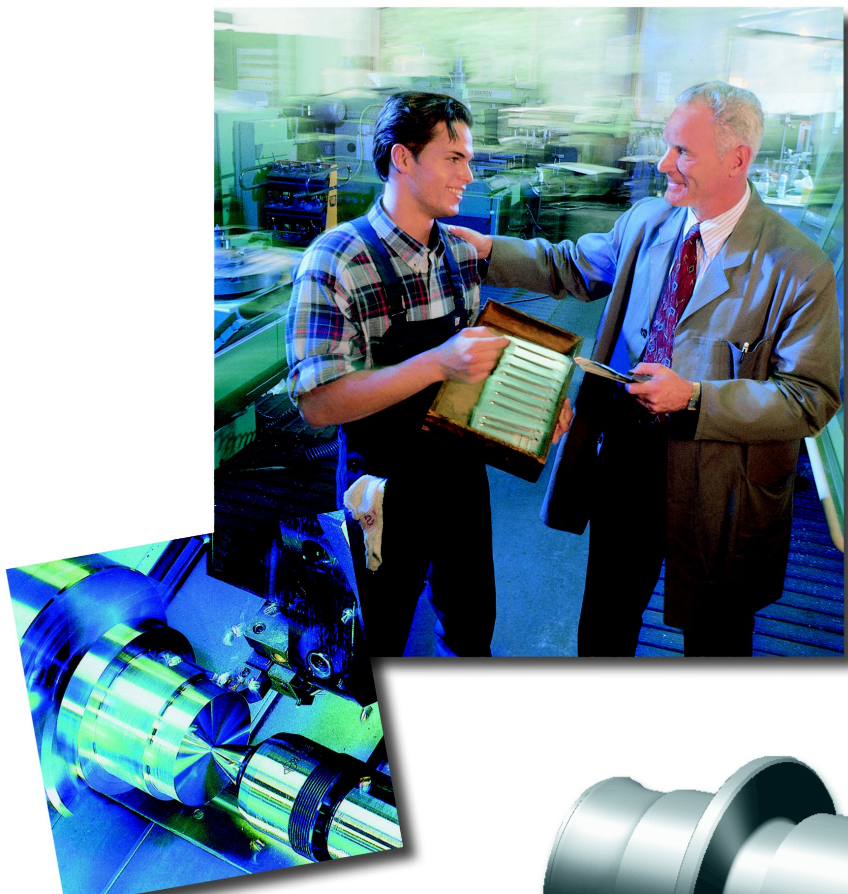
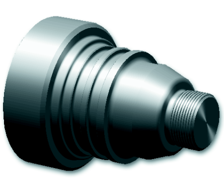
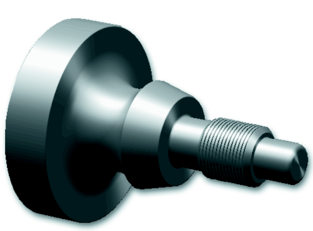
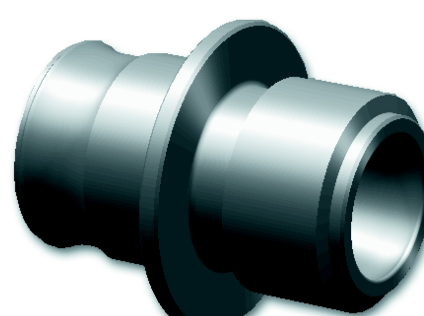
SIEMENS

Turning made easy with ShopTurn









First edition, 2001

This starter guide was compiled by

SIEMENS AG
Automation & Drvies
Motion Control Systems
P.O. Box 3180, D-91050 Erlangen/Germany.

in cooperation with

R. & S. KELLER GmbH
Siegfried Keller, Stefan Nover, Klaus Reckermann, Kai Schmitz
P.O. Box 131663, D-42043 Wuppertal/Germany.

Order No.: 6FC5095-0AA80-0BP0

Preface

Faster from the drawing to the workpiece–but how?

Previously, the NC production normally involved complicated, abstract-coded NC programs. Work that could be performed by specialists only. However, every skilled worker has learnt his craft and with his experience in the area of conventional cutting, is capable at any time of mastering even the most difficult tasks–even when the economics often suffered. These skilled workers needed to be given the possibility to use this knowledge efficiently with the help of CNC machine tools.

This is why with ShopTurn SIEMENS adopts a new path that saves the skilled worker from having to do any coding. Instead, SIEMENS gives these skilled workers a new generation of SINUMERIK control:

Creating a work schedule rather than programming is the solution.

The creation of this work schedule with easy-to-follow, skilled-worker-oriented handling sequences allows the ShopTurn user to make use of his actual knowledge, namely his know-how, for the cutting.

ShopTurn, with its integrated, powerful traverse path creation, allows even the most complicated contours and workpieces to be produced without difficulty. Consequently:

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

Although ShopTurn is indeed very easy to learn, this ShopTurn training document permits an even faster entry into this new world. However, before the actual work with ShopTurn is discussed, the first three chapters refer to important fundamentals:

- Initially, the advantages of working with ShopTurn are mentioned.
- The basics for operating follow.
- The geometric and technological fundamentals of the production are explained for newcomers.

After this theory, practical work with ShopTurn follows:

- Four examples are used to explain the machining capabilities with ShopTurn. The degree of difficulty increases continually with the examples. At the beginning, all key operations are specified; later, encouragement is made to act independently.
- You then learn how you can cut with ShopTurn in automatic operation.
- Finally, if you wish, you can test how fit you are with ShopTurn.

Please note the various situations present in the workshop mean that the technology data used here has only example character.

Just as ShopTurn was created with the help of skilled workers, this training guide was also produced by experts.
We wish you much enjoyment and success in working with ShopTurn.

The authors

Erlangen/Wuppertal, August 2001

Table of contents

1 Advantages When You Work With ShopTurn 5

1.1 You save learning time … 5
1.2 You save programming time … 6
1.3 You save production time … 8

2 To ensure everything operates without problems 10

2.1 Proven technology 10
2.2 The Machine Control Panel 11
2.3 The Contents of the Main Menu 13

3 Basics for Newcomers 18

3.1 Geometrical Basics 18
3.1.1 Axes and Planes 18
3.1.2 Points in the Working Range 18
3.1.3 Absolute and Incremental Dimensions 19
3.1.4 Cartesian and Polar Dimensions 20
3.1.5 Circular Movements 21
3.2 Technological Basics 22
3.2.1 Cutting Speed and Speeds 22
3.2.2 Feed 23

4 Well-Equipped 24

4.1 Tool Management 24
4.1.1 Tool List 24
4.1.2 Tool Wear List 25
4.1.3 Magazine List 25
4.2 The Tools Used 26
4.3 Tools in the Magazine 27
4.4 Calculate Tool Lengths 28
4.5 Set the Workpiece Zero Point 29

5 Example 1: Step Shaft 30

5.1 Program management and create program 31
5.2 Call Tool and Enter Traverse Path 33
5.3 Create Arbitrary Contours Using the Contour Calculator and Rough Cutting 35
5.4 Finishing 39
5.5 Thread Undercut 40
5.6 Threads 41
5.7 Grooves 42

6 Example 2: Drive Shaft 44
6.1 Facing 45
6.2 Create the Contour, Cutting and Residual Cutting 46
6.3 Thread 52

7 Example 3: Guide Shaft 54
7.1 Facing 55
7.2 Creating an Arbitrary Blank Contour 56
7.3 Create the Machined Part Contour and Cut 57
7.4 Cut Residue 62
7.5 Groove 64
7.6 Thread 67
7.7 Drilling 69
7.8 Mill Rectangular Pocket 72

8 Example 4: Hollow Shaft 74
8.1 Create the First Workpiece Side 75
8.1.1 Face Turning 75
8.1.2 Drilling 76
8.1.3 Blank Contour 77
8.1.4 Machined Part Contour of the First Side External 77
8.1.5 Undercut 81
8.1.6 Machined Part Contour of the First Side Internal 83
8.1.7 The Extended Editor 86
8.1.8 Copy a Contour 87
8.2 Create the Second Workpiece Side 88
8.2.1 Face Turning 88
8.2.2 Drilling 89
8.2.3 Adding the Blank Contour 90
8.2.4 Machined Part Contour of the Second Side Outside 90
8.2.5 Create an Asymmetric Groove 93
8.2.6 Machined Part Contour of the Second Side Inside 94

9 And Now We Produce 98
9.1 Approach refrence point 98
9.2 Clamp Workpiece 99
9.3 Set Workpiece Zero Point 99
9.4 Execute Work Schedule 100

10 How Fit Are You With ShopTurn? 102

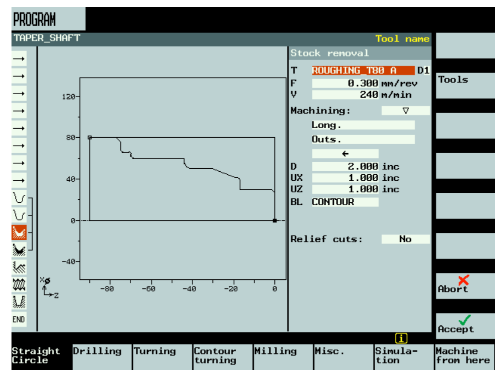
Index 106

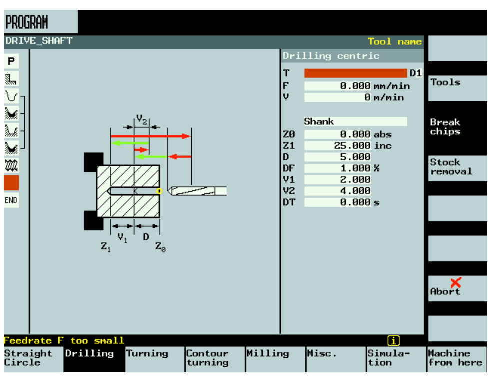
1 Advantages When You Work With ShopTurn

This chapter describes the special advantages you have working with ShopTurn.

1.1 You save learning time …

… because ShopTurn does not have any coding or use foreign-language terms that you would need to learn:
All required inputs are requested in plain text.

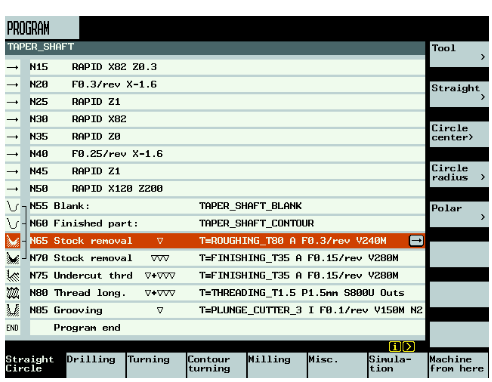


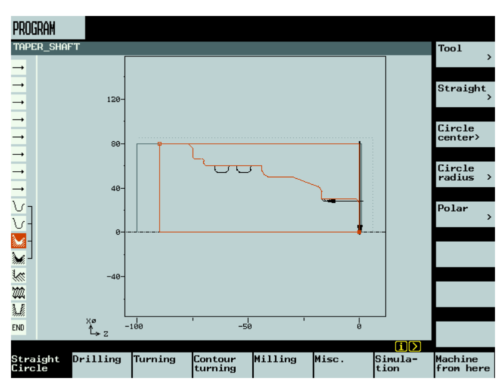


… because of colored pictures that help provide optimum support for ShopTurn

…because you can also integrate DIN/ISO commands in the *graphical work schedule*.

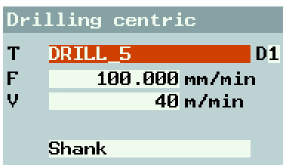
```
N100 G96 S320 LIMS=3000 M4 M8
N105 G18 G54 G90
N110 G0 X32 Z0
N120 G1 X-0.8 F0.1
N130 G0 Z2
N140 G0 G42 X22 Z2
N150 X30 Z-2
```



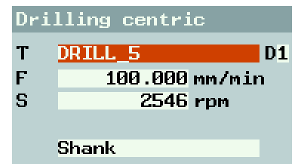


... because at any time you can switch between the individual machining step and the workpiece graphic while creating the work schedule.

1.2 You save programming time …

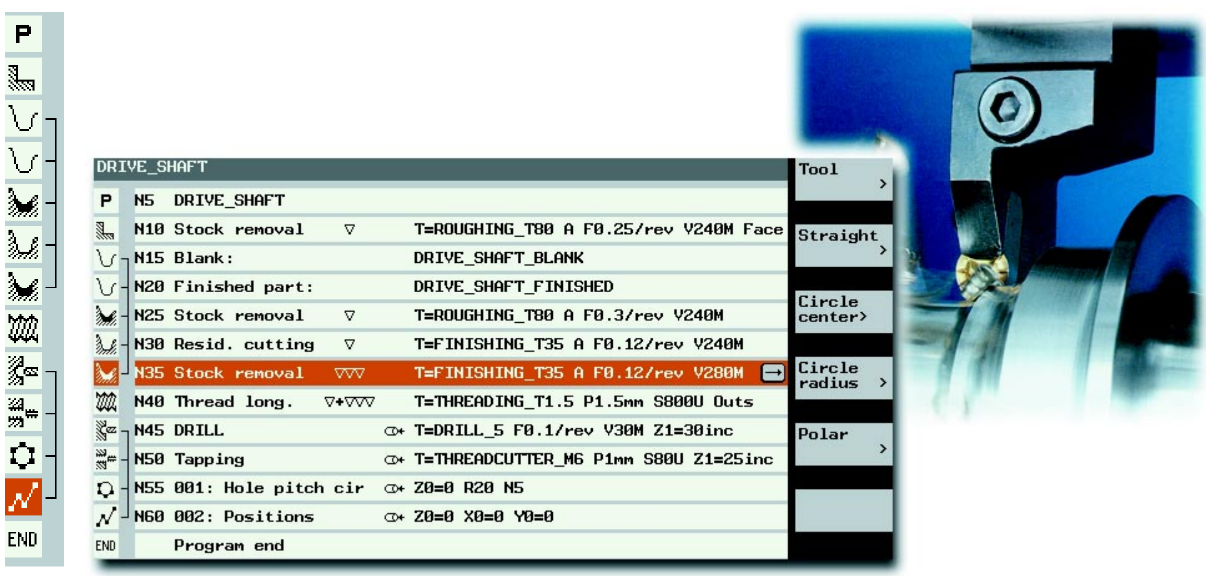


... because ShopTurn provides optimum support right from the input of the technological values: you only need to enter the *values for feedrate (or feed)* and *cutting speed*–ShopTurn automatically calculates the speed at the press of a button.



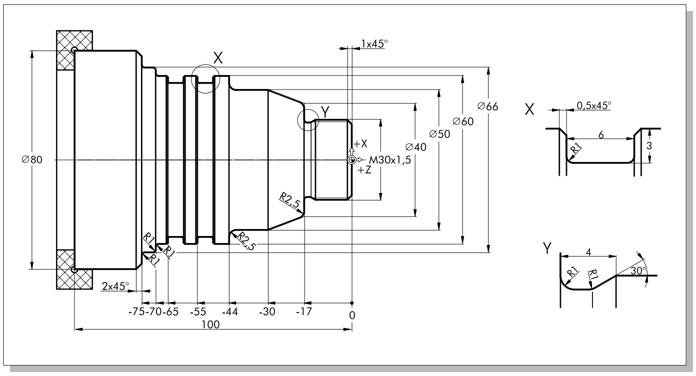
… because ShopTurn allows you to describe a complete machining with a single machining step and then automatically create the required positioning movements (here, from the tool change point to the workpiece and back).

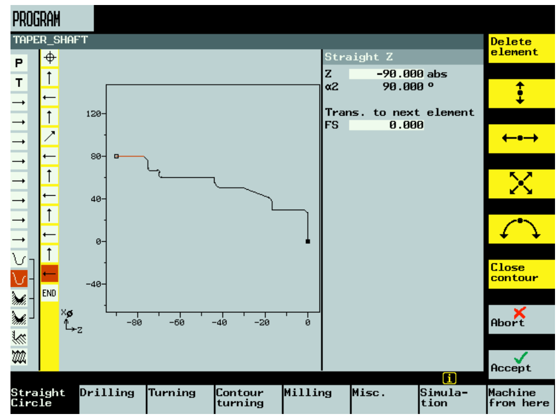
… because all *machining steps* in the ShopTurn graphical work schedule are displayed in a compact and clear manner. This gives you a complete overview and thus better editing capabilities, even for extensive production steps.

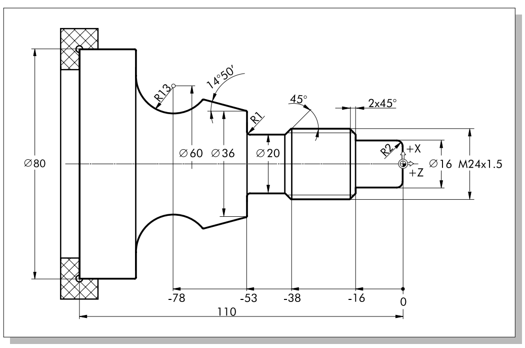


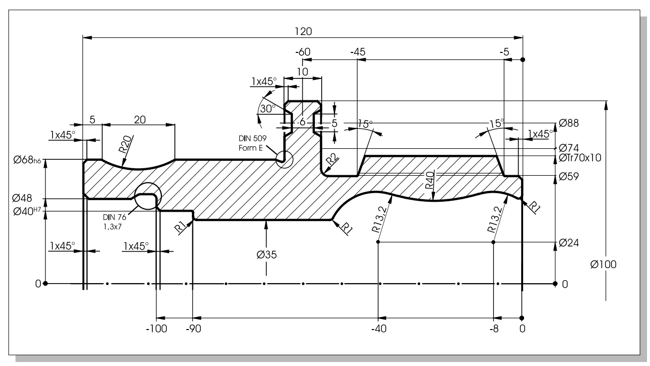
... because several machining operations and contours can be chained together for cutting.

… because the integrated contour computer can process all conceivable dimensioning even though its handling is very simple and clear–thanks to the pictograms and online graphs.

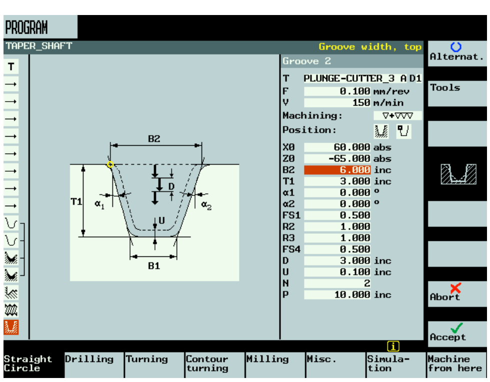


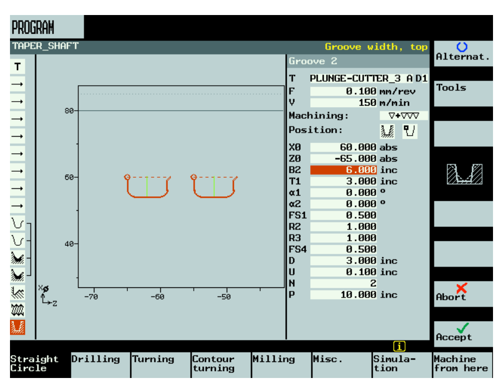






... because at any time you can use a button to switch between static help displays and dynamic online graphs. The online graph provides you with an immediate visual check of the entered values.

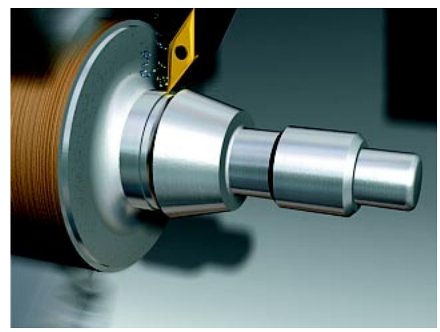




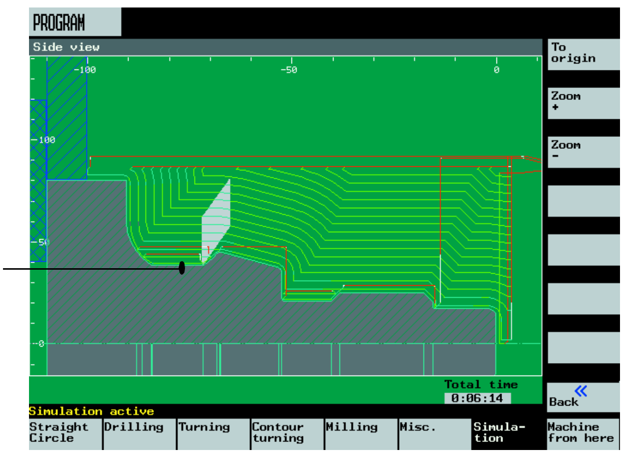
… because the creation of work schedules and production are not mutually exclusive: ShopTurn allows you to create a new work schedule in parallel with the production.

1.3 You save production time …

… because you can optimize the tool selection for the cutting of contours with regard to both time and technology:
Large volumes can be removed using roughing tools, the residual material is then recognized automatically and cut with a pointed tool.



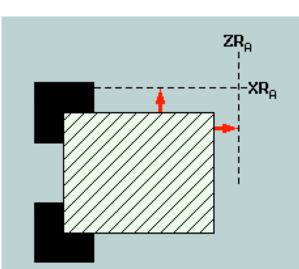
Residual material



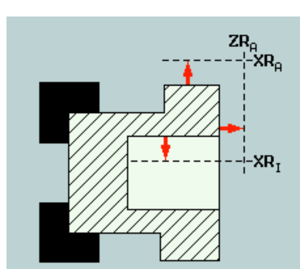
… because the exact specification of the selected return plane avoids the use of unnecessary traverse paths and so saves expensive production time. This is possible using the settings: *normal*, *extended* or *all*.

Help displays in ShopTurn

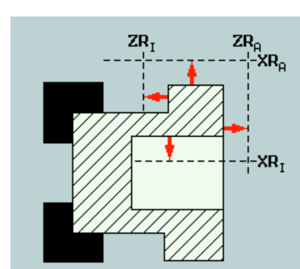
normal *return plane*



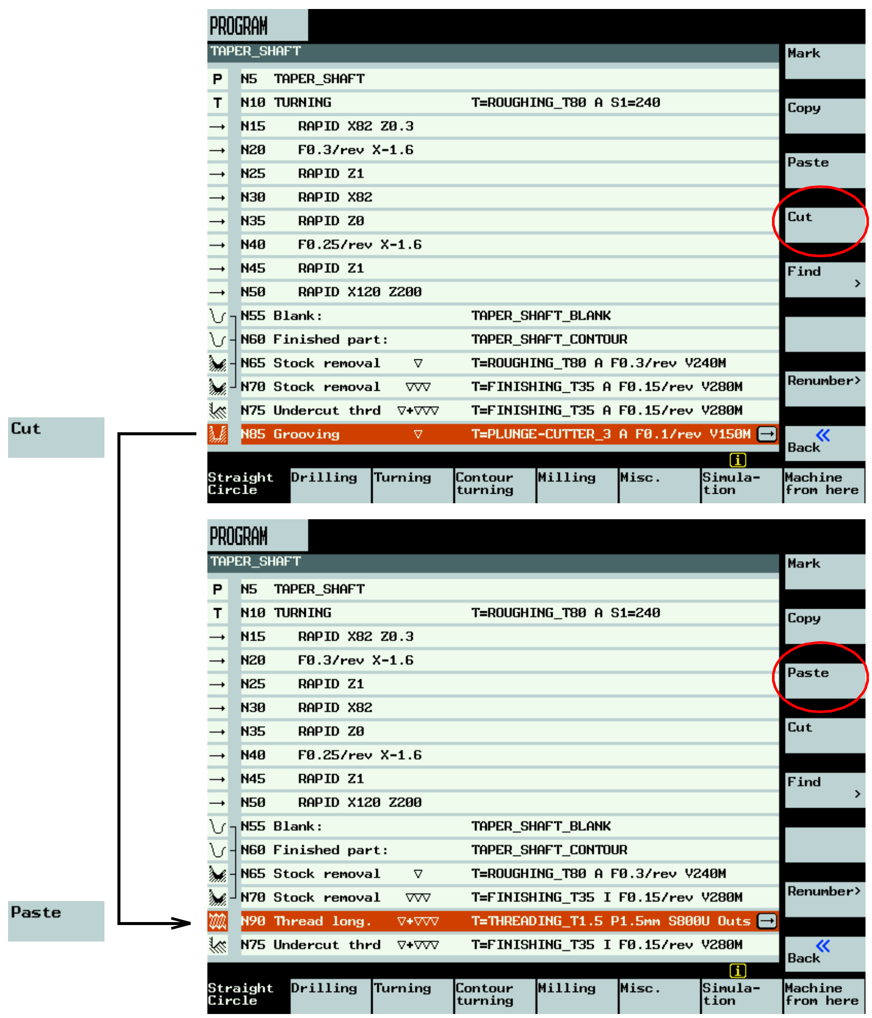
extended *return plane*



all *return plane*



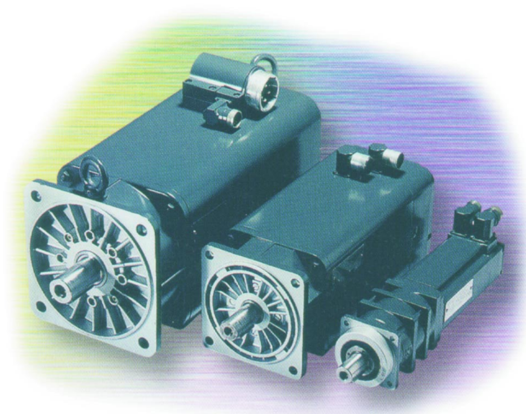
... because the compact structure of the work schedule allows you to optimize your machining sequence with a minimum of effort (e.g. by saving a tool change).



Original machining sequence

Optimizing machining sequence using *cut* & *paste* of the machining step

... because ShopTurn makes use of consistent digital technology (SIMODRIVE drives, ..., SINUMERIK controls) to achieve the maximum feedrates for optimum repeat accuracy.

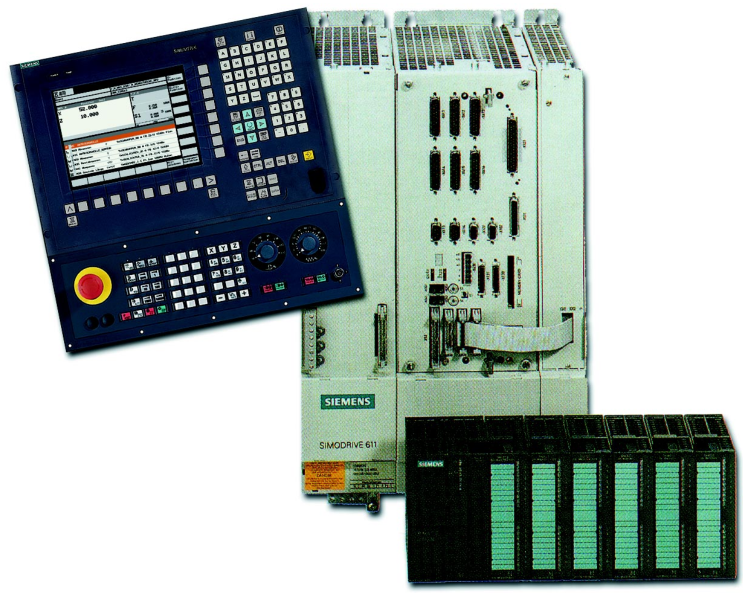


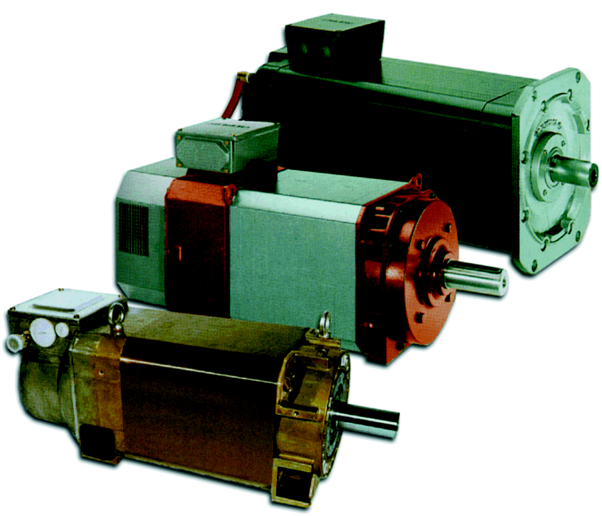
2 To ensure everything operates without problems

This chapter uses examples to describe the basics of the ShopTurn operation.

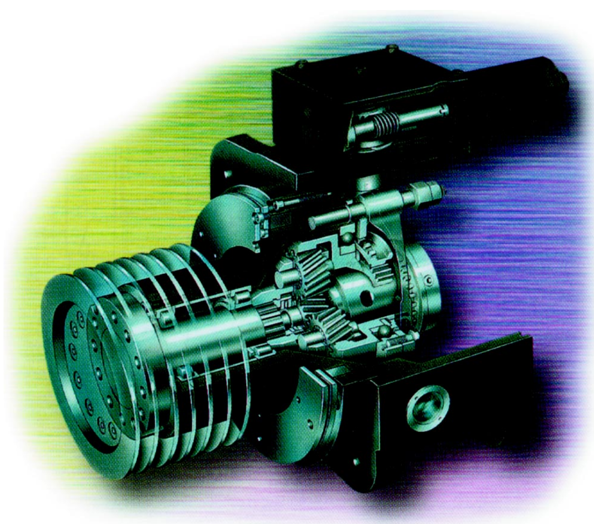
2.1 Proven technology

The SINUMERIK 810D that serves as the basis for ShopTurn is the low-cost entry in the future-oriented digital CNC and drive world for machine tools.





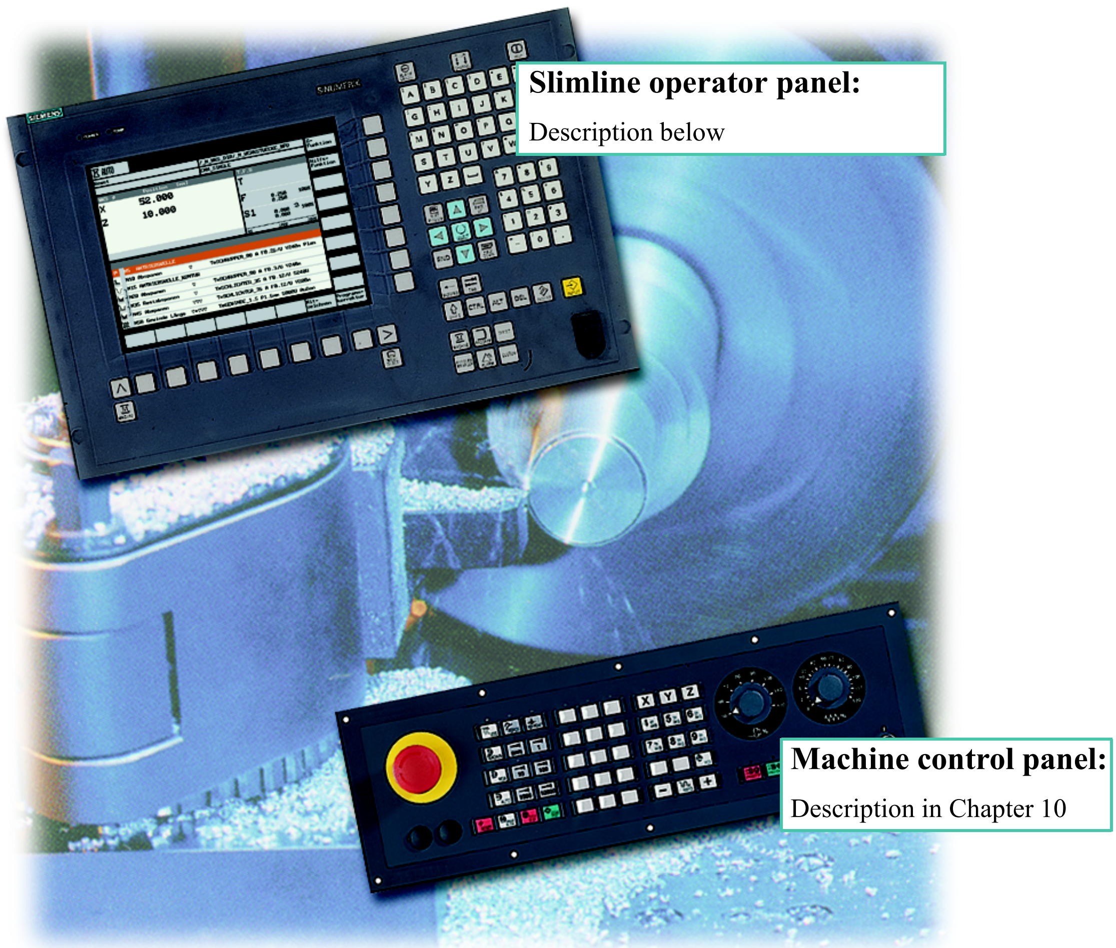
The use of SIEMENS three-phase current motors and ...



... the SIEMENS gearbox technology allows production not only with maximum speeds, but also the highest possible feed and rapid traverse speeds.

2.2 The Machine Control Panel

Although high-performance software is important, it must be easy to use.
ShopTurn's clear machine control panel ensures this. The control panel consists of 2 areas.



The most important buttons on the CNC full keyboard to navigate in ShopTurn are:

Alternative key (the same function as Alternat.)

The 4 arrow keys are used to move the cursor. The right-arrow key also opens machining steps.

The Input key accepts the value in an input field, ends a computing action or moves the cursor down.

The Information key can be used to switch between contour and help diagram or work schedule and workpiece.

This key deletes the inputs to the left.

Del This key deletes the value of an input field.

Scroll up or down by a page.

This key starts the calculator function for the current input field.

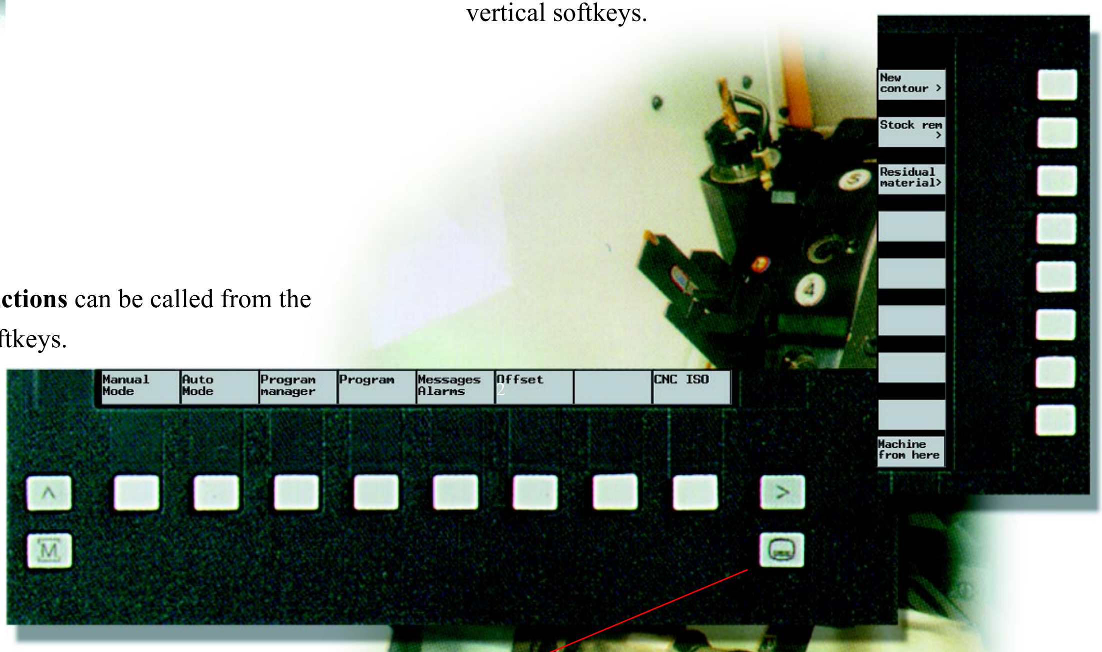
To help you familiarize yourself with ShopTurn, let us look more closely at the key groups.

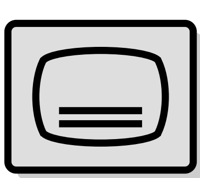
Softkeys

The actual function selection in ShopTurn is made using keys around the screen. Most of these are assigned directly to the individual menu items. Because the contents of the menu change, depending on the situation, these are called softkeys.

All ShopTurn **subfunctions** can be reached from the vertical softkeys.

All **main functions** can be called from the horizontal softkeys.



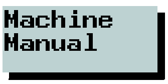


The main menu can be called at any time using this key – irrespective of the operating area in which you are currently working.

Main menu

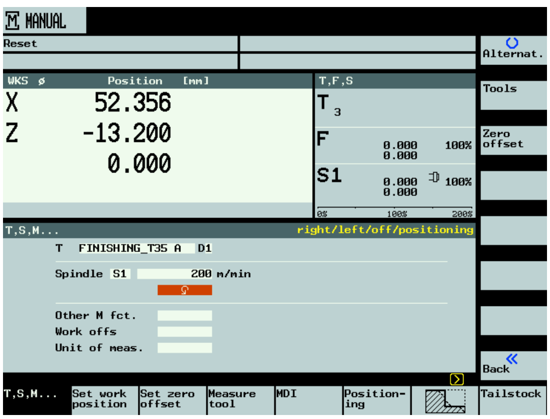
Manual Mode	Auto Mode	Program manager	Program	Messages Alarms	Offset		CNC ISO

2.3 The Contents of the Main Menu

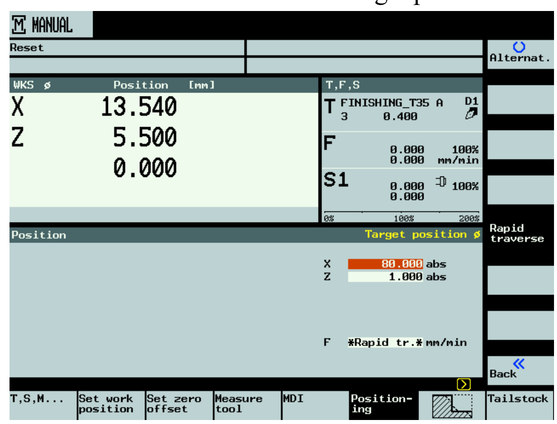


This is used to set up the machine, to traverse the tool in manual operation. Is is also possible to measure tools and set workpiece zero points.

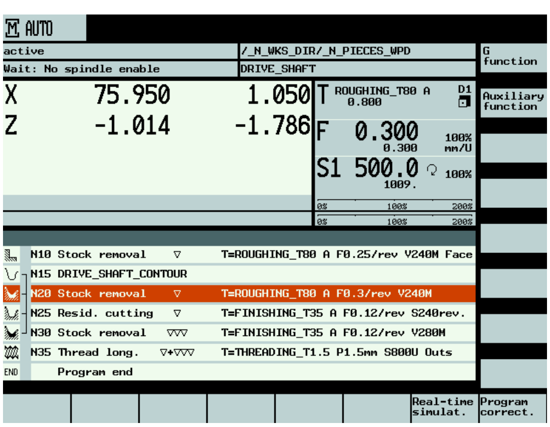
Call a tool and enter the technological values



Enter a target position

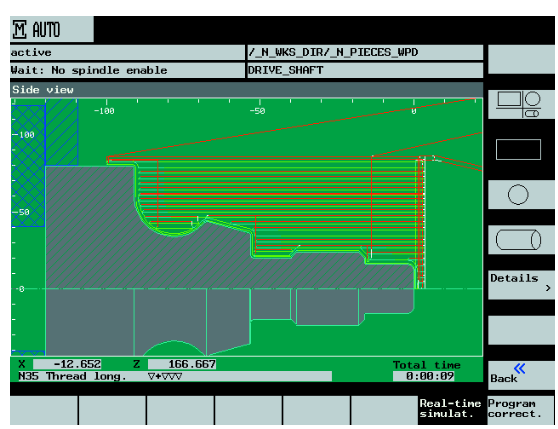


The current machining step is displayed during the production. A switch to the simulation running in parallel can be made at the press of a button. Machining steps can be added or a new work schedule can be started while a work schedule is being executed.



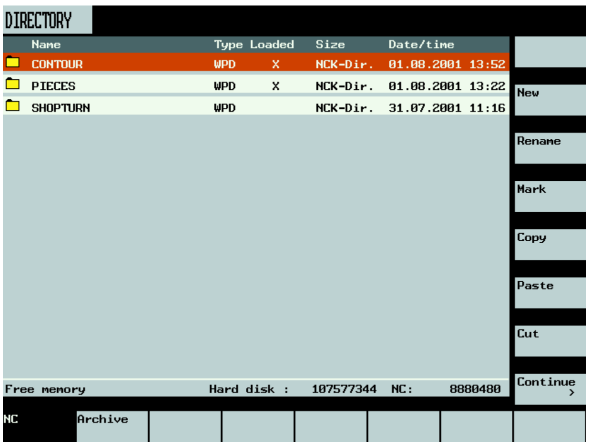
Display the machining steps and the current technology data ...

... or the simulation

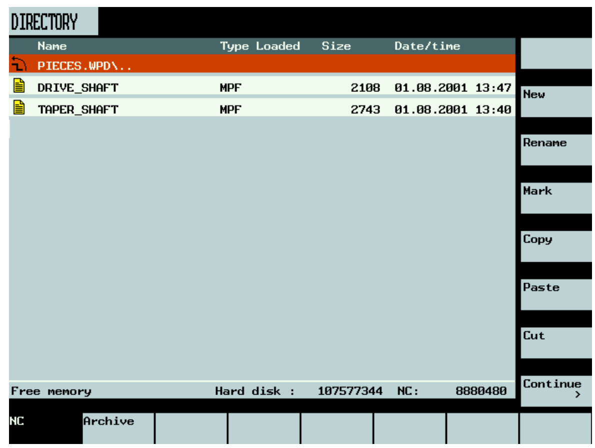


Program manager

The work schedules are managed here. Work schedules can also be exported and imported here.



To ensure that the work schedule list does not become too long and thus no longer easy to use, the *Program Manager* can be used to create any required directories.



The various work schedules can then be stored in the individual directories.

Execute Execute the selected work schedule in the *Auto Machine* mode.

New Recreate the folders and work schedules.

Rename Rename any existing work schedules.

Mark Group work schedules for moving or copying.

Copy Place the marked work schedules in buffer storage.

Paste Insert the contents of the buffer storage, for example in a different folder.

Cut Delete the marked work schedules or machining steps and store in buffer storage.

Continue >

Load HD->NC Move work schedules from the hard disk into the NC core.

