Training Document for
Integrated Automation Solutions
Totally Integrated Automation (TIA)

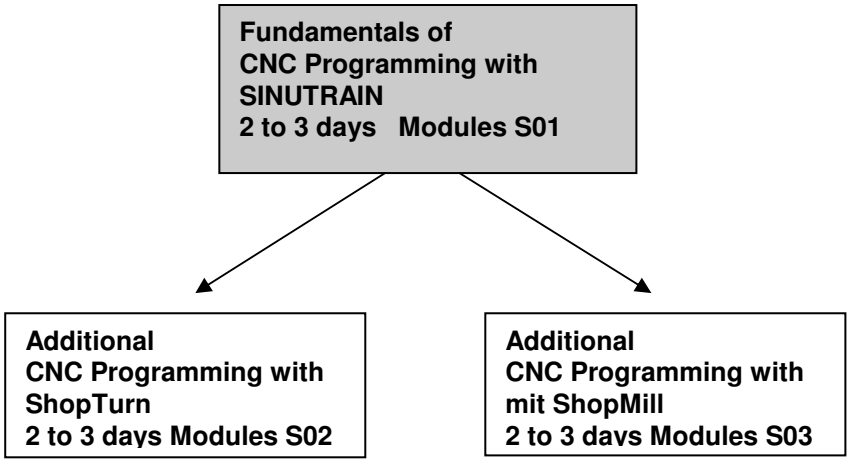
Module S01
Fundamentals of CNC Programming
with SinuTrain

We thank the Michael Dziallas Engineering corporation, the teachers of vocational schools others for their support during the preparation of this document.

Table of Contents

1 Preface

This training document **'Fundamentals of CNC Programming'** is the prerequisite for working with the additional modules with SINUTRAIN.



Today, CNC controllers are considered the most essential part of any automation. Depending on the problem definition, the most varied tasks in the areas of turning, milling, lasering, and grinding as well as in many other areas can be carried out economically with the controllers ShopMill and ShopTurn.
.

Training objective:

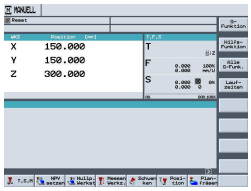
The CNC S01module is structured in a way that you have before you a completely programmed workpiece in G-code programming (DIN 66025).
The same workpiece is generated step by step with this document according to the machining step programming. In this way, you will learn the simple operation of this software, and control and optimize it with the simulation.

Required Hardware and Software

1 PC, operating system Windows XP Professional starting with SP1 with 500 MHz and 256 MB RAM, free disk storage approx. 400 MB, of that 50 MB on the system drive, 1GB for installing all products, MS Internet Explorer starting with 6.0

2 Software SINUTRAIN 802D/ 810D/840D/840Di/Programming &Training, SinuTrain/JopShop

1 PC



2. SINUTRAIN

2 Introduction

2.1 Development Phases of CNC Technology

- At the beginning of the eighties, first CNC machines with simple controllers
- In the middle of the eighties, more powerful controllers with cycles because of faster processors as well as machine tools with greater processing speed
- At the end of the eighties, machine tools with 5 and more axes and special software tools for external programming by using CAD/CAM systems
- At the beginning of the nineties, flexible manufacturing systems with extensive supplementary functions such as palette systems and multiple clamping with multiple spindle drives
- In the middle of the nineties, continued development of tool systems and the use of special tools for processing complex workpiece contours with only one tool
- End of the nineties: central programming systems for programming several different controllers at different machine tools

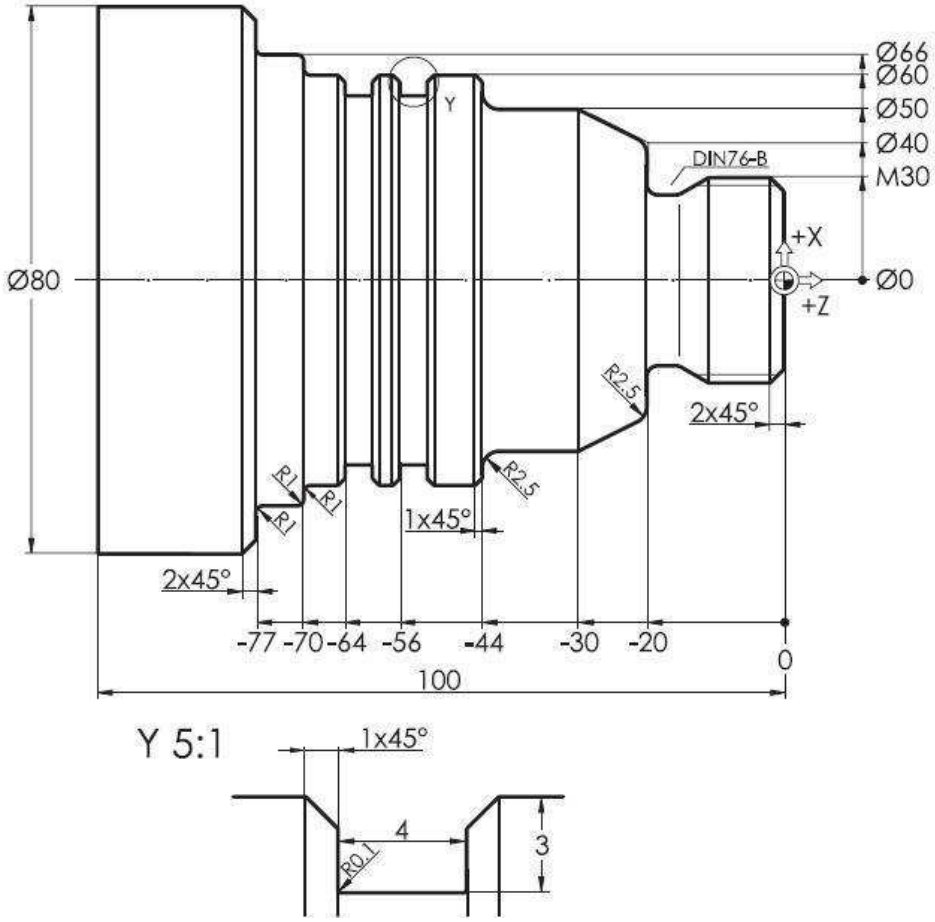
2.2 Controller Requirements for the New Millennium

- Openness: It is to be possible for the machine manufacturer or the user to configure and expand controllers according to their own requirements
- Independence: Programming by means of a uniform controller interface for the most varied CNC processing
- Equality: All machine data is to be available also at the external programming units. Programming at the external programming units is the same as on the machine tool.
- Saving programming time: With graphic machining plans and help displays, it is to be possible to generate complex workpiece contours very easily and quickly
- Editing capability: Extensive editing functions provide for fast and simple program changes/program expansion

2.3 Advantages of CNC Programming with SinuTrain SHOPMILL, SHOPTURN

The controller is continuously optimized and can be adapted any time to the individual requirements of the machine manufacturers. Moreover, cycles and functions can be integrated later.
Regardless of whether turning, milling, or any other type of processing is performed, always the same program interface and the same menus or functions are used.
Retrofit: This means: Siemens can retrofit also older CNC machines to ShopMill and ShopTurn.
Advantage: Operating the software and the menu structure has to be learned only once.
By transferring the machine data to the programming system of SINUTRAIN, programming at the external programming unit is the same as on the machine tool.
By using contour calculators and CAD readers, simple programming is possible without technical terms. By directly entering technological values, no external calculations have to be made beforehand. The integrated contour calculator is able to process all conceivable dimensions, yet is very simple to handle. Through work step programming and many online help functions, extensive programming tasks can be solved very quickly.
Convenient programming is possible with functions such as Copy, Cut, and Insert. Since the program is generated in the editor as a graphic machining plan by means of individual work steps, all editing steps are provided in a straightforward arrangement.

3 Programming a Turned Part in G-Code



WERKSTUECK_G_COD 30

```
T="SCHRUPPER_80"¶
G96 S250 LIMS=5000 M4 M8¶
G18 G54 G90¶
G0 X84 Z.2¶
G1 X-1.6¶
G0 X80 Z2¶
CYCLE95("Kontur",2.5,0.1,0.5,0.1,0.3,0.15,,1,0,0,1)¶
G0 X200 Z5 ¶
T="SCHLICHTER_35"¶
G96 S320 M4 M8¶
G0 X-1.8 Z1¶
G42¶
G1 X0 Z0¶
Kontur_Schl¶
G40¶
G0 X200 Z100¶
T="GEWINDESTAHL"¶
G95 M3 M8¶
G0 X30 Z8¶
CYCLE97(1.5,,0,-17,30,30,3,2,0.92,0.03,30,0,8,2,1,1,0)¶
G0 X200 Z100 ¶
T="STECHER"¶
G96 S120 F.07 M4 M8¶
G0 X62 Z-56¶
CYCLE93(60,-56,4,3,0,0,0,-1,-1,0.1,0.1,0.1,0.1,3,0.2,11,0)¶
G0 X200 Z100¶
M30¶
¶
==eof==
```

KONTUR

```
G18 G90 DIAMON¶
G1 X24 Z1¶
G1 X30 Z-2¶
G1 Z-20¶
G1 X40 RND=2.5¶
G1 X50 Z-30¶
G1 Z-44 RND=2.5¶
G1 X60 CHR=1¶
G1 Z-70 RND=1¶
G1 X66 RND=1 ¶
G1 Z-75 RND=1¶
G1 X76¶
G1 X82 Z-78¶
M17¶
```

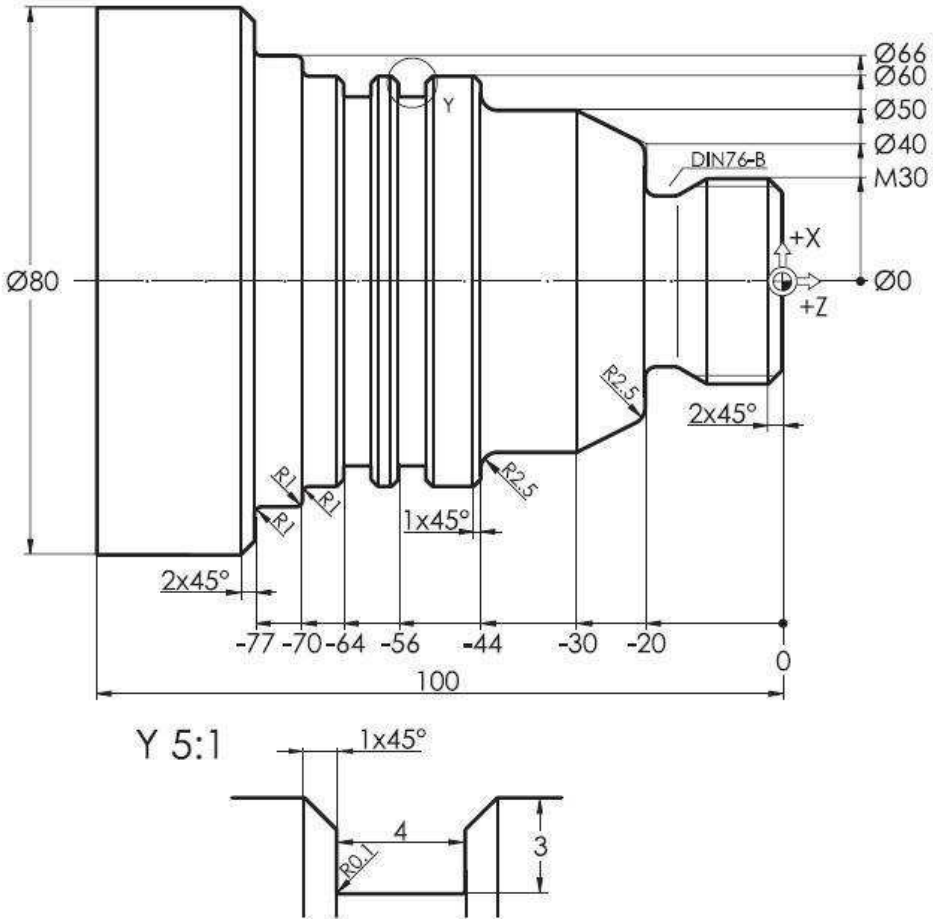
KONTUR_SCHL

```
G18 G90 DIAMON¶
G1 X0 Z0¶
G1 X26¶
G1 X30 Z-2¶
G1 Z-16¶
G1 X27.7 Z-18 RND=.8¶
G1 Z-20 RND=.8¶
G1 X40 RND=2.5¶
G1 X50 Z-30¶
G1 Z-44 RND=2.5¶
G1 X60 CHR=1¶
G1 Z-70 RND=1¶
G1 X66 RND=1 ¶
G1 Z-75 RND=1¶
G1 X76¶
G1 X82 Z-78¶
M17¶
```

Schrupper = roughing tool
Schlichter = finishing tool
Gewindestahl = threading tool
Stecher = grooving tool

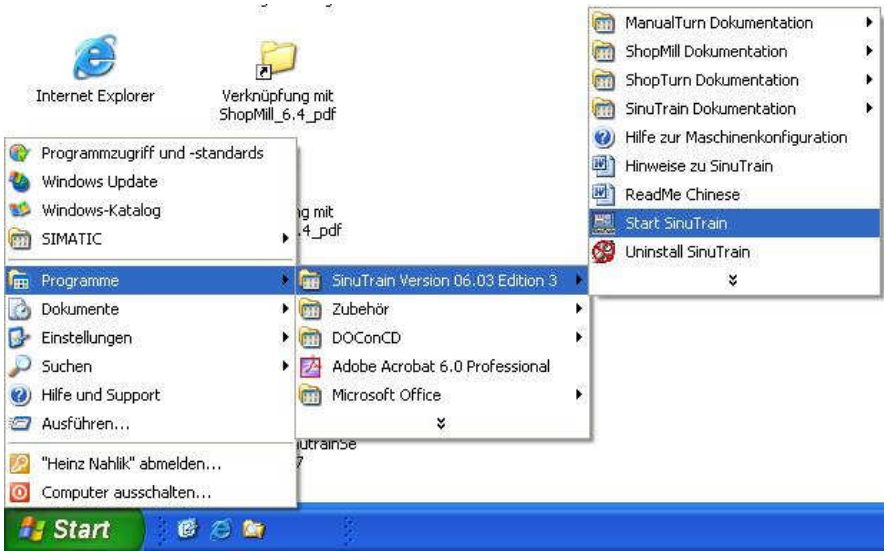
4 Programming a Turned Part according to the Machining Step Procedure

Using this drawing, we will generate a program in the machining step procedure.



4.1 Starting SINUTRAIN

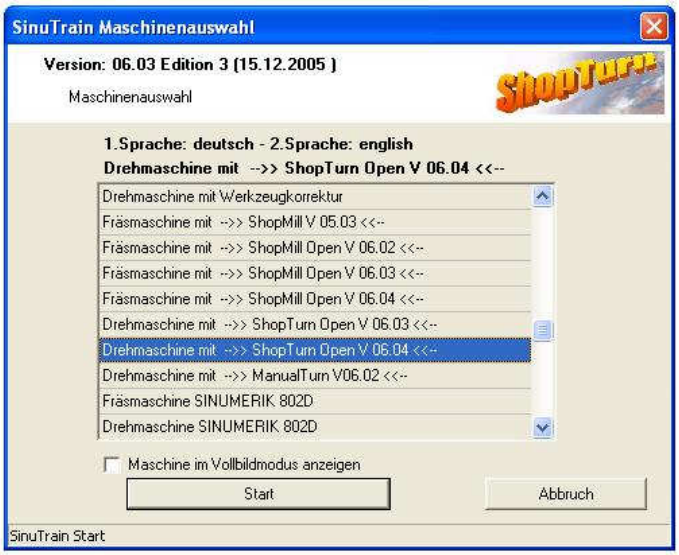
To start SinuTrain, double click on  Start SinuTrain ,
or start SinuTrain using the start bar.



4.2 Selecting the Machine

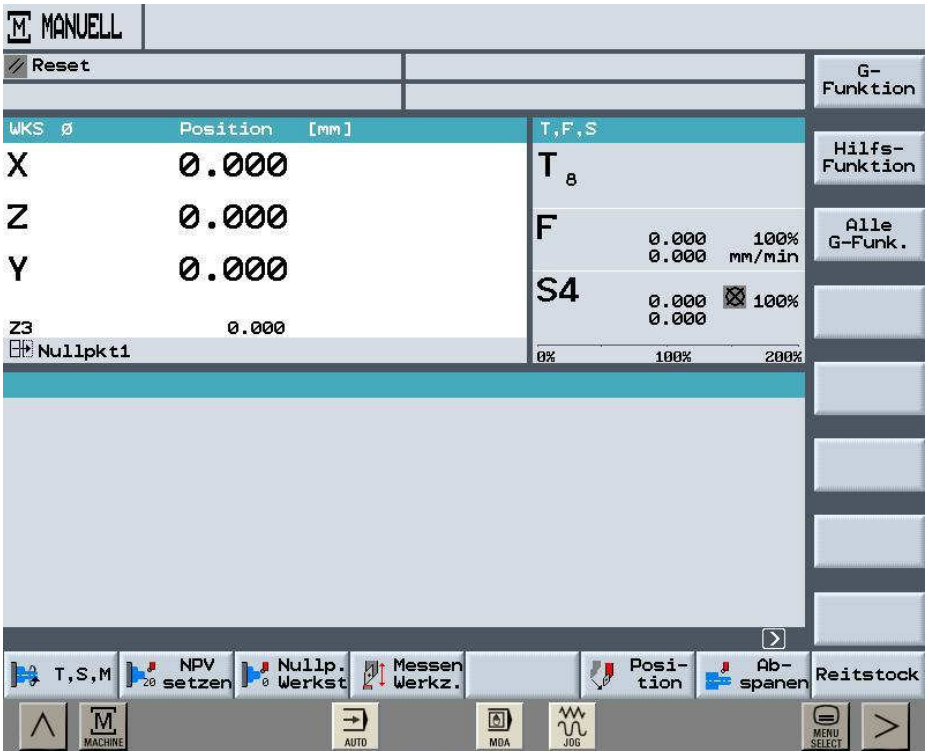
After booting up the computer, a menu is displayed with a selection of machines with the respective version number.

1. Select turning machine with >> ShopTurn Open V 06.04 <<
2. Click the Start button



4.3 SINUTRAIN Basic Display

After selecting the machine, the basic display of SINUTRAIN appears.
From here, all SINUTRAIN actions are started.



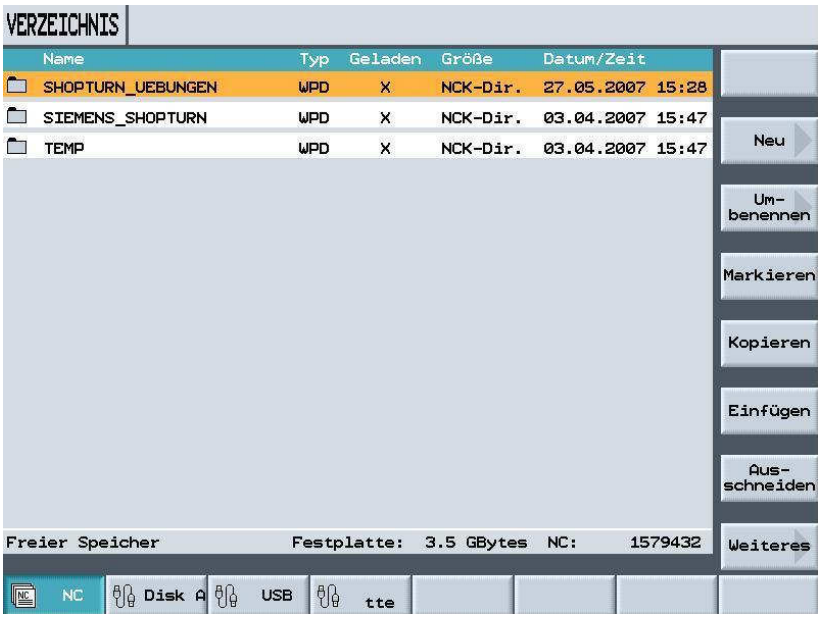
4.4 Generating the Turning Program

4.5 Setting Up a New Directory/Program Generation

Various actions can be performed in the Program Manager
(for example: New, Rename, Copy, etc.)

