SIEMENS

SCE Training Curriculums

Siemens Automation Cooperates with Education | 02/2016

CNC Technology Module 700-020
ShopTurn Basics

Cooperates with Education
Automation
SIEMENS

Suitable SCE trainer packages to accompany these training curriculums

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

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

In-service training courses

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

Additional information on SinuTrain

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

Further information on SCE

siemens.com/sce

Notes on use

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

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

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

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

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

Contents

1. Objective

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

2. Introduction

Faster from the drawing to the workpiece –but how?

The technological development of machine tools is highly dynamic. Particularly with the creation of NC programs, the range has extended from pure CAM system programming to programming directly at the CNC machine. Special, productive programming methods are available for each area. With ShopTurn, SIEMENS therefore offers a programming solution tailored to the workshop that allows quick programming of machining steps in line with real-world requirements, ranging from the machining of single parts up to small batches. In conjunction with SINUMERIK Operate, the new operator interface for the controller, intuitive and effective working in the workshop is made possible even for series production.

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

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

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

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

Although ShopTurn is easy to learn, this ShopTurn training curriculum allows you to get started in this world even faster. Before starting the actual work with ShopTurn, however, important basics are discussed in the first sections:

- First, we will show you the advantages of working with ShopTurn.
- We show then you the basics of the operation.
- The basics of geometry and technology for machining will be explained for beginners.
- A short introduction to tool management will be given in a further section.

The theory is followed by practical exercises with ShopTurn:

- Five examples have been chosen to explain the possibilities for machining with ShopTurn, whereby the degree of difficulty is increased continuously. At the beginning, all key strokes are specified. Later, you will be prompted to proceed on your own.
- You will then learn how to machine in Automatic mode using ShopTurn.

Note that the technology data used here only serves as an example due to the wide variety of situations in the workshop.

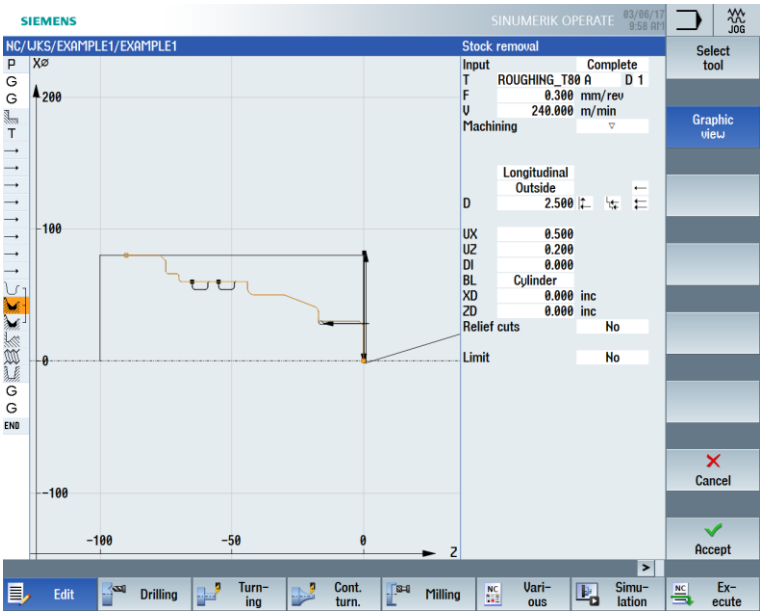
Just as ShopTurn was created with the help of skilled workers, this training curriculum was also elaborated by practitioners. In this sense, we hope you enjoy working with ShopTurn and wish you success.

3. Advantages of working with ShopTurn

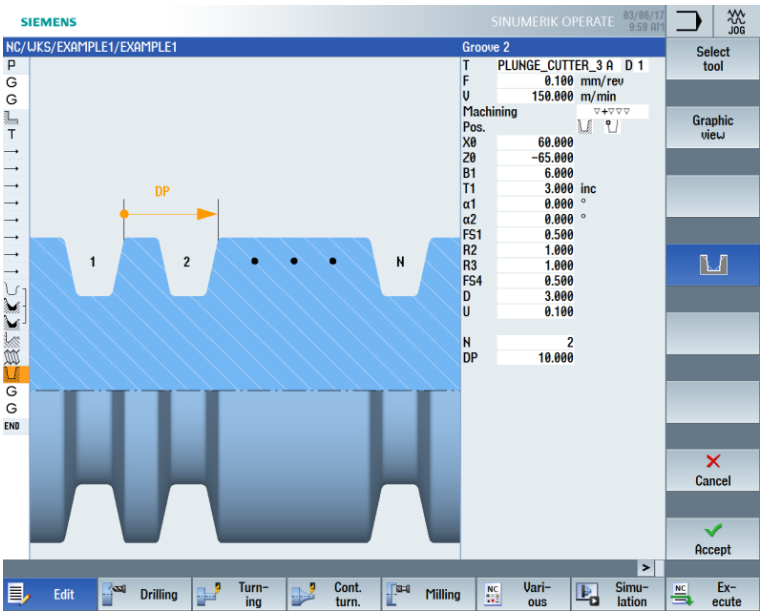
This section shows you the special advantages of working with ShopTurn.

3.1 You save training time:

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



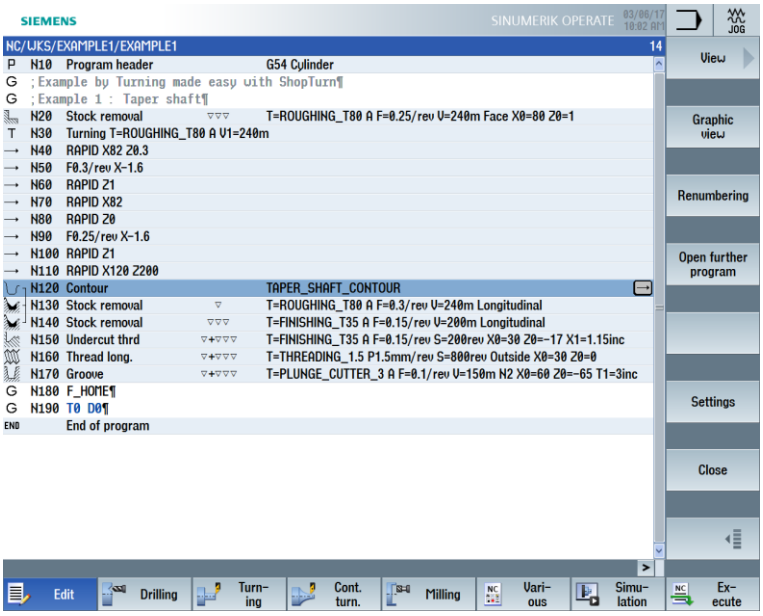
- When working with ShopTurn you are assisted optimally by colored help screens.

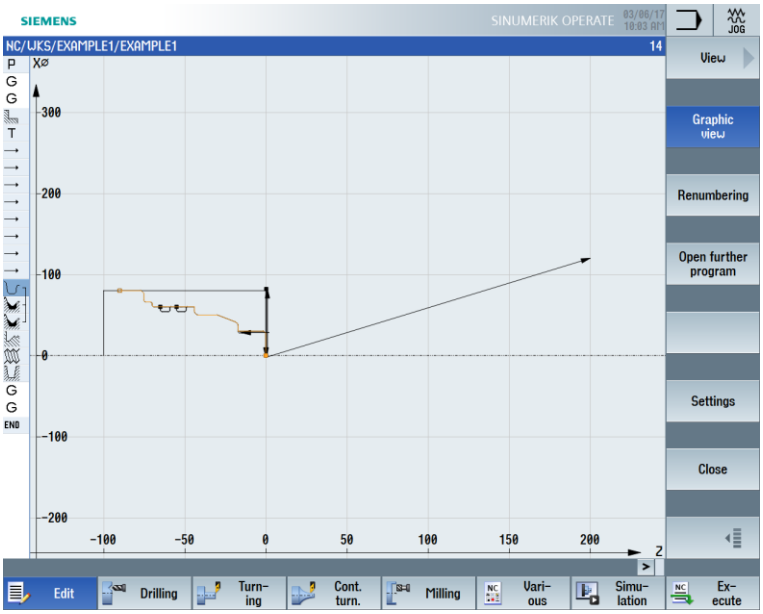


- You may also integrate DIN/ISO commands into the graphical machining plan of ShopTurn. You may also program in DIN/ISO 66025 and use DIN cycles.

```
G   G96 S320 LIMS=3000 M4 M8¶
G   G18 G54 G90¶
G   G0 X32 Z0¶
G   G1 X-1.6 F0.1¶
G   G0 Z2¶
G   G0 G42 X22 Z2¶
G   X30 Z-2¶
```

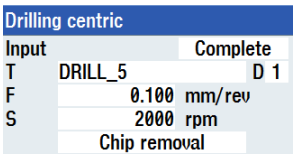
- You may switch between the individual machining step and the workpiece graphic (broken-line graphic) at any time when creating a machining plan.

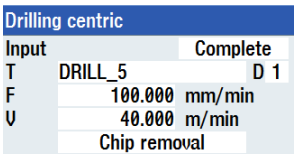




3.2 You save programming time:

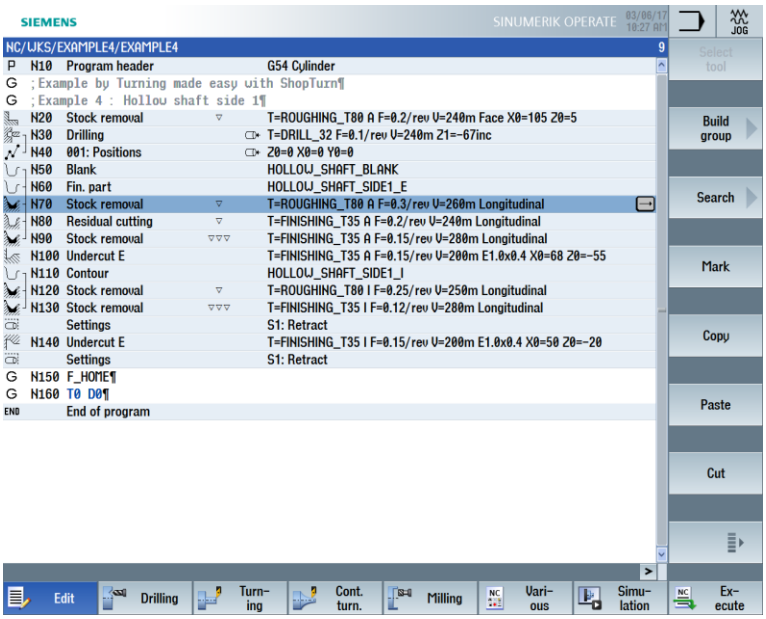
- ShopTurn assists you even when entering the technological values: You only have to enter the handbook values "feedrate" (or feed) and "cutting rate" – the speed is calculated by ShopTurn automatically.



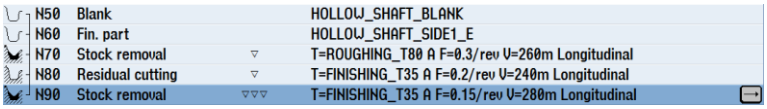


- ShopTurn enables you to describe a complete machining operation with one machining step, and the required positioning motions (in this case, from the tool change point to the workpiece and reverse) are created automatically.

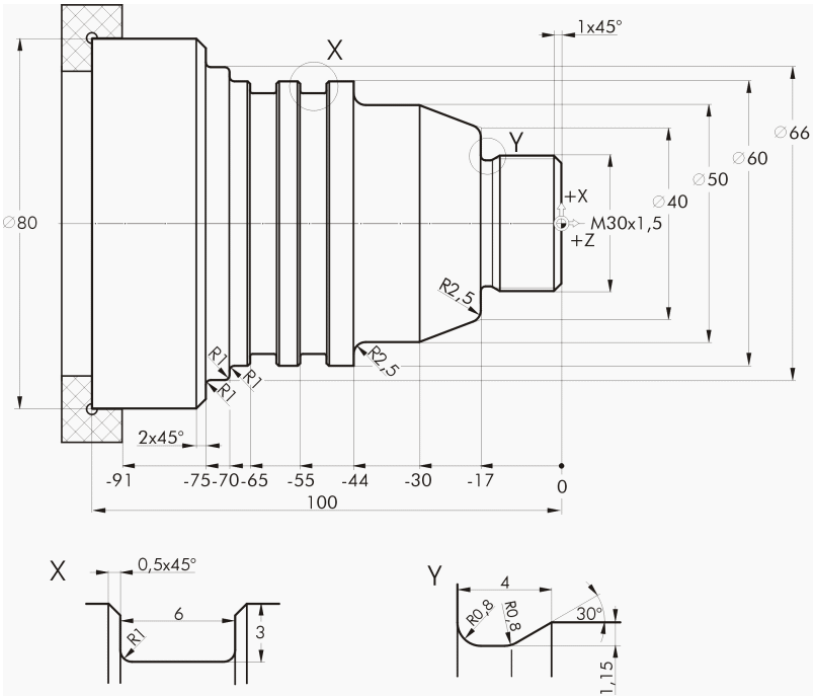
- All machining steps are represented by ShopTurn in a compact and clear fashion in the graphic machining plan. This provides you a complete overview and thus better editing possibilities even for comprehensive machining sequences.

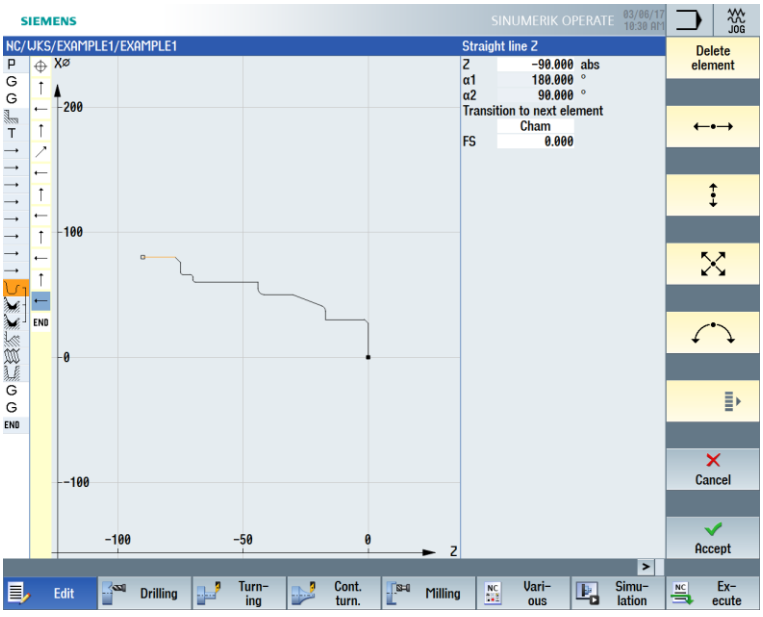


- For stock removal, for example, several machining operations and contours can be linked.

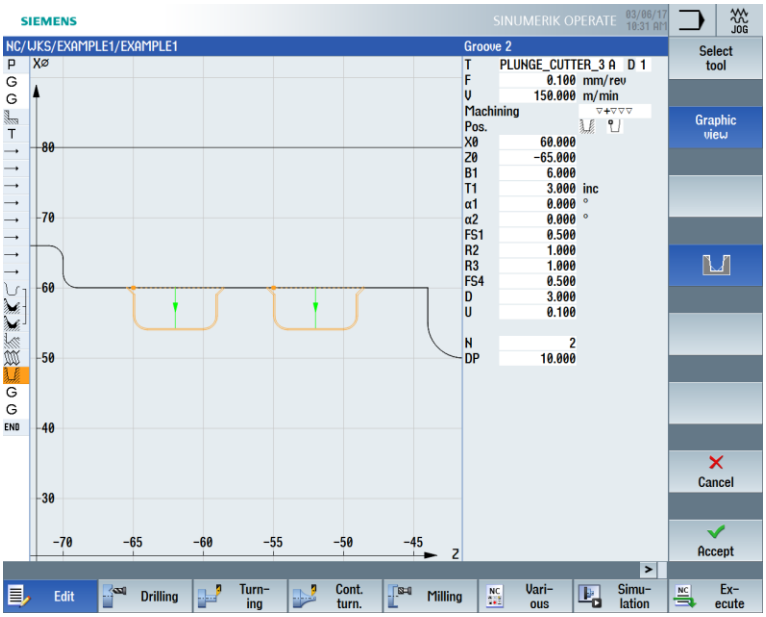


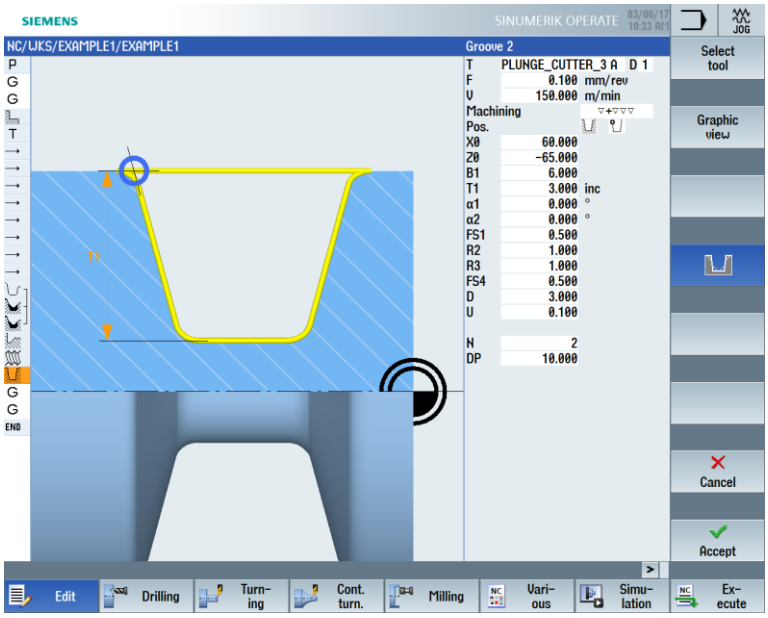
- The integrated contour calculator can process all standard dimensions (Cartesian and polar). It is nevertheless very easy to handle and understand – thanks to colloquial input and graphic support.





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

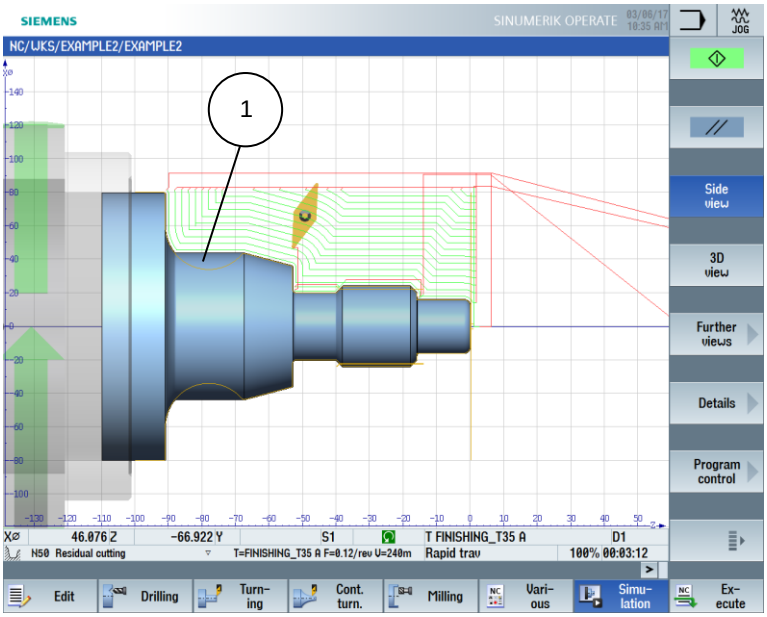




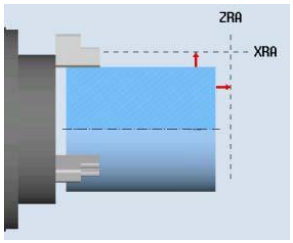
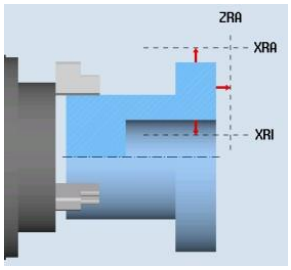
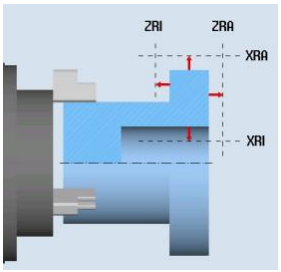
- The creation and the machining of the machining plan are not mutually exclusive. With ShopTurn, you can create a new machining plan in parallel with machining.

3.3 You save machining time:

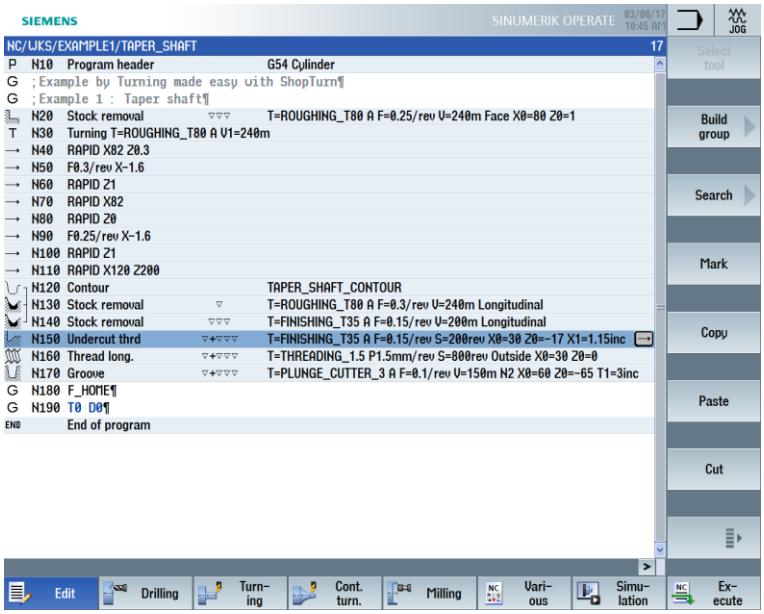
- You can optimize tool selection for the stock removal of contours: Large volumes are removed using roughing tools, and residual material ① is detected and removed automatically using a pointed tool.

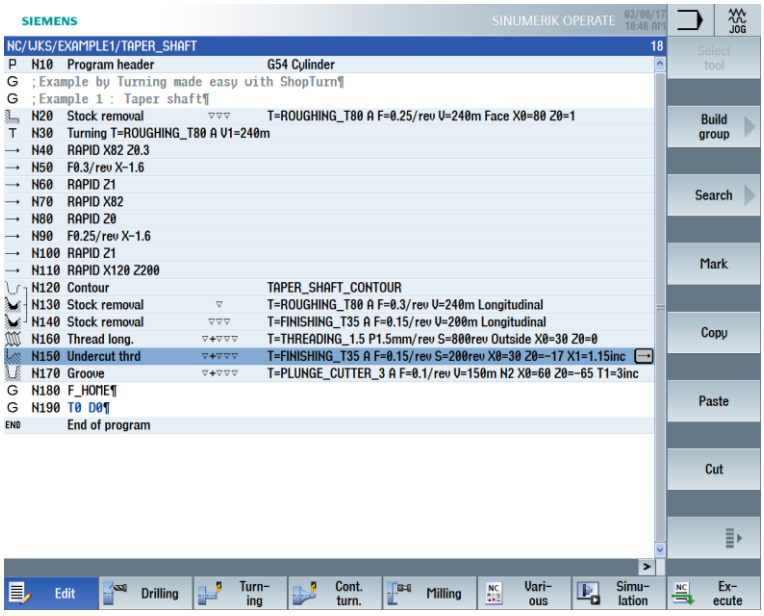


- The ability to pinpoint the selected retraction plane means that unnecessary traversing motions can be avoided, thereby saving valuable machining time. This is possible using the settings "Simple", "Extended" and "All".

Retraction plane: Simple	Retraction plane: Extended	Retraction plane: All
		

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





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

4. To ensure that everything function smoothly

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

4.1 The operation of ShopTurn

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

The operator panel consists of 3 parts – the flat operator panel ①, the CNC full keyboard ② and the machine control panel (MCP) ③.

The most important keys on the CNC full keyboard for navigation in ShopTurn are listed in the following:

Key	Function
	<HELP> • Calls the context-sensitive online help for the selected window.
	<SELECT> (also called Toggle key) • Selects a listed value.
	Cursor keys • The cursor is moved using the four cursor keys. • Use the <Cursor right> key shown here to open a directory or program (e.g. a cycle) in the editor in edit mode.
	<PAGE UP> • Scroll upwards in a menu screen.
	<PAGE DOWN> • Scroll downwards in a menu screen.
	<END> • Moves the cursor to the last text box in a menu screen or table.
	<DEL> • Edit mode: Deletes the first character to the right. • Navigation mode: Deletes all characters.
	<BACKSPACE> • Edit mode: Deletes a character selected to the left of the cursor. • Navigation mode: Deletes all of the selected characters to the left of the cursor.
	<INSERT> • Press the key to enter Edit mode. Press the key again to exit Edit mode and go to Navigation mode.
	<INPUT> • Complete input of a value in the text box. • Open a directory or program.

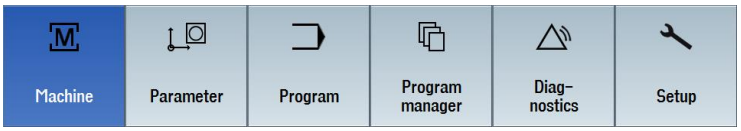
The actual function selection in ShopTurn is performed using the keys located around the screen. Most of them are assigned directly to the individual menu commands. Since the contents of the menus change depending on the situation, the term "softkeys" is used.

All main functions can be called using the horizontal softkeys.

All subfunctions in ShopTurn can be called using the vertical softkeys.

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

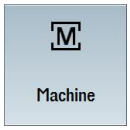
Main menu



4.2 The contents of the main menu

4.2.1 Machine

Machine – Manual



Select the "Machine" softkey.

Press the "JOG" key.

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

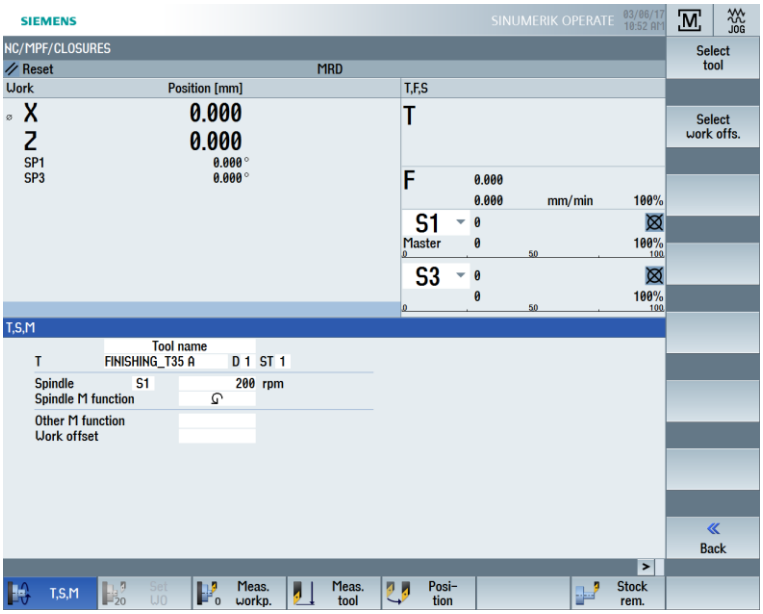


Figure 3-1 Call of a tool and input of technological values

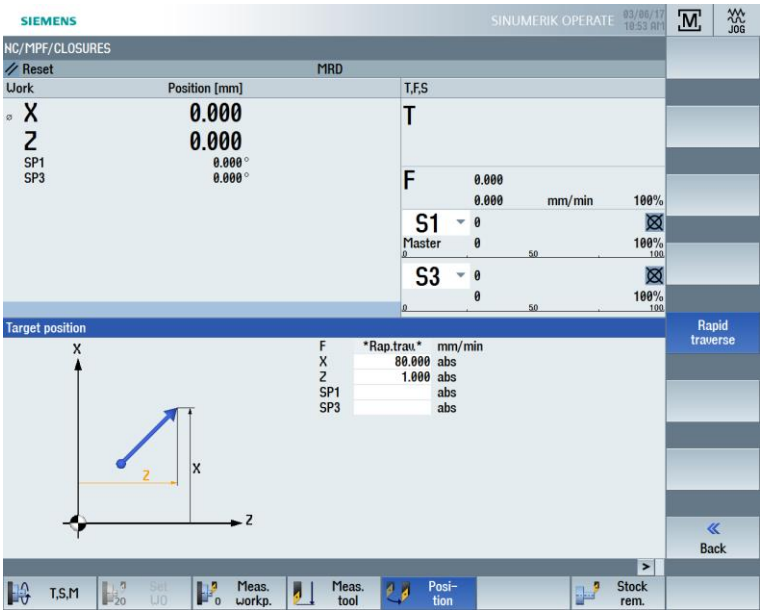
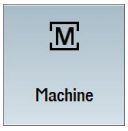


Figure 3-2 Specification of a target position

Machine – Auto



Select the "Machine" softkey.

Press the "AUTO" key.

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

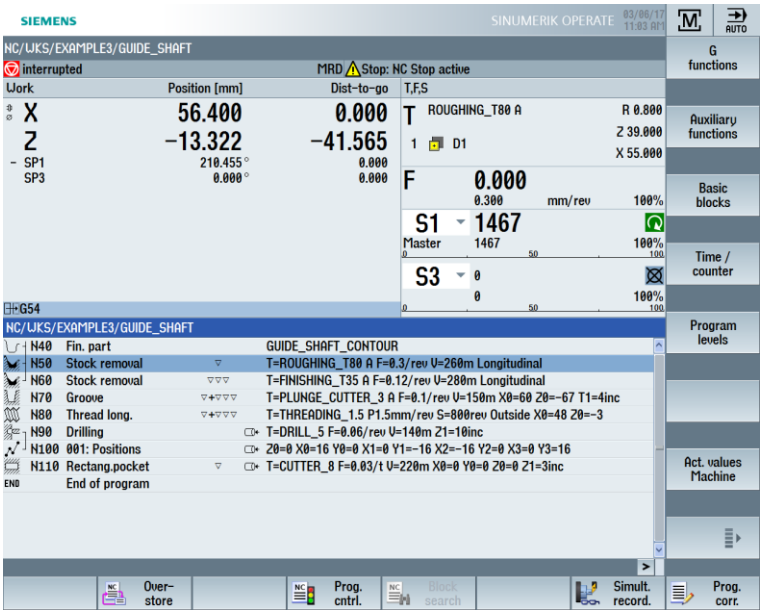


Figure 3-3 Executing the machining plan

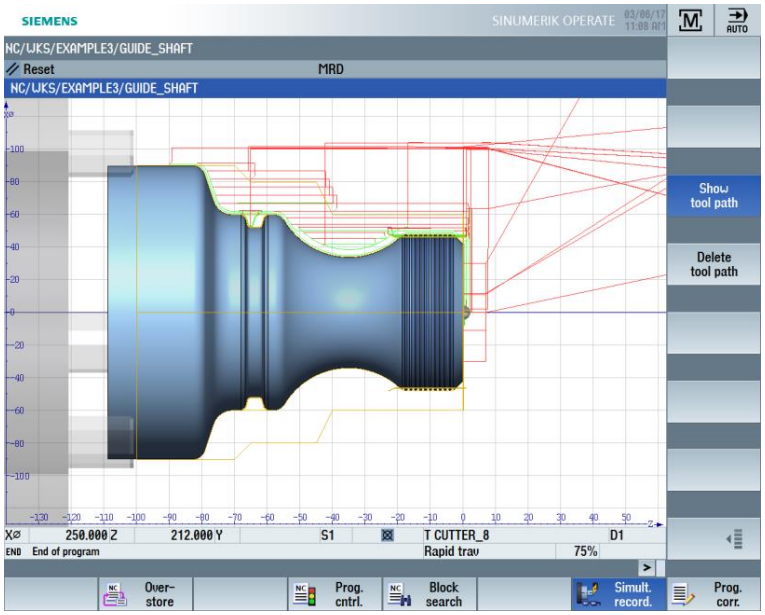
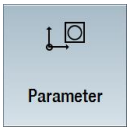


Figure 3-4 Simultaneous recording of the execution

4.2.2 Parameters

Parameter lists



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

Tool lists

Cutting is not possible without tools.
These can be managed in a tool list.

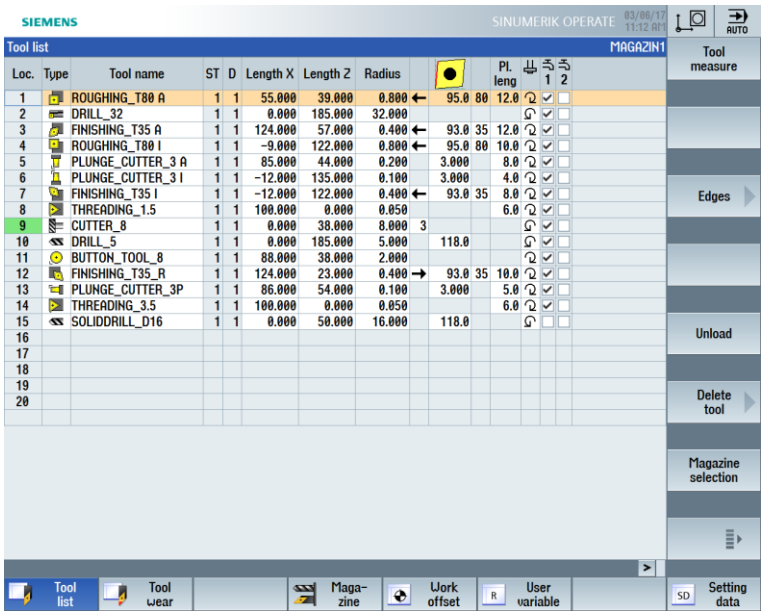


Figure 3-3 Tool list

Magazine

Tools can be organized in a magazine.

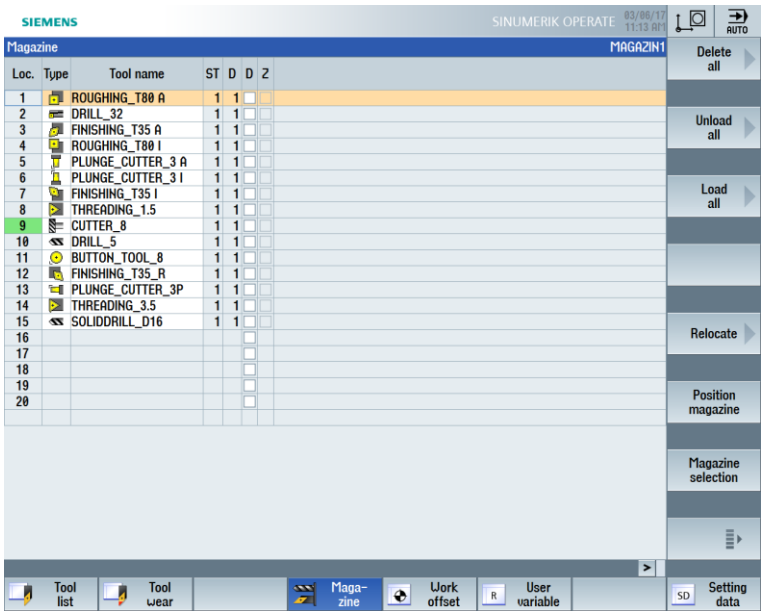


Figure 3-6 Magazine

Work offsets

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

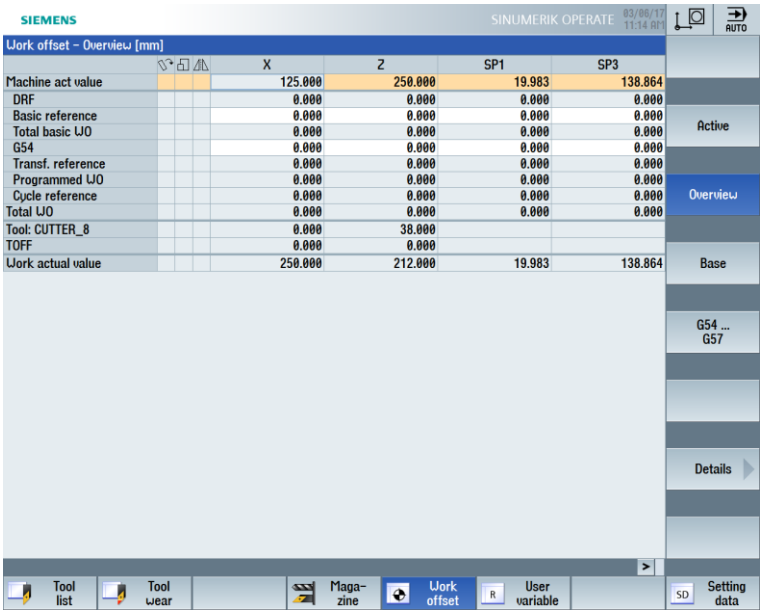


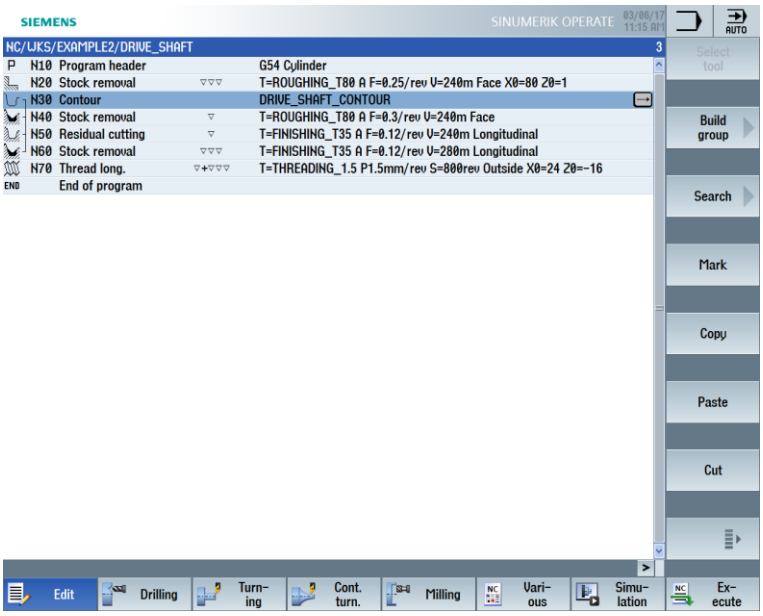
Figure 3-7 Work offsets

4.2.3 Program

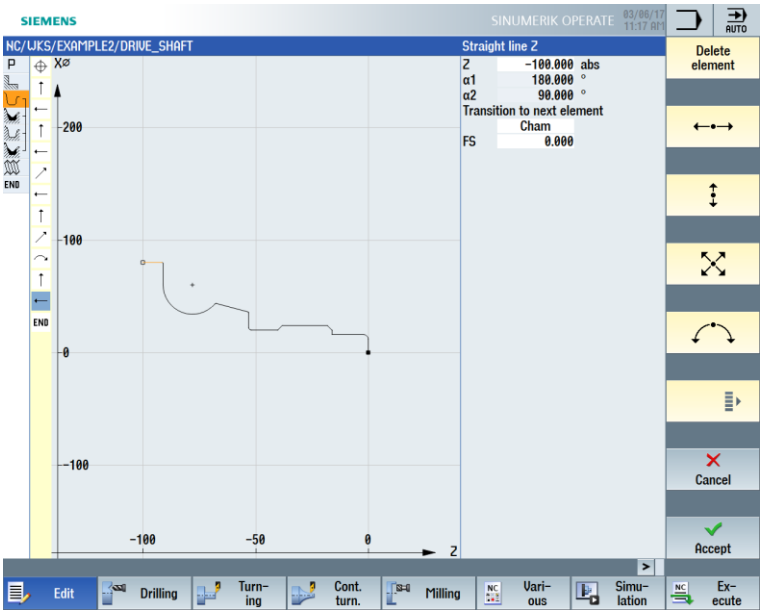
Editing programs

This key can be used to edit programs.

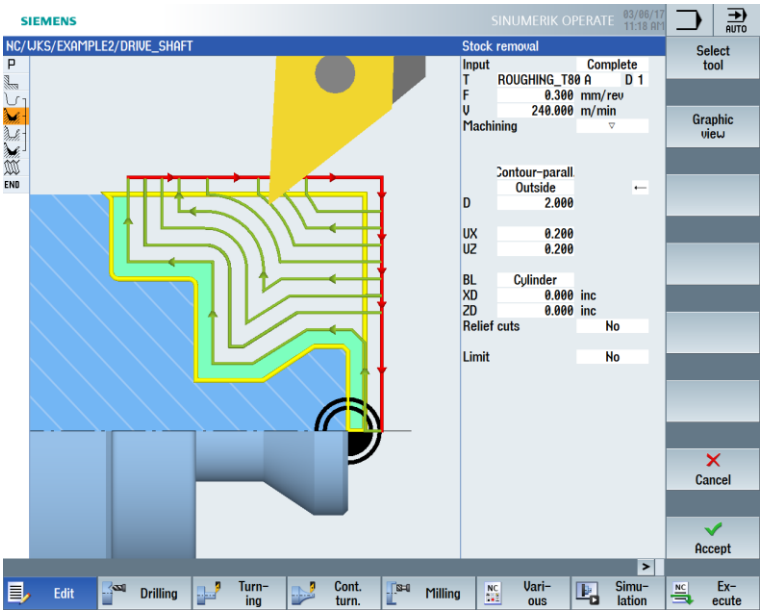
If you have created a ShopTurn program in the Program Manager, you can now create the machining plan with the complete machining sequence for the appropriate workpiece. Your practical knowledge is required to create the optimal sequence.



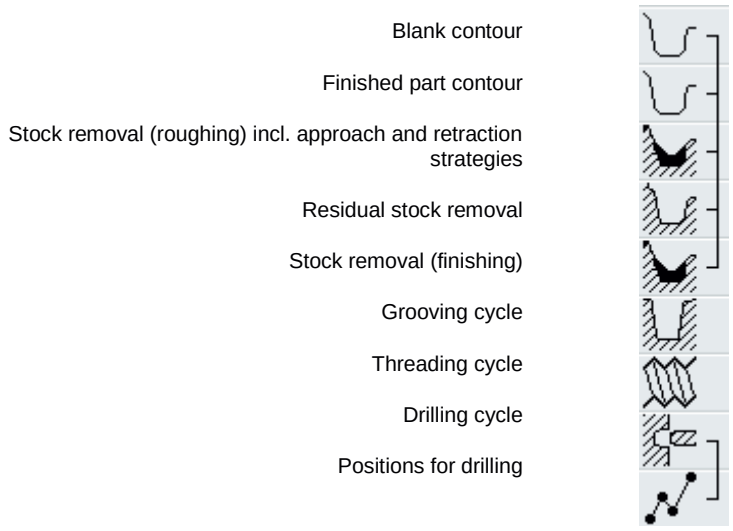
The contour to be machined is entered graphically as a machining step.



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



Example of the dovetailing of geometry and technology:



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

Simulating programs

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

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

The following views are available, among others, for simulation:

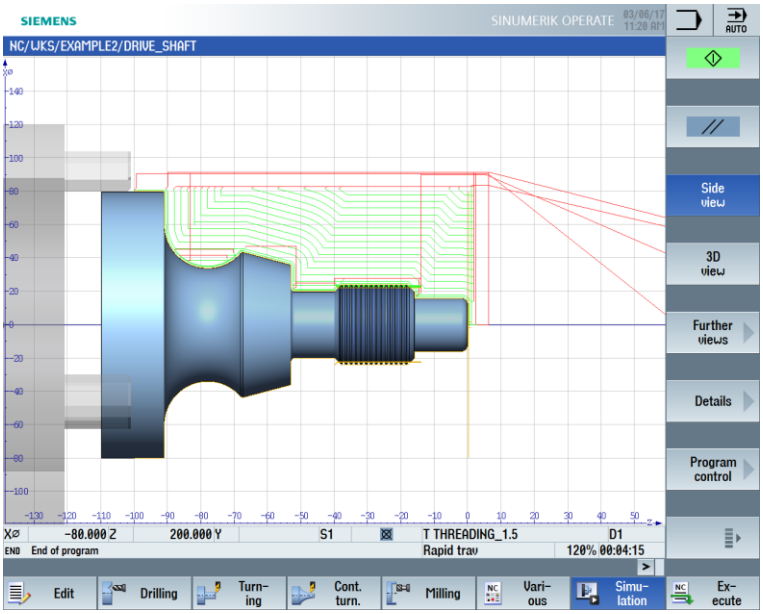


Figure 3-8 Side view (display tool path, activated)

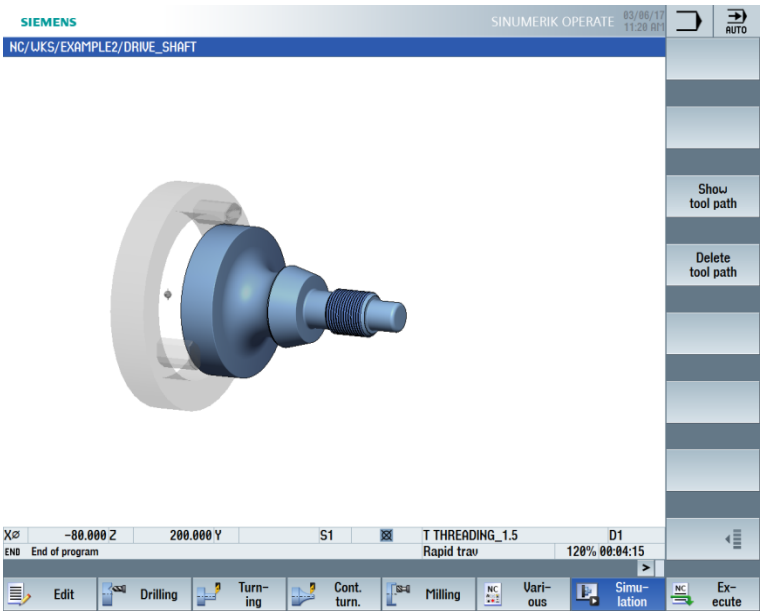


Figure 3-9 3D view (OPTION)

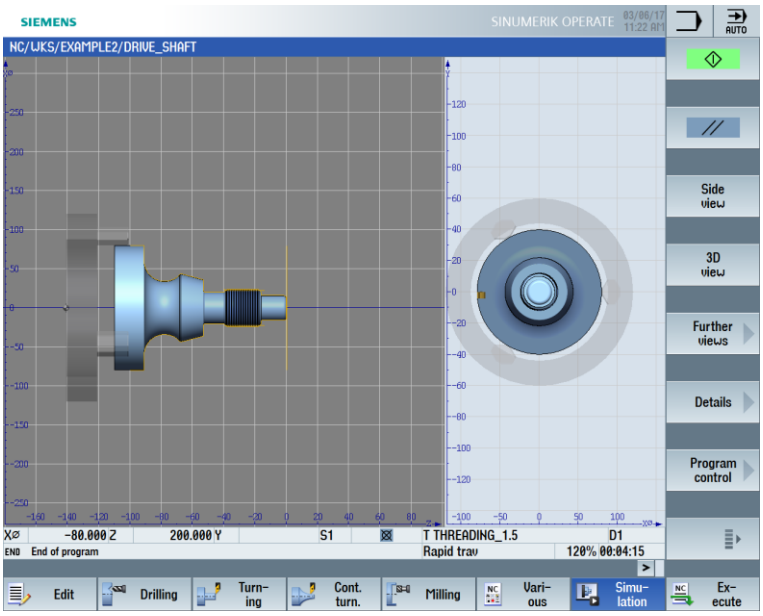
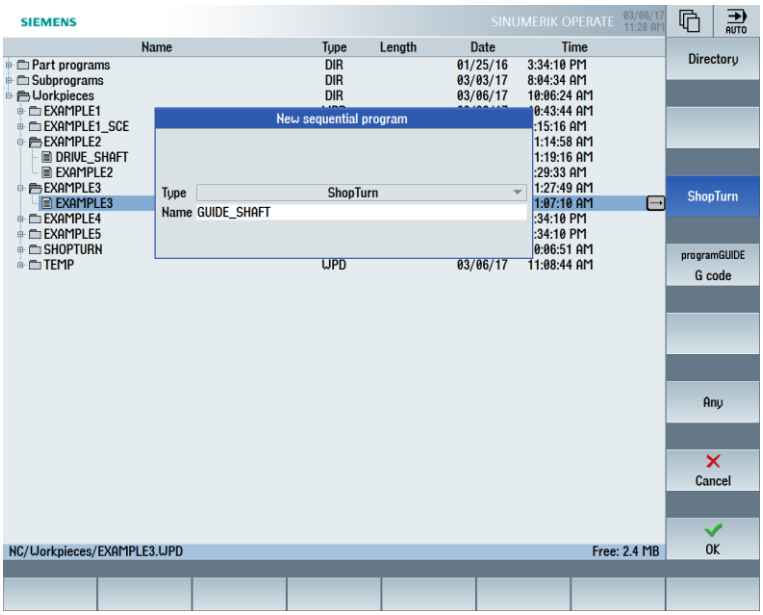


Figure 3-10 2-window view

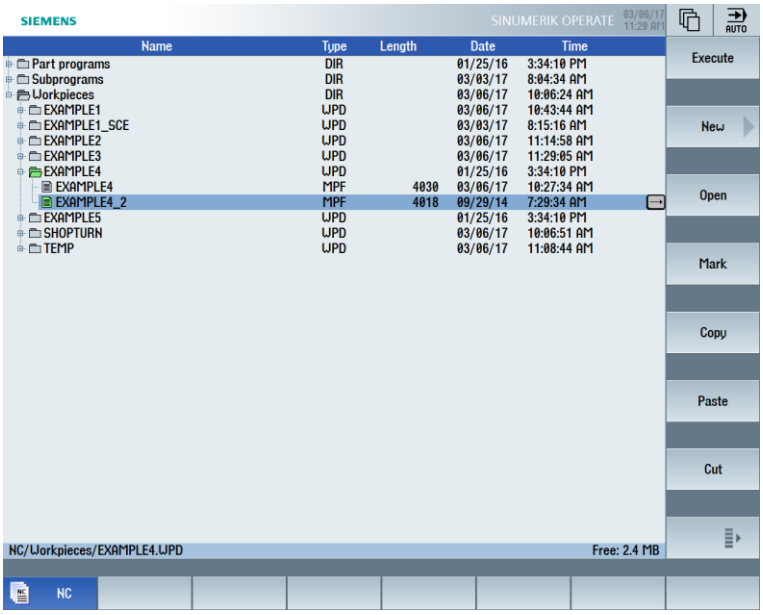
4.2.4 Program Manager

Managing programs

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



Active programs are marked with a green symbol.



USB

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

Creating a new workpiece

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

Creating a new program

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

ShopTurn

"ShopTurn" program

programGUIDE G-Code

"G-Code program"

4.2.5 Diagnostics

Alarms and messages

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

Diag-nostics

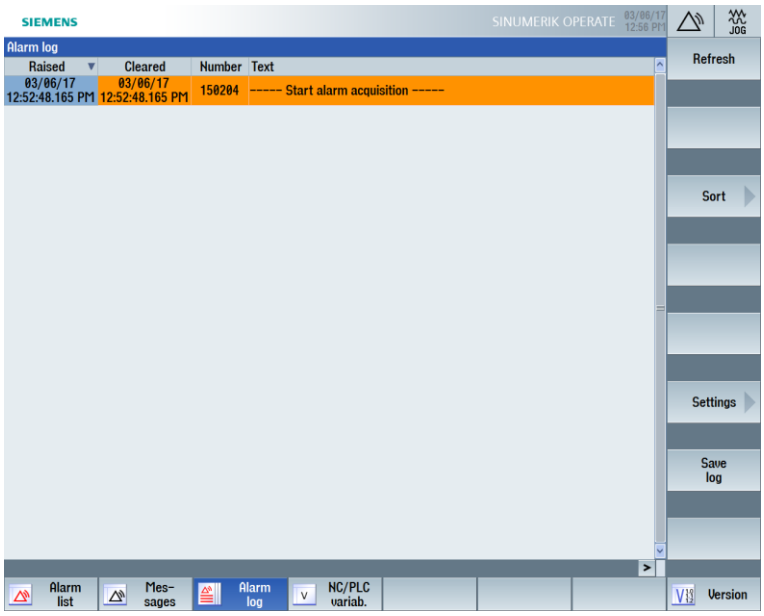


Figure 3-11 Alarm log

5. Basics for beginners

This section explains the general basics of the geometry and technology for turning. You do not have to enter anything in ShopTurn in this section.

5.1 Basics of geometry

5.1.1 Tool axes and machining planes

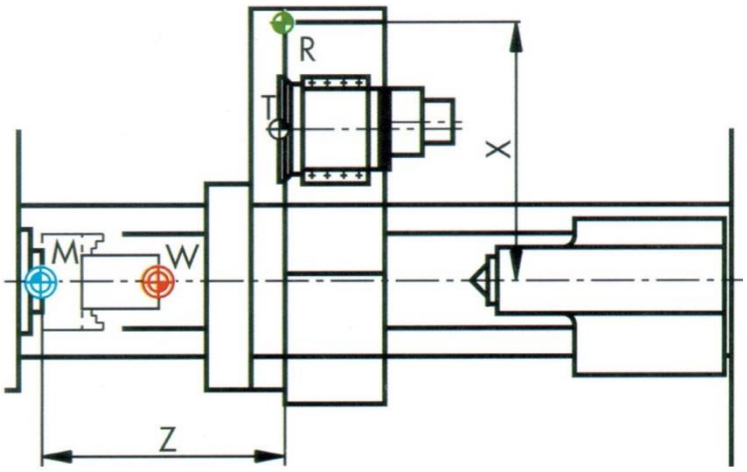
During turning, it is the workpiece and not the tool that rotates. The axis is the Z axis.

- Plane G18 = machining with turning tools
- Plane G17 = drilling and milling operations on the front face
- Plane G19 = drilling and milling operations on the peripheral surface

Since the diameters of the turned workpieces are relatively easy to check, the dimensions of the transverse axis are specified based on the diameter. This means that you can directly compare the actual dimensions with the dimensions in the drawing.

5.1.2 Points in the work area

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



Machine zero (M):

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

Workpiece zero (W):

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

Reference point (R):

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

Tool carrier reference point (T):

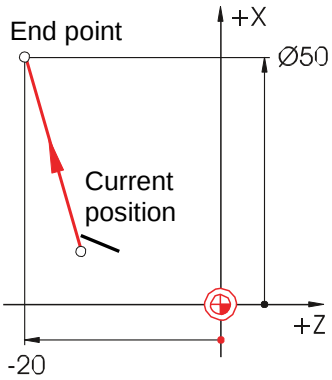
The tool carrier reference point (T) is relevant for setting up in the case of machines with tool turrets and preset tools. Its position and locating hole enable setting up with tool holders for shank tools in accordance with DIN 69880 and VDI 3425.

