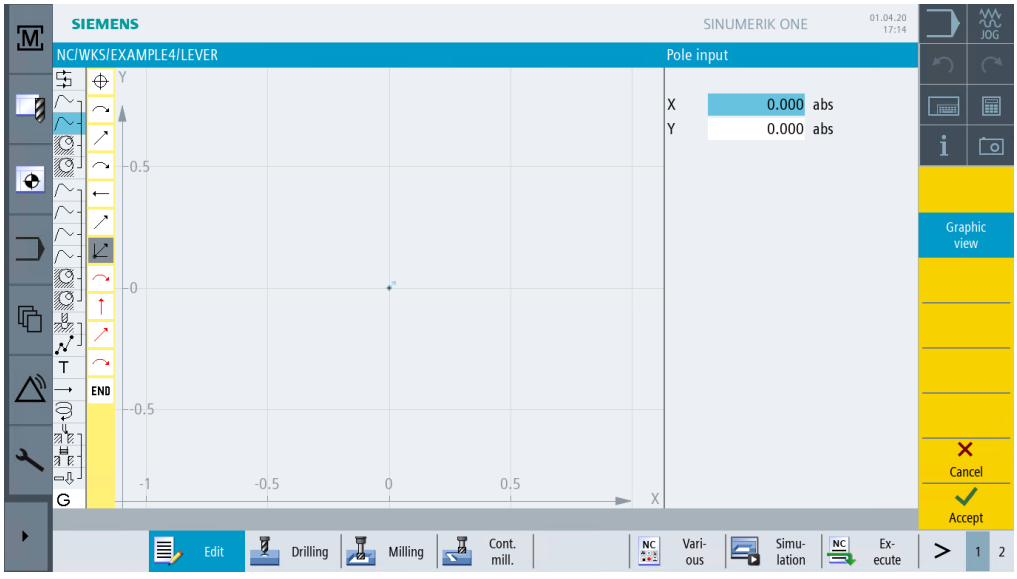


Figure 10-63 Specifying the pole

Press "Accept" to apply your input.

Subsequently, change the values matching the arc:

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

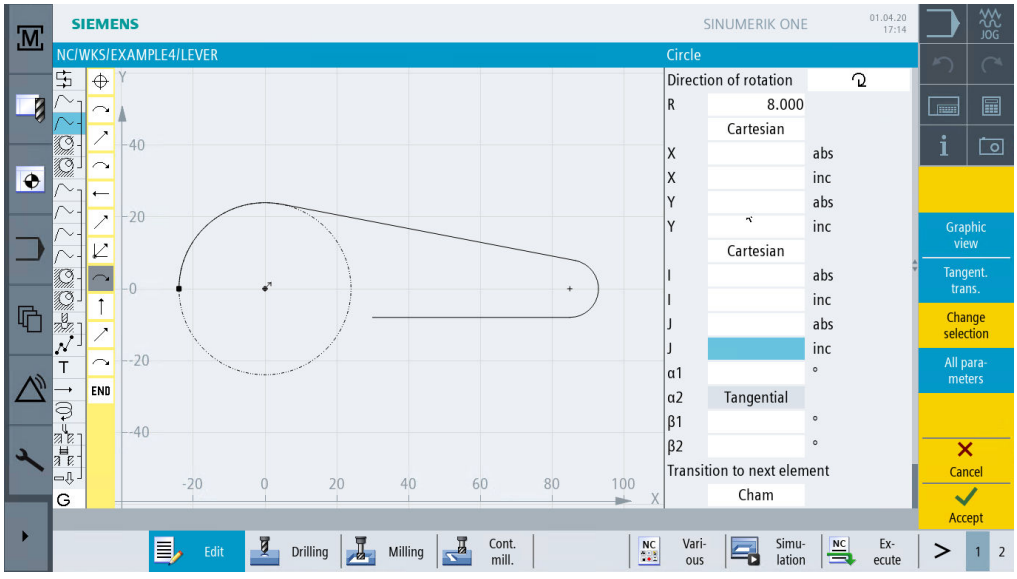


Figure 10-64 Deleting the values

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

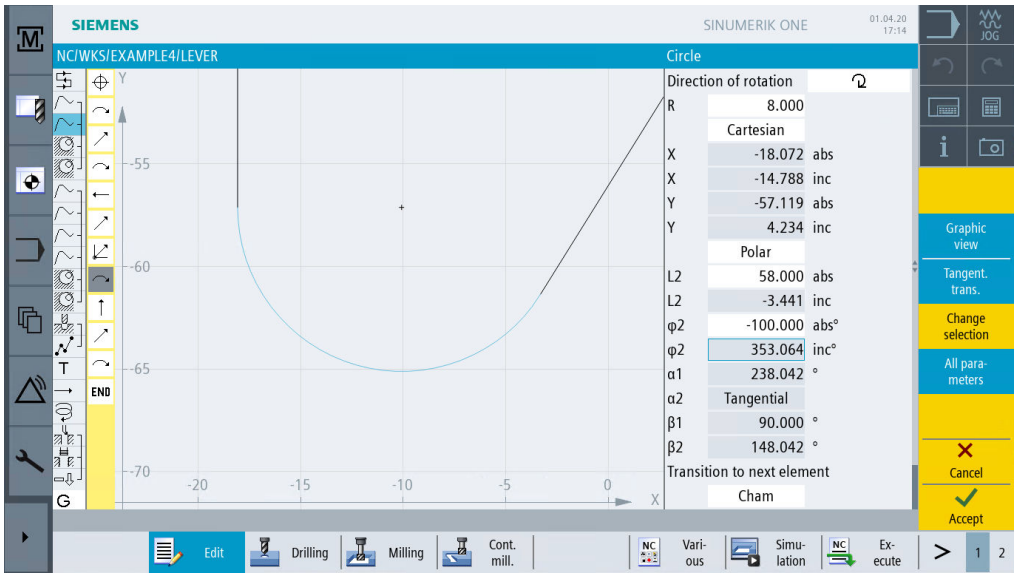


Figure 10-65 Entering the distance to the pole and specifying the polar angle

Press "Accept" to apply your input.

Apply the change.

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

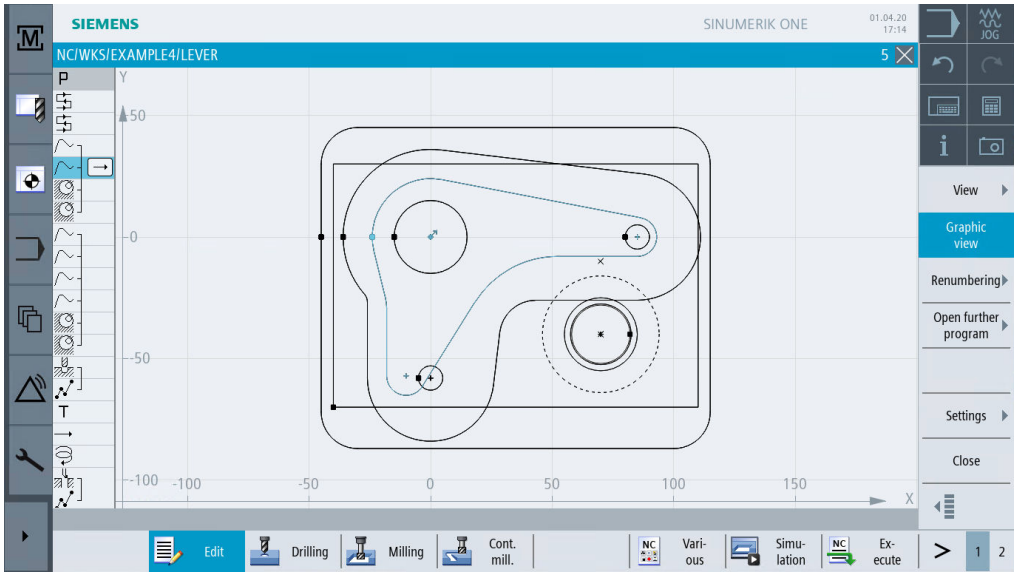


Figure 10-66 Broken-line graphics after shifting

Change these two contours without help. Note the following:

Note

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

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

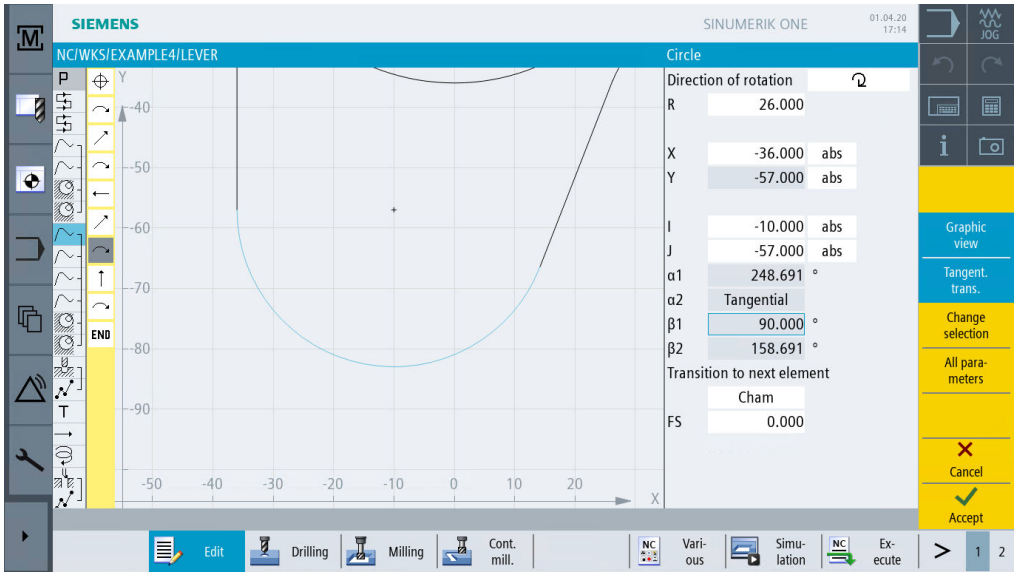


Figure 10-67 Adapting the edge

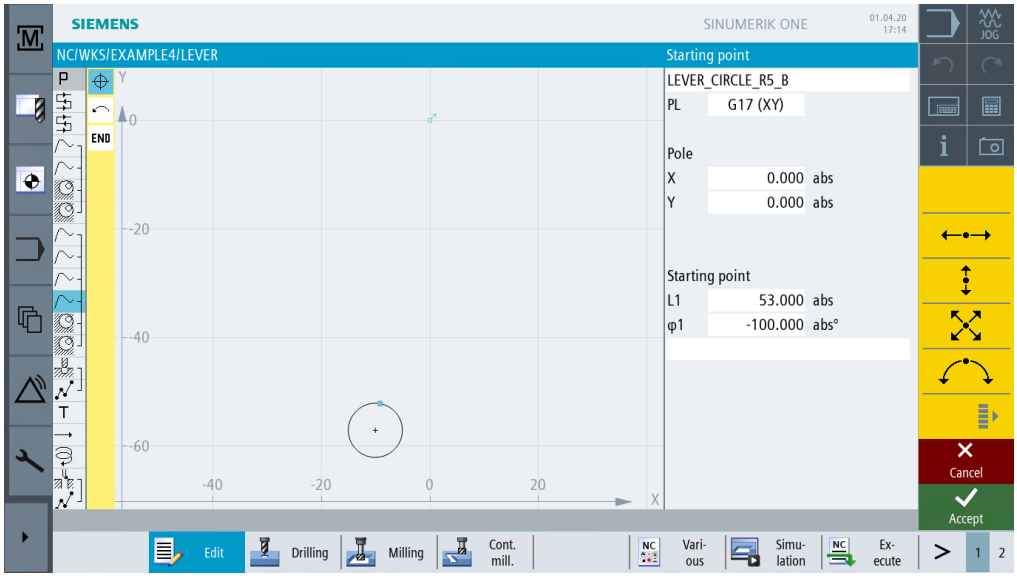


Figure 10-68 Defining the start point as a pole

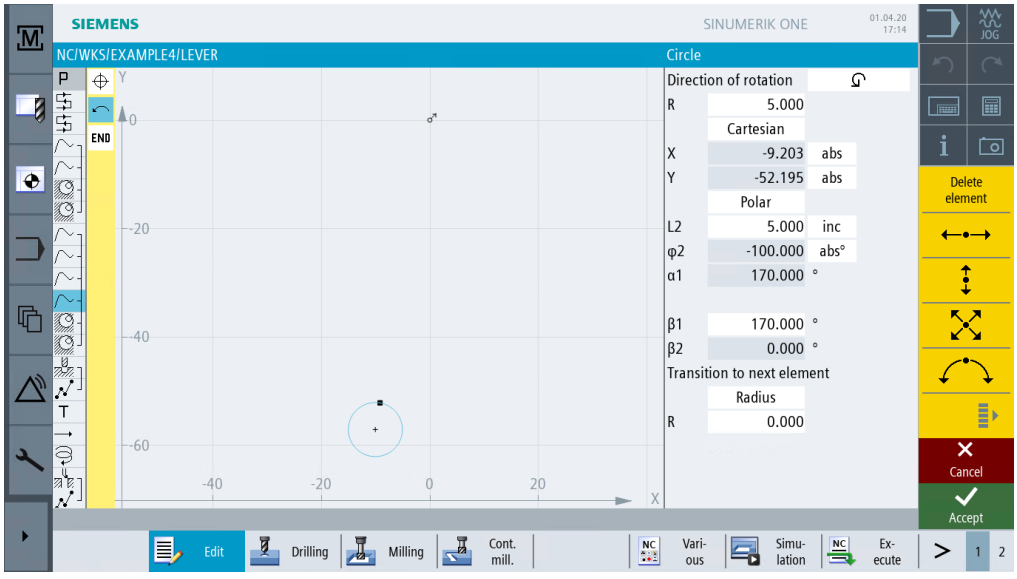


Figure 10-69 Adapting the circular island

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

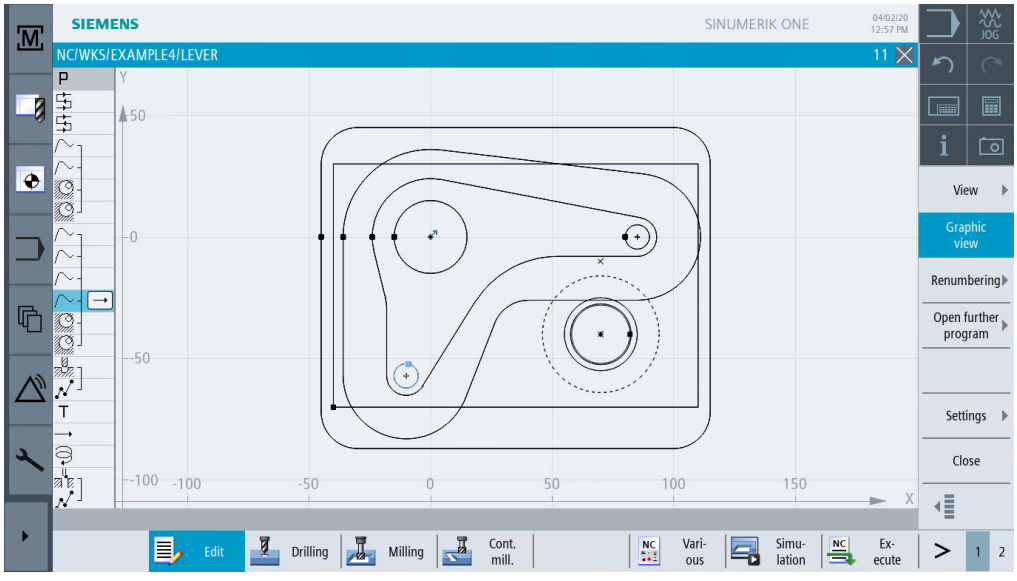


Figure 10-70 Broken-line graphics

Example 5: Flange

11

11.1 Overview

Learning objectives

In this chapter you will learn how to ...

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

Task

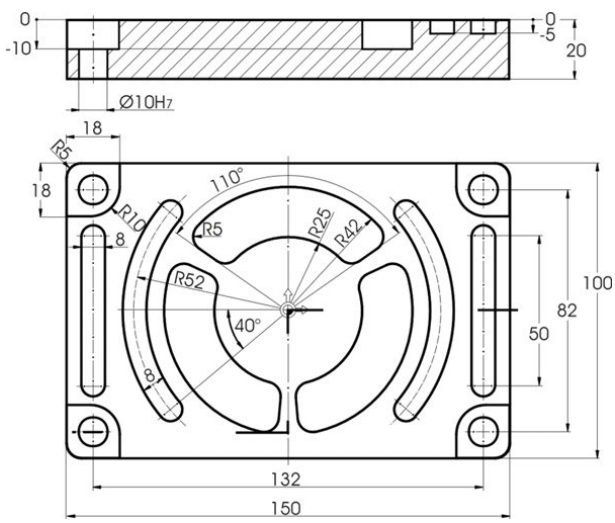


Figure 11-1 Workshop drawing - Example 5:

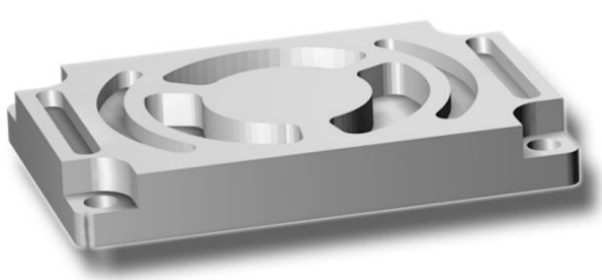


Figure 11-2 Workpiece - Example 5:

Note

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

11.2 Creating a subroutine

Operating sequences

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

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

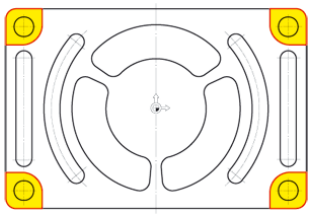


Figure 11-3 Contour of the four corners

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

Figure 11-4 Creating a subroutine

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

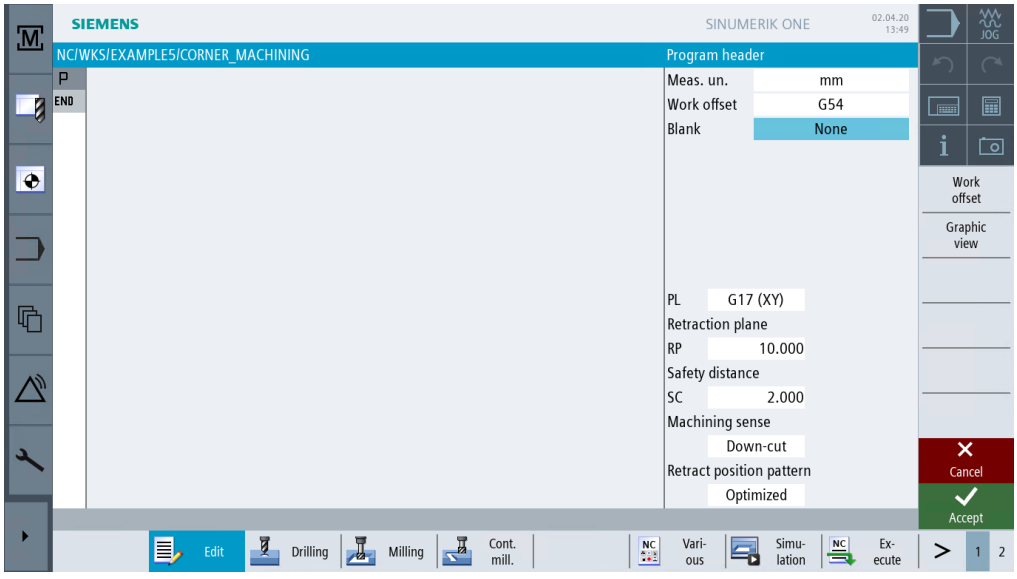


Figure 11-5 Entering the data for the subroutine program header

Press "Accept" to apply the values entered.

Select the **Contour milling** softkey.

Create a new contour with the name CORNER_M_SURFACE .

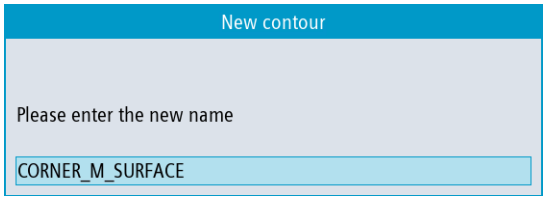


Figure 11-6 Creating the contour

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

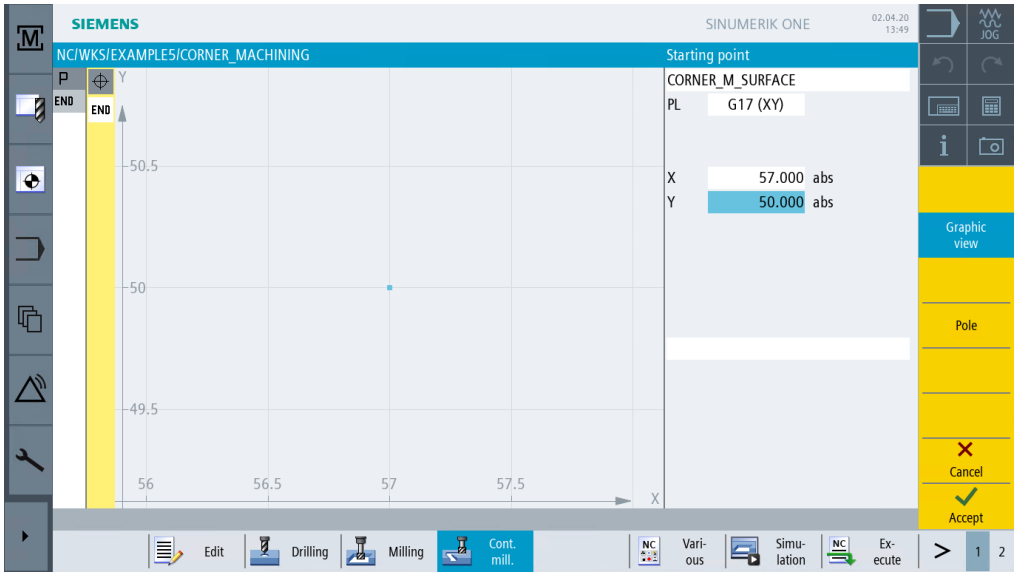


Figure 11-7 Specifying the starting point

Press "Accept" to apply the values entered.

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

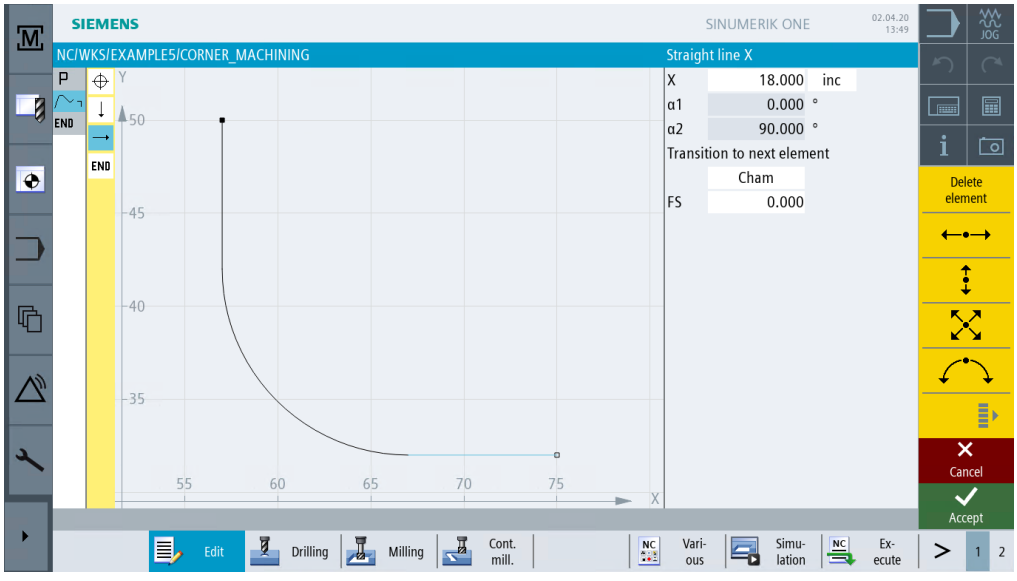


Figure 11-8 Contour subroutine, top right corner

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

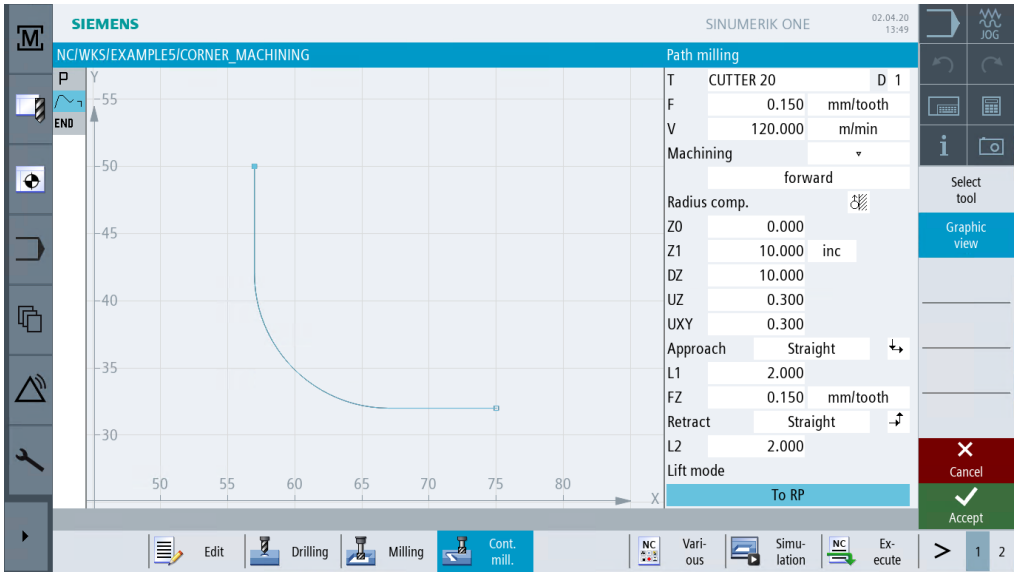


Figure 11-9 Roughing the contour

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

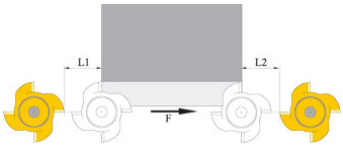


Figure 11-10 Approach and retract paths along a straight line

Press "Accept" to apply the values entered.