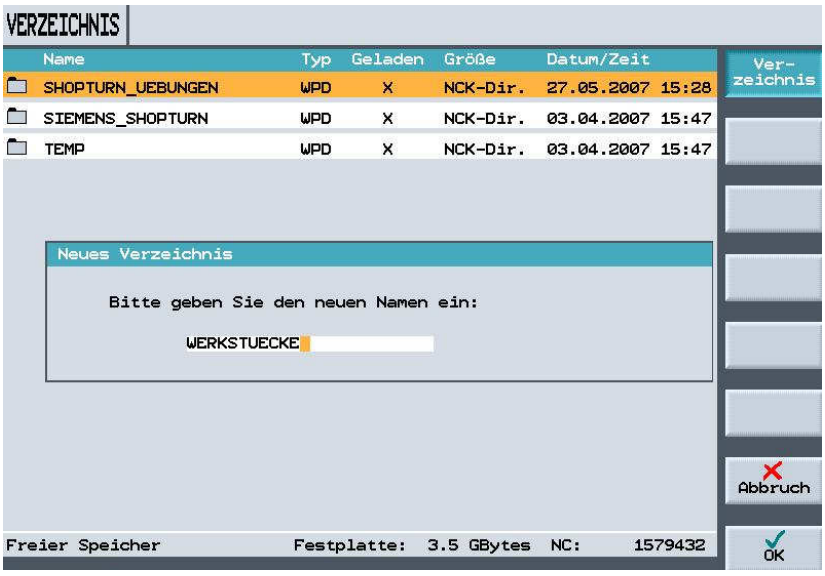
By pressing the softkeys 

, the available directories are opened in the operating area **DIRECTORY** <<**VERZEICHNIS**>> under ShopTurn.



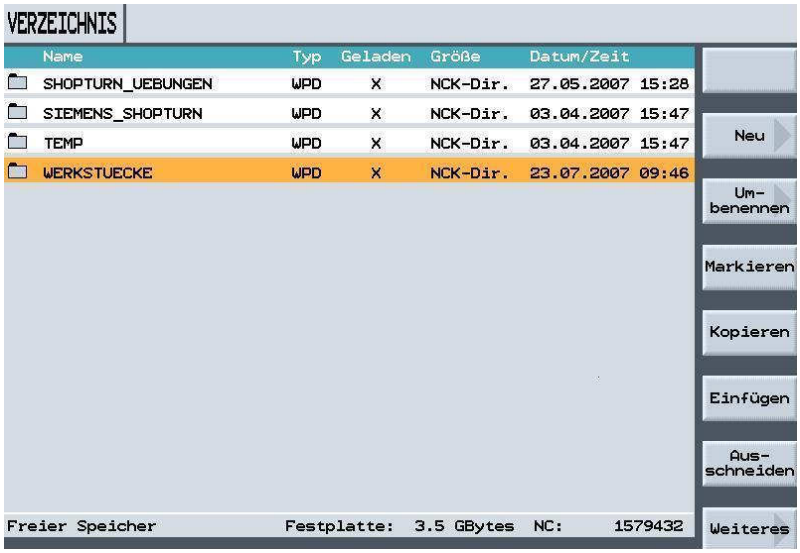
With the softkey 

, we are now setting up a new directory with the name "Work pieces“ <<Werkstuecke>>.



Confirm the input with **OK**.

The directory with the name “Work pieces“ is set up.



By pressing the arrow key  on the CNC keyboard or with the cursor key toward the right, the selected directory opens.

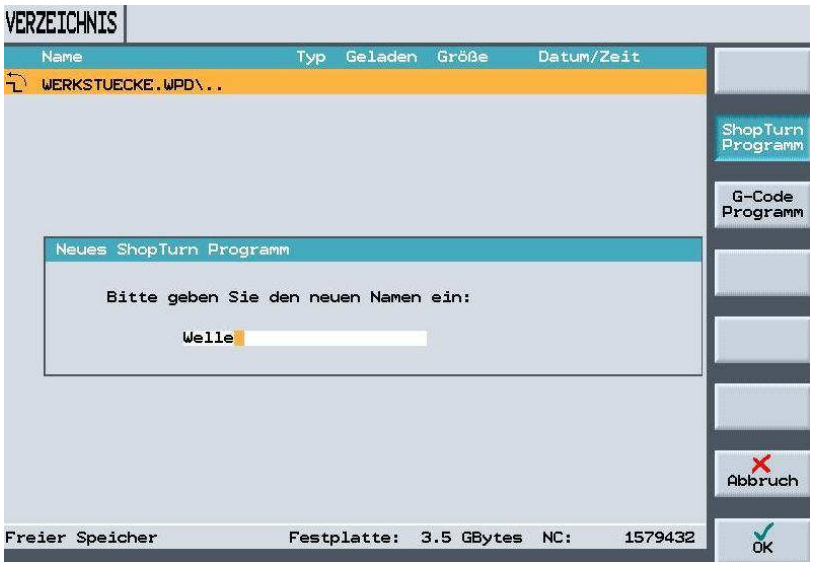
Note
The cursor keys can be used for navigating in SinuTrain.

4.6 Setting Up a New Workpiece Program

After the directory is opened, existing programs are displayed, or new programs can be set up.

By pressing the softkey 

and the selection Shopturn or G-Code (vertical softkeys), a new program name “Name of the machining plan“ is entered,
in our case “Shaft“ <<“Welle“>>.



Confirm input with **OK**.

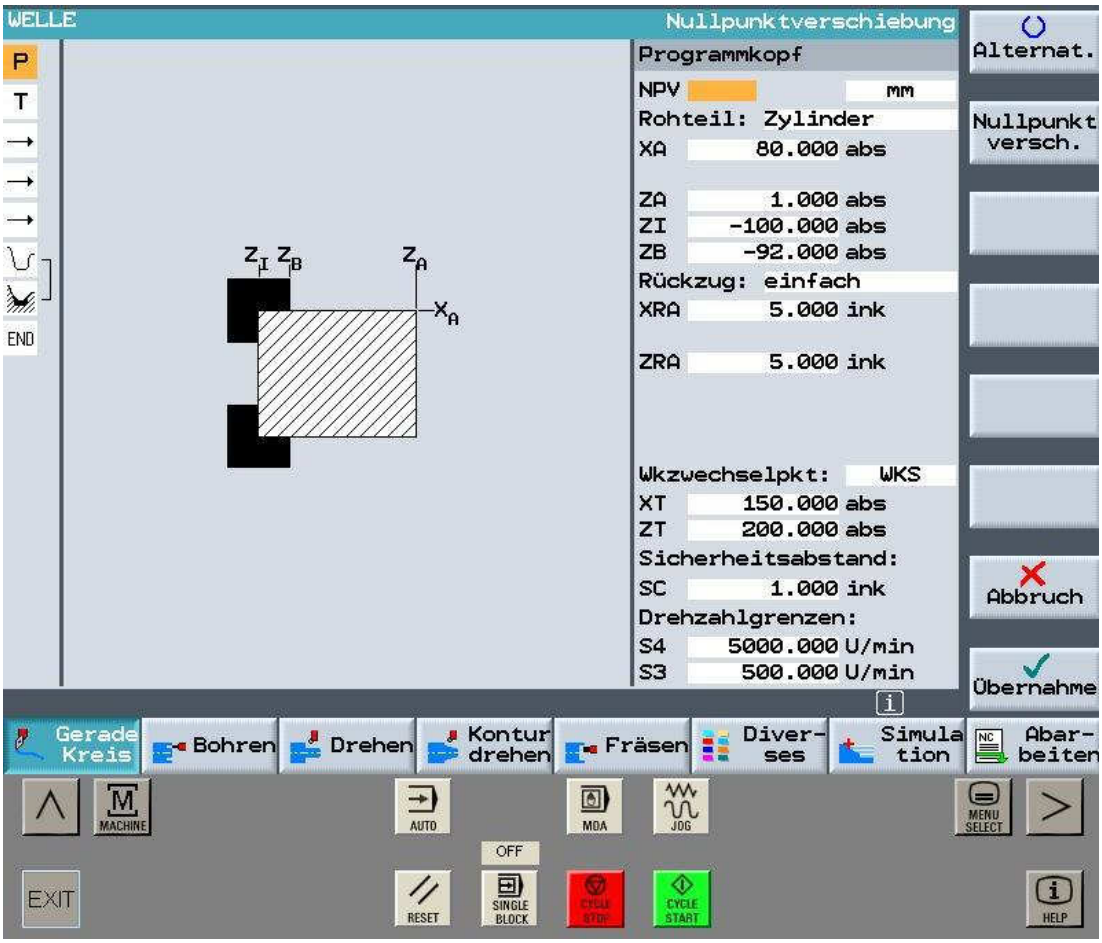
4.7 Program Header

The program header of the new program opens automatically. In the program header, the following is entered: the workpiece data, zero shift, raw part pattern, distance to the chuck, retraction planes, tool change point, safety clearance incremental or absolute, and the spindle speed limitation.

.

With the softkey 

you can now call the associated help display.



Make all inputs as indicated in the display above

and accept 

the input into the machining plan.

Note

With the softkey 

you can alternate, for example, between different zero points, units of mass, mm, inch, raw part shapes, cylinder, pipe, rectangle, corner, etc.
The blue line above always shows you the current input type.

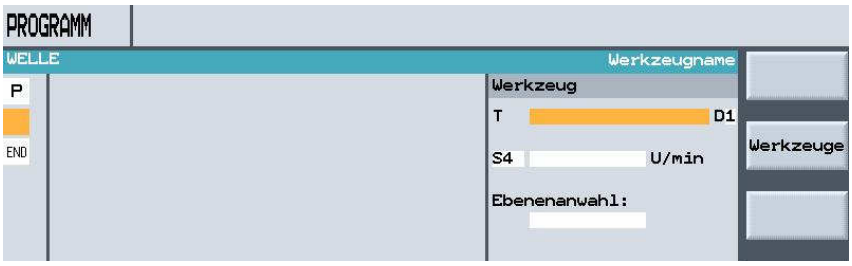
4.8 Program Editor

Entered/edited in the program editor in the form of machining steps.

4.9 Tool Selection

First, you need a tool for face turning and for roughing the outside contour.

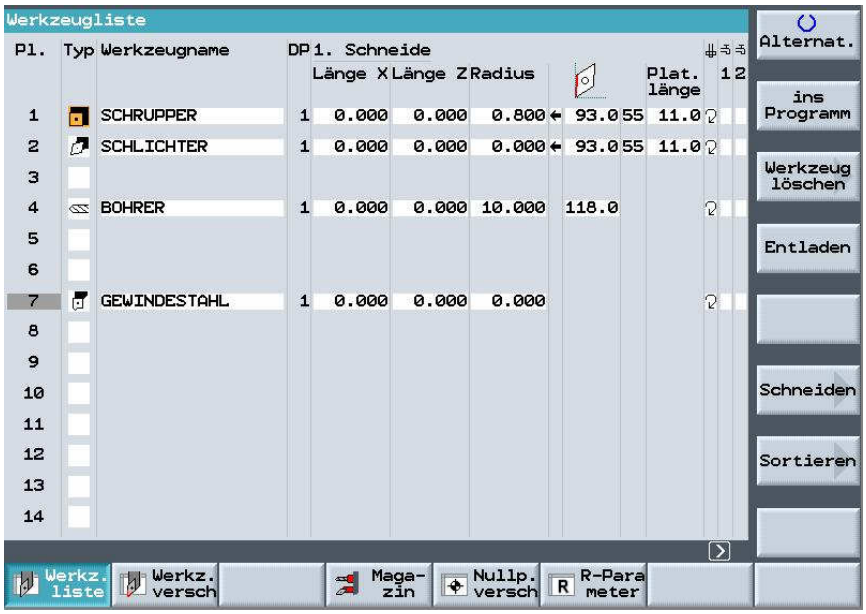
Click on 



Then, by clicking on “Tools“ again, call the tool list. Here, the tools that have already been set up are provided. But you can also create new ones.
Accept the roughing tool. The finishing tool and the threading tool are added later from the tool list. The grooving tool has to be set up, however.

Click on 

ROUGHING TOOL 



Now, enter the data for the constant cutting speed and the plane.

Accept the values with the input key.

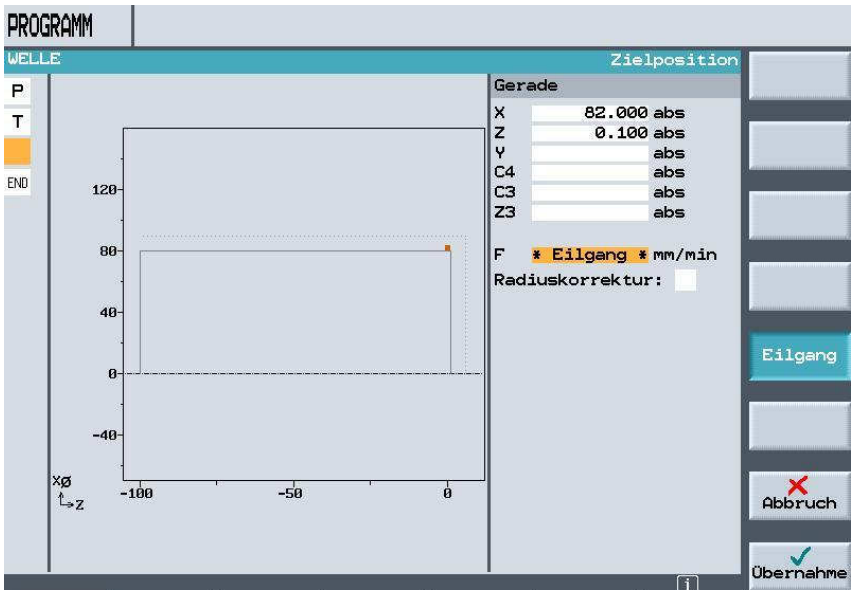
If needed, change the unit or selection with 

and finally confirm your input with 

The roughing tool is included in the program, with all the data.

4.10 Face Turning

Positioning for face turning in rapid feed.
By using line/circle, we can program a travel movement.

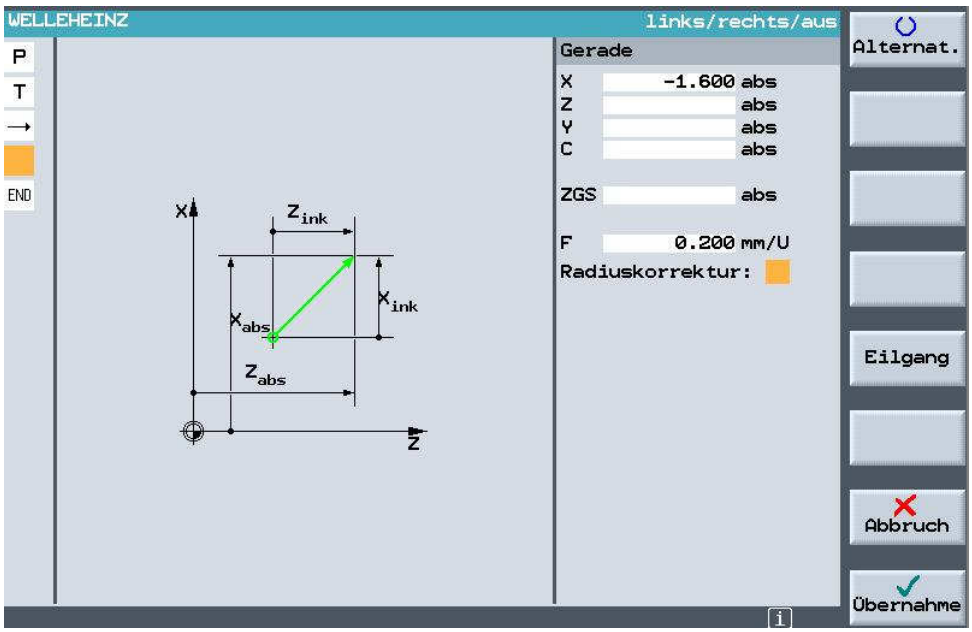


Click on 

. Enter X80 Z0.1 and click on 

Confirm with 

Next, face the front of the workpiece to X-1.6 (2 x plate radius size R=0.8)
so no residual material remains in the center.

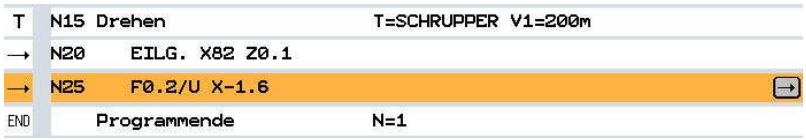


Click on 

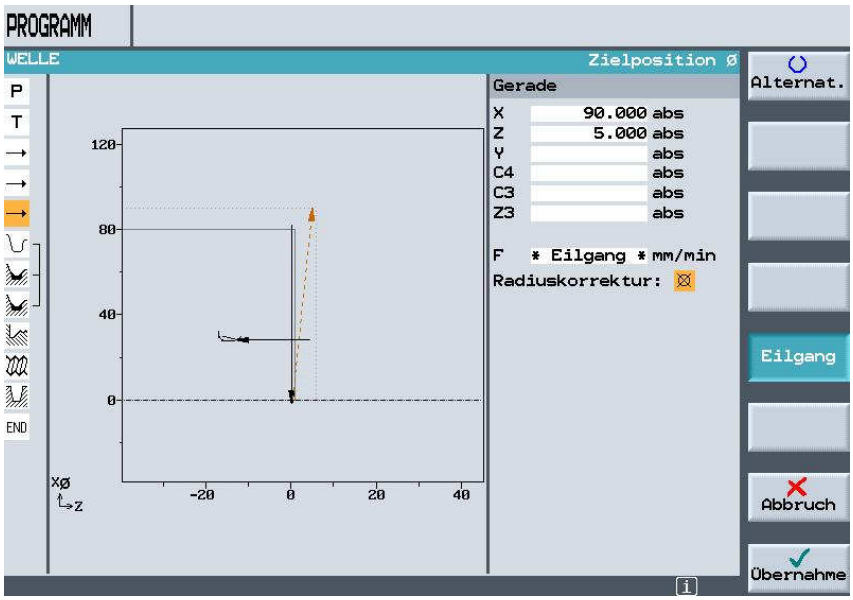
and enter X-1.6 and F0.3 and 

mm/U

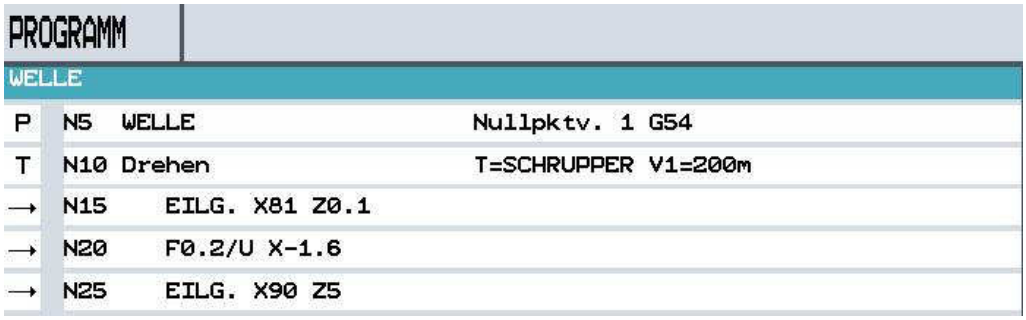
Confirm with 



Now, position on X90 Z5 at rapid feed.

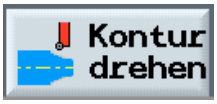


After accepting, you are back in the program.