5.1.3 Absolute and incremental dimensioning

Absolute input

The entered values are relative to the workpiece zero.

* G90 Absolute dimensions



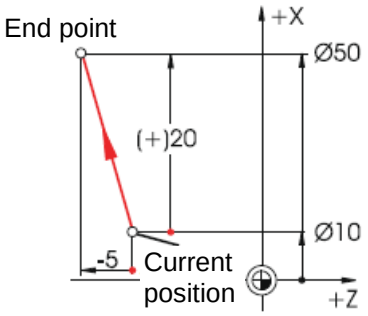
With absolute specifications, the absolute coordinate values of the end point must always be entered (the current position is not considered).

Incremental input

The entered values are relative to the current position.

Straight		
X	20.000	inc
Y		abs
Z	-5.000	inc

* G91 Incremental dimensions

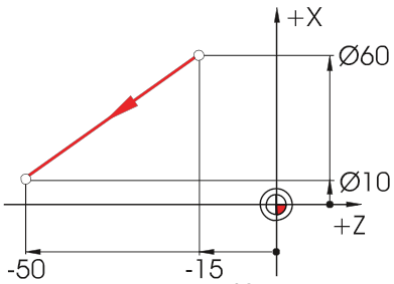


With incremental specifications, the difference values between the current position and the end point must always be entered while taking the direction into account.

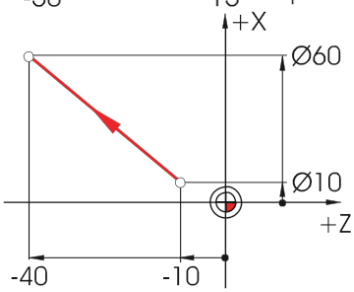
Switching between absolute and incremental input is possible at any time using the SELECT key.

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

Straight		
X	10.000	abs
Y		abs
Z	-35.000	inc



Straight		
X	25.000	inc
Y		abs
Z	-40.000	abs

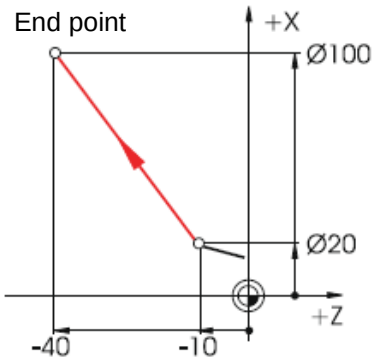


5.1.4 Cartesian and polar dimensions

Cartesian input

The coordinates X and Z are entered. The gray values in the example were calculated automatically.

Straight ZX		
X	100.000	abs
X	40.000	inc
Z	-40.000	abs
Z	-30.000	inc
L	50.000	
α1	126.870	°
α2	320.906	°



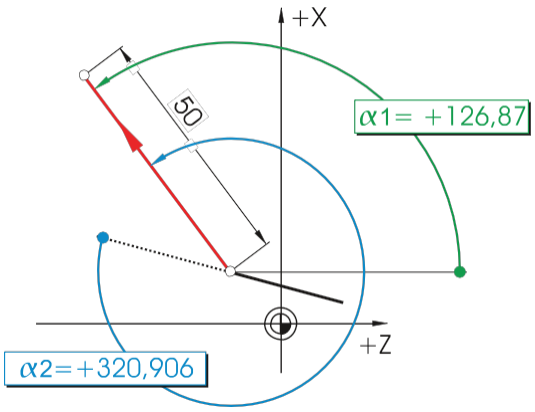
With absolute specifications, the absolute coordinate values of the end point must always be entered (the current position is not considered).

Polar input

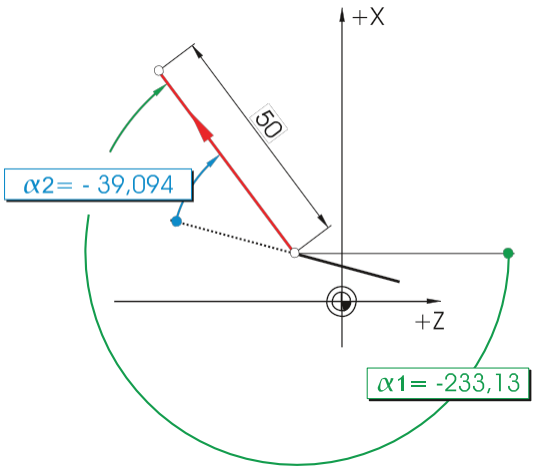
Specification of the length and angle. The gray values in the example were calculated automatically.

Straight ZX		
X	100.000	abs
X	40.000	inc
Z	-40.000	abs
Z	-30.000	inc
L	50.000	
α1	126.870	°
α2	320.906	°

The angles can be entered as positive angles…



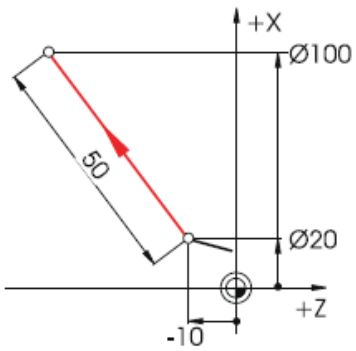
…and/or as negative angles.



You can also combine Cartesian and polar inputs. Two examples:

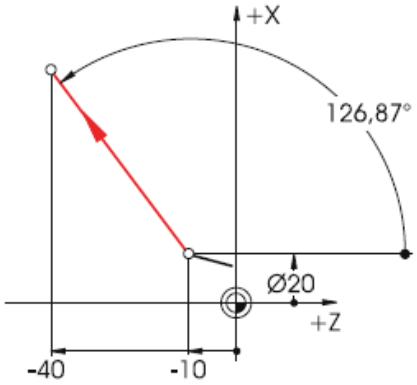
Specification of the end point in X and the length

Straight ZX		
X	100.000	abs
X	40.000	inc
Z	-40.000	abs
Z	-30.000	inc
L	50.000	
α1	126.870	°
α2	320.906	°



Specification of the end point in Z and an angle

Straight ZX		
X	100.000	abs
X	40.000	inc
Z	-40.000	abs
Z	-30.000	inc
L	50.000	
α1	126.870	°
α2	320.906	°



5.1.5 Circular motions

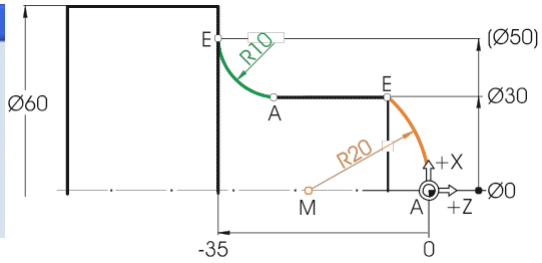
According to DIN, for circular arcs the end point of the arc (X and Z coordinates in the G18 plane) is specified along with the center point (I and K in the G18 plane).

The ShopTurn contour calculator gives you the freedom to use any dimension from the drawing for circular arcs, without having to carry out conversions.

The following example shows two – initially only partially defined – circular arcs.

Specification of the R10 arc:

Circle		
Direction of rotation		
R	10.000	
X	50.000	abs
Z	-35	abs
I		abs
K		abs
α1		°



Specification of the R20 arc:

Circle		
Direction of rotation		
R		
X	30.000	abs
Z		abs
I	0.000	abs
K	-20	abs
α1	-90.000	°

After the input:

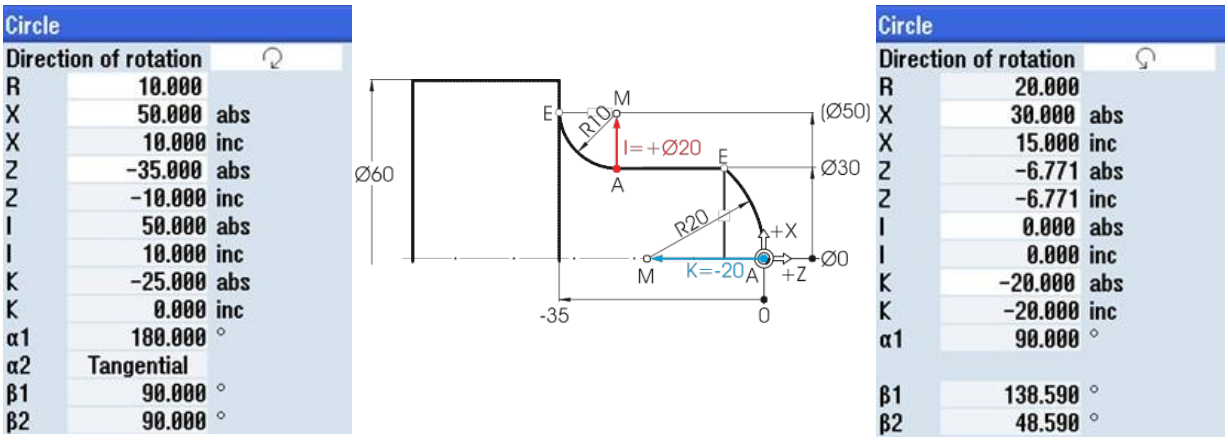
Circle		
Direction of rotation		
R	10.000	
X	50.000	abs
Z	-35.000	abs
I	50.000	abs
K	-25.000	abs
α1	180.000	°
α2	Tangential	

After the input:

Circle		
Direction of rotation		
R	20.000	
X	30.000	abs
X	15.000	inc
Z	-6.771	abs
Z	-6.771	inc
I	0.000	abs
I	0.000	inc
K	-20.000	abs
K	-20.000	inc
α1	90.000	°
β1	138.590	°
β2	48.590	°

All para-meters

The following displays of all values appear when you have entered all known dimensions and pressed the "All parameters" softkey in the input window of the respective arc.



In DIN format:
G2 X50 Z-35 CR=10

In DIN format:
G3 X30 Z-6.771 K-20

5.2 Basics of technology

5.2.1 Cutting rate and speeds

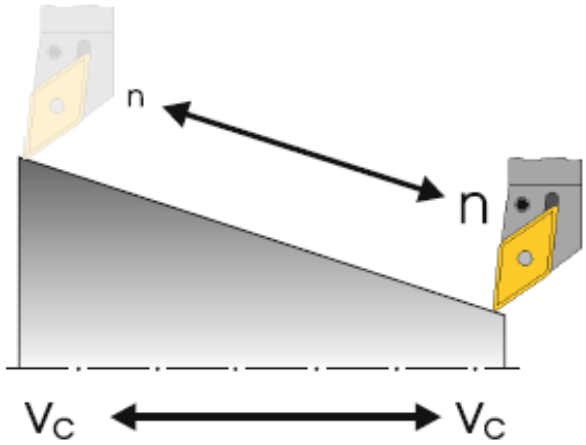
Most of the time the cutting rate is directly programmed for turning, in particular for roughing, finishing and grooving. The speed is programmed only for drilling and (most of the time) for thread cutting.

Determination of the cutting rate

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

Material of the tool:	Hard metal
Material of the workpiece:	Machining steel
Value:	Vc = 180 m/min

Constant cutting rate v_c (G96) for roughing, finishing and grooving:



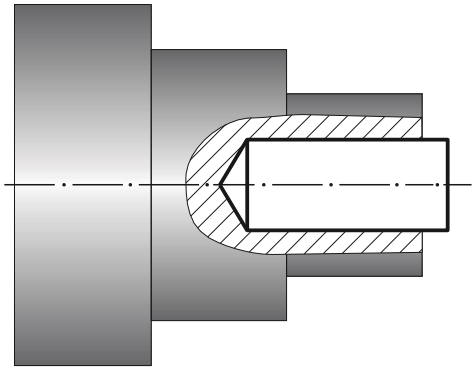
To ensure that the selected cutting rate is the same at every workpiece diameter, the speed is adjusted by the controller with command G96 = Constant cutting rate. This is carried out by DC motors or frequency-controlled three-phase motors. As the diameter decreases, the speed increases theoretically to infinity. To avoid danger due to excessive centrifugal forces, a speed limitation of n = 3000 rpm for example, must be programmed.
In DIN format, the block would have the following form:
G96 S180
LIMS = 3000 (LIMS = limit).

Constant speed n (G97) for drilling and thread cutting:

Since a constant speed is used for drilling, the command G97 = Constant speed must be used here.

The speed is dependent on the desired cutting rate (120 m/min selected here) and the tool diameter.

Therefore, the inputs are: G97 S1900.



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

d = 20mm (tool diameter)

$$n = \frac{120mm \cdot 1000}{20mm \cdot \pi \cdot min}$$

$$n \approx 1900 \frac{1}{min}$$

5.2.2 Feed

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

Determination of the feed

Like the cutting rate, the value for the feed is also obtained from the handbook, the documents of the tool manufacturer or practical knowledge.

Cutting material of the tool:	Hard metal
Material of the workpiece:	Machining steel
Determined value (handbook):	f = 0.2 – 0.4 mm
The mean value is selected:	f = 0.3 mm

Relationship between feed and feedrate:

The constant feed f and the respective speed n yield the feedrate:

$V_f = f * n$

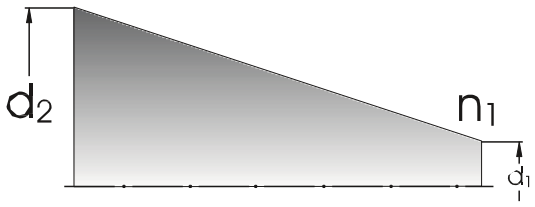
$V_c = 180$ m/min

$d_2 = 80$ mm

$n_2 \approx 710$ rpm

$V_{f2} \approx 710$ rpm * 0.3 mm

$V_{f2} \approx 210$ mm/min



$V_c = 180$ m/min

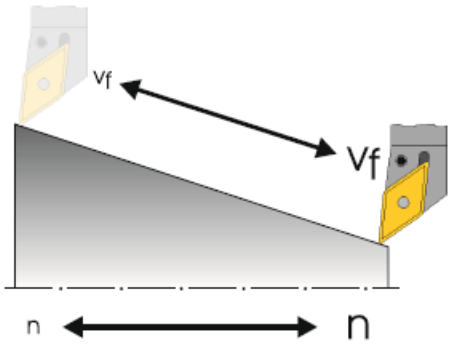
$d_2 = 20$ mm

$n_2 \approx 2800$ rpm

$V_{f2} \approx 2800$ rpm * 0.3 mm

$V_{f2} \approx 840$ mm/min

Since the speed is different, the feedrate also differs for the various diameters, despite having the same feed.



6. Effective setup

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

6.1 Tool management

ShopTurn offers three lists for tool management:

- Tool list
- Tool wear list
- Magazine list

6.1.1 Tool list

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

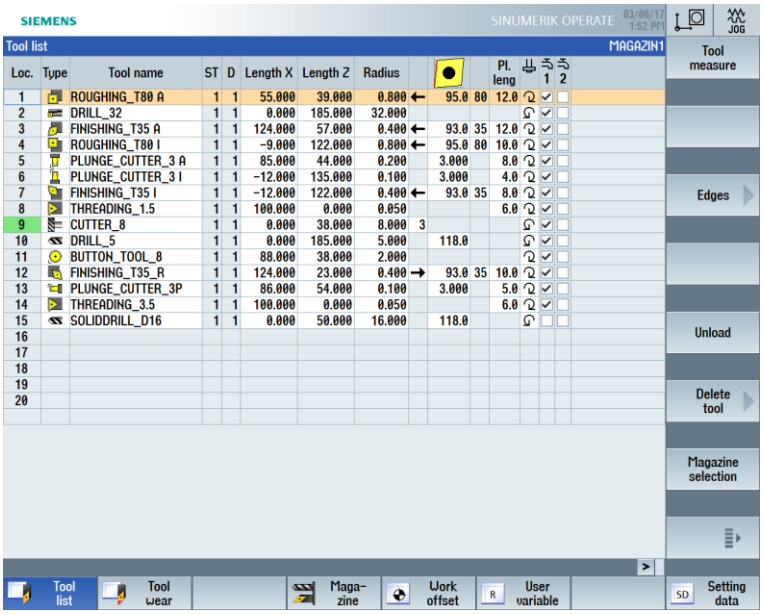


Figure 5-1 Example of tool list

Meanings of the most important parameters:

Loc.	Location number
Type	Tool type
Tool name	The tool is identified by the name and the sister tool number. You may enter the name as a text or number.
ST	Sister tool number (for replacement tool strategy)
D	Cutting edge number
Length X	Geometry data, length X
Length Z	Geometry data, length Z
Diameter	Tool diameter
Mounting bracket, tip angle, insert width	Mounting bracket (roughing tool and finishing tool), tip angle (drill) and insert width (grooving tool)
	Direction of spindle rotation
	Coolants 1 and 2 (e.g. internal and external cooling)

ShopTurn provides various tool types (favorites, milling cutters, drills, turning tools and special tools). Tools can be created in the tool list by means of a predefined tool catalog. There are various mounting positions and geometrical parameters (e.g. mounting bracket), depending on the tool type.

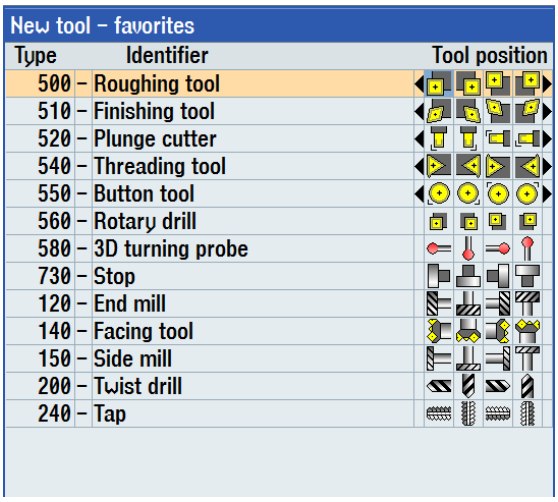


Figure 5-2 Example of Favorites list

6.1.2 Tool wear list

The wear data for the respective tools is defined here.

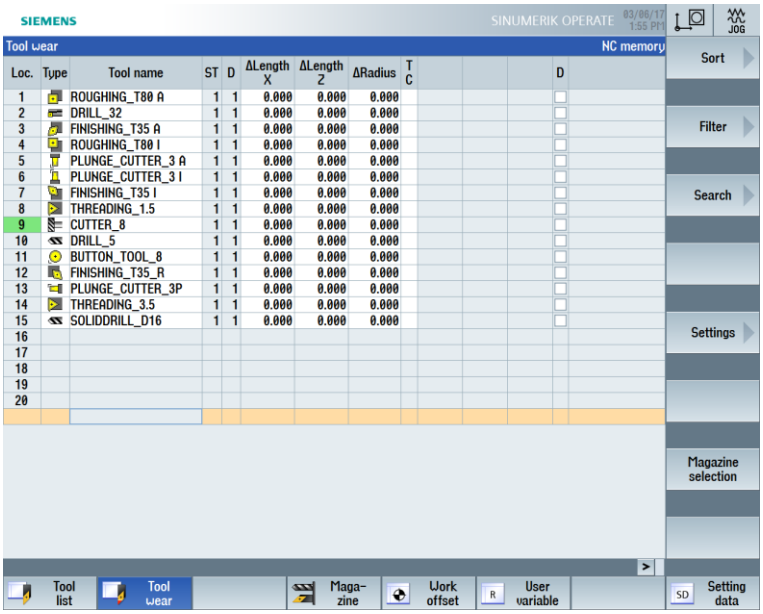


Figure 5-3 Tool wear list

The most important tool wear parameters are:

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

6.1.3 Magazine list

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

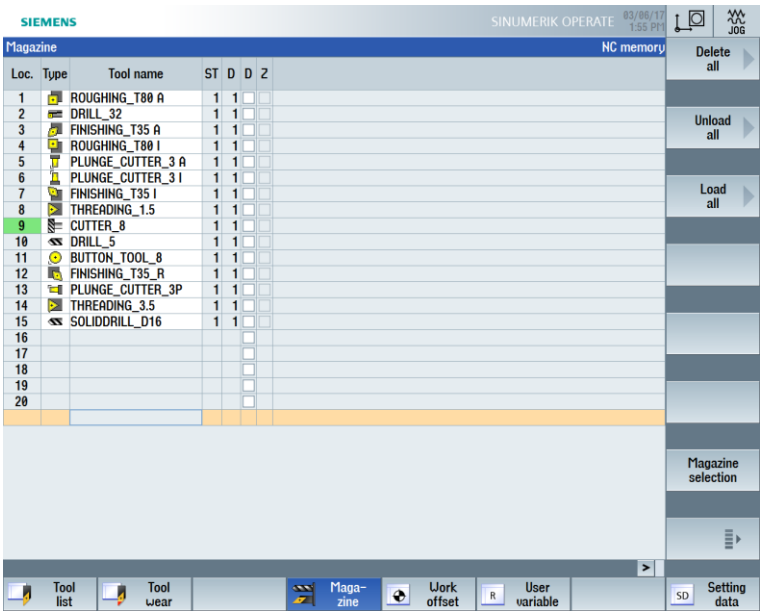
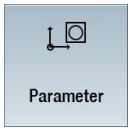


Figure 5-4 Magazine list

Meanings of the most important parameters:

D	Locking of the magazine location
Z	Marking of a tool as oversized. The tool occupies two half locations left, two half locations right, one half location top and one half location bottom in a magazine.
L	Fixed location coding The tool is permanently assigned to this magazine location.

6.2 Tools used



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

Select the "Parameter" area in the main menu.

Select the "Tool list" softkey.

To create a new tool, go to a free location in the tool list ("New tool" softkey must be displayed).

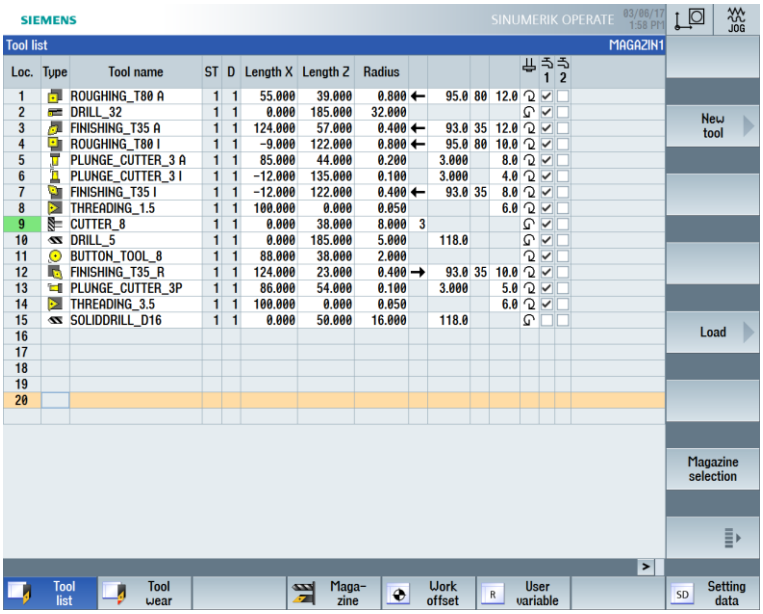


Figure 5-5 Tool list – Free location

Select the "New tool" softkey.

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

Note:

The milling cutter with diameter 8 (CUTTER_8) must be insertable, as it will be used for milling a pocket.

6.3 Tools in the magazine

In the following, you will learn how to load the tools into the magazine.
In the tool list, select a tool without a location number.

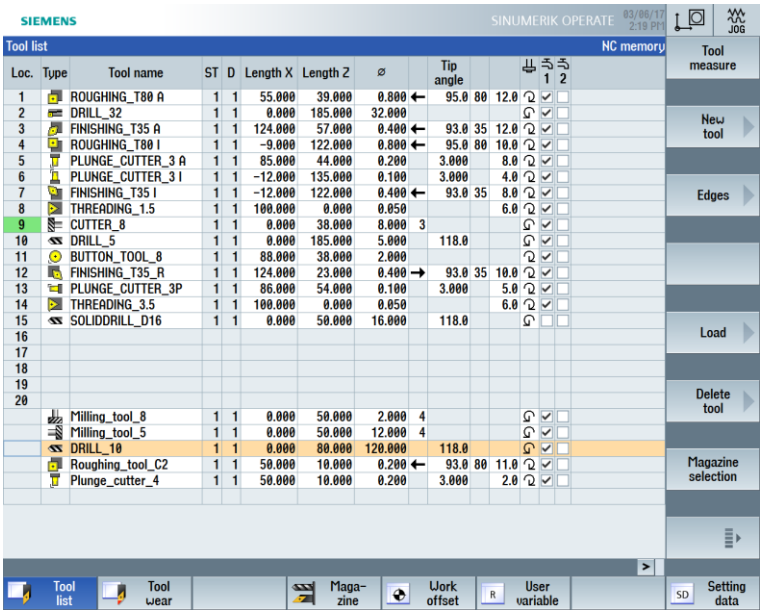


Figure 5-6 Selecting the tool in the magazine

Load

Press the "Load" key. The following dialog offers the first free magazine location, which you can change or accept as is.

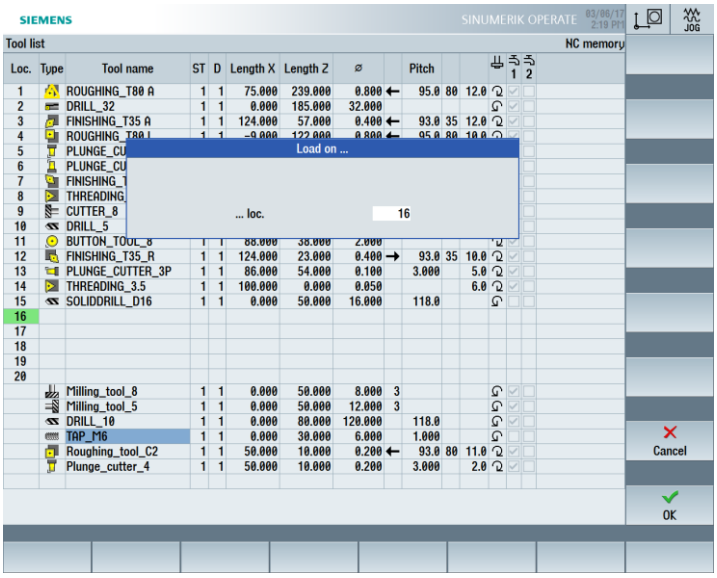


Figure 5-7 Entering and/or accepting a magazine location

Once applied, the tool list may look like this.

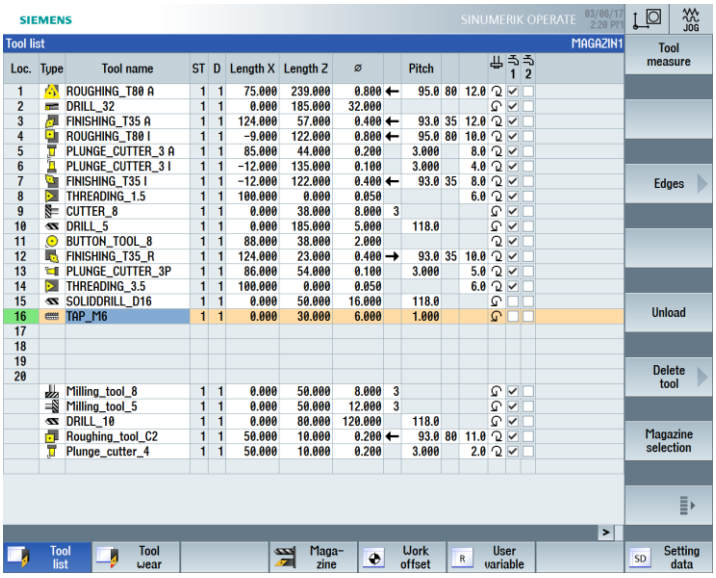


Figure 5-8 Tool list once applied

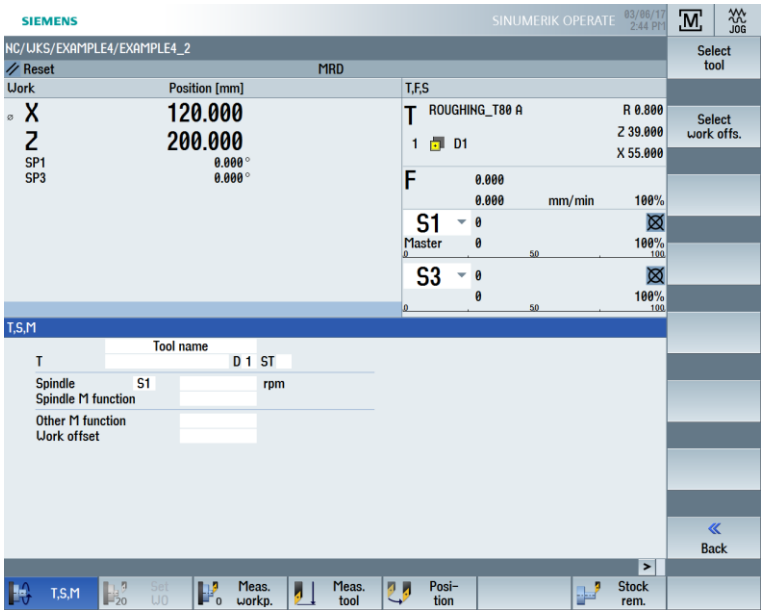
6.4 Measuring tools

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

Procedure

T,S,M

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



Meas. tool

Then go to the "Meas. tool" menu.

Manually

Enter the probed or turned diameter.

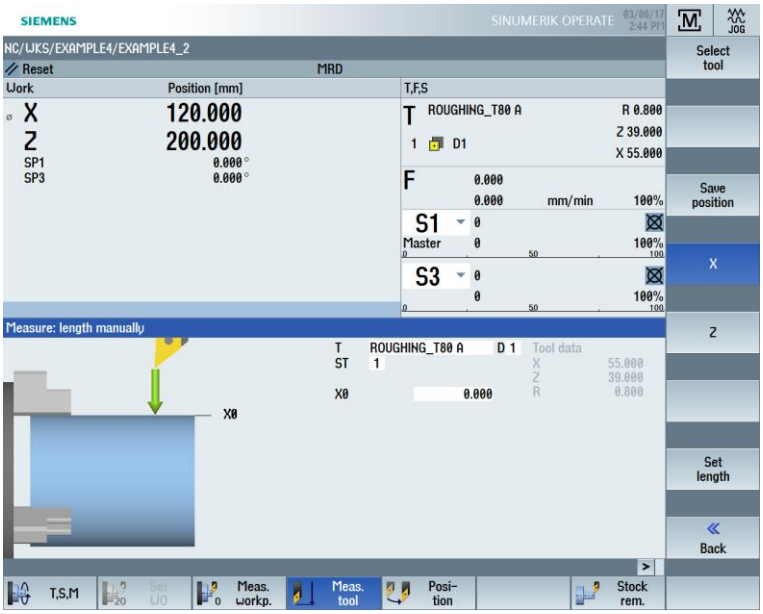


Figure 5-9 Tool measurement – Specification of the X value

Set length

The current position of the tool is offset taking into account the workpiece diameter.

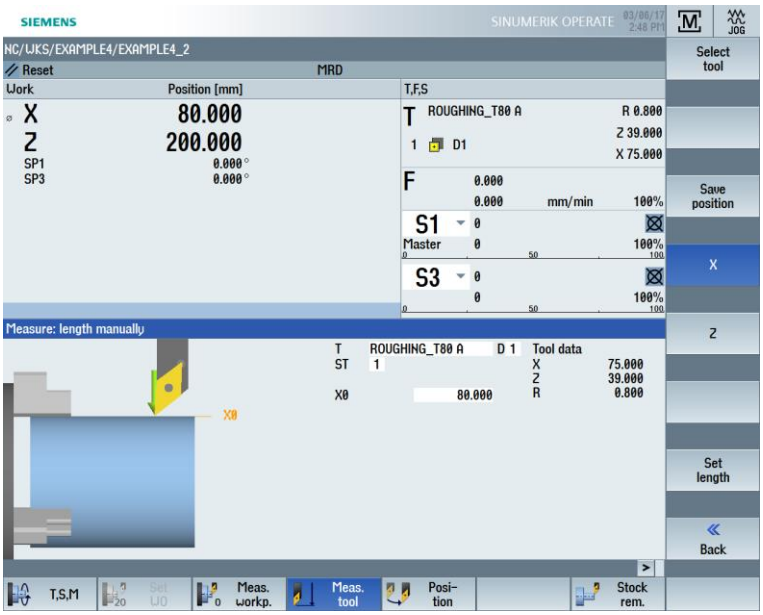


Figure 5-10 Tool measurement – Setting length X

You must now repeat this operation for Z.

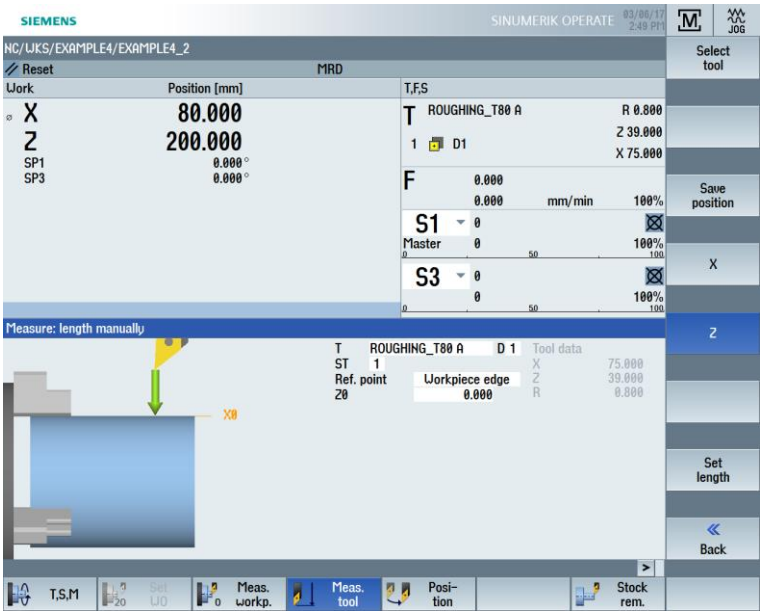


Figure 5-11 Tool measurement – Setting length Z

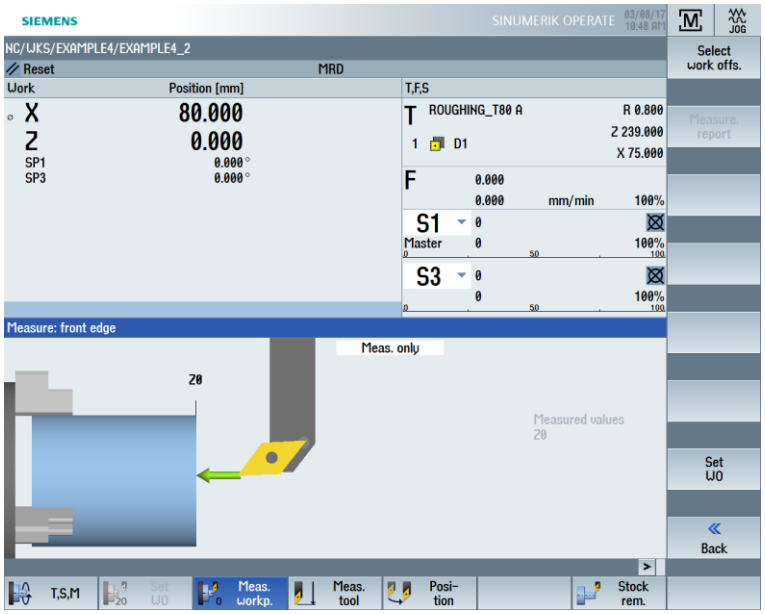
6.5 Setting the workpiece zero

In the following, you will learn how to set the workpiece zero.

Procedure

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

Shift the workpiece zero if this does not lie on the end face of the workpiece.



Accept your input.

7. Example 1: Taper shaft

7.1 Overview

Learning objectives

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

- Manage and create programs
- Call tools
- Enter traversing paths
- Create any contours with the contour calculator
- Rough and finish contours
- Create a thread undercut
- Create a thread
- Create grooves

Task

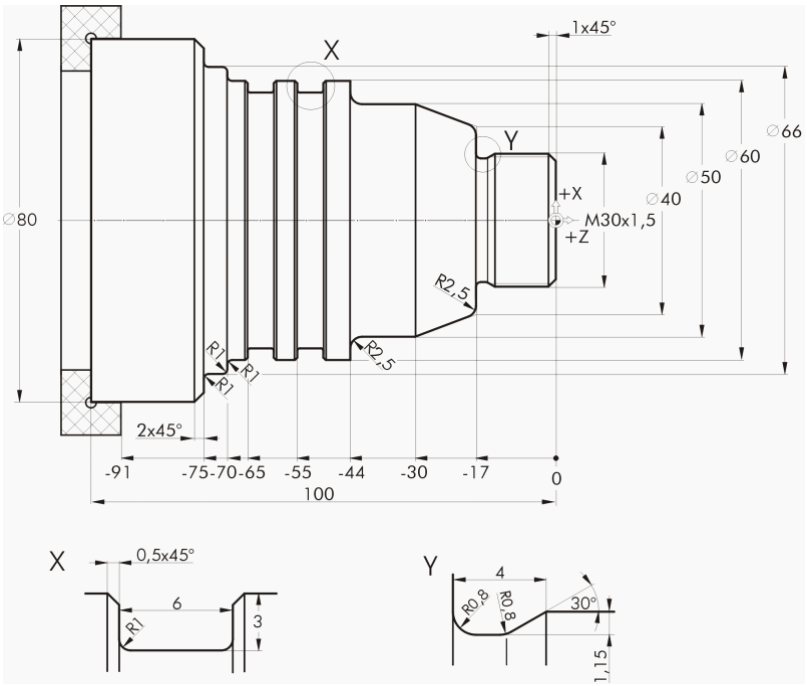


Figure 6-1 Workshop drawing – Example 1

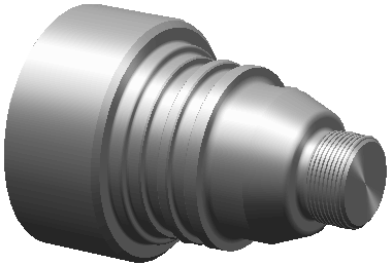


Figure 6-2 Workpiece – Example 1

Note:

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

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

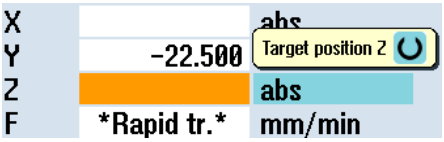


Figure 6-3 Example of toggle field

7.2 Program management and creating programs

Operating sequences

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

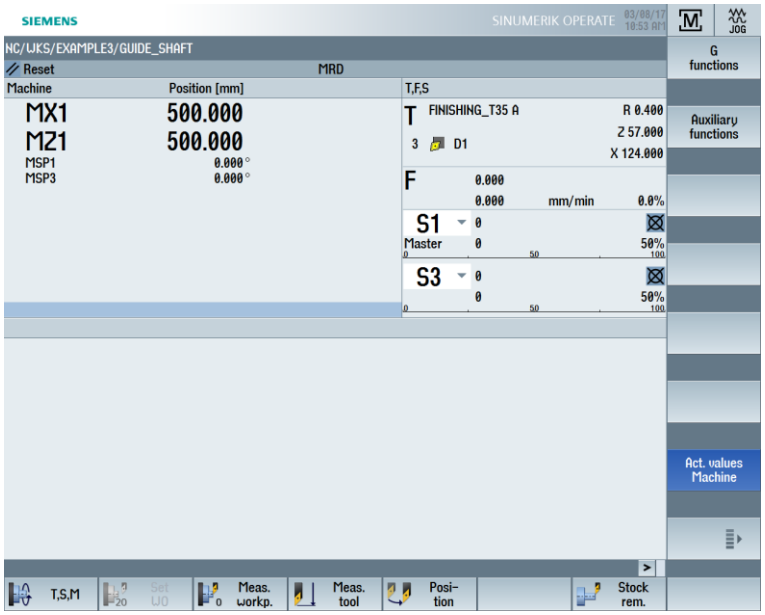


Figure 6-4 Main screen

Open the main menu using the MENU SELECT key. You can open the various areas of ShopTurn from the main menu.

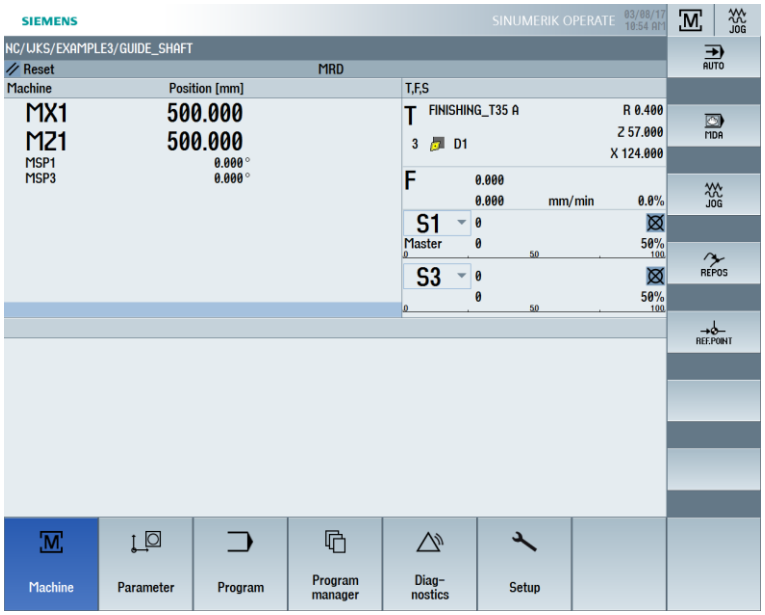


Figure 6-5 Main menu

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

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

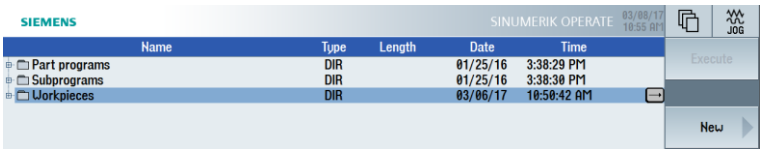


Figure 6-6 Program Manager

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

Open the "Workpieces" directory.

Enter the name 'EXAMPLE1' for the workpiece.

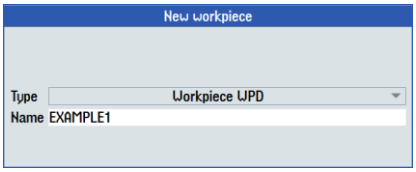


Figure 6-7 Creating the workpiece

Confirm your entry. The following dialog box opens.

Figure 6-8 Creating the sequential program

You can select the input format with the "ShopTurn" and "programGUIDE" G code softkeys.

You specify the program type via the "ShopTurn" softkey.

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

Accept your input.

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

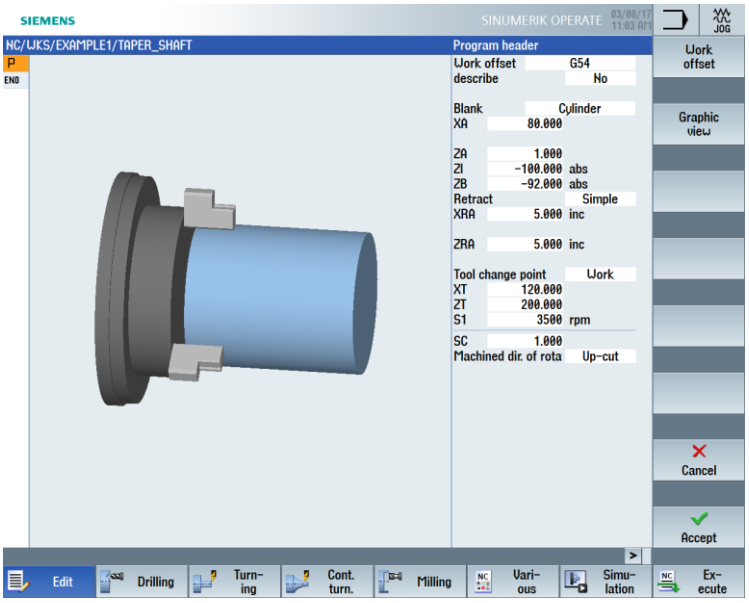


Figure 6-9 Program header – Help screen

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

Enter the following details:

Field	Value	Selection via toggle key	Notes
Unit of measurement	mm	X	
Work offset		X	
Blank	Cylinder	X	Select the blank shape (here, cylinder) using the toggle key.
XA	80		
ZA	1		
ZI	-100 abs	X	
ZB	-92 abs	X	The value ZB indicates the distance from the chuck.
Retract	Simple	X	See below Retraction
XRA	5 inc	X	The dimensions of the retraction planes (absolute or incremental) and the tool change point are entered here.
ZRA	5 inc	X	
Tool change point	Workpiece	X	
XT	120		
ZT	200		
Safety clearance SC	1		
Speed limits S1	3500		
Machined direction of rotation	Down-cut	X	

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

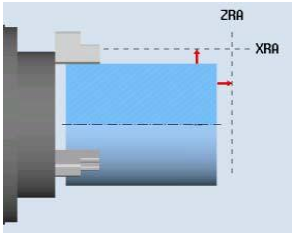
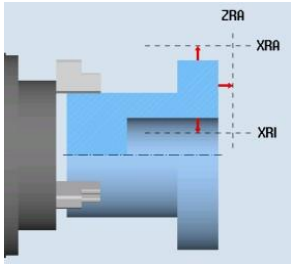
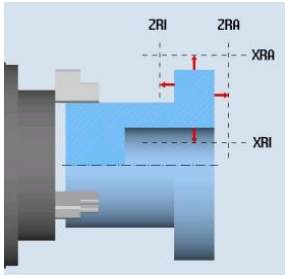
Figure 6-10 Program header, example 1 – Machining step editor

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

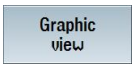
You can open the program header again at any time to make changes or check the values.

Retraction

It is possible to switch the retraction plane between "Simple", "Extended" and "All". Depending on the retraction setting, the associated fields are enabled for the input of the distances.

"Simple" (for simple cylinders)	"Extended" (for complex workpieces with internal machining)	"All" (for complex workpieces with internal machining and/or relief cuts)
		
Retract Simple XRA 5.000 inc ZRA 5.000 inc	Retract Extended XRA 5.000 inc XRI 5.000 inc ZRA 5.000 inc	Retract All XRA 5.000 inc XRI 5.000 inc ZRA 5.000 inc ZRI 0.000

Softkeys



Use this softkey to change to the online graphic of the workpiece (see figure below).

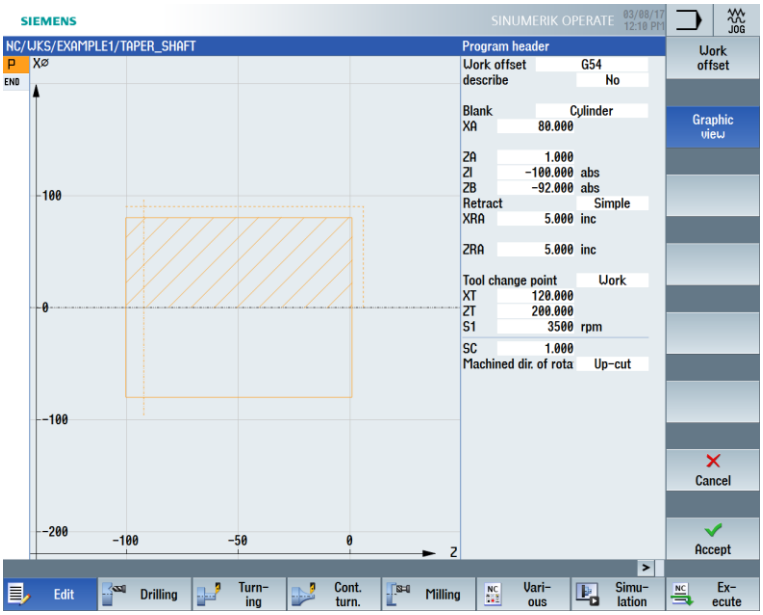


Figure 6-11 Program header – Graphic view

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

7.3 Opening a tool

Operating sequences

Follow the steps below to call the required tool:

Use this key to extend the horizontal softkey menu.

Select the "Straight Circle" softkey.

Select the "Tool" softkey.

Select tool

Open the tool list.

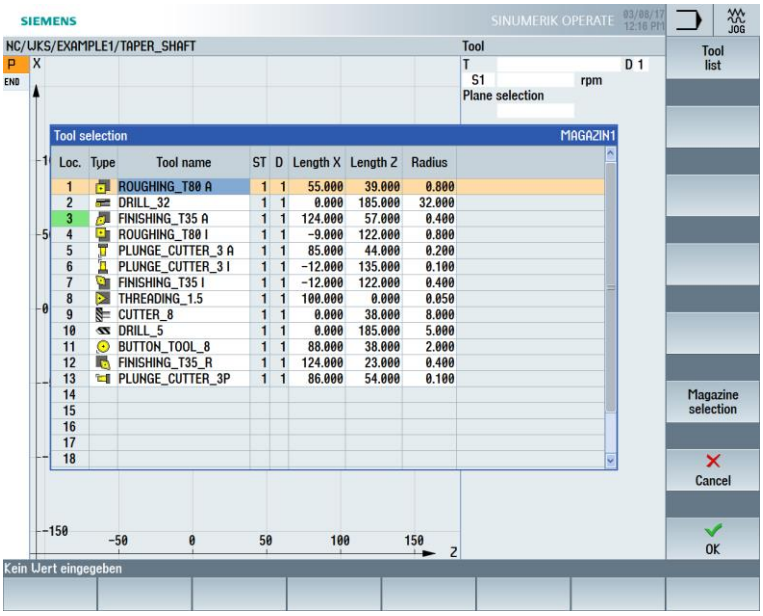


Figure 6-12 Tool list

Use the cursor key to select the "ROUGHING_T80 A" tool.

OK

Apply the tool to the program. After the tool has been applied, enter the following values in the screen form
(if necessary, change the unit using the toggle key):

Field	Value	Selection via toggle key	Notes
Spindle	V1	X	Select the main spindle V1.
Cutting rate	240 m/min	X	
Plane selection	Turning	X	

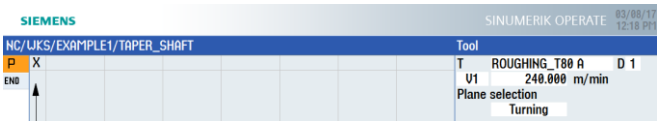


Figure 6-13 Tool – Input

Accept

Accept the entered value.

7.4 Entering the traversing path

Operating sequences

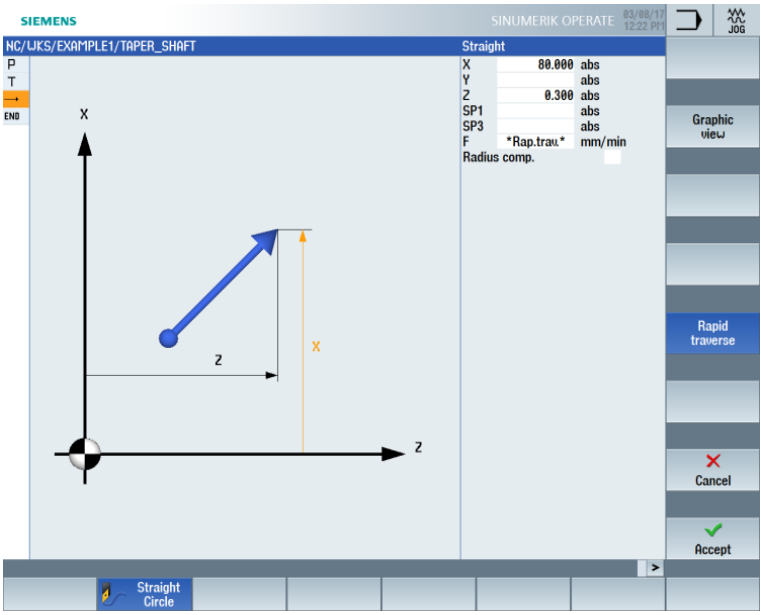
Now enter the traversing paths:

Select the "Straight" softkey.

Select the "Rapid traverse" softkey.

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

Field	Value	Selection via toggle key	Notes
X	82 abs	X	
Z	0.3 abs	X	



Enter the starting point for the traversing path

Accept the entered values.

Select the "Straight" softkey.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
X	-1.6 abs	X	The tool has a 0.8 mm radius and must therefore be traversed to the diameter X –1.6.
F	0.3 mm/rev	X	

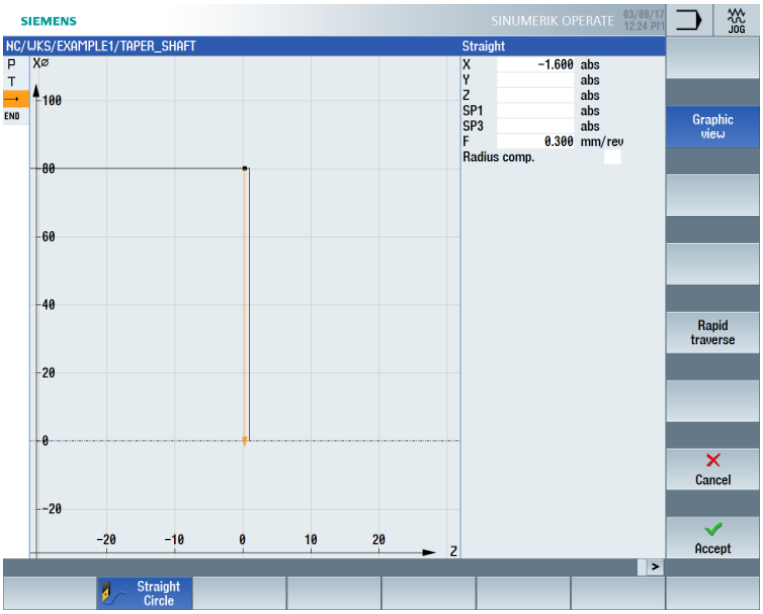
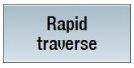


Figure 6-14 Entering the traversing path

Accept the entered values.

Select the "Straight" softkey.



Select the "Rapid traverse" softkey. Move the tool away from the end face in rapid traverse.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
Z	1 abs	X	

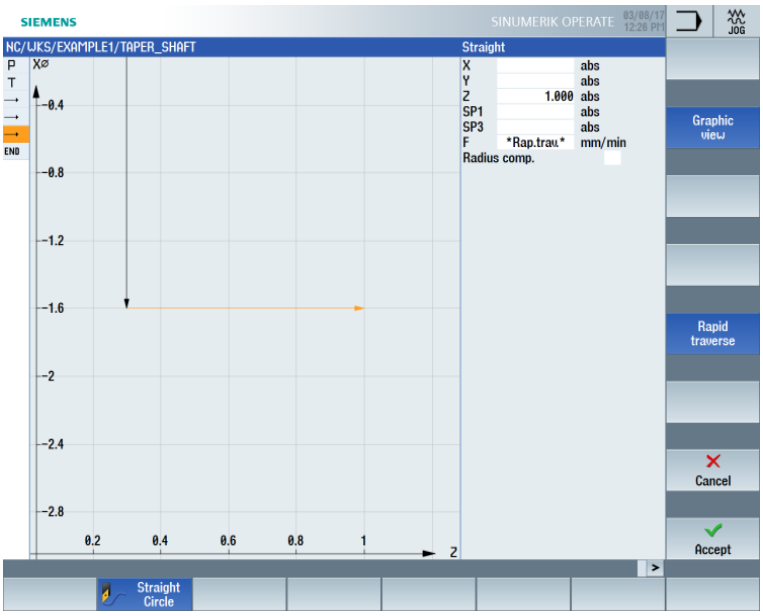
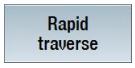


Figure 6-15 Enter the traversing path – Moving away from the end face

Accept the entered values.

Select the "Straight" softkey.



Select the "Rapid traverse" softkey.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
X	82 abs	X	This entry returns the tool to the starting point.

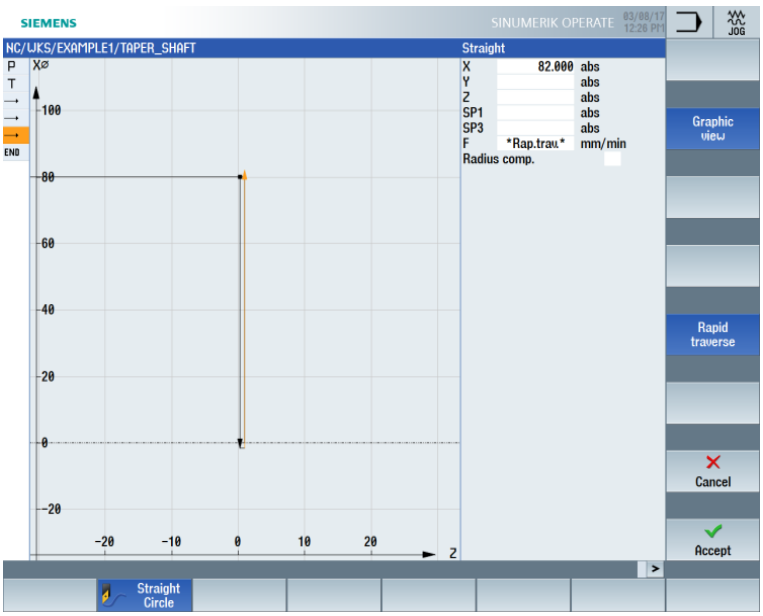


Figure 6-16 Entering the traversing path – Returning to the starting point

Accept the entered values.

Select the "Straight" softkey.

Create the other four traversing paths according to the following list of machining steps.

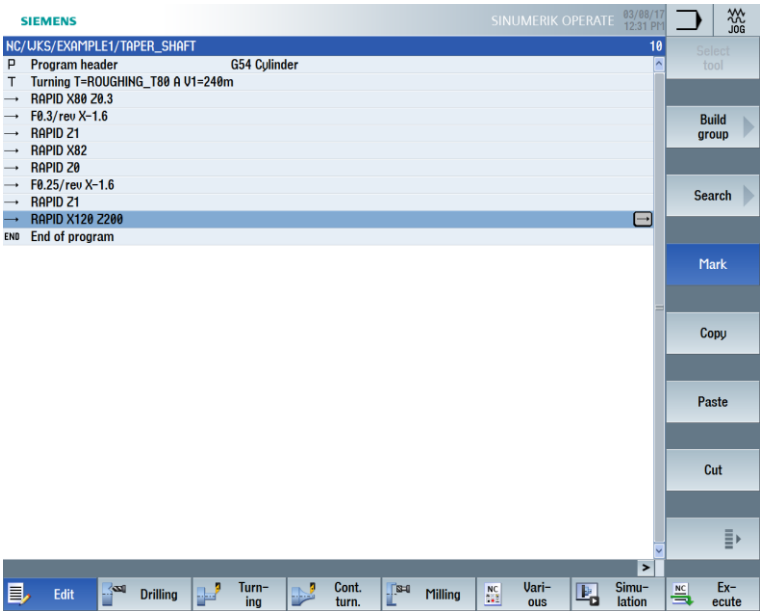


Figure 6-17 Entering the traversing path – Other four traversing paths

Start the simulation.

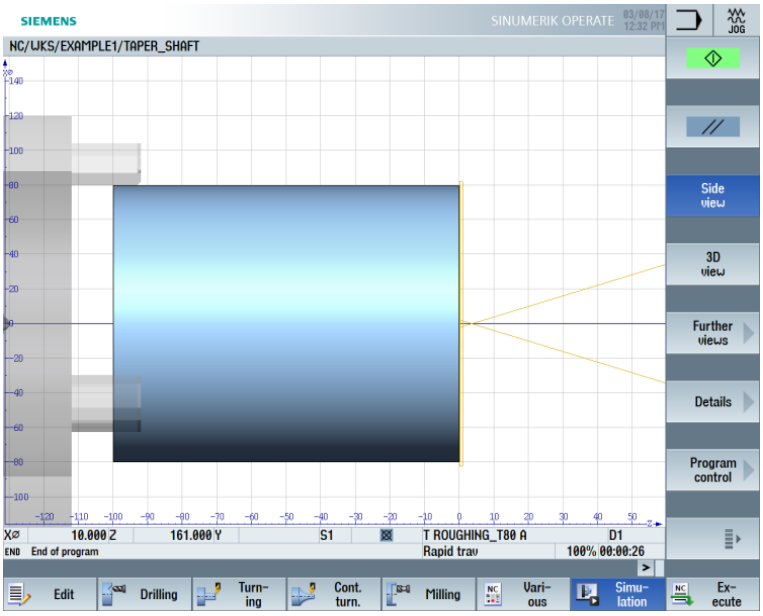


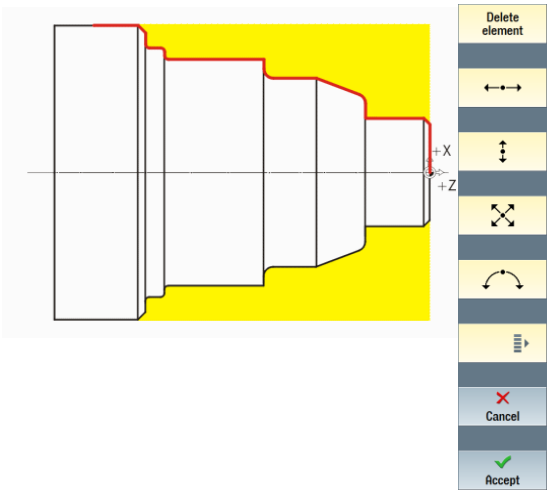
Figure 6-18 Simulation – Side view

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

7.5 Creating contours with the contour calculator and machining

Contour calculator

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



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

Operating sequences

Follow the steps below to enter the contour: Select the "Cont. turn." softkey.

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

Each contour is assigned its own name. This makes programs easier to read.

Figure 6-19 Creating the "TAPER_SHAFT_CONTOUR" contour

Accept your input.

You can accept the starting point for the contour line without making any changes (see figure below).

Note:

The contour definition is the roughing limit on the one hand and the finishing distance on the other hand.

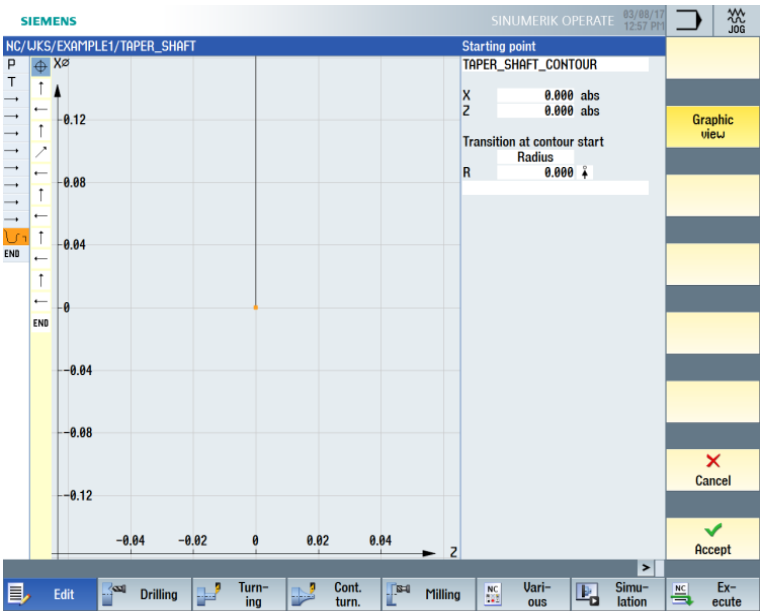


Figure 6-20 Entering the starting point

Note:

If you deselect the "Graphic view" softkey, detailed help screens are displayed.

Accept your input.

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

Field	Value	Selection via toggle key	Notes
X	30 abs	X	
Transition to next element	Cham	X	Attach the chamfer (FS) directly to the straight line as a transition element.
FS	1.5		

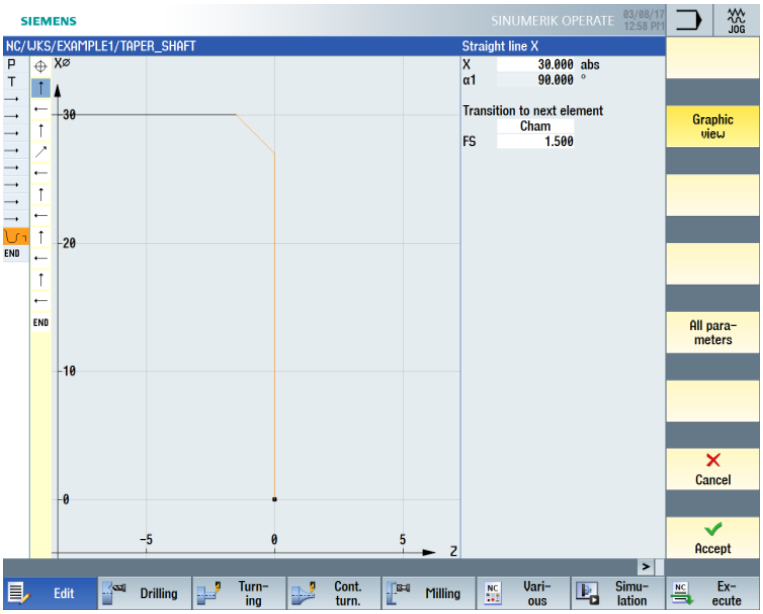
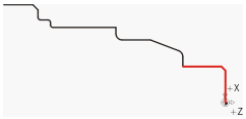


Figure 6-21 Entering the vertical straight-line segment for the contour

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
Z	-17 abs	X	A straight line follows to Z-17.
Transition to next element	Cham	X	
FS	0		The thread undercut will be inserted later as an individual element.

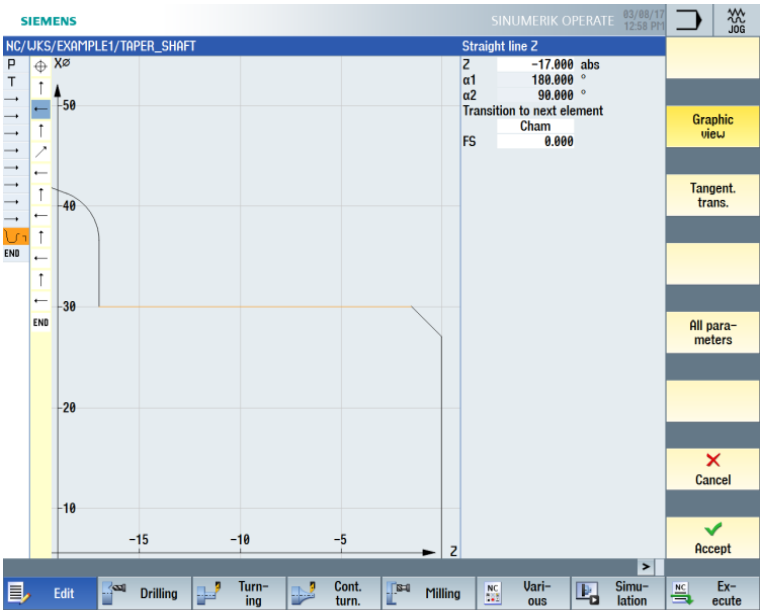
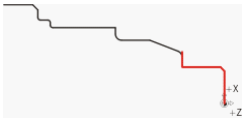


Figure 6-22 Entering the horizontal straight-line segment for the contour

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
X	40 abs	X	Construct the vertical straight-line segment up to the dimensioned intersection including the rounding to the next element. 
Transition to next element	Radius	X	
R	2.5		

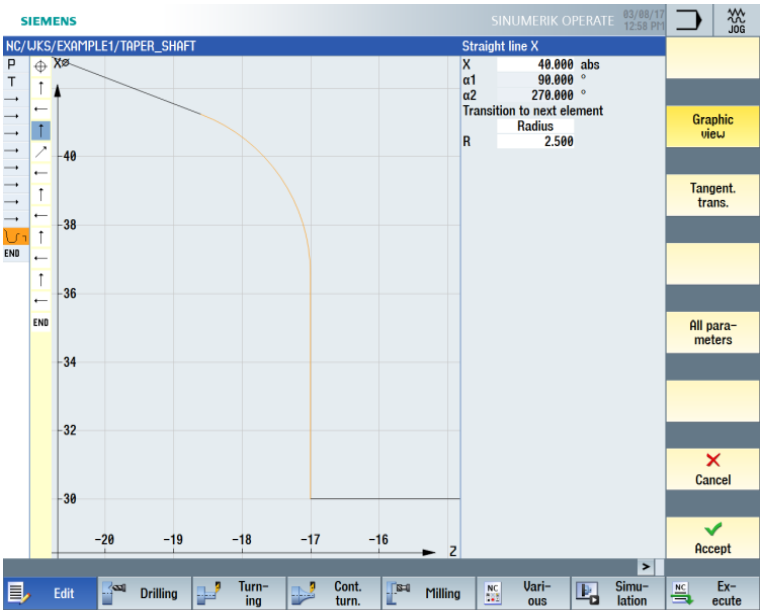
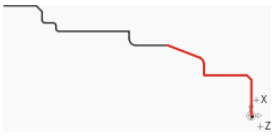


Figure 6-23 Entering the vertical straight-line segment for the contour

Accept the entered values.

Enter the following values for the end point of the inclined straight line in the screen form:

Field	Value	Selection via toggle key	Notes
X	50 abs	X	
Z	-30 abs	X	
Transition to next element	Cham	X	
FS	0		

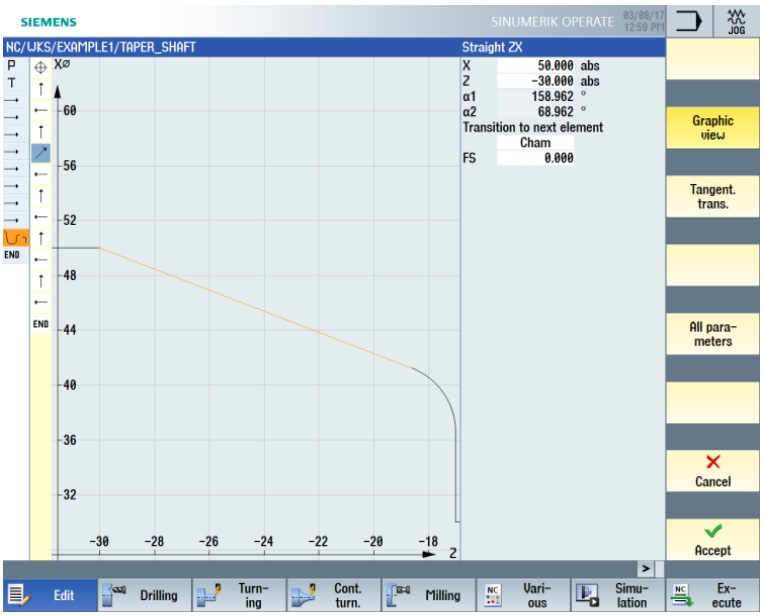
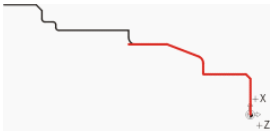


Figure 6-24 Entering the end point of the inclined straight line for the contour

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
Z	-44 abs	X	
Transition to next element	Radius	X	
R	2.5		

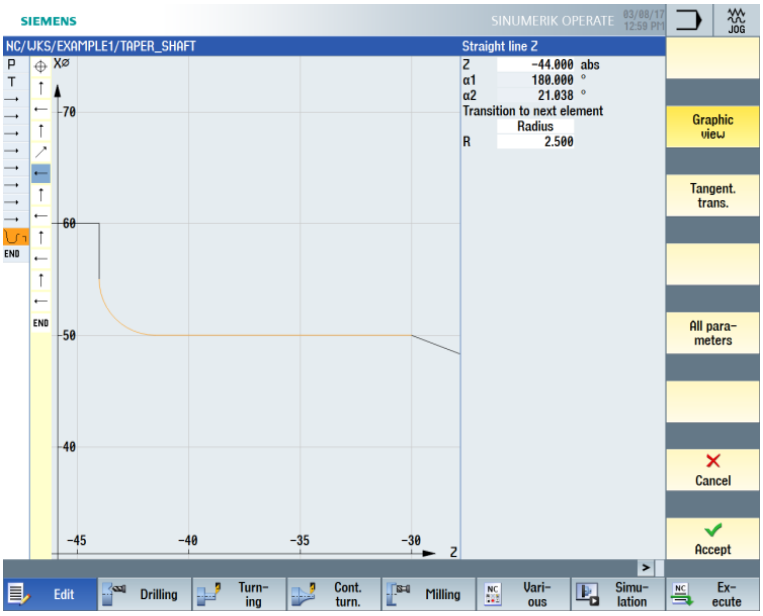
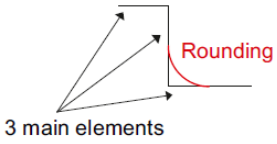


Figure 6-25 Entering the horizontal straight-line segment for the contour

Übernehmen

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
X	60 abs	X	The line segments (= main elements) are not tangential. 

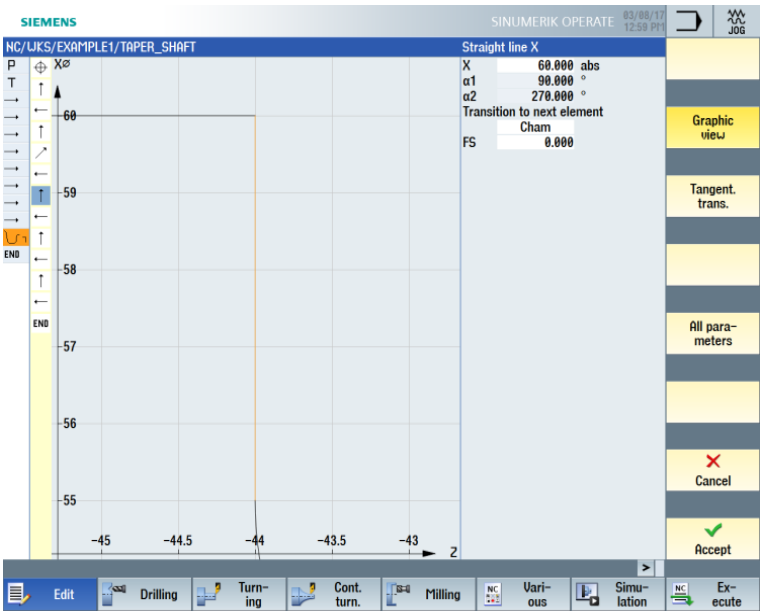


Figure 6-26 Entering the vertical straight-line segment for the contour

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
Z	-70 abs	X	Similar to the thread undercut, the grooves are entered later as individual elements. 
Transition to next element	Radius	X	
R	1		

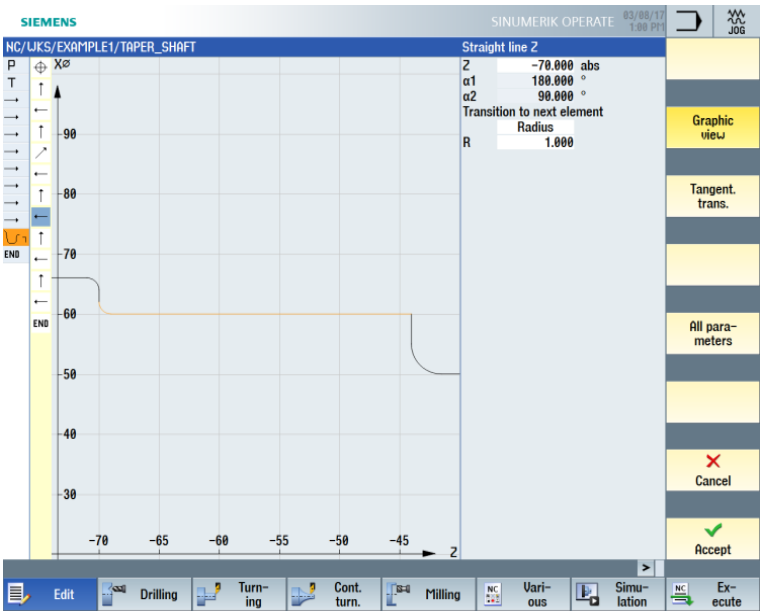


Figure 6-27 Entering the horizontal straight-line segment for the contour

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
X	66 abs	X	
Transition to next element	Radius	X	
R	1		

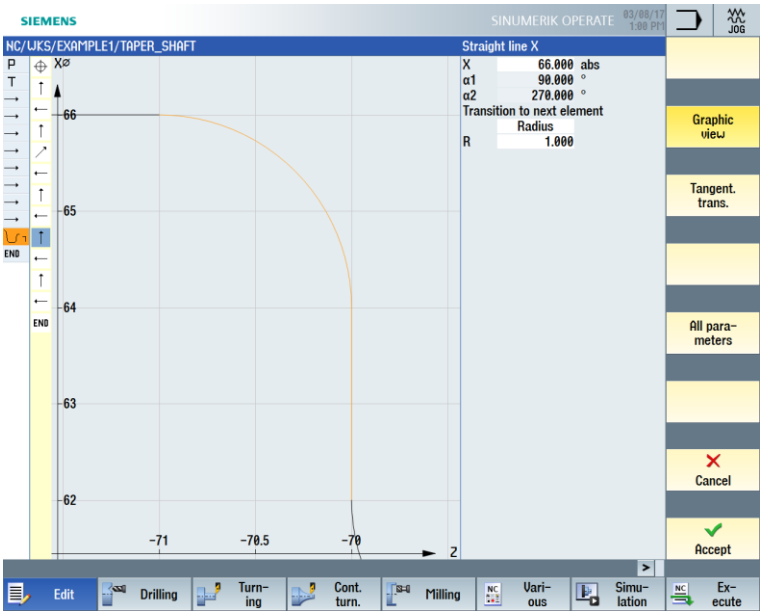
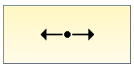


Figure 6-28 Entering the vertical straight-line segment for the contour

Accept

Accept the entered values.



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

Field	Value	Selection via toggle key	Notes
Z	-75 abs	X	
Transition to next element	Radius	X	
R	1		

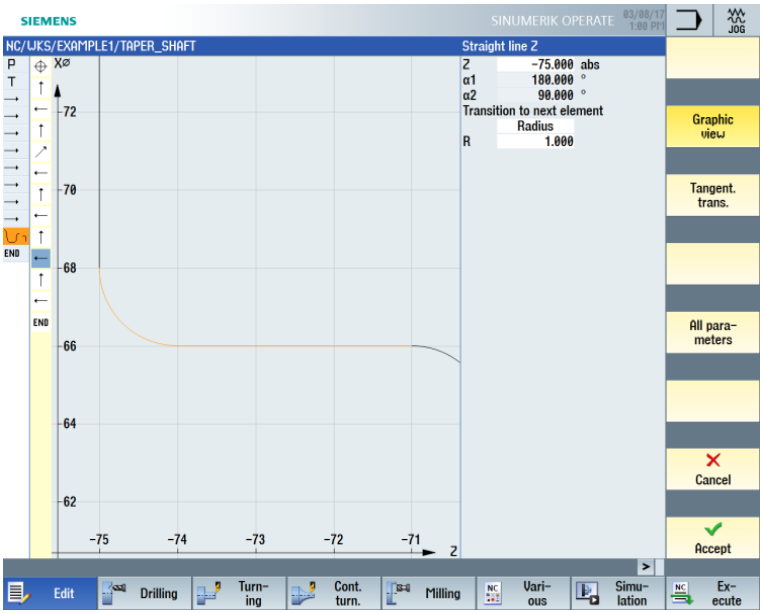


Figure 6-29 Entering the horizontal straight-line segment for the contour

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
X	80 abs	X	End point X80 with a 2 x 45° cham 
Transition to next element	Cham	X	
FS	2		

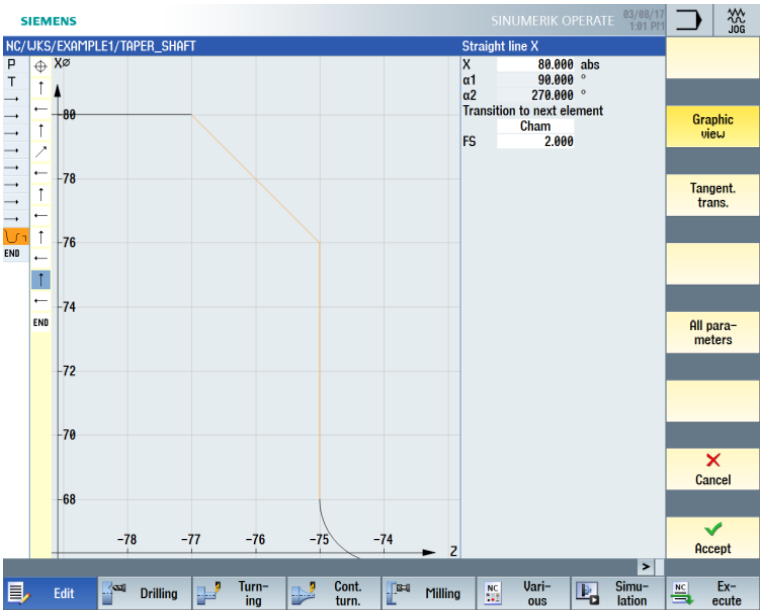


Figure 6-30 Entering the vertical straight-line segment for the contour

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
Z	-90 abs	X	
Transition to next element	Cham	X	
FS	0		The contour end point lies at X80 and Z-90 (2 mm in front of the chuck).

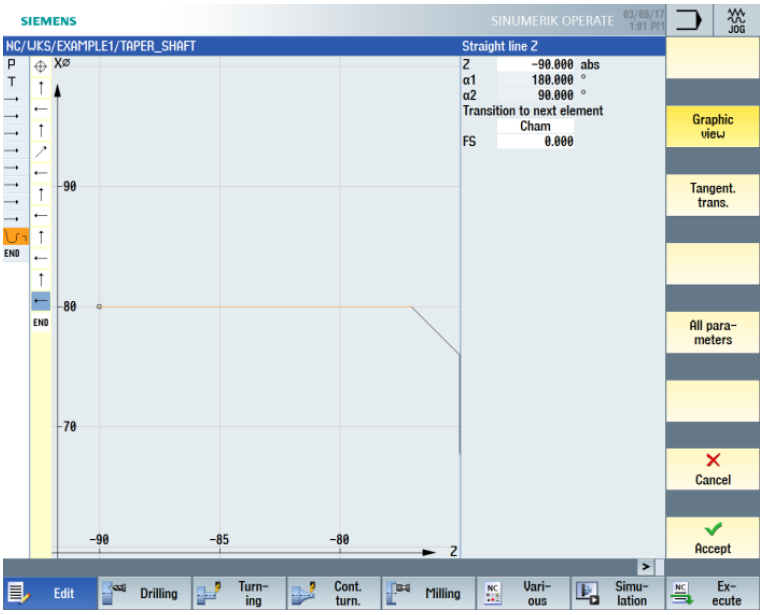


Figure 6-31 Specifying the contour end point

Accept

Accept the entered values.

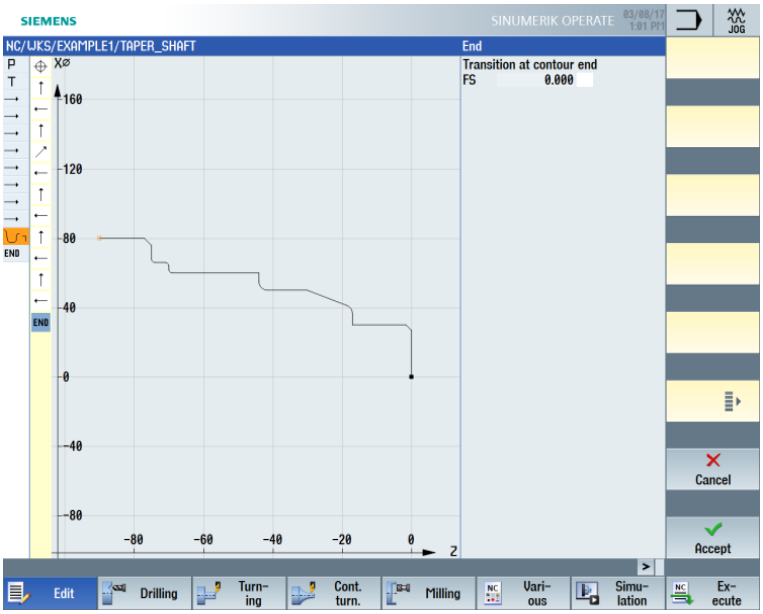


Figure 6-32 Complete contour

Accept

Accept the contour to apply it to your machining plan.

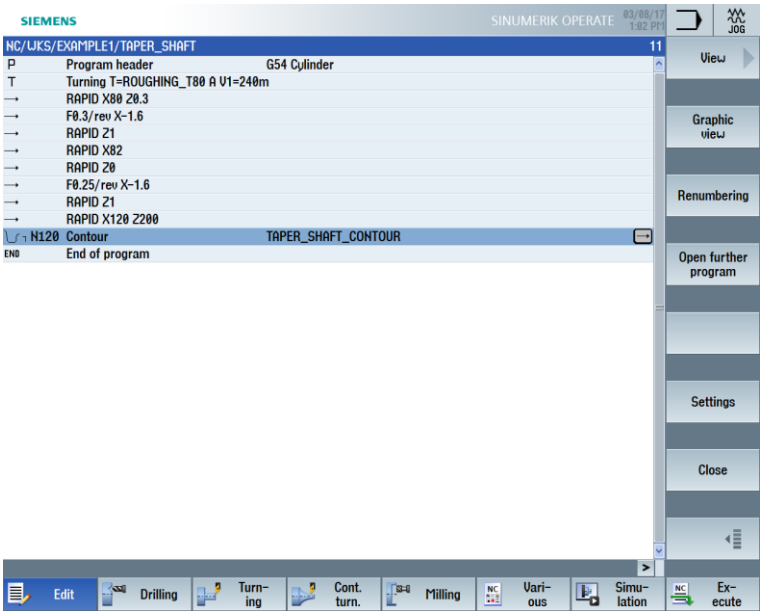


Figure 6-33 Contour in the machining plan

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

Stock removal

Select the "Stock removal" softkey.

Select tool

Open the tool list and select "ROUGHING_T80 A".

OK

Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.3		
V	240 m/min	X	
Machining	Roughing Longitudinal Outside	X X	
D	2.5		
UX	0.5		
UZ	0.2		
DI	0.0		
BL	Cylinder	X	
XD	0.0 inc	X	
ZD	0.0 inc	X	
Relief cuts	No	X	
Limit	No	X	

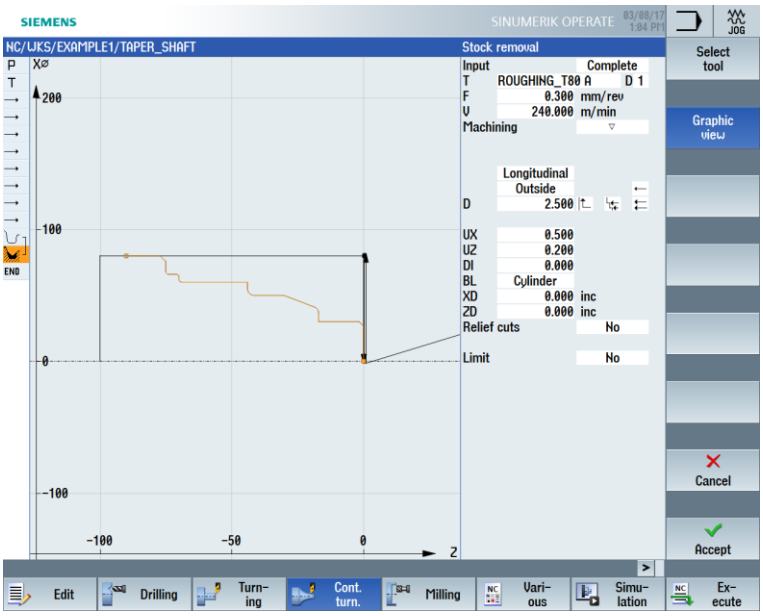
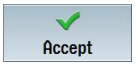


Figure 6-34 Roughing the contour



Accept the entered values.

Select the "Stock removal" softkey.

Open the tool list and select "FINISHING_T35 A".

Apply the tool to the program.

Enter the following values for finishing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	200 m/min	X	
Machining	Finishing	X	

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

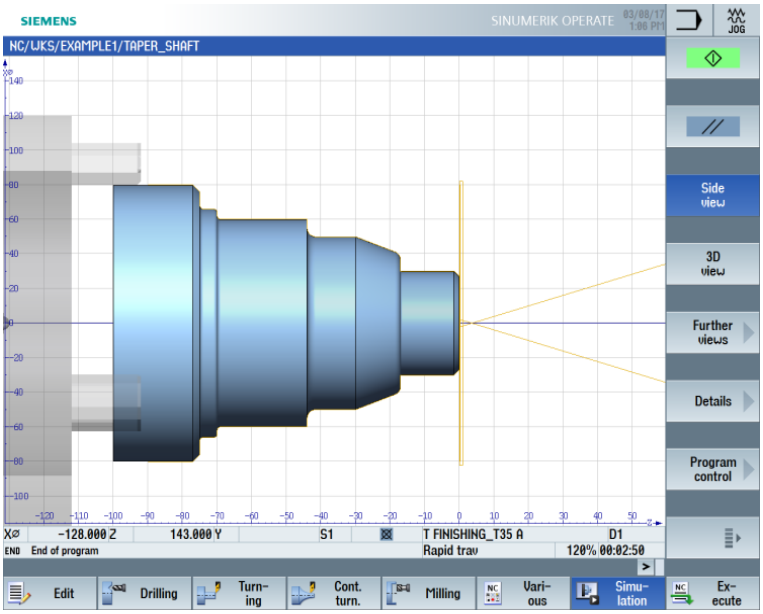


Figure 6-37 Simulation side view

7.6 Thread undercut

Operating sequences

Follow the steps below to create the thread undercut:

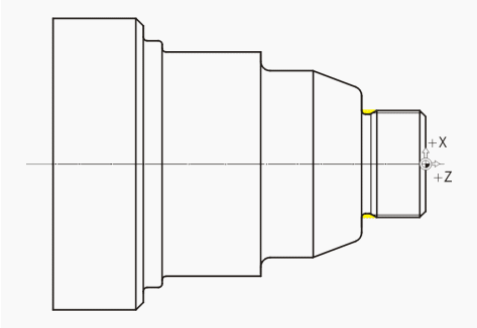
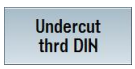


Figure 6-38 Thread undercut

Select the "Turning" softkey.

Select the "Undercut" softkey.



Select the "Undercut thread" softkey.

Open the tool list and select the finishing tool "FINISHING_T35 A".

Apply the tool to the program.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	200 m/min	X	
Machining	Roughing / Finishing Longitudinal	X X	
Pos.		X	See figure above
X0	30		
Z0	-17		
X1	1.15 inc	X	
Z1	4.5 inc	X	
R1	0.8		
R2	0.8		
α	30		
VX	1 inc	X	
D	0.8		
UX	0.1	X	
UZ	0.1	Z	

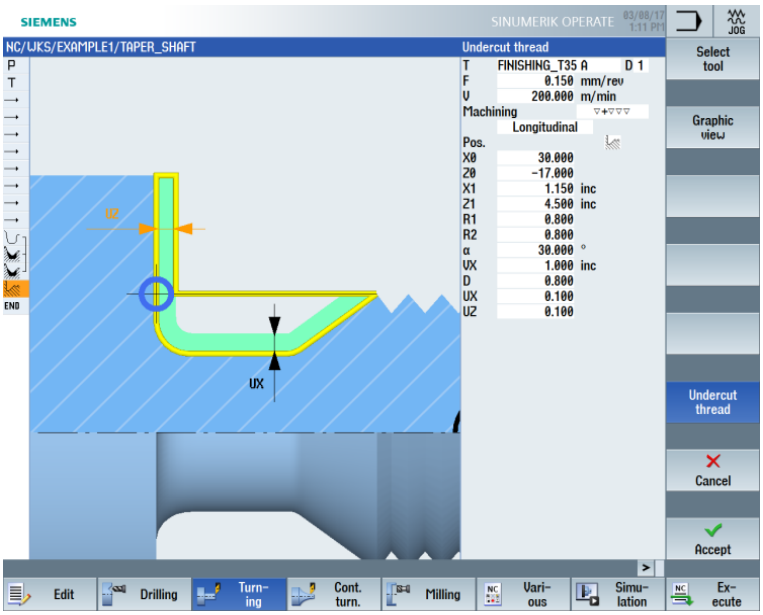


Figure 6-39 Thread undercut

Switch between the graphic view and the help screen as required.

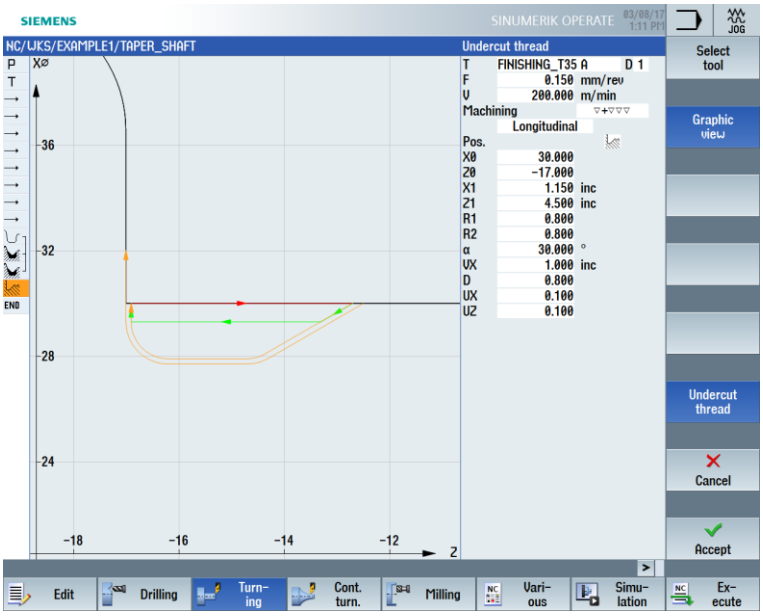
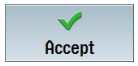


Figure 6-40 Thread undercut – Graphic view



Accept the entered values.

Select the "Simulation" softkey. Check the thread undercut via the detailed view in the 3D view, for example.

Select the "3D view" softkey.

Select the "Details" softkey. You can manipulate the display as appropriate using the "Zoom +", "Zoom -", "Zoom", etc. softkeys.

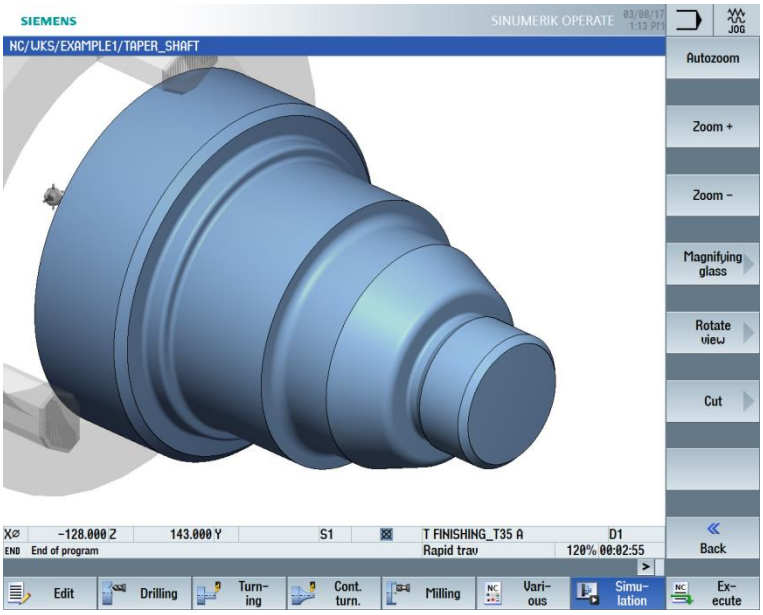


Figure 6-41 Detailed view of the simulation in the 3D view

7.7 Thread

Operating sequences

Follow the steps below to create the thread:

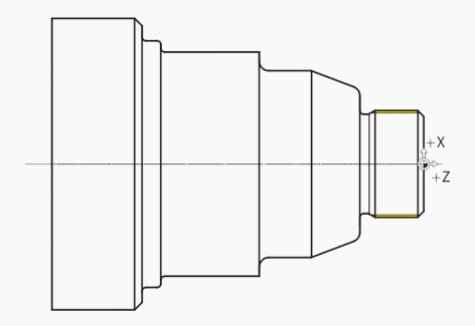


Figure 6-42 Thread

Select the "Thread" softkey.

Open the tool list and select the "THREADING_T1.5" drill.

Apply the tool to the program.

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

Field	Value	Selection via toggle key	Notes
L	1.5 mm/rev	X	
D	0		
S	800 rpm	X	
Machining	Roughing/Finishing Linear External thread	X X X	
X0	30	X	You define the thread geometry with the following inputs.
Z0	0		
Z1	-16 abs	X	
LW	2		
LR	1		
H1	0.92		
αP	29	X	
ND	8		

Field	Value	Selection via toggle key	Notes
U	0.1		
NN	0		
VR	2		
Multiple threads	No	X	
α0	0		

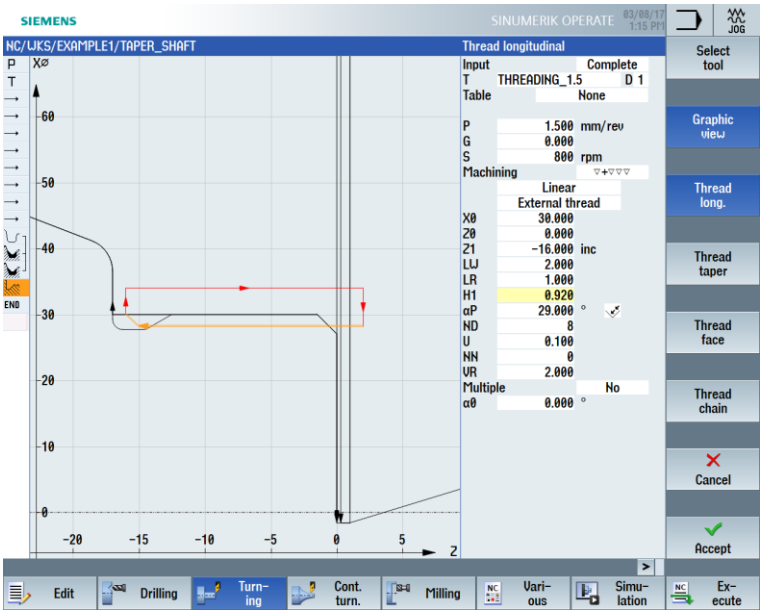


Figure 6-43 Thread – Graphic view

Switch to the help screen when necessary.

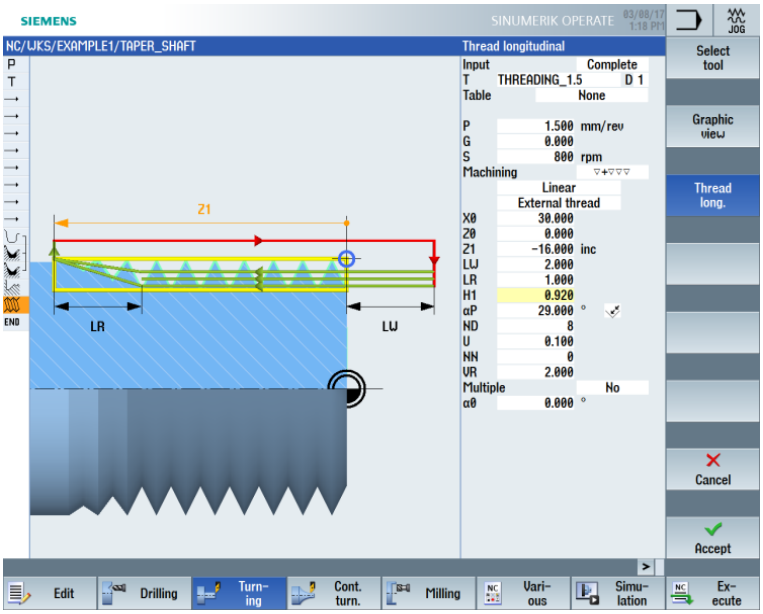
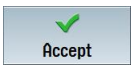


Figure 6-44 Thread – Help screen



Accept the entered values.

Start the "Simulation".

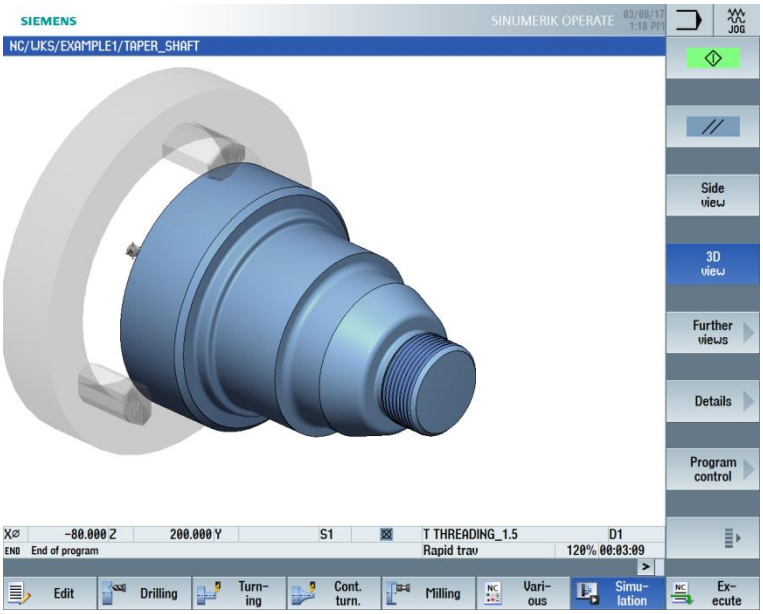


Figure 6-45 Simulation of thread

7.8 Grooves

Operating sequences

Follow the steps below to create the two grooves:

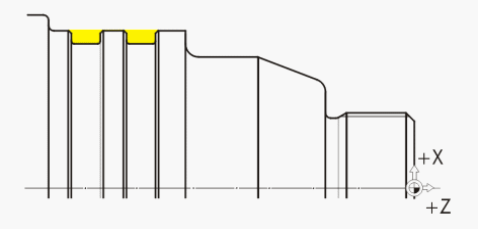


Figure 6-46 Grooves

Select the "Groove" softkey.

Select the "Groove 2" softkey.

Open the tool list and select the grooving tool "PLUNGE_CUTTER_3 A".

Apply the tool to the program.

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

Field	Value	Selection via toggle key	Notes
F	0.1		
V	150 m/min	X	
Machining	Roughing/ Finishing		
Pos.			See figure above
X0	60		You define the geometry of the grooves with the following inputs.
Z0	-65		
B1	6	X (field)	
T1	3 inc	X	
α1	0		
α2	0		
FS1	0.5	X (field)	
R2	1	X (field)	
R3	1	X (field)	
FS4	0.5	X (field)	
D	3		
U	0.1	X (field)	
N	2		
DP	10		

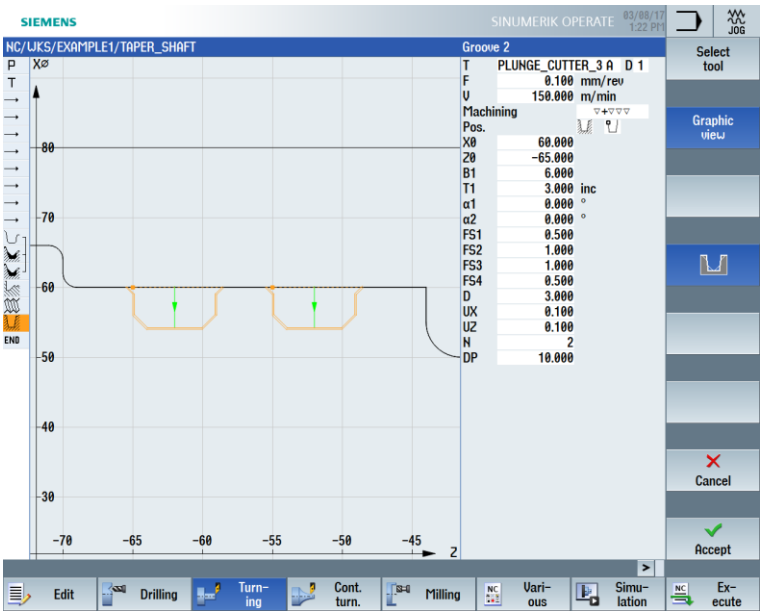


Figure 6-47 Grooves – Graphic view

Switch to the help screen when necessary.

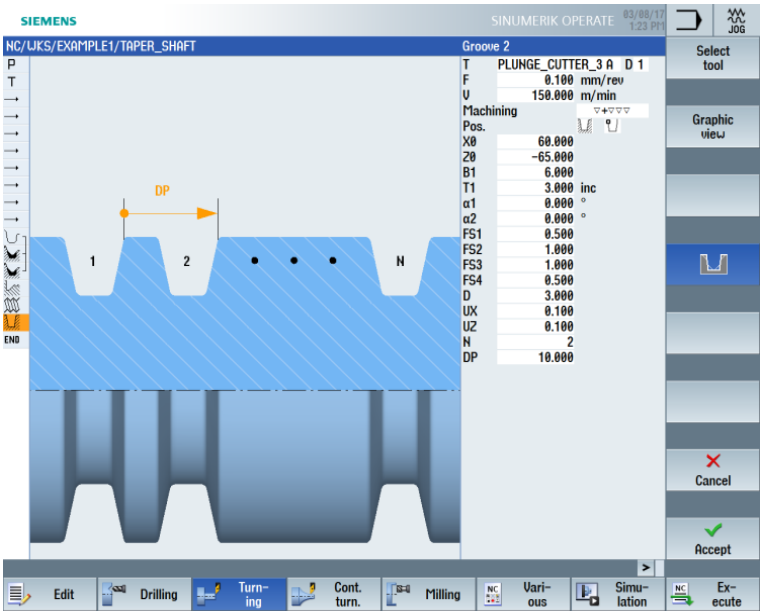


Figure 6-48 Grooves – Help screen

Accept the entered values.

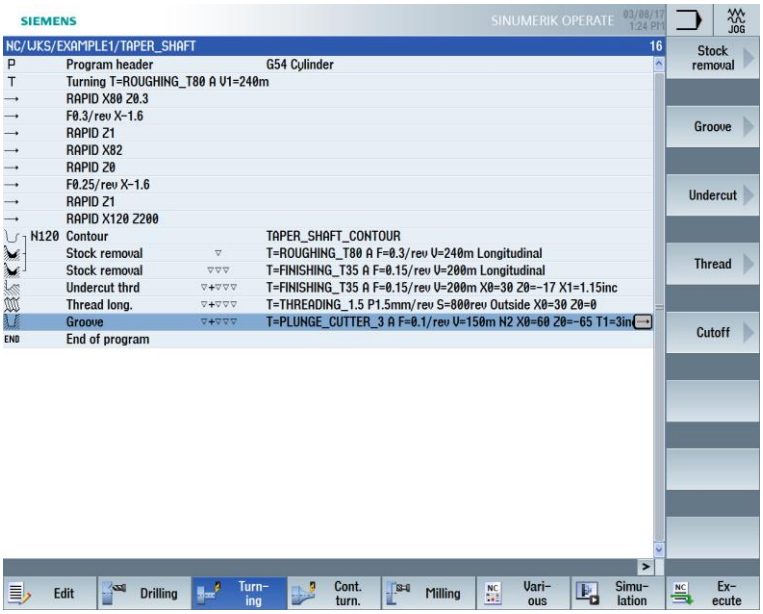
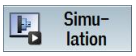


Figure 6-49 Machining plan with grooves



Start the "Simulation" in the side view or in the 2-window view, for example.

Select the "Side view" softkey.

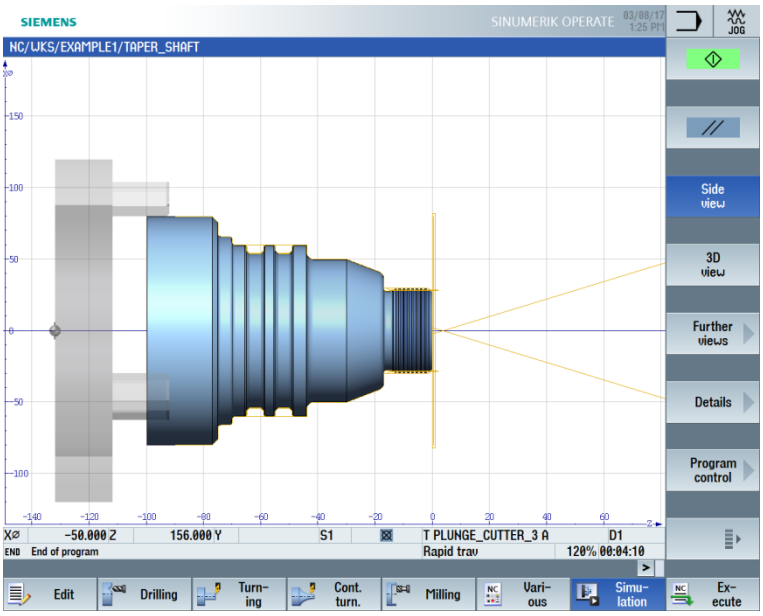


Figure 6-50 Simulation – Side view

Select the "Further views" softkey.

Select the "2 windows" softkey.

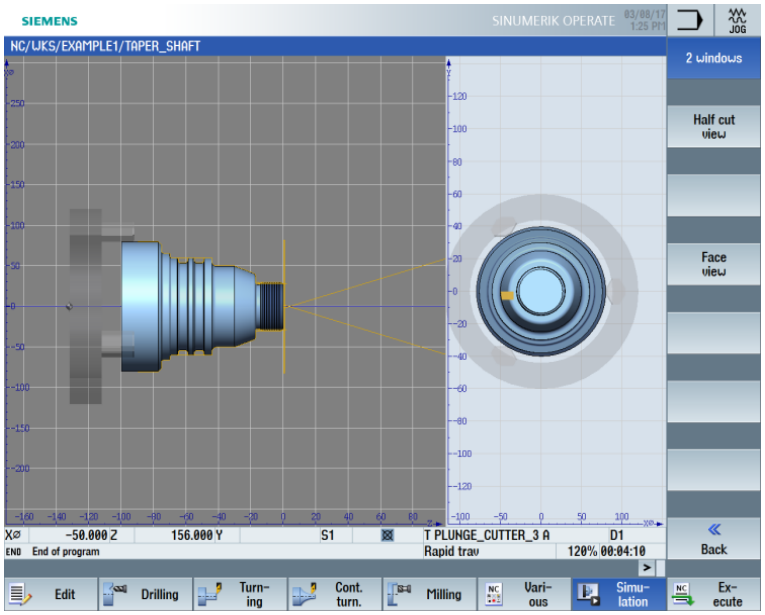


Figure 6-51 Simulation – 2-window view

8. Example 2: Drive shaft

8.1 Overview

Learning objectives

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

- Perform face turning
- Work with the contour calculator (advanced application)
- Machine residual material

Task

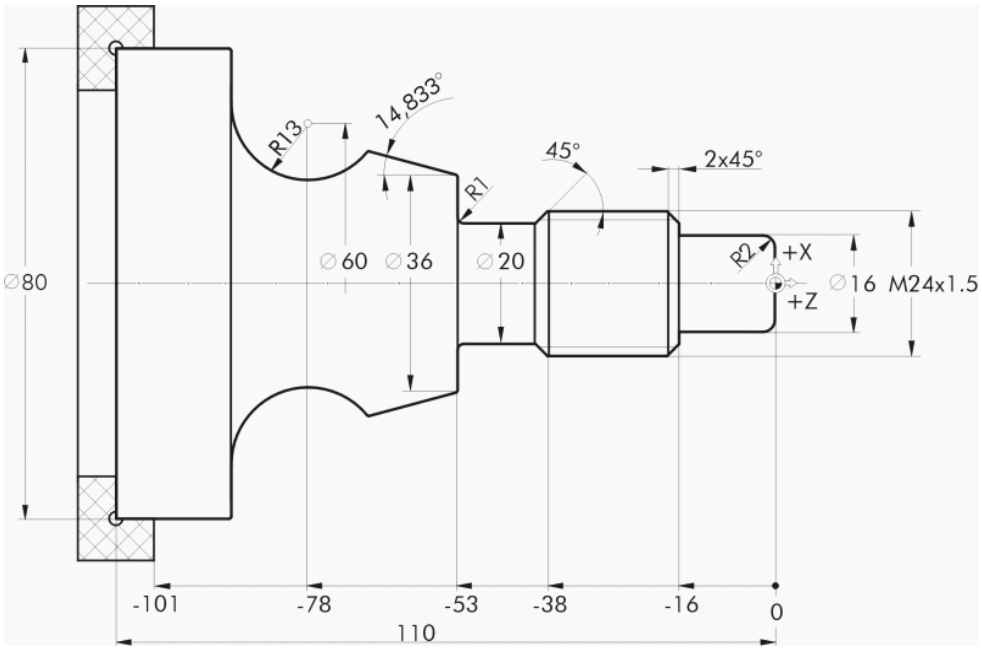


Figure 7-1 Workshop drawing – Example 2

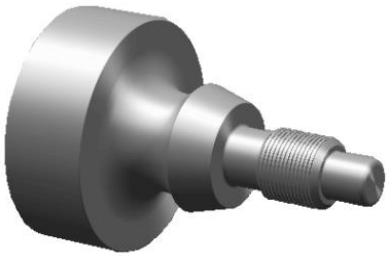


Figure 7-2 Workpiece – Example 2:

Preparations

Perform the following steps on your own:

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

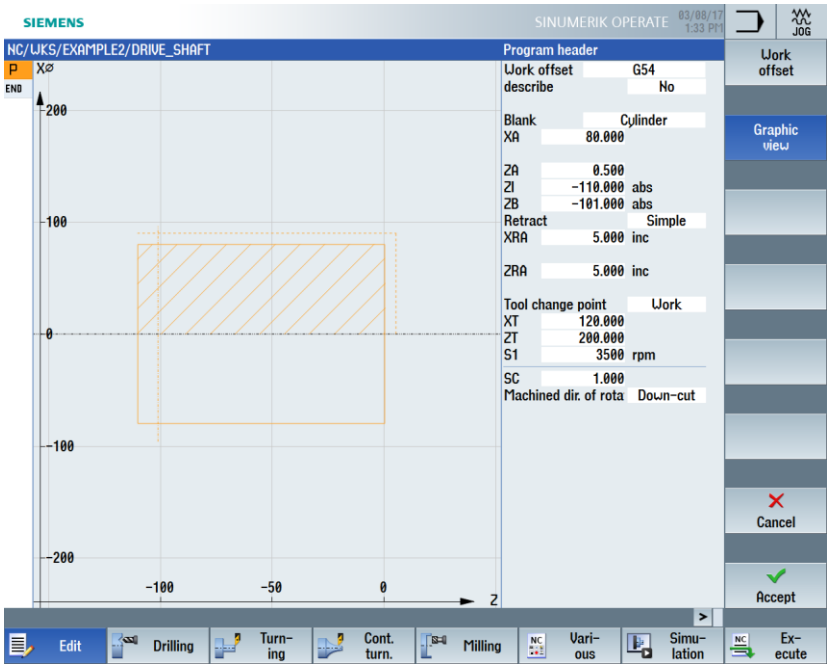


Figure 7-3 Creating the program header

After creation of the program header, the machining plan looks like this:

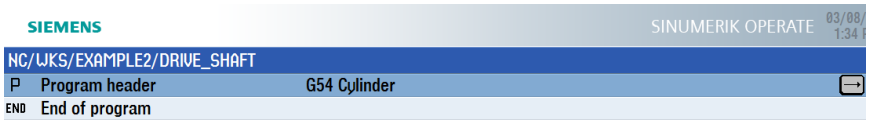


Figure 7-4 Machining step program

8.2 Face turning

Operating sequences

Follow the steps below to face the workpiece:

Select the "Turning" softkey.

Select the "Stock removal" softkey.

Since face turning is to be completed in one cut, switch the machining to finishing. Select the "ROUGHING_T80 A" tool and enter the following values.

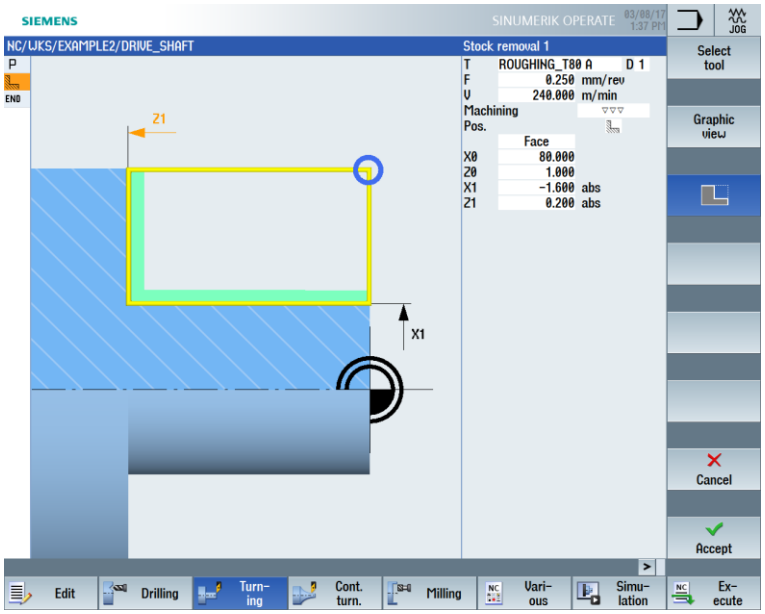
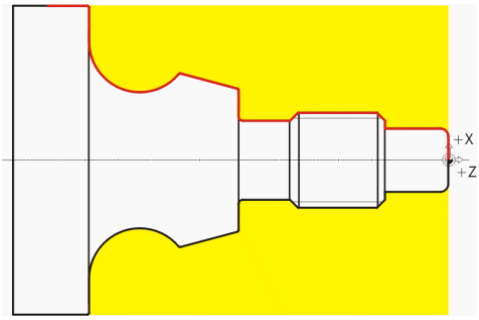


Figure 7-5 Facing the workpiece

8.3 Creating the contour, stock removal and residual stock removal

Operating sequences

Follow the steps below to enter the contour:



Select the "Cont. turning" softkey.

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

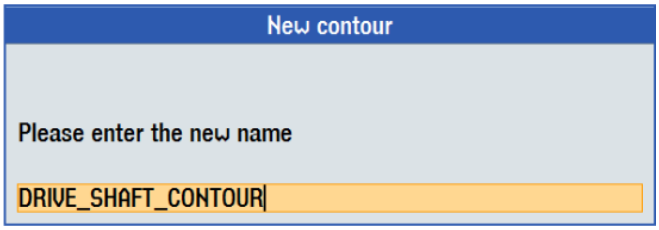


Figure 7-6 Creating the contour

Accept your input.

You can accept starting point X0/Z0 directly (see figure below).

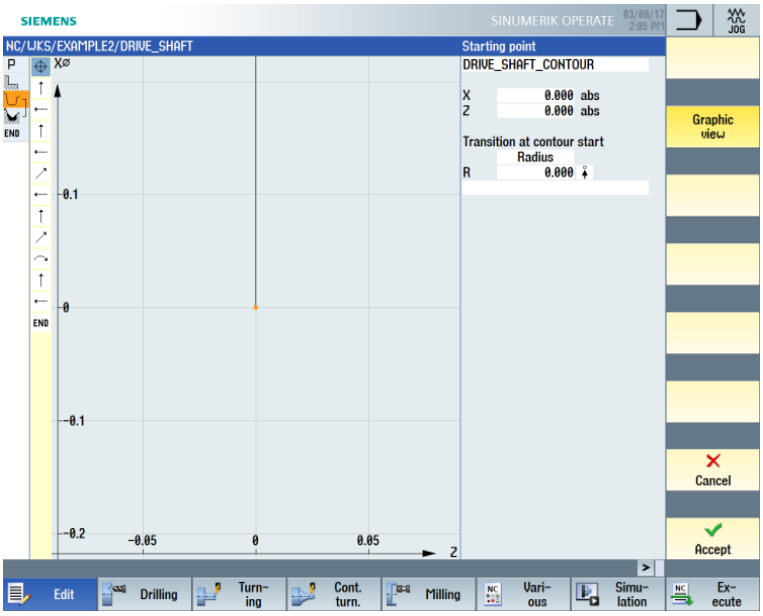
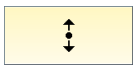


Figure 7-7 Applying the starting point

Accept your input.



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

Field	Value	Selection via toggle key	Notes
X	16 abs	X	
Transition to next element	Radius	X	
R	2		

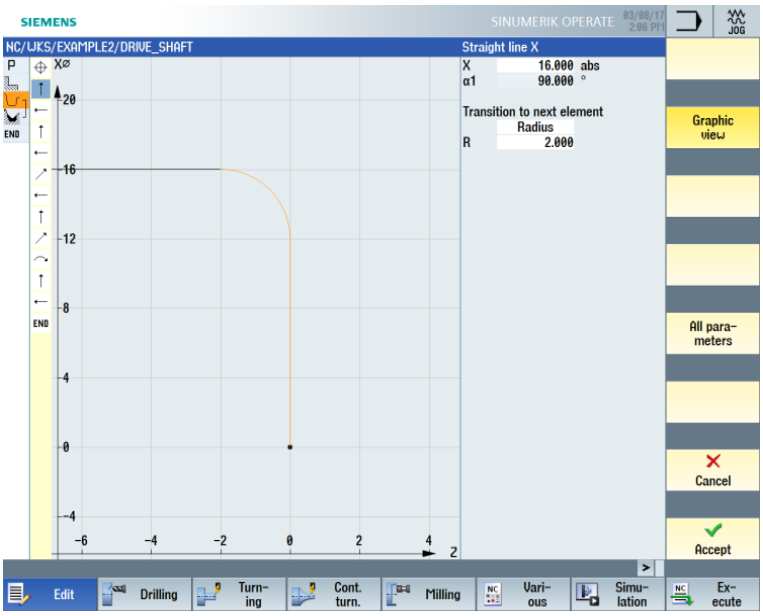


Figure 7-8 Entering the vertical straight-line segment for the contour

Accept

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
Z	-16 abs	X	
Transition to next element	Cham	X	
FS	0		

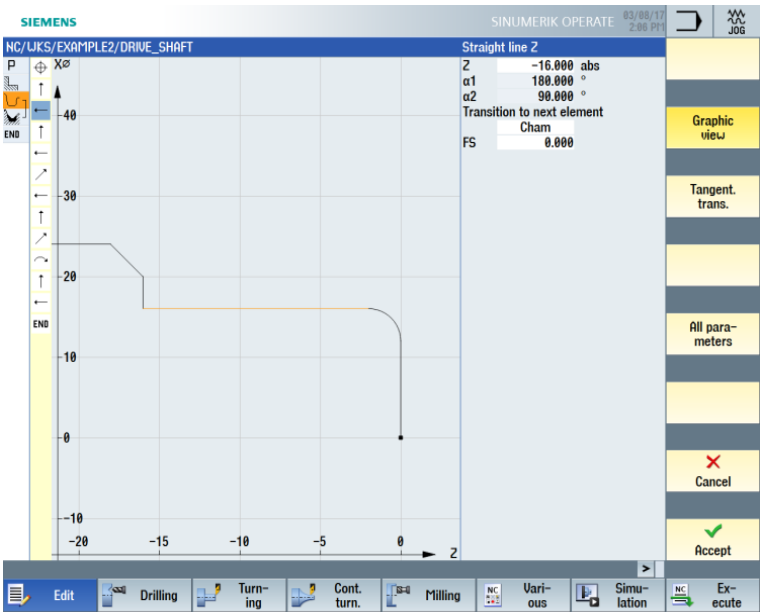


Figure 7-9 Entering the horizontal straight-line segment for the contour

Accept

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
X	24 abs	X	
Transition to next element	Cham	X	
FS	2		

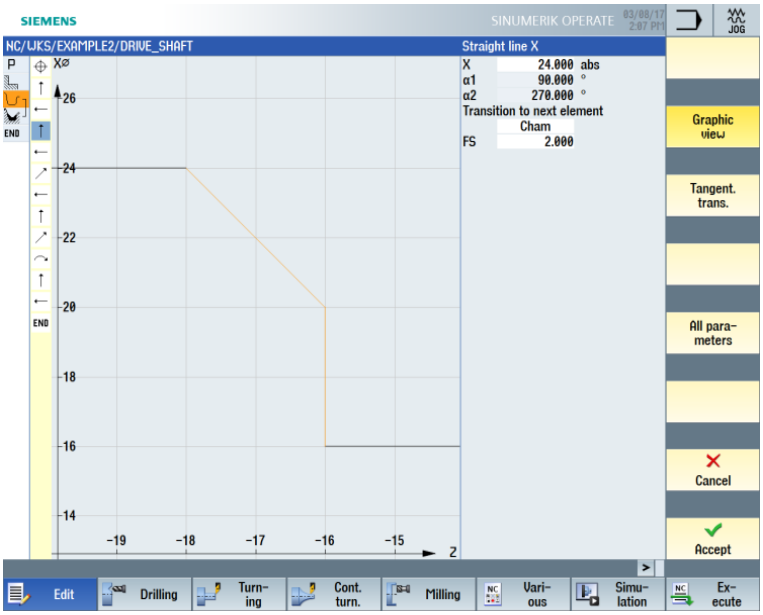


Figure 7-10 Entering the vertical straight-line segment for the contour

Accept

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
Z	-38 abs	X	
Transition to next element	Cham	X	
FS	0		

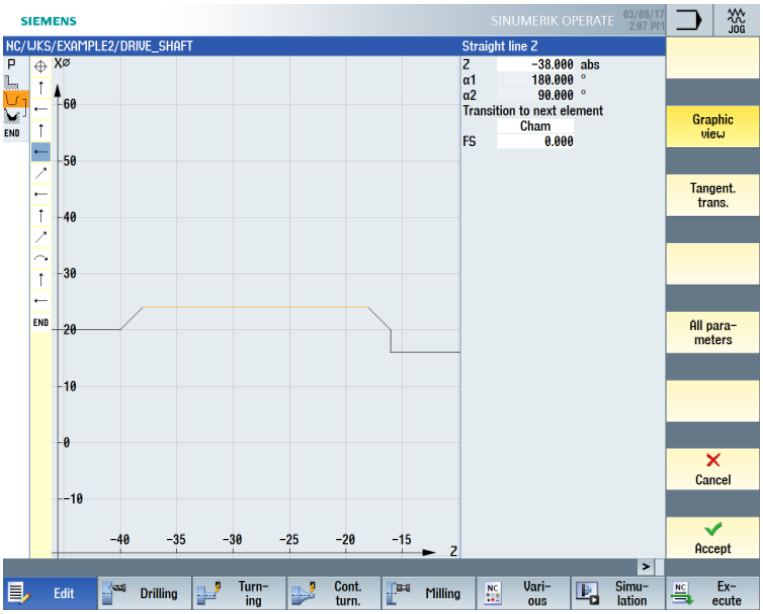
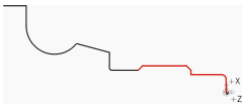


Figure 7-11 Entering the horizontal straight-line segment for the contour

Accept the entered values.

Enter the following values for the sloping straight-line segment in the screen form:

Field	Value	Selection via toggle key	Notes
X	20 abs	X	 The entered angle is relative to the preceding element.
α2	45	X	
Transition to next element	Cham	X	
FS	0		

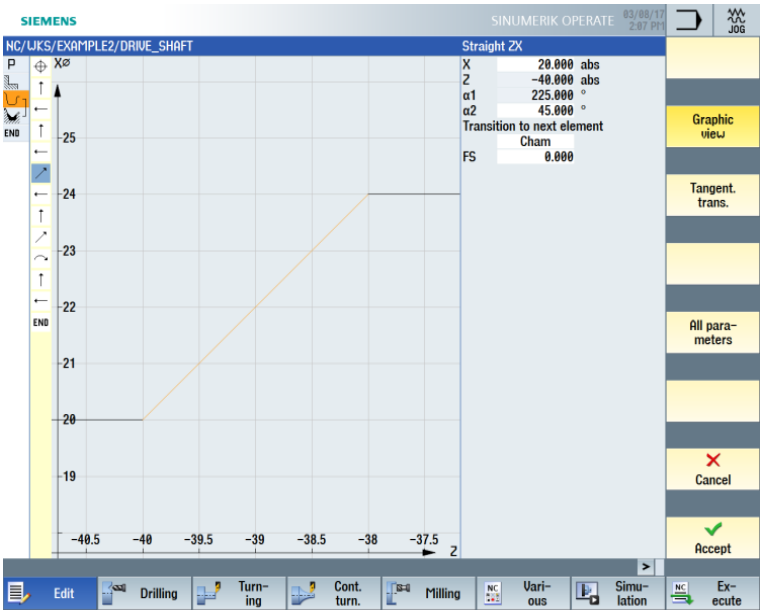
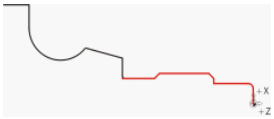


Figure 7-12 Entering the sloping straight-line segment for the contour

Accept

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
Z	-53 abs	X	
Transition to next element	Radius	X	
R	1		

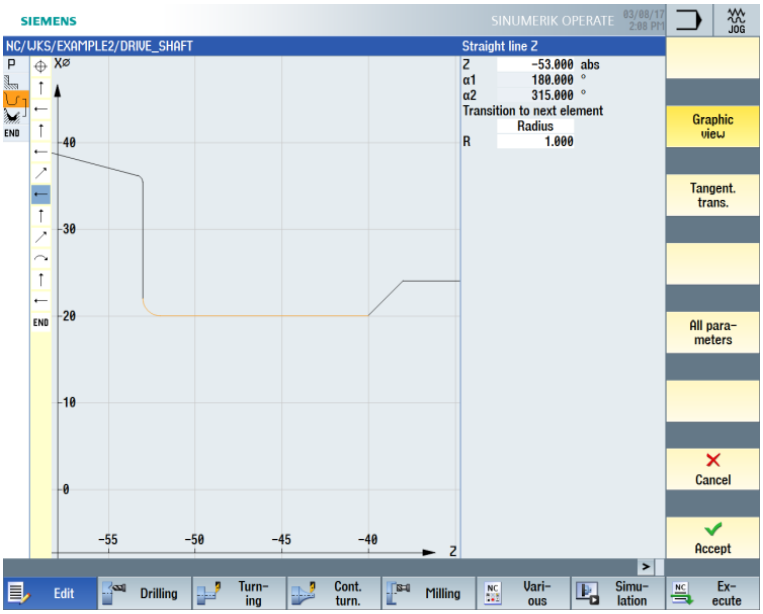
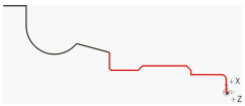


Figure 7-13 Entering the horizontal straight-line segment for the contour

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
X	36 abs	X	 Round the transition to the next element with R0.4.
Transition to next element	Radius	X	
R	0.4		

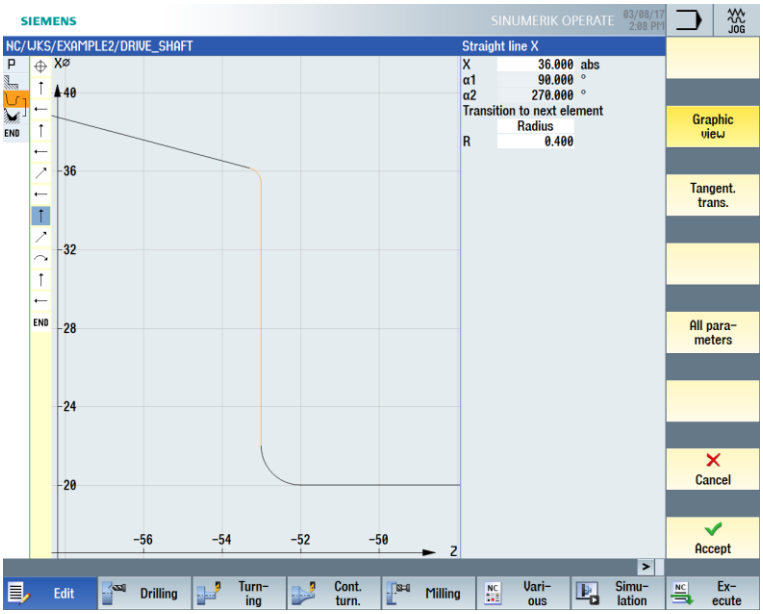


Figure 7-14 Entering the vertical straight-line segment for the contour

Accept

Accept the entered values.

Enter the following values for the next segment in the screen form:

Field	Value	Selection via toggle key	Notes
X		X	 From the line segment, nothing further is known than the angle relative to the Z-axis of 165.167°. In such cases, continue the construction with the next element.
Z		X	
α1	165.167°		
Transition to next element	Radius	X	
R	0.4		

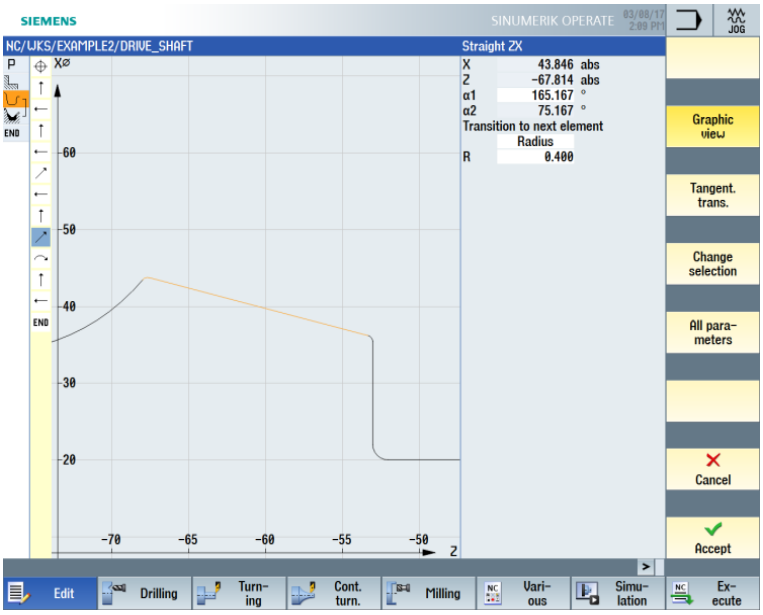


Figure 7-15 Entering the inclined straight line for the contour

Accept the entered values.

Enter the following values for the next segment in the screen form:

Field	Value	Selection via toggle key	Notes
Direction of rotation	right	X	 The missing points of the previous contour element are calculated using the known dimensions of the arc. Since there are several possibilities, you must select the correct one.
R	13		
X			
Z			
I	60 abs	X	
K	-78 abs	X	
Transition to next element	Cham	X	
R	0		

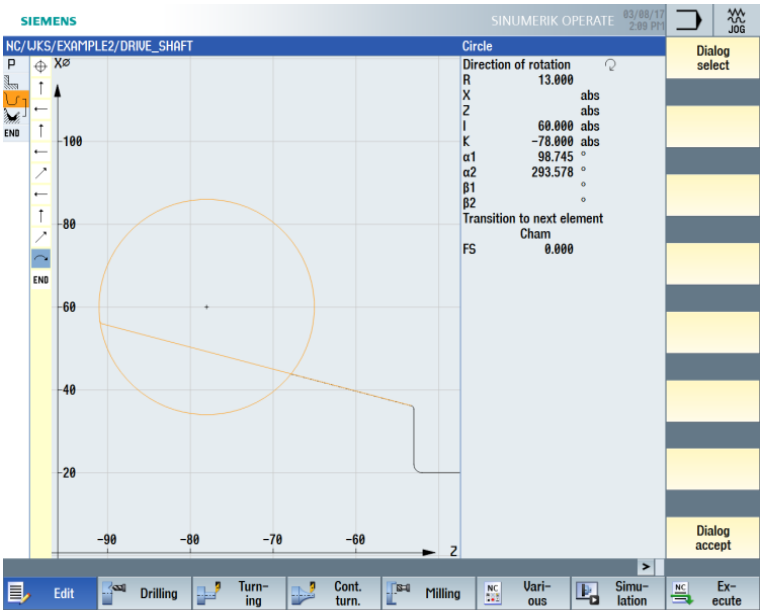


Figure 7-16 Entering the arc for the contour

Dialog select

Select the suggested solution corresponding to the figure below:

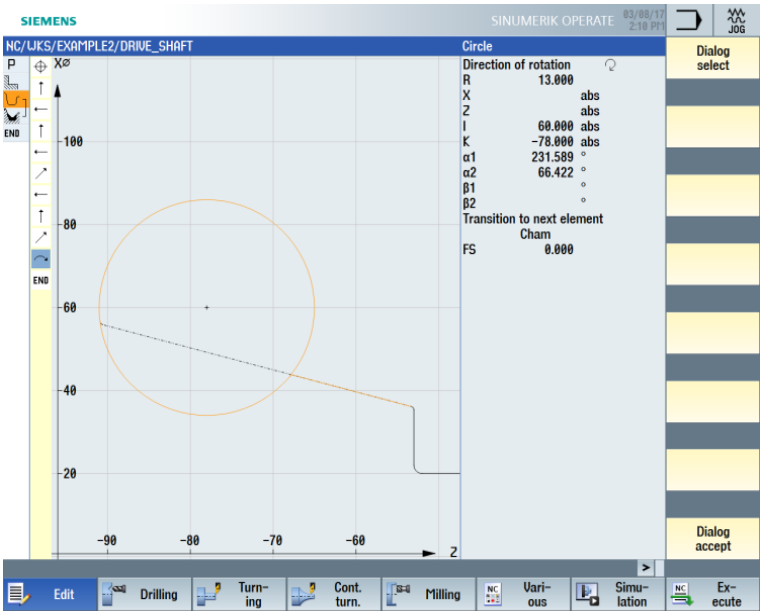


Figure 7-17 Confirming the contour selection

Dialog accept

Once you have selected the desired construction, accept it.

Since the end point of the arc is not known, continue the construction. You could also use the "All parameters" softkey to enter the runout angle here.

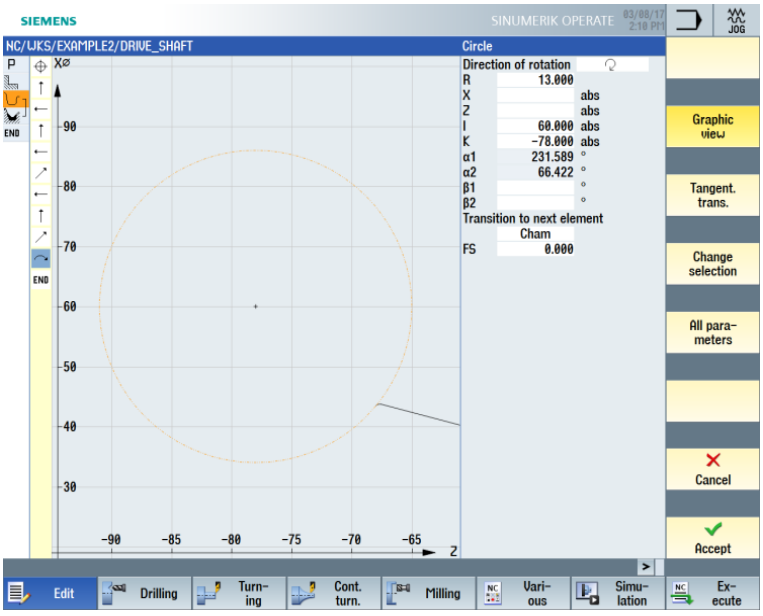
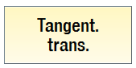


Figure 7-18 Applying the arc for the contour

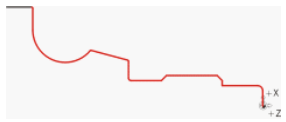
Accept

Accept the contour segment.

A tangential line segment follows.



Select the "Tangent trans." softkey.

Field	Value	Selection via toggle key	Notes
X	80 abs	X	
Transition to next element	Radius	X	
R	0.4		

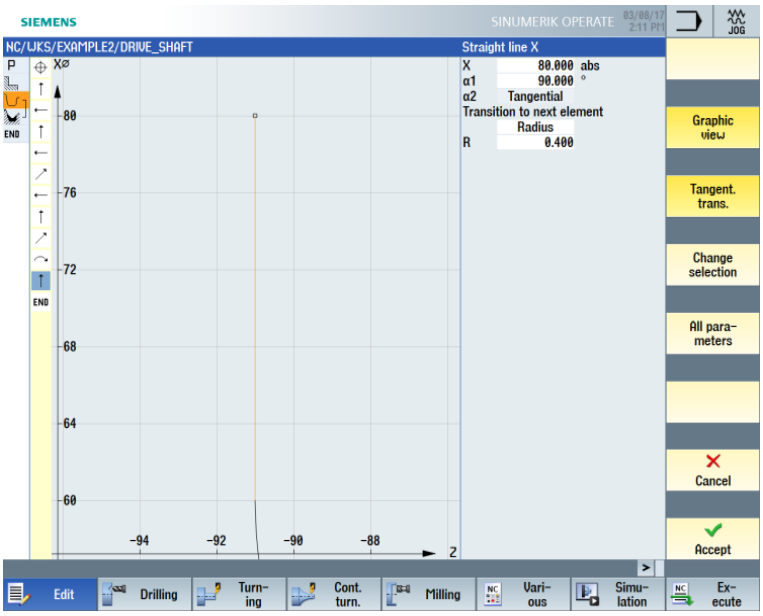


Figure 7-19 Entering the vertical straight-line segment for the contour

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
Z	-100 abs	X	 The end point of the contour is at Z-100.
Transition to next element	Cham	X	
FS	0		

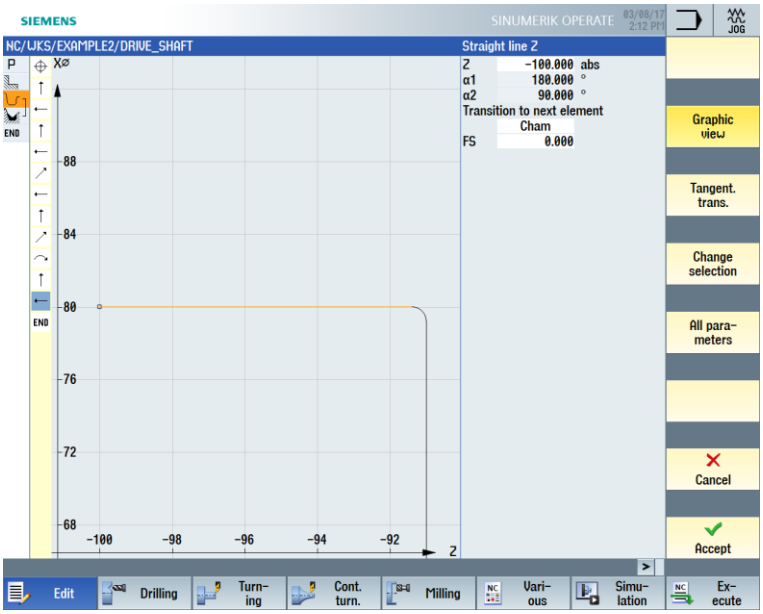


Figure 7-20 Entering the horizontal straight-line segment for the contour

Accept

Accept the entered values.

Accept

Accept the contour to apply it to your machining plan.

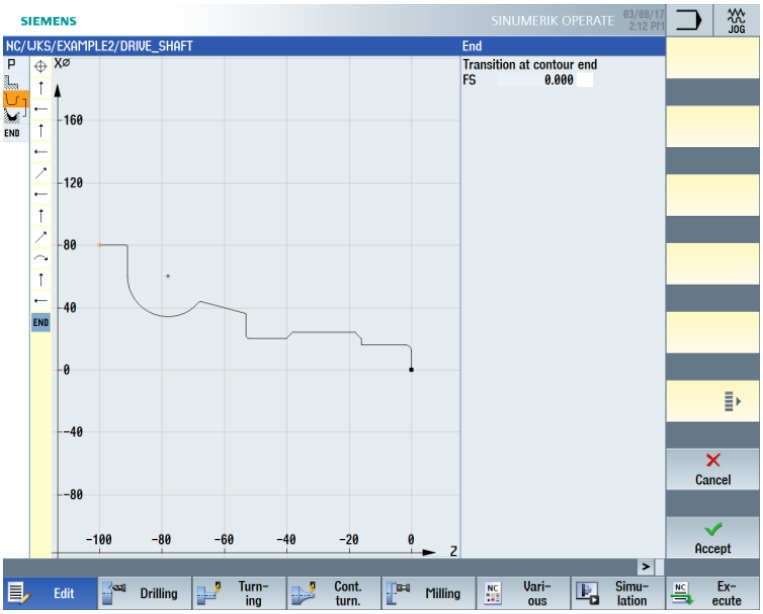


Figure 7-21 Applying the contour

Stock removal, residual stock removal and finishing

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

Stock removal

Select the "Stock removal" softkey.

Select tool

Open the tool list and select the "ROUGHING_T80 A" tool.

OK

Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.3		
S	240 rpm	X	
Machining	Roughing Contour-parallel Outside	X X X	The machining of the contour is performed parallel to the contour here, for example.
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		

Field	Value	Selection via toggle key	Notes
BL	Cylinder	X	
XD	0.0 inc	X	
ZD	0.0 inc	X	
Relief cuts	No	X	
Limit	No	X	

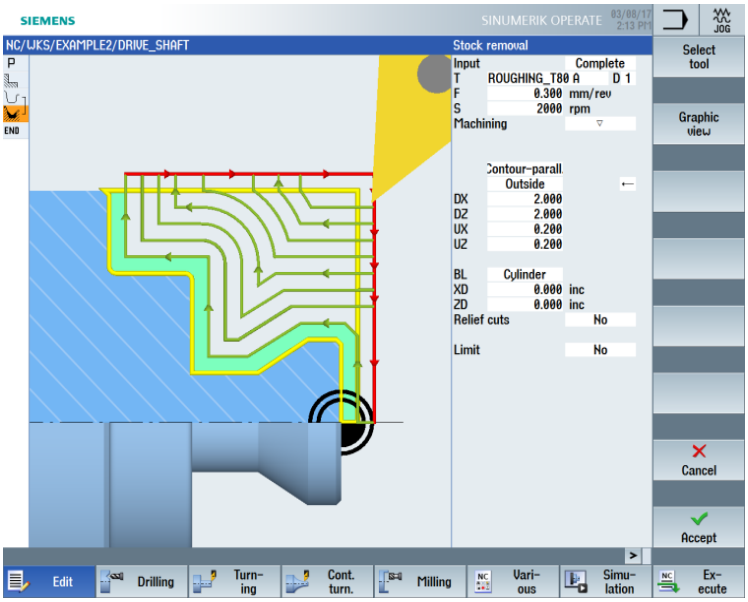


Figure 7-22 Roughing the contour

Accept the entered values.

Select the "Simulation" softkey.

Select the "Side view" softkey.

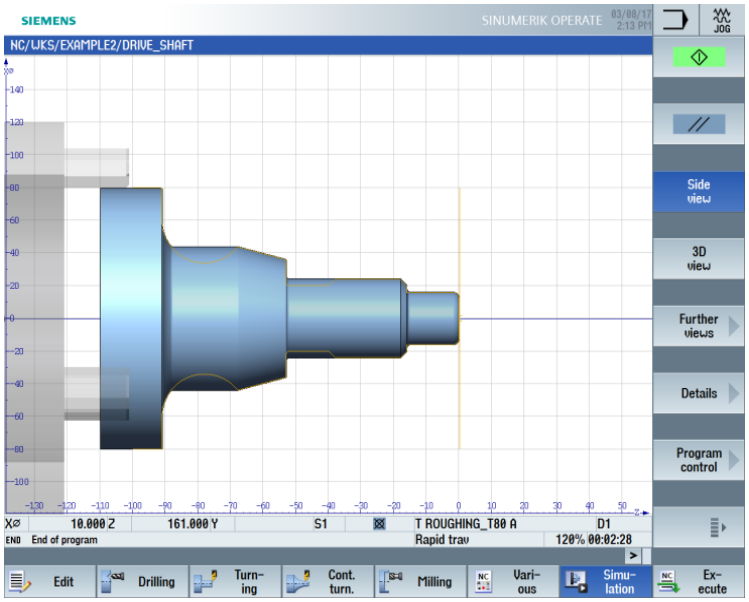


Figure 7-23 Roughing the contour – Simulation side view

Select the "Cont. turning" softkey.

Select the "Cut resid stock" softkey.

Open the tool list and select the "FINISHING_T35 A" tool.

Apply the tool to the program.

Enter the following values for residual stock removal in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.12		
V	240 m/min	X	
Machining	Roughing Longitudinal Outside	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
Relief cuts	Yes	X	To be able to cut all residual material, you must toggle the text box to “Yes”.
FR	0.2		
Limit	No	X	

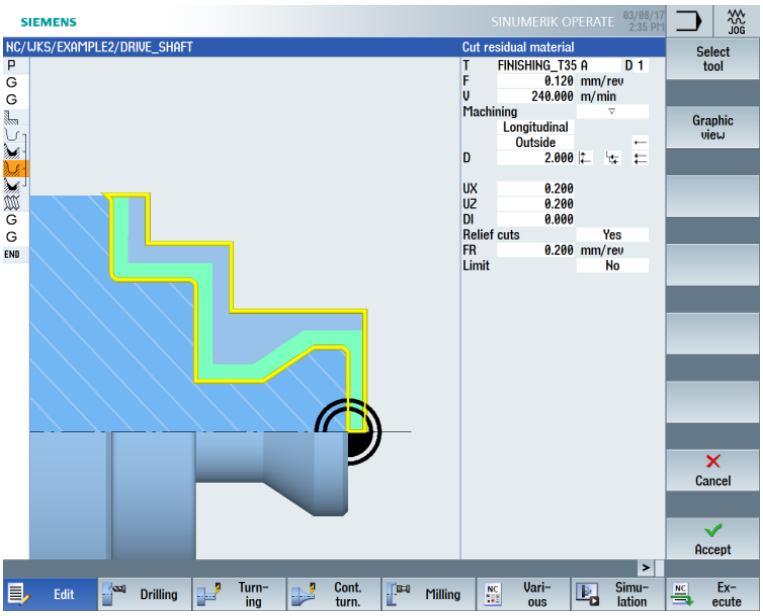
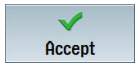


Figure 7-24 Contour: Cutting the residual material



Accept the entered values.

Select the "Simulation" softkey.

Expand the menu.

Activate the display of traversing paths.

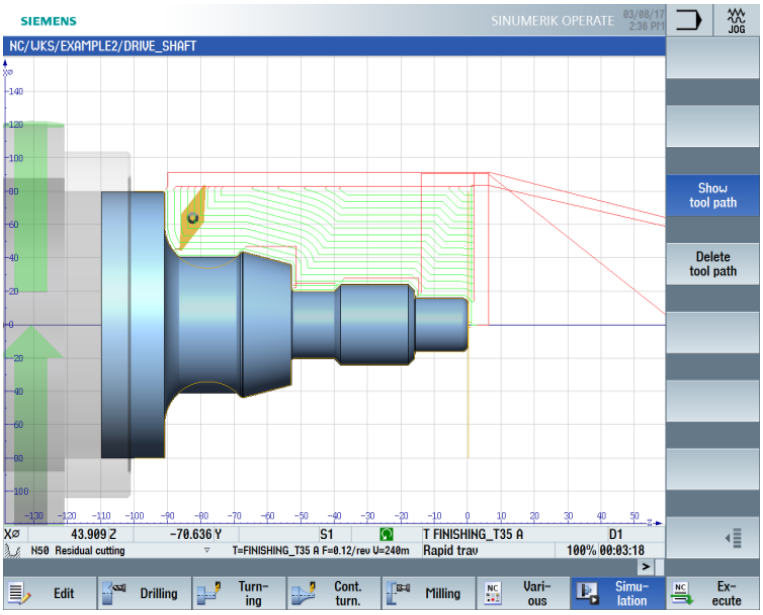


Figure 7-25 Cutting residual material – Simulation side view

Select the "Cont. turning" softkey.

Stock removal

Select the "Stock removal" softkey.

Tool

Open the tool list and select the "FINISHING_T35 A" tool.

OK

Apply the tool to the program.

Enter the following values for finishing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.12		
S	280 rpm	X	
Machining	Finishing Longitudinal Outside	X X X	
Allowance	No	X	
Relief cuts	Yes	X	
Limit	No	X	

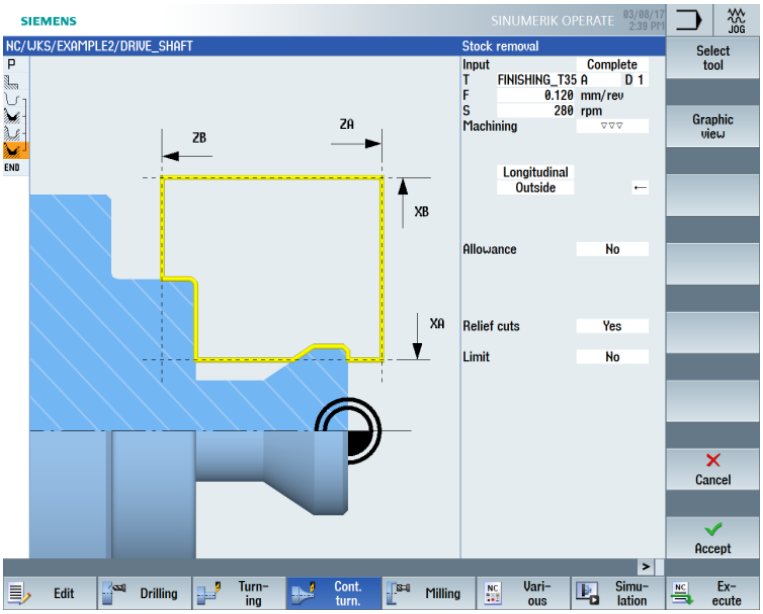


Figure 7-26 Finishing the contour

Accept the entered values. Once applied, the machining plan looks like this:

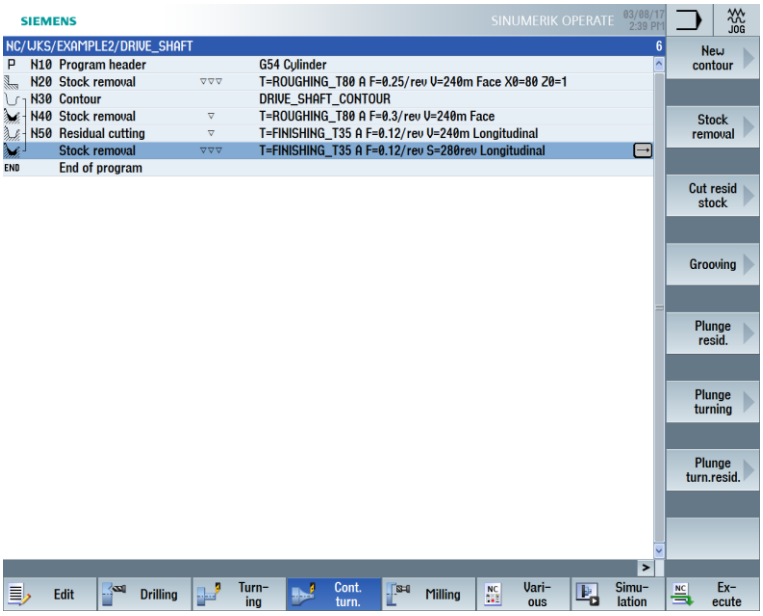
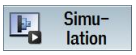


Figure 7-27 Machining plan



Start the "Simulation".

Select the "Details" softkey. Here, you can zoom into or out from the view.

The "Zoom +" softkey enlarges the view.

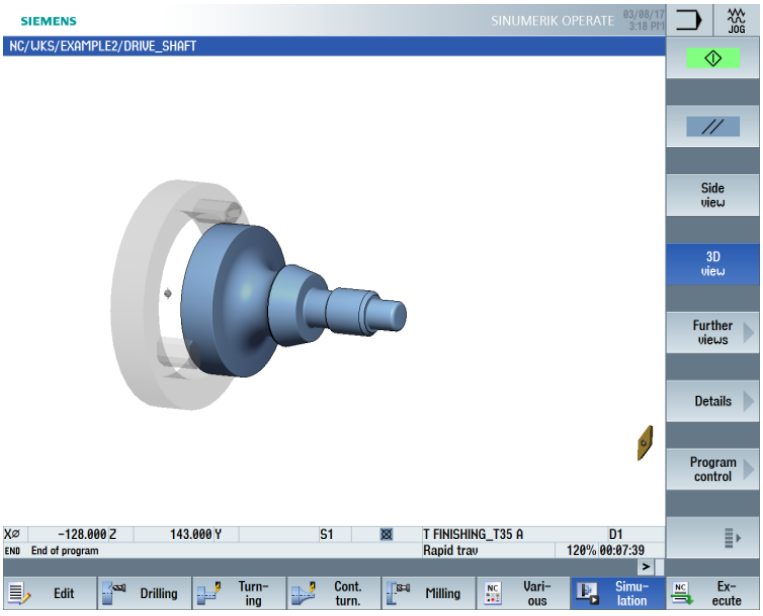


Figure 7-28 Simulation 3D view – Details

8.4 Thread

Operating sequences

Select the "Thread" softkey.

Open the tool list and select the "THREADING_T1.5" drill.

Apply the tool to the program.

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

Field	Value	Selection via toggle key	Notes
P	1.5 mm/rev	X	
G	0		
S	800 rpm	X	
Machining	Roughing / Finishing Linear External thread	X X X	
X0	24		
Z0	-16		
Z1	-40 abs	X	
LW	2		
LR	1		
H1	0.92		
αP	29 Infeed with alternating flank	X X	
ND	8		
U	0.1		
NN	0		
VR	2		
Multiple threads	No	X	
α0	0		

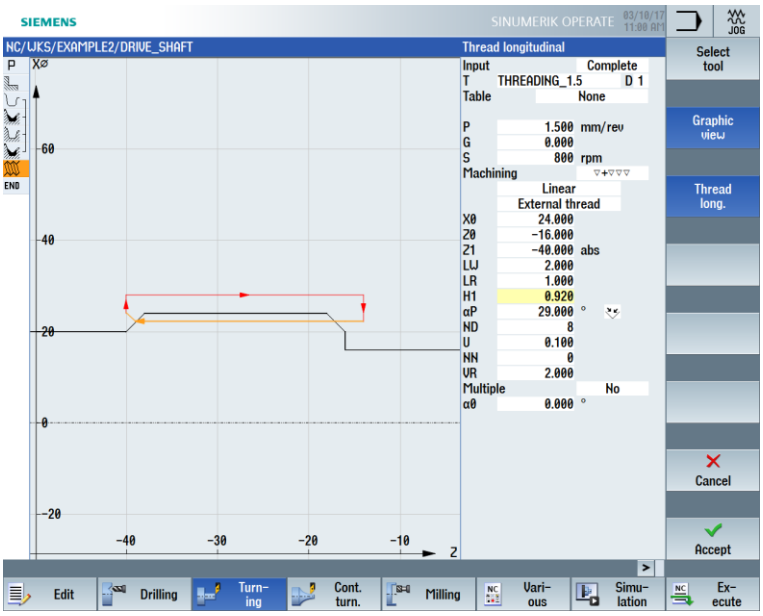
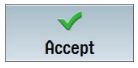


Figure 7-29 Producing the thread



Accept the entered values.

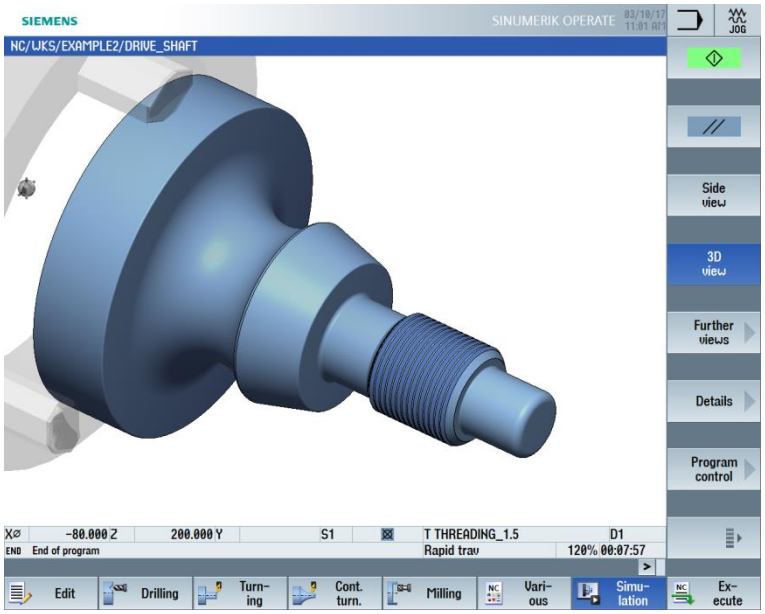


Figure 7-30 Simulation 3D view – Details

9. Example 3: Guide shaft

9.1 Overview

Learning objectives

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

- Create a random blank shape
- Remove the difference in material between the blank and the machined part
- Drill on the front face
- Mill on the front face

Task

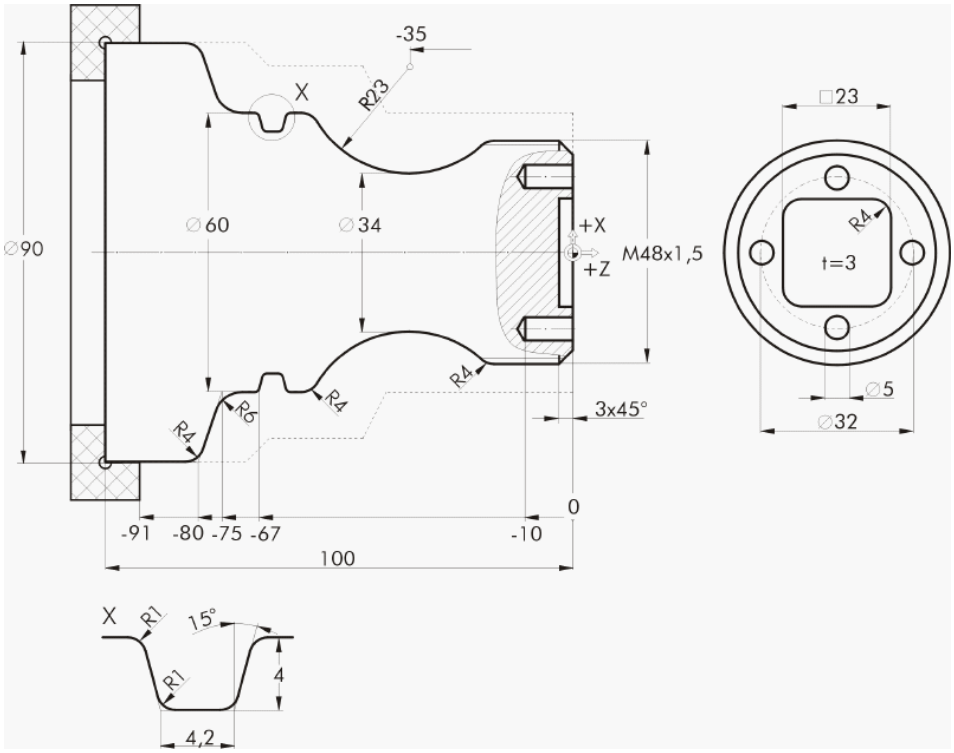


Figure 8-1 Workshop drawing – Example 3

Preparations

Perform the following steps on your own:

1. Create a new workpiece with the name "EXAMPLE3".
2. Create a new sequential program with the name "GUIDE_SHAFT".
3. Fill in the program header (see figure below).

Note:

Despite the random blank shape, select the Cylinder blank shape here. ShopTurn ignores this input and orients itself to the random blank shape.

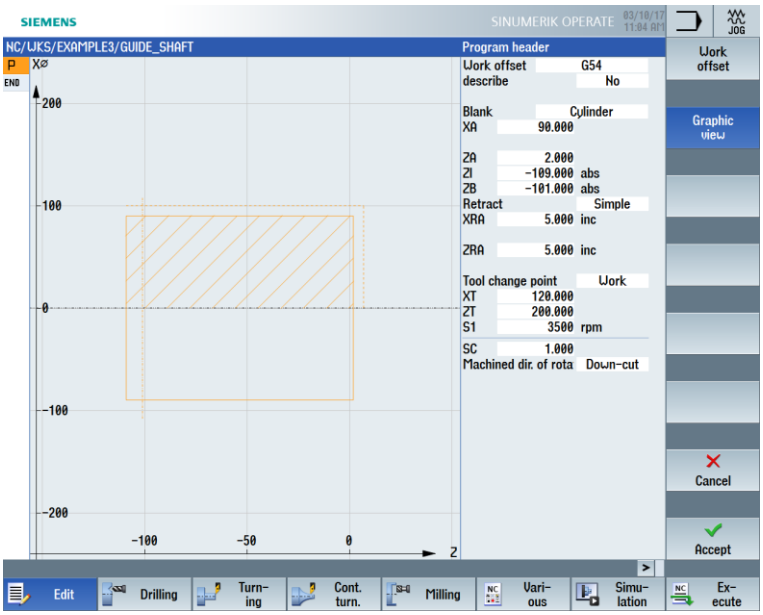


Figure 8-2 Creating the program header

9.2 Face turning

Operating sequences

Follow the steps below to create a new program and face the blank to Z0:

Select the "Turning" softkey.

Select the "Stock removal" softkey.

Select the "ROUGHING_T80 A" tool.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.25		
V	240 m/min	X	
Machining	Finishing	X	
Pos.	See figure below	X	
Machining direction	Face	X	
X0	60		Because the random blank shape has a diameter of 60 mm, you also need to set dimension X0 to 60 in this machining step.
Z0	2		
X1	-1.6 abs	X	
Z1	0.0 abs	X	
D	1.5		
UX	0.0		
UZ	0.2		

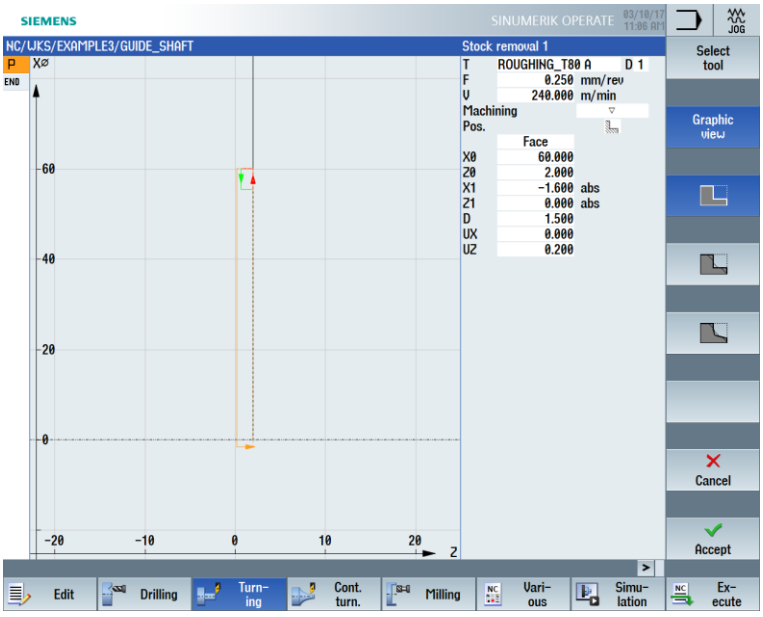


Figure 8-3 Facing the workpiece

Accept the entered values.

Start the "Simulation" to check the machining step.

You can activate the display of traversing paths via the expanded menu.

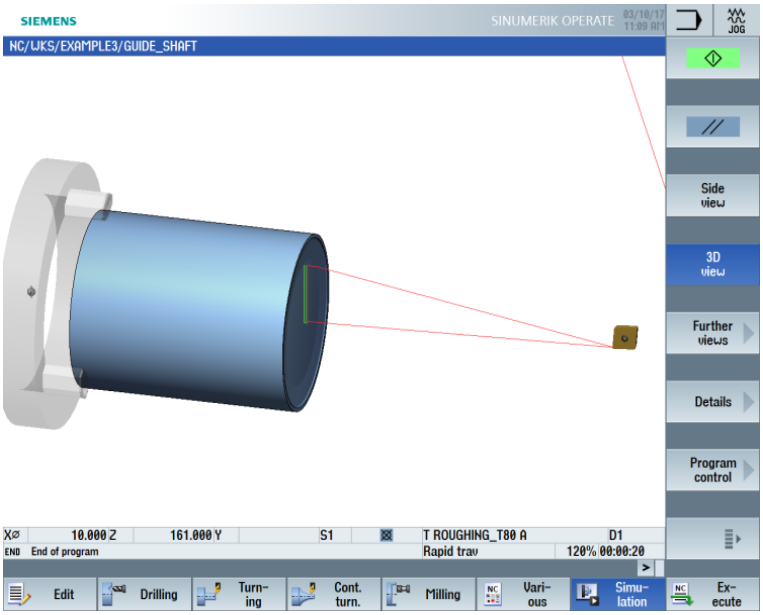
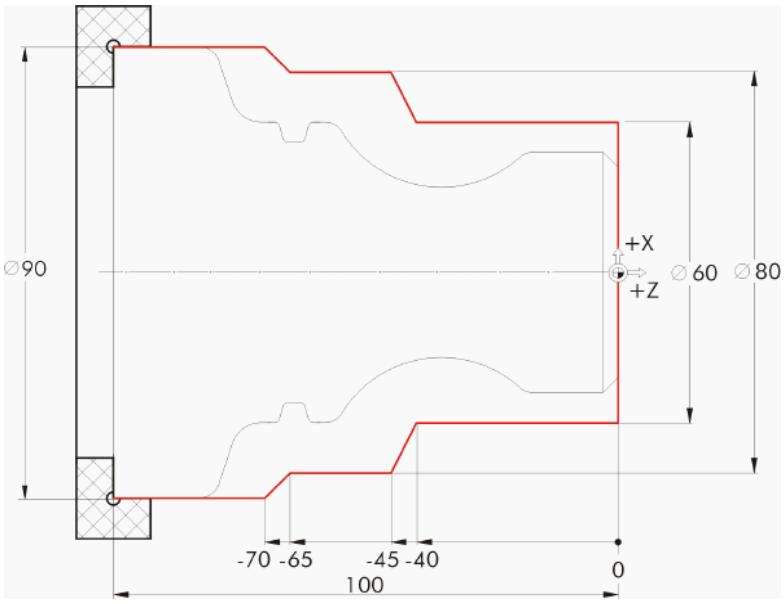


Figure 8-4 Face turning simulation

9.3 Creating a random blank shape contour

Operating sequences

Enter the following blank contour on your own:



New contour

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

Figure 8-5 Creating the contour

In the contour calculator, create the blank contour with the starting point at X0/Z0 (see figure below).

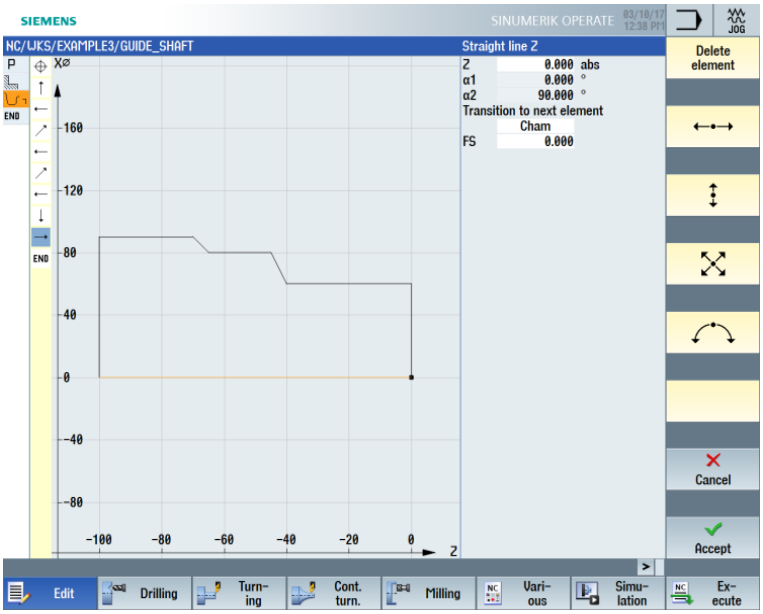


Figure 8-6 Random blank shape contour

Note:

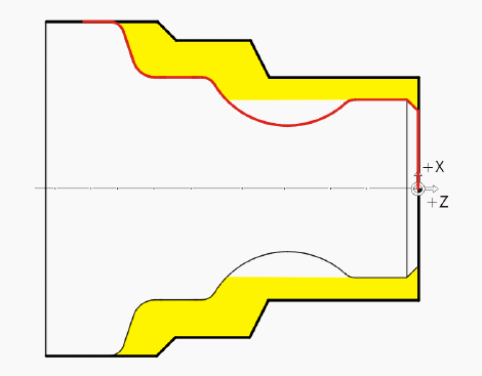
Close contour

The contour must be closed.

9.4 Creating the finished part contour and stock removal

Operating sequences

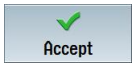
Follow the steps below to enter the finished part contour:



Select the "Cont. turning" softkey.

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

Figure 8-7 Creating the contour



Accept your input.

Since the blank was planned at Z0 in the first machining step, you can accept the starting point X0/Z0 directly (see figure below).

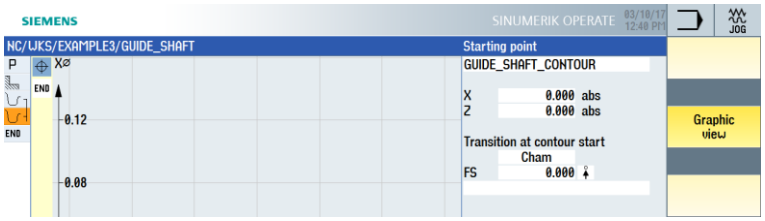


Figure 8-8 Specifying the contour starting point

Accept your input.

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

Field	Value	Selection via toggle key	Notes
X	48 abs	X	
Transition to next element	Cham	X	
R	3		

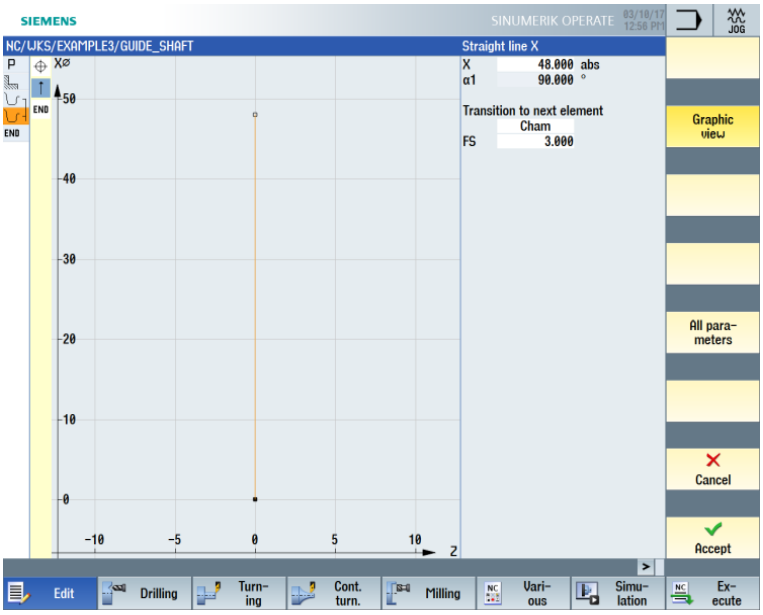


Figure 8-9 Entering the vertical straight-line segment for the contour

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
Z			 The end point of the horizontal straight-line segment is unknown. Enter only the transition to the next element with R4. The end point of the straight-line segment is calculated automatically from the subsequent constructions of the contour.
Transition to next element	Radius	X	
R	4		

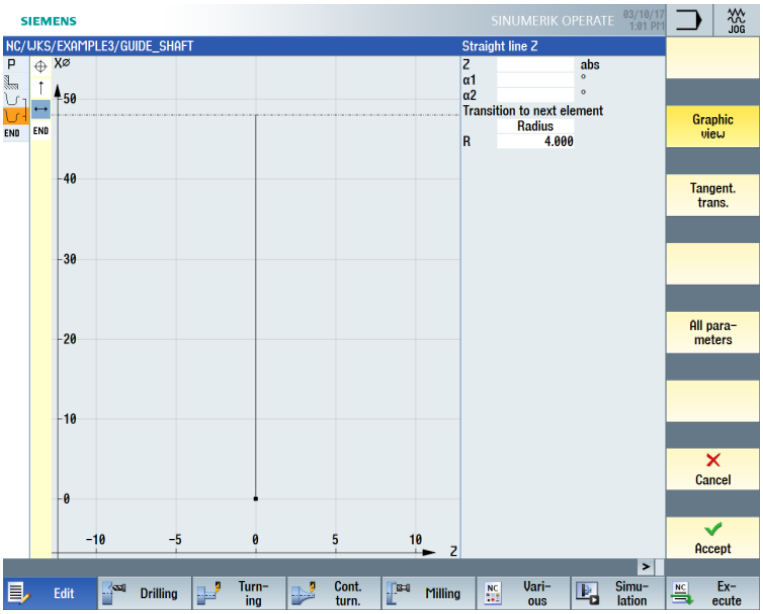
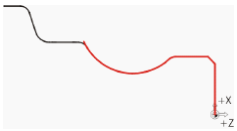


Figure 8-10 Entering the horizontal straight-line segment for the contour

Accept the entered values.

Enter the following values for the next segment in the screen form:

Field	Value	Selection via toggle key	Notes
Direction of rotation	right	X	
R	23		
X	60 abs	X	
Z			
I	80 abs	X	If several solutions are possible when contour data is entered (e.g. in the case of circular arcs here), you can select these via the "Dialog select" softkey.

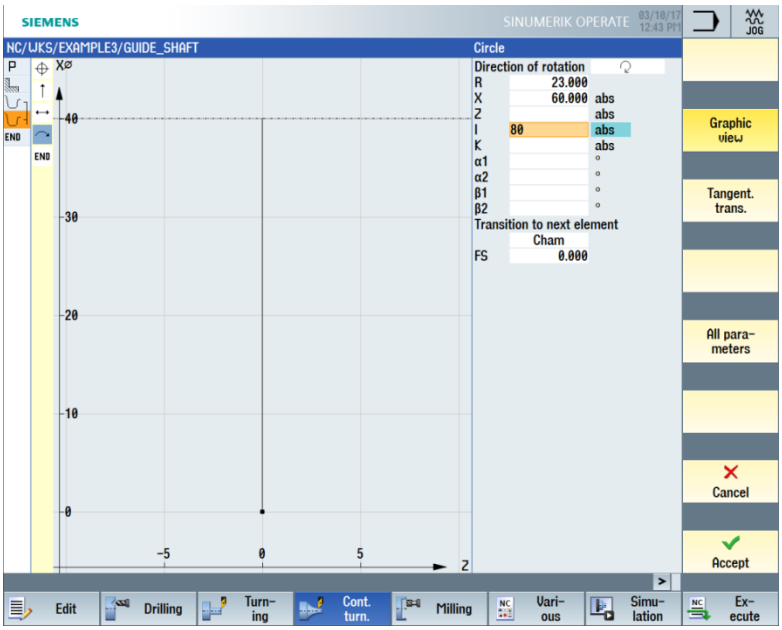


Figure 8-11 Entering the arc for the contour

Dialog select

Select the suggested solution corresponding to the figure below:

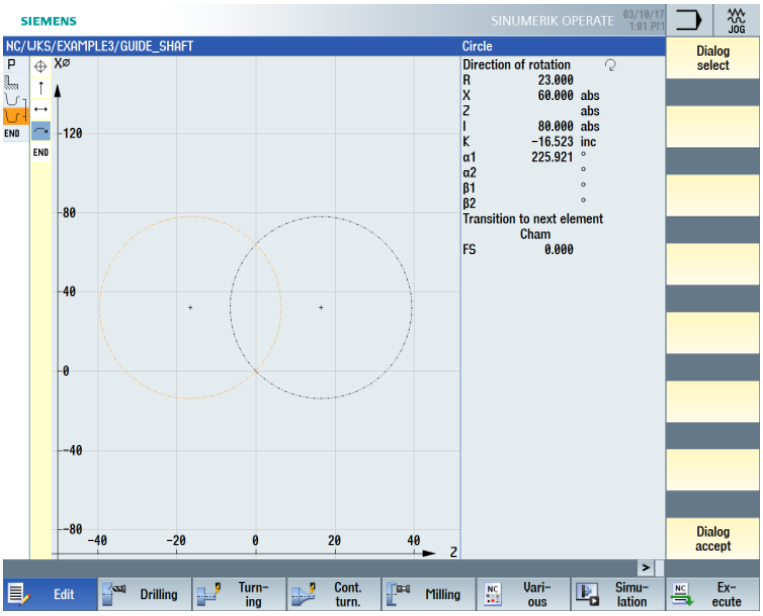


Figure 8-12 Selecting an arc for the contour

Dialog accept

Once you have selected the desired construction, accept it.

Dialog select

Select the suggested solution corresponding to the figure below:

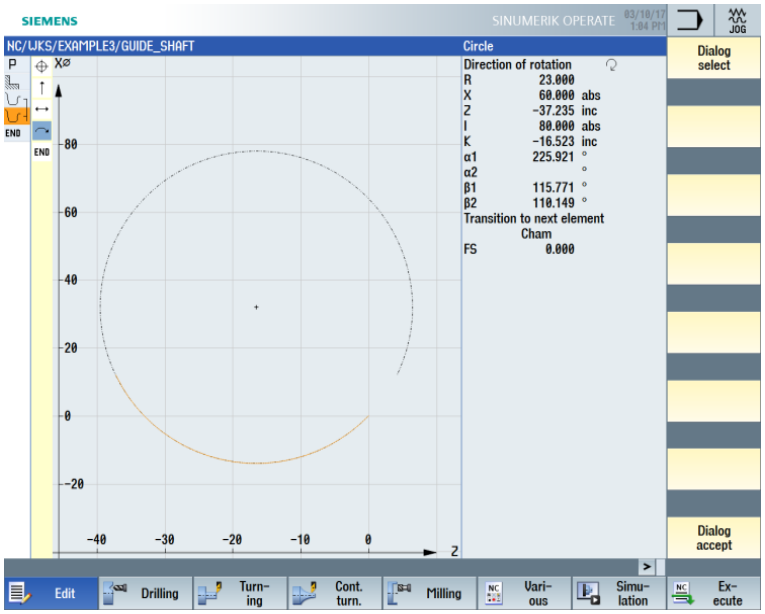


Figure 8-13 Selecting an arc for the contour

Dialog accept

Once you have selected the desired construction, accept it.

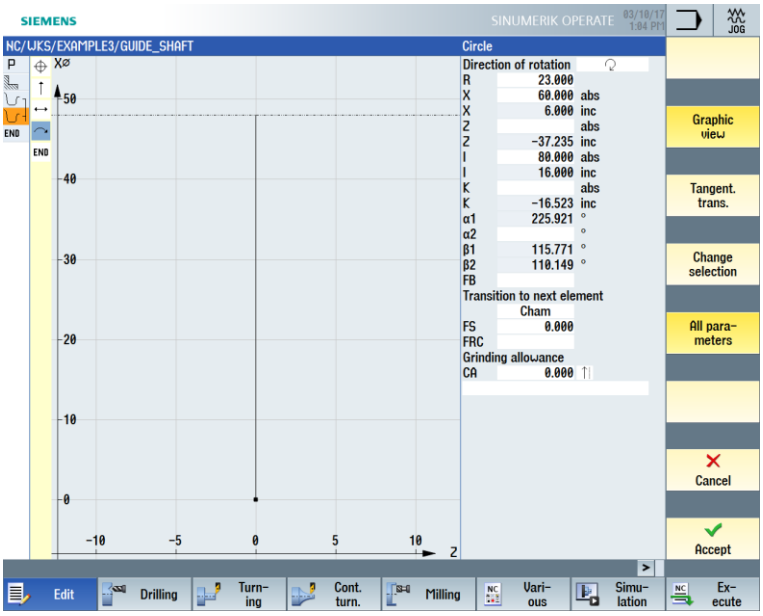


Figure 8-14 Applying the selected arc for the contour

Follow the steps below to complete the arc:

1. Enter center point K-35 (absolute dimension).

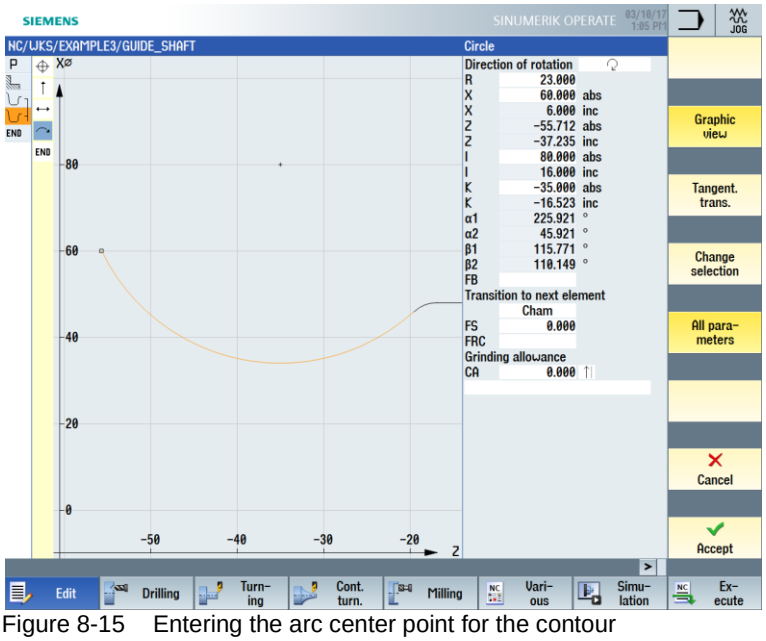


Figure 8-15 Entering the arc center point for the contour

2. Enter the transition to the next element with R4.

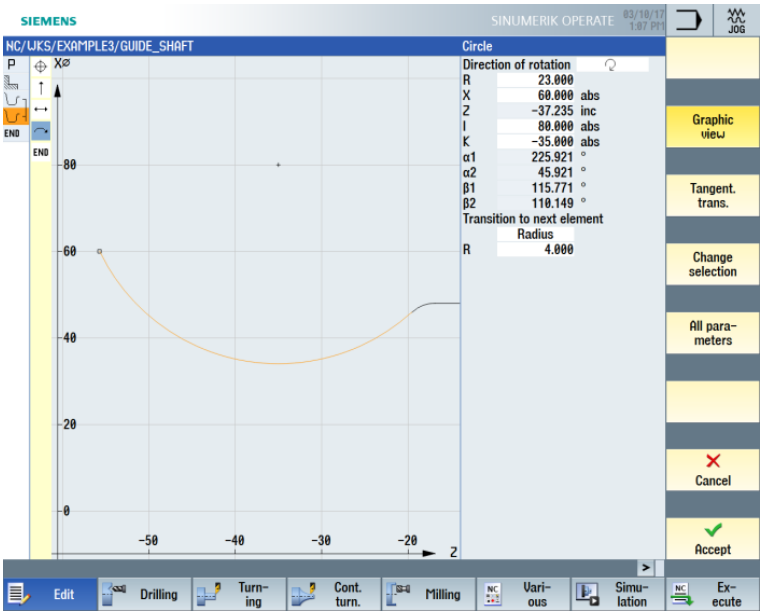
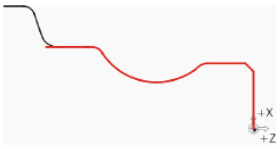


Figure 8-16 Entering the arc radius for the contour

You could use the contour data and the calculated selection options to construct the arc and the straight-line segment (with unknown end point).

Accept the contour segment.

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

Field	Value	Selection via toggle key	Notes
Z	-75 abs	X	
Transition to next element	Radius	X	
R	6		

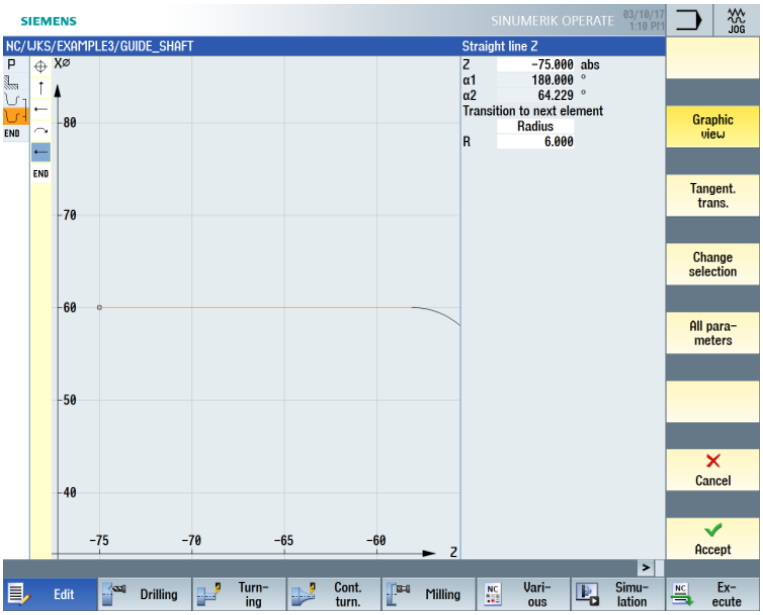
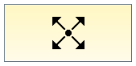
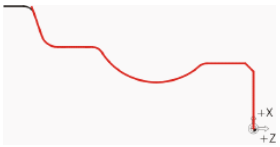


Figure 8-17 Entering the horizontal straight-line segment for the contour

Accept the entered values.



Enter the following values for the inclined straight-line segment in the screen form:

Field	Value	Selection via toggle key	Notes
X	90 abs	X	
Z	-80 abs	X	
Transition to next element	Radius	X	
R	4		

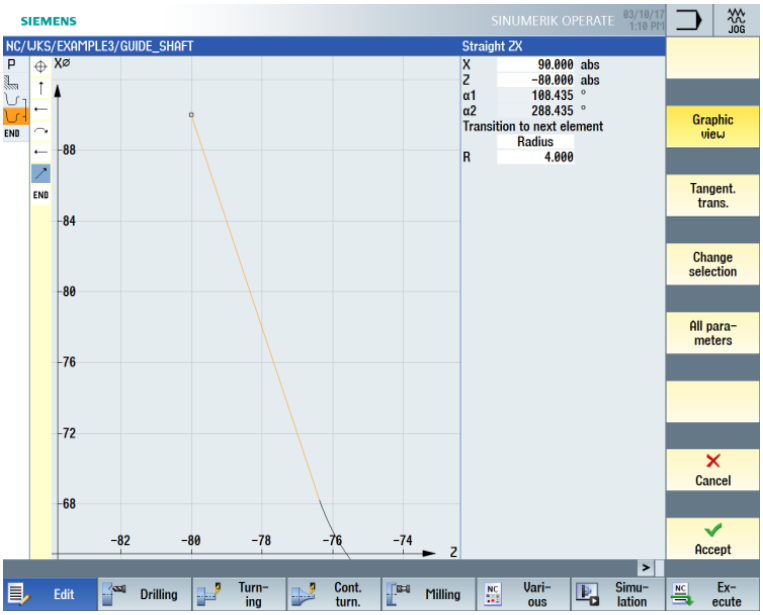
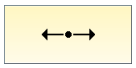
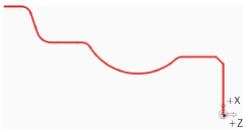


Figure 8-18 Entering the inclined straight-line segment for the contour

Accept the entered values.



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

Field	Value	Selection via toggle key	Notes
Z	-90 abs	X	
Transition to next element	Cham	X	
FS	0		To prevent damage to the chuck, stop the construction at Z-90.

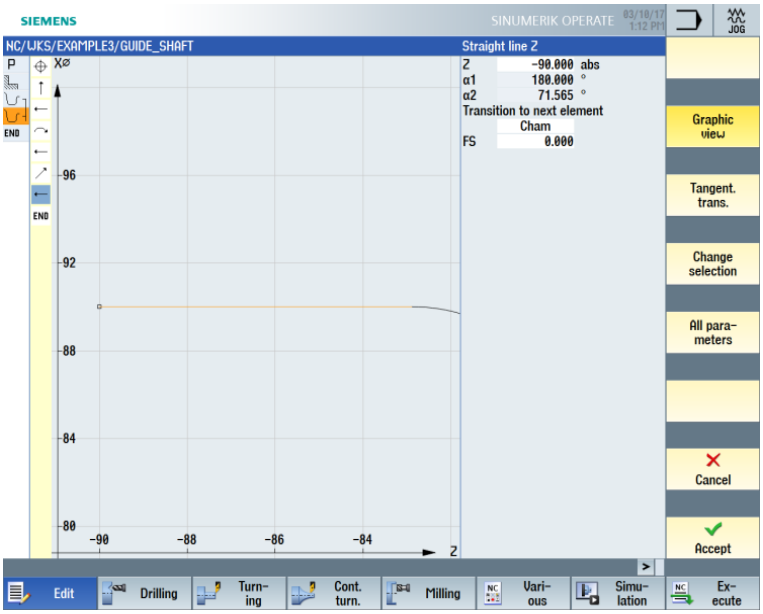


Figure 8-19 Entering the horizontal straight-line segment for the contour

Accept the entered values.

Accept the contour to apply it to your machining plan.

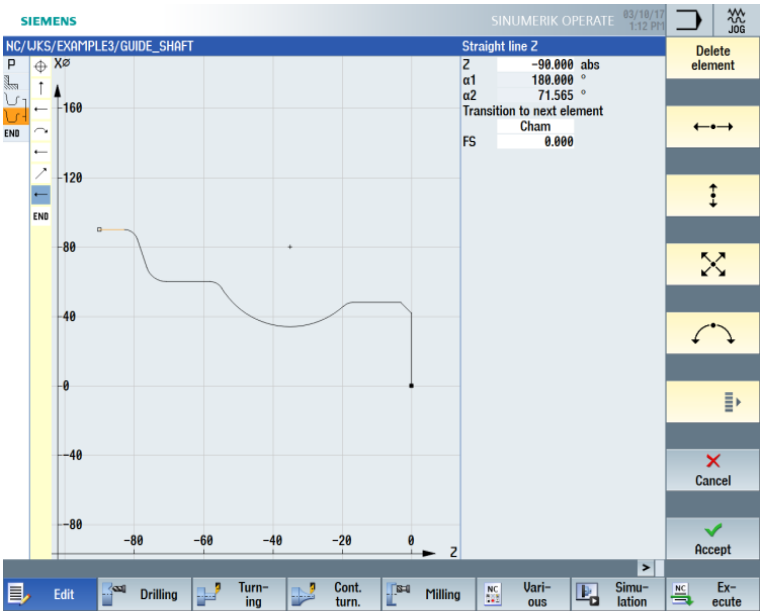


Figure 8-20 Applying the contour

Stock removal

You machine the contour in the next machining step.
Proceed as follows:

Select the "Stock removal" softkey.

Open the tool list and select the "ROUGHING_T80 A" tool.

Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.3		
V	260 m/min	X	
Machining	Roughing Longitudinal Outside	X X X	
D	2.5		
UX	0.2		
UZ	0.2		
DI	0.0		

Field	Value	Selection via toggle key	Notes
BL	Contour	X	Change the blank description to Contour.
Relief cuts	No	X	To ensure that the recess of radius 23 is not machined, you must toggle this setting to "No".
Limit	No	X	

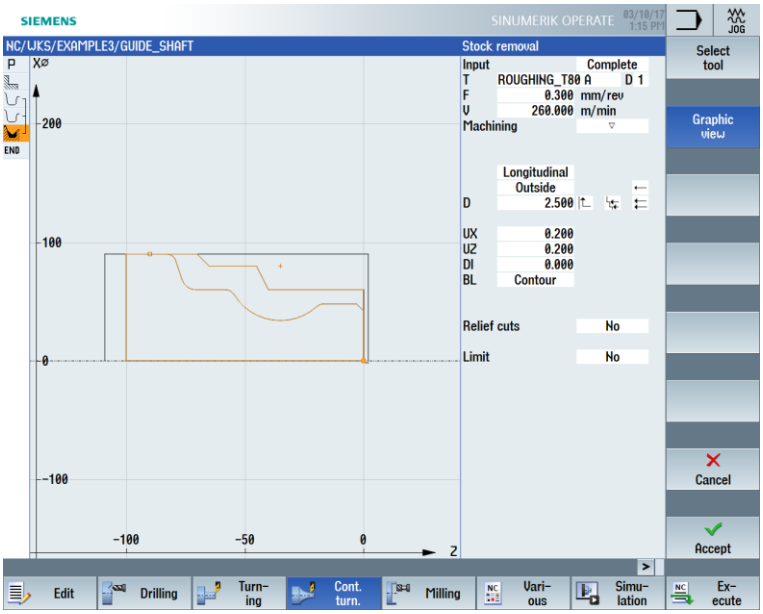
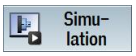


Figure 8-21 Removing stock from the contour

Accept the entered values. Once applied, the two contours and the machining step are linked.



Select the "Simulation" softkey.

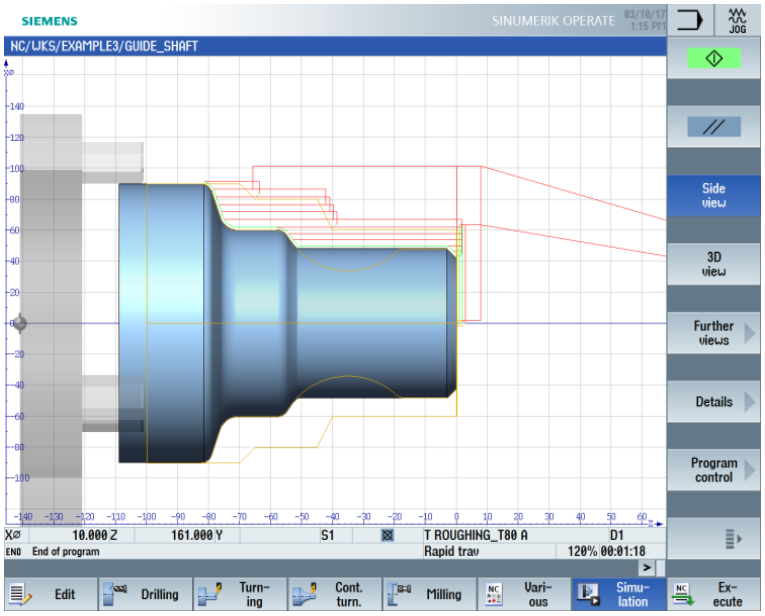


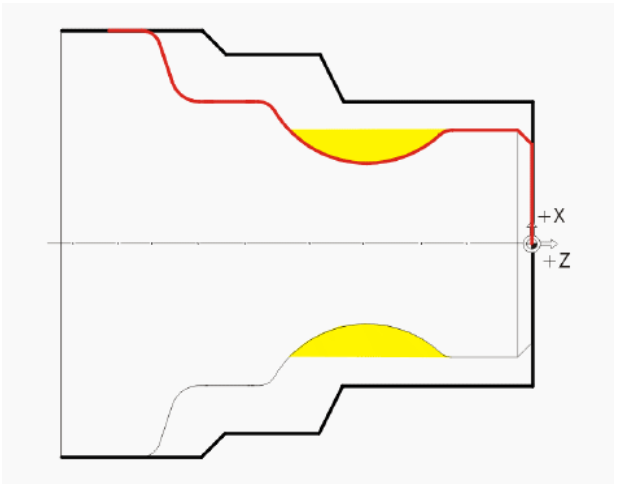
Figure 8-22 Contour stock removal simulation (with display of the traversing paths).

The traversing paths in the simulation clearly show how the previously constructed blank is taken into consideration.

9.5 Residual stock removal

Operating sequences

Follow the steps below to cut the residual material:



The following figure shows the machining plan up to and including the roughing machining:

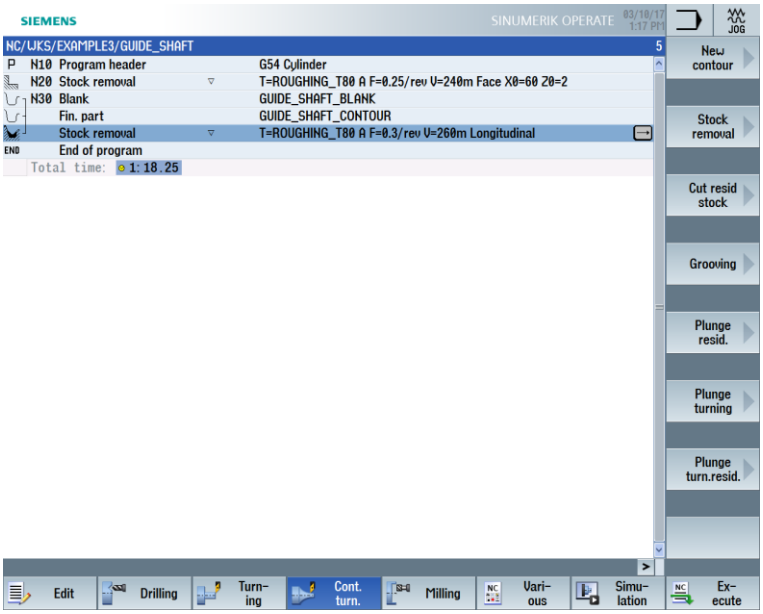


Figure 8-23 Machining plan including roughing machining

Select the "Cont. turning" softkey.

Select the "Cut resid stock" softkey.

Open the tool list and select the "BUTTON_TOOL_8" tool.

Apply the tool to the program.

Enter the following values for residual stock removal in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.25		
V	240 m/min	X	
Machining	Roughing Longitudinal Outside	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		

Field	Value	Selection via toggle key	Notes
Relief cuts	Yes	X	Toggle the setting for machining with relief cuts to "Yes".
FR	0.2		
Limit	No	X	

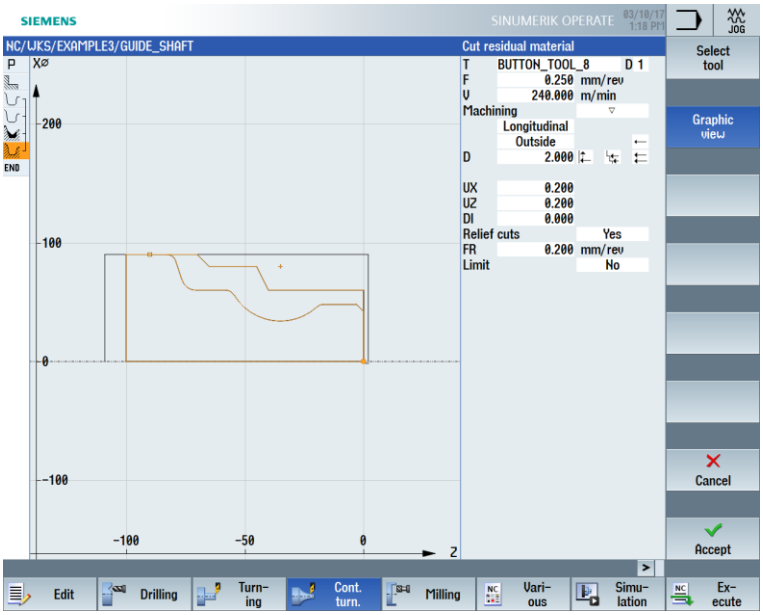


Figure 8-24 Cutting the residual material from the contour

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

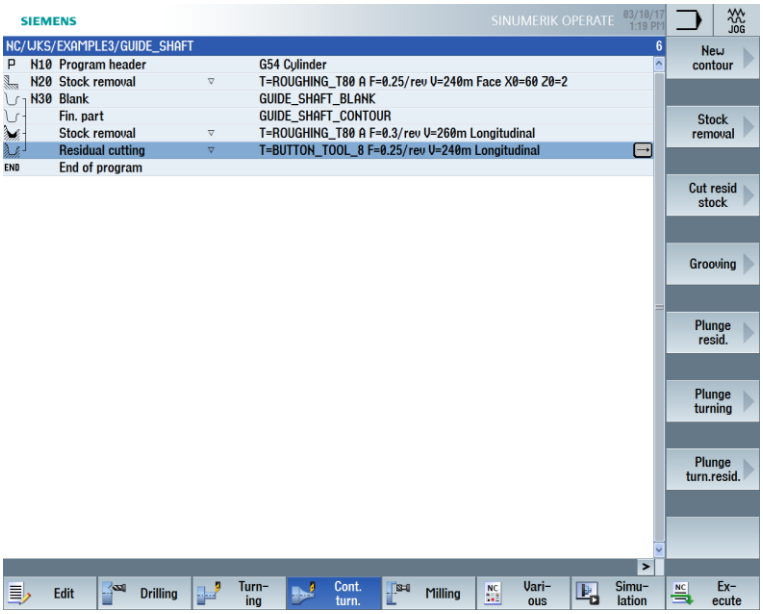


Figure 8-25 Machining plan with residual stock removal

Start the simulation.

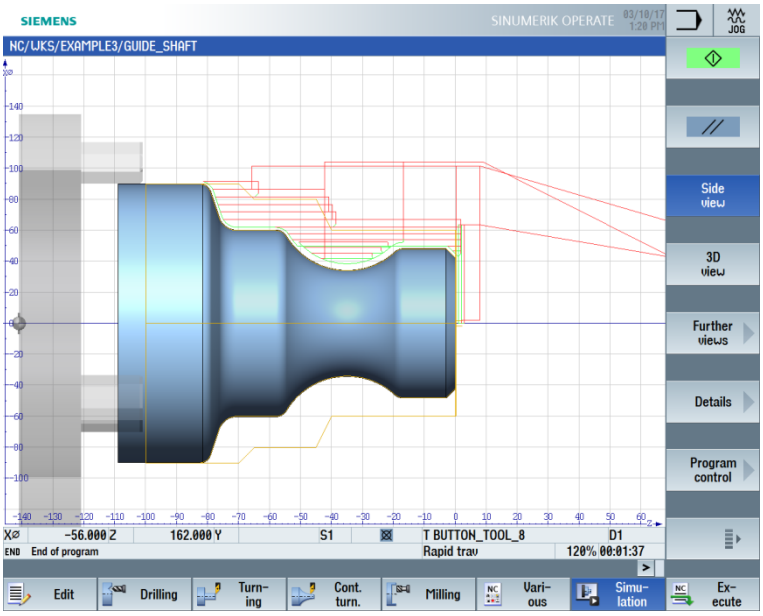


Figure 8-26 Residual stock removal simulation

After roughing the contour, it must then be finished.

Select the "Cont. turning" softkey.

Select the "Stock removal" softkey.

Open the tool list and select the "FINISHING_T35 A" tool.

Apply the tool to the program.

Enter the following values for finishing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.12		
S	280 m/min	X	
Machining	Finishing Longitudinal Outside	X X X	
Allowance	No	X	
Relief cuts	Yes	X	
Limit	No	X	

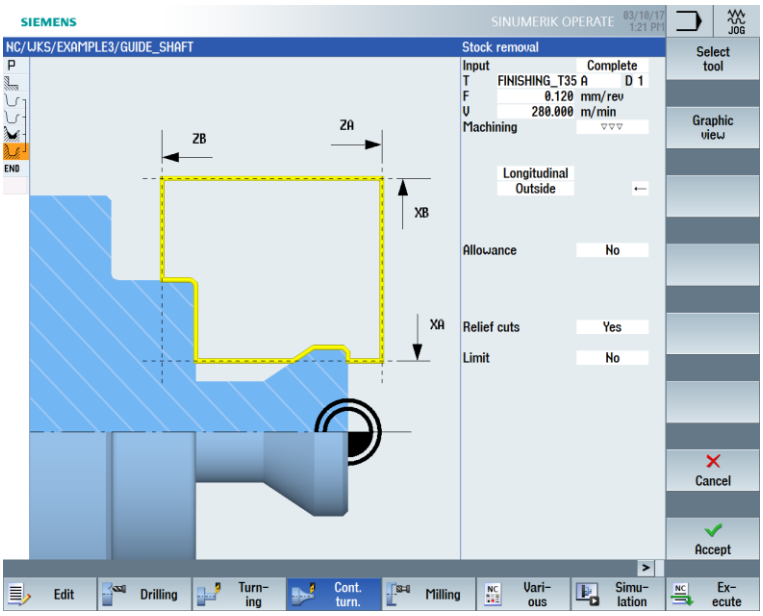


Figure 8-27 Finishing the contour

Accept the entered values.

Start the "Simulation".

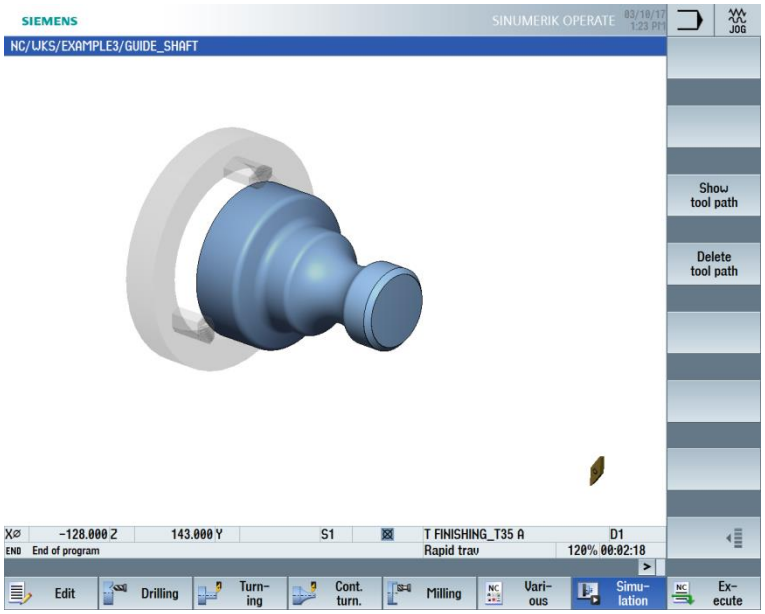
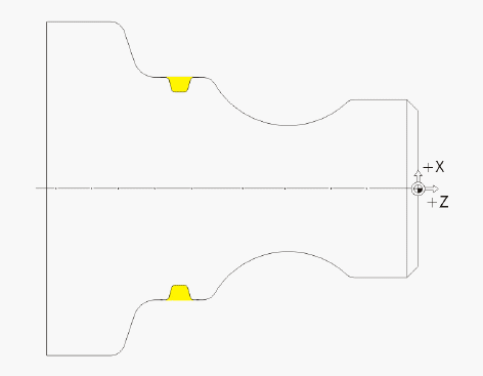


Figure 8-28 Finishing simulation – 3D view

9.6 Groove

Operating sequences

Follow the steps below to create the groove:



After residual stock removal, the list of machining steps looks like this:

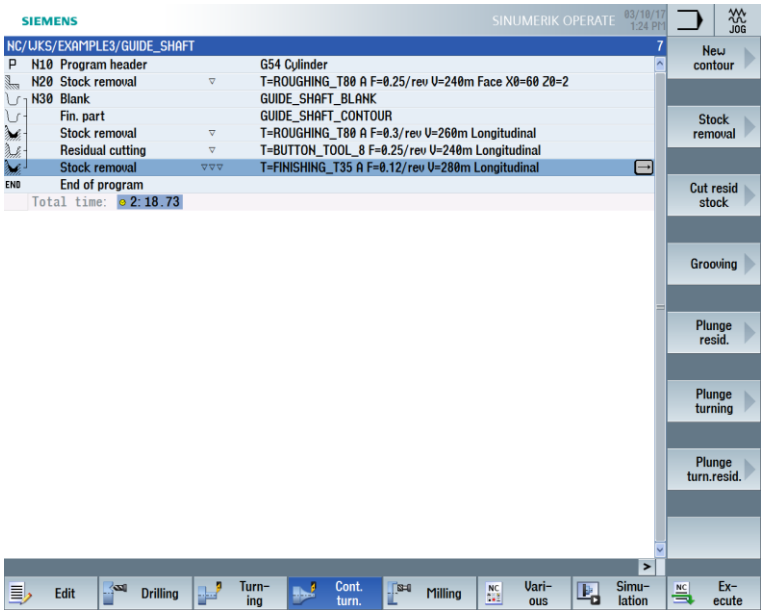


Figure 8-29 Machining plan after stock removal

Select the "Turning" softkey.

Select the "Groove" softkey.

Select the second offered groove shape (Groove 2).

Open the tool list and select the grooving tool "PLUNGE_CUTTER_3 A".

Apply the tool to the program.

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

Field	Value	Selection via toggle key	Notes
F	0.1 mm/rev		
V	150 m/min	X	
Machining	Roughing / Finishing	X	
Pos.	See figure below	X	
X0	60		Here, you enter the position and allowance of the groove.
Z0	-67		
B1	4.2	X (field)	
T1	4 inc	X	
α1	15		Here, you enter the flank angle and the rounding at the corners.
α2	15		
FS1	1	X (field)	
R2	1	X (field)	
R3	1	X (field)	
FS4	1	X (field)	
D	4		
U	0.2	X (field)	
N	1		

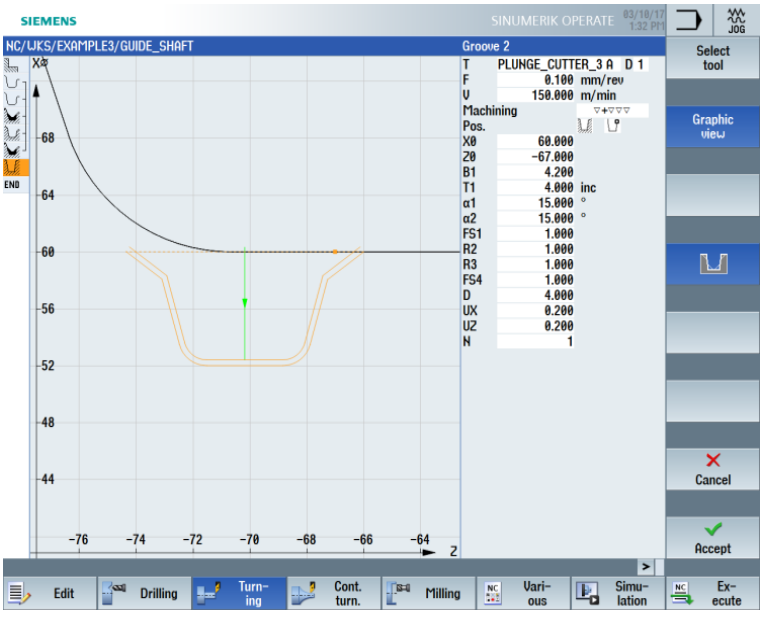


Figure 8-30 Creating the groove

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

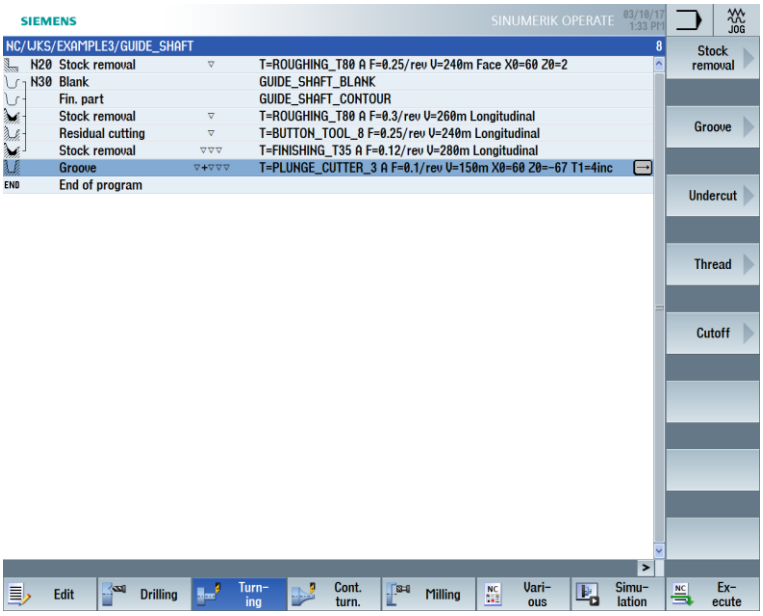
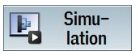


Figure 8-31 Machining plan including groove



Start the "Simulation". You can check subareas of the workpiece using the "Zoom" softkey.

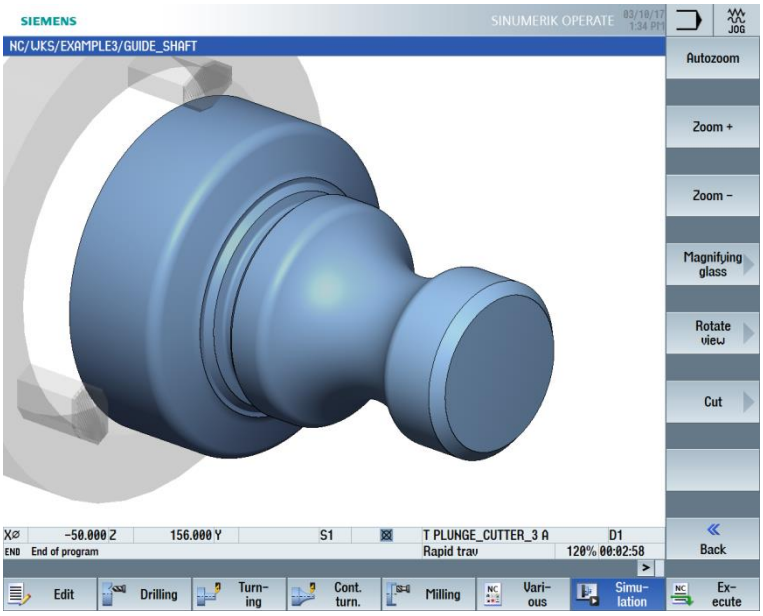
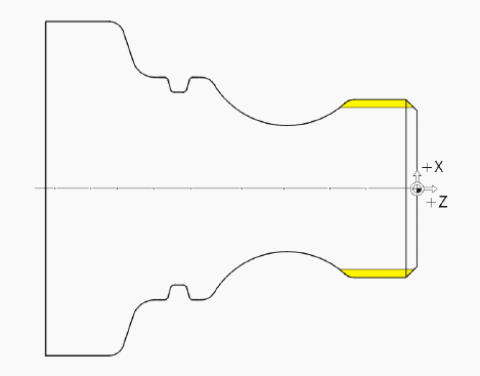


Figure 8-32 Simulation – 3D view (Zoom)

9.7 Thread

Operating sequences

Follow the steps below to create the thread:



Select the "Turning" softkey.

Select the "Thread" softkey.

Open the tool list and select the "THREADING_T1.5" thread cutting tool.

Apply the tool to the program.Enter the following values for the thread in the screen form:

Field	Value	Selection via toggle key	Notes
L	1.5 mm/rev	X	
D	0		
S	800 rpm	X	
Machining	Roughing Degressive External thread	X X X	The thread will be created with the Degressive setting. This setting causes the cut segmentation to be reduced for each cut, and so ensures that the cutting cross-section remains constant.
X0	48		
Z0	-3		
Z1	-23 abs	X	

Field	Value	Selection via toggle key	Notes
LW	4	X (field)	
LR	2		
H1	0.92		
αP	29 Infeed with alternating flank	X (field) X	
ND	8	X (field)	
U	0.1		
VR	2		
Multiple threads	No	X	
α0	0		

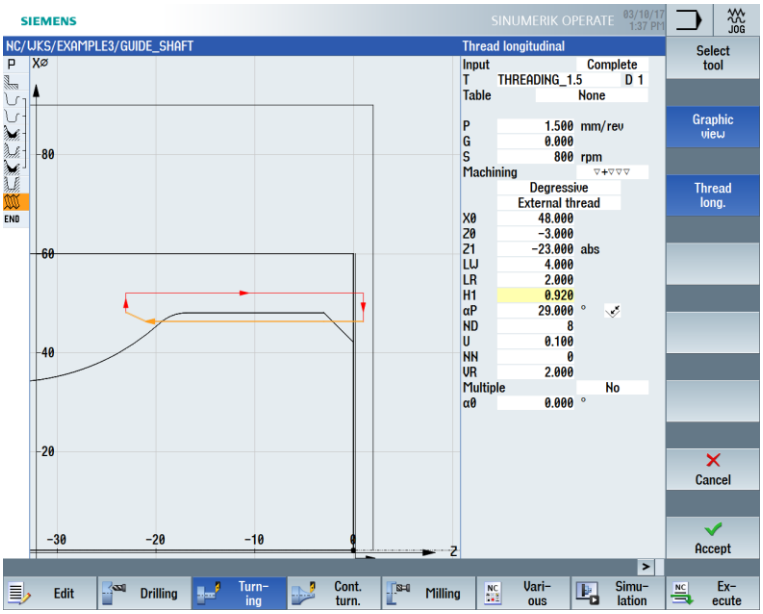


Figure 8-33 Producing the thread

Switch to the help screen when necessary.

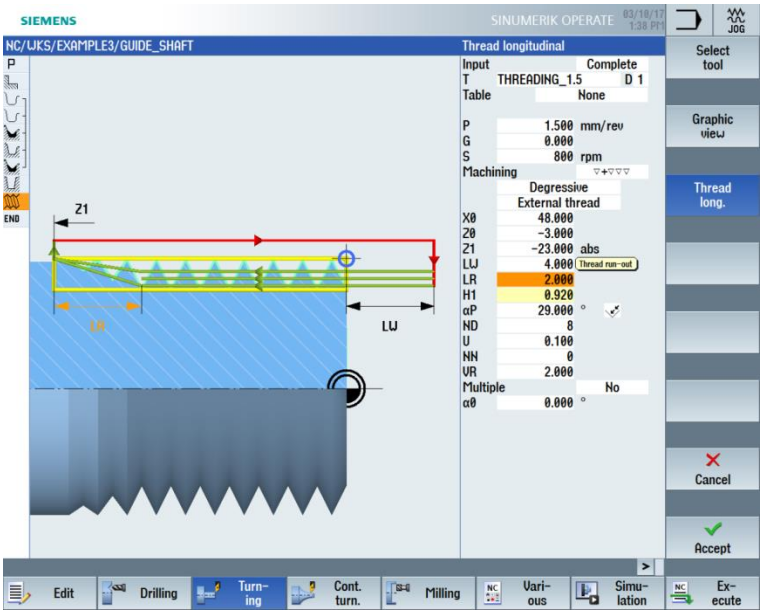


Figure 8-34 Help screen - thread run-out

Accept the entered values.

Start the "Simulation". You can check subareas of the workpiece using the Details softkey.

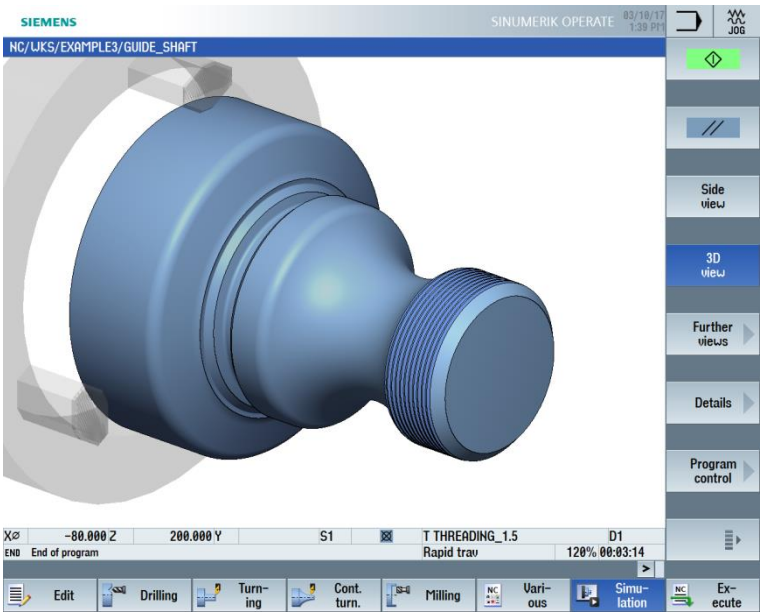
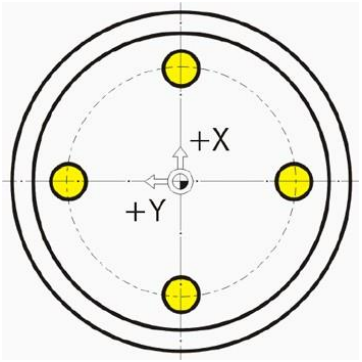


Figure 8-35 Simulation 3D view – Details

9.8 Drilling

Operating sequences

Follow the steps below to create drill holes on the front face (C axis or complete machining sequence):



After the thread machining has been added, the list of machining steps looks like this:

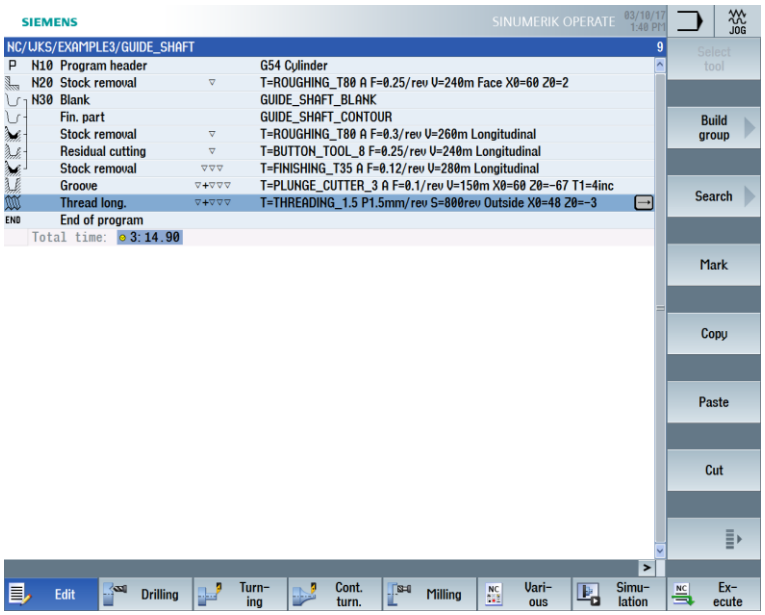
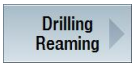


Figure 8-36 Machining plan after thread machining

Select the "Drill." (horizontal) softkey.



Select the "Drilling Reaming" softkey. The workpiece is drilled directly, i.e. without centering.

Select the "Drilling" (vertical) softkey.

Select tool

Open the tool list and select the "DRILL_5" twist drill.

OK

Apply the tool to the program.

Enter the following values for the drill hole in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.06 mm/rev	X	
V	140 m/min	X	
	Face Shank	X X	The depth reference is switched to Shank.
Z1	10 inc	X	The hole depth can be entered as 10 mm incremental or -10 mm absolute.
DT	0 s	X	Dwell time

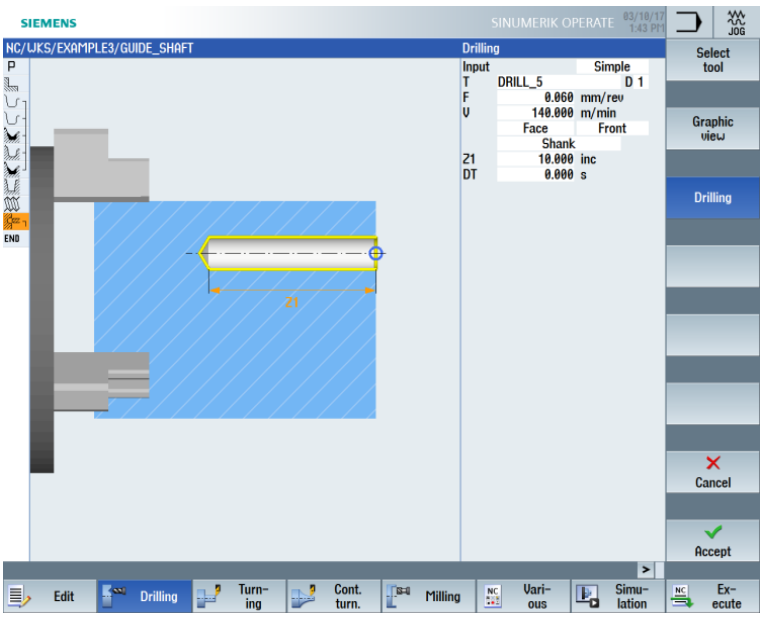


Figure 8-37 Drilling

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

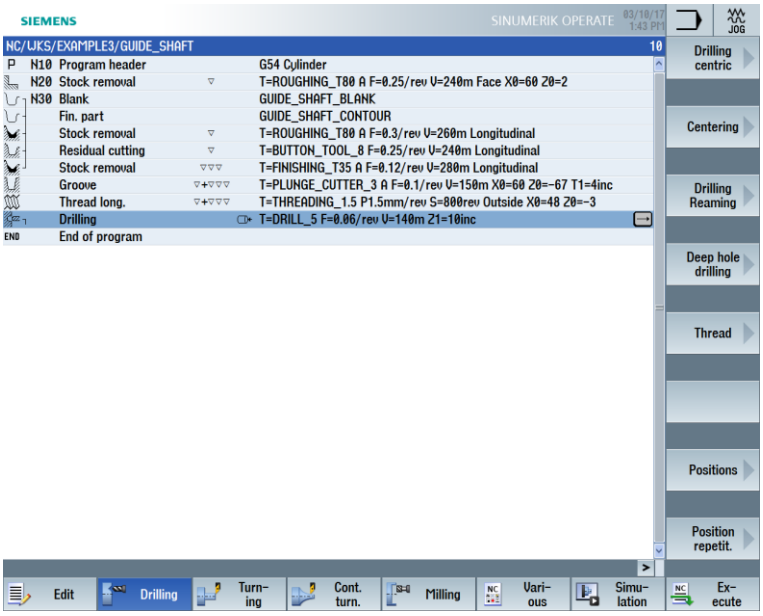


Figure 8-38 Machining plan following drilling

For the drilling machining step, you may notice an open link in the list of machining steps. This will be automatically linked to the drilling positions in the next step.

Select the "Positions" softkey.

For the purposes of the exercise, the four drill holes are entered as single positions. The simpler solution would be to use the position circle.

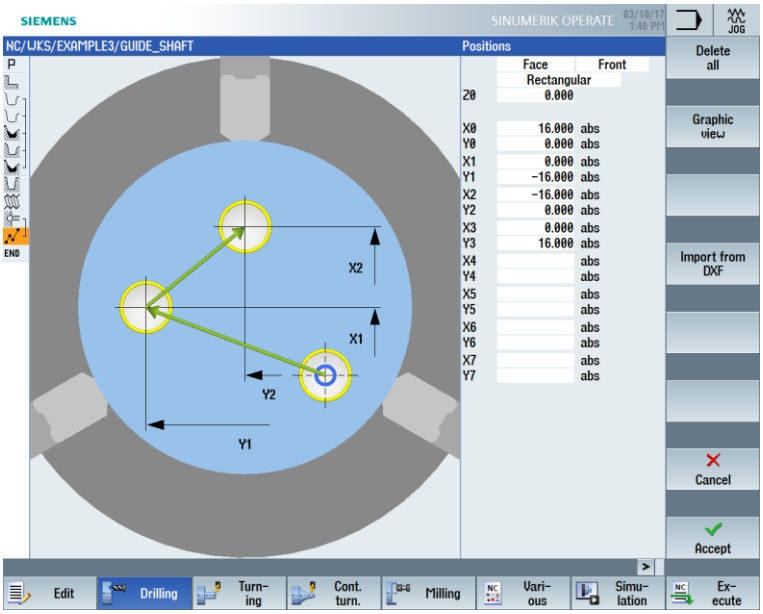


Figure 8-39 Entering the positions

Accept

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

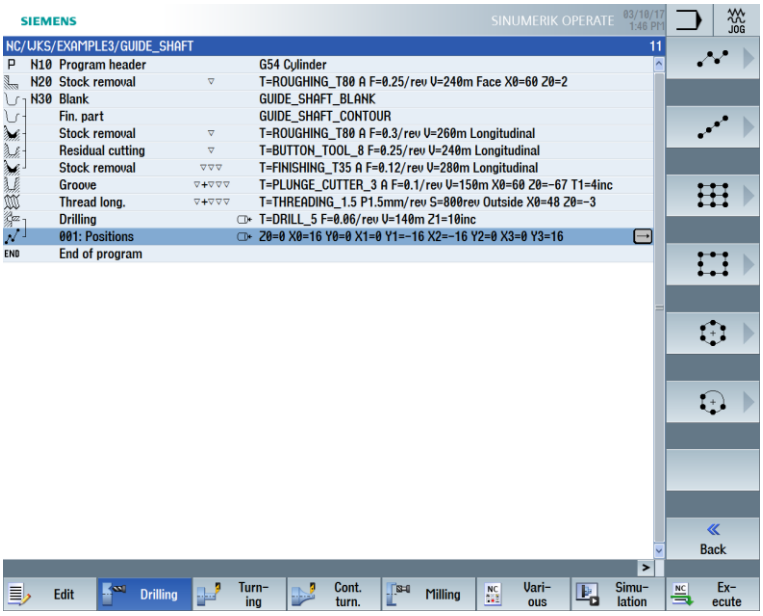


Figure 8-40 Machining plan after input of the position pattern

The drill holes are now linked to the drilling positions.

Simu-lation

Start the "Simulation".

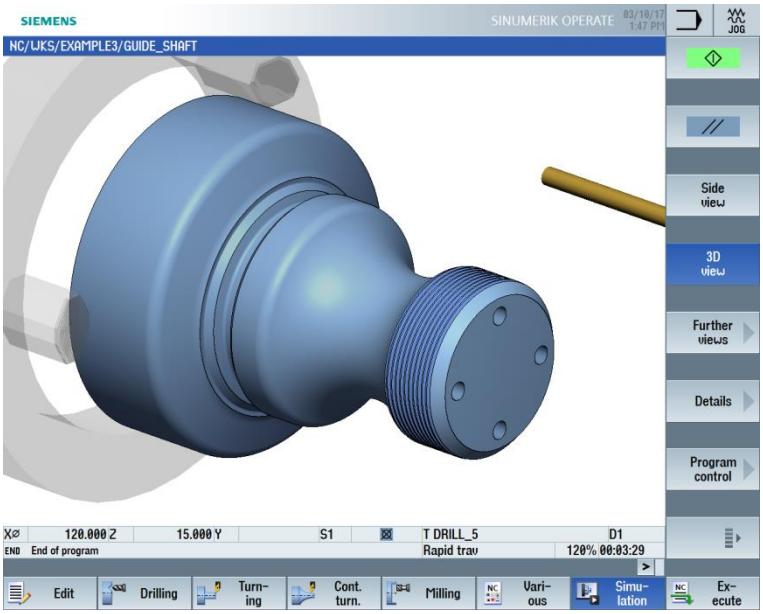


Figure 8-41 Simulation – 3D view

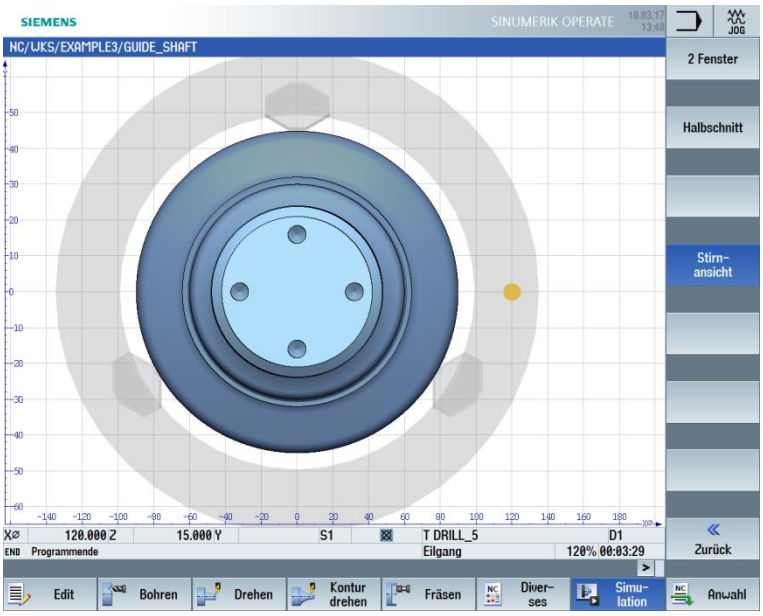
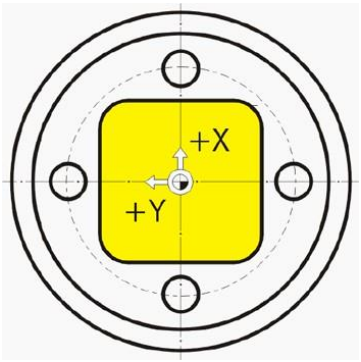


Figure 8-42 Simulation - Face view

9.9 Milling the rectangular pocket

Operating sequences

Follow the steps below to create the rectangular pocket on the front face (with the C axis and complete machining sequence).



Select the "Mill." softkey.

Select the "Pocket" softkey.

Select the "Rectang. pocket" softkey.

Open the tool list and select the "CUTTER_8" milling cutter.

Apply the tool to the program.

Enter the following values for the rectangular pocket in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.03 mm/tooth	X	
V	220 m/min	X	
	Face	X	
Machining	Roughing Single position	X X	
X0	0	X (field)	
Y0	0	X (field)	
Z0	0		
W	23		
L	23		
R	4		
α0	0		
Z1	3 inc	X	
DXY	75%	X	
DZ	1.5		
UXY	0		
UZ	0		
Insertion	Helical	X	Insertion see below
EP	1		
ER	7		

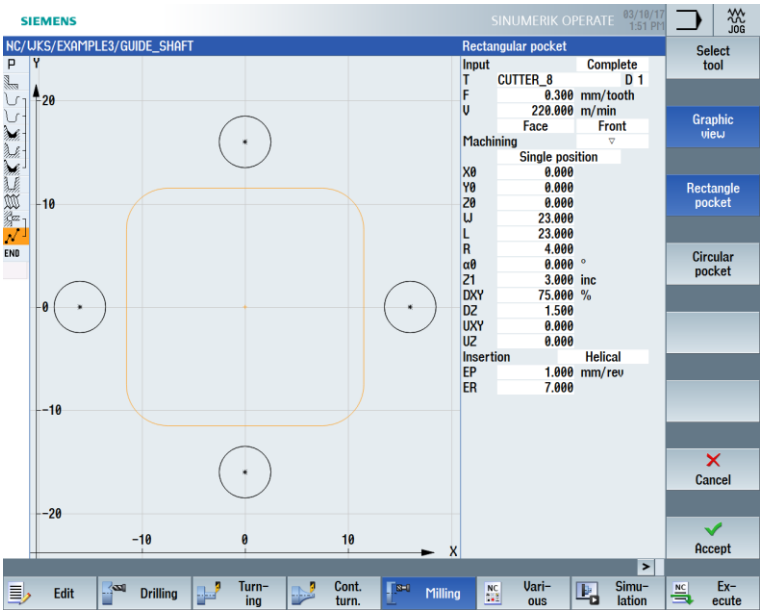
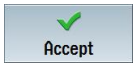


Figure 8-43 Producing a rectangular pocket



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

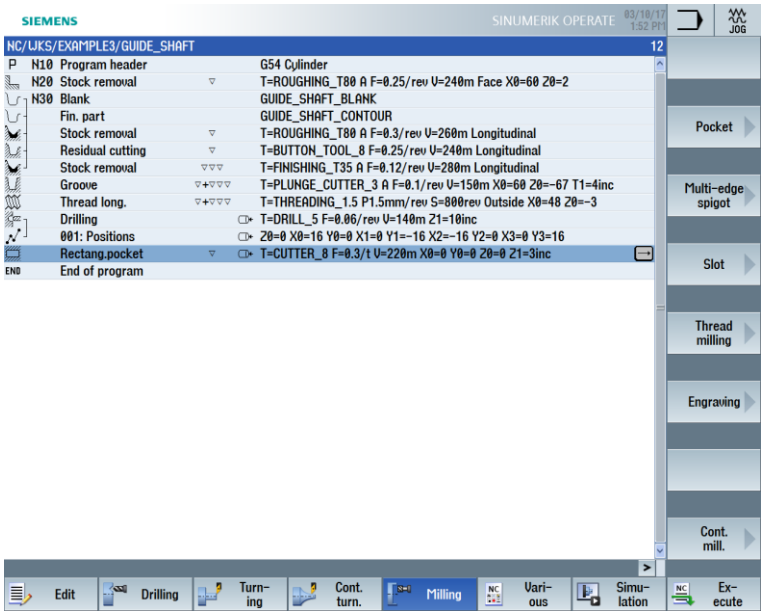


Figure 8-44 Machining plan after rectangular pocket

Start the "Simulation".

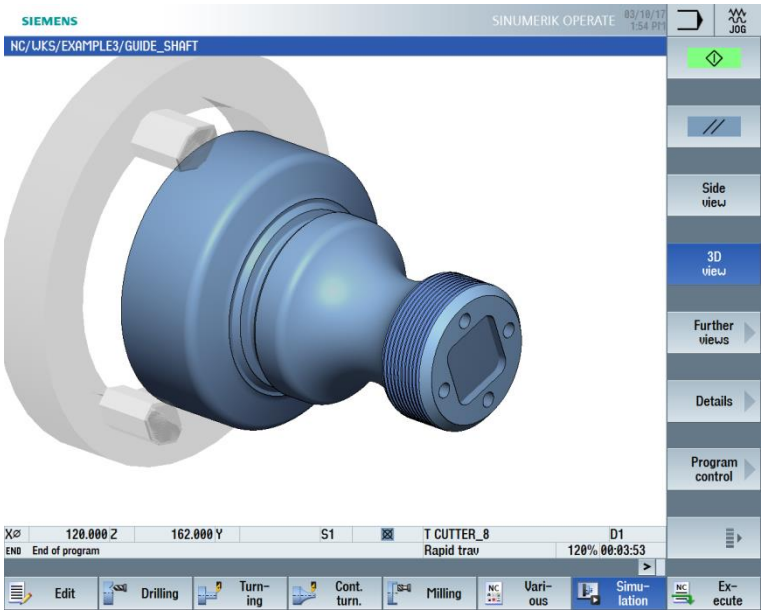
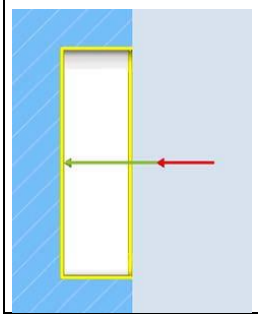
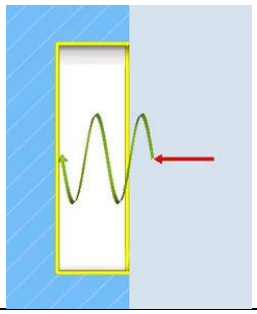
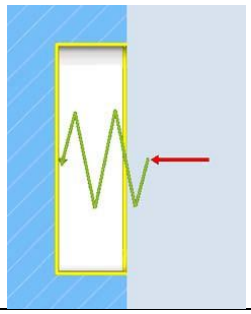


Figure 8-45 Simulation 3D view

Insertion

Perpendicular insertion	Helical insertion	Oscillating insertion
		

10. Example 4: Hollow shaft

10.1 Overview

Learning objectives

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

- Perform internal machining on workpieces
- Work with the machining step editor
- Create an undercut
- Create an asymmetrical groove

Task

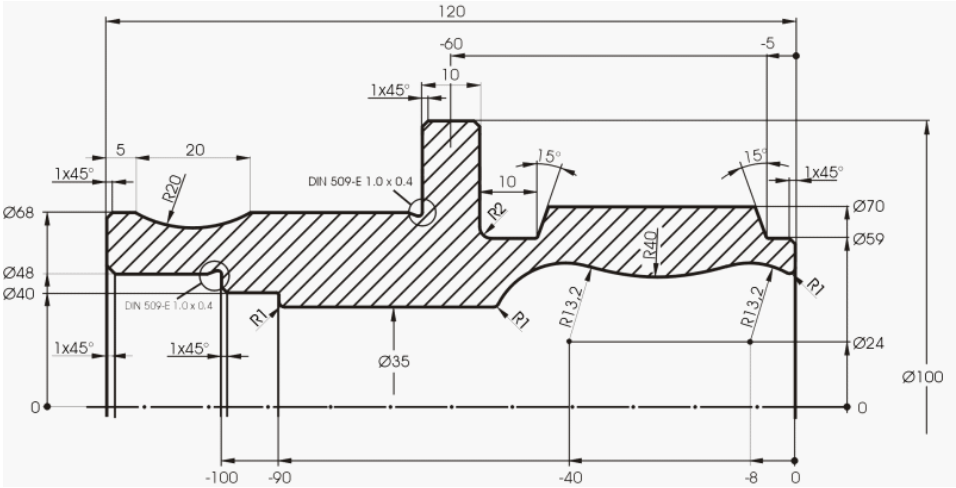


Figure 9-1 Workshop drawing – Example 4

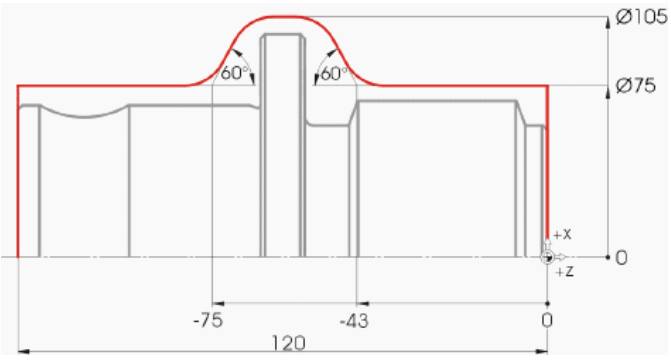


Figure 9-2 Blank contour

All non-dimensioned radii are R10

Note:

Side 1 is machined first on account of the better clamping possibility.

10.2 Creating the first workpiece side

Creating the machining plan

Since the workpiece is to be machined from two sides (it will be machined without counterspindle), two machining plans must be created.

Create the machining plan for the left side first ("HOLLOW_SHAFT_SIDE1")

Operating sequences

Create the program "HOLLOW_SHAFT_SIDE1" on your own.

Figure 9-3 Creating the ShopTurn program

Enter the following data in the program header (see figure).

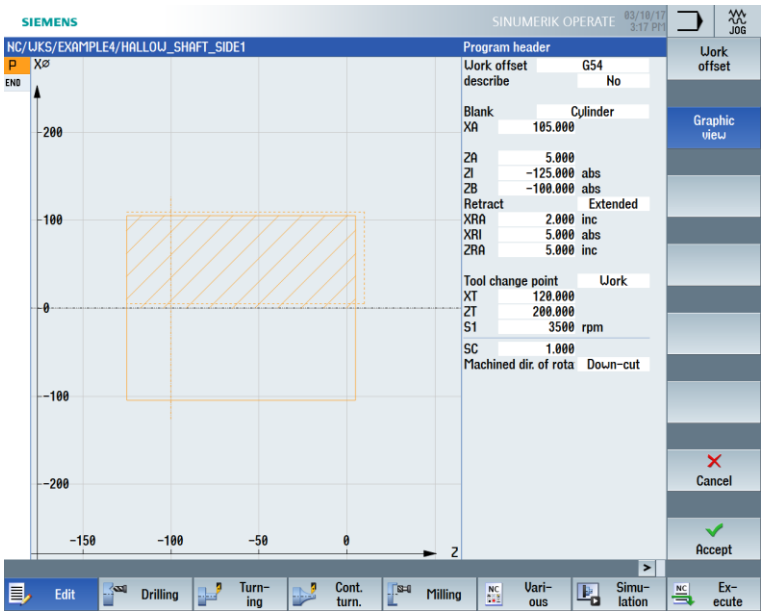


Figure 9-4 Workpiece dimensions in the program header

10.2.1 Face turning

Operating sequences

Follow the steps below to face the blank to Z0:

Select the "Turning" softkey.

Select the "Stock removal" softkey.

Select the "ROUGHING_T80 A" tool.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.2		
V	240 m/min	X	
Machining	Roughing	X	Since a large amount of material (5 mm) remains on the front face, set the machining to roughing.
Pos.	See figure below	X	
Machining direction	Face	X	
X0	105		
Z0	5		
X1	-1.6 abs	X	
Z1	0 abs	X	
D	2.5		
UX	0.0		
UZ	0.2		

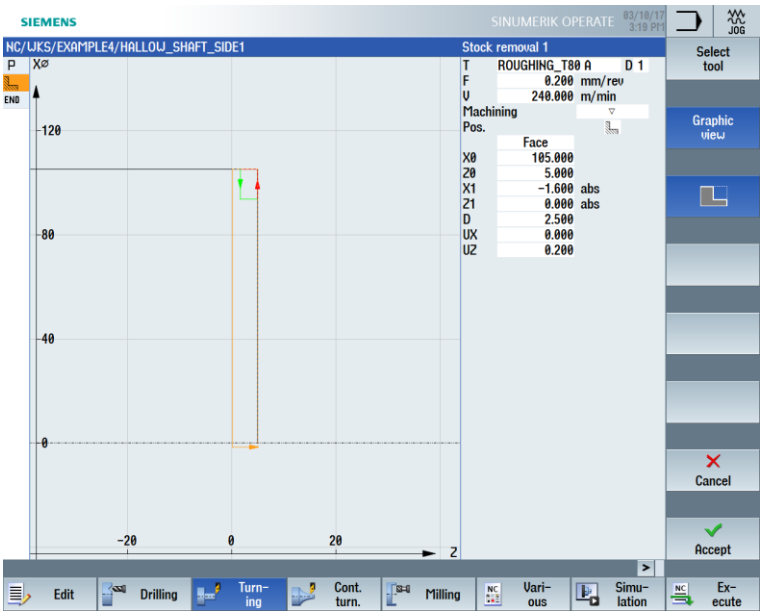


Figure 9-5 Facing the workpiece

Accept the entered values. Once applied, your machining step program looks like this:

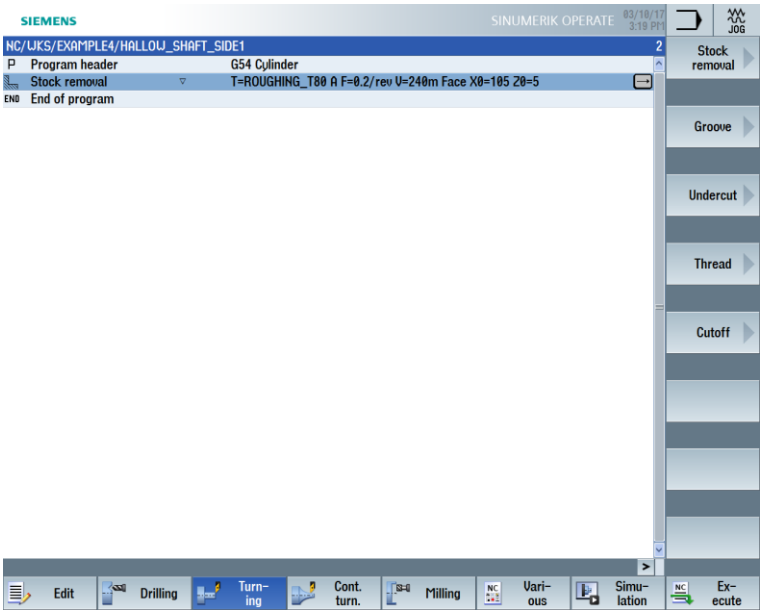


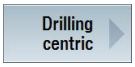
Figure 9-6 Machining plan after face turning

10.2.2 Drilling

Operating sequences

Follow the steps below to drill the workpiece in the center.

Select the "Drill." softkey.



Select the "Drilling centric" softkey.

Select tool

Open the tool list and select the "DRILL_32" solid drill.

OK

Apply the tool to the program.

Enter the following values for the drill hole in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.1 mm/rev	X	
S	2500 rpm	X	
	Chip removal	X	
Z0	0		
	Tip	X	
Z1	-57 inc	X	
D	57		

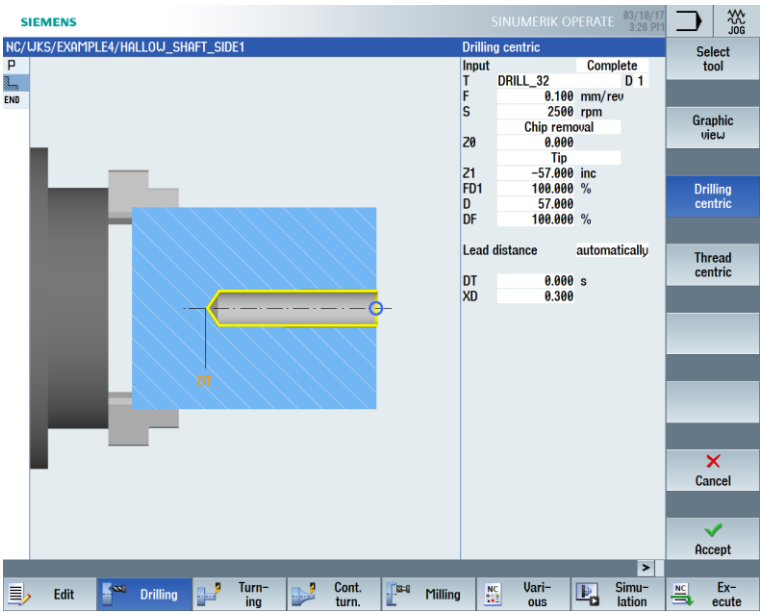


Figure 9-7 Drilling

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

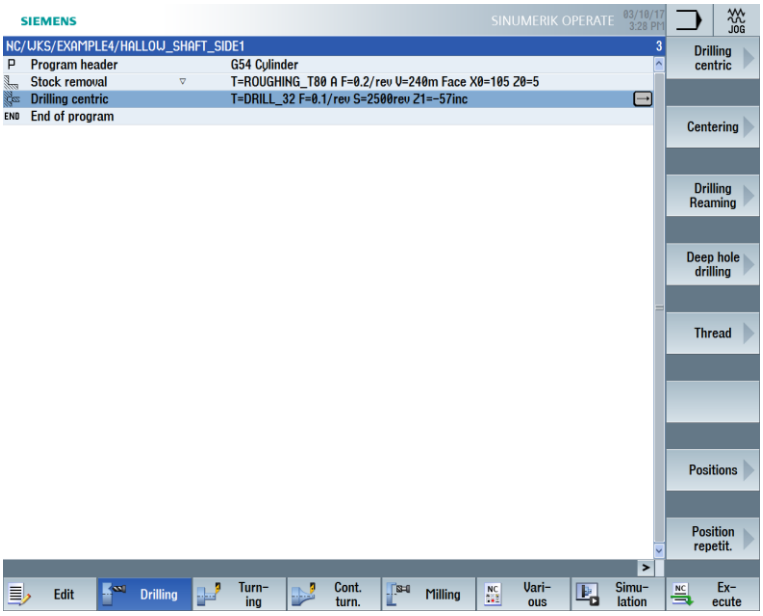
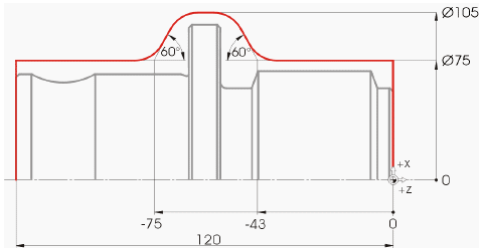


Figure 9-8 Machining plan after drilling

10.2.3 Blank contour

Operating sequences

Enter the following blank contour on your own: Since the workpiece will be machined only from one side per machining plan, the blank contour is to be constructed only to Z-65.



Select the "New contour" softkey. Enter the name “HOLLOW_SHAFT_BLANK“ for the contour.

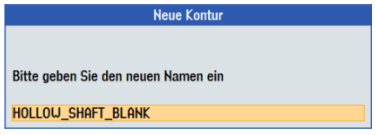


Figure 9-9 Creating the contour

Create the blank contour in the contour calculator (see figure below).

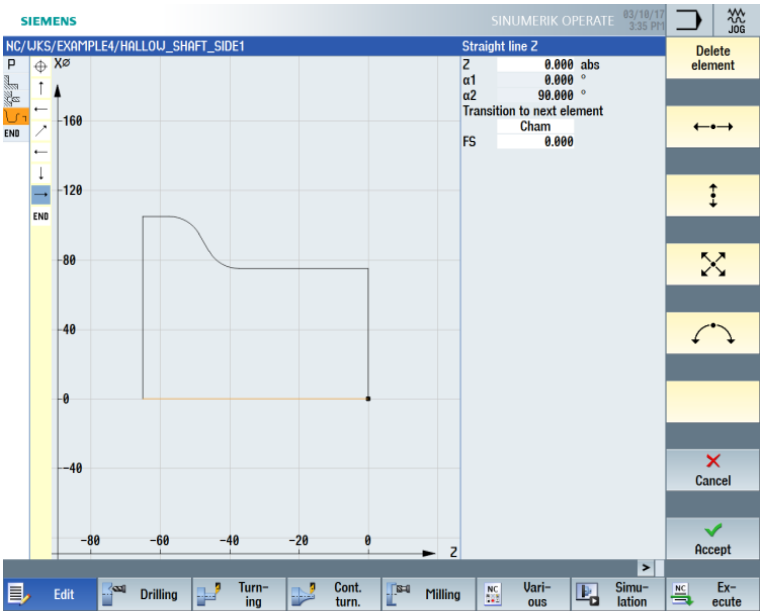
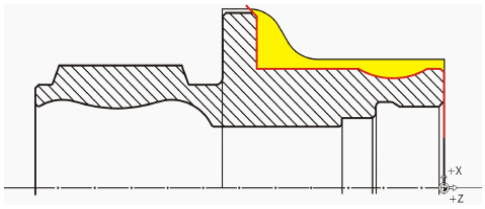


Figure 9-10 Creating the blank contour

10.2.4 Machined part contour of the first side, external

Operating sequences

Follow the steps below to enter the finished part contour:



Note:

The (red) contour of the machined part intentionally does not correspond to the drawing. The machined part contour serves, on the one hand, as the boundary for the roughing machining limit, but more importantly it specifies the precise traversing path for finishing. The construction therefore begins here at the drill hole diameter. This ensures that the end face is finished cleanly. The contour end is an extension of the chamfer beyond the blank. The large diameter is machined only in the second clamping.

Select the "Cont. turning" softkey.

enSelect the "New contour" softkey. Enter the name "HOLLOW_SHAFT_SIDE1_E" for the contour.

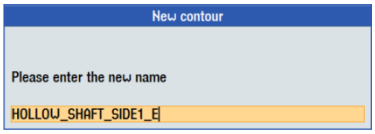


Figure 9-11 Creating the contour

Accept your input and set the starting point to X32/Z0.

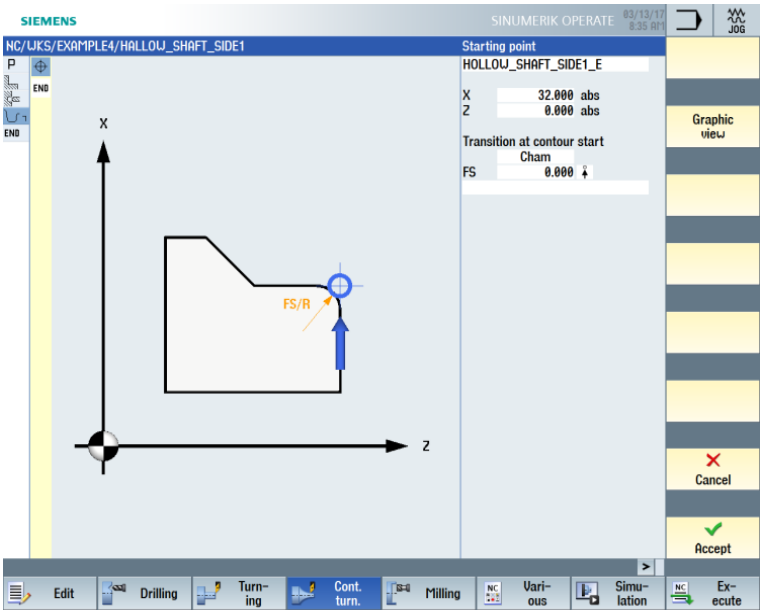
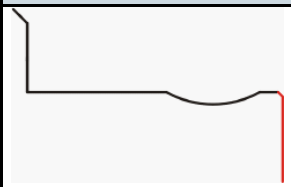


Figure 9-12 Entering the starting point for the contour

Accept your input.

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

Field	Value	Selection via toggle key	Notes
X	68 abs	X	
Transition to next element	Cham	X	
F	1		

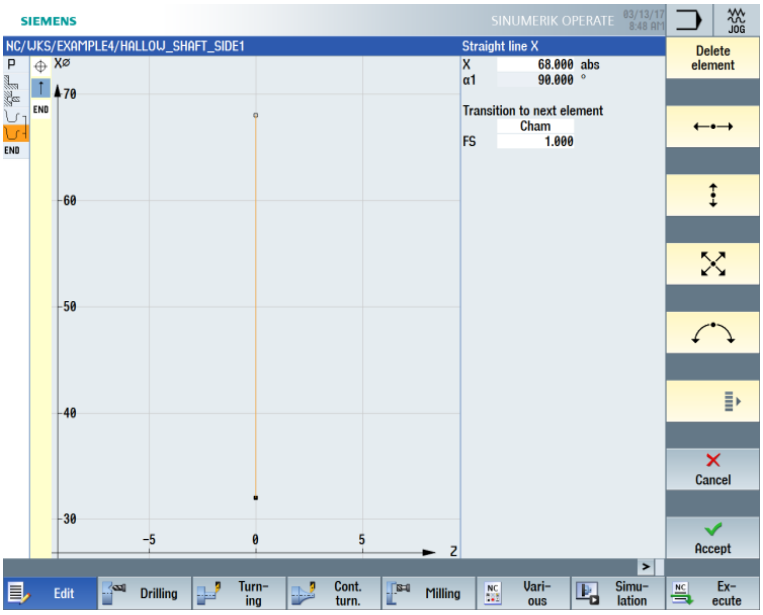
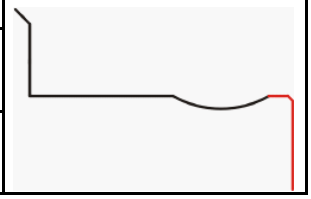


Figure 9-13 Entering the vertical straight-line segment for the contour

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
Z	-5 abs	X	
Transition to next element	Cham	X	
FS	0		

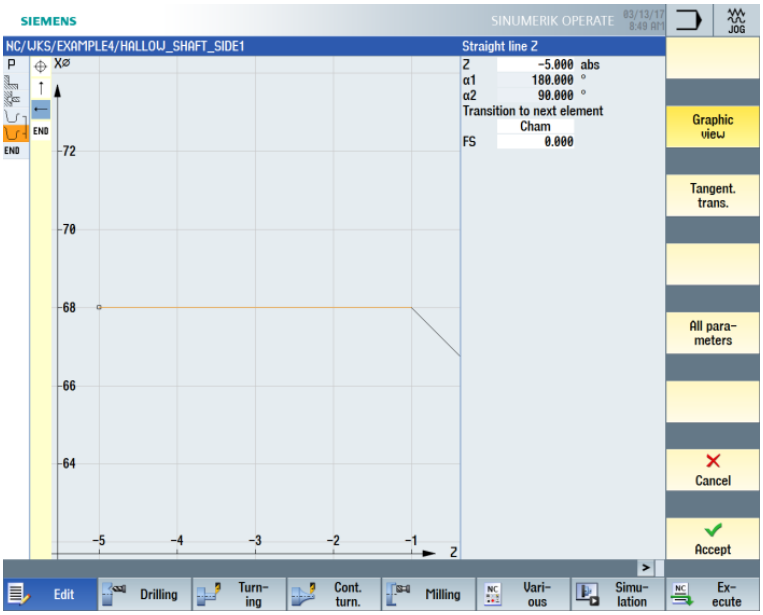


Figure 9-14 Entering the horizontal straight-line segment for the contour

Accept the entered values.

Enter the following values for the next segment in the screen form:

Field	Value	Selection via toggle key	Notes
Direction of rotation	right	X	
R	20		
X	68 abs	X	
Z	-25 abs	X	
Transition to next element	Cham	X	
FS	0		

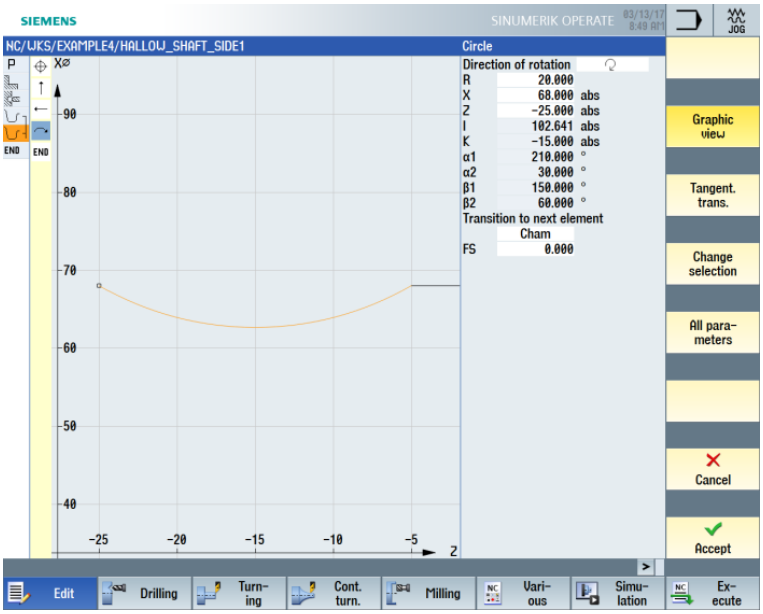


Figure 9-15 Entering the arc for the contour

Dialog select

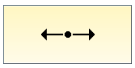
Select the desired construction.

Dialog accept

Accept your selection.

Accept

Accept the contour segment.



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

Field	Value	Selection via toggle key	Notes
Z	-55 abs	X	 The undercut is inserted later as an individual element.
Transition to next element	Cham	X	
FS	0		

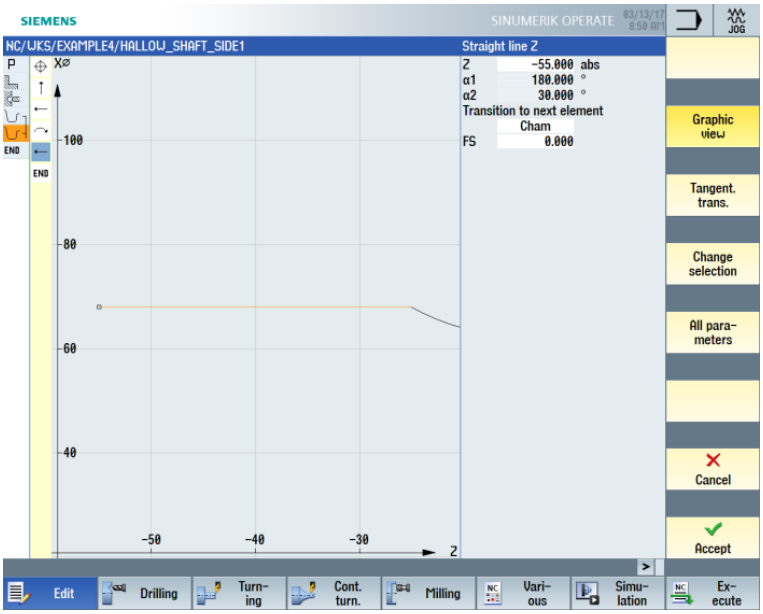


Figure 9-16 Entering the horizontal straight-line segment for the contour

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
X	98 abs	X	 The inclined straight line remains as chamfer after the second side has been machined.
Transition to next element	Cham	X	
FS	0		

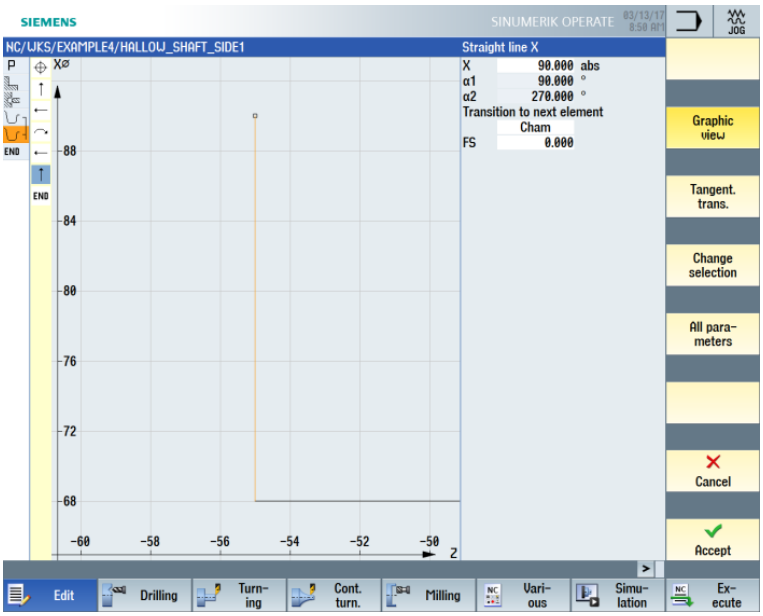
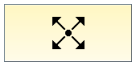


Figure 9-17 Entering the vertical straight-line segment for the contour

Accept the entered values.



Enter the following values for the inclined straight-line segment in the screen form:

Field	Value	Selection via toggle key	Notes
X	106 abs	X	
α1	135	X	
Transition to next element	Cham	X	
FS	0		

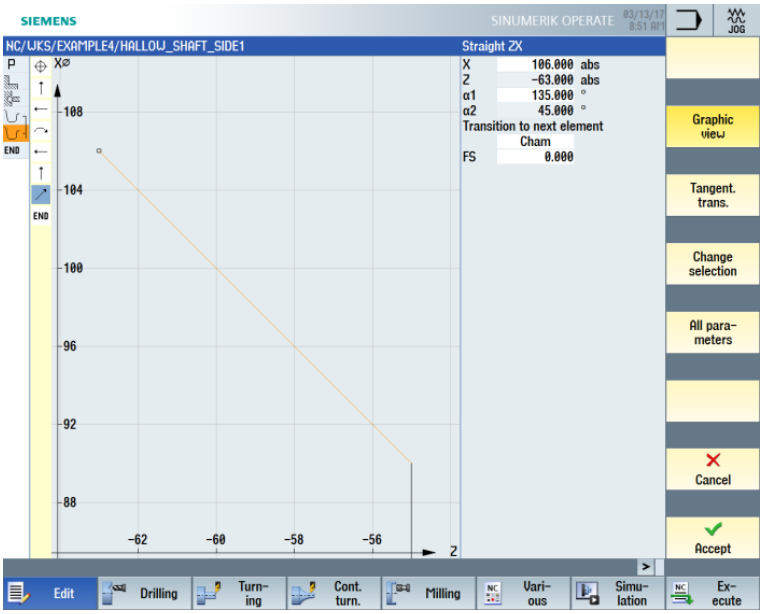
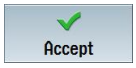


Figure 9-18 Entering the inclined straight-line segment for the contour

Accept the entered values.



Accept the contour to apply it to your machining plan.

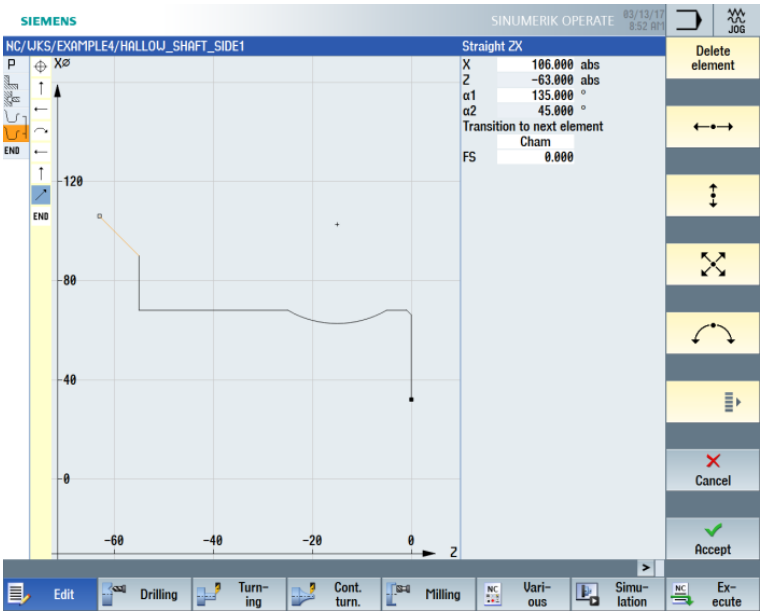


Figure 9-19 Contour in the contour calculator

Once applied, the machining plan looks like this: The two contours are linked automatically.

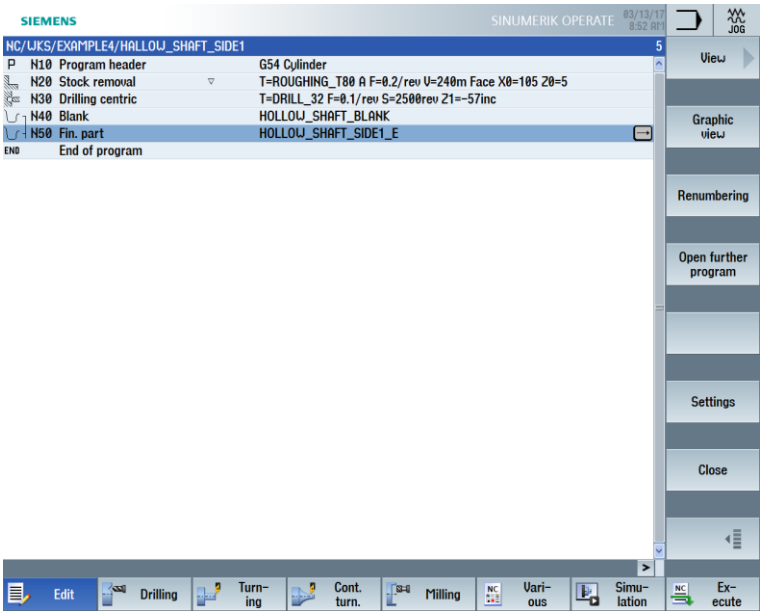


Figure 9-20 Machining plan after input of the contours

Stock removal, residual stock removal and finishing

You machine the contour in the next machining step.
Proceed as follows:

Select the "Stock removal" softkey.

Open the tool list and select the "ROUGHING_T80 A" tool.

Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.3		
V	260 m/min	X	
Machining	Roughing Longitudinal Outside	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Contour	X	You can choose between the following settings in the blank descriptions: Cylinder: Blank = cylinder Contour: Blank = constructed contour Allowance: Blank = constructed contour with defined allowance
Relief cuts	No	X	The roughing tool is not appropriate for insertion. Therefore, switch the Relief cuts field to" No".
Limit	No	X	

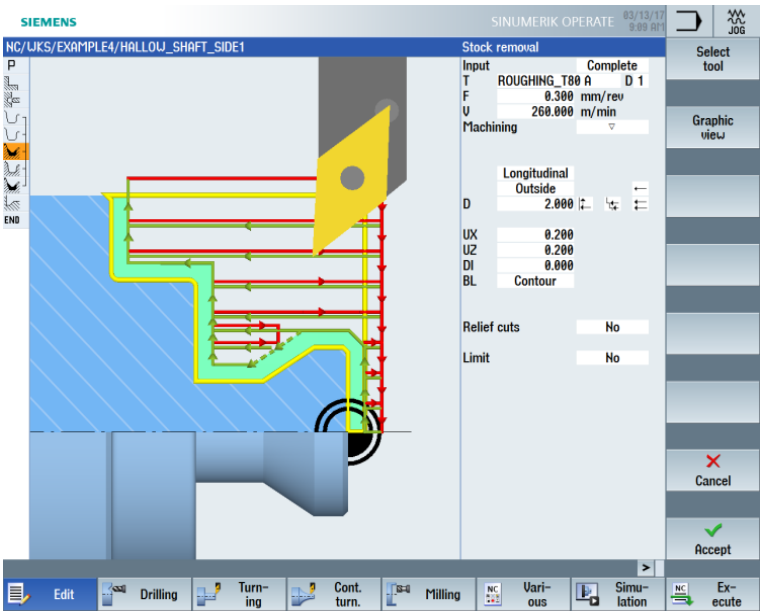


Figure 9-21 Roughing the contour

Accept the entered values.

Select the "Cut resid stock" softkey.

Open the tool list and select the "FINISHING_T35 A" tool.

Apply the tool to the program. Before the finishing, the residual material is cut in the hollow groove in this machining step.

Enter the following values for stock removal of residual material in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.2		
V	240 m/min	X	
Machining	Roughing Longitudinal Outside	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
Relief cuts	Yes	X	So that the hollow groove is taken into consideration, the Relief cuts field must be switched to "Yes".
FR	0.2		
Limit	No	X	

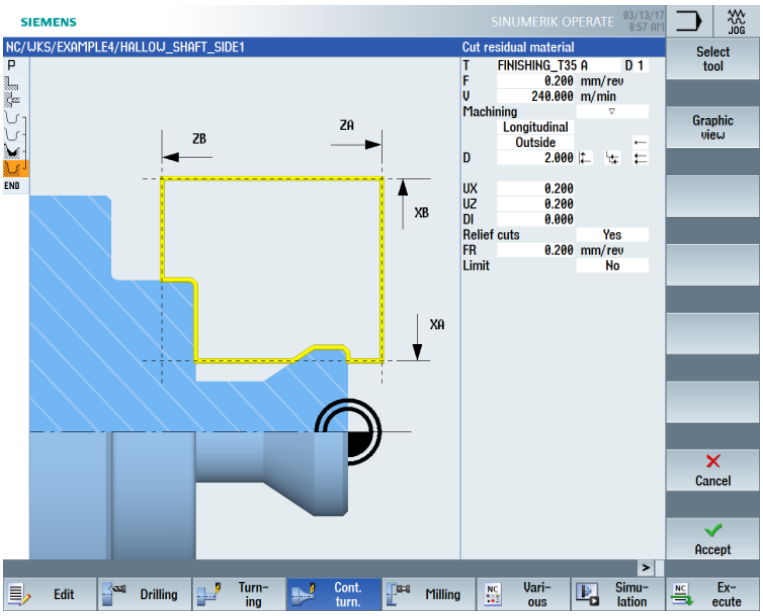


Figure 9-22 Stock removal of residual material from contour

Accept the entered values.

Select the "Stock removal" softkey.

Open the tool list and select the "FINISHING_T35 A" tool.

Apply the tool to the program.

Enter the following values for finishing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	280 m/min	X	
Machining	Finishing Longitudinal Outside	X X X	
Allowance	No	X	
Relief cuts	Yes	X	Switch Relief cuts to "Yes".
Limit	No	X	

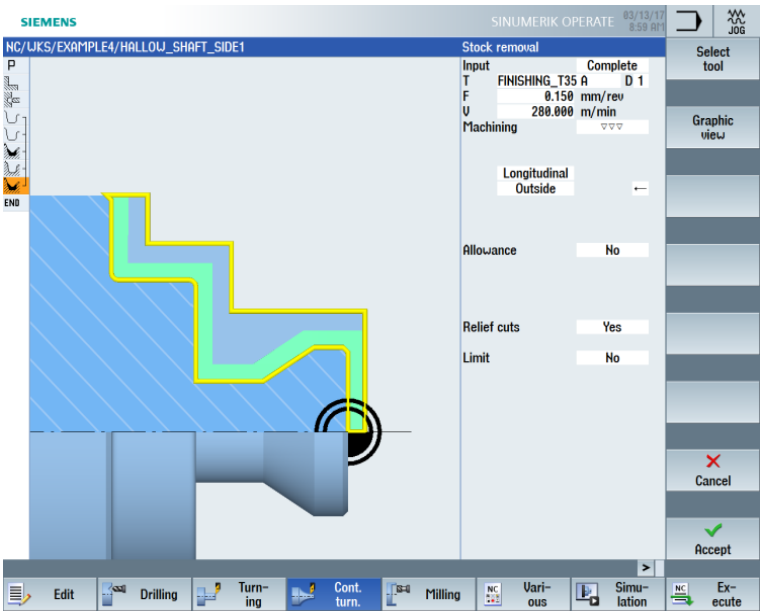


Figure 9-23 Finishing the contour

Accept the entered values. Once applied, your machining step program looks like this: The contours are automatically linked to the stock removal machining steps.

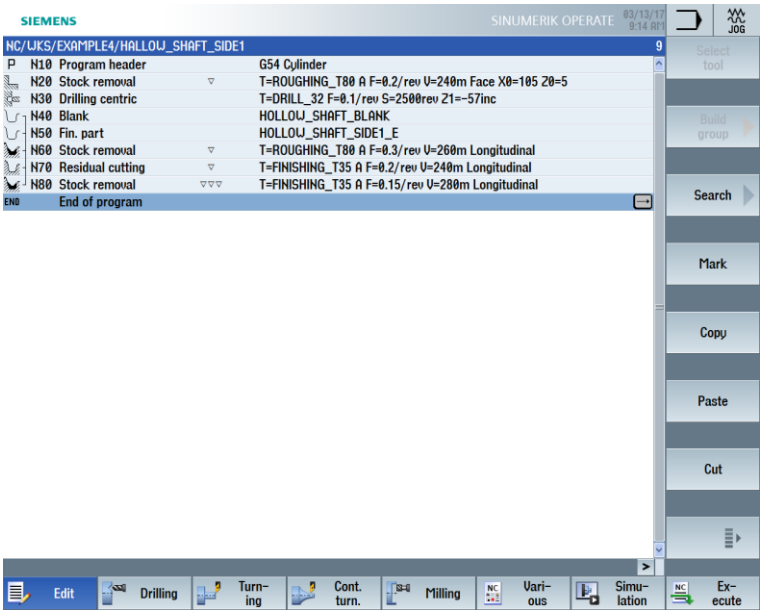
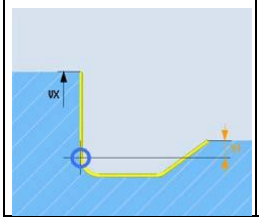
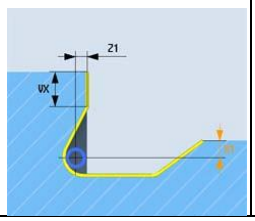
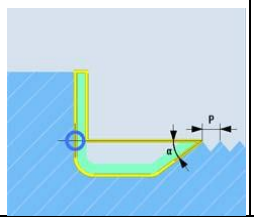
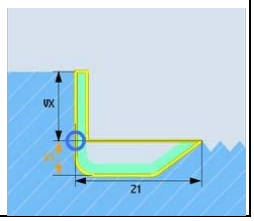


Figure 9-24 Machining plan after machining the contour

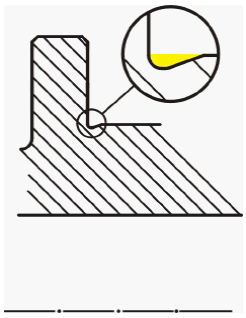
10.2.5 Undercut

You can select from four different types of undercut:

Undercut form E	Undercut form F	Undercut thread DIN	Undercut thread
			

Operating sequences

Follow the steps below to create the undercut:



After residual stock removal, the list of machining steps looks like this:

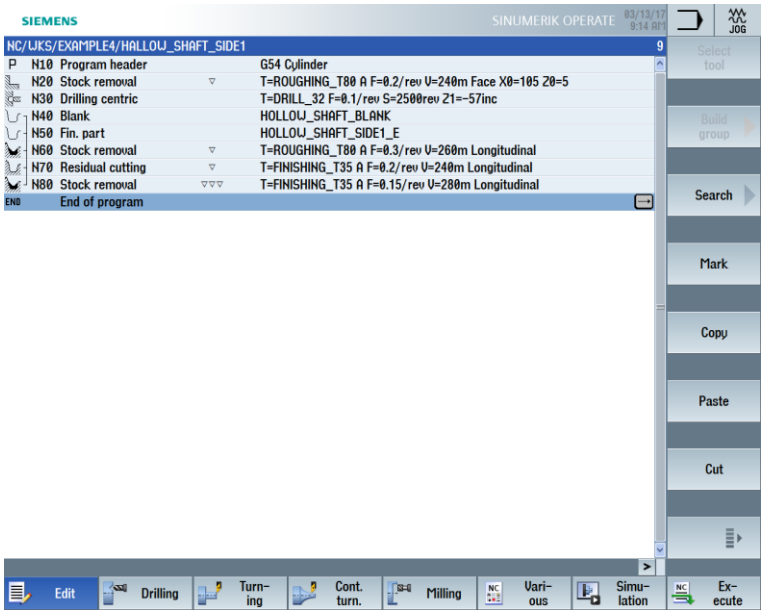
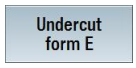


Figure 9-25 Machining plan after stock removal

Select the "Turning" softkey.

Select the "Undercut" softkey.



Select the "Undercut form E" softkey.

Open the tool list and select the "FINISHING_T35 A" tool.

Apply the tool to the program.

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

Field	Value	Selection via toggle key	Notes
F	0.15		
V	200 m/min	X	
Pos.	See figure below E 1.0 x 0.4	X X	
X0	68		
Z0	-55		
X1	0 inc	X	
VX	70 abs	X	

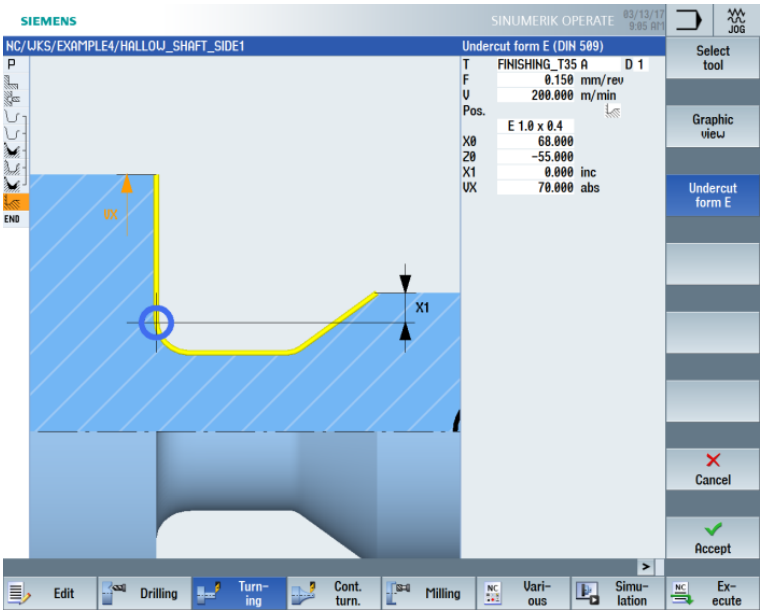


Figure 9-26 Specifying the undercut

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

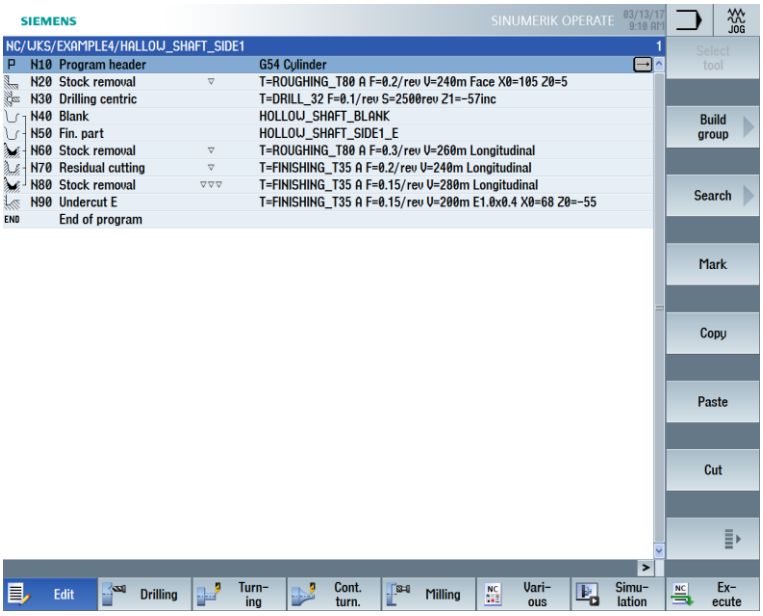
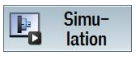


Figure 9-27 Machining plan with undercut



Start the "Simulation".

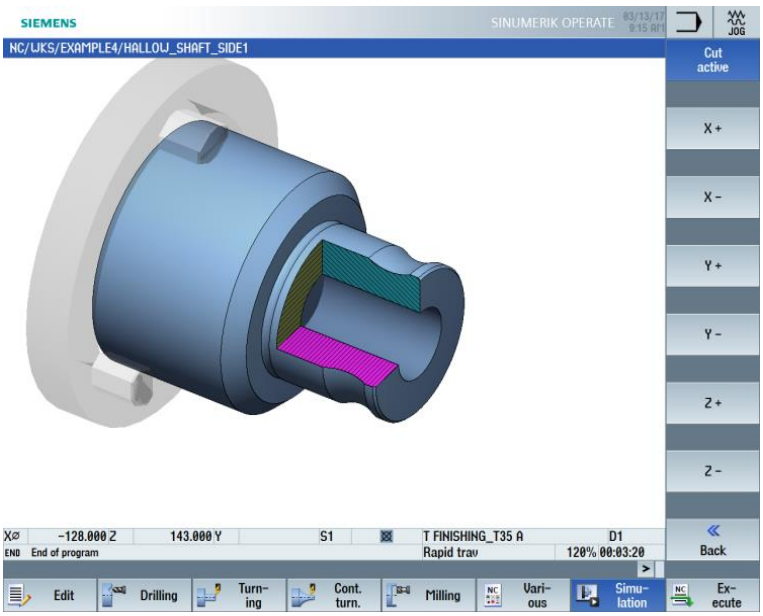


Figure 9-28 Simulation – Cut active

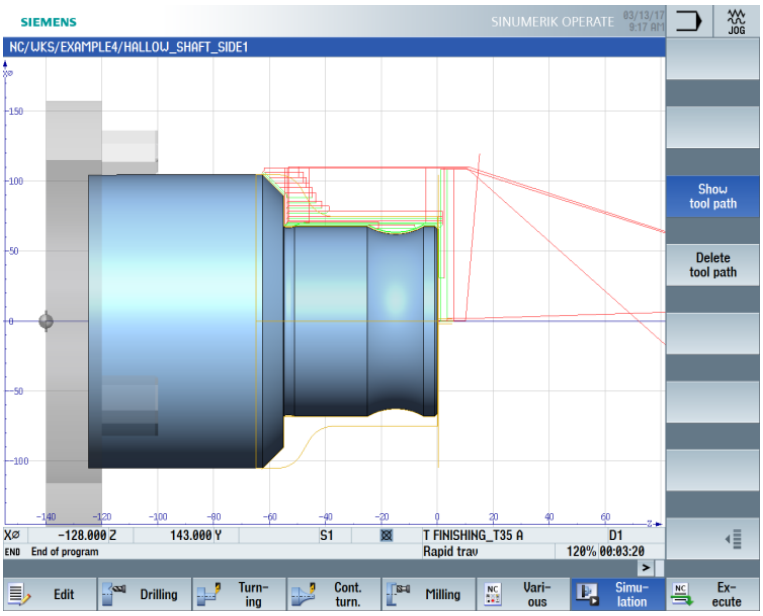
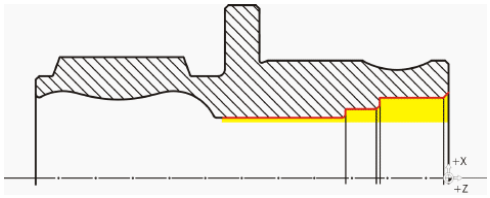


Figure 9-29 Simulation – Side view with display of the traversing paths

10.2.6 Finished part contour of the first side, internal

Operating sequences

Follow the steps below to enter the finished part contour:



Select the "Cont. turning" softkey.

Select the "New contour" softkey. Enter the name "HOLLOW_SHAFT-SIDE1_I" for the contour.

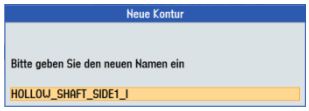


Figure 9-30 Creating the contour

Accept your input.

Set the starting point to X50/Z0.

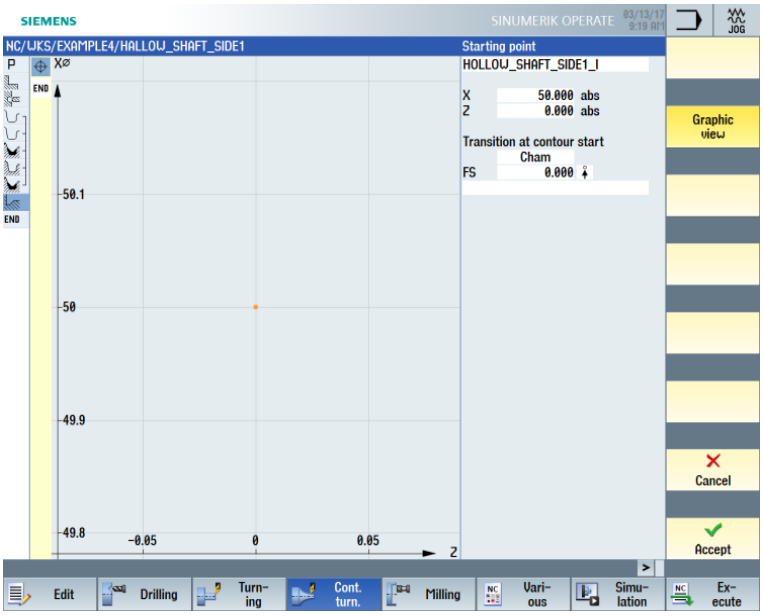


Figure 9-31 Entering the starting point for the contour

Accept your input.

Create the contour on your own (see figure below).

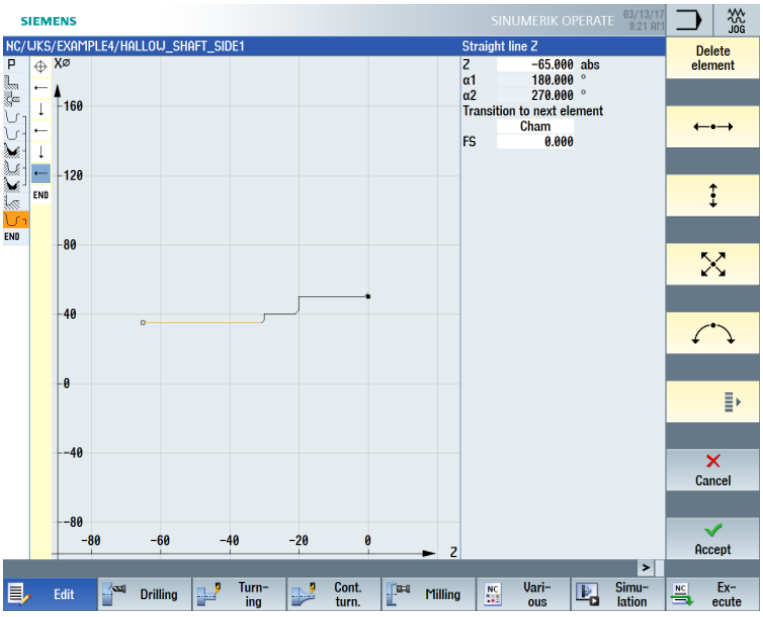


Figure 9-32 Finished part contour of the first side, internal

Stock removal, residual stock removal and finishing

You machine the contour in the next machining step. The geometries are present in your machining plan as follows:

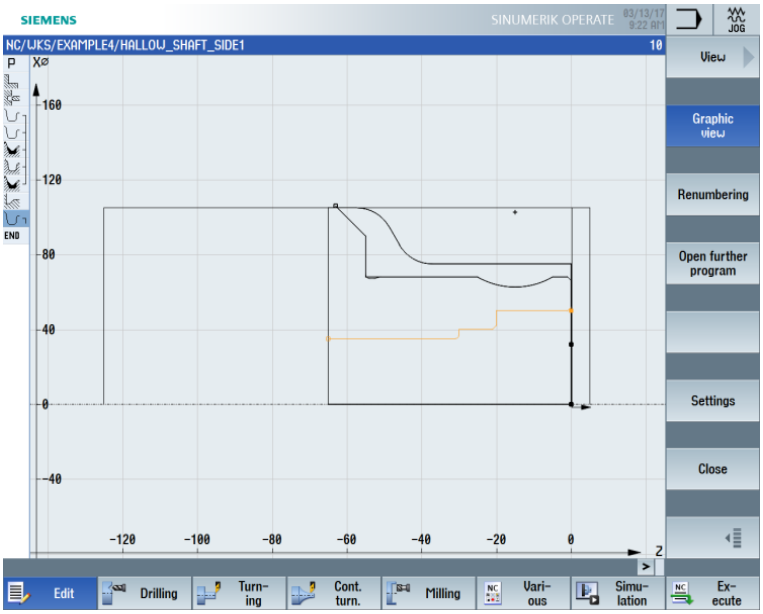


Figure 9-33 Broken-line graphic

Select the "Stock removal" softkey.

Open the tool list and select the "ROUGHING_T80 I" tool.

Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.25		
V	250 m/min	X	
Machining	Roughing Longitudinal Inside	X X X	You must switch the machining to "Inside".
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Cylinder	X	Because drilling has already been performed, you do not have to take a black contour into account for the internal machining. Switch to Cylinder.

Field	Value	Selection via toggle key	Notes
XD	32 abs	X	
ZD	0 inc	X	
Relief cuts	No	X	
Limit	No	X	

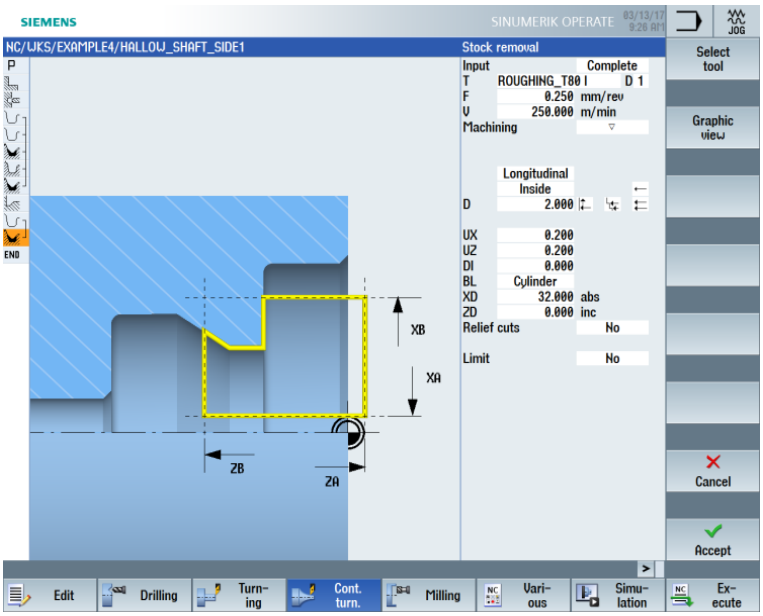


Figure 9-34 Roughing the contour

Accept the entered values.

Select the "Stock removal" softkey.

Open the tool list and select the "FINISHING_T35 I" tool.

Apply the tool to the program.

Enter the following values for finishing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.12		
V	280 m/min	X	
Machining	Finishing Longitudinal Inside	X X X	
Allowance	No	X	
Relief cuts	No	X	
Limit	No	X	

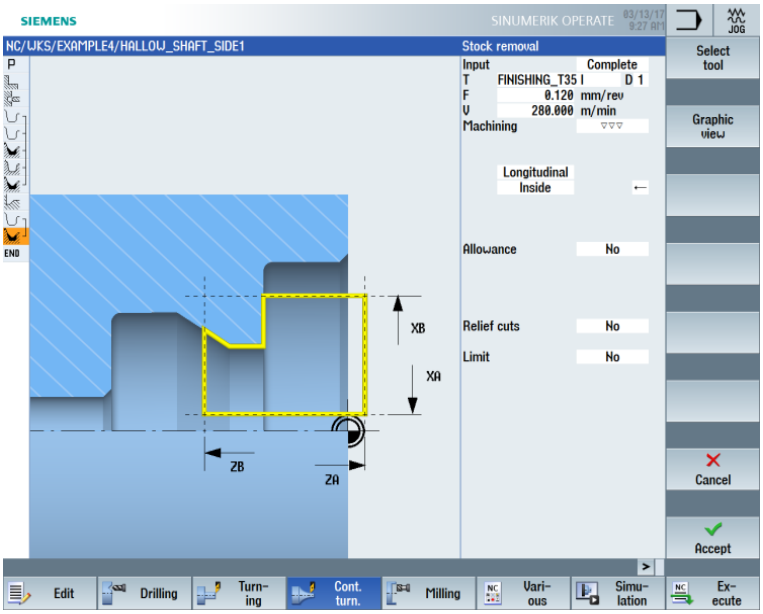


Figure 9-35 Finishing the contour

Accept the entered values.

Start the simulation to perform checks.

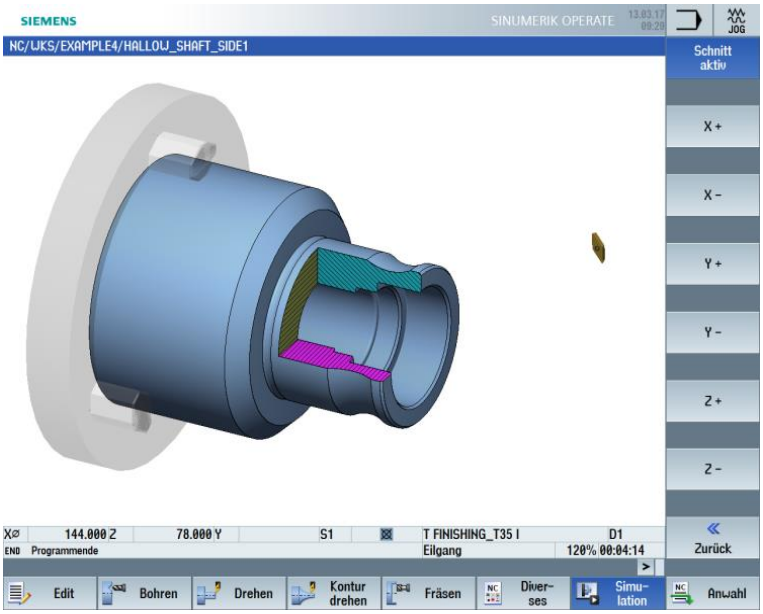
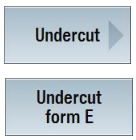


Figure 9-36 Simulation – Cut active

Undercut



Follow the steps below to create the undercut:

Select the "Undercut" softkey.

Select the "Undercut form E" softkey.

Create the undercut (see figure below).

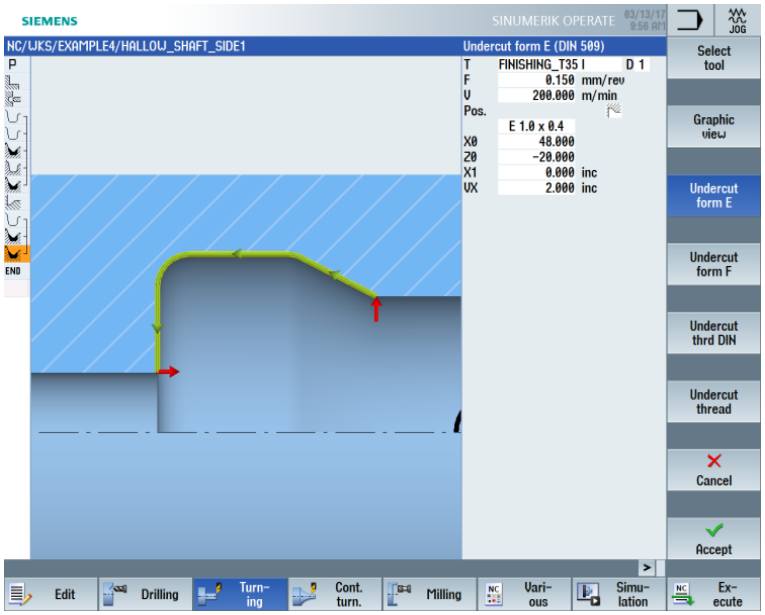


Figure 9-37 Creating an undercut

Note:

Make sure that the undercut is in the correct position!

Start the simulation.

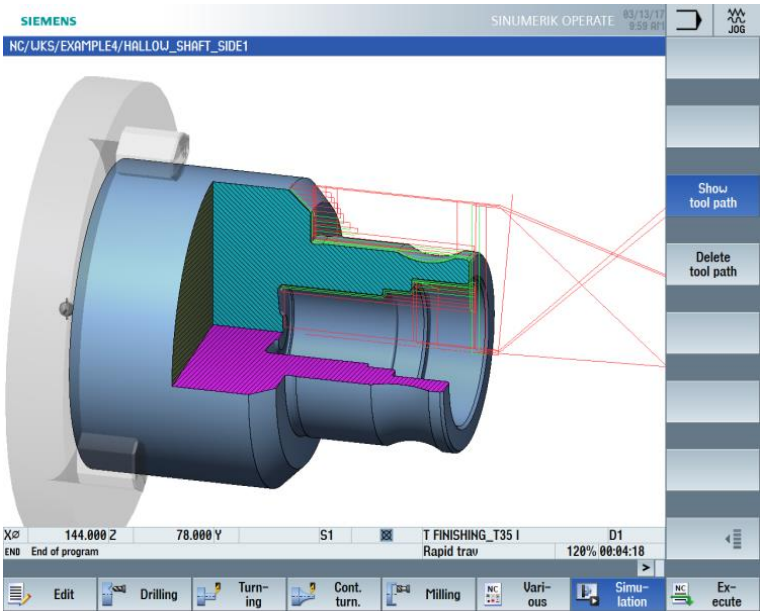


Figure 9-38 Undercut simulation (with display of the traversing paths)

The machining plan for the first side of the workpiece looks like this:

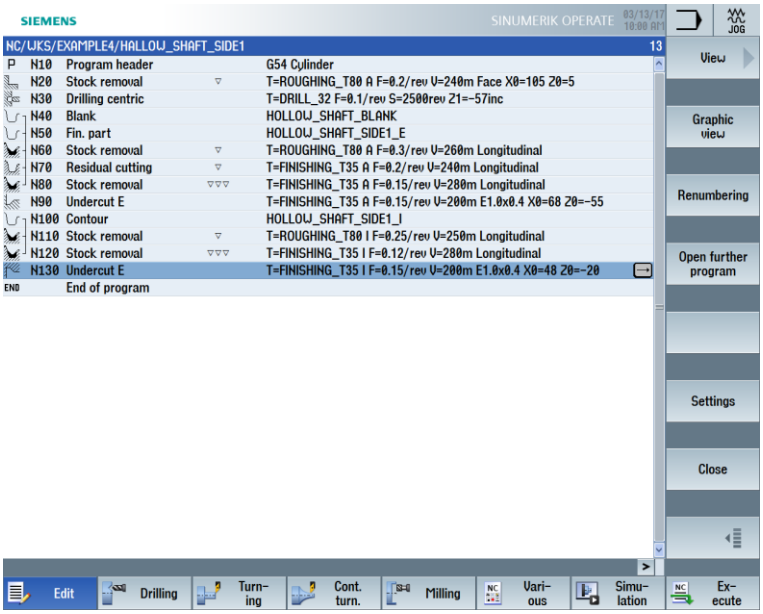
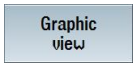


Figure 9-39 Machining plan with undercut

10.2.7 The work step editor

Functions of the machining step editor

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



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

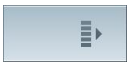
You can use the "Search" softkey to search for texts in the program.

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

You can use this softkey to copy work steps to the clipboard.

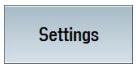
You can use the "Paste" softkey to paste machining steps from the clipboard to the machining plan. The machining step(s) are always inserted after the currently marked machining step.

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

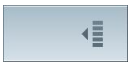


Use this softkey to go to the expanded menu.

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



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



Use this softkey to return to the previous menu.

You will need some of these functions in order to reuse the blank contour of the first side in the machining plan for the second side of the workpiece. You copy the blank contour to the clipboard and insert it in the machining plan for the second side.

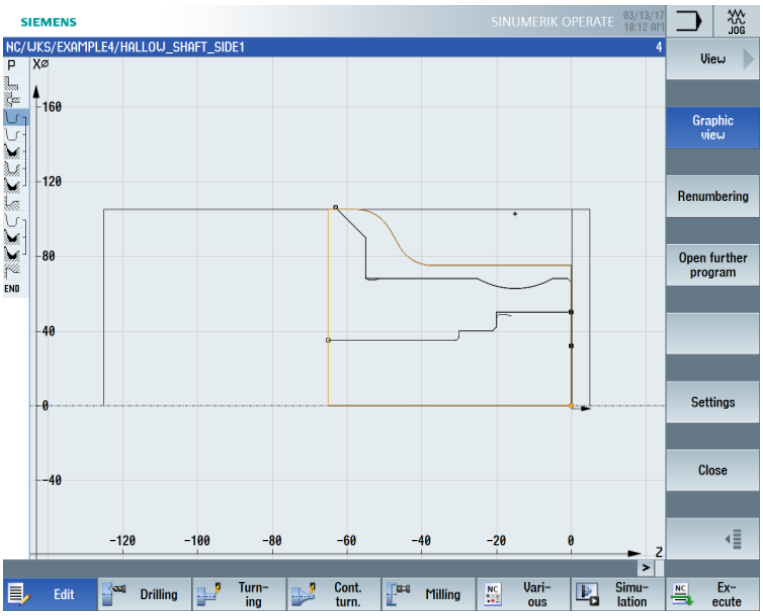


Figure 9-40 Blank contour

10.2.8 Copying the contour

Operating sequence

Proceed as follows to copy the blank contour to the clipboard: Navigate to the "HOLLOW_SHAFT_BLANK" contour.

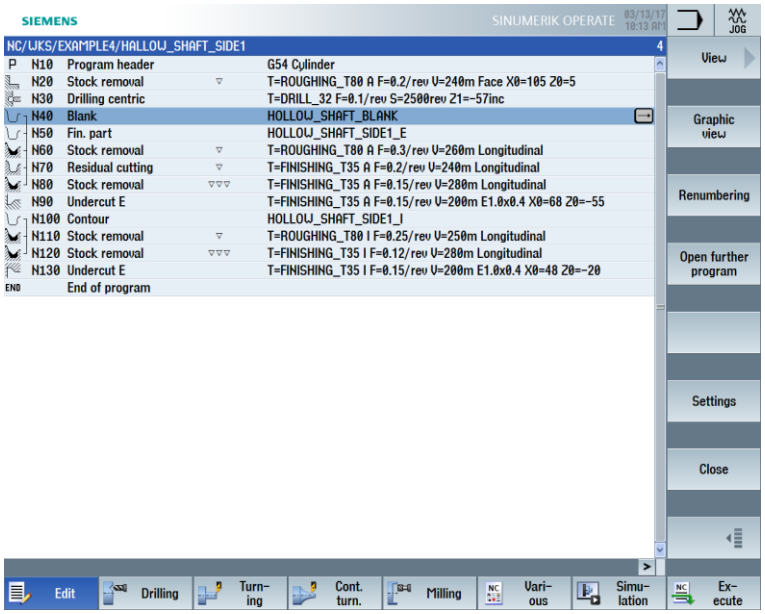


Figure 9-41 Copying the contour to the clipboard

Copy the blank contour to the clipboard. The contour remains on the clipboard until you copy another machining step to the clipboard or shut down the controller.

10.3 Creating the second workpiece side

Creating the machining plan

Proceed as follows to create a machining plan for the second side of the workpiece:

Operating sequences

Create the program "HOLLOW_SHAFT_SIDE2" on your own.

Figure 9-42 Creating the ShopTurn program

Enter the following data in the program header (see figure):

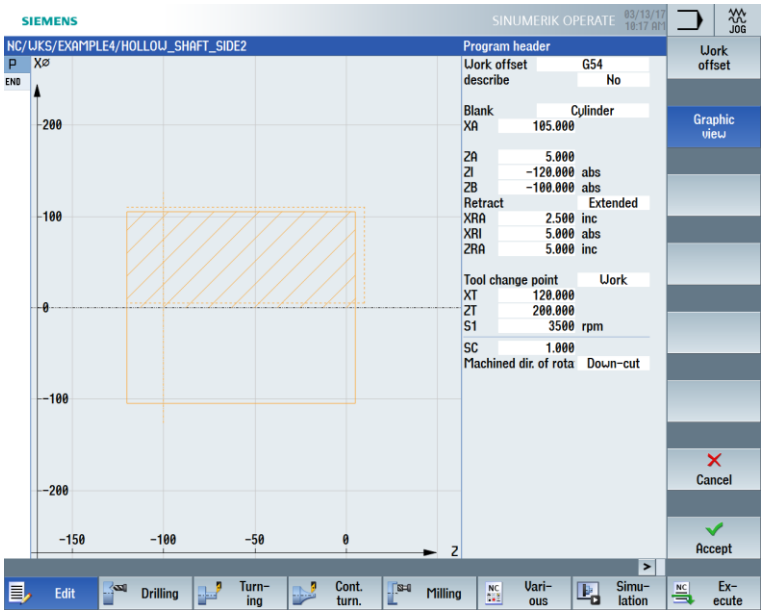


Figure 9-43 Workpiece dimensions in the program header

10.3.1 Face turning

Operating sequences

Follow the steps below to face the blank to X-1.6 and Z0:

Select the "Turning" softkey.

Select the "Stock removal" softkey.

Accept

Select the "ROUGHING_T80 A" tool.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.2		
V	240 m/min	X	
Machining	Roughing	X	Since a large amount of material (5 mm) remains on the front face, set the machining to roughing.
Pos.	See figure below	X	
Machining direction	Face	X	
X0	105		
Z0	5		
X1	-1.6 abs	X	
Z1	0 abs	X	
D	2.5		
UX	0.0		
UZ	0.2		

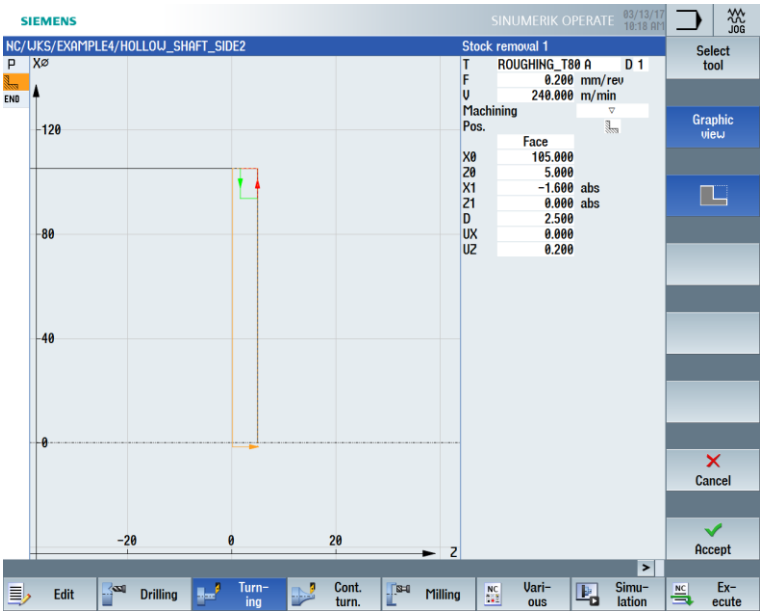


Figure 9-44 Facing the workpiece

Accept the entered values. Once applied, your machining step program looks like this:

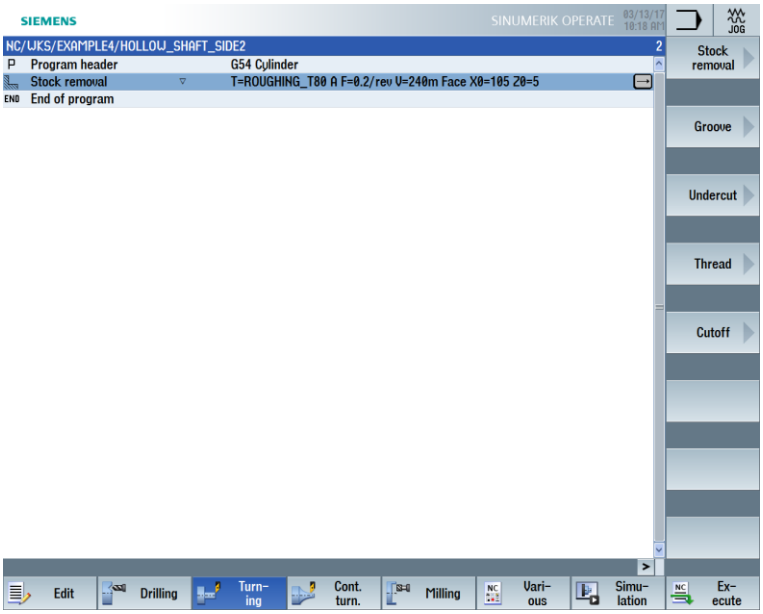


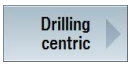
Figure 9-45 Machining plan after face turning

10.3.2 Drilling

Operating sequences

Follow the steps below to drill the workpiece in the center.

Select the "Drill." softkey.



Select the "Drilling centric" softkey.

Open the tool list and select the "DRILL_32" solid drill.

Apply the tool to the program.

Enter the following values for the drill hole in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.1 mm/rev	X	
S	2500 rpm	X	
	Chip removal	X	
Z0	0		
	Tip		
Z1	-67 abs	X	
D	67		
DT	0	X	

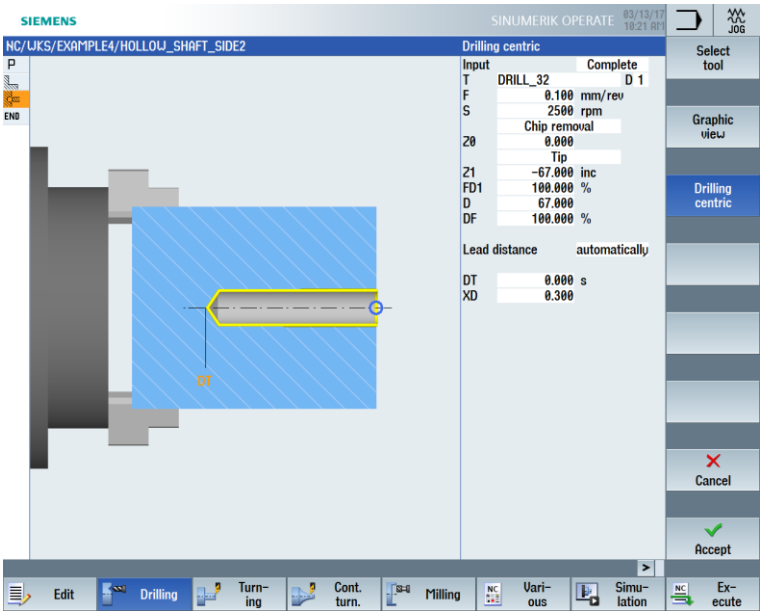


Figure 9-46 Drilling

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

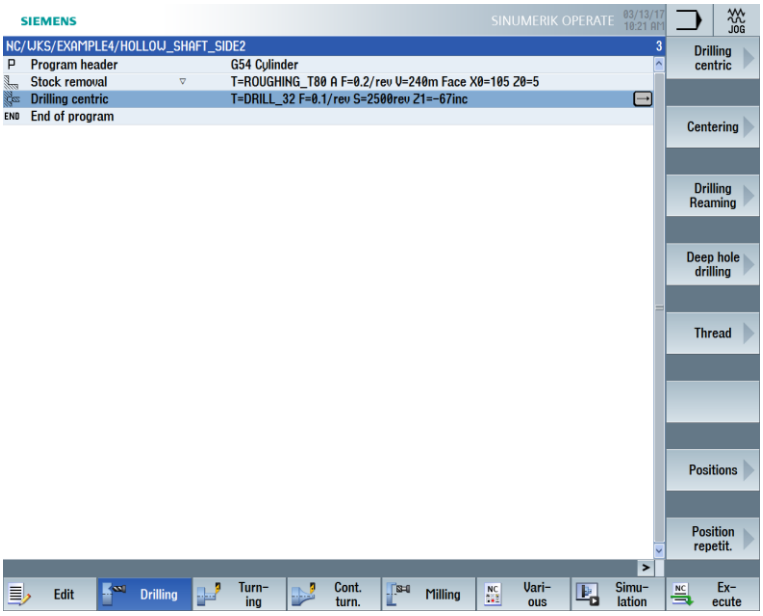


Figure 9-47 Machining plan after input of the drilling position

10.3.3 Specifying a blank contour

Operating sequences

Follow the steps below to paste the blank contour from the clipboard into your machining plan:

First, navigate in the machining plan to the most recently entered machining step (see figure).

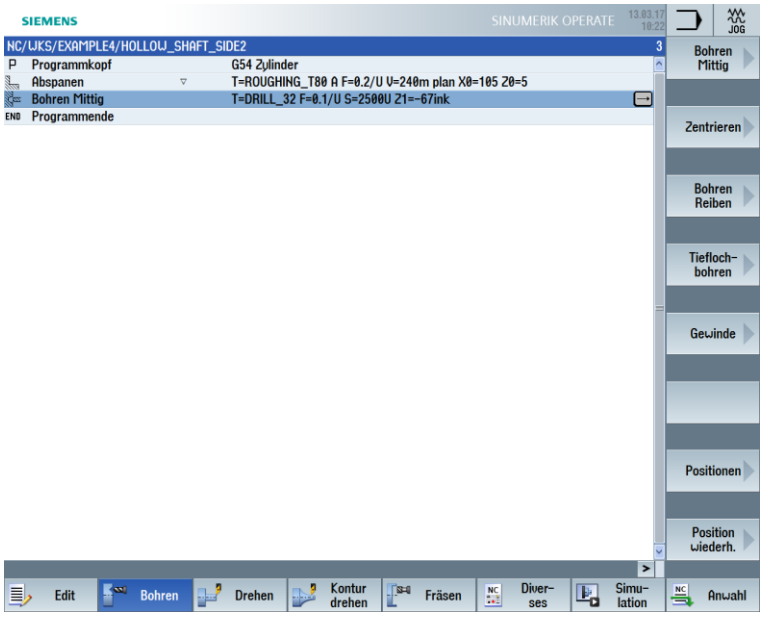


Figure 9-48 Position for inserting the blank contour

Paste the blank contour from the clipboard. After the contour is pasted, your machining plan should look like this:

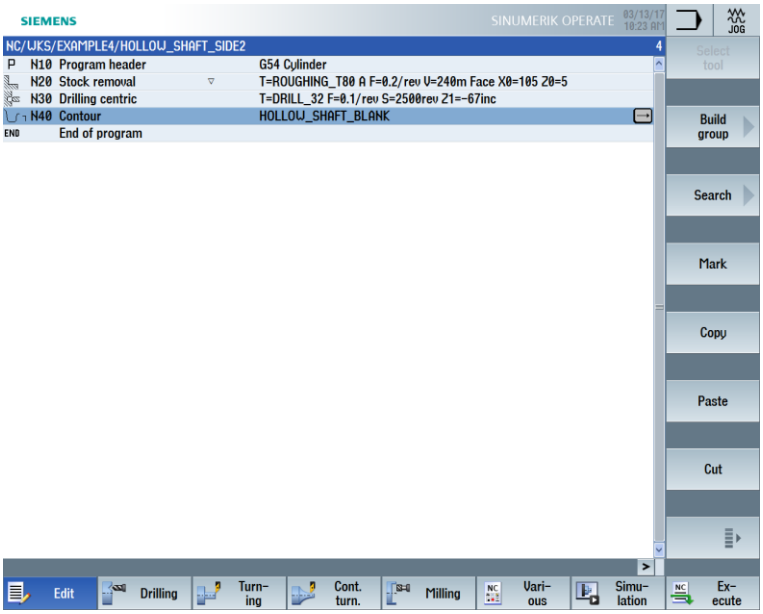
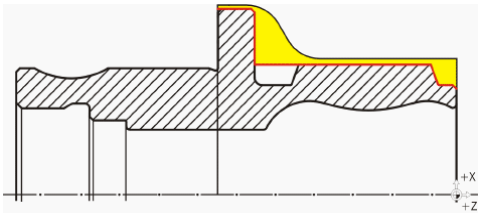


Figure 9-49 Pasting the contour

10.3.4 Finished part contour of the second side, external

Operating sequences

Follow the steps below to enter the finished part contour:



Note:

The asymmetrical groove is machined later.

Select the "Cont. turning" softkey.

Select the "New contour" softkey. Enter the name “HOLLOW_SHAFT_SIDE2_E“ for the contour.

Accept your input.

Set the starting point to X57/Z0.

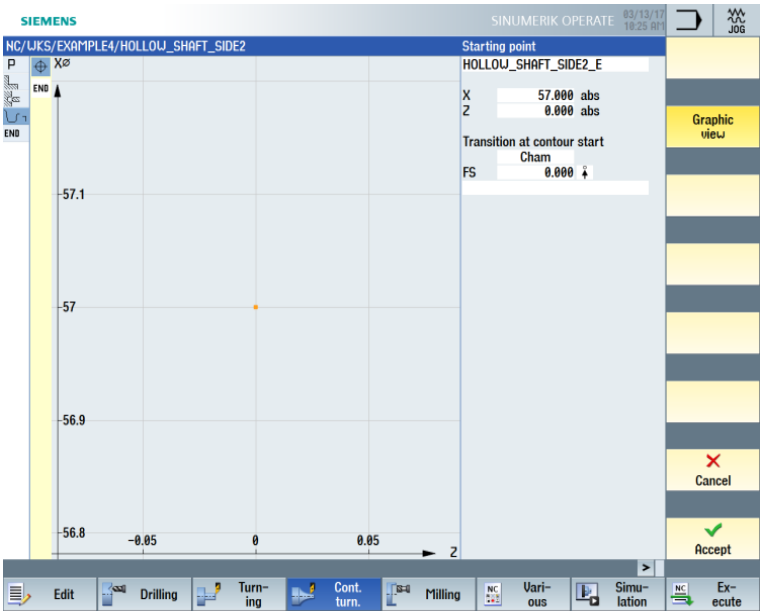


Figure 9-50 Entering the starting point for the contour

Accept your input.

Create the contour up to the end point at Z-65 and X100 on your own (see figure below).

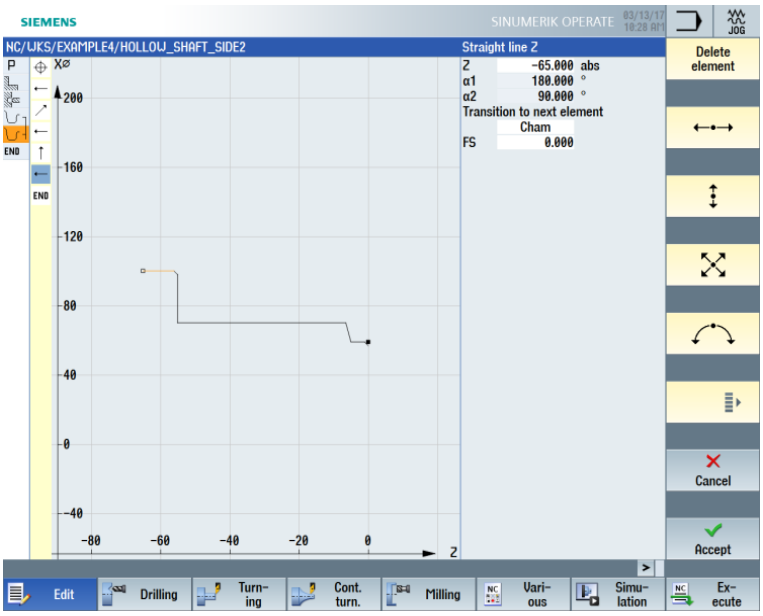
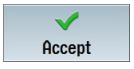


Figure 9-51 Contour in the contour calculator



Accept the contour to apply it to your machining plan.

Stock removal and finishing

You machine the contour in the next machining step. Proceed as follows:

Select the "Stock removal" softkey.

Open the tool list and select the "ROUGHING_T80 A" tool.

Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.3		
V	260 m/min	X	
Machining	Roughing Longitudinal Outside	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Contour	X	
Relief cuts	No	X	
Limit	No	X	

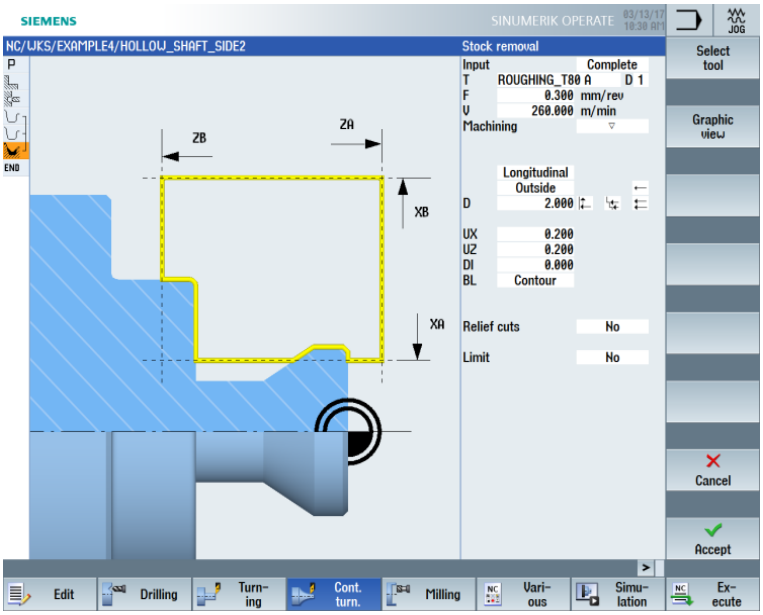


Figure 9-52 Roughing the contour

Accept

Accept the entered values. Once applied, your machining step program looks like this:

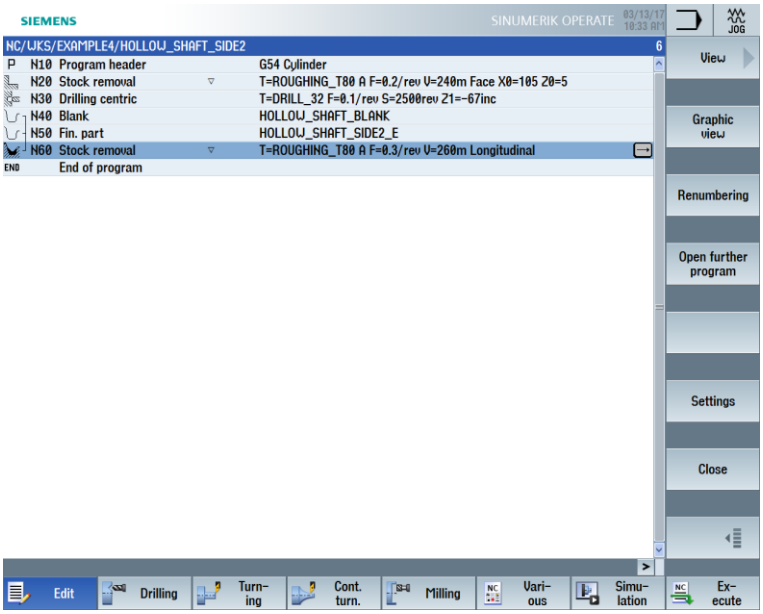


Figure 9-53 Machining plan after roughing

Stock removal

Select the "Stock removal" softkey.

Select tool

Open the tool list and select the "FINISHING_T35 A" tool.

OK

Apply the tool to the program.

Enter the following values for finishing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	200 m/min	X	
Machining	Finishing Longitudinal Outside	X X X	
Allowance	No	X	
Relief cuts	No	X	
Limit	No	X	

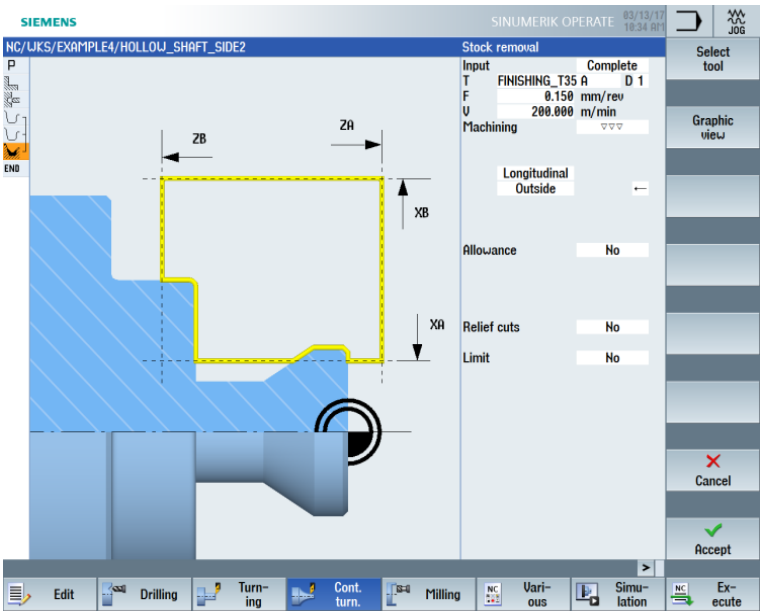


Figure 9-54 Finishing the contour

Accept the entered values. Once applied, your machining step program looks like this:

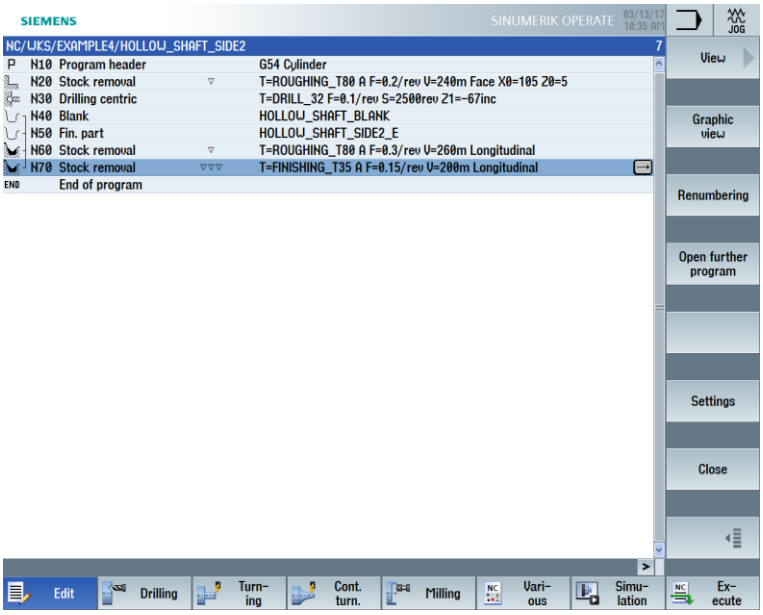
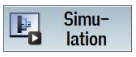


Figure 9-55 Machining plan after machining the contour



Start the "Simulation".

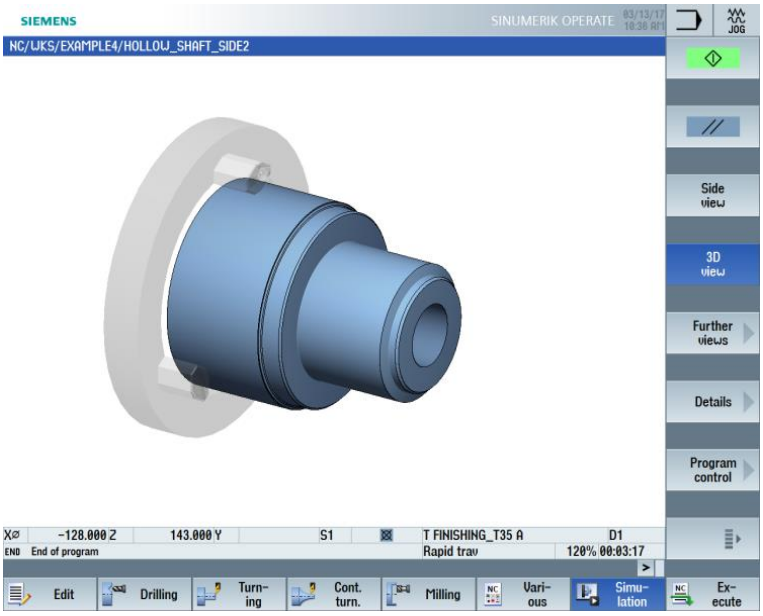
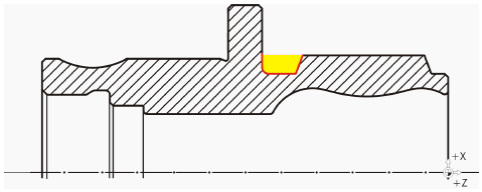


Figure 9-56 Simulation – 3D view

10.3.5 Creating the asymmetrical groove

Operating sequences

Follow the steps below to create the asymmetrical groove:



Select the "Turning" softkey.

Select the "Groove" softkey.

Select the "Groove 2" softkey.

Open the tool list and select the "PLUNGE_CUTTER_3 A" tool.

Apply the tool to the program.

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

Field	Value	Selection via toggle key	Notes
F	0.08		
V	180 m/min	X	
Machining	Roughing/ Finishing	X	
Pos.	See figure below	X	
X0	70		
Z0	-55		
B1	10	X (field)	
T1	5.5 inc	X	
α1	0		
α2	15		
R1	0	X (field)	
R2	2	X (field)	
R3	0	X (field)	
R4	0	X (field)	
D	3		
U	0.2	X (field)	
N	1		

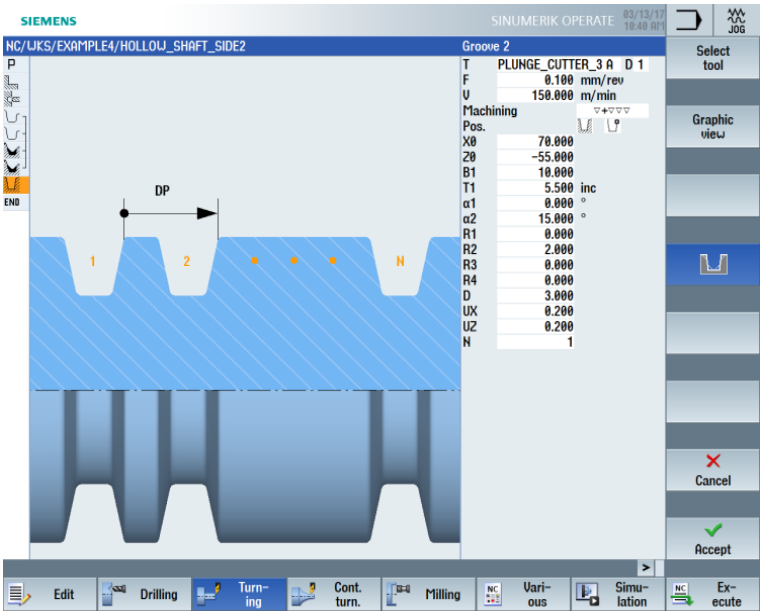


Figure 9-57 Specifying the groove

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

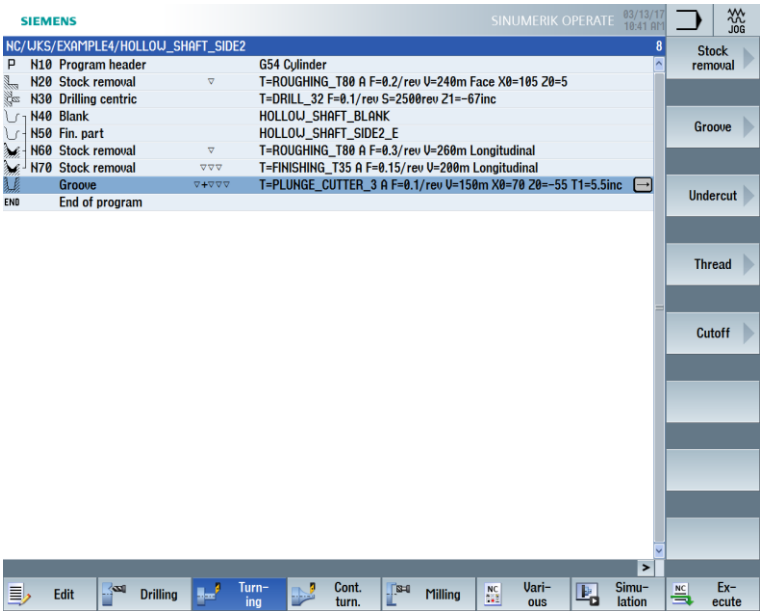
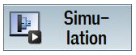


Figure 9-58 Machining plan after groove



Start the "Simulation".

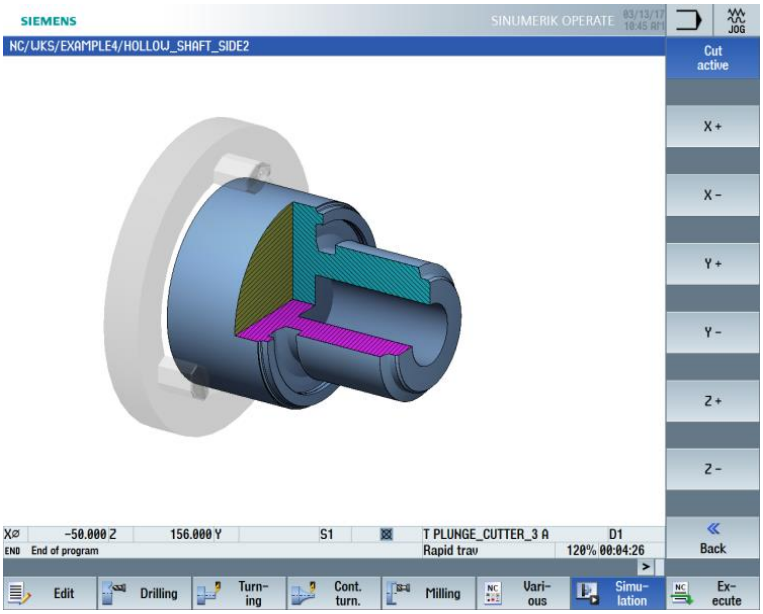
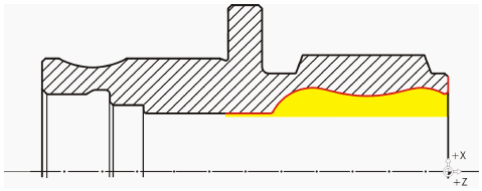


Figure 9-59 Simulation – 3D view (cut active)

10.3.6 Finished part contour of the second side, internal

Operating sequences

Follow the steps below to enter the finished part contour:



Select the "Cont. turning" softkey.

Select the "New contour" softkey. Enter the name 'HOLLOW_SHAFT_SIDE2_I' for the contour.

Accept your input.

Set the starting point to X57/Z0.

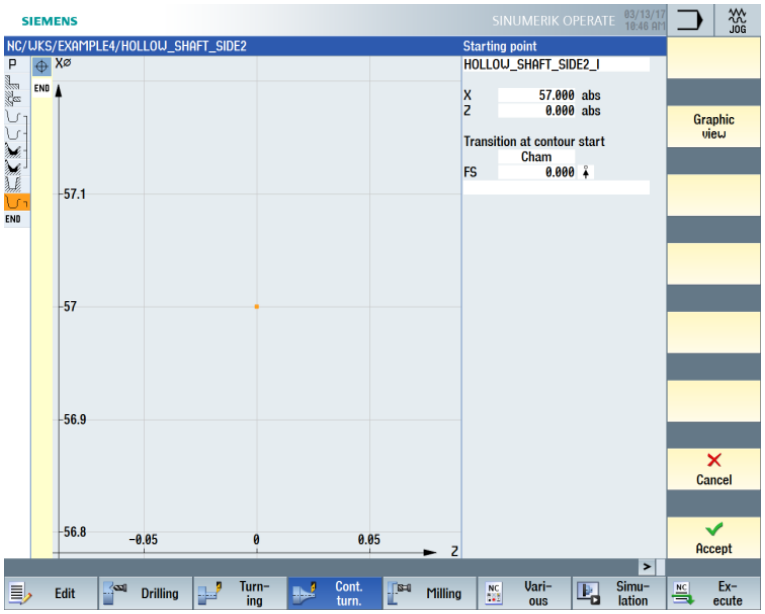


Figure 9-60 Entering the starting point for the contour

Accept your input.

Create the contour on your own (see figure below).

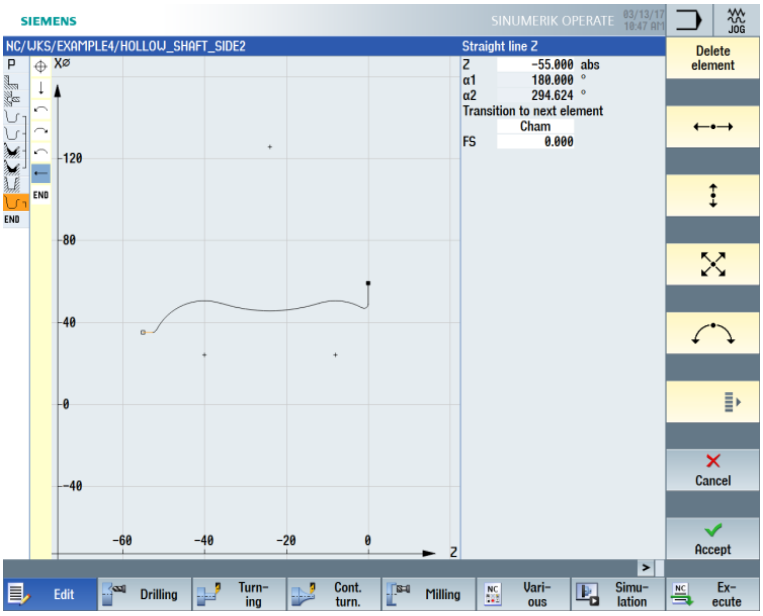
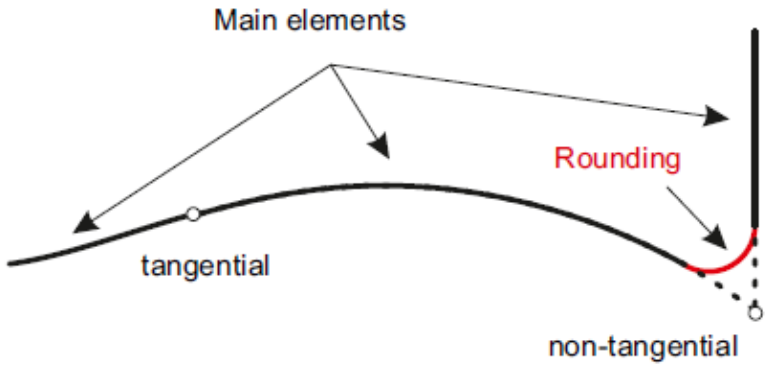


Figure 9-61 Finished part contour of the second side, internal

Note:

When creating the contour, ensure that the arc elements merge tangentially.

The tangential transition applies only to main elements, i.e. the rounding is attached to the main element (see figure below).



Accept

Accept the contour. Once applied of the contour, your work step program looks like this:

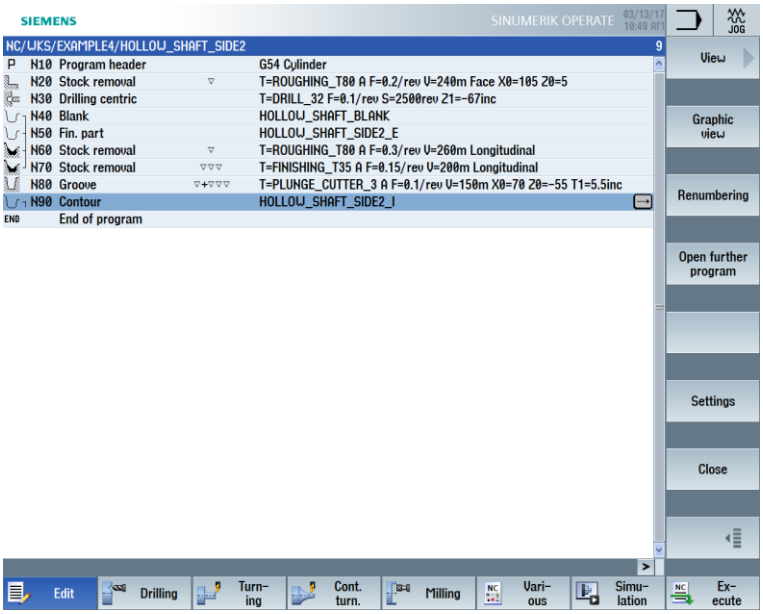


Figure 9-62 Machining plan after input of the contour

Stock removal, residual stock removal and finishing

You machine the contour in the next machining step:

Stock removal

Select the "Stock removal" softkey.

Select tool

Open the tool list and select the "ROUGHING_T80 I" tool.

OK

Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.25		
V	280 m/min	X	
Machining	Roughing Longitudinal Inside	X X X	You must switch the machining to "Inside".
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		

Field	Value	Selection via toggle key	Notes
BL	Cylinder	X	Because drilling has already been performed, you do not have to take a black contour into account for the internal machining. Switch to Cylinder.
XD	32 abs	X	
ZD	0 abs	X	
Relief cuts	No	X	
Limit	No	X	

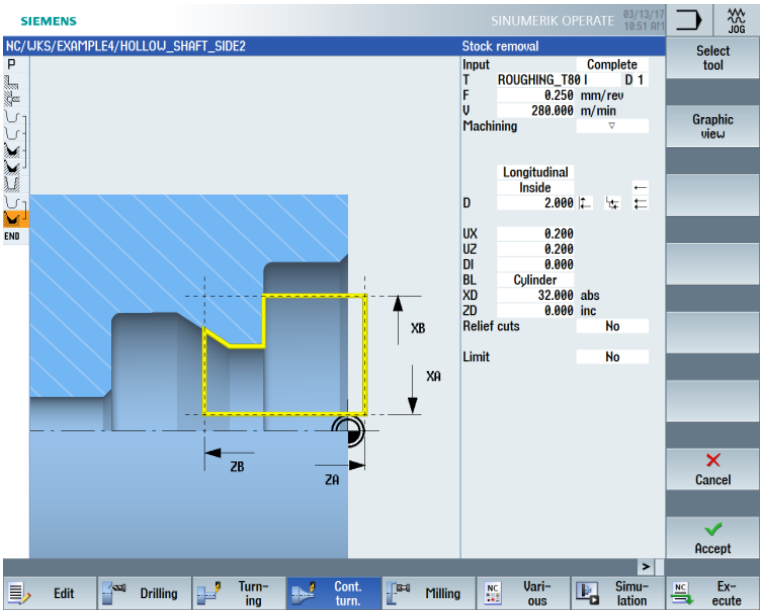


Figure 9-63 Roughing the contour

Accept the entered values.

Select the "Cut resid stock" softkey.

Open the tool list and select the "FINISHING_T35 I" tool.

Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.2		
V	240 m/min	X	
Machining	Finishing Longitudinal Inside	X X X	
Allowance	No	X	
Relief cuts	Yes	X	
FR	0.2		
Limit	No	X	

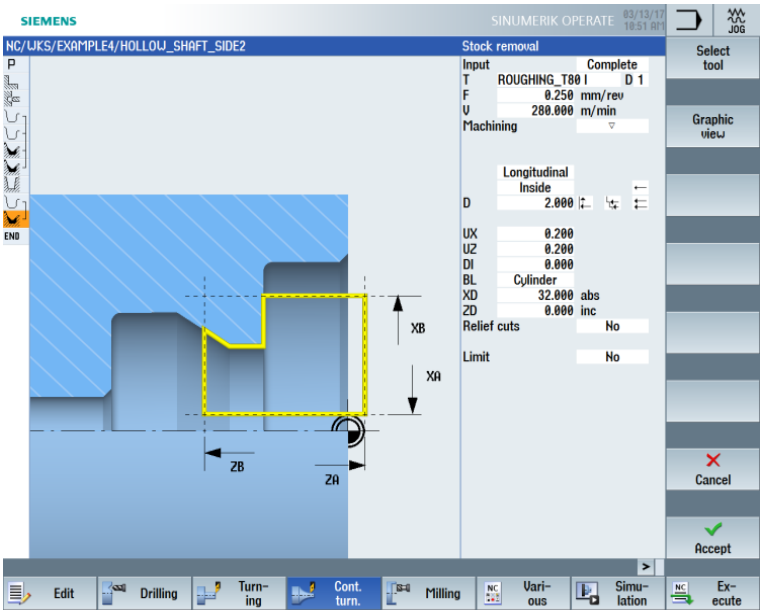


Figure 9-64 Residual stock removal from contour

Accept the entered values.

Select the "Stock removal" softkey.

Open the tool list and select the "FINISHING_T35 I" tool.

Apply the tool to the program.

Enter the following values for finishing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.12		
V	280 m/min	X	
Machining	Finishing Longitudinal Inside	X X X	
Allowance	No	X	
Relief cuts	Yes	X	
Limit	No	X	

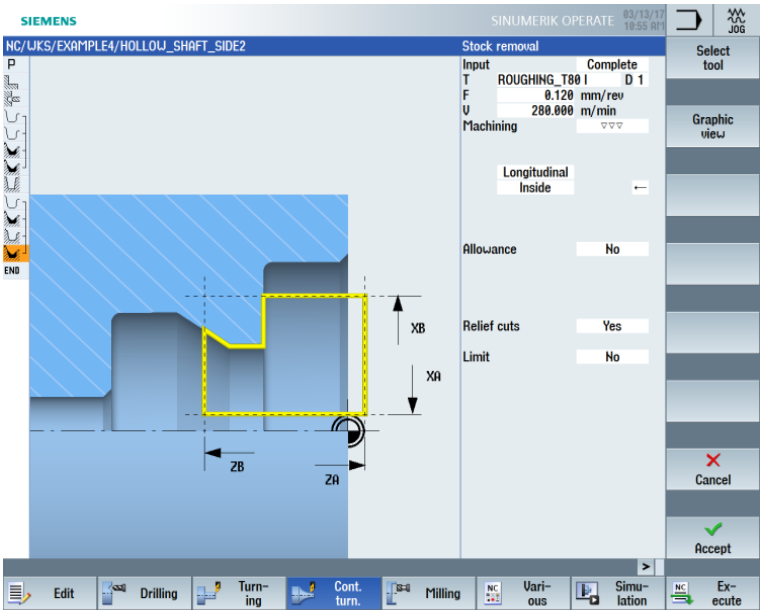


Figure 9-65 Finishing the contour

Accept the entered values.

Start the simulation to perform checks.

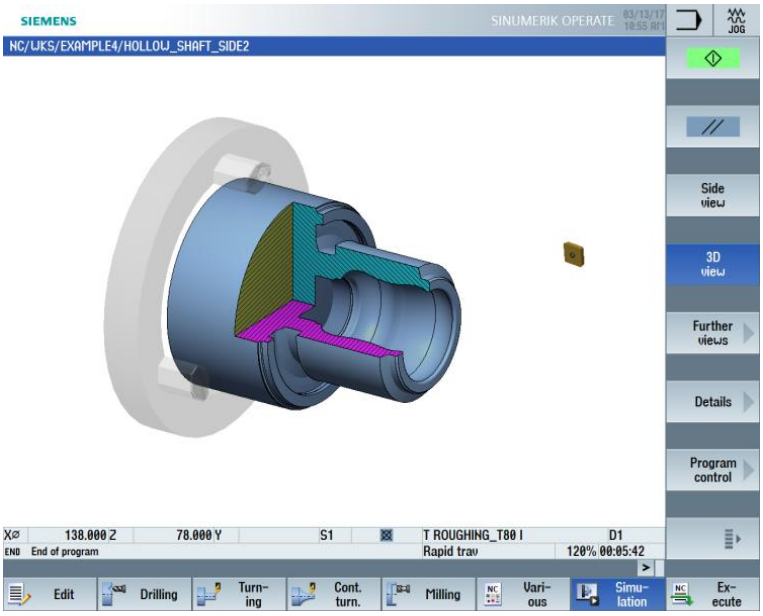


Figure 9-66 Simulation – 3D view (cut active)

11. Example 5: Plunge-turning

11.1 Overview

Learning objectives

In this section, you will learn how to use the plunge-turning functions.

Task

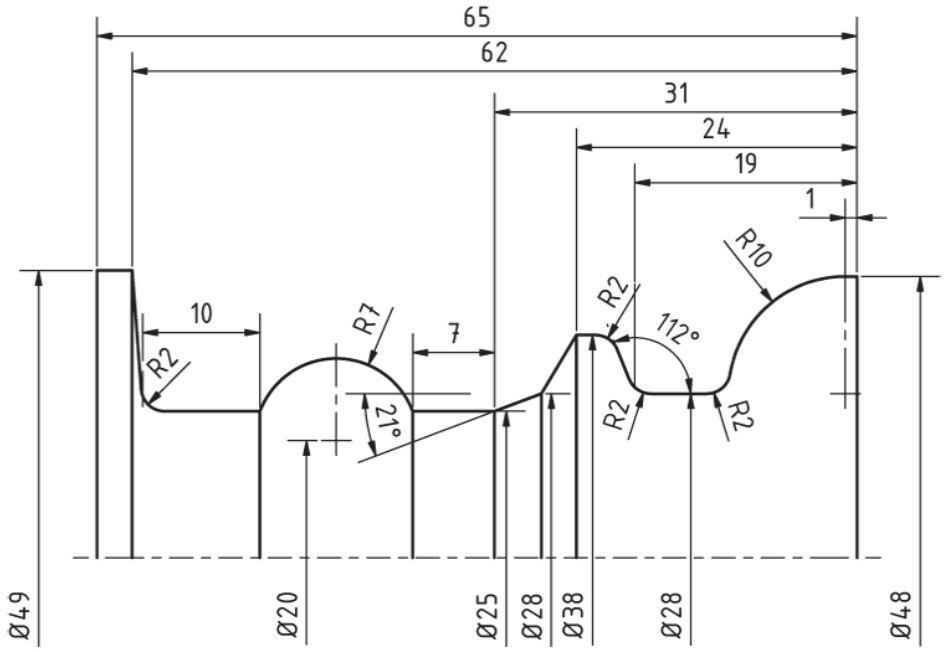


Figure 10-1 Workshop drawing – Example 5

Preparations

Perform the following steps on your own:
1. Create a new workpiece with the name "EXAMPLE5".
2. Create a new sequential program with the name "PLUNGE_TURNING".
3. Fill in the program header (see figure below).

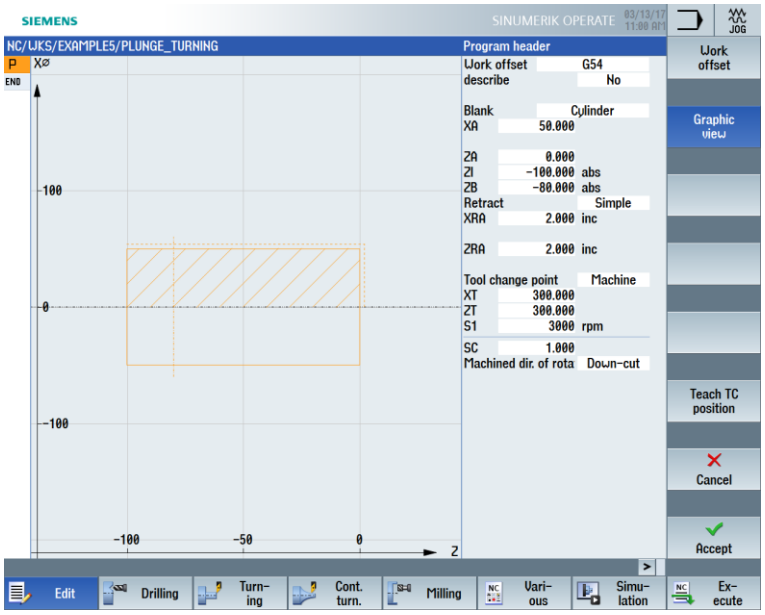


Figure 10-2 Creating the program header

11.2 Plunge-turning

The level of productivity that can be achieved during turning is limited, among other things, by the possible number of tools in the turret and the frequent tool changes required for effective turning machining. Not all possible contours can be produced with standard turning tools alone. The residual material machining, therefore, is often performed using grooving. For the complete machining of a contour, therefore, it is always necessary to switch between standard turning tools and grooving tools.

The aim of the plunge-turning cycle is to reduce the number of tool changes and to avoid empty cuts, such as those that occur during the backward movements of the turning tool, for example.

As a general rule, there are hardly any empty cuts during the plunge-turning cycle, as stock is removed during both forward and backward movement. This must be taken into account during program creation. ShopTurn offers optimum support for this. As you already know, you only need to describe the contour of the turned part, and for the stock removal cycle you can select whether you want to machine with conventional methods or with grooving or plunge-turning. ShopTurn automatically calculates the cuts and traversing movements of the tool based on the cycle. This means that empty cuts are eliminated to a great extent.

During the simulation, you can effectively analyze the calculated traversing movements of the tool. Even a combination of conventional turning machining and plunge-turning is possible. For example, a standard tool can be used for roughing, while plunge-turning can be used for machining residual material. This means the contour can be machined completely and without damage.

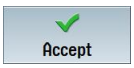
11.3 Creating the contour

Operating sequences

Create the contour on your own.

Select the "Cont. turning" softkey.

Select the "New contour" softkey. Enter the name 'CONTOUR_E' for the contour.



Accept your input.

Set the starting point to X48/Z0.

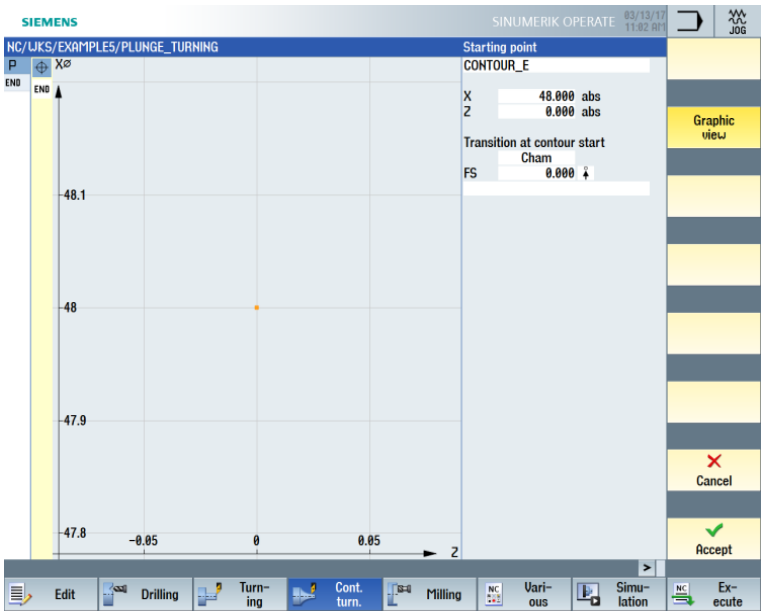


Figure 10-3 Entering the starting point for the contour

Create the contour (see figure below).

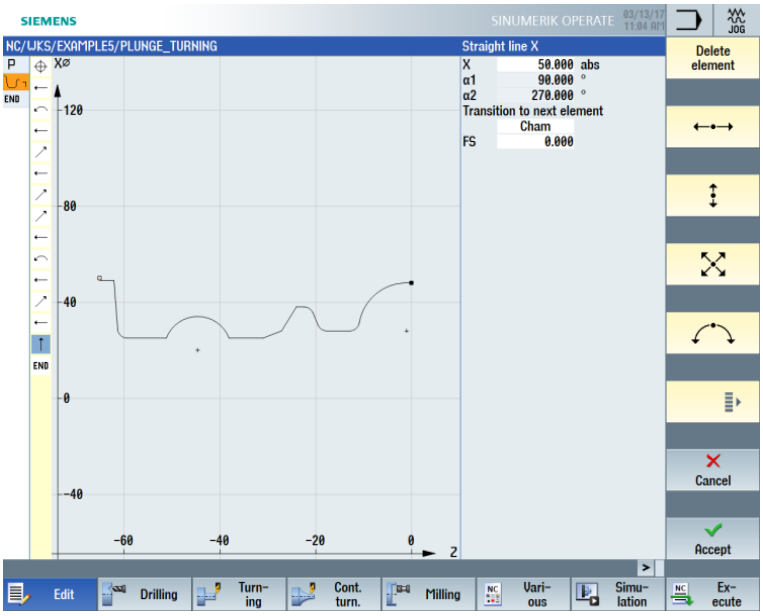


Figure 10-4 Contour in the contour calculator

11.4 Stock removal with the plunge-turning cycle

Operating sequences

You machine the contour in the next machining step.

Proceed as follows:

Select the "Cont. turning" softkey.

Select the "Plunge turning" softkey.

Open the tool list and select the "PLUNGE_CUTTER_3 A" tool.

Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
FX	0.2		
FZ	0.25		
V	150 m/min	X	
Machining	Roughing Longitudinal Outside	X X X	
D	2.5		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Cylinder	X	
XD	50 abs	X	
ZD	0 abs	X	
Limit	No	X	
N	1		

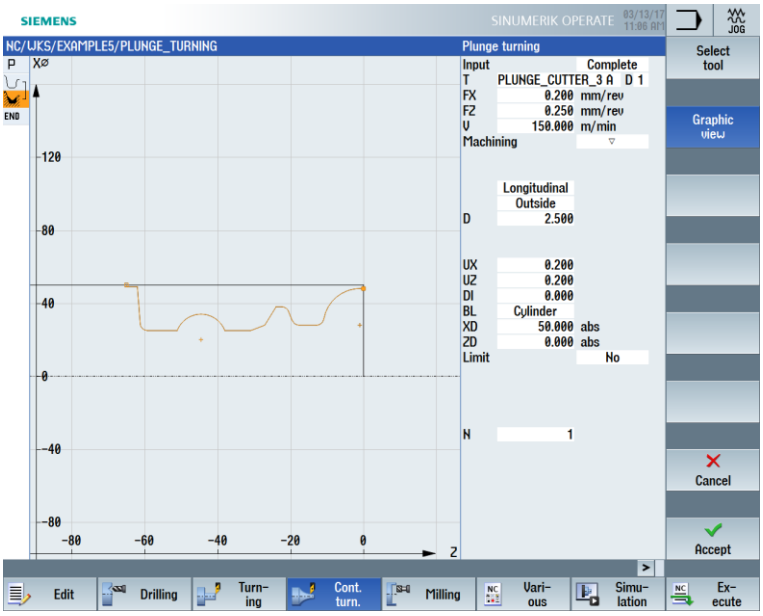


Figure 10-5 Roughing the contour

Accept the entered values.

Select the "Part" softkey.

Open the tool list and select the "PLUNGE_CUTTER_3 A" tool.

OK

Apply the tool to the program.

Enter the following values for finishing in the screen form:

Field	Value	Selection via toggle key	Notes
FX	0.15		
FZ	0.15		
V	200 m/min	X	
Machining	Finishing Longitudinal Outside	X X X	
Allowance	No	X	
Limit	No	X	
N	1		

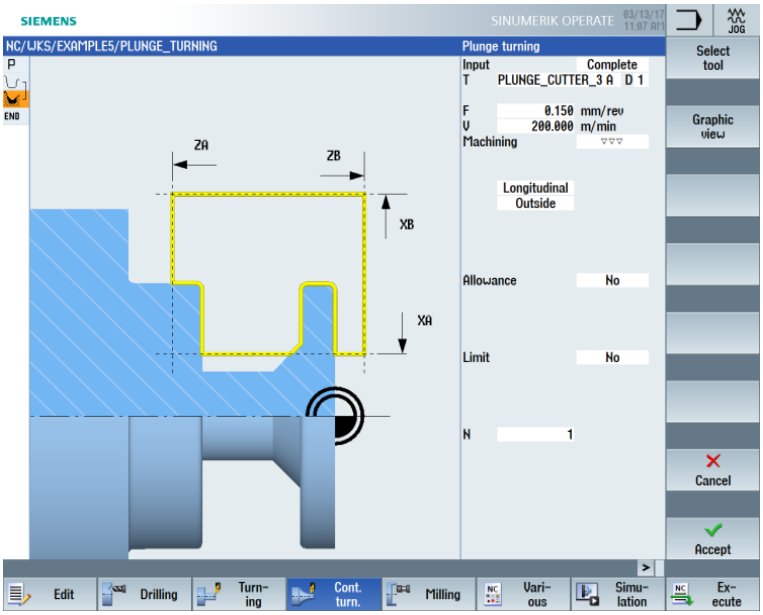


Figure 10-6 Finishing the contour

Accept the entered values. Once applied, your machining step program looks like this:

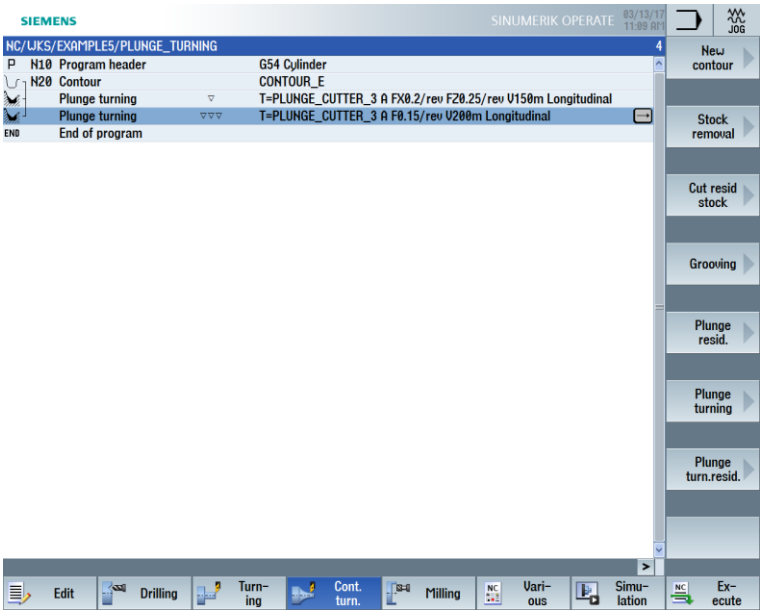


Figure 10-7 Machining step program

Select the "Simulation" softkey.

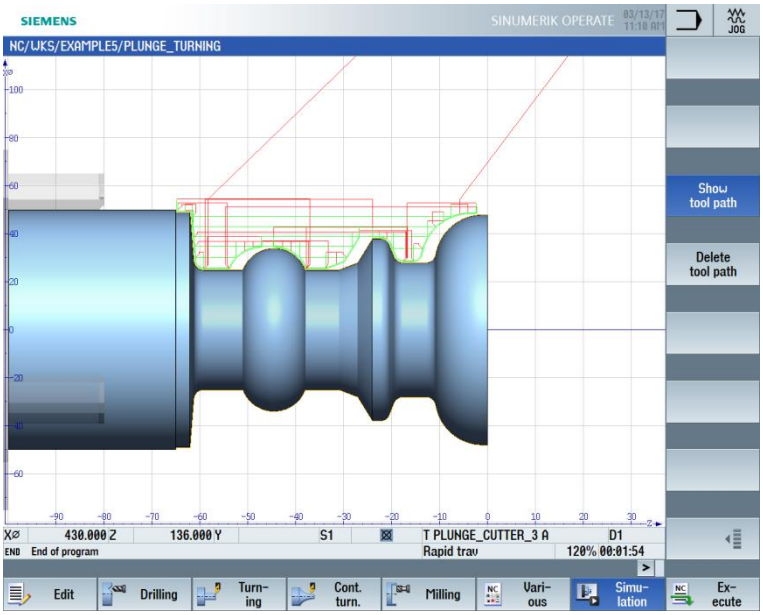


Figure 10-8 Simulation – Side view (with display of the traversing paths)

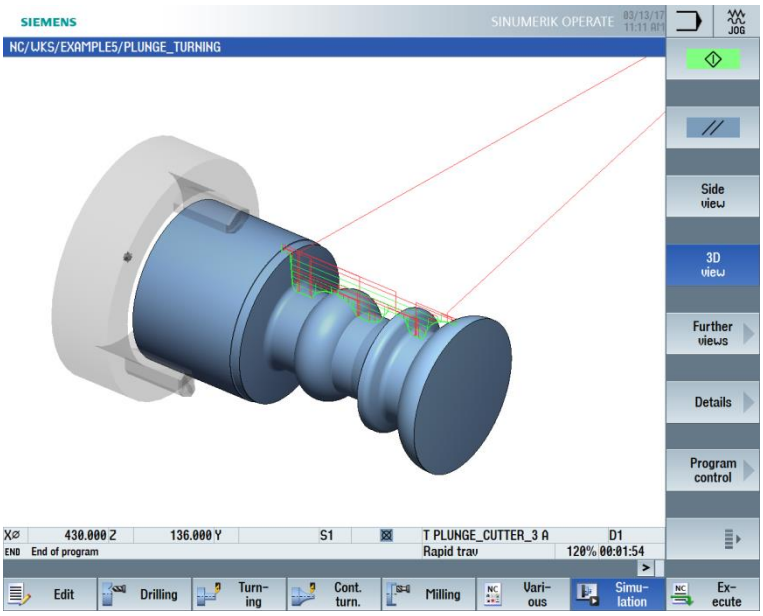


Figure 10-9 Simulation – 3D view (with display of the traversing paths)

12. Machining the workpieces

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

The steps described below are required for the machining:

Reference point approach

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

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

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

Clamping the workpiece

For dimensionally-correct machining and, naturally, also for your own safety, secure chucking that is appropriate for the workpiece is required. A three-jaw chuck is usually used for this.

Setting the workpiece zero

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

The workpiece zero is usually determined in the Z axis through scratching with an offset tool.

Executing the machining plan

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

First, select the program you want to execute in the Program Manager, e.g. "HOLLOW_SHAFT_SIDE2".

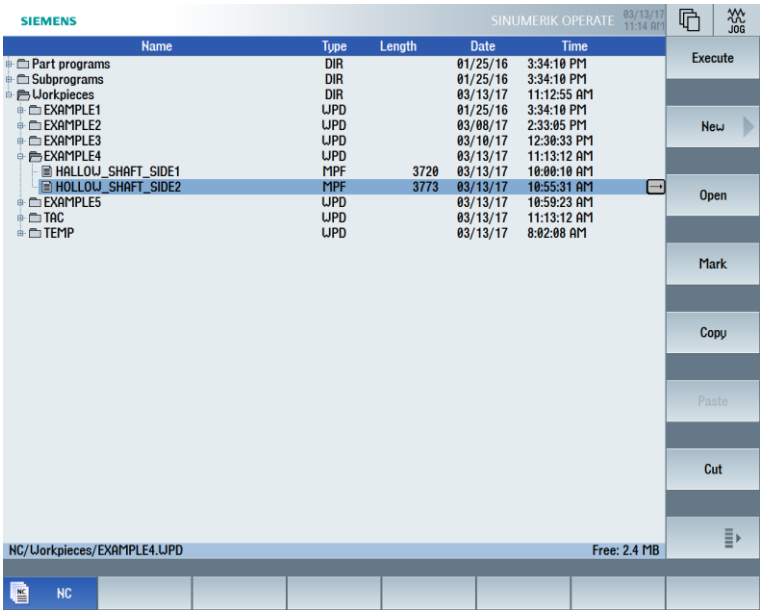


Figure 11-1 Selecting the program

Open the program.

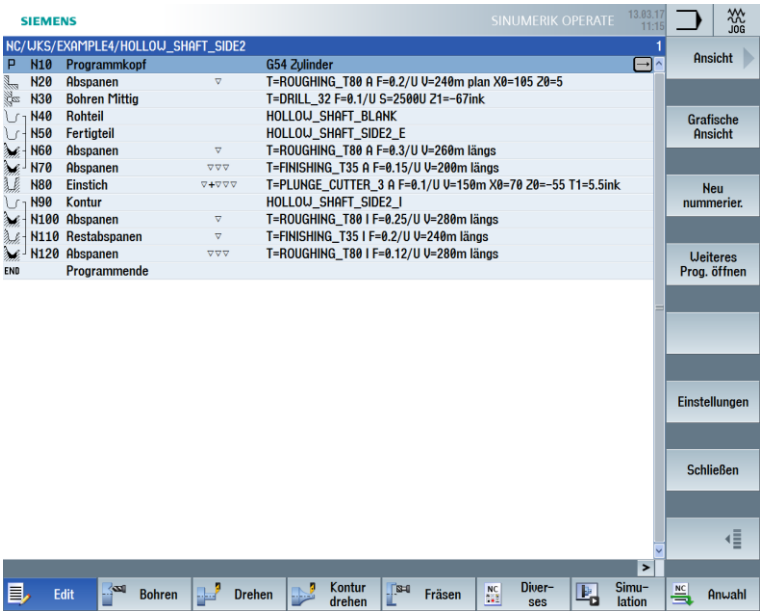


Figure 11-2 Opening the machining plan

Select the "NC Execute" softkey.

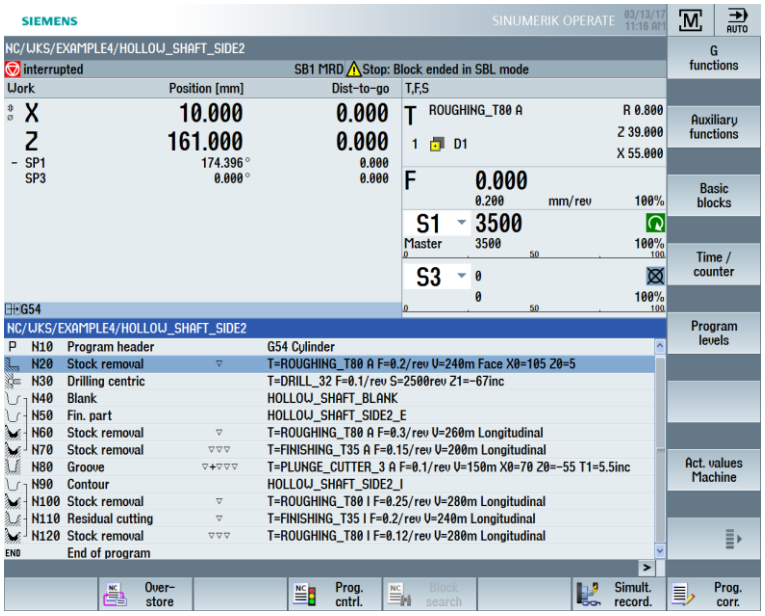
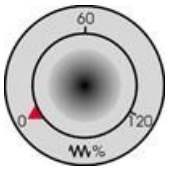


Figure 11-3 Executing



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

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

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

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

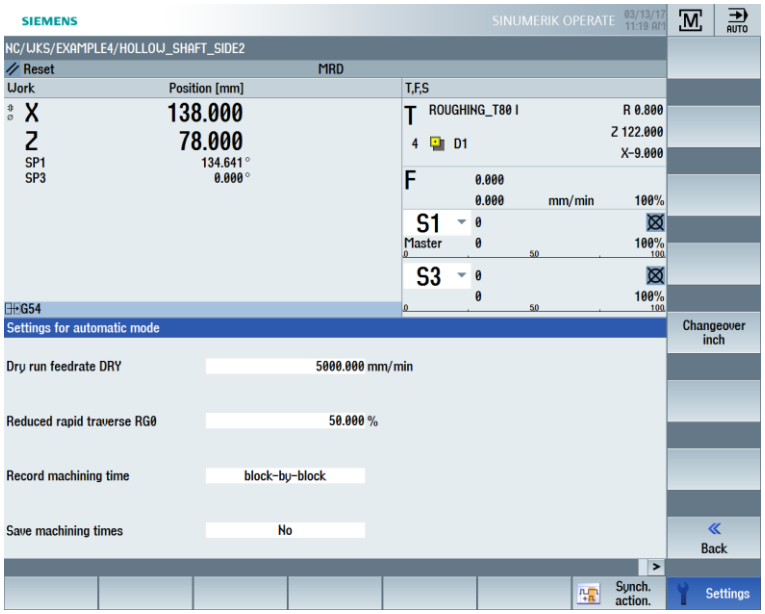


Figure 11-4 Setting RG0

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

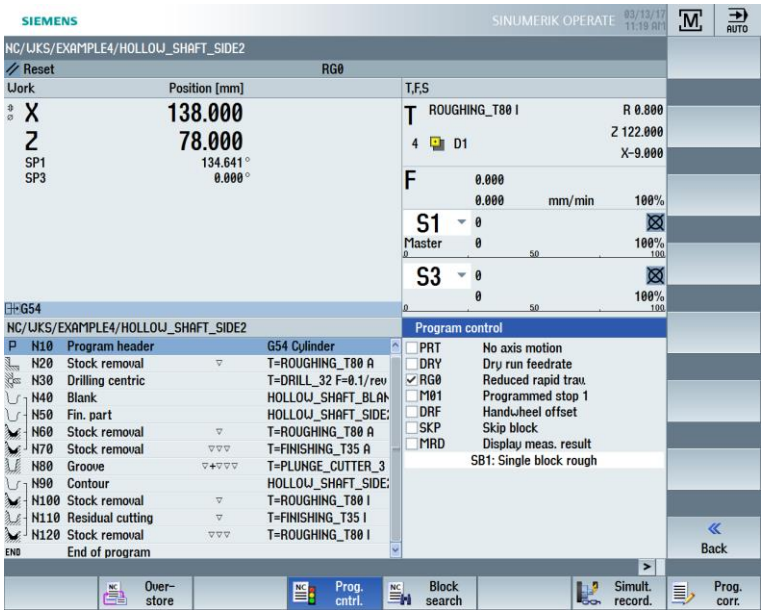


Figure 11-5 Selecting RG0

OPERATE keyboard shortcuts

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

Figure 11-6 Keyboard shortcuts

Keyboard shortcuts, continued

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

Figure 11-7 Keyboard shortcuts

Keyboard shortcuts, continued

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

Figure 11-8 Keyboard shortcuts