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

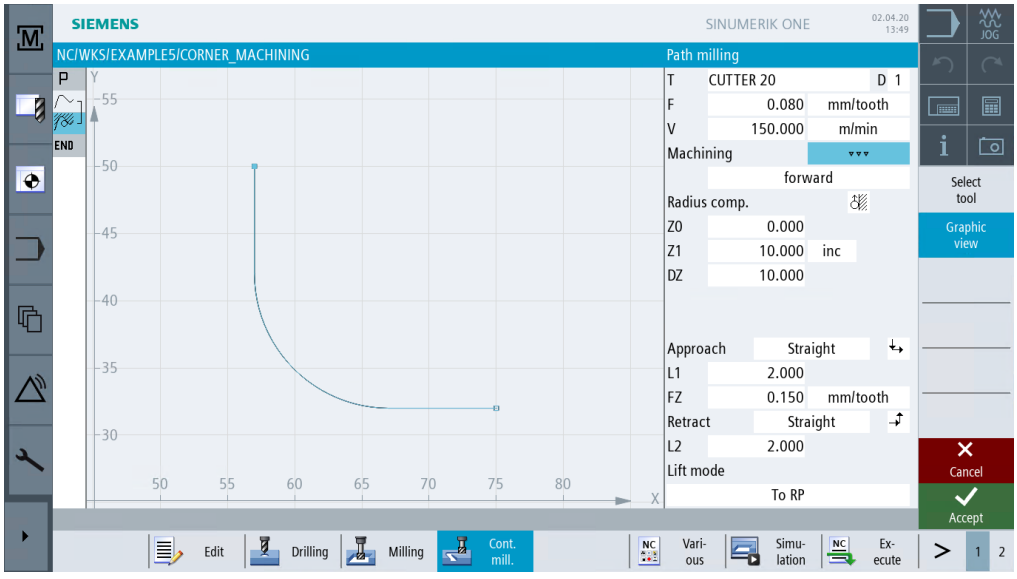


Figure 11-11 Finishing the contour

Press "Accept" to apply the values entered.

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

Select the **Contour milling** softkey.

Create a new contour with the name CORNER_M_ARC .

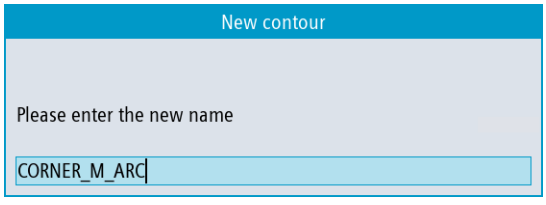


Figure 11-12 Creating the contour

Specify the starting point.

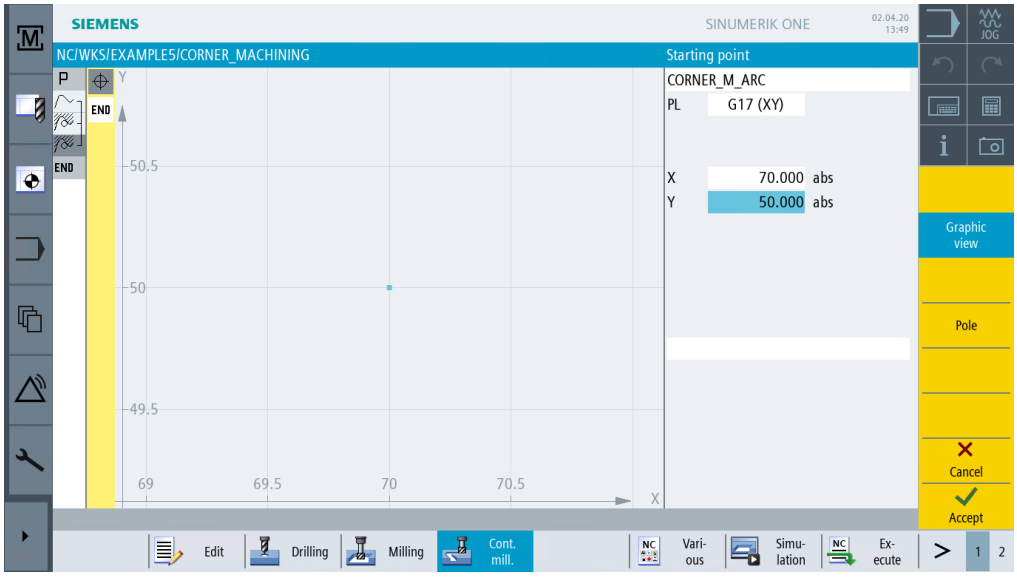


Figure 11-13 Specifying the starting point

Press "Accept" to apply the values entered.

Subsequently, specify the contour and the relevant work steps:

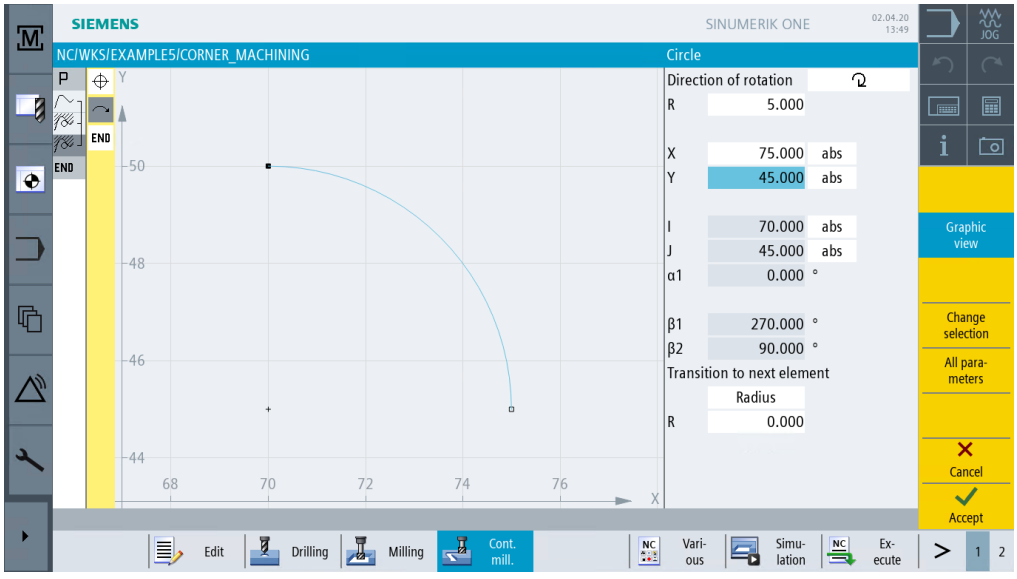


Figure 11-14 Specifying the geometry

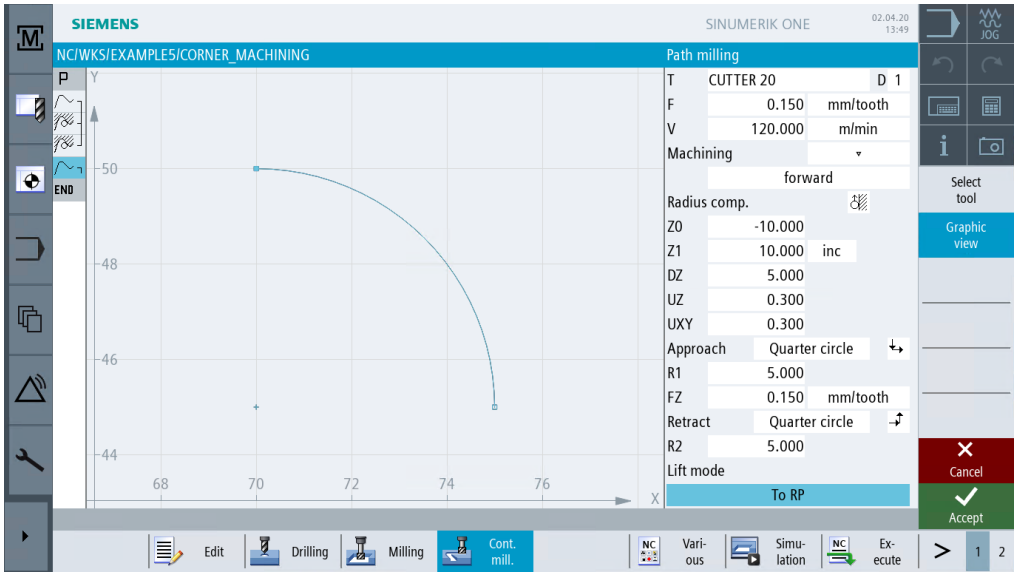


Figure 11-15 Roughing the contour

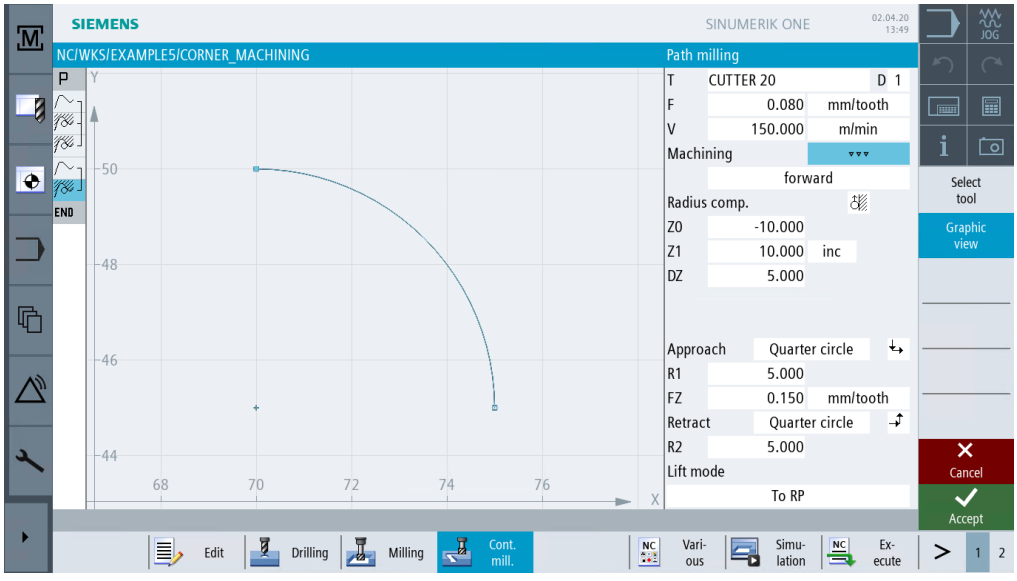


Figure 11-16 Finishing the contour

P	N10 Program header		G54 Blank: None
	N20 Contour		CORNER_M_SURFACE
	N30 Path milling	▾	T=CUTTER 20 F=0.15/t V=120m Z0=0 Z1=10inc
	N40 Path milling	▾▾▾	T=CUTTER 20 F=0.08/t V=150m Z0=0 Z1=10inc
	N50 Contour		CORNER_M_ARC
	N60 Path milling	▾	T=CUTTER 20 F=0.15/t V=120m Z0=-10 Z1=10inc
	N70 Path milling	▾▾▾	T=CUTTER 20 F=0.08/t V=150m Z0=-10 Z1=10inc
END	End of program		

Figure 11-17 Complete subroutine in the work step editor

11.3 Mirroring work steps

Task

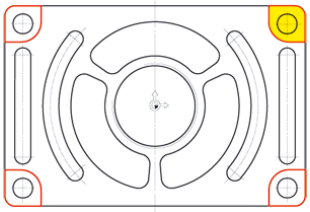
When you have finished the subroutine, create the main program. You may use the subroutine for all workpiece corners using the "Mirror" function in the "Transformation" menu.

Mirroring can be performed in two different ways:

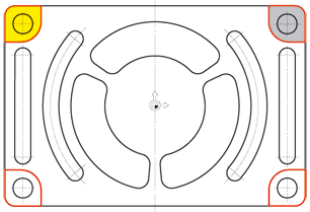
- New:
  Mirroring is performed from the point at which the 1st processing operation was performed.
- Additive:
  Mirroring is performed from the point last machined.

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

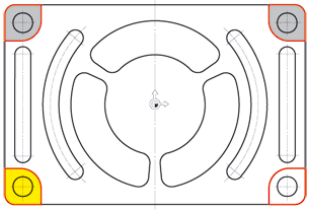
1. Processing (see subroutine)



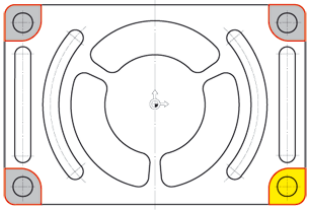
2. Processing: Mirroring the X axis
(the X values are mirrored here)



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



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



Operating sequences

Create the main program with the name FLANGE .

Figure 11-18 Creating the main program

Enter the program header.

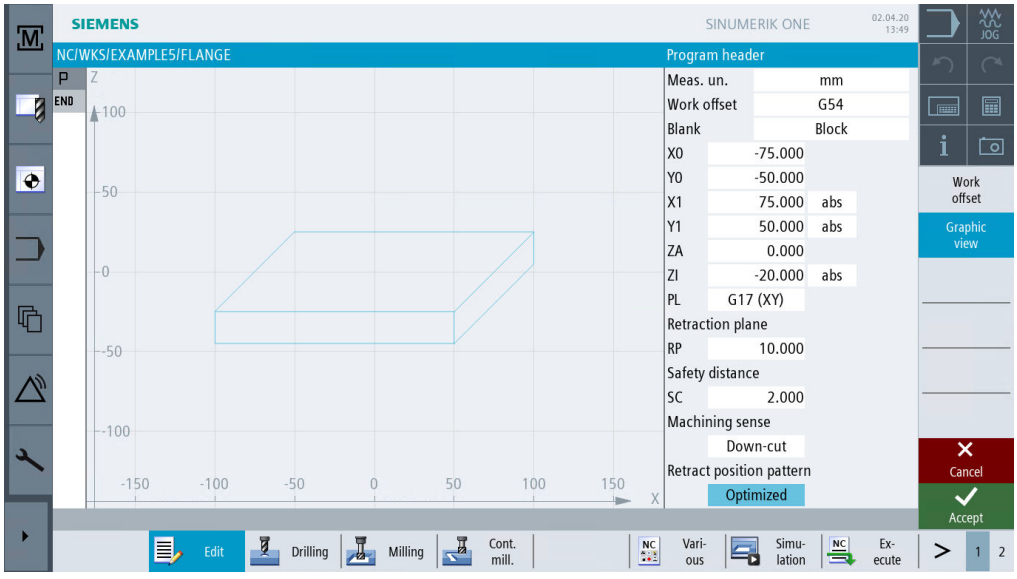
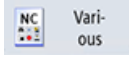


Figure 11-19 Specifying the program header of the main program

Press "Accept" to apply the values entered.



Select the **Miscellaneous** softkey.

Select the CORNER_MACHINING subprogram.

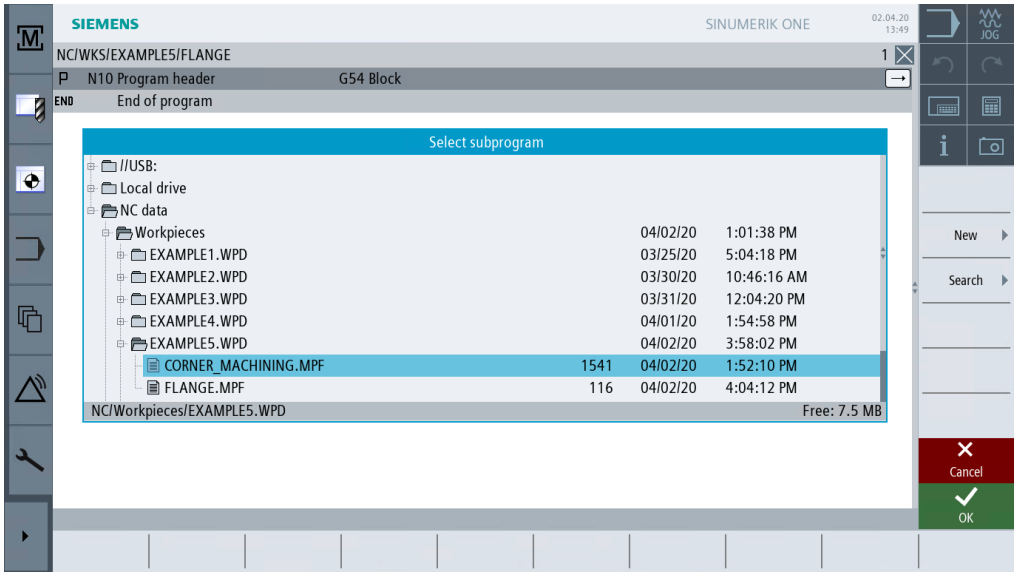


Figure 11-20 Selecting the subprogram

Accept your selection. After acceptance, your work step program looks like this:

Figure 11-21 Subroutine pasted into the main program

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

Preparing the 2nd processing: Mirror the X values.

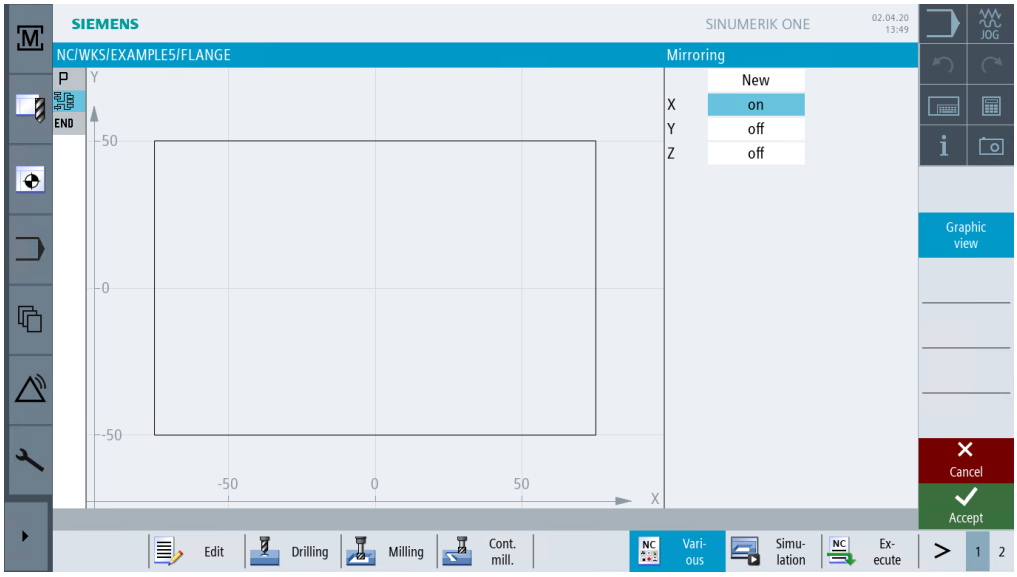


Figure 11-22 Mirroring

Press "Accept" to apply your input.

To mirror the remaining processing operations, proceed as follows:

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

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

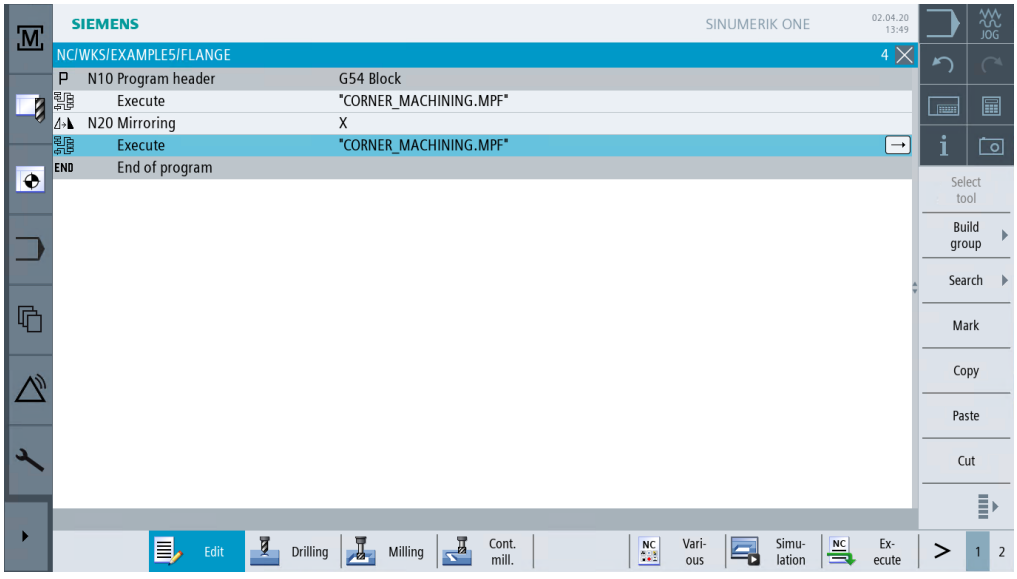


Figure 11-23 Copying the subroutine

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

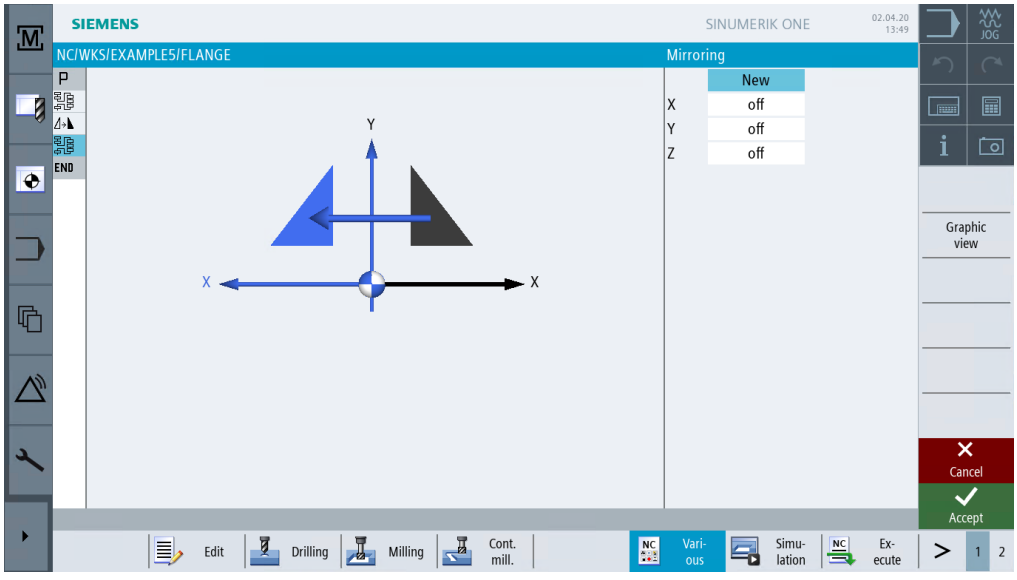


Figure 11-24 Mirroring help display

Your process plan will look as follows:

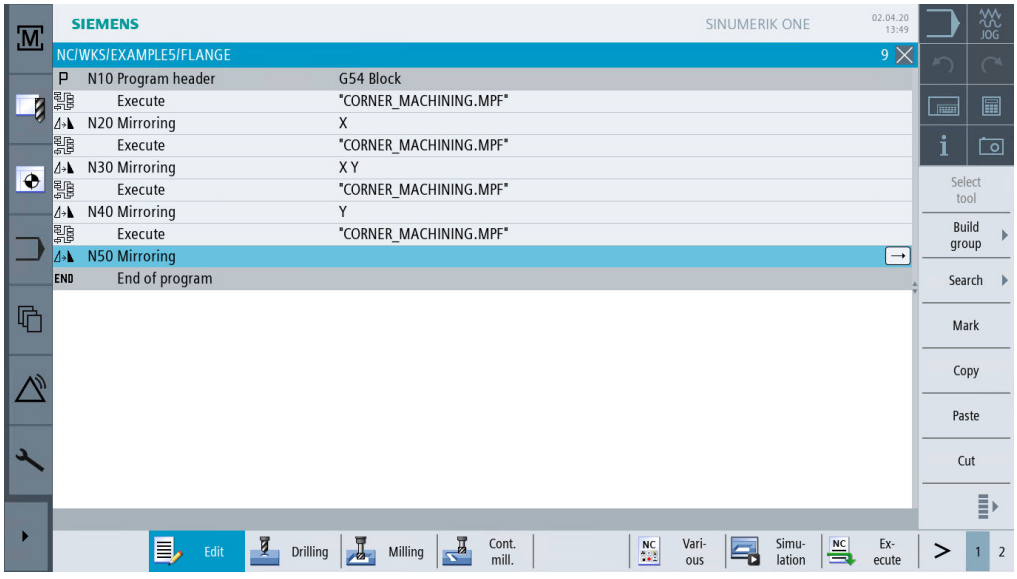


Figure 11-25 Complete mirroring in the work step editor

Check your work by now using the simulation.