4.11 Workpiece Contour

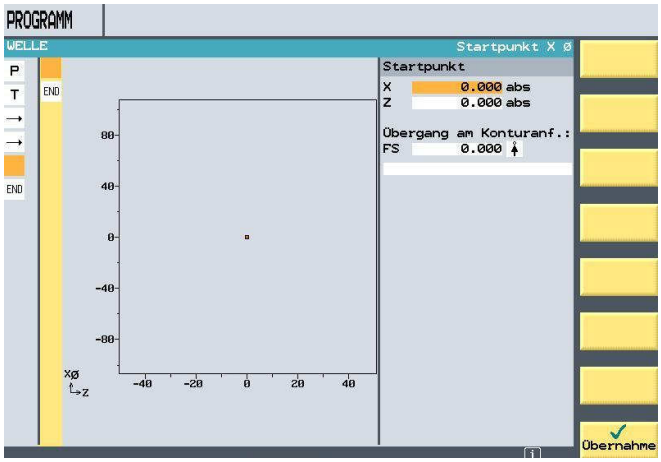
The contour is generated with the contour calculator.

Click on the button 

below and then in the upper right on 

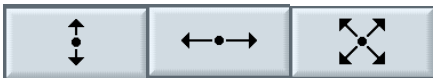
Assign a name to the new contour; then the contour calculator opens.

The first input specifies the starting point.



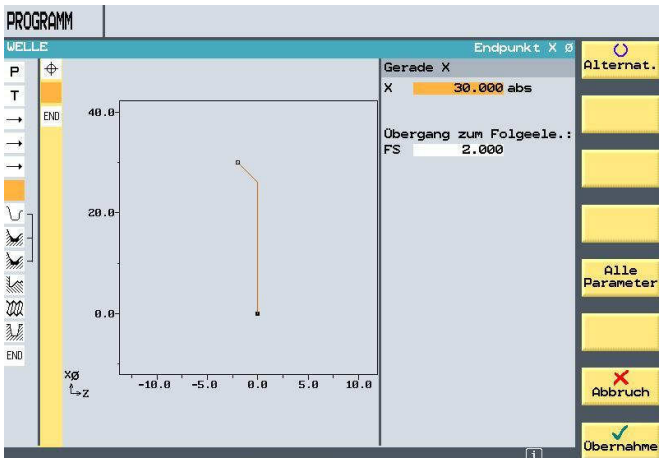
EnterX0 Z0 and confirm with 

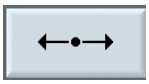
Note

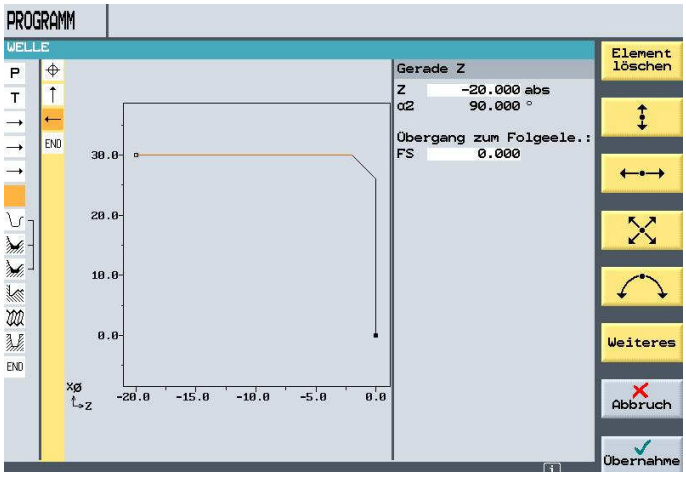
With the buttons  we describe the contour.

Here, the first diameter of the finished part contour is described.
A chamfer 2 x 45 degrees as transition element is appended to the end of the element.

Click on the button  and enter X30 and FS 2.

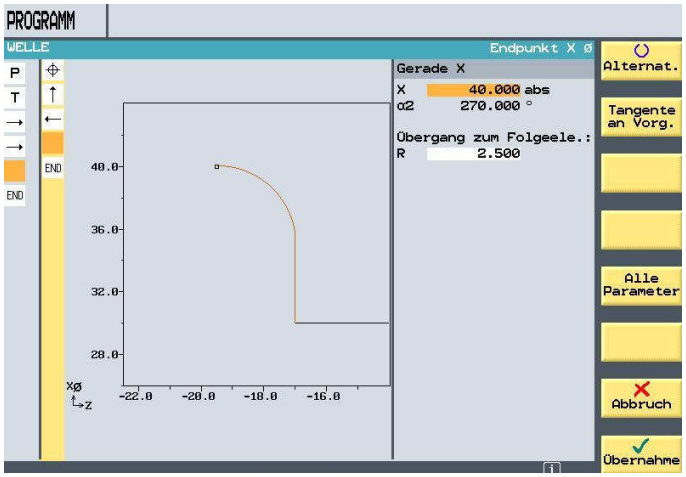


Click on the button  and enter Z-20. 

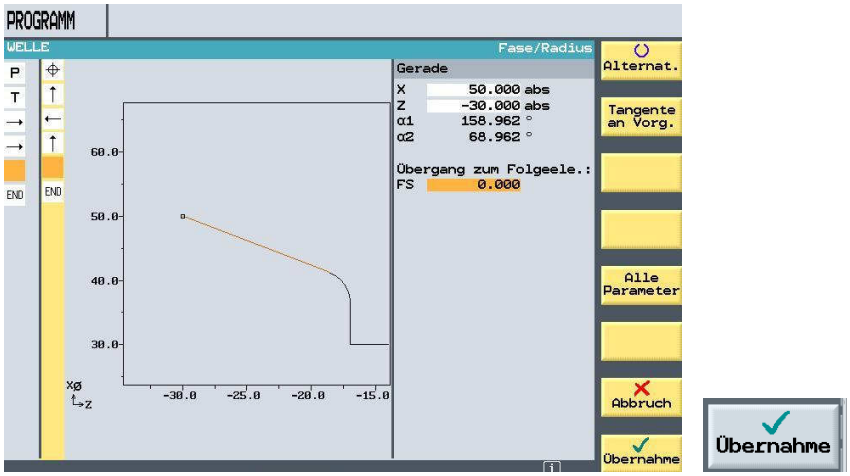


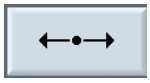
Click on the button  and enter X40 

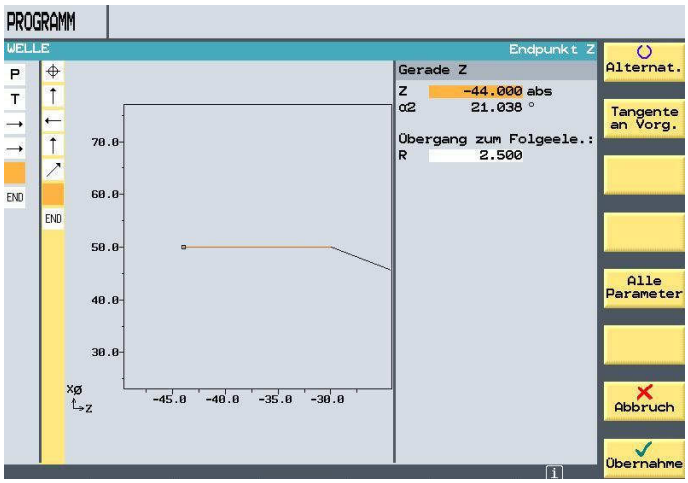
R2.5.



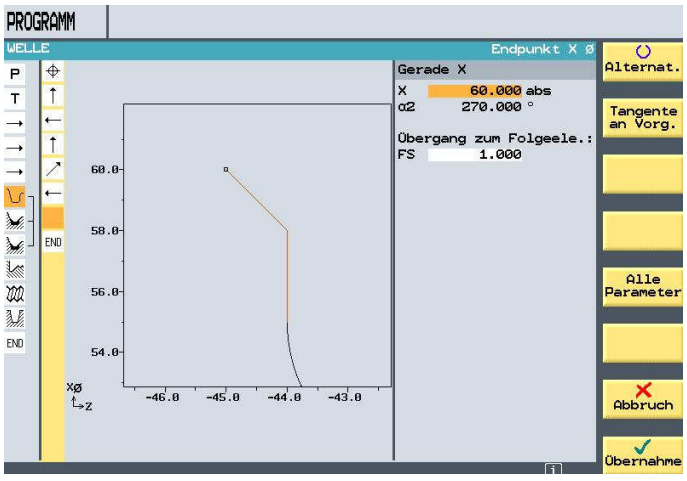
Click on the button  and enter X50 Z-30.



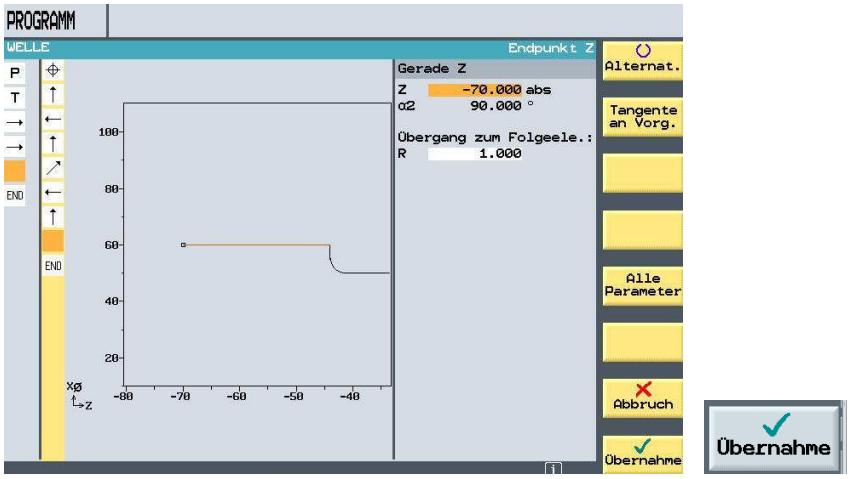
Click on the button  and enter Z-44  R2.5.



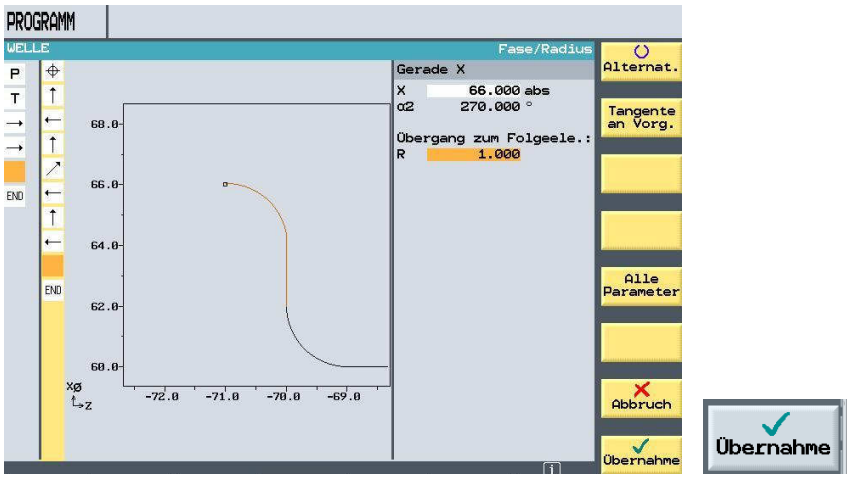
Click on the button  and enter X60  FS1.

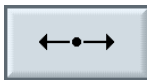


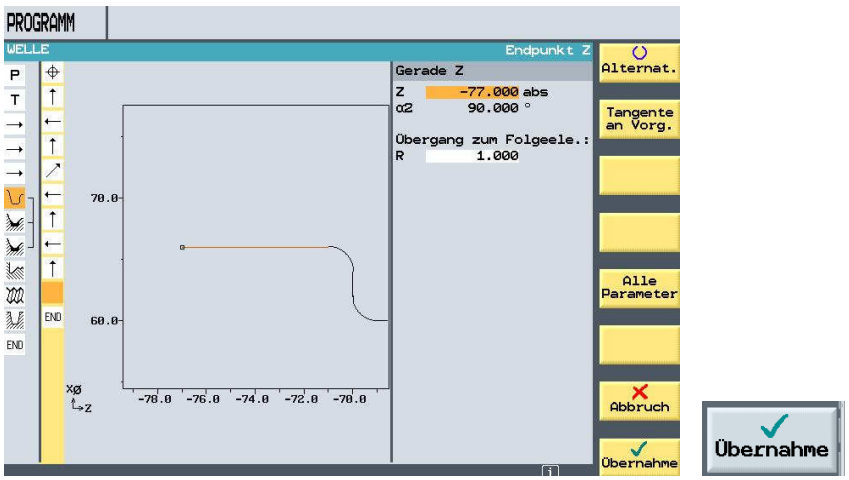
Click on the button  and enter Z-70 and R1.



Click on the button  and enter X66 and R1.

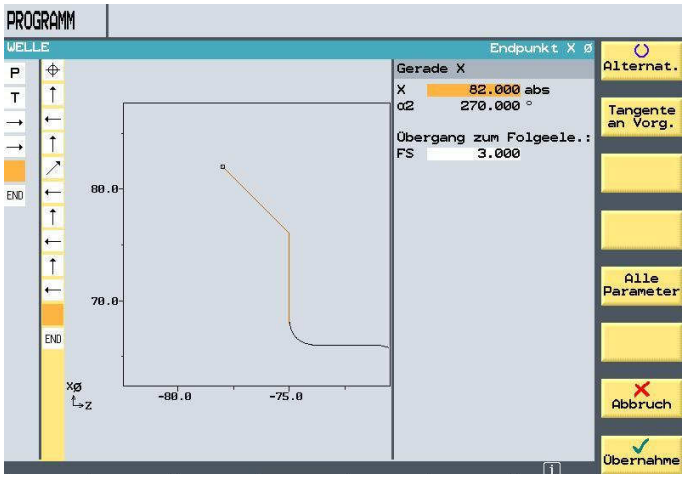


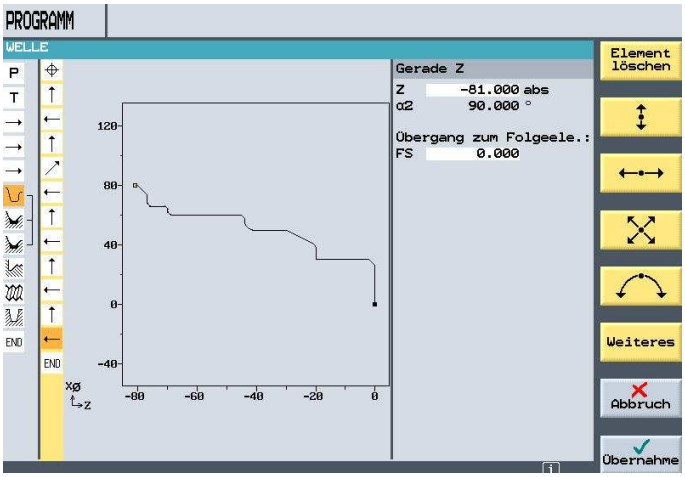
Click on the button  and enter Z-77 and R1.



Click on the button  and enter X82 and FS3.

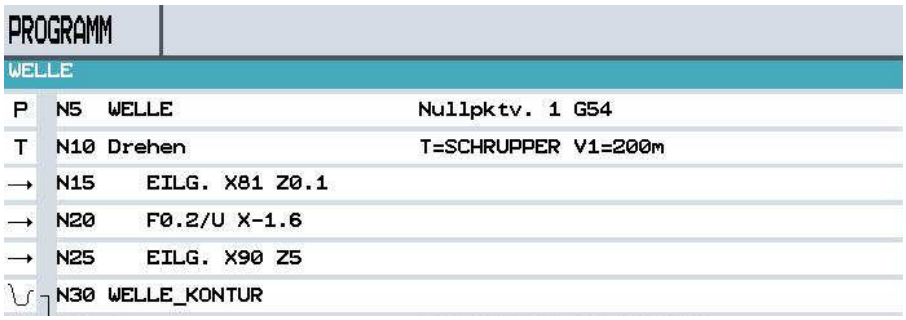
The raw part is diameter 80 in the program and is run to 82, in order to gain distance from the tool. For that reason, the chamfer size has to be programmed 1mm larger.





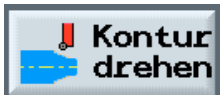
The finished part contour has now been described, and is accepted into the program with  .

After that, you are back in the program editor.



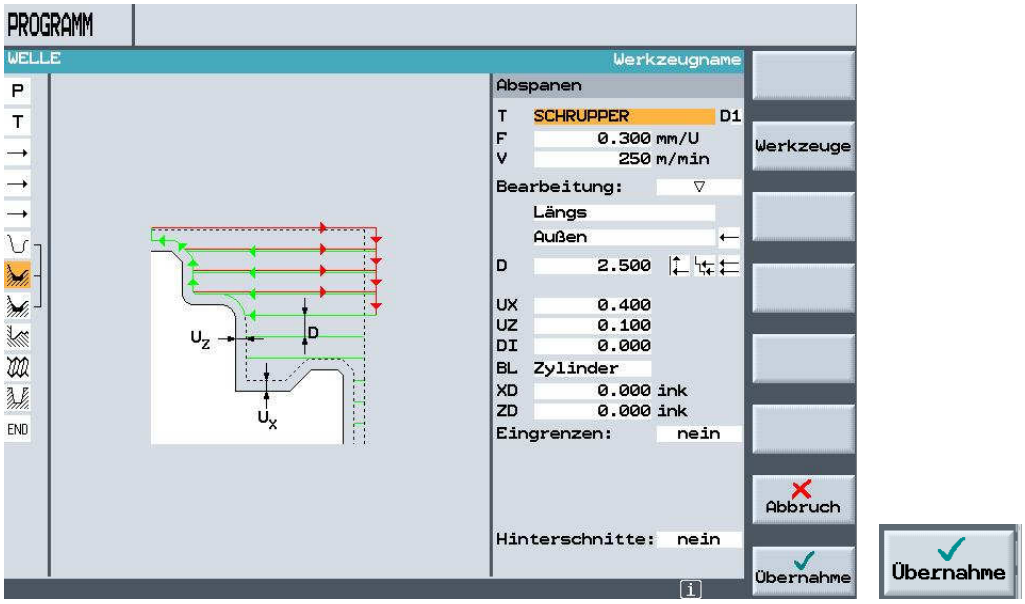
4.12 Stock Removal Cycle Roughing

The stock removal of a contour is performed in the area Contouring and Stock Removal.
Using the “Help Key“ brings up the help display.

Click on the button 

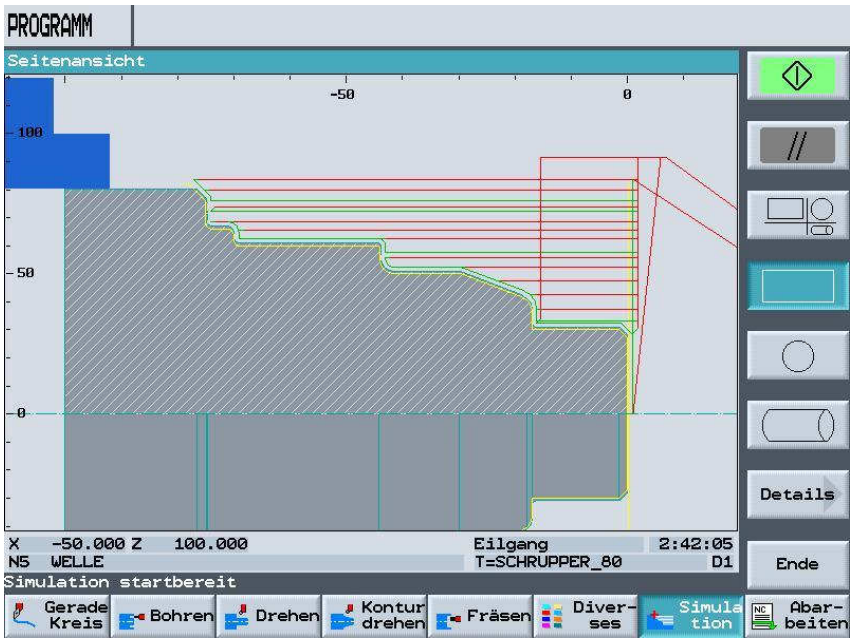
and 

.
Select the tool and enter the cutting data.
Accept all other data in the picture.