Unload NC->HD Move work schedules from the NC core to the hard disk.

Multiple clamping>

Save data > Store the tool data and the zero points in a file.

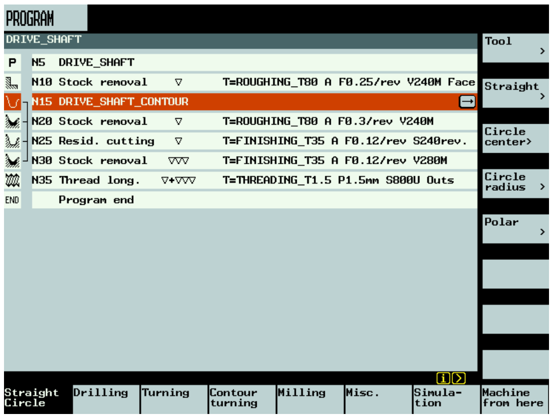
Read out Export the work schedules to an external storage.

Read in Import the work schedules from an external storage.

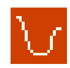
« Back

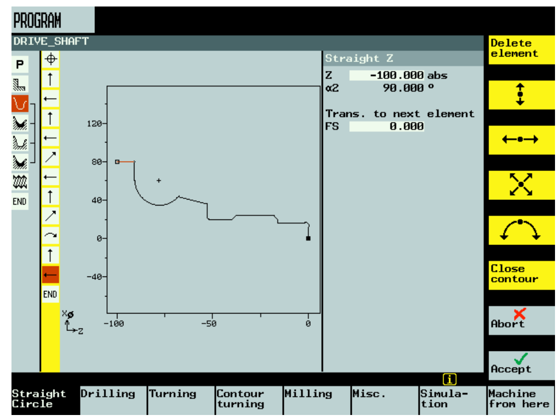
The *Next* and *Return* softkeys can be used at anytime to switch between the two vertical softkey bars.

The work schedule with its complete machining sequence for the associated workpiece is created here. Prerequisite for the optimum sequence is the knowledge of the skilled worker.



Contour

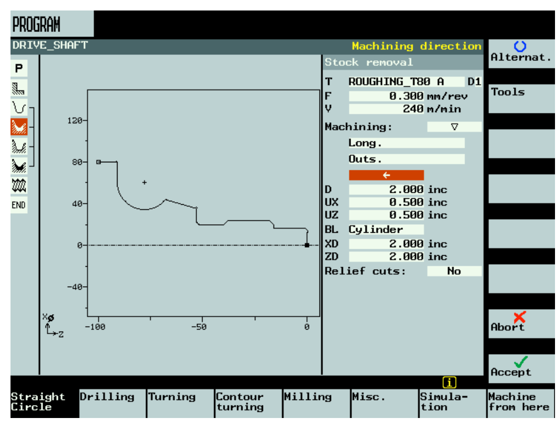




The contour to be machined is entered graphically ...

... and then converted directly into chips: the geometry and technology are fully interconnected.

Cutting machining



Contour

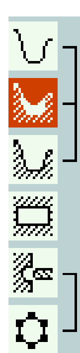
Cutting incl. approach and retract strategies

Residual machining incl. technology

Rectangular pocket incl. technology and position

Drilling technology

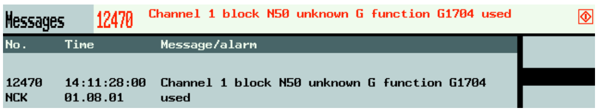
Positions for drilling



Example of the interaction of geometry and technology

This geometric-technological connection is shown very clearly in the graphical display of the machining steps by "bracketing" the corresponding symbols. Here, the "bracketing" means a chaining of geometry and technology to produce a machining step.

Messages Alarms

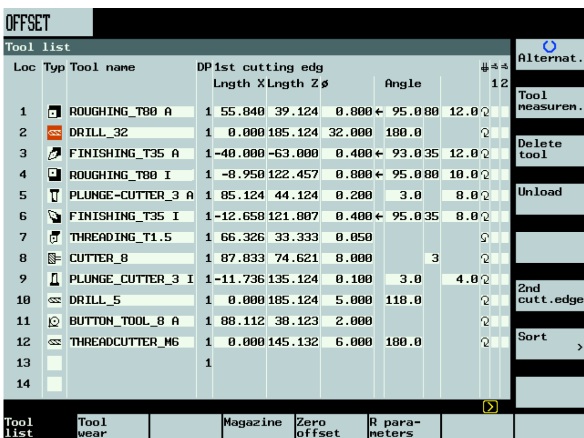


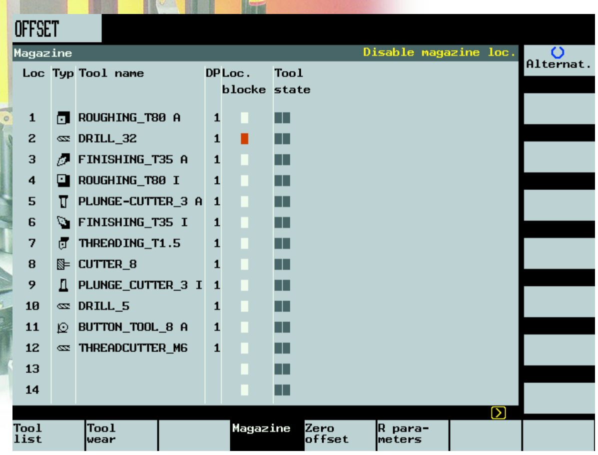
All currently pending messages and alarms with corresponding error number, the time of occurrence of the error and other explanations are displayed here.

The ShopTurn user documentation contains a listing of the messages and alarms.

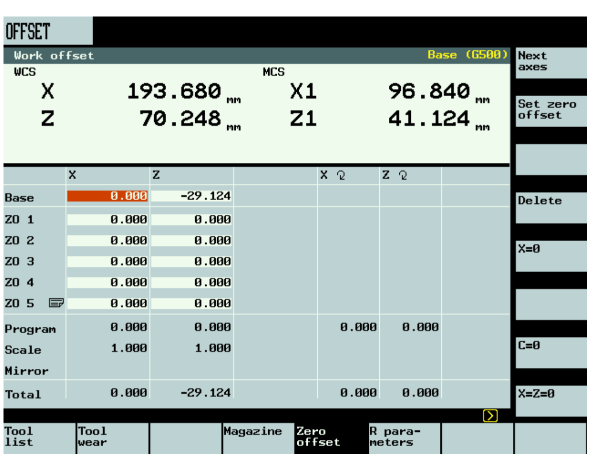
tool zerooffs.

No cutting without tools. These can be managed in a tool list...



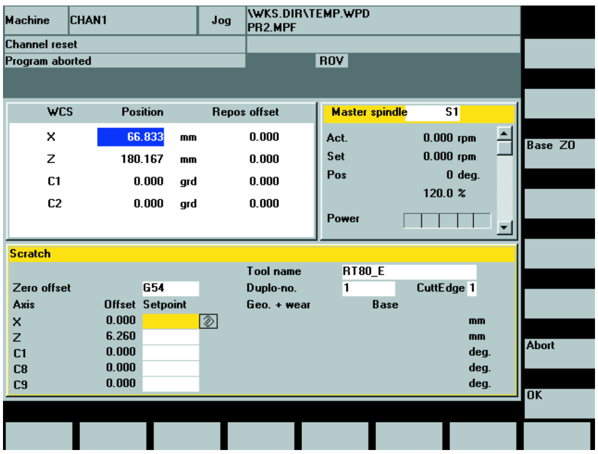


... and grouped to produce a magazine.

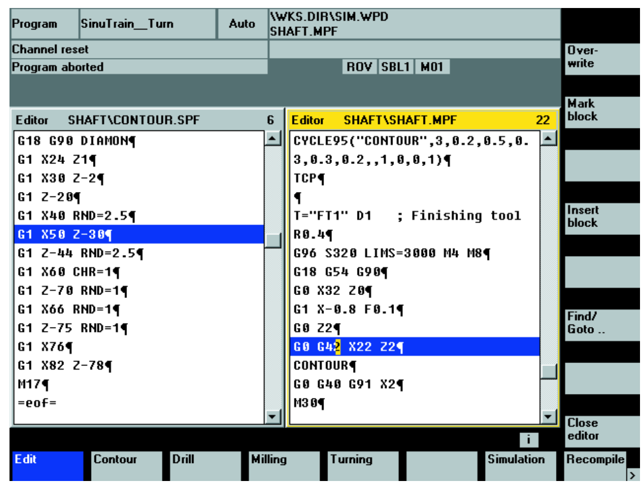


The zero points are stored in a clear zero point table.

The ShopTurn user interface is based on the proven SINUMERIK 810D control. The *CNC ISO* key can be used to switch to the SINUMERIK level. The production can then run exactly as on other 810D/840D controls.



The combination of ShopTurn with the SINUMERIK 810D results in large flexibility in the CNC production.



People familiar with SINUMERIK 810D will know these pictures.

3 Basics for Newcomers

This chapter discusses the general basics of the geometry and the technology for turning. No ShopTurn inputs are made yet.

3.1 Geometrical Basics

3.1.1 Axes and Planes

In turning, the tool does not rotate but the workpiece. The axis is the Z axis.

G18 plane = machining with turning tools

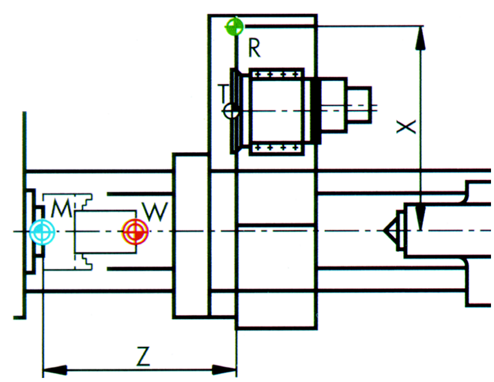
G17 plane = drilling and milling operations on the front face

G19 plane = drilling and milling operations on the peripheral surface

Because the diameter of turned workpieces is relatively easy to control, the dimensions of the transverse axis are based on the diameter. Thus, the skilled worker can directly compare the actual dimensions with the dimensions specified on the drawing.

3.1.2 Points in the Working Range

To allow a CNC control—such as the SINUMERIK 810D with ShopTurn—to orient itself using the measuring system on the existing working range, there are several important reference points there.



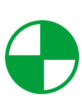
Machine zero point M

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

Workpiece zero point W

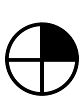
The workpiece zero point W, also called program zero point, is the origin of the workpiece coordinate system. It can be freely selected and should be placed at the point in the drawing where most dimensions originate.

Reference point R



Because the machine zero point normally cannot be approached, the reference point R is approached to zero the measuring system. In this way the control finds its counting begin in the position measuring system.

Toolholder reference point T

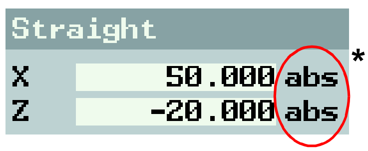


The toolholder reference point T is used to set up machines with tool revolvers with default tools. Its position and location hole permit the setup with cutter holders for shank tools in accordance with DIN 69880 and VDI 3425.

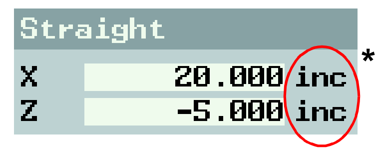
3.1.3 Absolute and Incremental Dimensions

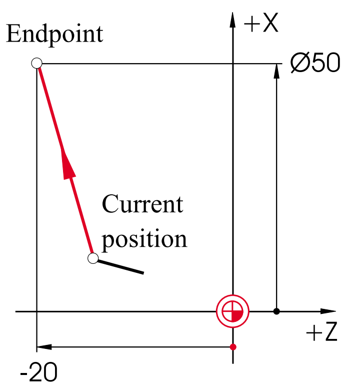
Absolute inputs:
The entered values use the workpiece zero point as reference.

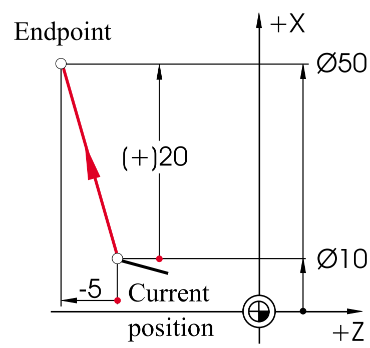
Increment inputs:
The entered values are relative to the current position.



The softkey Alternat. or the key can be used at any time to switch.







*G90 Absolute dimensions

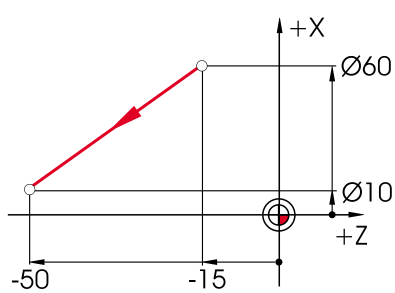
The **absolute** coordinates values of the **end point** in the active coordinate system must always be entered for absolute inputs (the current position is not considered).

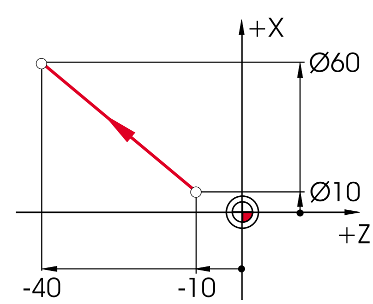
*G91 Incremental dimensions

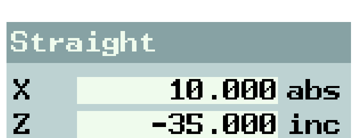
The **differential** values between the **current position** and the **end point** with regard to the **direction** must always be entered for incremental inputs.

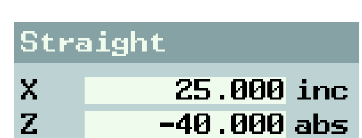
The inputs can also be made as an absolute/incremental combination.

Here two examples:









3.1.4 Cartesian and Polar Dimensions

Two quantities are needed to determine the end point of a straight line. These can have the following form:

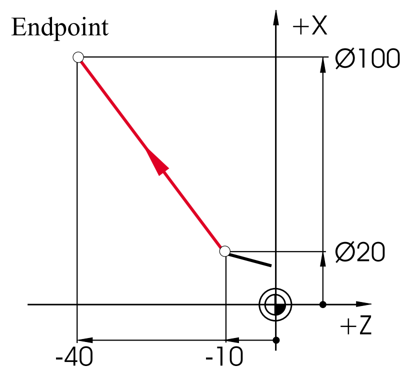
Cartesian: Input of the X and Z coordinates

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

All gray values were calculated automatically.

Polar: Input of the length and an angle

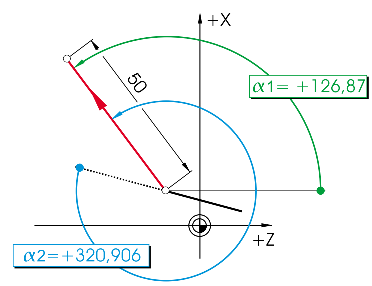
Straight		
X	40.000	inc
X	100.000	abs
Z	-30.000	inc
Z	-40.000	abs
L	50.000	
α1	126.870	°
α2	320.906	°



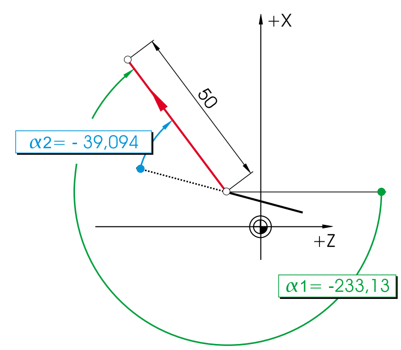
α1 Angle to the positive Z axis

α2 Angle to the previous element

The angles can be **positive** and/or ...

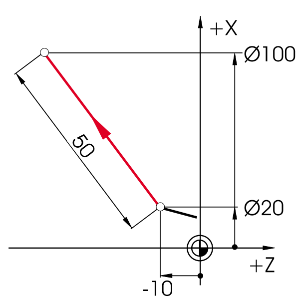


... **negative**.



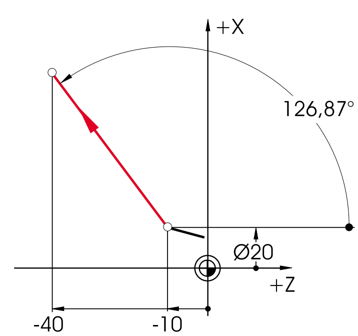
The Cartesian and polar inputs can be combined.
Here two examples:

Input of the end point in X and the length



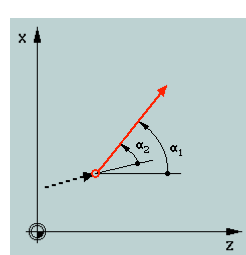
Straight		
X	40.000	inc
X	100.000	abs
Z	-30.000	inc
Z	-40.000	abs
L	50.000	
α1	126.870	°
α2	320.906	°

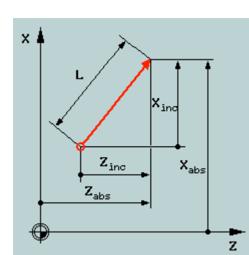
Input of the end point in Z and an angle



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

The context-sensitive help pictures can be called during the input and show the designations of the individual input fields.





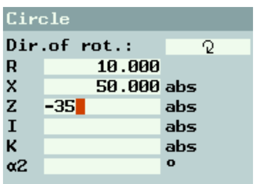
3.1.5 Circular Movements

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).

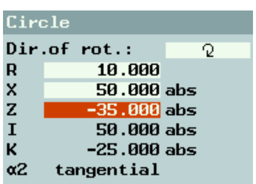
The ShopTurn contour computer also gives you the freedom to use any dimension from the drawing without conversion effort for circular arcs.

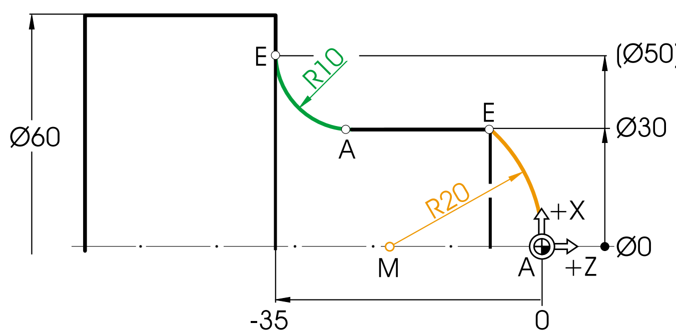
The following example shows two circular arcs, one of which, however, is only partially determined.

Input of the arc R10:

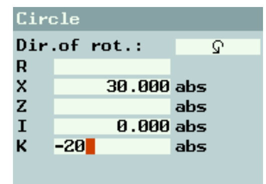


After input:

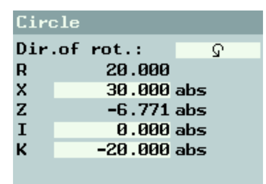




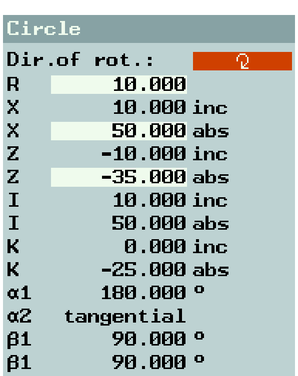
Input of the arc R20:

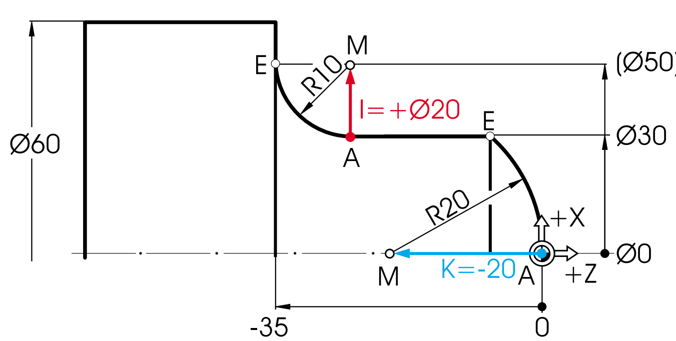


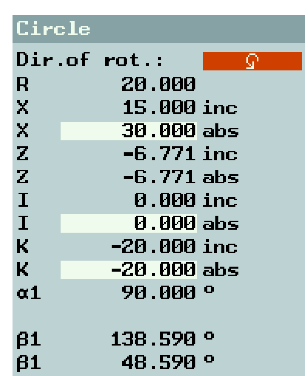
After input:



The following displays of all values result once you have entered all known dimensions and pressed the softkey in the input window of the All parameters associated arc..







The inputs of the arcs in DIN format would be:

G2 X50 Z-35 CR=10 **G3 X30 Z-6.771 I0 K-20**

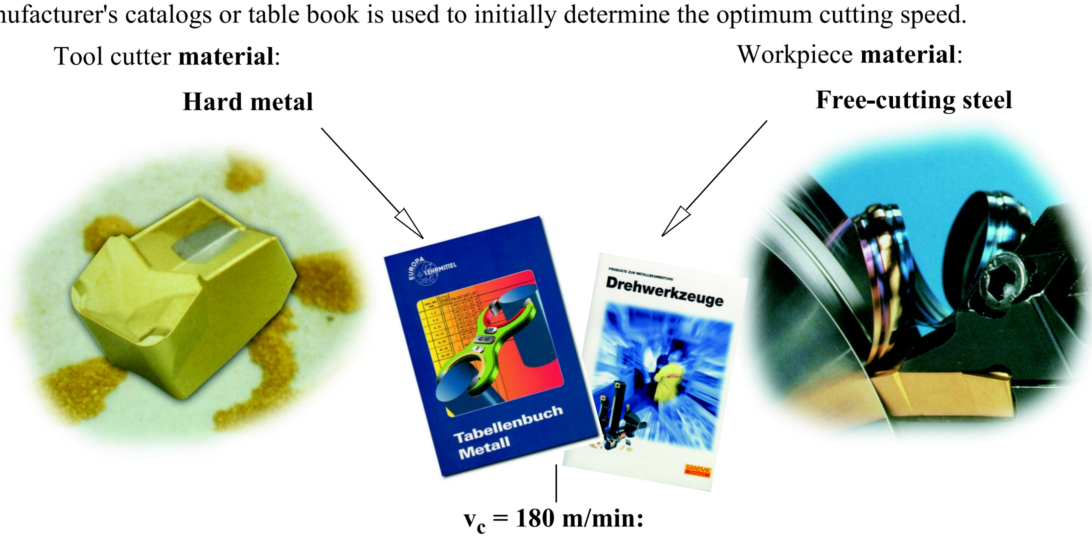
3.2 Technological Basics

3.2.6 Cutting Speed and Speeds

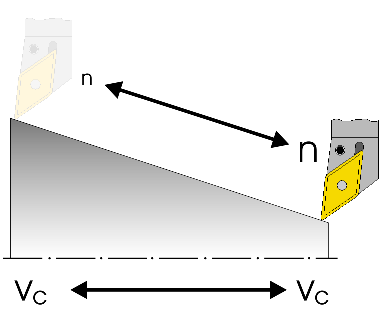
The cutting speed is normally directly programmed for turning, in particular for rough cutting, finishing and plunge-cutting. The speed is programmed only for drilling and (normally) thread cutting.

Determining the cutting speed:

The manufacturer's catalogs or table book is used to initially determine the optimum cutting speed.

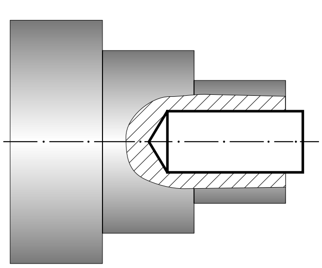


Constant cutting speed v_c, (G96) for rough cutting, finishing and plunge-cutting:



To ensure that the selected cutting speed remains constant for each workpiece diameter, the control uses the G96 command (= constant cutting speed) to adapt the associated speed. This is done using direct-current- or frequency-controlled three-phase motors. As the diameter reduces, the speed will theoretically increase to infinity. Consequently, to prevent danger resulting from excessive centrifugal forces, a speed limitation must be programmed, for example, n = 3000 rpm. In DIN format, the block would have the following form: **G96 S180 LIMS=3000** (from Limes = limit).

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



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

d = 20mm (tool diameter)

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

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

Because a constant speed is used for drilling, the command G97 (= constant speed) must be used here.

The speed depends on the required cutting speed (selected here as 120m/min) and the tool diameter.

The inputs are then G97 S1900.

3.2.7 Feed

You learnt on the previous page how to determine the cutting speed and to calculate the speeds. To make the tool cut, a feed for the tool must be assigned to the cutting speed and to the speed.

Determine the feed:

As for the cutting speed, the value for the feed is obtained from the table book or the tool manufacturer's documents or values gained from experience.

Tool cutter **material**: **Hard metal**

Workpiece **material**: **Free-cutting steel**

Feed f = 0.2–0.4mm:

The middle value **f = 0,3mm** is chosen (often called mm per revolution in the workshop).

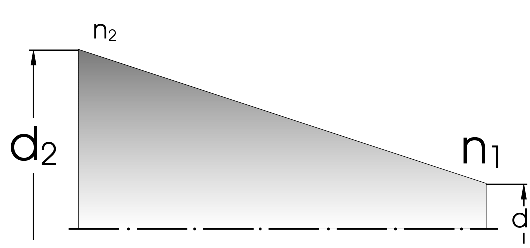
The input is then F0.3

Relationship between feed and feedrate:

The constant feed **f** and the associated speed **n** produce the feedrate $\mathbf{v_f}$.

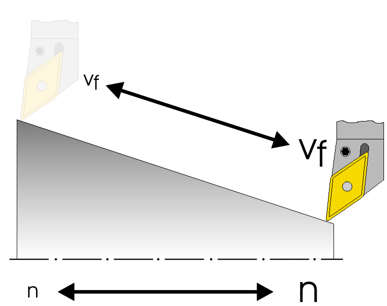
$$v_f = f \cdot n$$

$$v_c = 180\frac{m}{min}$$
$$d_2 = 80mm$$
$$n_2 \approx 710\frac{1}{min}$$
$$v_{f2} = 710\frac{1}{min} \cdot 0,3mm$$
$$v_{f2} \approx 210\frac{mm}{min}$$



$$v_c = 180\frac{m}{min}$$
$$d_1 = 20mm$$
$$n_1 \approx 2800\frac{1}{min}$$
$$v_{f1} = 2800\frac{1}{min} \cdot 0,3mm$$
$$v_{f1} = 840\frac{mm}{min}$$

Because the speed is different, the feedrate (despite the same feed) is different for the various diameters.



4 Well-Equipped

This chapter describes how the tools for the examples in the following chapters are set up. The calculation of the tool lengths and the setting of the workpiece-zero point is also explained as an example.

4.1 Tool Management

ShopTurn offers three lists for tool management.

4.1.1 Tool List

All tools present in the control and their offset data are entered and displayed here. This is also irrespective of whether or not the tools are assigned to a magazine location.

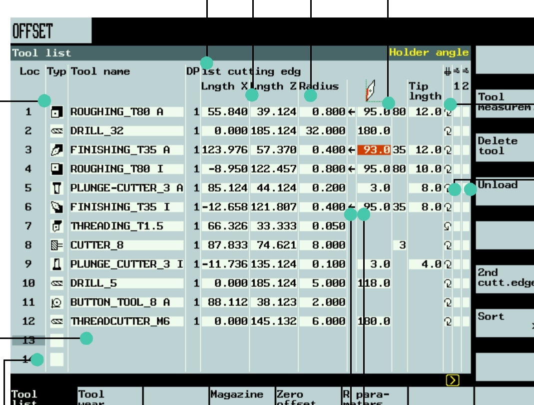
Radius or diameter of the tool

Length of the tool

Seven tool types and a stop are provided. Depending on the tool type, there are various mounting positions and geometrical parameters (e.g. fixing bracket).

DP = Duplo-Number (this creates a replacement tool with the same name)

Inputs:
Fixing bracket (roughing tool and finishing tool, incl. pictogram display) and point angle (drill) and board width (plunge-cutter)



Rotational direction of the spindles or the tool

Switch on / off coolant supply 1 and 2

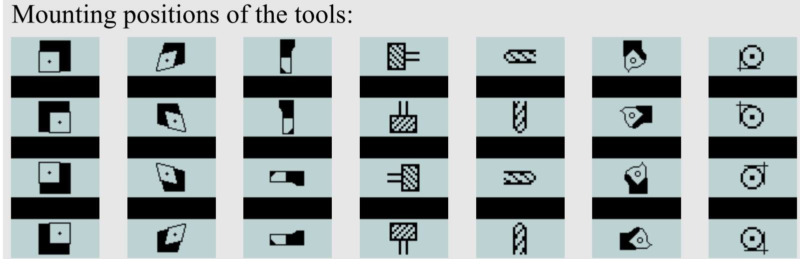
The tool name is automatically suggested based on the selected tool type. Although this name can be freely changed, its length must not exceed 17 characters. All letters (other than accented characters), digits and underscores are permitted as input.

The location number indicates whether and where the tool is installed in the magazine.

Plate angle or number of teeth for milling tools

Main cutting direction of the tool

Mounting positions of the tools:

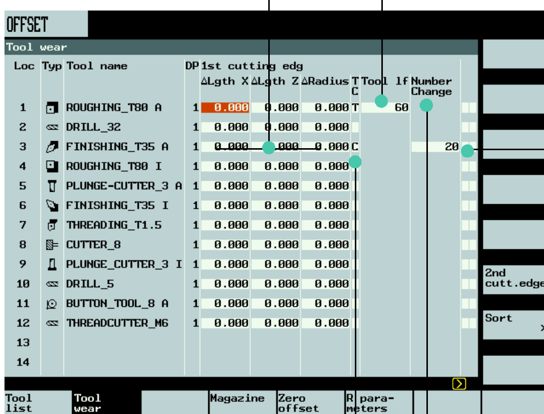


4.1.2 Tool Wear List

The wear data for the associated tools is specified here.

Enter here the tool wear based on the differential values of the tool length or the tool diameter.

Enter here the service life in minutes provided this function was previously enabled.



These switch fields can be used to specify the following properties:

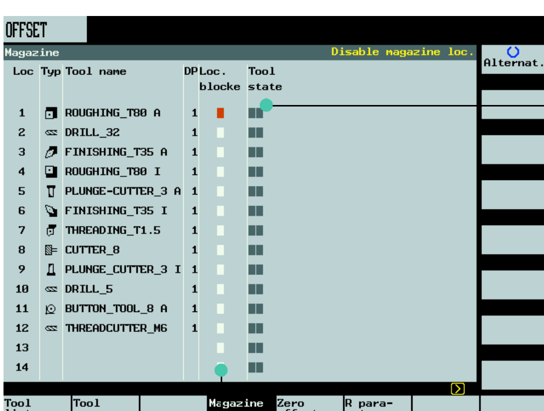
1. Lock tool
2. Tool too large

Enter here the number of tool replacements provided this function was previously activated.

Specify here the monitoring of the tool based on the service life or the number of tool replacements. T means that the service life is monitored; C means that the number of replacements is monitored.

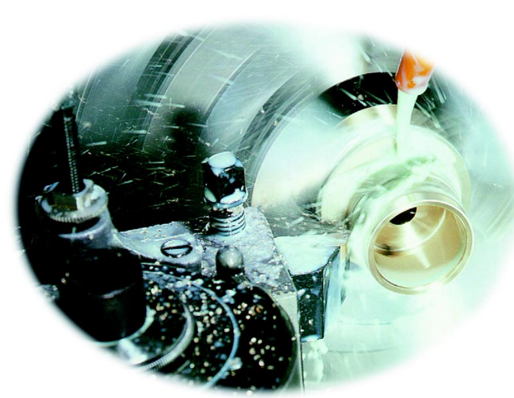
4.1.3 Magazine List

The magazine list contains all tools that are assigned to one or more tool magazine(s). This list displays the status of each tool. In addition, individual magazine locations can be reserved or locked for planned tools.



The current tool state is displayed here.

The location inhibit is activated here.



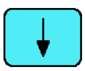
4.2 The Tools Used

Enter in the following tool list the tools that are required for subsequent machining of the examples.

Create tool:

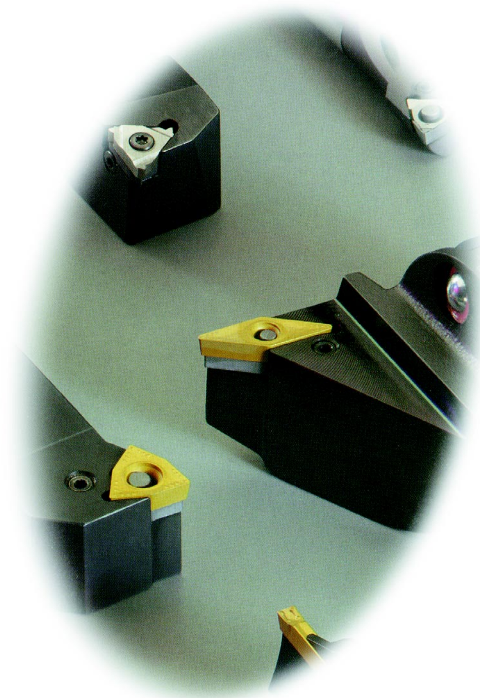
tool zerooffs.

Tool list



... search for empty location

New tool>



Select tool type ...

Roughing tool

Finishing tool

Plunge cutter

Milling tool

Drill

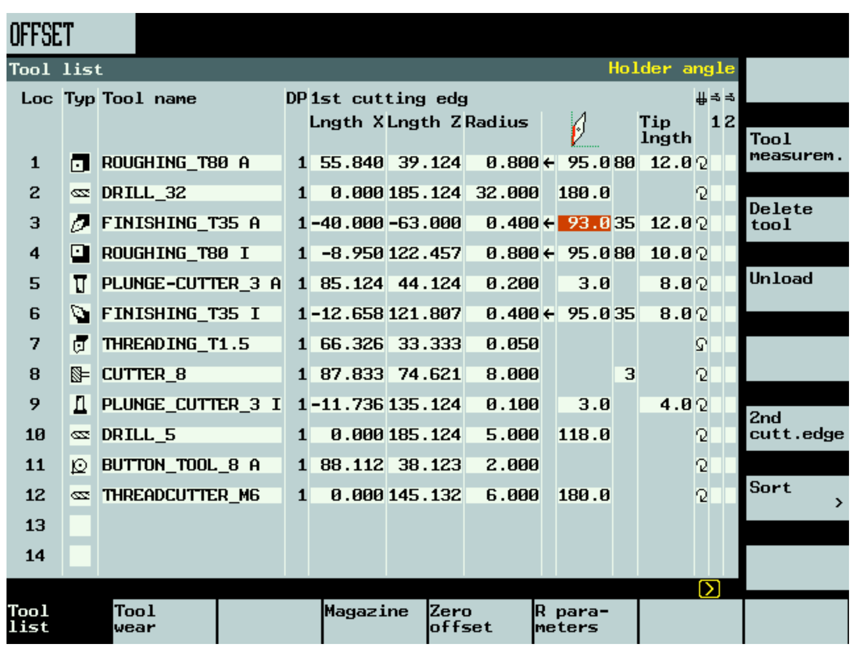
more

Threading tool

Button tool

Dead stop

...and enter the data



Loc	Tool name	DP	Lngth X	Lngth Z	Radius		Holder angle		Tip lngth
1	ROUGHING_T80 A	1	55.840	39.124	0.800	←	95.0	80	12.0
2	DRILL_32	1	0.000	185.124	32.000		180.0		
3	FINISHING_T35 A	1	-40.000	-63.000	0.400	←	93.0	35	12.0
4	ROUGHING_T80 I	1	-8.950	122.457	0.800	←	95.0	80	10.0
5	PLUNGE-CUTTER_3 A	1	85.124	44.124	0.200		3.0		8.0
6	FINISHING_T35 I	1	-12.658	121.807	0.400	←	95.0	35	8.0
7	THREADING_T1.5	1	66.326	33.333	0.050				
8	CUTTER_8	1	87.833	74.621	8.000			3	
9	PLUNGE_CUTTER_3 I	1	-11.736	135.124	0.100		3.0		4.0
10	DRILL_5	1	0.000	185.124	5.000		118.0		
11	BUTTON_TOOL_8 A	1	88.112	38.123	2.000				
12	THREADCUTTER_M6	1	0.000	145.132	6.000		180.0		
13									
14									

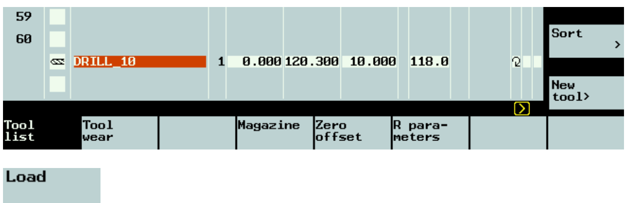
Note: Because milling tool 8 is used to mill a pocket, it must be able to be inserted.

4.3 Tools in the Magazine

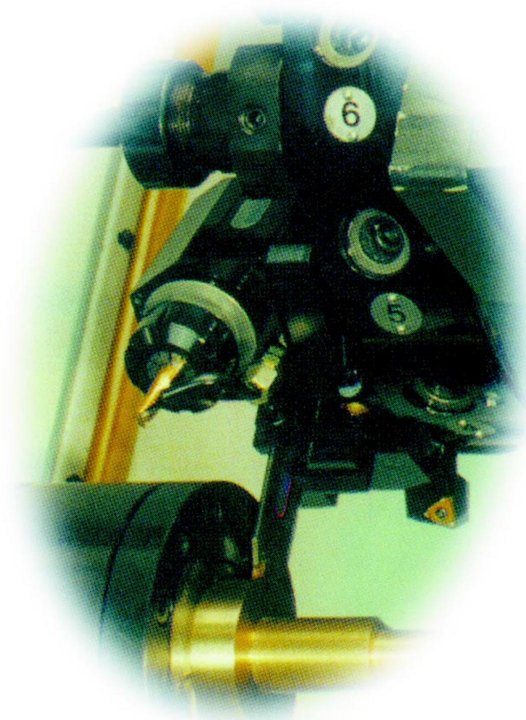
The tools are placed in the magazine in the following section.

Load magazine:

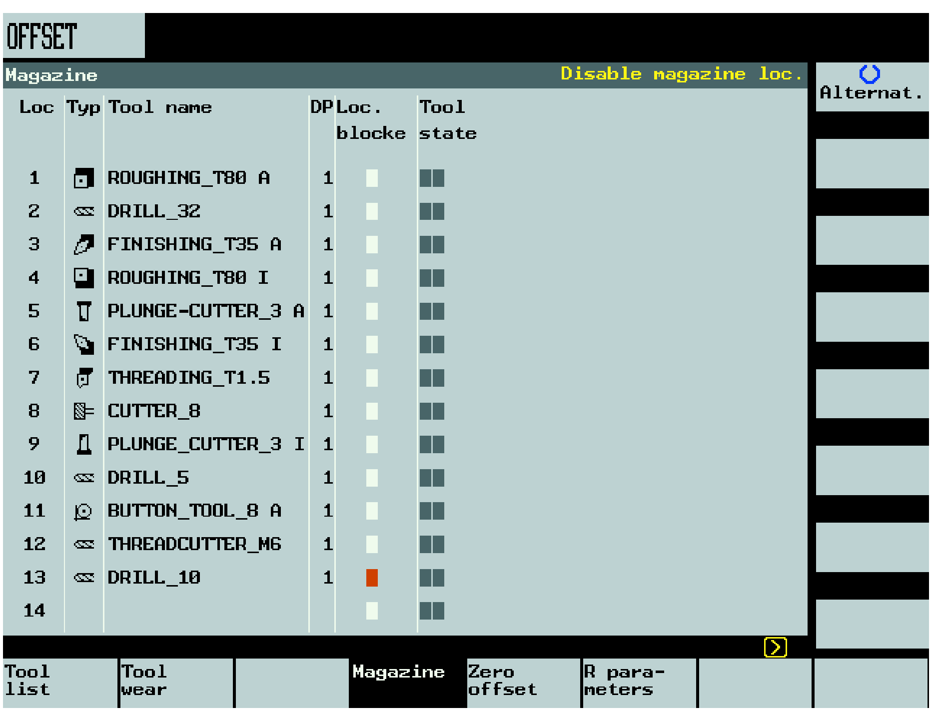
Select a tool shown without location number in the tool list.



The following dialog offers you the first free magazine location. You can change it or accept it directly.

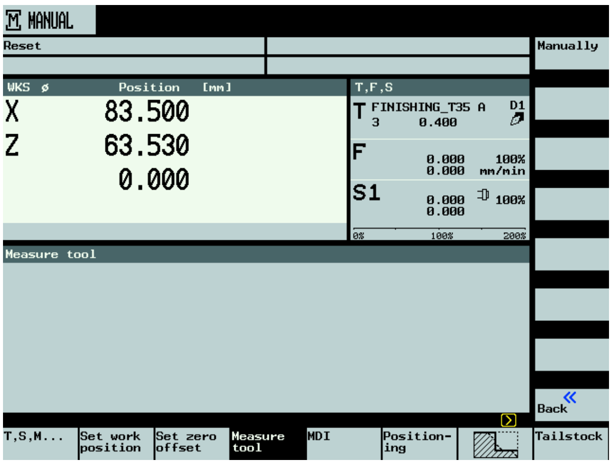


Thus, the magazine for the following exercises could appear as follows.



4.4 Calculate Tool Lengths

To calculate the tool lengths, the mode must be switched to *Manual Machine* in the main menu.



The submenu of the *Measure Tool* option provides two possibilities (*manual* or *zoom*) to calculate the tool.

For example, the tool is now calculated using the function (Manually).

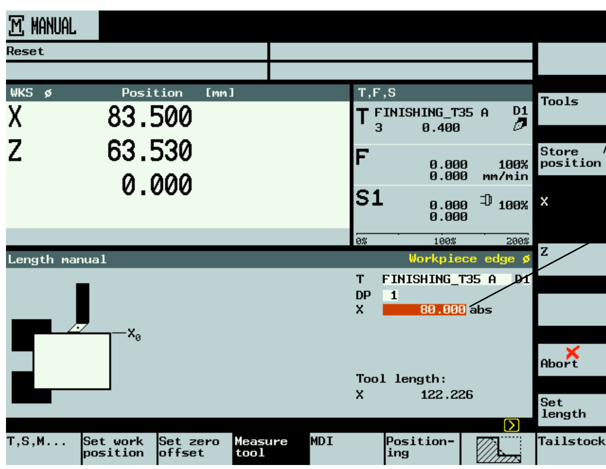
This key marks a position that is then used for the length calculation.

Procedure:

1. Manually

2. Measure the diameter 80.

3. Enter the X-value 80.

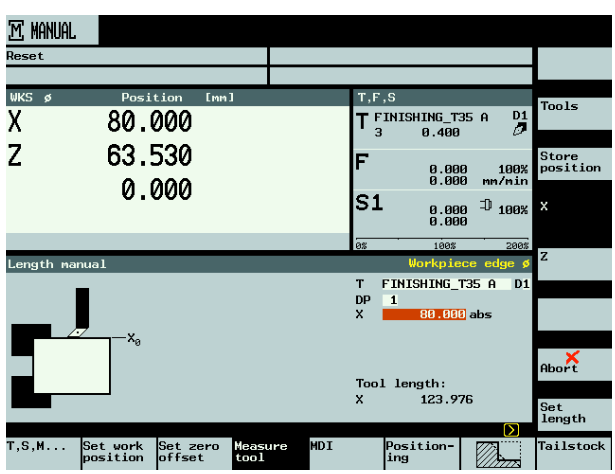


Enter the probed or turned diameter.

4. Set length

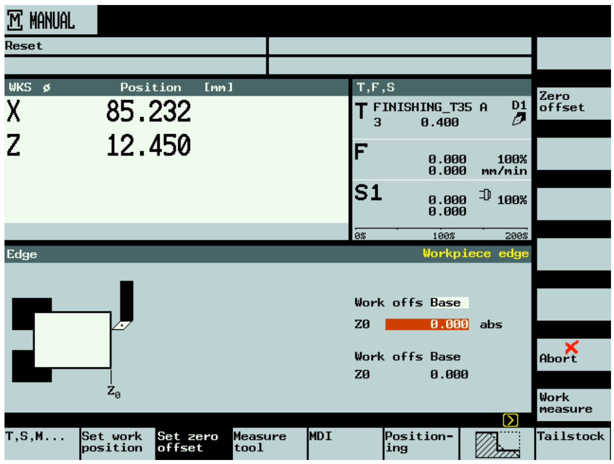
Calculate the workpiece using the diameter.

This calculation action must now be repeated for Z.



4.5 Set the Workpiece Zero Point

To set the workpiece zero point, the mode must be switched to *Manual Machine* in the main menu.



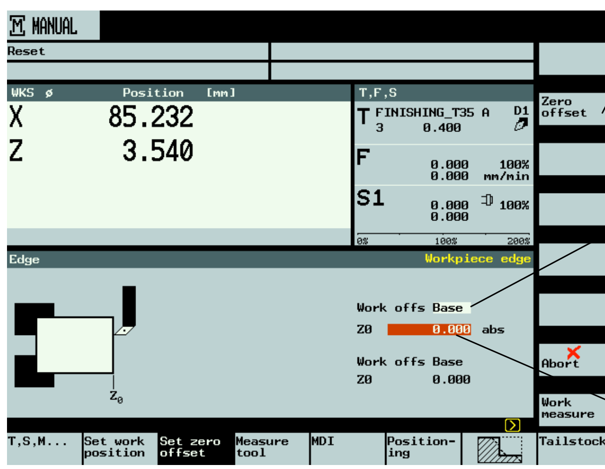
Set the *workpiece zero point* in the submenu of the zero point option.

This key calls the list of the zero point offsets that can then be set in the zero point offset field.

Procedure:

1.Probe the end face.

2. Enter any offset of the workpiece zero point.

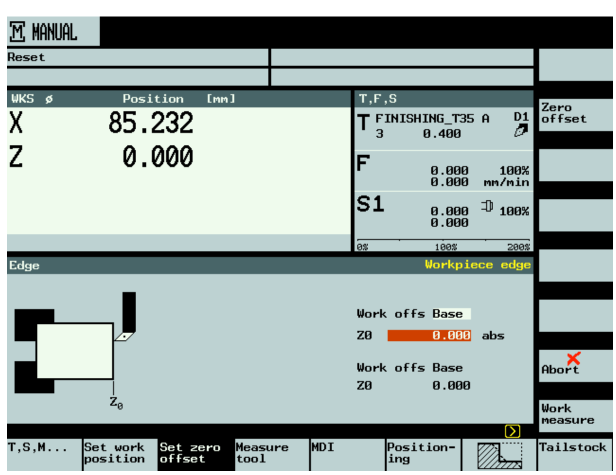


Enter a zero point offset.

The offset of the workpiece zero point if this is not to lie on the end face of the workpiece.

3. Set length

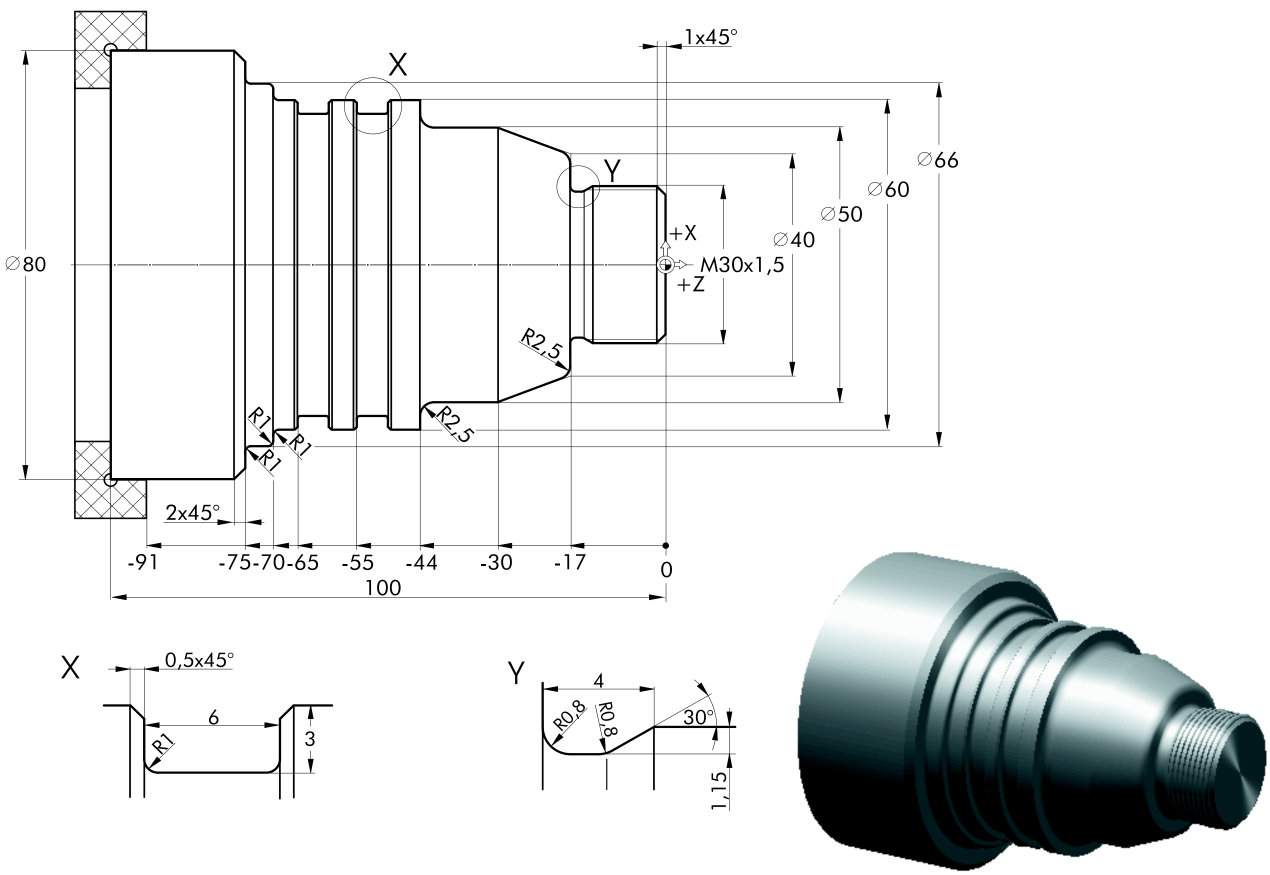
Set the workpiece zero point.



5 Example 1: Taper Shaft

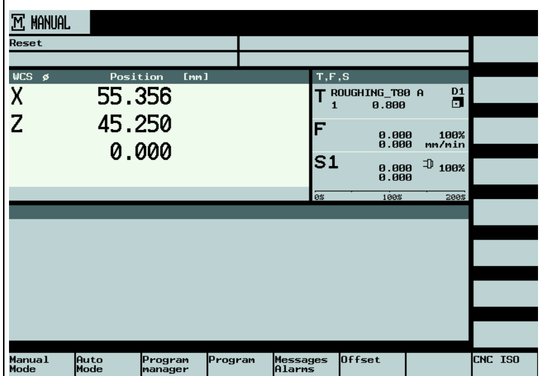
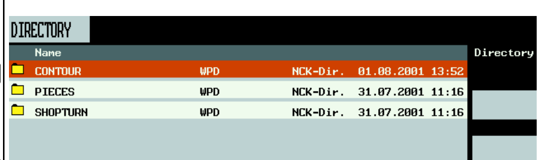
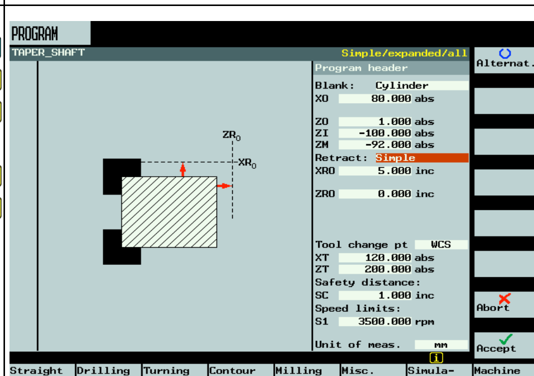
This chapter describes in detail the first steps with ShopTurn:

- Program management and create program
- Invoke tool and enter the traverse path
- Create arbitrary contours using the contour calculator and rough cutting
- Finish
- Thread undercut
- Thread
- Grooves



Note: As ShopTurn always saves the last setting which was set with the key ⟳ or the softkey Alternat. , you must take care that all units, texts and symbols are set the same in all input and switching fields as in the dialog boxes of all the examples shown.
The switching possibility is available when softkey Alternat. is diplayed.

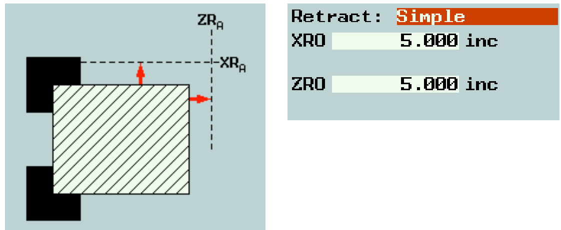
5.1 Program management and create program

Keys		Screen	Explanations
Program manager			• The various areas of ShopTurn can be called in the main menu (refer to Chapter 2). • A list of the existing ShopTurn directories is displayed in the Program Manager.
New	W...		• Create a new directory to separately store the work schedules of the next chapter. Assign it the name "Workpieces".
	...		• The management of the work schedules and contours is organized in the Program Manager (e.g. *New*, *Open*, *Copy* ...). • is used to place the cursor at the WORKPIECES directory; the key is used to open the directory.
New	S...		• Enter here the name of the work schedule, in this case "Taper_Shaft". • is used to accept the name. • The *ShopTurn Program* and *G Code Program* softkeys are used to select the input format.
	80 1 -100 -92		• Enter in the program header the workpieces data and general details about the program. • The key can be used to switch the blank form between *cylinder* and *pipe* . werden. • The ZB value specifies the distance from the chuck. • The key can be used at any time to call help pictures.

The return plane can be switched between *simple*, *extended* and *all*:

Simple

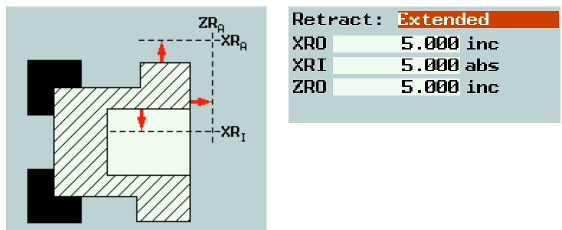
(for simple cylinder)



Enter here the dimensions of the return plane (absolute or incremental) and the tool change point.

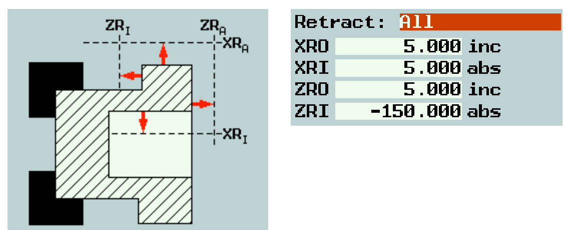
Extended

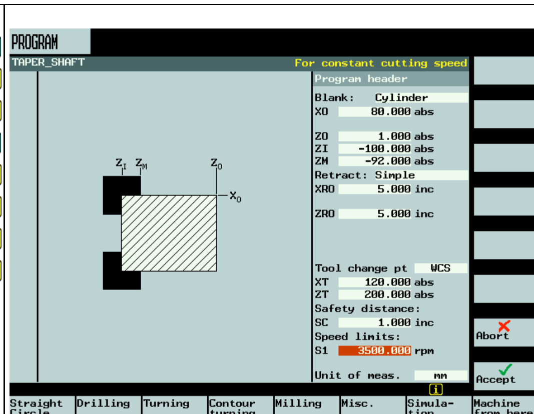
(for complicated workpieces with inside machining)



All

(for most complicated workpieces with inside machining and/or relief cuts)



Accept	5 5 120 200 1 3500		• Enter here the dimensions of the return plane (absolute or incremental) and the tool change point. • The safety clearance and the *speed limit*can also be entered in the program header. • The Accept softkey causes all values of the associated dialog window to be accepted.
			• The created program header is marked with the P symbol. • re-invokes the program header, e.g. to make a change.

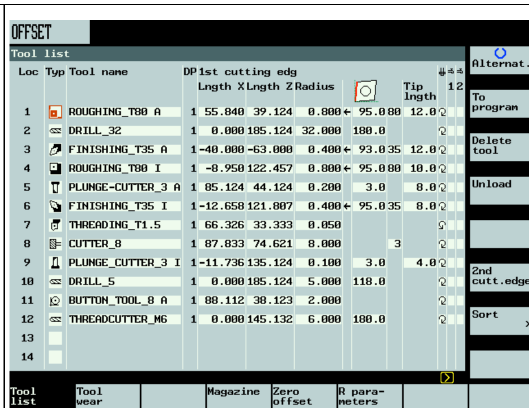
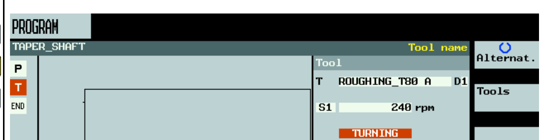
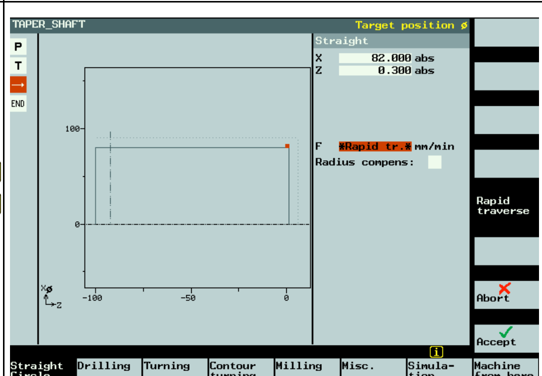
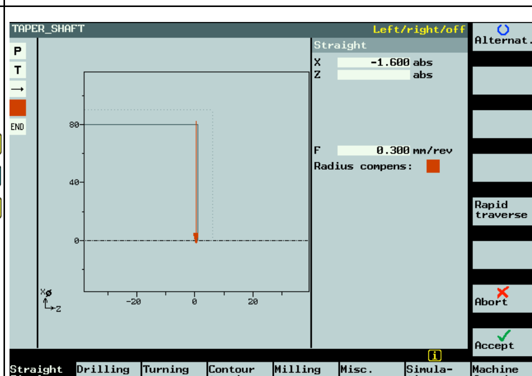
The program to be used as the basis for further machining steps is now created.

It has a name, a program header and a program end (which the "END" symbol represents).

The individual machining steps and contours are stored successively in the program. The subsequent execution is then performed from top to bottom.

5.2 Call Tool and Enter Traverse Path

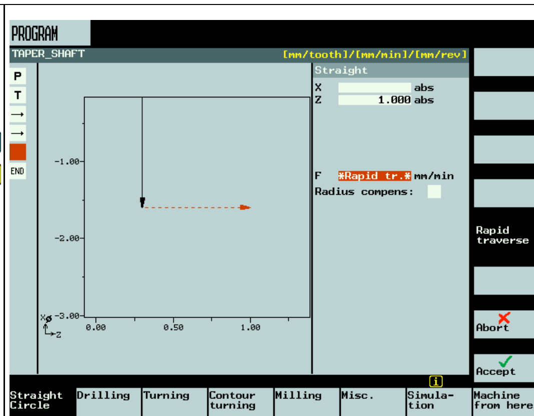
The workpiece is to be faced. You learn here how you can use ShopTurn to create the individual traverse paths.

Tool > Tools To program			• Select and accept the Roughingtool_80_A in the *tool list*. • Because the cursor has already been placed on the tool, the To program softkey can use it directly in the tool call.
Accept	240		• After making the tool selection, *S1* is used to select the main spindle and 240 m/min entered for the cutting rate. • The *S2* spindle is the tool spindle used for driven tools. • The key switches between the *Sheath/Front/Turn* functions and the Turn *machining*.
Straight > Rapid traverse Accept	82 0.3		• The workpiece is faced in two steps. The starting point for rough cutting is entered first (X82 und Z0.3).
Straight > Accept	-1.6 0.3		• The tool has a 0.8 radius. This means it must be traversed to diameter X -1.6. 

Straight >

Rapid traverse

1



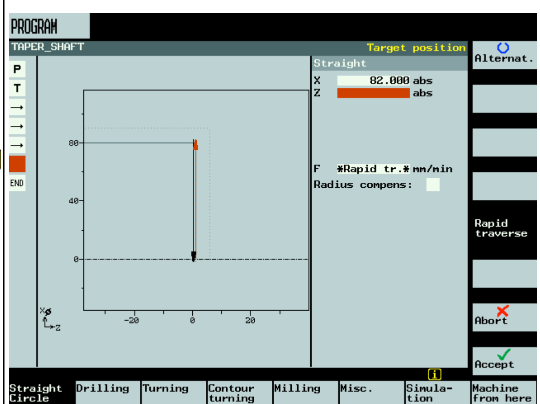
Accept

- Retract the tool in rapid traverse from the end face.

Straight >

Rapid traverse

82



Accept

- Traverse the tool back to the start diameter.

Straight >

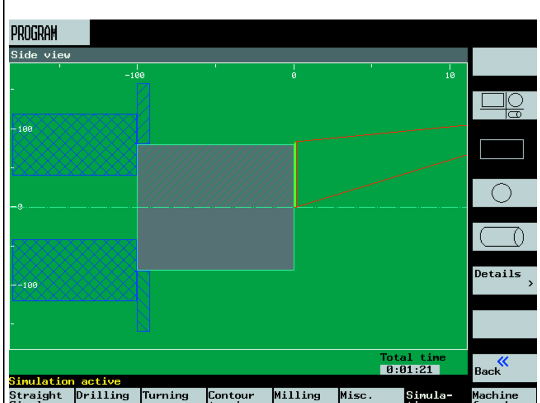
•••

...

```
TAPER_SHAFT
P    N0   TAPER_SHAFT
T    N5   TURNING                 T=ROUGHING_T80 A S1=240
→    N10    RAPID X82 Z0.3
→    N15    F0.3/rev X-1.6
→    N20    RAPID Z1
→    N25    RAPID X82
→    N30    RAPID Z0
→    N35    F0.25/rev X-1.6
→    N40    RAPID Z1
→    N45    RAPID X120 Z200
END       Program end
```

- As an exercise, you should now create yourself the four traverse paths shown with a red frame.

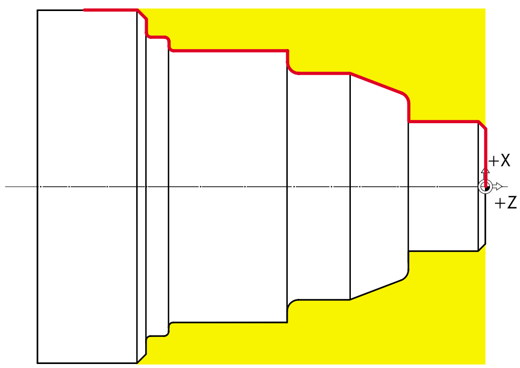
Simula- tion



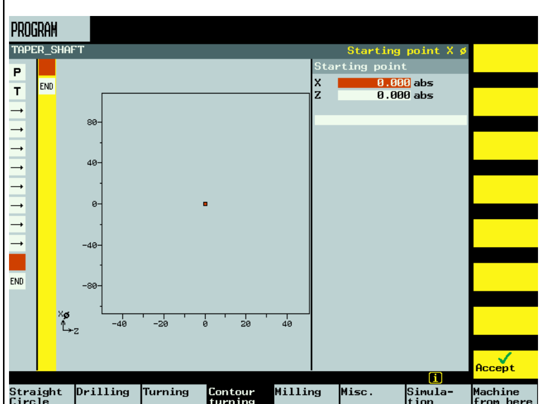
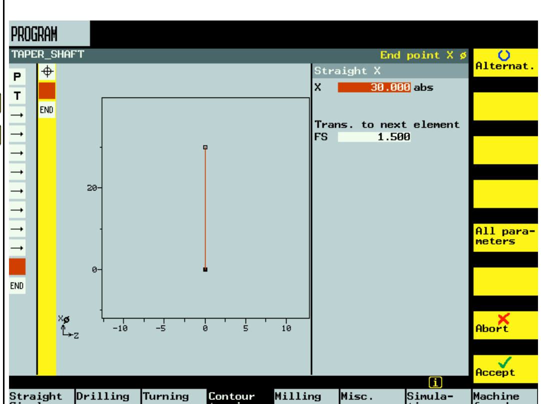
Simula- tion

- Start simulation with the Simula- tion softkey.
- The simulation can also be invoked in the next examples even when it is not expressly shown.
- Further information about simulation is available at the end of this chapter.
- Simula- tion, Back or any softkey from the horizontal softkey menu is used to exit the simulation.

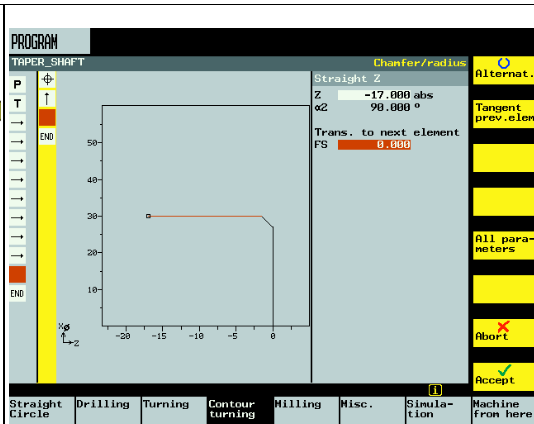
5.3 Create Arbitrary Contours Using the Contour Calculator and Rough



In the following section, use the contour calculator to create the workpiece contour shown in red (path/arc-pictograms). Then rough cut against the contour and finish.

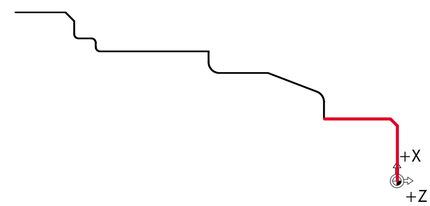
Contour turning New contour >	K ...		• The new contours can be created in the Turn Contour submenu. This requires that a contour name is assigned, in this case TAPER_SHAFT_CONTOUR.
Accept			• The *starting point* of the contour definition can be accepted without change. • Note: The contour definition serves both as the limit for rough cutting and the finishing path.
Accept	30 1.5		• The first contour definition is a vertical path to the end point X30. • The *chamfer (FS)* is appended directly to the straight line as transition element. The key or *Alternat.* switches the transition between *chamfer* and *radius*.

-17

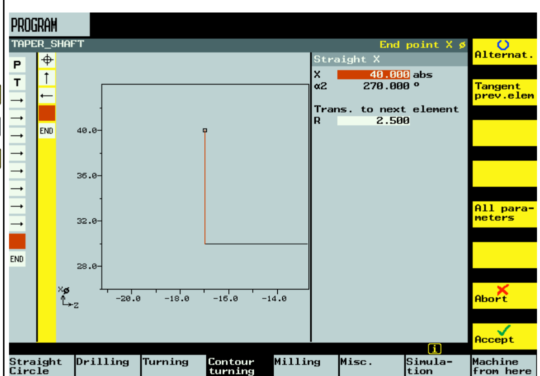


Accept

• A straight line to Z-17 follows.
The thread undercut will be added later as a single element.

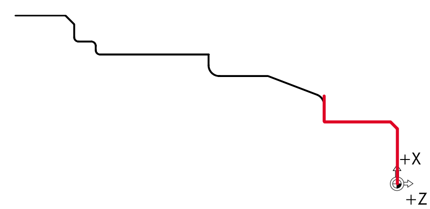


40
2.5

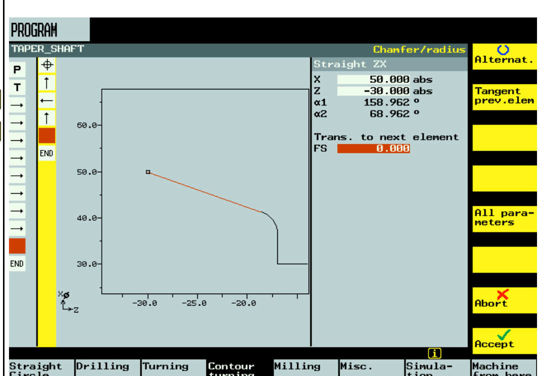


Accept

• Construct the vertical path to the dimensioned intersection including the rounding to the following element.

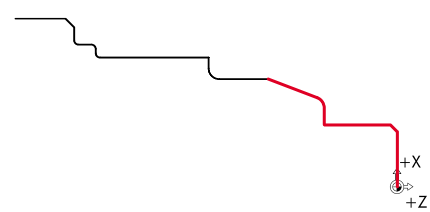


50
-30

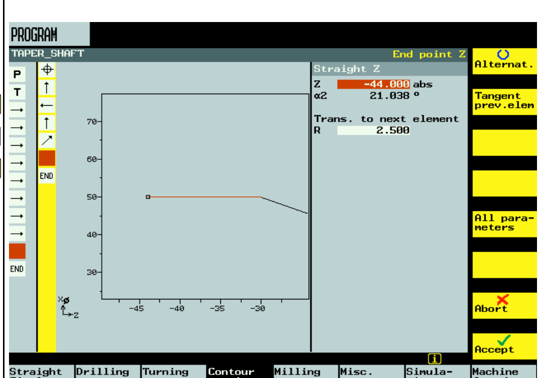


Accept

• The end point of the slope is at X50 and Z-30.

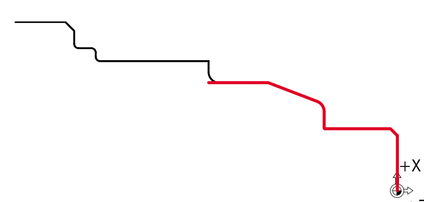


-44
2.5



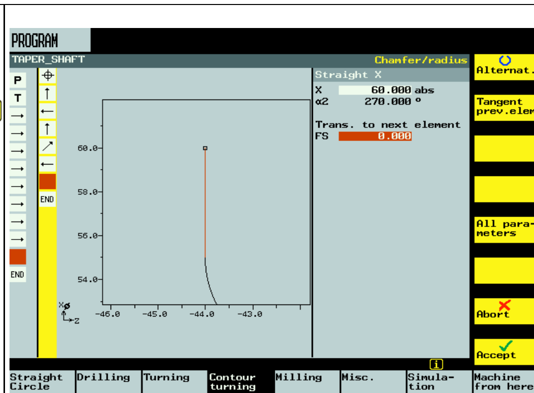
Accept

• A horizontal path to Z-44 with a radial transition (R2.5) to the next element follows.

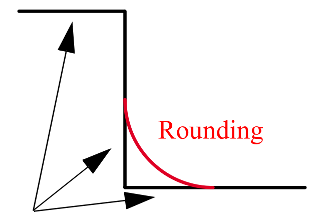


60

Accept



- A path with the end point X60 follows.
- Caution: The paths (= main elements) do not meet **tangentially.**.

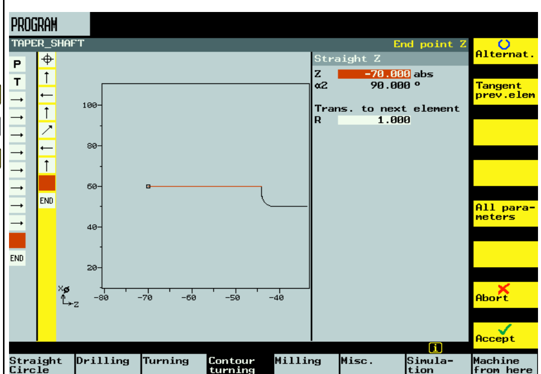


3 Main elements

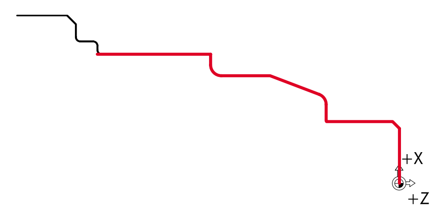
-70

1

Accept



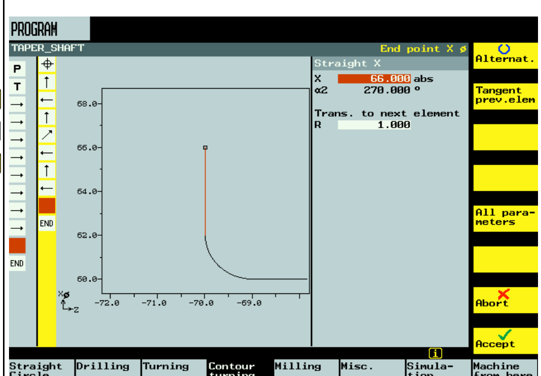
- The grooves are entered later, the same as the thread undercut, as single elements and so are not considered here.



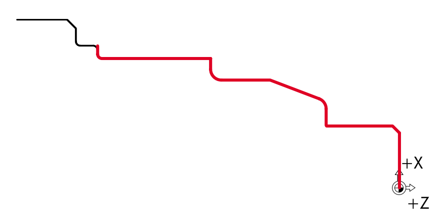
66

1

Accept



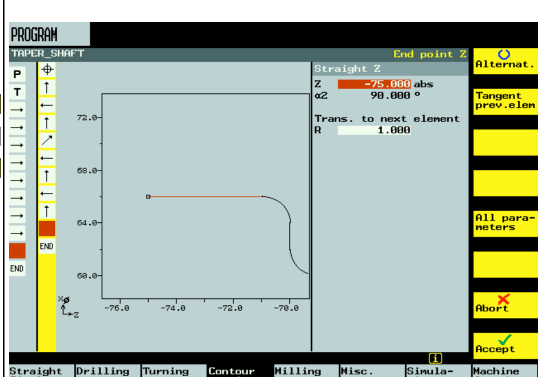
- A vertical path to X66 with a radial transition (R1) to the next element follows.



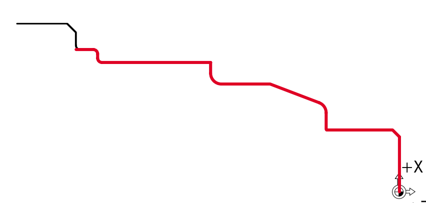
-75

1

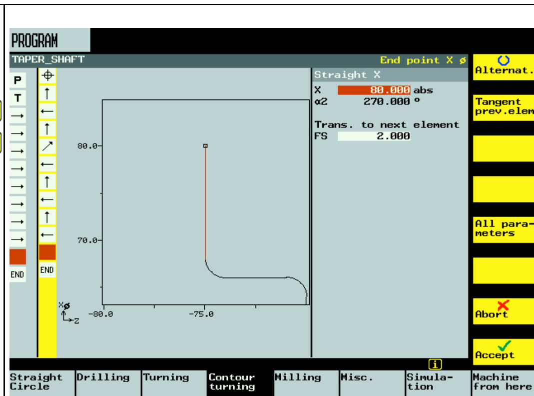
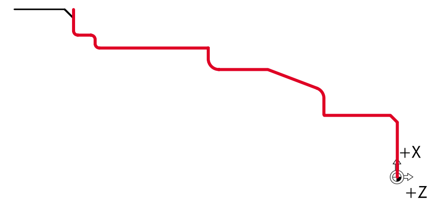
Accept

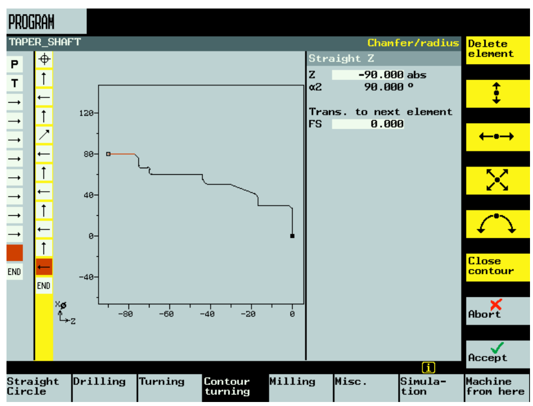
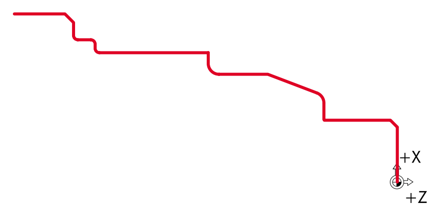


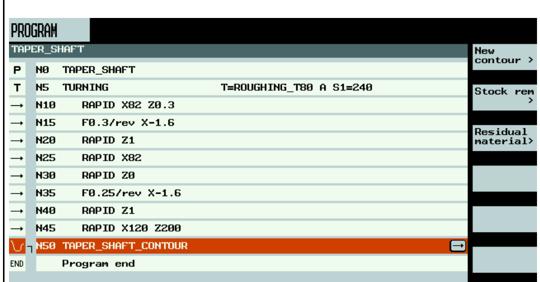
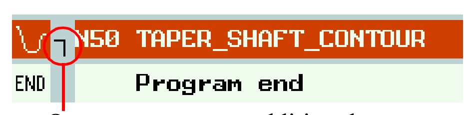
- End point Z-75 with radial transition R1

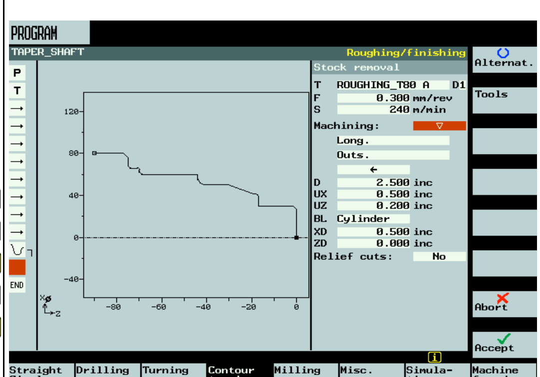


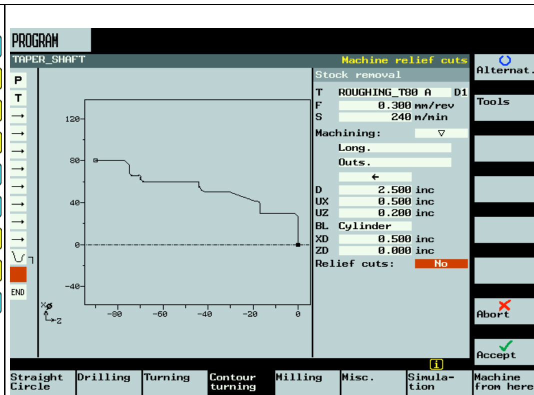
<table>
<tr>
<td>↕

Accept</td>
<td>80 ⇨
2 ⇨</td>
<td>
</td>
<td>• End point X80 with a 2x45° chamfer

</td>
</tr>
<tr>
<td>←•→

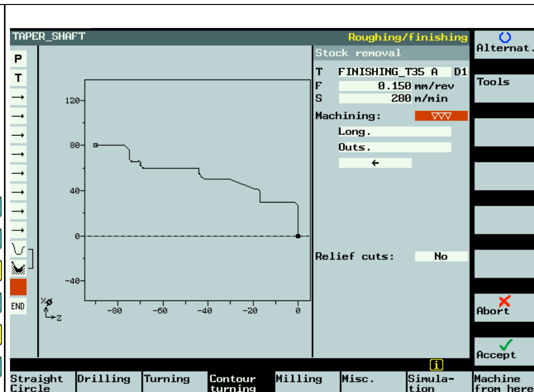
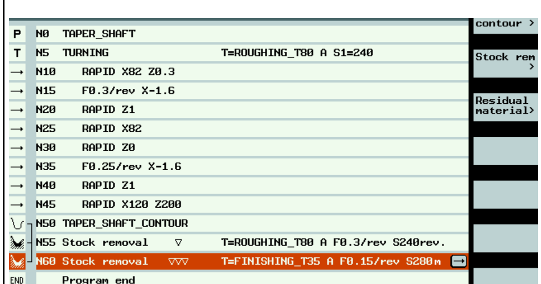
Accept
Accept</td>
<td>-90 ⇨</td>
<td>
</td>
<td>• The contour end point lies at X80 and Z-90 (2mm in front of the chuck).


• The contour is accepted in the work schedule.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>
</td>
<td>• The line open at the bottom is used to connect this contour with additional contours or machining steps.


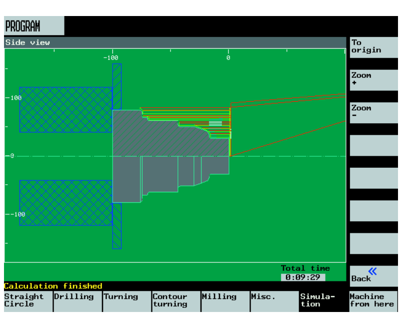
Open connector to additional contours or machining steps</td>
</tr>
<tr>
<td>Stock rem >
Tools
To program</td>
<td>↓
↻
0.3 ⇨
↻
240 ⇨</td>
<td>
</td>
<td>• Cut the contour with a feed of 0.3 mm/rev and a cutting rate of 240m/min.
• Rough cut against the contour in the first machining step (▽) .
</td>
</tr>
</table>

Accept	4x ↓ 2.5 0.5 0.2 ↓ 0 0		• Enter here the *cutting direction*, the *external machining*, the *machining direction*, the *infeed depth* and the *final machining* allowance. • The description of the *blank* is also selected here (*cylinder, allowance, contour*). • Because this contour does not have any relief cuts, the *Relief Cut* field can remain as *no*.

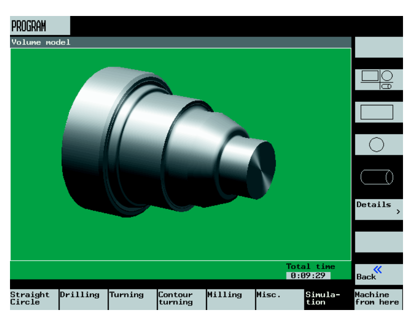
5.4 Finishing

Stock rem > Tools To program Accept	↓ 0.15 280		• The FINISHINGTOOL_35_A tool is used to finish the contour. This requires the tool to be loaded from the magazine. • After modifying the technological data, switch the machining to finishing (▽▽▽).
			• After accepting the machining step, the work schedule should now look like this. • Now check the work schedule by simulating it.

Simula-tion

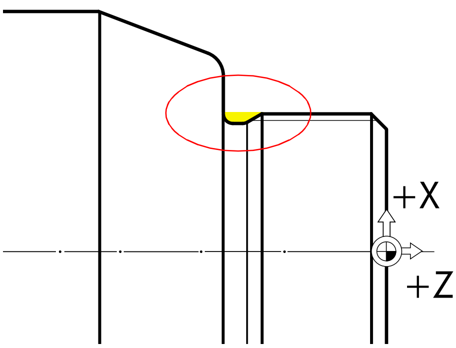


Further information about these variations of the workpiece display can be found at the end of this chapter.

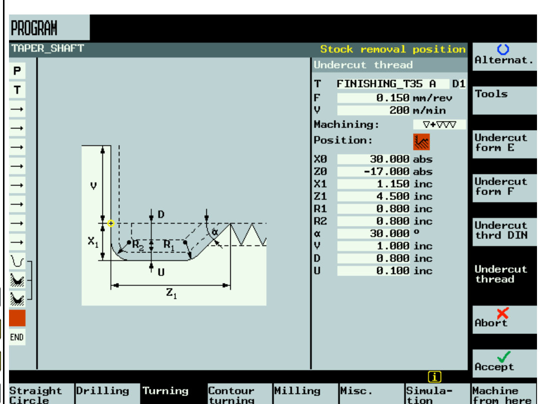
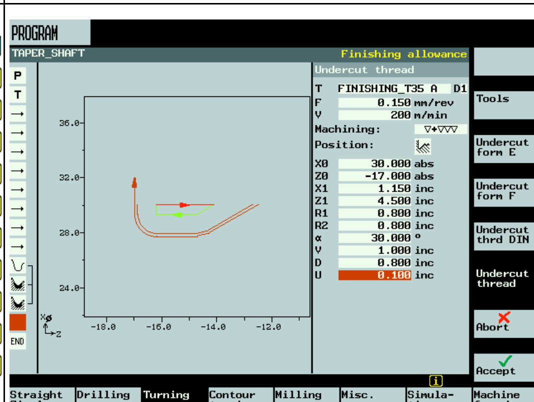
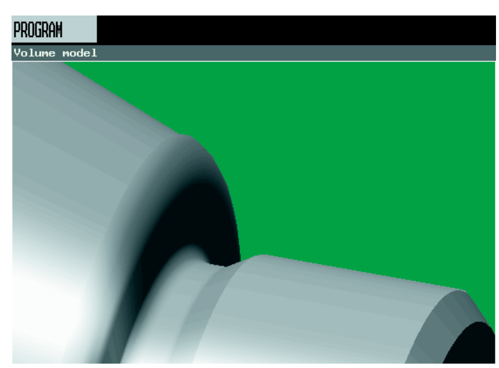


Simula-tion

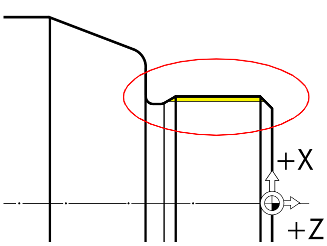
5.5 Thread Undercut



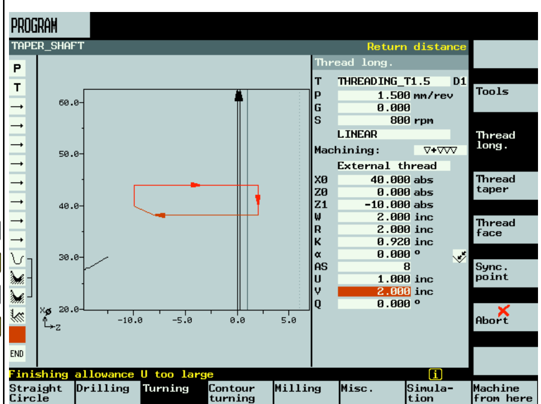
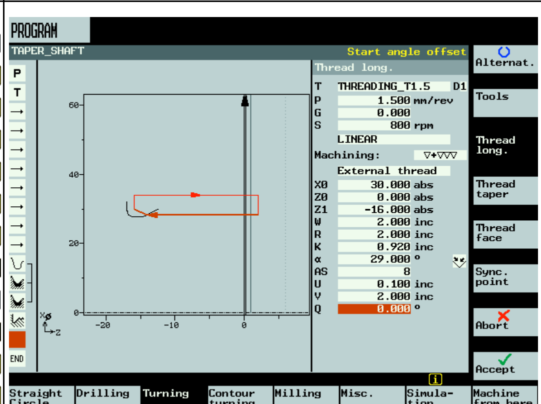
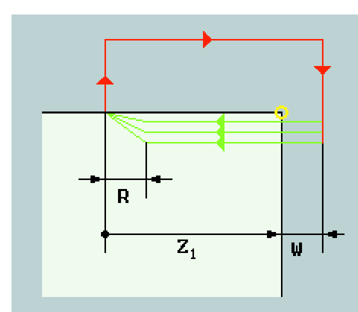
The thread undercut with previously swung in finishing tool is created in this section.

Keys		Screen	Explanations
Turning Undercut Undercut thread Tools To program	↓ 0.15 ⇔ 200 ⇔ 2x ◡ ↓ ◡		• If the finishing tool has not yet been selected in the machining step, it must be used now. • Enter the technological data. Also change to *Rough Cutting / Finishing* machining. • Select the undercut position.
Accept	↓ 30 ⇔ -17 ⇔ 1.15 ⇔ 4.5 ⇔ 0.8 ⇔ 0.8 ⇔ 30 ⇔ 1 ⇔ 0.8 ⇔ 0.1 ⇔		• Make the following inputs to geometrically define the undercut. 

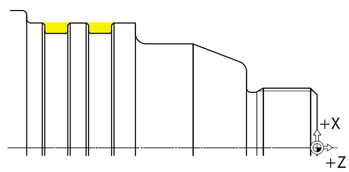
5.6 Threads



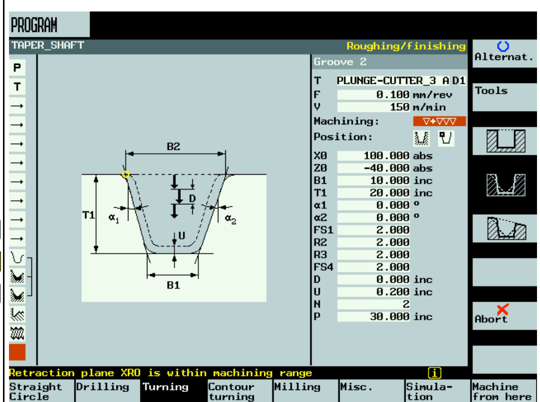
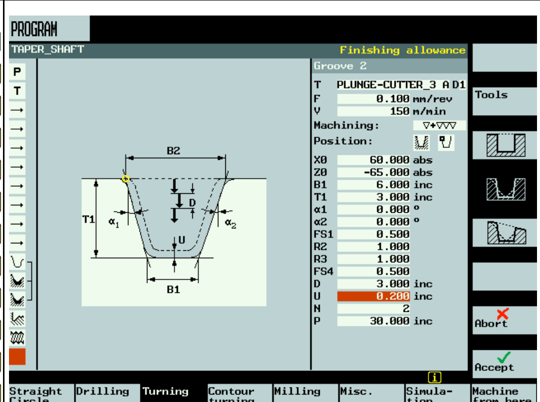
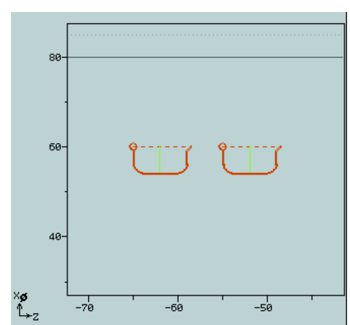
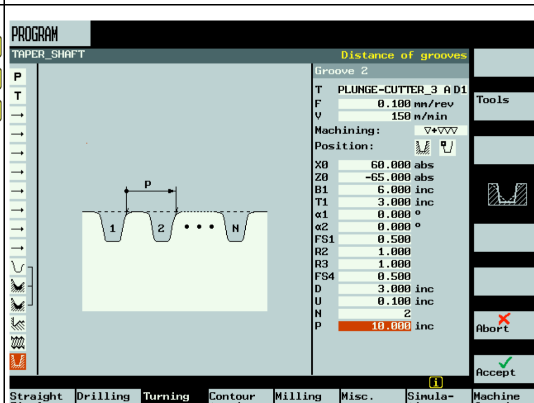
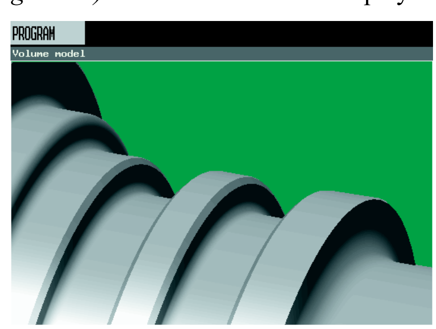
The thread is created in this section.

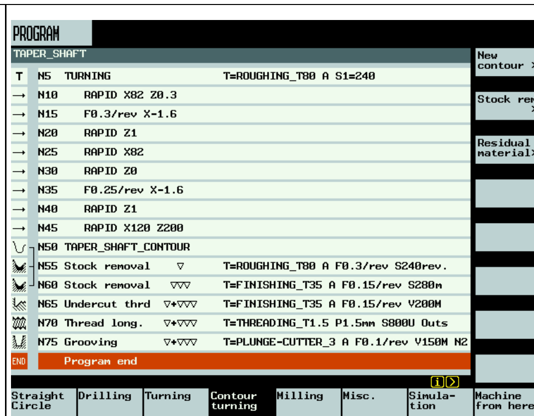
Keys		Screen	Explanations
Thread > Tools To program	↓ 1.5 ⇨ ↓ 800 ⇨		• Use the thread cutting tool in the machining step. • The following inputs can be made in the *P* input field: 1. Thread pitch in mm/rev 2. Thread pitch in inch/rev 3. Cycles / inch 4. Module
Accept	3x ↓ 30 ⇨ 0 ⇨ -16 ⇨ 2 ⇨ 1 ⇨ 0.92 ⇨ → ○ ← 29 ⇨ 8 ⇨ 0.1 ⇨ 0 ⇨		• Make the following inputs to geometrically define the thread. 
			• This "photograph" of a virtual production (and also the "photographs" on pages 33, 87 and 98) are taken from the ShopTurn multimedia CD-*ROM*.

5.7 Grooves



The two grooves are created in this section.

Keys		Screen	Explanations
Groove >  Tools To program	0.1 ↓ 150 ⇒ 2x ↻		• PLUNGECUTTER_3_A is required to create the grooves. • The (i) key is used to call pictures to help.
	2x ↓ 60 ⇒ -65 ⇒ 6 ⇒ 3 ⇒ 0 ⇒ 0 ⇒ 0.5 ⇒ 1 ⇒ 1 ⇒ 0.5 ⇒ 3 ⇒		• Make the following inputs to geometrically define the two grooves. 
Accept	0.1 ⇒ 2 ⇒ 10 ⇒		• If the value 1 is entered as the number of grooves *N*, the *P field (distance between the grooves)* is removed from the display. 

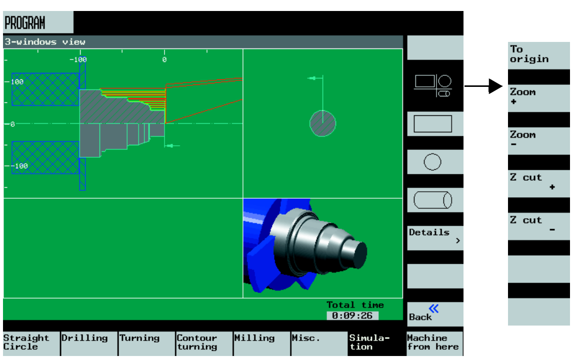


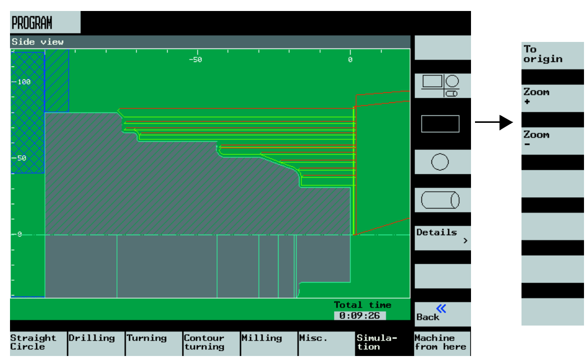
• The complete work schedule should now look like this.

Further information about the display of the workpiece:
The simulation can run in the *3-window-view,* in the *side view* or in the *end face* view.
The workpiece can subsequently by viewed as a *volume model* in a 3D view.

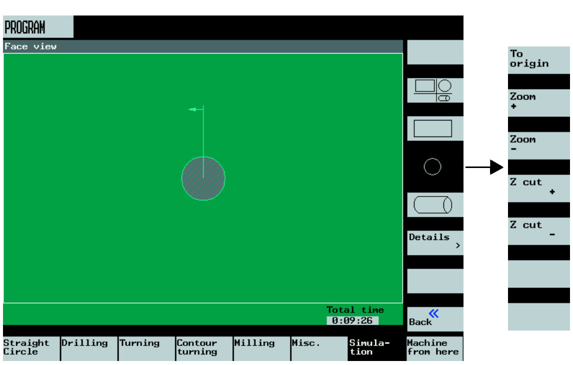
Simula- tion

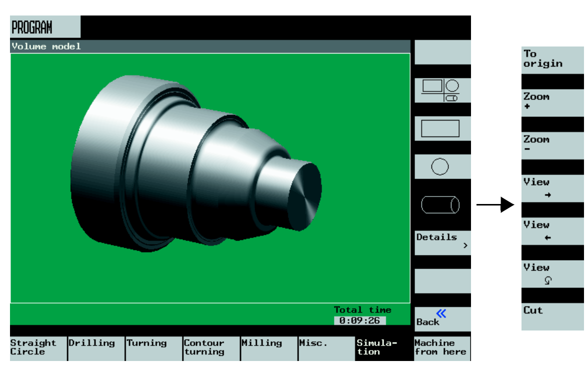
The [3-window key], [side view key], [face view key] or [volume model key] key can be used during the simulation to switch to the other representation.





If the **Details >** key is pressed in the various views, further submenus appear to modify the displays (e.g. *Zoom+* or *Cuts*).



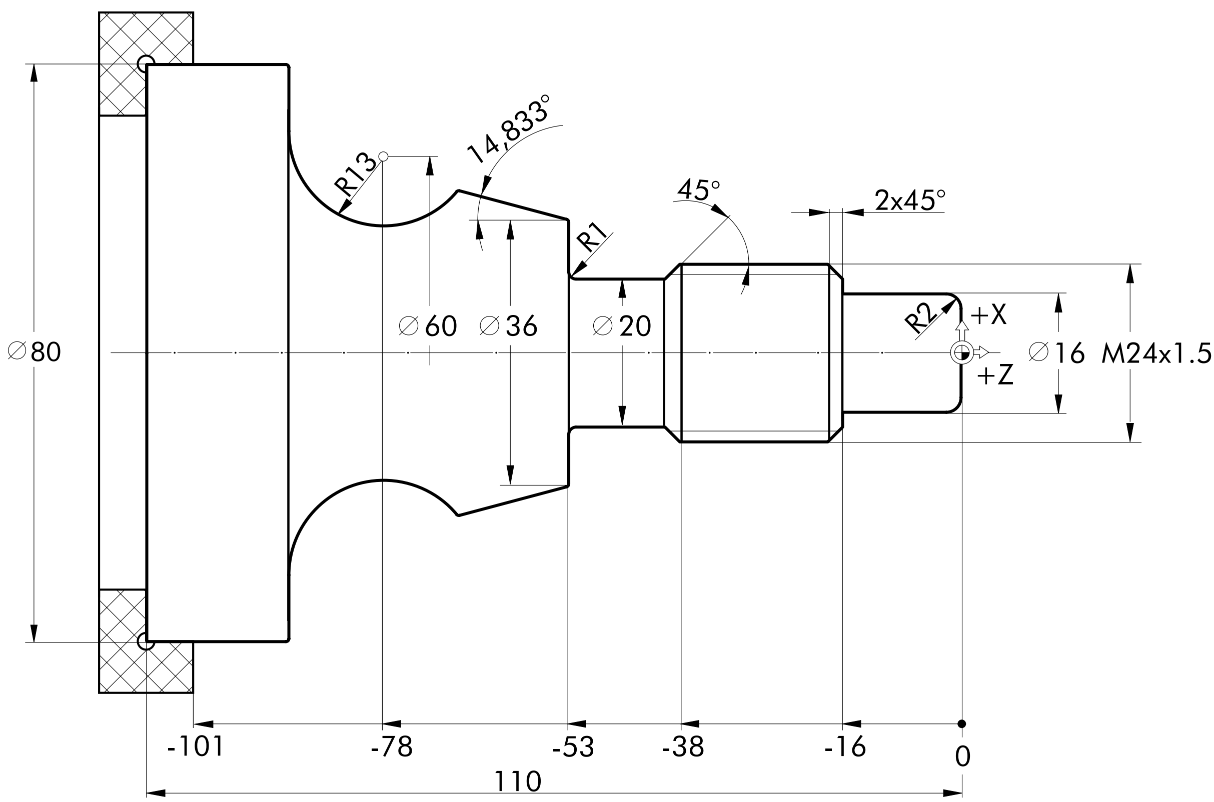


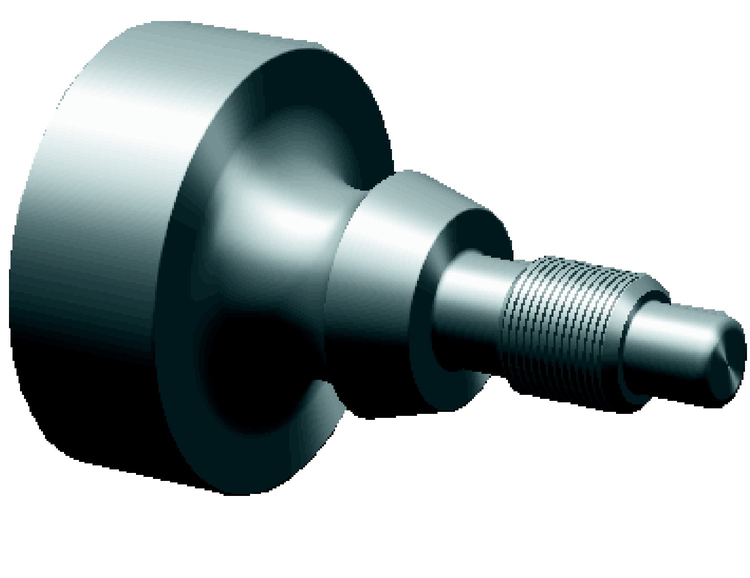
Simula- tion

6 Example 2: Drive Shaft

This chapter familiarizes you with the following new functions:

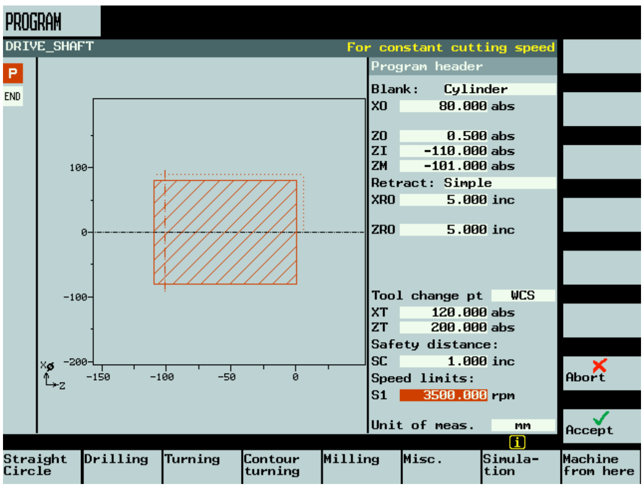
- Facing
- Extended use of the contour calculator
- Machining of residual material



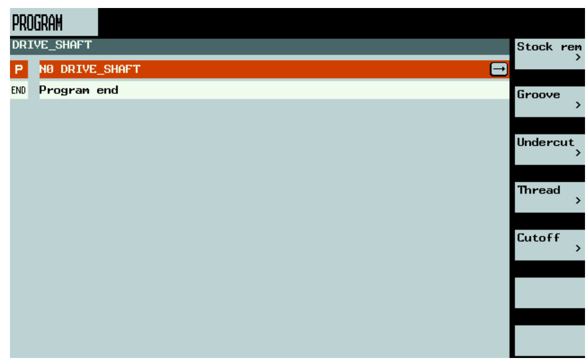


Create work schedule

First create a new work schedule with the name "Drive shaft"; the dimensions of the blank are entered at the same time (refer to the "Step Shaft" chapter for the procedure involved).



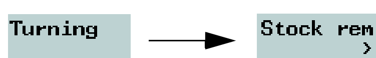
Once the program header has been created, the work schedule should look like this.

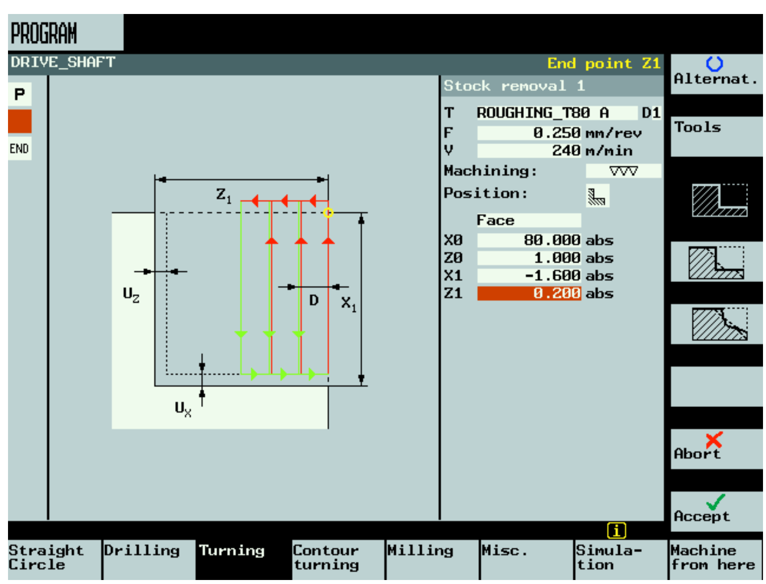


6.1 Facing

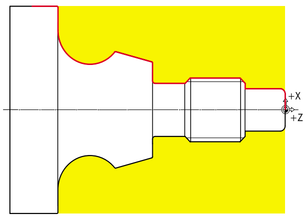
The workpiece should now be faced. Select *Turning machining* in the main menu and *Cutting* in the submenu.

Because the facing should be performed in a single step, the machining should be switched to finishing.



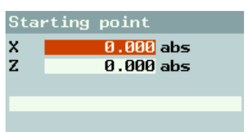
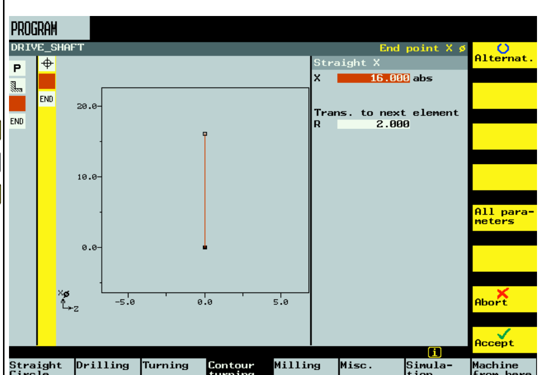
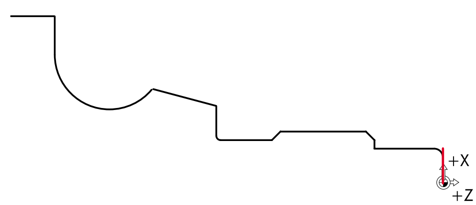
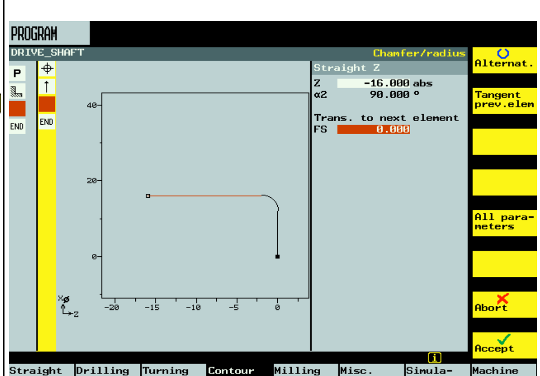
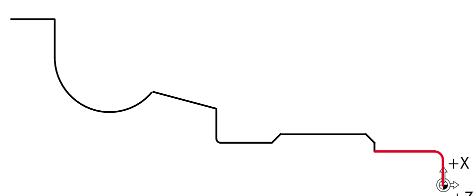


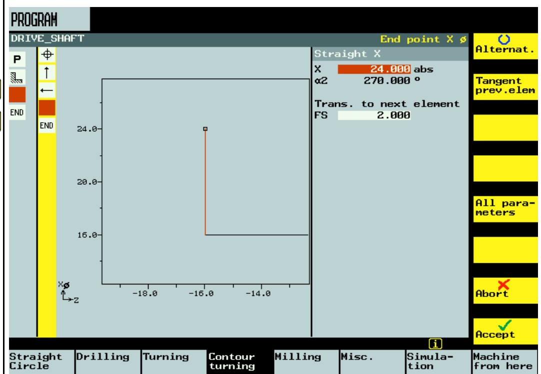
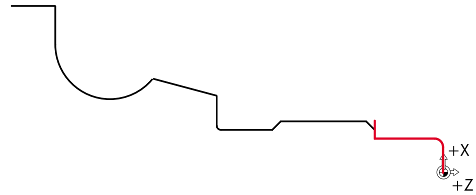
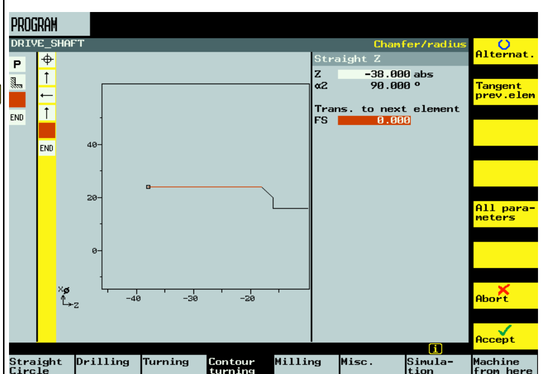
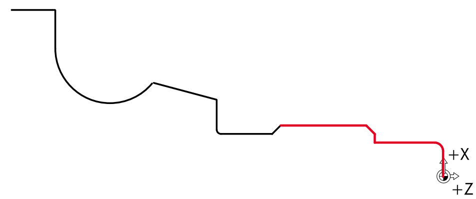
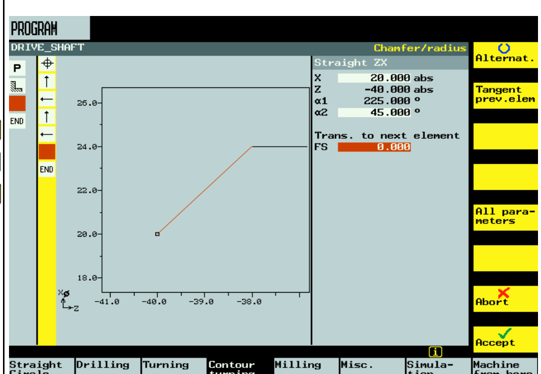
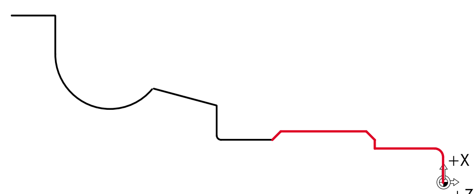
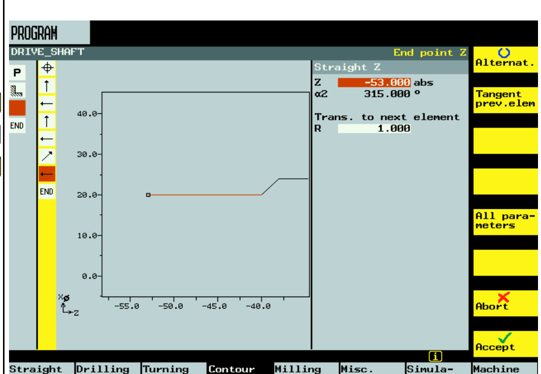
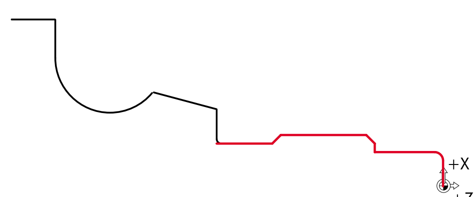
6.2 Create the Contour, Cutting and Residual Cutting

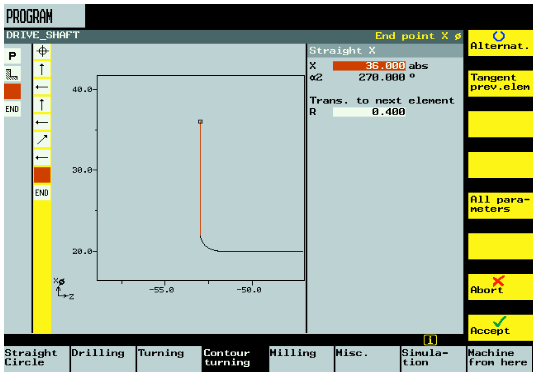
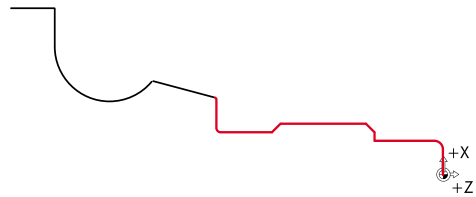
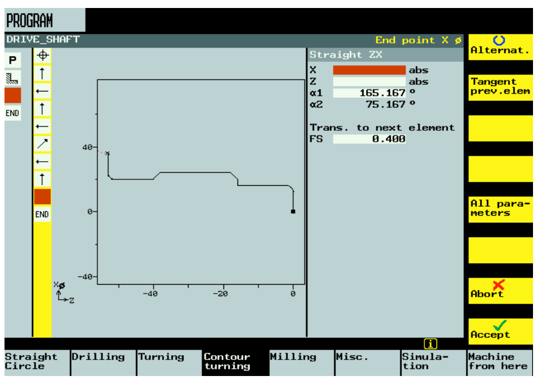
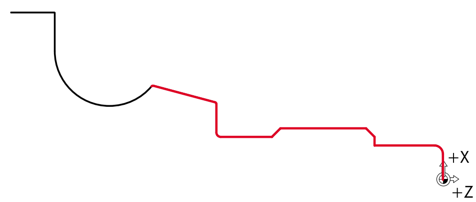
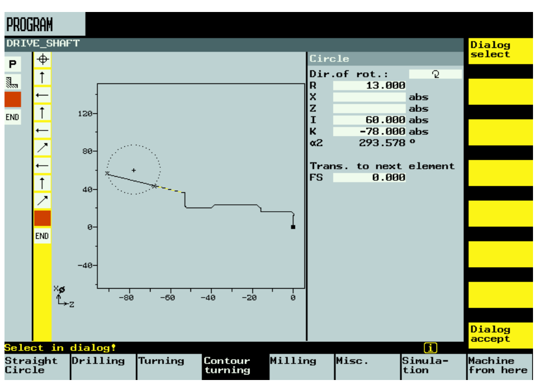
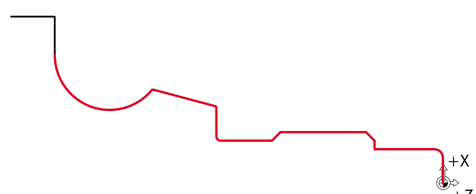
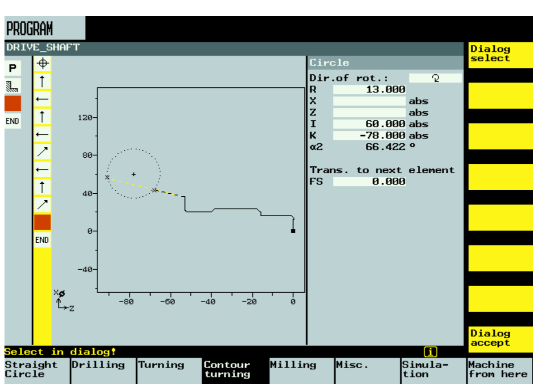
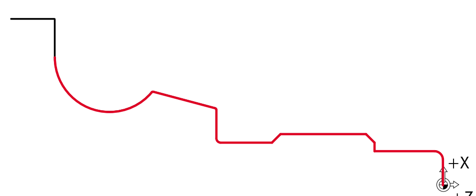


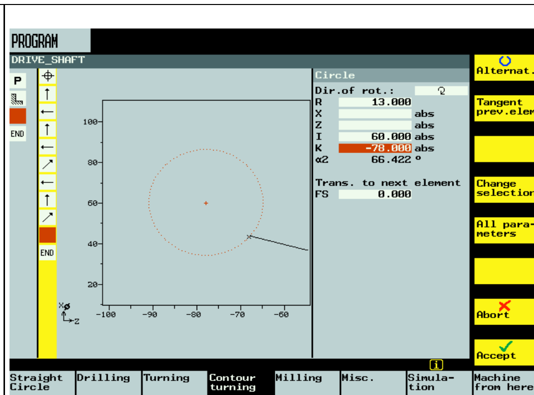
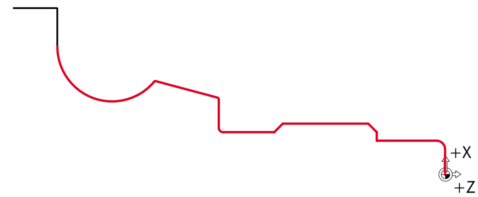
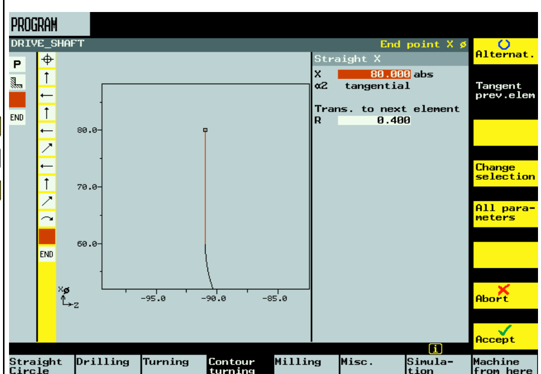
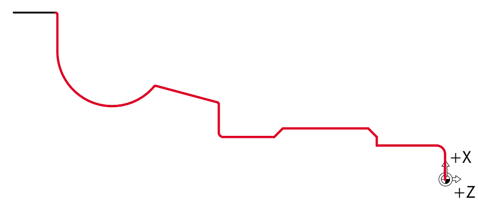
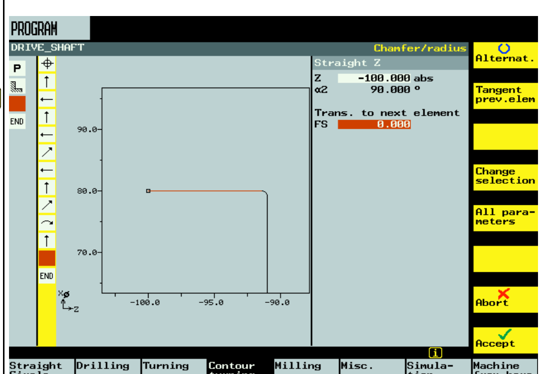
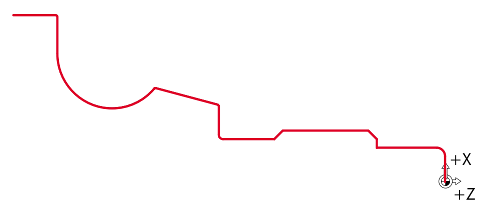
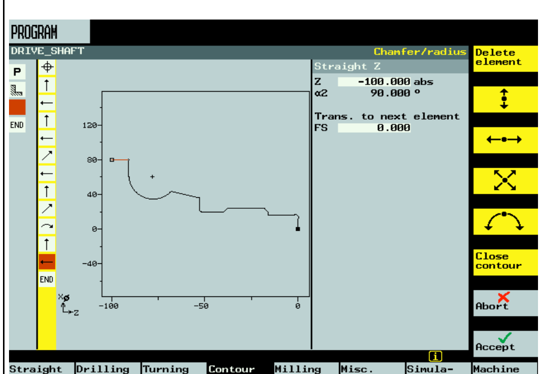
In the following section, the contour is created, turned longitudinal with an 80° insert, and then the residual material rough cut using a pointed tool.

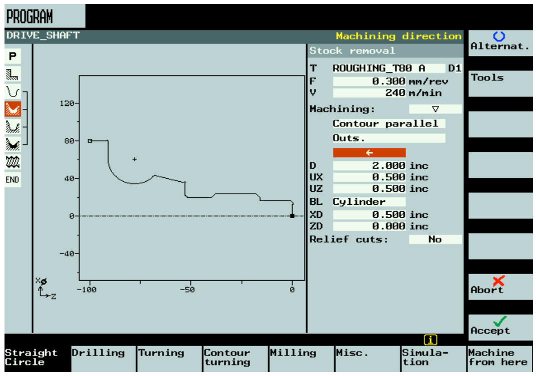
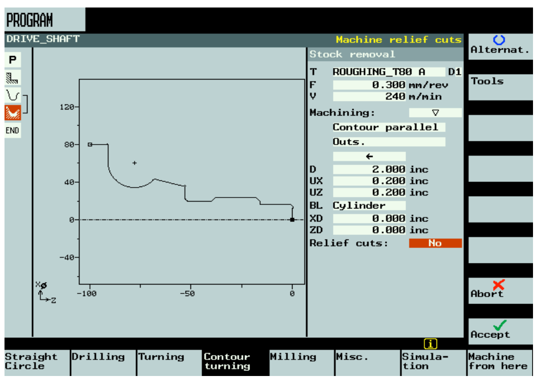
It is then finished. Finally, the thread is tapped.

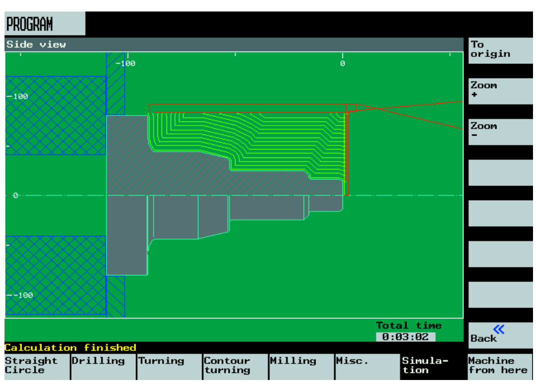
<table>
<tr><th colspan="2">Keys</th><th>Screen</th><th>Explanations</th></tr>
<tr><td>Contour turning
New contour ></td><td>K...</td><td>
</td><td>• Assign the name DRIVE_SHAFT_Contour to the contour.</td></tr>
<tr><td>Accept</td><td></td><td>
</td><td>• The starting point XO/ZO is accepted directly.</td></tr>
<tr><td>Accept</td><td>16
2</td><td>
</td><td>• The contour begins with a vertical path to X16 and a radius 2 as transition element.

</td></tr>
<tr><td>Accept</td><td>-16</td><td>
</td><td>• A horizontal path follows.

</td></tr>
</table>

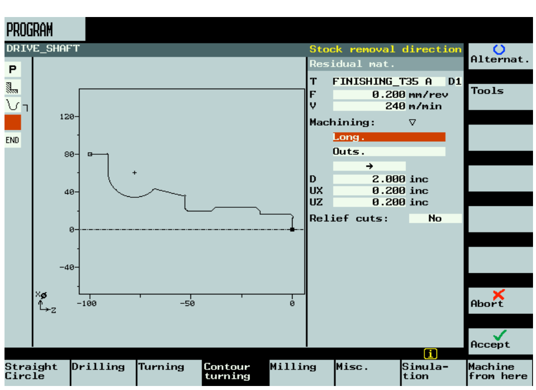
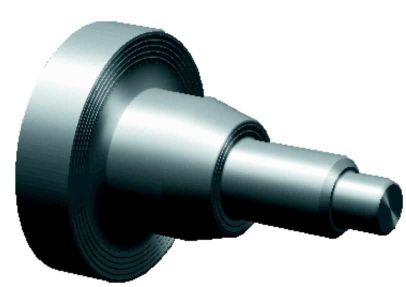
 Accept	24 2		• A vertical path with a chamfer as transition to the next element follows. 
 Accept	-38		• A horizontal path to Z-38 follows. 
 Accept	20 2x 45		• A falling path to X20 follows. The entered Alpha 2 angle is relative to the previous element (refer to Chapter 3). 
 Accept	-53 1		• A horizontal path with radius 1 as transition to the *next element follows*. 

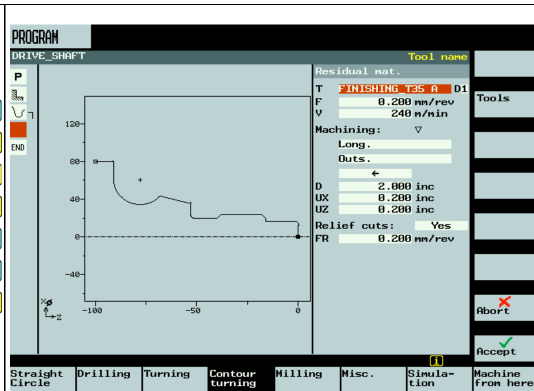
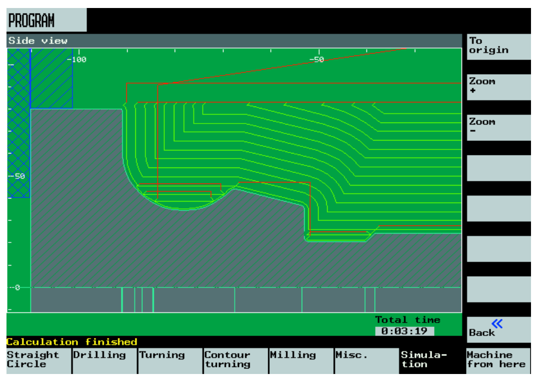
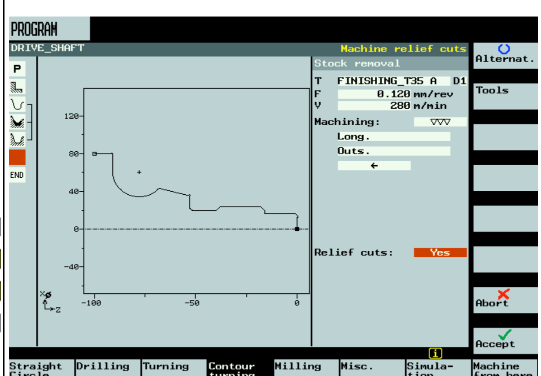
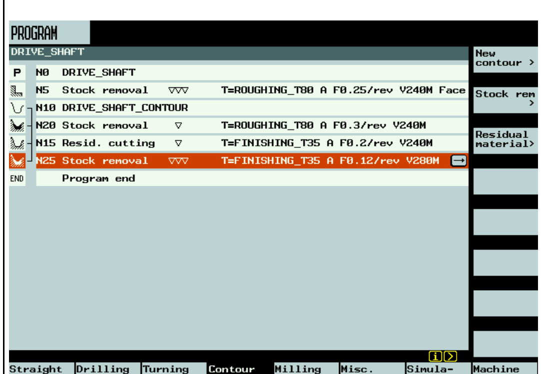
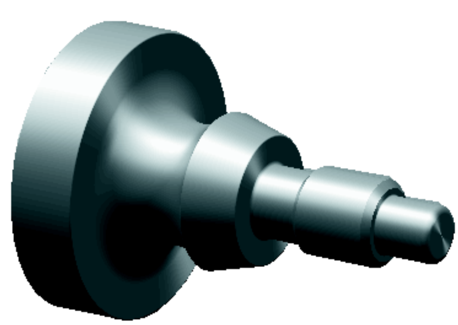
 Accept	36 0.4		• A path to a diameter X36 follows. • The transition to the next element is rounded with R0.4. 
 Accept	2x ... 0.4		• The only information known about the path is the angle to the Z-axis, 165.167°. In such cases simply continue the construction with the next element. 
 Dialog select	13 2x 60 -78		• The known dimensions of the arc allow the missing points of the previous contour element to be calculated. • If several possibilities are present, the correct one must be selected first. 
Dialog accept			• Once the required construction has been selected, it can be accepted. 

Accept			• Because the end point of the arc is not known, the construction is simply continued. • The *All Parameters* function could also be used here to enter the exit angle. 
↕ Tangent prev.elem Accept	80 0.4		• A tangential path follows. • The transition to the next element is rounded with a radius 0.4. 
←•→ Accept	-100		• The end point of the contour lies at Z-100. 
Accept			• Transfer the complete contour to the work schedule.

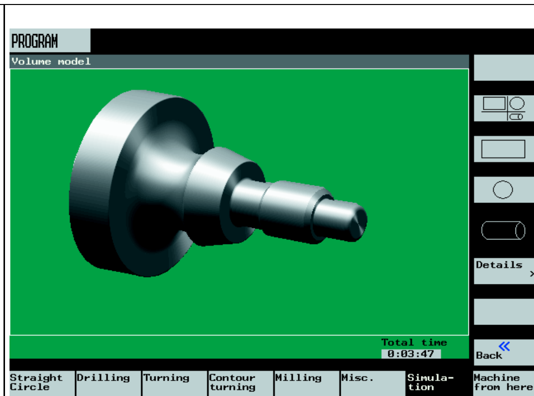
<table>
<tr>
<td>Stock rem >
Tools
To program</td>
<td>↓
0.3
240
...
↓
...
↓
...
↓
...</td>
<td>
</td>
<td>• To cut the contour, the ROUGHCUTTERTOOL_80_A must be loaded into the machining step.</td>
</tr>
<tr>
<td>Accept</td>
<td>↓
2
0.2
0.2
↓
0
0</td>
<td>
</td>
<td>• The machining of the contour, for example, is performed parallel to the contour here.</td>
</tr>
<tr>
<td>Simula- tion
Details >
Zoom +

Simula- tion</td>
<td></td>
<td>
</td>
<td>• The Zoom+ and Zoom- softkeys can be used to enlarge or reduce the simulation.</td>
</tr>
<tr>
<td>Contour turning
Residual material>
Tools
To program

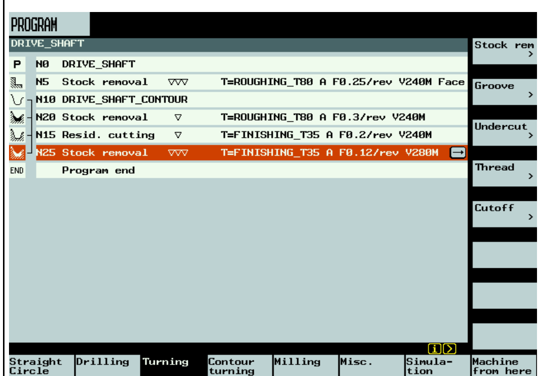
Accept</td>
<td>↓
0.12
240</td>
<td>
</td>
<td>• The FINISHINGTOOL_35_A tool is now used to cut the residual material.
</td>
</tr>
</table>

Accept	3x 2 0.2 0.2 0.2		• To cut all residual material, the input field for *relief cuttingn* must be switched to *yes*.
Simula- tion Simula- tion			• The simulation clearly shows the traverse paths for machining the residual material.
Contour turning Stock rem > Tools To program Accept	0.12 280		• The contour is finished in this machining step. This requires that the technological data is modified and the machining switched to finishing.
Simula- tion			• The work schedule should now look like this. 

Simula-tion



• The *volume model* shows here the current production status.



• Finally, tap the thread.

6.3 Thread

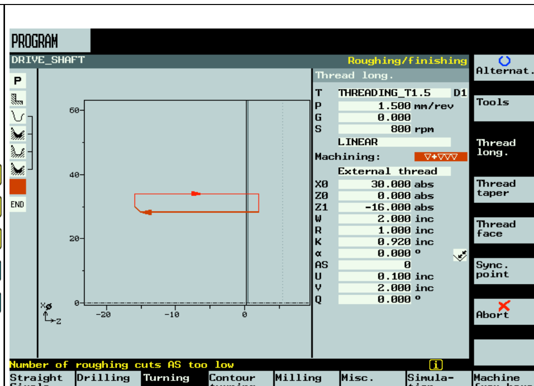
Turning

Thread

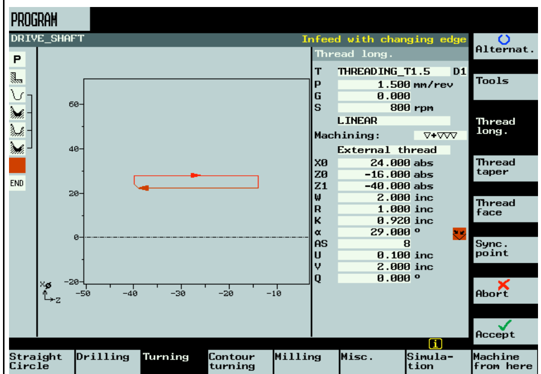
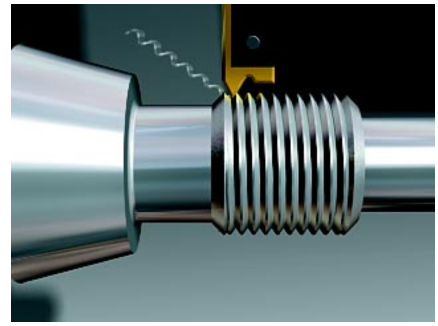
1.5

0

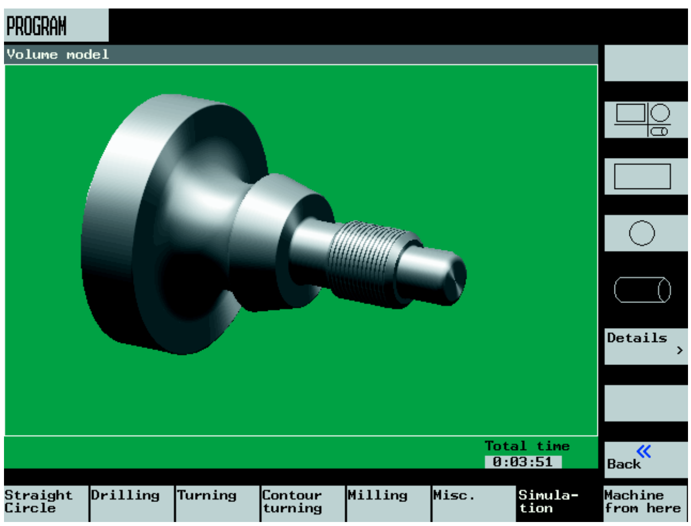
800



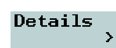
• Enter the thread data.

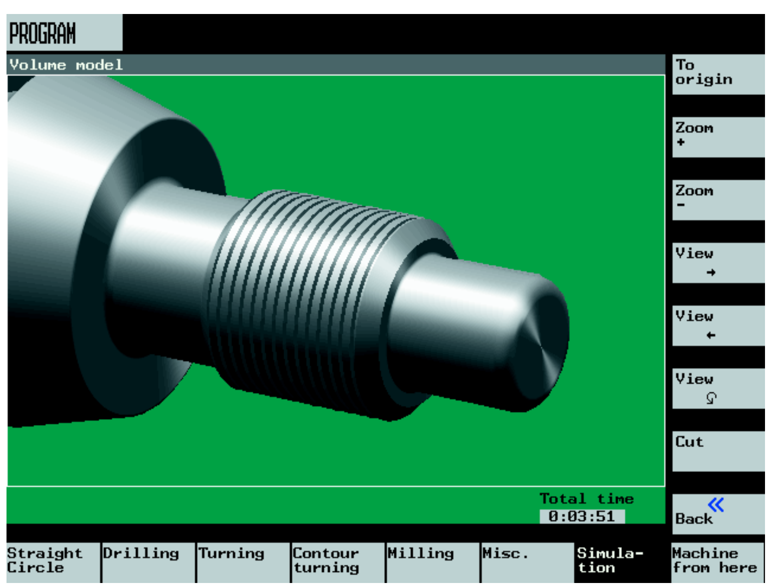
	...		• Complete the lower input fields. 

The work schedule is now simulated ...



... whereby the *Details* softkey can be used to check subareas of the workpiece .

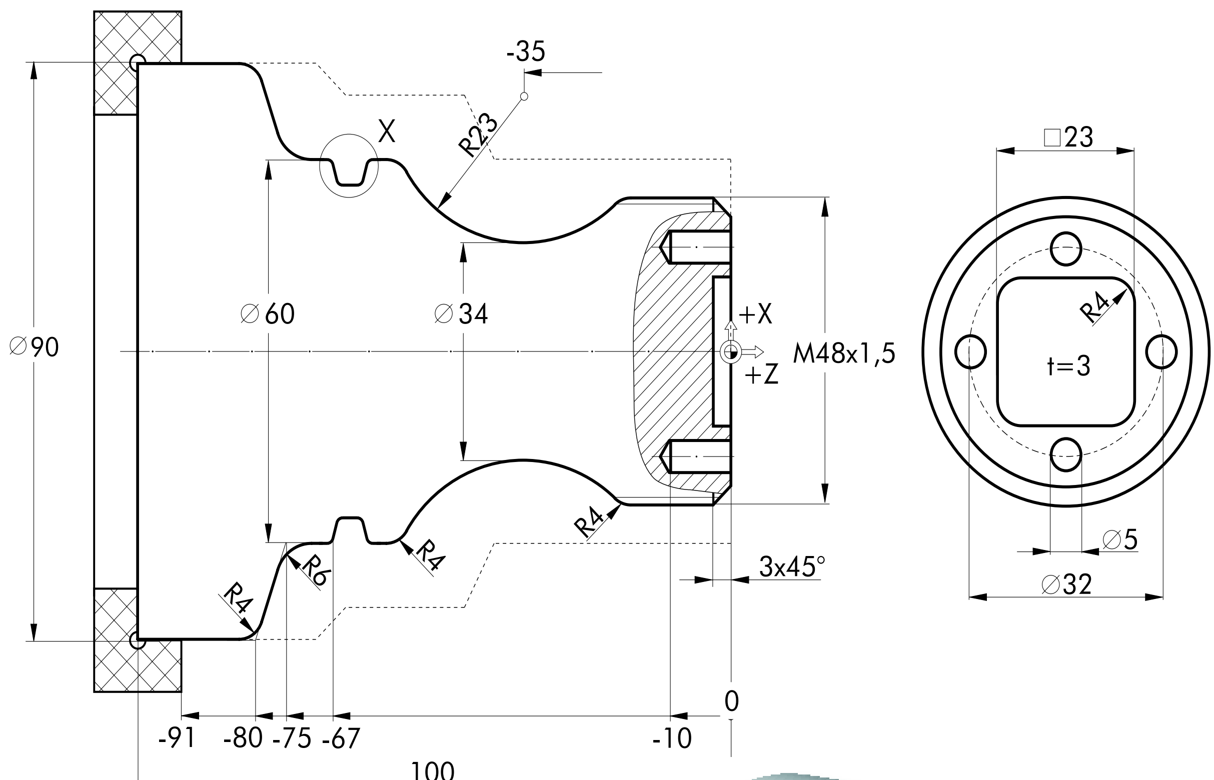


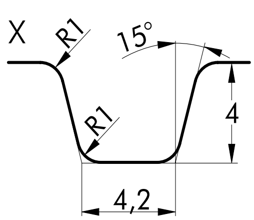


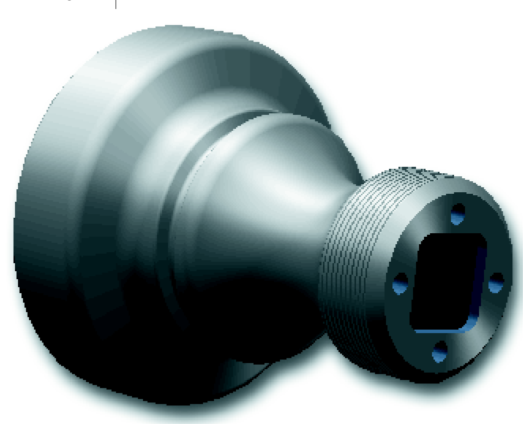
7 Example 3: Guide Shaft

This chapter familiarizes you with further important ShopTurn functions:

- Creating an arbitrary blank
- Cutting the difference material between the blank and the machined part
- Drilling on the front face
- Milling on the front face

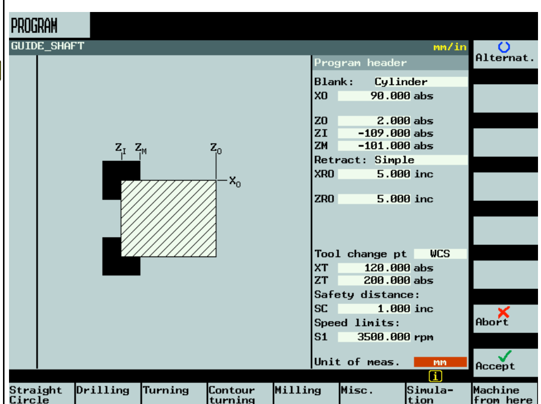
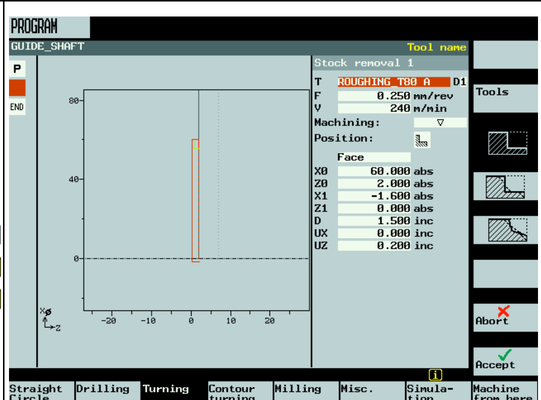
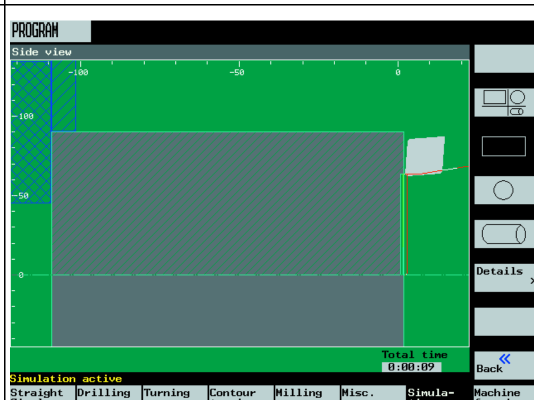




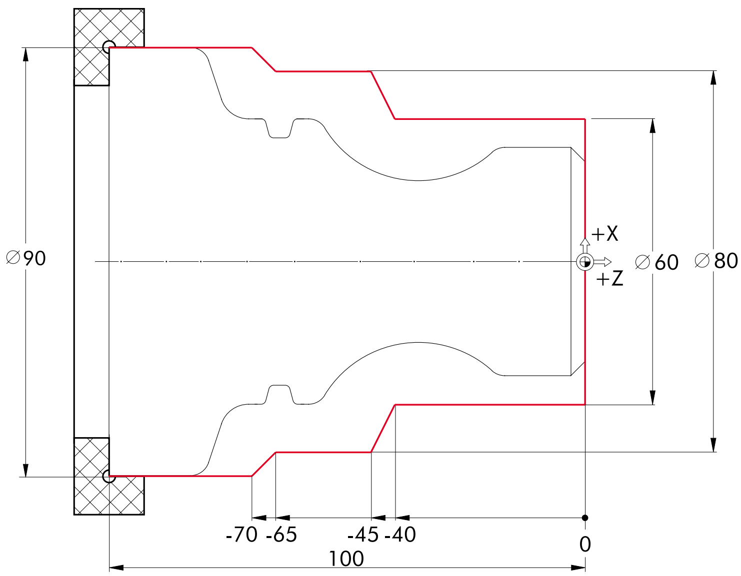


7.1 Facing

In this section, the new program is created and the blank faced to ZO.

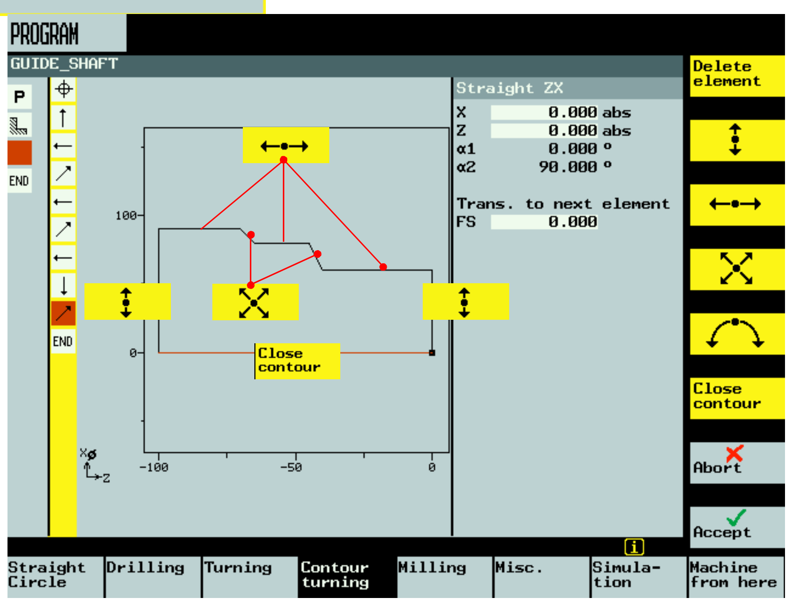
<table>
<tr><th>Keys</th><th></th><th>Screen</th><th>Explanation</th></tr>
<tr><td>Program manager
New ></td><td>U...</td><td>
</td><td>• Create the new program with the name "GUIDE_SHAFT" in the "WORKPIECES" directory.</td></tr>
<tr><td>Accept</td><td>...</td><td>
</td><td>• Complete the program header as shown on the left-hand side.
• Despite the arbitrary blank, select the Cylinder blank form here. ShopTurn ignores this input and orients itself on the arbitrary blank. This is constructed in the following.</td></tr>
<tr><td>Turning
Stock rem >
Tools
To program
Accept</td><td>0.25
240
...</td><td>
</td><td>• Complete the dialog fields as shown here.
• Because the arbitrary blank has a diameter of 60 mm, the XO dimension must also be set to 60 in this machining step.</td></tr>
<tr><td>Simula- tion
Simula- tion</td><td></td><td>
</td><td>• Test the machining step by performing the simulation.</td></tr>
</table>

7.2 Creating an Arbitrary Blank Contour



The blank contour must first be constructed with the contour calculator before ShopTurn can make use of it.

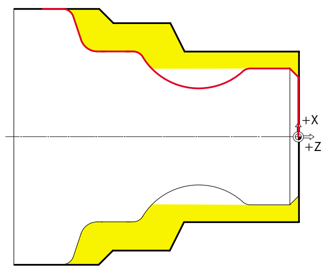
Create the GUIDE_SHAFT_BLANK blank contour with the starting point at XO/Z0 as shown above.



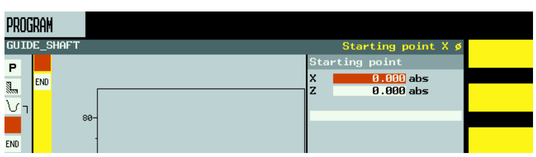
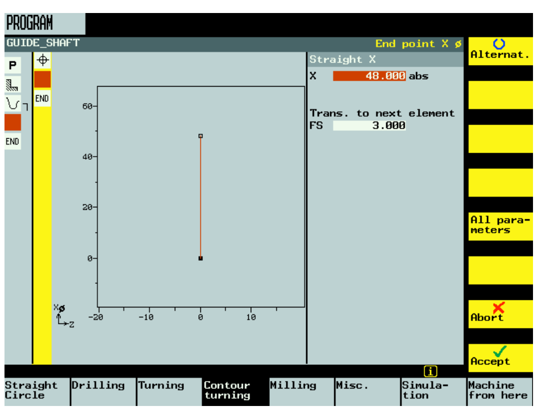
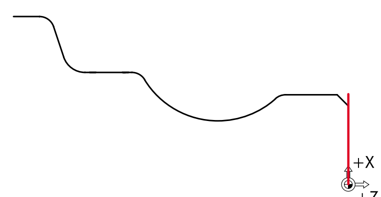
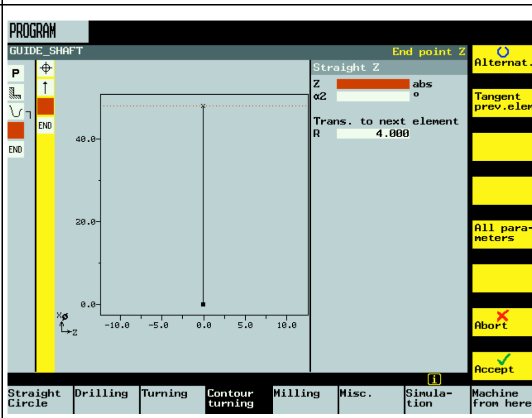
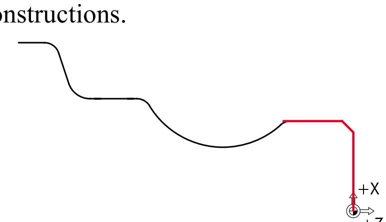
For emphasis, the softkeys that can be used to create the contour are shown here.

Caution: The contour must be closed!

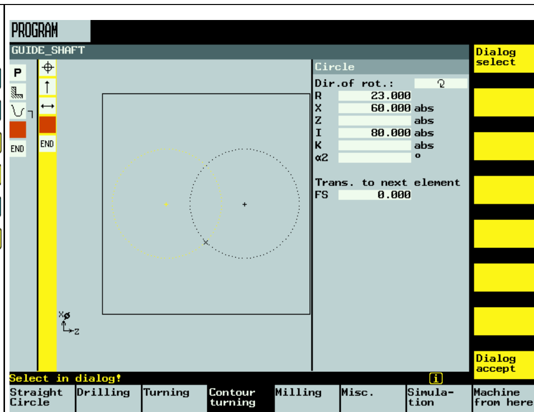
7.3 Create the Machined Part Contour and Cut



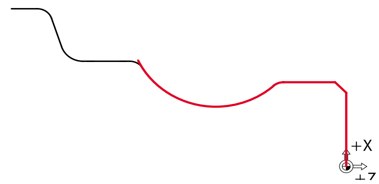
The machined part contour is entered in this section.

Keys		Screen	Explanation
Contour turning New contour >	K ... ⇔		• Assign the name "GUIDE_SHAFT_CONTOUR" to the contour.
Accept			• Because the blank has already been faced to ZO in the first machining step (refer to page 55), the machined part contour can begin at XO/ZO.
↕ Accept	48 ⇔ 3 ⇔		• The contour begins with a vertical path. • Enter the next chamfer as the subsequent element. 
←•→ Accept	2x ↓ ↻ 4 ⇔		• A horizontal path with unknown end point follows the chamfer. In this case, enter only the transition to the next element (radius 4). The end point of the path is then calculated automatically from the subsequent contour constructions. 

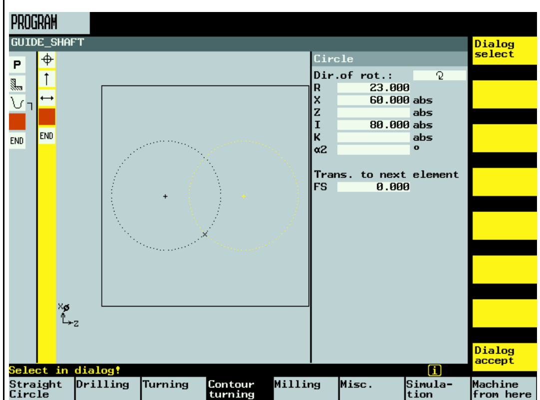
Dialog select



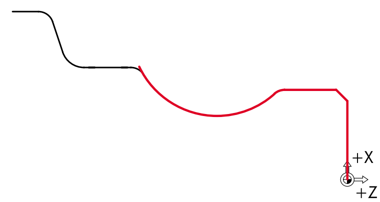
- If several solutions are to be possible as input for the contour data (e.g. here for the circular arc), they can be selected using the *Dialog select* softkey.
- The second solution is selected here.



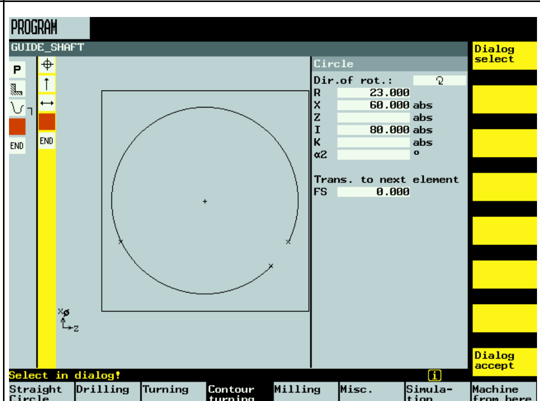
Dialog accept



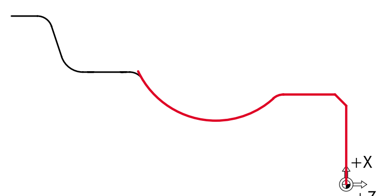
- Click the *Dialog accept* softkey to accept the solution.



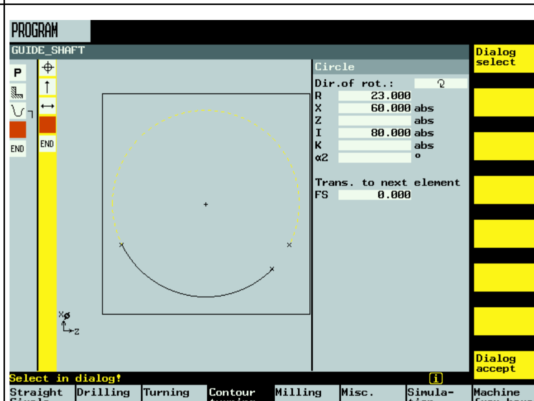
Dialog select



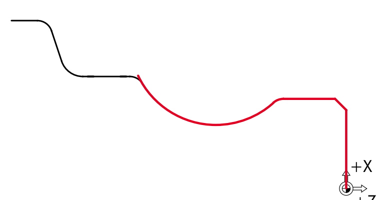
- The second solution is also selected here.

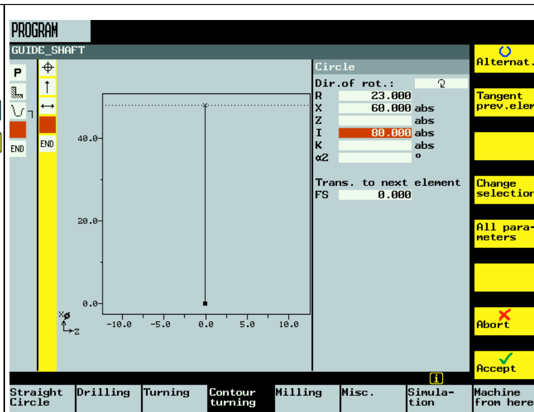
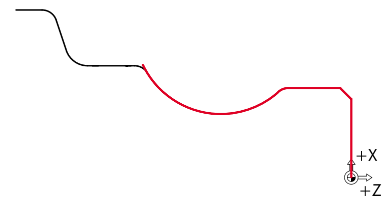
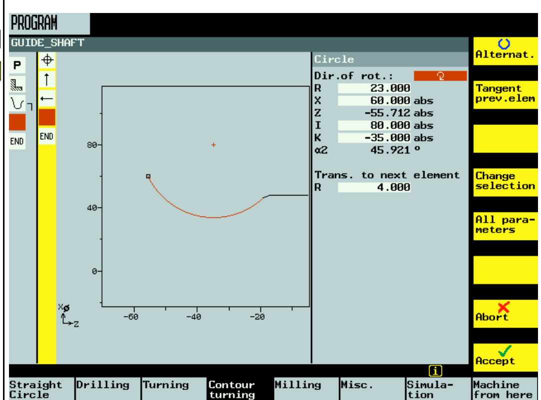
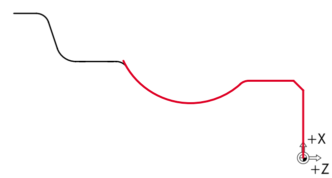
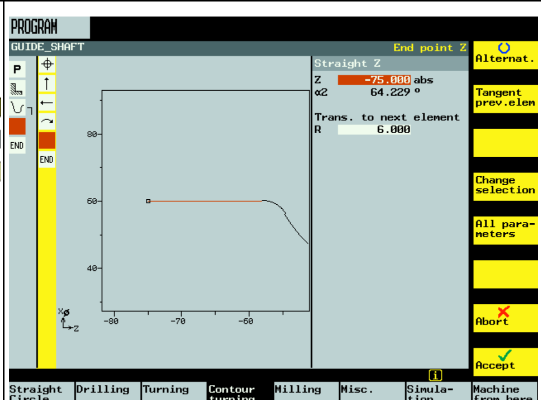
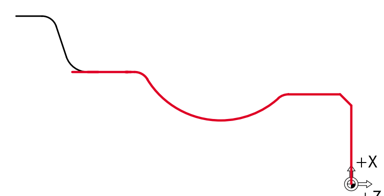
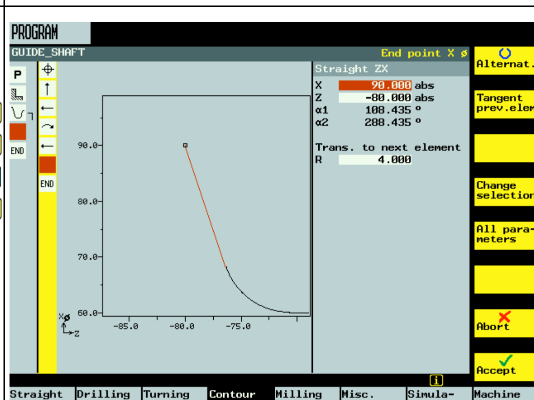
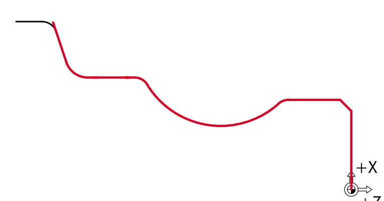


Dialog accept



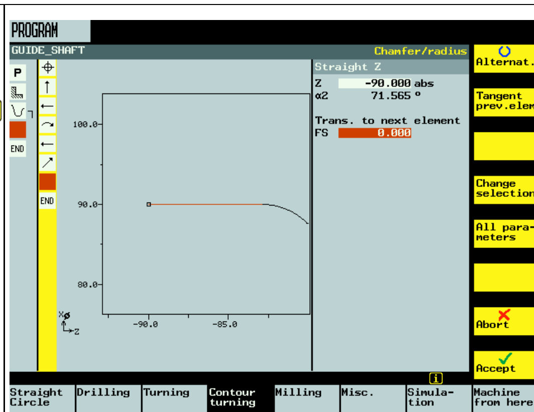
- The solution is also accepted with the *Dialog accept* softkey.



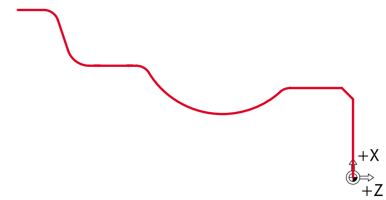
	-35		• The value K (center in the absolute dimension) is known. 
Accept	4		• The available contour data and the computed selection possibilities now allow the arc and the path (with unknown end point) to be constructed. • The transition to the next element is rounded with R4. 
Accept	-75 6		• A horizontal path with known end point at Z-75 and a radial transition with 6mm follows. 
Accept	90 -80 4		• A path with known end point follows. 

Accept

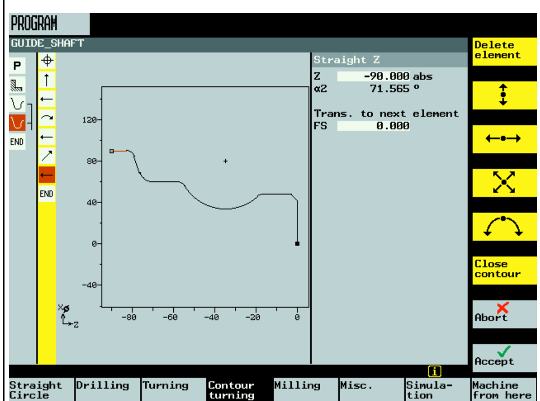
-90



• To prevent damage to the chuck, the construction already ends at Z-90.



Accept



• Accept the contour.

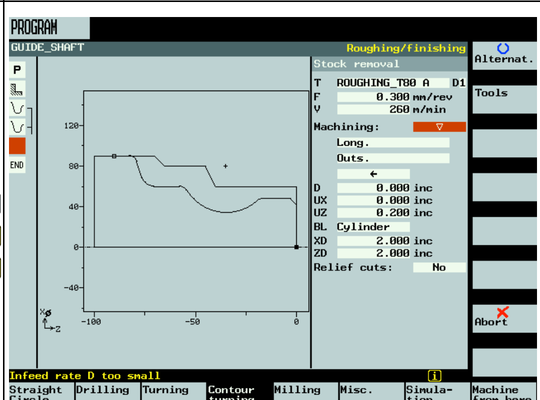
Stock rem >

Tools

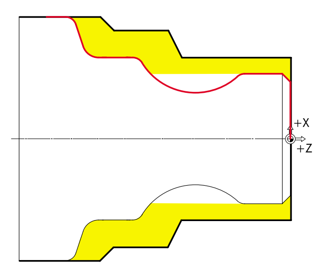
To program

0.3

260



• The ROUGHINGTOOL_80_A is used in this machining step to cut the contour.

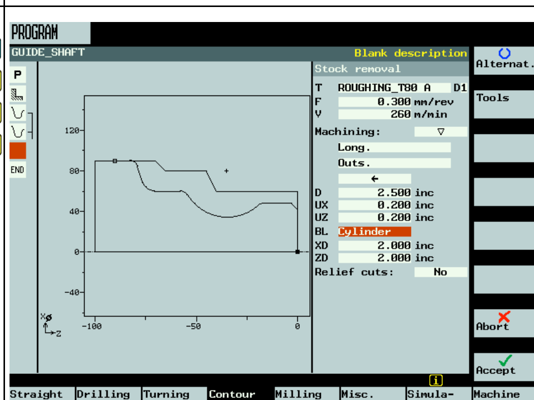


4x

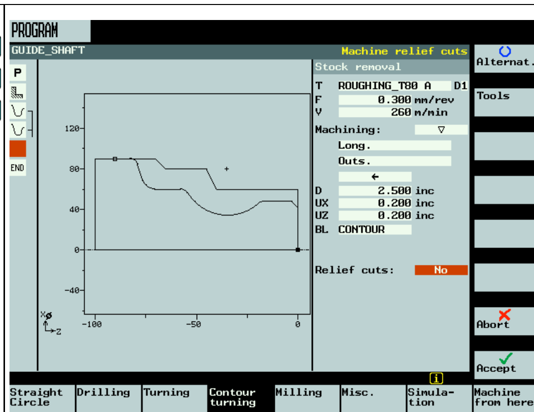
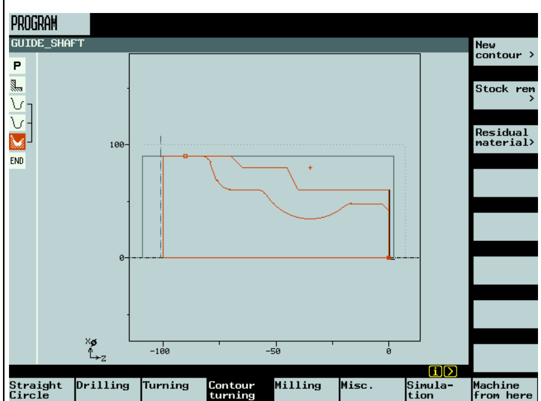
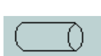
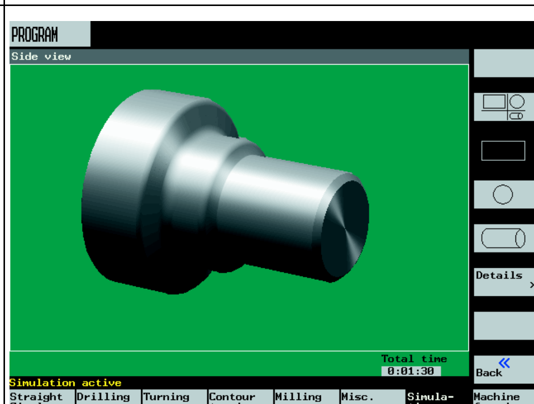
2.5

0.2

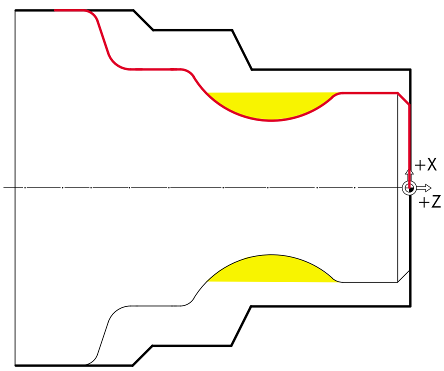
0.2



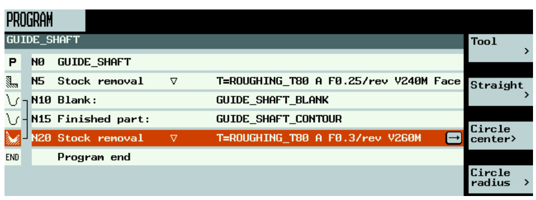
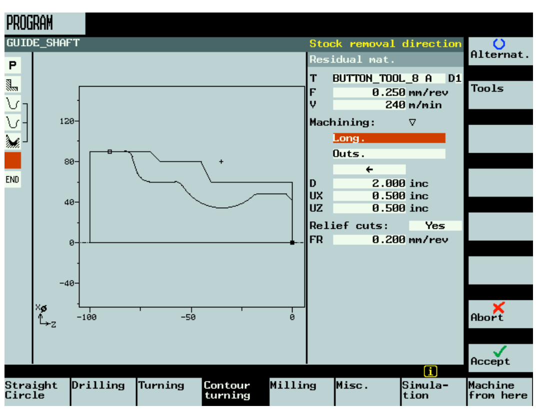
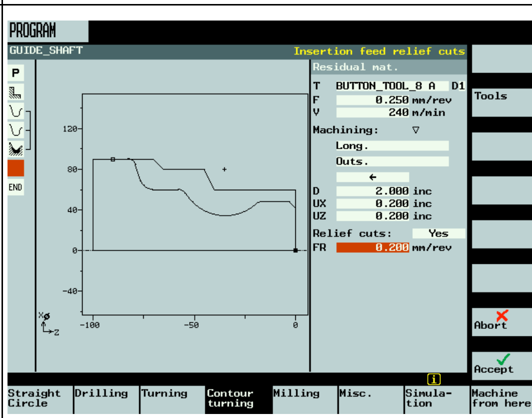
• Enter here the machining directions, the infeed amounts and the finishing allowances.

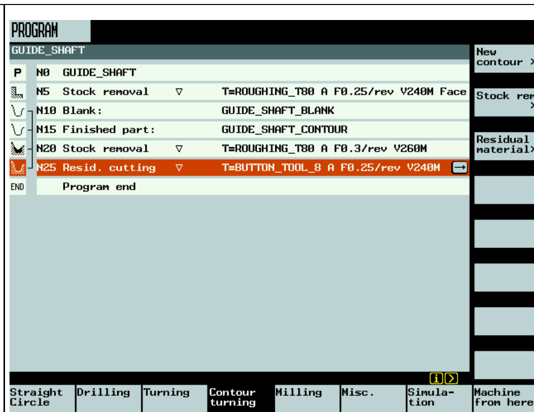
Accept	2x 		• The blank description must be switched to *Contour* here. • To ensure the recess of the Radius 23 does not get machined, change the *Relief Cut* field to *No*.
Simula-tion			• Once the machining step has been accepted, the two contours and the machining step are linked with each other. The contours displayed red also indicate this link.
			• The traverse paths in the simulation clearly show how the previously constructed blank is taken into consideration.
Simula-tion			• The *Volume model* shows the current production status.

7.4 Cut Residue

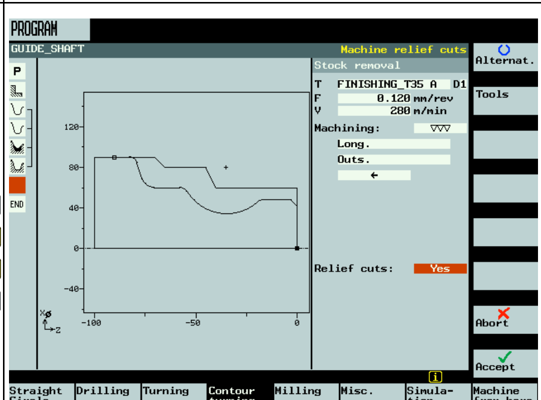
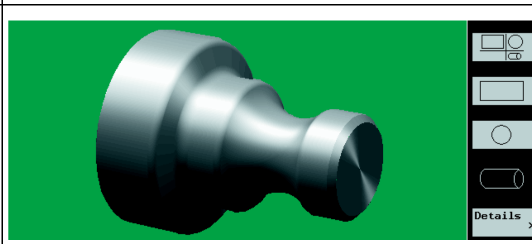


The residual material is cut in this section.

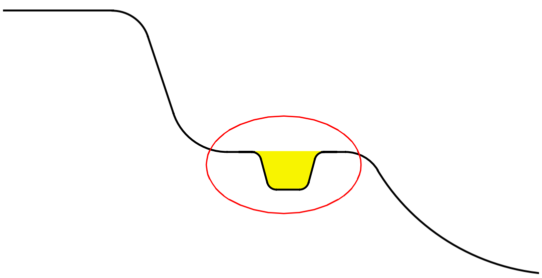
Keys	Screen	Explanation
		• This is the work schedule through to the roughing machining.
Contour turning Residual material> Tools To program ↓ 0.25 ⇒ 240 ⇒		• The BUTTON_TOOL_8_A is used to machine the residual material. • Enter the feeds and the cutting rate.
3x ↓ 2 ⇒ 0.2 ⇒ 0.2 ⇒ ↻ ↓ 0.2 ⇒ Accept		• The machining with relief cuts must be set to *Yes* here.

Simula-tion			• After the acceptance of the machining step, the work schedule should look like this. • The work schedule is simulated.
Simula-tion			• The complete production time is also shown in the simulation.

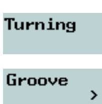
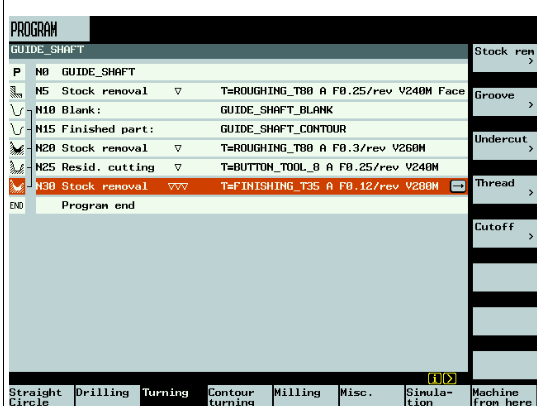
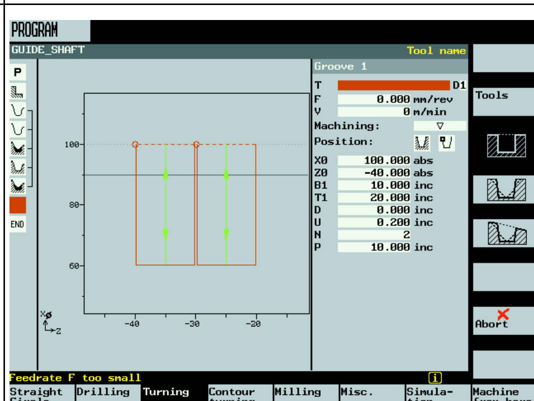
After the contour has been rough cut, it still needs to be finished.

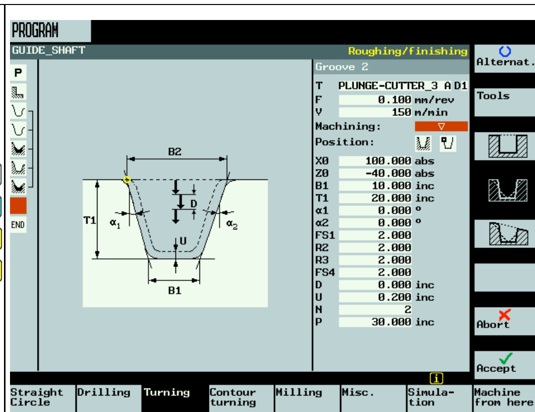
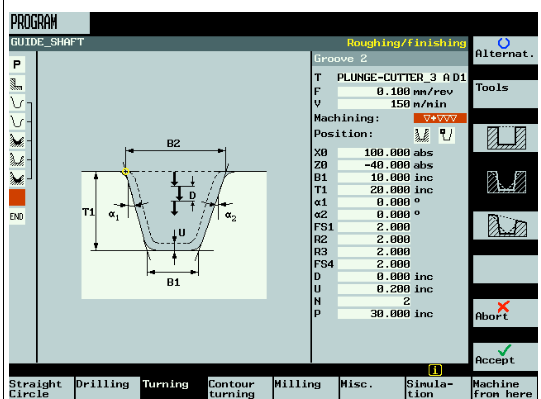
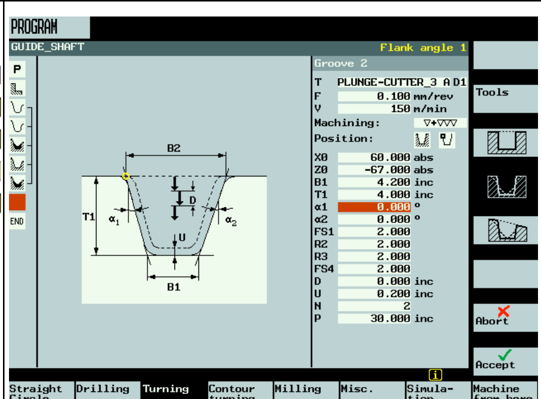
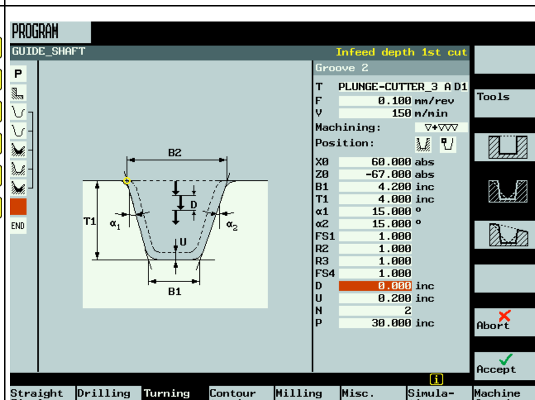
Keys		Screen	Explanation
Contour turning Stock rem Tools To program Accept	0.12 280		• Select the FINISHING_T35 A tool • The technological data must be modified to finish the contour. • The machining must also be changed to finishing. This results in the input fields for the finishing allowances being omitted.
Simula-tion Simula-tion			• The *Volume model* shows the current production status.

7.5 Groove



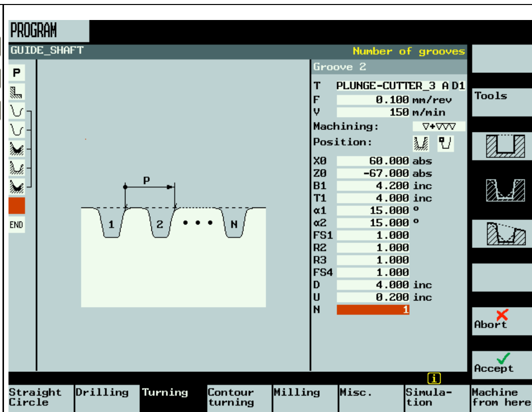
The groove is produced in this section.

Keys	Screen	Explanation
		• Click the *Turn*ing softkey in the horizontal softkey menu. • Click the *Groove* softkey in the vertical softkey menu.
		• Select the second of the three offered groove forms.

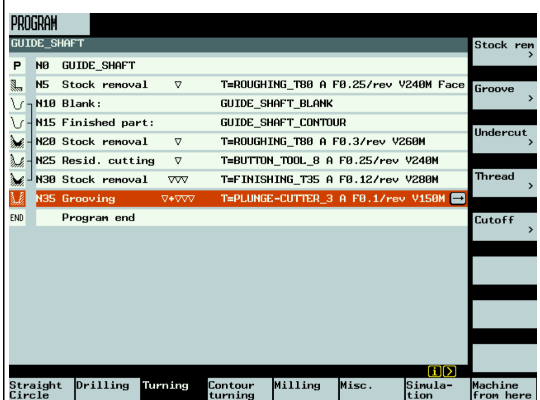
<table>
<tr>
<td>Tools
To program</td>
<td>0.1
150</td>
<td>
</td>
<td>• The PLUNGE-CUTTER_3 A grooving tool is used in the machining step.
• Enter the feeds and the cutting rate.</td>
</tr>
<tr>
<td></td>
<td>2x</td>
<td>
</td>
<td>• Change the machining to Rough Cutting / Finishing.</td>
</tr>
<tr>
<td></td>
<td>2x
60
-67
4.2
4</td>
<td>
</td>
<td>• Enter the position and the allowance of the groove.</td>
</tr>
<tr>
<td></td>
<td>15
15
1
1
1
1</td>
<td>
</td>
<td>• Enter the flank angle and the rounding at the corners.</td>
</tr>
</table>

4
0.2
1

Accept

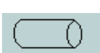


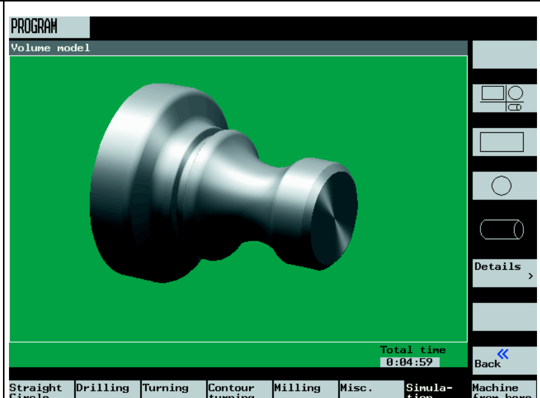
• Once all the data has been entered, the geometry of the groove must appear in the graphic.



• The groove has been added to the work schedule.

Simula-tion

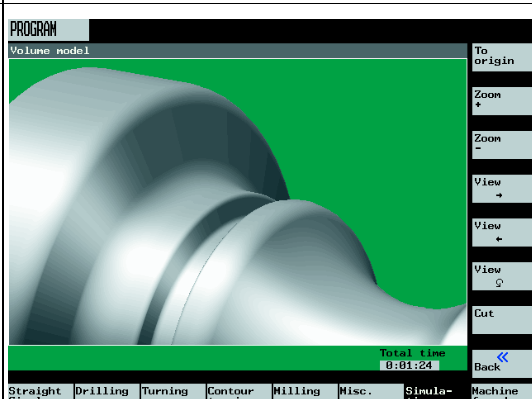




• The *Volume model*

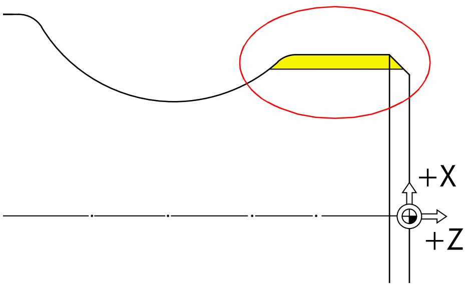
Details

Simula-tion

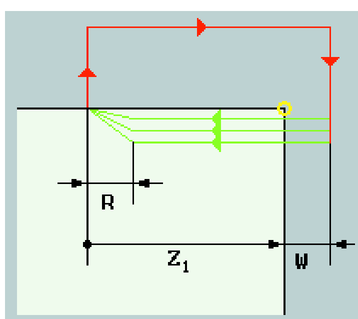
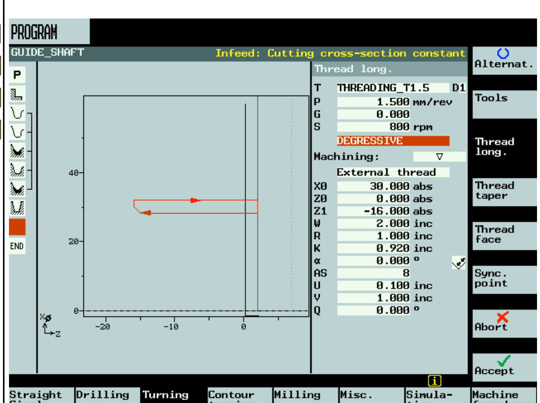
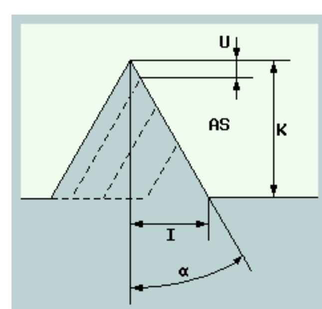


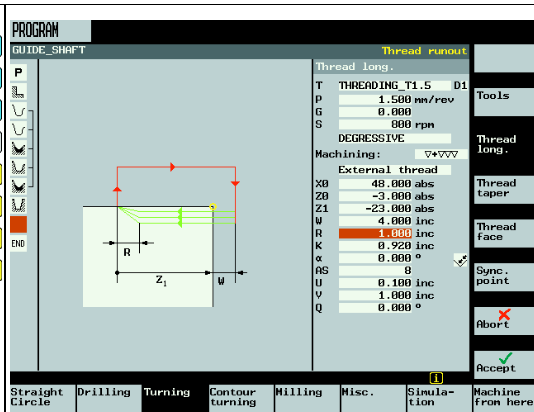
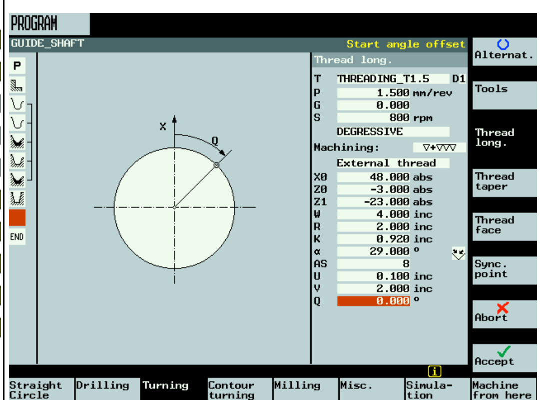
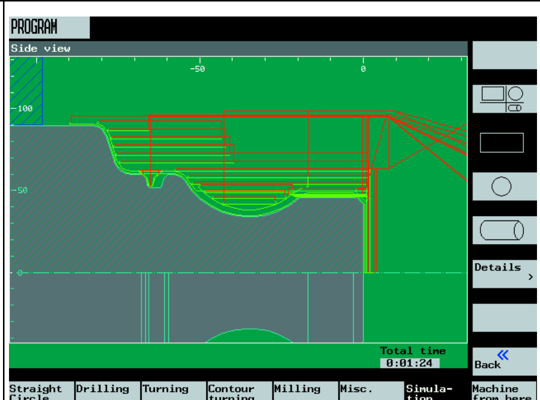
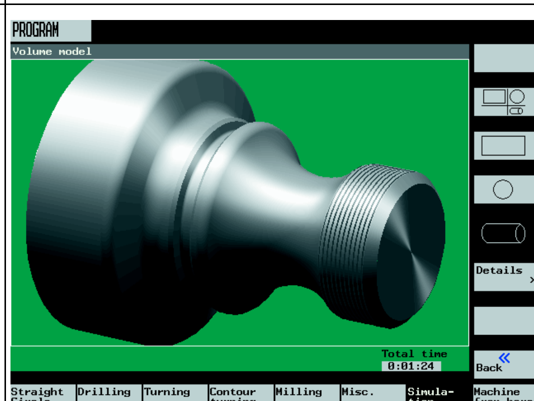
• The zoom view can be produced with the *Zoom* + and *Zoom* - softkeys.

7.6 Thread

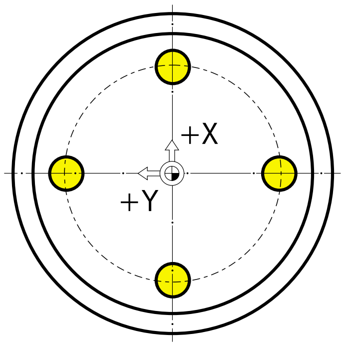


The thread is produced in this section.

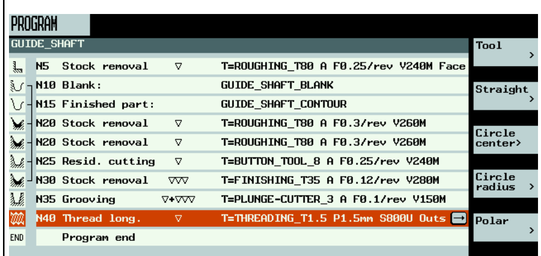
Keys	Screen	Explanation
Turning Thread Tools To program ↓ 1.5 0 800	 	• Select here the *longitudinal thread*. • The thread is created with the *DEGRESSIVE* setting. This setting causes the cutting division to be reduced for each cut and so ensures that the cutting cross-section remains constant.  • Various infeed strategies can be selected when the cursor is placed on *infeed slope (angle)*.

	… 2x 48 -3 -23 4		• Change the machining to *Rough Cutting / Finishing*.
Accept	2 0.92 29 8 0.1 2 0		• Enter the data for the thread. • An "infeed with alternating flank" is set.
Simula-tion			• The *Side view*
Simula-tion			• The *Volume model*

7.7 Drilling



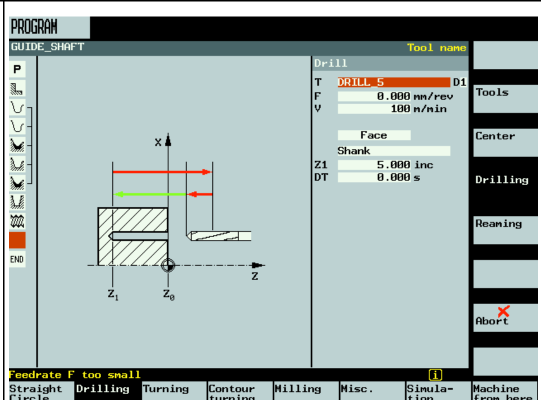
The holes on the front face are produced in this section (C axis or complete machining).

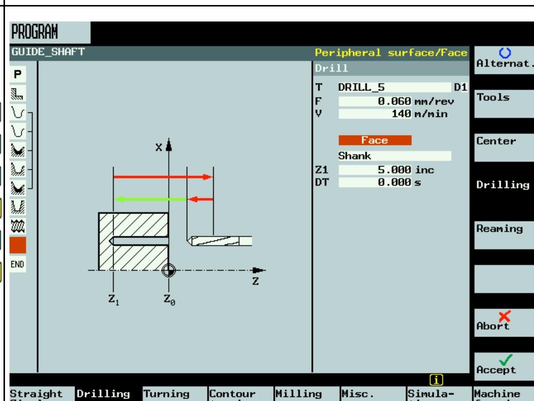
<table>
<tr><th colspan="2">Keys</th><th>Screen</th><th>Explanation</th></tr>
<tr><td></td><td></td><td>
</td><td>• Once the thread has been created, the work schedule should look like this:</td></tr>
<tr><td>Drilling

Drilling Reaming >

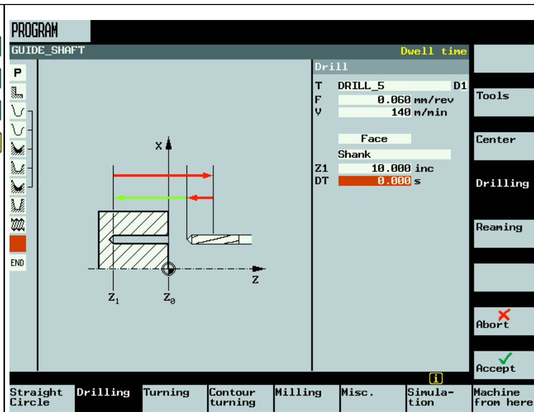
Drilling

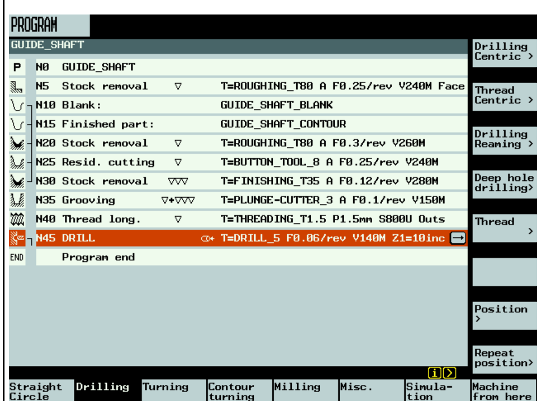
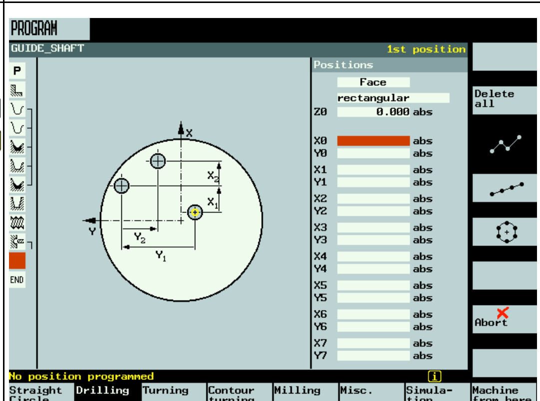
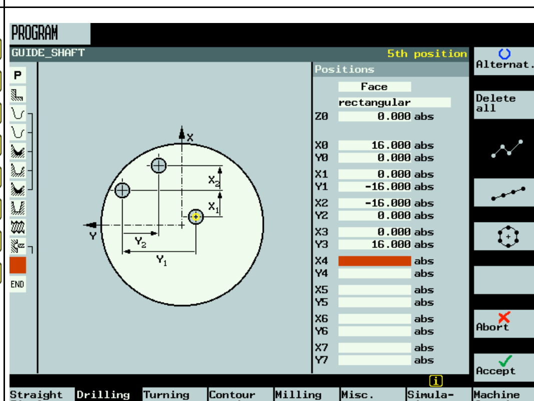
Tools

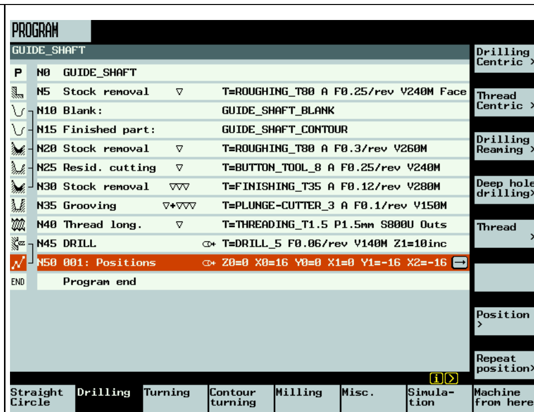
To program</td><td></td><td>
</td><td>• The workpiece is drilled directly, i.e. without centering. Consequently select the Drilling option. The DRILL_5 is used here.</td></tr>
<tr><td></td><td>
0.06

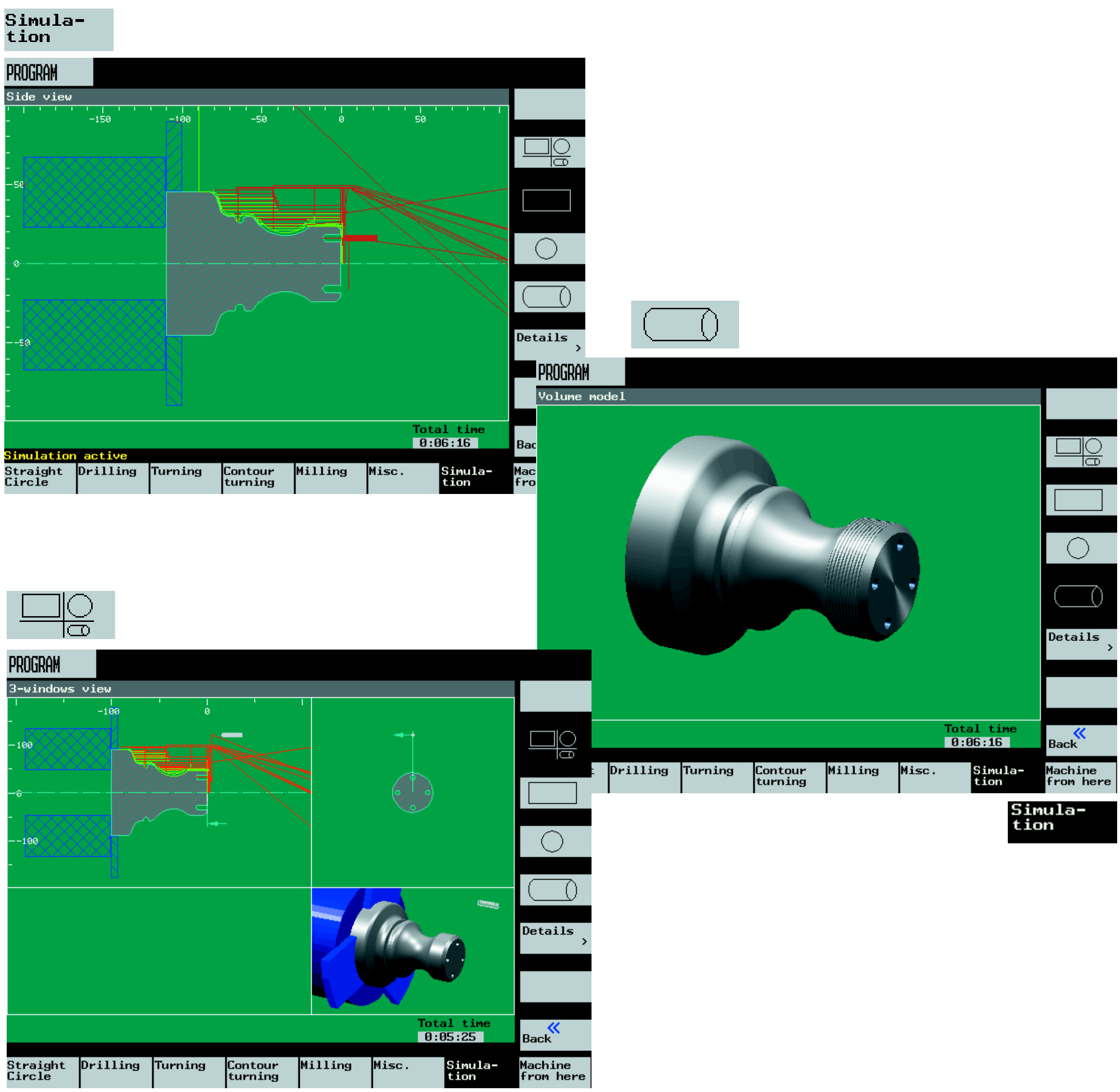
140</td><td>
</td><td>• Enter the technological data.

• Set the unit for F to mm/rev and the unit for V to m/min.</td></tr>
</table>

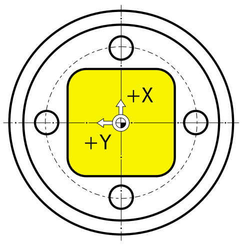
<table>
<tr>
<td>Accept</td>
<td>10</td>
<td>
</td>
<td>• Change the depth reference to shank

• Enter the hole depth either as 10mm incremental or -10mm absolute.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>
</td>
<td>• Once the hole machining step has been accepted, an open connector appears at the bottom. This will be linked automatically with the hole positions later.</td>
</tr>
<tr>
<td>Position ></td>
<td>2x
0</td>
<td>
</td>
<td>• As an exercise, enter the four holes as individual positions.
The simpler solution for the positioning would be possible using the softkey (holes on hole circle).</td>
</tr>
<tr>
<td>Accept</td>
<td>16
0
0
-16
-16
0
0
16</td>
<td>
</td>
<td>• Enter the hole positions here</td>
</tr>
</table>

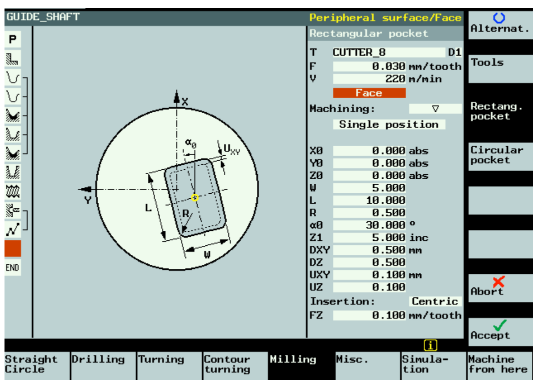
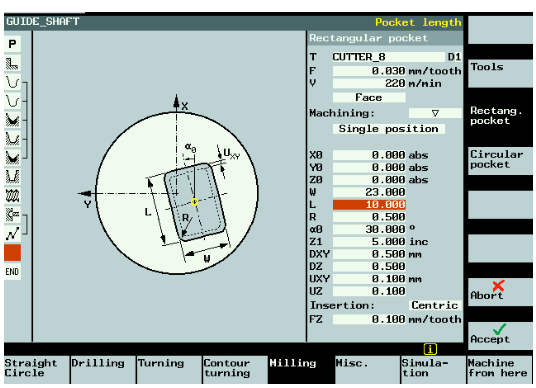
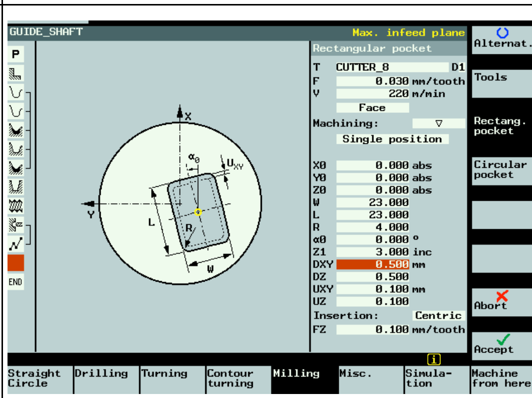
			• After accepting the holes, the hole positions are automatically linked with the previous technology block.



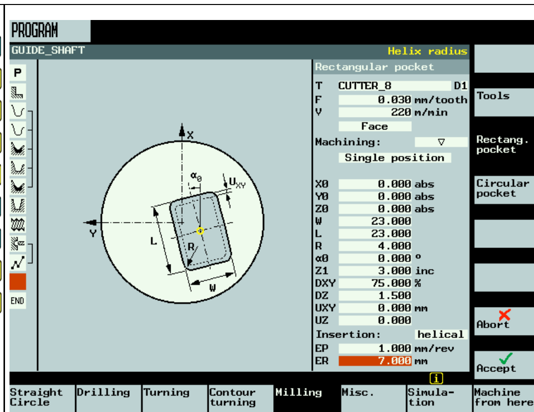
7.8 Mill Rectangular Pocket



The rectangular pocket on the front face is produced in this section (C-axis or complete machining).

Keys		Screen	Explanation
Milling Pocket > Tools To program	↓ 0.03 ⇒ 220 ⇒		• The rectangular pocket is machined using the size 8 milling tool. Insert the tool and enter the associated technological data.
	3x ↓ 0 ⇒ 0 ⇒ 0 ⇒ 23 ⇒		• Once the technological data has been entered, enter the geometric data.
	23 ⇒ 4 ⇒ 0 ⇒ 3 ⇒		• Further geometric data

75
1.5
0
0
1
7



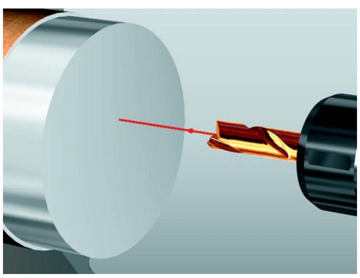
• Finally, determine the insertion type from the three possibilities: *centered*, *helical* and *reciprocating*. Select *helical* in this case.

• *EP = pitch of the helix*

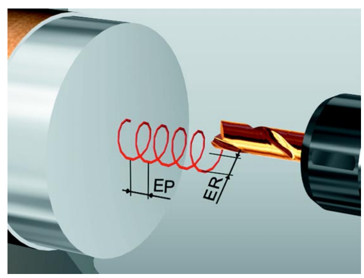
• *ER = radius of the helix*

• *RW = insertion angle (for reciprocating)*

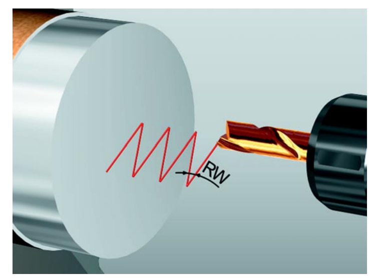
Centered

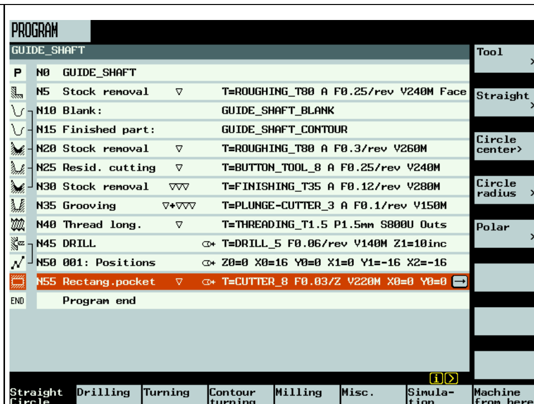


Helical

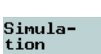


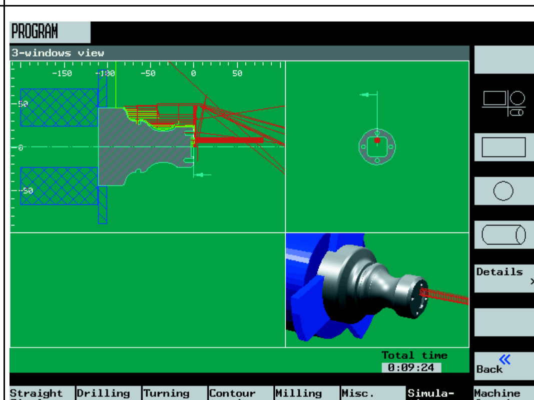
Reciprocating





• The completed work schedule should now look like this:



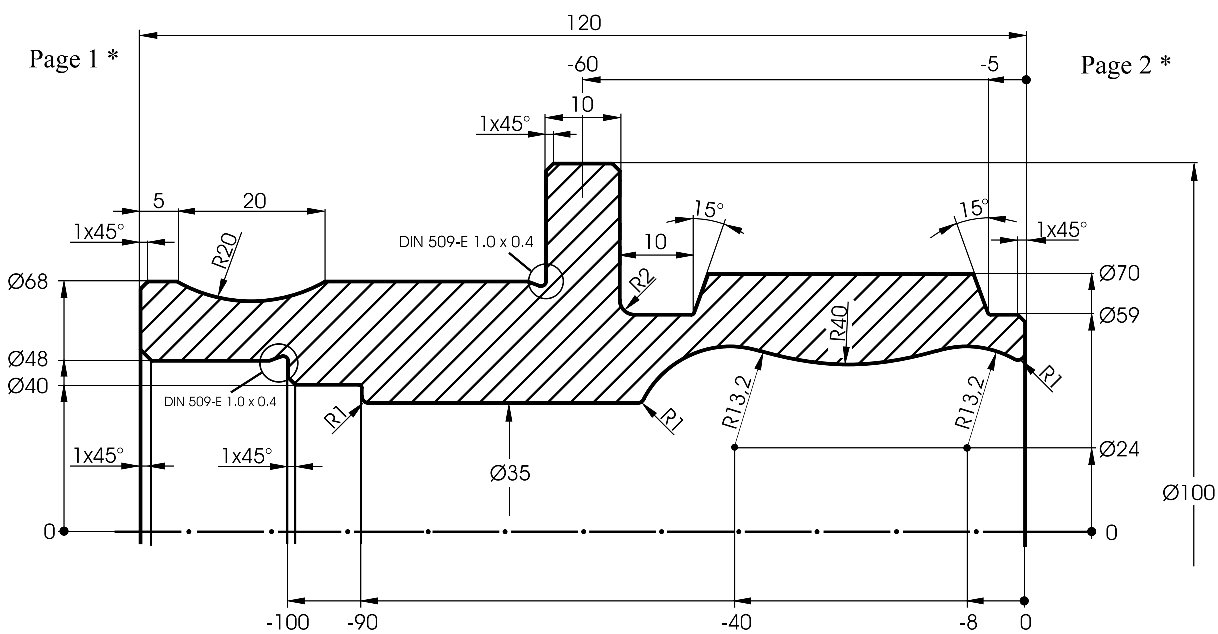


• Workpiece in the *3-window view*

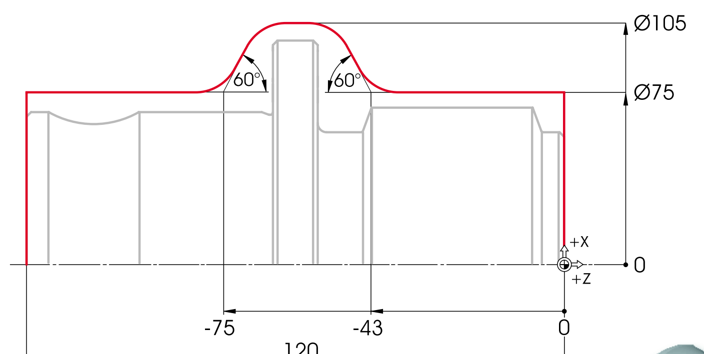
8 Example 4: Hollow Shaft

This chapter familiarizes you with further important ShopTurn functions:

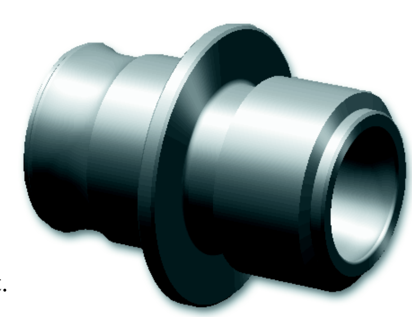
- Internal machining of workpieces
- Extended editor
- Undercut form E
- Asymmetrical groove



Rohteil:



Alle nicht bemaßten Radien R10

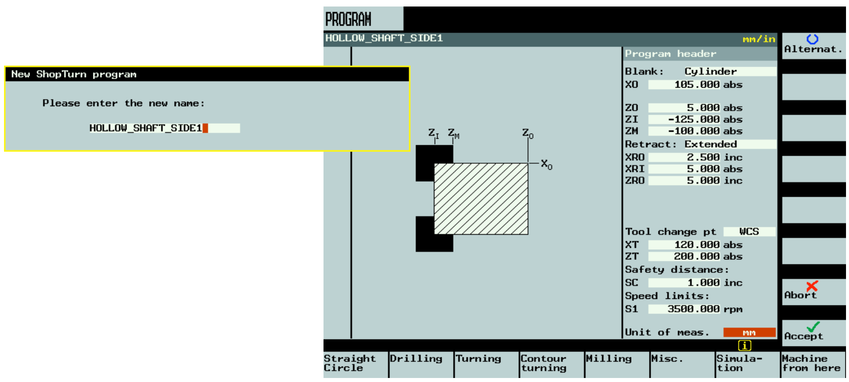


* Wegen der besseren Spannmöglichkeit wird zuerst die Seite 1 gefertigt.

8.1 Create the First Workpiece Side

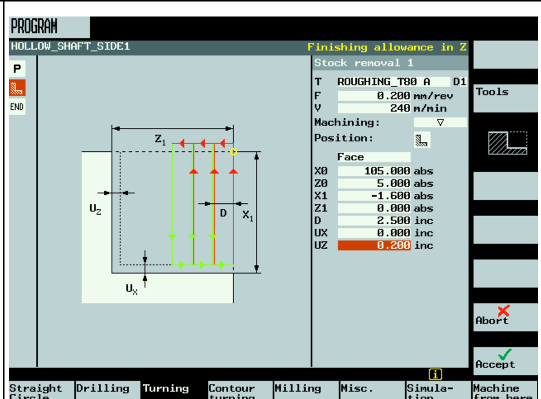
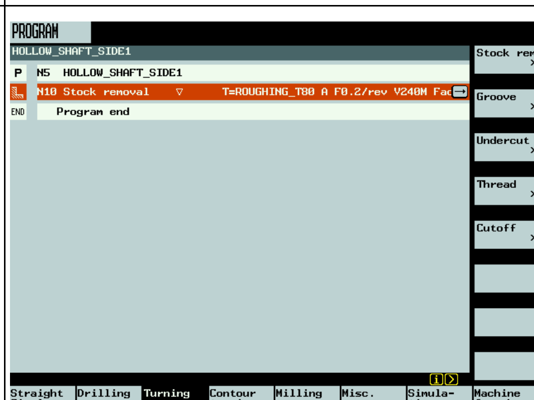
Create work schedule

Because the workpiece is to be machined from two sides (and produced without counterspindle), two work schedules must be created. For technical production reasons, the work schedule is initially created for the left-hand "HOLLOW_SHAFT_SIDE1" side.



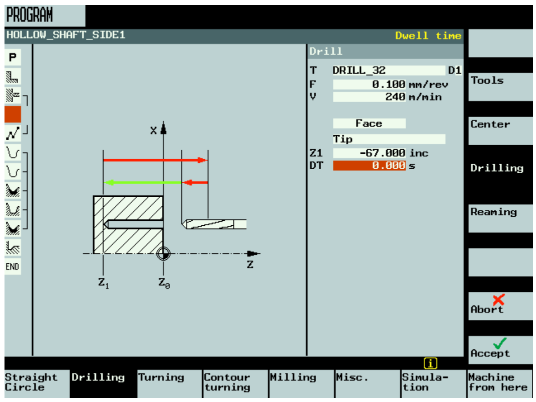
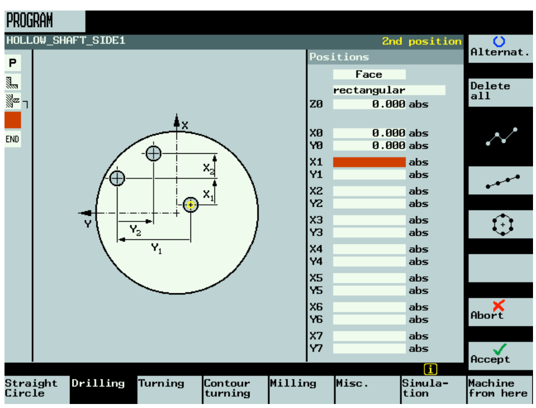
The program header can be accepted once the data shown on the left-hand side has been entered.

8.1.1 Face Turning

Keys	Screen	Explanations
Turning Stock rem > Tools To program ... Accept		• The blank is faced to X-1.6 and ZO here. Because still a great amount of material (5 mm) remains on the front face, the *Machining* field remains here at ▽ . • The allowance of 0.5mm will be finished later.
		• View of the work schedule after acceptance of the first machining step.

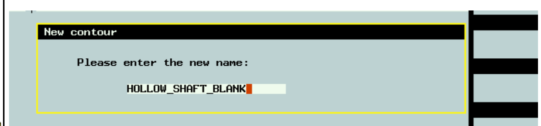
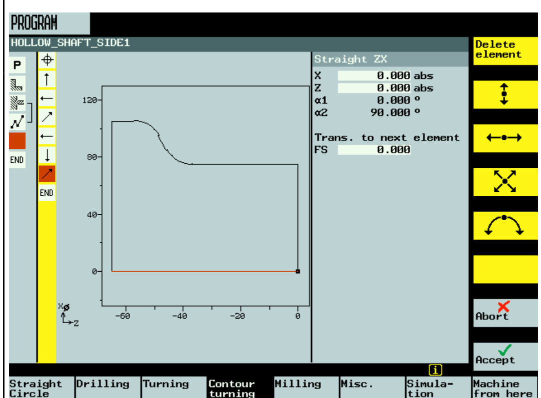
8.1.2 Drilling

The workpiece is now drilled at the center.

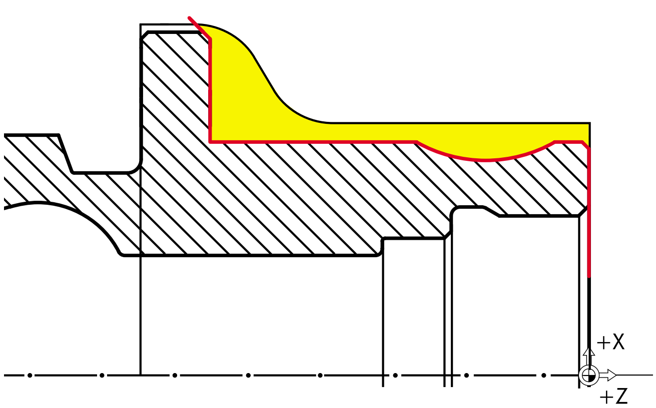
Keys		Screen	Explanations
Drilling Drilling Reaming > Drilling Accept	...		• Enter the technological and geometric data for the drilling as shown in the picture on the left-hand side.
			• The work schedule should now appear as follows.
Position > Accept	...		• Enter the position data for drilling as shown in the picture on the left-hand side.
			• The drilling technology and geometric data is automatically linked to each other in the work schedule.

8.1.3 Blank Contour

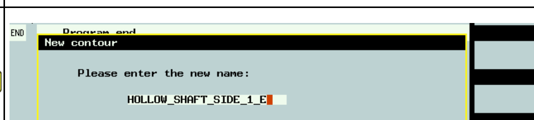
The blank contour of the workpiece is entered in this section. Because the workpiece is only machined from one side for each work schedule, it suffices only to construct the blank contour to Z-65.

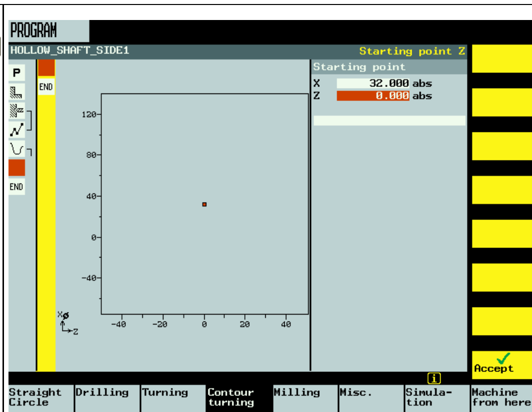
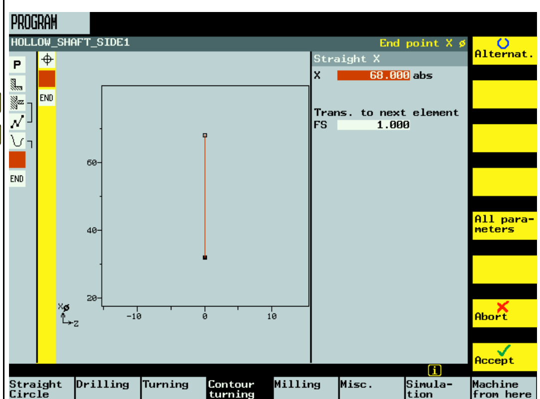
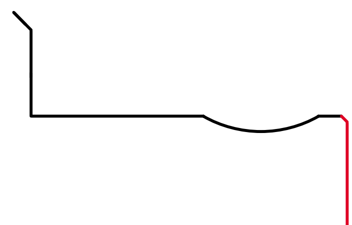
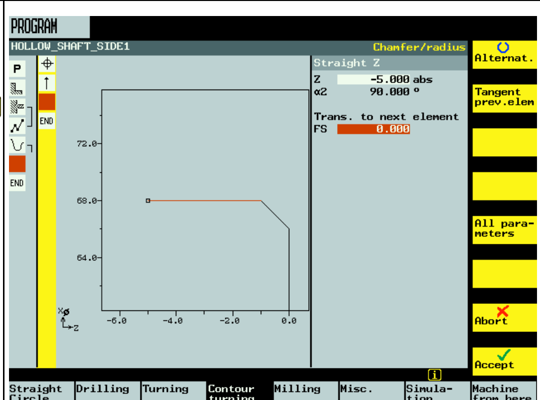
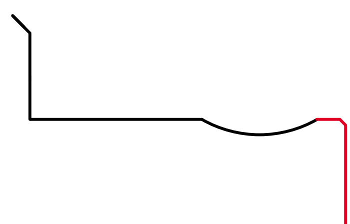
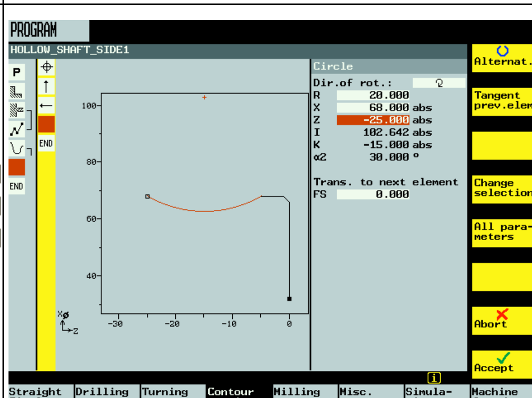
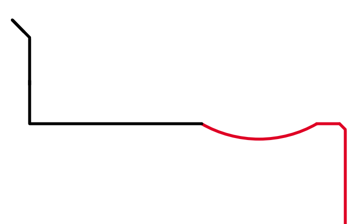
Keys		Screen	Explanations
Contour turning New contour >	R...		• Assign the name "HOLLOW_SHAFT_BLANK" to the blank contour.
	...		• In accordance with the previous drawing, construct the contour from the starting point X0/Z0 shown on the left-hand side. • Construct the contour to a maximum value of Z-65. • Close the contour at X0/Z0.

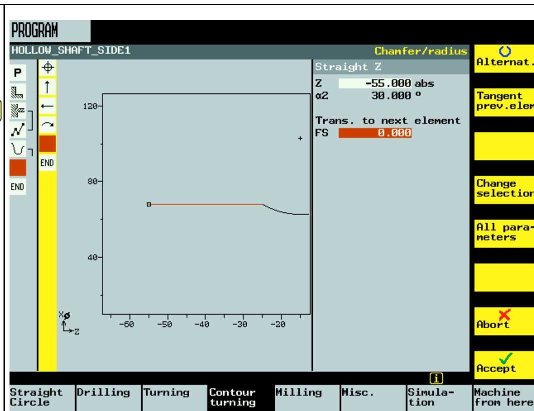
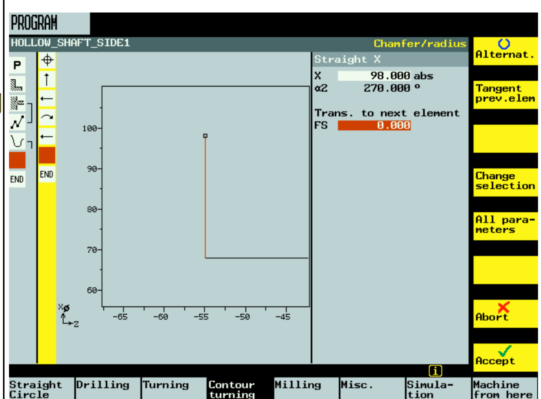
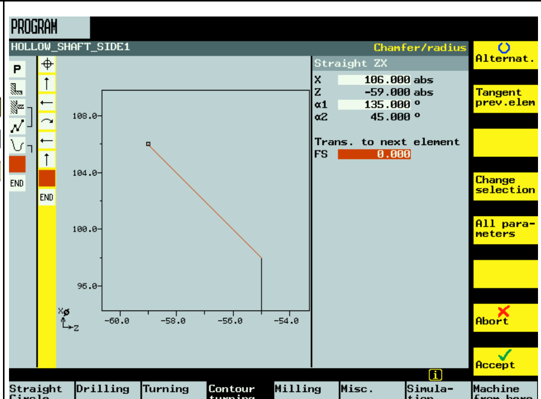
8.1.4 Machined Part Contour of the First Side External

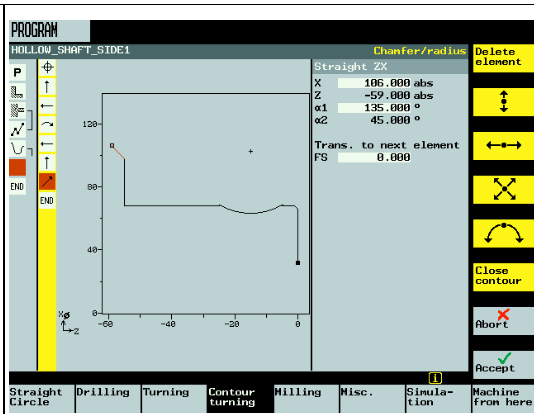
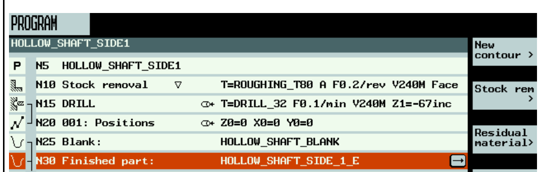


The (red) contour of the machined part intentionally does not correspond to the drawing. The machined part contour serves both as a limit for the rough-cut machining, but also, and what is much more important, it specifies the exact traverse path for the finishing. Thus, the construction begins here at the 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.

Keys		Screen	Explanations
New contour >	1_E		• Assign the name "HOLLOW_SHAFT_SIDE_1_E" to the contour.

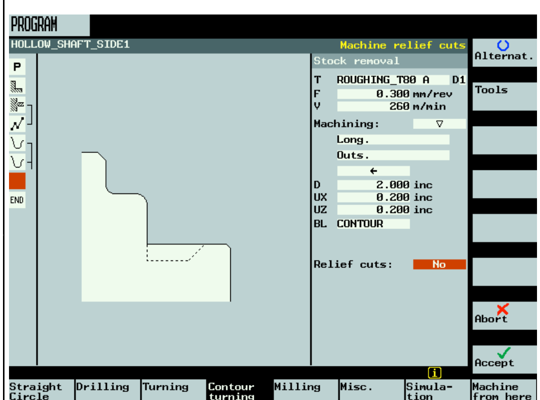
Accept	32		• Set the starting point to X32/Z0.
Accept	68 1		• This path ends at X68 and has a chamfer as transition to the next element. 
Accept	-5		• The horizontal path ends at Z-5. 
Dialog select Dialog accept Accept	20 68 -25		• A circular arc in clockwise direction follows. 

<table>
<tr>
<td>
Accept</td>
<td>-55</td>
<td>
</td>
<td>• A straight line to Z-55 follows.
• The undercut will be inserted later as a single element.
</td>
</tr>
<tr>
<td>
Accept</td>
<td>98</td>
<td>
</td>
<td>• A vertical path to X98, the initial diameter of the chamfer, follows.
</td>
</tr>
<tr>
<td>
Accept</td>
<td>106
135</td>
<td>
</td>
<td>• A slope extended outside the blank diameter follows. This remains as chamfer after the second side has been machined.
</td>
</tr>
</table>

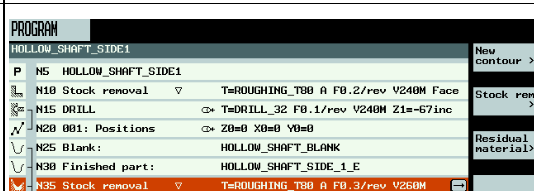
<table>
<tr>
<td>Accept</td>
<td></td>
<td>
</td>
<td>• The contour is accepted in the work schedule.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>
</td>
<td>• Both contours are automatically linked in the work schedule and, in accordance with their sequence, marked as a blank or machined part in the work schedule.</td>
</tr>
<tr>
<td>Stock rem >

Tools

To program

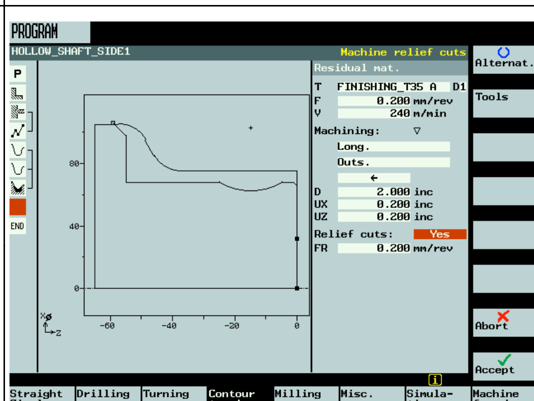
Accept</td>
<td>...</td>
<td>
</td>
<td>• Now machine the machined part contour using the cutting machining step.

• One of three settings can be selected for the description of the blank:
1. Cylinder: blank = cylinder
2. Contour: blank = constructed contour
3. Allowance: blank = constructed contour with defined allowance

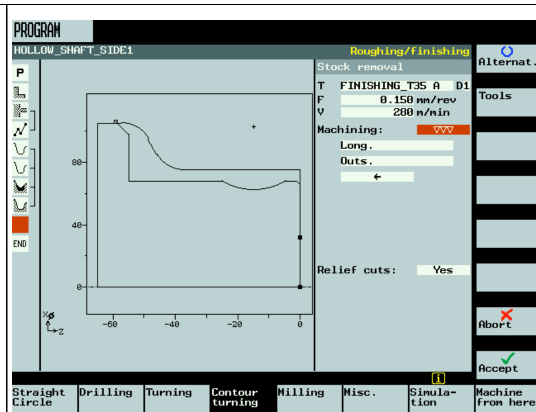
• It is not sensible to insert using the roughing tool. Thus, the Relief cut field is changed to No.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>
</td>
<td>• The contours are automatically linked with the cutting machining step in the work schedule.</td>
</tr>
<tr>
<td>Residual material>

Tools

To program

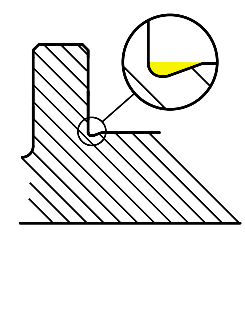
Accept</td>
<td>...</td>
<td>
</td>
<td>• Prior to the finishing, in this machining step the residual material is cut in the concave fillet.

• The Relief cut field must be changed to Yes to allow the concave fillet to be taken into consideration.</td>
</tr>
</table>

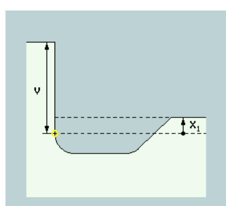
Stock rem > Tools To program Accept	...		• Finally, finish the contour. • If necessary, load the appropriate tool in the machining step if this is not suggested automatically. • The *Relief cut* field must also be changed to *Yes* here.

8.1.5 Undercut

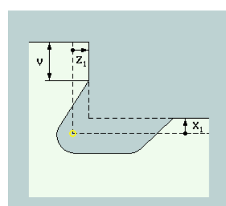
Four different undercut types are available for selection:



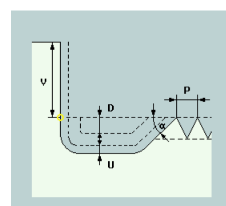
Undercut form E



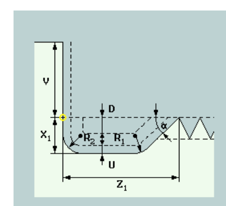
Undercut form F



Undercut DIN thread

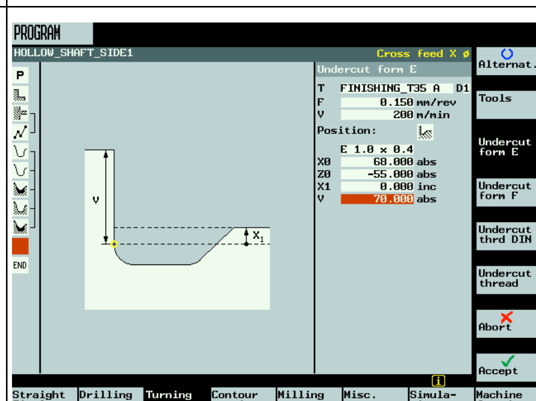


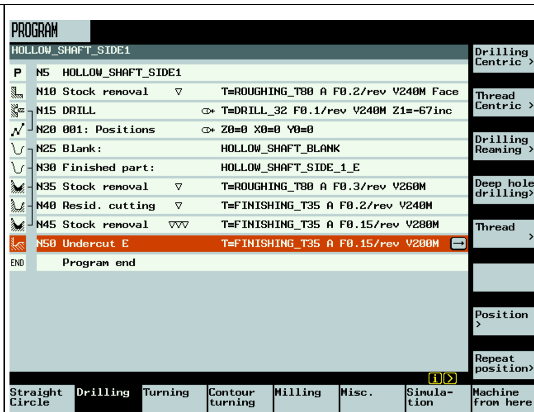
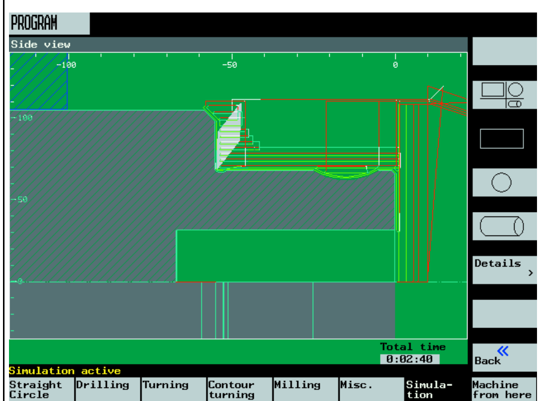
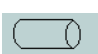
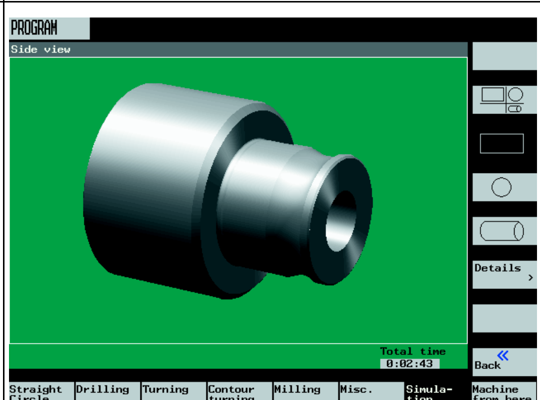
Undercut thread

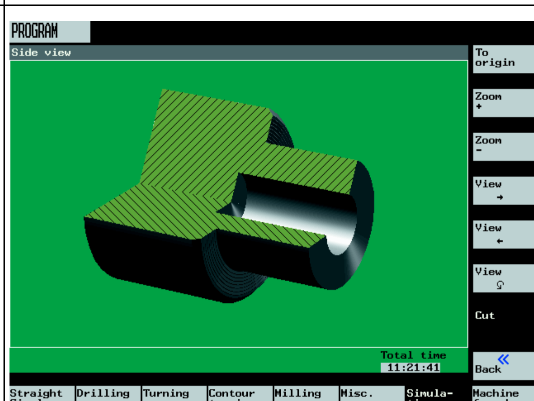


Once the outside contour has been completely machined, produce the undercut.Now create, as specified in the drawing, the undercut form E.

Now create, as specified in the drawing, the undercut form E.

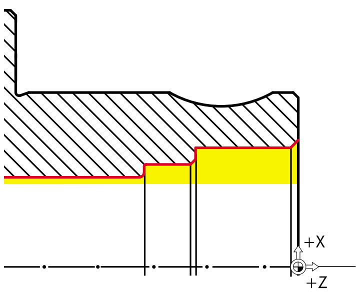
Keys		Screen	Explanations
Turning Undercut > (i) Accept	...		• Enter the data to create the undercut.

<table>
<tr>
<td></td>
<td></td>
<td>
</td>
<td>• This completes the 1st external side of the workpiece.</td>
</tr>
<tr>
<td>Simula-tion
Details >
Zoom +</td>
<td></td>
<td>
</td>
<td>• To check, simulate the work schedule.
• Increase the view using the Details > or Zoom + key.</td>
</tr>
<tr>
<td>Back
</td>
<td></td>
<td>
</td>
<td>• The Volume model</td>
</tr>
<tr>
<td>Details >
Cut

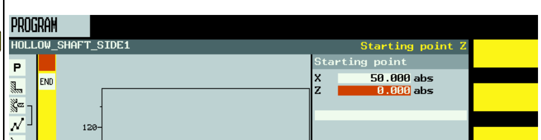
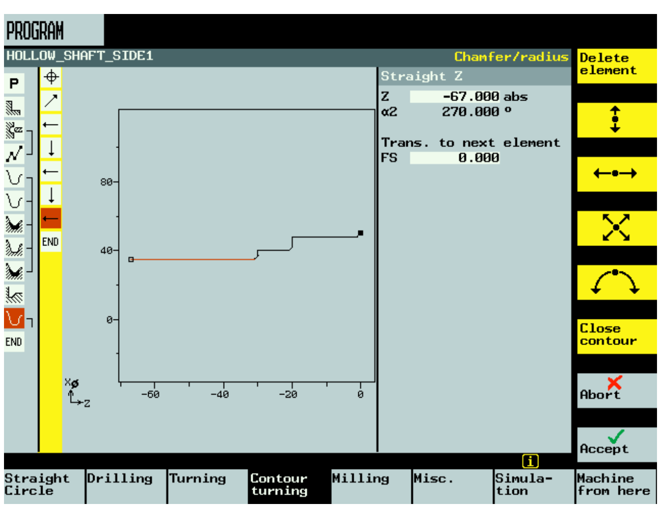
Simula-tion</td>
<td></td>
<td>
</td>
<td>• The Cut key is used to display the workpiece in the cut view.

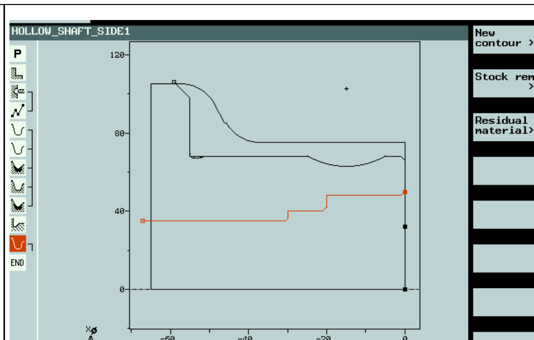
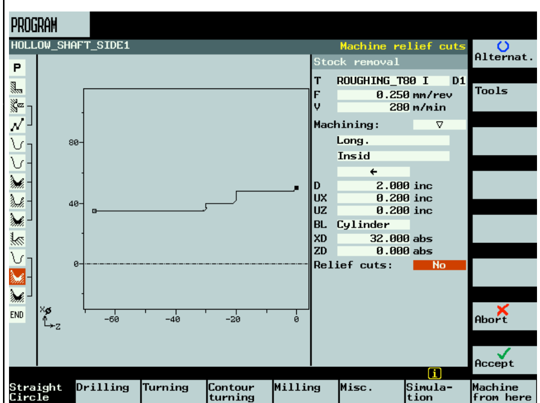
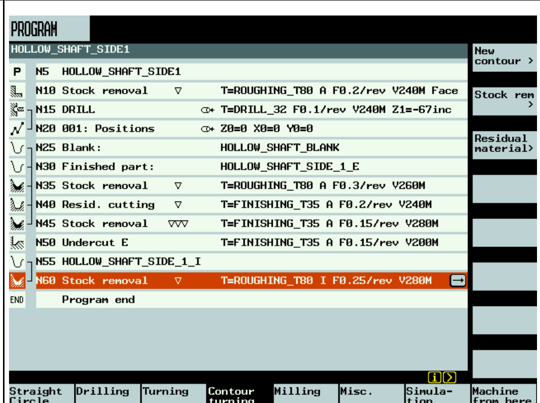
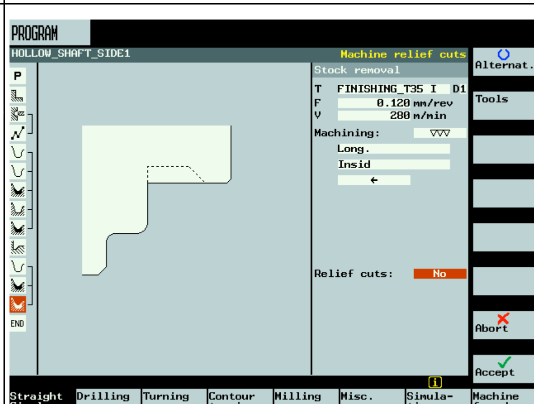
• End the simulation.</td>
</tr>
</table>

8.1.6 Machined Part Contour of the First Side Internal



Once the work schedule has been completed for the first external side, now construct the internal contour (red contour) of the first side.

Keys	Screen	Explanations
Contour turning New contour > 1_I		• Assign the name "HOLLOWSHAFT_SIDE_1_I" to the contour.
50 Accept		• Because the workpiece has already been faced, the starting point can be set to X50/Z0. This ensures that the areas already faced are not machined again.
		• Construct the internal contour to the Z value -67 shown on the left. • The chamfers and radiuses are constructed as a transition to the next element (because the first chamfer does not have any previous element, it is created as a path). • Finally, accept the contour in the work schedule.

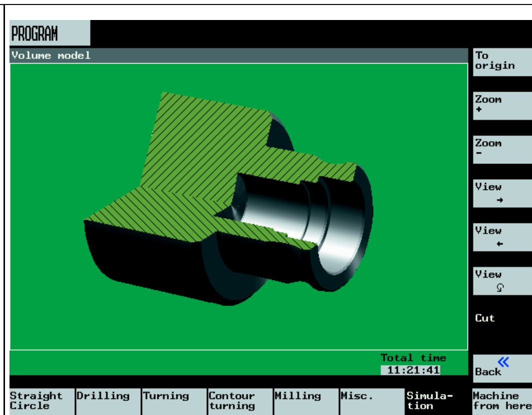
			• The geometric values should be present in the work schedule in this form.
Stock rem > Accept	…		• In this machining step, the size 80 inside roughing tool is used to cut to the constructed inside contour. • The machining must be changed to *inside*. • Because drilling has already been done there, no blank contour needs to be taken into consideration for the internal machining. Consequently, the *Blank description* field is switched to *Cylinder*.
			• The work schedule shows the important details of the machining steps. This permits a fast checking of the technological data.
Stock rem > Accept	…		• Finish the previously rough-cut contour here.

Simula-tion

Details >

Cut

Simula-tion



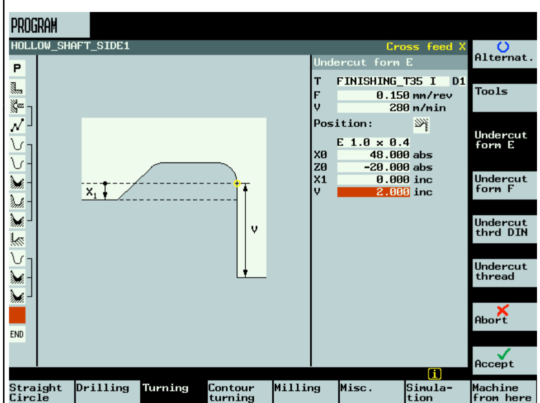
• Check the workpiece as produced this far by "playing" with the individual simulation views.

Turning

Undercut >

Accept

...



• The next step of the undercut form E is added to the end of this work schedule. Ensure the correct position of the undercut.

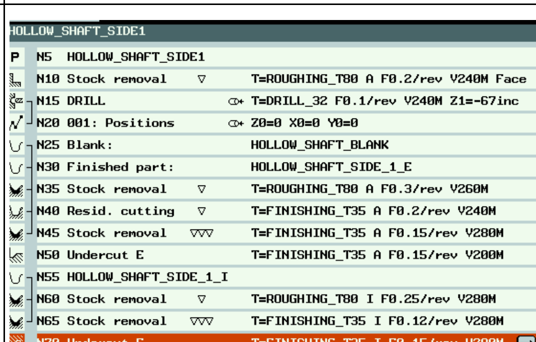
Simula-tion

Details >

Zoom +

Simula-tion

• Zooming the side view allows the traverse paths to be checked.



HOLLOW_SHAFT_SIDE1

P	N5 HOLLOW_SHAFT_SIDE1		
	N10 Stock removal	▽	T=ROUGHING_T80 A F0.2/rev V240M Face
	N15 DRILL		T=DRILL_32 F0.1/rev V240M Z1=-67inc
	N20 001: Positions		Z0=0 X0=0 Y0=0
	N25 Blank:		HOLLOW_SHAFT_BLANK
	N30 Finished part:		HOLLOW_SHAFT_SIDE_1_E
	N35 Stock removal	▽	T=ROUGHING_T80 A F0.3/rev V260M
	N40 Resid. cutting	▽	T=FINISHING_T35 A F0.2/rev V240M
	N45 Stock removal	▽▽▽	T=FINISHING_T35 A F0.15/rev V280M
	N50 Undercut E		T=FINISHING_T35 A F0.15/rev V200M
	N55 HOLLOW_SHAFT_SIDE_1_I		
	N60 Stock removal	▽	T=ROUGHING_T80 I F0.25/rev V280M
	N65 Stock removal	▽▽▽	T=FINISHING_T35 I F0.12/rev V280M
	N70 Undercut E		T=FINISHING_T35 I F0.15/rev V280M

• The complete work schedule for the first side of the workpiece looks like this:

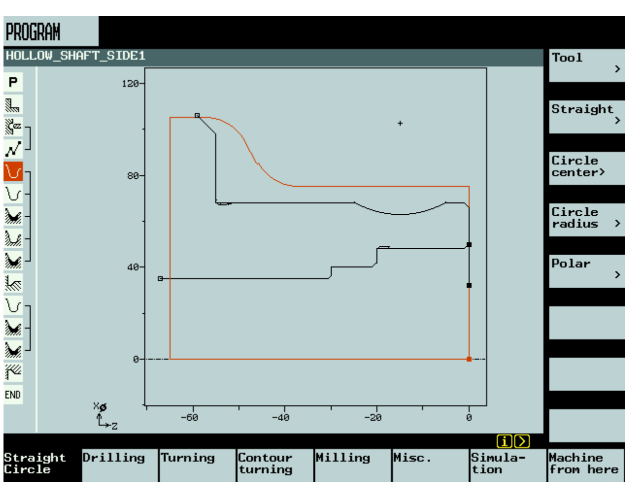
8.1.7 The Extended Editor

ShopTurn offers a number of special functions that allow the reuse and management of parts of the work schedule. These special functions can be reached at any time using the > key on the slimline operator panel or with the Shift+F9 key combination on the PC keyboard.

The descriptions of these functions follow:

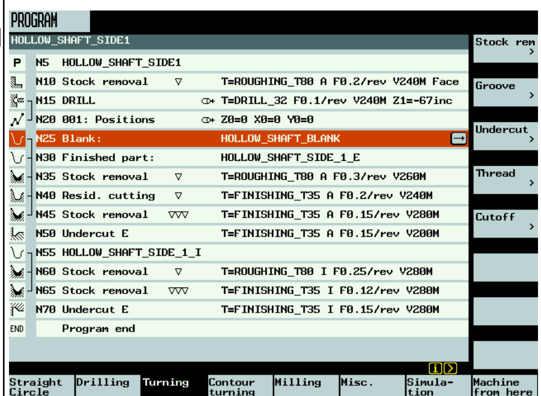
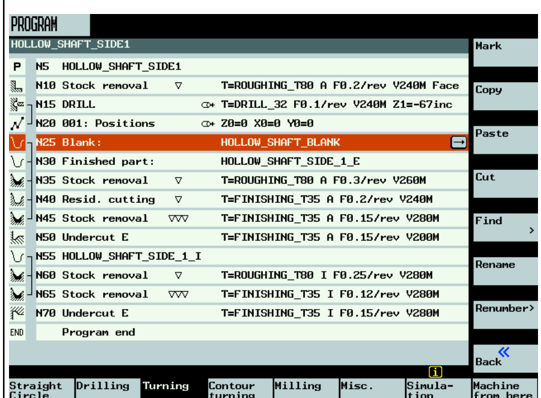
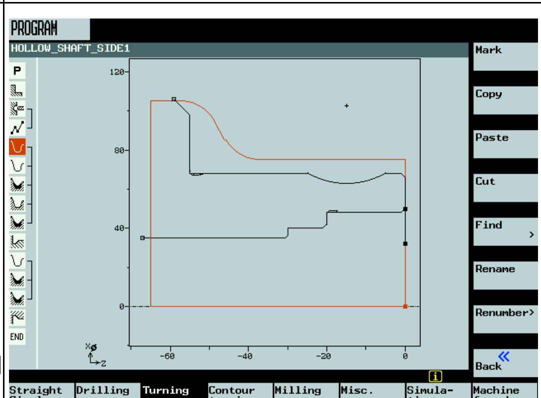
Mark	The *Mark* function allows several machining steps to be selected for further machining (e.g. *Copy* or *Cut*).
Copy	The *Copy* function copies machining steps to the intermediate storage.
Paste	The *Paste* function adds machining steps from the intermediate storage into the work schedule. The insertion is always made after the currently marked machining step.
Cut	The *Cut* function copies machining steps to the intermediate storage and at the same time deletes the original entry. This softkey can also be used for "pure" deletion.
Find >	The *Find* function searches for text in the program.
Rename	The *Rename* function changes the names of contours, directories or work schedules.
Renumber	The *Renumbers* function renumbers the machining steps.
« Back	The *Back* function returns to the previous menu.

One of the previously described functions is applied in the following section to also use the red blank contour in the following work schedule for the second side of this workpiece.



This (red) blank contour should now be copied into the intermediate storage of the control.

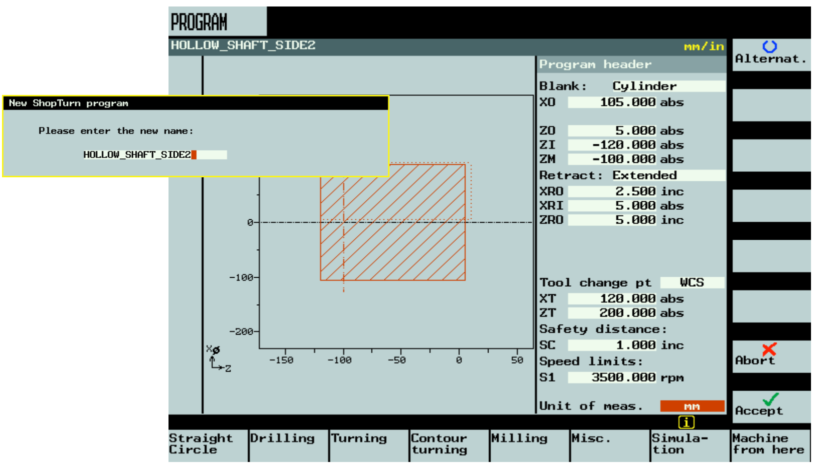
8.1.8 Copy a Contour

Keys	Screen	Explanations
↑		• Use the <arrow>-key to select the blank contour.
>		• The extended editor opens.
ⓘ Copy ↓		• Pressing the Copy key copies the selected contour into the control's intermediate storage. It remains stored there until the *Copy* or *Cut* key is pressed or the control is switched off.

8.2 Create the Second Workpiece Side

Create work schedule

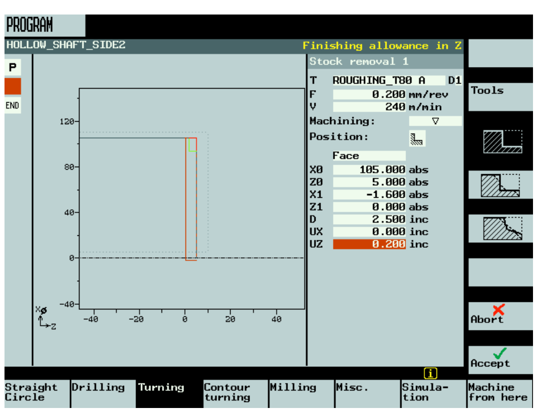
The work schedule for the second side of the workpiece is created in this section. A new work schedule with the name "HOLLOW_SHAFT_SIDE2" is created.

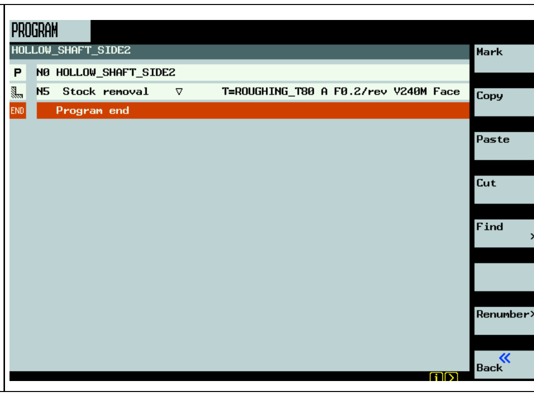


The program header is accepted after entering the data shown on the left.

8.2.1 Face Turning

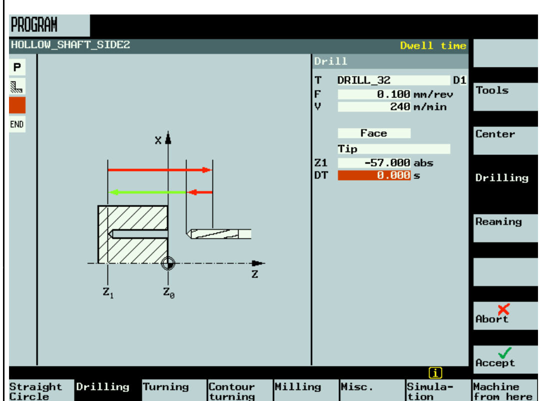
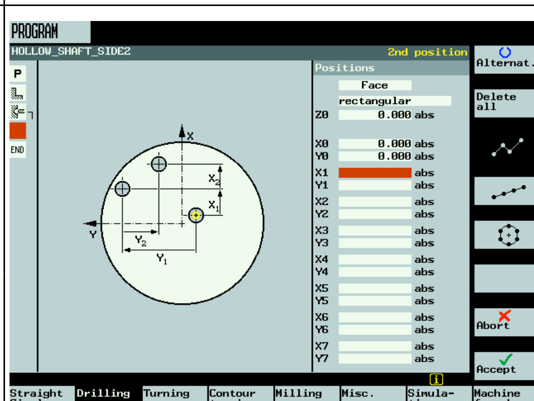
The workpiece is first faced to Z0 with an allowance of 0.5mm.

Keys		Bildschirm	Explanations
Turning Stock rem > Tools To program Accept	...		• The blank is faced to X- 1.6 and ZO. The finishing of the end face is performed later by machining the inside contour. 

			• View of the work schedule after accepting the first machining step.

8.2.2 Drilling

The workpiece is now drilled at the center.

Keys		Screen	Explanations
Drilling Drilling Reaming > Drilling Accept	...		• Enter the technological and geometric data for drilling as shown on the left in the picture. Because the first side has already been drilled to Z-67, drilling only needs to be done to Z-57.
			• The work schedule should now look like this:
Position > Accept	...		• Enter the position data for drilling as shown on the left in the picture.

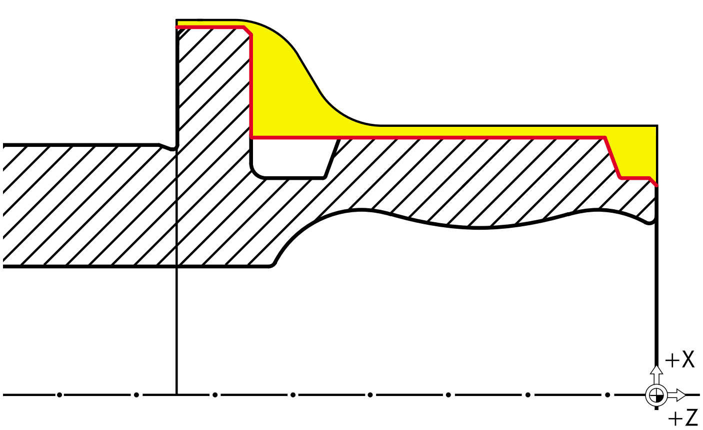
<table>
<tr><td></td><td></td><td>
</td><td>• Drilling technological and geometric data is automatically chained to each other in the work schedule.</td></tr>
</table>

8.2.3 Adding the Blank Contour

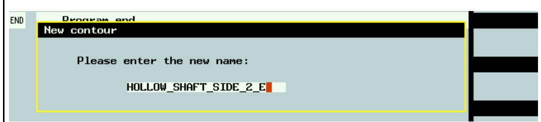
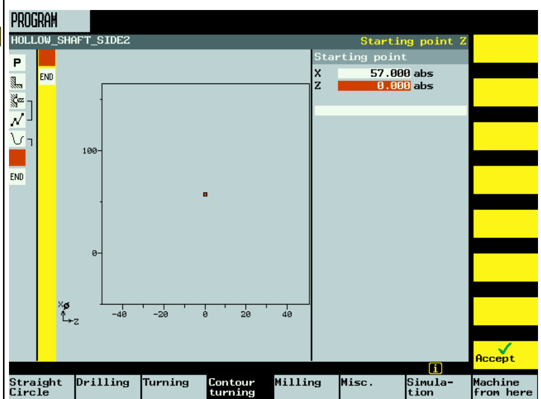
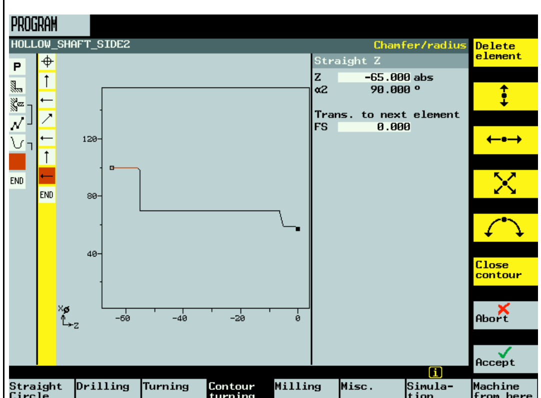
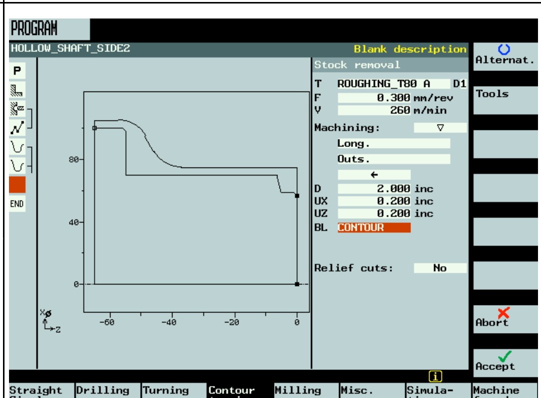
The blank contour contained in the intermediate storage is added in this section. If this is no longer in the intermediate storage, it must be re-copied from the first work schedule.

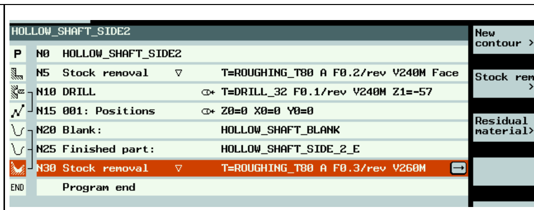
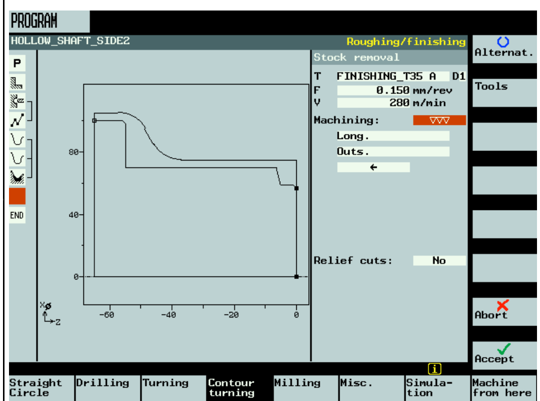
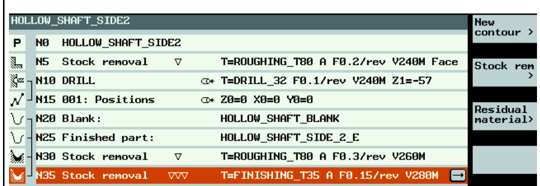
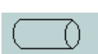
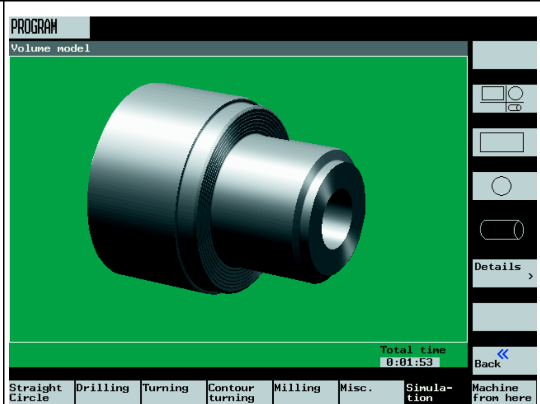
<table>
<tr><th colspan="2">Keys</th><th>Screen</th><th>Explanations</th></tr>
<tr><td>
Paste</td><td></td><td>
</td><td>• After calling the extended editor, the Paste key adds the blank contour to the work schedule.</td></tr>
<tr><td></td><td></td><td>
</td><td>• The machining steps are always added after the activated machining step.</td></tr>
</table>

8.2.4 Machined Part Contour of the Second Side Outside

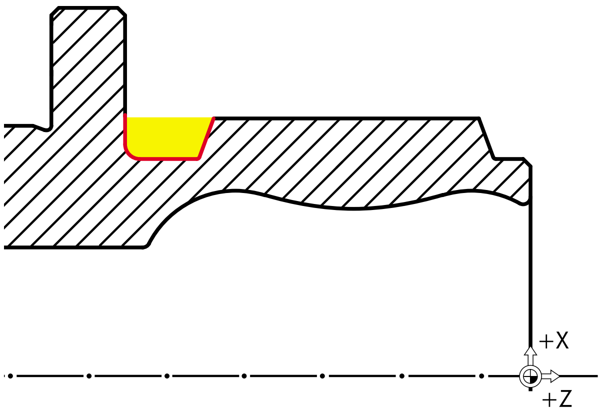


Once the blank contour has been added, the outer part of the workpiece is constructed.
The asymmetric groove will be produced later.

Keys		Screen	Explanations
Contour turning New contour >	2_E ⇒		• Assign the name "HOLLOW_SHAFT_SIDE_2_E" to the contour of the second outer side.
Accept	57 ⇒		• The starting point of the contour lies at the start of the chamfer (X57/Z0).
Accept			• Construct the contour yourself to the end point at Z-65 and X100.
Stock rem > Tools To program Accept	...		• Enter the data for the rough cutting of the outside contour.

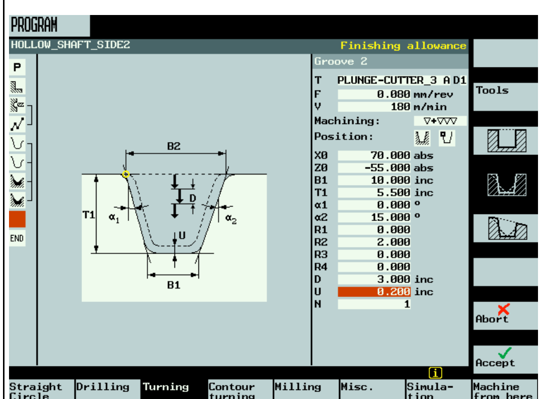
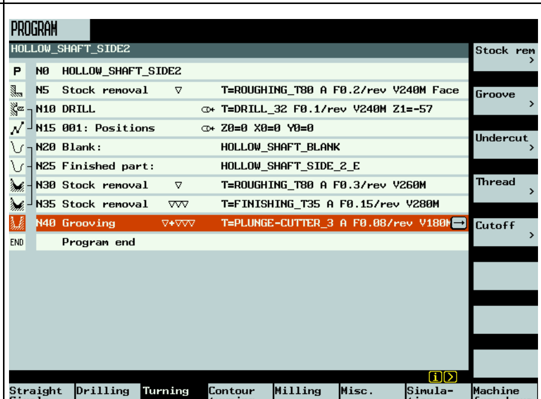
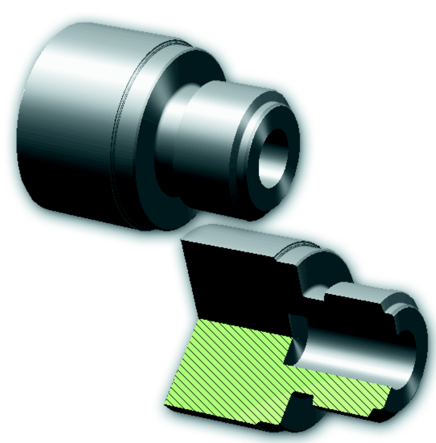
			• The cutting machining step is again automatically appended to the contours.
Stock rem > Tools To program Accept	...		• Enter the data for *finishing* the outside contour.
			• Append the finishing machining step to the rough-cutting machining step.
Simula-tion  Simula-tion			• Representation of the second side in the *Volume model*

8.2.5 Create an Asymmetric Groove

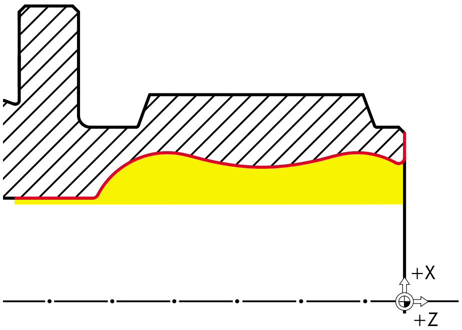


The yellow area is produced as an asymmetric groove in this section.

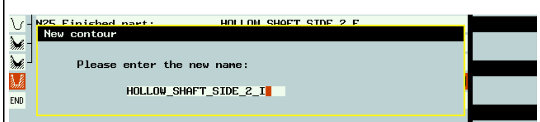
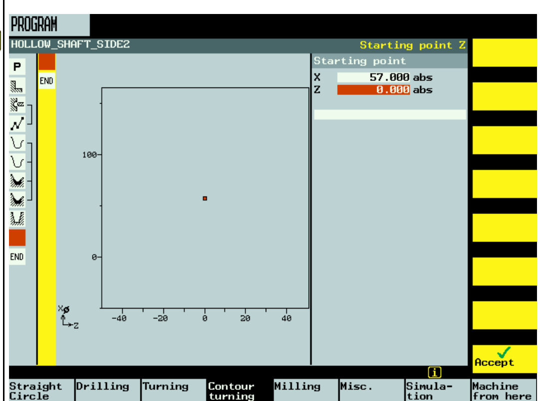
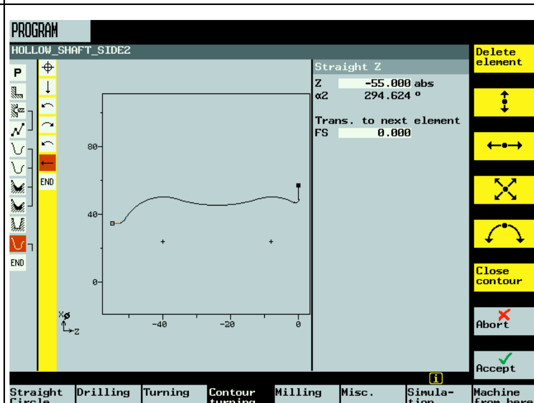
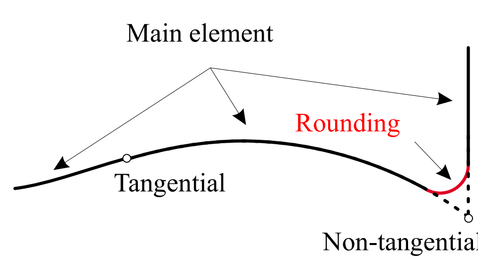
<table>
<tr><th colspan="2">Keys</th><th>Screen</th><th>Explanations</th></tr>
<tr><td>Turning
Groove >

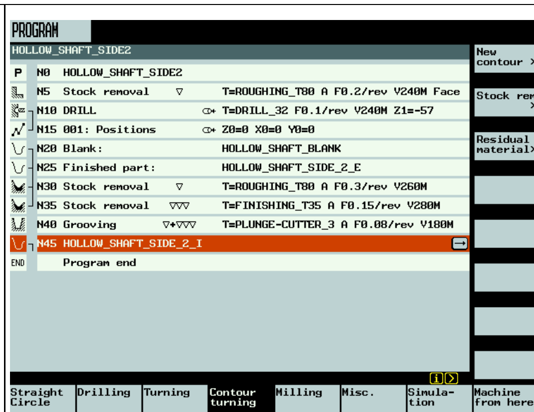
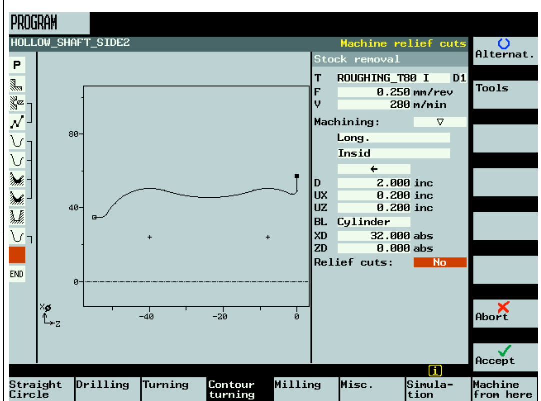
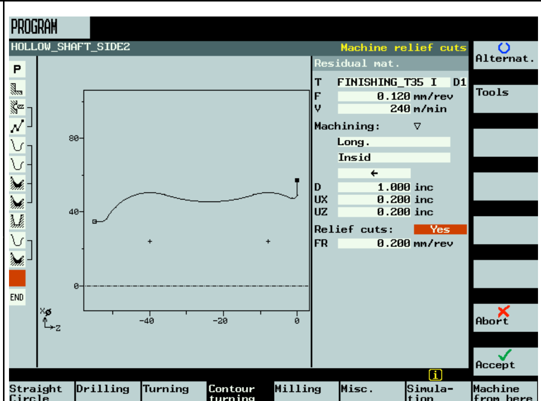
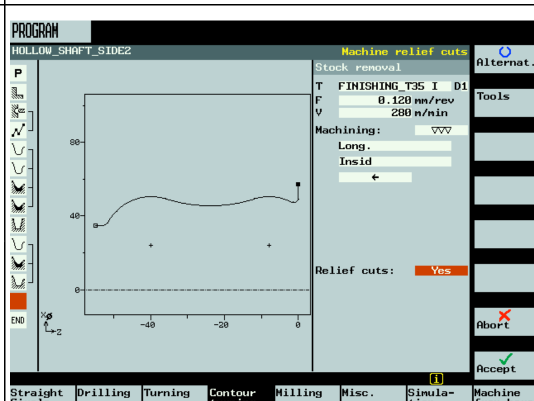
...
Accept</td><td></td><td>
</td><td>• Select the appropriate tool and enter the values shown on the left for the groove.</td></tr>
<tr><td></td><td></td><td>
</td><td>• This machining step now completes the outside machining of the second workpiece side.
</td></tr>
</table>

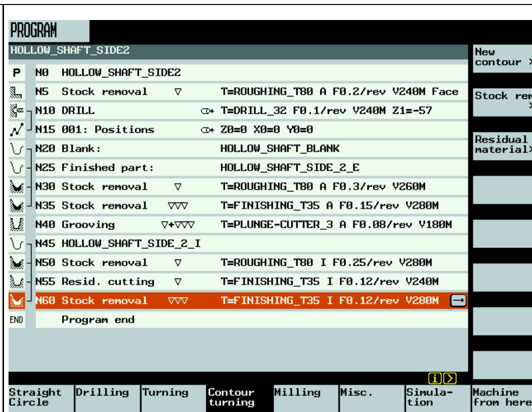
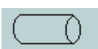
8.2.6 Machined Part Contour of the Second Side Inside

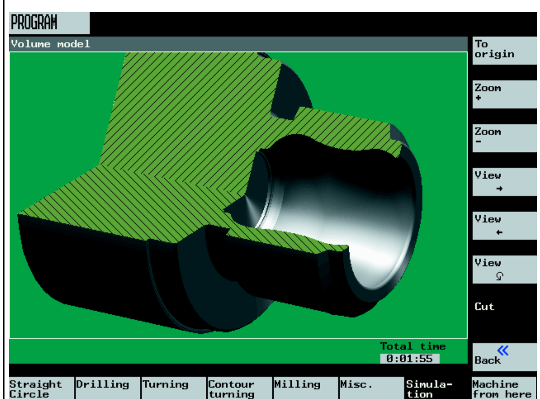


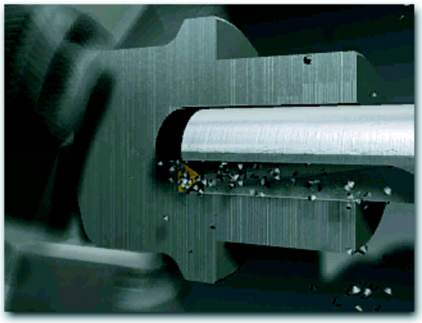
The inside of the previously drilled part is machined in this section. This requires the creation of the inside contour.

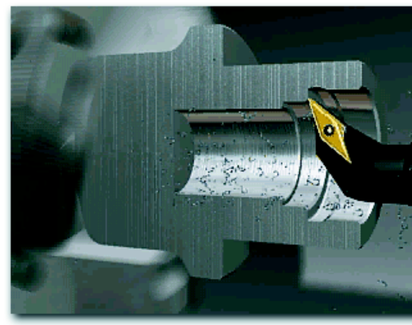
Keys		Screen	Explanations
Contour turning New contour >	2_I ⇔		• Assign the contour the name "HOLLOW_SHAFT_SIDE_2_I".
Accept	57 ⇔		• Because the end face still has a finishing allowance, the starting point is placed at X57 and Z0. In this way, the end face is then machined for the subsequent contour turning. Because the first side has already been machined to Z-67, the contour ends at Z-55.
Accept			• Care must be taken during the creation of the contour that the arc elements merge tangentially. • Note: The tangential transition applies only to the main elements, i.e. a rounding is appended to the main element. 

Softkey		Screen	Description
			• The inside contour has been accepted in the work schedule.
Stock rem > Tools To program Accept	…		• Use the FINISHING_TOOL_80_I tool for the inside rough cutting. • Switch the machining to *Inside*. • To ensure that the 80°-insert does not insert, the *Relief cut* setting should be changed to *No*.
Residual material> Tools To program Accept	…		• Use the FINISHING_T35_I tool for the residual material.
Stock rem > Tools To program Accept	…		• Then use the FINISHING_T35_I tool to finish the inside contour.

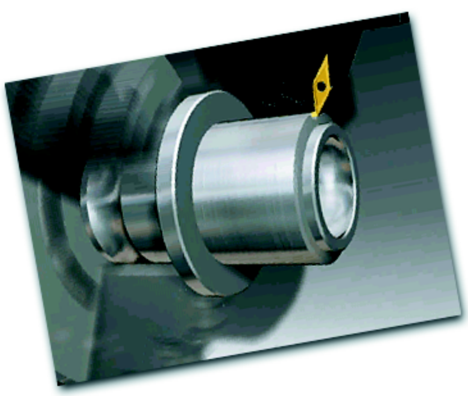
<table>
<tr>
<td>Simula-
tion</td>
<td></td>
<td>
</td>
<td>• Now simulate the complete work schedule.</td>
</tr>
<tr>
<td>
Details >
Cut

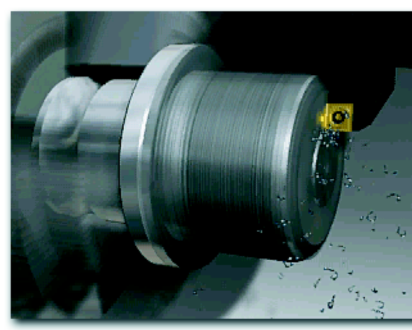
Simula-
tion</td>
<td></td>
<td>
</td>
<td>• To check the internal machining, the cut workpiece can be viewed.</td>
</tr>
</table>



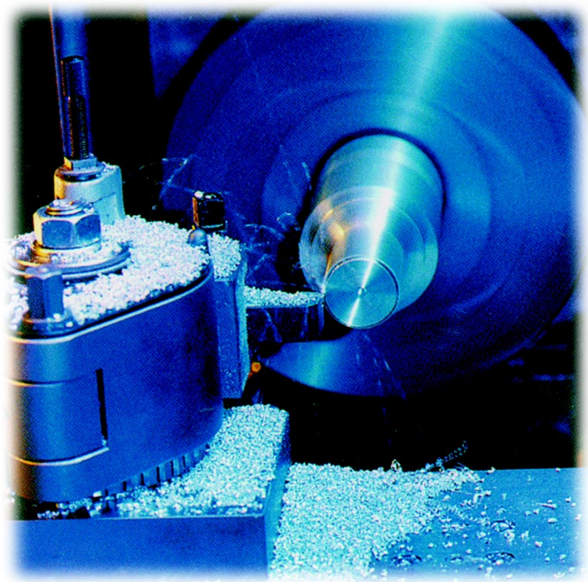


Finally, some photographs
of a hollow shaft in production ...





9 And Now We Produce



Now having worked through the examples, you have a consolidated knowledge of the creation of work schedules in ShopTurn. The production of the workpieces follows.

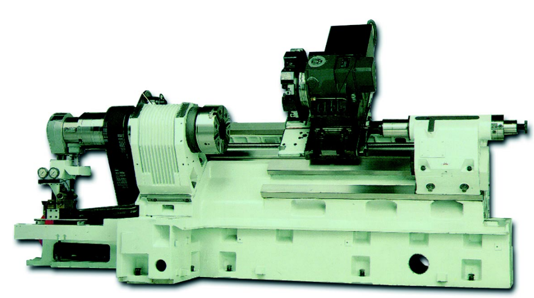
9.1 Approach refrence point

After switching on the control, the reference point of the machine must be approached before running the work schedules or before traversing manually. This enables ShopTurn to find the counting begin in the position measuring system of the machine.

Because the approaching of the reference point depends on the machine type and manufacturer, only a few general notes can be given here:

1. Fahren Sie das Werkzeug ggf. auf eine freie Stelle im Arbeitsraum, von wo aus in alle Richtungen kollisionsfrei verfahren werden kann. Achten Sie dabei darauf, dass das Werkzeug danach nicht bereits hinter dem Referenzpunkt der jeweiligen Achse liegt (da das Anfahren des Referenzpunktes je Achse nur in einer Richtung erfolgt, kann dieser Punkt sonst nicht erreicht werden).

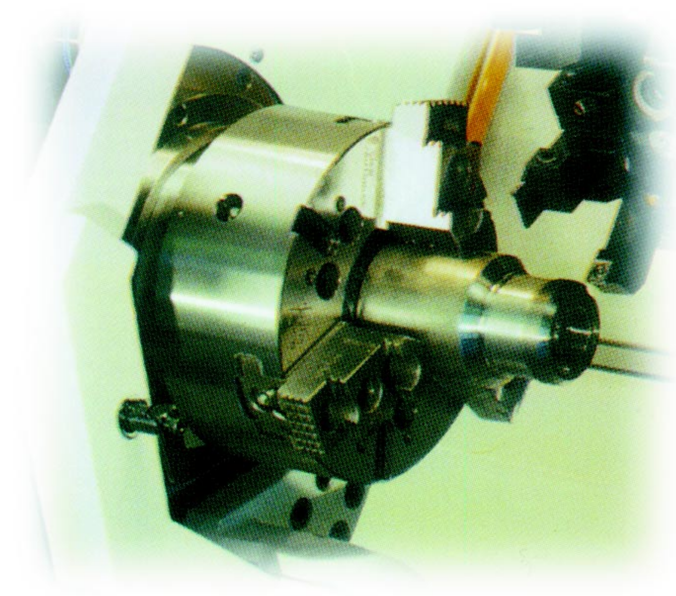
2. Führen Sie das Anfahren des Referenzpunktes exakt nach den Angaben des Maschinenherstellers durch.



9.2 Clamp Workpiece

To ensure the production adheres to the specified dimensions and obviously also for your safety, a fixed clamping appropriate for the workpiece is necessary.

Normally a three-block chuck is used here.

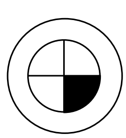


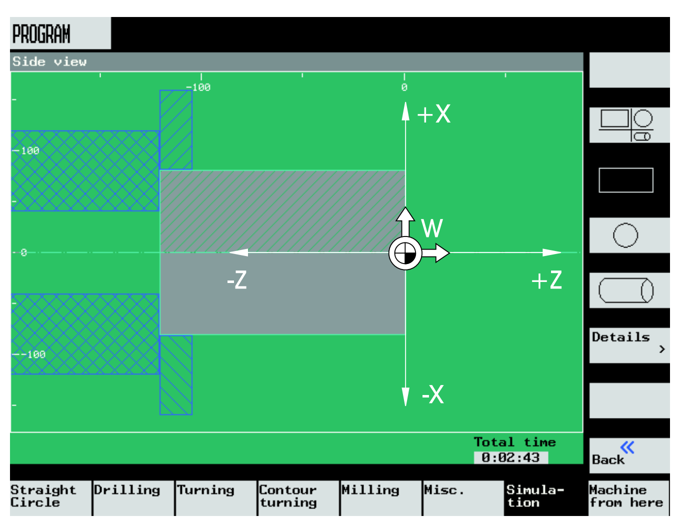
9.3 Set Workpiece Zero Point

Because the position of the workpiece in the Z-direction is not known, you must determine the workpiece zero point in Z. Because it is a rotating part, the center point is always X0 relative to the X-axis.

In the Z-axis, the workpiece zero point is normally determined by scratching using a calculated tool.

Symbol for the workpiece zero point W

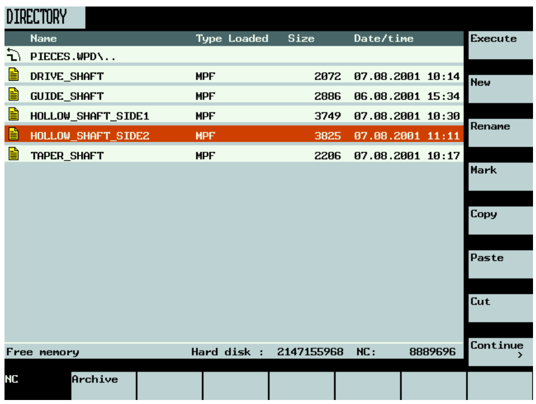


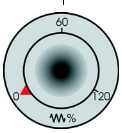


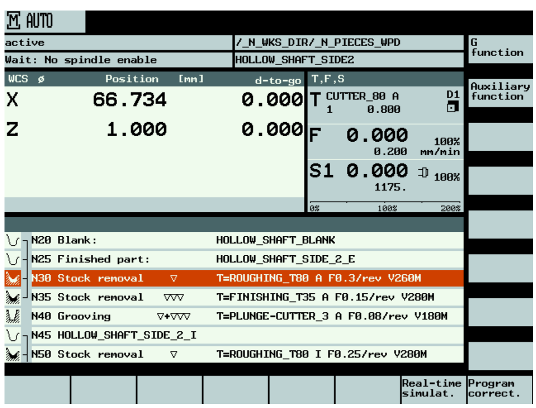
9.4 Execute Work Schedule

The machine has now been prepared, the workpiece is setup and the tools have been measured (refer to Chapter 4). Now at last we can start:

<table>
<tr><th>Keys</th><th>Screen</th><th>Explanations</th></tr>
<tr><td>
Program manager

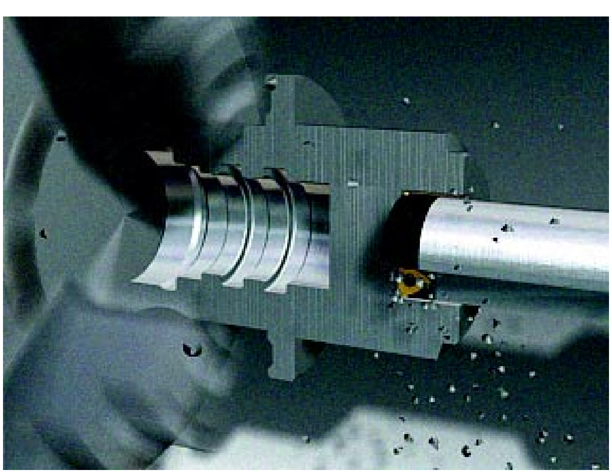
Execute</td><td>
</td><td>• Select the directory that contains the required work schedule. This is the WERKSTUECKE (=WORKPIECES) directory for the examples in this document.

• The Execute key loads the work schedule in the AUTO mode.</td></tr>
<tr><td>
Record

</td><td>
</td><td>• Because the work schedule has not yet been run in a controlled manner, set the feed potentiometer to zero so that you have "everything under control" from the beginning.

• If you also want to view a simulation during the production, the Trace function must have been selected prior to the start. This is essential to ensure that all traverse paths are displayed.

• Start the production with the  key and use the feed potentiometer to check the speed of the tool movements.</td></tr>
</table>



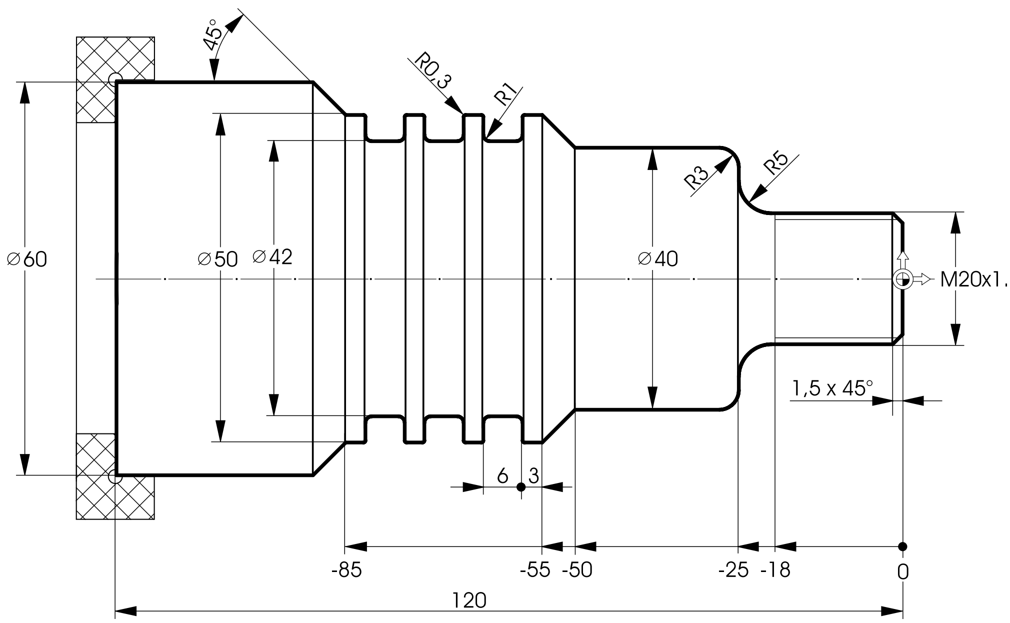
As easily and as fast as you have created these workpieces with ShopTurn...

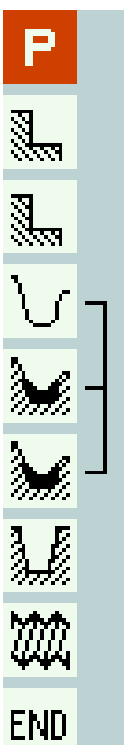
... you will now also be able to create YOUR workpieces with ShopTurn.

10 How Fit Are You With ShopTurn?

The following 4 exercises form the basis for your personal test for working with ShopTurn. As an aid, a possible work schedule is displayed for each of the exercises. The mentioned times are based on the procedure appropriate for this work schedule. Please consider the mentioned times as a general orientation for your answer to the above question.

Exercise 1: Can you manage this with ShopTurn in 10 minutes?

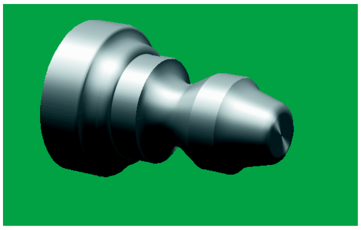


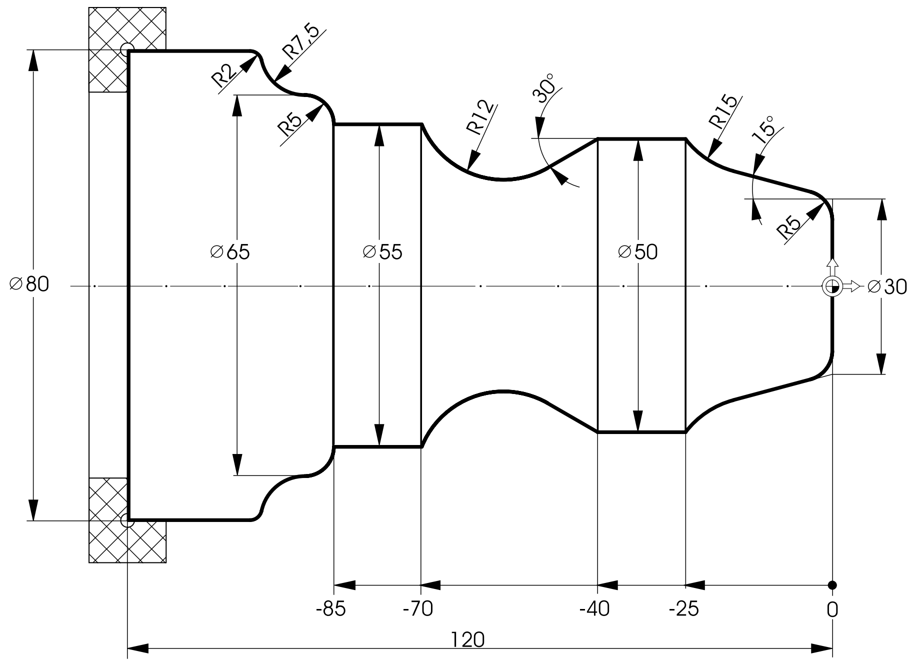


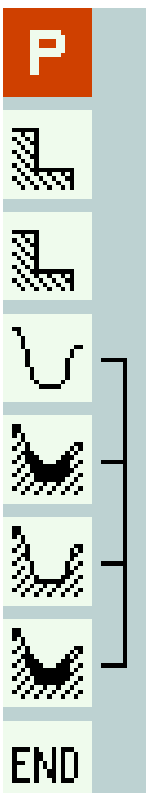
EXERCISE_1			
P N0	EXERCISE_1		
N5	Stock removal	▽	T=ROUGHING_T80 A F0.25/rev V240M Face
N10	Stock removal	▽▽▽	T=ROUGHING_T80 A F0.2/rev V280M Face
N15	CONTOUR_1		
N20	Stock removal	▽	T=ROUGHING_T80 A F0.3/rev V260M
N25	Stock removal	▽▽▽	T=FINISHING_T35 A F0.15/rev V280M
N30	Grooving	▽+▽▽▽	T=PLUNGE-CUTTER_3 A F0.1/rev V150M N3
N35	Thread long.	▽	T=THREADING_T1.5 P1.5mm S800U Outs
END	Program end		

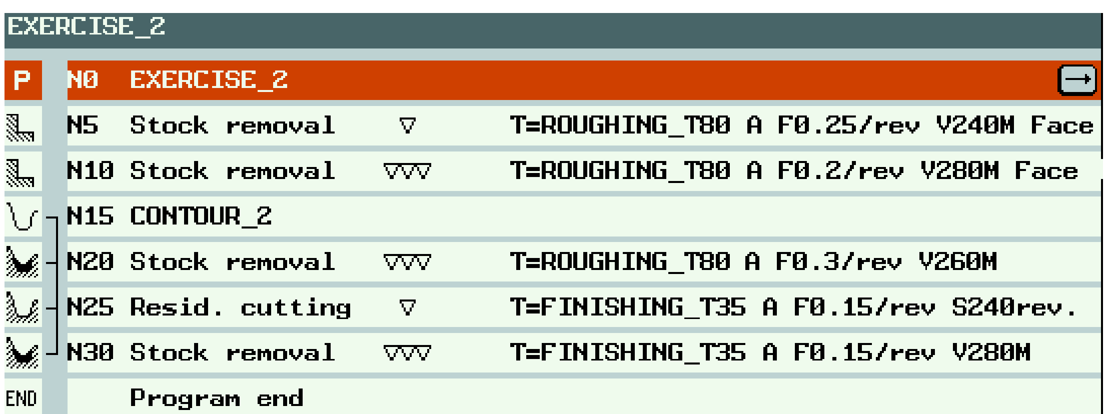
In this work schedule, the workpiece is milled to size in two machining steps. This means that the starting point of CONTOUR_1 can be placed at the start of the first chamfer.

Exercise 2: Can you manage this with ShopTurn in 10 minutes?







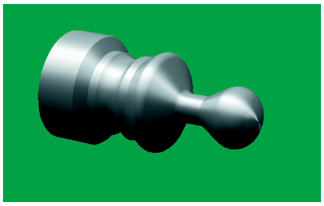


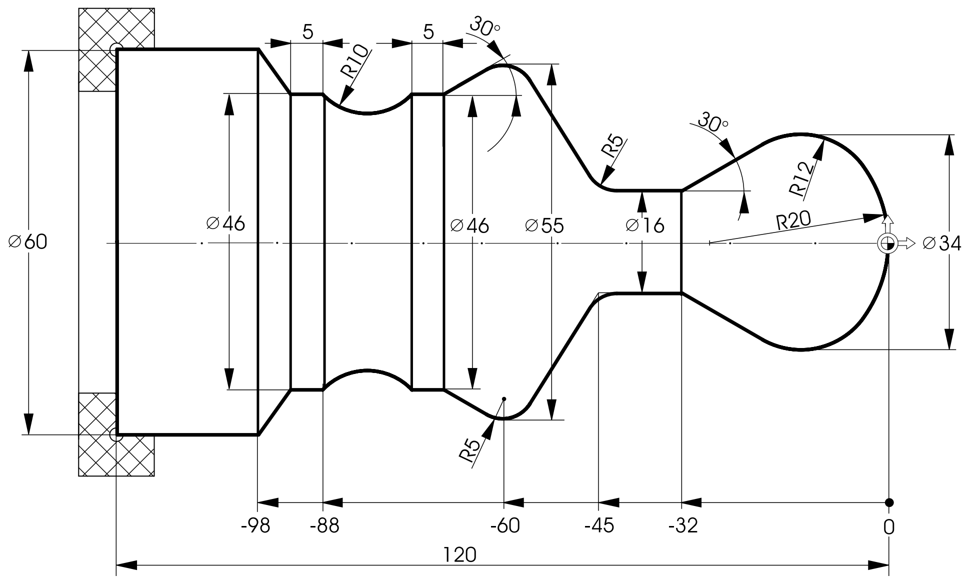
EXERCISE_2

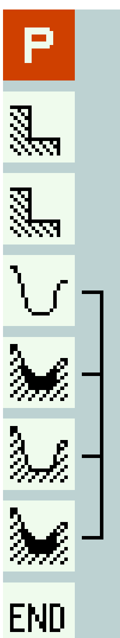
P	N0	EXERCISE_2		
	N5	Stock removal	∇	T=ROUGHING_T80 A F0.25/rev V240M Face
	N10	Stock removal	∇∇∇	T=ROUGHING_T80 A F0.2/rev V280M Face
	N15	CONTOUR_2		
	N20	Stock removal	∇∇∇	T=ROUGHING_T80 A F0.3/rev V260M
	N25	Resid. cutting	∇	T=FINISHING_T35 A F0.15/rev S240rev.
	N30	Stock removal	∇∇∇	T=FINISHING_T35 A F0.15/rev V280M
END		Program end		

The automatic cutting of residual material can be used optimally here.

Exercise 3: Can you manage this with ShopTurn in 10 minutes?





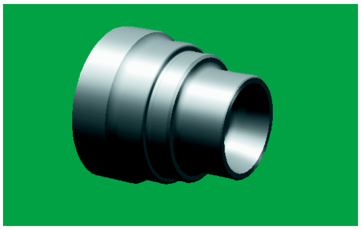


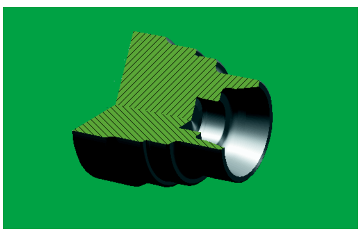
EXERCISE_3

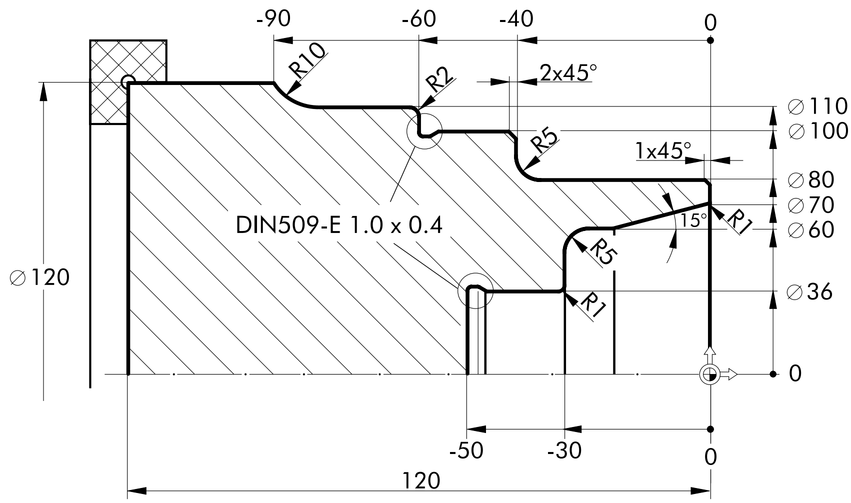
P	N0	EXERCISE_3		
	N5	Stock removal	▽	T=ROUGHING_T80 A F0.25/rev V240M Face
	N10	Stock removal	▽▽▽	T=ROUGHING_T80 A F0.25/rev V280M Face
	N15	CONTOUR_3		
	N20	Stock removal	▽	T=ROUGHING_T80 A F0.3/rev V260M
	N40	Resid. cutting	▽	T=BUTTON_TOOL_8 A F0.2/rev S240rev.
	N25	Stock removal	▽▽▽	T=FINISHING_T35 A F0.15/rev V280M
END		Program end		

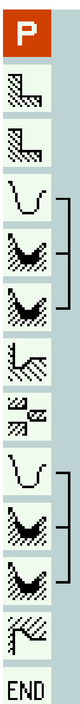
Note: Create the radius 5 in two steps.

Exercise 4: Can you manage this with ShopTurn in 15 minutes?









EXERCISE_4		
P	N0 EXERCISE_4	
	N5 Stock removal ▽	T=ROUGHING_T80 A F0.25/rev V240M Face
	N10 Stock removal ▽▽▽	T=ROUGHING_T80 A F0.2/rev V280M Face
	N15 CONTOUR_4	
	N20 Stock removal ▽	T=ROUGHING_T80 A F0.3/rev V260M
	N25 Stock removal ▽▽▽	T=FINISHING_T35 A F0.15/rev V280M
	N30 Undercut E	T=FINISHING_T35 A F0.15/rev V200M
	N35 Drilling centric	T=DRILL_32 F150/min V120M Z0=0
	N35 CONTOUR_4_I	
	N45 Stock removal ▽	T=ROUGHING_T80 I F0.3/rev V260M
	N50 Stock removal ▽▽▽	T=FINISHING_T35 I F0.15/rev V280M
	N55 Undercut E	T=FINISHING_T35 I F0.15/rev V200M
END	Program end	

In this work schedule, the end face is initially rough cut and finished. The complete outside area, including the undercut, is then produced.

The inner part of the contour is then machined. The starting point of the inside contour is then placed at X70/ Z0. The extended editor can use cut & paste to simply swap the outside and inside machining.

Index

Numerics
810D/840D control 17

A
Absolute dimensions 19
Accept Dialog 58
Advantages of ShopTurn 5
Alarms 16
All Parameters 49
Alternative key 11
approach refrence point 98
Arbitrary blank 56
Axes 18

B
Back 86
Blank Contour 77
Blank description 61
Blank form 31

C
Call Tool 33
Cartesian 20
C-axis 69
Centering 69
Chaining 15
Clamp Workpiece 99
CNC ISO 17
Complete machining 69
Contour 15
Contour calculator 56
Contour computer 7
Coolant 24
Copy 86
Copying work schedules 14
Create program 31
Create work schedule 45
Cut 9, 86
cut view 82
Cutting 15, 50
Cutting speed 6
Cylinder 31

D
Degressive 67
Depth reference point 70
Details 53
Dialog acceptance 48
Dialog call 32
Dialog selection 48, 58
Directory 14, 31
Drilling 69, 76, 89
Duplo-Number 24

E
End face view 43
Enter Traverse Path 33
Error number 16
Execute Work Schedule 100
Exporting files 14

F
Facing 45, 88
Flank angle 65
form blank 80

G
Geometrical Basics 18
Graphical work schedule 6
Groove 64

H
Hole positions 70

I
Importing files 14
Incremental dimensions 19
Information key 11
Innen-Bearbeitung 84
Input key 11
Insert 86
Insertion type 73
inside contour 94
intermediate storage 87

L

Learning time 5
Location inhibit 25

M

Machine Control Panel 11
Machine control panel 11
Machine zero point 18
Magazine 16, 27
Magazine List 25
Main menu 12
Manual operation 13
Mark 86
Measure tool 28
Measuring tools 13
Messages 16
Moving work schedules 14

N

New 86

P

Paste 9
Pictures 5
Pipe 31
Plandrehen 75
Planfräsen 75
Points in the Working Range 18
Polar 20
potentiometer 100
production 98
Production time 8
Program header 31
Program management 30, 31
Program Manager 14, 31
Programming time 6

R

Rectangular Pocket 72
Reference point 18
Relief cut 61
Relief cuts 62
Relief cutting 51
Rename 86
Renaming work schedules 14
Replacement tool 24
Residual material 8, 50, 62
Return plane 8, 32
Rotational direction 24

S

Safety clearance 32
Search 86
Setting up the machine 13
Shank 70
Side view 43
Simulation 34
Simulation - details 43
Slimline operator panel 11
Softkeys 12
Speed limitation 22
Start-Taste 100

T

Table book 22, 23
Thread 52, 67
Tool List 24
Tool list 16
Tool Management 24
Tool name 24
Tool state 25
Tool type 24
Tool types 26
Tool Wear List 25
Tools for the examples 26
Trace 100

U

Undercut form E, F, DIN thread 81

V

Values 6
Volume model 43

W

Window view 43
Work schedule managing 14
Workpiece Zero Point 99
Workpiece zero point 13, 18, 29

Z

Zero point offset 28, 29
Zero points 16
Zoom 43

Index of figures

We are grateful to

DMG

Verlag Europa-Lehrmittel

Iscar

Krupp-Widia

Röhm

Sandvik

Seco

Walter AG

for providing the pictures on pages 6, 11, 12, 16, 22, 23, 25, 26, 27, 96, 97 and 99.

Siemens AG

Automation and Drives

Motion Control Systeme

P.O. Box 3180, D-91050 Erlangen

Germany

Siemens Aktiengesellschaft Order No.: 6FC5095-0AA80-0BP0