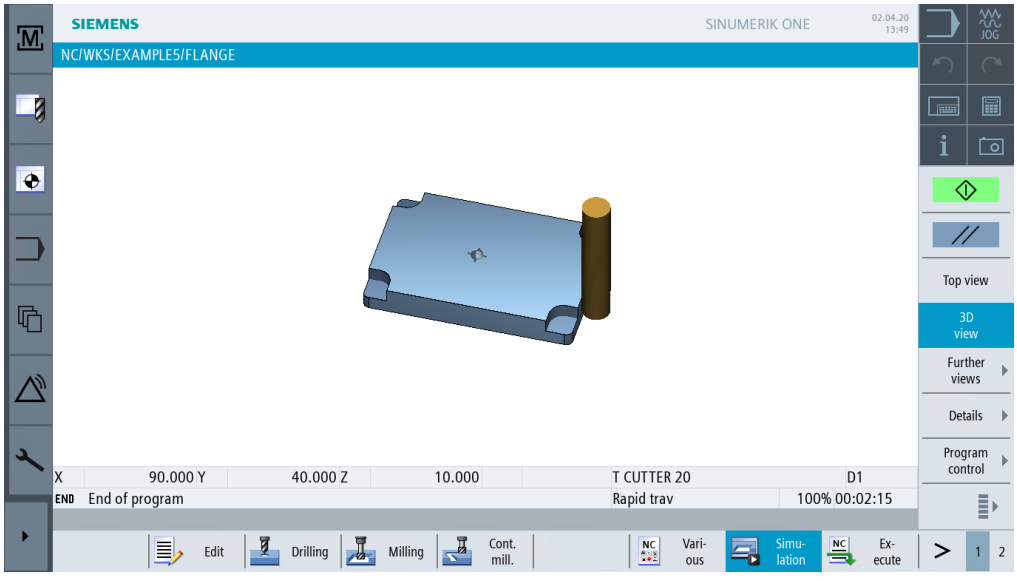


Figure 11-26 Simulation in 3D display

11.4 Holes

Operating sequences

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

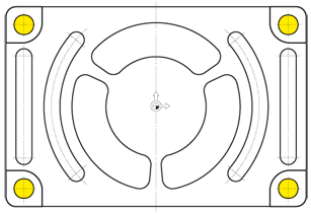


Figure 11-27 Holes

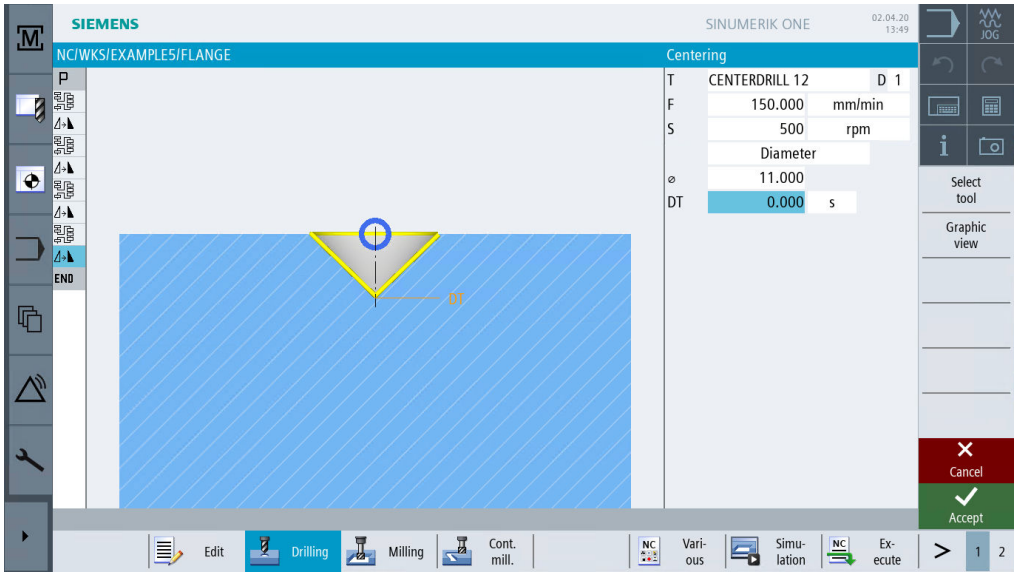


Figure 11-28 Centering

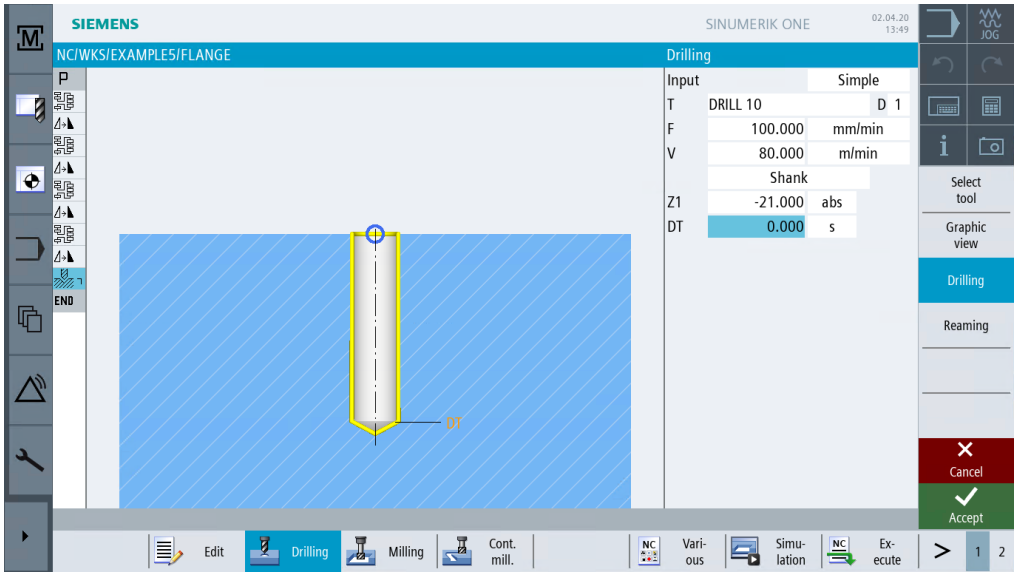


Figure 11-29 Drilling

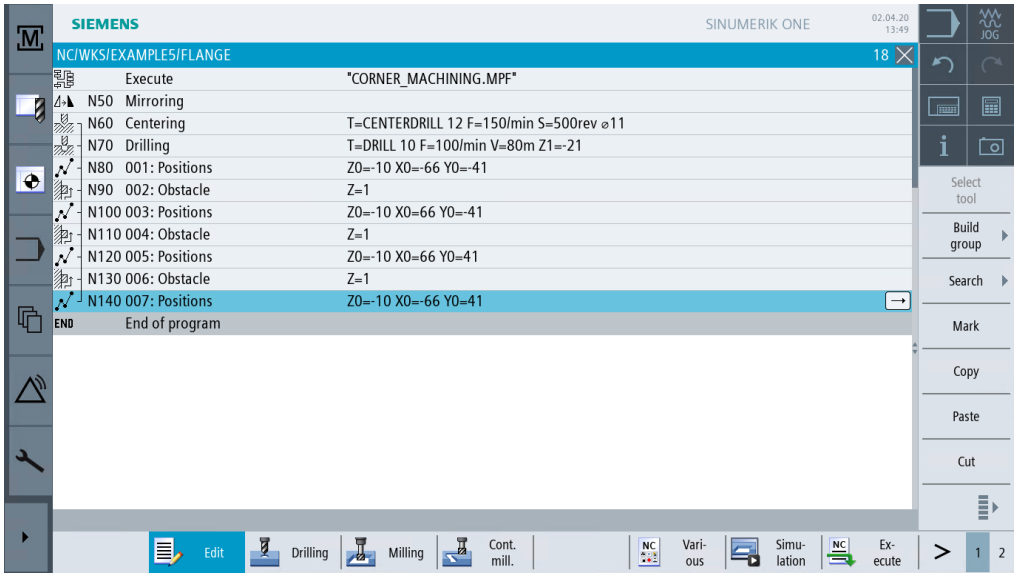


Figure 11-30 Specifying the positions of the obstacles

11.5 Rotation of pockets

Operating sequences

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

Select the **Contour milling** softkey.

Create a new contour with the name 'FLANGE_NODULE' .

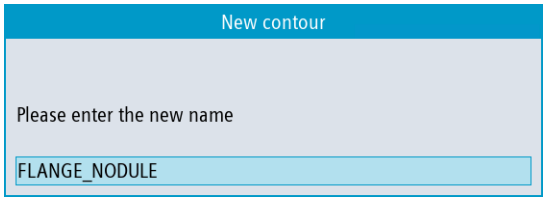


Figure 11-31 Creating a new contour

Specify the starting point.

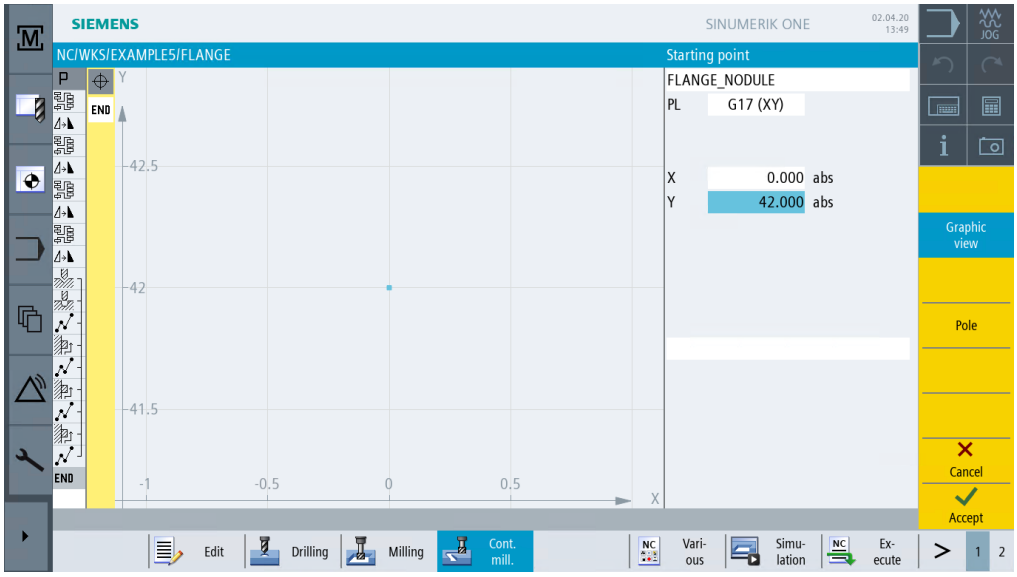


Figure 11-32 Specifying the starting point

Press "Accept" to apply the values entered.

Select the **Arc** softkey.

Select the **All parameters** softkey.

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

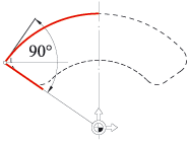
Figure 11-33 Specifying the arc

Press "Accept" to apply the values entered.

Select the **Diagonal** softkey.

Select the **All parameters** softkey.

Create the diagonal straight line.



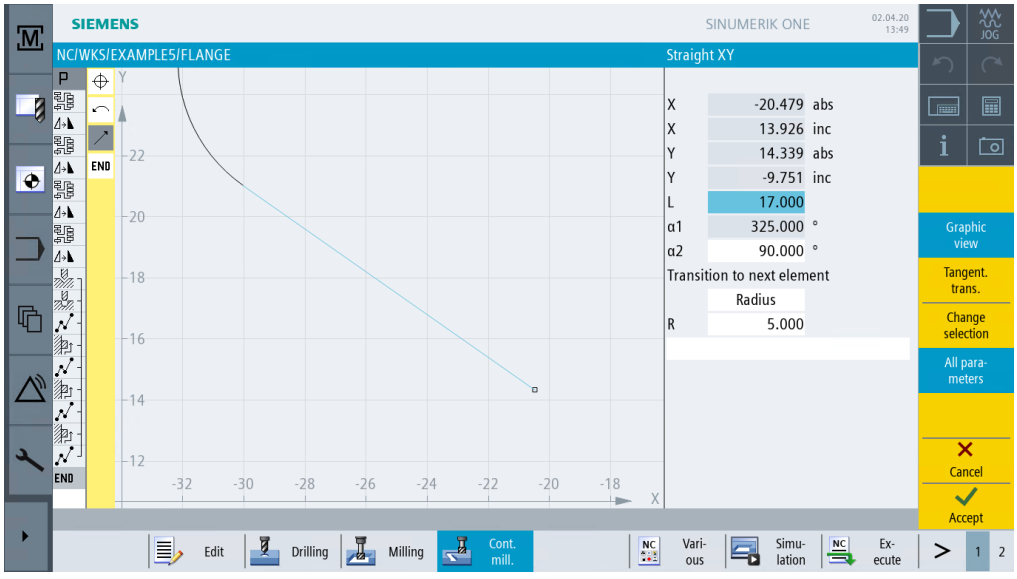


Figure 11-34 Specifying the diagonal

Press "Accept" to apply the values entered.

Select the **Arc** softkey.

Select the **All parameters** softkey.

Create the 2nd arc.

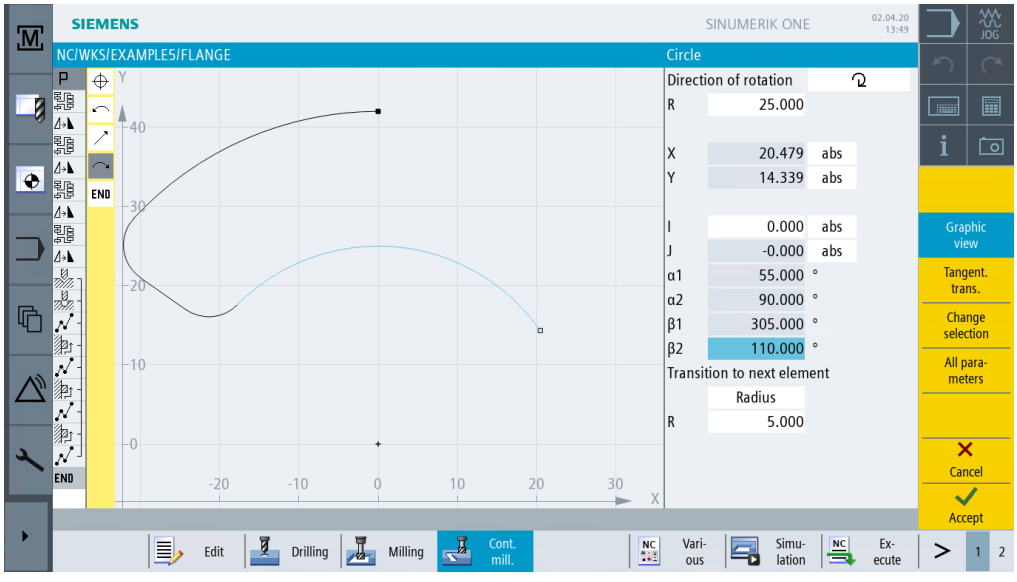


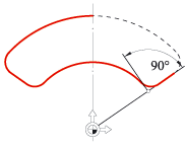
Figure 11-35 Specifying the arc

Press "Accept" to apply the values entered.

Select the **Diagonal** softkey.

Select the **All parameters** softkey.

Create the 2nd diagonal straight line.



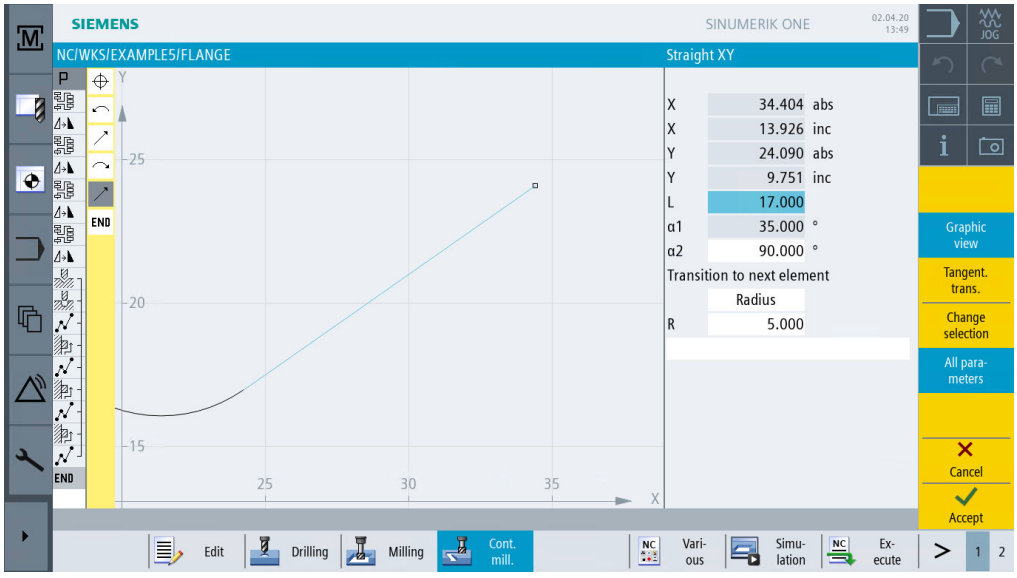


Figure 11-36 Specifying the diagonal

Press "Accept" to apply the values entered.

Select the **Arc** softkey.

Create the final arc.

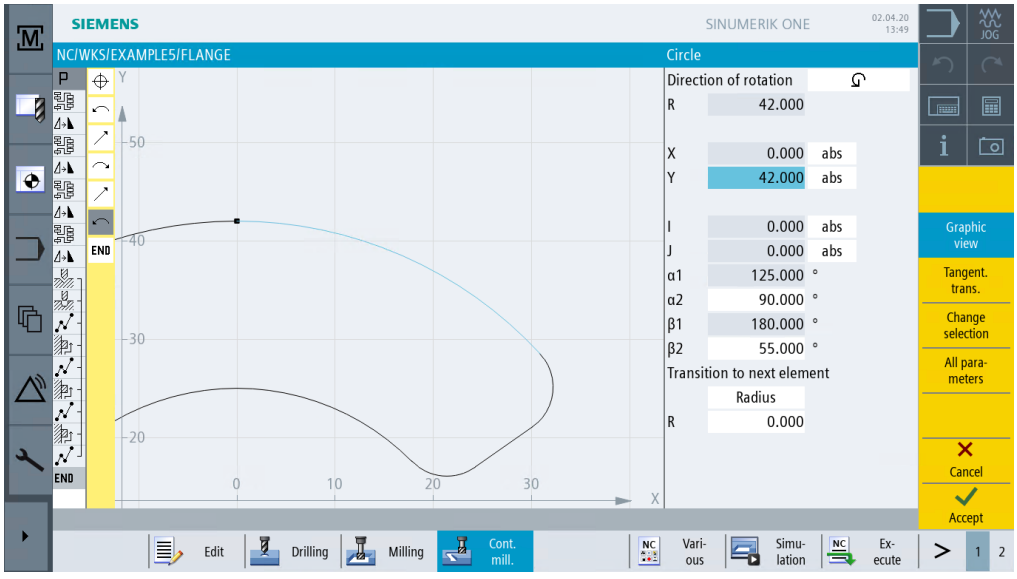


Figure 11-37 Specifying the final arc

Press "Accept" to apply the values entered.

Accept the contour pocket into your process plan.

Create the following work steps without help:

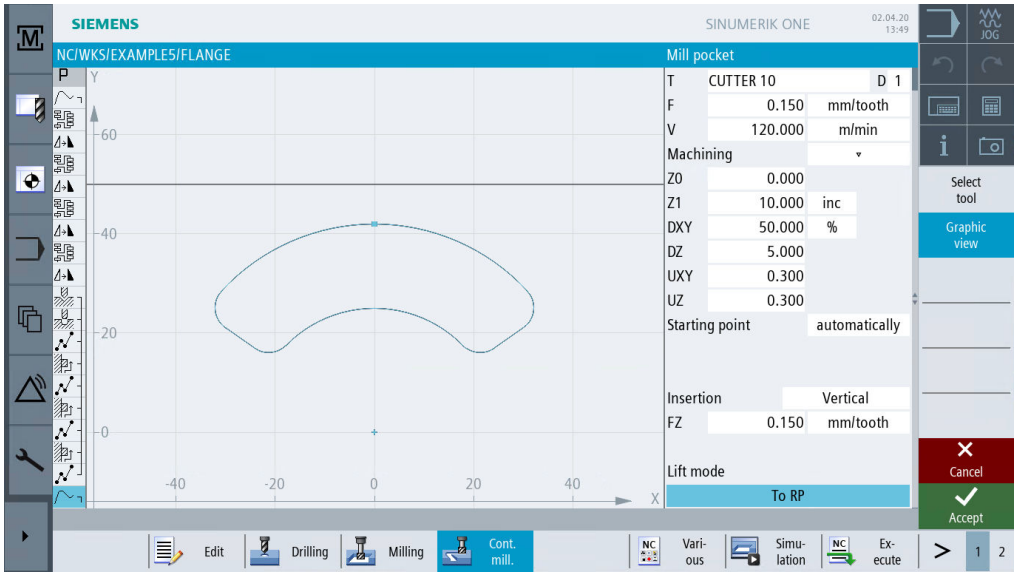


Figure 11-38 Roughing pockets

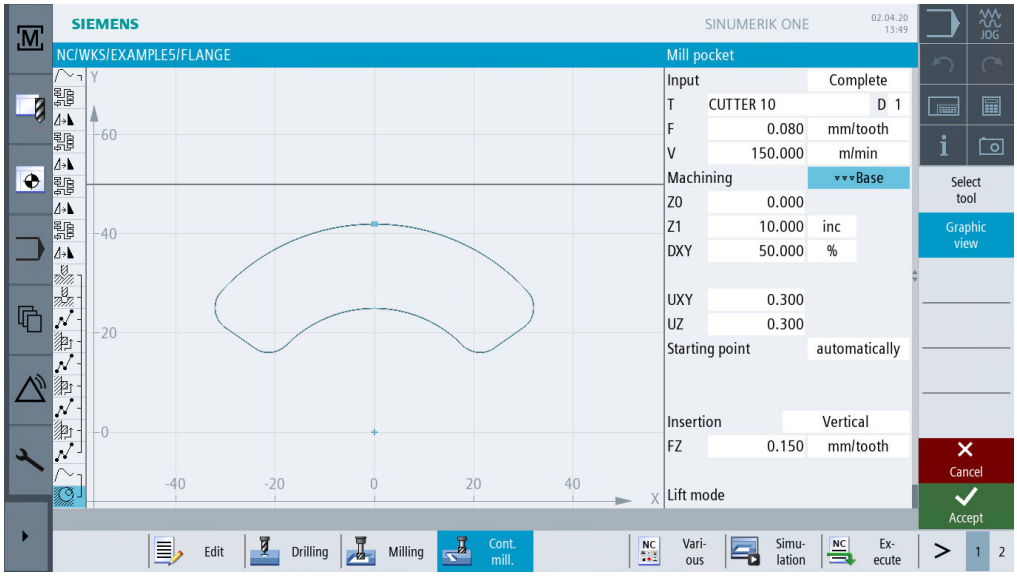


Figure 11-39 Finishing the pocket base

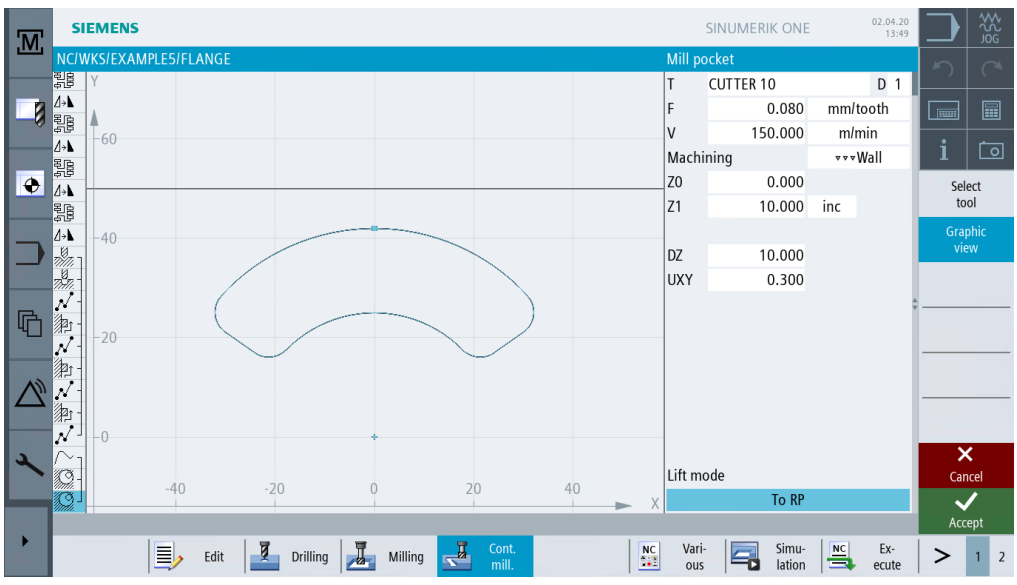


Figure 11-40 Finishing the pocket edge

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

Mark

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

Copy

Copy the work step sequence to the clipboard.