Here, you should test the program with a simulation display.

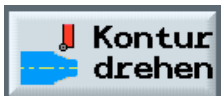
Click on the button 



Clicking on the button "End“ takes you back to the program editor.

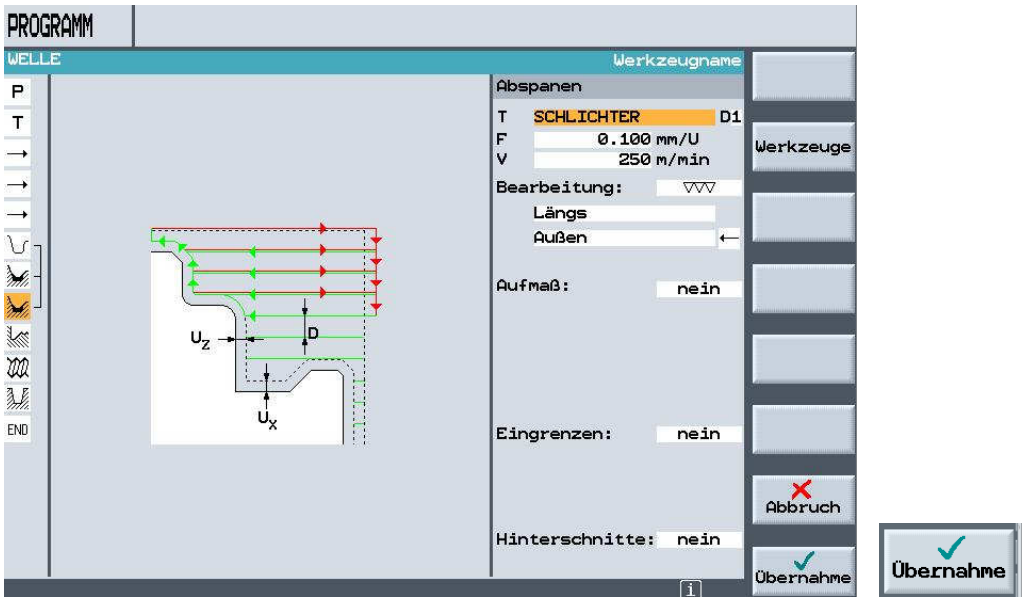
4.13 Finishing the Outside Contour

The outside contour is executed with the finishing tool contour parallel.

Click on the button 

and 

Select the tool and enter the cutting data.
Accept all other data from the figure.



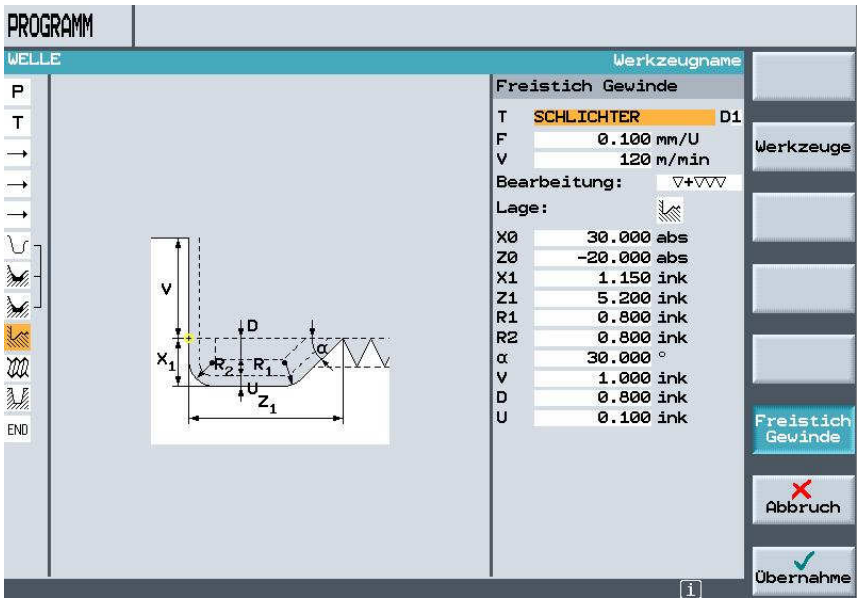
4.14 Thread Undercut

We incorporate the undercut DIN 76

Click on the button 

and 

and then on 



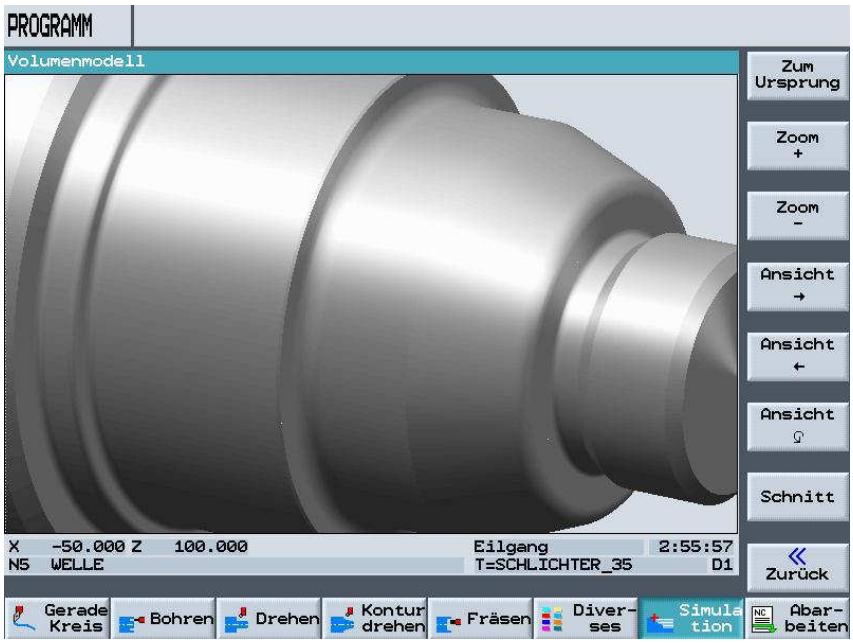
Use all data from the figure and exit the menu with 

.

We can see here that the contour during roughing and finishing is connected with stock removal. The thread undercut was generated with its own cycle.

N30	WELLE_KONTUR		
N35	Abspanen	▽	T=SCHRUPPER F0.3/U V250m
N40	Abspanen	▽▽▽	T=SCHLICHTER F0.1/U V250m
N45	Freistich Gew.	▽+▽▽▽	T=SCHLICHTER F0.1/U V120m X0=30
END	Programmende		N=1

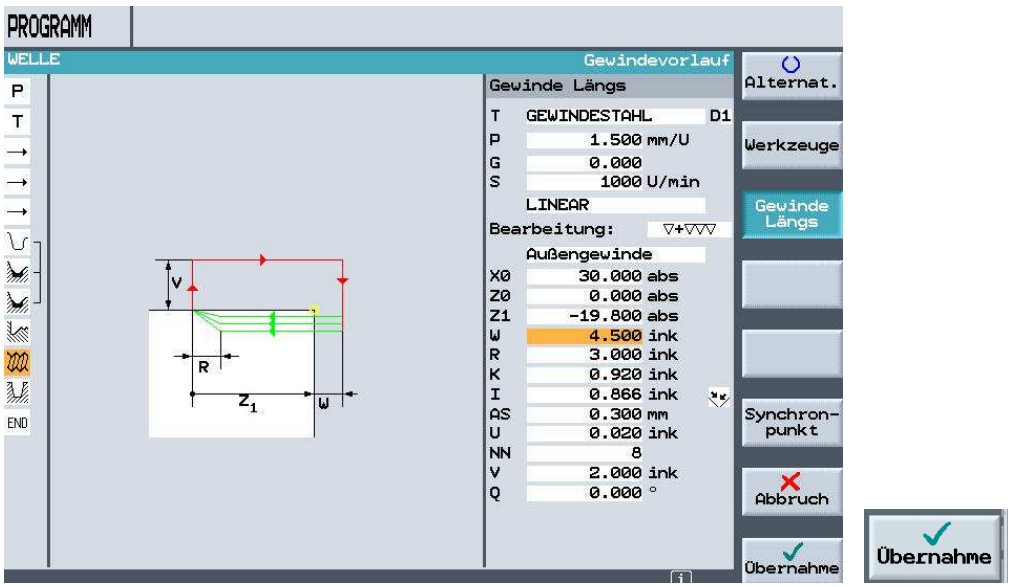
Here, you can test your program with the simulation display.



4.15 Thread Cutting

A thread of M30 X 1.5 is to be manufactured.

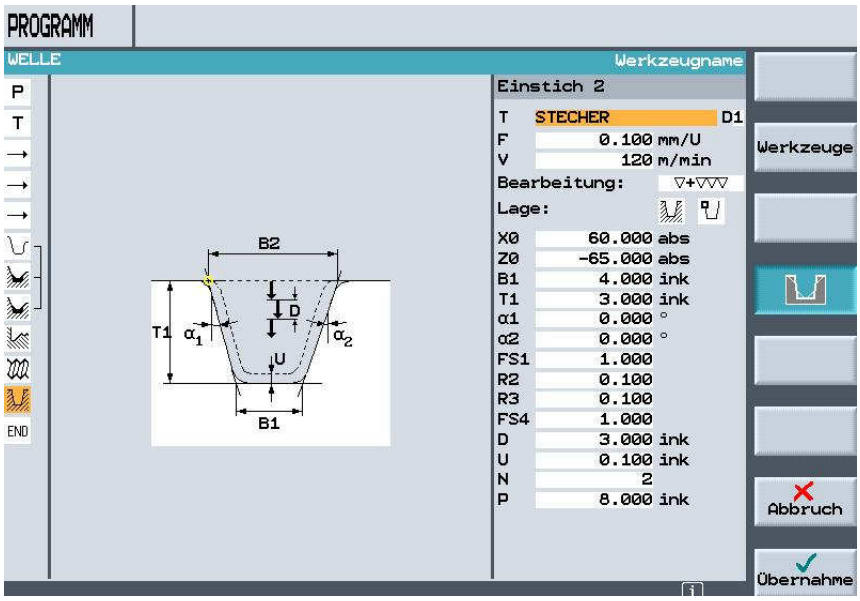
Click on **Drehen** and on **Gewinde** and enter the values.

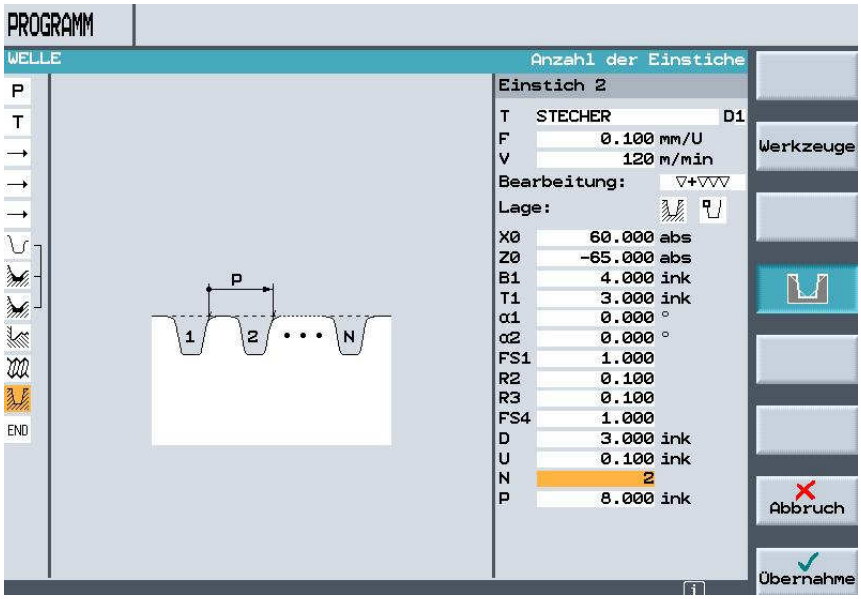


4.16 Making Grooves

Finally, we are making two grooves.

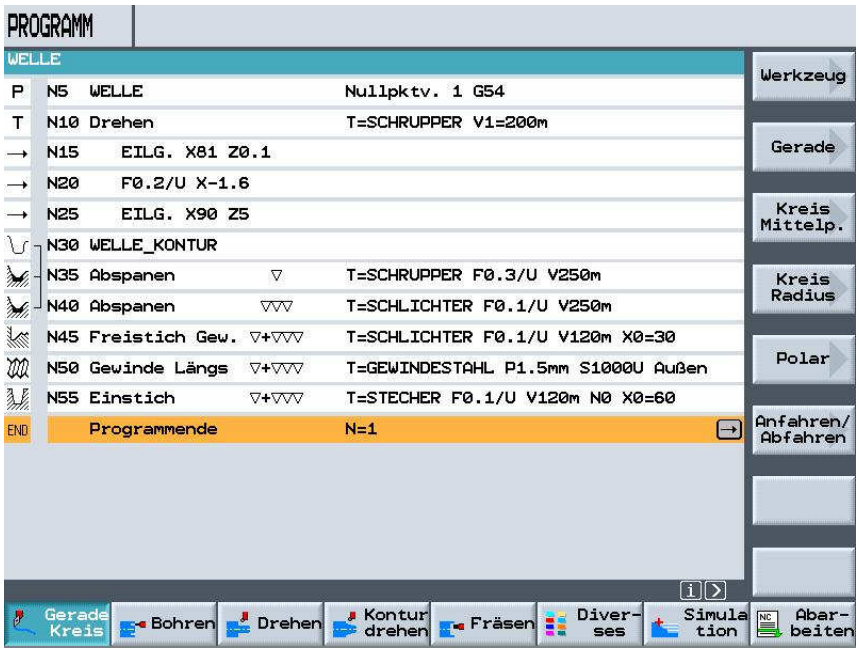
Click on Drehen and on Einstich and enter the values.





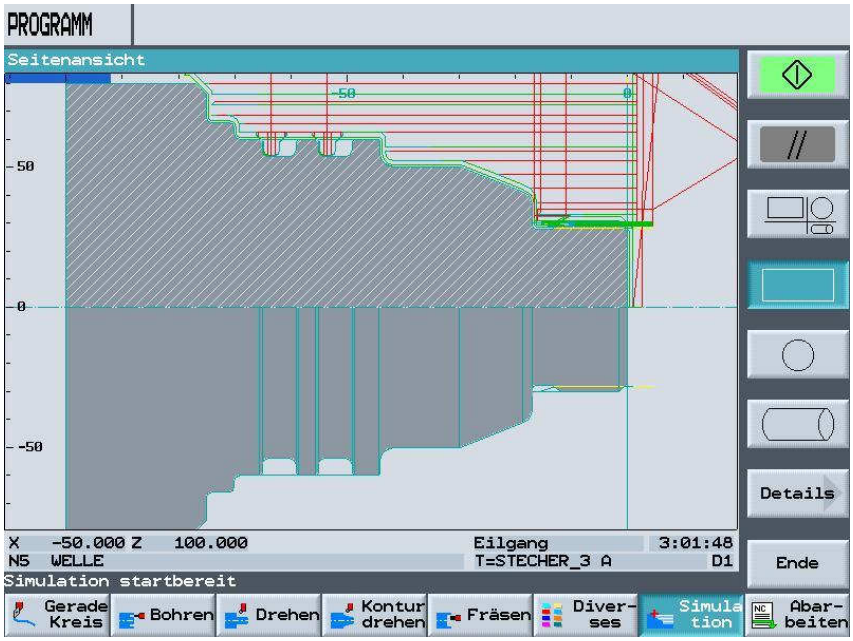
Exit menu with Übernahme.

We are done with our machining step program.

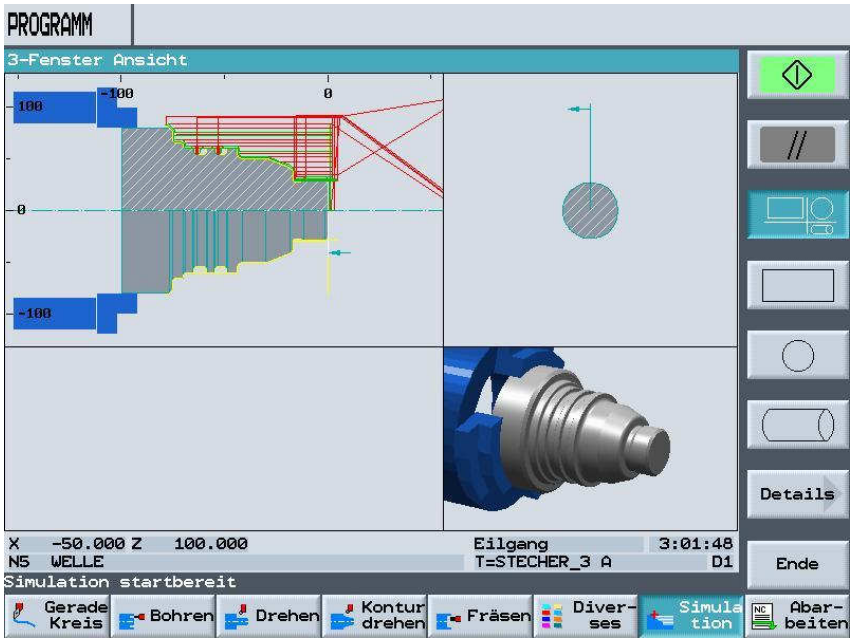


4.17 Simulation

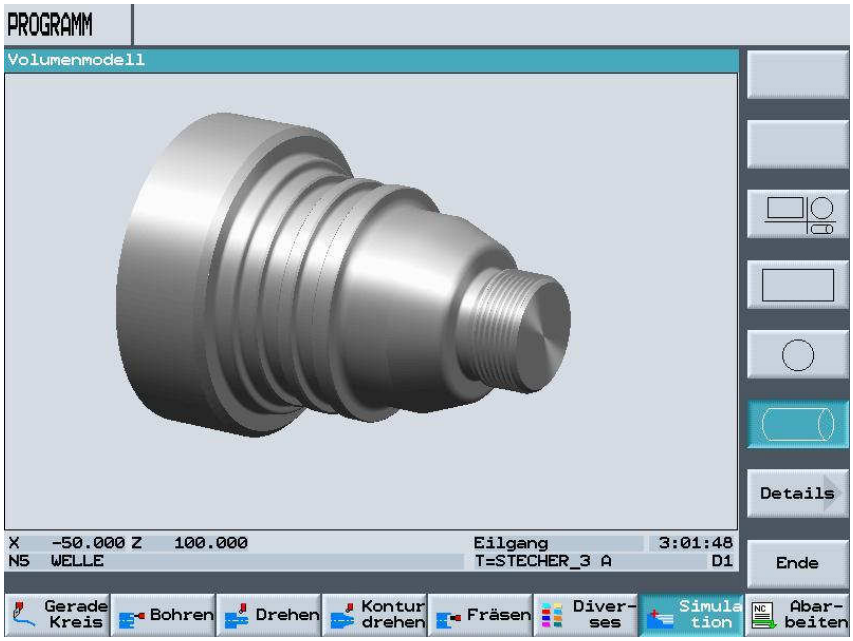
Finally, test your program with a simulation display.
First in the side view.



