SIEMENS

SINUMERIK Operate

SinuTrain
Turning made easy with ShopTurn

Training Documents

09/2011

6FC5095-0AB80-1BP1

Legal information

Warning notice system

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

⚠ DANGER

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

⚠ WARNING

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

⚠ CAUTION

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

CAUTION

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

NOTICE

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

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

Qualified Personnel

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

Proper use of Siemens products

Note the following:

⚠ WARNING

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

Trademarks

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

Disclaimer of Liability

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

Siemens AG
Industry Sector
Postfach 48 48
90026 NÜRNBERG
GERMANY

Order number: 6FC5095-0AB80-1BP1
℗ 09/2011

Table of contents

Introduction 1

Faster from the drawing to the workpiece - but how?

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

The solution is: Creating a process plan instead of programming

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

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

Simpler and faster from the drawing to the workpiece - with ShopTurn.

Although ShopTurn is easy to learn, these ShopTurn Training Documents allow you to enter this world even faster. Before, however, it comes to actually working with ShopTurn, the first sections cover a few important basics:

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

The theory is followed by practical exercises with ShopTurn:

- Five examples have been chosen to explain the machining possibilities with ShopTurn, whereby the degree of difficulty is increased continuously. At the beginning, all key actions are specified; later you will be prompted to proceed without help.
- Then you will learn how to cut in AUTOMATIC mode using ShopTurn.
- If you wish, at the end you may test your knowledge of ShopTurn.

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

Just as ShopTurn itself was created with the help of skilled workers, these Training Documents were also drawn up by practical users. On that note, we wish you much pleasure and success in your work with ShopTurn.

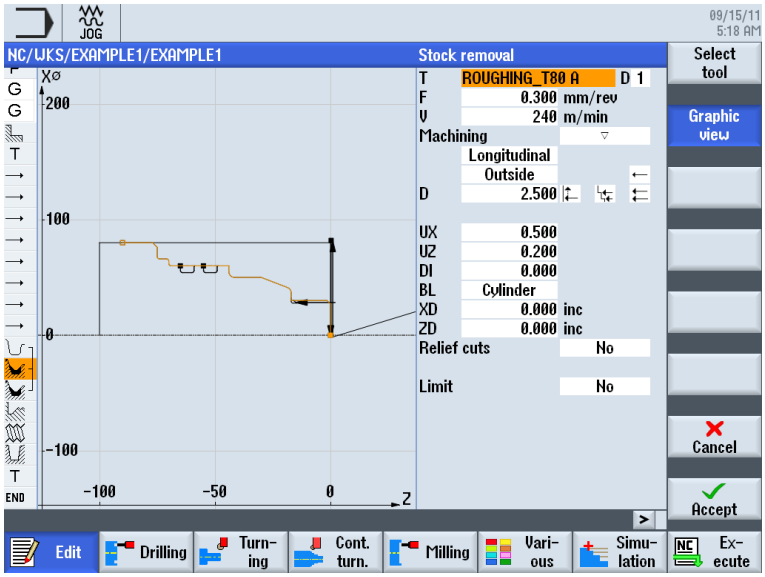
Advantages of working with ShopTurn

2

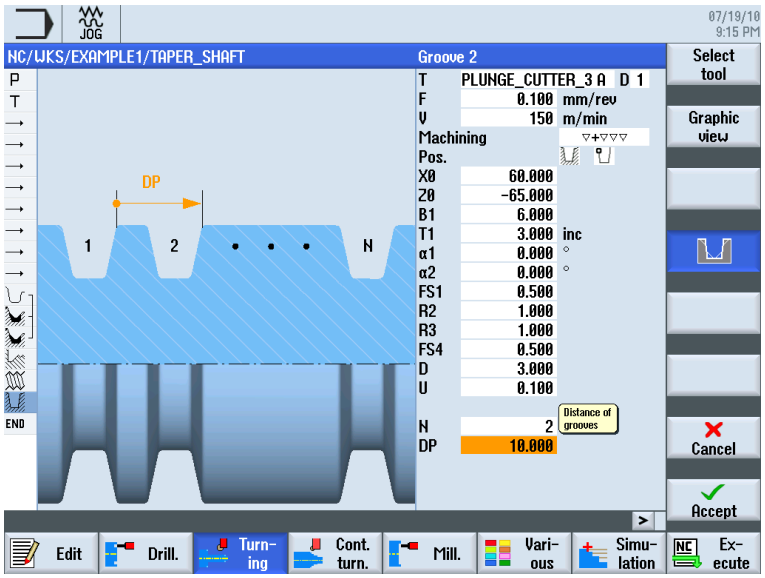
This section shows you the special advantages of working with ShopTurn.

2.1 You will save time for training....

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



- When working with ShopTurn, you are offered clear assistance by colored help displays.



- You can also integrate DIN/ISO commands into the **Graphical Process 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 work step and the workpiece graphic (broken-line graphics) at any time during the creation of a process plan.

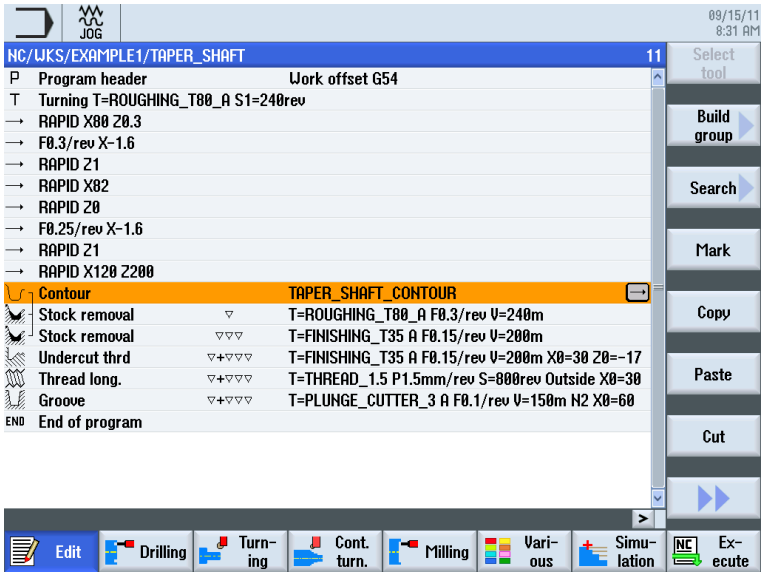


Figure 2-1 Work step in a process plan

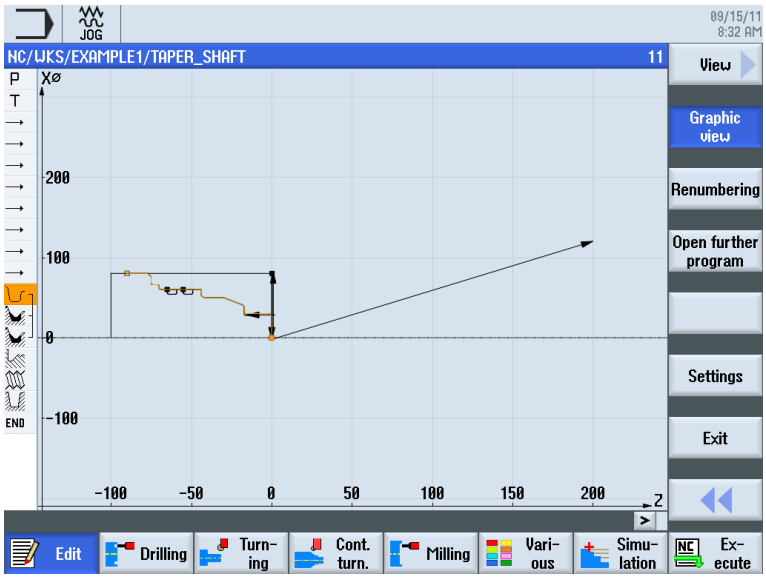
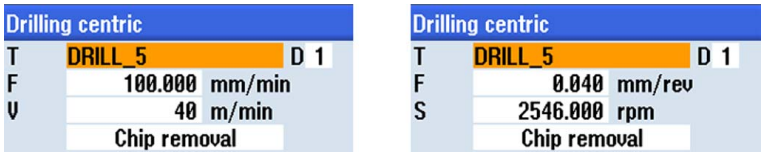


Figure 2-2 Broken-line graphics

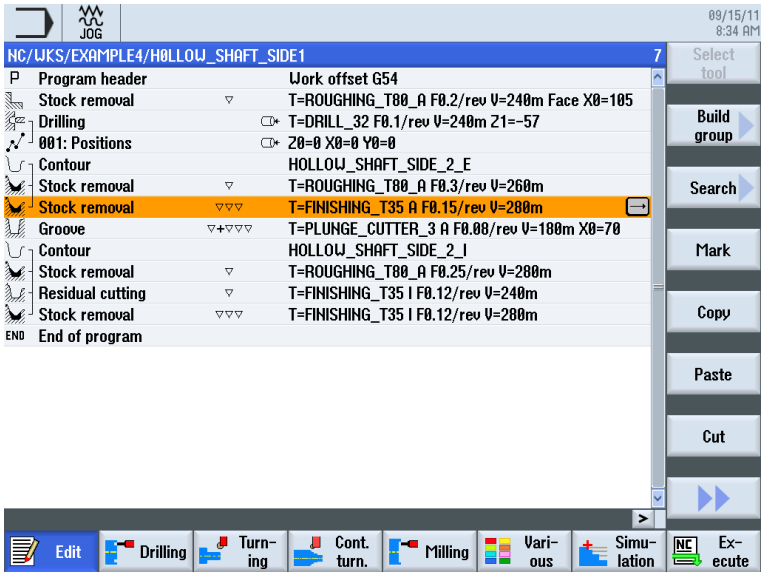
2.2 You will save time for programming...

- ShopTurn offers optimum assistance when it comes to entering 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 sequence using only one work step, and the required positioning motions (in this case, from the tool change point to the workpiece and back) are created automatically.

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



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

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

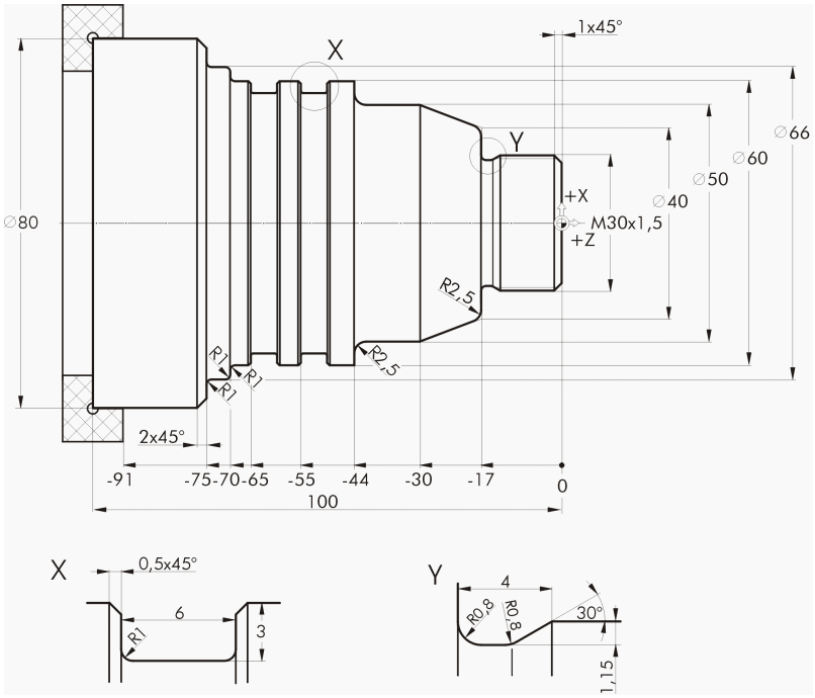


Figure 2-3 Technical drawing

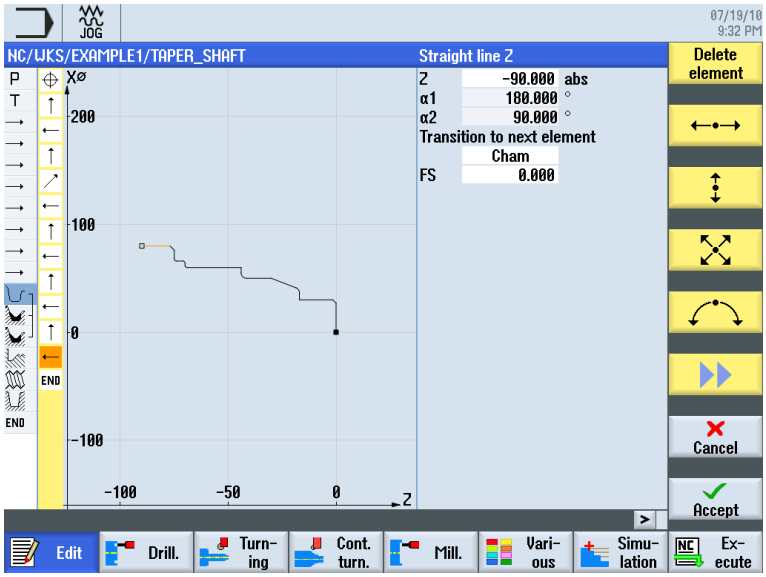


Figure 2-4 Screenform

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

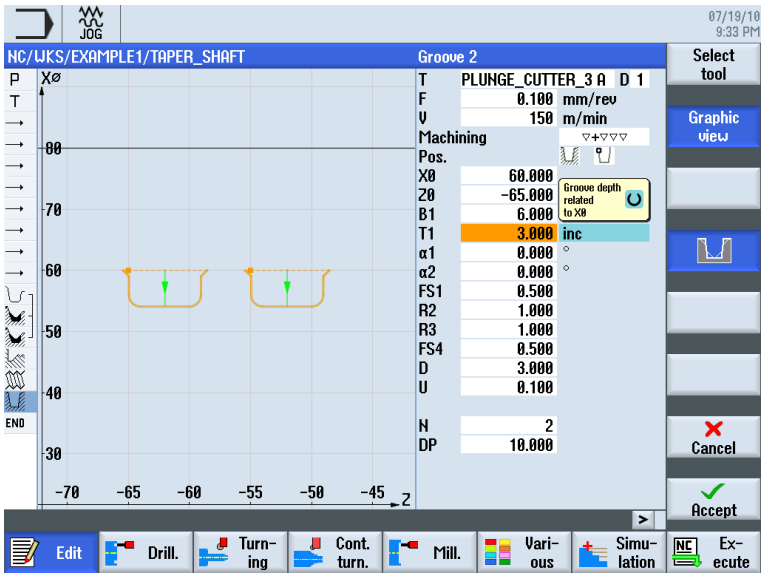


Figure 2-5 Graphic view

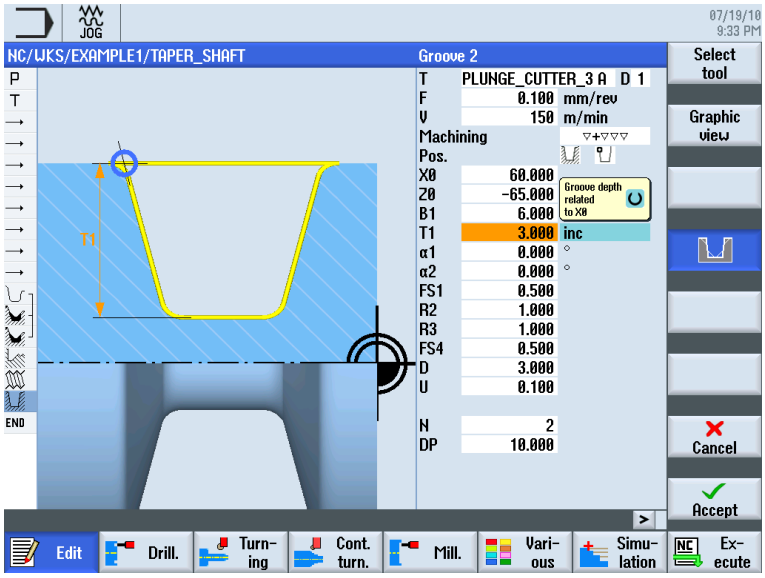


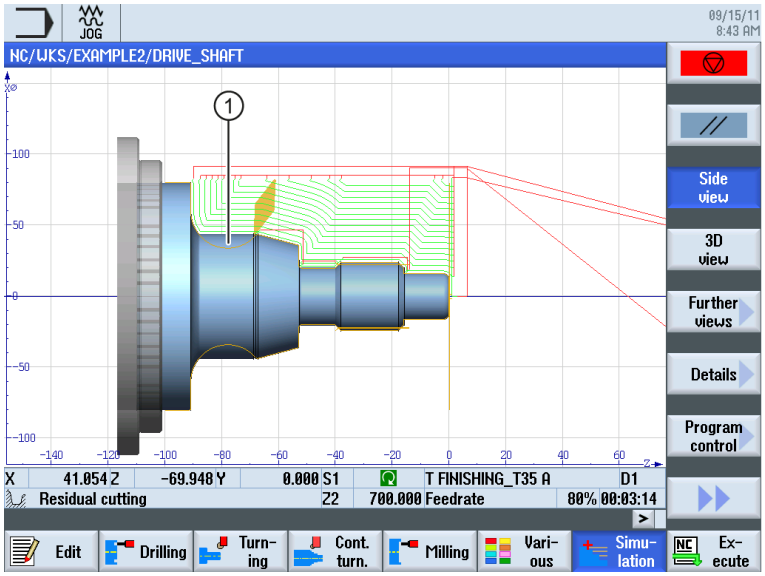
Figure 2-6 Help display

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

2.3 You will save time for manufacturing...

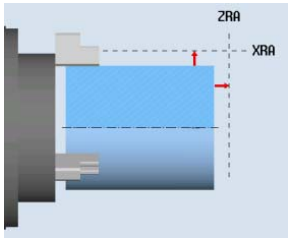
- You can optimize tool selection during the stock removal of contours:

  Large volumes are removed using roughing tools, any residual material ① is then detected and removed automatically using a pointed tool.

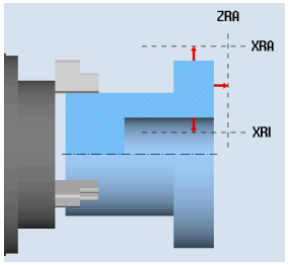


- The exact specification of the selected retraction plane avoids the use of unnecessary traversing paths and thereby saves valuable manufacturing time. This is possible using the settings **normal**, **extended** and **all**.

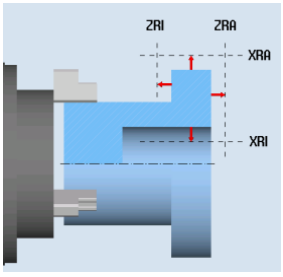
Retraction plane: normal



Retraction plane: extended



Retraction plane: all



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

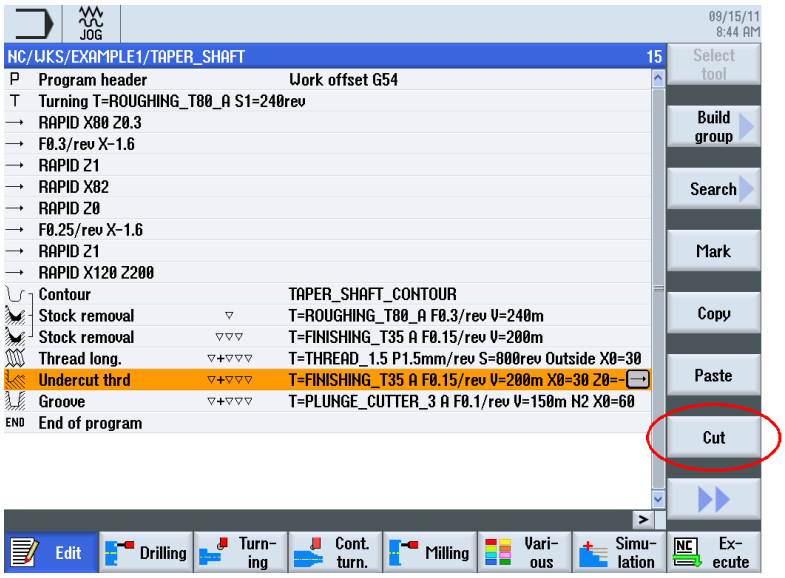


Figure 2-7 Original machining sequence

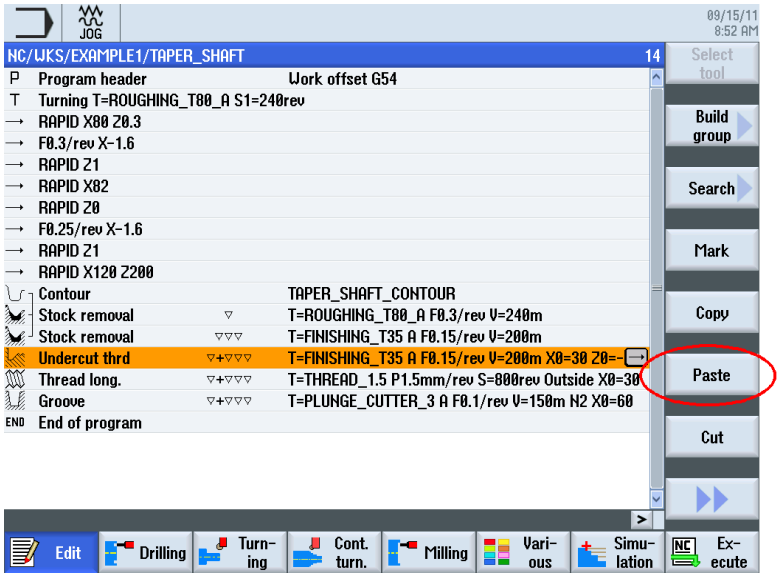


Figure 2-8 Optimized machining sequence with **cutting** and **pasting**

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

Ensuring everything runs smoothly

3

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

3.1 Using ShopTurn

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

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

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

All main functions can be called using the horizontal softkeys.

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

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

Main menu

3.2 The contents of the main menu

3.2.1 Machine

Machine - Manual

Select the "Machine" softkey.

Press the "JOG" key.

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

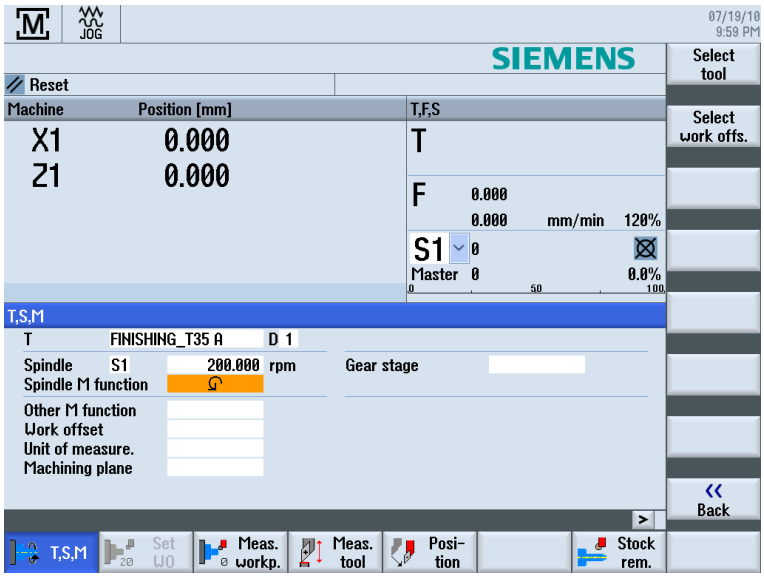


Figure 3-1 Call of a tool and input of technological values

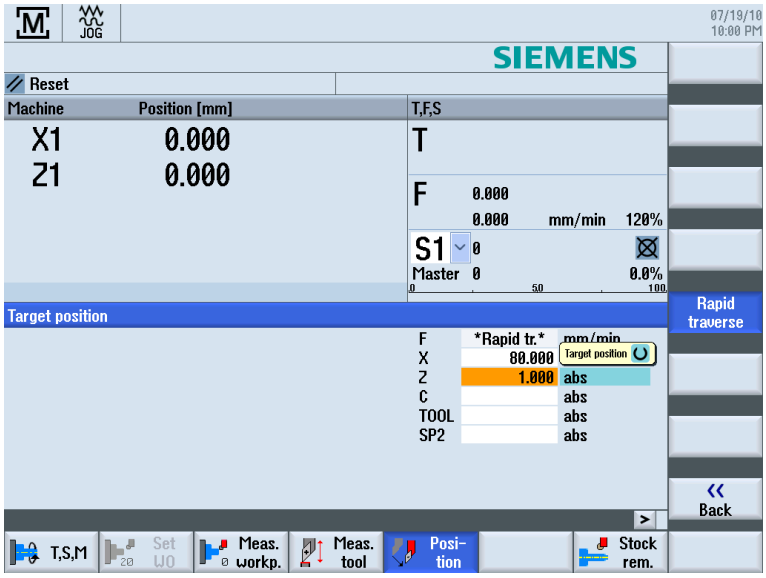


Figure 3-2 Input of a target position

Machine - AUTO

Select the "Machine" softkey.

Press the AUTO key.

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

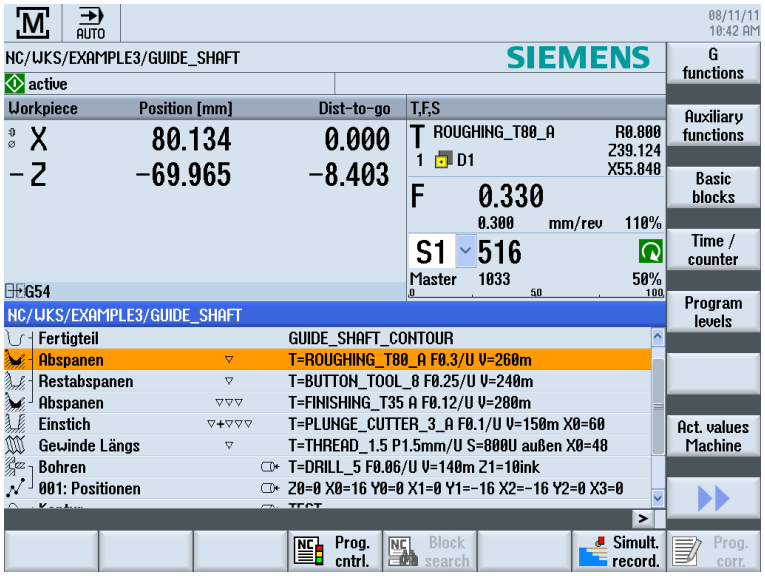


Figure 3-3 Executing a process plan

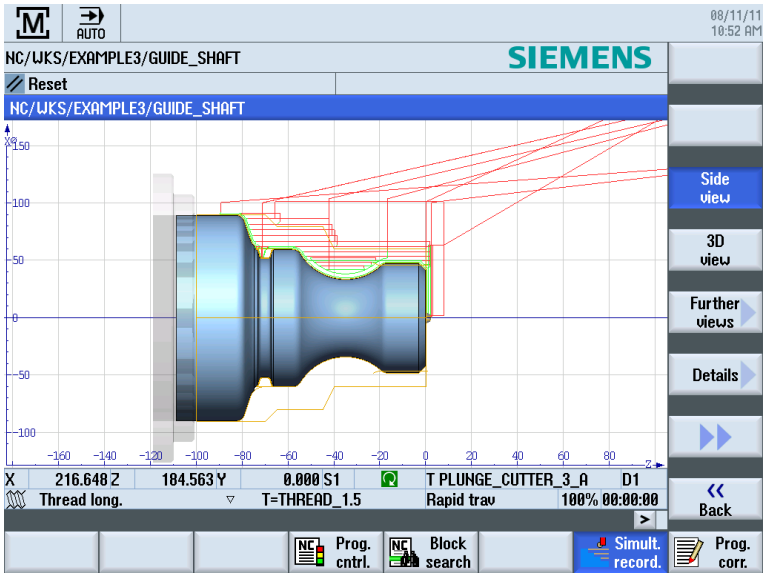
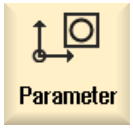


Figure 3-4 Simultaneous recording of the execution

3.2.2 Parameters

Parameter lists



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

Tool lists

No cutting without tools.

The tools can be managed in a tool list.

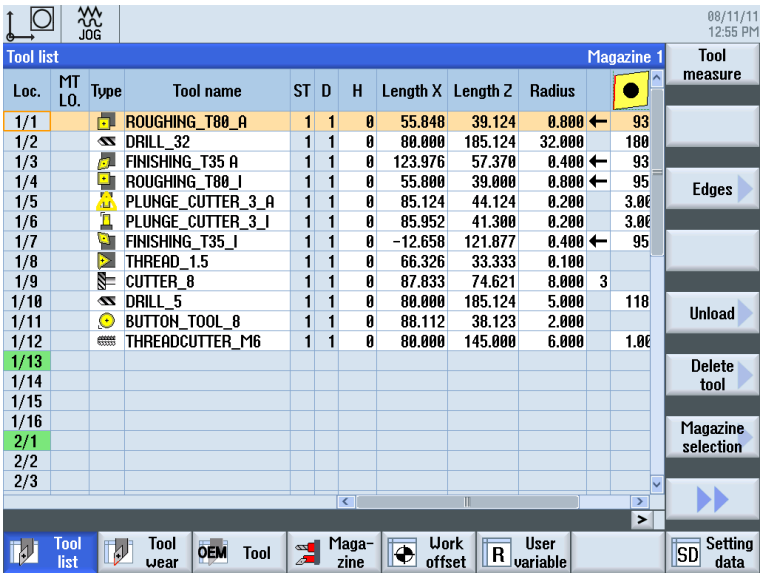


Figure 3-5 Tool list

Magazine

Tools can be organized into a magazine.

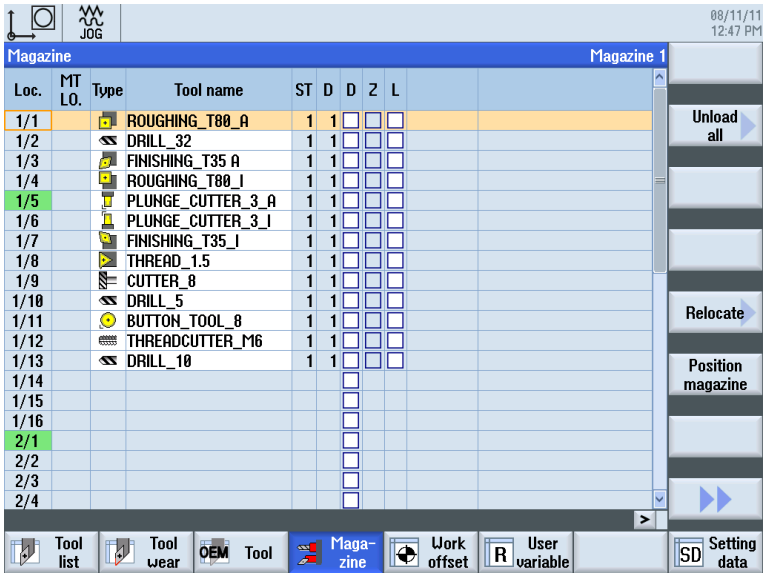


Figure 3-6 Magazine

Work offsets

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

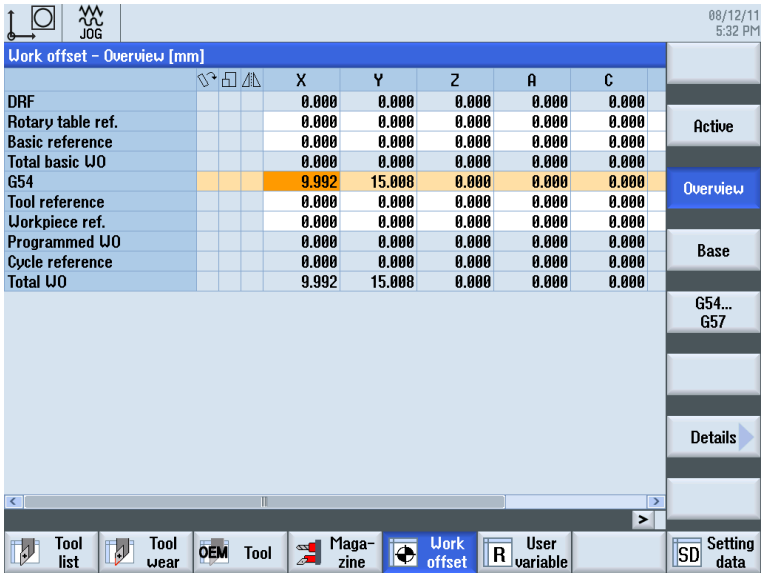


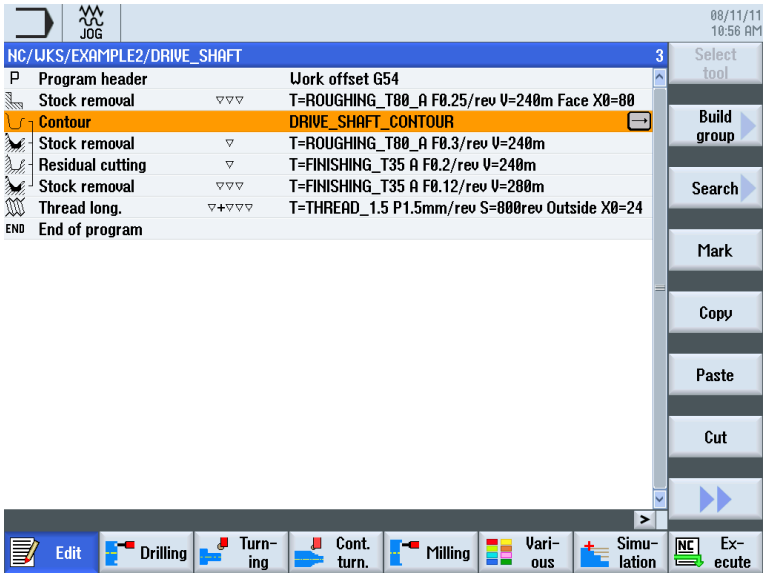
Figure 3-7 Work offsets

3.2.3 Program

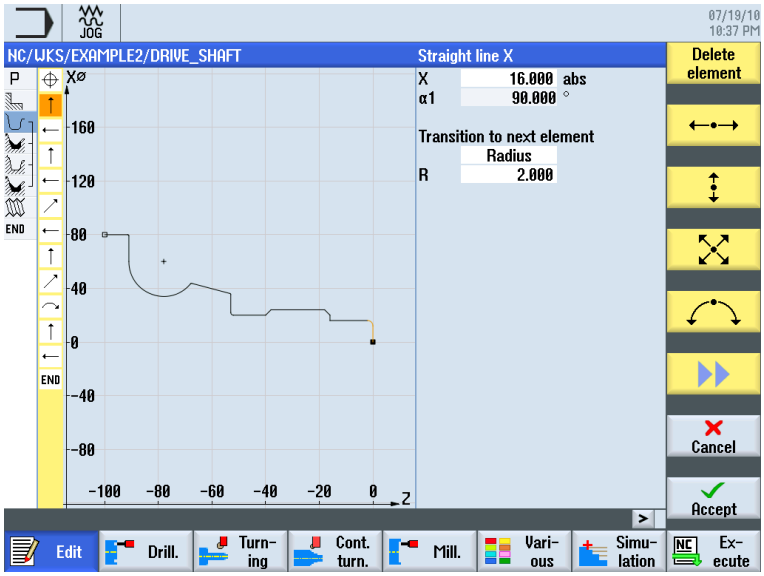
Editing programs

This key can be used to edit programs.

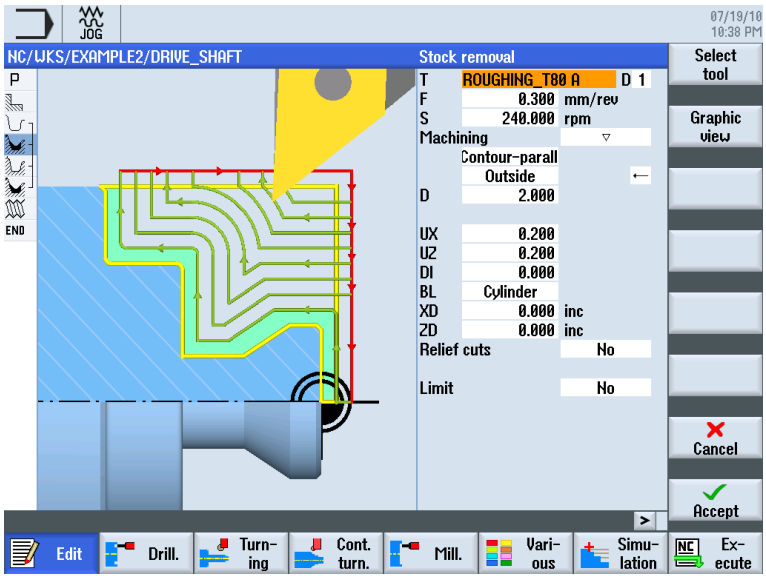
If you have created a **ShopTurn program** in the Program Manager, you can now create the process plan with the complete machining sequence for the appropriate workpiece. An optimum sequence can only be achieved by a skilled worker with suitable experience.



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



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



Example for the dovetailing of geometry and technology:

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

Simulating programs

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

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

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

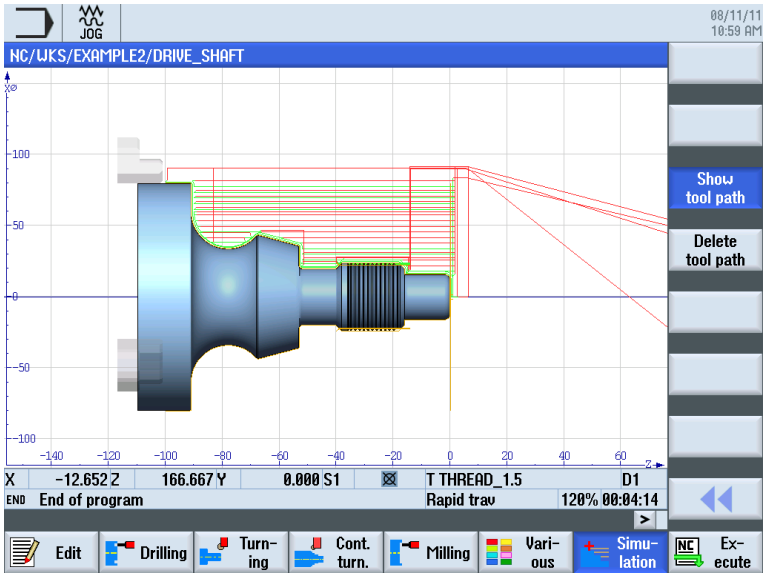


Figure 3-8 Side view (display tool path, activated)

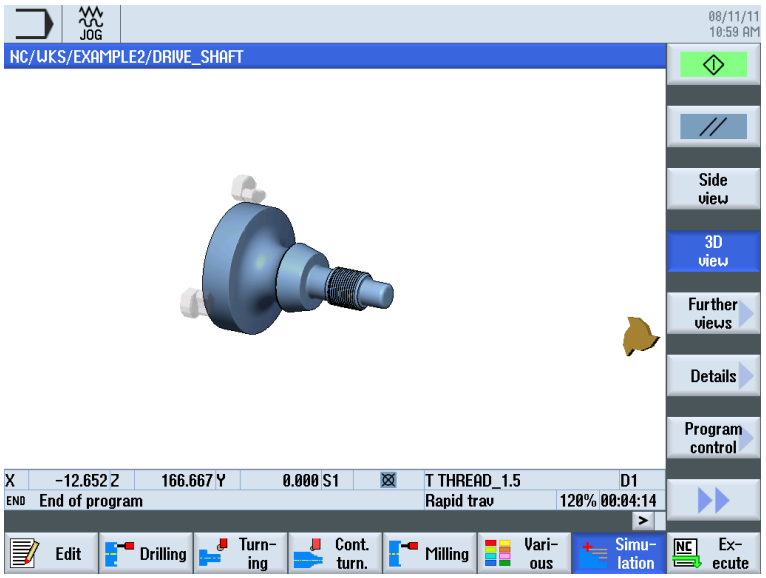


Figure 3-9 3D view

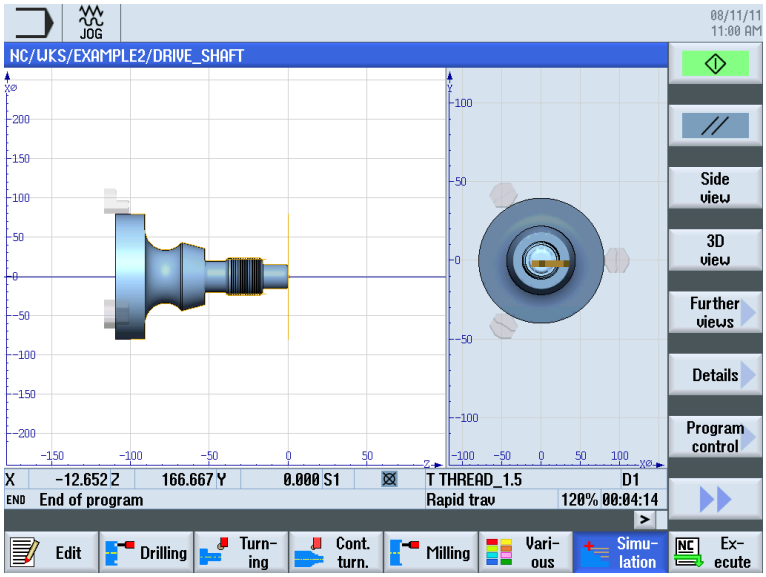
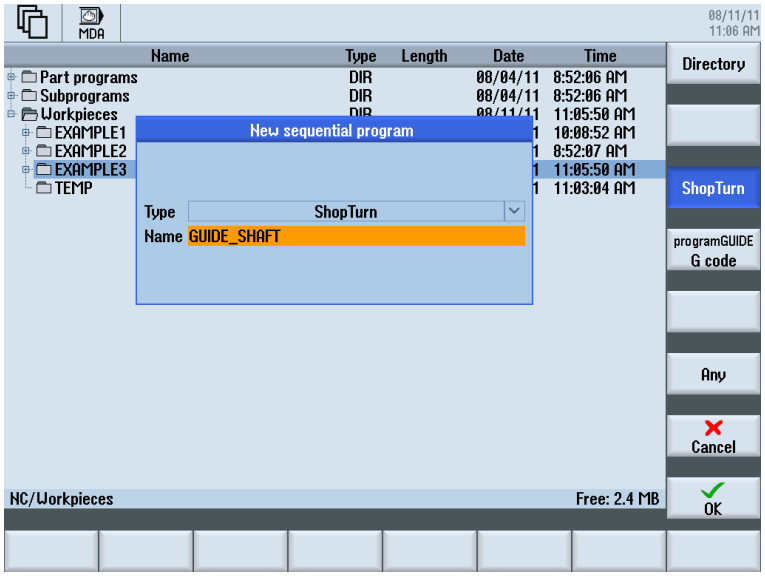


Figure 3-10 2-window view

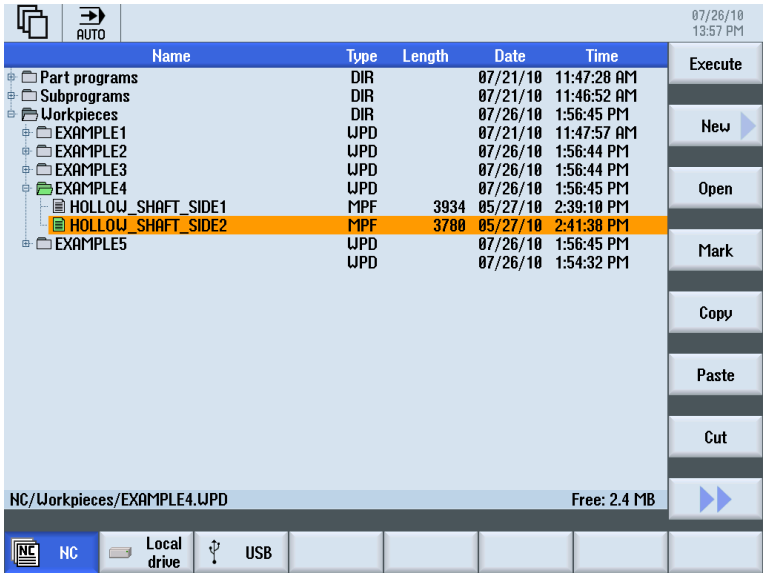
3.2.4 Program Manager

Managing programs

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



Active programs are marked with a green symbol.