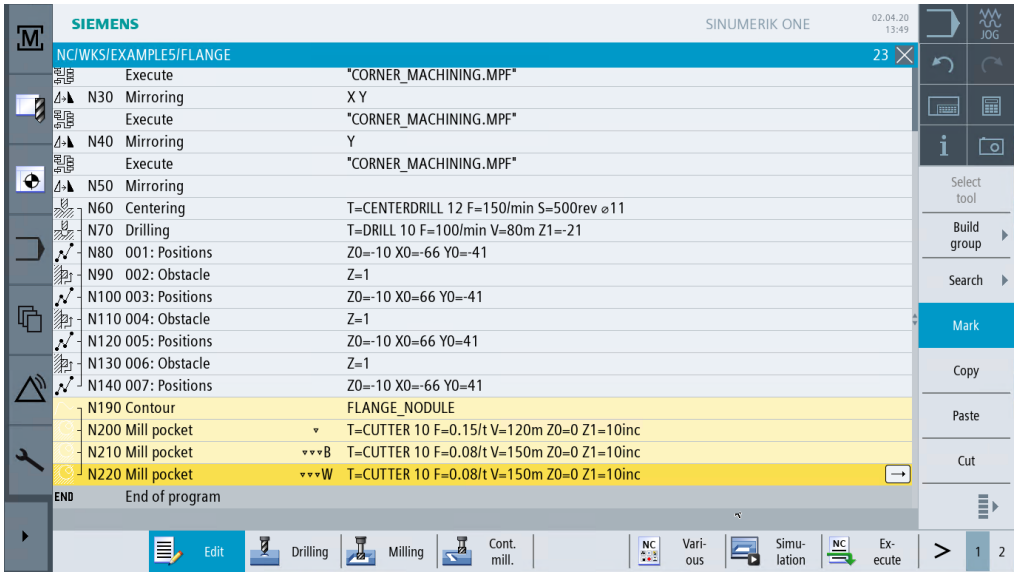


Figure 11-41 Copying the work steps

Select the **Miscellaneous** softkey.

Select the **Transformations** softkey.

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

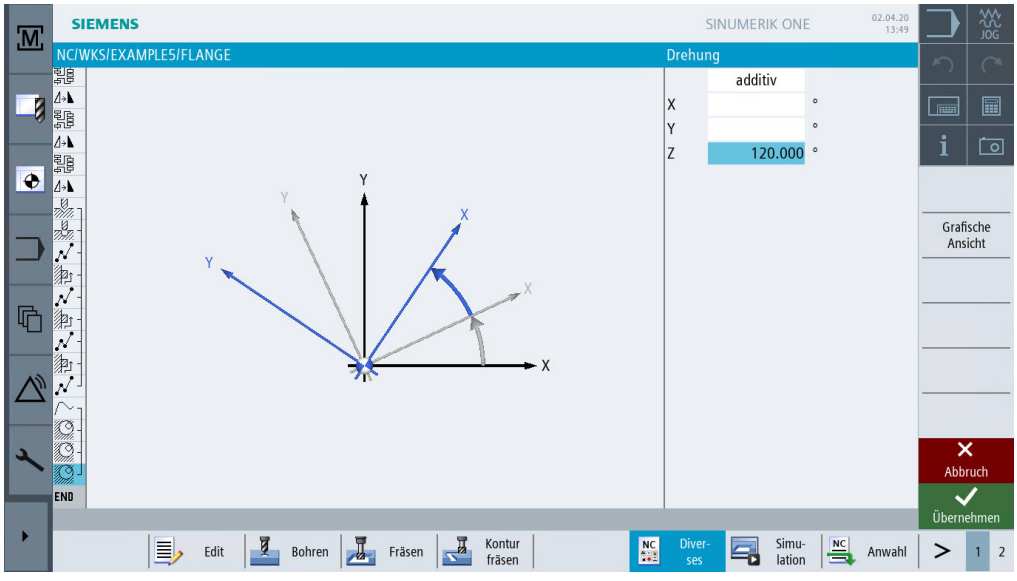


Figure 11-42 Rotation around the Z axis

Press "Accept" to apply your input.

Paste the copied work steps.

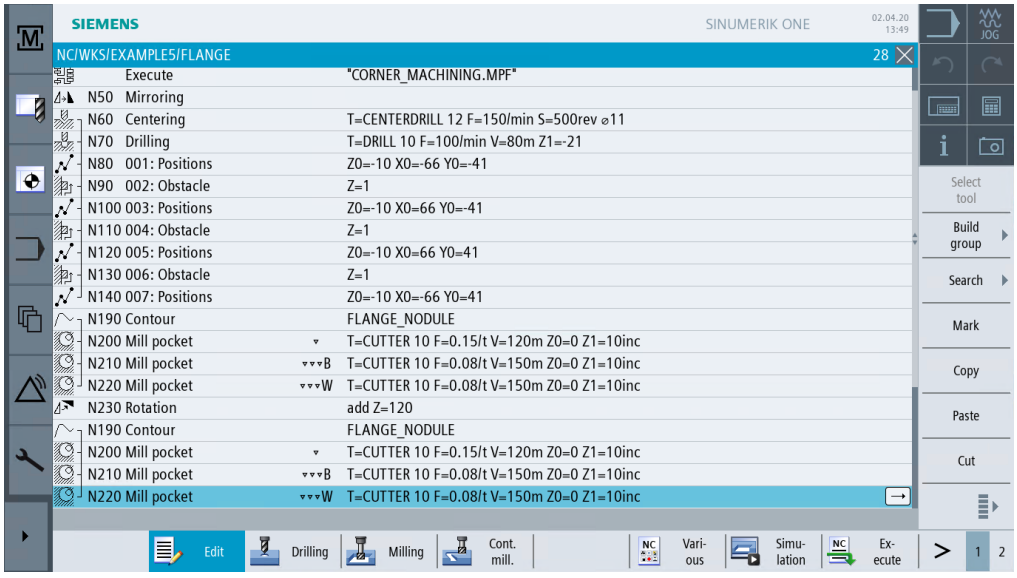


Figure 11-43 Pasting the copied work steps

Select the **Transformations** softkey.

Enter another rotation by 120°.

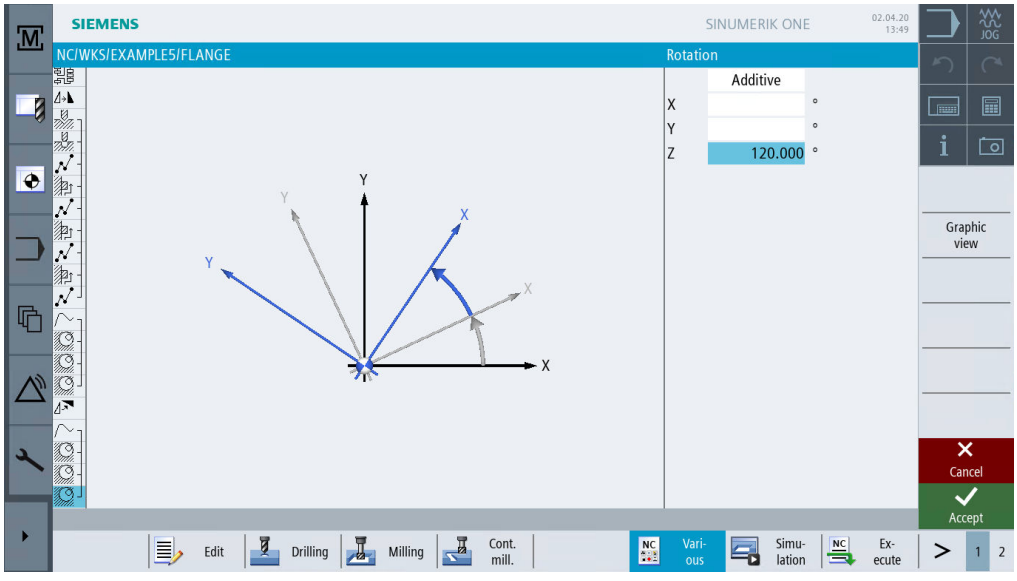


Figure 11-44 Rotation around the Z axis

Press "Accept" to apply your input.

Paste the copied work steps.

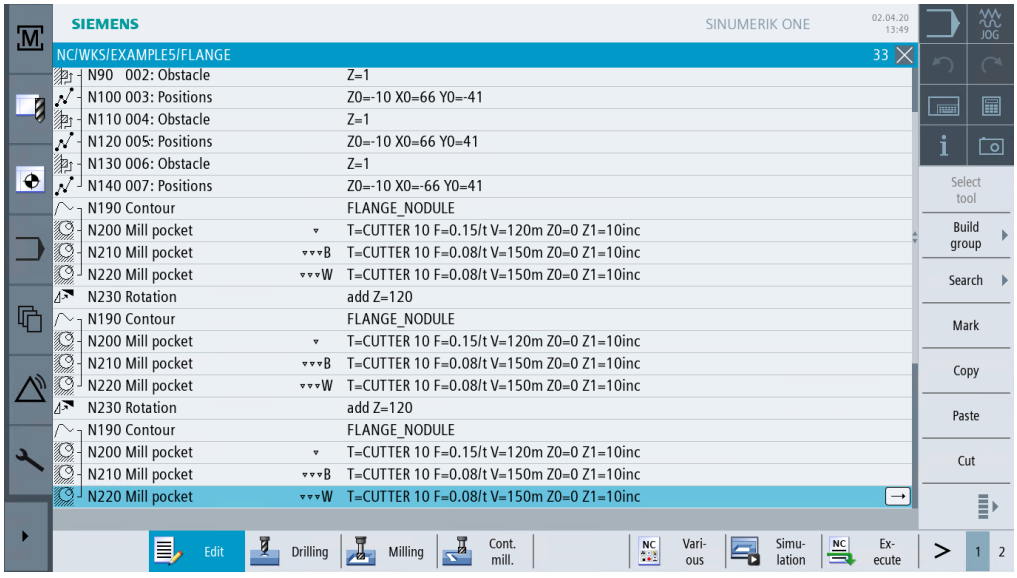


Figure 11-45 Pasting the copied work steps

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

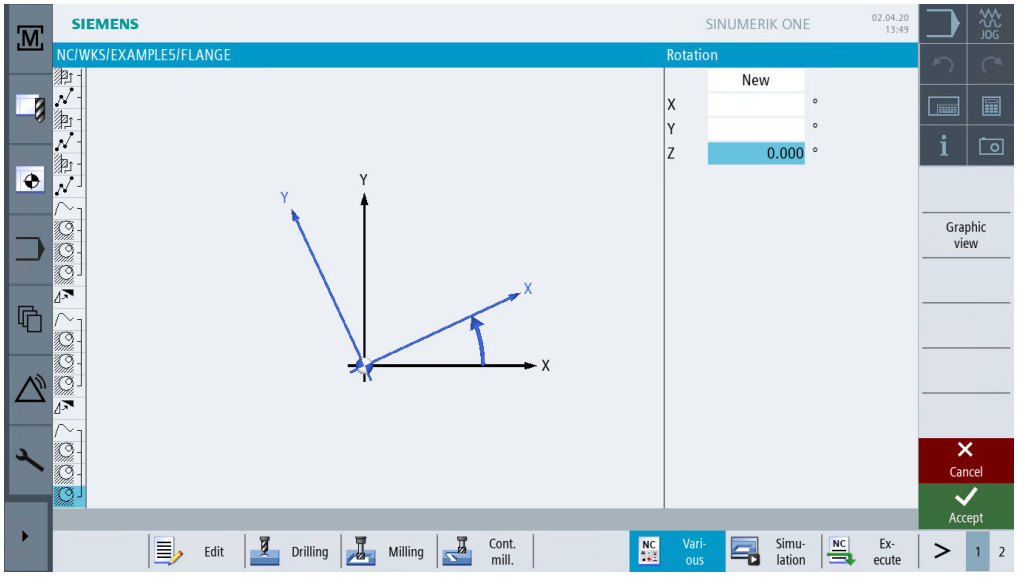


Figure 11-46 Undoing the rotation

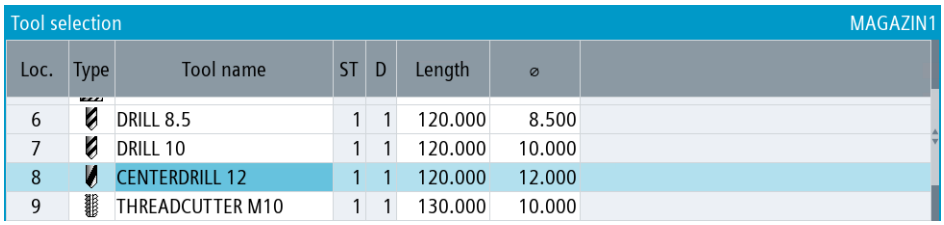
Press "Accept" to apply your input.

11.6 Chamfering contours

Operating sequences

Chamfer the circular pocket last machined without help.

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



Loc.	Type	Tool name	ST	D	Length	⌀
6		DRILL 8.5	1	1	120.000	8.500
7		DRILL 10	1	1	120.000	10.000
8		CENTERDRILL 12	1	1	120.000	12.000
9		THREADCUTTER M10	1	1	130.000	10.000

Figure 11-47 Center drill

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

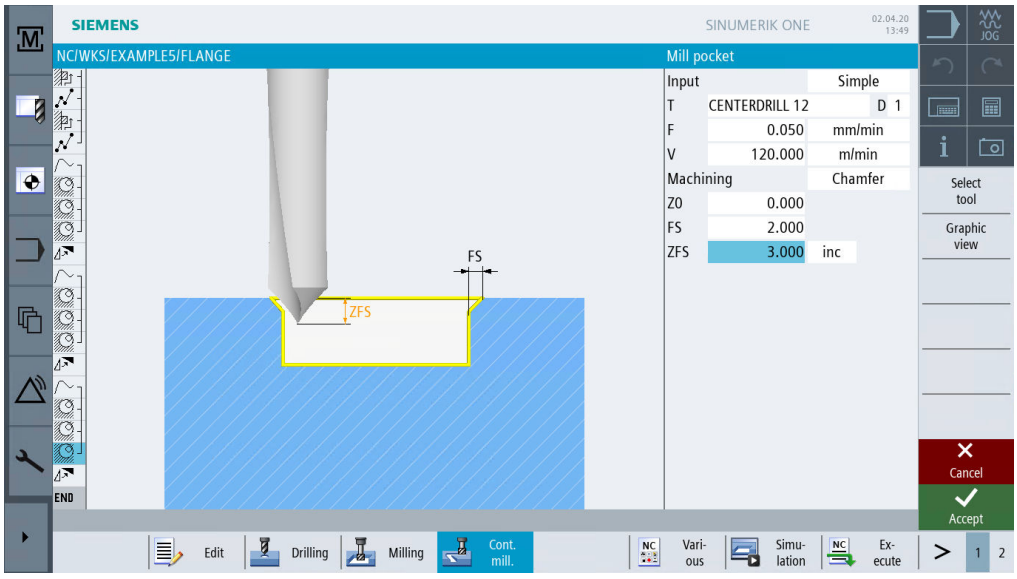


Figure 11-48 Chamfering

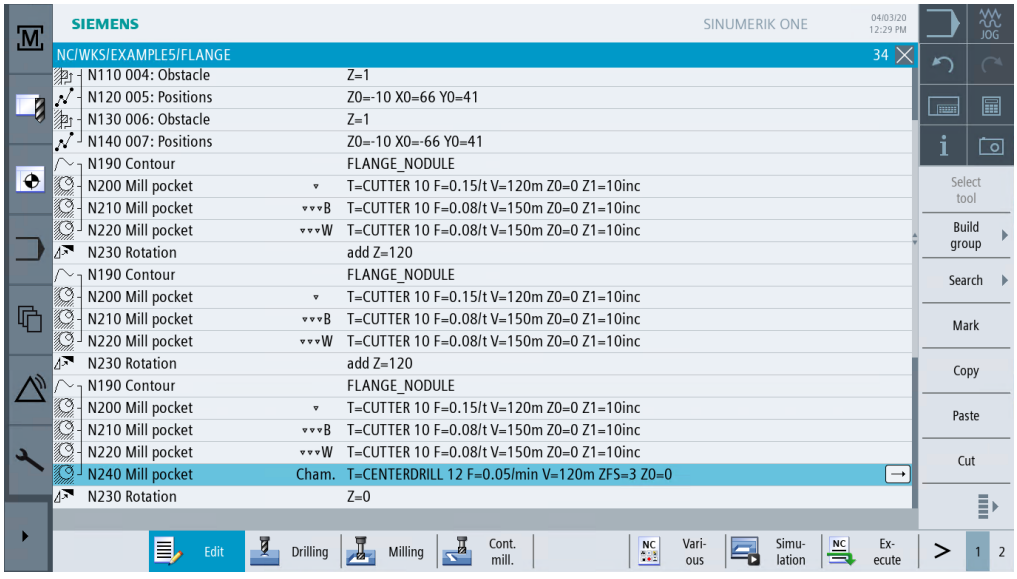


Figure 11-49 Work step "Chamfering" in the work step editor

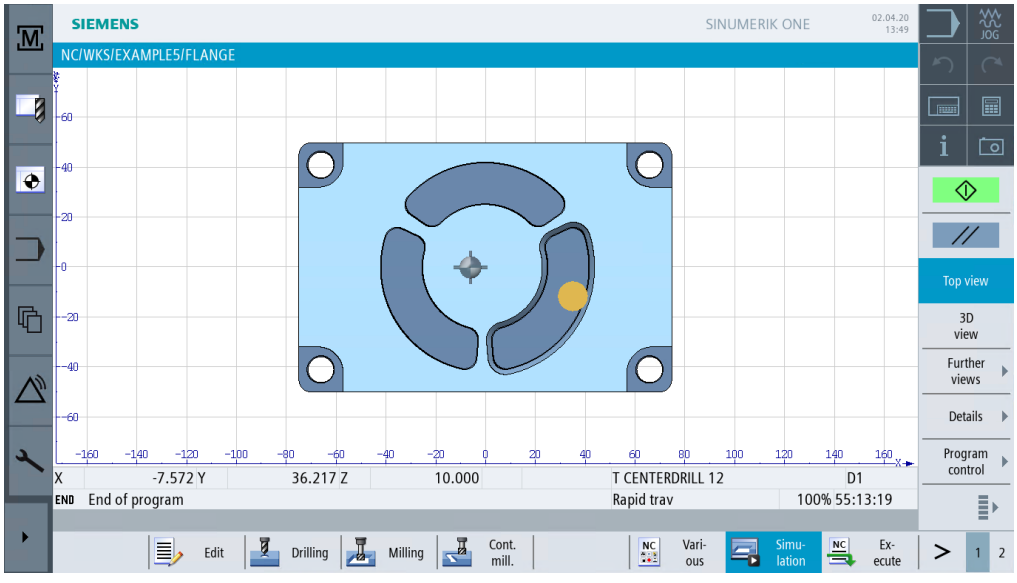


Figure 11-50 Top view on the chamfered contour

11.7 Longitudinal and circular grooves

Operating sequences

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

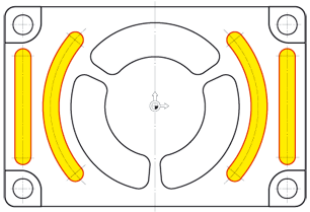


Figure 11-51 Longitudinal and circular grooves

Select the **Milling** softkey.

Select the **Groove** softkey.

Use the tool CUTTER 6 (F = 0.08 mm/tooth and v = 120 m/min) for the roughing of the longitudinal grooves.

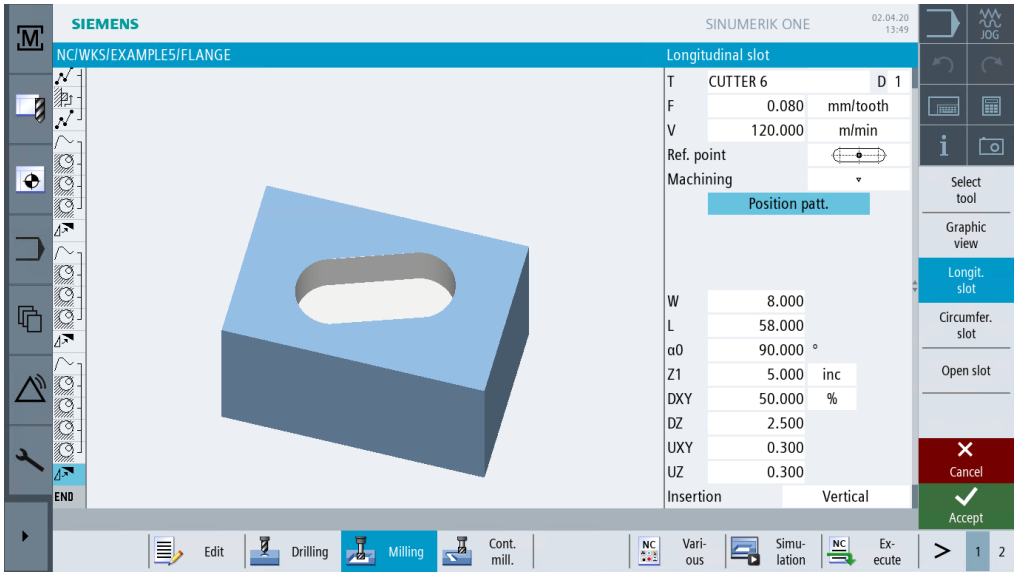


Figure 11-52 Roughing a longitudinal groove

Press "Accept" to apply the values entered.

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

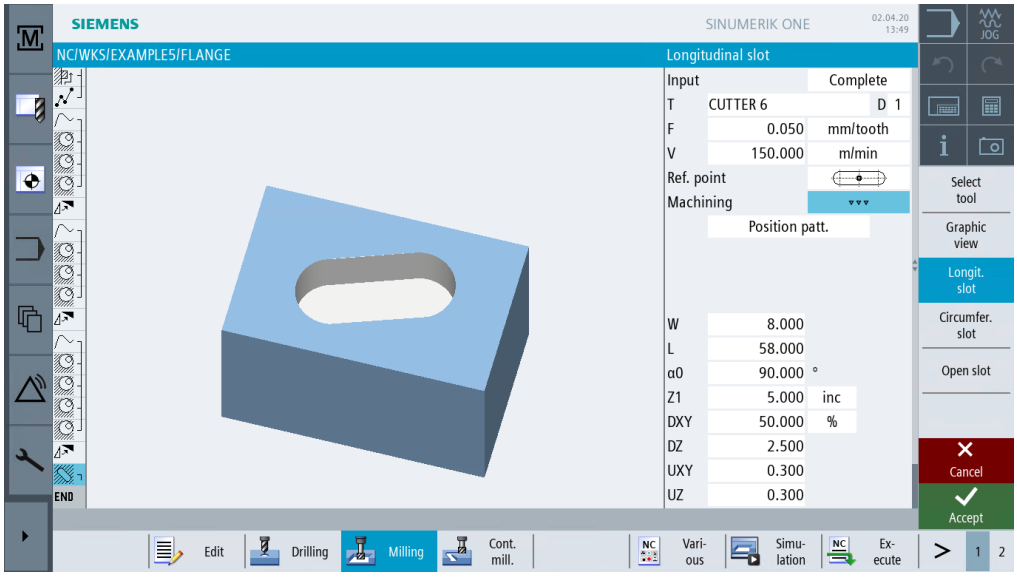


Figure 11-53 Finishing a longitudinal groove

Press "Accept" to apply the values entered.

Select the **Drilling** softkey.

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

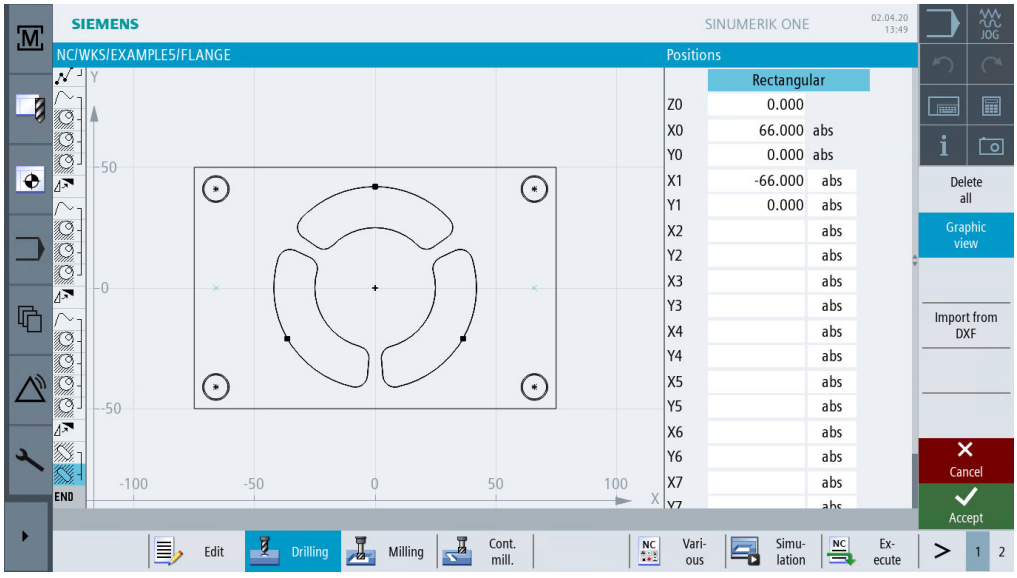


Figure 11-54 Specifying the positions for the longitudinal groove

Press "Accept" to apply the values entered.