Next in the 3 window view

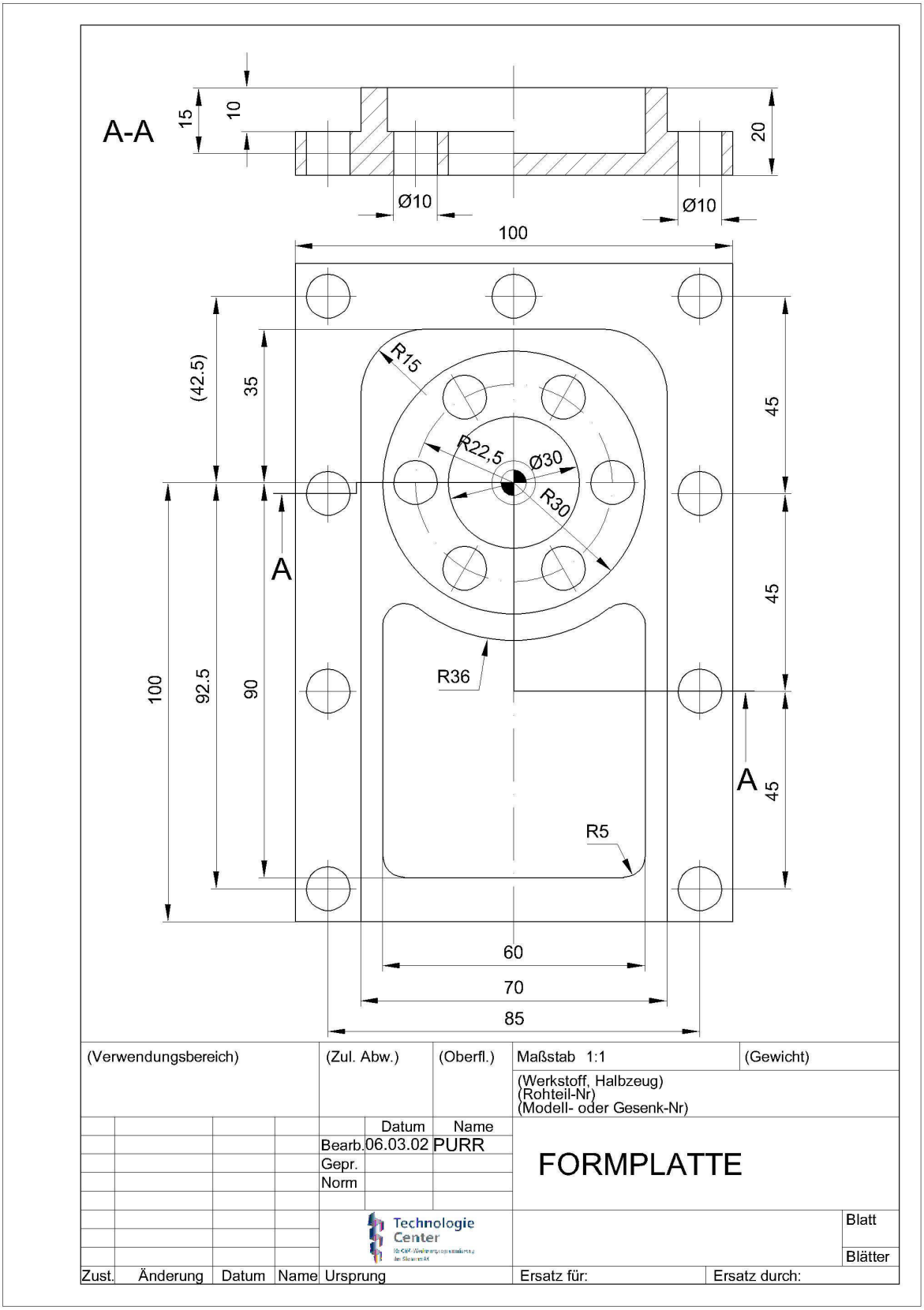


and in the volume model.



Using **EXIT**, we exit the ShopTurn software.

5 Programming a Milled Part in G-Code

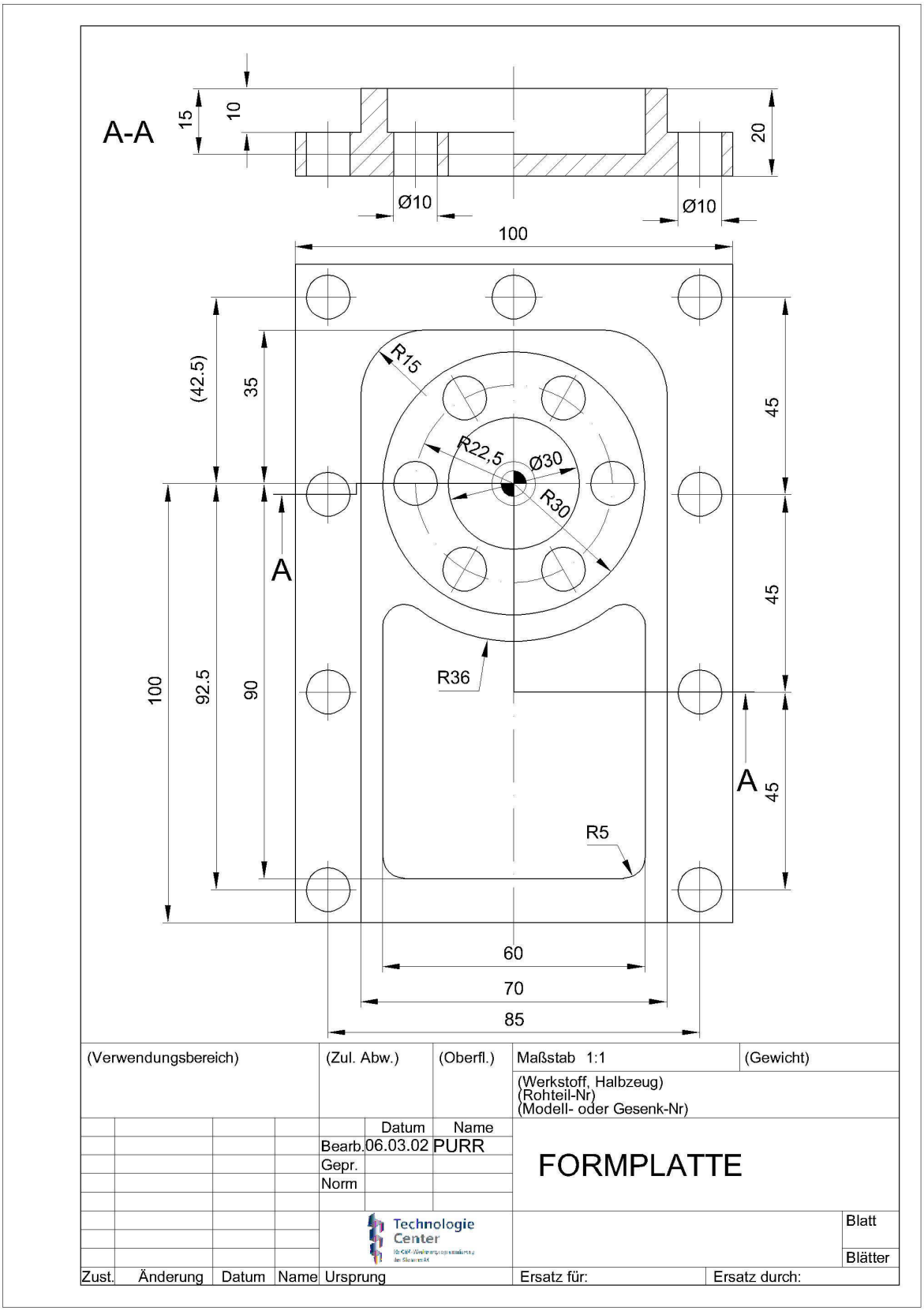


```
FORMPLATTE_G_CODE
N1 G0 G90 G40 G17¶
N2 T="PLANFRAESER63"¶
N3 M6¶
N4 G94 S400 F300 M3 M8¶
N5 G54 G0 X-20 Y-135 Z3 ¶
N8 CYCLE71(3,2,2,0,-50,-100,100,150,0,1.8,55,2,0.2,300,22,)¶
N10 G0 Z150¶
N11 G0 X200 Y200¶
N12 T="FRAESER32"¶
N13 M6¶
N14 G94 S1000 M3 M8 F200¶
N15 G0 X-51 Y-112 Z1¶
N16 G0 Z-10¶
N18 G17 G90 DIAMOF ;*GP*¶
N19 G1 X-35 Y-100 G41 ;*GP*¶
N20 Y35 RND=15 ;*GP*¶
N21 X35 RND=15 ;*GP*¶
N22 Y-120 ;*GP*¶
N23 G40 ¶
N32 G0 Z100¶
N33 X200 Y150¶
N34 T="FRAESER10"¶
N35 M6 M8¶
N36 G94 S2000 M3 M8 F200¶
N37 G0 X0 Y-84 Z1¶
N38 G1 Z-15 ¶
N40 G17 G90 G41 DIAMOF ;*GP*¶
N41 G1 X0 Y-90 ;*GP*¶
N42 X30 RND=5 ;*GP*¶
N43 Y-19.9 RND=5 ;*GP*¶
N44 G2 X-30 I=AC(0) J=AC(0) RND=5 ;*GP*¶
N45 G1 Y-90 RND=5 ;*GP*¶
N46 X0 ;*GP*¶
N47 G40¶
N48 G0 Y-84¶
N49 G0 Z1¶
N64 _ZSD[2]=1 ;*RO*¶
N65 POCKET3(,0,1,-15,60,54,5,-30,-90,0,5,0.2,0.2,200,100,0,31,6,,,,
12,)¶
N70 _ZSD[2]=1 ;*RO*¶
N71 POCKET3(2,0,1,-15,60,54,5,-30,-90,0,5,0.2,0.2,200,100,0,32,6,,,
,12,)¶
N73 G0 Z100¶
N74 G0 X200 Y150¶
N75 T="FRAESER20"¶
N76 G94 S1000 M3 M8 F200¶
N79 POCKET4(2,0,1,-10,30,0,0,5,0.2,0.2,200,100,0,21,10,,,12,5)¶
N83 POCKET4(2,0,1,-10,30,0,0,5,0.2,0.2,200,100,0,22,10,,,12,5)¶
N87 POCKET4(2,-10,1,-21,15,0,0,5,0.2,0,200,100,0,21,10,,,12,5)¶
N91 POCKET4(2,-10,1,-21,15,0,0,5,0.2,0,200,100,0,22,10,,,12,5)¶
N93 G0 Z100¶
N94 G0 X200 Y150¶
N95 T="ZENTRIERER12"¶
N96 G94 S2000 M3 M8 F200¶
N97 G0 Z2¶
N100 MCALL CYCLE82(2,0,2,-15.3,,0.1)¶
```

```
N104 HOLES1(-42.5,-92.5,90,0,45,4)¶
N106 G0 X0 Y42.5¶
N109 HOLES1(42.5,42.5,-90,0,45,4)¶
N111 MCALL¶
N114 G0 Z1¶
N118 MCALL CYCLE82(2,0,2,-15.3,,0.1)¶
N122 HOLES2(0,0,22.5,0,60,6)¶
N124 MCALL¶
N125 G0 Z100¶
N126 G0 X200 Y150¶
N127 T="BOHRER10"¶
N128 G94 S2000 M3 M8 F200¶
N131 MCALL CYCLE83(2,0,1,-24,,-15,,5,,3,1,1,3,10,,,1)¶
N135 HOLES1(-42.5,-92.5,90,0,45,4)¶
N137 G0 X0 Y42.5¶
N140 HOLES1(42.5,42.5,-90,0,45,4)¶
N142 MCALL¶
N143 G0 Z2¶
N146 MCALL CYCLE83(2,0,1,-24,,-15,,5,,3,1,1,3,10,,,1)¶
N150 HOLES2(0,0,22.5,0,60,6)¶
N152 MCALL¶
N153 G0 Z100¶
N154 G0 X200 Y150¶
N155 M30¶
```

PLANFRAESER = facing tool
FRAESER = milling tool
ZENTRIERER = centering tool

6 Programming a Milled Part according to the Machining Step Process



6.1 Start SINUTRAIN

To start SinuTrain, double click on or start SinuTrain using the start bar.

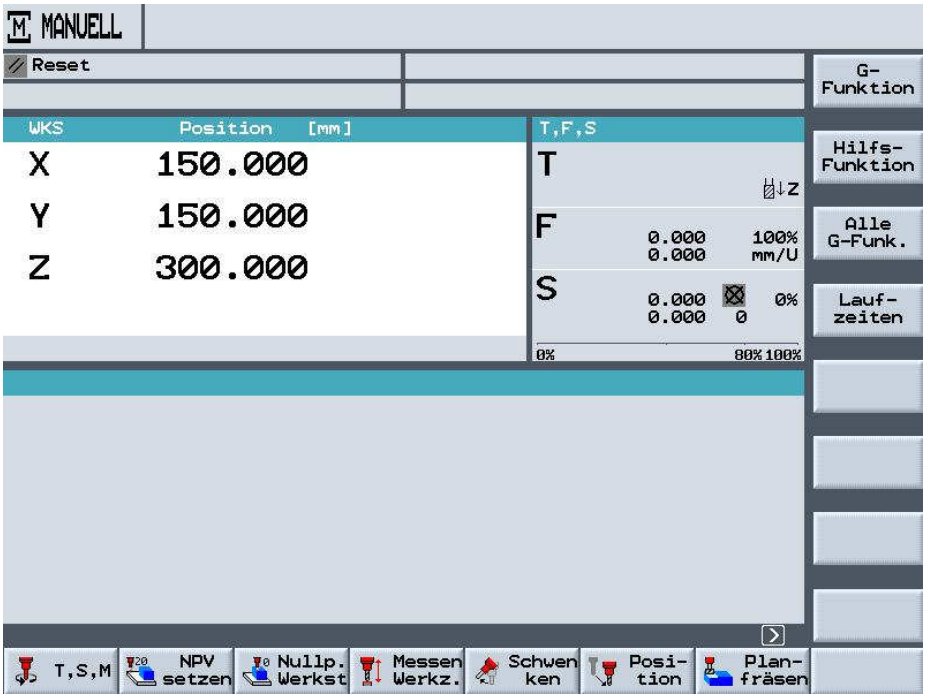
6.2 Selecting the Machine

After you booted the computer, a menu is displayed with the selection of machines with the respective version number.

1. Select milling machine >> ShopMill Open V 06.04 <<.
2. Click on the Start button.

6.3 SINUTRAIN Basic Display

After you selected the machine, the basic display of SINUTRAIN appears.
All SINUTRAIN actions are started from here.

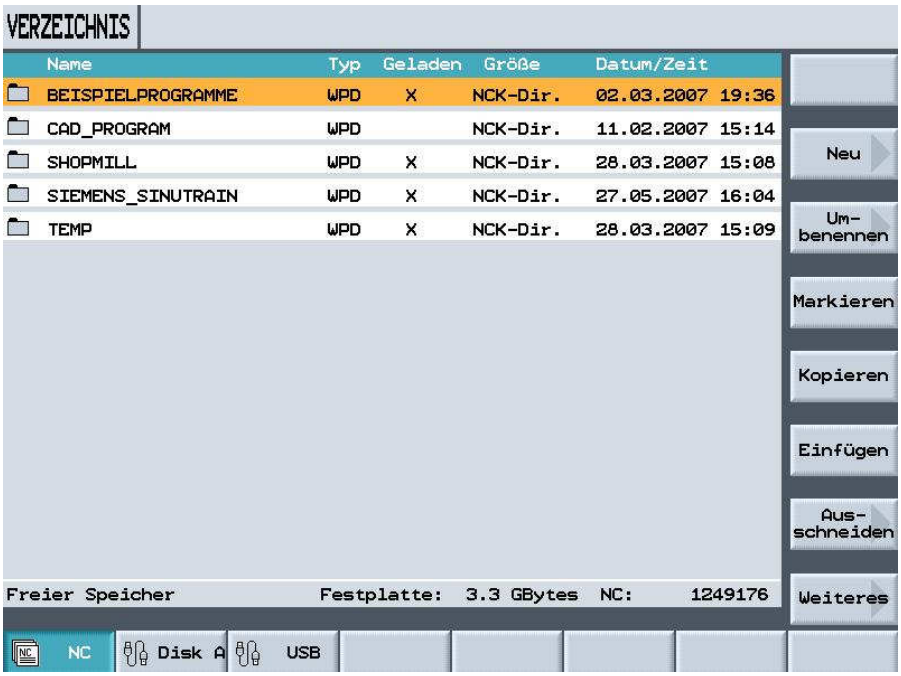


6.4 Setting Up a New Directory/Program Generation

Different actions can be performed in the program manager.

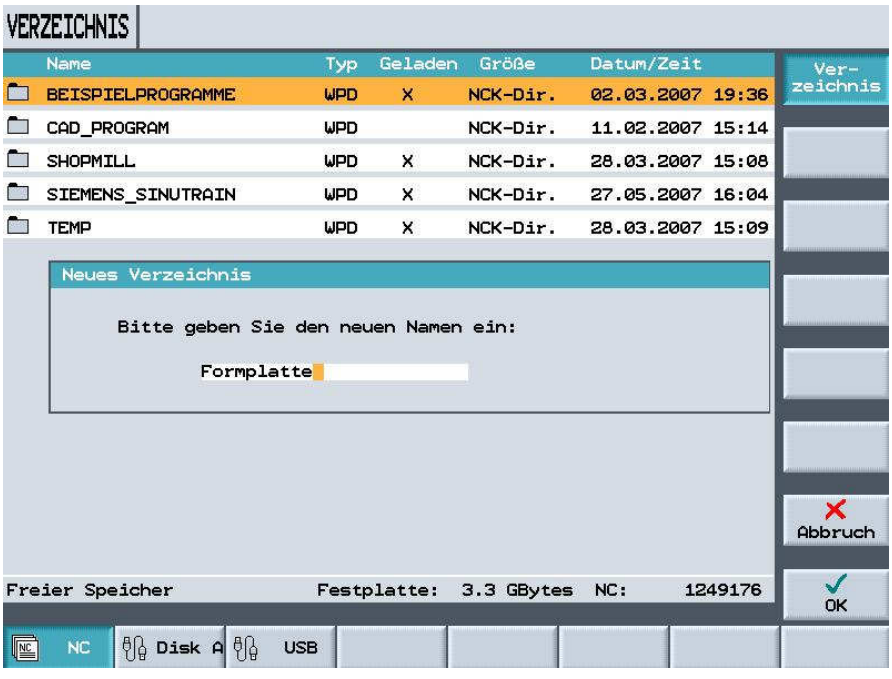
By clicking on the softkeys 

, the available directories are opened under ShopMill, in the operating area **DIRECTORY**



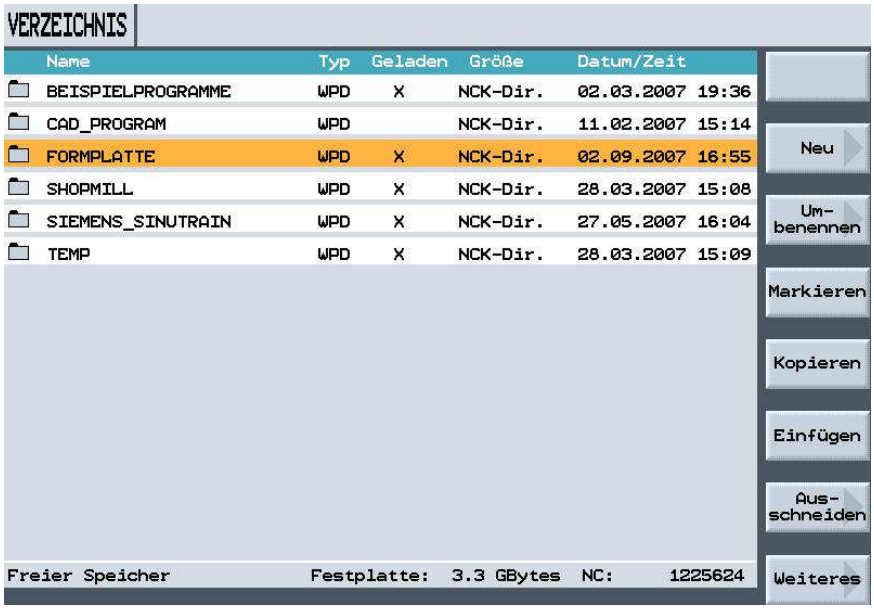
With the softkey 

we are now setting up a new directory called "Pattern plate“.



Confirm the input with **OK**.