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

Creating a new workpiece

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

Creating a new program

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

ShopTurn program

G code program

3.2.5 Diagnosis

Alarms and messages

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

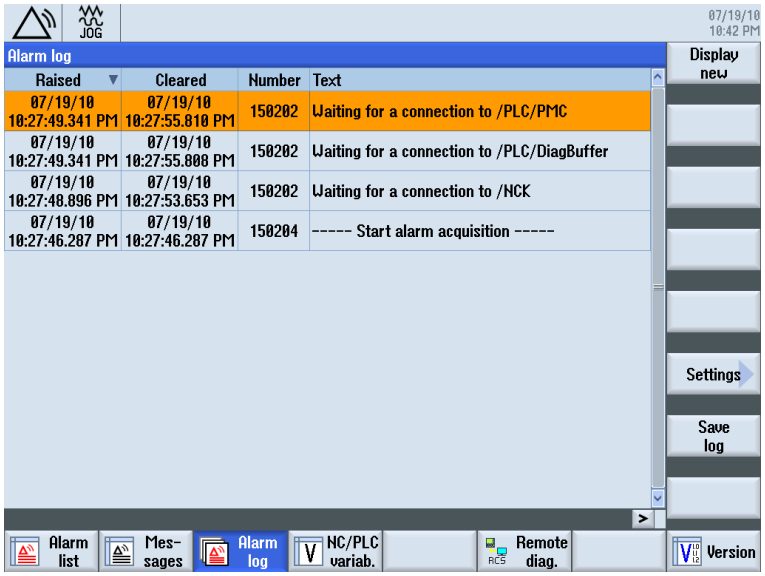


Figure 3-11 Alarm log

Basics for beginners

4

This section explains the general basics of the geometry and technology for turning. No inputs for ShopTurn are planned yet.

4.1 Geometrical basics

4.1.1 Tool axes and work planes

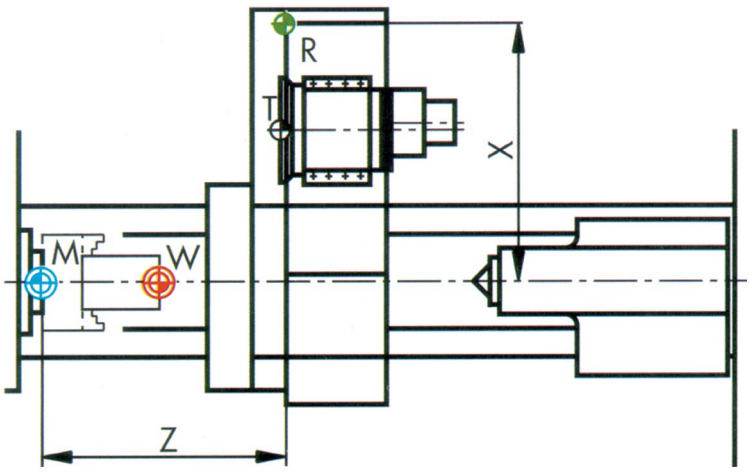
During turning, it is the workpiece, and not the tool, that rotates. The axis is the Z axis.

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

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

4.1.2 Points in the work space

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



Machine zero (M)

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

Workpiece zero (W)

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

Reference point (R)

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

Toolholder reference point T

Toolholder reference point T is used to set up machines with tool revolvers with default tools. Its position and location hole permit setup with cutter holders for shank tools in accordance with DIN 69880 and VDI 3425.

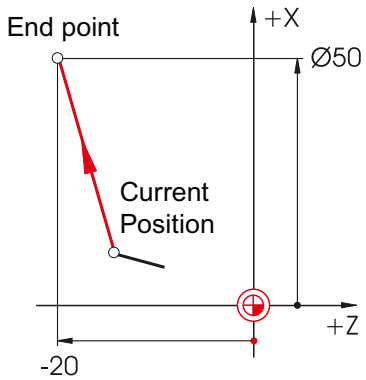
4.1.3 Absolute and incremental dimensioning

Absolute input

The entered values refer to the workpiece zero.

Straight		
X	50.000	abs
Y		abs
Z	-20.000	abs

* G90 Absolute dimensions



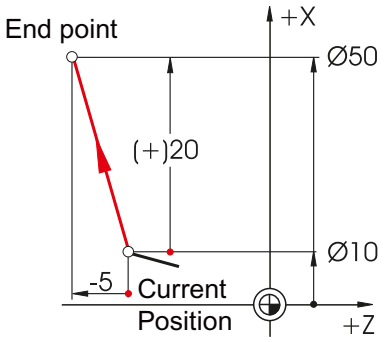
With absolute inputs, the **absolute** coordinate values of the **end point** must always be entered (the current position is not taken into account).

Incremental input

The entered values refer to the current position.

Straight		
X	20.000	inc
Y		abs
Z	-5.000	inc

* G91 Incremental dimensions

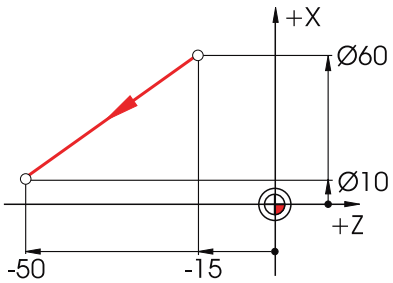


With incremental inputs, the **difference** values between the **current position** and the **end point** must always be entered, observing the **direction**.

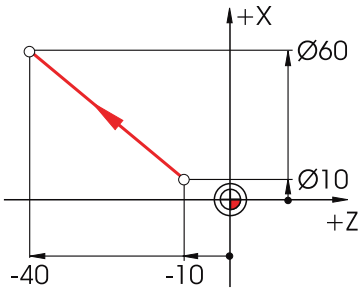
Switching between absolute and incremental input is possible at any time using the SELECT key.

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

Straight		
X	10.000	abs
Y		abs
Z	-35.000	inc



Straight		
X	25.000	inc
Y		abs
Z	-40.000	abs

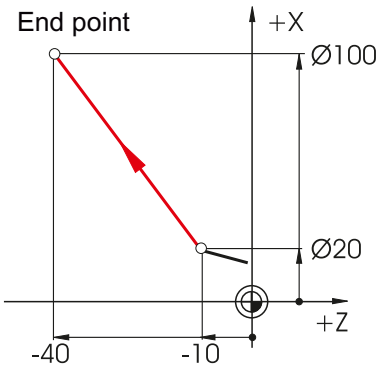


4.1.4 Cartesian and polar dimensions

Cartesian input

Input of the X and Z coordinates. 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 inputs, the **absolute** coordinate values of the **end point** must always be entered (the current position is not taken into account).

Polar input

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

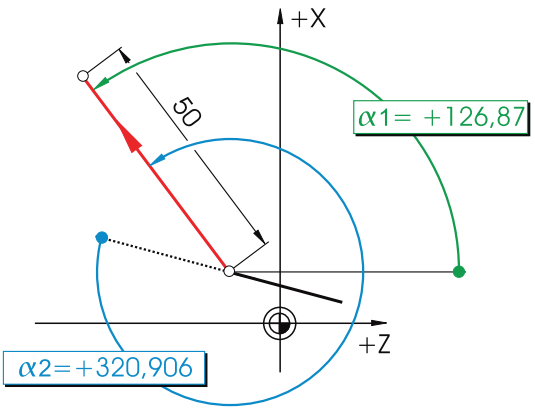
Straight ZX		
X	100.000	abs
X	40.000	inc
Z	-40.000	abs
Z	-30.000	inc
L	50.000	
α1	126.870	°
α2	320.906	°

α1 = Angle to the positive Z axis

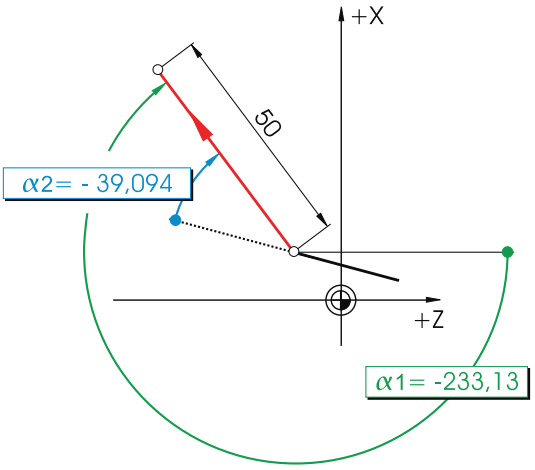
α2 = Angle to the preceding element

The angles can be entered...

positive and/or...



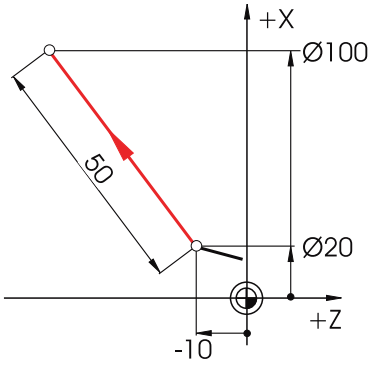
... **negative**.



The Cartesian and polar inputs can also be combined. Here are two examples:

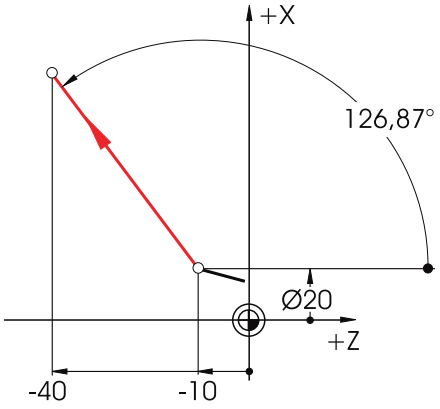
Input of the end point in X and the length

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	°



Input 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	°



4.1.5 Circular motions

In accordance with DIN, circular arcs are specified with the end point of the arc (X and Z coordinates in the G18 plane) and 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.

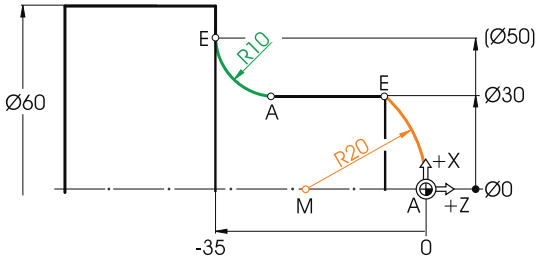
The following example shows two circular arcs, one of which is only partially determined.

Input of the R10 arc:

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

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	



Input 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	°

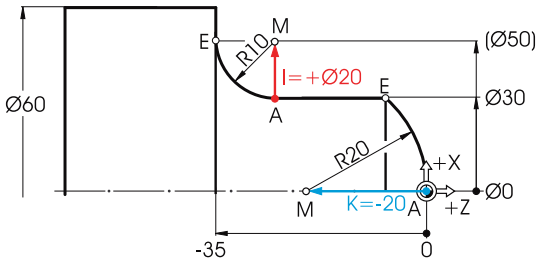
After the input:

Circle		
Direction of rotation		
R	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 parameters

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.

Circle		
Direction of rotation		
R	10.000	
X	50.000	abs
X	10.000	inc
Z	-35.000	abs
Z	-10.000	inc
I	50.000	abs
I	10.000	inc
K	-25.000	abs
K	0.000	inc
α1	180.000	°
α2	Tangential	
β1	90.000	°
β2	90.000	°



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	°

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

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

4.2 Technological basics

4.2.1 Cutting rate and speeds

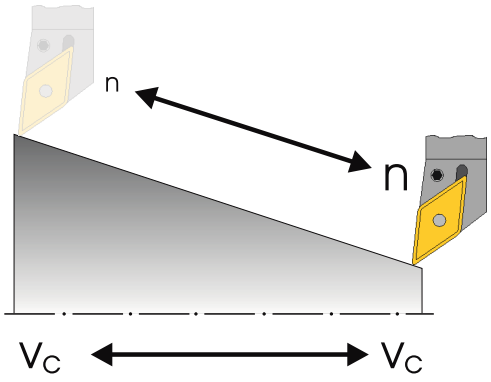
The cutting rate is usually directly programmed for turning, in particular for roughing, finishing and plunge-cutting. The speed is programmed only for drilling and (usually) for thread cutting.

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 rough cutting, finishing and plunge-cutting:



To ensure that the selected cutting rate remains constant for each workpiece diameter, the control uses the G96 command (= constant cutting rate) to adjust the appropriate speed. This is performed using direct-current or frequency-controlled three-phase motors. As the diameter reduces, the speed will theoretically increase to infinity. To prevent danger caused by 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:

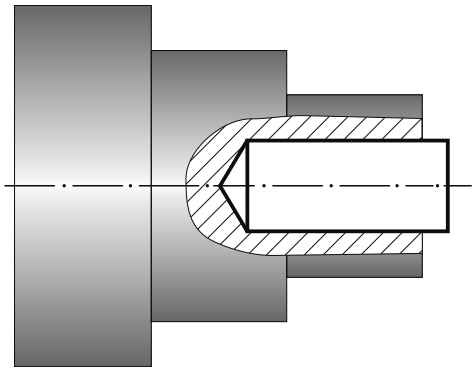
G96 S180 LIMS=3000
(LIMS = limit).

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.

The inputs are then G97 S1900.



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

d = 20mm (tool diameter)

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

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

4.2.2 Feed

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

Determination of the feed

Like the cutting rate, the value for the feed is also taken from the handbook, the documents of the tool manufacturer or from empirical 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 will be chosen:	**f = 0.3 mm**

Relationship between feed and feedrate:

The constant feed **f** and the associated speed **n** produce the feedrate:

$v_f = f \cdot n$

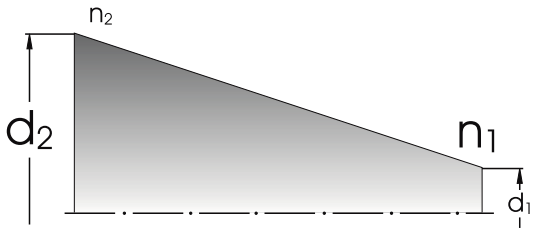
$v_c = 180 \frac{m}{min}$

$d_2 = 80mm$

$n_2 \approx 710 \frac{1}{min}$

$v_{f2} = 710 \frac{1}{min} \cdot 0.3mm$

$v_{f2} \approx 210 \frac{mm}{min}$



$v_c = 180 \frac{m}{min}$

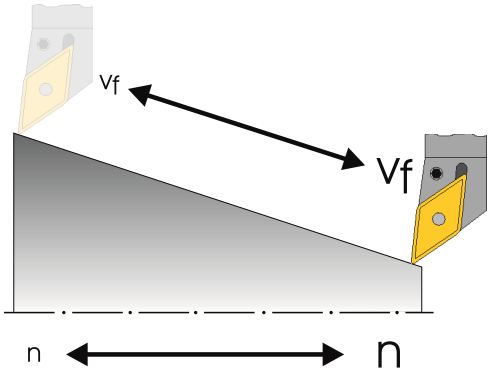
$d_1 = 20mm$

$n_1 \approx 2800 \frac{1}{min}$

$v_{f1} = 2800 \frac{1}{min} \cdot 0.3mm$

$v_{f1} \approx 840 \frac{mm}{min}$

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



Well equipped

5

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

5.1 Tool management

ShopTurn offers three lists for tool management:

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

5.1.1 The tool list

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

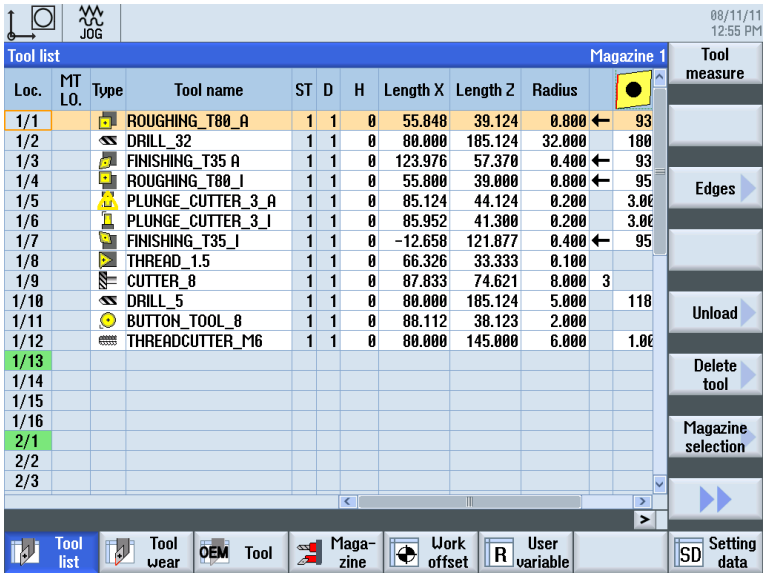


Figure 5-1 Example for tool lists

Meanings of the most important parameters:

Location	Location number
Type	Tool type
Tool name	The tool is identified by the name and the replacement tool number. You may enter the names as text or numbers.
ST	Replacement tool number (for replacement tool strategy)
D	Cutting edge number
Length X	Geometry data, length X
Length Z	Geometry data, length Z
Diameter	Tool diameter
Holder angle, point angle, board width	Holder angle (roughing tool and finishing tool), point angle (drill) and board width (plunge-cutter)
	Direction of spindle rotation
	Coolants 1 and 2 (e.g. internal and external cooling)

ShopTurn features 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. Various mounting positions and geometrical parameters exist (e.g. holder angle), depending on the tool type.

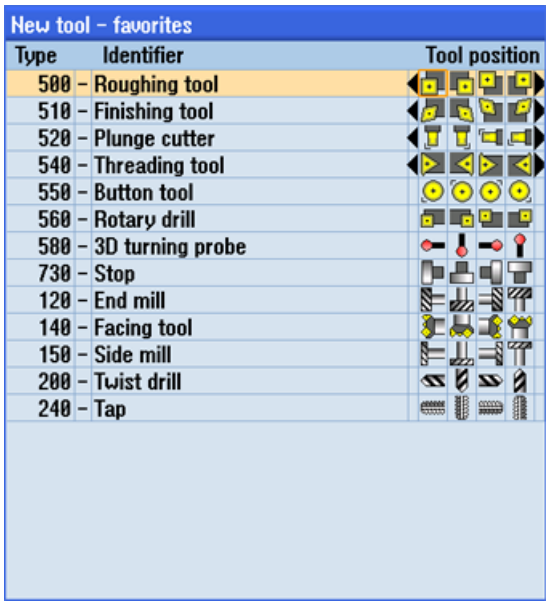


Figure 5-2 Example of Favorites list

5.1.2 The tool wear list

The wearing data for the appropriate tools are defined here.

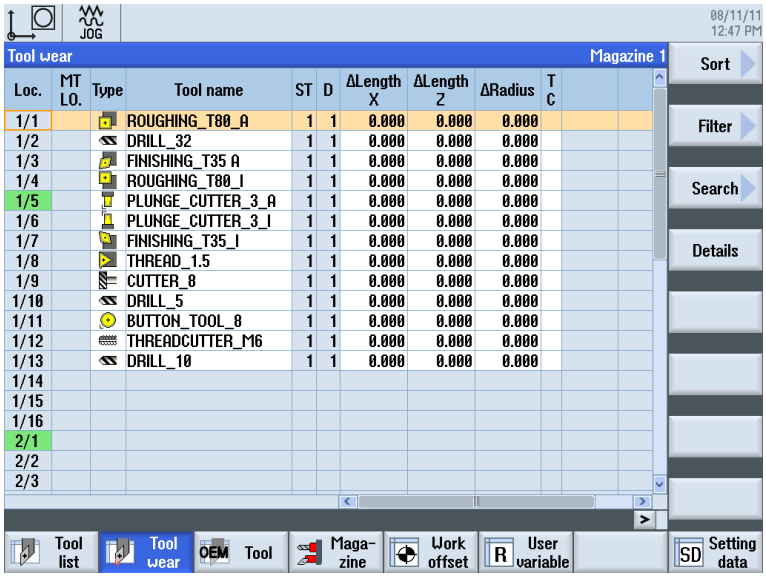


Figure 5-3 Tool wear list

The most important tool wearing parameters are:

Δ Length 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 Number of workpieces Tool wear
Setpoint	Setpoint for tool life, workpiece count, or wear
Prewarning limit	Specification of the tool life, workpiece count or wear at which a warning is displayed.
G	The tool is disabled when the checkbox is selected.

5.1.3 Magazine list

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

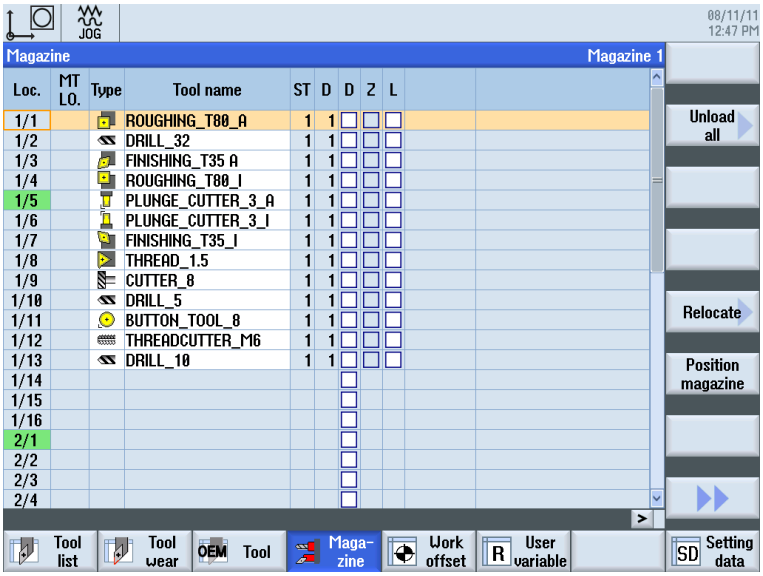


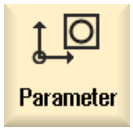
Figure 5-4 Magazine list

Meanings of the most important parameters:

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

5.2 Tools used

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



Select the "Parameters" area in the main menu.

Select the "Tool list" softkey.

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

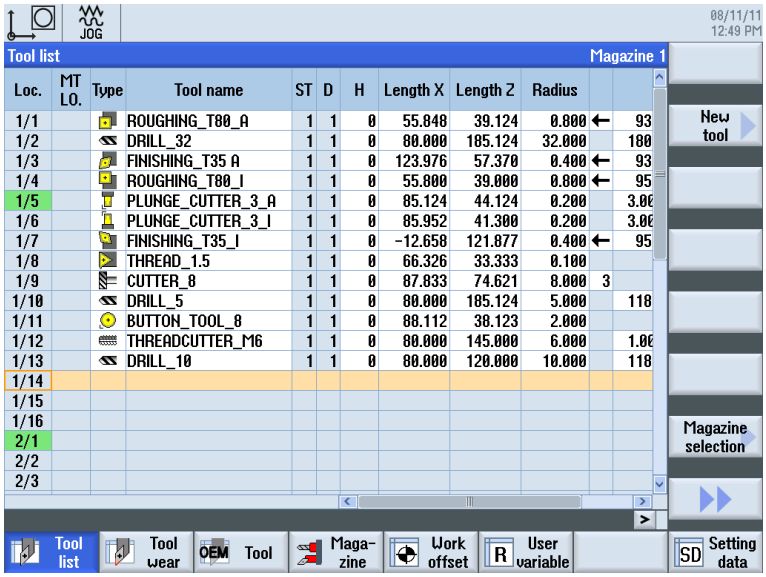


Figure 5-5 Tool list - free location

Select the "New tool" softkey.

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

Note

Milling cutter 8 (CUTTER_8) must immerse, as it is used for milling a pocket.

5.3 Tools in the magazine

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

In the tool list, select a tool without a location number.

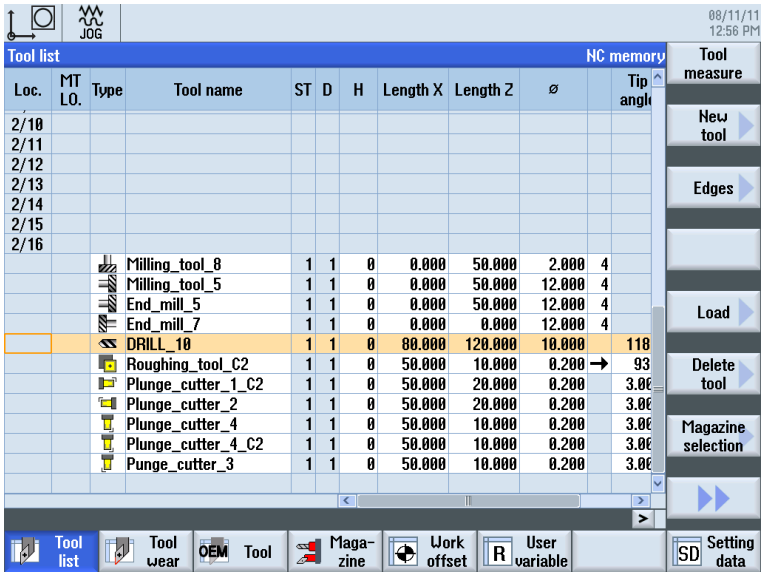


Figure 5-6 Select the tool in the magazine

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

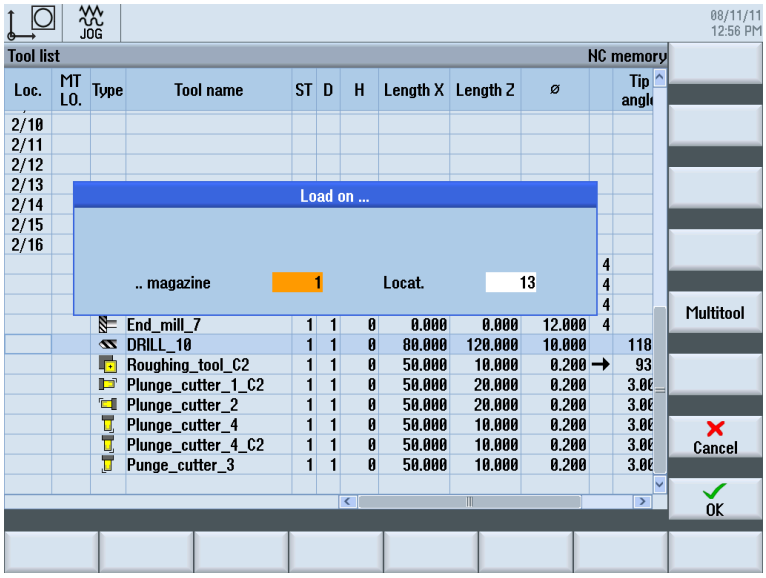


Figure 5-7 Enter and/or accept a magazine location

After acceptance, the tool list may look like this.

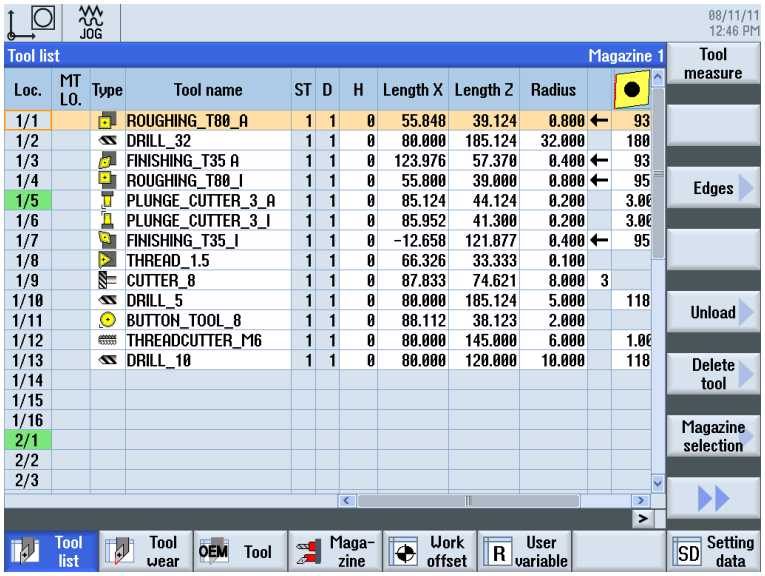


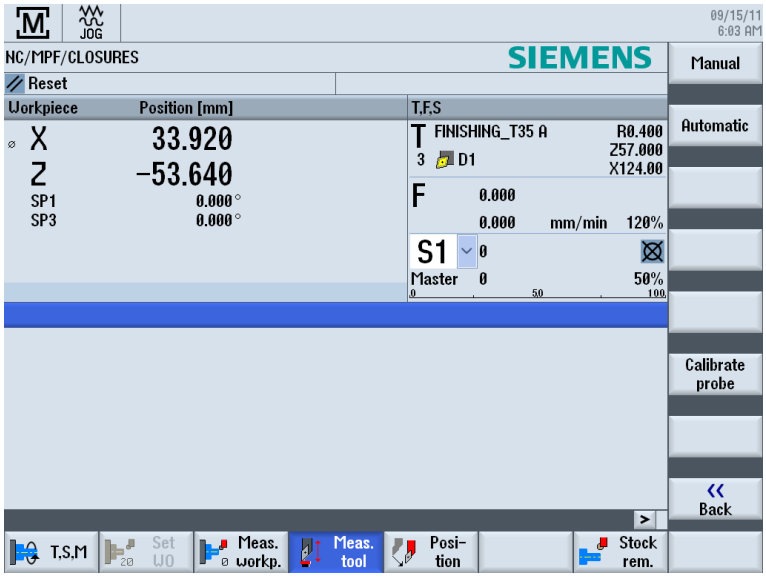
Figure 5-8 Tool list following acceptance

5.4 Gauging tools

In the following you will learn how to calculate tools.

Procedure

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



Meas. tool

Then switch to the "Gauge tool" menu.

Manual

Enter the probed or turned diameter.

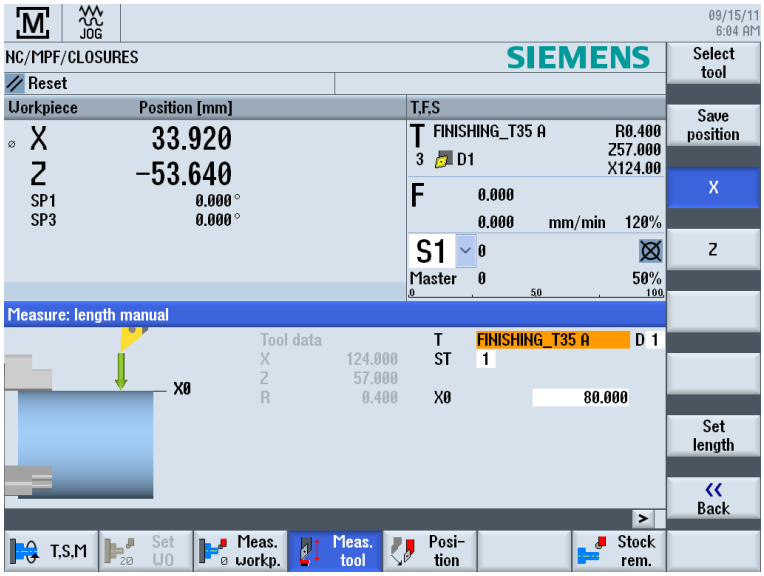


Figure 5-9 Tool measurement - input of the X value

Set length

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

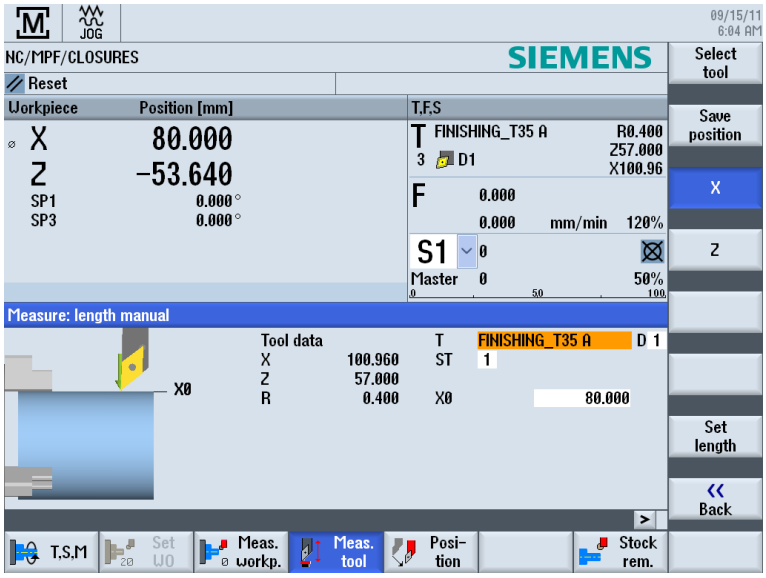


Figure 5-10 Tool measurement - set length X

Repeat for Z.

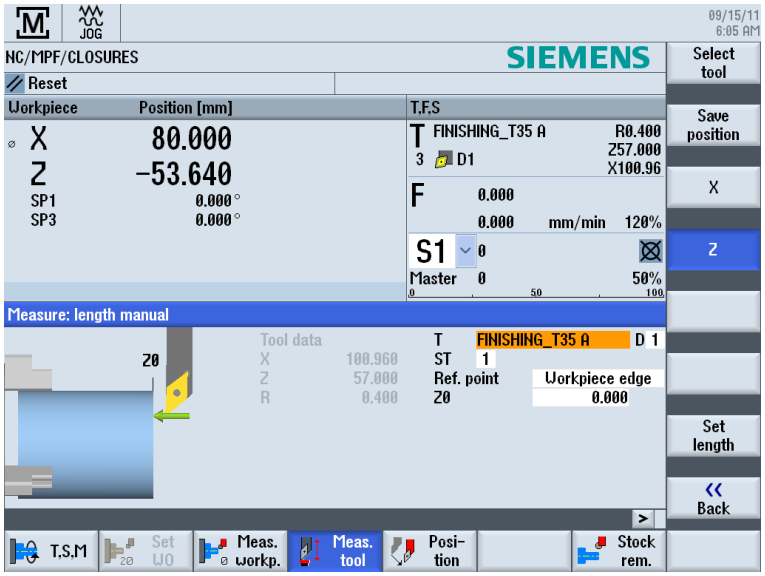


Figure 5-11 Tool measurement - set length Z

5.5 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.

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

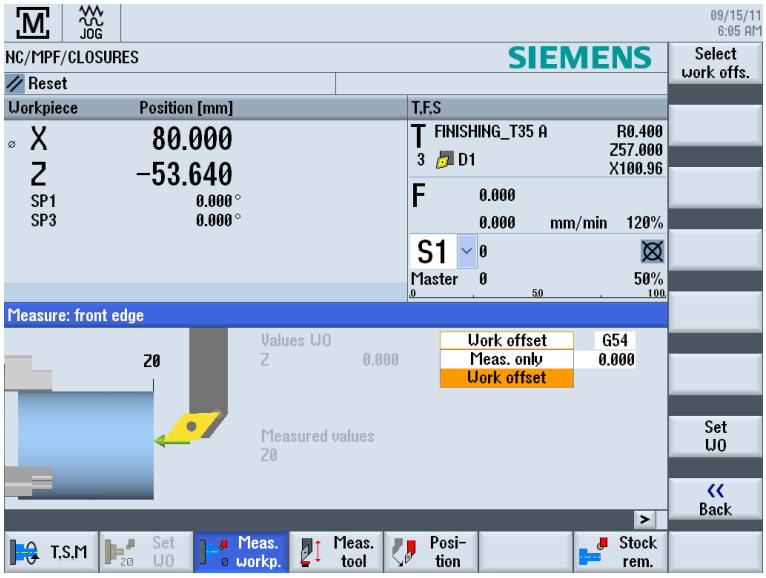


Figure 5-12 Enter the work offset

Set WO

Confirm your input.

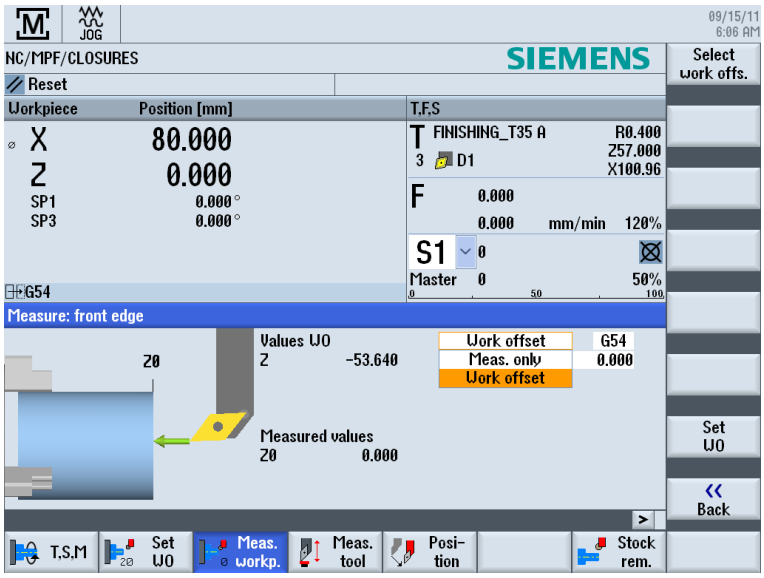


Figure 5-13 Work offset set

Example 1: Taper shaft

6

6.1 Overview

Learning objectives

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

- create and manage programs;
- call tools;
- enter traversing paths;
- create any contours with the contour calculator;
- rough and finish contours;
- create a thread undercut,
- thread and
- grooves.

Task

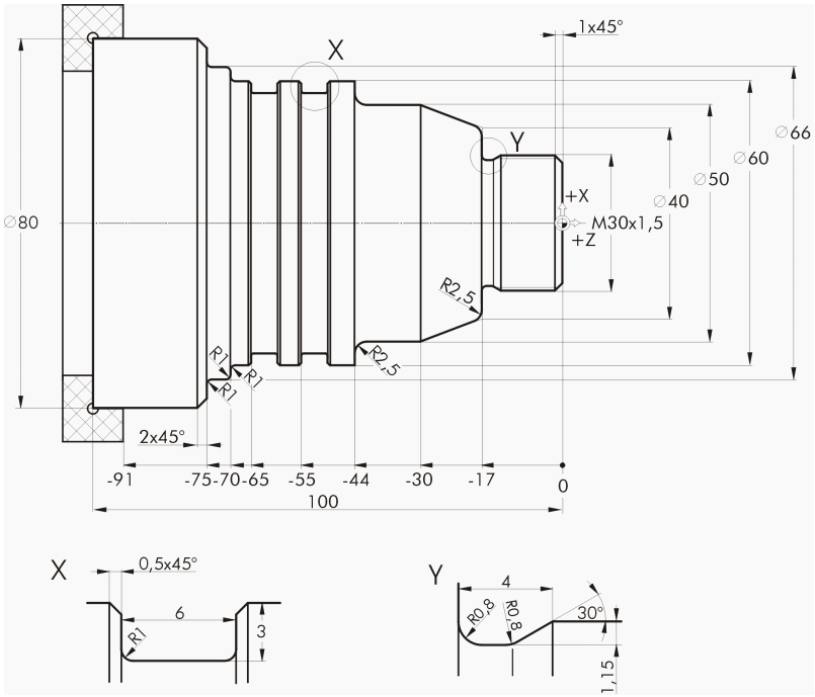


Figure 6-1 Workshop drawing - Example 1:

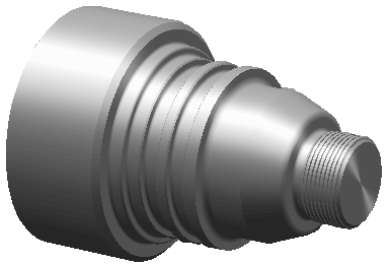


Figure 6-2 Workpiece - Example 1:

Note

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

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

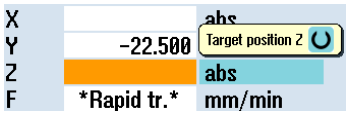


Figure 6-3 Example toggle field

6.2 Program management / creating programs

Operating sequences

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

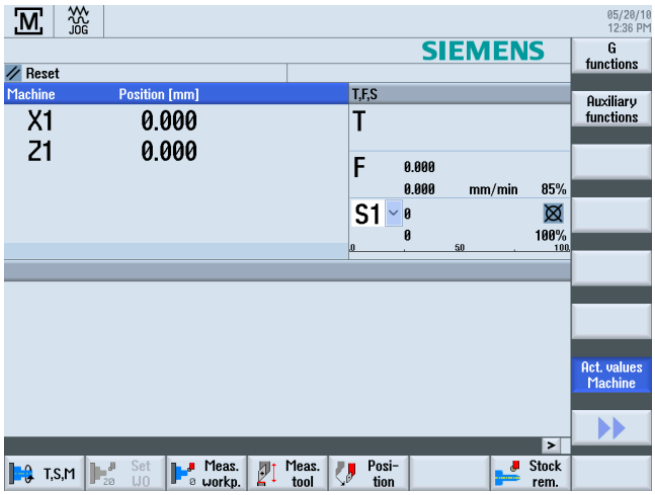


Figure 6-4 Main screen

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

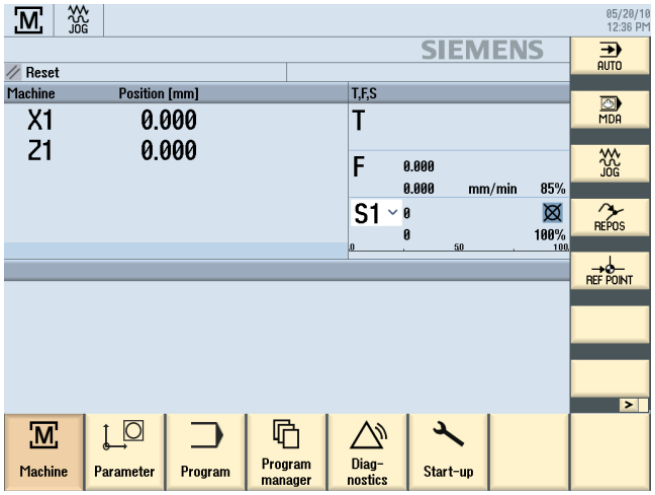


Figure 6-5 Main menu

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

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

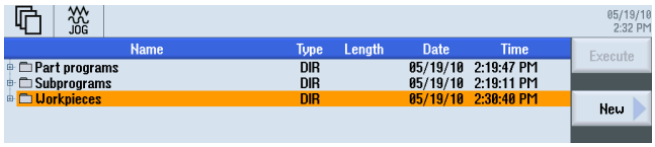


Figure 6-6 Program Manager

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 a workpiece

Confirm your input. The following dialog box is opened:

Figure 6-8 Creating a step sequence program

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

The program type can be specified via the **ShopTurn** softkey.

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

Confirm your input.

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

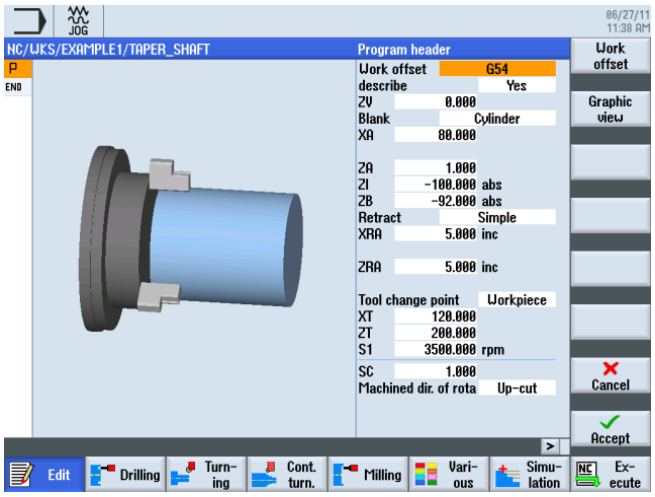


Figure 6-9 Program header - Help display

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

Enter the following values:

Field	Value	Selection via toggle key	Notes
Unit of measurement	mm	X	
Work offset		X	
Blank	Cylinder	X	Select the blank shape (here cylinder) using the toggle button.
XA	80		
ZA	1		
ZI	-100 abs	X	
ZB	-92 abs	X	The value ZB indicates the distance from the chuck.
Retraction	normal	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	WCS	X	
XT	120		
ZT	200		
Safety clearance SC	1		
Speed limits S1	3500		
Machining direction	Synchronous	X	

Accept

Accept the entered values. After confirming, the program header is displayed.

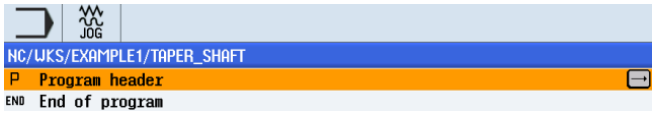


Figure 6-10 Program header, example 1 - Work step editor

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

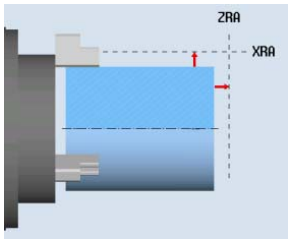
You may call the program header again at any time to make changes or check the values.

Retraction

It is possible to switch the retraction plane between normal, extended and all. Depending on the retraction setting, the associated fields are enabled for the input of the distances.

normal

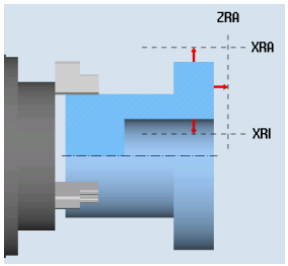
(for simple cylinders)



Retract	Simple	
XRA	5.000	inc
ZRA	5.000	inc

extended

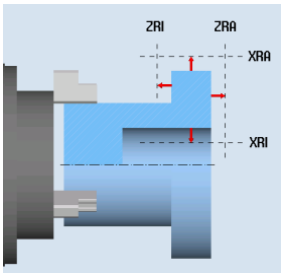
(for complex workpieces with internal machining)



Retract	Extended	
XRA	5.000	inc
XRI	5.000	inc
ZRA	5.000	inc

all

(for complex workpieces with internal machining and/or relief cuts)



Retract	All	
XRA	5.000	inc
XRI	5.000	inc
ZRA	5.000	inc
ZRI	0.000	

Softkeys

Graphic view

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

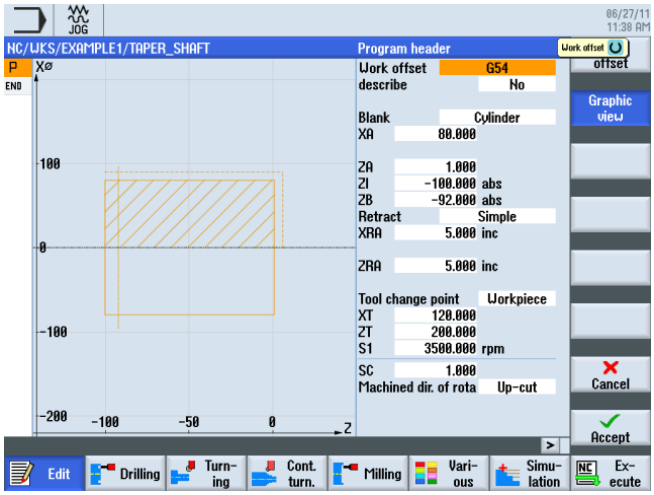


Figure 6-11 Program header - graphical view

Graphic view

Use this softkey to switch back to the help display.

6.3 Calling a tool

Operating sequences

To call the required tool, proceed as follows:

Use this key to extend the horizontal softkey menu.

Gerade Kreis

Select the **Straight line Circle** softkey.

Tool

Select the **Tool** softkey.

Select tool

Open the tool list.

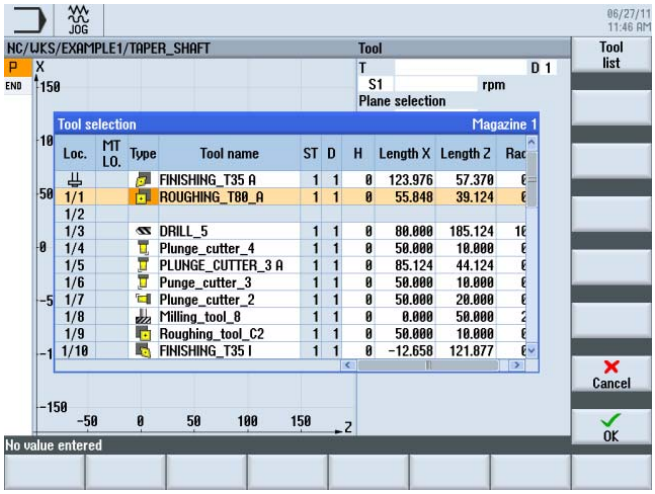


Figure 6-12 Tool list

Use the cursor key to select the ROUGHING_T80 A tool.

To program

Accept the tool into your program. After accepting the tool, enter the following values in the interactive screenform (if necessary, change the unit using the toggle key):

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

Figure 6-13 Tool - input

Accept

Accept the entered value.

6.4 Specifying the distance to be traversed

Operating sequences

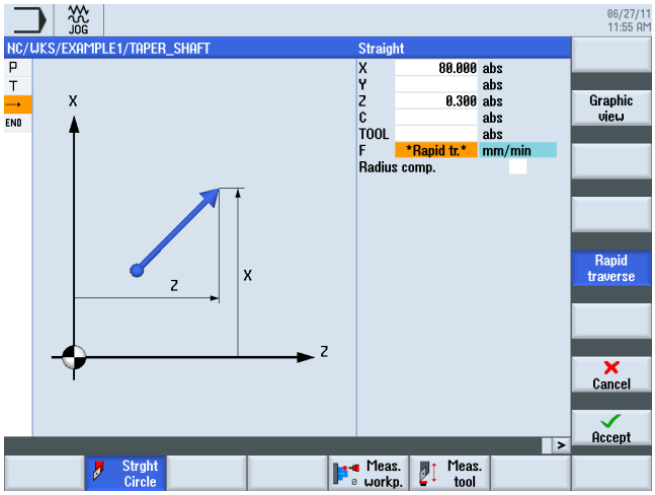
Now enter the distances to be traversed:

Select the Straight line softkey.

Select the "Rapid traverse" softkey.

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

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



Enter the starting point for the traversing path

"Apply" the set values.

Select the Straight line softkey.

Enter the following values in the interactive screenform:

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

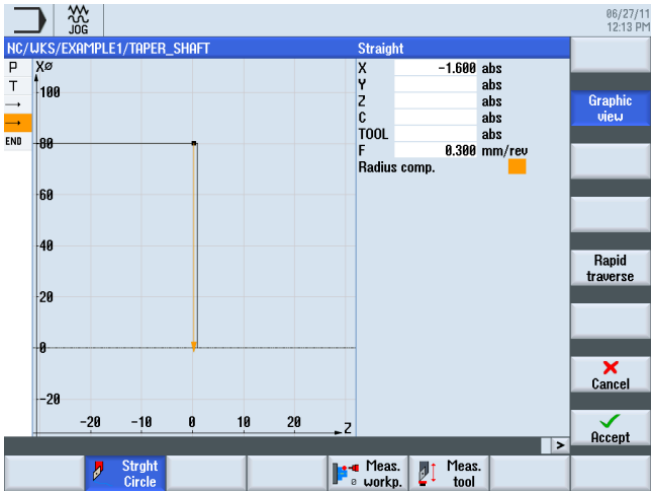


Figure 6-14 Specify the traversing path

"Apply" the set values.

Select the Straight line softkey.

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

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z	1 abs	X	

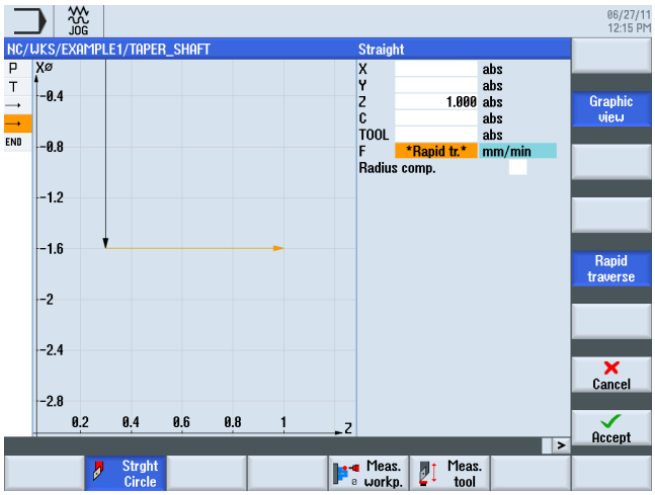


Figure 6-15 Enter the traversing path - moving away from the end face

"Apply" the set values.

Select the Straight line softkey.

Select the "Rapid traverse" softkey.

Enter the following values in the interactive screenform:

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

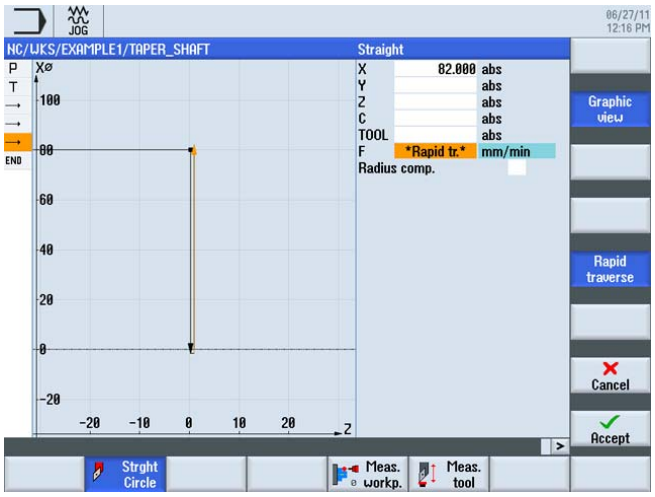


Figure 6-16 Enter the traversing path - returning to the starting point

"Apply" the set values.

Select the Straight line softkey.

Create the four further traversing paths according to the following list of work steps.

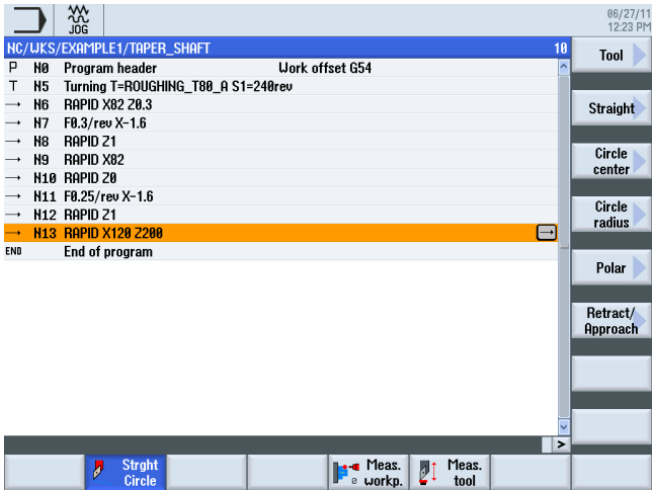


Figure 6-17 Enter the traversing path - four further traversing paths

Start the simulation.

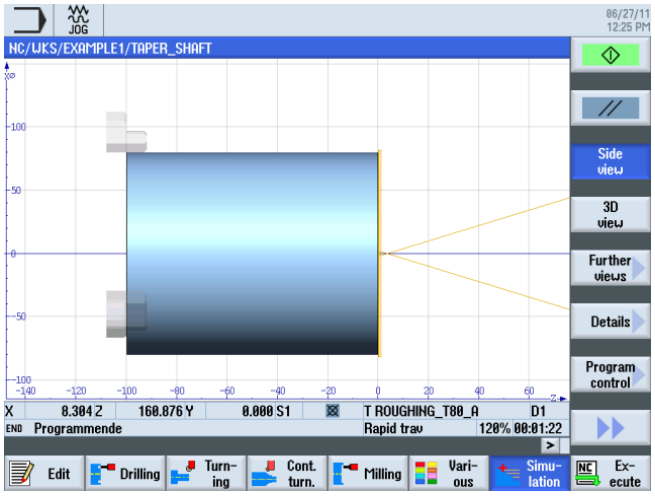


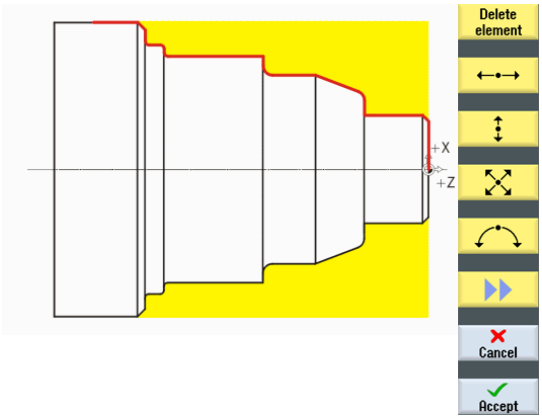
Figure 6-18 Simulation – side view

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

6.5 Creating contours with the contour calculator and machining sequence

Contour calculator

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



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

Operating sequences

Proceed as follows to enter the contour:

Select the **Contour turning** softkey.

Select the **New contour** softkey. Enter the name 'TAPER_SHAFT_CONTOUR' for the contour.

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

Figure 6-19 Creating the 'TAPER_SHAFT_CONTOUR' contour

Confirm your input.

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

Note

The contour line is in one sense the roughing limit and in another the finishing path.

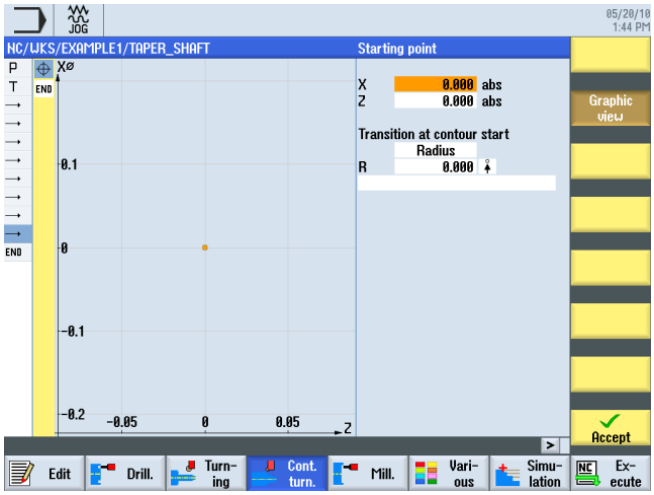


Figure 6-20 Specifying the starting point

Note

If you deselect the **Graphic view** softkey, detailed help displays are shown.

Confirm your input.

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

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

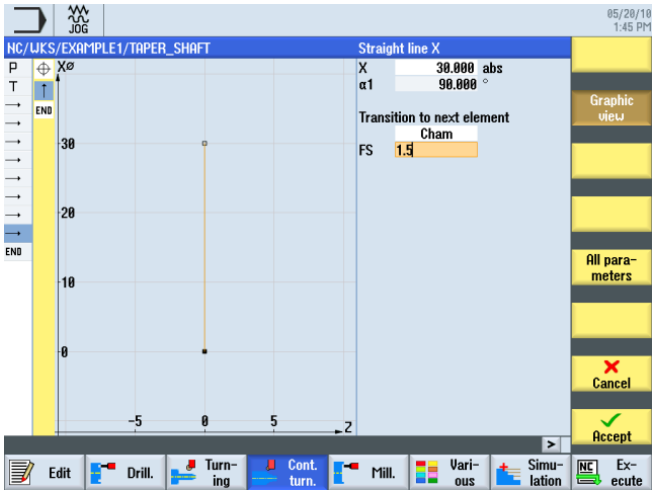
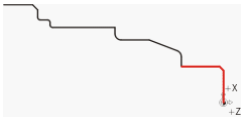


Figure 6-21 Specifying the vertical straight line contour

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
Z	-17 abs	X	A straight line appears up to Z-17.
Transition to next element	Chamfer	X	
FS	0		 The thread undercut is inserted later as an individual element.

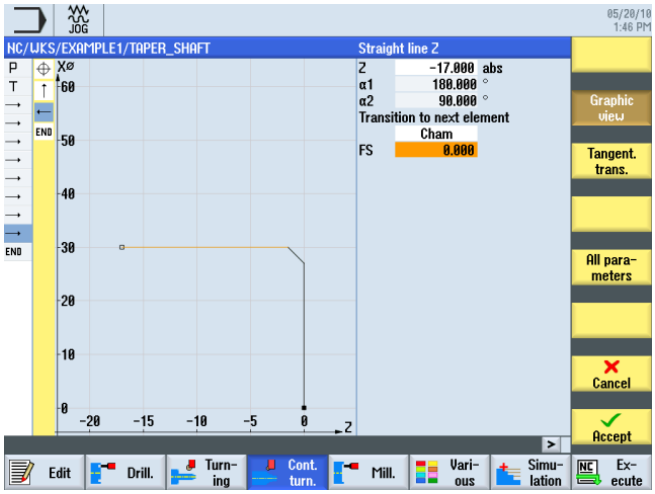
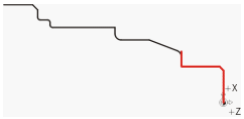


Figure 6-22 Specifying the horizontal straight line contour

Accept the entered values.

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

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

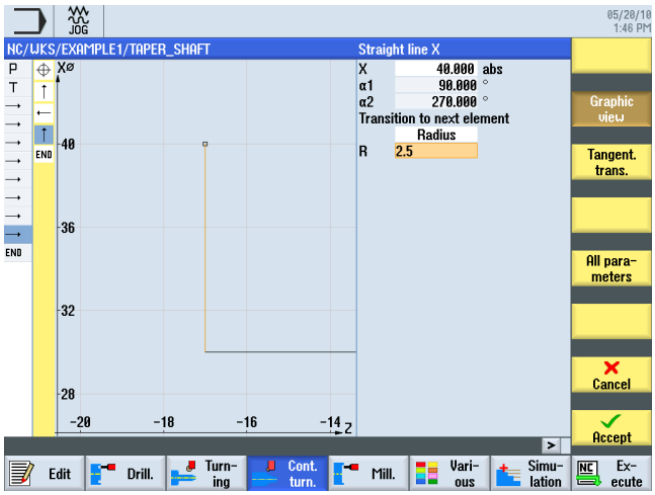


Figure 6-23 Specifying the vertical straight line contour

Accept the entered values.

In the interactive screenform, enter the following values for the end point of the inclined straight line:

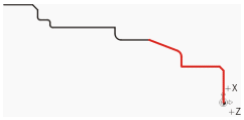
Field	Value	Selection via toggle key	Notes
X	50 abs	X	
Z	-30 abs	X	
Transition to next element	Chamfer	X	
FS	0		

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

Accept the entered values.

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

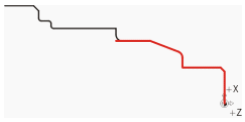
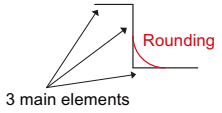
Field	Value	Selection via toggle key	Notes
Z	-44 abs	X	
Transition to next element	Radius	X	
R	2.5		

Figure 6-25 Specifying the horizontal straight line contour

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
X	60 abs	X	The paths (= main elements) do not meet **tangentially**. 

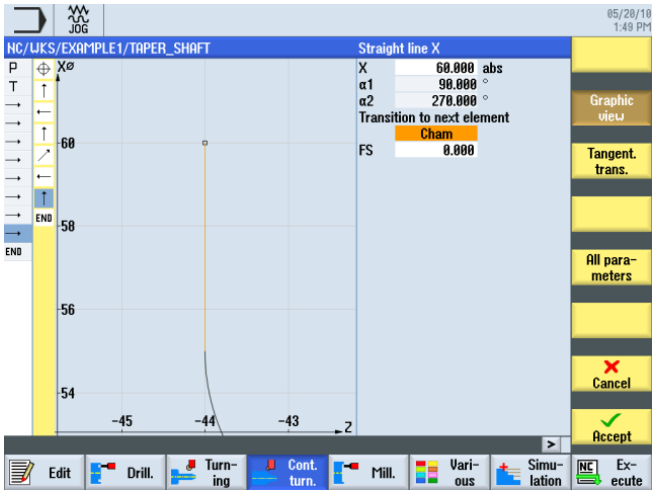


Figure 6-26 Specifying the vertical straight line contour

Accept the entered values.

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

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

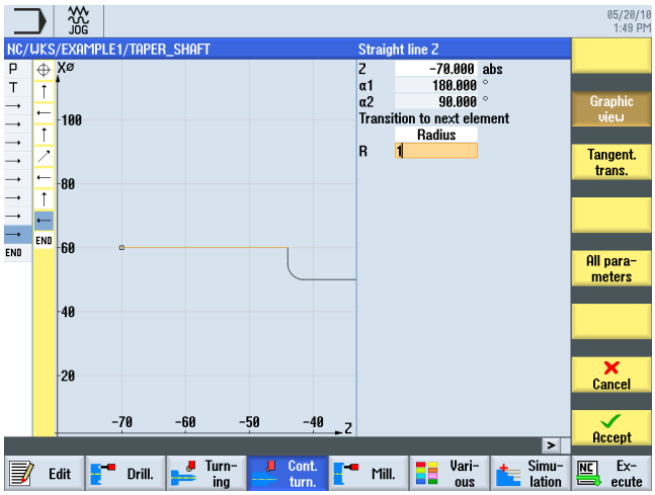


Figure 6-27 Specifying the horizontal straight line contour

Accept the entered values.

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

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

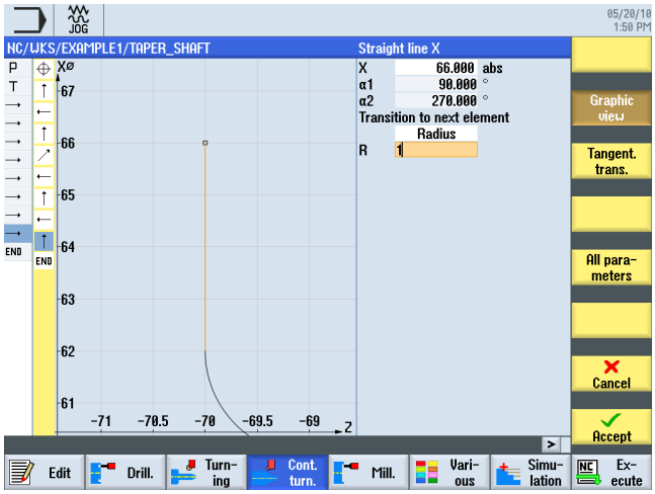


Figure 6-28 Specifying the vertical straight line contour

Accept the entered values.

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

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

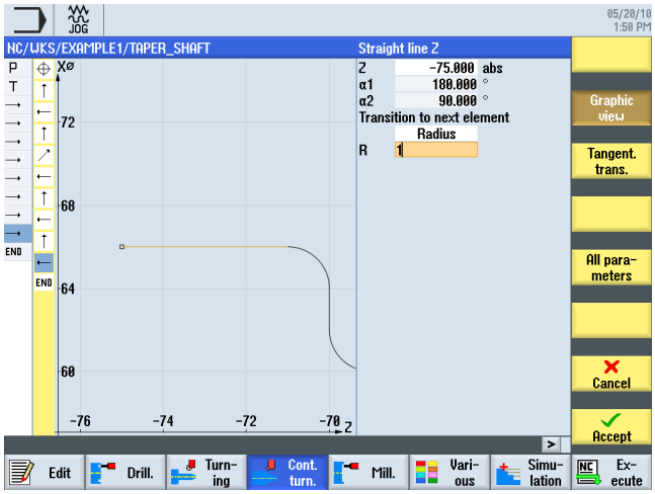


Figure 6-29 Specifying the horizontal straight line contour

Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
X	80 abs	X	End point X80 with a 2x45° chamfer 
Transition to next element	Chamfer	X	
FS	2		

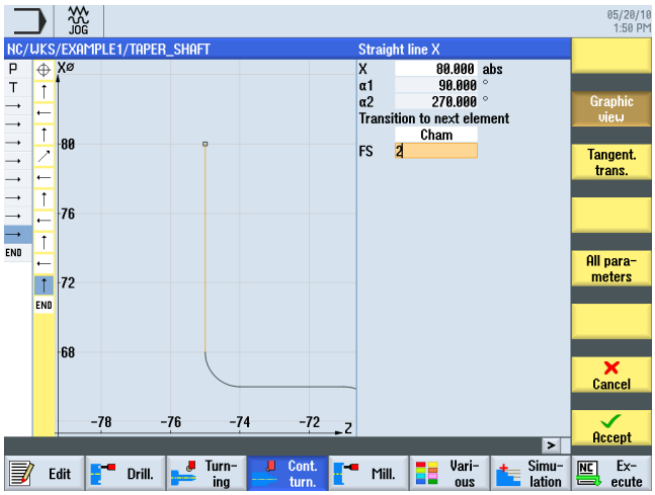


Figure 6-30 Specifying the vertical straight line contour

Accept the entered values.

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

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

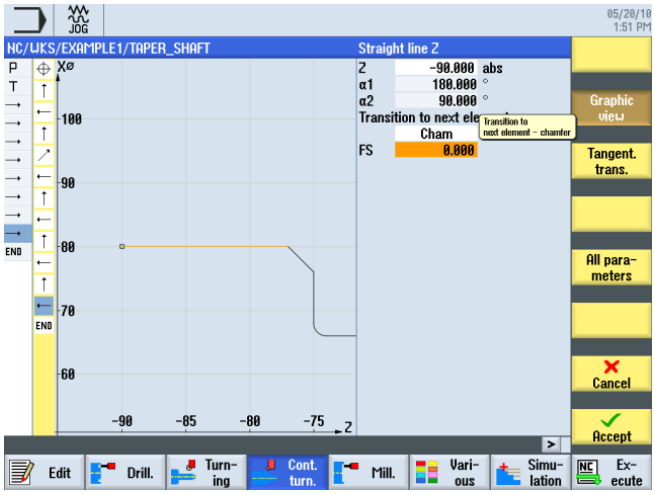


Figure 6-31 Specifying the contour end point

Accept the entered values.

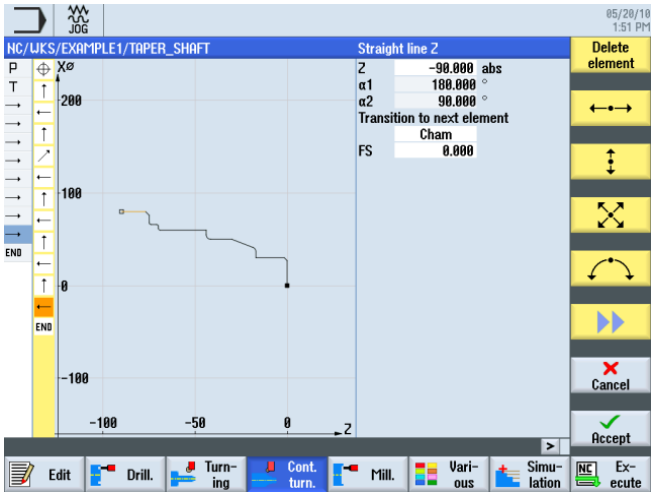


Figure 6-32 Full contour

Accept the contour into your process plan.

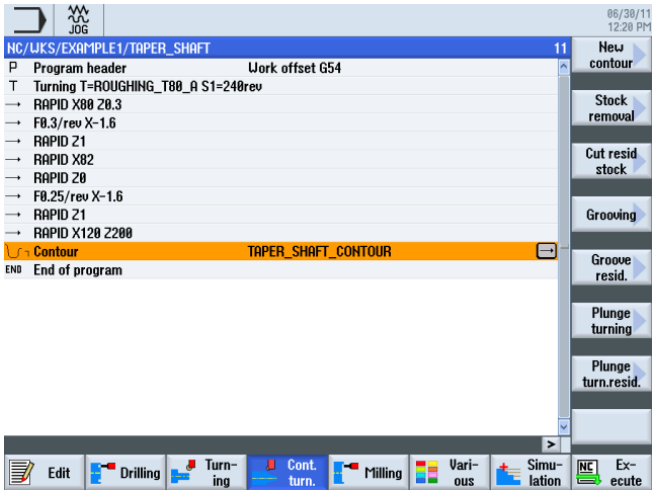


Figure 6-33 Contour in the process plan

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

Select the **Stock removal** softkey.

Open the tool list and select ROUGHING_T80 A.

Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.3		
V	240 m/min	X	
Machining	longitudinal external roughing	X 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	
Set machining area limits	No	X	

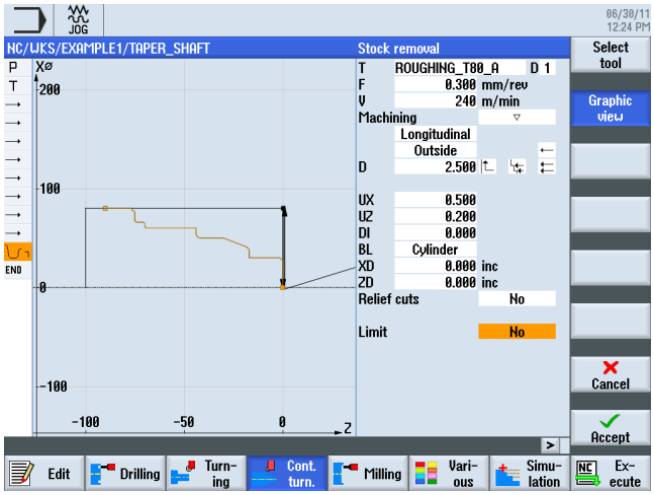


Figure 6-34 Roughing the contour

Accept

Accept the entered values.

Stock removal

Select the **Stock removal** softkey.

Select tool

Open the tool list and select FINISHING_T35 A.

To program

Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

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

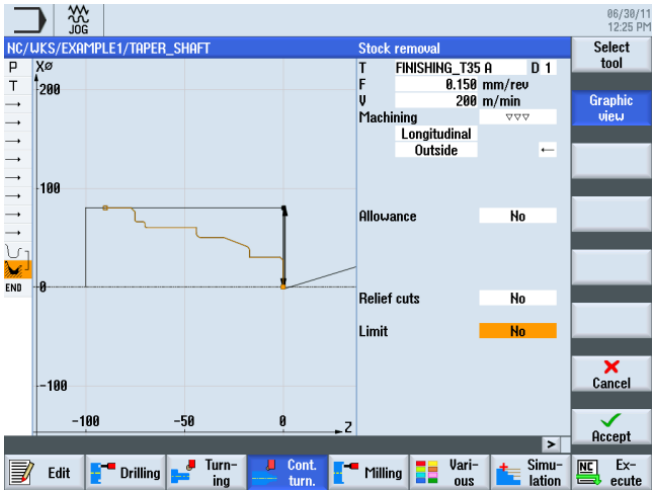


Figure 6-35 Finishing the contour

Accept

Accept the entered values.

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

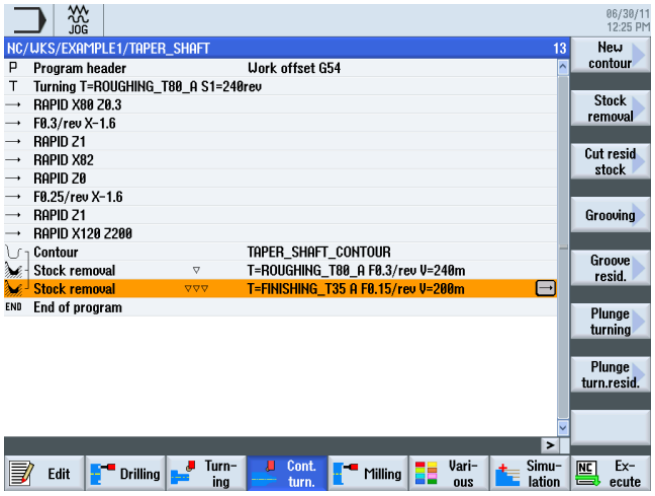


Figure 6-36 Linking of the work steps in the process plan

Simu-lation

Select the **Simulation** softkey.

Side view

Select the **Side view** softkey.

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

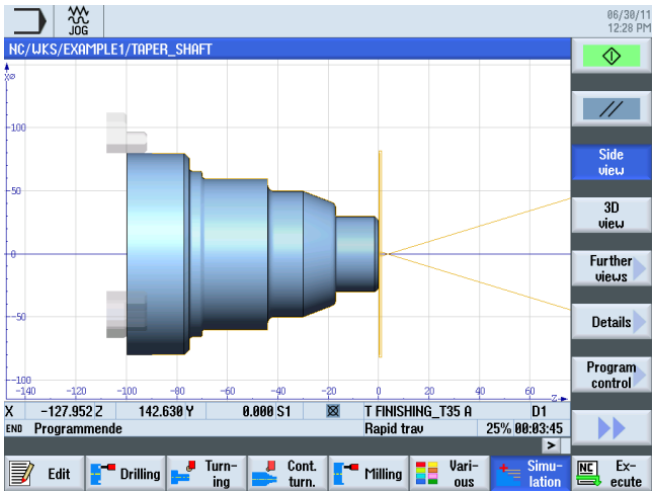


Figure 6-37 Simulation – side view

6.6 Thread undercut

Operating sequences

Proceed as follows to create a thread undercut:

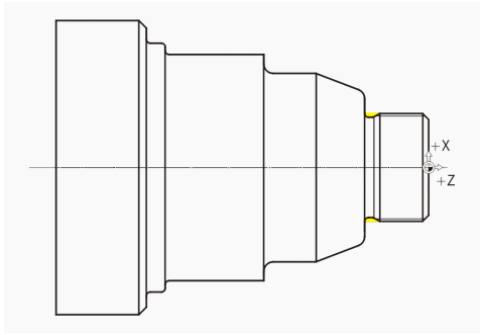


Figure 6-38 Thread undercut

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 .

To program

Accept the tool into your program.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	200 m/min	X	
Machining	Roughing/ finishing Longitudinal	X X	
Position		X	(see illustration 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		
U	0.1	X (field)	

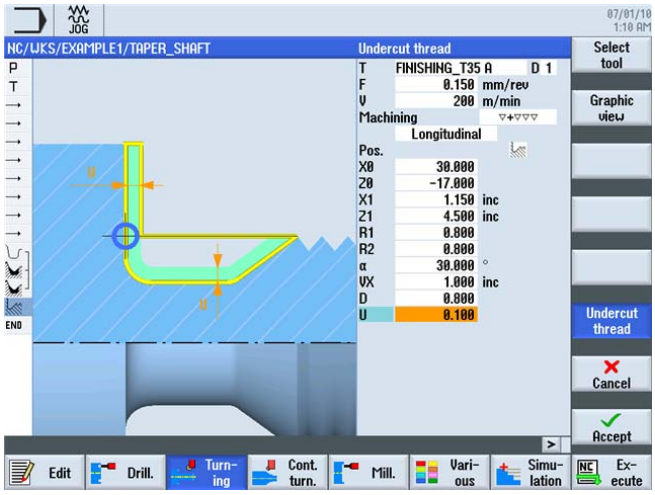


Figure 6-39 Thread undercut

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

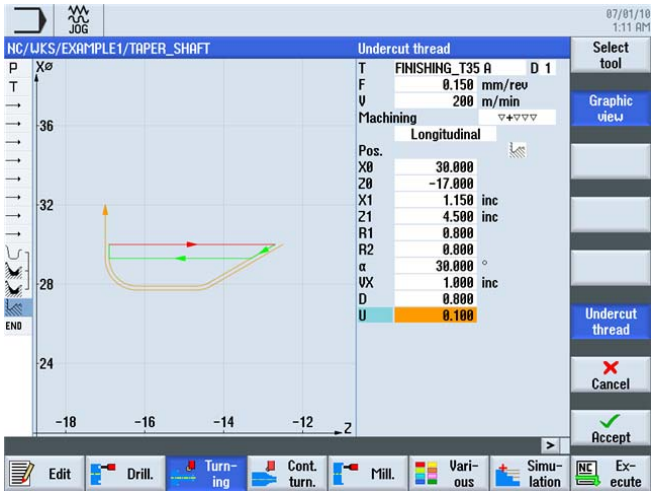
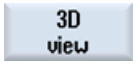


Figure 6-40 Thread undercut - graphic view

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 using the Zoom +, Zoom -, Magnifying glass, etc. softkeys.

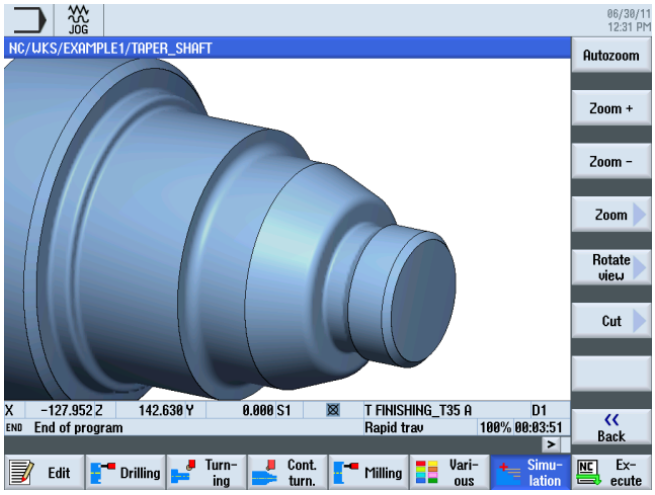


Figure 6-41 Detailed view of the simulation in the 3D display

6.7 Thread

Operating sequences

Proceed as follows to create a thread:

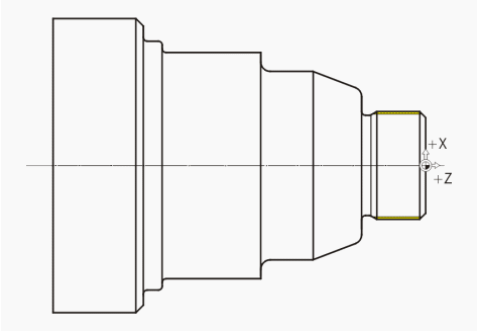


Figure 6-42 Thread

Select the **Thread** softkey.

Open the tool list and select THREADING_T1.5.

Accept the tool into your program.

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

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	30	X	The following inputs define the thread geometrically.
Z0	0		
Z1	-16 abs	X	
LW	2		
LR	1		
H1	0.92		
αP	29	X	
ND	8		
U	0.1		
NN	0		
VR	2		
Multiple threads	No	X	
α0	0		

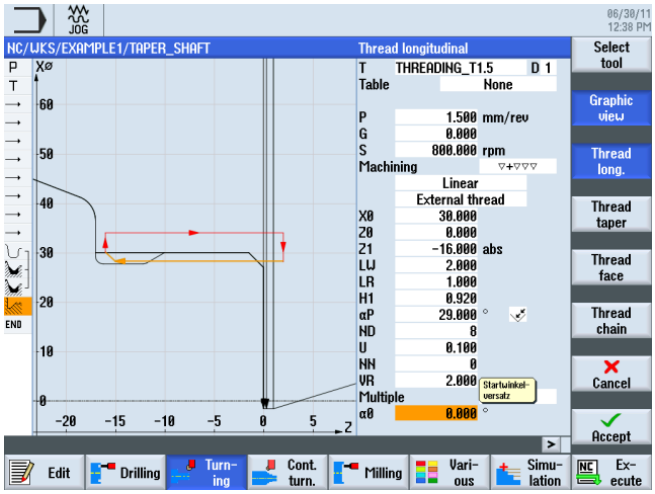


Figure 6-43 Thread - graphic view

Switch to the help display when necessary.

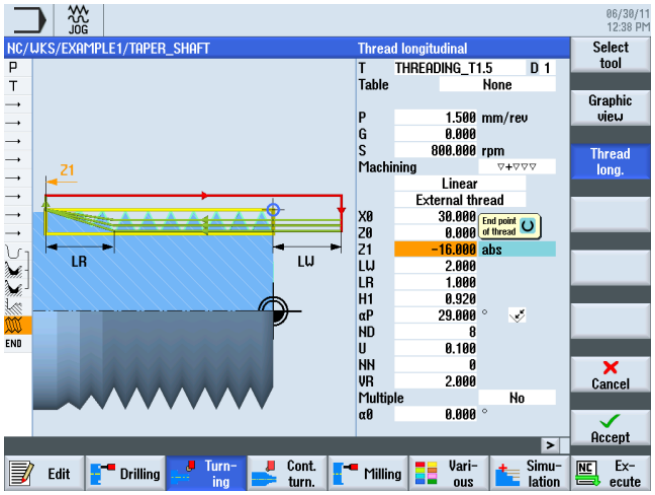


Figure 6-44 Thread - help display

Accept the entered values.

Simu-lation

Start the simulation.

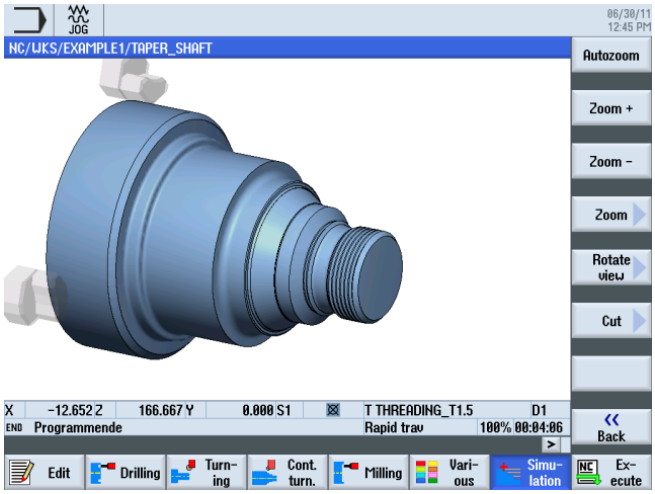


Figure 6-45 Thread simulation

6.8 Grooves

Operating sequences

Proceed as follows to create the two grooves:

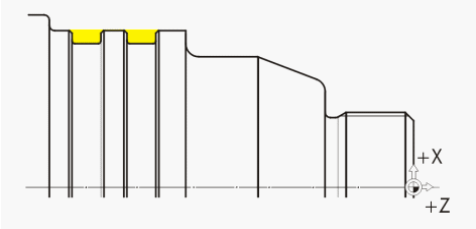


Figure 6-46 Grooves

Groove

Select the **Groove** softkey.

Select the **Groove 2** softkey.

Select tool

Open the tool list and select the PLUNGE_CUTTER_3 A plunge cutter.

To program

Accept the tool into your program.

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

Field	Value	Selection via toggle key	Notes
F	0.1		
V	150 m/min	X	
Machining	Roughing/finishing		
Position			(see illustration above)
X0	60		The following inputs define the grooves geometrically.
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		

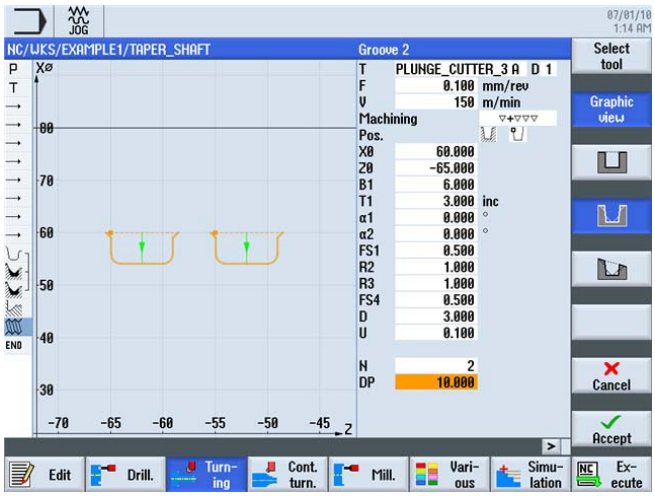


Figure 6-47 Grooves - graphic view

Switch to the help display when necessary.

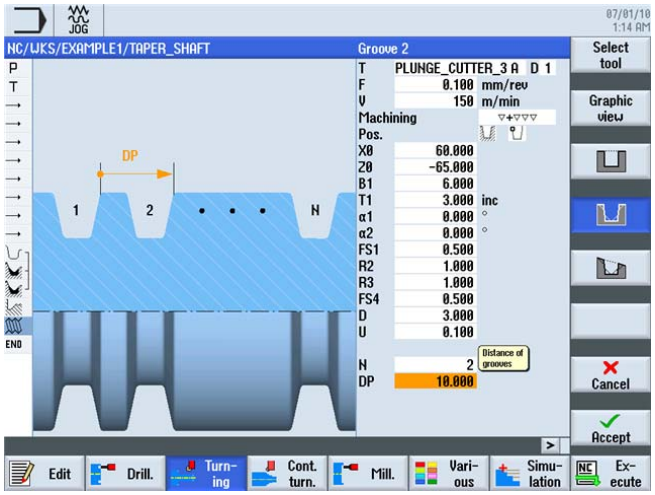


Figure 6-48 Grooves - help display

Accept the entered values.

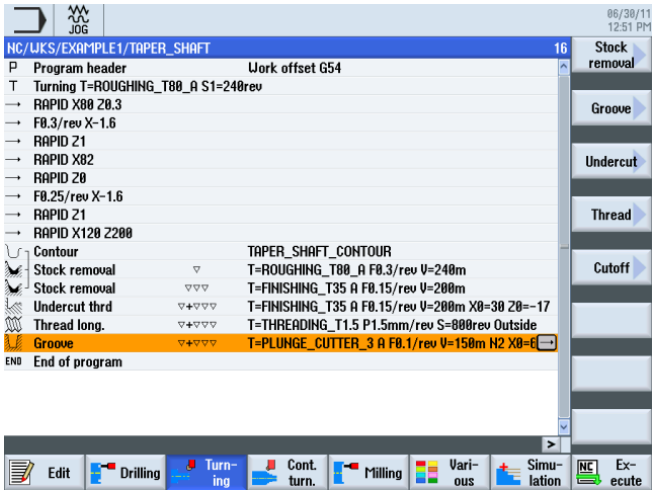


Figure 6-49 Process plan with grooves

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

Select the **Side view** softkey.

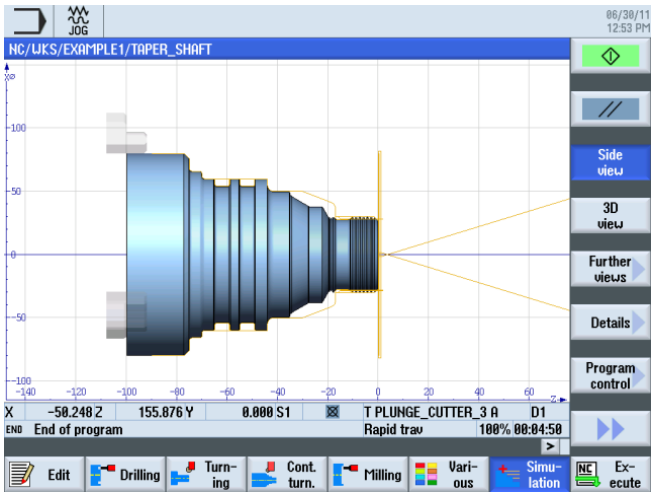


Figure 6-50 Simulation – side view

Select the **Further views** softkey.

Select the **2-window** softkey.

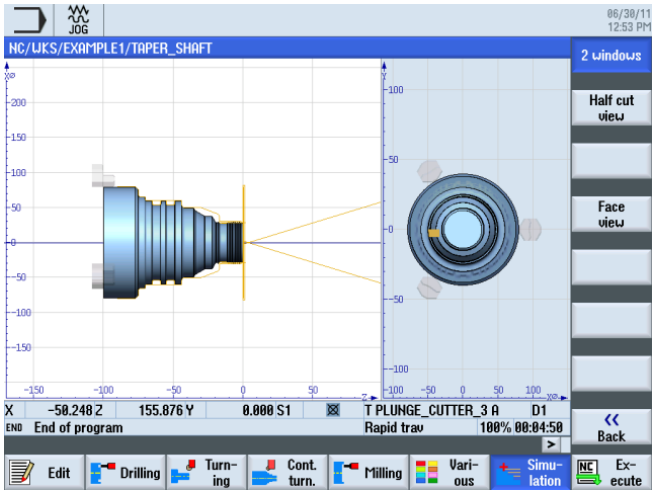


Figure 6-51 Simulation – 2-window view

Example 2: Input shaft

7

7.1 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

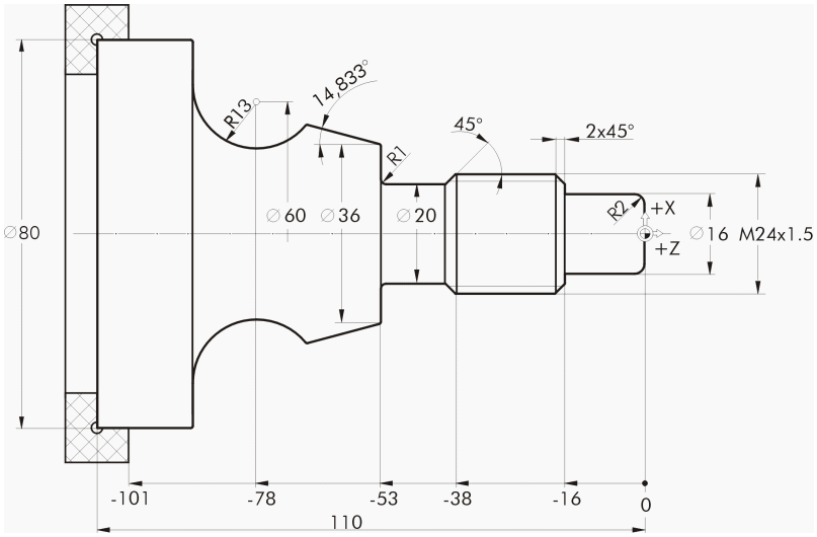


Figure 7-1 Workshop drawing - Example 2:

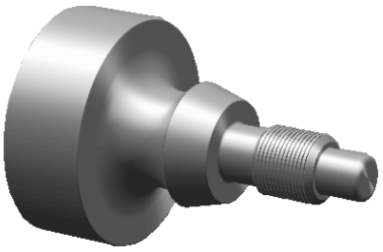


Figure 7-2 Workpiece - Example 2:

Preparation

Perform the following steps without help:

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

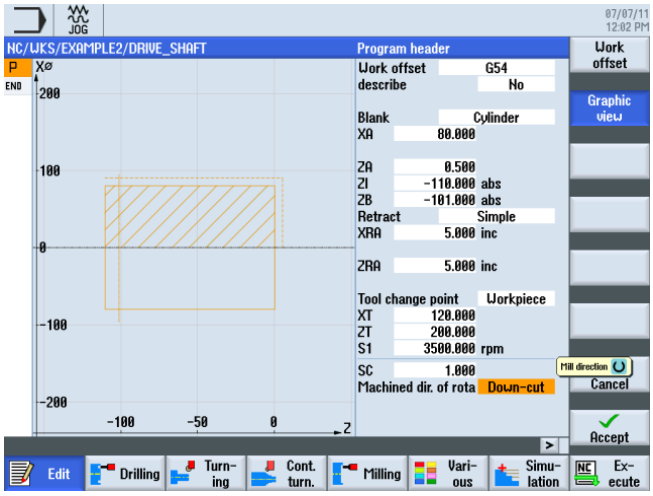


Figure 7-3 Creating a program header

Following creation of the program header, the process plan looks like this.

Figure 7-4 Work step program

7.2 Face turning

Operating sequences

Proceed as follows to face turn the workpiece:

Select the **Turning** softkey.

Select the **Stock removal** softkey.

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

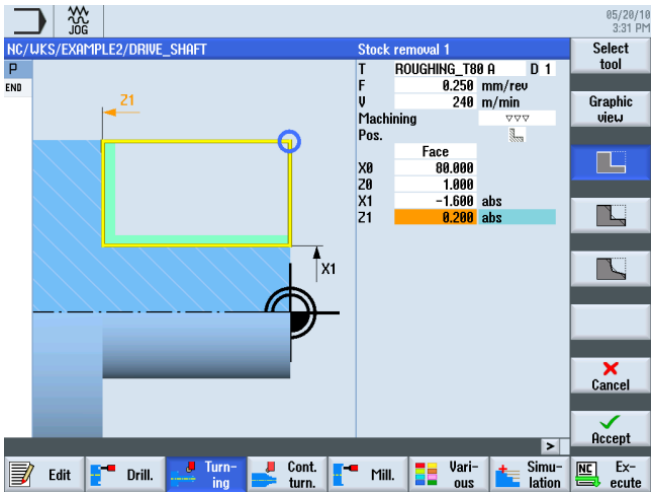
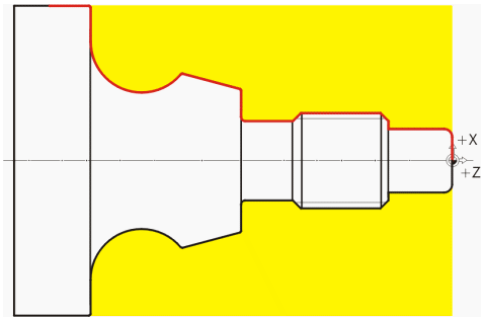


Figure 7-5 Face turning the workpiece

7.3 Creating the contour, stock removal and residual stock removal

Operating sequences

Proceed as follows to enter the contour:



Select the **Contour turning** softkey.

Select the **New contour** softkey. Enter the name 'DRIVE_SHAFT_CONTOUR' for the contour.

Figure 7-6 Creating the contour

Confirm your input.

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

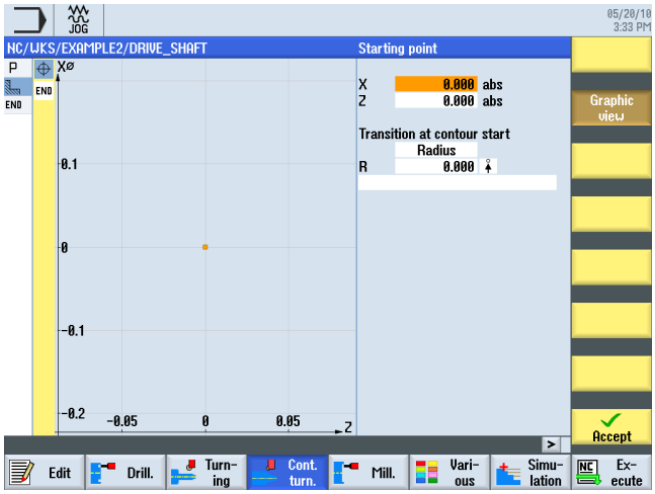


Figure 7-7 Accepting the starting point

Confirm your input.

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

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

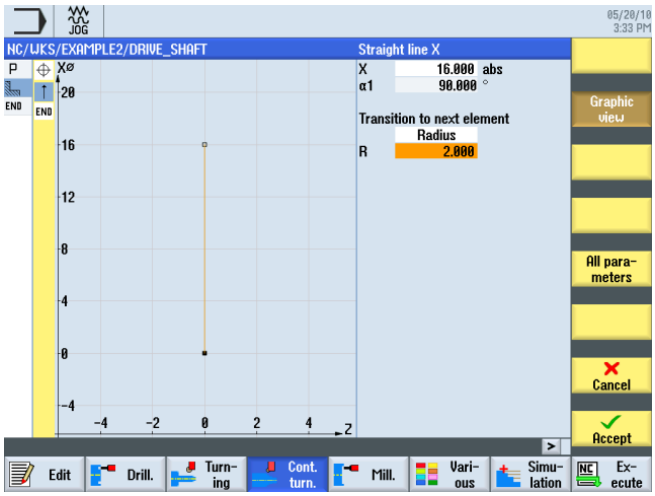


Figure 7-8 Specifying the vertical straight line contour

Accept the entered values.

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

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

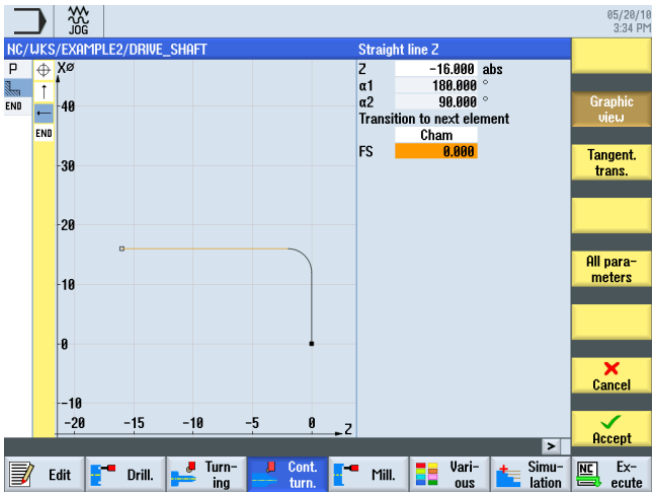


Figure 7-9 Specifying the horizontal straight line contour

Accept the entered values.

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

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

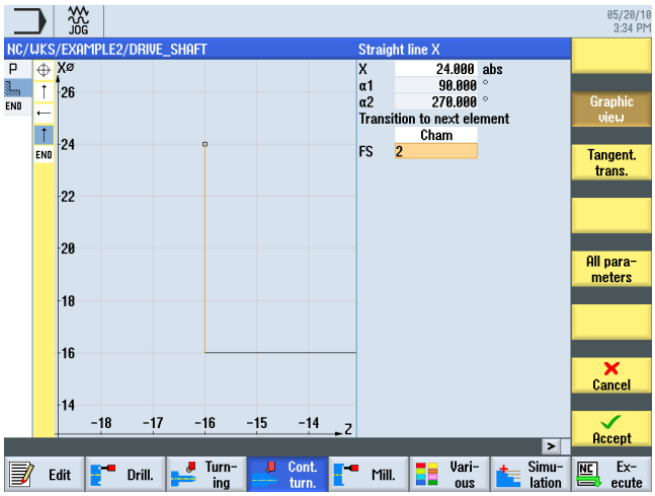
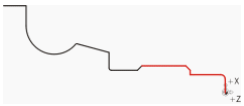


Figure 7-10 Specifying the vertical straight line contour

Accept the entered values.

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

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

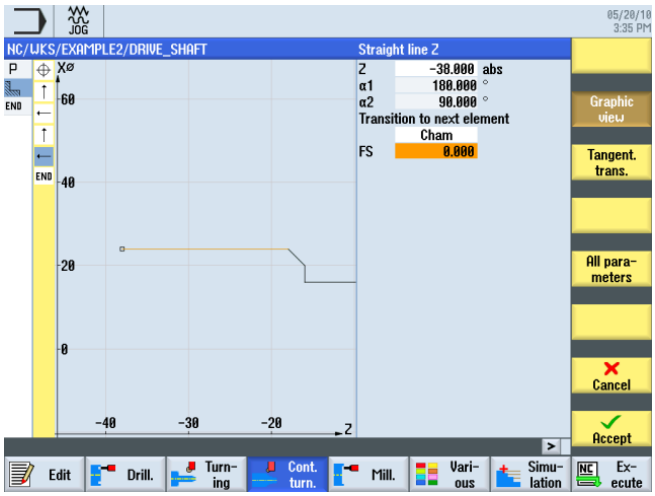
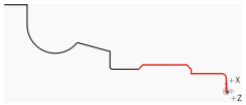


Figure 7-11 Specifying the horizontal straight line contour

Accept the entered values.

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

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

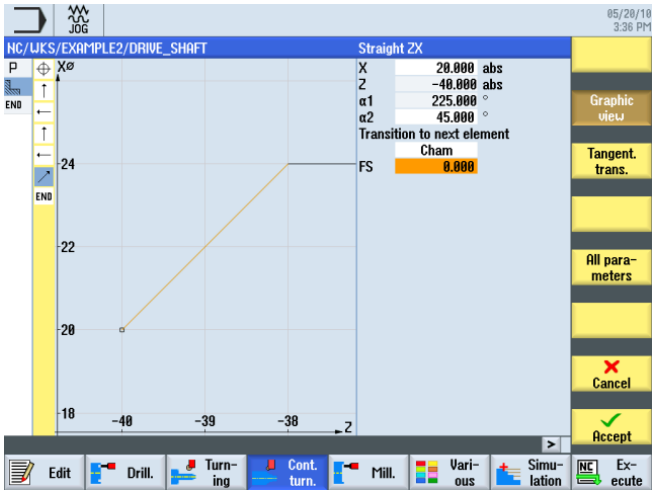
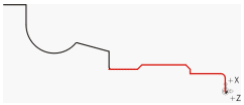


Figure 7-12 Specifying the sloping straight line contour

Accept the entered values.

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

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

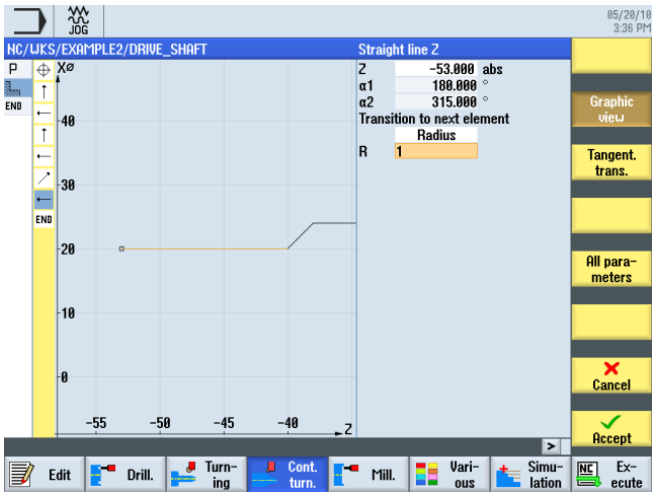
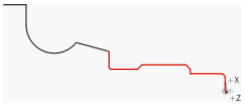


Figure 7-13 Specifying the horizontal straight line contour

Accept the entered values.

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

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		

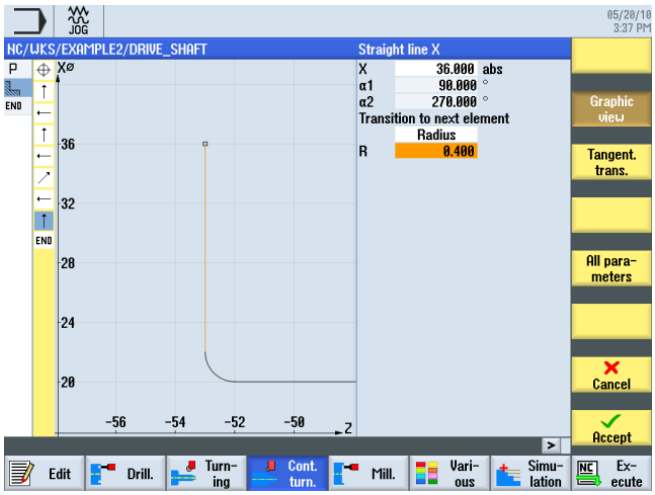
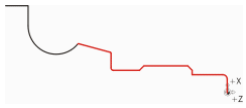


Figure 7-14 Specifying the vertical straight line contour

Accept the entered values.

Enter the following values for the next section in the interactive screenform:

Field	Value	Selection via toggle key	Notes
X		X	 Nothing more is known of the path than the angle to the Z axis, 165.167°. In such cases, simply continue the construction with the next element.
Z		X	
α1	165.167°		
Transition to next element	Radius	X	
R	0.4		

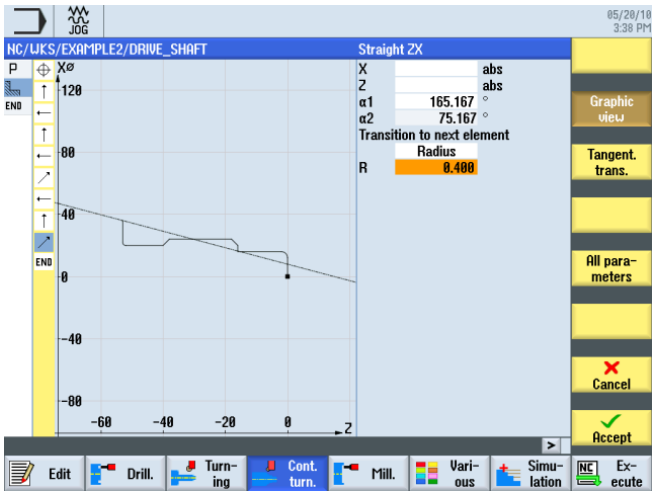


Figure 7-15 Specifying the inclined straight line contour

Accept the entered values.

Enter the following values for the next section in the interactive screenform:

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, it is important that the correct selection is made.
R	13		
X			
Z			
I	60 abs	X	
K	-78 abs	X	
Transition to next element	Chamfer	X	
R	0		

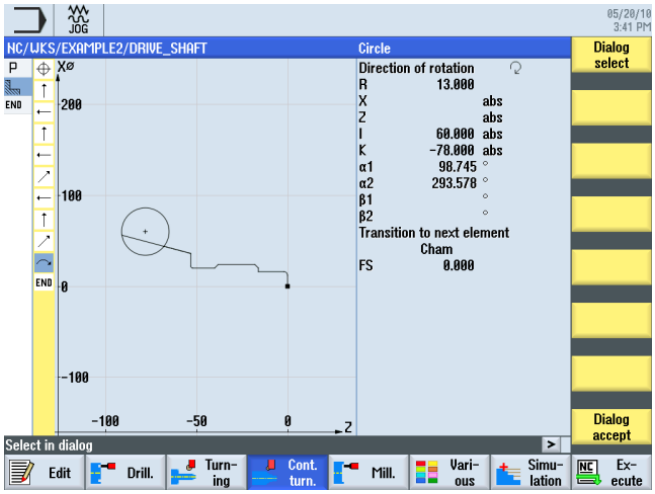


Figure 7-16 Specifying the arc contour

Select the recommended solution in accordance with the below illustration.

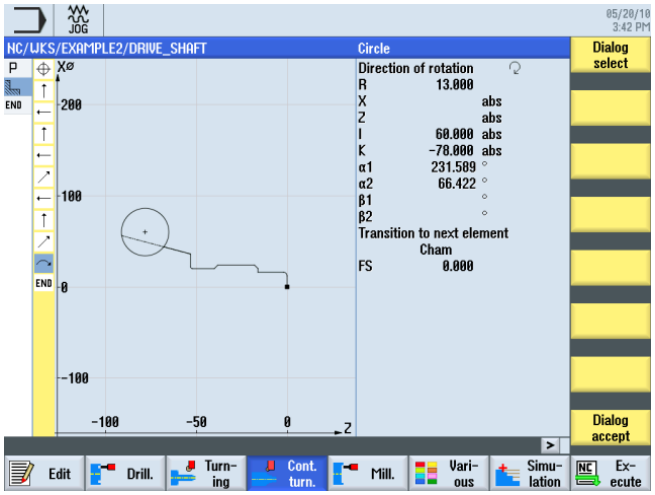


Figure 7-17 Confirming the contour selection

Dialog accept

Select the desired construction, then accept.

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

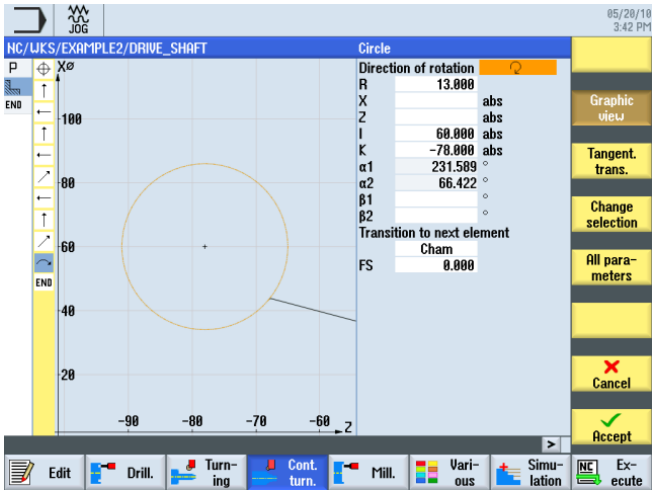


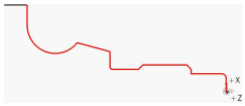
Figure 7-18 Accepting the arc contour

Accept

Accept the contour section.

A tangential path follows.

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

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

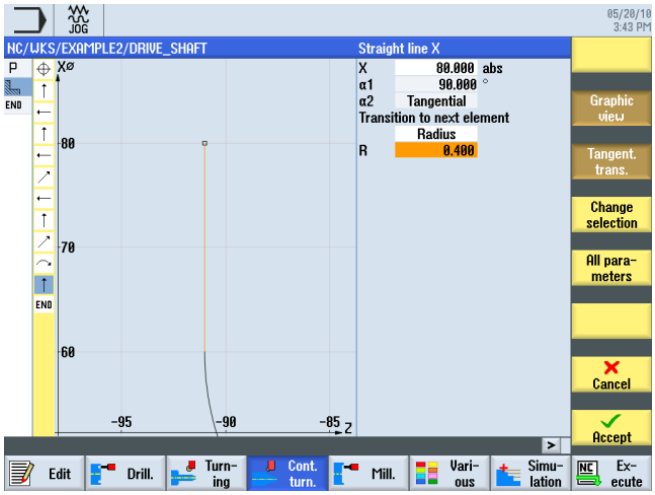


Figure 7-19 Specifying the vertical straight line contour

Accept the entered values.

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

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	Chamfer	X	
FS	0		

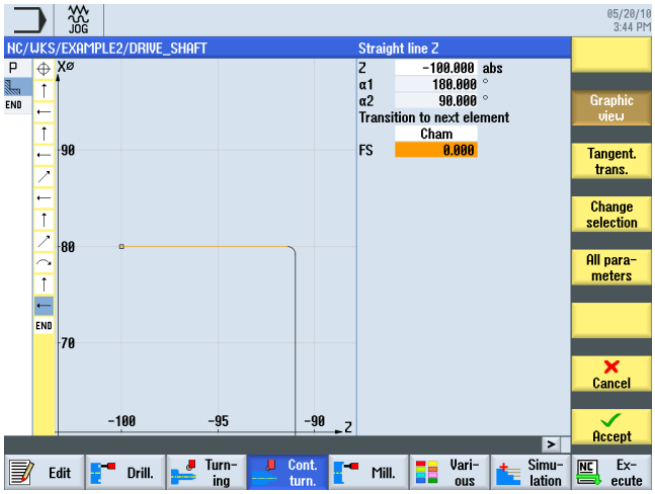


Figure 7-20 Specifying the horizontal straight line contour

Accept the entered values.

Accept the contour into your process plan.

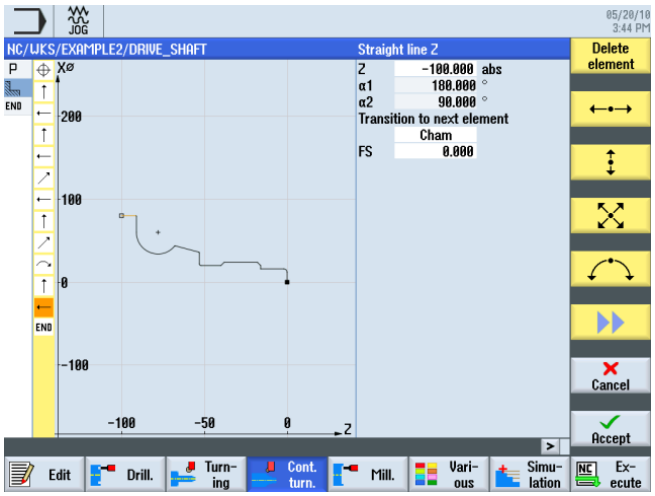


Figure 7-21 Accepting the contour

Stock removal, residual stock removal and finishing

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

Select the **Stock removal** softkey.

Open the tool list and select the ROUGHING_T80 A tool.

Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.3		
S	240 rpm	X	
Machining	contour-parallel external roughing	X X X	The machining of the contour, for example, is performed parallel to the contour here.
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Cylinder	X	
XD	0.0 inc	X	
ZD	0.0 inc	X	
Relief cuts	No	X	
Set machining area limits	No	X	

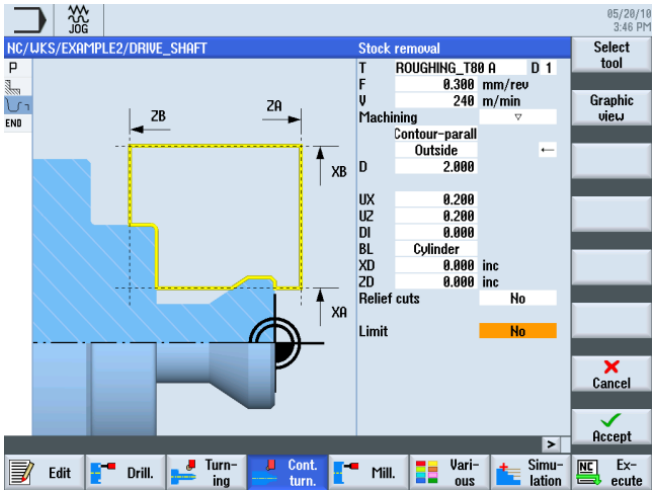


Figure 7-22 Roughing the contour

Accept the entered values.

Select the **Simulation** softkey.

Select the **Side view** softkey.

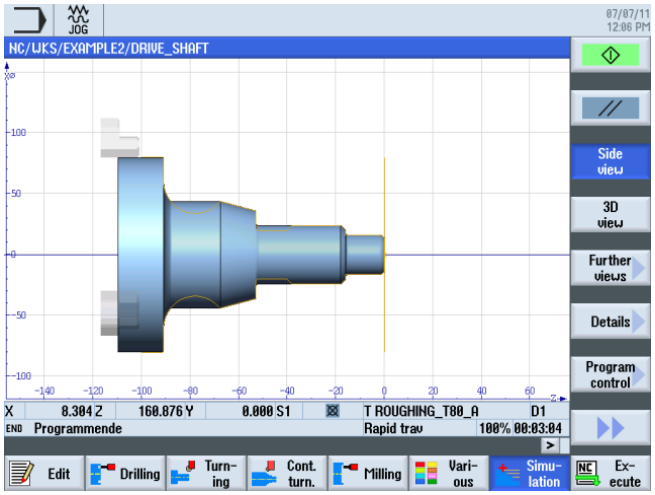


Figure 7-23 Roughing the contour - side view simulation

Select the **Contour turning** softkey.

Select the **Residual stock removal** softkey.

Open the tool list and select the FINISHING_T35 A tool.

Accept the tool into your program.

Enter the following values for residual stock removal in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.12		
V	240 m/min	X	
Machining	longitudinal external roughing	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 switch the input field to *Yes* .
FR	0.2		
Set machining area limits	No	X	

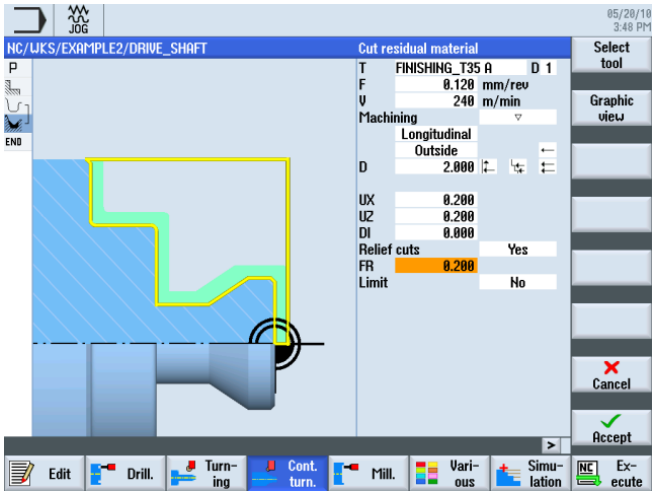


Figure 7-24 Cutting contour residual material

Accept the entered values.

Select the **Simulation** softkey.

Extend the menu.

Show tool path

Activate the display of traversing paths.

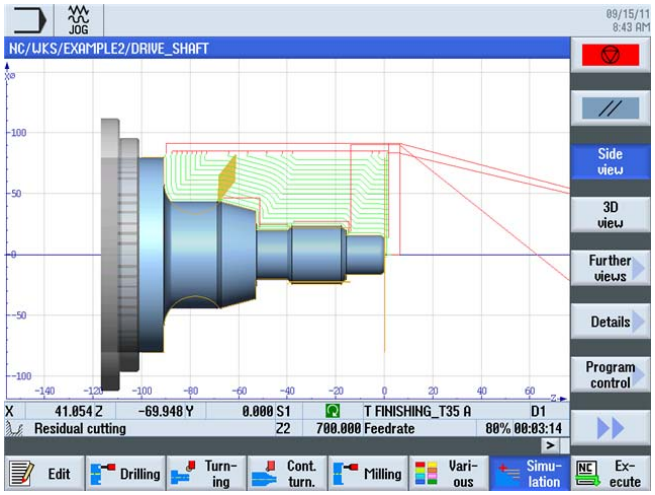


Figure 7-25 Cutting residual material - side view simulation

Cont. turn.

Select the **Contour turning** softkey.

Stock removal

Select the **Stock removal** softkey.

Tool

Open the tool list and select the FINISHING_T35 A tool.

To program

Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.12		
S	280 rpm	X	
Machining	longitudinal external finishing	X X X	
Allowance	No	X	
Relief cuts	Yes	X	
Set machining area limits	No	X	

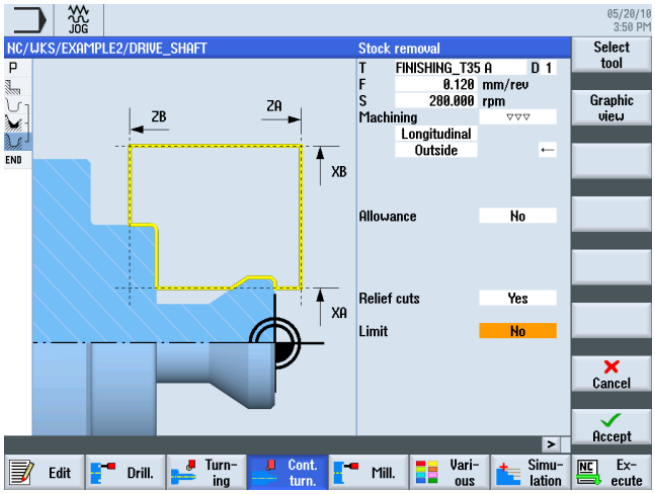


Figure 7-26 Finishing the contour

Accept

Accept the entered values. After acceptance, the process plan looks like this.

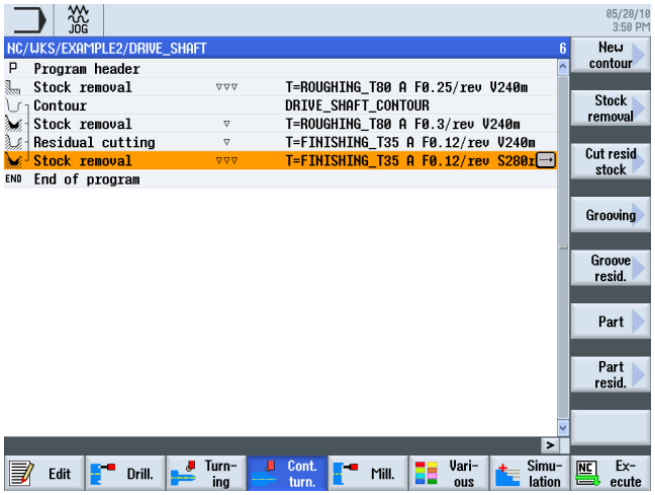


Figure 7-27 Process plan

Simu-lation

Start the simulation.

Details

Select the **Details** softkey. Here you can expand or minimize the view.

Zoom +

The **Zoom +** softkey expands the view.

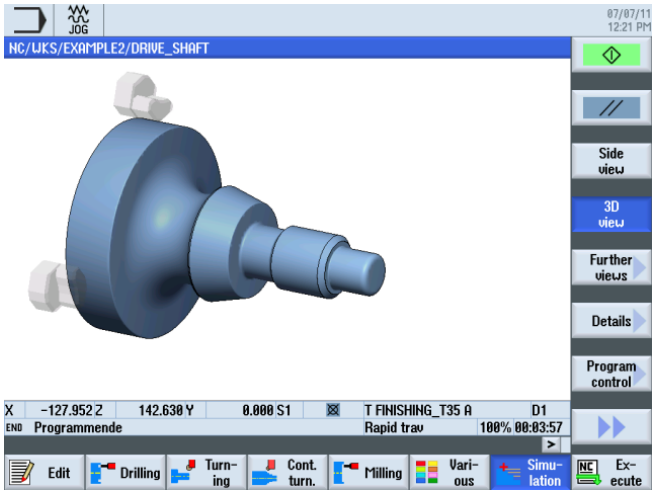


Figure 7-28 3D view simulation - details

7.4 Thread

Operating sequences

Proceed as follows to create a thread.

Thread

Select the **Thread** softkey.

Select tool

Open the tool list and select THREADING_T1.5.

To program

Accept the tool into your program.

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

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 flanks	X X	
ND	8		
U	0.1		
NN	0		
VR	2		
Multiple threads	No	X	
α0	0		

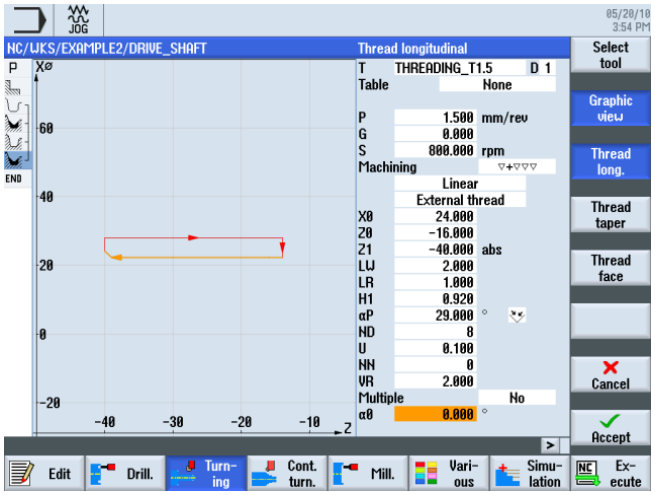


Figure 7-29 Producing a thread

Accept the entered values.

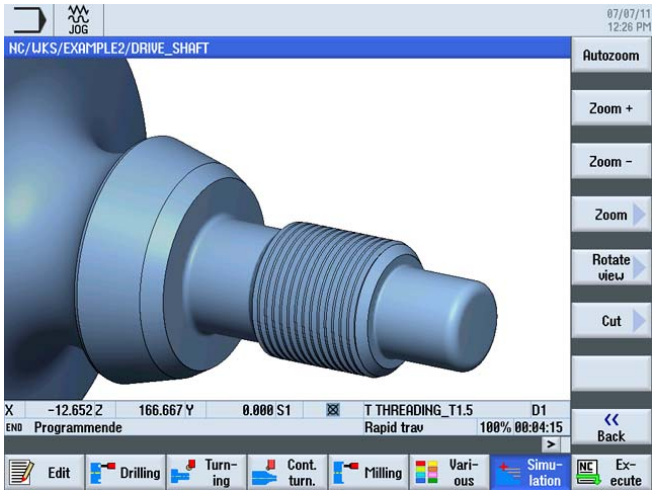


Figure 7-30 3D view simulation - details

Example 3: Guide shaft

8

8.1 Overview

Learning objectives

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

- create any blank;
- perform stock removal of the difference material between the blank and the machined part;
- drill on the front face;
- mill on the front face;

Task

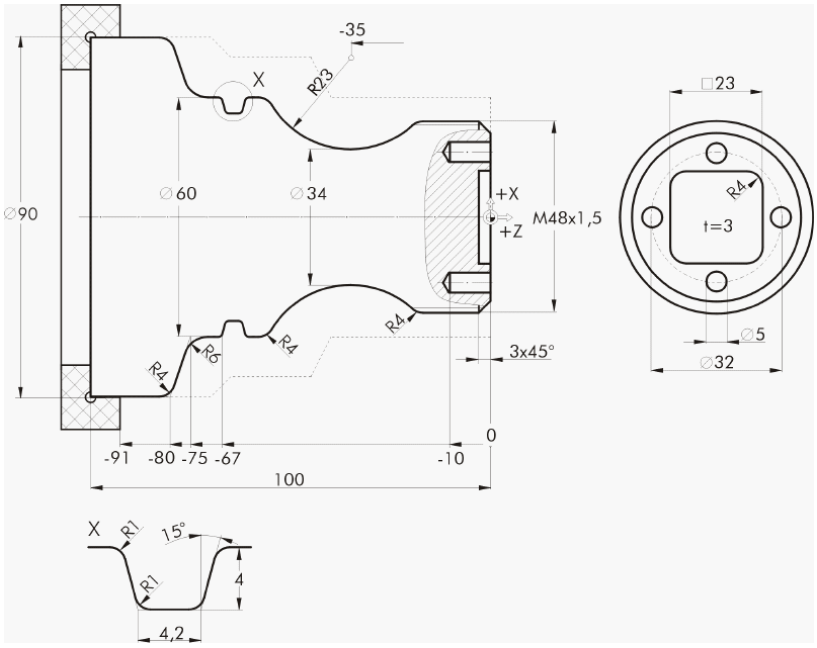


Figure 8-1 Workshop drawing - Example 3:

Preparation

Perform the following steps without help:

1. Create a new workpiece with the name 'EXAMPLE3'.
2. Create a new step sequence program with the name 'GUIDE_SHAFT'.
3. Fill in the program header (see illustration below).

Note

Although you can do this for any blank, select the *Cylinder* blank here. ShopTurn ignores this input and selects an arbitrary blank.

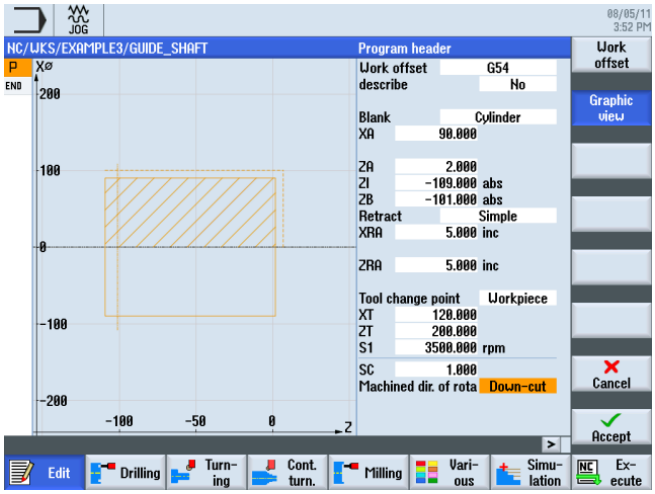


Figure 8-2 Creating a program header

8.2 Face turning

Operating sequences

Proceed as follows to create a new program and face turn 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 interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.25		
V	240 m/min	X	
Machining	Finishing	X	
Position	(see illustration below)	X	
Machining direction	Chart	X	
X0	60		Since the blank has a diameter of 60mm, you must also set dimension X0 to 60 in this work step.
Z0	2		
X1	-1.6 abs	X	
Z1	0.0 abs	X	
D	1.5		
UX	0.0		
UZ	0.2		

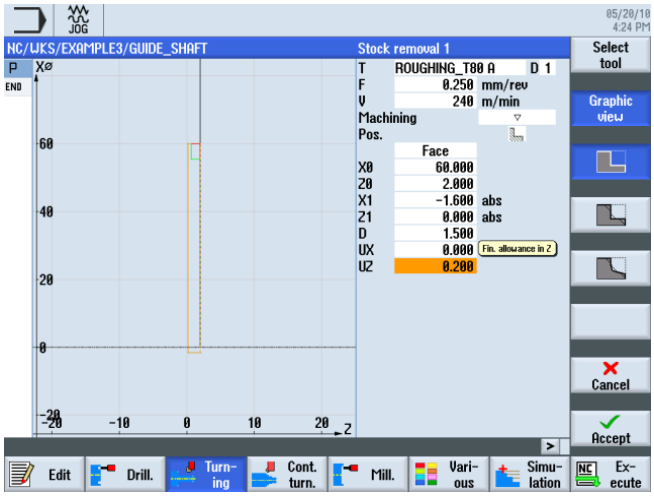


Figure 8-3 Face turning the workpiece

Accept the entered values.

Start the simulation to check the work step.

Show tool path

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

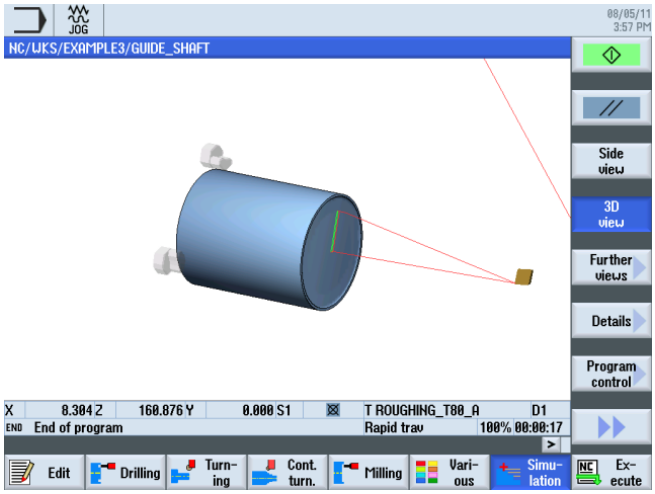
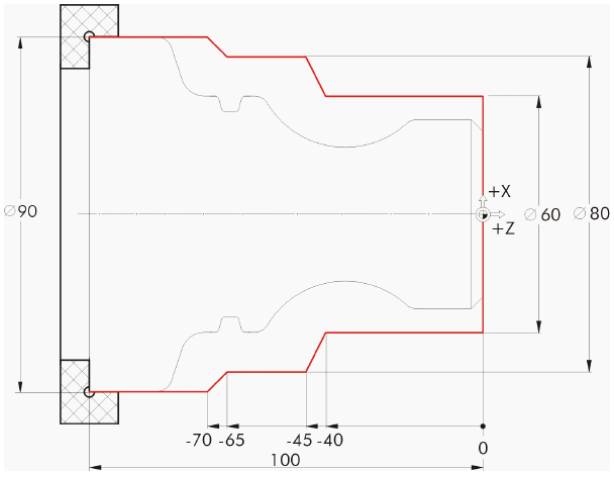


Figure 8-4 Face turning simulation

8.3 Creating any blank contour

Operating sequences

Enter the following blank contour without help:



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 illustration below).

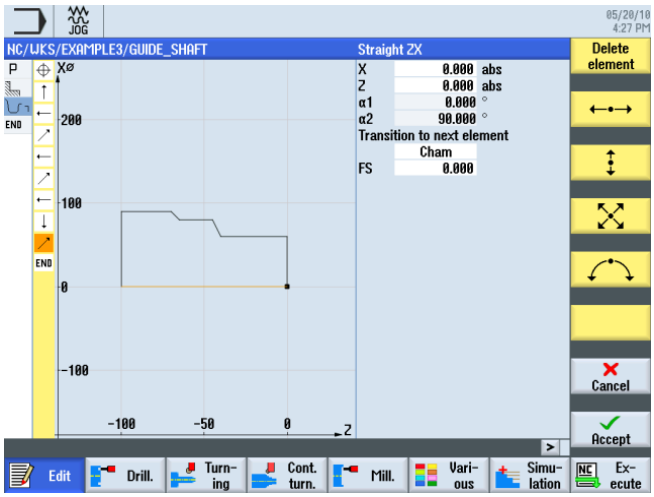


Figure 8-6 Any blank contour

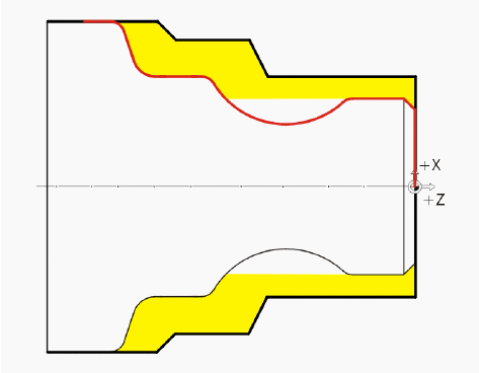
Note

The contour must be closed.

8.4 Creating the machined part contour and stock removal

Operating sequences

Proceed as follows to enter the machined part contour:



Select the **Contour turning** softkey.

Select the **New contour** softkey. Enter the name 'GUIDE_SHAFT_CONTOUR' for the contour.

Figure 8-7 Creating the contour

Confirm your input.

Since the blank was assigned to Z0 in the first work step, you can accept the starting point X0/Z0 directly (see illustration below).

Figure 8-8 Specifying the contour starting point

Confirm your input.

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

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

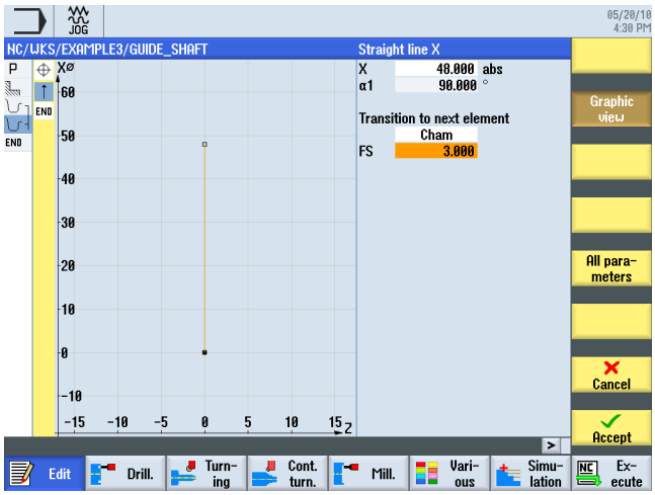


Figure 8-9 Specifying the vertical straight line contour

Accept the entered values.

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

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

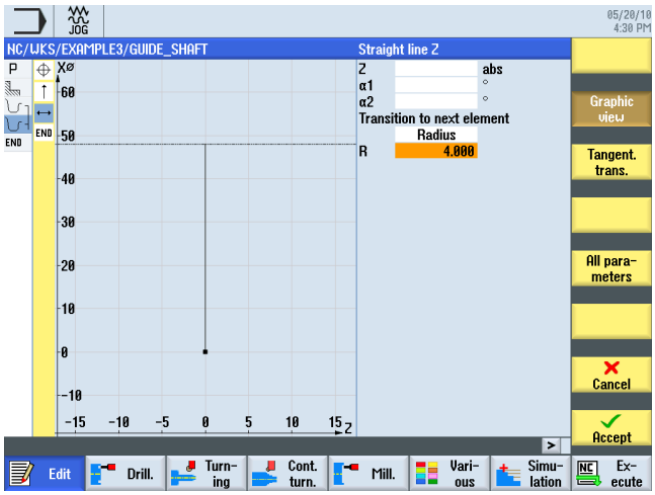
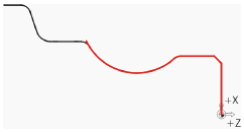


Figure 8-10 Specifying the horizontal straight line contour

Accept the entered values.

Enter the following values for the next section in the interactive screenform:

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 entering the contour data (e.g. in the case of circular arcs here), you can select these via the *Selection dialog* softkey.

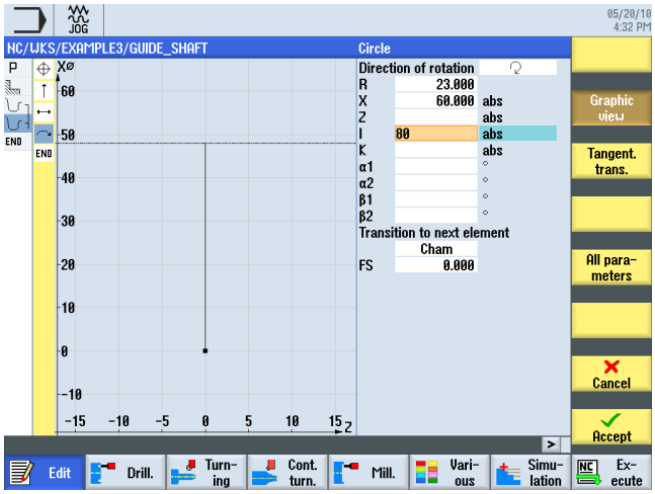


Figure 8-11 Specifying the arc contour

Dialog select

Select the recommended solution in accordance with the below illustration.

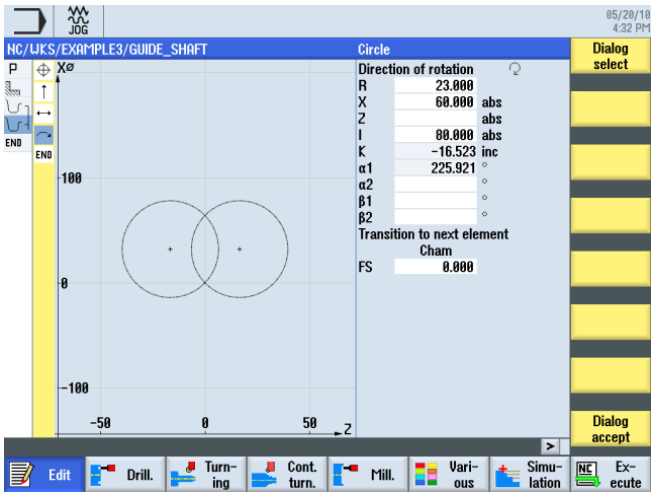


Figure 8-12 Selecting an arc contour

Dialog accept Select the desired construction, then accept.

Dialog select Select the recommended solution in accordance with the below illustration.

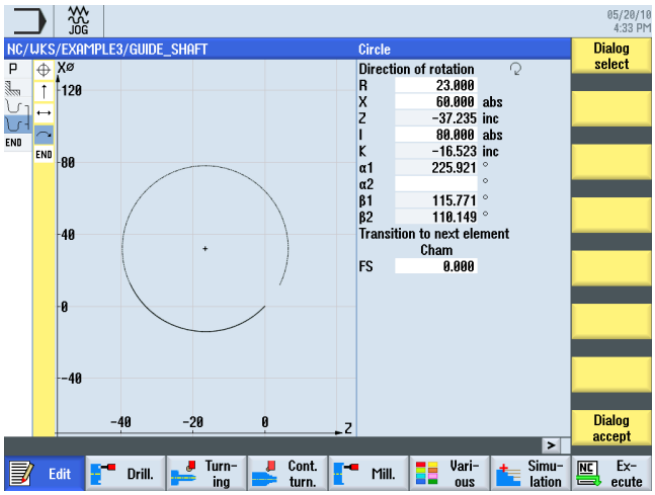


Figure 8-13 Selecting an arc contour

Dialog accept Select the desired construction, then accept.

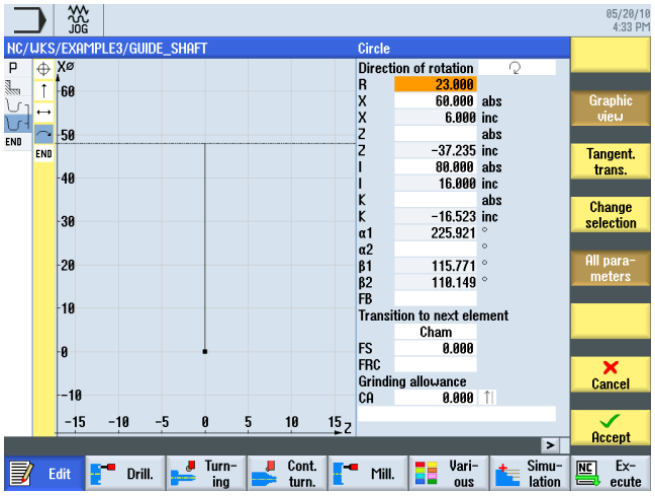


Figure 8-14 Accepting the selected arc contour

To complete the arc, proceed as follows:

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

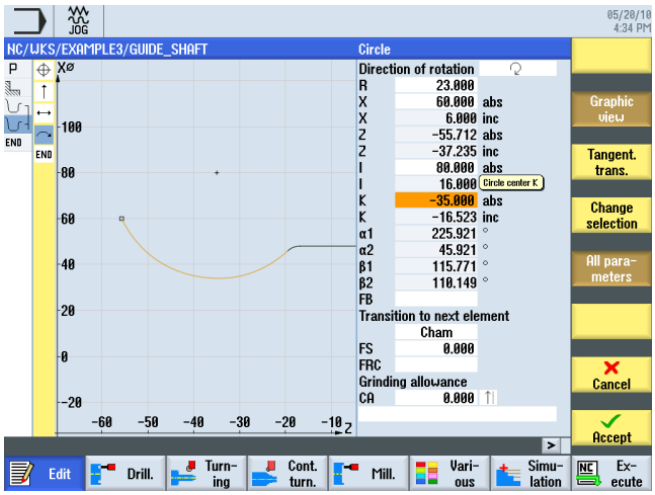


Figure 8-15 Specifying the arc contour center point

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

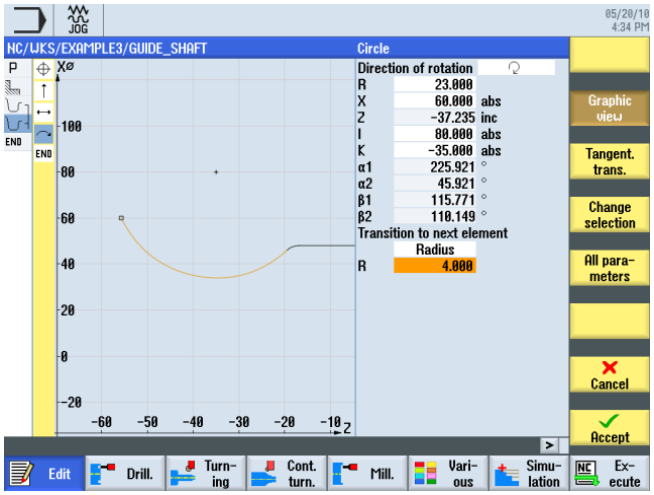
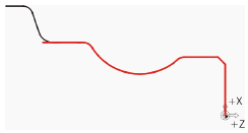


Figure 8-16 Specifying the arc contour radius

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

Accept the contour section.

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

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

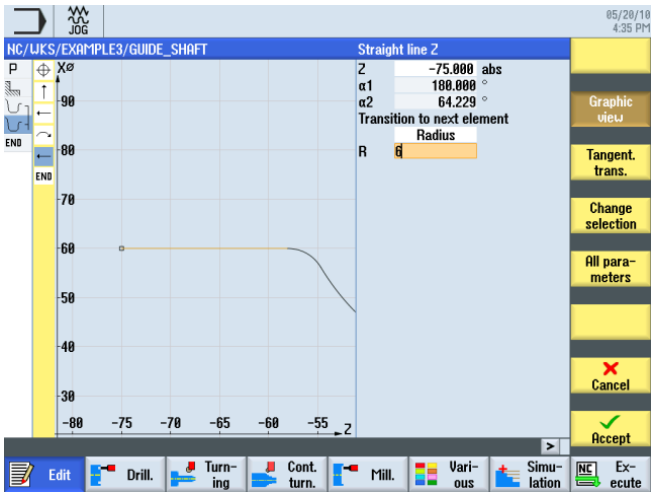
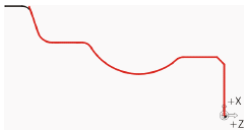


Figure 8-17 Specifying the horizontal straight line contour

Accept the entered values.

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

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

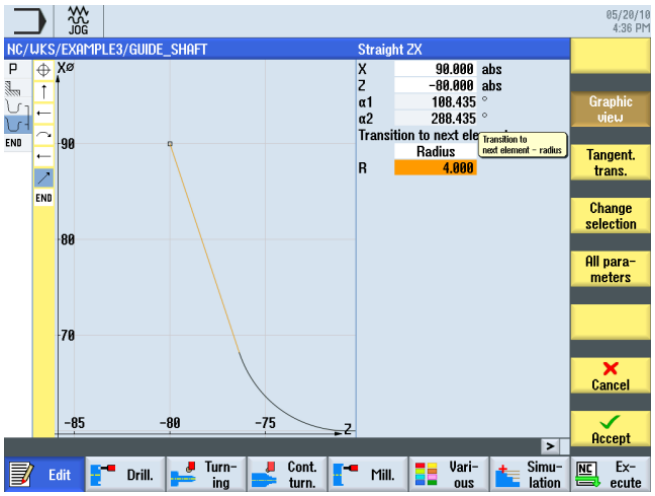
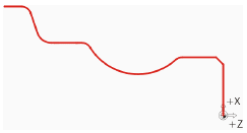


Figure 8-18 Specifying the inclined straight line contour

Accept

Accept the entered values.

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

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

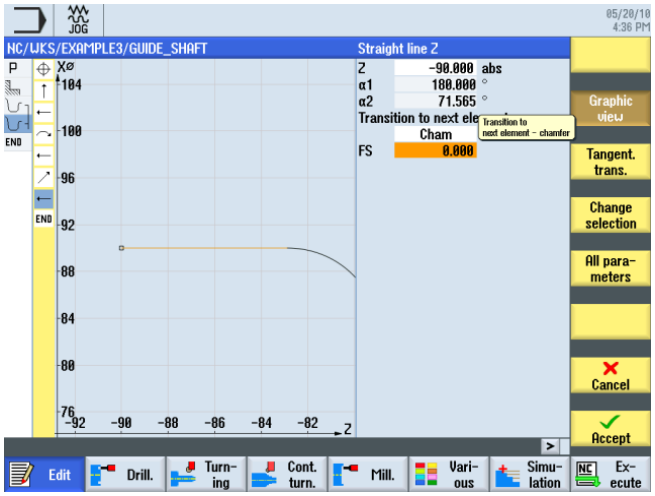


Figure 8-19 Specifying the horizontal straight line contour

Accept the entered values.

Accept the contour into your process plan.

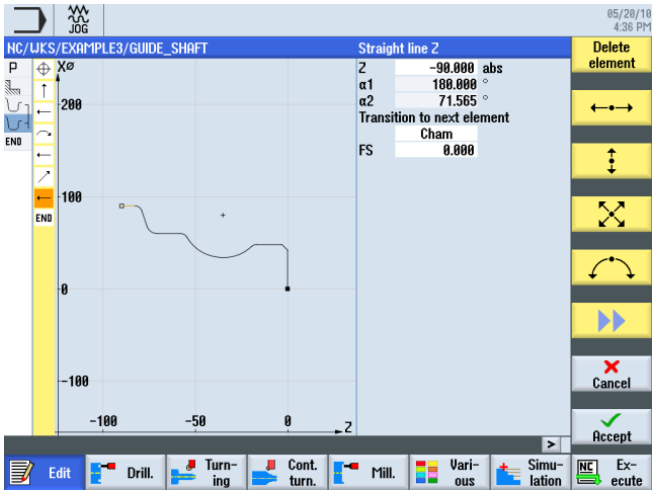


Figure 8-20 Accepting the contour

Stock removal

In the following work step you will perform contour stock removal.

To this end, proceed as follows:

Select the **Stock removal** softkey.

Open the tool list and select the ROUGHING_T80 A tool.

Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.3		
V	260 m/min	X	
Machining	longitudinal external roughing	X X X	
D	2.5		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Contour	X	The blank description must be switched to contour here.
Relief cuts	No	X	To ensure that the recess of radius 23 is not machined, you must switch to *No*.
Set machining area limits	No	X	

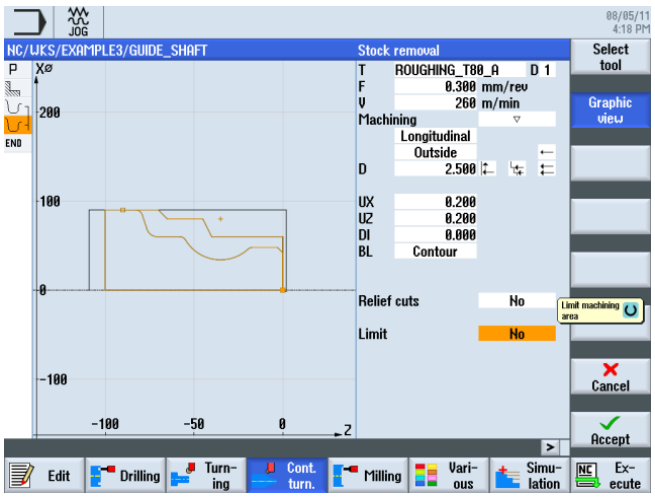


Figure 8-21 Contour stock removal

Accept the entered values. Both contours and the work step are linked together following acceptance.

Select the **Simulation** softkey.

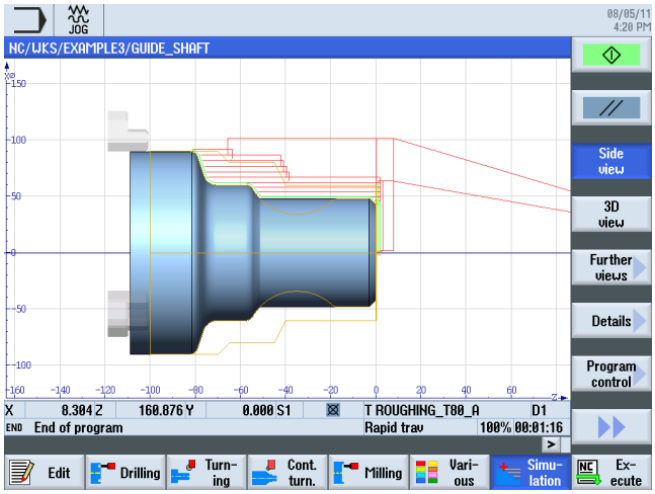


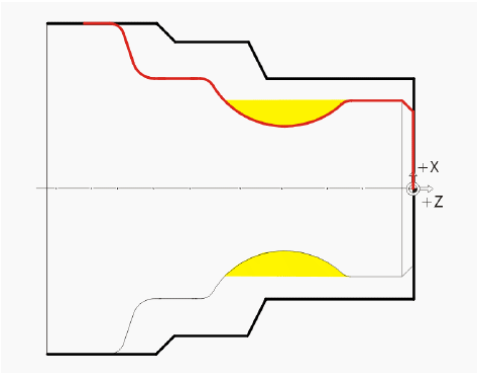
Figure 8-22 Contour stock removal simulation (with display of the traversing paths).

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

8.5 Residual stock removal

Operating sequences

Proceed as follows to cut the residual material:



The following illustration shows the process plan up to roughing machining:

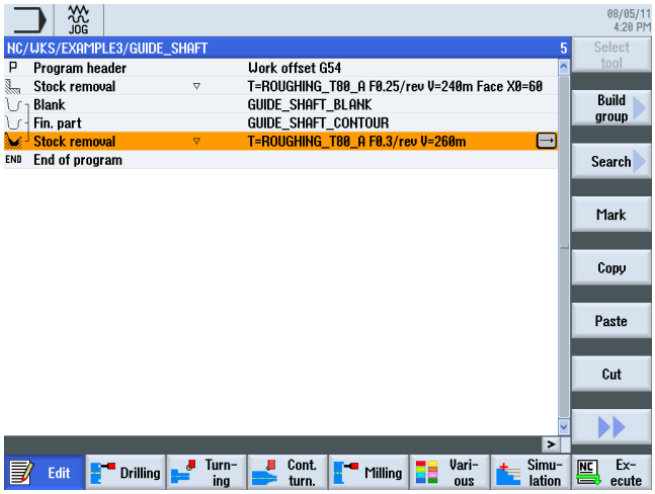


Figure 8-23 Process plan including roughing machining

Select the **Contour turning** softkey.

Select the **Residual stock removal** softkey.

Open the tool list and select the BUTTON_TOOL_8 tool.

Accept the tool into your program.

Enter the following values for residual stock removal in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.25		
V	240 m/min	X	
Machining	longitudinal external roughing	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
Relief cuts	Yes	X	Machining with relief cuts must be switched to *Yes* here.
FR	0.2		
Set machining area limits	No	X	

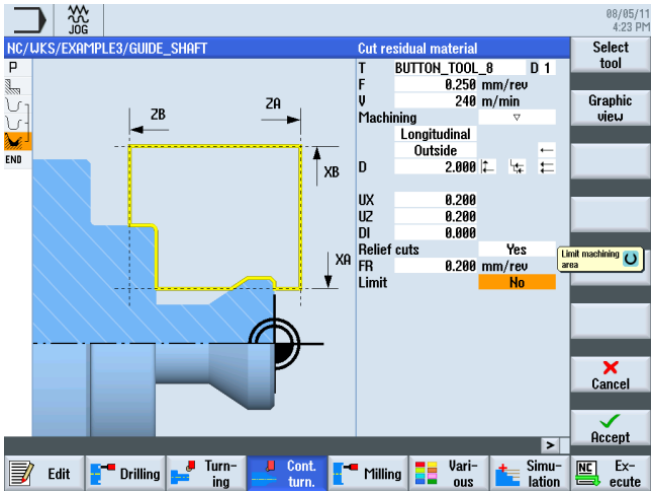


Figure 8-24 Cutting contour residual material

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

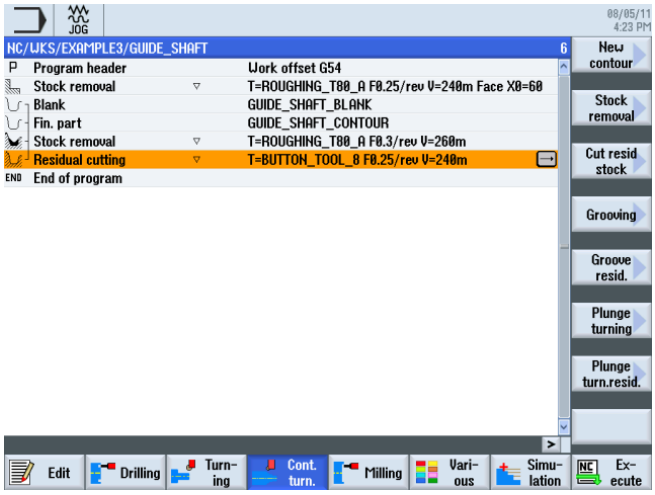


Figure 8-25 Process plan with residual stock removal

Simu-lation

Start the simulation.

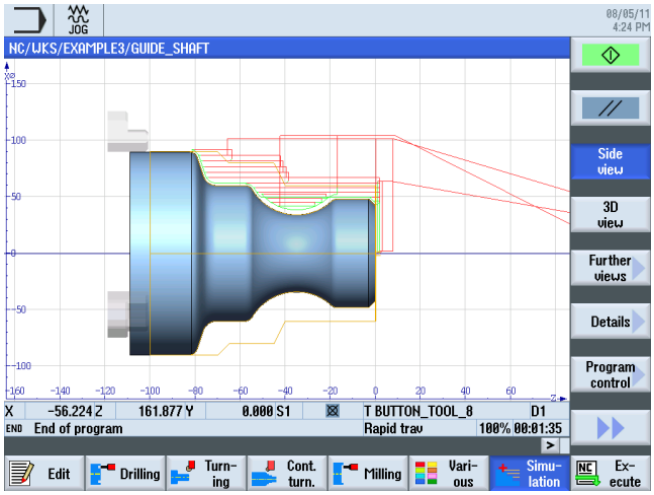


Figure 8-26 Residual stock removal simulation

After roughing the contour, it must then be finished.

Cont. turn.

Select the **Contour turning** softkey.

Stock removal

Select the **Stock removal** softkey.

Tool

Open the tool list and select the FINISHING_T35 A tool.

To program

Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.12		
S	280 m/min	X	
Machining	longitudinal external finishing	X X X	
Allowance	No	X	
Relief cuts	Yes	X	
Set machining area limits	No	X	

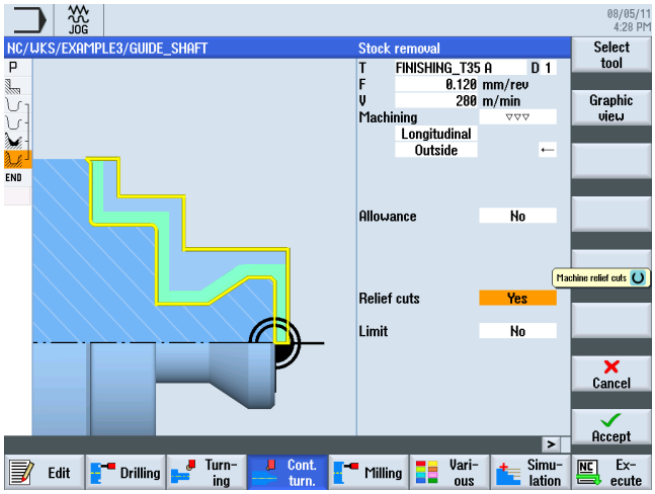


Figure 8-27 Finishing the contour

Accept the entered values.

Start the simulation.

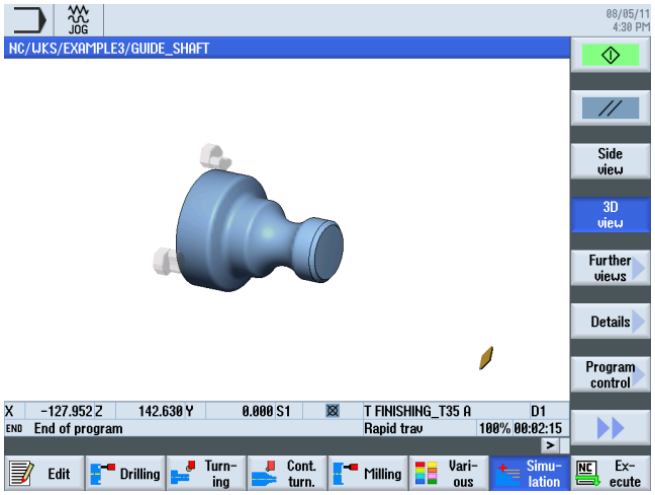
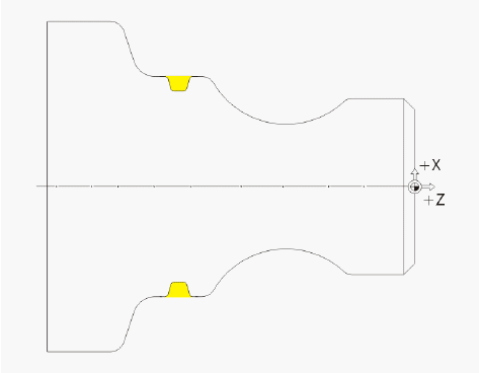


Figure 8-28 Finishing simulation - 3D view

8.6 Groove

Operating sequences

Proceed as follows to create a groove:



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

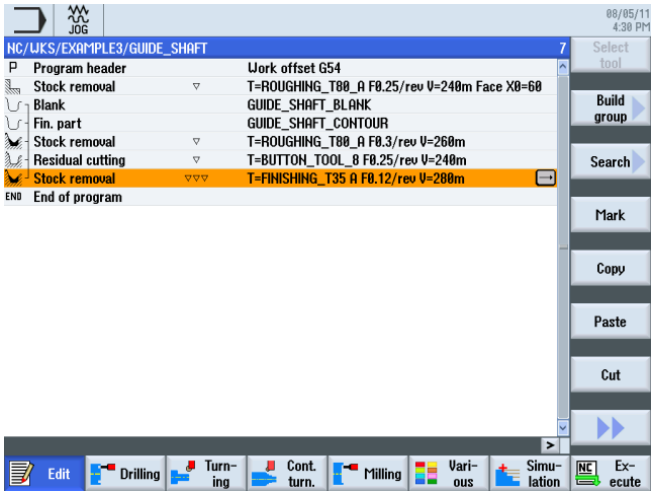


Figure 8-29 Process plan following stock removal

Select the **Turning** softkey.

Select the **Groove** softkey.

Select the second groove shape (groove 2).

Open the tool list and select PLUNGE_CUTTER_3 A.

To program

Accept the tool into your program.

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

Field	Value	Selection via toggle key	Notes
F	0.1 mm/rev		
V	150 m/min	X	
Machining	Roughing + finishing	X	
Position	see illustration 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 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		

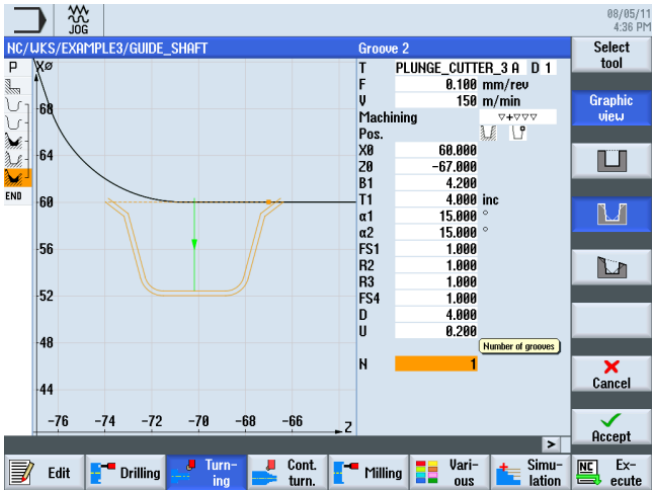


Figure 8-30 Creating a groove

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

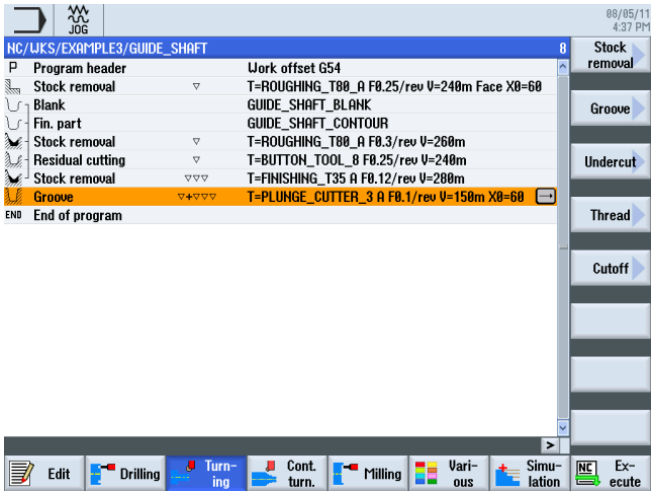


Figure 8-31 Process plan including groove

Start the simulation. You can check subareas of the workpiece using the **Magnifying glass** softkey.

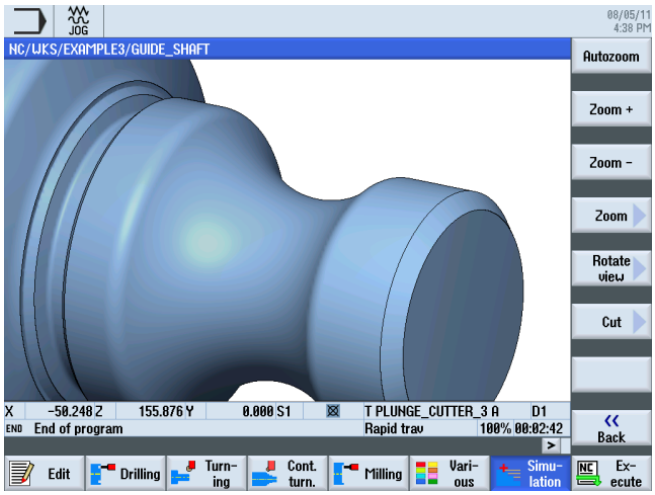
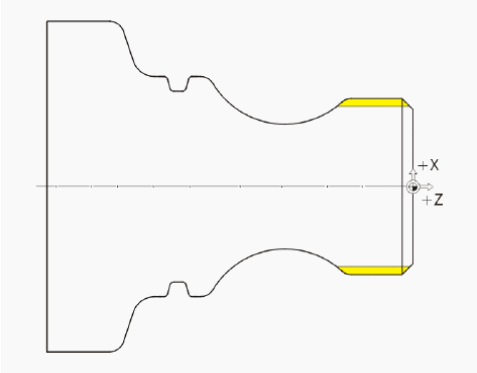


Figure 8-32 Simulation - 3D view (magnifying glass)

8.7 Thread

Operating sequences

Proceed as follows to create a thread.



Select the **Turning** softkey.

Select the **Thread** softkey.

Open the tool list and select THREADING_T1.5.

Accept the tool into your program.

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

Field	Value	Selection via toggle key	Notes
P	1.5 mm/rev	X	
G	0		
S	800 rpm	X	
Machining	Roughing Diminishing External thread	X X X	The thread is created with the *diminishing* setting. This setting causes the cutting division to be reduced for each cut, and so ensures that the cutting cross-section remains constant.
X0	48		
Z0	-3		
Z1	-23 abs	X	
LW	4	X (field)	
LR	2		

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

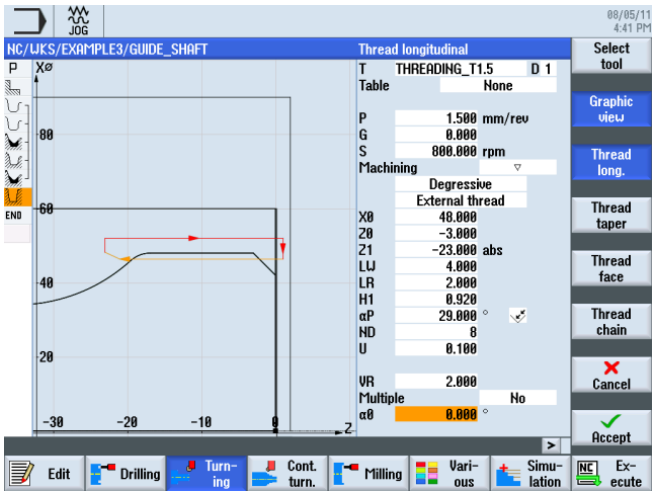


Figure 8-33 Producing a thread

Switch to the help display when necessary.

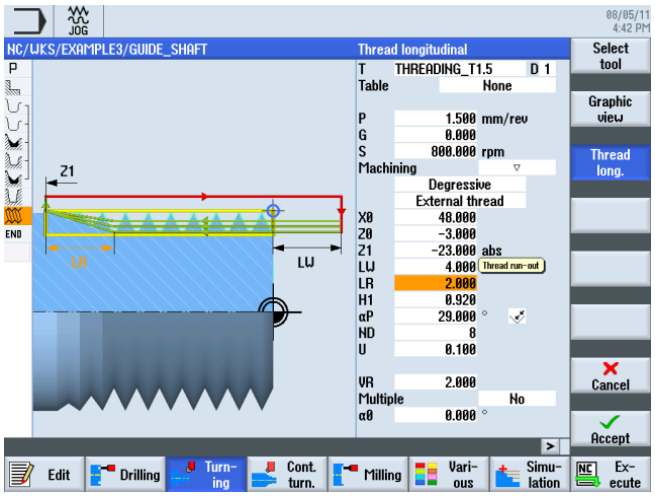


Figure 8-34 Help display - thread exit

Accept the entered values.

Start the simulation. You can check subareas of the workpiece using the **Details** softkey.

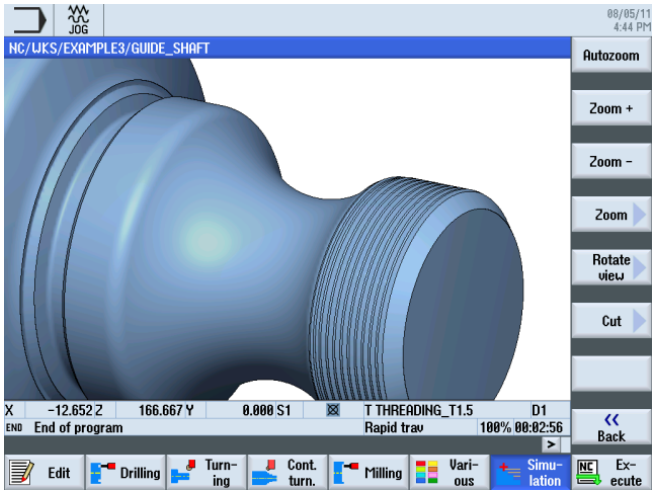
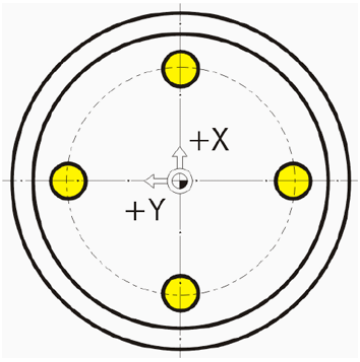


Figure 8-35 3D view simulation - details

8.8 Drilling

Operating sequences

Proceed as follows to create drill holes on the front face (C axis or complete machining sequence).



After the thread has been created, the list of work steps looks like this:

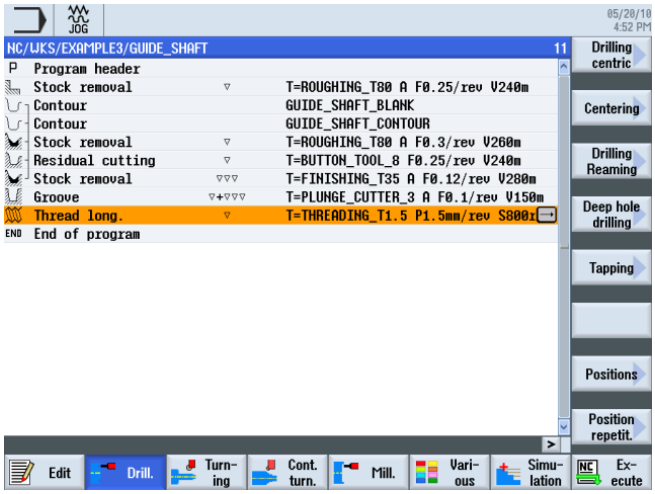


Figure 8-36 Process plan following thread creation

Select the **Drilling** softkey.

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

Select the **Drilling** softkey.

Open the tool list and select DRILL_5.

Accept the tool into your program.

Enter the following values for the drill hole in the interactive screenform:

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 10mm incremental or -10mm absolute.
DT	0 s	X	

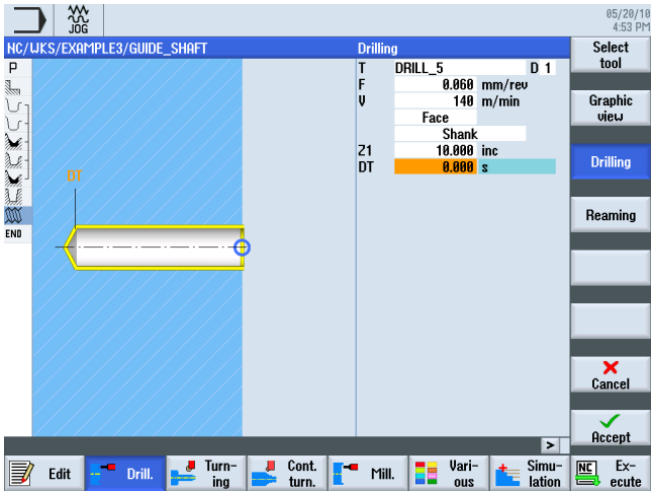


Figure 8-37 Drilling

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

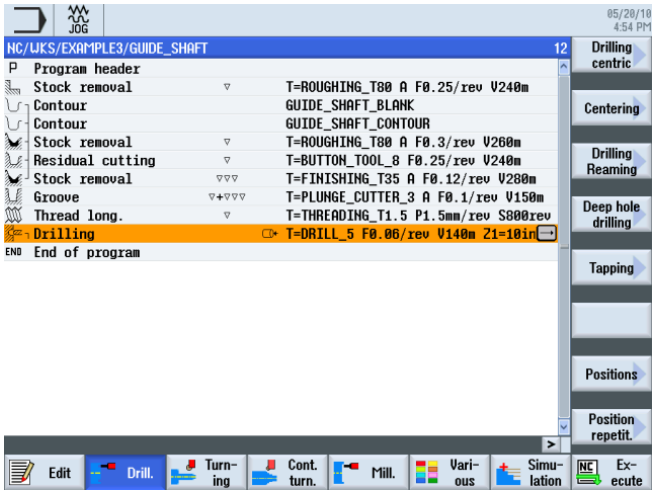


Figure 8-38 Process plan following drilling

During the drilling work step, an open connector appears in the list of work steps. This is 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. A simpler solution here would be to use the position circle.

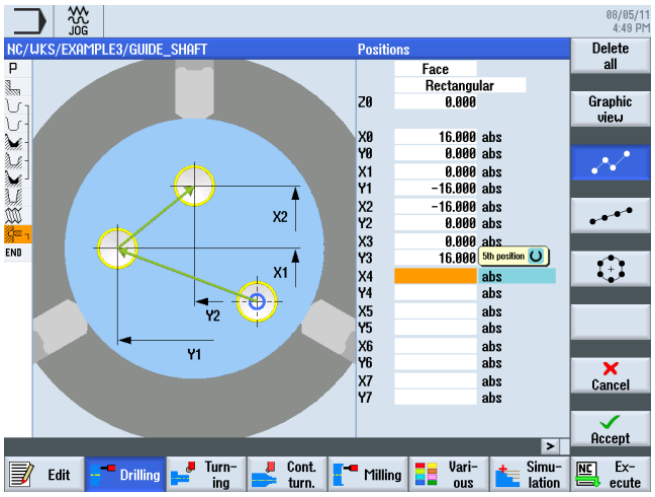


Figure 8-39 Entering the positions

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

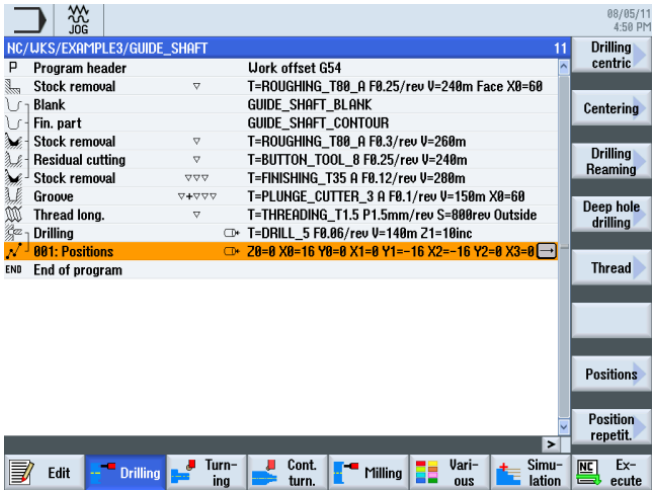


Figure 8-40 Process plan following input of the position pattern

The drill holes are now linked to the drilling positions.

Start the simulation.

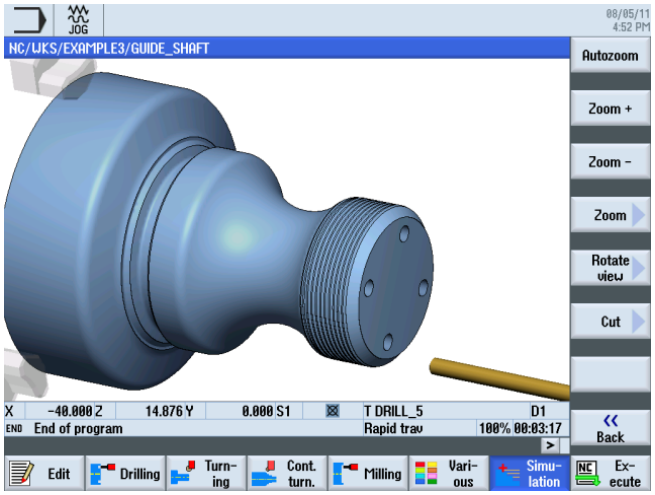


Figure 8-41 Simulation - 3D view

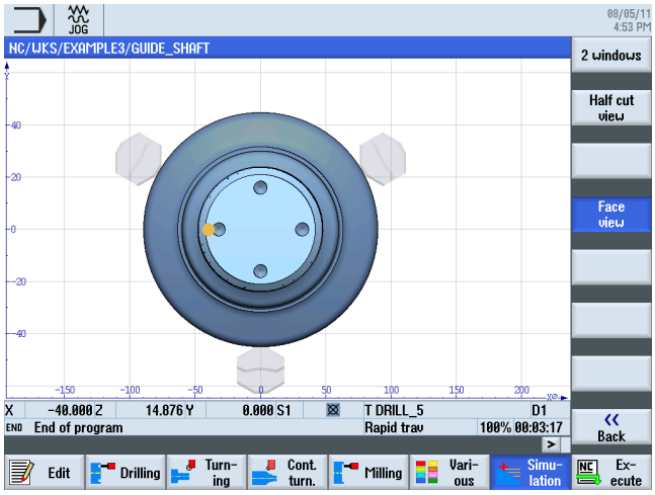
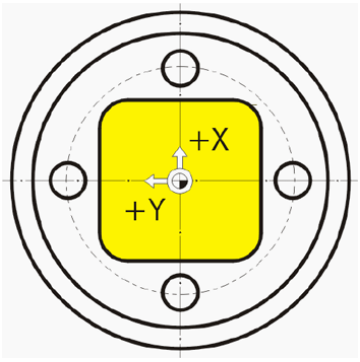


Figure 8-42 Simulation - front view

8.9 Milling the rectangular pocket

Operating sequences

Proceed as follows to create a rectangular pocket on the front face (C axis or complete machining sequence).



Select the **Milling** softkey.

Select the **Pocket** softkey.

Select the **Rectangular pocket** softkey.

Open the tool list and select CUTTER_8.

Accept the tool into your program.

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

Field	Value	Selection via toggle key	Notes
F	0.03 mm/tooth	X	
V	220 m/min	X	
	Face	X	
Machining	Roughing Single positions	X X	
X0	0	X (field)	
Y0	0	X (field)	
Z0	0		
W	23		
L	23		
R	4		

Field	Value	Selection via toggle key	Notes
α0	0		
Z1	3 inc	X	
DXY	75%	X	
DZ	1.5		
UXY	0		
UZ	0		
Insertion	Helical	X	see insertion below
EP	1		
ER	7		

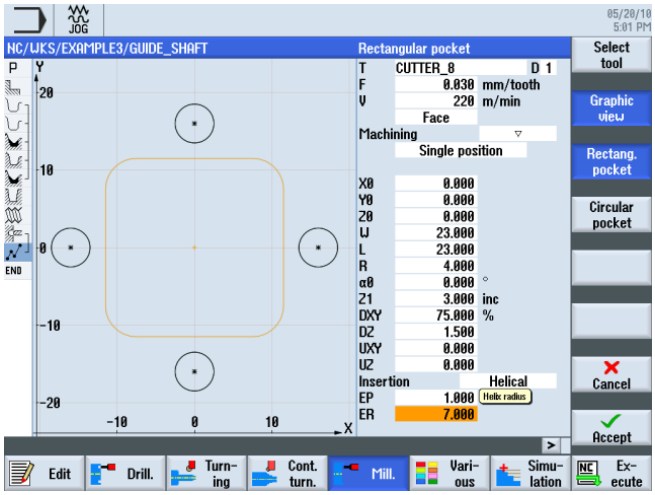


Figure 8-43 Producing a rectangular pocket

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

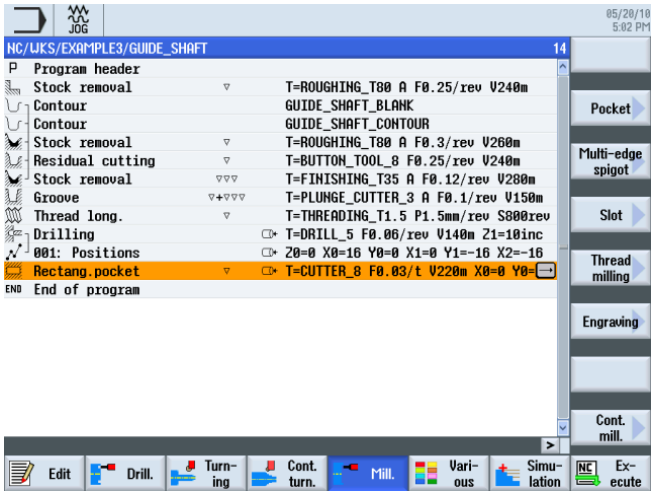


Figure 8-44 Process plan following rectangular pocket

Start the simulation.

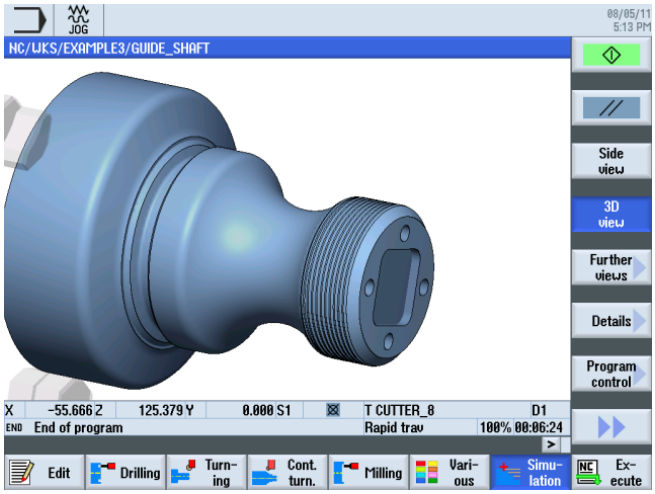
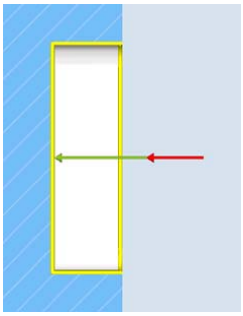
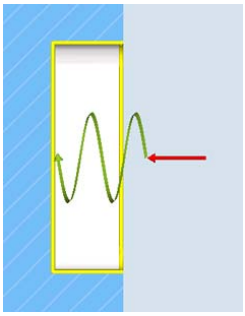
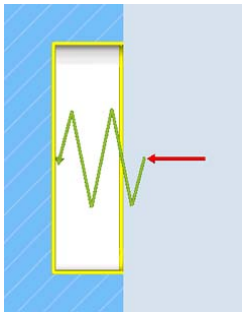


Figure 8-45 Simulation - 3D view

Insertion

Vertical insertion	Helical insertion	Oscillating insertion
		

Example 4: Hollow shaft

9

9.1 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 work step editor;
- create an undercut and
- an asymmetrical groove.

Task

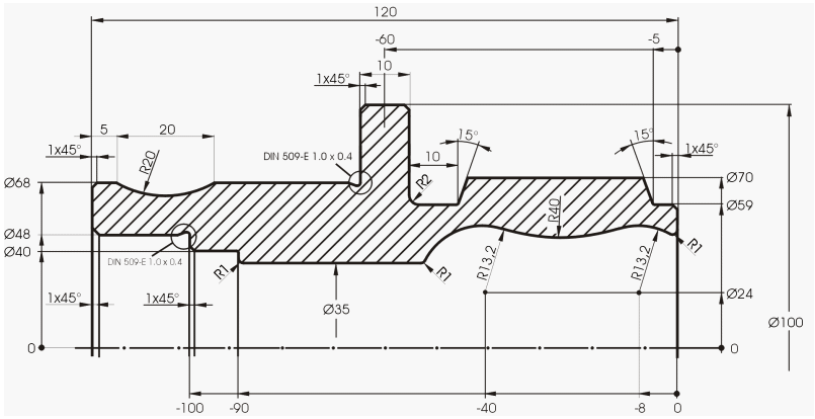


Figure 9-1 Workshop drawing - Example 4:

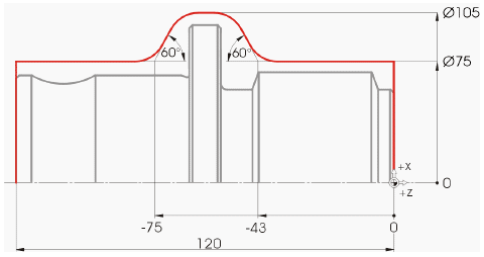


Figure 9-2 Blank contour

All non-dimensioned radii R10.

Note

On account of improved clamping, side 1 is manufactured first.

9.2 Creating the first workpiece side

Creating a process plan

Since the workpiece is to be machined from two sides (and produced without counterspindle), two process plans must be created.

Create the process plan for the left-hand side first ('HOLLOW_SHAFT_SIDE1')

Operating sequences

Create the program 'HOLLOW_SHAFT_SIDE1' without help.

Figure 9-3 Creating a ShopTurn program

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

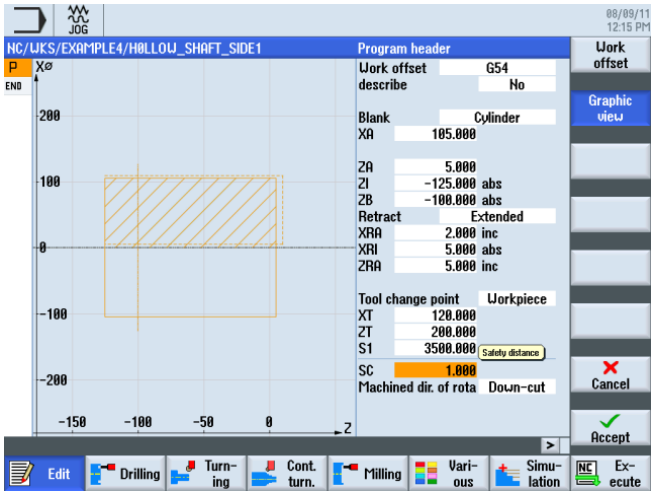


Figure 9-4 Workpiece dimensions in the program header

9.2.1 Face turning

Operating sequences

Proceed as follows to face turn 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 interactive screenform:

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

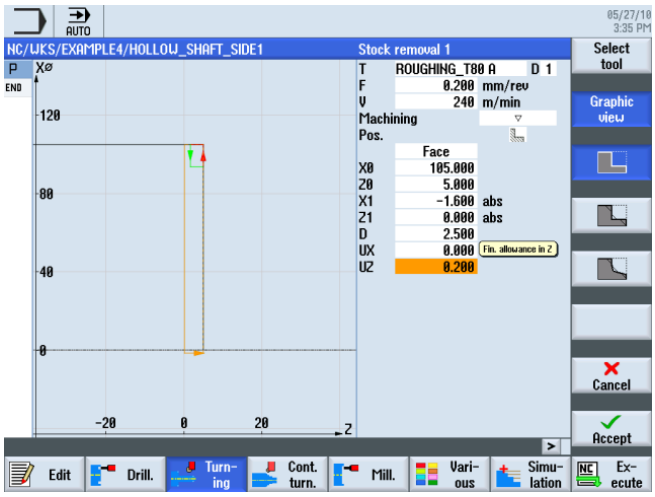


Figure 9-5 Face turning the workpiece

Accept the entered values. After acceptance, your work step program looks like this:

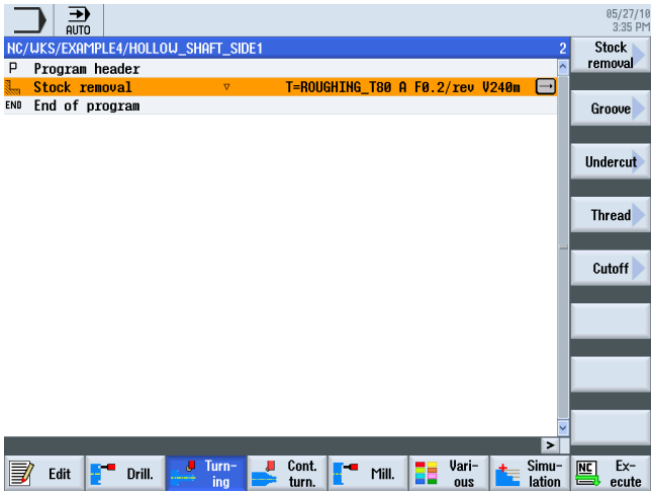


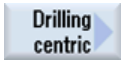
Figure 9-6 Process plan following face turning

9.2.2 Drilling

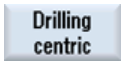
Operating sequences

Proceed as follows to drill the workpiece at the center.

Select the **Drilling** softkey.



Select the **Drilling Centric** softkey.



Select the **Drilling Centric** softkey.

Open the tool list and select DRILL_32.

Accept the tool into your program.

Enter the following values for the drill hole in the interactive screenform:

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		

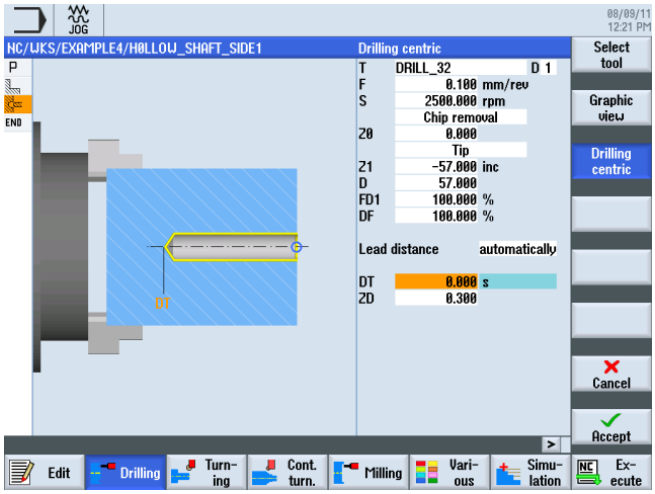


Figure 9-7 Hole

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

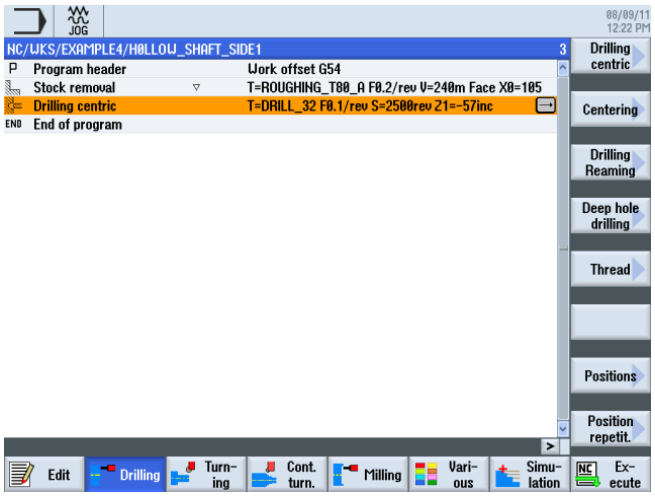
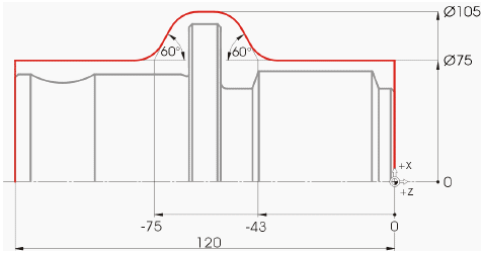


Figure 9-8 Process plan following drilling

9.2.3 Blank contour

Operating sequences

Enter the following blank contour without help. Since the workpiece is only machined on one side for each process plan, the blank contour only needs to be constructed 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 illustration below).

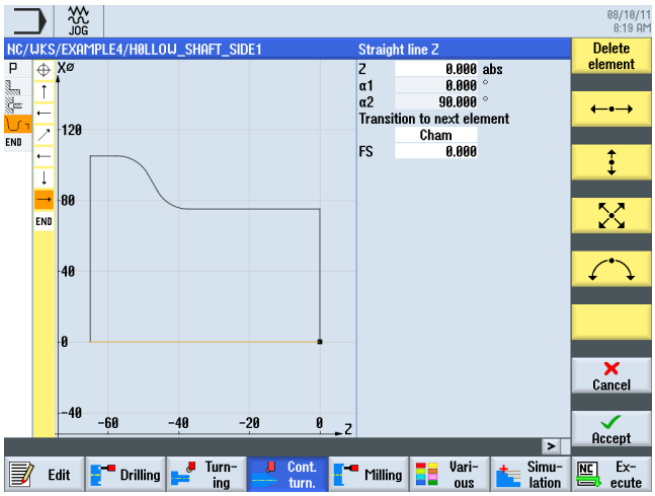
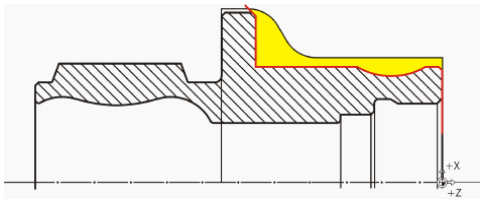


Figure 9-10 Creating a blank contour

9.2.4 Machined part contour of the first side, external

Operating sequences

Proceed as follows to enter the machined part contour:



Note

The (red) contour of the machined part is intended not to correspond to the drawing. The machined part contour serves as the roughing machining limit, but more importantly it specifies the precise traversing path for finishing. Thus, the construction 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 extending outside the blank. The large diameter is produced only in the second clamping.

Select the **Contour turning** softkey.

Select the **New contour** softkey. Enter the name 'HOLLOW_SHAFT_SIDE1_E' for the contour.

Figure 9-11 Creating the contour

Confirm your input.

Set the starting point to X32/Z0.

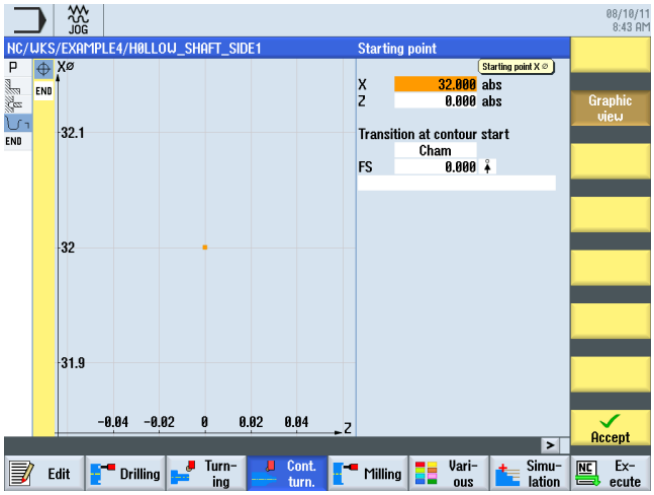


Figure 9-12 Specifying the contour starting point

Confirm your input.

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

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

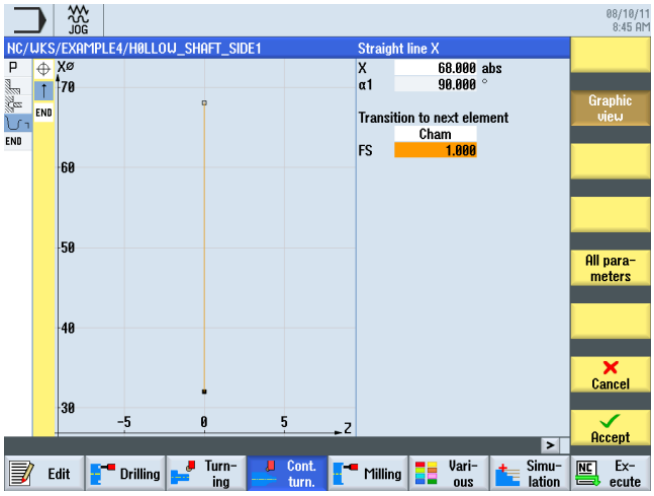


Figure 9-13 Specifying the vertical straight line contour

Accept the entered values.

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

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

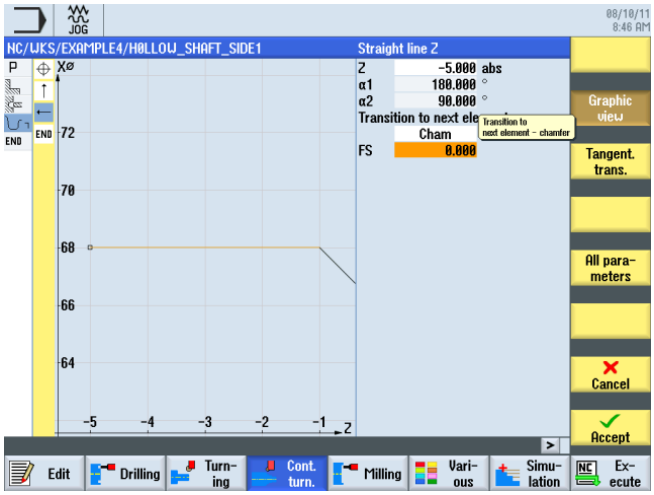


Figure 9-14 Specifying the horizontal straight line contour

Accept the entered values.

Enter the following values for the next section in the interactive screenform:

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	Chamfer	X	
FS	0		

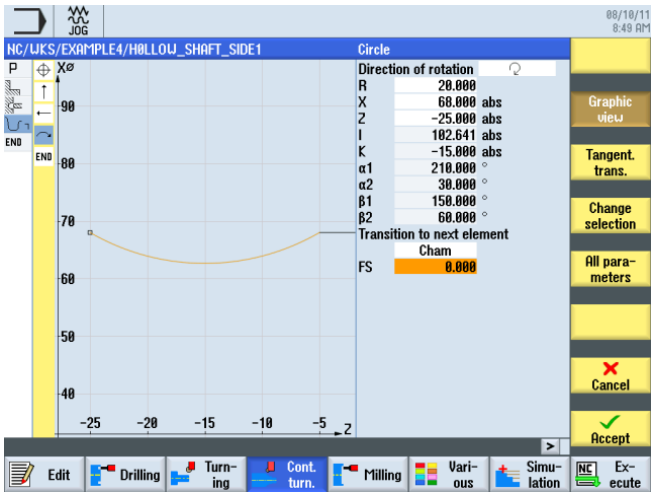


Figure 9-15 Specifying the arc contour

Select the desired construction.

Accept your selection.

Accept the contour section.

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

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

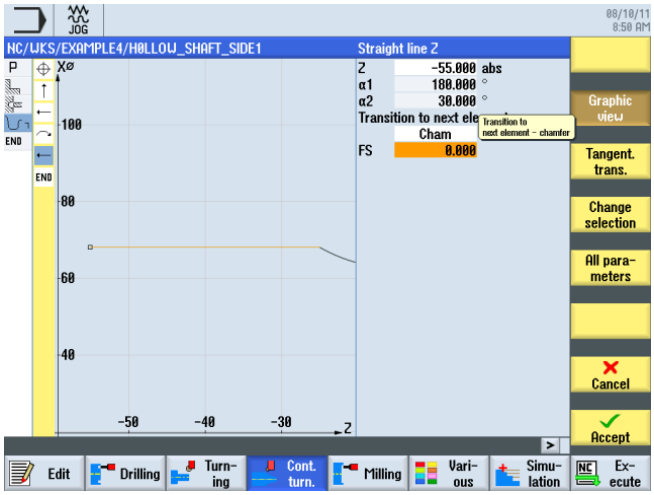


Figure 9-16 Specifying the horizontal straight line contour

Accept the entered values.

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

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	Chamfer	X	
FS	0		

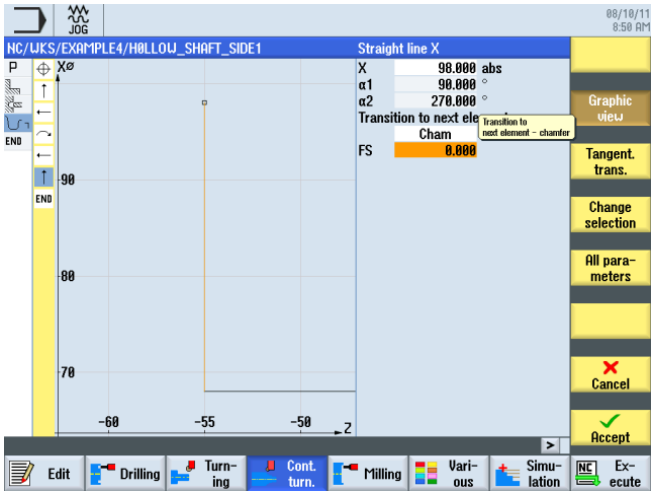


Figure 9-17 Specifying the vertical straight line contour

Accept the entered values.

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

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

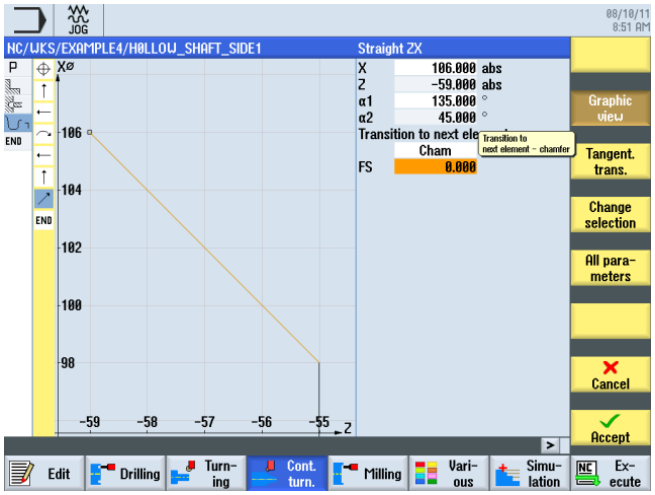


Figure 9-18 Specifying the inclined straight line contour

Accept the entered values.

Accept the contour into your process plan.

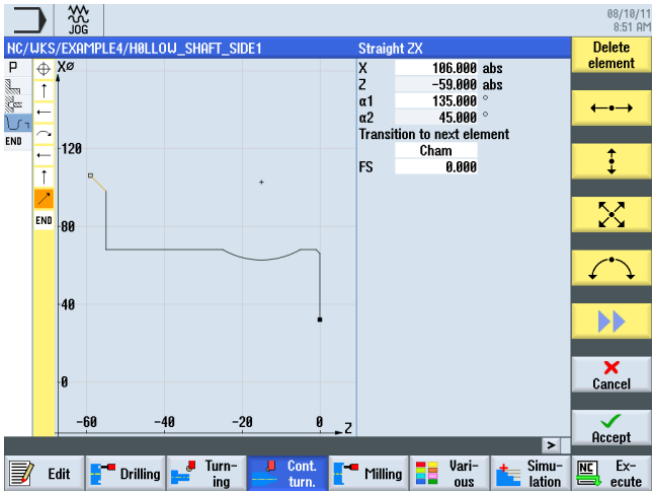


Figure 9-19 Contour in the contour calculator

After acceptance, the process plan looks like this. Both contours are automatically linked together.

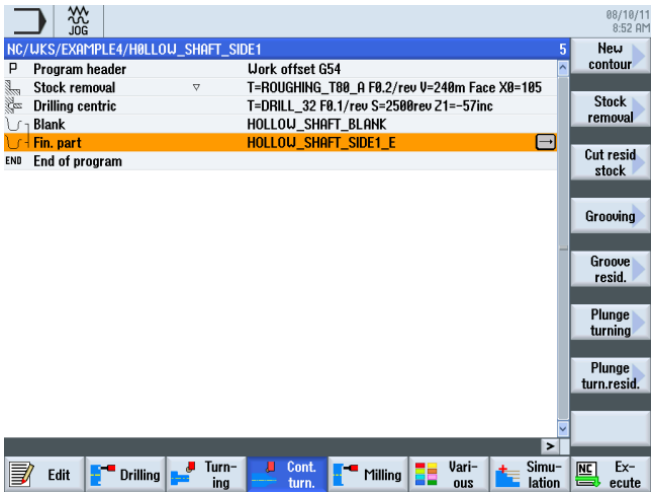


Figure 9-20 Process plan following input of the contours

Stock removal, residual stock removal and finishing

In the following work step you will perform contour stock removal.

To this end, proceed as follows:

Select the **Stock removal** softkey.

Open the tool list and select the ROUGHING_T80 A tool.

Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.3		
V	260 m/min	X	
Machining	longitudinal external roughing	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		

Field	Value	Selection via toggle key	Notes
BL	Contour	X	In the blank descriptions you can choose between the following settings: *Cylinder:* Blank = cylinder *Contour:* Blank = constructed contour *Allowance:* Blank = constructed contour with defined allowance
Relief cuts	No	X	Insertion with the roughing tool is not recommended. Therefore, switch the Relief cuts field to *No*.
Set machining area limits	No	X	

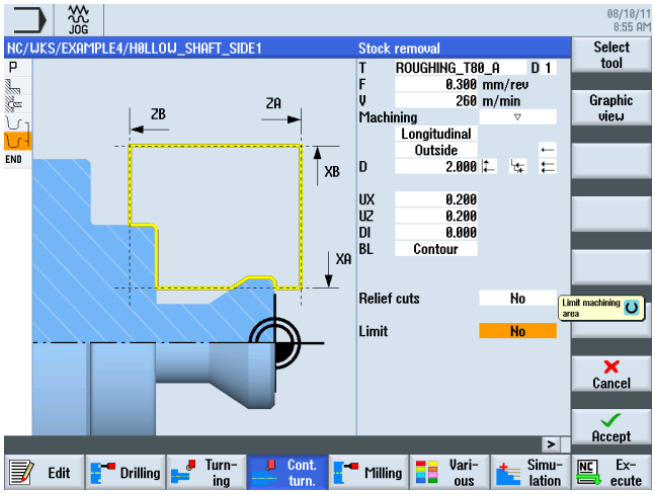


Figure 9-21 Roughing the contour

Accept	Accept the entered values.
Cut resid stock	Select the **Residual stock removal** softkey.
Select tool	Open the tool list and select the FINISHING_T35 A tool.
To program	Accept the tool into your program. Before finishing, the residual material is cut in the concave fillet.

Enter the following values for stock removal of residual material in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.2		
V	240 m/min	X	
Machining	longitudinal external roughing	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
Relief cuts	Yes	X	To ensure that the concave fillet is taken into consideration, the Relief cuts field must be switched to *Yes*.
FR	0.2		
Set machining area limits	No	X	

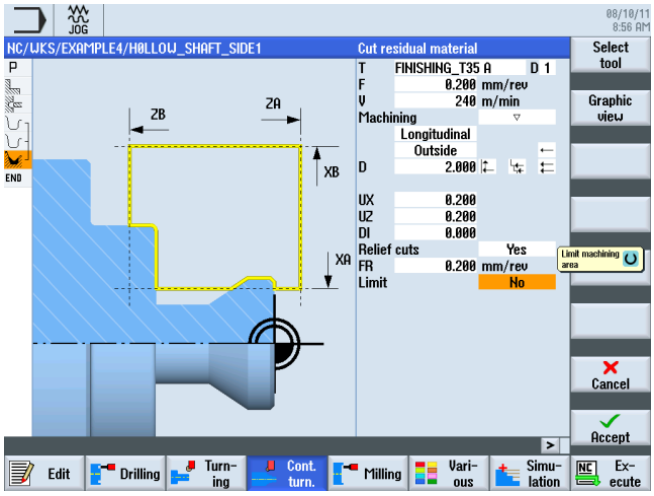


Figure 9-22 Stock removal of residual material contour

Accept the entered values.

Select the **Stock removal** softkey.

Open the tool list and select the FINISHING_T35 A tool.

To program

Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	280 m/min	X	
Machining	longitudinal external finishing	X X X	
Allowance	No	X	
Relief cuts	Yes	X	Switch Relief cuts to *Yes* here, too.
Set machining area limits	No	X	

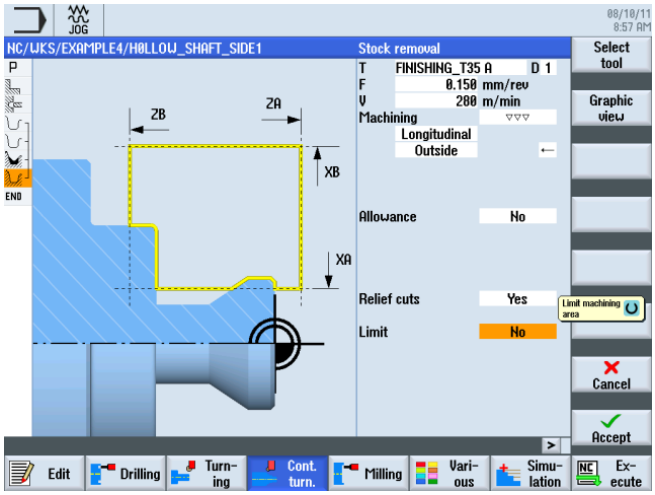


Figure 9-23 Finishing the contour

Accept the entered values. After acceptance, your work step program looks like this: The contours are automatically linked to the stock removal work steps.

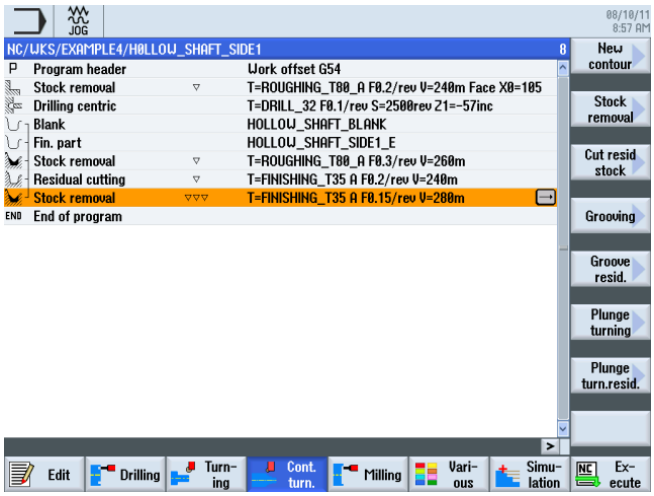
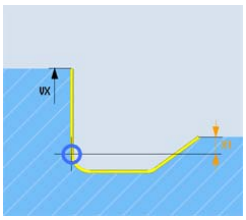
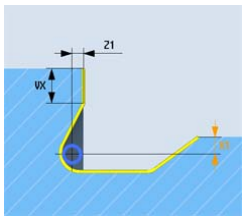
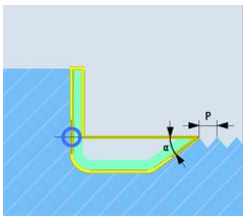
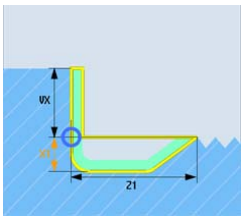


Figure 9-24 Process plan following stock removal of the contour

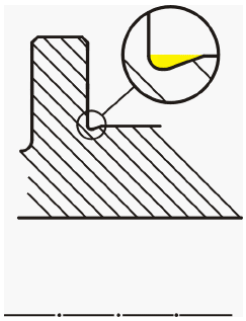
9.2.5 Undercut

You can select from four different types of undercut:

Undercut form E	Undercut form F	Undercut DIN thread	Undercut thread
			

Operating sequences

Proceed as follows to create an undercut.



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

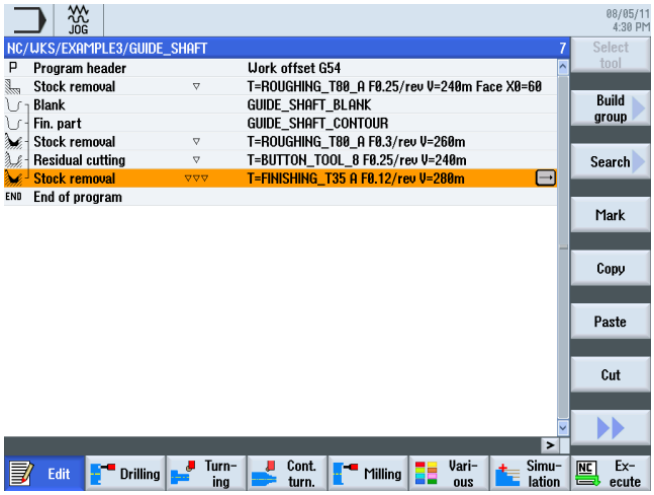


Figure 9-25 Process plan following stock removal

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.

Accept the tool into your program.

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

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

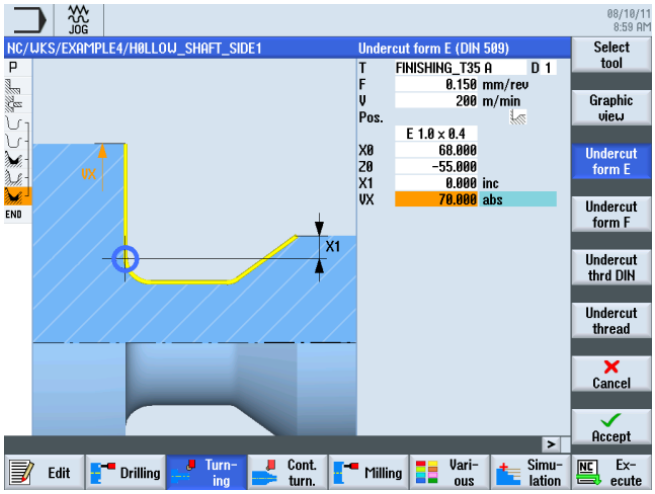


Figure 9-26 Specifying the undercut

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

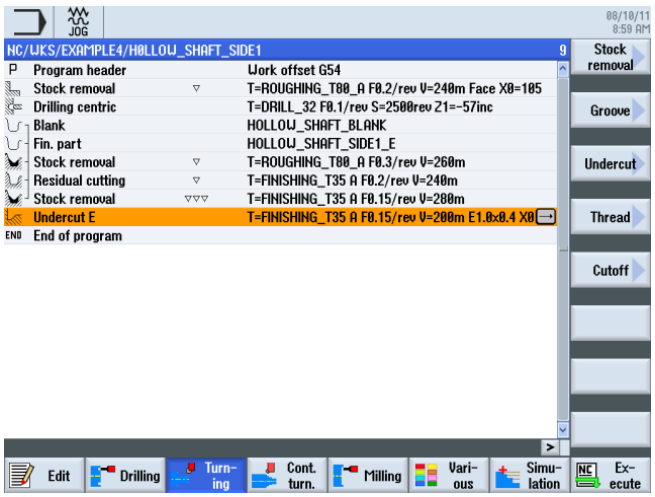


Figure 9-27 Process plan with undercut

Start the simulation.

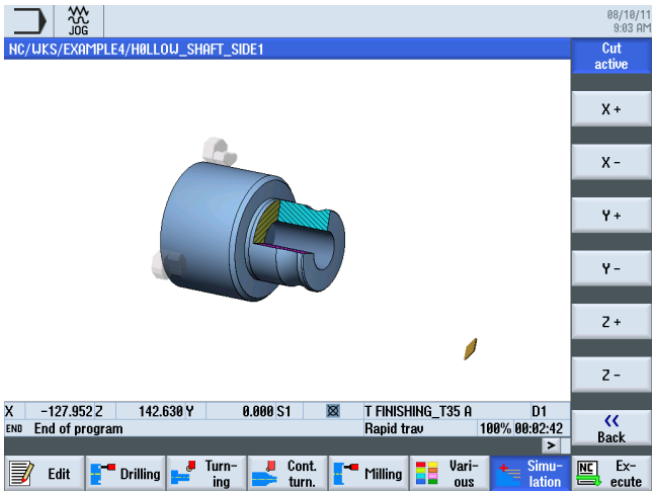


Figure 9-28 Simulation - Cut active

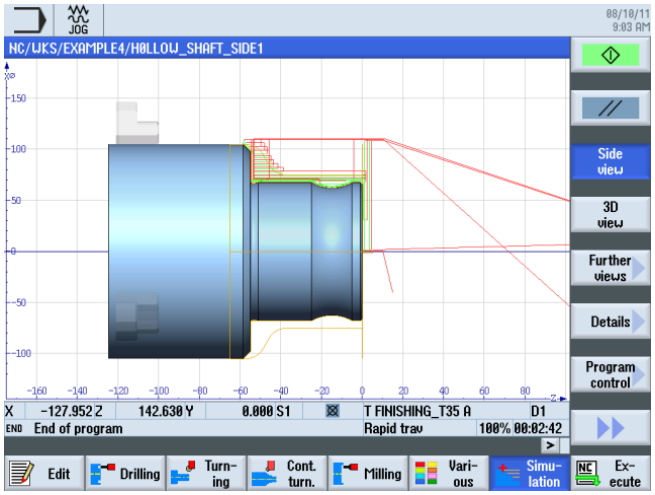
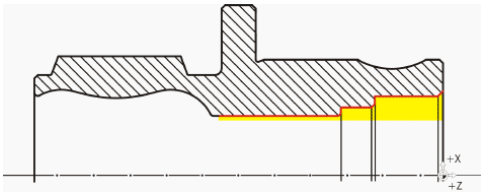


Figure 9-29 Simulation - Side view with display of the traversing paths

9.2.6 Machined part contour of the first side, internal

Operating sequences

Proceed as follows to enter the machined part contour:



Select the **Contour turning** softkey.

Select the **New contour** softkey. Enter the name 'HOLLOW_SHAFT_SIDE1_I' for the contour.

Figure 9-30 Creating the contour

Confirm your input.

Set the starting point to X50/Z0.

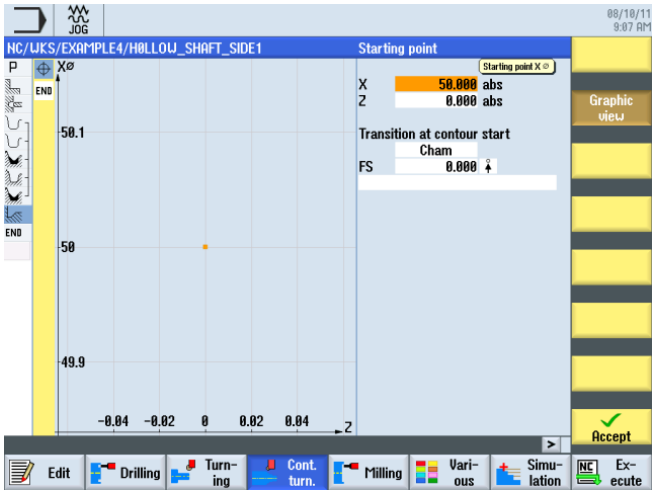


Figure 9-31 Specifying the contour starting point

Confirm your input.

Create the contour without help (see illustration below).

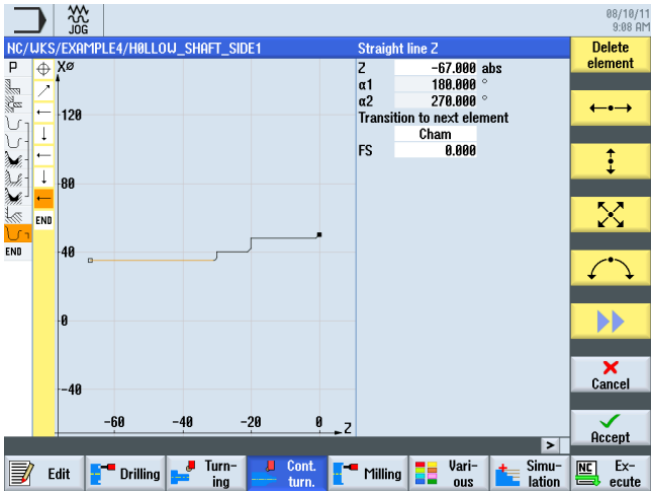


Figure 9-32 Machined part contour of the first side, internal

Stock removal, residual stock removal and finishing

In the following work step you will perform contour stock removal. Enter the geometries into your process plan as follows.

Figure 9-33 Broken-line graphics

Select the **Stock removal** softkey.

Open the tool list and select the ROUGHING_T80 I tool.

Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.25		
V	250 m/min	X	
Machining	longitudinal internal roughing	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	Since drilling has already taken place, you do not need to take a blank contour into consideration for internal machining. Switch to *Cylinder*.
XD	32 abs	X	
ZD	0 inc	X	
Relief cuts	No	X	
Set machining area limits	No	X	

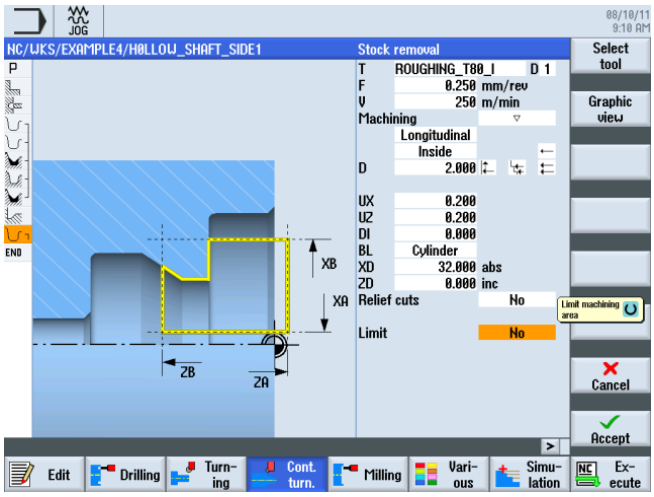


Figure 9-34 Roughing the contour

Accept the entered values.

Select the **Stock removal** softkey.

Select tool

Open the tool list and select the FINISHING_T35 I tool.

To program

Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.12		
V	280 m/min	X	
Machining	longitudinal internal finishing	X X X	
Allowance	No	X	
Relief cuts	No	X	
Set machining area limits	No	X	

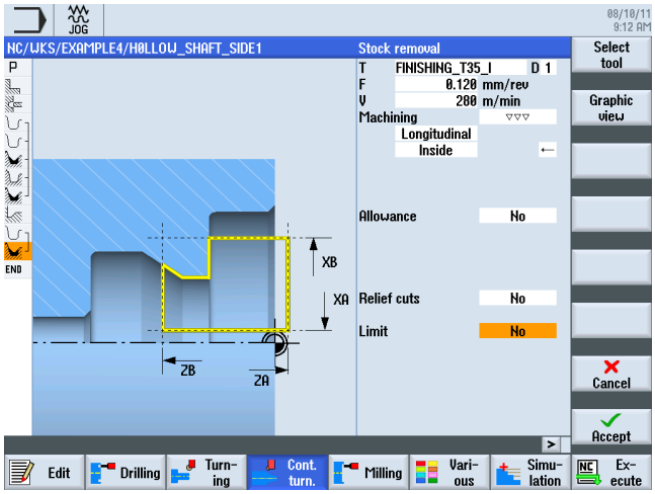


Figure 9-35 Finishing the contour

Accept

Accept the entered values.

Start the simulation to perform checks.

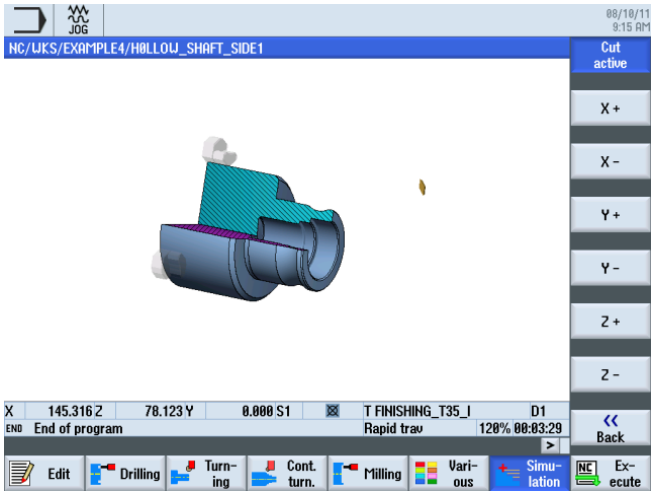


Figure 9-36 Simulation - Cut active

Undercut

Proceed as follows to create an undercut:

Select the **Undercut** softkey.

Select the **Undercut form E** softkey.

Create the undercut (see illustration below).

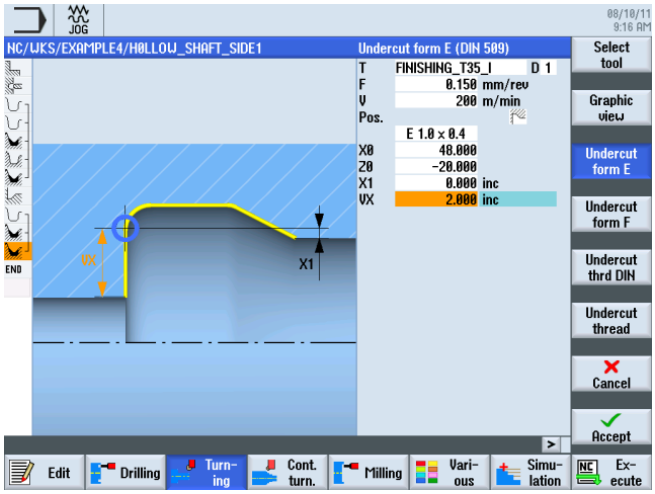


Figure 9-37 Creating an undercut

Note

Make sure that the undercut is in the correct position.

Start the simulation.

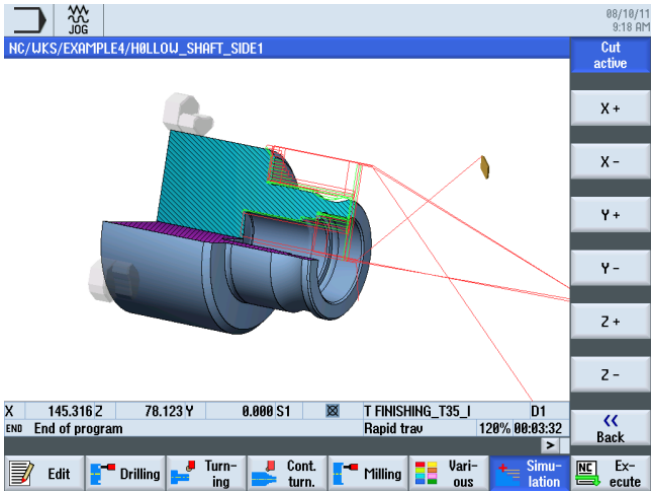


Figure 9-38 Undercut simulation (with display of the traversing paths)

The process plan for the first side of the workpiece looks like this.

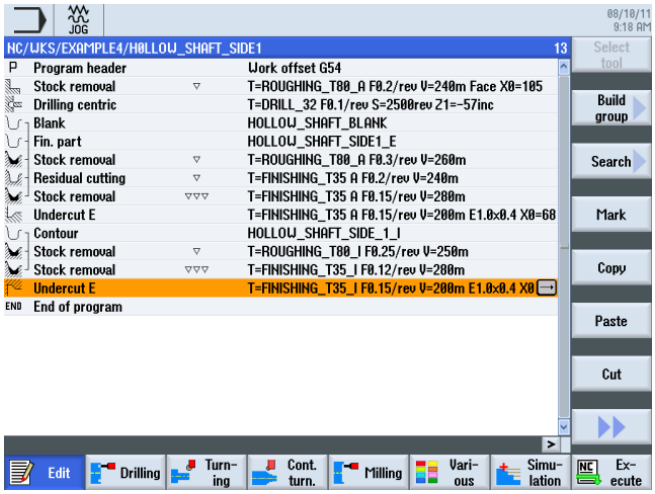


Figure 9-39 Process plan with undercut

9.2.7 The work step editor

Functions of the work step editor

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

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

Use this softkey to search for texts in the program.

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

Use this softkey to copy work steps to the clipboard.

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

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

Use this softkey to switch to the extended menu.

Use this softkey to renumber the work steps.

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

Use this softkey to return to the previous menu.

You will need some of these functions in order to reuse the blank contour of the first side in the process plan for the second side of the workpiece. The blank contour will be copied to the clipboard and pasted into the process plan for the second side.

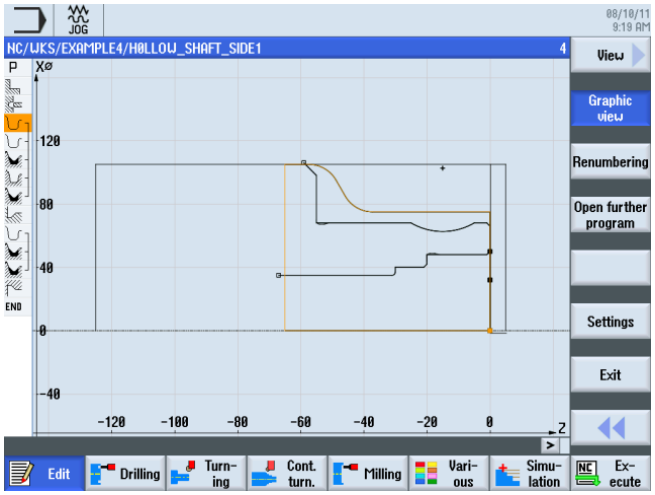


Figure 9-40 Blank contour

9.2.8 Copying the contour

Operating sequence

Proceed as follows to copy the blank contour to the clipboard:

Navigate to the 'HOLLOW_SHAFT_BLANK' contour.

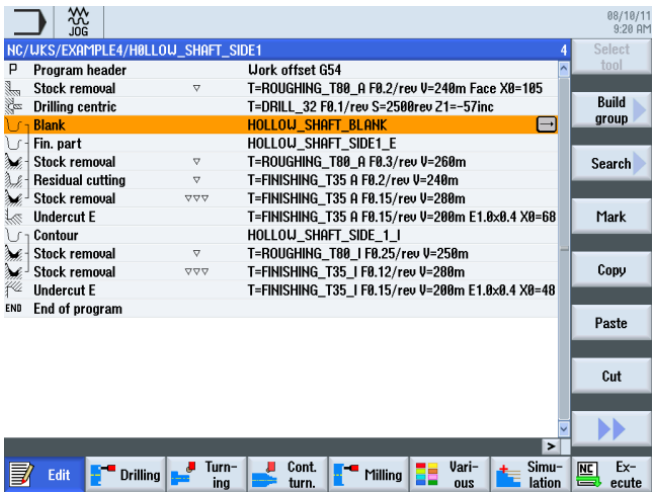


Figure 9-41 Copying the contour to the clipboard.

Copy the blank contour to the clipboard. The contour remains on the clipboard until you copy another work step to the clipboard or shut down the control system.

9.3 Creating the second workpiece side

Creating a process plan

Proceed as follows to create a process plan for the second side of the workpiece.

Operating sequences

Create the program 'HOLLOW_SHAFT_SIDE2' without help.

Figure 9-42 Creating a ShopTurn program

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

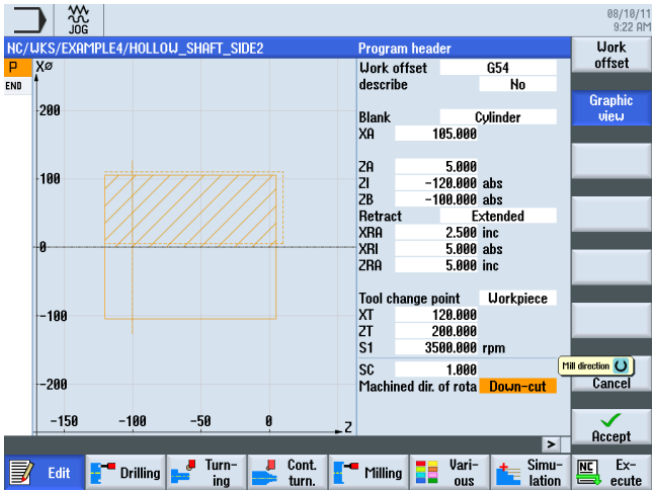


Figure 9-43 Workpiece dimensions in the program header

9.3.1 Face turning

Operating sequences

Proceed as follows to face turn the blank to X-1.6 and Z0:

Select the **Turning** softkey.

Select the **Stock removal** softkey.

Select the ROUGHING_T80 A tool.

Enter the following values in the interactive screenform:

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

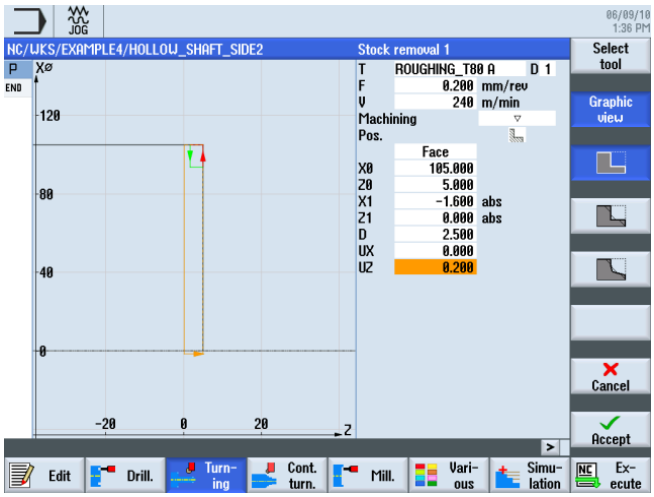


Figure 9-44 Face turning the workpiece

Accept the entered values. After acceptance, your work step program looks like this:

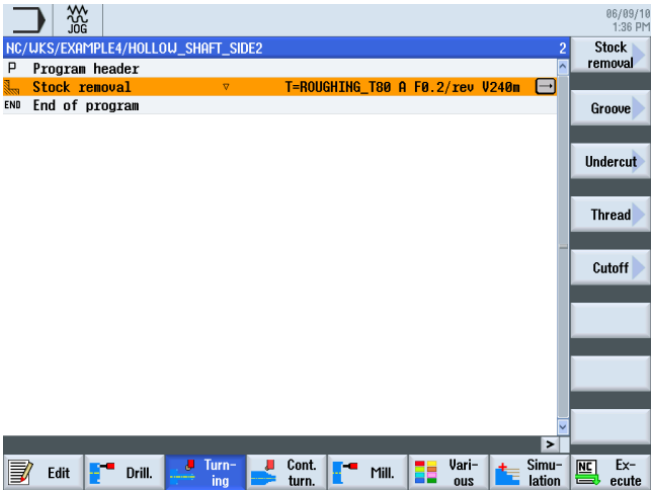


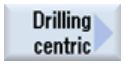
Figure 9-45 Process plan following face turning

9.3.2 Drilling

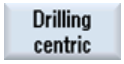
Operating sequences

Proceed as follows to drill the workpiece at the center.

Select the **Drilling** softkey.



Select the **Drilling Centric** softkey.



Select the **Drilling Centric** softkey.

Open the tool list and select DRILL_32.

Accept the tool into your program.

Enter the following values for the drill hole in the interactive screenform:

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	

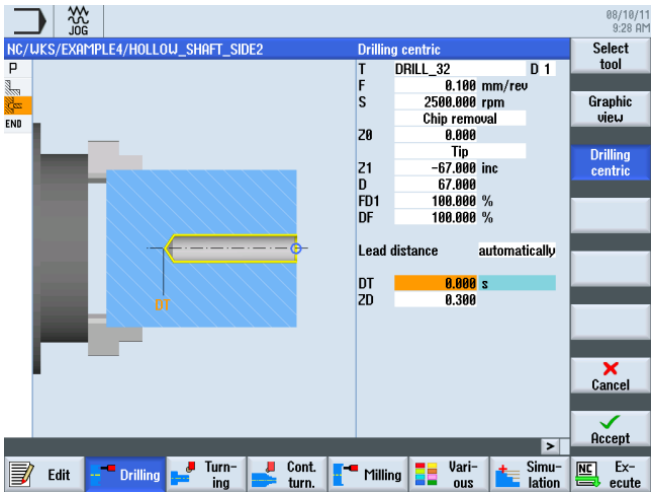


Figure 9-46 Hole

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

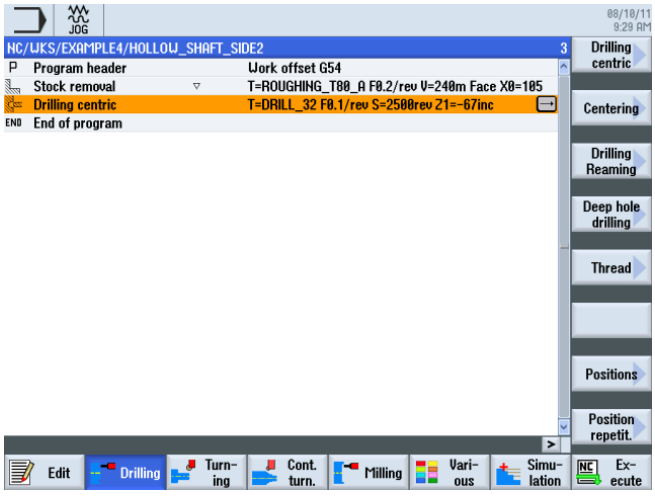


Figure 9-47 Process plan following input of the drilling position

9.3.3 Specifying a blank contour

Operating sequences

Proceed as follows to paste the blank contour from the clipboard into your process plan:

First, navigate in the process plan to the most recently entered work step (see illustration).

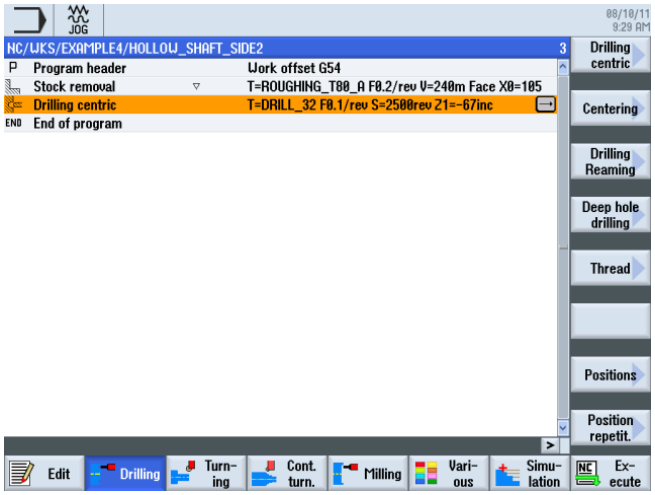


Figure 9-48 Position for inserting the blank contour

Paste

Paste the blank contour from the clipboard. After pasting the contour, your process plan should look like this.

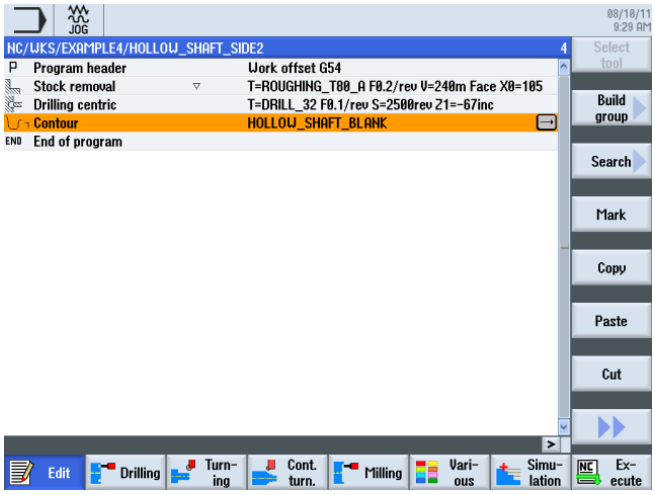
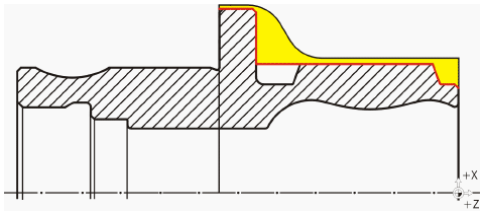


Figure 9-49 Pasting the contour

9.3.4 Machined part contour of the second side, external

Operating sequences

Proceed as follows to enter the machined part contour:



Note

The asymmetrical groove is produced later.

Select the **Contour turning** softkey.

Select the **New contour** softkey. Enter the name 'HOLLOW_SHAFT_SIDE2_E' for the contour.

"Apply" your input.

Set the starting point to X57/Z0.

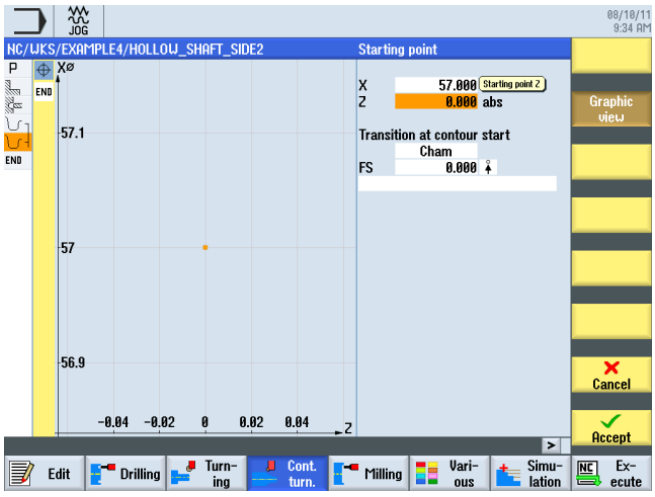


Figure 9-50 Specifying the contour starting point

"Apply" your input.

Create the contour up to the end point to Z-65 and X100 without help (see illustration below).

Figure 9-51 Contour in the contour calculator

Accept

Accept the contour into your process plan.

Stock removal and finishing

In the following work step you will perform contour stock removal.

To this end, proceed as follows:

Stock removal

Select the **Stock removal** softkey.

Select tool

Open the tool list and select the ROUGHING_T80 A tool.

To program

Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.3		
V	260 m/min	X	
Machining	longitudinal external roughing	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Contour	X	
Relief cuts	No	X	
Set machining area limits	No	X	

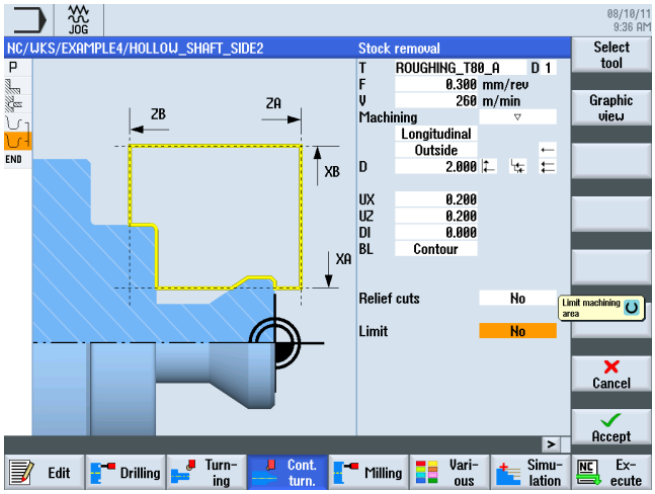


Figure 9-52 Roughing the contour

Accept

"Apply" the set values. After acceptance, your work step program looks like this:

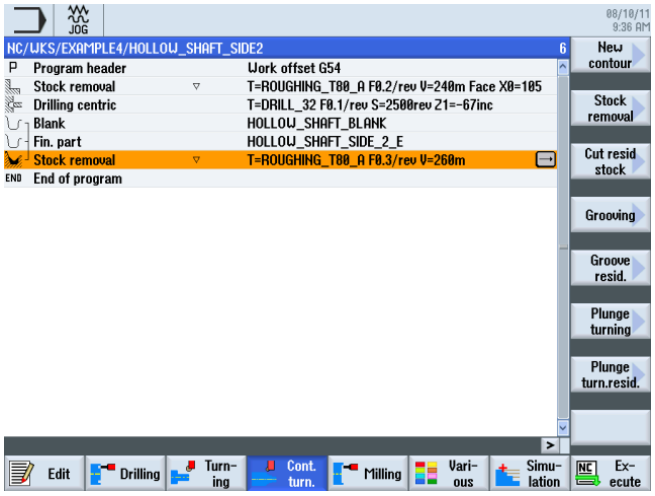


Figure 9-53 Process plan following roughing

Stock removal

Select the **Stock removal** softkey.

Select tool

Open the tool list and select the FINISHING_T35 A tool.

To program

Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	200 m/min	X	
Machining	longitudinal external finishing	X X X	
Allowance	No	X	
Relief cuts	No	X	
Set machining area limits	No	X	

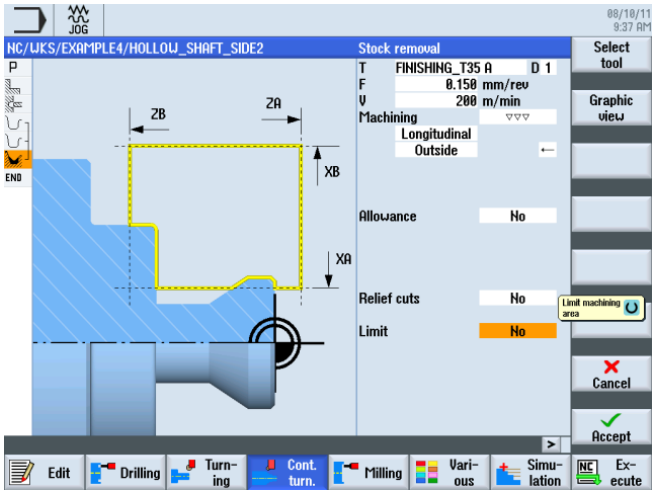


Figure 9-54 Finishing the contour

"Apply" the set values. After acceptance, your work step program looks like this:

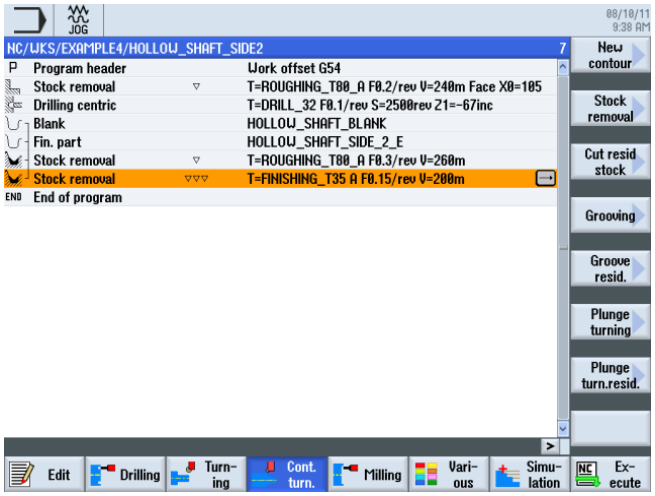


Figure 9-55 Process plan following stock removal of the contour

Start the simulation.

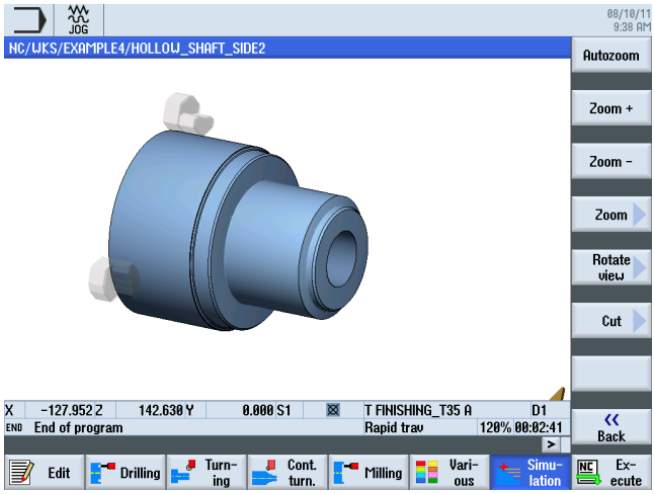
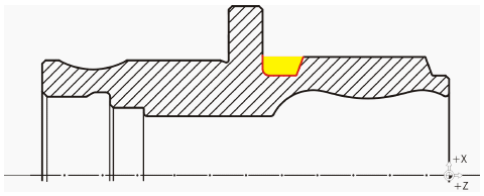


Figure 9-56 Simulation - 3D view

9.3.5 Creating an asymmetrical groove

Operating sequences

Proceed as follows to create an 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.

Accept the tool into your program.

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

Field	Value	Selection via toggle key	Notes
F	0.08		
V	180 m/min	X	
Machining	Roughing + finishing	X	
Position	see illustration below	X	
X0	70		
Z0	-55		
B1	10	X (field)	
T1	5.5 inc	X	
α1	0		
α1	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		

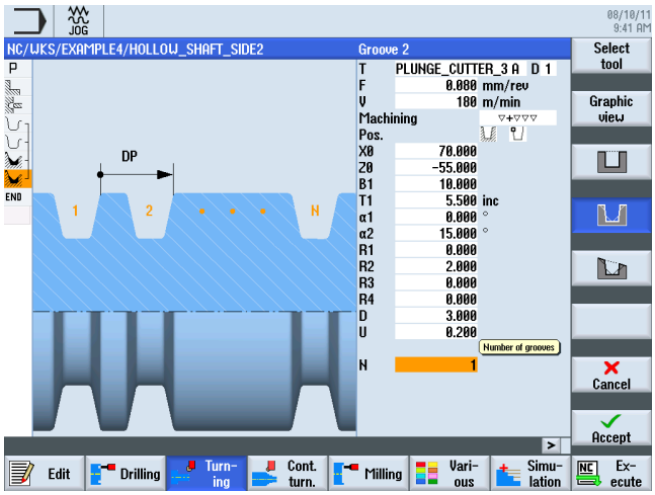


Figure 9-57 Specifying a groove

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

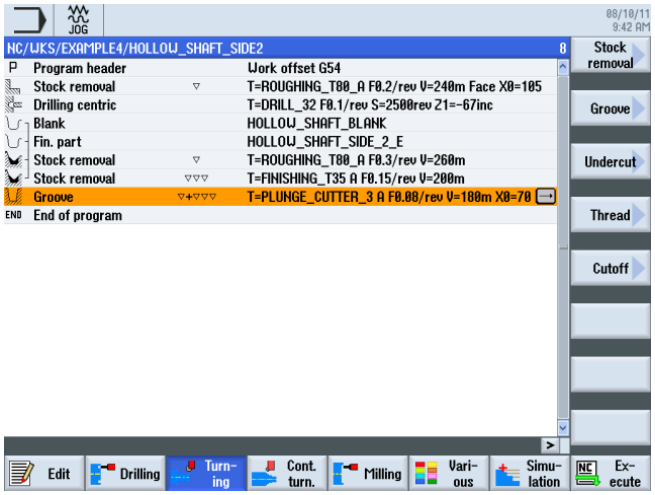


Figure 9-58 Process plan following groove

Start the simulation.

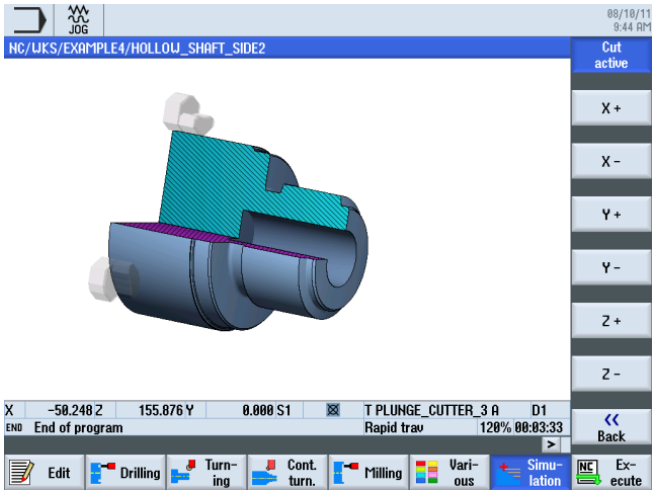
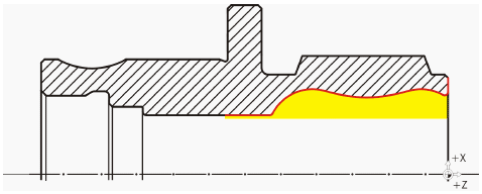


Figure 9-59 Simulation - 3D view (cut active)

9.3.6 Machined part contour of the second side, internal

Operating sequences

Proceed as follows to enter the machined part contour:



Select the **Contour turning** softkey.

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

"Apply" your input.

Set the starting point to X57/Z0.

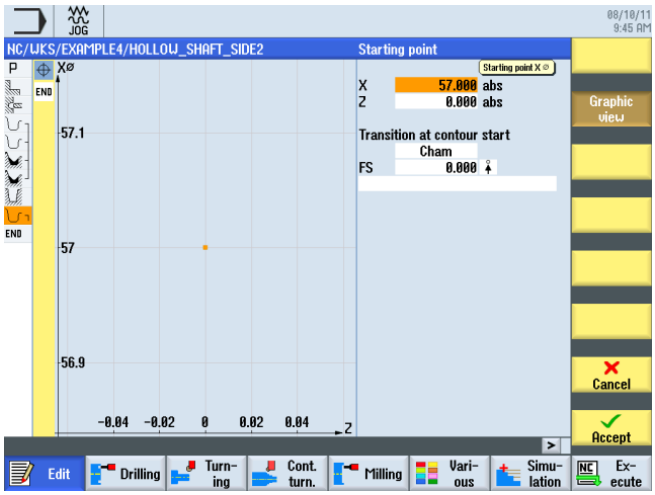


Figure 9-60 Specifying the contour starting point

"Apply" your input.

Create the contour without help (see illustration below).

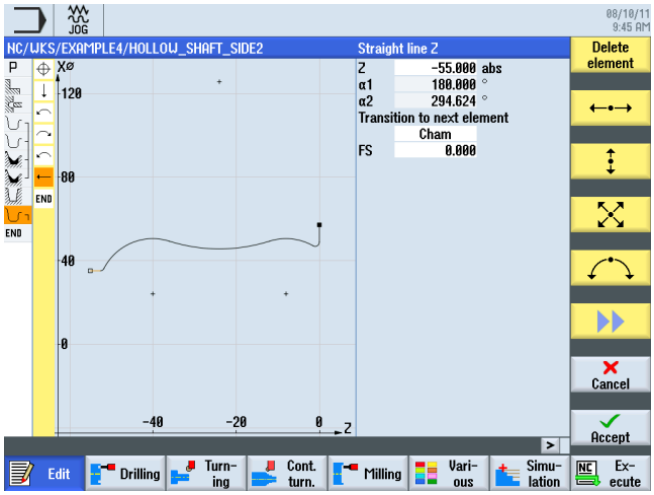


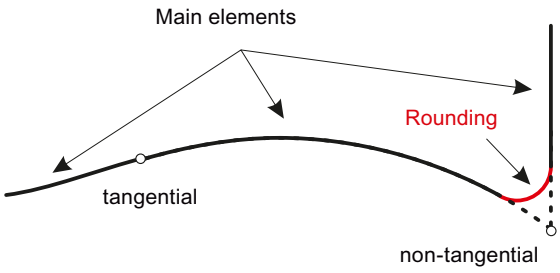
Figure 9-61 Machined part contour of the second side, internal

Note

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

Tangential merging applies only to main elements, i.e. the rounding is attached to the main element.

(See illustration below)



Accept

Accept the contour. After accepting the contour, your work step program looks like this.

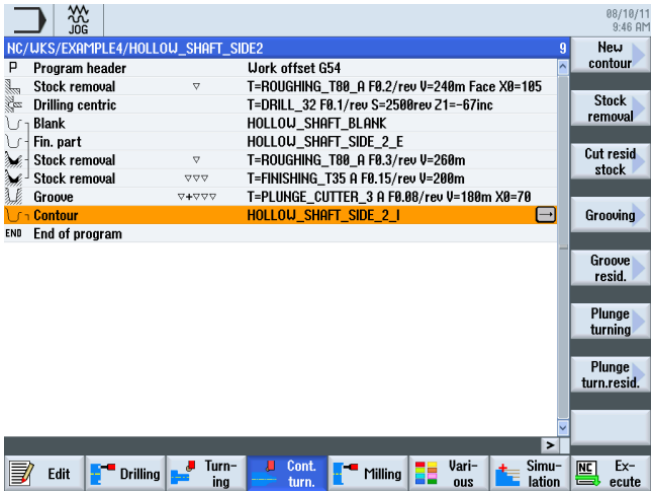


Figure 9-62 Process plan following input of the contour

Stock removal, residual stock removal and finishing

In the following work step you will perform contour stock removal.

Stock removal

Select the **Stock removal** softkey.

Select tool

Open the tool list and select the ROUGHING_T80 I tool.

To program

Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.25		
V	280 m/min	X	
Machining	longitudinal internal roughing	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	Since drilling has already taken place, you do not need to take a blank contour into consideration for internal machining. Switch to *Cylinder*.

Field	Value	Selection via toggle key	Notes
XD	32 abs	X	
ZD	0 abs	X	
Relief cuts	No	X	
Set machining area limits	No	X	

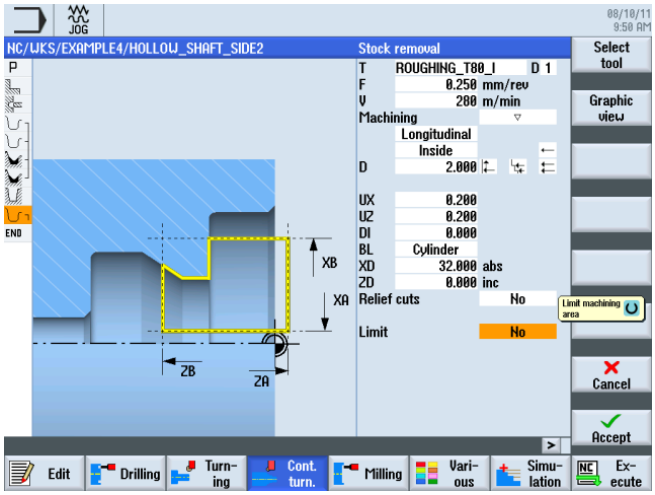


Figure 9-63 Roughing the contour

"Apply" the set values.

Select the **Residual stock removal** softkey.

Open the tool list and select the FINISHING_T35 I tool.

Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.2		
V	240 m/min	X	
Machining	longitudinal internal finishing	X X X	
Allowance	No	X	
Relief cuts	Yes	X	
FR	0.2		
Set machining area limits	No	X	

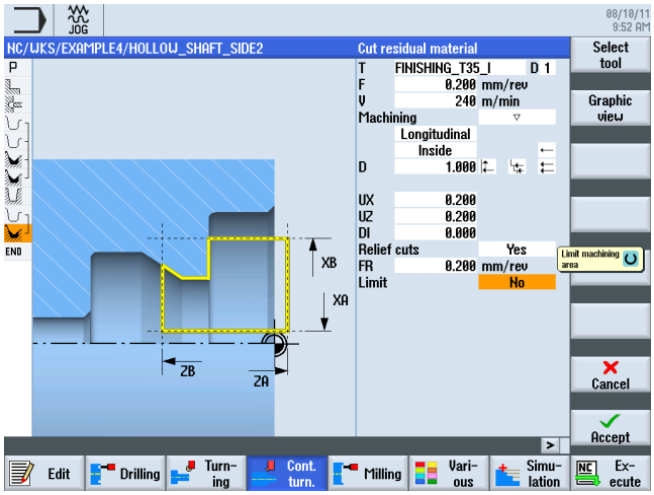


Figure 9-64 Contour residual stock removal

"Apply" the set values.

Select the **Stock removal** softkey.

Open the tool list and select the FINISHING_T35 I tool.

Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.12		
V	280 m/min	X	
Machining	longitudinal internal finishing	X X X	
Allowance	No	X	
Relief cuts	Yes	X	
Set machining area limits	No	X	

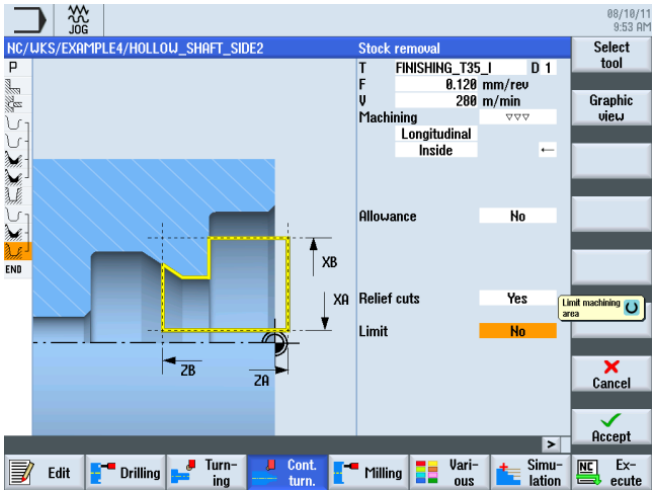


Figure 9-65 Finishing the contour

"Apply" the set values.

Start the simulation to perform checks.

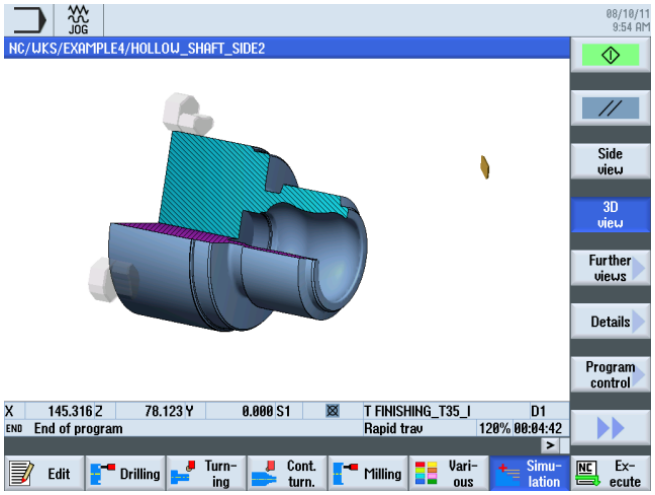


Figure 9-66 Simulation - 3D view (cut active)

Example 5: Plunge-turning

10

10.1 Overview

Learning objectives

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

Task

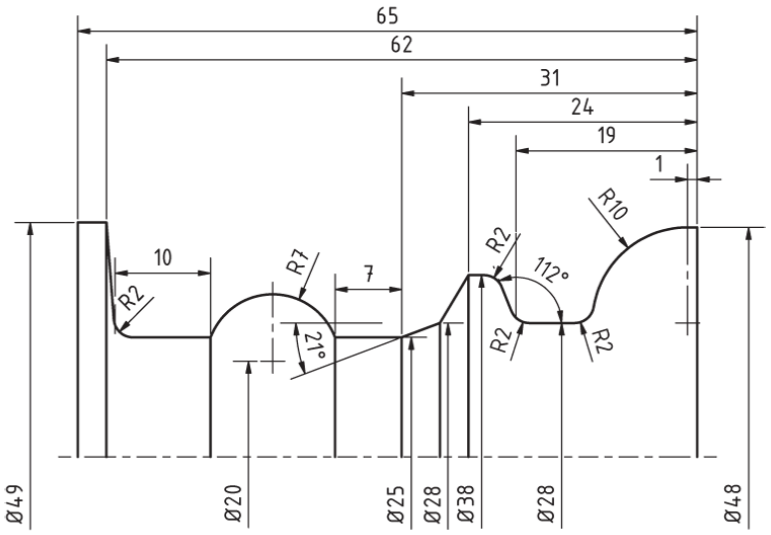


Figure 10-1 Workshop drawing - Example 5:

Preparation

Perform the following steps without help:

1. Create a new workpiece with the name 'EXAMPLE5'.
2. Create a new step sequence program with the name 'PLUNGE_TURNING'.
3. Fill in the program header (see illustration below).

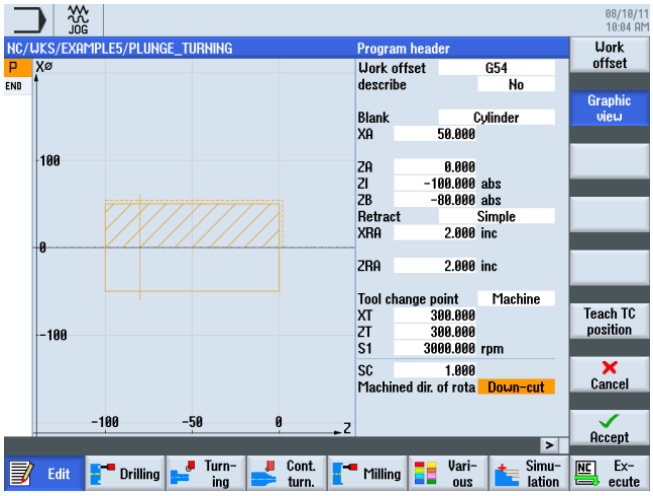


Figure 10-2 Creating a program header

10.2 Plunge-turning

The achievable level of productivity during turning is limited, among other things, by the possible number of tools in the revolver and the frequent tool changes required for effective turning machining. With standard turning tools alone, not all possible contours can be produced, and residual material machining, therefore, is often performed by means of plunge-cutting. For the complete machining of a contour, therefore, it is always necessary to change between standard turning tools and plunge-cutting 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 movement of the turning tool, for example.

As a general rule, hardly any empty cuts exist during the plunge-turning cycle, as stock removal is performed 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 turning part and during the stock removal cycle you can select whether you wish to perform stock removal using a conventional procedure or by means of plunge-cutting 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 clearly analyze the calculated traversing movements of the tool. Even a combination of conventional turning machining and plunge-turning is possible, i.e. a standard tool is used for roughing, while plunge-turning is used for machining residual material, meaning that the contour can be thoroughly machined without the risk of damage.

10.3 Creating the contour

Operating sequences

Create the contour without help.

Select the **Contour turning** softkey.

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

Confirm your input.

Set the starting point to X48/Z0.

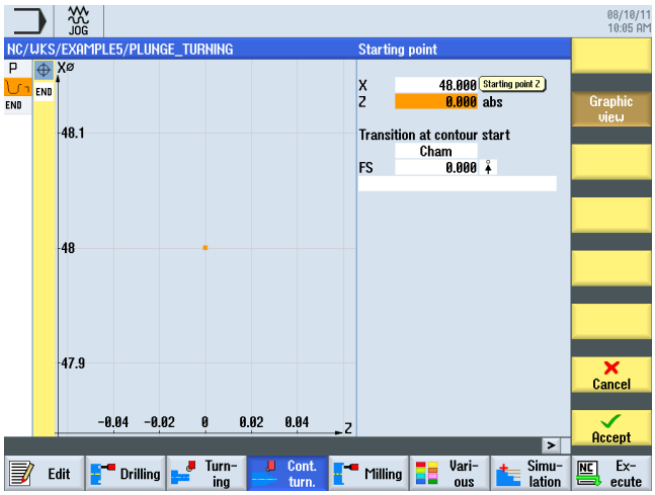


Figure 10-3 Specifying the contour starting point

Create a contour (see illustration below).

Figure 10-4 Contour in the contour calculator

10.4 Stock removal with the plunge-turning cycle

Operating sequences

In the following work step you will perform contour stock removal.

To this end, proceed as follows:

Select the **Contour turning** softkey.

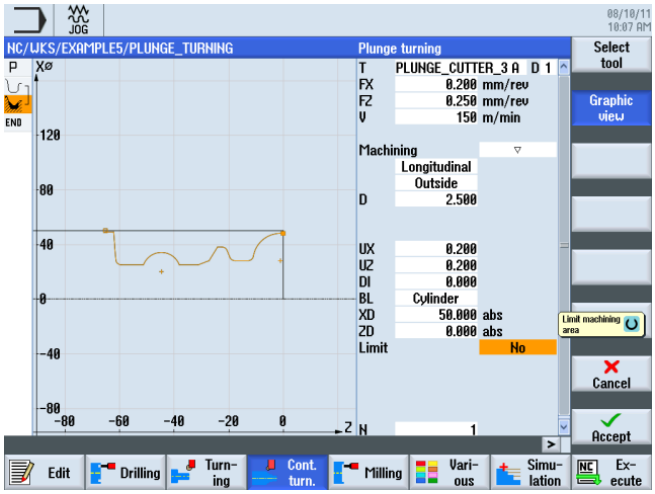
Select the **Plunge-turning** softkey.

Open the tool list and select the PLUNGE_CUTTER_3 A tool.

Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
FX	0.2		
FZ	0.25		
V	150 m/min	X	
Machining	longitudinal external roughing	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	
Set machining area limits	No	X	
N	1		



Roughing the contour

Accept the entered values.

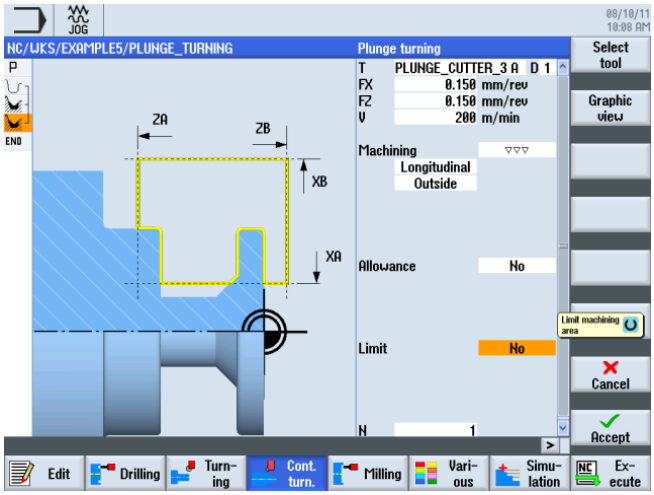
Select the **Plunge-turning** softkey.

Open the tool list and select the PLUNGE_CUTTER_3 A tool.

Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
FX	0.15		
FZ	0.15		
V	200 m/min	X	
Machining	longitudinal external finishing	X X X	
Allowance	No	X	
Set machining area limits	No	X	
N	1		



Finishing the contour

Accept the entered values. After acceptance, your work step program looks like this:

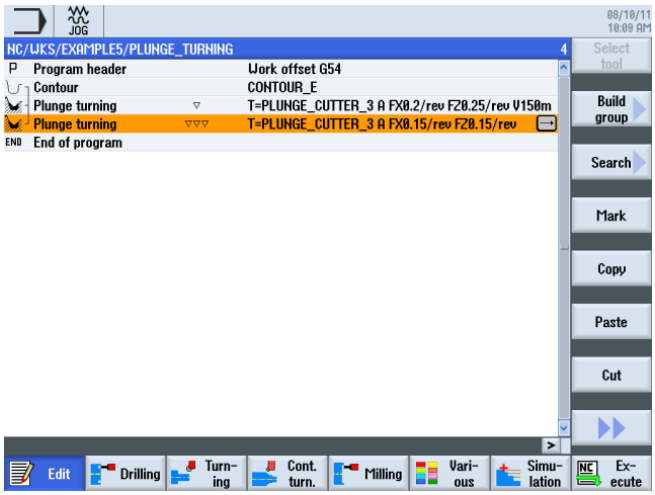


Figure 10-5 Work step program

Select the **Simulation** softkey.

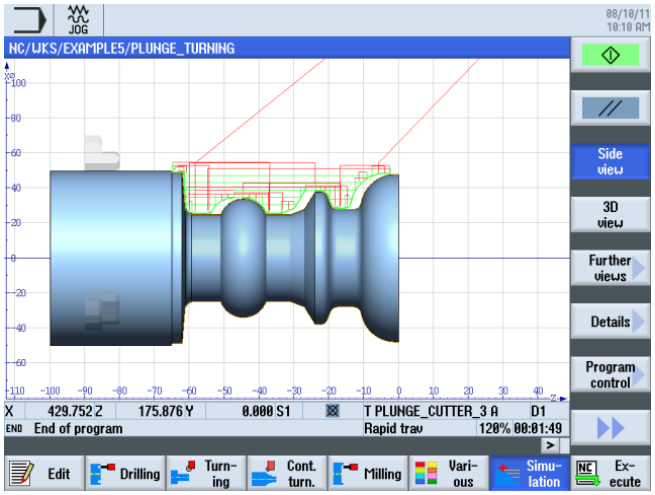


Figure 10-6 Simulation - Side view (with display of the traversing paths)

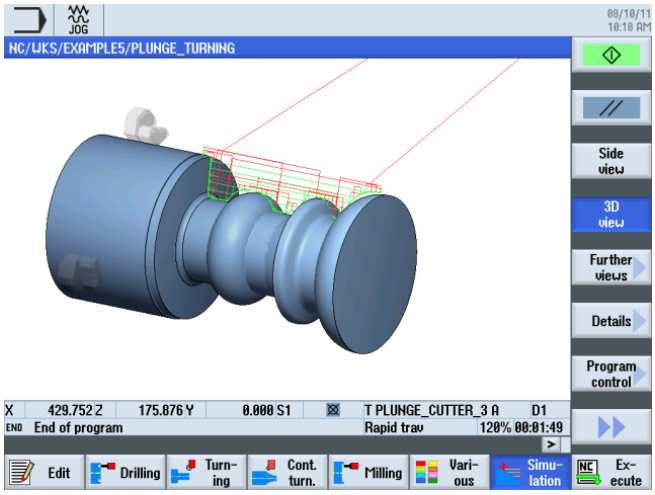


Figure 10-7 Simulation - 3D view (with display of the traversing paths)

And now you can start manufacturing! 11

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

To machine a workpiece, proceed as follows:

Approaching reference point

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

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

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

Clamping the workpiece

To guarantee machining in accordance with the specified dimensions and, naturally, also for your own safety, it is imperative to clamp the workpiece tightly. A three-block 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 through sampling with a calculated tool in the Z axis.

Executing the process plan

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

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

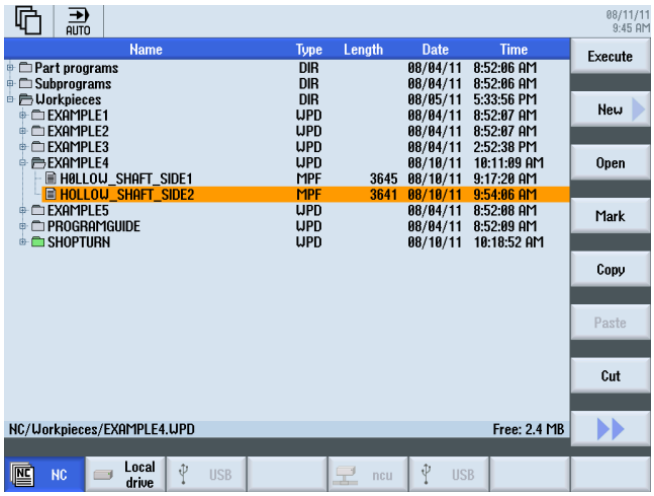


Figure 11-1 Selecting the program

Open the program.

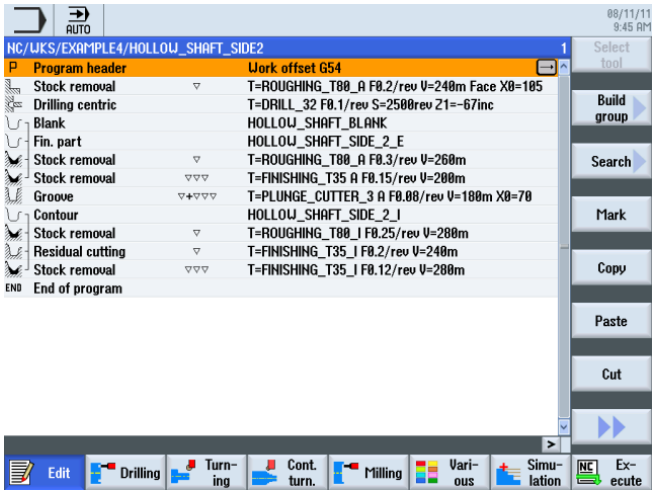
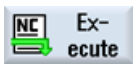


Figure 11-2 Opening the process plan



Select the **NC selection** softkey.

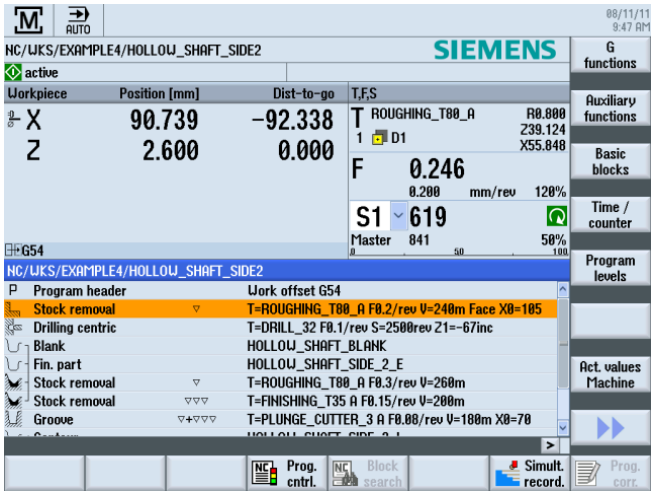
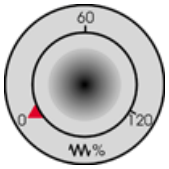


Figure 11-3 Executing



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

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

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

How accomplished are you in the use of ShopTurn? 12

12.1 Exercise 1

Will you manage this task using ShopTurn within 10 minutes?

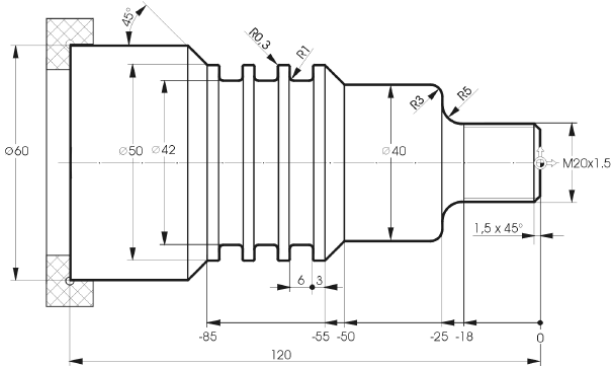


Figure 12-1 Workshop drawing DIYS1

Notes

In the process plan, the workpiece is milled to size in two work steps (see model below). For this reason, you can assign the the starting point of the CONTOUR_1 contour to the start of the first chamfer.

Model

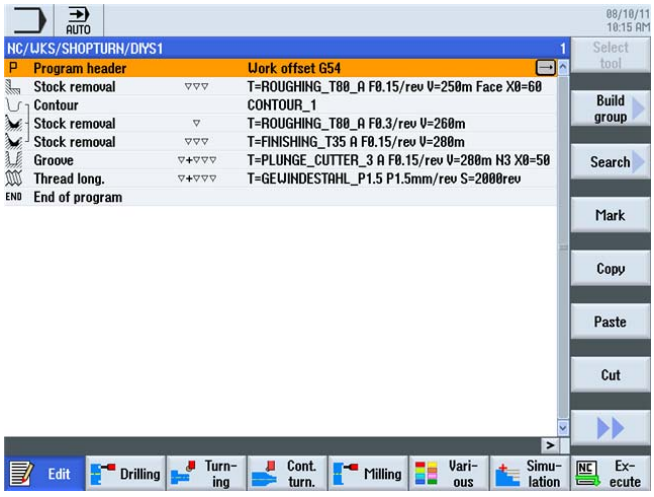


Figure 12-2 Process plan

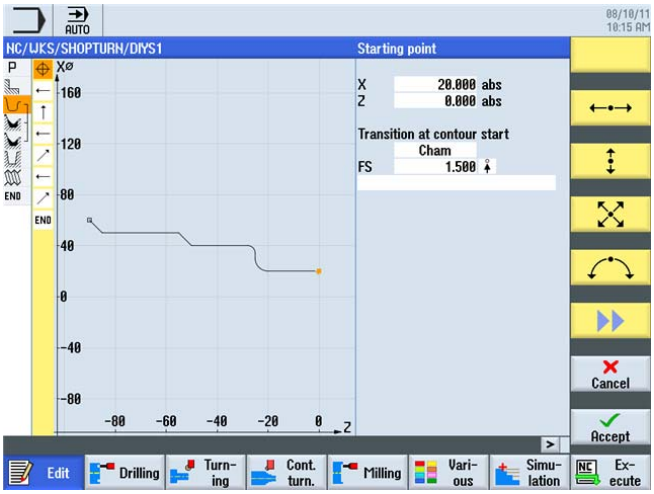


Figure 12-3 Contour in the contour calculator

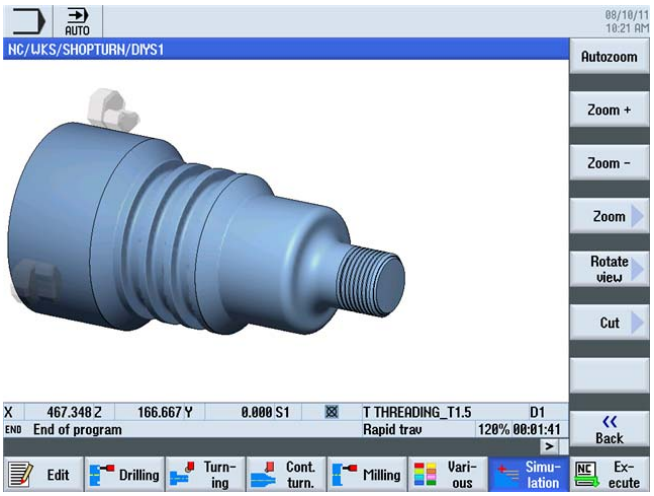


Figure 12-4 Workpiece simulation

12.2 Exercise 2

Will you manage this task using ShopTurn within 10 minutes?

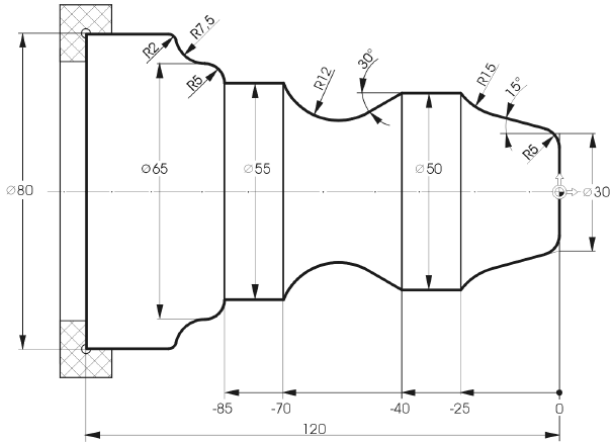


Figure 12-5 Workshop drawing DIYS2

Notes

You can use the automatic cutting of residual material to great effect here.

Model

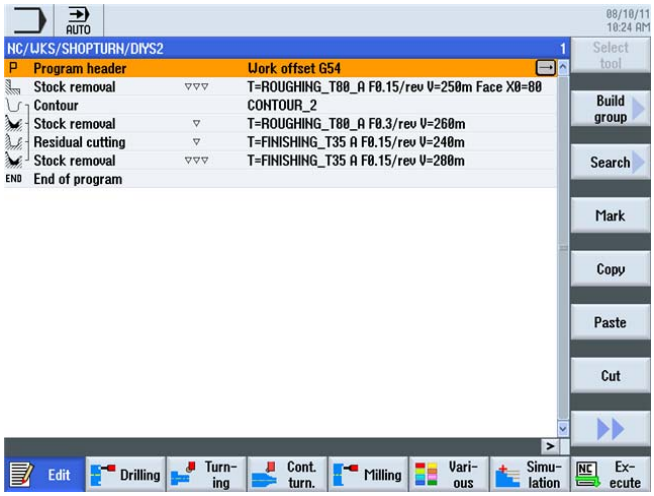


Figure 12-6 Process plan

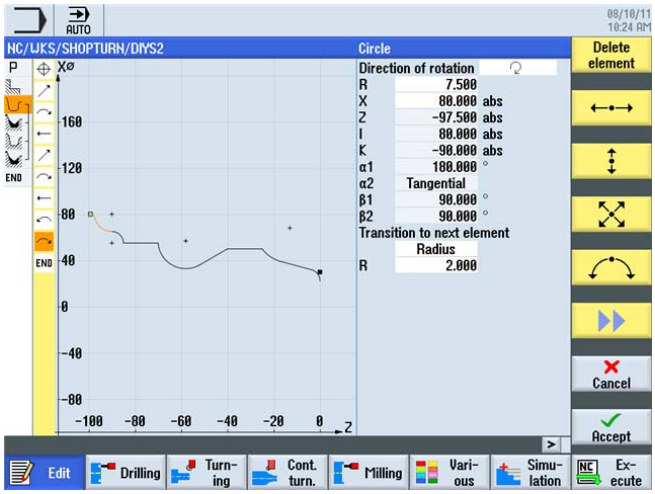


Figure 12-7 Contour in the contour calculator

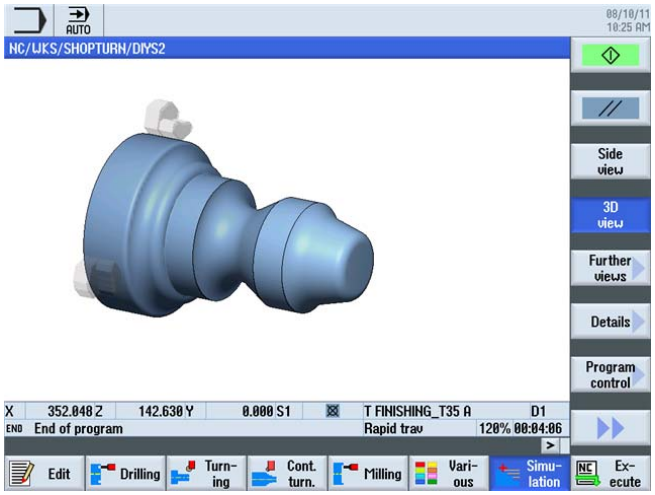


Figure 12-8 Workpiece simulation

12.3 Exercise 3

Will you manage this task using ShopTurn within 10 minutes?

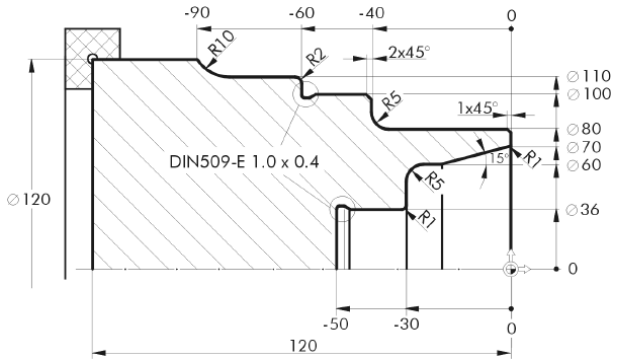


Figure 12-9 Workshop drawing DIYS3

Notes

Construct radius 5 in two steps.

Model

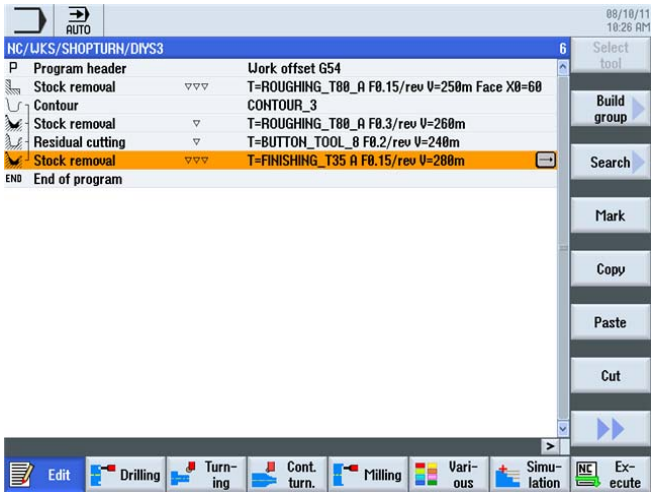


Figure 12-10 Process plan

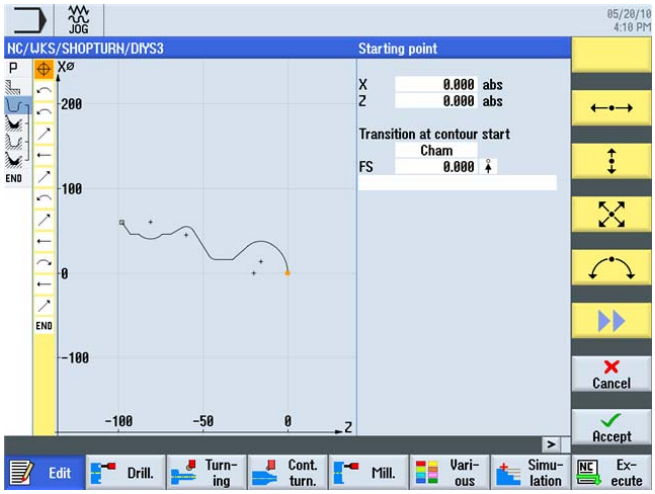


Figure 12-11 Contour in the contour calculator

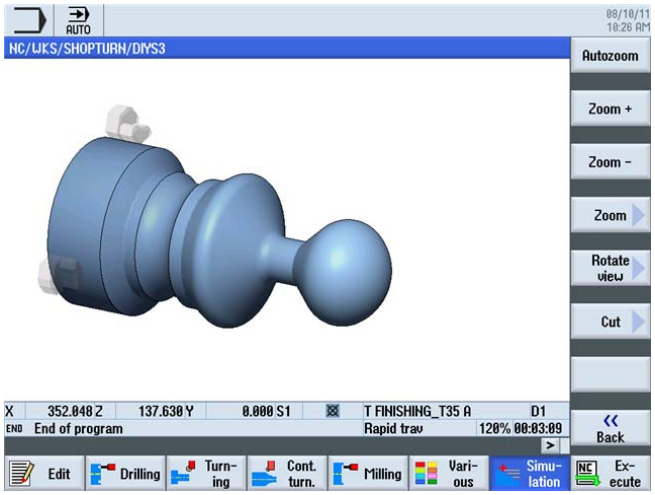


Figure 12-12 Workpiece simulation

12.4 Exercise 4

Will you manage this task using ShopTurn within 15 minutes?

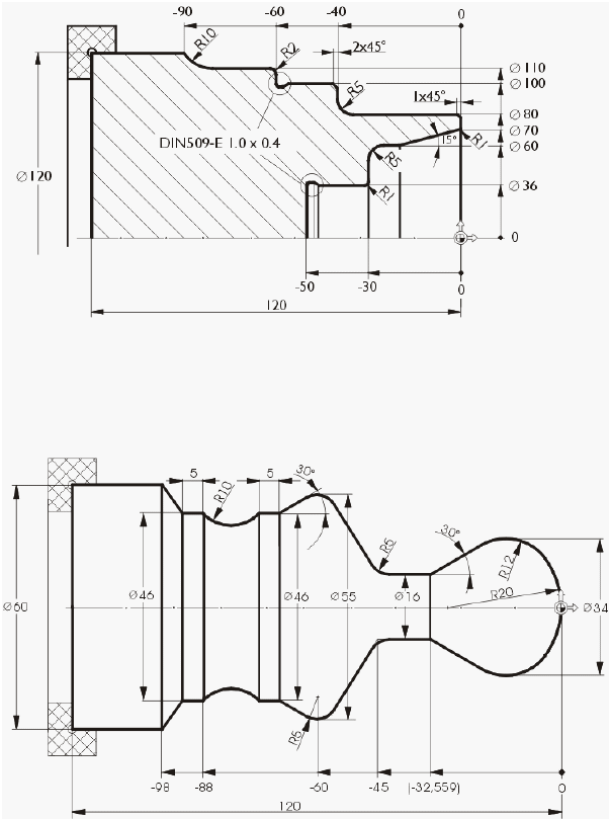


Figure 12-13 Workshop drawing DIYS4

Notes

In the process plan, the end face is roughed and finished first (see model below). Then the entire external area, including the undercut, is produced. Following this, the internal part of the contour is machined. The starting point of the internal contour is set to X70/Z0. You can copy the external and internal machining sequences using the work step editor by means of cut and paste.

Model

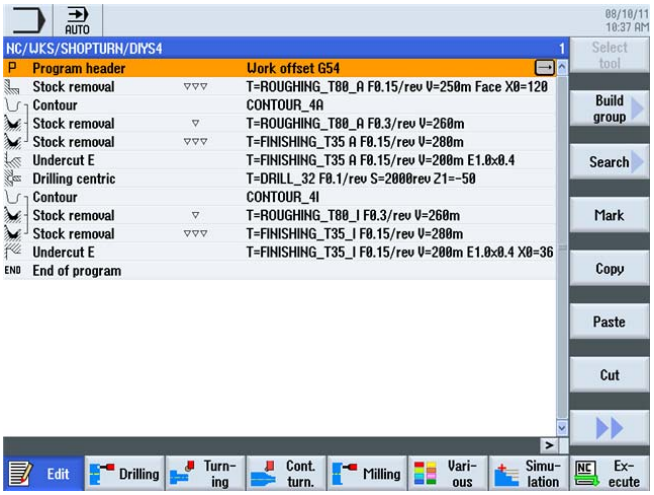


Figure 12-14 Process plan

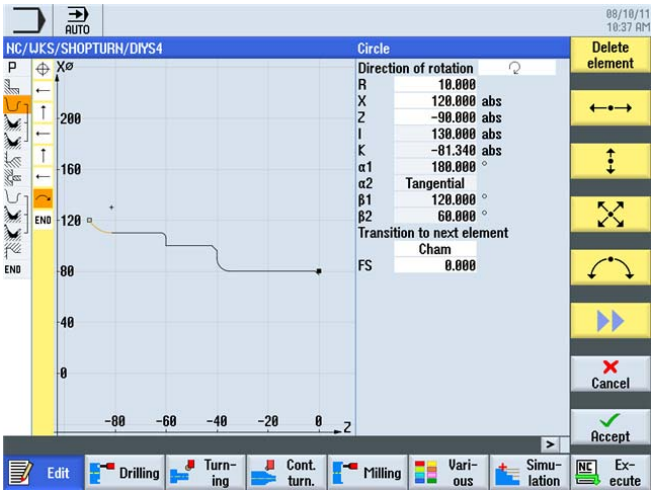


Figure 12-15 External contour in the contour calculator

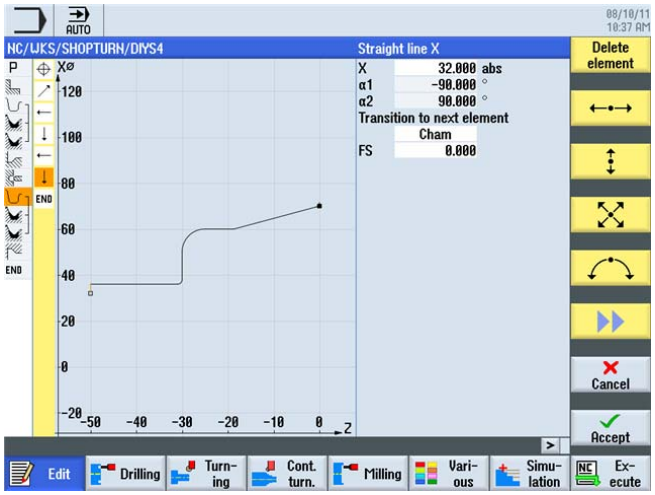


Figure 12-16 Internal contour in the contour calculator

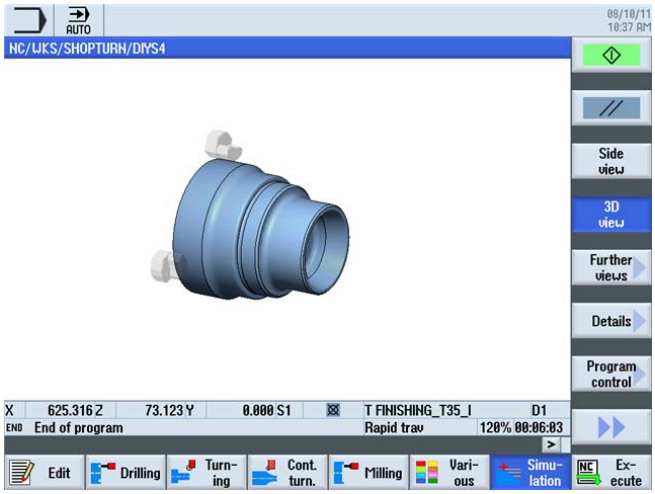


Figure 12-17 Workpiece simulation

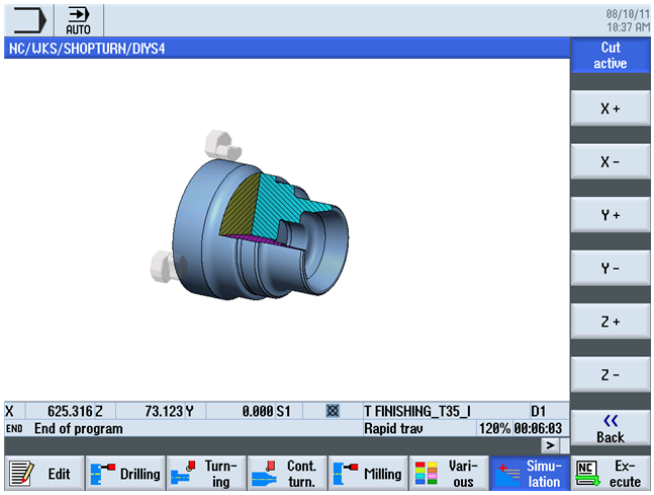


Figure 12-18 Workpiece simulation - Cut active

Index

S

T

U

W