Select the **Milling** softkey.

Select the **Groove** softkey.

Use the tool CUTTER 6 (F = 0.08 mm/tooth, FZ = 0.08 mm/tooth and v = 120 m/min) for the roughing of the circular grooves.

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

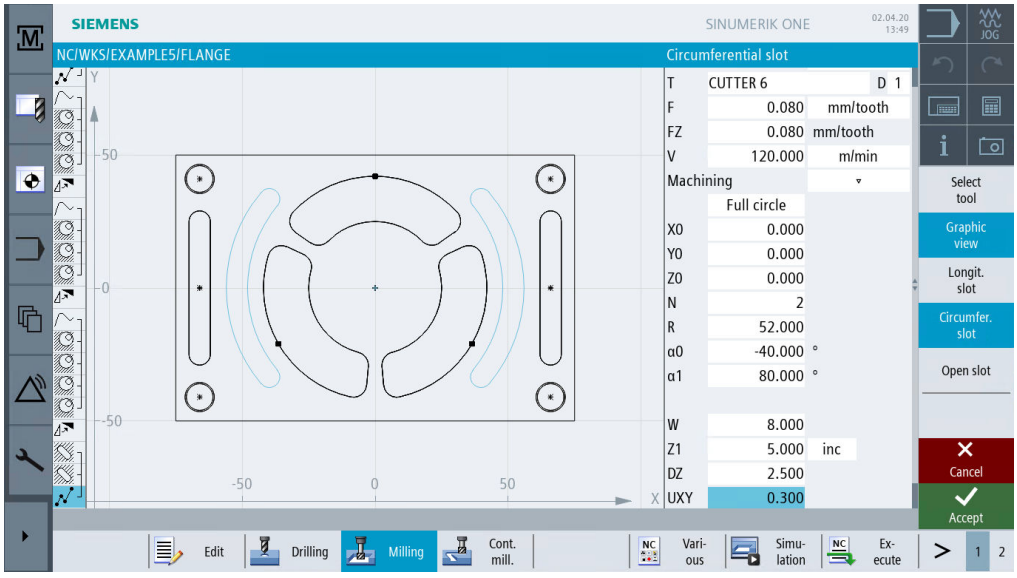
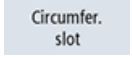


Figure 11-55 Roughing a circular groove

Press "Accept" to apply the values entered.

Select the **Groove** softkey.



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

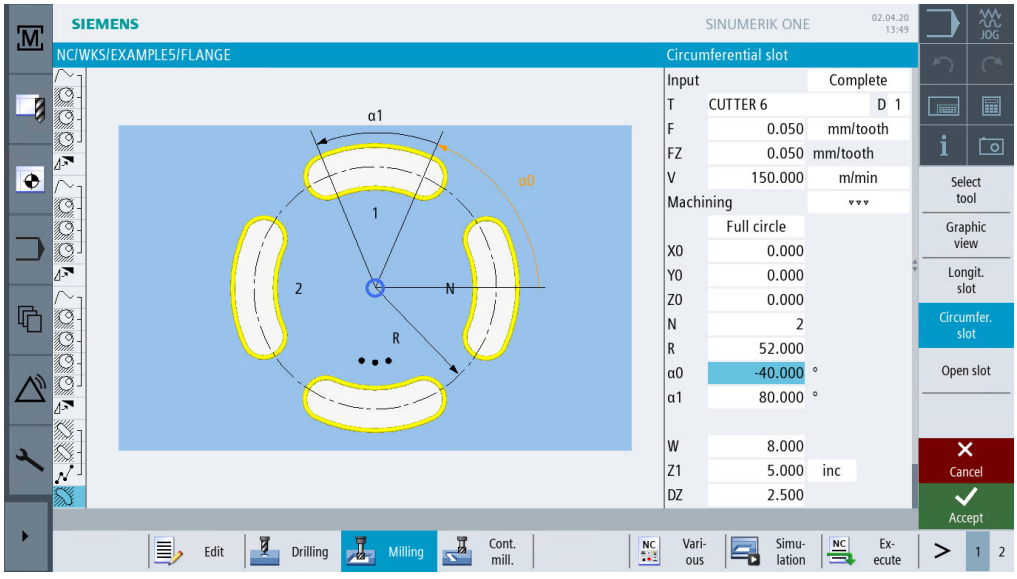


Figure 11-56 Finishing the circular groove

Press "Accept" to apply the values entered.

Process plan

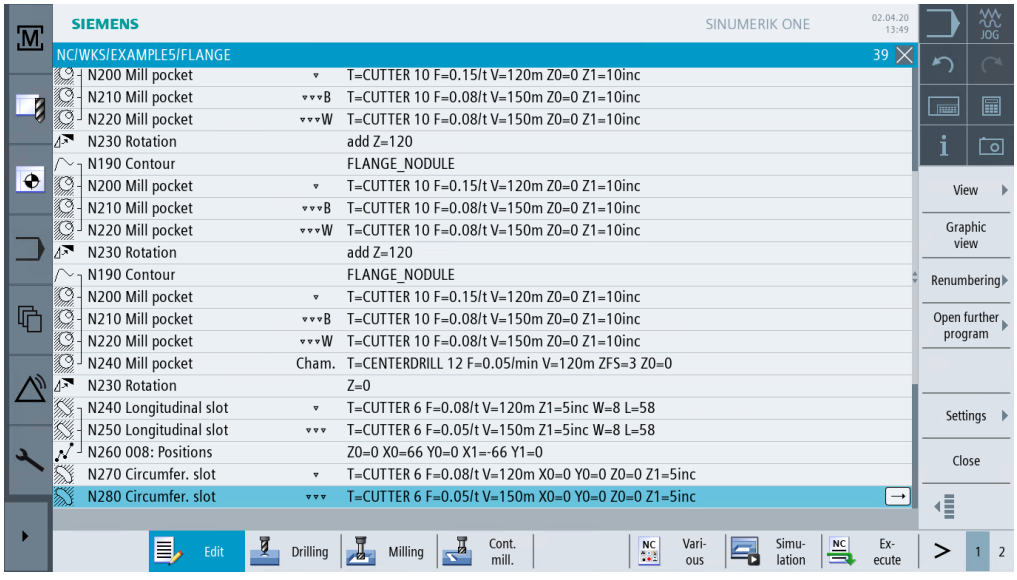


Figure 11-57 Extract from process plan

Broken-line graphics

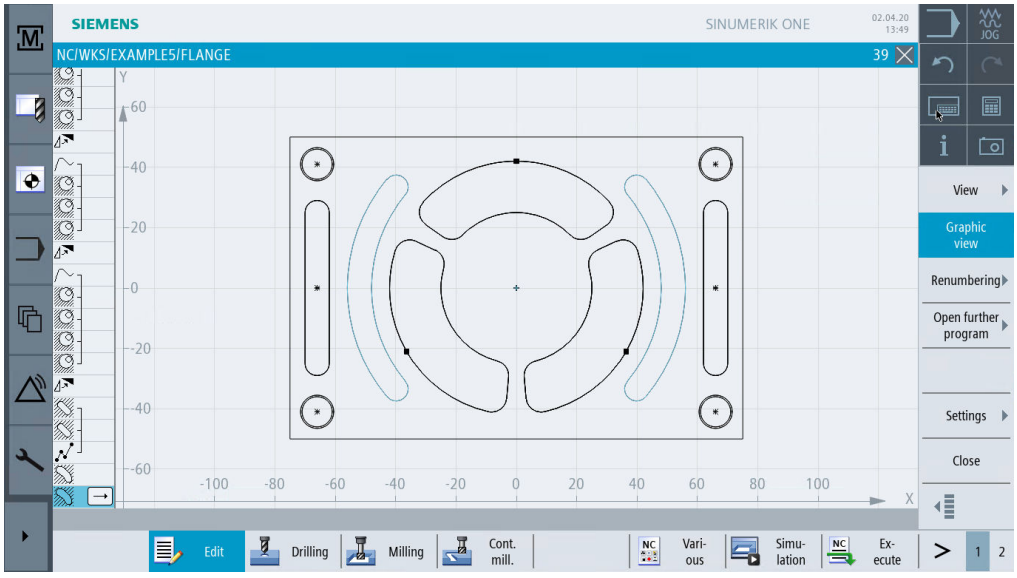


Figure 11-58 Broken-line graphics

Simulation in 3D display

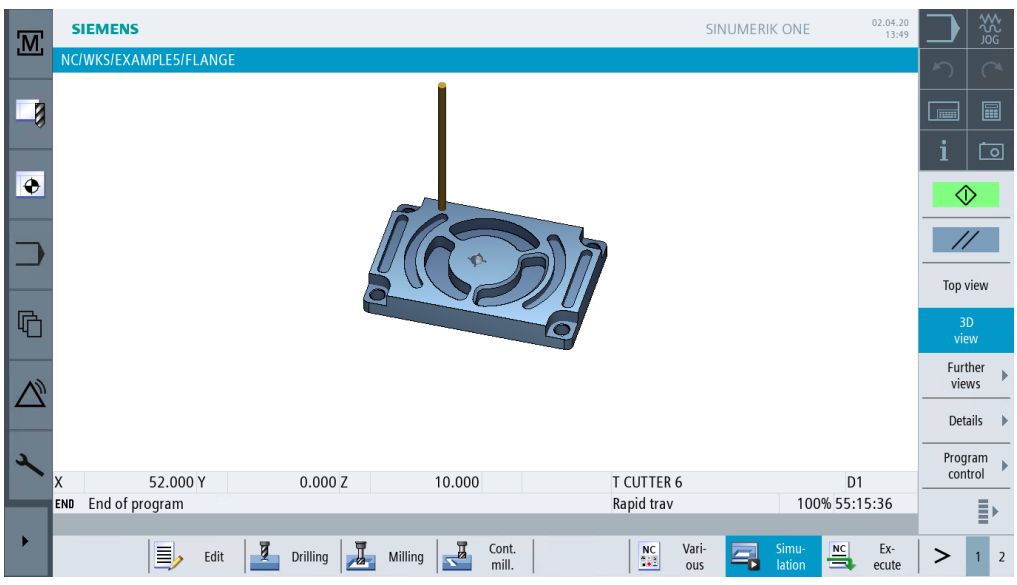


Figure 11-59 3D view

Execution in SINUMERIK Operate 12

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

To machine a workpiece, proceed as follows:

Approaching reference point

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

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

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

Clamping the workpiece

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

Setting the workpiece zero

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

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

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

through sampling.

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

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

.

Note

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

Executing the process plan

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

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

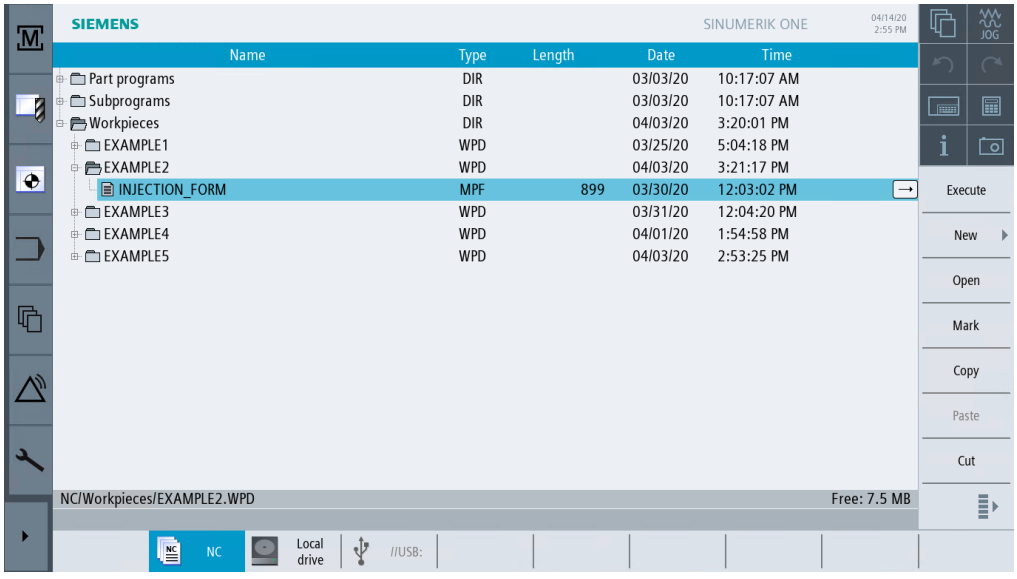


Figure 12-1 Selecting the program

Open the program.

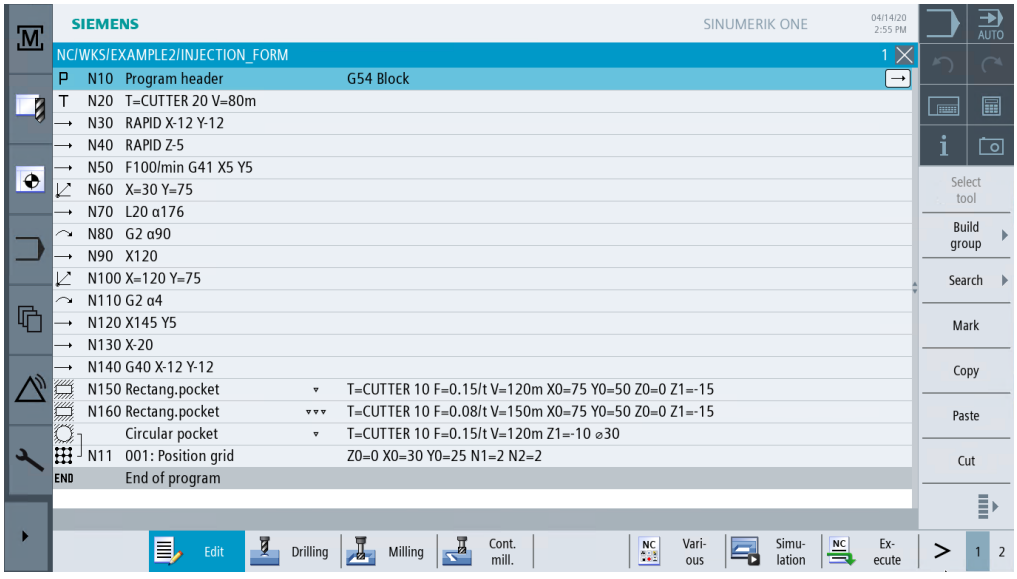


Figure 12-2 Opening the process plan

Select the **NC selection** softkey.

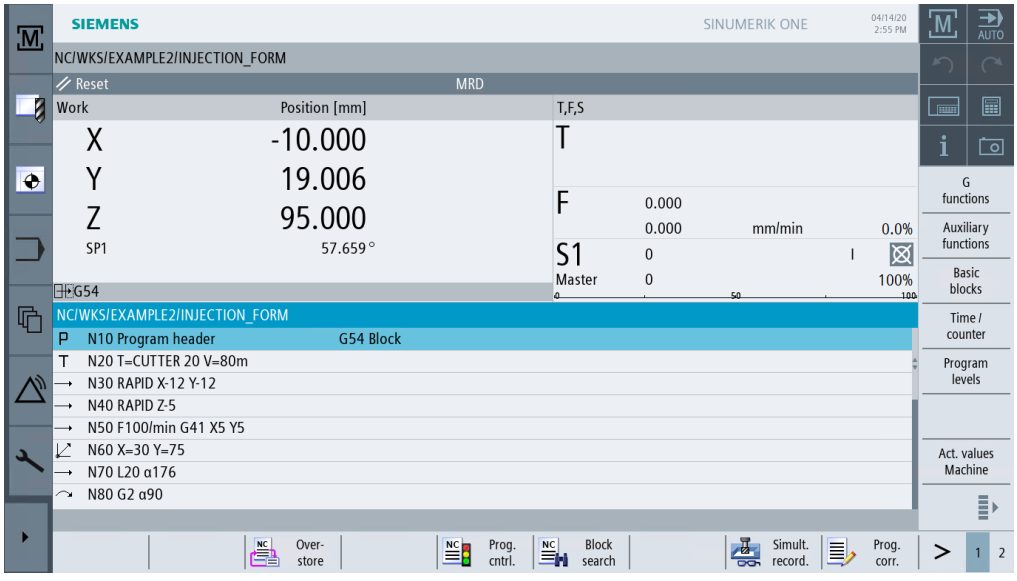


Figure 12-3 Executing

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

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

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

Execution in Run MyVirtual Machine /3D

13

13.1 Overview

Learning objectives

In this chapter you will learn ...

- how you create a machine project in Run MyVirtual Machine using a template and how you start the machine project,
- about the function areas of Run MyVirtual Machine /3D and the components required for the 3D simulation,
- how you activate the setup for the 3D simulation that matches the NC program,
- how you activate collision detection.

Task

1. Based on the template project of a 3-axis machine, you create a machine project in Run MyVirtual Machine and start the machine project.
2. You familiarize yourself with the fundamentals of Run MyVirtual Machine /3D.
3. In order that you can subsequently execute the program example using the 3D simulation, you must first activate the setup that matches the NC program and collision detection.
4. Start execution using 3D simulation.

Note

You require the appropriate licenses for "Run MyVirtual Machine /Operate" and "Run MyVirtual Machine /3D". You can purchase the software through the Industrial Software Store (https://www.dex.siemens.com). Here, you can also apply for your individual free-of-charge test license!

Result

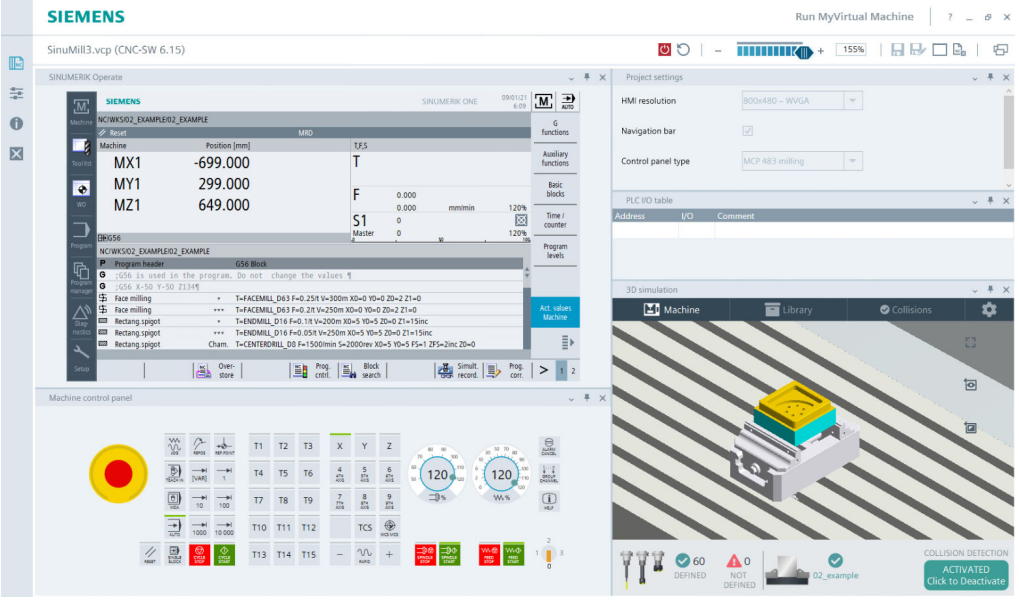


Figure 13-1 Result after execution

13.2 Creating and starting the machine project

Create a machine project in Run MyVirtual Machine based on the template project of a 3-axis machine "SinuMill3.vcp".

Operator workflow

1. Start Run MyVirtual Machine. Project management is displayed after the start.

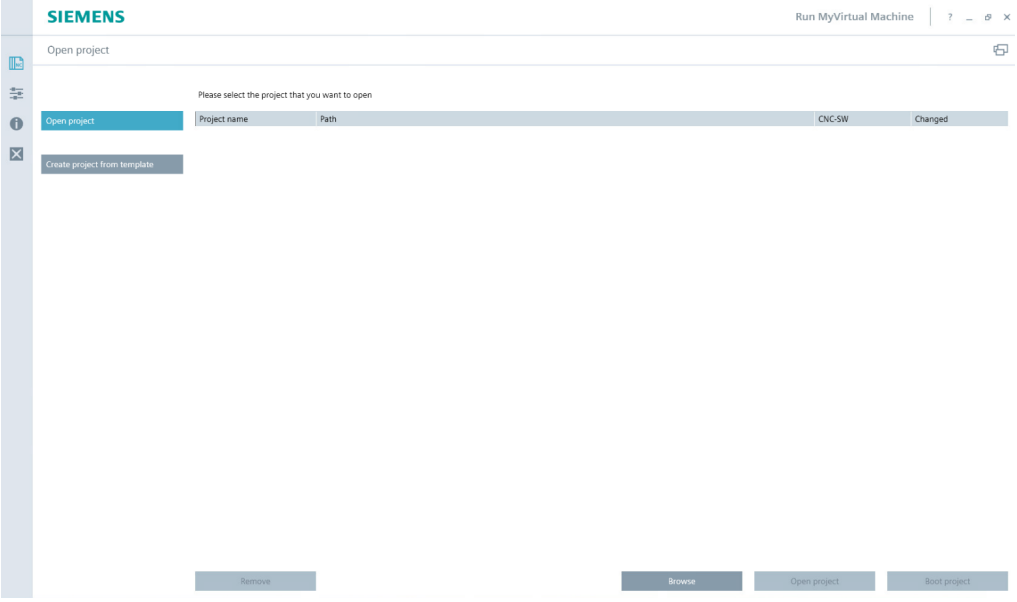


Figure 13-2 Project management

2. Click on "Create project from template". The list of template projects is displayed.
3. In the template project, select "SinuMill3.vcp".

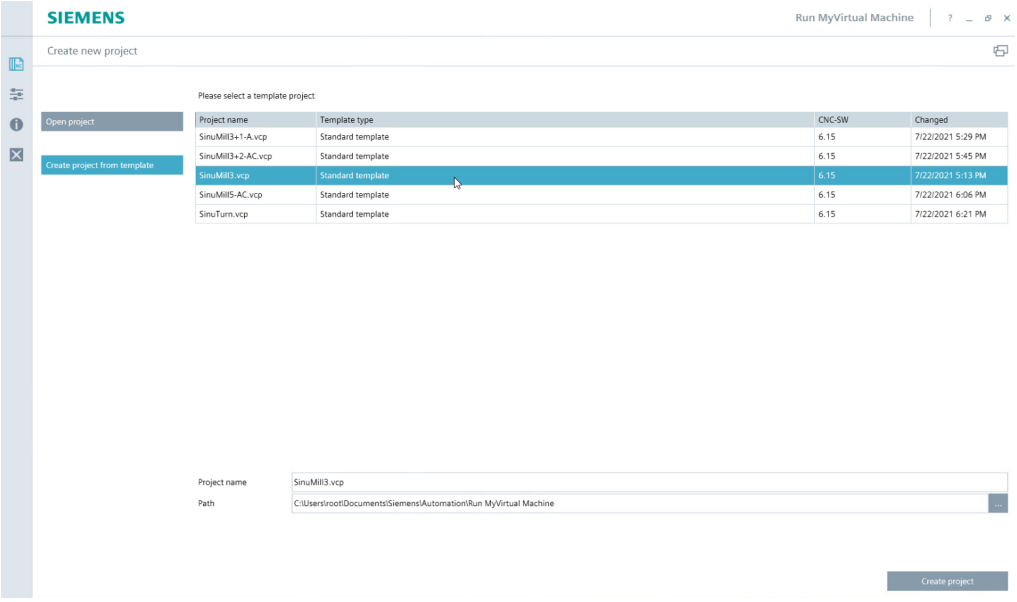


Figure 13-3 Selecting a template project

4. Enter a project name.
5. Select the storage path.
6. Confirm with "Create project". The project opens.

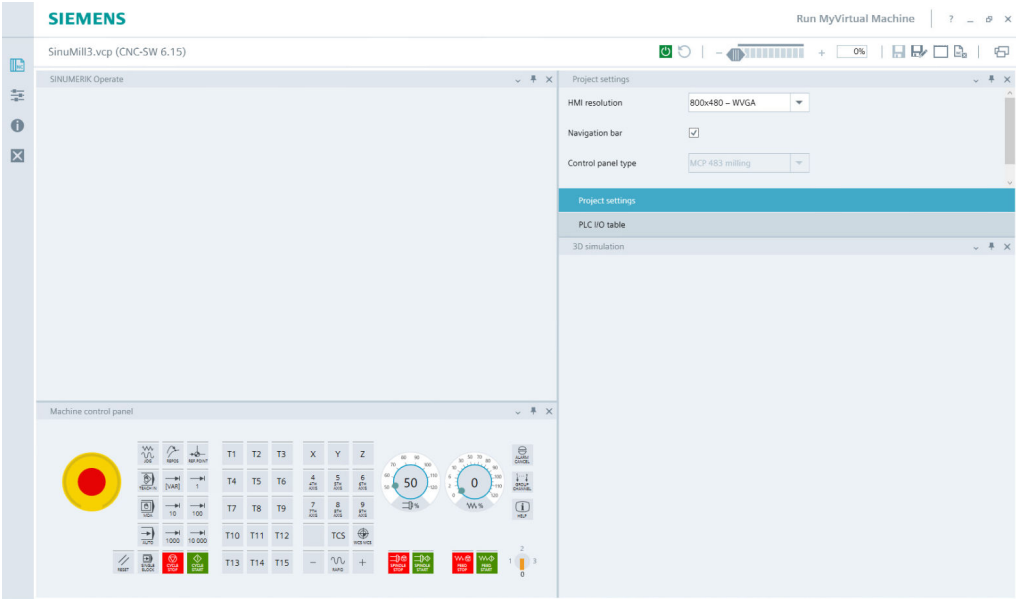


Figure 13-4 Machine project opened

7. Click on button "Start machine" to start the machine project.

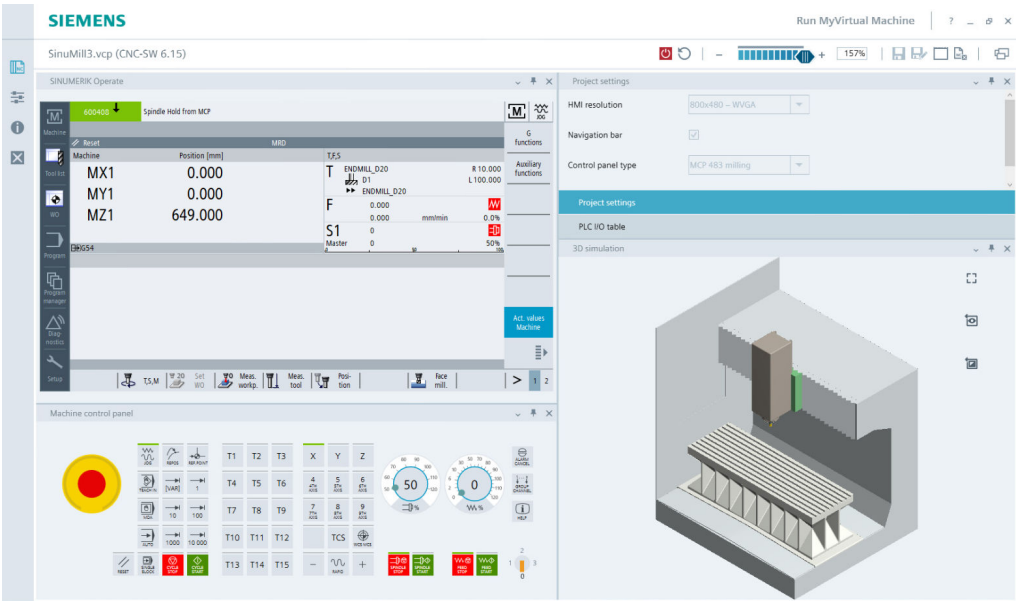


Figure 13-5 Starting the machine project

The machine project is automatically added to the list of last opened projects.