The selected directory opens by pressing the arrow key  on the CNC keyboard, or with the cursor key toward the right.

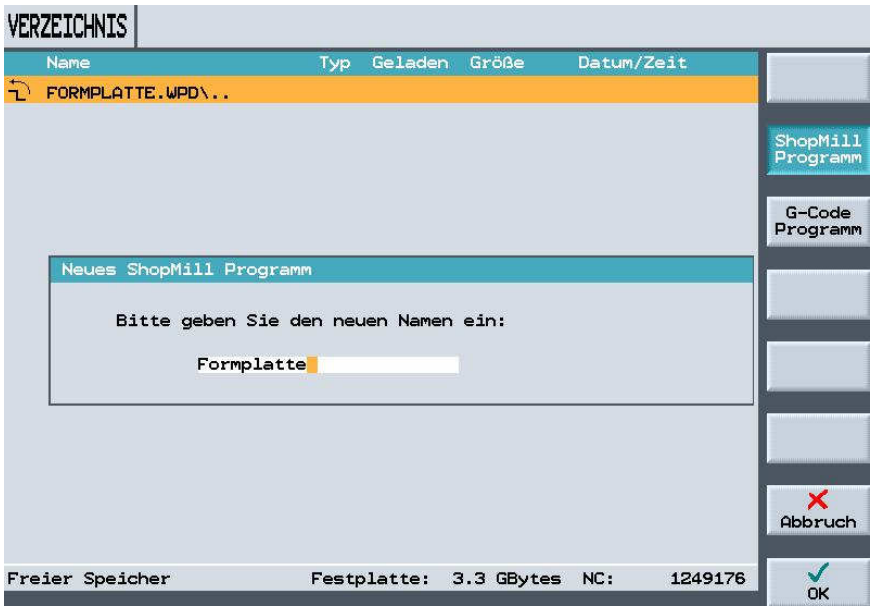


6.5 Setting Up a New Workpiece Program

After opening the directory, existing programs are displayed, or new programs can be set up.

By pressing the softkey 

and selecting ShopMill or G-Code (vertical softkeys), a new program name “Name of the work plan“ is entered; in our case “Pattern plate“.



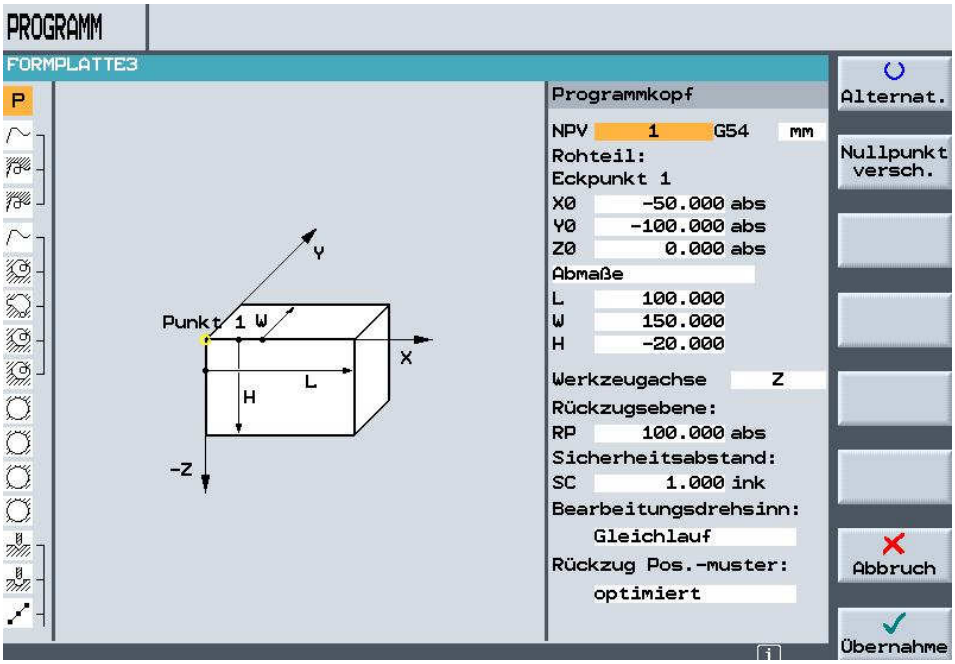
Confirm the input with **OK**.

6.6 Program Header

The program header of the new program opens automatically. The following is entered in the program header: the workpiece data, zero shift, raw part pattern, distance to the chuck, retraction planes, tool change point, safety clearance incremental or absolute, and the spindle speed limitation.

With the softkey 

you can now call the associated help display.



Enter all data as in the figure above

and accept the inputs in the machining plan with 

.

Note

With the softkey 

you can alternate, for example, between different zero points and measurement units, mm, inch. The blue line above always shows the current input type.

6.7 Outside Contour

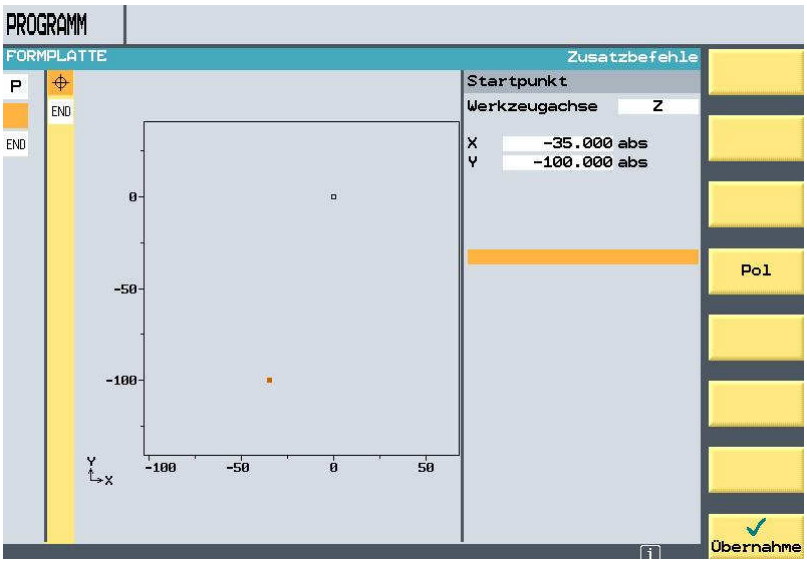
The contour is generated with the contour calculator.

Click on the button 

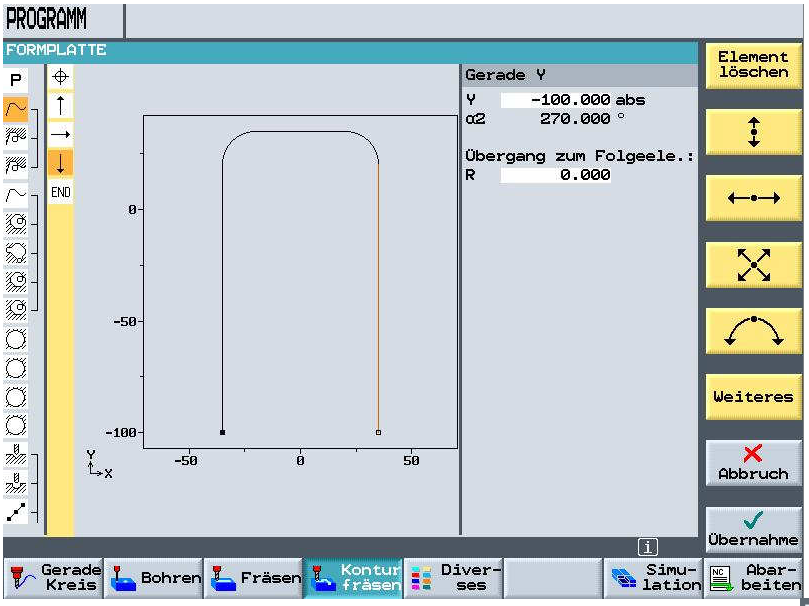
below and then in the upper right on 

.
Assign a name to the new contour; after that, the contour calculator opens.

The first input specifies the starting point.



Complete programming the outside contour with the contour calculator.



The outside contour is now described, and is accepted into the program with 

.

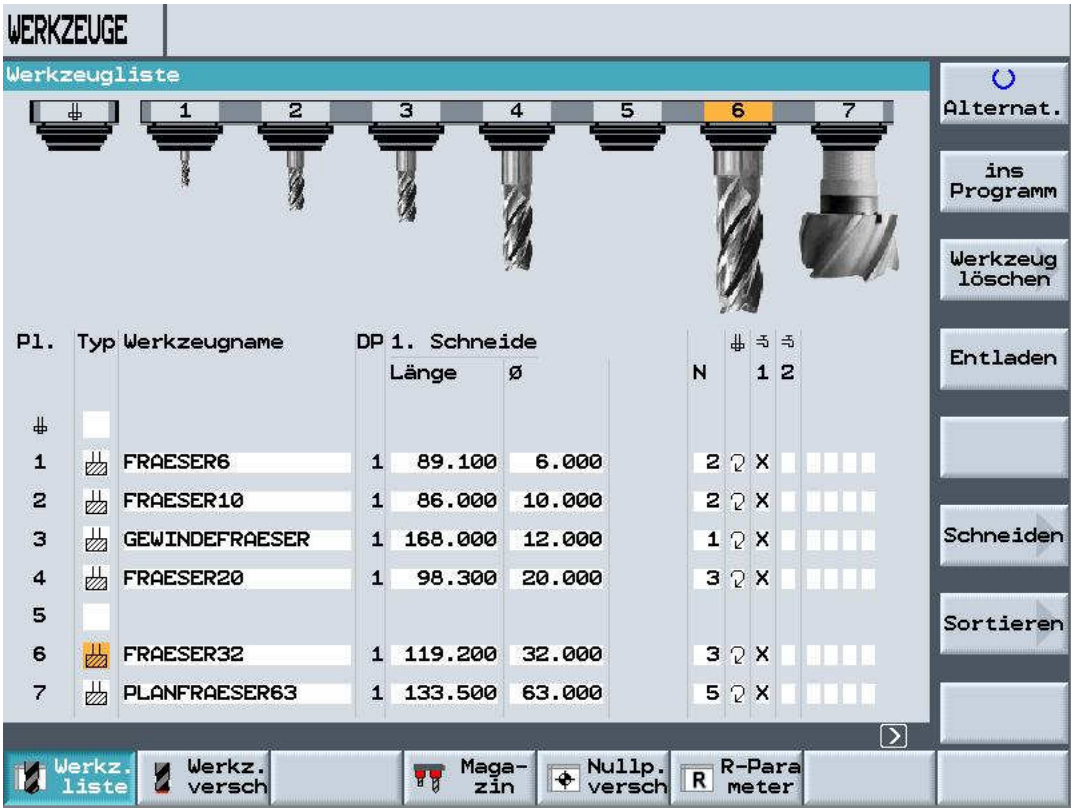
6.8 Roughing the Outside Contour

To perform the machining step roughing on the outside, with contour compensation on the left, do the following:

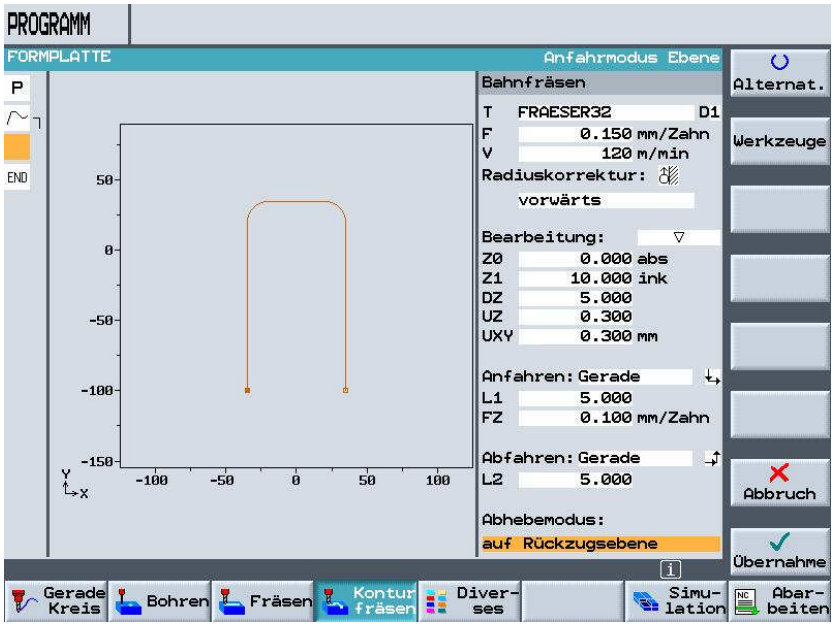
Click on Bahn- fräsen and on Werkzeuge and enter the values.

6.9 Selecting the Tool

Here the tools that have already been set up are provided. But you can also generate new ones. Here, take the MillingTool32.



Click on ins Programm to accept the selected tool.



Accept all data from the graphic and exit the menu with 

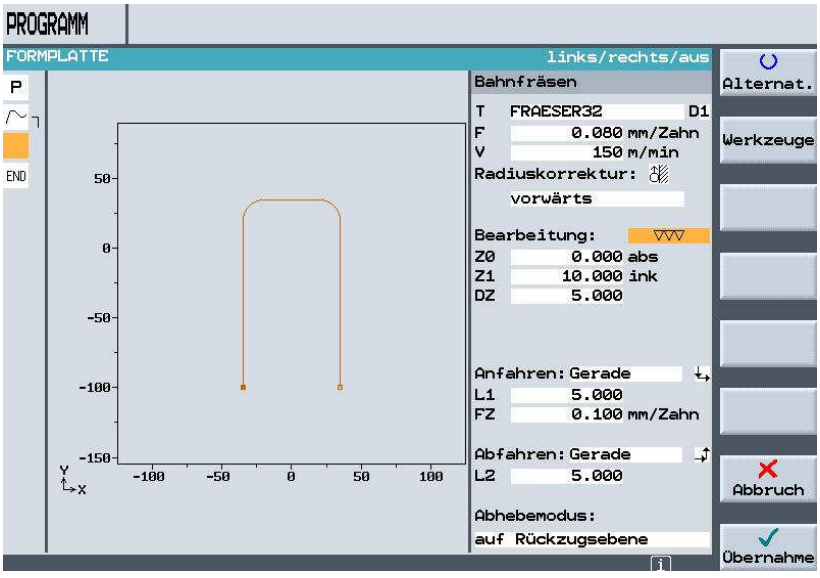
.

6.10 Finishing the Outside Contour

To perform the machining step Finishing Outside, with contour compensation to the left, do the following:

Click on 

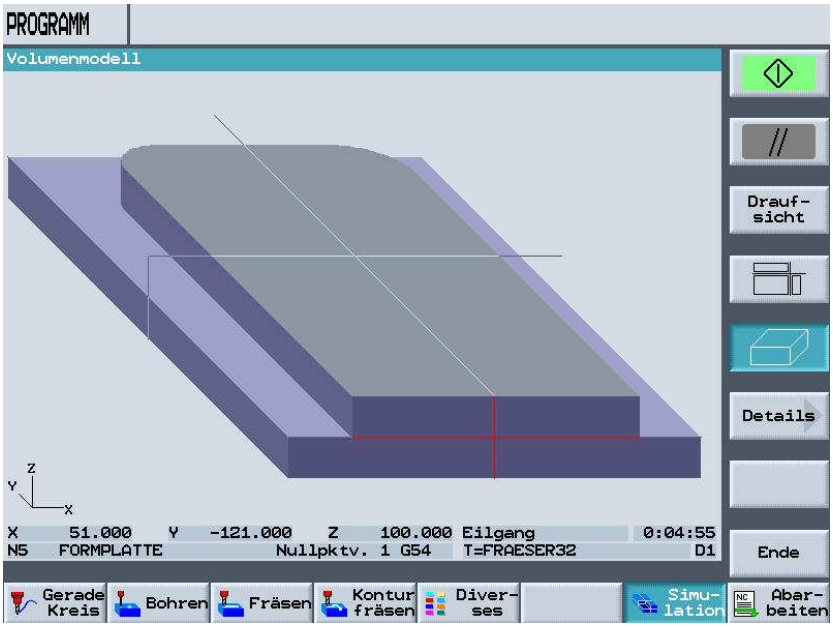
and enter the values.



Accept all values from the figure, and exit the menu with 

.

Here, you can test your program with the simulation display.



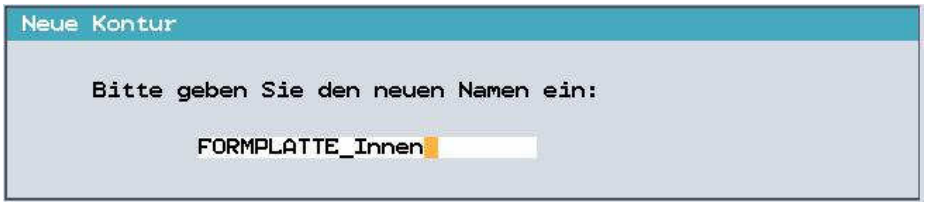
6.11 Stock Removal, Residual Material and Finishing of Contour Pockets and Inside Contour

The contour is generated with the contour calculator.

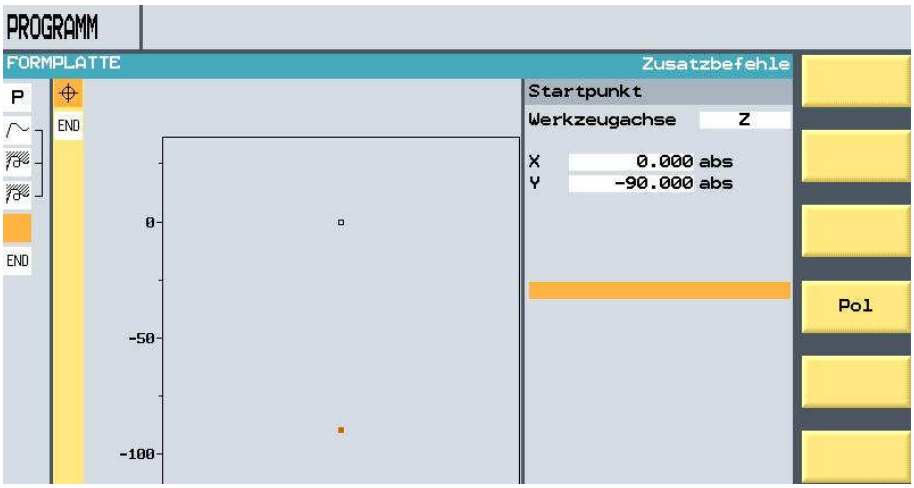
Below, click on the button 

and then in the upper right on 

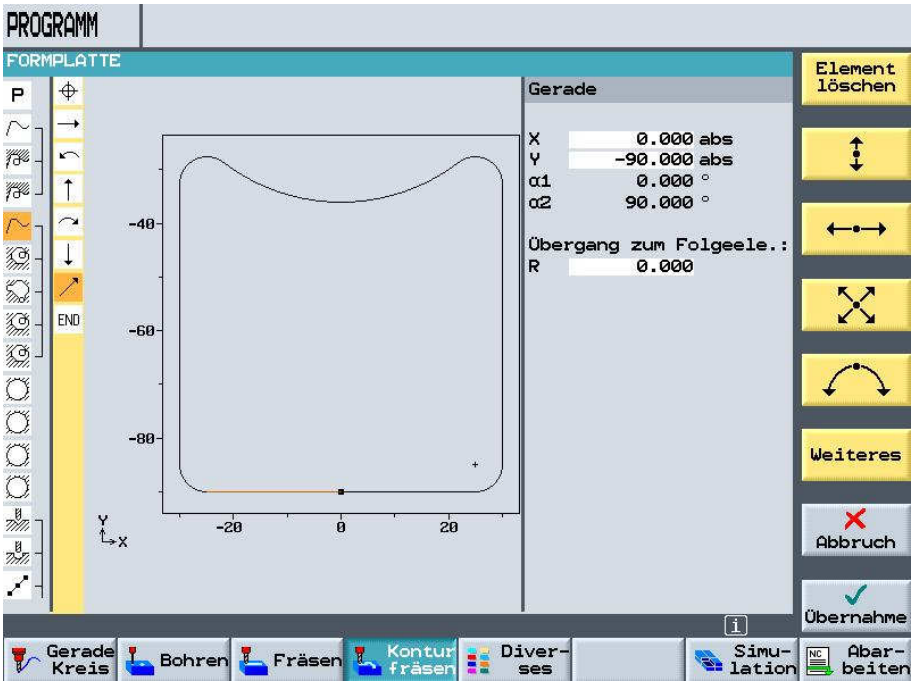
.
Assign a name to the new contour; then the contour calculator opens.