13.3 Opening the program example

Open the program example "02_EXAMPLE.MPF". This workpiece is machined in Section "Start 3D simulation (Page 243)".

Operator workflow

Open the basic menu.

Press the "Program Manager" softkey.

Open folder "02_EXAMPLE". The workshop drawing is saved as PDF.

1. Open workshop drawing "02_DRAWING_EXAMPLE.PDF". You can close the preview using softkey "Close".

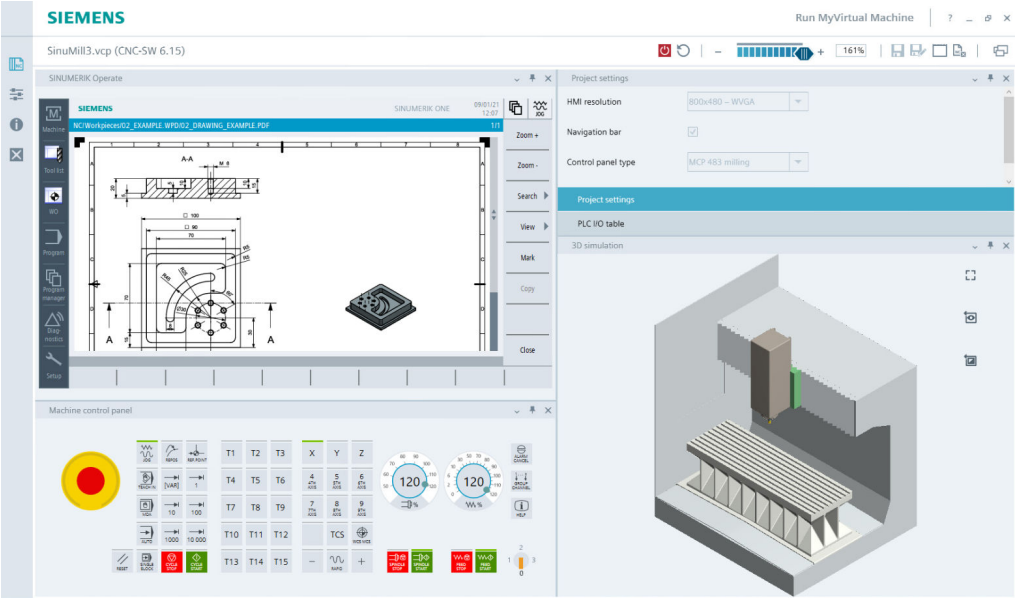


Figure 13-6 PDF preview: Workshop drawing

2. Open ShopMill program "02_EXAMPLE.MPF".

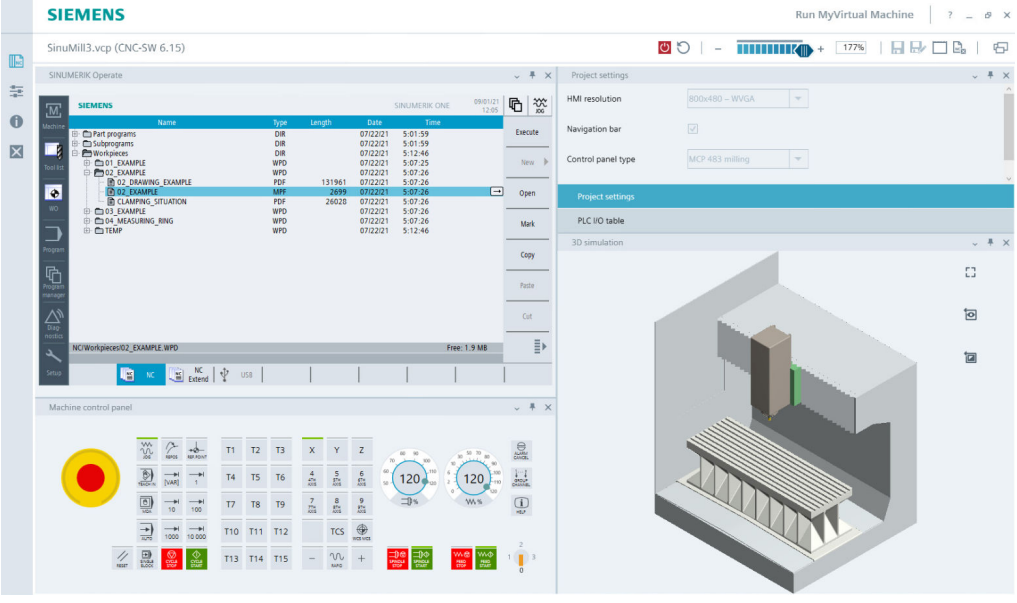


Figure 13-7 Program Manager: Open workpiece

Result

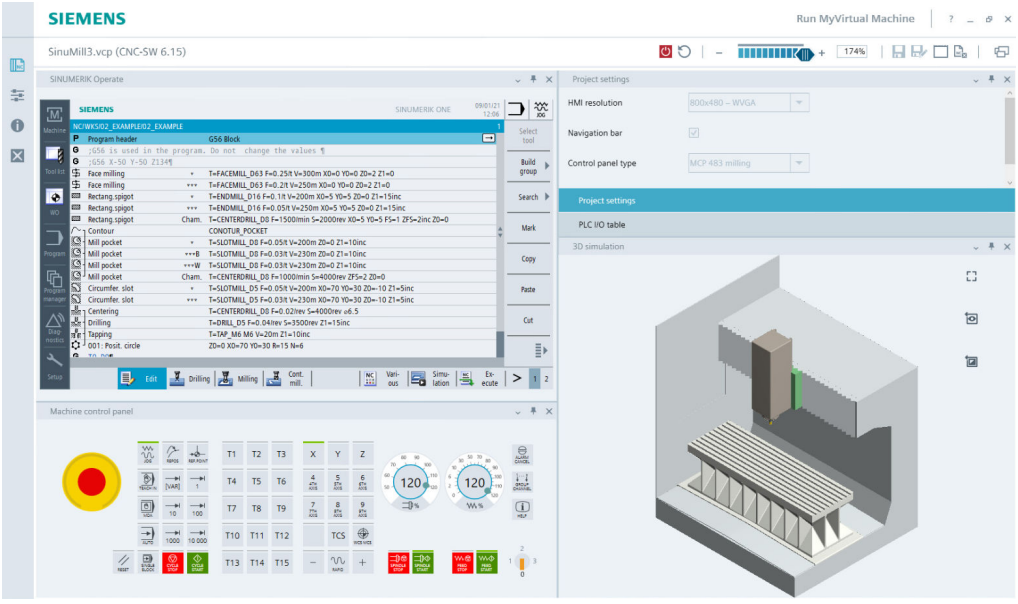


Figure 13-8 ShopMill program "02_EXAMPLE.MPF"

13.4 Fundamentals Run MyVirtual Machine /3D

13.4.1 Overview

In Run MyVirtual Machine you can visualize the machining process and machine movements using 3D simulation. You can simulate the execution of NC programs in the AUTOMATIC mode, for example, or manual traversing movements and tool changes in the JOG mode. You obtain an insight into the user interface of Run MyVirtual Machine /3D, and learn about the components required for the 3D simulation.

Note

You require the appropriate licenses for "Run MyVirtual Machine /Operate" and "Run MyVirtual Machine /3D". You can purchase the software through the Industrial Software Store (https://www.dex.siemens.com). Here, you can also apply for your individual free-of-charge test license!

Overview of 3D simulation

An overview of the function areas of Run MyVirtual Machine /3D is subsequently provided.

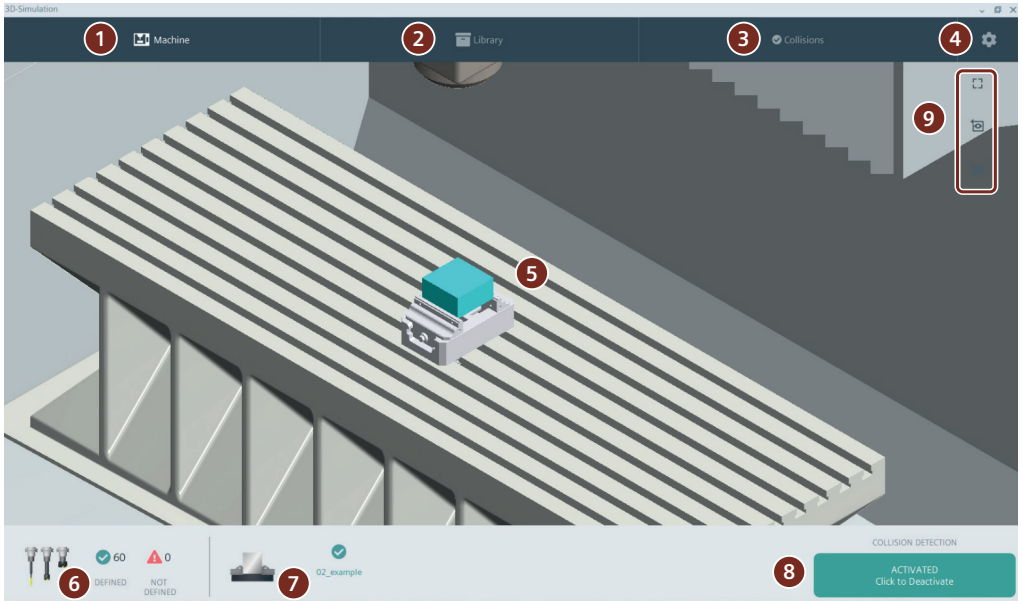


Figure 13-9 Overview of 3D simulation

You can perform the following tasks in the areas:

- ① Machine
  In the "Machine" tab, you configure the simulation components and visualize the machining process.
  - ⑤ Simulation of the machining process and machine motion
  - ⑥ Definition of tools
  - ⑦ Selection and activation of the setup (clamping)
  - ⑧ Activation of collision detection
  - ⑨ Show/hide machine menu, reset view/workpiece
- ② Library
  In "Library" you manage the components required for simulation.
  - Tool components/holders
  - Setup (clamping operation)
  - Blanks
  - Protection area (workholder)
- ③ Collisions
  In the "Collisions" tab, the detected collisions are logged during the execution of an NC program or during the manual procedure.
- ④ Settings
  - Language switchover
  - Toolholder diameter
  - Export/import of archives

13.4.2 Components in the library

In the library you manage the components tool component, setup, blank and protection area for the 3D simulation.

Overview of library

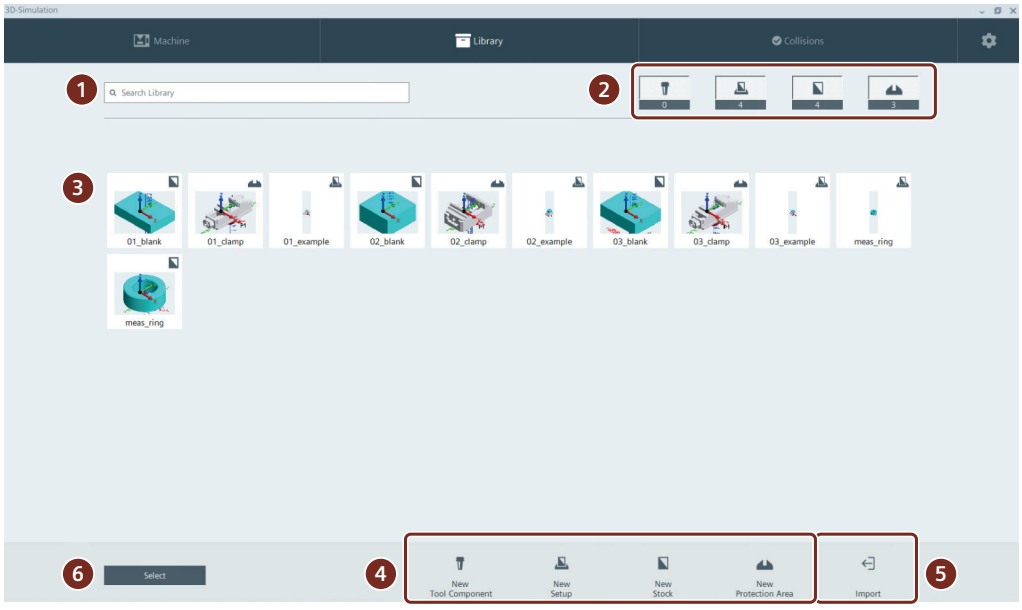


Figure 13-10 Managing 3D simulation components in the library

Operating areas:

① Search for configured components: Here you enter the term to be found in the library and press the ENTER key. The hits are displayed in the overview. Click on the "X" in the search field to delete the search term.

② Here the configured components of the library are displayed with icon and number. The buttons also serve as a filter option for the components. If you click on the button, it is greyed out and the components of this type are not shown in the overview. Click on the button again to show the components.

③ Here the overview of all configured components is displayed with preview image, name and symbolic icon. To edit, click on the component.

④ By clicking the buttons you create new components of the selected type.

⑤ Use Import to import library elements as "*.zip" file into the library. The imported components are extracted and displayed in the library. With this function you can import exported components from other machine projects.

The "*.zip" files in the storage folder are available for import.

⑥ Click "Select" to select one or more components from the library. You can, for example, delete the selected components or export them as "*.zip".

The exported component is saved as a file in the storage folder.

Component types

In the following table the components are shown with icon and description.

Symbol	Component	Description
	Tool component	Tool adapter, tool holder For example, holders for tools with shanks with a steep or hollow taper.
	Setup	The setup comprises a protection area and the matching blank. In a setup, you can also configure protection areas and blanks, e.g. clamping towers or also machine vises.
	Blank	Workpiece blank e.g. cube, cylinder
	Protection area	A protection area fixes the blank. Protection areas include, e.g. machine vises, chucks or clamping jaws.

Components are described as "*.stl file" (3D model). You create an "*.stl file" using an external program and import it when you create a new component.

The components are already defined in the project templates for program examples.

Tool component

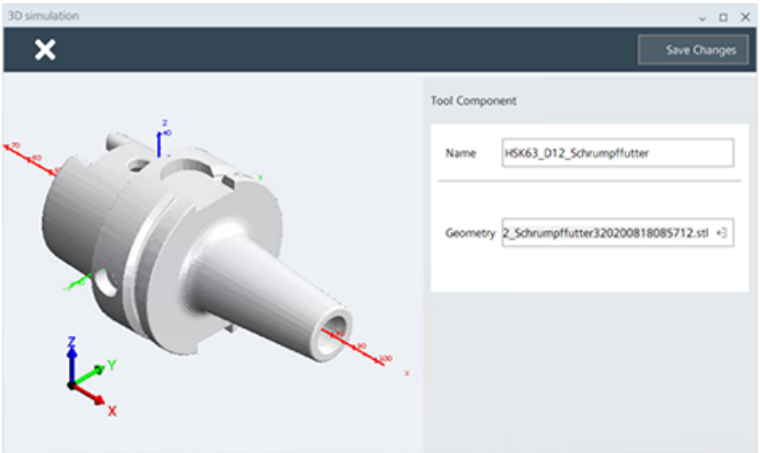


Figure 13-11 Example of a tool component

Setup

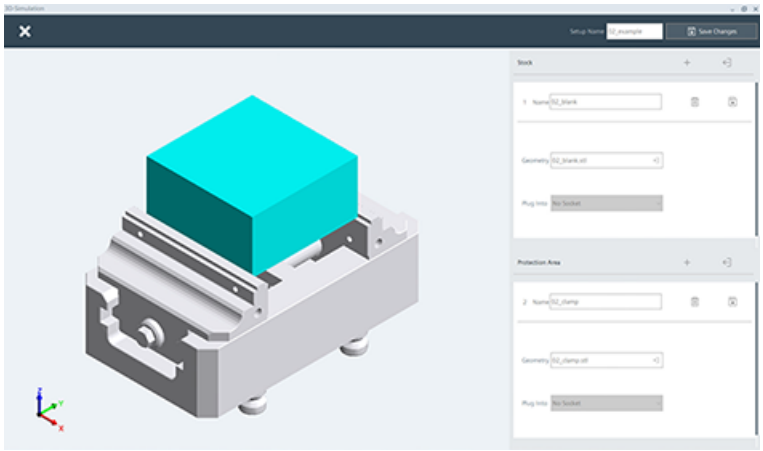


Figure 13-12 Example of a setup

Blank

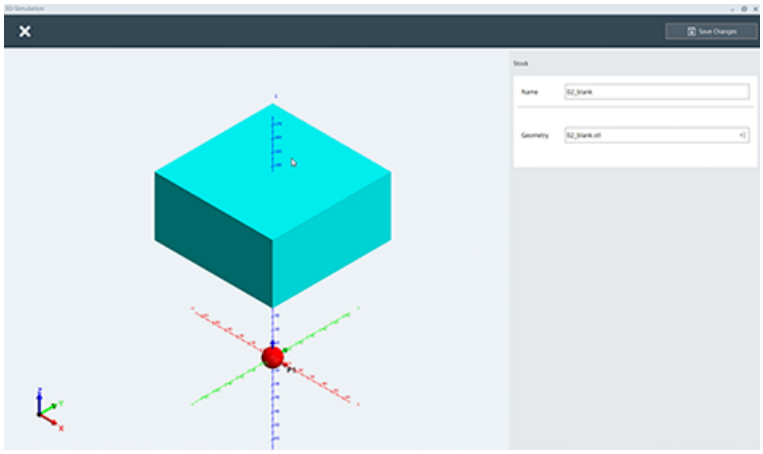


Figure 13-13 Example of a blank

Protection area

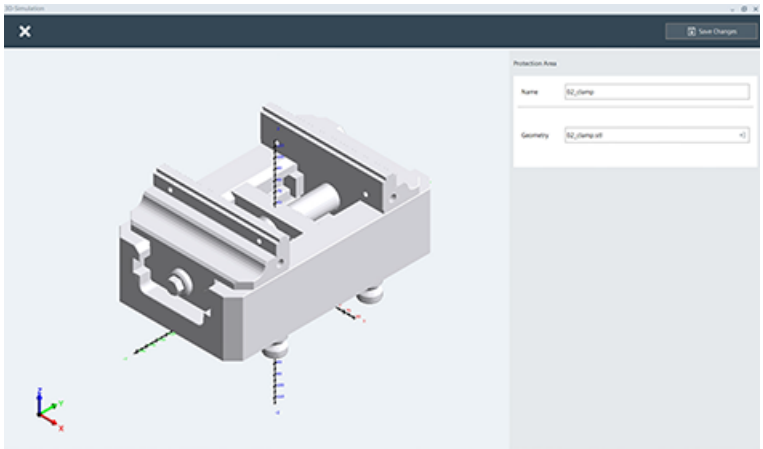


Figure 13-14 Example of a protection area

Note

A protection area can also consist of several components, e.g. a stationary and the movable part of a machine vise.

13.4.3 Tool manager

The tool manager is displayed when you click on the tool icon on the start page of the 3D simulation.

Managing tools for 3D simulation

All of the tools, with their IDs, tool names and preview image are displayed in a horizontal list. The tool manager is synchronized with the tool list in SINUMERIK Operate and contains the tools defined there.

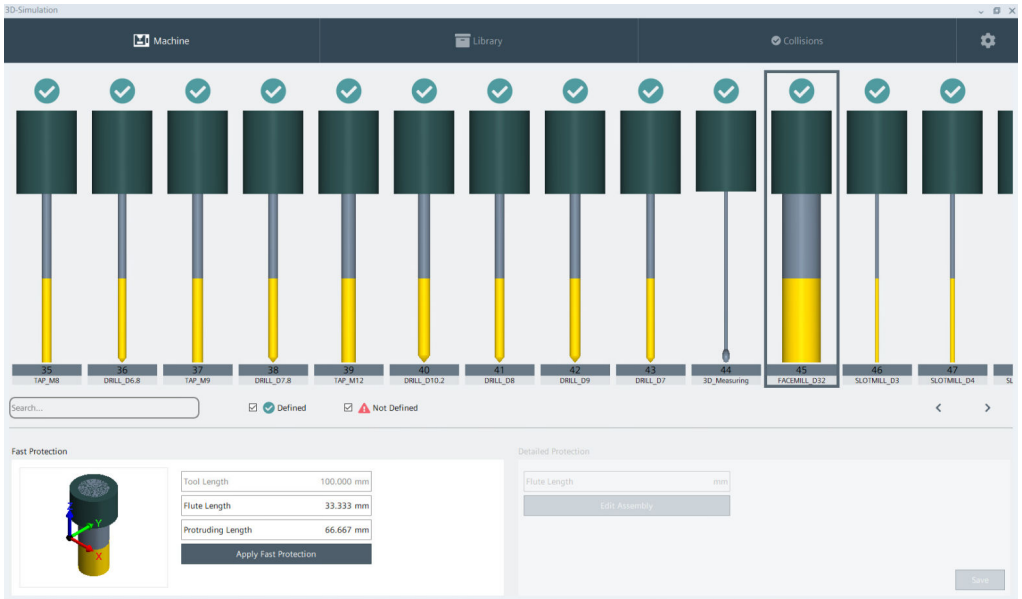


Figure 13-15 Example: Tools with defined protection are marked with a green symbol

Meaning of the protection

In addition to the geometry data from the SINUMERIK Operate tool list, you also define the protection variant in Run MyVirtual Machine /3D. The protection describes the tool holder geometry (workholder) and other geometry data of the cutting edge or the tool. Thus a realistic simulation with collision detection is achieved. Tools with a defined protection are marked with a green checkmark.

13.4.4 Setup manager (clamping)

The setup manager is displayed when you click the "Setup" icon on the 3D simulation start page.

Active setup (clamping)

In the setup manager, activate/deactivate the setup effective for 3D simulation.

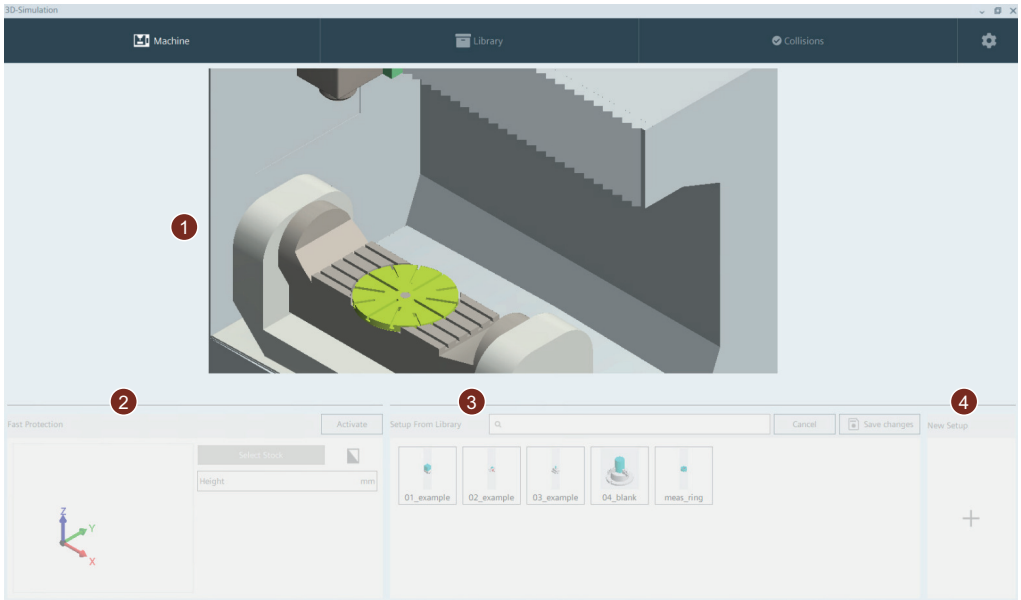


Figure 13-16 Example: Setup manager before activating the setup

The possible setups are shown in gray. Click on the area 2-4 to parameterize the corresponding setup.

① Preview of the machine space with activated setup
② Activate setup as "Fast protection"
③ Activate setup as "Detailed protection" from the library
④ Creating a new setup
The newly created setup is saved in the library and can then be activated (see ③)

Note

- With the "fast protection" you define a protection area around a blank geometry. An "*.stl file" of the workholder is not required.
- To define "detailed protection", activate a setup (clamping) from the library.

13.4.5 Collisions

During 3D simulation of the machining process, collisions that occur while collision detection is active are logged and visually displayed in the 3D simulation. The collisions are marked by colored highlighting.

- Orange: Safety clearance of the components violated, but no collision has occurred
- Red: collision of the components

Collision log

In the "Collisions" tab, all collisions that have occurred are displayed in tabular form and with a 3D image.

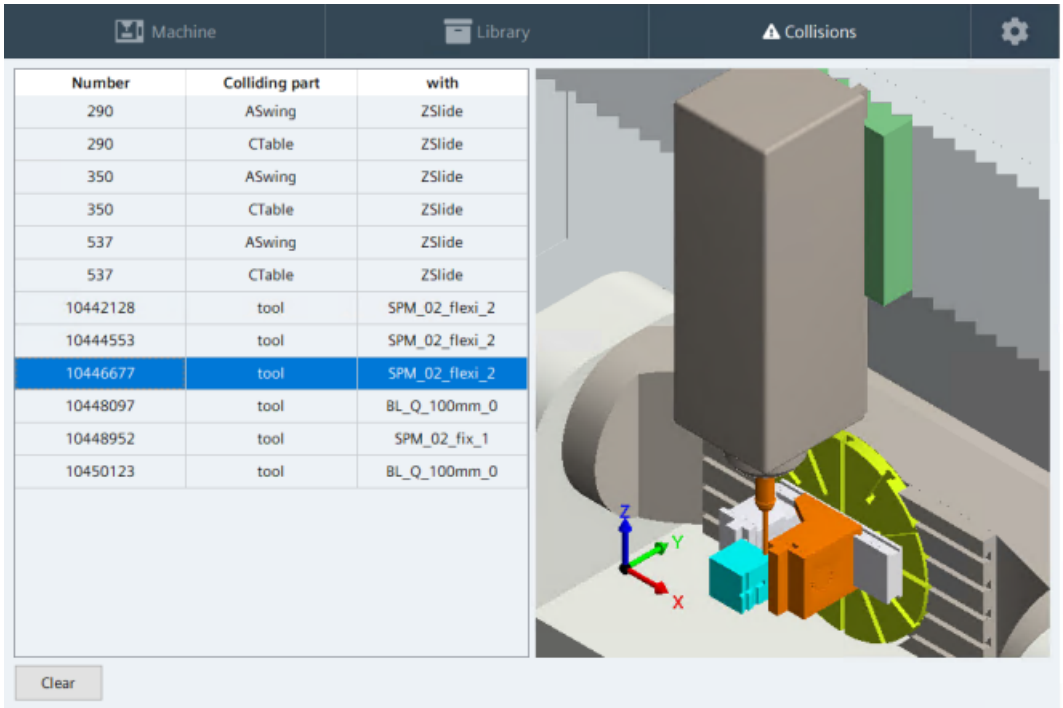


Figure 13-17 Example of a violation of the protection area of the tool with workholder (marked in orange)

The following data is displayed in the table for each collision:

- Number
  Unique number of the collision
- Collision component
  Which component caused the collision?
- Collision with component
  Which component was involved in the collision?
- 3D image of the collision
  A 3D image of the collision is displayed in the right section of the tab.

13.5 Starting execution in the 3D simulation

In Section "Open program example (Page 231)" you opened ShopMill program "02_EXAMPLE.MPF". Execute the 3D simulation in Run MyVirtual Machine /3D based on this example.

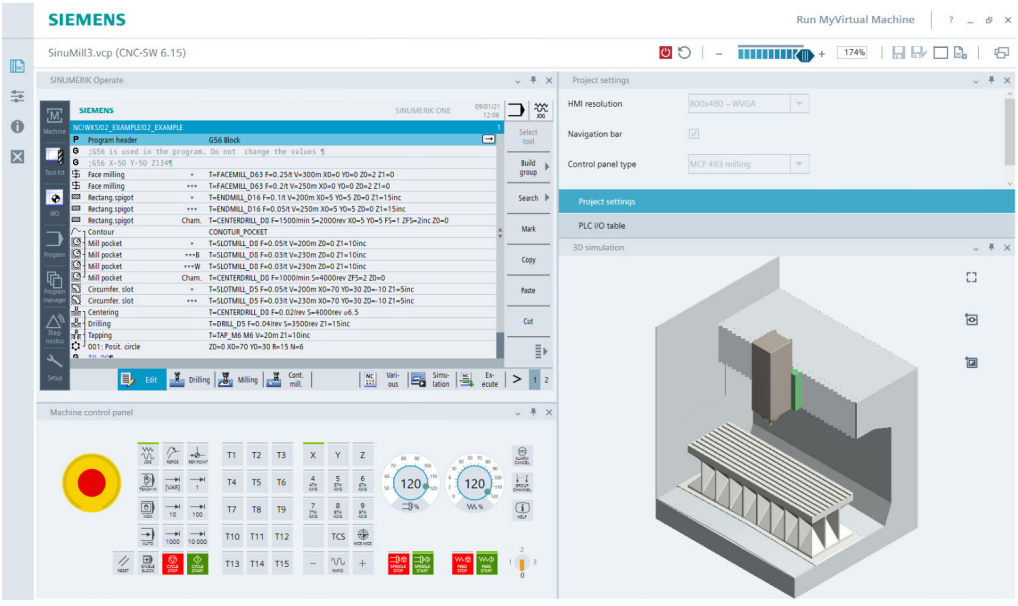


Figure 13-18 02_EXAMPLE.MPF program

Requirements for 3D simulation

The following preconditions must be satisfied in order to use 3D simulation with collision detection:

- Machine project with integrated machine model
- Tool component, setup, blank, protection area are defined in the library
- Protection function and protection areas (workholders) of the tools used are defined

These preconditions are already satisfied for the project templates. You must still execute the following steps:

- Activate the setup that matches the NC program
- Activate collision detection

Operator workflow

1. You can remove the viewlets as windows with the buttons and dock them at any other position in Run MyVirtual Machine. For example, you can display the 3D simulation in a separate window.

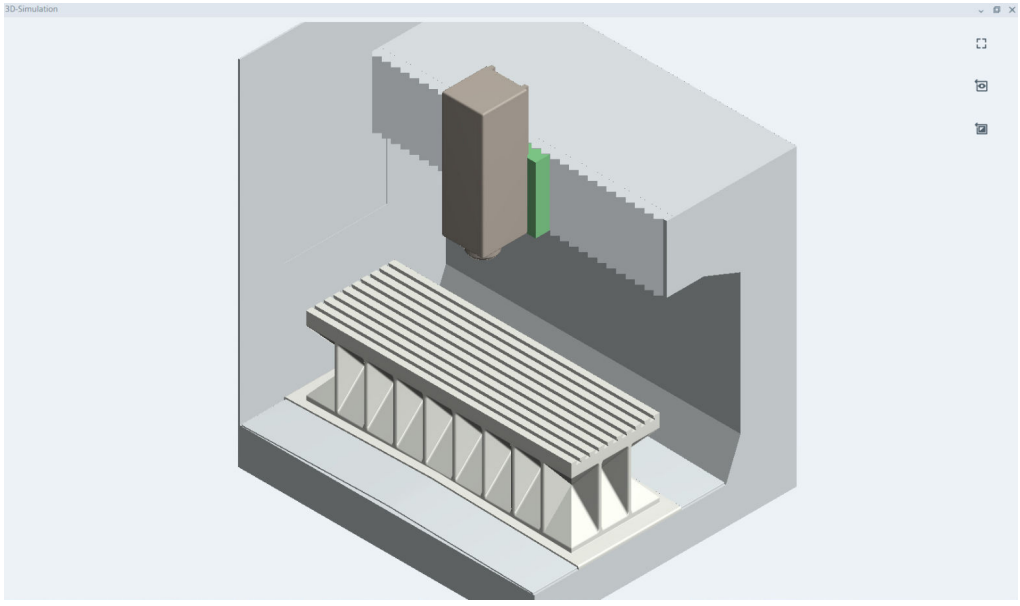


Figure 13-19 3D simulation – window removed

2. Click on icon to display the Machine menu of the 3D simulation.

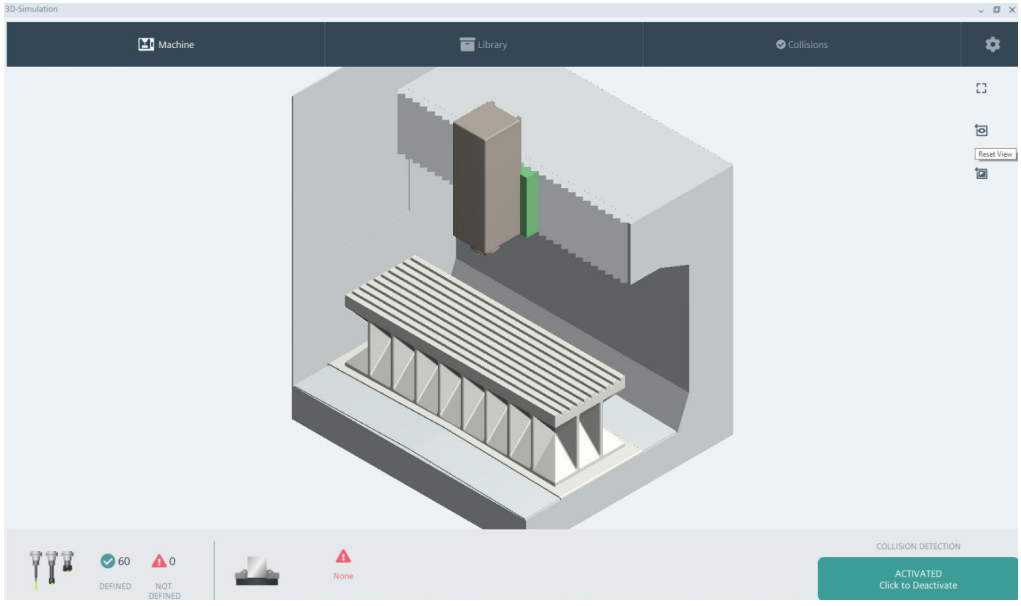


Figure 13-20 Machine menu

3. For the 3D simulation, activate/deactivate the effective setup in the "Setup" tab. The window is displayed when you click the "Setup" icon on the 3D simulation start page.

4. In window Setup, select setup "02_example".

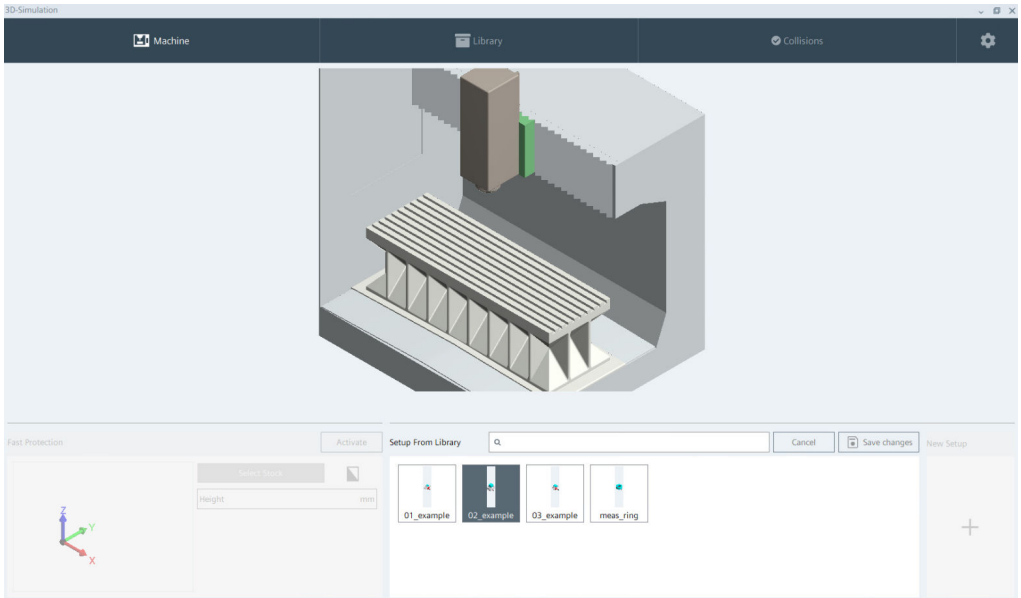


Figure 13-21 Selecting setup "02_example"

5. This is accepted by double-clicking on the selected setup.

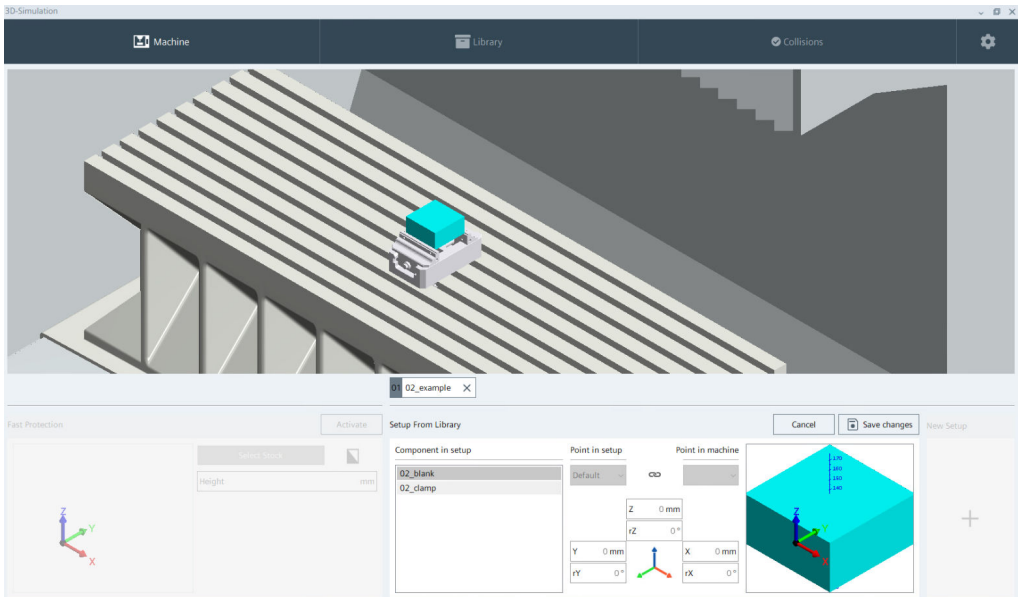


Figure 13-22 Accepting setup "02_example"

6. Confirm by clicking on "Save changes".

7. Check whether collision detection is activated. If collision detection is not active, activate this by clicking on "DEACTIVATED click to activate".

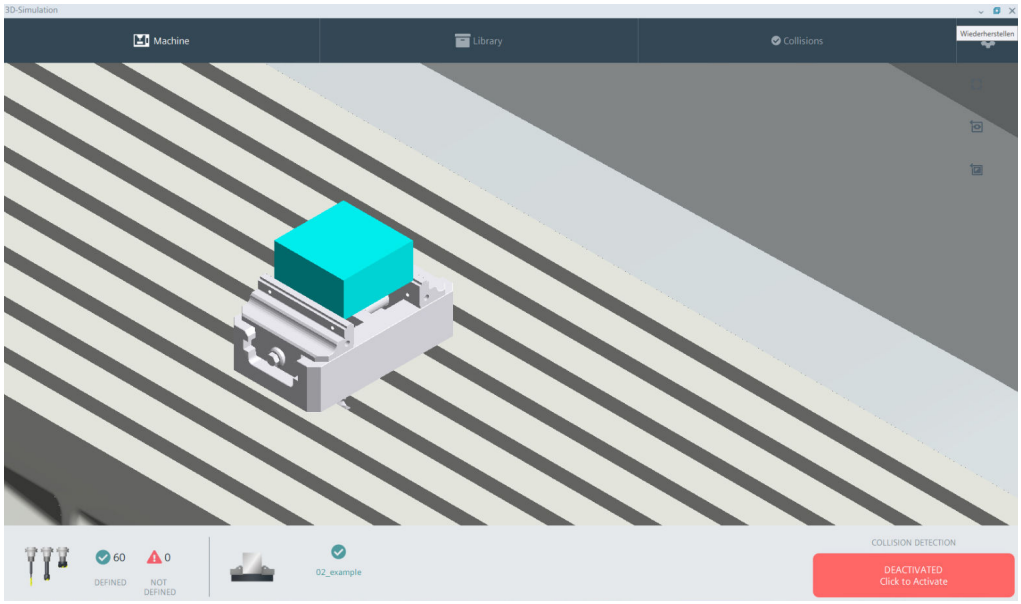


Figure 13-23 Activate collision detection

8. You can now start activation. In SINUMERIK Operate, select softkey "NC selection".

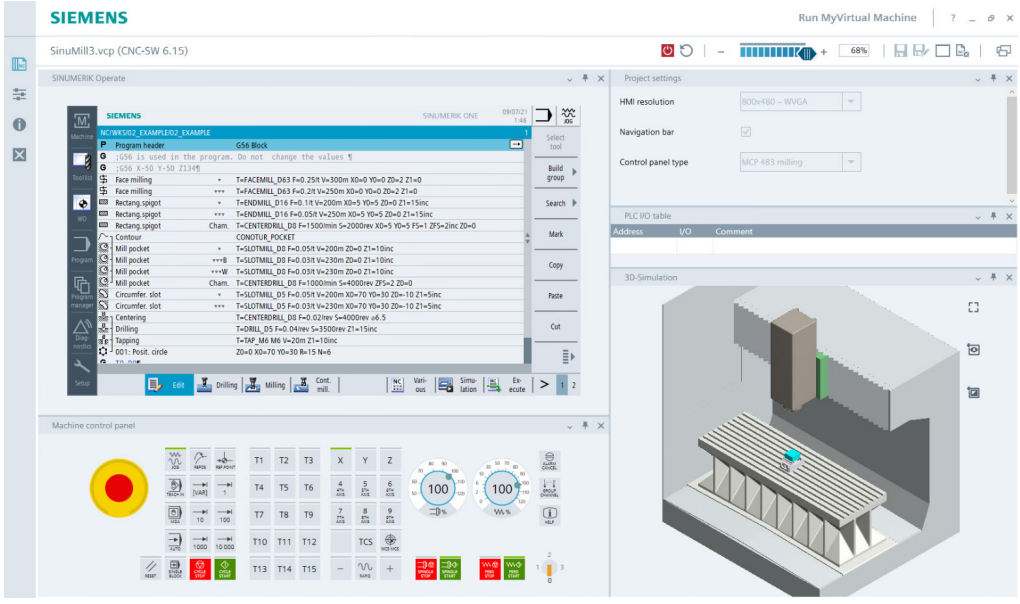


Figure 13-24 NC selection

9. Set the feedrate and spindle speed and enable these via "SPINDLE START" and "FEED START".

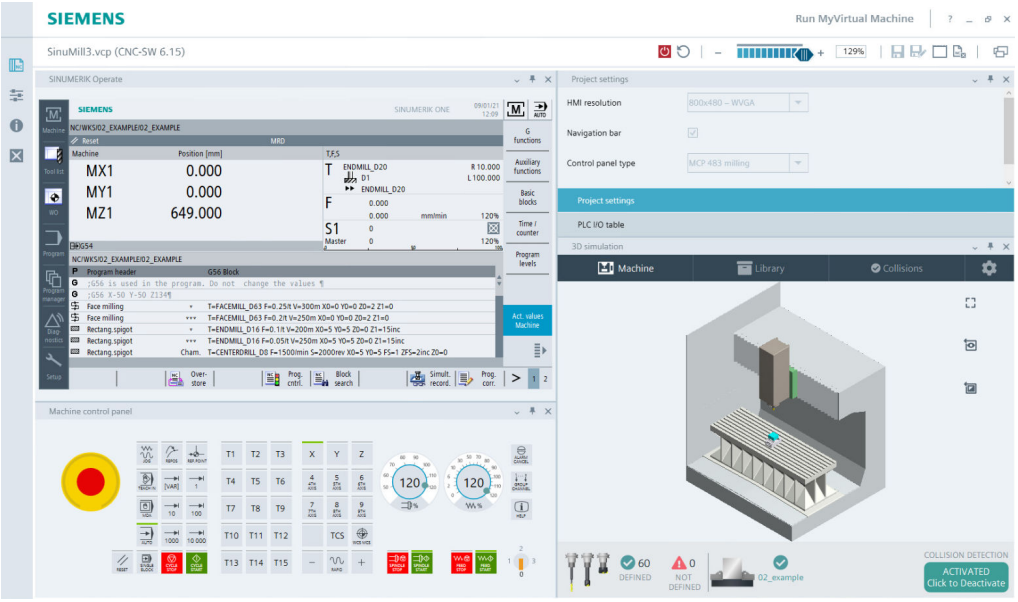


Figure 13-25 Setting the feedrate and spindle speed

10. Click on "CYCLE START" to start execution.
Execution of the workpiece is visualized in the 3D simulation.

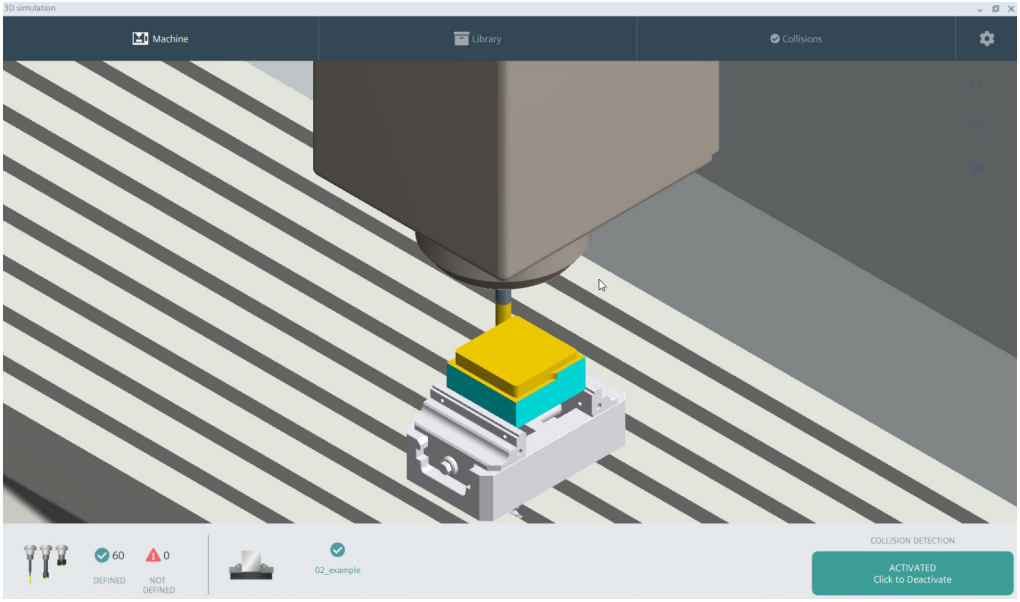


Figure 13-26 Executing in 3D simulation

Result

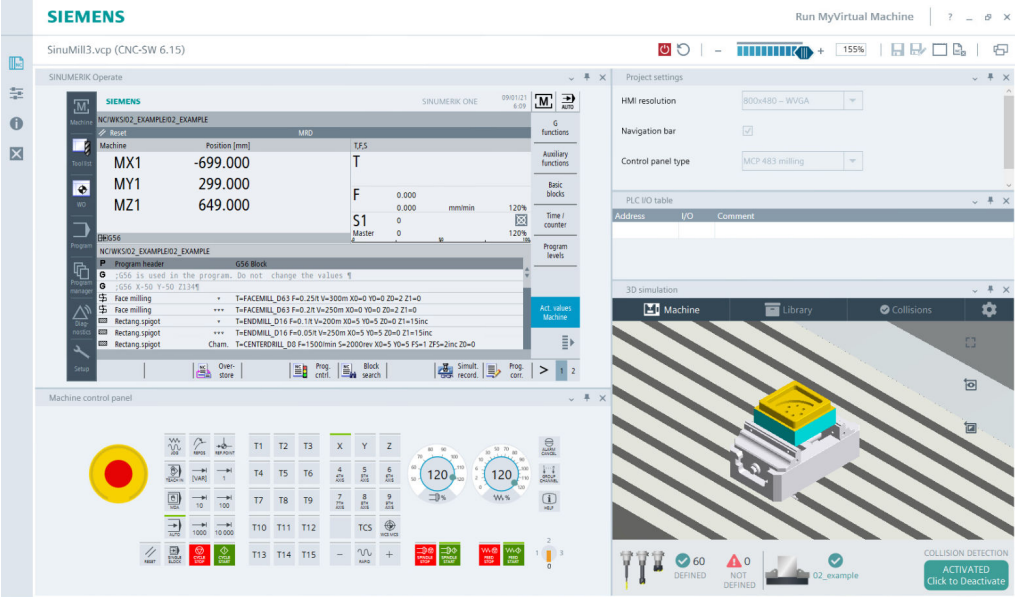


Figure 13-27 Result after execution

How fit are you in ShopMill? 14

14.1 Introduction

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

14.2 Exercise 1

Will you manage this task using ShopMill within 15 minutes?

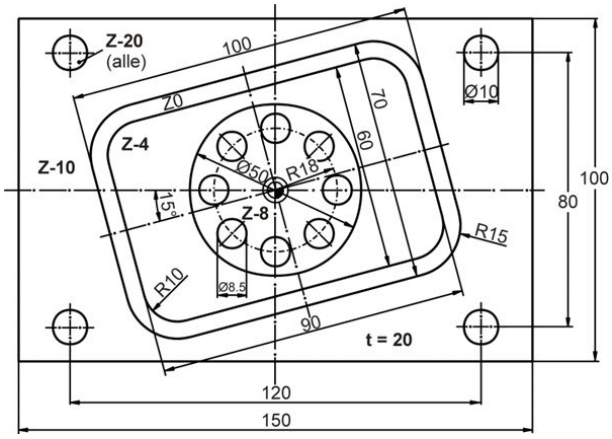


Figure 14-1 Workshop drawing DIYS1

Notes

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

Model

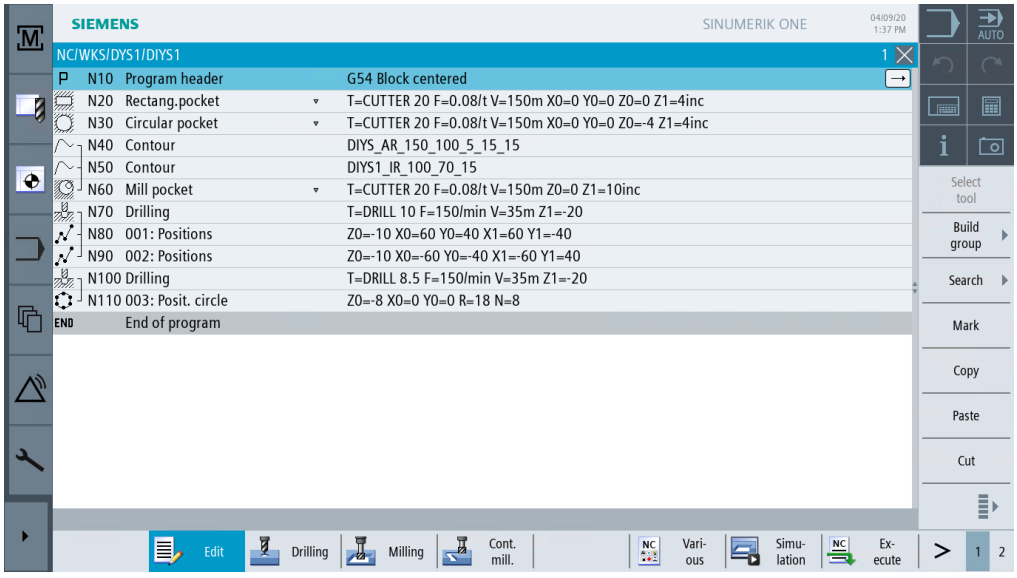


Figure 14-2 Process plan

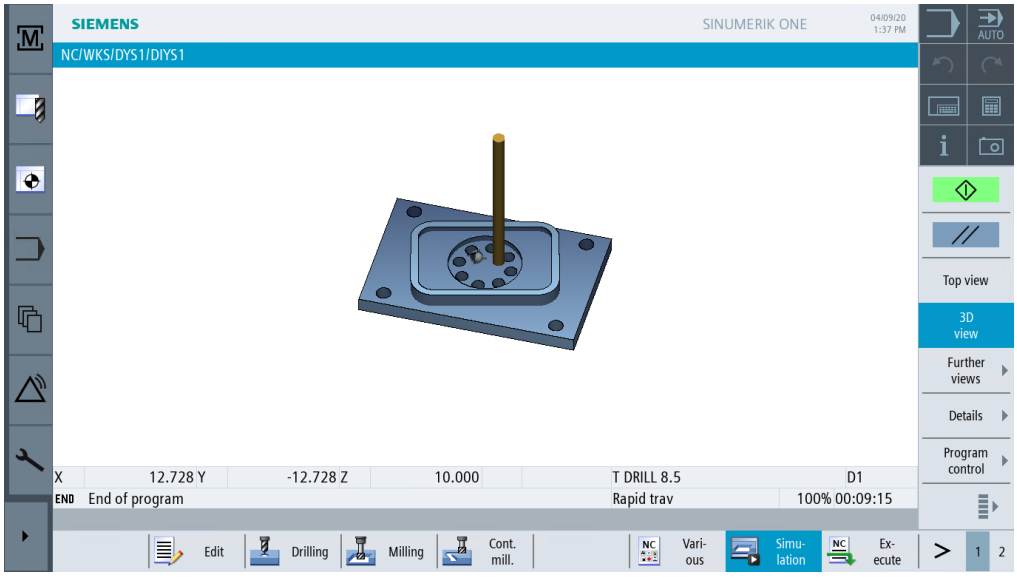


Figure 14-3 Workpiece simulation

14.3 Exercise 2

Will you manage this task using ShopMill within 20 minutes?

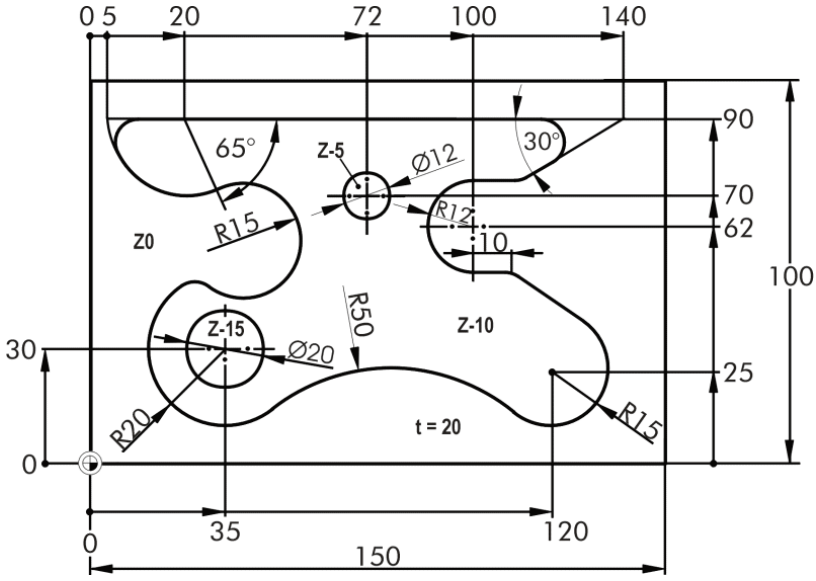


Figure 14-4 Workshop drawing COMPLEX_POCKET

Notes

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

Notes regarding the contour:

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

Model

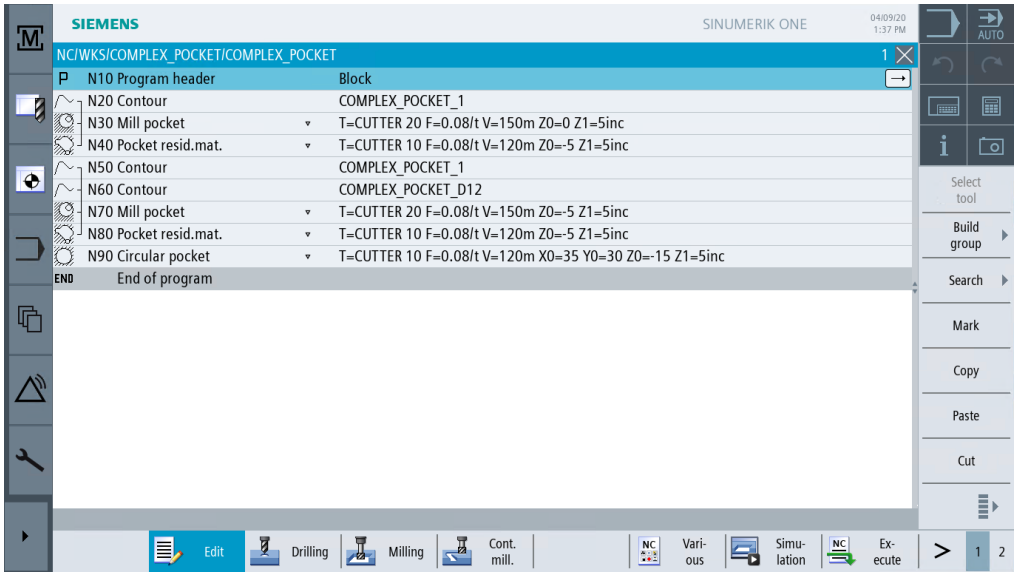


Figure 14-5 Process plan

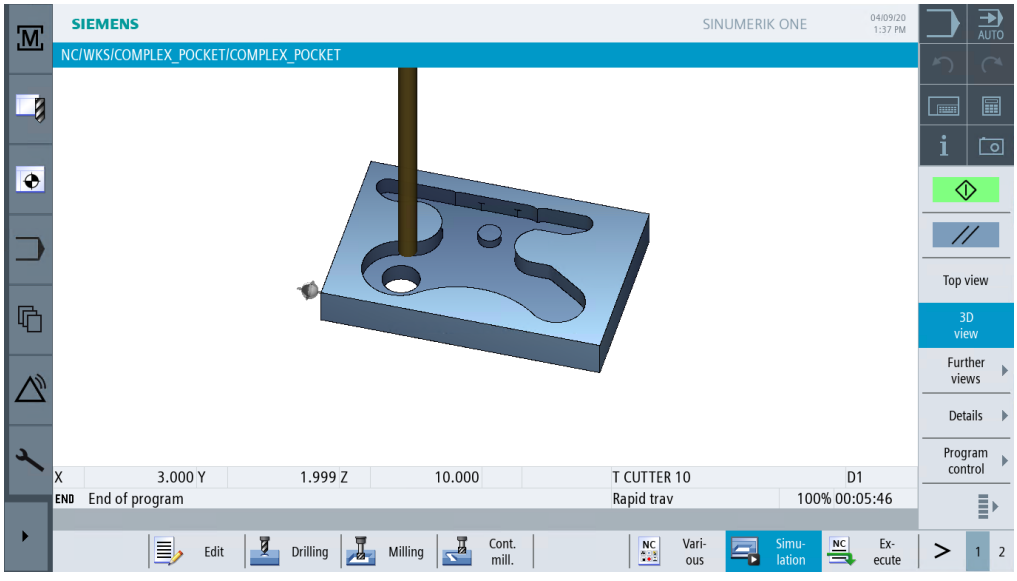


Figure 14-6 Workpiece simulation

14.4 Exercise 3

Will you manage this task using ShopMill within 30 minutes?

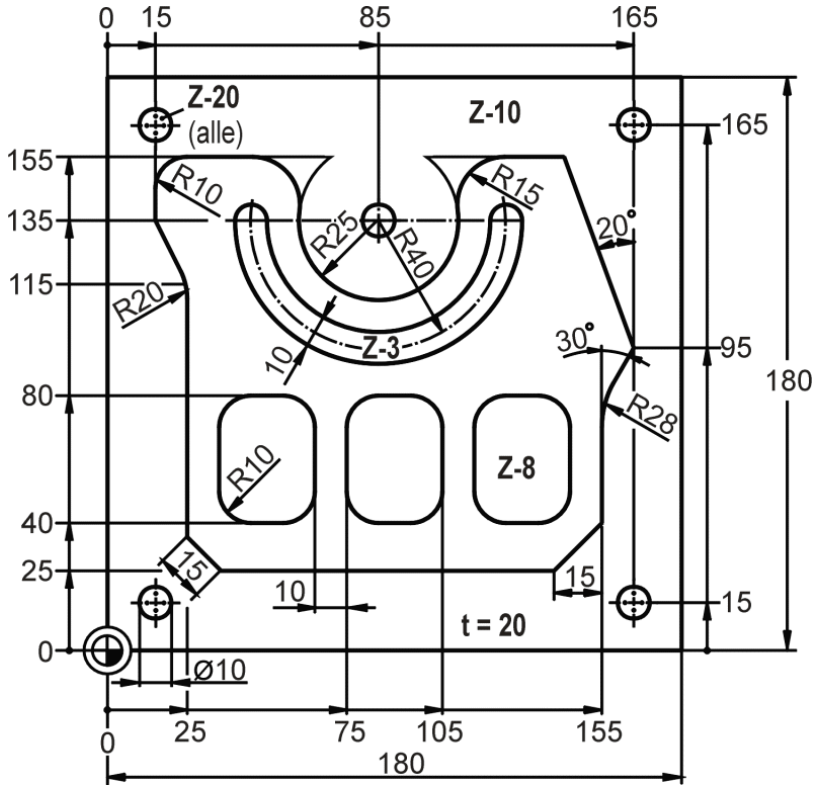


Figure 14-7 Workshop drawing PLATE

Notes

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

Model

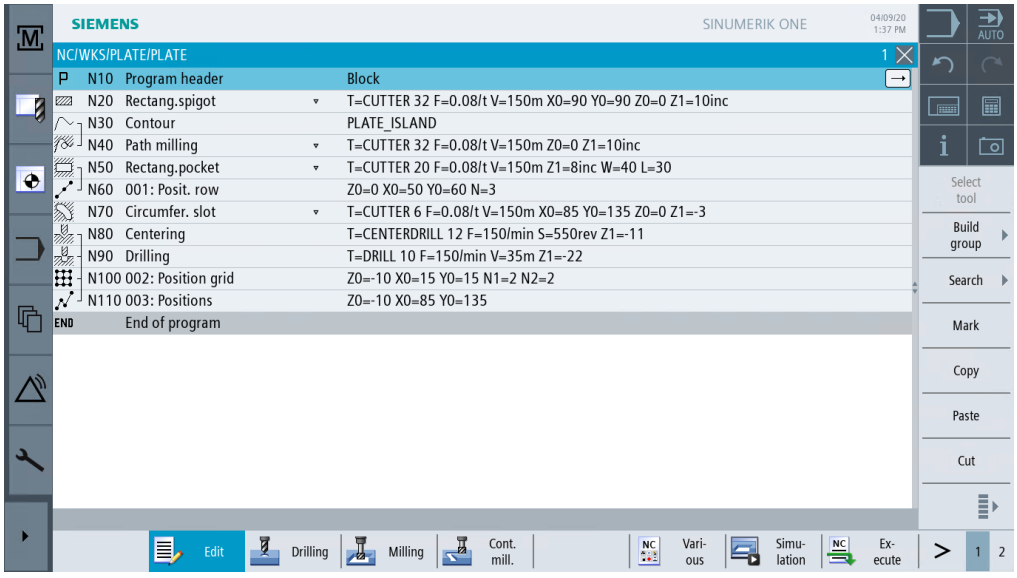


Figure 14-8 Process plan

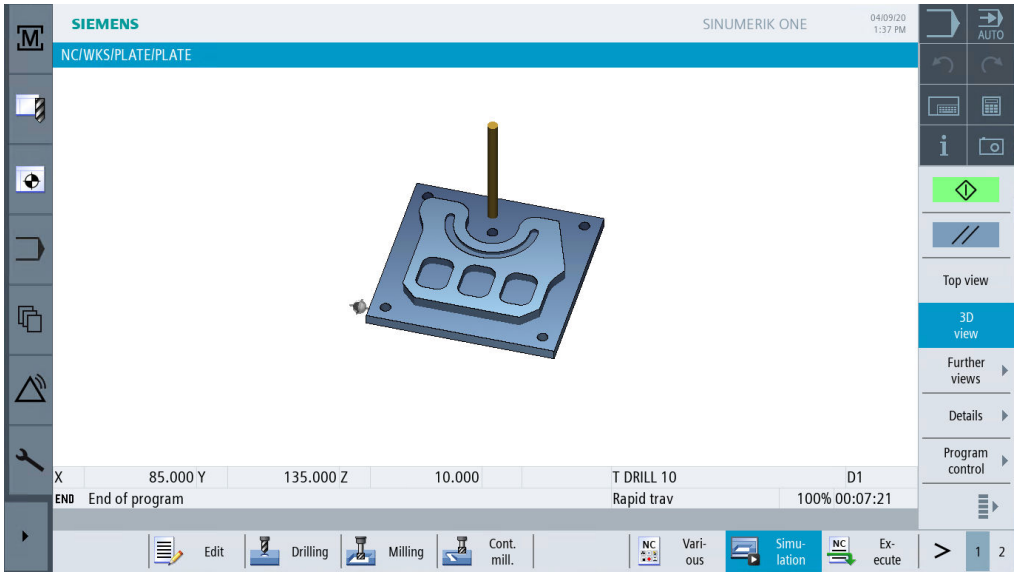


Figure 14-9 Workpiece simulation

14.5 Exercise 4

Will you manage this task using ShopMill within 30 minutes?

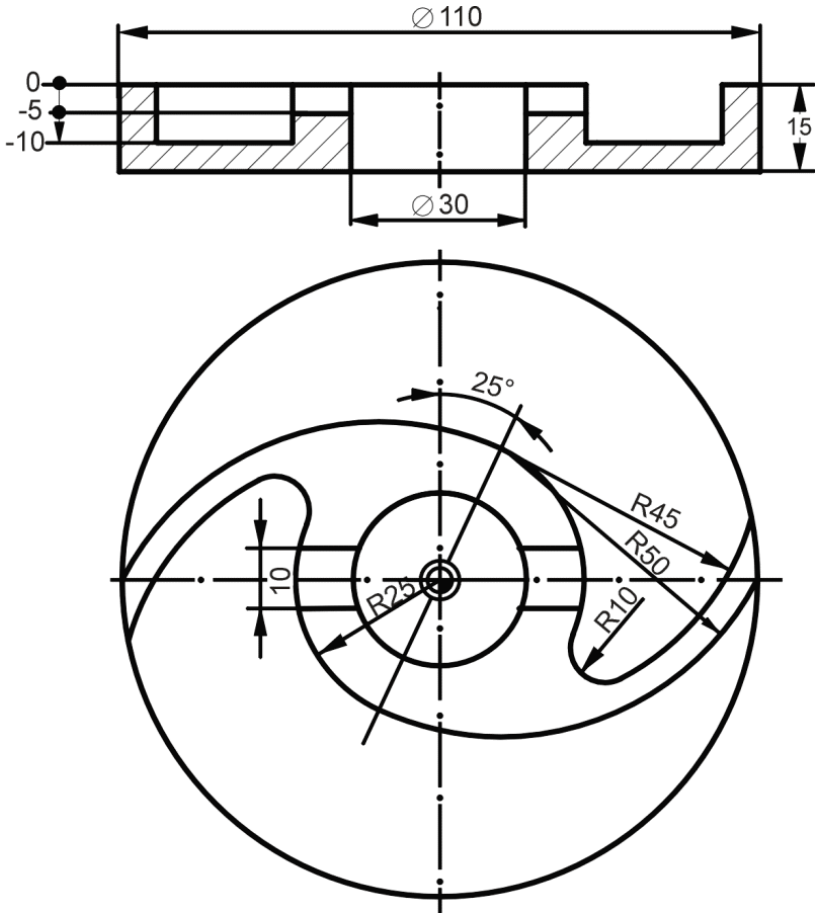


Figure 14-10 Workshop drawing WING

Notes

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

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

Model

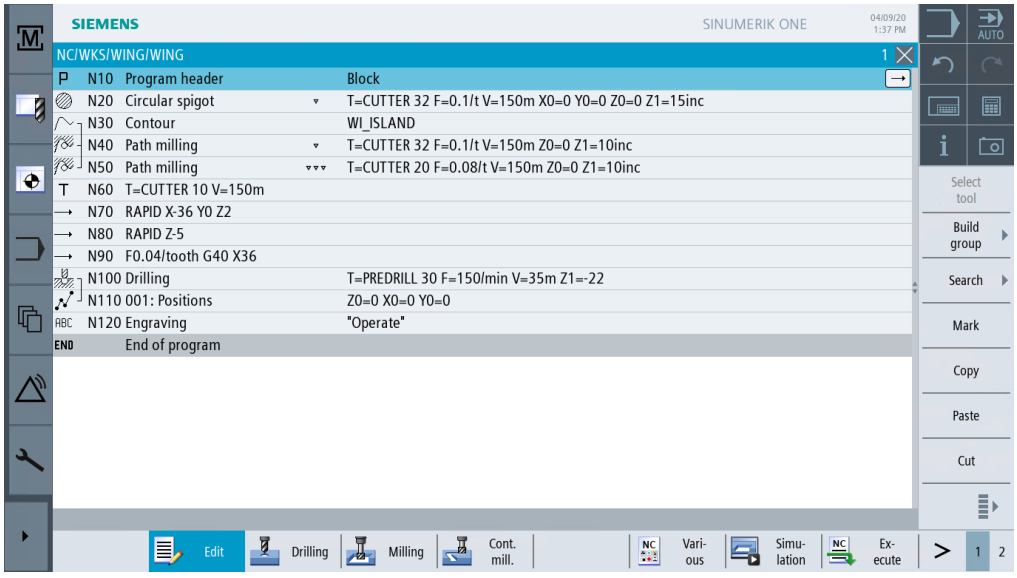


Figure 14-11 Process plan

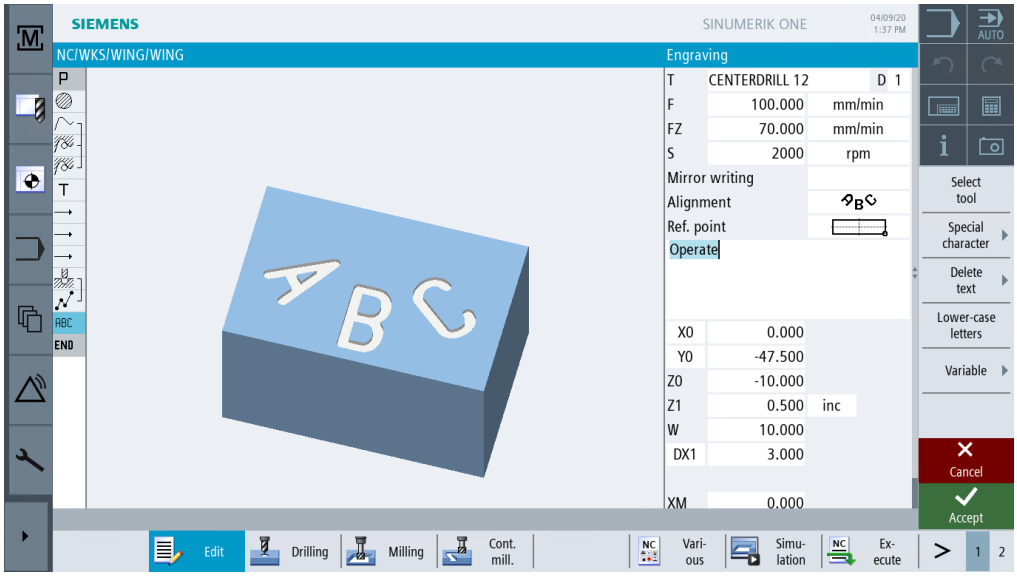


Figure 14-12 Specifying engraving

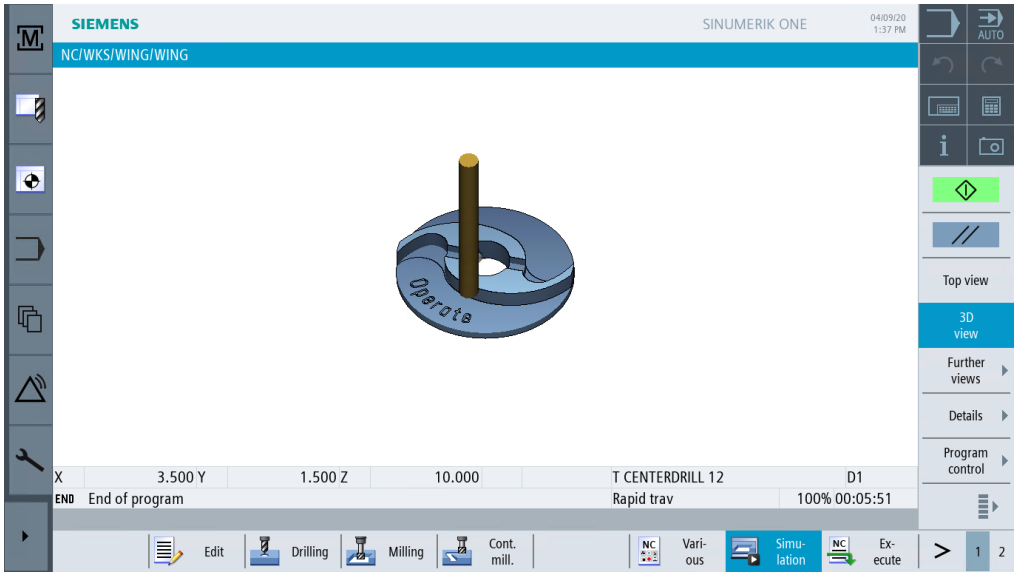


Figure 14-13 Workpiece simulation

Index

R

S

T

V

W