The first input specifies the starting point.



With the contour calculator, complete programming the inside contour.



The inside contour is now described and is accepted into the program with 

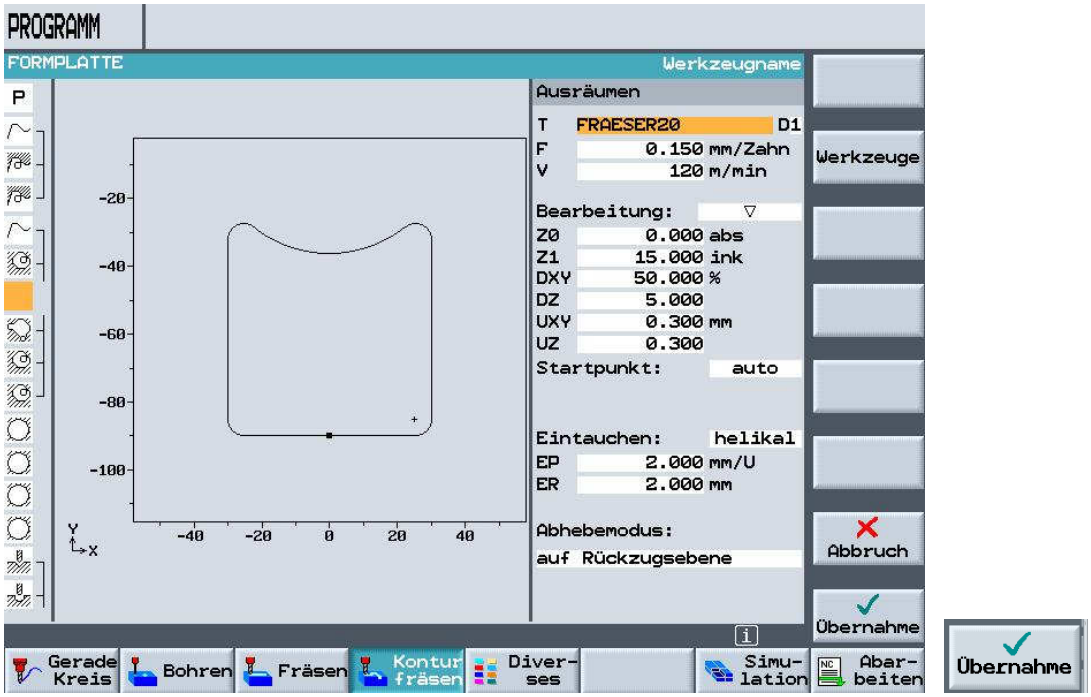
.

6.12 Stock Removal of Contour Pocket Roughing

Click on 

and on 

and enter the values.
Select the tool and enter the cutting data.
Accept all additional data from the figure.

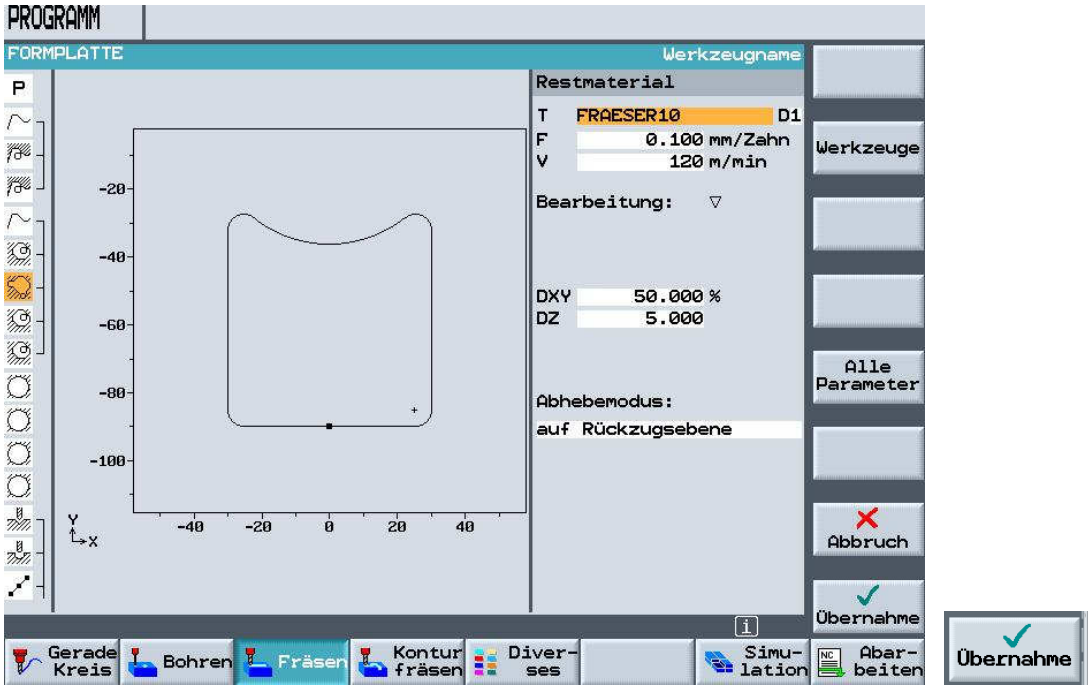


6.13 Removing Residual Material from the Corners

Click on 

and on 

and enter the values.
Select the tool and enter the cutting data.
Accept all additional data from the figure.

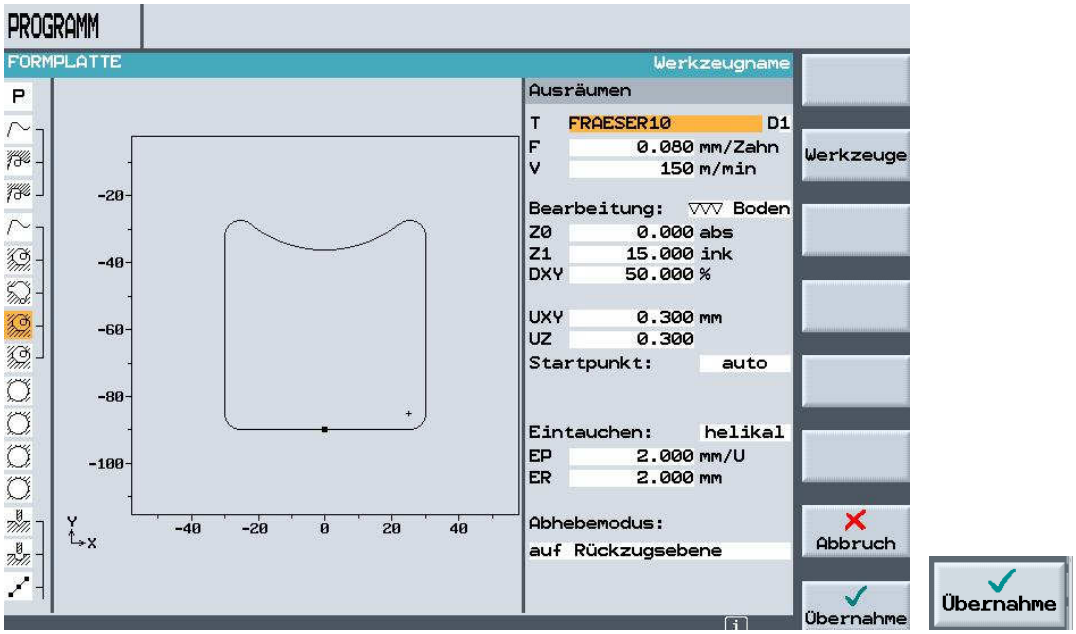


6.14 Finishing the Base

Click on 

and on 

and enter the values.
Select the tool and enter the cutting data.
Accept all additional data from the picture.

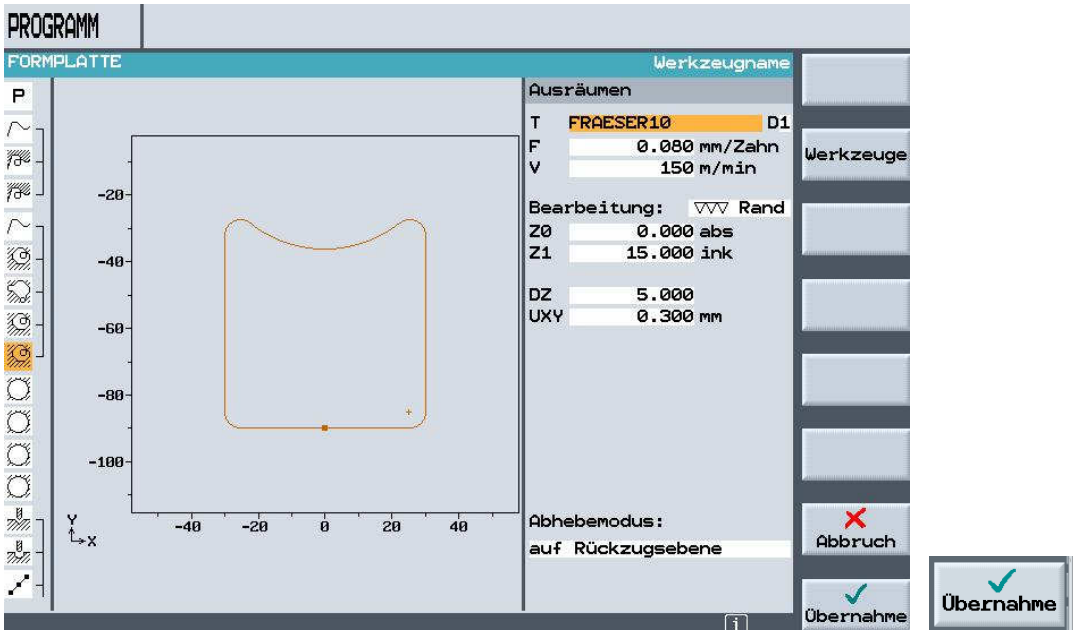


6.15 Finishing the Edge

Click on 

and on 

and enter the values.
Select the tool and enter the cutting data.
Accept all additional data from the picture.
.

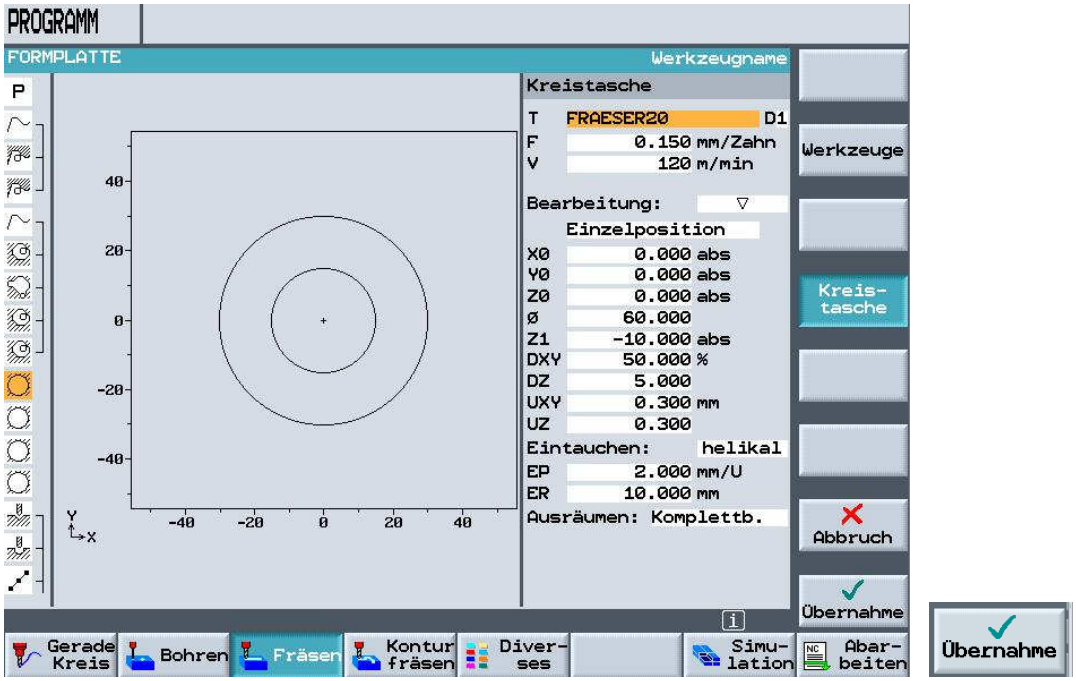


6.16 Roughing the Circular Pocket Diameter 60

Click on 

and on 

and enter the values.
Select the tool and enter the cutting data.
Accept all additional data from the picture.

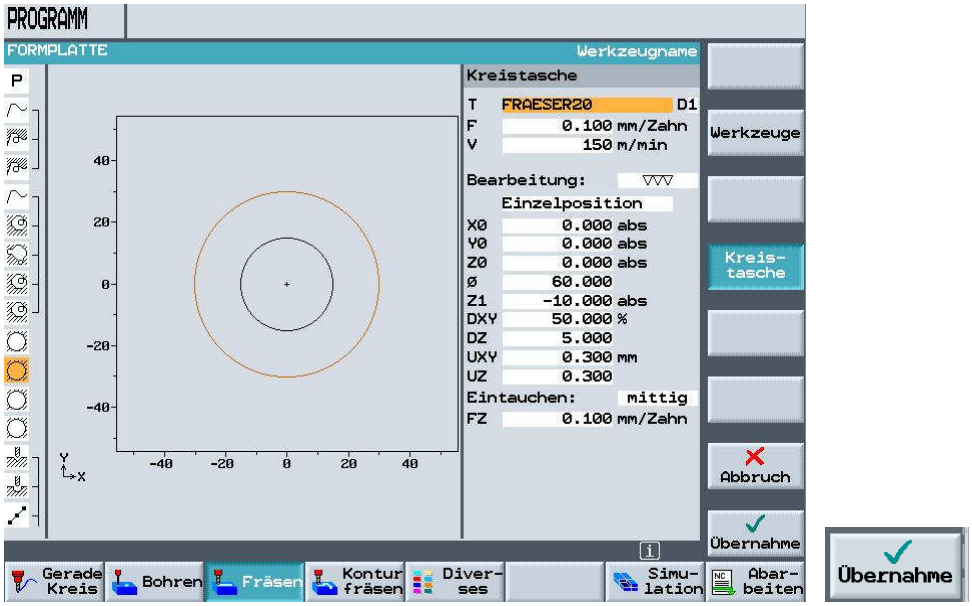


6.17 Finishing the Circular Pocket Diameter 60

Click on 

and on 

and enter the values.
Select the tool and enter the cutting data.
Accept all additional data from the picture.



6.18 Roughing the Circular Pocket Diameter 30

Rough the circular pocket yourself.
Important: The starting depth is no longer at “0“ but at “-10mm“

6.19 Finishing the Circular Pocket Diameter 30

Rough the circular pocket yourself.
Important: The starting depth is no longer at “0“ but at “-10mm“

6.20 Centering and Drilling taking Obstructions into Account

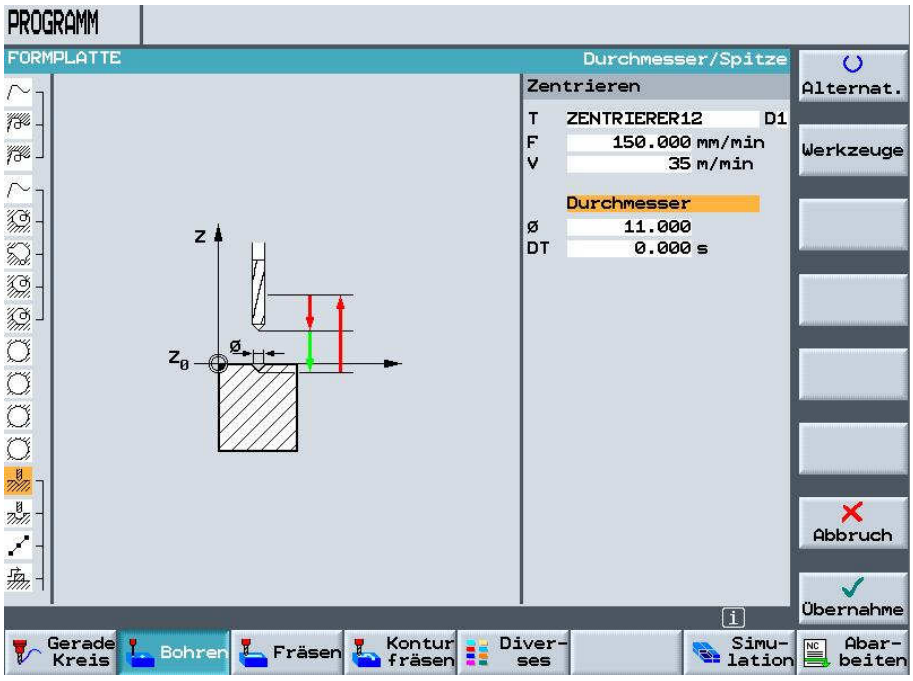
For centering and drilling, first the corresponding machining steps are programmed, and then the individual positions are specified.
The obstructions have to be taken into account when positioning.

6.21 Machining Step Centering

Click on 

and on 

and enter the values.
Select the tool and enter the cutting data.
Accept all additional data from the picture.



6.22 Machining Step Drilling

Click on 

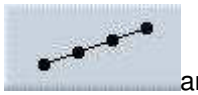
and on 

and enter the values.
Select the correct tool for drilling, enter the cutting data

and exit the menu with 

6.23 Inputting the Line of Holes to the Left

Click on 

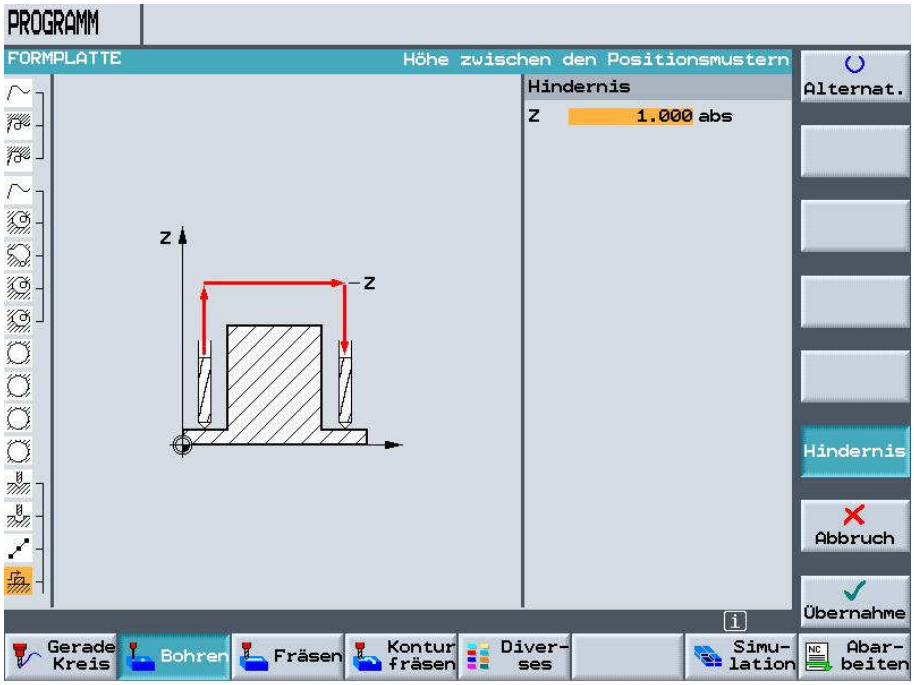

and enter the values for the

"left" side. 

6.24 First Obstruction

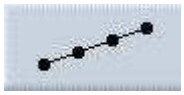
Click on 

and enter the Z value. 



6.25 Inputting the Line of Holes to the Right

Click on 

 and enter the values for the "right" side. 

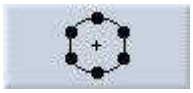
6.26 Second Obstruction

Click on 

and enter the Z value. 

6.27 Drilling the Full Circle

Click on 

 and enter the values for the right hole circle. 

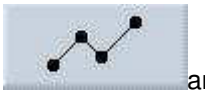
6.28 Third Obstruction

Click on 

and enter the Z value. 

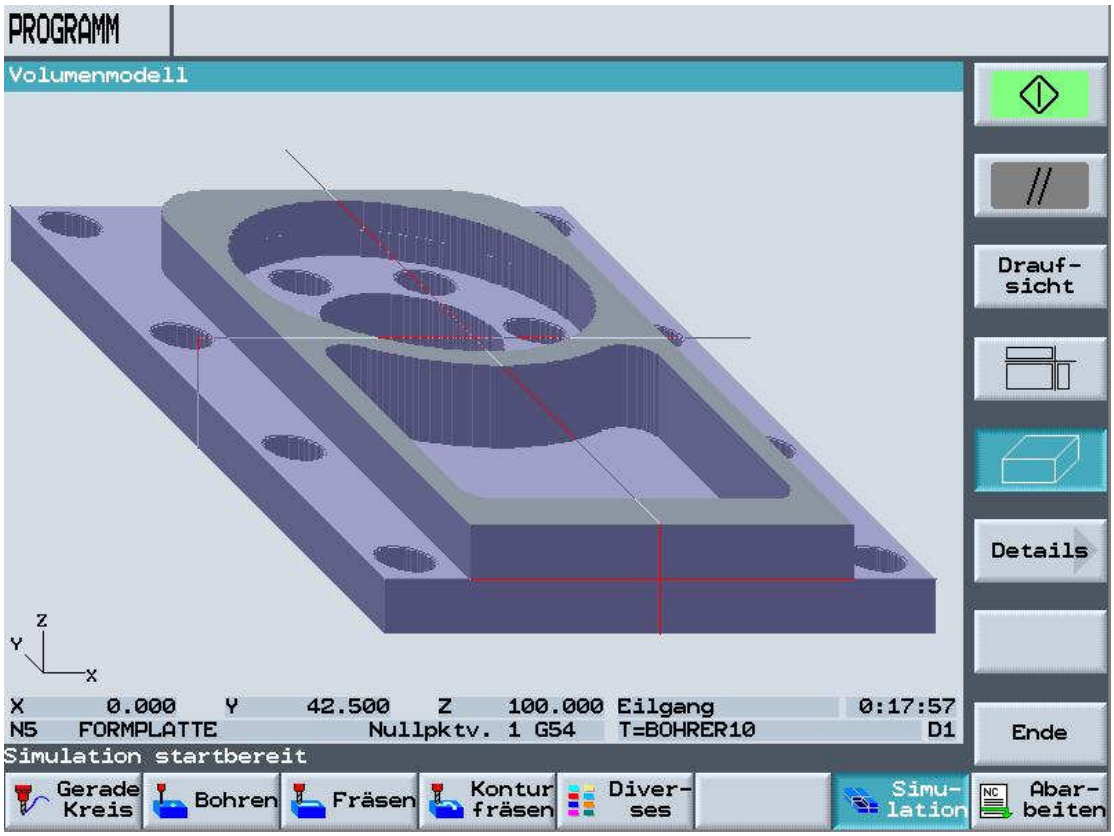
6.29 Drilling a Single Hole

Click on 

 and enter the values. 

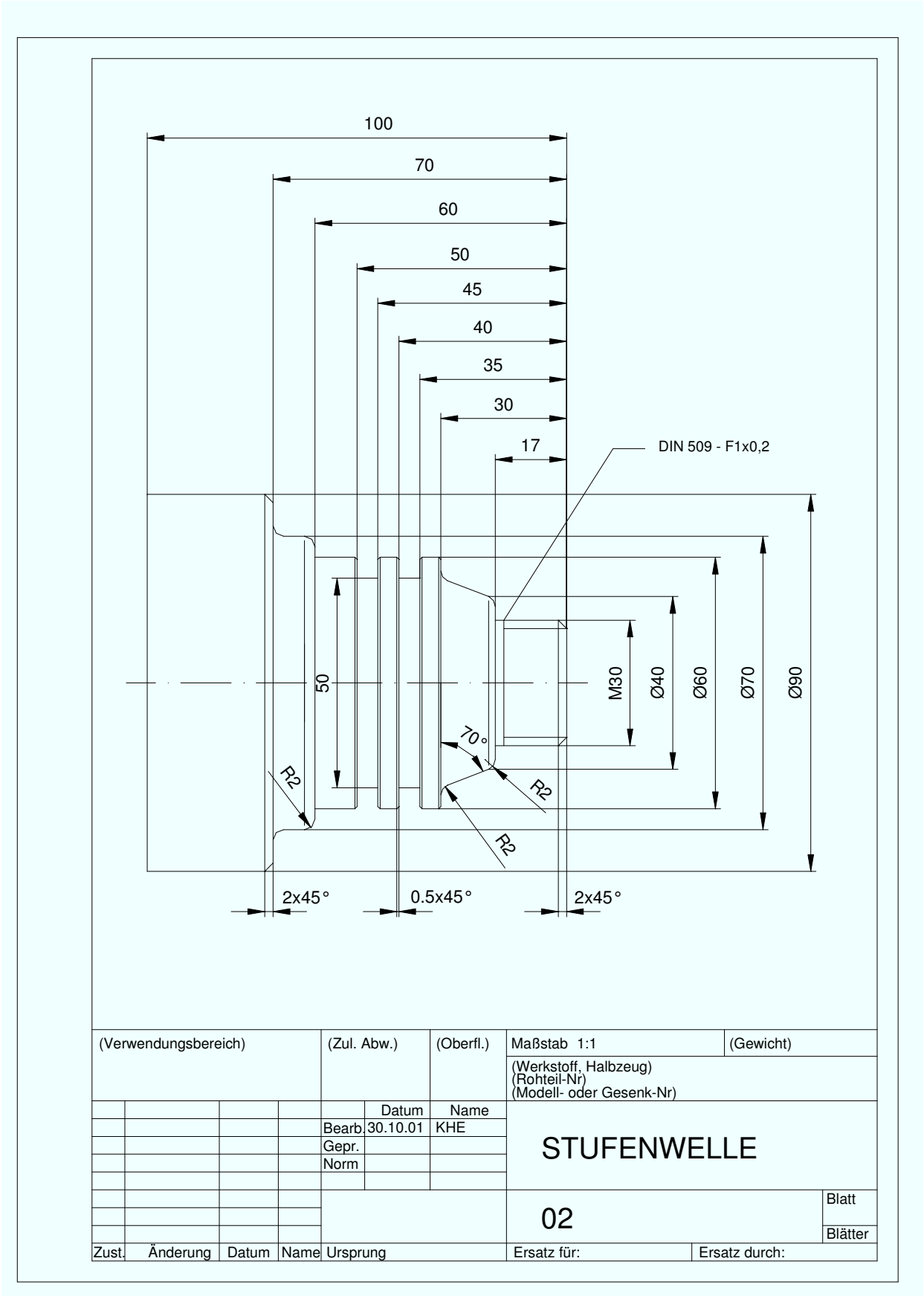
6.30 Simulation

Finally, test your program by using the different simulation displays.

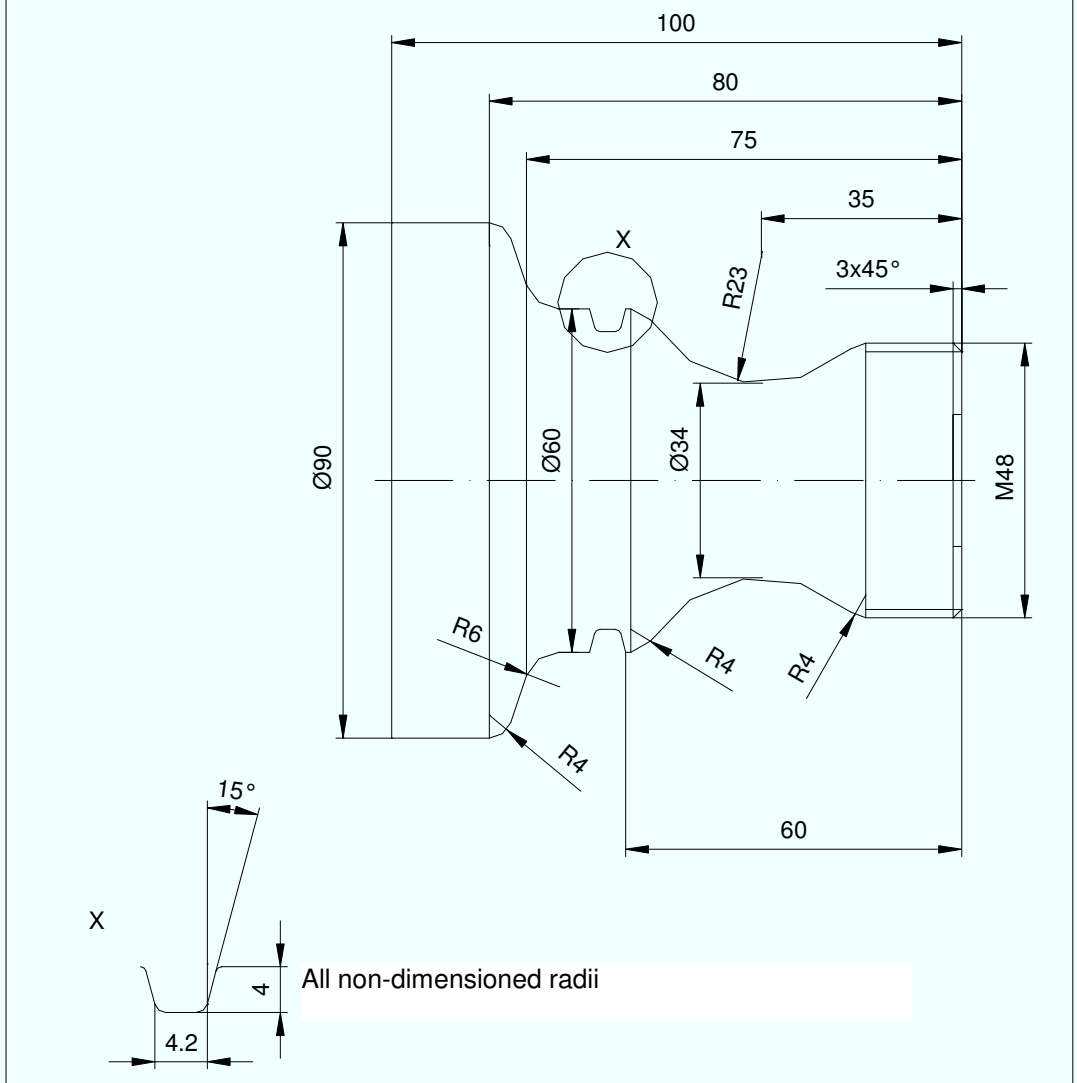


7 Additional Tasks for Turning

7.1 Step Shaft



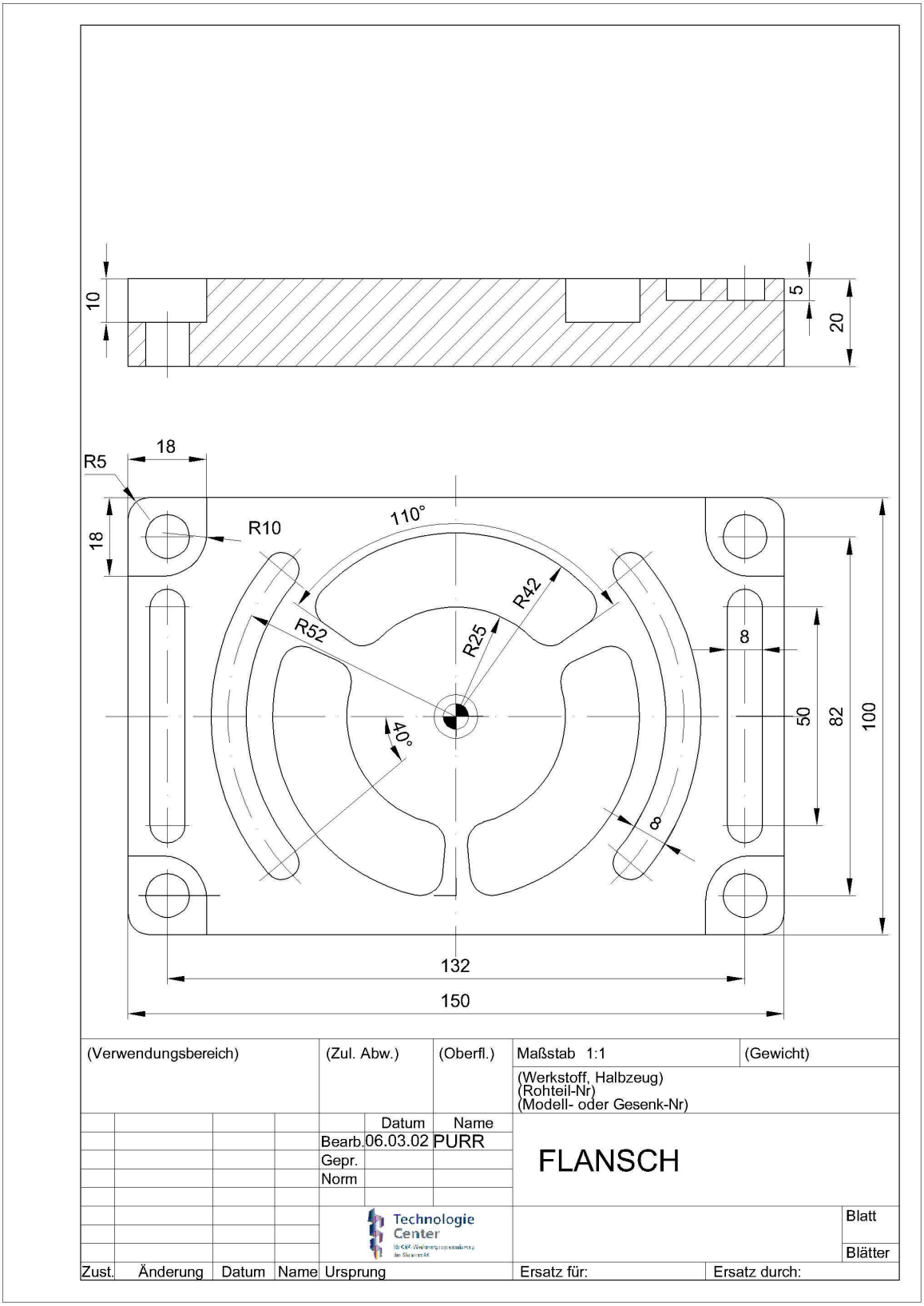
7.2 Shaft02



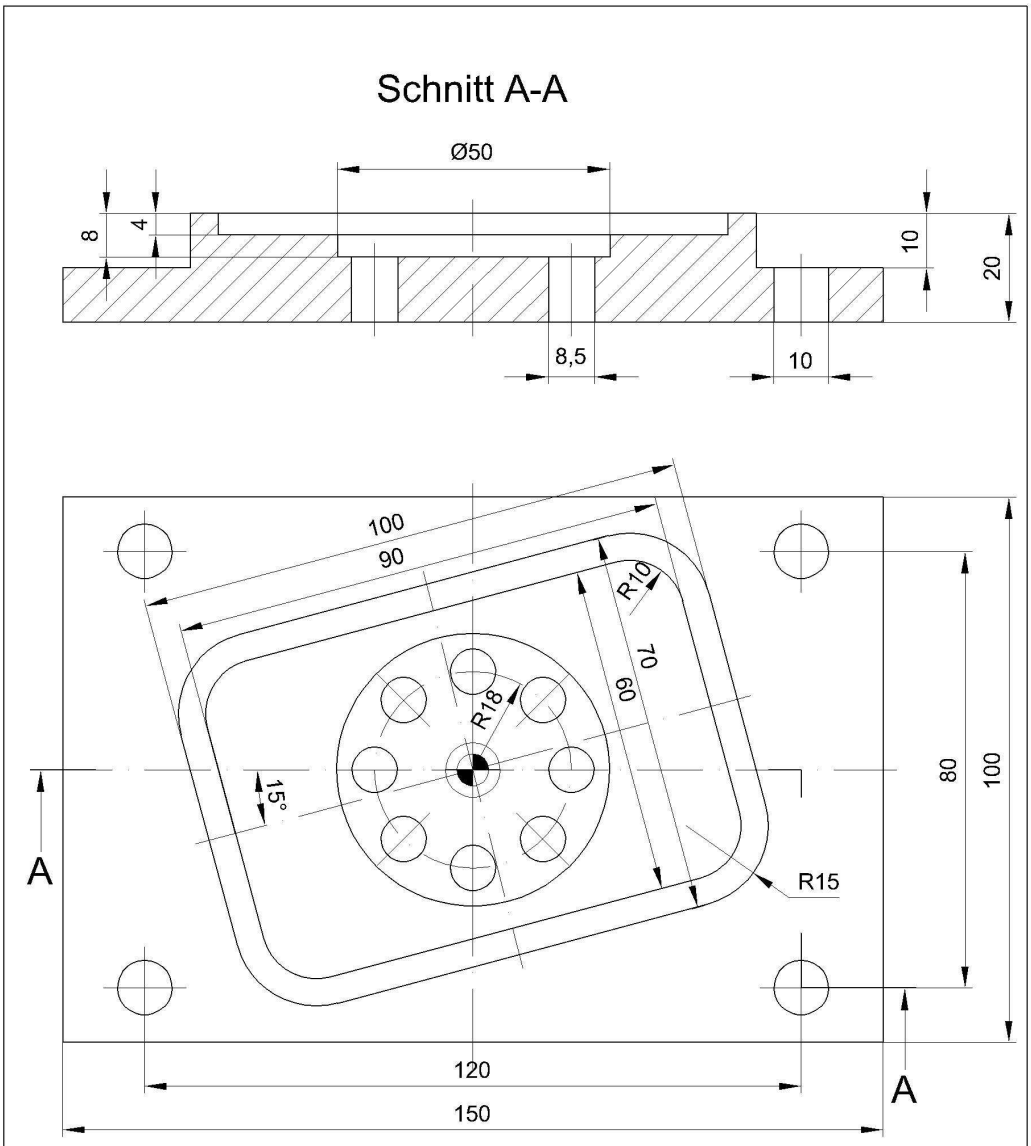
(Verwendungsbereich)				(Zul. Abw.)		(Oberfl.)	Maßstab 1:1	(Gewicht)
							(Werkstoff, Halbzeug) (Rohteil-Nr) (Modell- oder Gesenk-Nr)	
					Datum	Name	WELLE02	
				Bearb.	30.10.01	KHE		
				Gepr.				
				Norm				
							05	Blatt
								Blätter
Zust.	Änderung	Datum	Name	Ursprung			Ersatz für:	Ersatz durch:

8 Additional Tasks Milling

8.1 Flange



8.2 Exercise11



(Verwendungsbereich)				(Zul. Abw.)		(Oberfl.)	Maßstab 1:1	(Gewicht)
							(Werkstoff, Halbzeug) (Rohteil-Nr) (Modell- oder Gesenk-Nr)	
					Datum	Name	UEBUNG 11	
				Bearb.	06.03.02	PURR		
				Gepr.				
				Norm				
				Technologie Center				Blatt
								Blätter
Zust.	Änderung	Datum	Name	Ursprung			Ersatz für:	Ersatz durch: