Training Document for Integrated Automation Solutions

Totally Integrated Automation (TIA)

Module S02

CNC Programming Turning

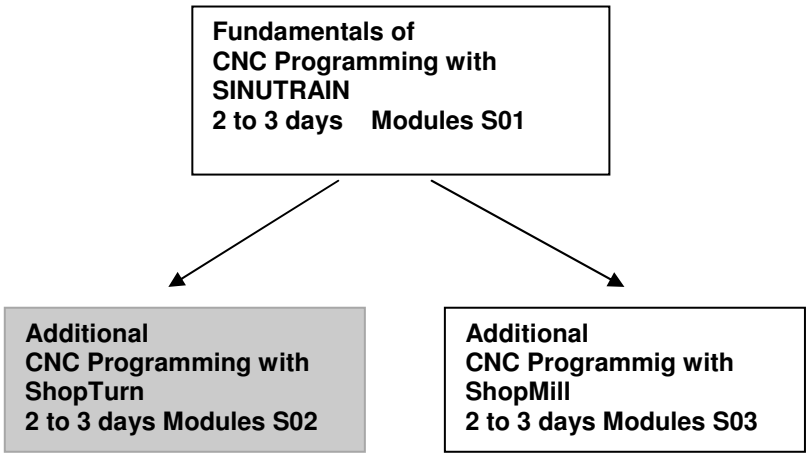
ShopTurn

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

The training document **"Programming with ShopTurn'** is used for getting to know the software.



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 performed economically with the controllers ShopMill and ShopTurn.

Training Objective:
Module S02 shows you step by step how to program with ShopTurn.
Subsequently, the reader is to be able to solve the tasks provided.

Prerequisites:

To successfully work through Module S02, the following knowledge is assumed:

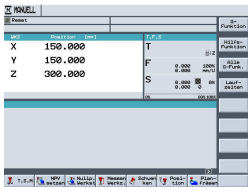
- Knowledge in handling Windows
- Fundamentals of CNC programming with Sinutrain (for example, Module S01)

Hardware and software required

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 installation of 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 in 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 Requirements for Controllers in 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

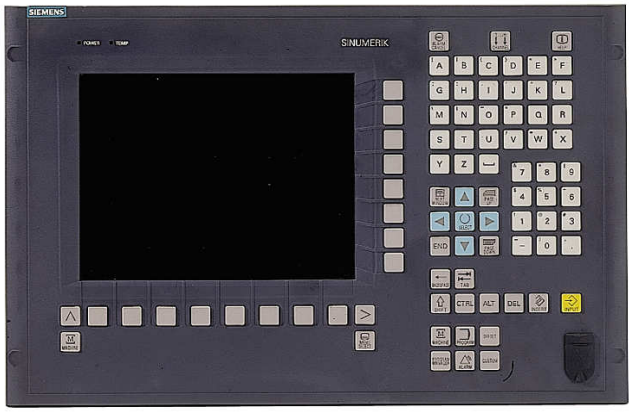
2.3 Advantages of CNC Programming with SinuTrain SHOPMILL, SHOPTURN

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, it always takes place with the same program interface and the same menus or functions.
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 machining 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 work plan by means of individual machining steps, all editing steps are provided in a straightforward manner.

3 Operator Components

This chapter shows the basic operator components. These components are to be considered examples, and are not necessarily included on the machine in the design described.
Take note of the information by the machine manufacturer.

Here an example of an operator panel of the type OP010C.



This operator panel consists of a screen with horizontal

and
vertical

softkeys.
They are used for calling individual cycles, programs and functions.

Depending on the operator panel, an alpha/numeric block and a correction block is provided on the side.

Here an example of a machine control panel.

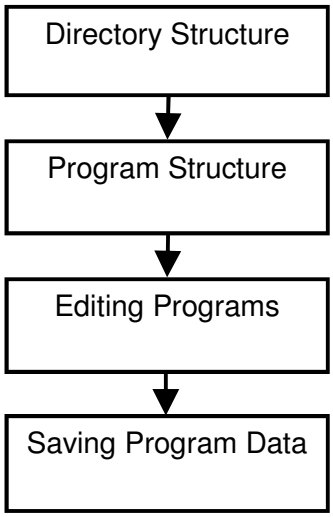
Machining the workpiece is started by means of the machine control panel, for example.

In this chapter, the keys preassigned by Siemens and their functions are not discussed any further, since they are described in detail in the operating instructions "Operation/Programming“.

4 Program Management Turning

The following is described in detail: the structure, the management and saving programs under ShopMill <<should be ShopTurn?>>.

Content of Module S02:

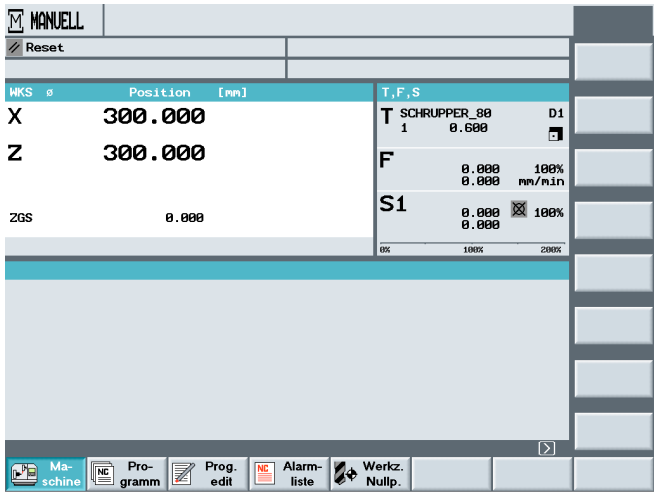


4.1 Directory

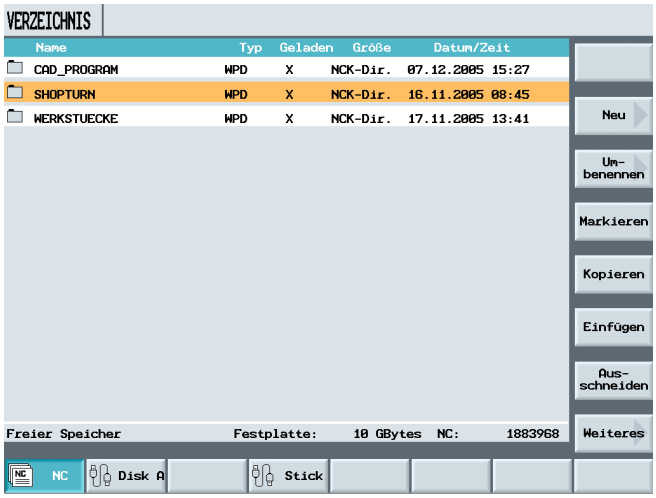
In the directories, programs can be stored. This keeps the program memory clearly arranged.

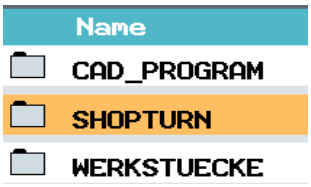
By pressing the softkey

the available directories under ShopTurn are opened in the operating area

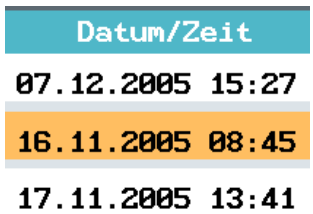


The following is displayed: the names of the directories,





the directory type,

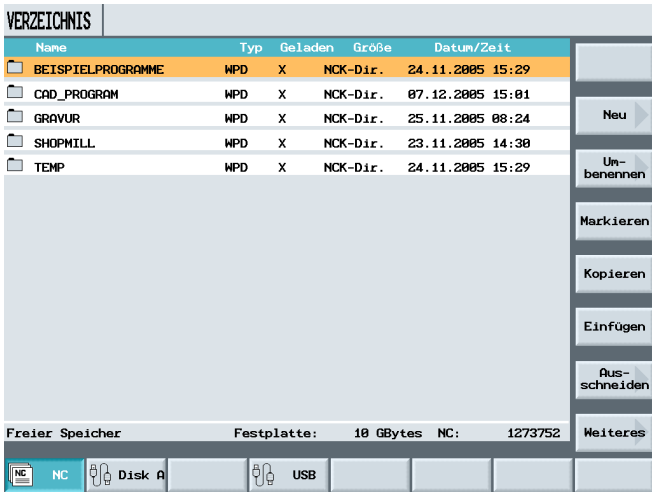


as well as the generation date and time.

Directory Structure

The size of the directory is not displayed.
The X in the area “Loaded“ means that the directory is loaded on the NC of the machine.

Geladen	Größe
X	NCK-Dir.
X	NCK-Dir.



4.2 Program Structure

By opening a directory, existing programs can be accessed, or new ones can be set up.

By pressing the arrow key

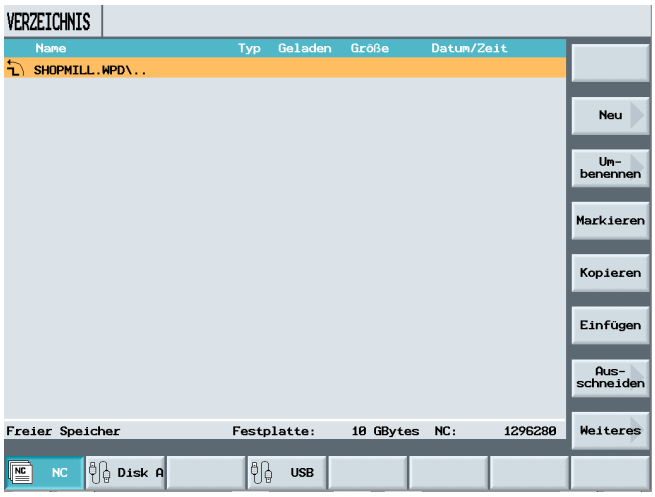
on the CNC keyboard,

the selected directory opens

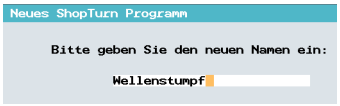
In ShopTurn, only main directories can be created - mpf – no subdirectories - spf -.

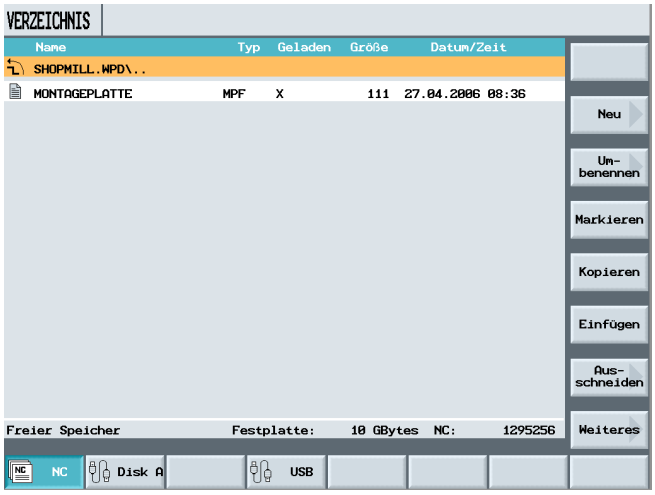
After opening the directory, programs existing in the directory are displayed. New programs can be created.

By pressing the softkey



new programs are created that are managed in the opened directory.





Programming and the program structure are described in the chapter Program Structure.

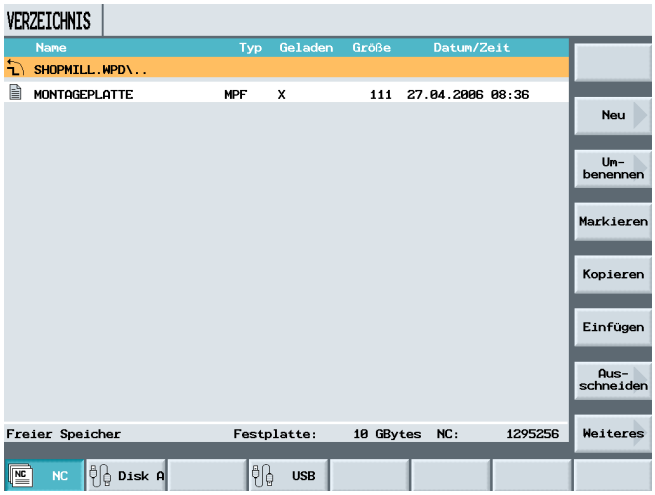
Pressing the arrow key

closes the directory. You are returned to the directory overview.

4.3 Editing Programs

The same functions are available as in Word.

With the softkeys



directories, programs, or individual program parts can be edited.

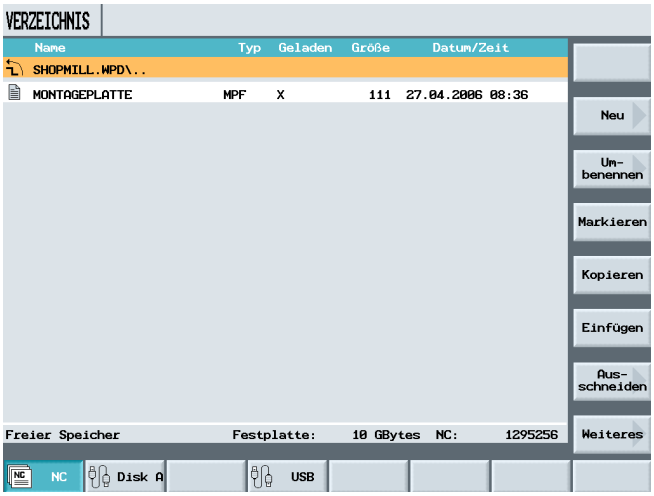
The individual softkeys are not described in detail since they occur in the exercises below..

4.4 Saving Program Data

Here, all important program data such as tools, zero points, etc. can be stored.

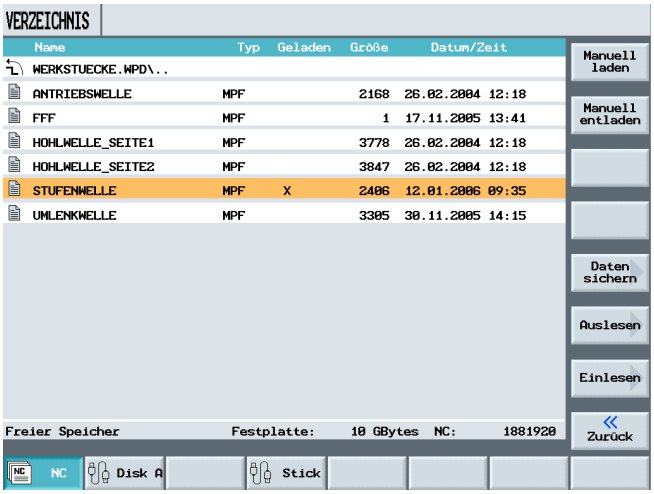
By pressing the softkey

in the operating area



the softkey below appears in the vertical softkey bar

With "Save data“, the relevant machining data of the currently selected program can be saved.

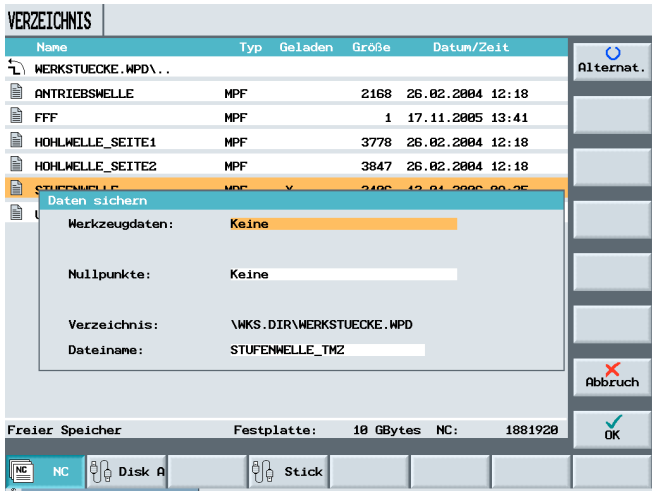


Pressing this softkey

opens a dialog field.

By pressing the softkey

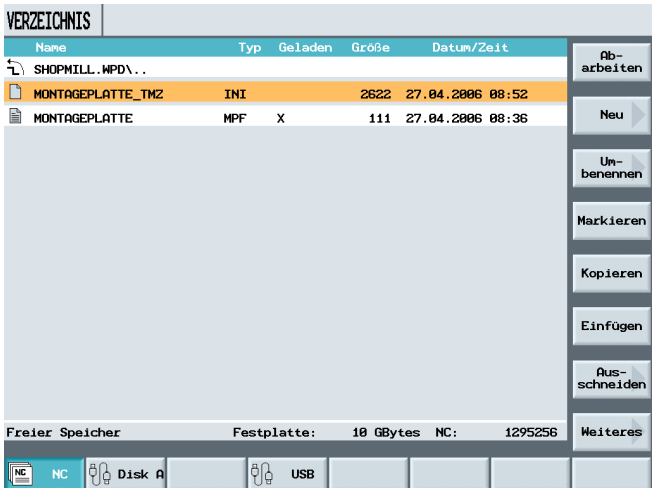
the desired data is selected.



By pressing the softkey

an "ini“ file is generated in the in the operating area

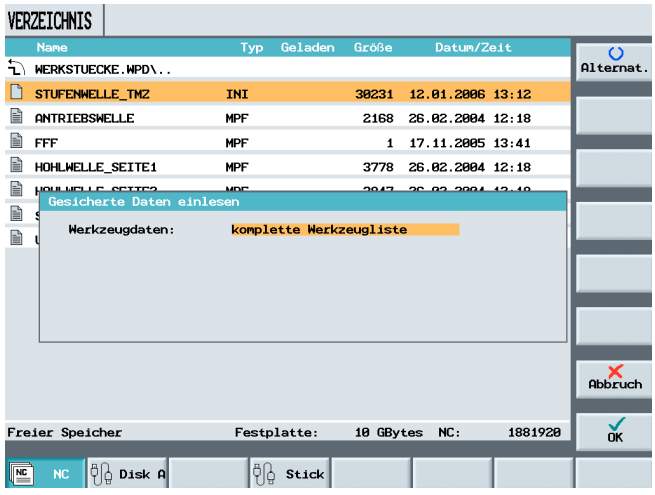
with the program name whose data was saved in the file.



The program with the correcponing “INI“ file can now be saved externally. With the program and the INI file, all relevant data for manufacturing the workpiece is saved, and can be called any time.

When an “INI file“ is selected, all saved data is read in again.

When reading in the tools used in the program, a query is displayed regarding tools with the same name: whether the current tool is to be overwritten.



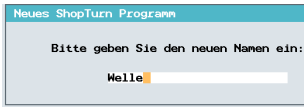
Here, the “complete tool list“, or the tools “used in the program“ can be stored.
If the “complete tool list“ is read in, all existing tools are deleted and replaced with the stored tools.

5 Program Structure Turning

5.1 Program Header

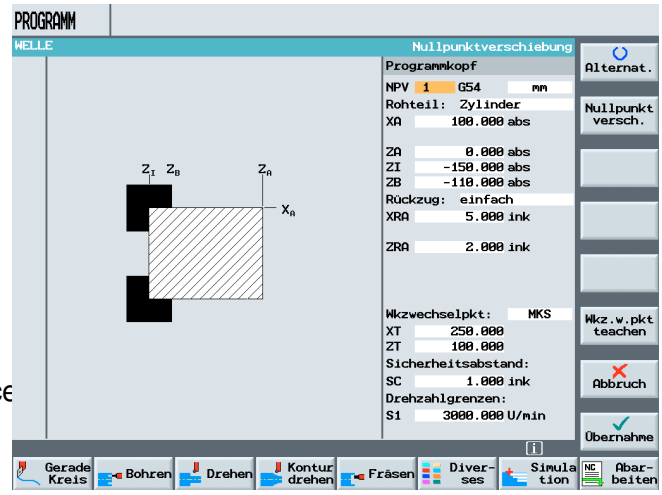
In the program header, the basic settings are described.

After entering the program name



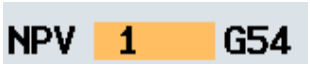
and confirming the input with

the program header of the new program is opened automatically.

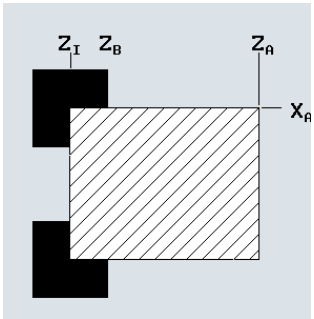


Here, the basic settings for the program sequence are made.

As the first input, a zero point shift can be programmed for the program directly in the program header.

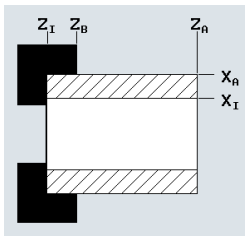


In the next input field, the raw part type is defined.

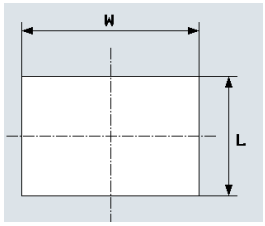


We differentiate between

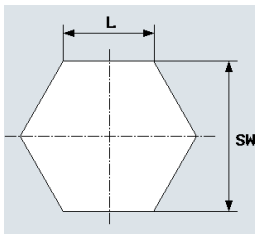
Rohr



Rechteck

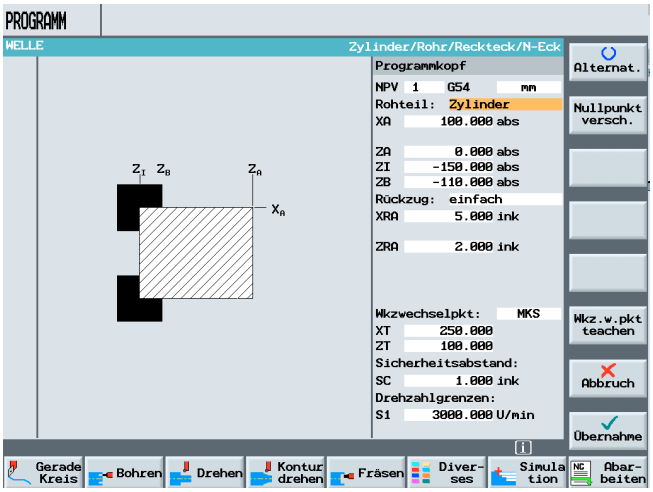


N-Eck



With the next input fields, the deviations of the unmashined part are defined.

ZA	0.000 abs
ZI	-150.000 abs
ZB	-110.000 abs



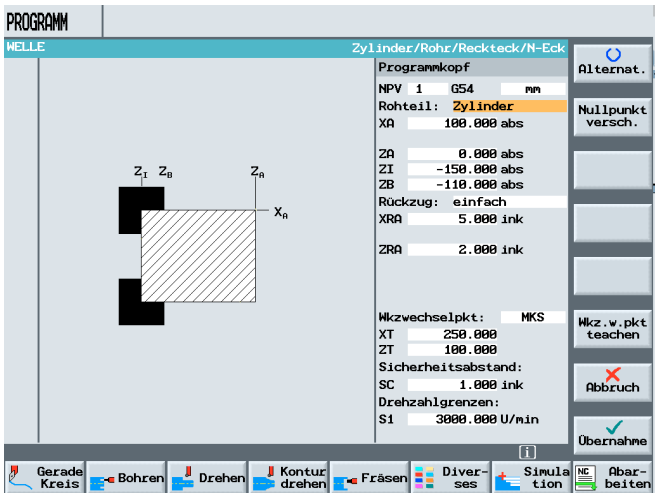
The unmachined part definition is necessary only for simulation.

With the retraction plane

Rückzug: einfach
XRA 5.000 ink

ZRA 2.000 ink

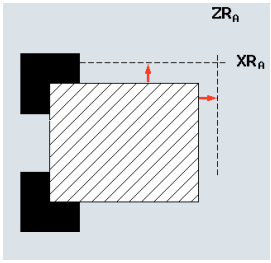
we define the distance that the tool travels with only one axis motion from the workpiece to the tool change point.



In the case of ShopTurn, we differentiate between

Rückzug: einfach

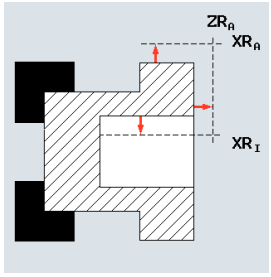
return motion single for external machining;



return motion expanded

Rückzug: erweitert

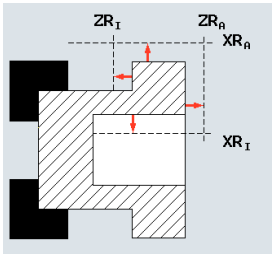
for outside and inside machining and



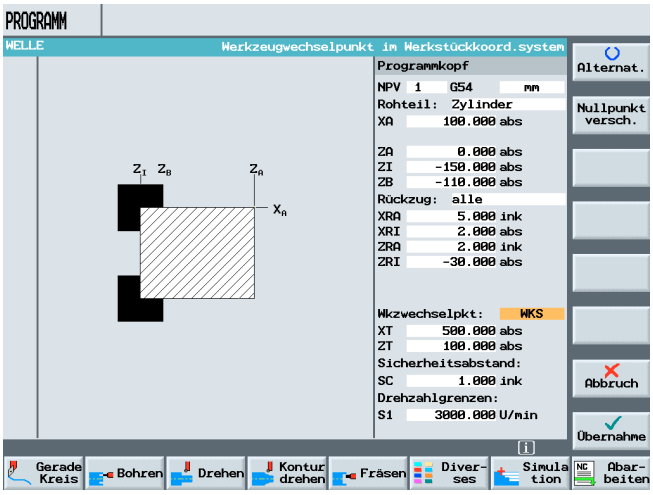
For each machining, a return motion has to be specified.

Rückzug: alle

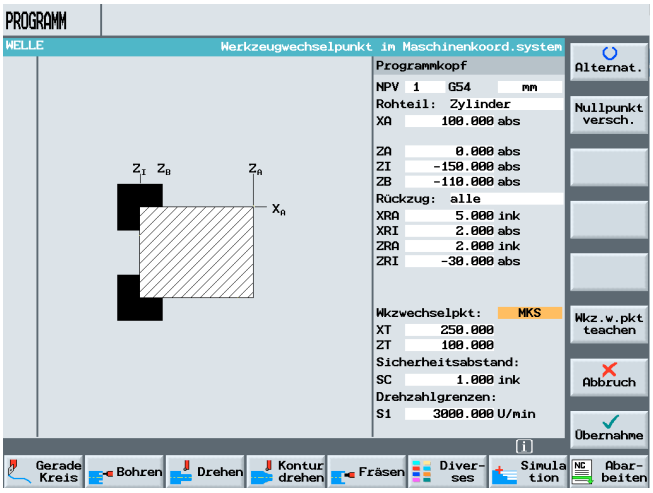
for outside, inside and backside machining.



The tool change point can be entered in the workpiece coordination system,



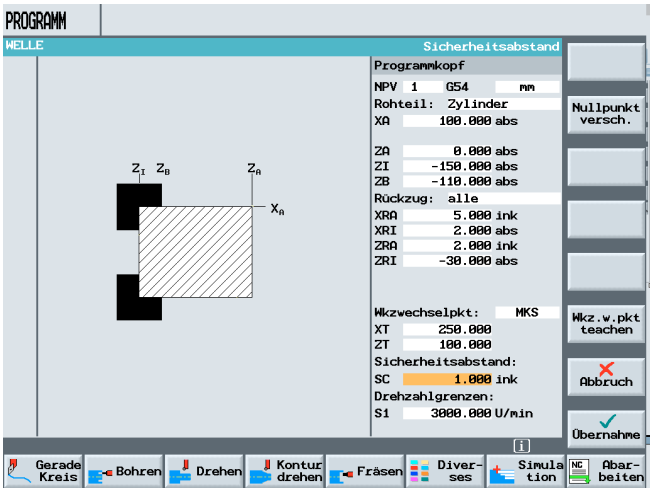
or in the machine coordination system.



If the tool change point is specified in the MCS, the current positon of the tool turret is accepted by pressing the softkey

.

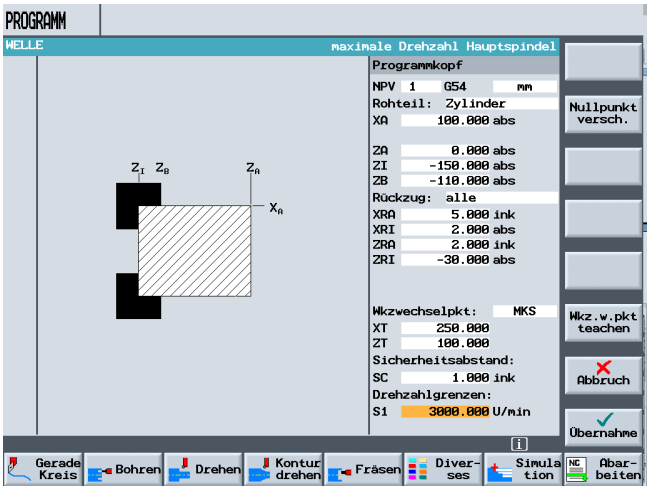
In the next input field the safety distance is defined.



Sicherheitsabstand:
SC 1.000 ink

The safety distance indicates how close the tool travels to the workpiece at its approach at rapid feed. During retraction, the safety distance specifies the distance with which the tool travels from the workpiece in the feed.

In the lower input fields, the speed limit for the main spindle and (if present) the counter spindle is entered.



Drehzahlgrenzen:
S1 3000.000 U/min

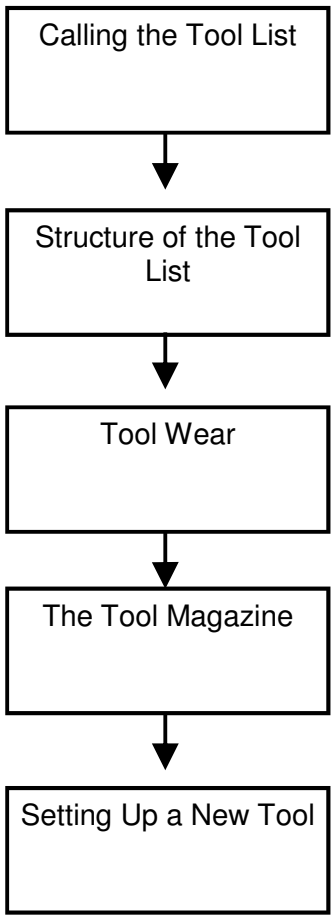
By pressing the softkey

the input is accepted in the machining plan.

6 Tool Management

In this module, the structure of tool management with the individual programming options is described.

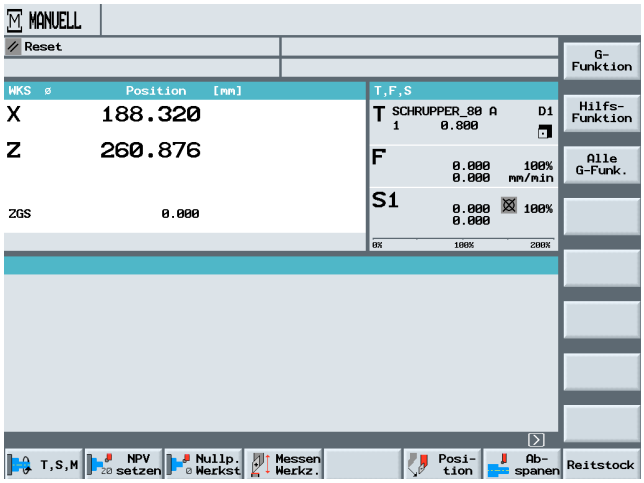
Content of the module:



6.1 Calling the Tool List

After starting ShopTurn, the operating area

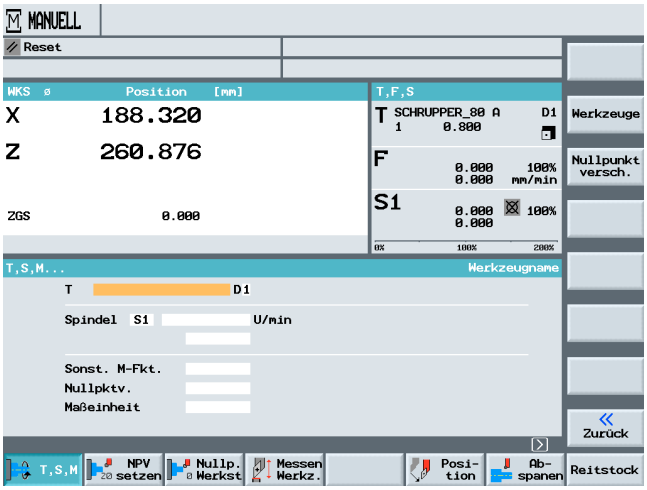
is active.



By pressing the softkey

and

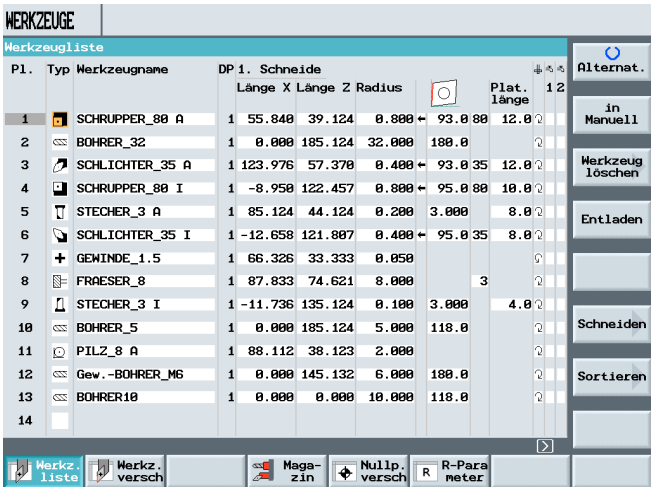
,



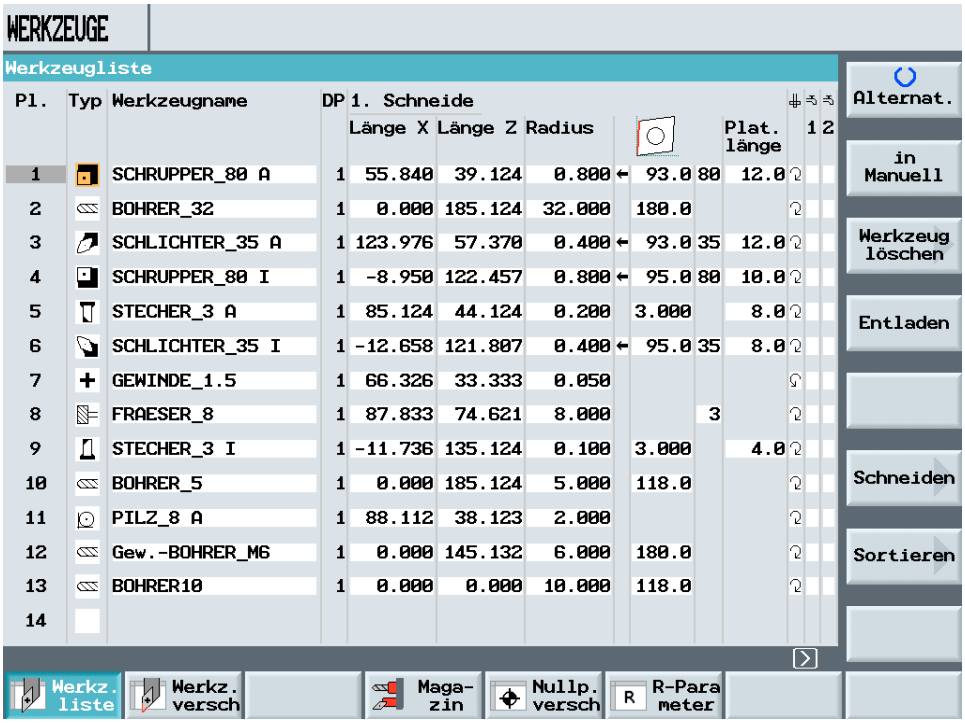
in the area

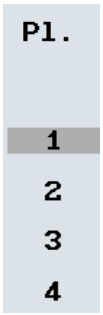
the

is called.



6.2 Structure of the Tool List





The location number:

The locaton number describes the magazine location. If a tool is located after the location number, the tool is active; that is, it is available in the magazine.

Tools that don't have a location number are not active in the magazine. They are in the "drawer". The correction data continues to be active.

The Type:

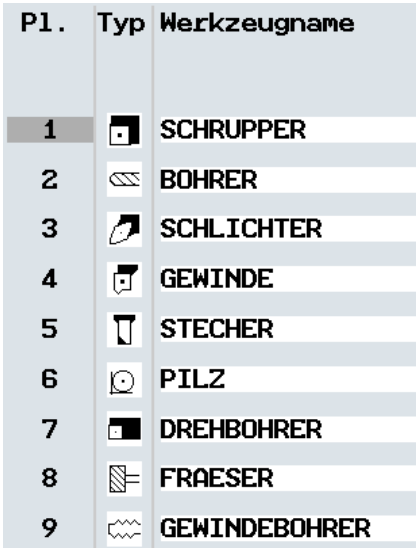
Here, a symbol is assigned to the respective tool type, as well as the corresponding tool edge position.

Example of tool edge position using a finishing tool:

6.3 Structure of the Tool List

The following tool types with the correspoding symbols are available:

In addition, there is the 3D button and the fixed stop as symbol.

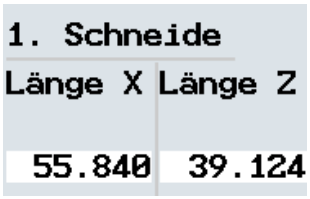


The Tool Name:

In this field, a name is entered to identify the tool.
Letters, numbers, and special characters can be entered.

The Duplo Number:

If an additional tool is set up with a name that already exists, it becomes a duplo tool (sister tool).



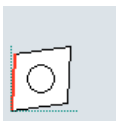
The Tool Edge Length

If a tool is measured, the tool lengths are available in the tool list.

Up to nine edges can be measured and assigned to a tool.

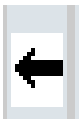
The Radius

Here, the tool radius is entered in the list.



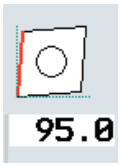
Example: Plate radius 0.8 mm.

6.4 Structure of the Tool List



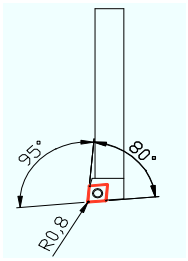
The Reference Direction for the Support Angle

This refers to the tool's main cutting direction.



The Supprt Angle

This defines the angle of the tool support.



The Tool Tip Angle

Here, the angle of the tool tip is entered.

Number of Tool Edges

If the tool that has been set up new as a milling tool, the number of tool eges is specified in this input field, not an angle.

Length of the Plate

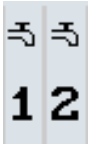
Here, the length of the tool tip is defined.

Direction of Spindel Rotation

In the case of driven tools, the DOR refers to the tool spindle, in the case of turning tools to the main or counter spindle.

Right:  i Left:  and  spindle stop

Cooling Water Supply



Under ShopTurn, inner and outer coolant supply can be activated, for example.

6.5 Tool Wear

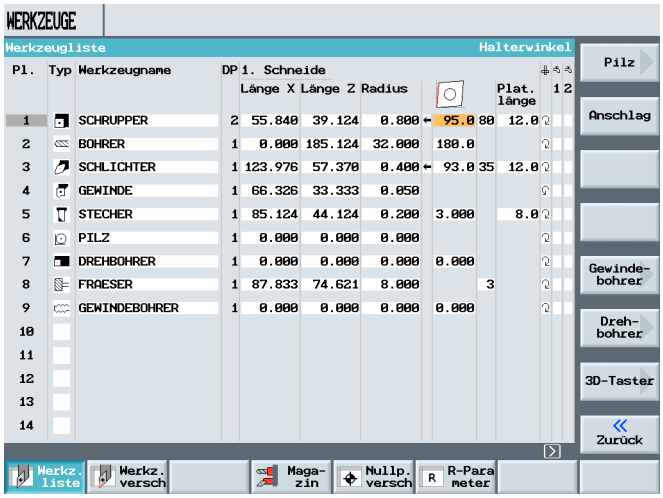
In the area

-by pressing the softkey

-

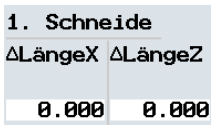
the input fields for tool wear

are activated.



Here, to the edge of a tool,

a wear value can be assigned for the length



and for the tool radius.

Starting with ShopTurn SW Version 6.4, an allowance can be entered directly in the input screen form “Contour finishing“.
It is advisable to prepare, by means of this “allowance“, exact measurements such as fitted measurements, and not to implement this by using tool wear.
The value entered in the input screen form “Contour finishing“ **only** refers to the generated program; the value entered in the tool wear, on the other hand, applies to **all** programs.

6.6 Tool Wear

The operation duration of a tool can be defined with the tool life

T Standzeit [min];

or the number if units.

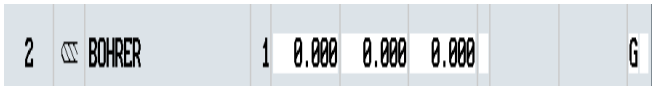
C Stückzahl

For both options, a prewarning limit can be used.

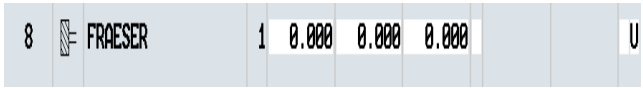
Vorwarn grenze	Stand- zeit
25.0	30.0

After the prewarning limit is reached, a message appears that the tool life or the number of loadings is soon reached.

Tools can be defined as “blocked“



or as “oversize“



G

U

6.7 The Magazine

In the area

-by pressing the softkey

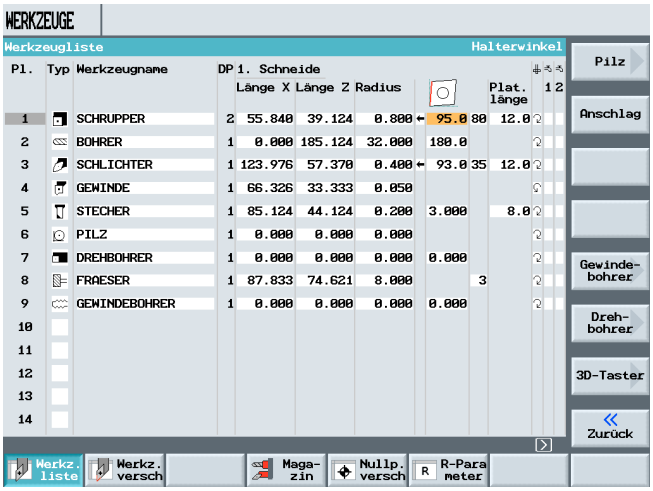
-
the input fields for the

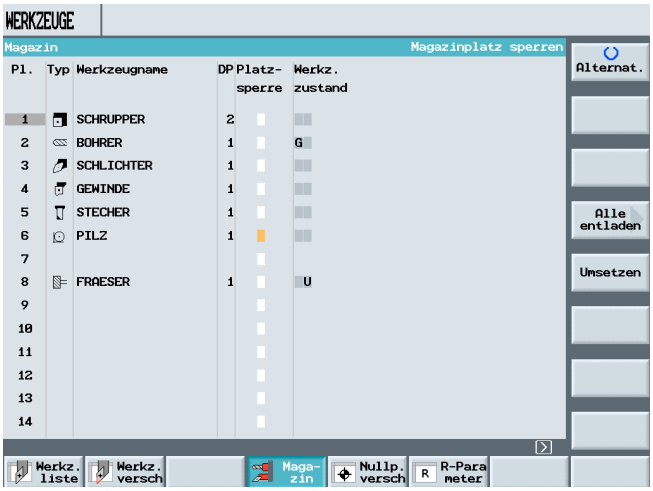
are activated.

Here, a magazine can be blocked.

The tool status

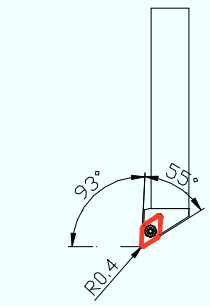
- that is, whether a tool is blocked or oversize- is displayed here.

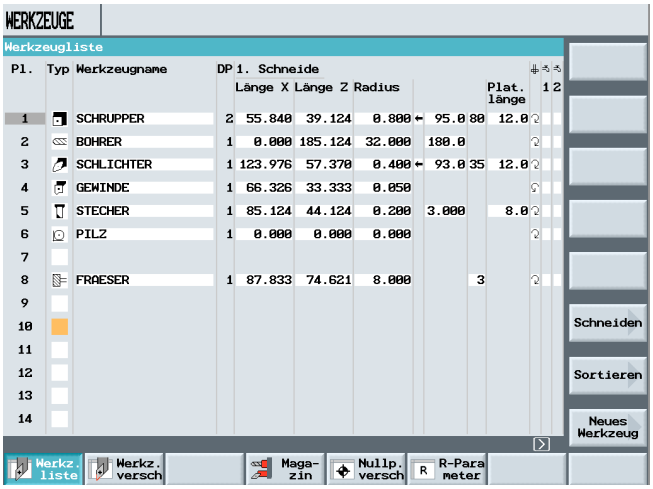




6.8 Setting Up a New Tool

As an example, the following tool is set up.





In this case, the cursor has to be on a free input field of the tool list.

By pressing the softkey New Tool,

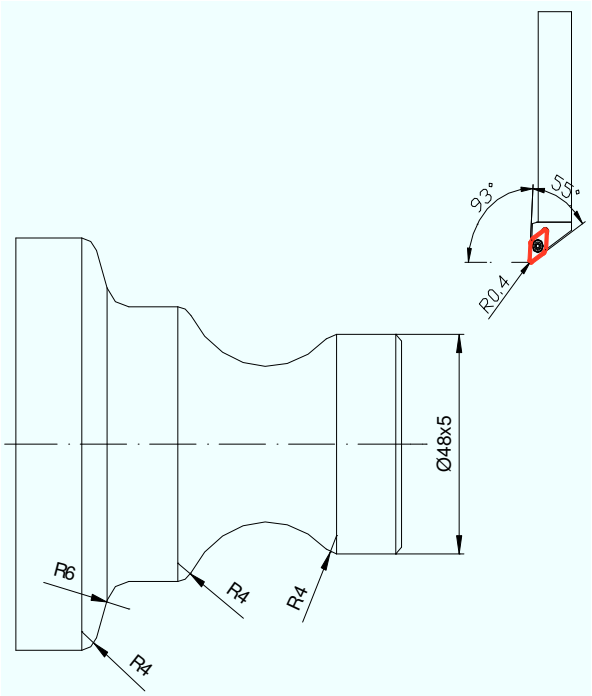
the different tool types are displayed in the vertical softkey bar.

After selecting the tool type Roughing Tool

the tool position is selected.

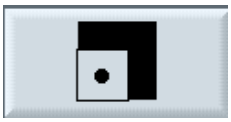
Since the machining takes place behind the turning center,

Example:

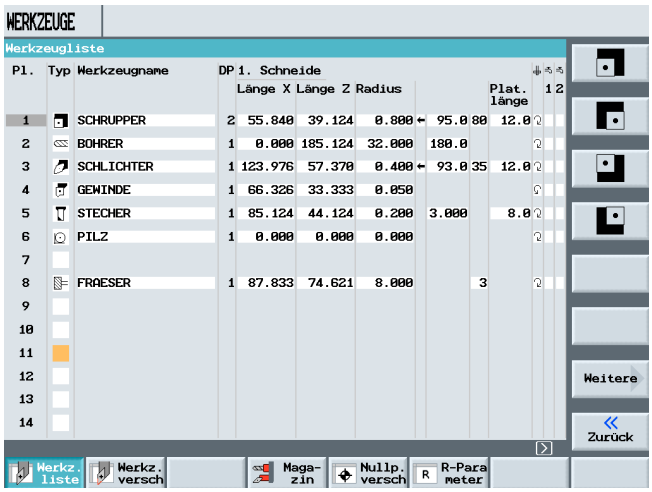


6.9 The Tool Edge Position

by pressing the softkey

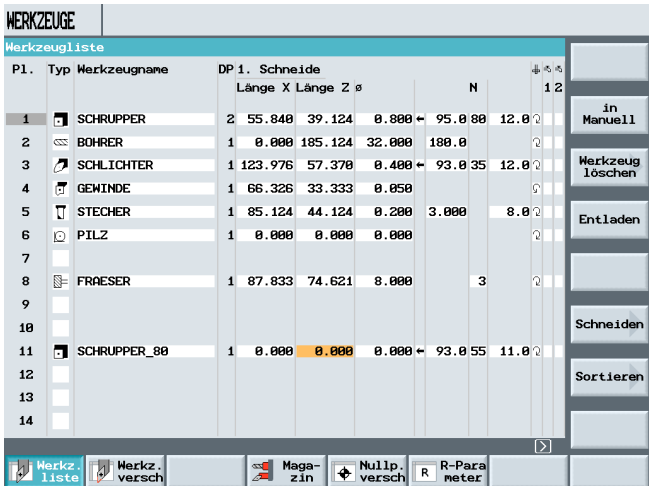


the corresponding edge position is selected.



When selecting the edge position, the tool name of the selected tool type is entered automatically in the name field.

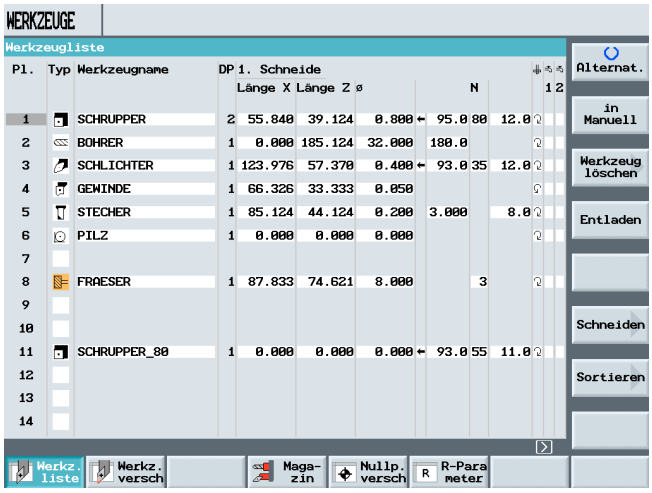
The new tool is set up.



When working with tools of the same type, unique names should be assigned since duplo tools are generared from tools with the same name without a warning.

6.10 Deleting Tools

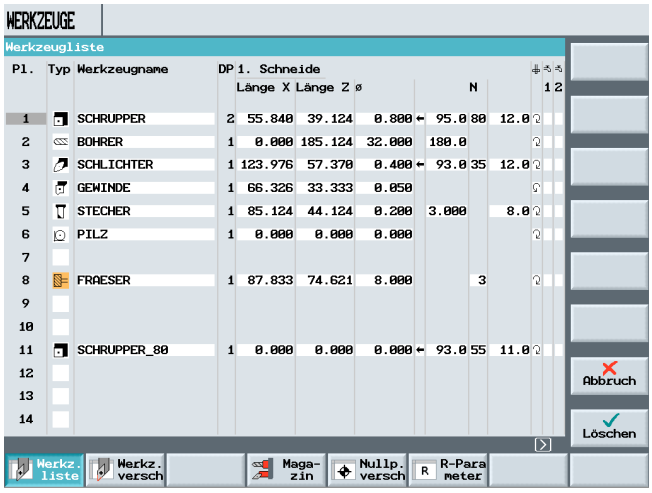
To delete a tool from the tool list, select the location with the tool.



By pressing the softkey Delete Tool

the softkeys Cancel/Delete are displayed:

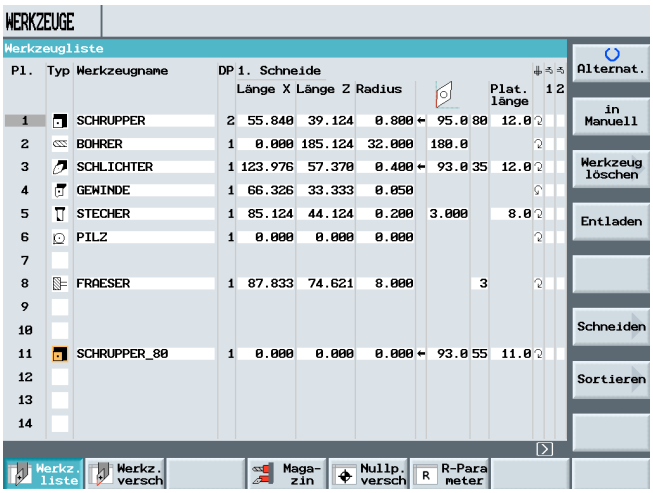
By pressing the softkey



The selected tool is deleted from the tool list.

Please note the tool manufacturer's information. In the case of some manufacturers, active tools can not be deleted.

A tool with several edges is set up as described above.

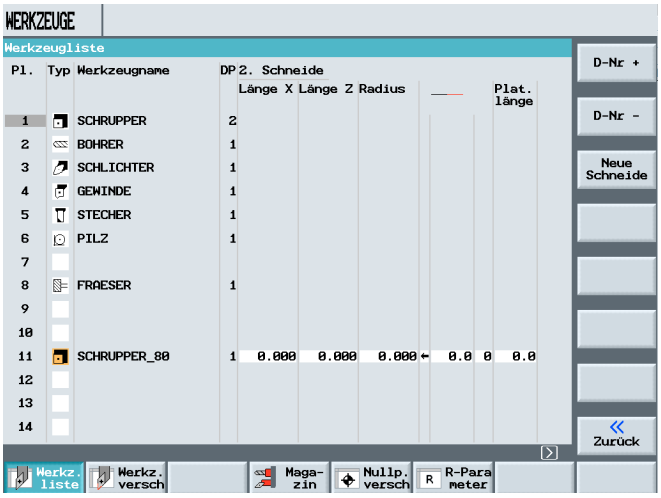


By pressing the softkey Edges

an input screen form opens where several edges can be defined.

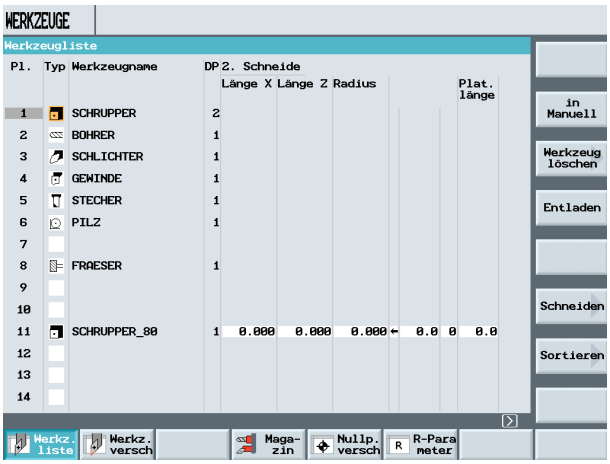
By pressing the softkey New Edge

an additional edge can be set up for the selected tool.

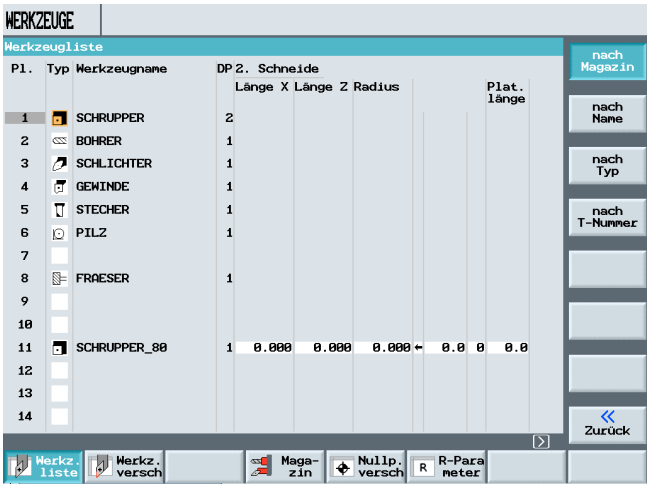


6.11 Sorting the Tools

When calling the tool list and pressing the softkey



four options are displayed for sorting tools:
according to the magazine, the name, the type, the T-number



The active softkey is darkened.

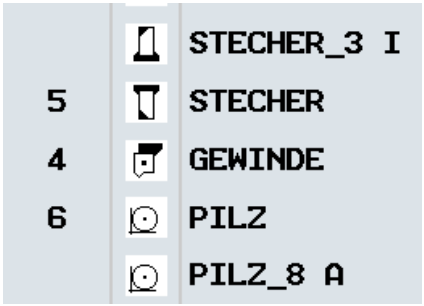
When sorting according to Magazine

the tools are shown in ascending order according to their magazine position.

When sorting

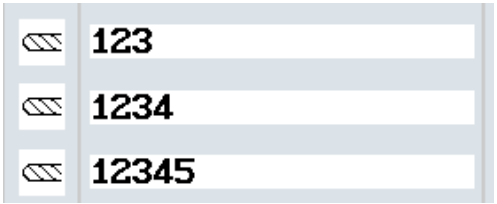
the tools are shown in the alphabetical sequence of their name.

When sorting



the tools are sorted according to type.

When sorting

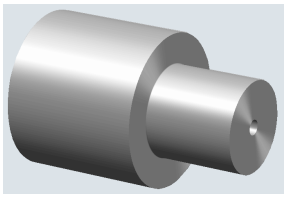


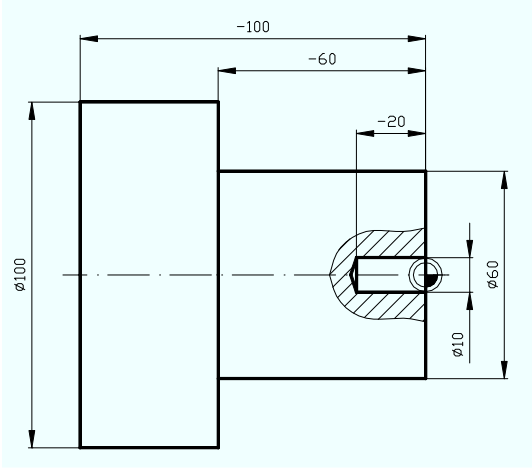
the tool names that are defined by numbers are sorted in the ascending mode.

7 Programming with Machining Cycles

Using an example, working with the machining cycles under ShopTurn is described below.

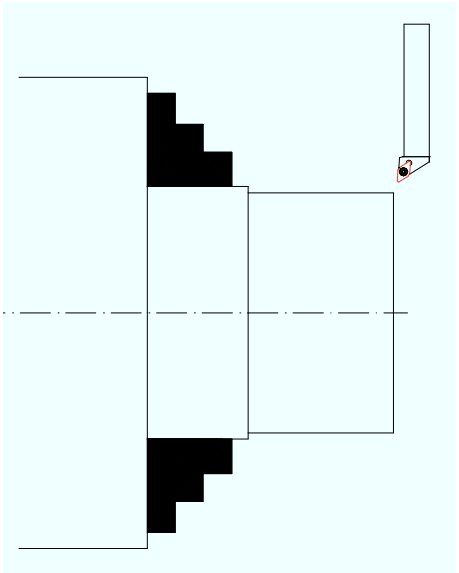
This workpiece is to be generated using the machining cycle.





After setting up a program and pressing the softkey Turning

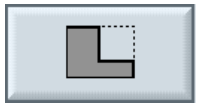
, the cycles for machining are opened.

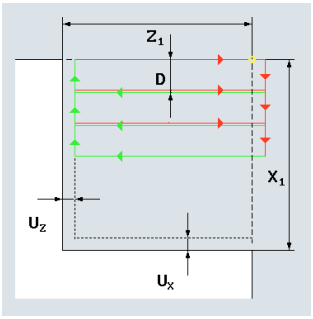


As a first step, the front face of the tool is to be faced.

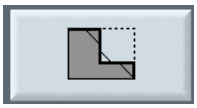
ShopTurn provides three machining cycles.

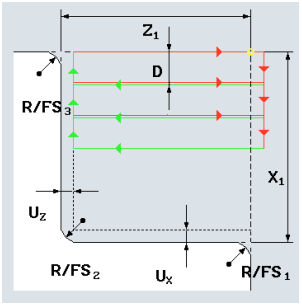
Machining without radius and chamfer



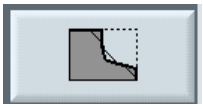


Machining with radius and chamfer,

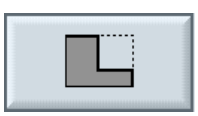


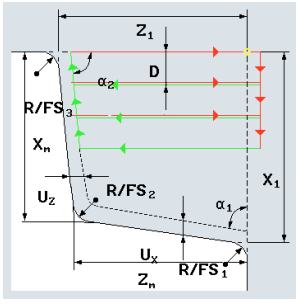


Machining with a slope, radius, and chamfer.

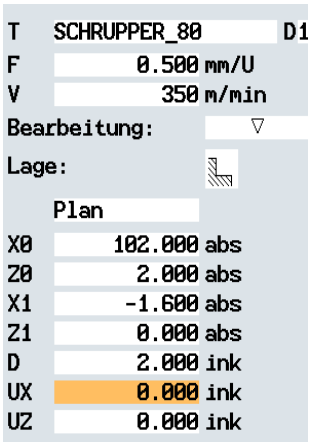


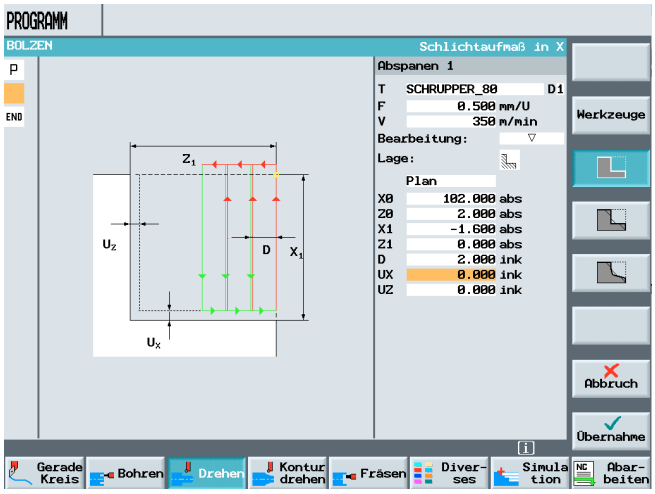
For facing the frontal face, the machining cycle is used without radius and chamfer.





The following values are to be entered in the input screen form.

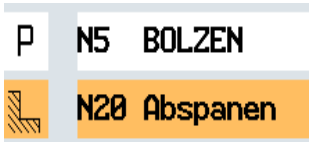


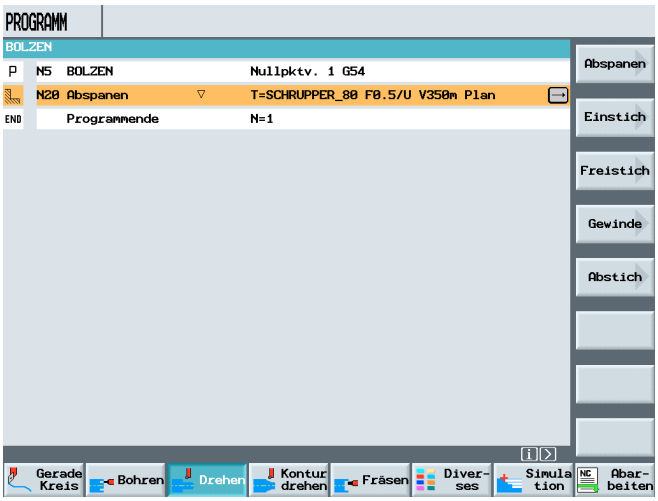


For facing, the value for the depth X-1.6 should correspond to 2x plate radius size (0.8mm) of the tool; because otherwise, material remains in the center.

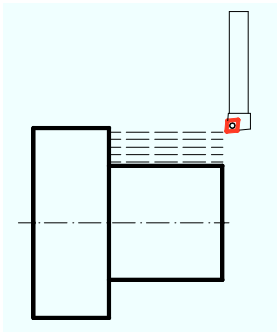
After accepting the input in the machining plan,

next the length is machined.

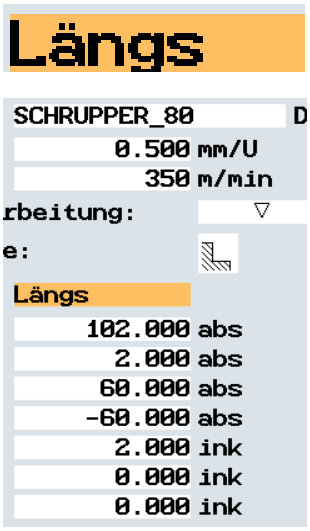


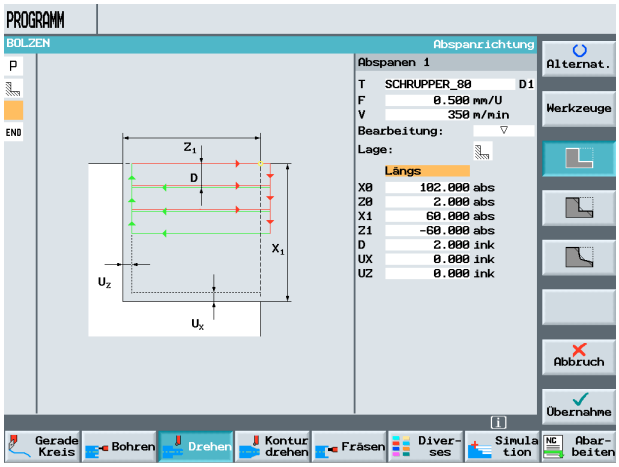


has to be defined
<<something missing here?>>.



Machining is done again with the same machining cycle, only with machining the

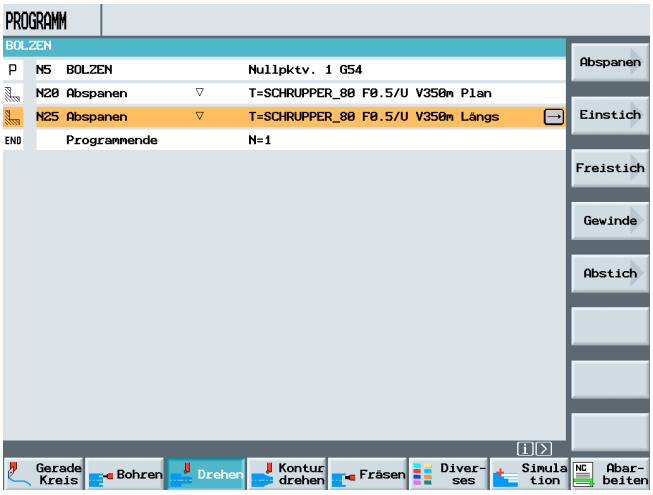




The input values can be retained modally. The values entered last appear again in the input screen form after the cycle is called again. Thus, in the example, the correct starting point appears as value in the screen after again calling the cycle for longitudinal machining.

By pressing the softkey Accept

the values in the input screen form are accepted into the machining plan.



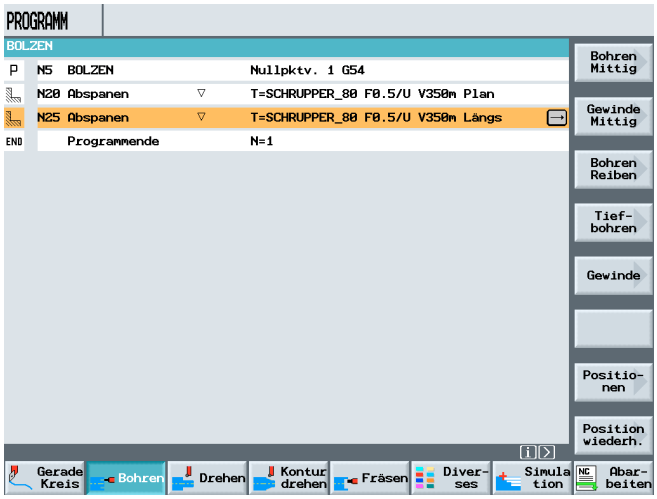
Since the workpiece is only to be roughened, the turning operation is done.

Next, center hole is made.

By pressing the softkey

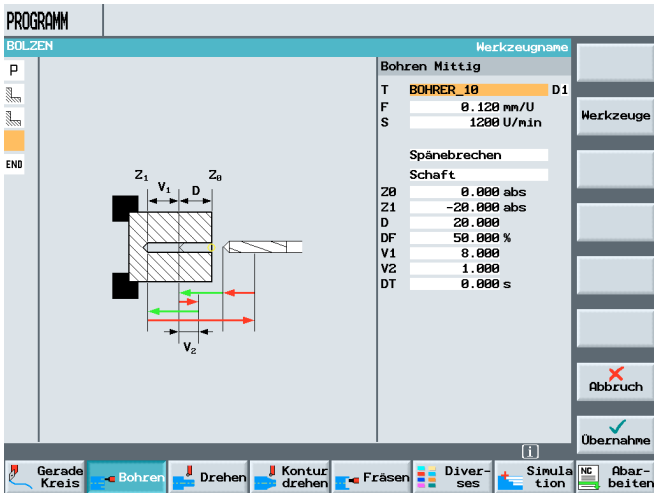
and

the input screen form is opened.

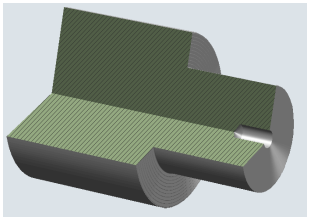


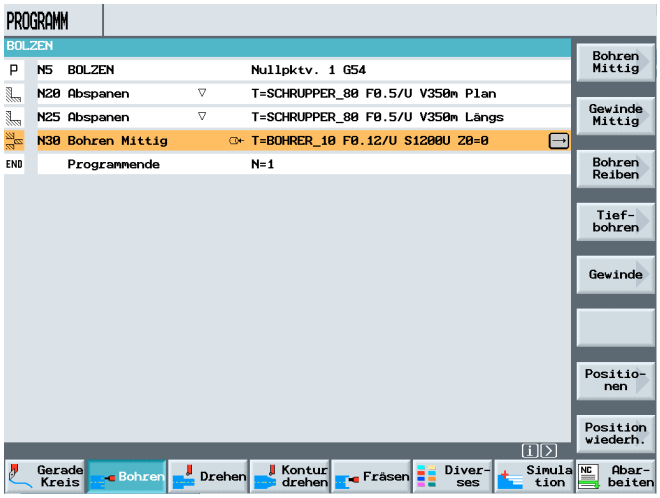
After entering the corresponding values

T	BOHRER_10	D1
F	0.120	mm/U
S	1200	U/min
	Spänebrechen	
	Schaft	
Z0	0.000	abs
Z1	-20.000	abs
D	10.000	
DF	50.000	%
V1	8.000	
V2	1.000	
DT	0.000	s



and acceptaing them in the machining plan of ShopTurn, the program is done.

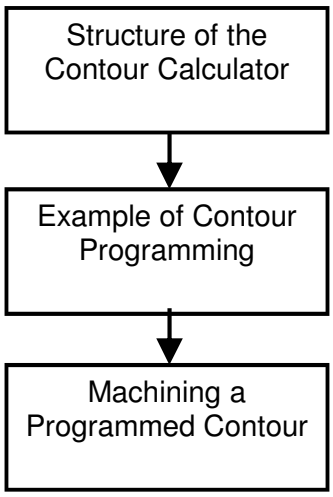




8 Contour Programming Turning

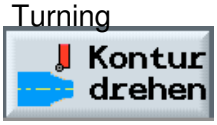
After finishing this module, you will be able to follow the operating philosophy of the contour calculator under ShopTurn.

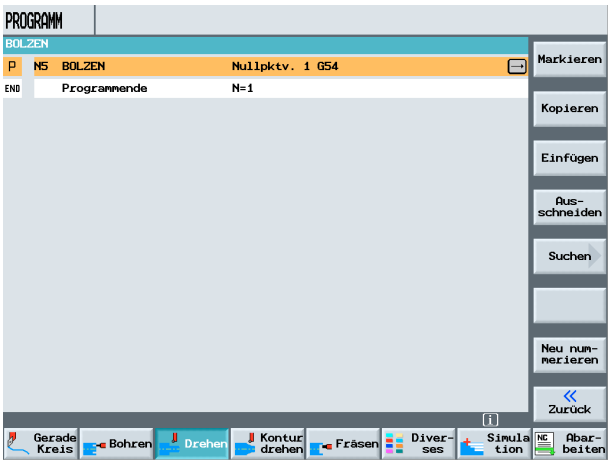
Using an example, the contour calculator
and the subsequent machining is described.



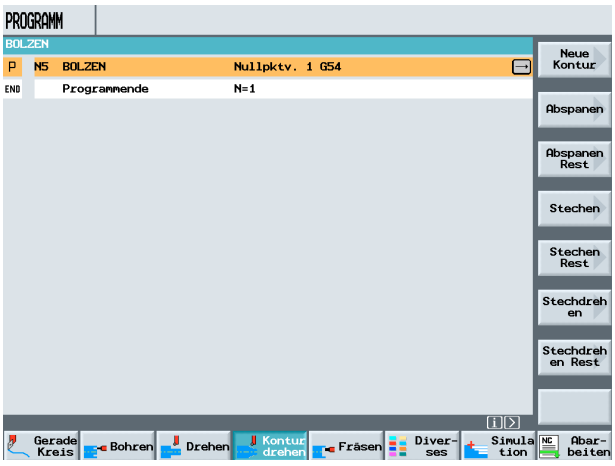
8.1 Structure of the Contour Calculator

After setting up a program, pressing the softkeys Contour Turning

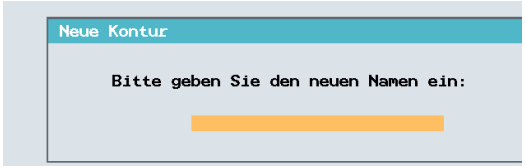




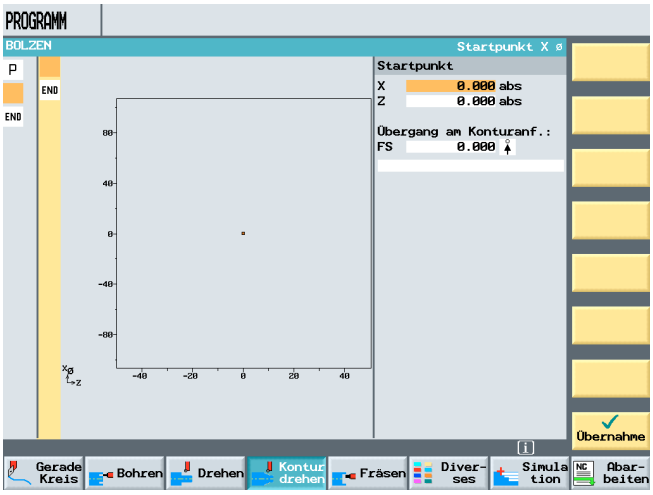
and



an input window is opened where we enter the name of the contour we want to describe.



After entering the name, the contour calculator opens.



First, the starting point of the contour is entered.

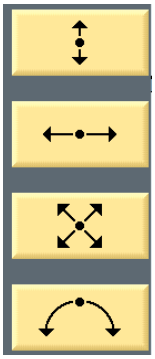
In this case, the direct diameter can be entered, without applying a starting chamfer or radius.

The transition at the contour start is defined directly with the starting point.

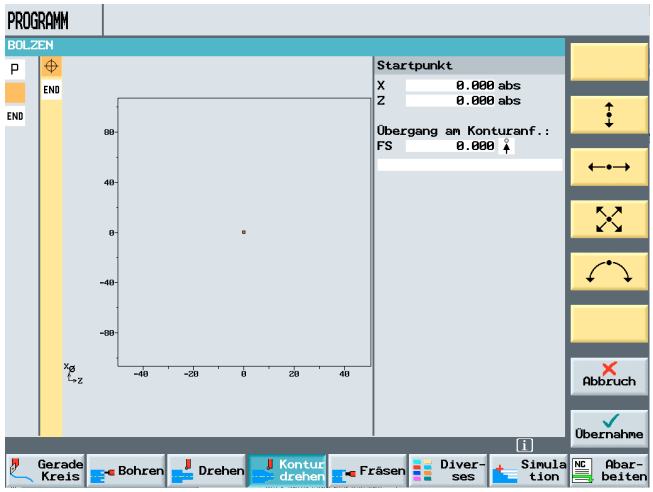
Also, the approach strategy to the contour can be selected here.

After accepting the starting point by pressing the softkey Accept,

the contour is described with the softkeys



.



When the contour is created, it can be generated completely, even if the workpiece is generated later in different machining steps and in different working areas.

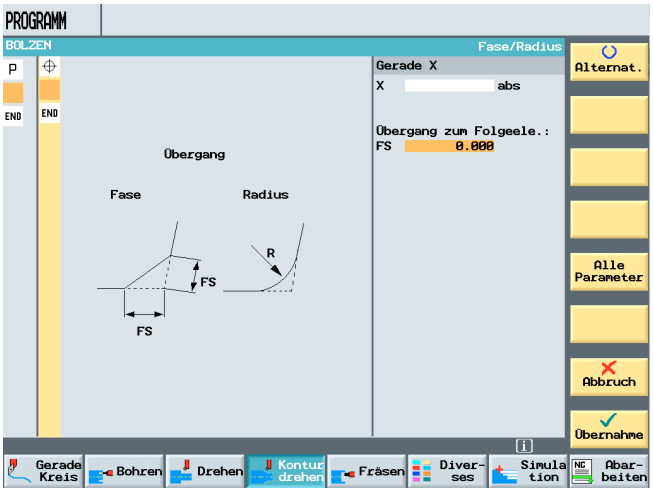
By pressing the softkey

a straight line in X direction can be defined.

In addition to the input field for the length of the contour element,

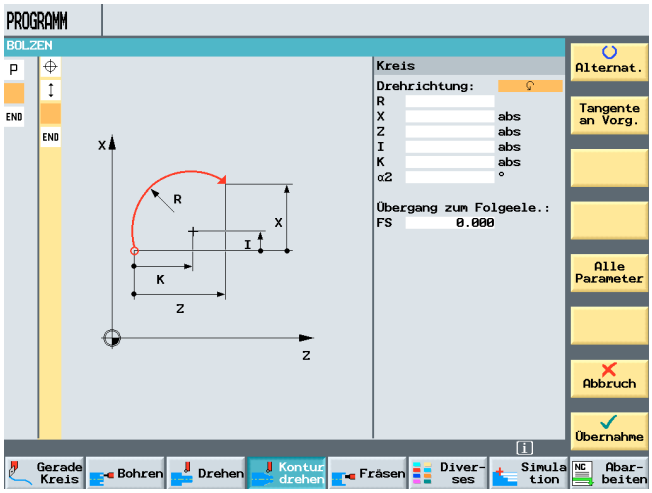
the transition to the subsequent element can also be defined.

The transition can be defined as chamfer or as radius.



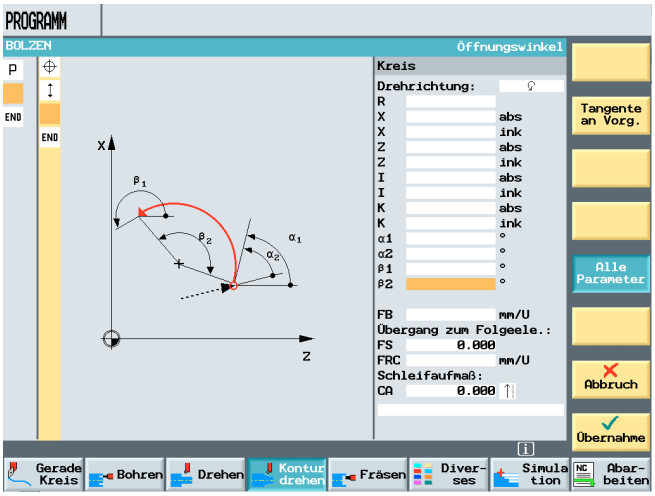
By pressing the softkey

the input options are expanded.



In addition to the expanded input options -for example, the angle input-

there is also the option to enter an element-related feed function for finishing



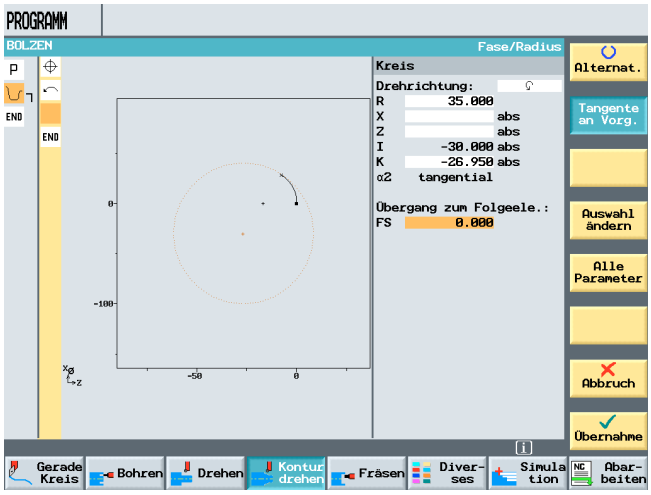
and the input of a feed for the transition elements .

Likewise, a grinding allowance can be defined here.

In a free input field, any instructions in the form of G-code can be called.

As a rule, a contour element is defined by determining the end position.

By pressing the softkey

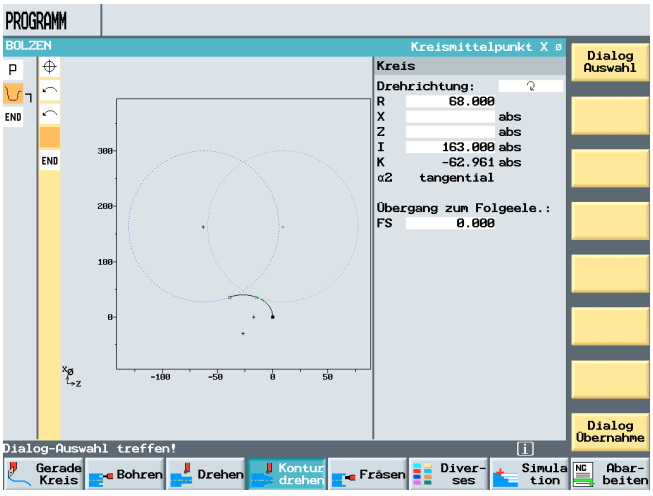


not the endposition of the elements is calculated, but the tangential contact point of two elements. In this way, tangential transitions can be generated.

If the contour calculator calculates several position options, we can select among the different options by using the softkey

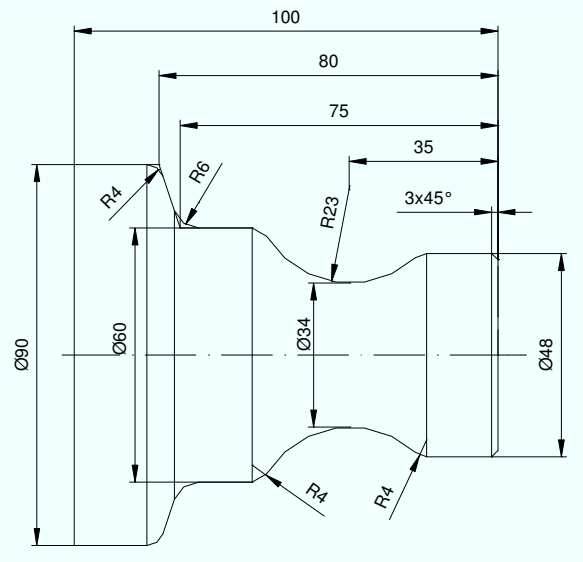
The selected element is accepted with

.

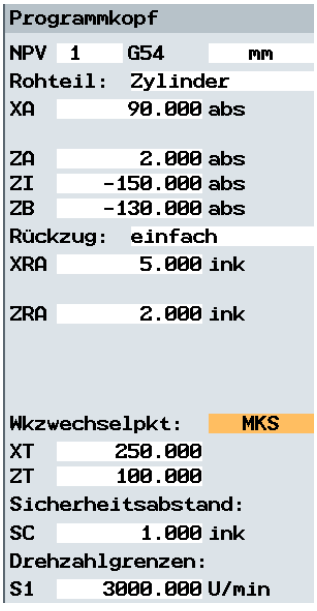


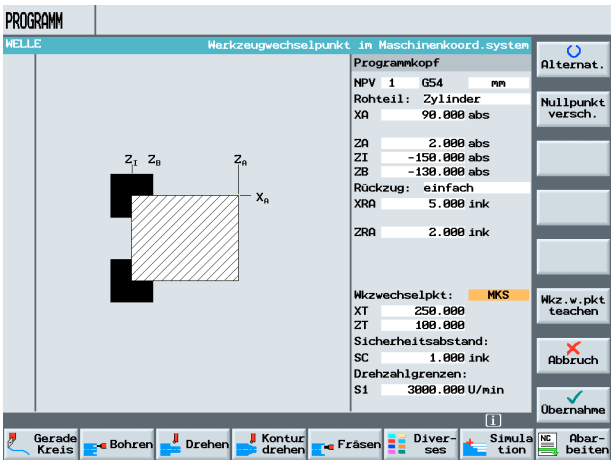
8.2 Example of Contour Programming

The example below provides more detail for working with the contour calculator.

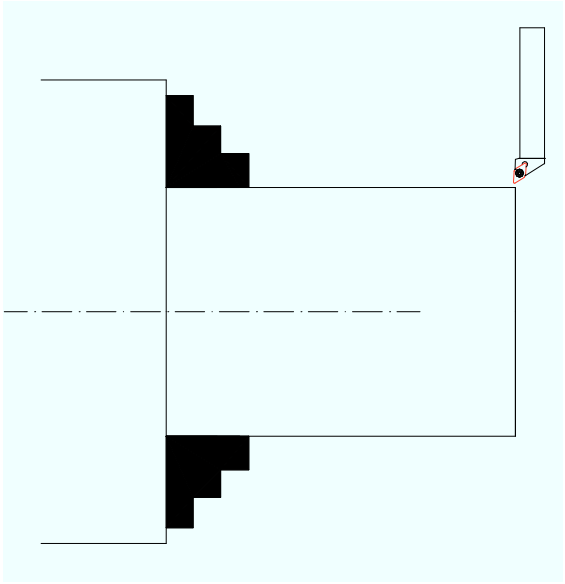


After setting up a program with the name Shaft, the following values are entered in the program header, and accepted.

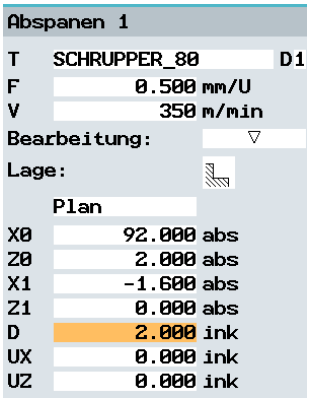




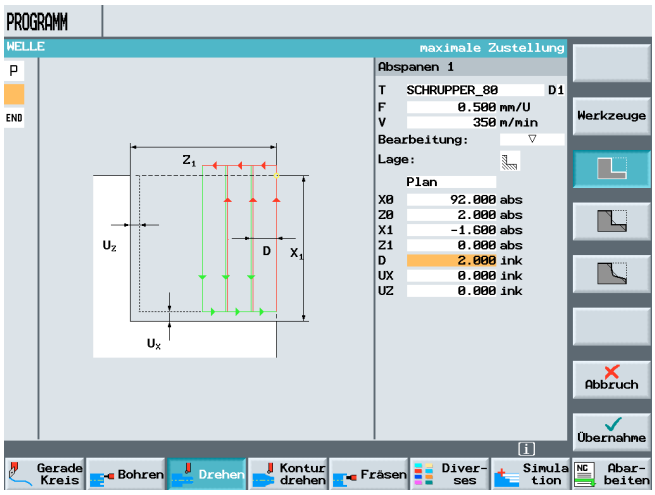
The first machining step is facing the frontal area using the machining cycle.



After entering the values, the machining cycle is accepted in the machining plan.



Processing the frontal area is completed.

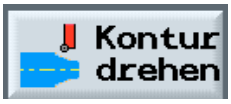


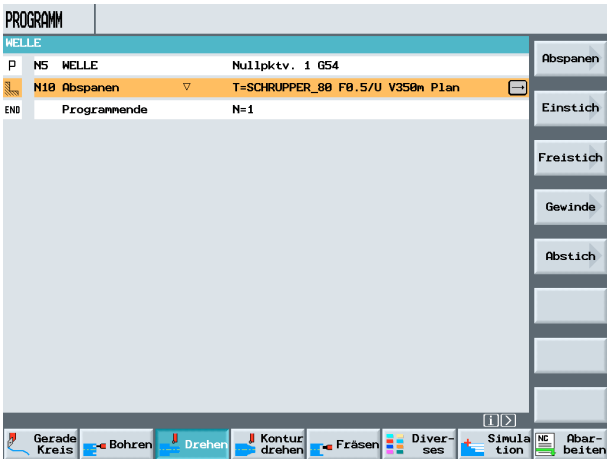
Through facing, the plane surface becomes the reference point for the Z-axis.

Topic: Example of Contour Programming

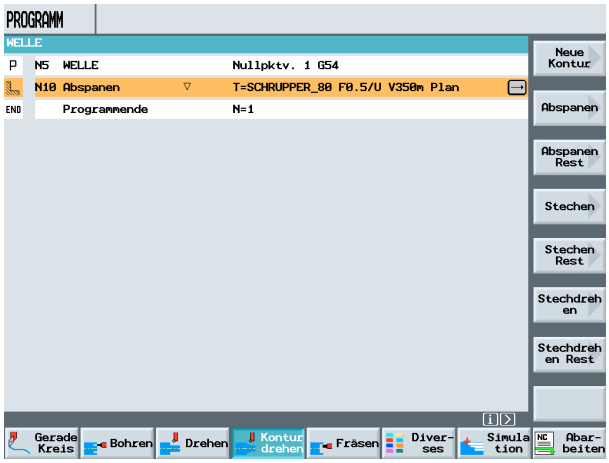
After accepting the machining in the machining plan,

by pressing the softkey

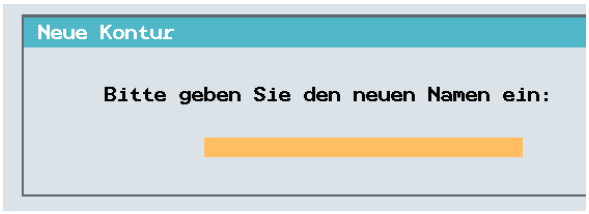




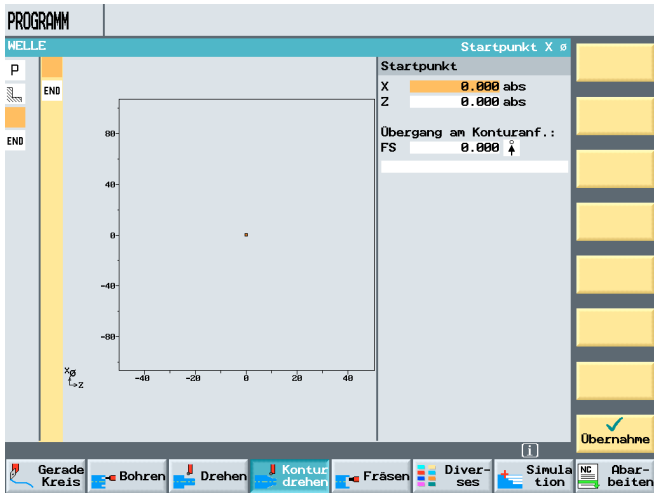
and



an input field opens where the new contour is assigned a name.



After the name is accepted, the contour calculator opens under ShopTurn.



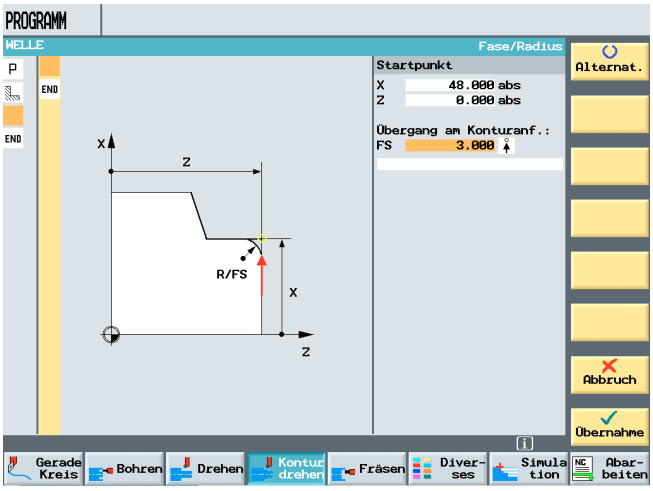
The first input defines the starting point.

Here, the first diameter of the .finished part contour is described.

X	48.000 abs
Z	0.000 abs

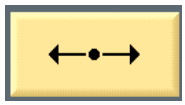
With the contour calculator, a transition element can be defined directly at the contour start.

Übergang am Konturanf.	
FS	3.000

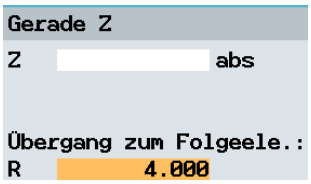


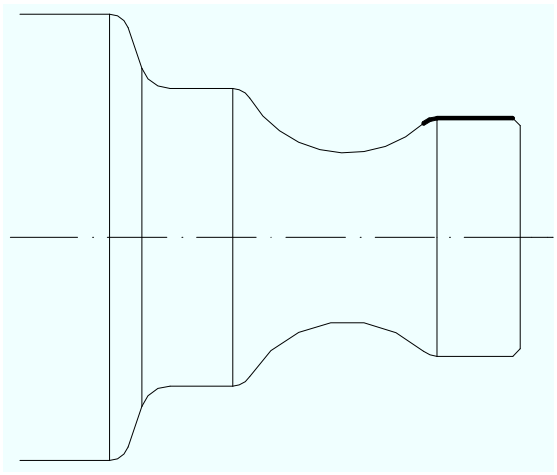
The transition at the contour start, programmed at the starting point, is taken into account automatically when the actual machining starting point is calculated.

Next follows a

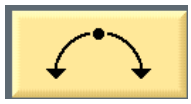


whose end position is unkown. Only the transition to the subsequent element is known.

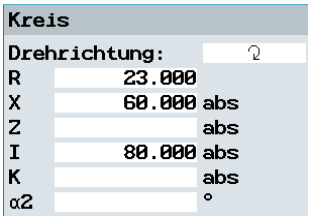


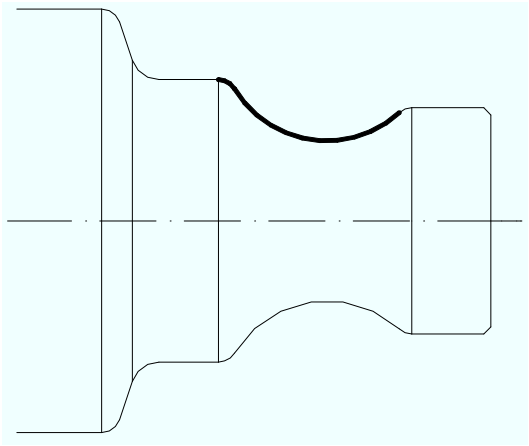


The next element is a

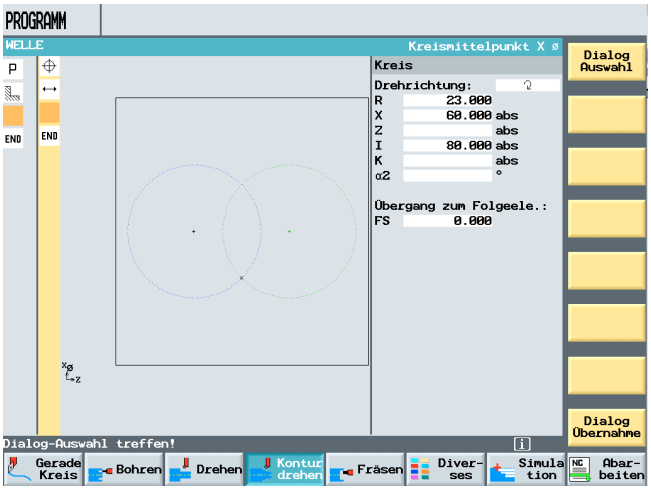


After entering the following values





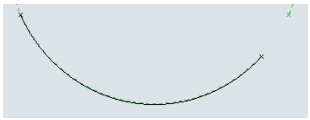
a dialog field opens.
The contour calculator offers a selection of possible contour geometries that can be selected by pressing the softkey

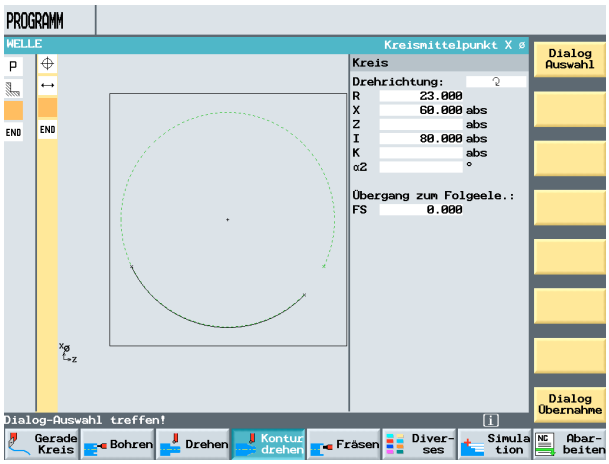


and accepted by pressing

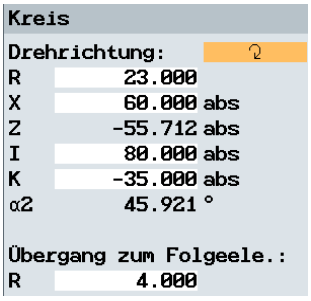
.

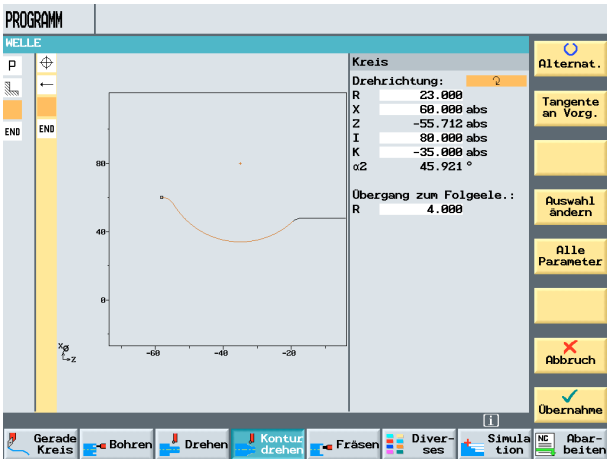
The following geometry is shown in the “Dialog selection“.



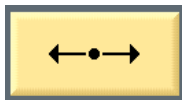


After accepting and entering the remaining values,



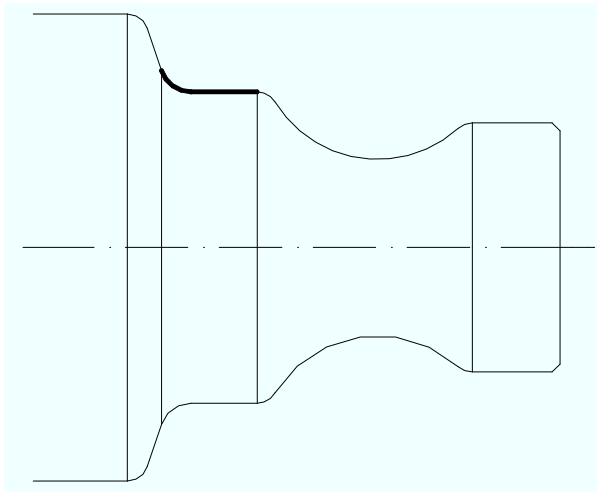


next the following is entered

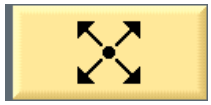


, with the values.

Gerade Z		
Z	-75.000	abs
α2	64.229	°
Übergang zum Folgeele.:		
R	6.000	

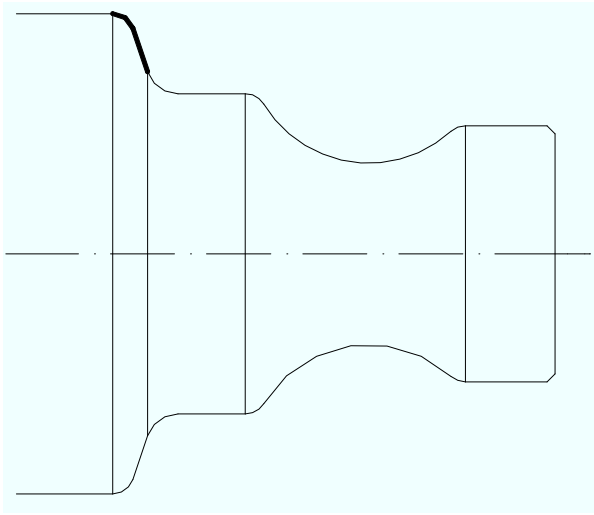


.A

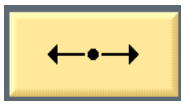


follows with the values:

Gerade		
X	90.000	abs
Z	-80.000	abs
α1	108.435	°
α2	288.435	°
Übergang zum Folgeele.:		
R	4.000	

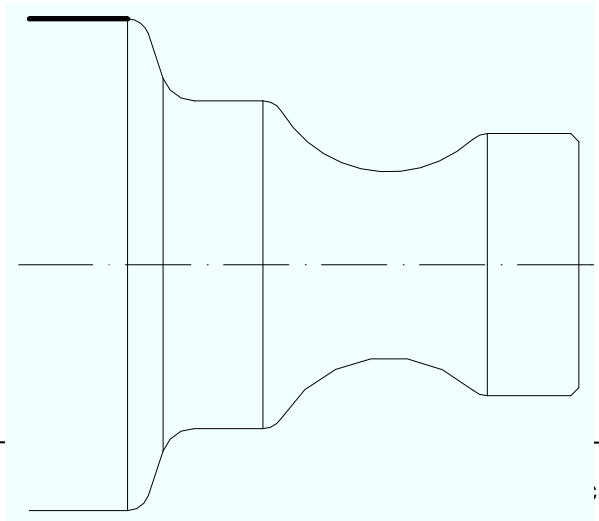


The last contour element is a

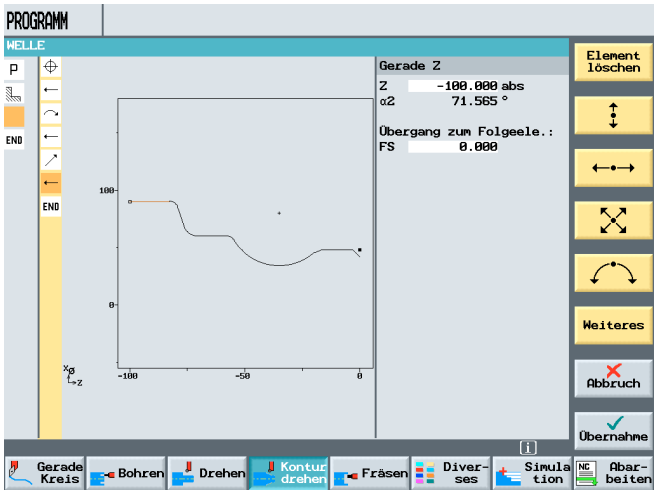


with the value

Gerade Z		
Z	-100.000	abs
α2	71.565	°
Übergang zum Folgeele.:		
FS	0.000	



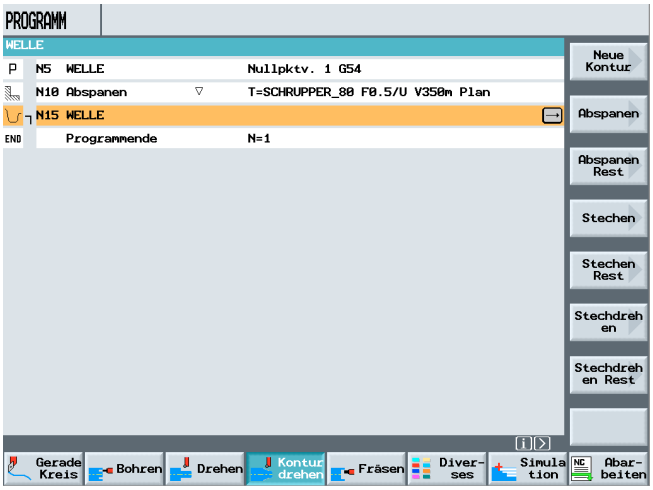
The finished part contour has now been described, and is included in the machining plan by pressing the softkey



.

An open bracket is shown at the contour in the machining plan.

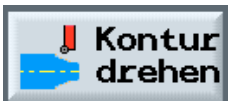
The contour has been described. In the next machining steps, the machining for the contour is defined.



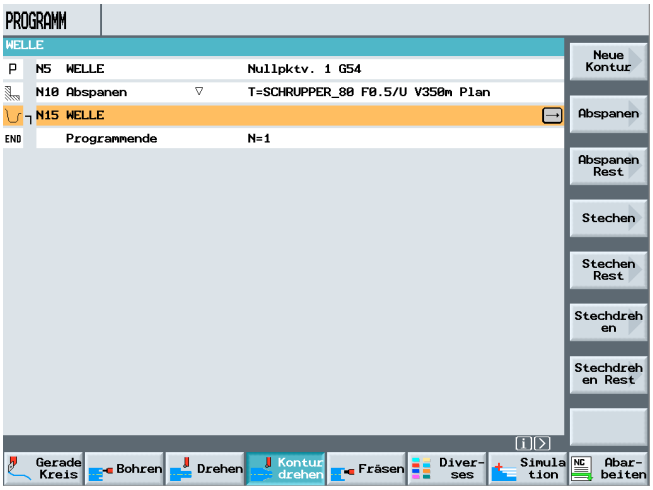
The programmed contours under ShopTurn can be changed or supplemented any time.

8.3 Contour Machining

A contour of rotation is machined in the area Contouring

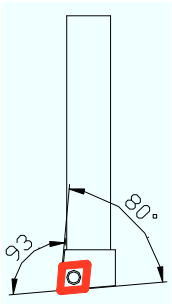


By pressing the softkey

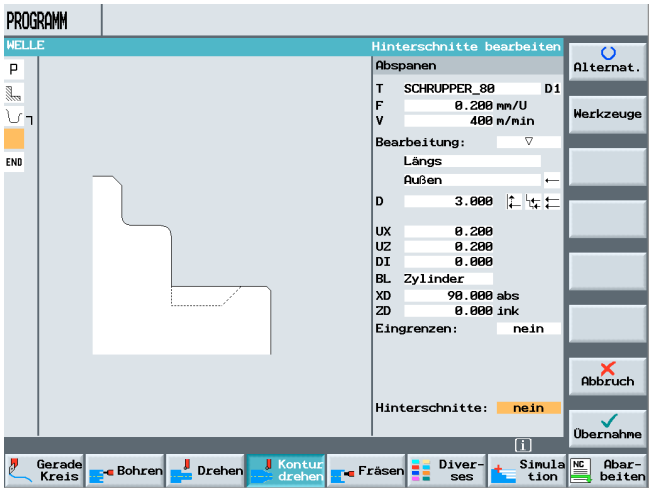


Under ShopTurn, the softkey “Machining“ exists twice. On the one hand, there is the autonomous cycle “Machining“ for facing and longitudinal turning (bracket in the machining plan is not cosed). Then there is the softkey “Machining“ in the area “Contouring“ (bracket in the machining plan is closed).

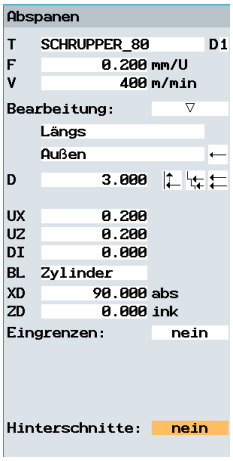
The contour is to be machined wit a roughing tool 80° degrees



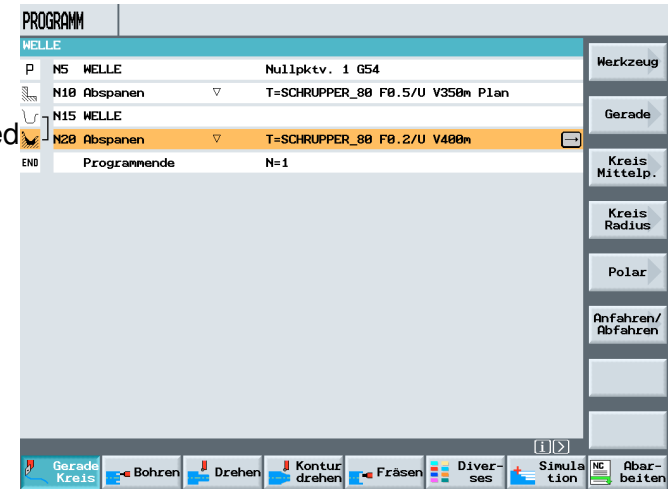
without relief cuts

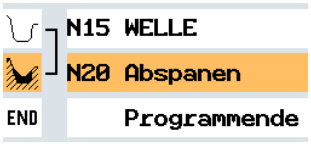


After entering the corresponding values

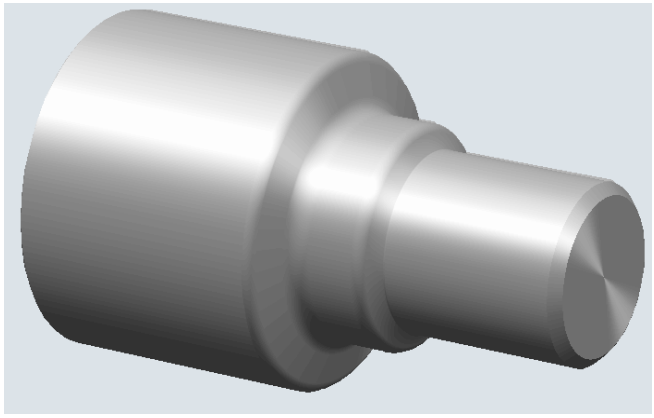


and acceptance into the machining plan, machining was appended to the contour. The bracket is closed.





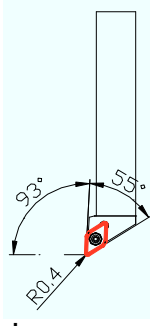
The workpiece is machined as follows.

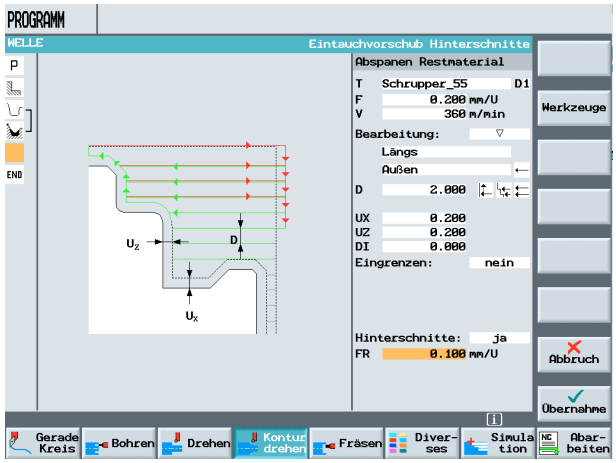


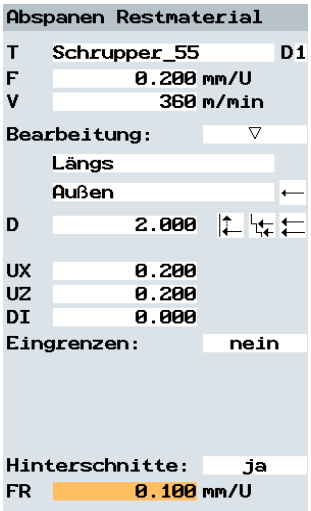
8.4 Contour Machining Residual Material

To machine the remaining material from the circular arc to the finishing allowance, the function

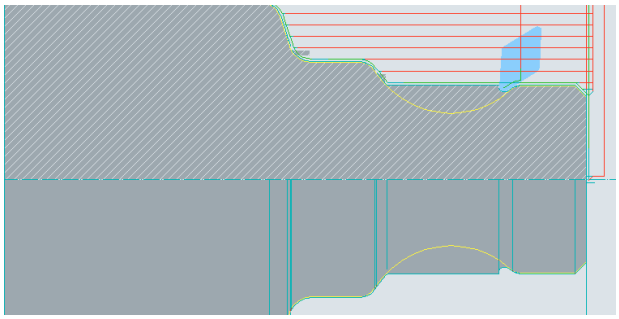
is provided.
The residual material is machined with a tool having a different edge geometry; for example, roughing tool 55°





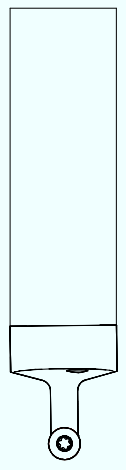


With the cycle “Residual material“, only the material is machined that is present beyond the defined finishing allowance.



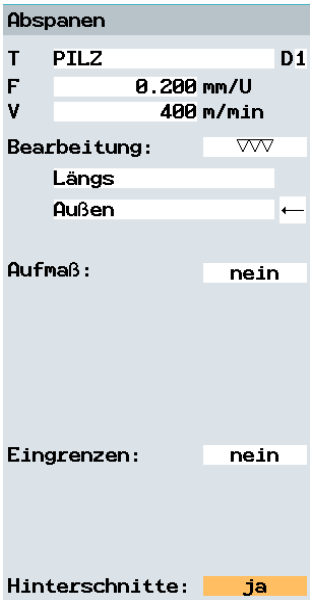
If necessary, this cycle can be performed several times consecutively with different tools.

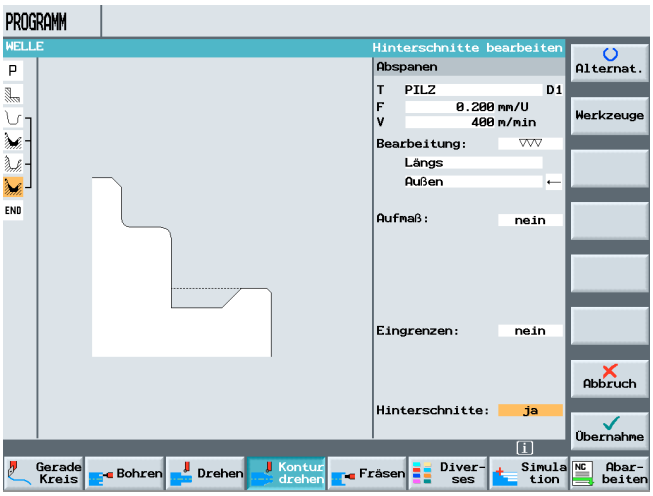
The last step consists of machining the finish. This is done with the mushroom tool.



By pressing the softkey

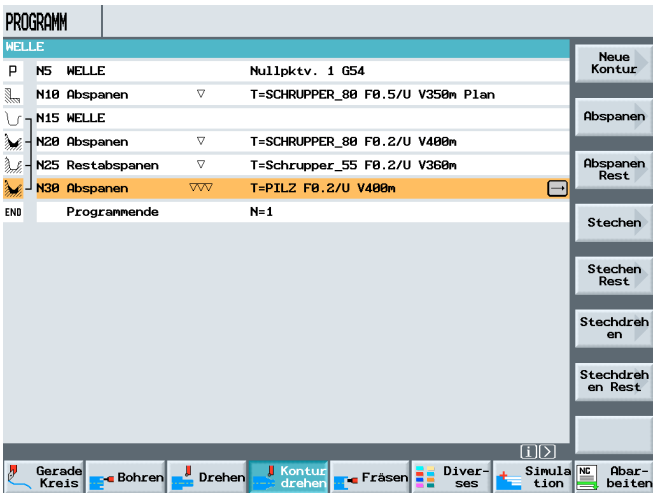
an input is made for machining the finish

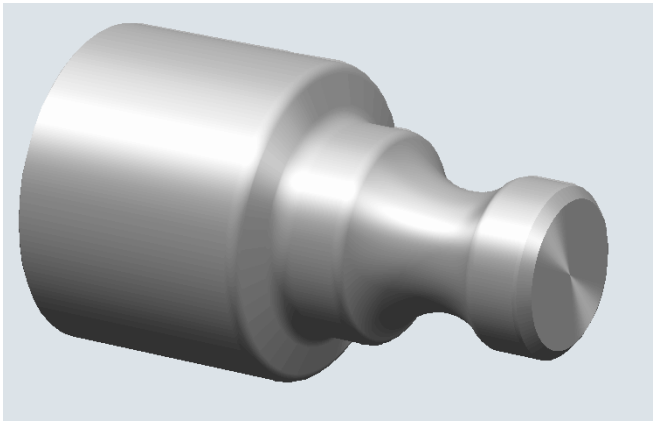




In the finishing cycle for finishing the contour, an allowance can be defined directly; for example for fit manufacturing.
Advantage: The value for tool wear remains untouched when a fit is manufactured, for example, and does not have to be reset to the old actual value for subsequent machining.

After accepting the input into the machining plan, the program is complete. It can be simulated and executed.





9 Working with a Raw Part Contour

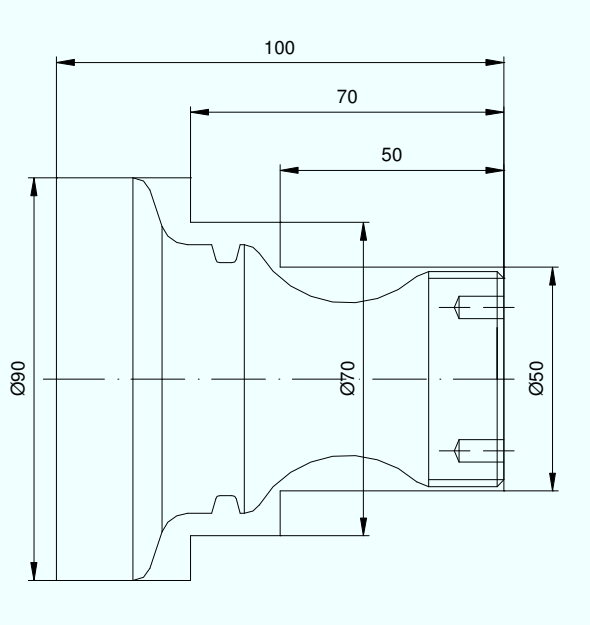
In this module, working with a raw part contour under ShopTurn is explained, based on an example.

In practice, the shape of a raw part is often not a cylinder.

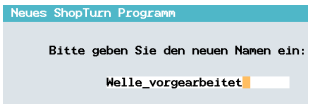
If this is the case, the raw part contour can be described under ShopTurn.

By defining a raw and a finished part contour, machining can take place contour-parallel; that is, without idle passes.

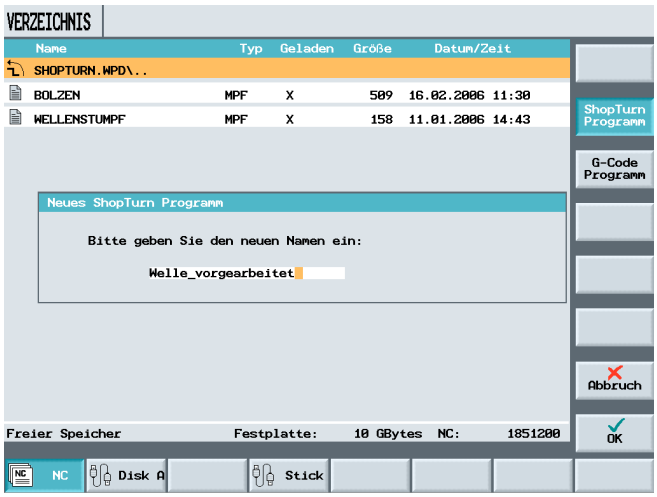
The following raw part contour is to be programmed under ShopTurn.



A program with the name
Shaft Premachined



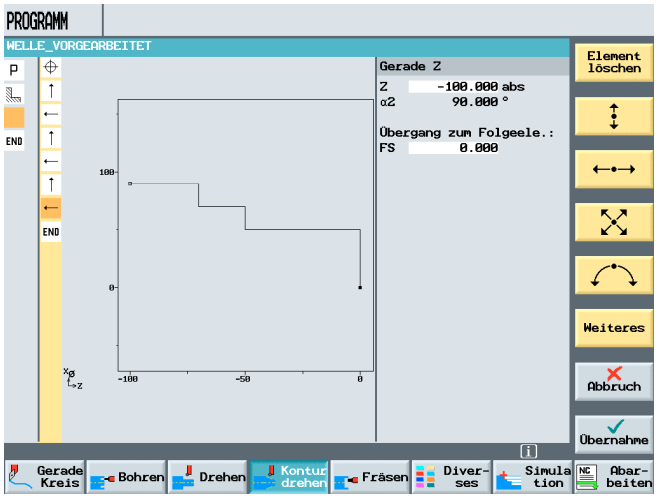
is created.

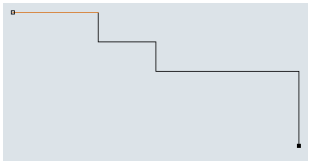


After face turning, the raw part contour is programmed.

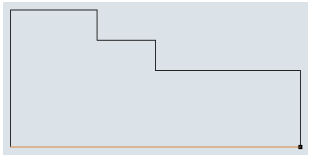
In this case, there is a special consideraton.

This contour description would be correct for a finished part contour.

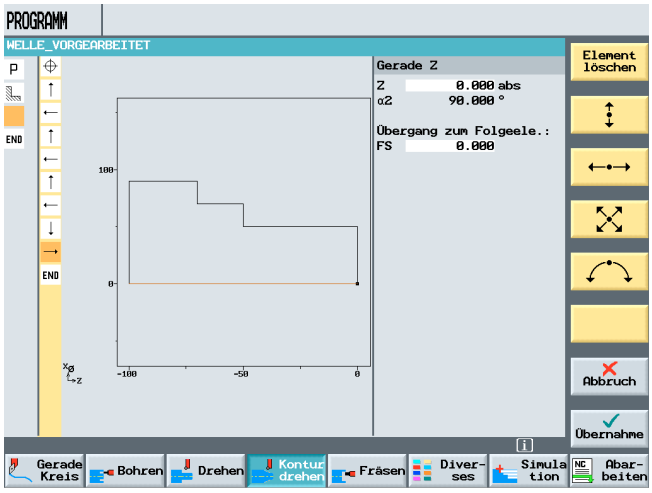




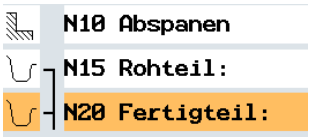
The raw part contour has to be closed, and surround the finished part contour.

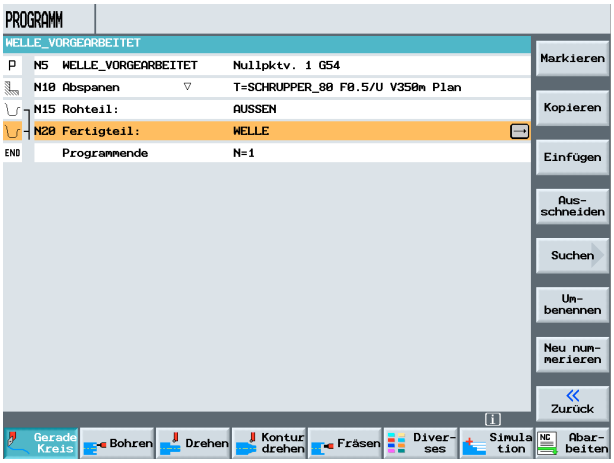


After accepting the contour in the machining plan, the finished part contour is programmed.

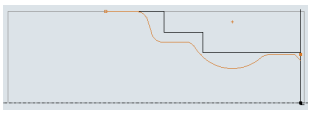


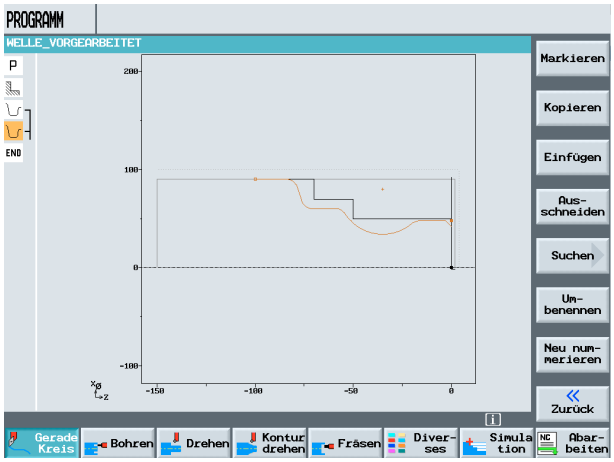
By accepting the finished part contour in the machining plan, ShopTurn automatically defines the first contour as the raw part contour and the second as the finished part contour.





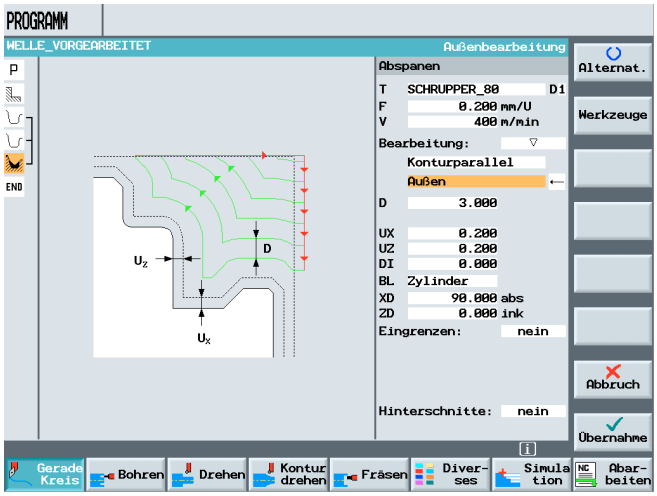
We have to check that the finished part contour is located within the raw part contour





During the subsequent machining cycle for roughing, machining takes place

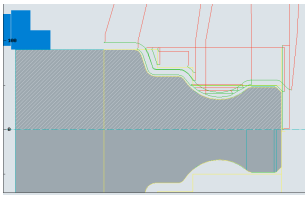
Konturparallel
Außen

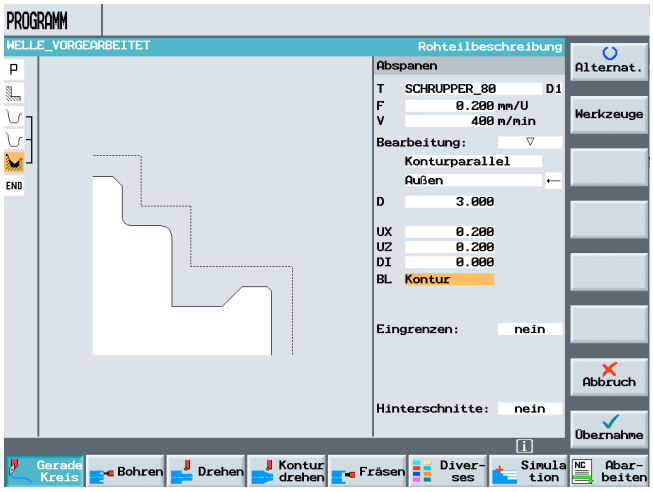


In addition, a

BL	Kontur

has to be defined as raw part.
Then, machining takes place contour-parallel.



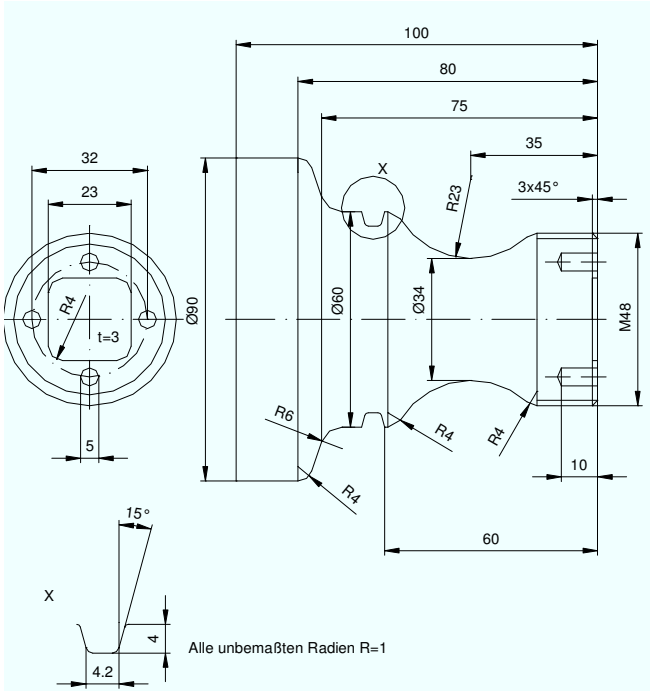


When working with raw part and finished part contours and subsequent longitudinal machining, relief cuts of contours where the tool is not operating move at rapid feed.

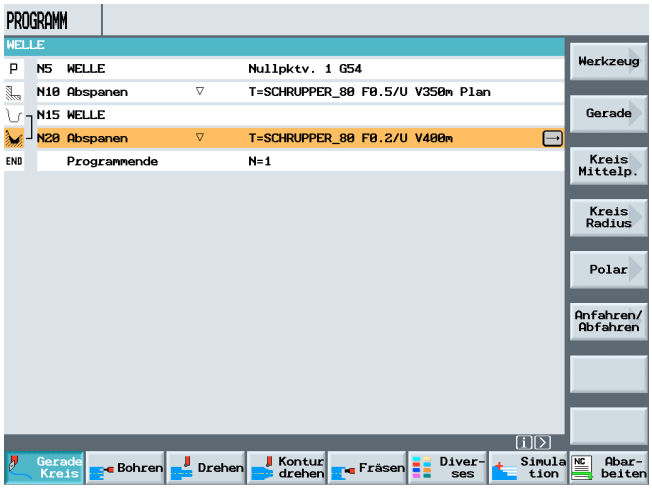
10 Drilling and Milling Positions

Drilling and milling positions under ShopTurn are explained, based on an example.

Holes are added to the front face of the programmed workpiece.

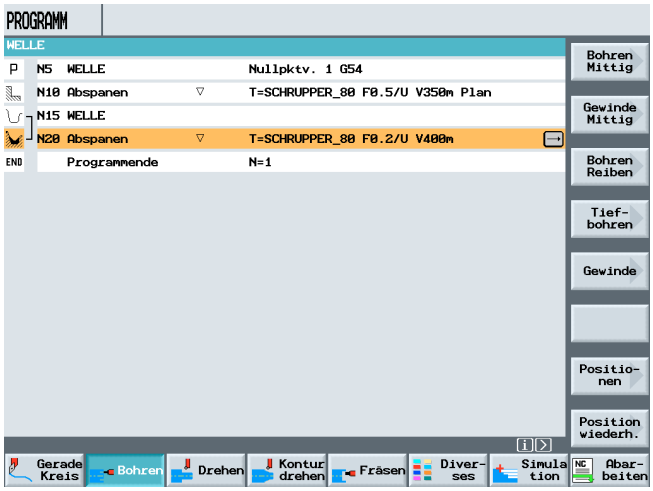


By pressing the softkey Drill



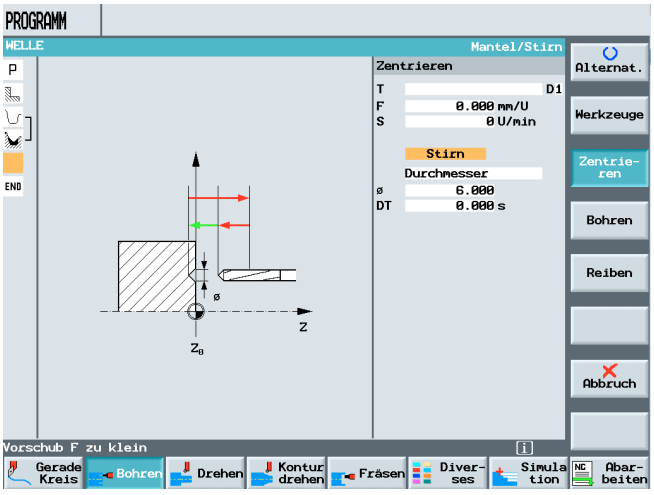
and

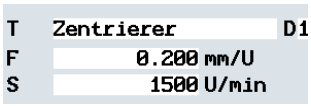
for off center machining,



the input screen form is displayed for off center hole machining.

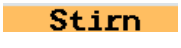
After entering the tool and the technology,





the machining plane

Front

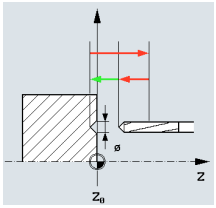


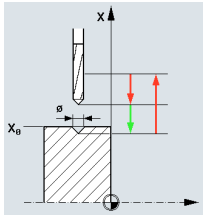
or

Shell

is selected.

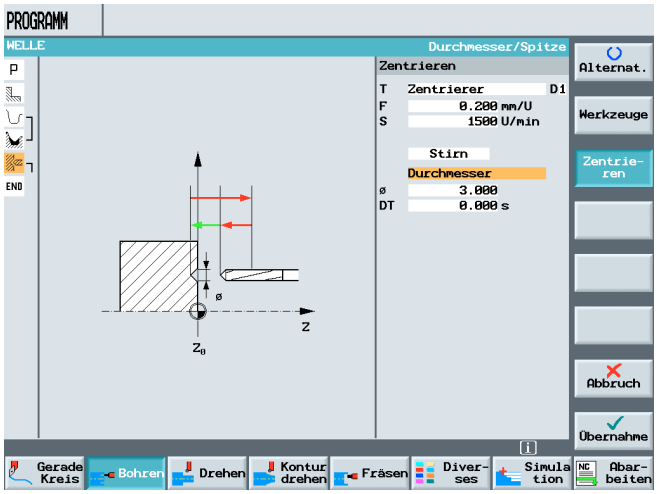
In this example, machining on the frontal face is selected.





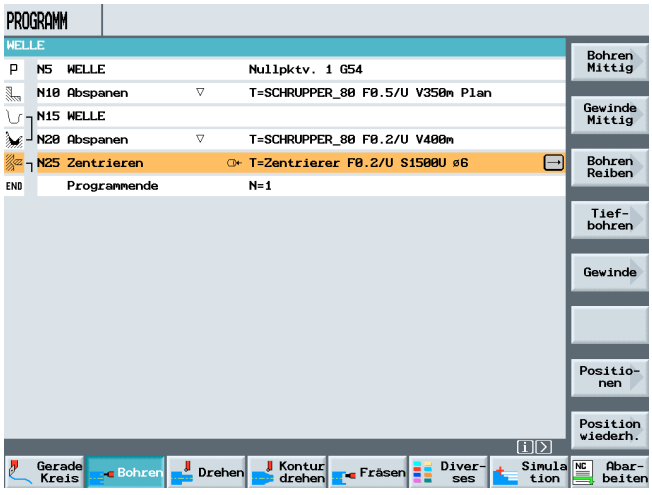
Since this is a centering task, it can be entered by taking the diameter

into account.



After pressing the softkey

to save the values entered,

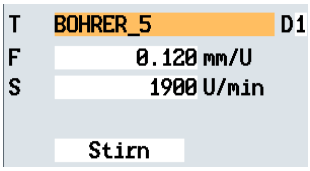


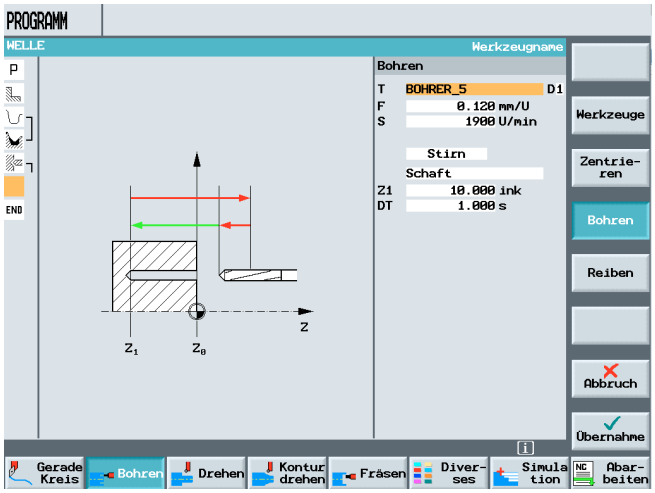
an open bracket is shown in the machining plan, as in the case of contour programming.

By again pressing the softkey

and

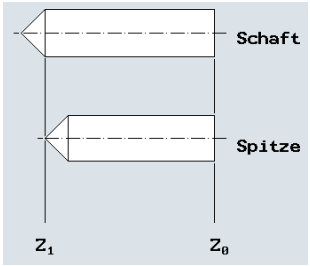
, the input screen form for drilling is displayed.





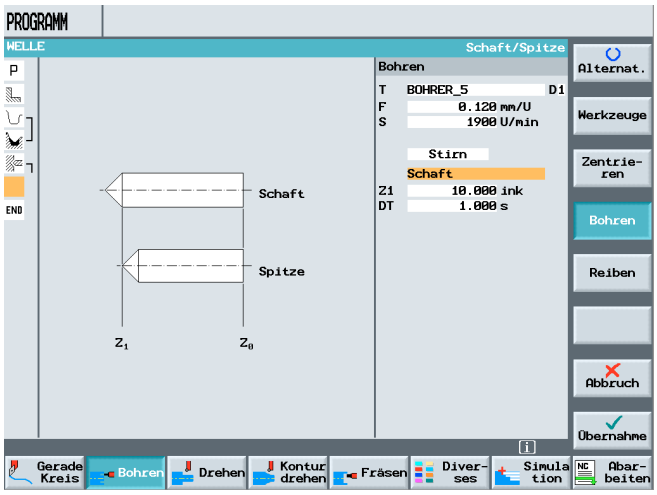
After selecting the tool, the technology and the machining plane, we can select between

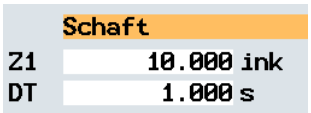
shaft and tip



for drilling.

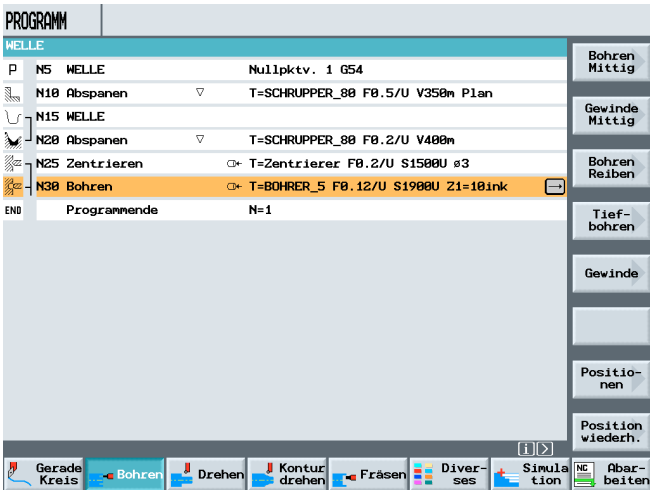
After selecting the reference point



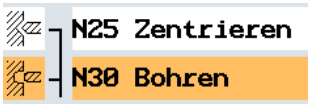


and entering the depth and dwell time - basically 1 second- the input values are accepted into the

machining plan by pressing the softkey



The bracket was expanded by one position

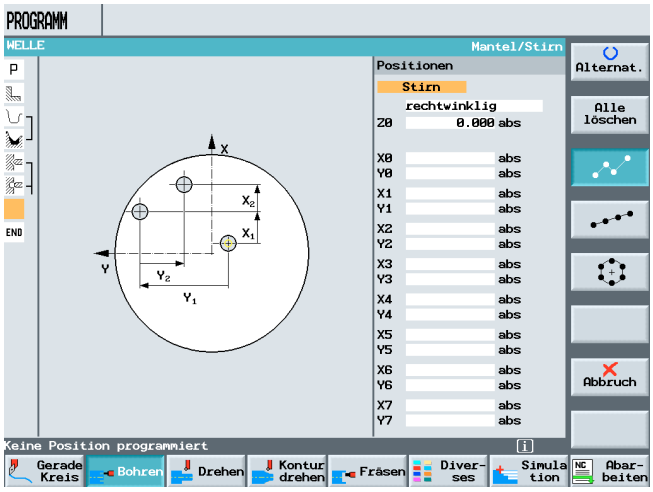


but not yet closed.

Next, the position for machining is defined.

By pressing the softkey

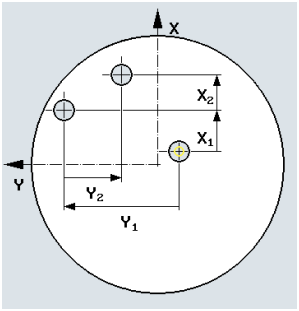
, the position patterns are opened.



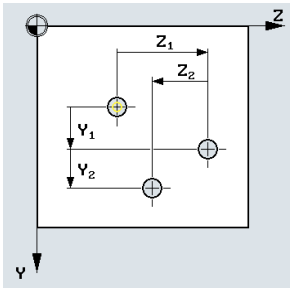
The following position patterns are provided in ShopTurn:

Position pattern

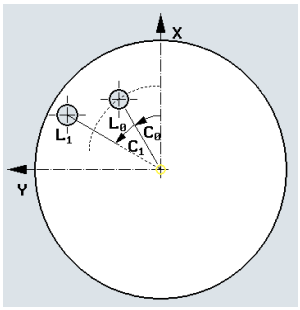
Stirn
rechtwinklig



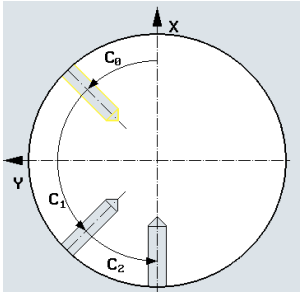
Mantel
rechtwinklig



Stirn
polar



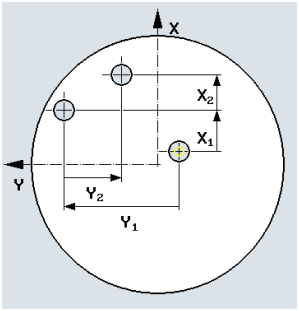
Mantel
zylindrisch



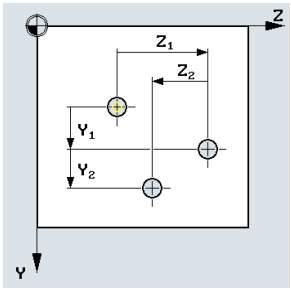
The following position patterns are provided in ShopTurn

Position pattern

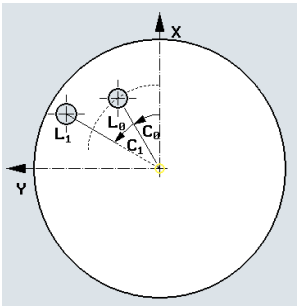
Stirn
rechtwinklig



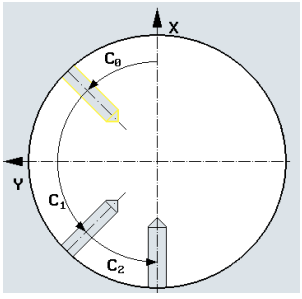
Mantel
rechtwinklig



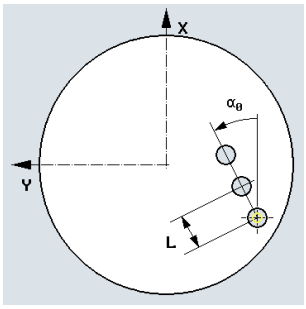
Stirn
polar



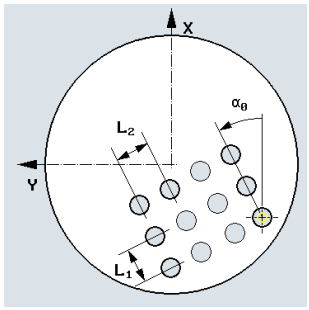
Mantel
zylindrisch



Linie

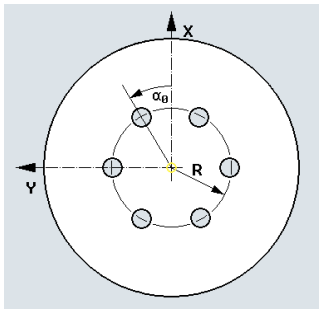


Gitter



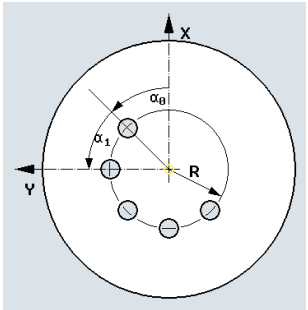
Stirn
mittig
Vollkreis

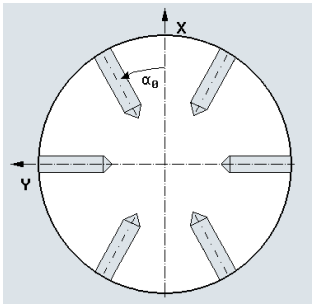
Stirn
außermittig
Vollkreis



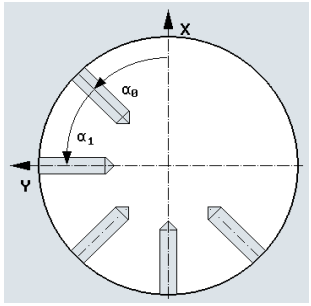
Stirn
mittig
Teilkreis

Stirn
außermittig
Teilkreis

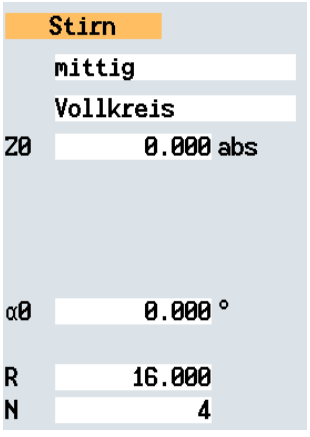


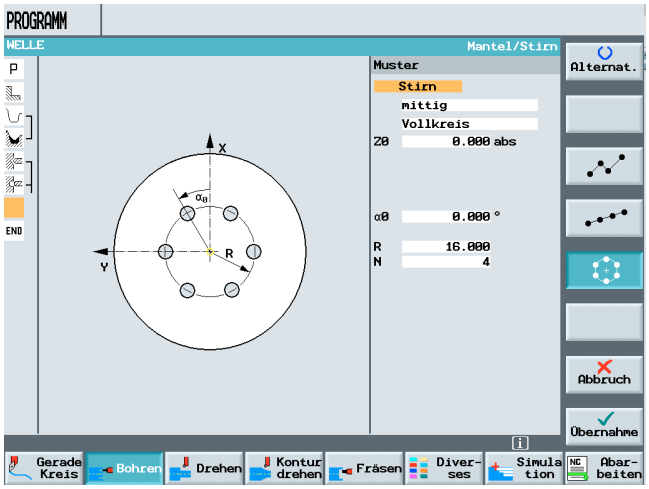


and

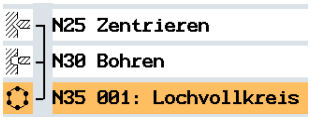


After entering the values for the holes in the program example



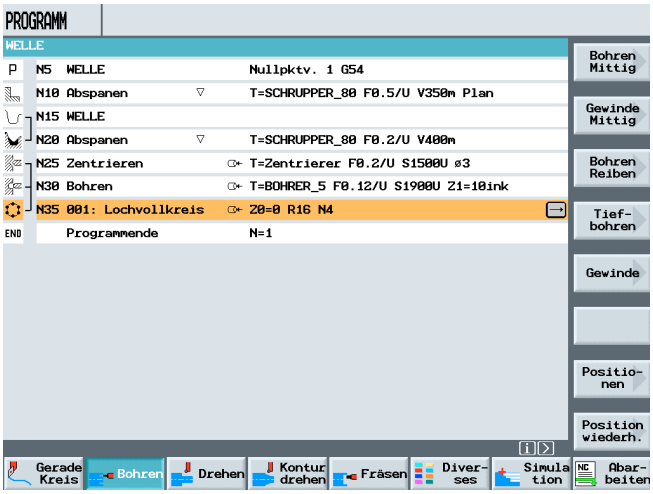


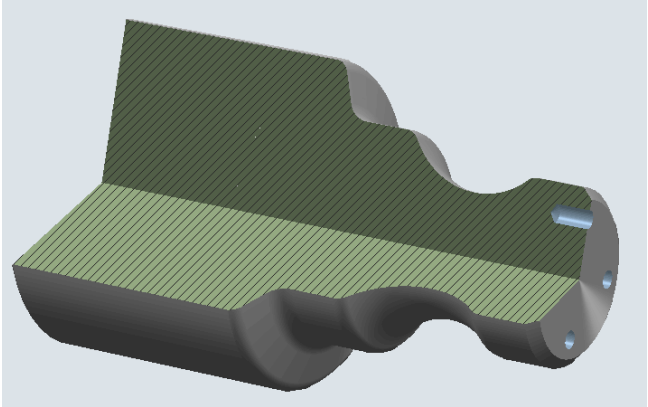
and accepting them in the machining plan,



the bracket is closed through the indication of the machining position.

The programmed machining step can be simulated and executed.



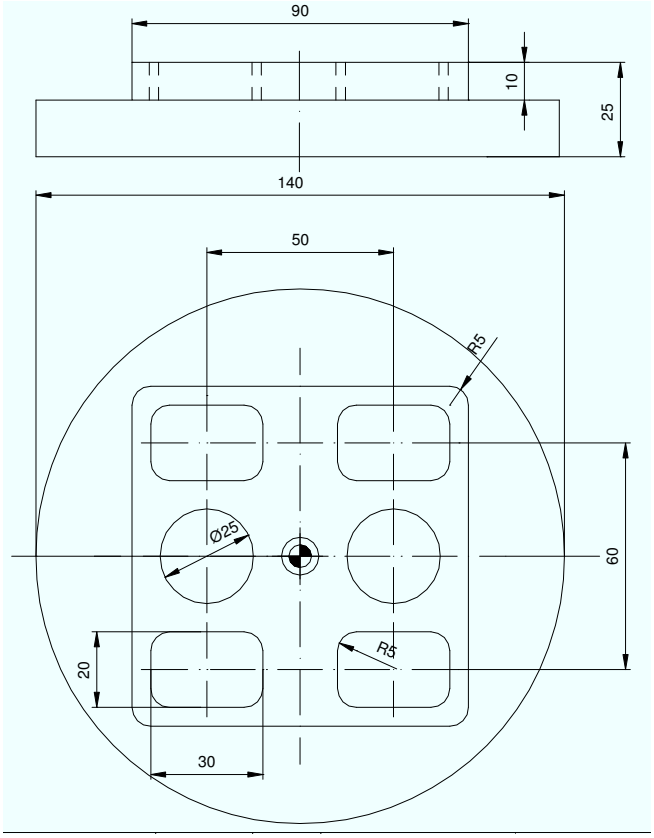


The positions described here are used for hole machining as well as milling with standard cycles.

11 Milling with Standard Milling Cycles

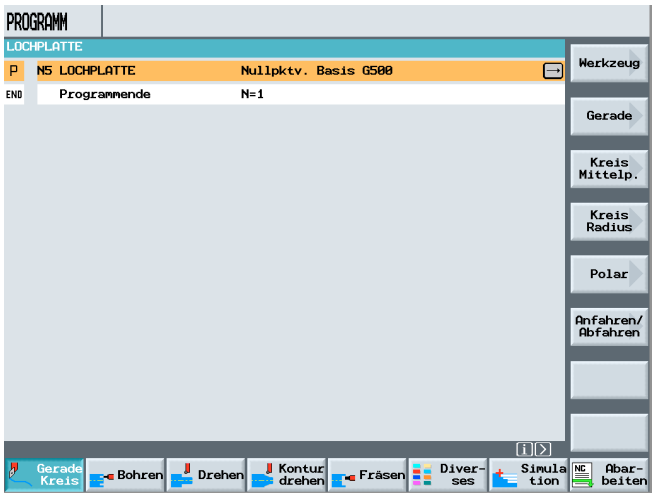
Standard milling cycles of ShopTurn are explained based on an example.

Using the example “hole insert“, the standard milling cycles of ShopTurn are explained.



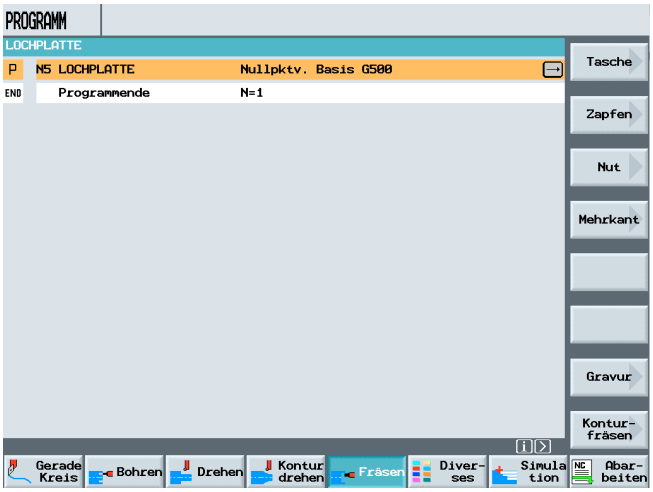
First, we make a rectangular 90mm spigot.

After setting up a program

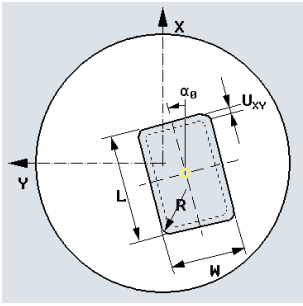


and pressing the softkey

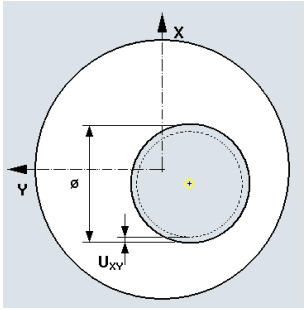
the following standard milling cycles are available.



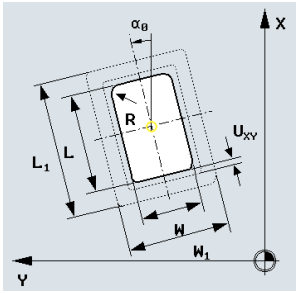
Rechteck-tasche

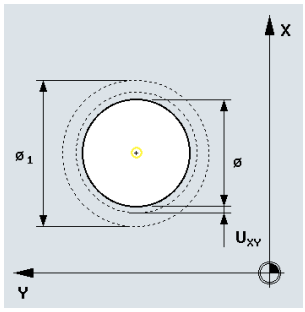


Kreis-tasche

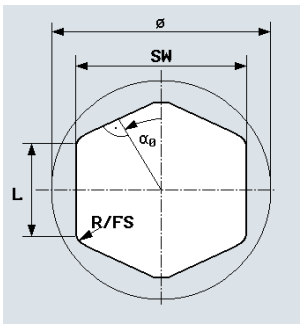


Rechteck-zapfen

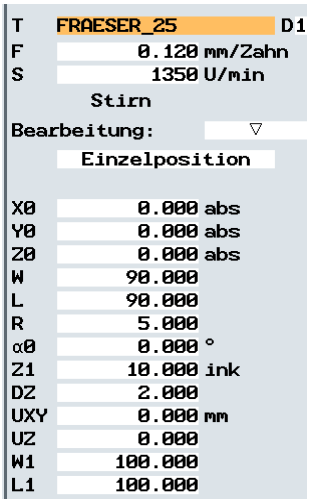


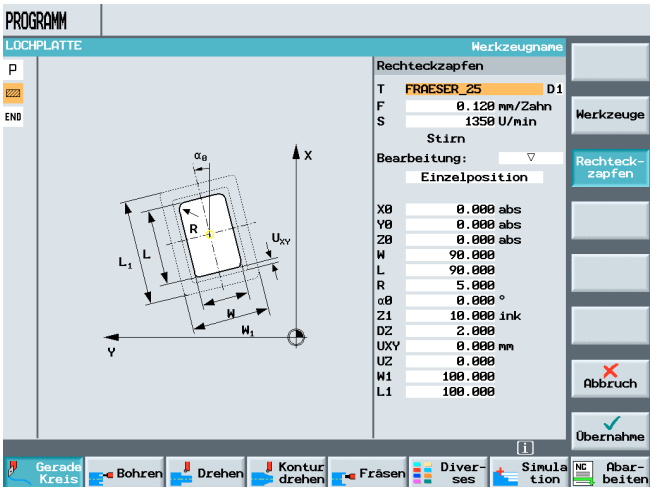


and



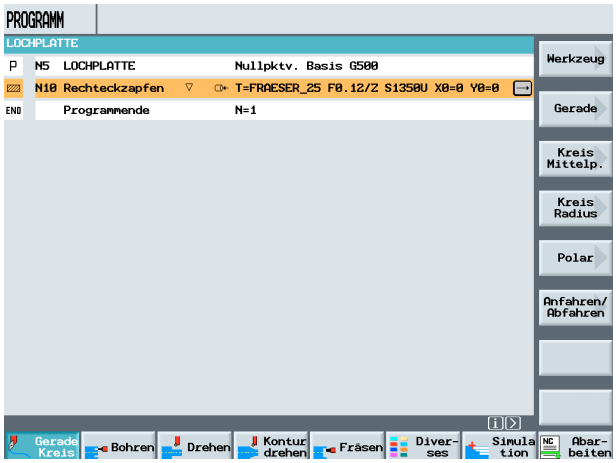
After selecting the cycle “Rectangular spigot“ and entering the values,



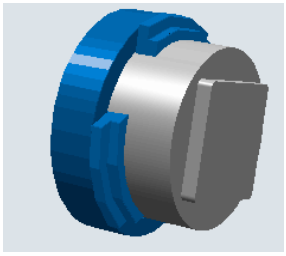


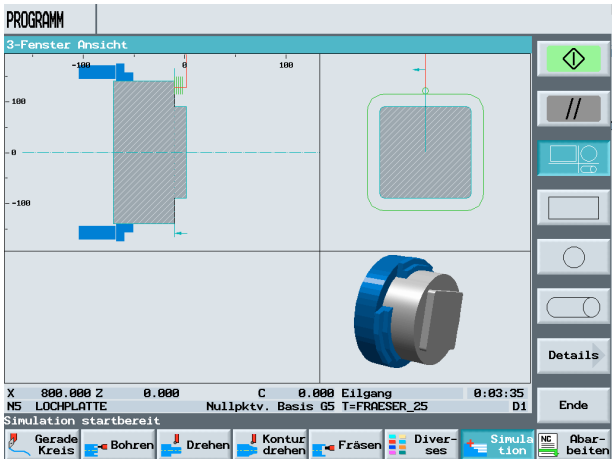
they are included in the machining plan by pressing the softkey

.



The rectangle is created and can be simulated.

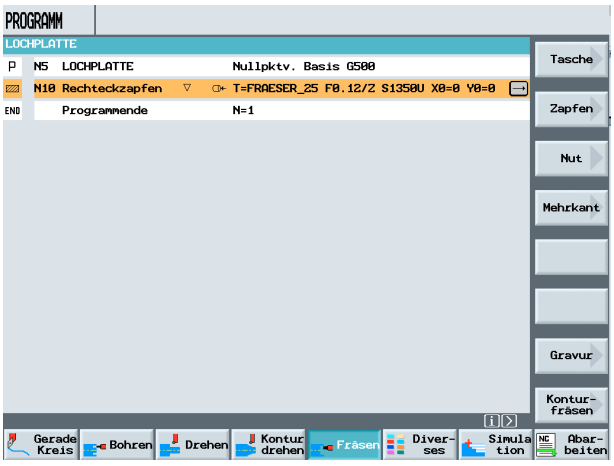




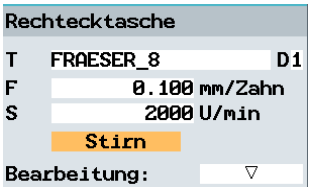
Next, the four pockets on the frontal face are made.

By pressing the softkey Pocket

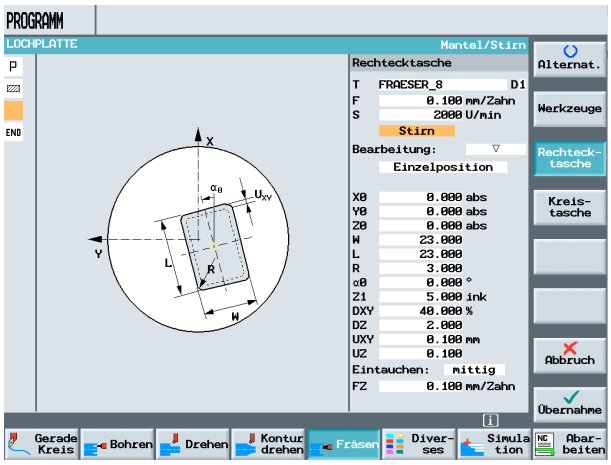
in the area Milling,

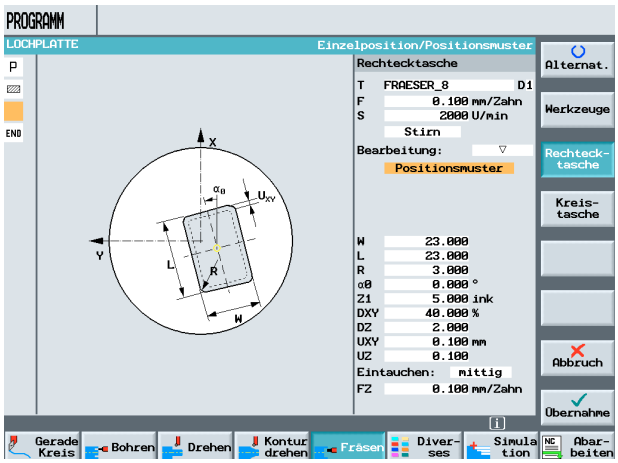


the input screen form for pocket milling is displayed. After selecting the cycle for making a rectangular pocket, select the tool and enter the technology data.

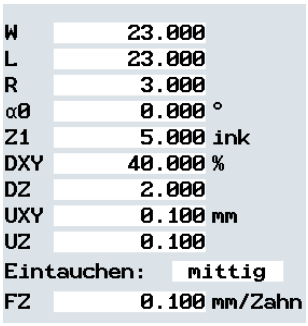


Since it is a question of several pockets, the following selection is made





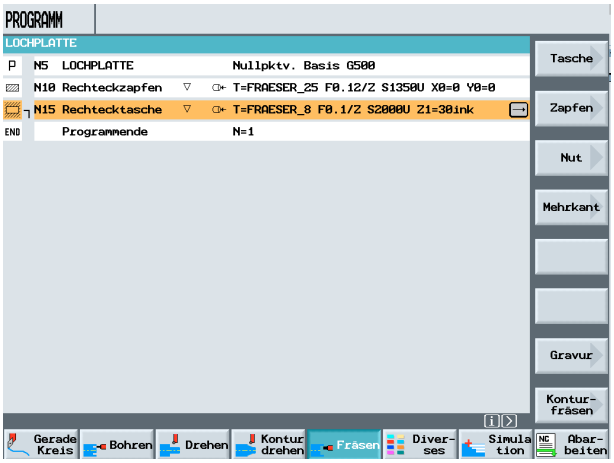
After entering the pocket geometry with the selected insertion strategy,



a bracket opens after the values have been accepted into the machining plan.

Then we define the machining position.

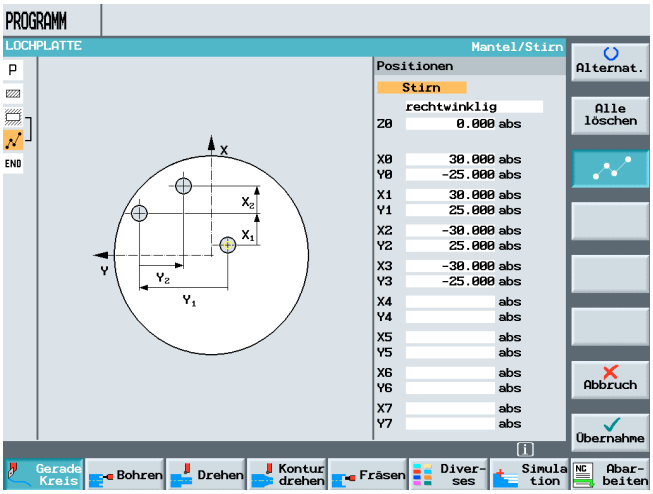
By pressing the softkey



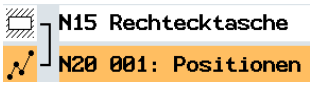
The positons described here are used for hole machining as well as milling with standard cycles.

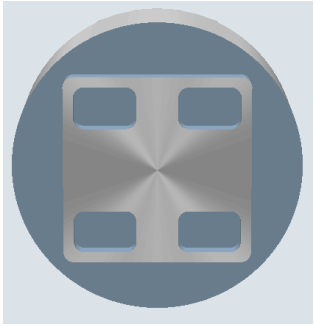
, selecting the corresponding position pattern and entering the values, the position pattern is accepted in the machining plan by

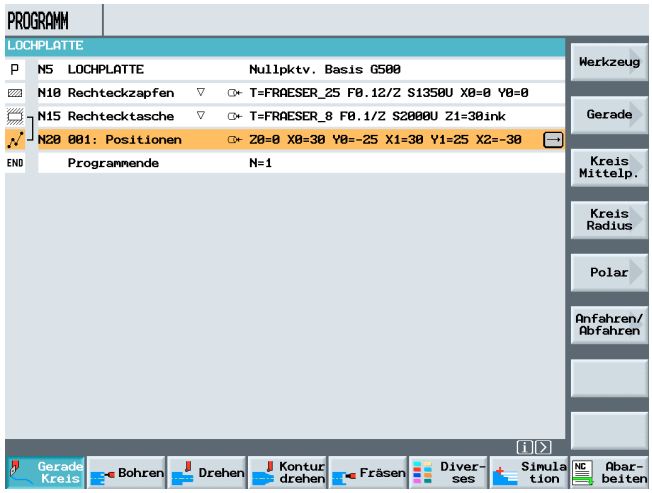
pressing Accept.



The bracket is closed.



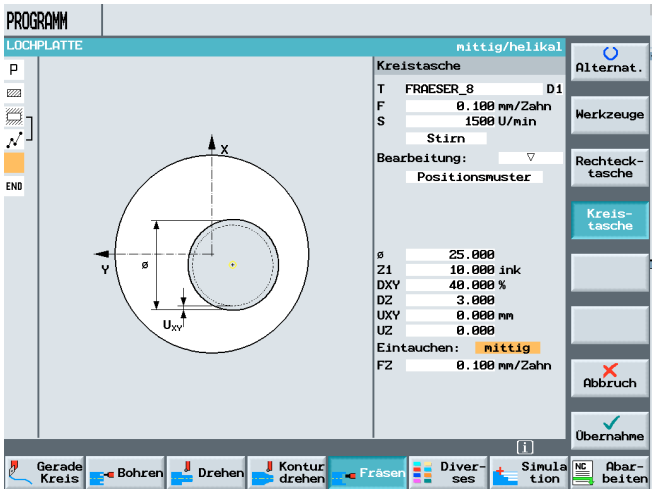




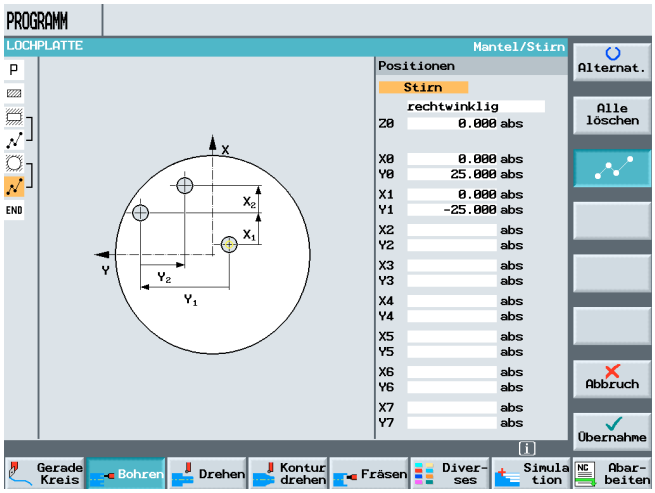
As the last machining step, the two circular pockets are programmed.

After calling the corresponding cycle, here also machining takes place with the “Position pattern“.

T	FRAESER_8	D1
F	0.100	mm/Zahn
S	1500	U/min
	Stirn	
Bearbeitung:		
	Positionsmuster	
ø	25.000	
Z1	10.000	ink
DXY	40.000	%
DZ	3.000	
UXY	0.000	mm
UZ	0.000	
Eintauchen:	mittig	
FZ	0.100	mm/Zahn

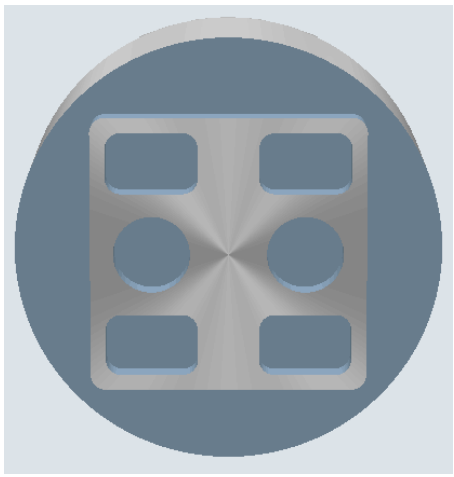


After entering the values, the cycle is accepted into the machining plan.



After entering the corresponding position, the program done.

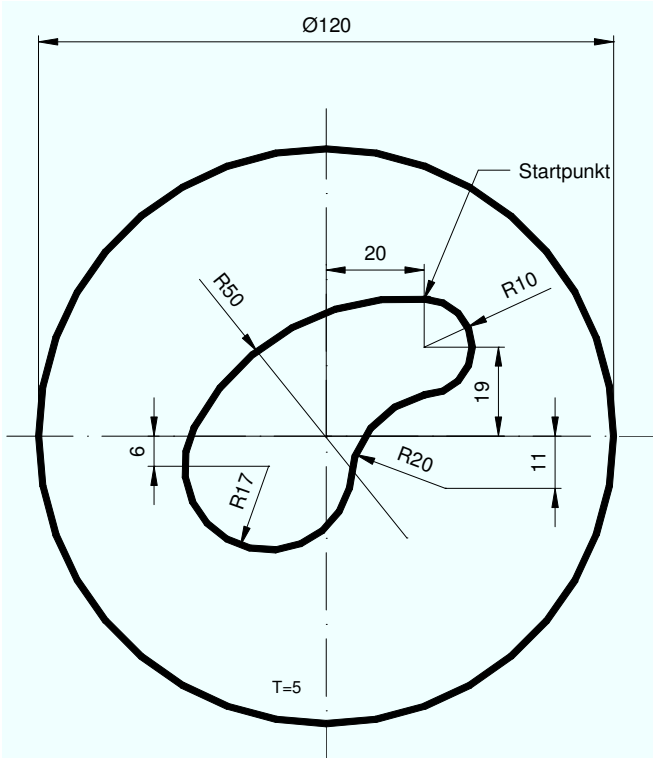
Stirn		
	rechtwinklig	
Z0	0.000	abs
X0	0.000	abs
Y0	25.000	abs
X1	0.000	abs
Y1	-25.000	abs



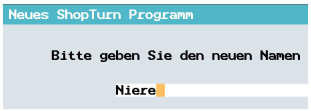
12 Face Milling with Free Contour Input

Face milling with free contour input under ShopTurn is explained, based on an example.

Using the “kidney“ as an example, free contour input under ShopTurn is explained.

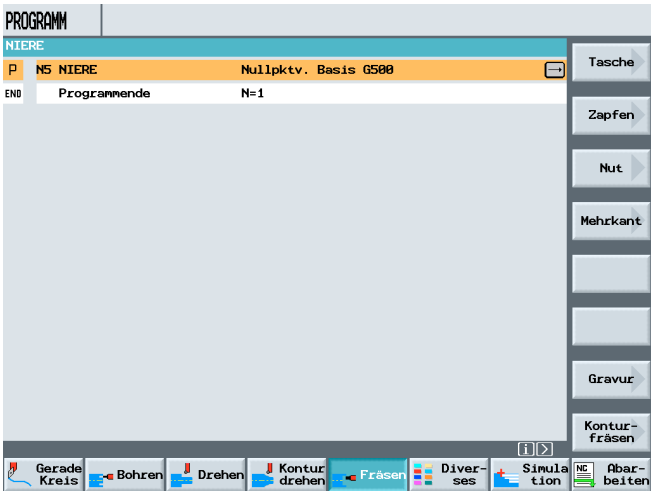


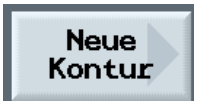
After setting up a program with the name
Kidney



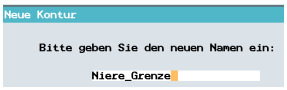
the contour calculator is opened

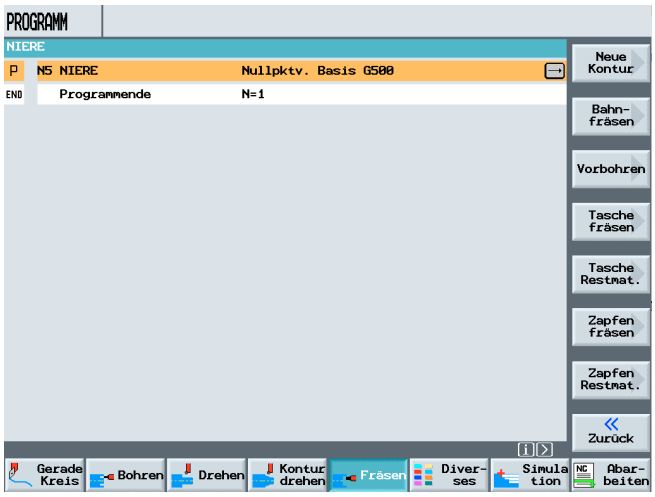
in the area Milling by pressing the softkey





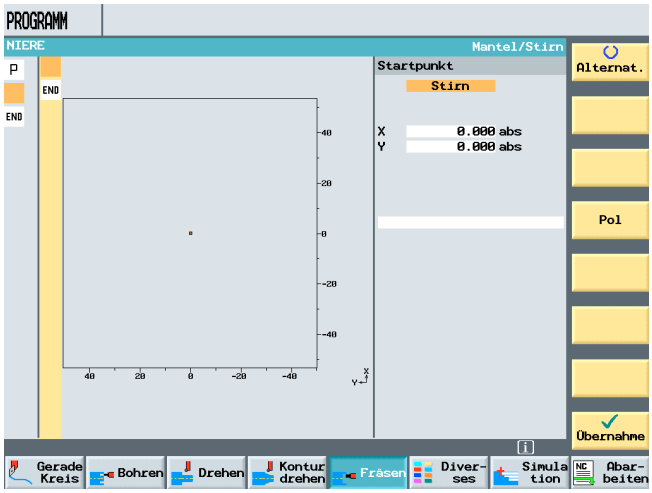
and entering the contour name





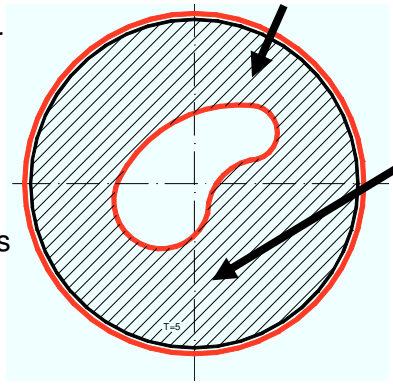
.

In this example, the contour of a kidney is to leap out as an “island“. For that reason, two contours have to be defined:



Boundary contour

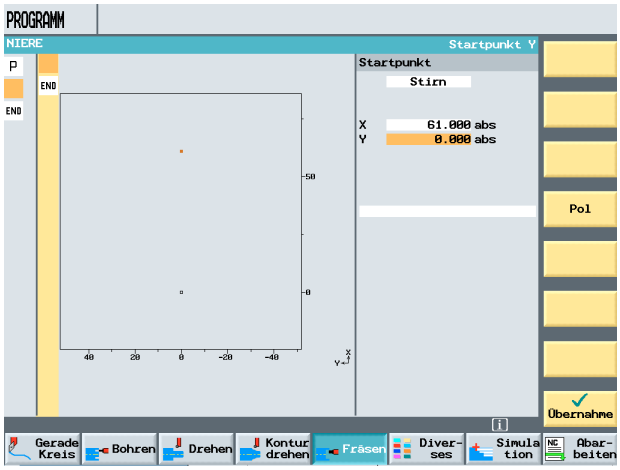
First, a boundary contour that surrounds the raw part in the example, and then the contour in the form of a kidney.
The hatched area between the two contours is to be machined.



Free form contour

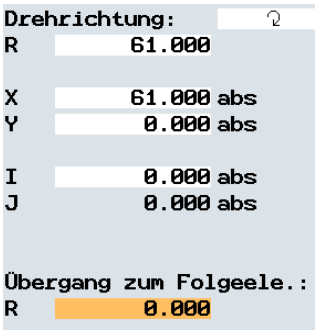
First, the boundary contour is described.

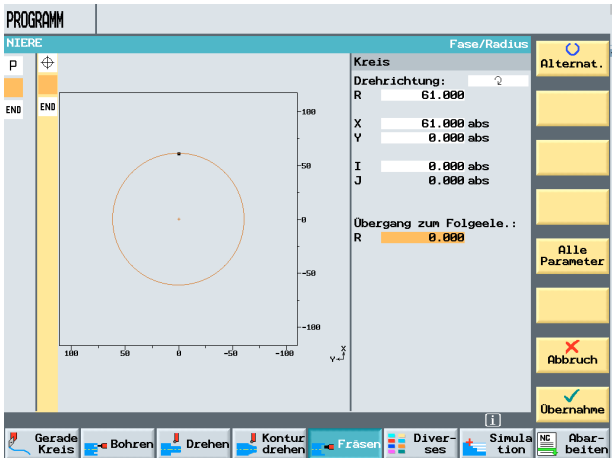
The starting point is entered “radius related“.



The boundary contour should have a larger diameter than the raw part so that no burr remains.
However, the diameter should not be too large since the outer path represents the longest machining path.

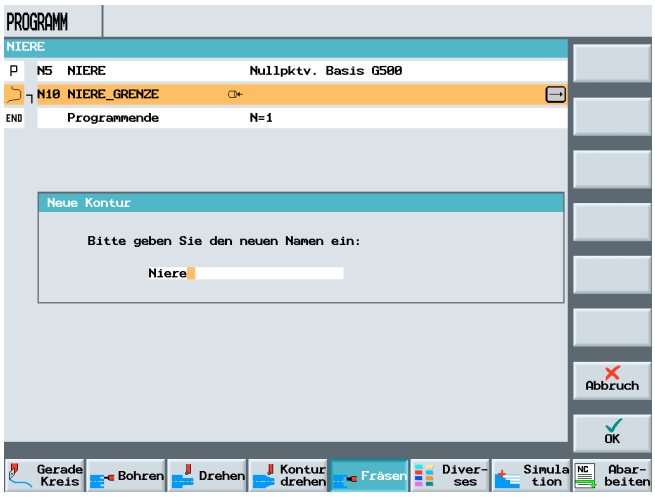
After entering the values for the boundary circle



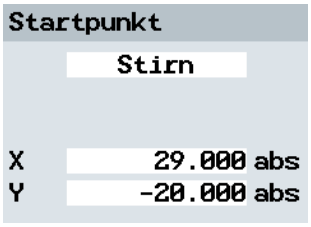


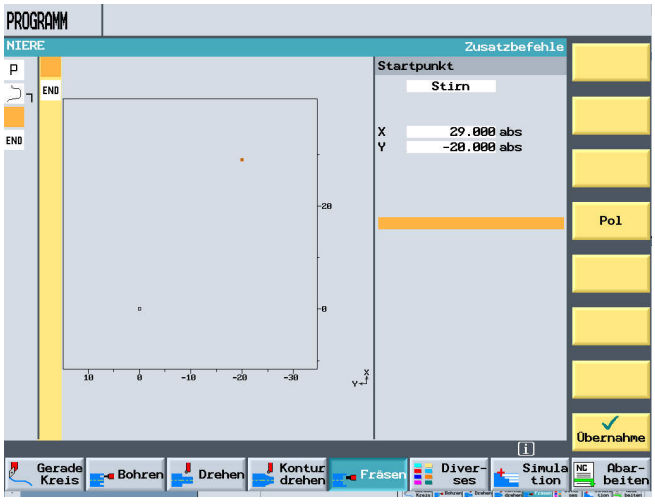
The machine manufactorer/commissioning engineer can freely define the coordinate cross in the contour calculator. This has to be taken into account during programming.

and accepting the values into the machining plan, next another contour is set up.

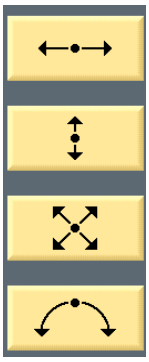


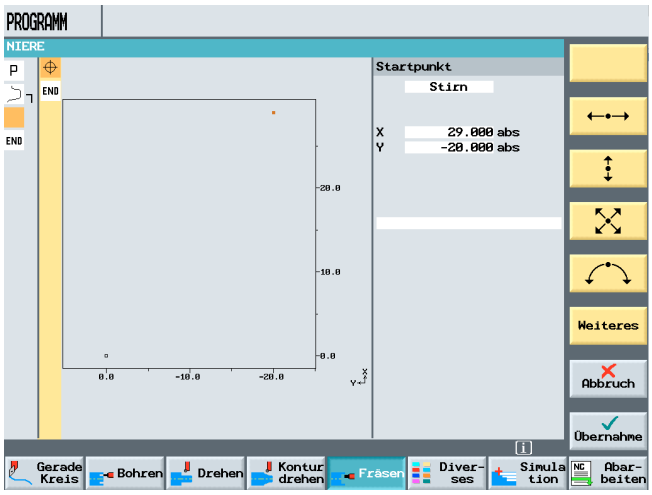
After entering the contour starting point





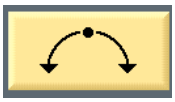
the milling contour is generated by means of these four softkeys

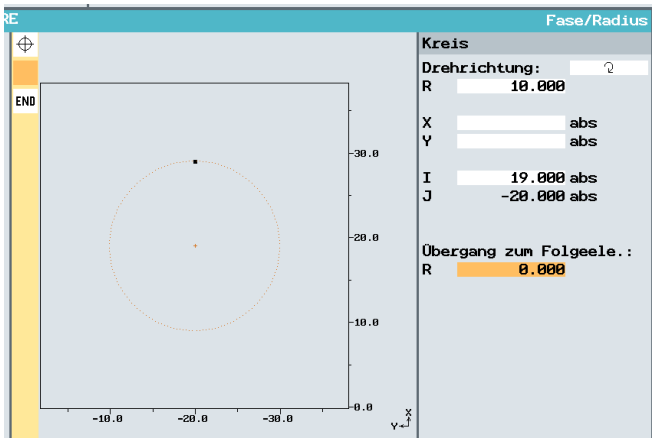




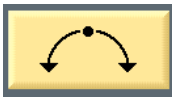
as in the case of the contour description for turning.

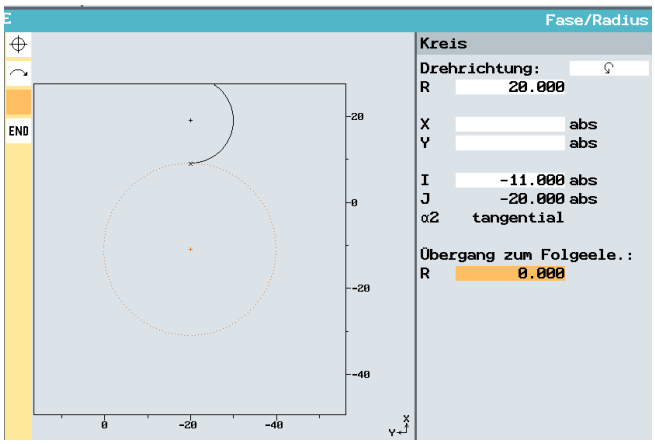
Element 1



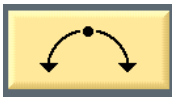


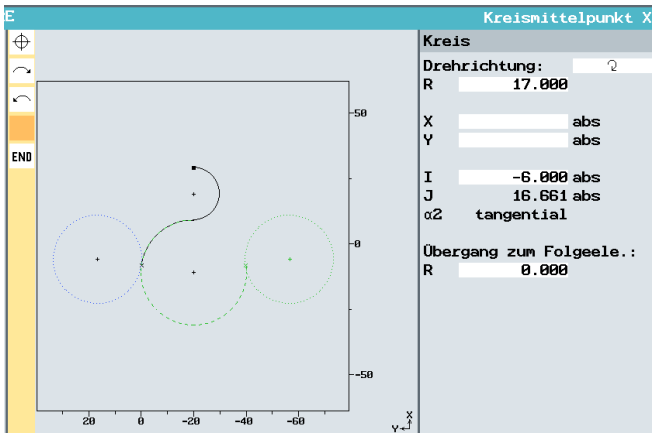
Element2



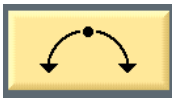


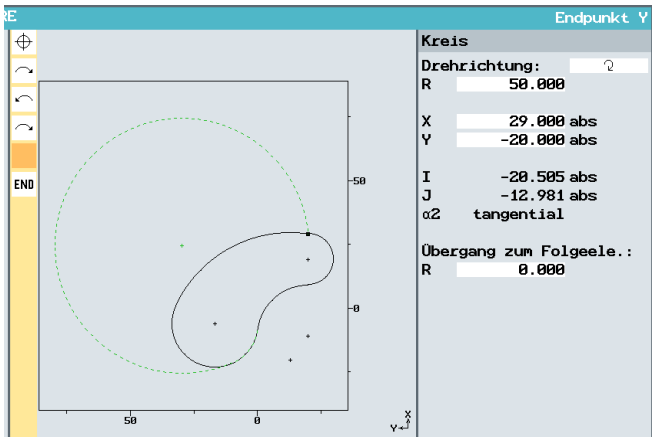
Element3



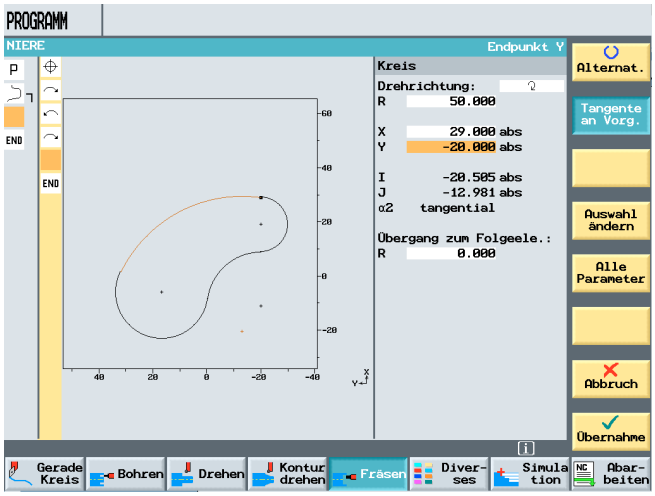


Element 4

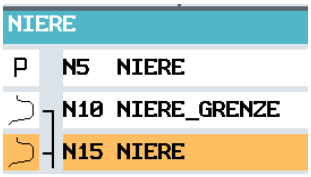


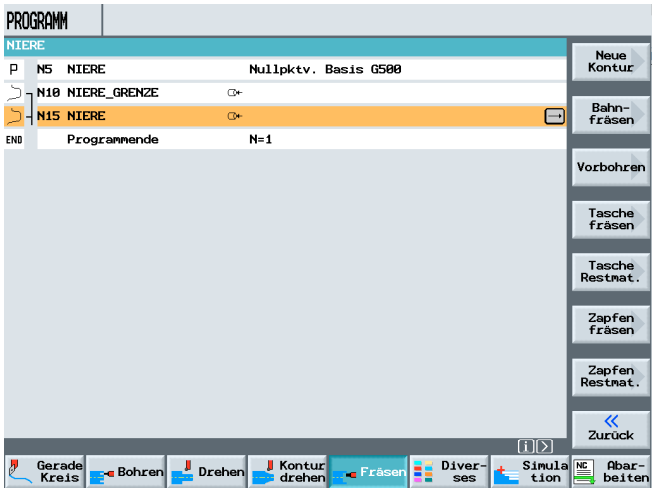


The contour is done and by pressing the softkey

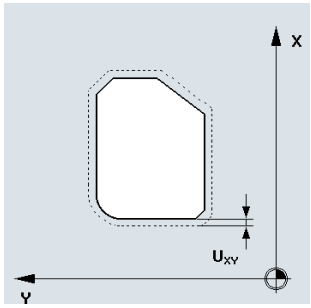


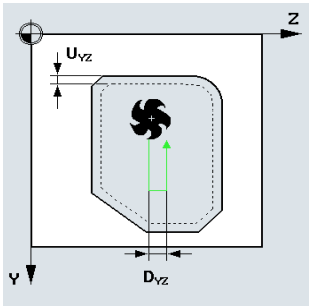
is accepted in to the machining plan.

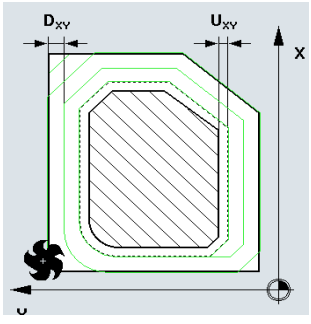




The following options are provided for machining milling contours.

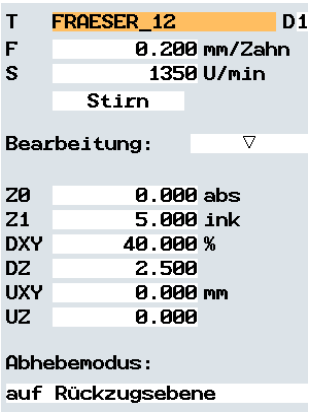




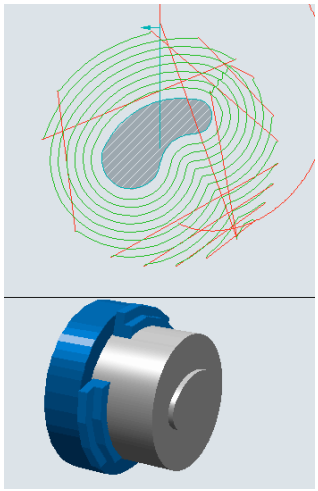


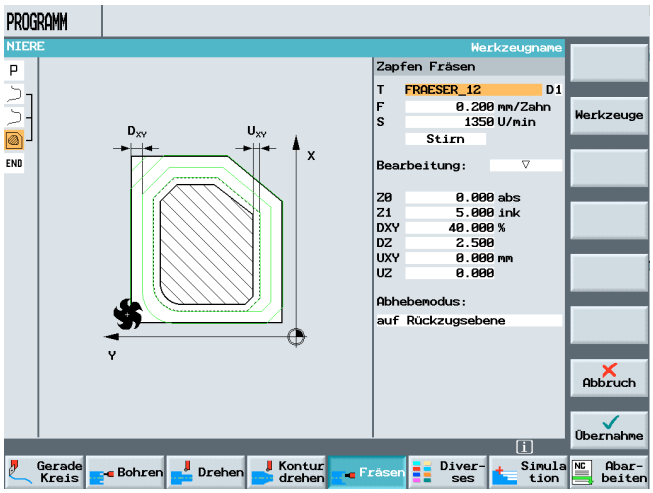
After selecting the machining

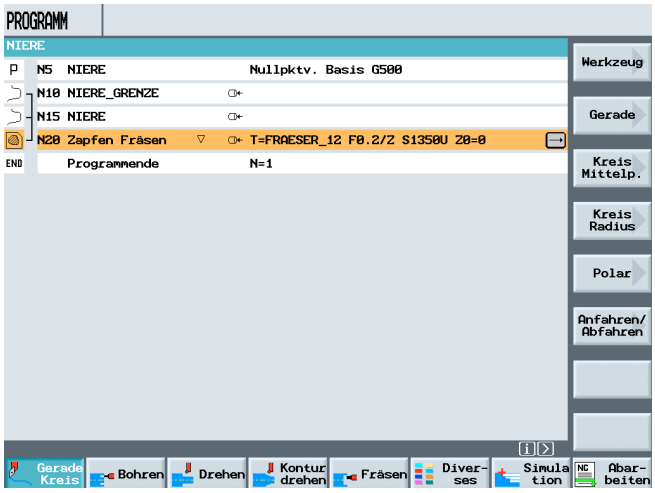
and entering the values in the input screen form



, these actions are included in the machining plan and can be simulated.







The milling strategy of ShopTurn looks like this: Milling starts outside the programmed boundary contour. The machining pattern corresponds to the final pattern of the "island contour“, only on a larger scale. Machining continues up to the final dimension. By using this strategy, even hard and tough materials can be “peeled“ to the desired finishing dimension.

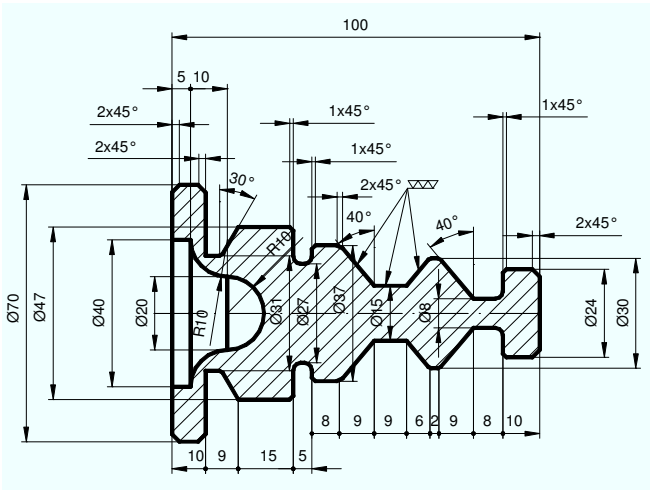
13 Delimitation and Counter Spindle

Delimitation and machining with a counter spindle under ShopTurn is explained based on an example.

Based on the drawing, the following is explained:

- Groove turning
- Delimitation
- Machining with the counter spindle

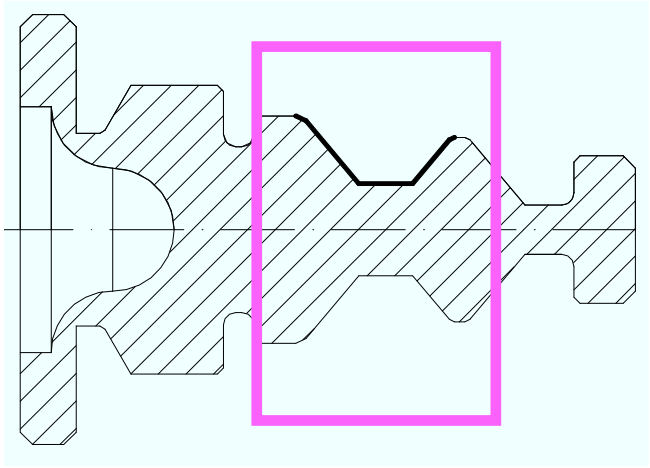
.

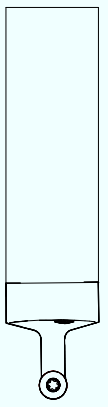


The following is to be noted regarding the workpiece called "Fitted Shaft“ <<?>>:

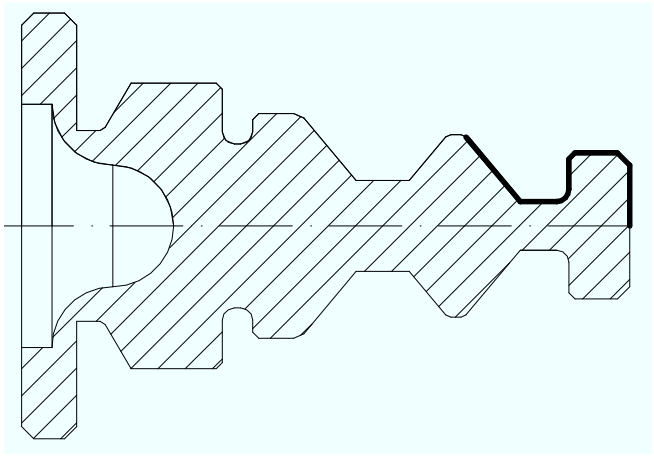
During finishing, the marked area is to be machined with a special tool.

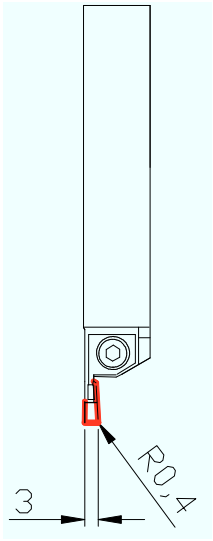
This special tool is a mushroom. The radii generated at the contour transitions are within the tolerance.



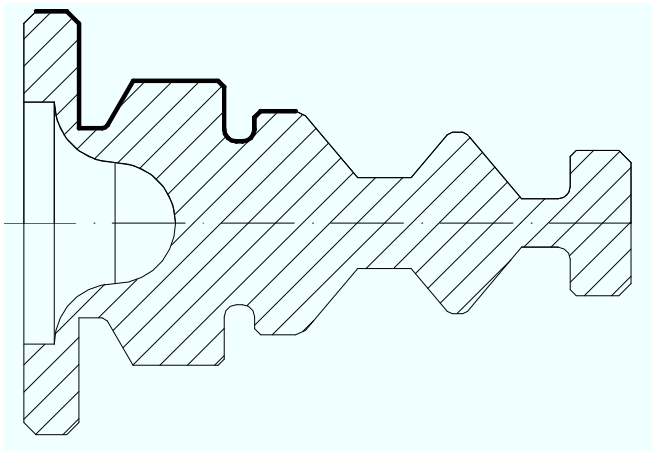


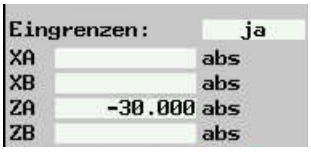
The two other areas are finished with a grooving lathe tool.





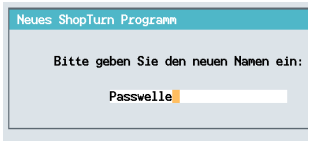
Here, the function “delimitation“ is used.

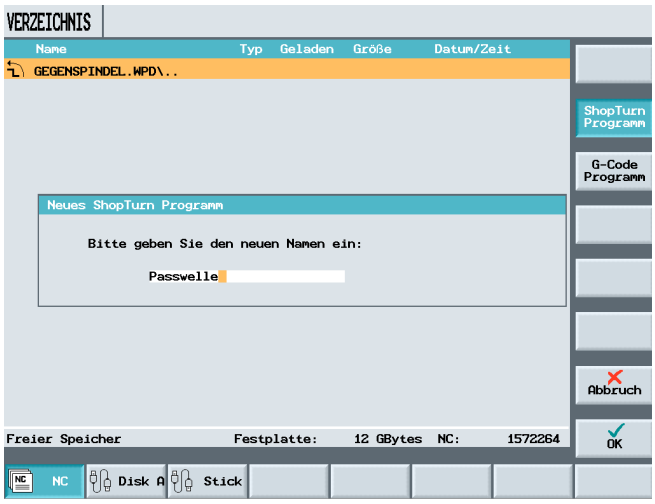




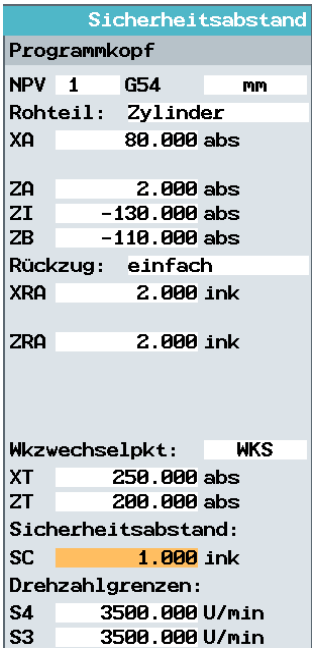
After entering the program name

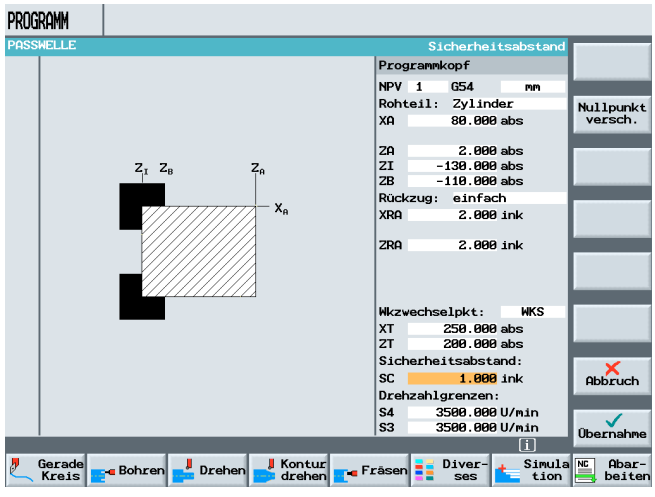
Fitted Shaft





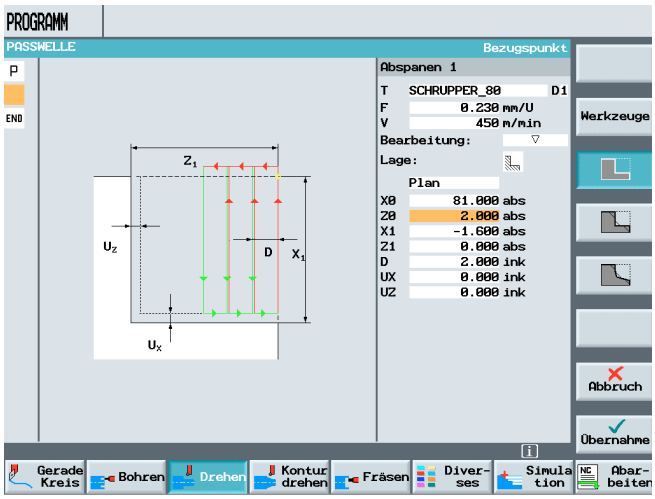
the following values are entered in the program header.

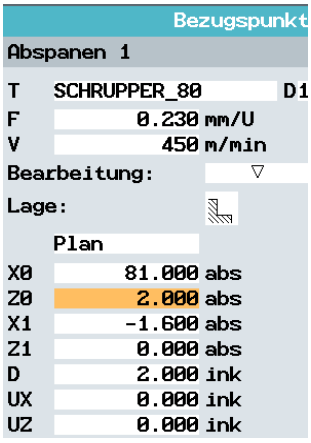


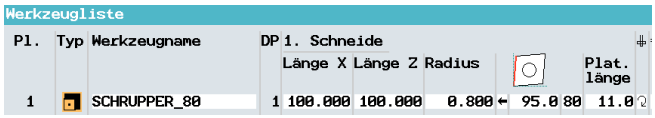


The frontal face is faced using the machining cycle.

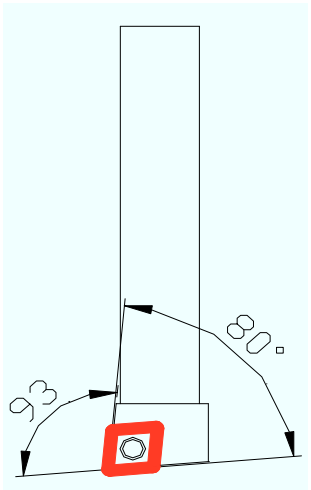
No finishing allowance is defined.

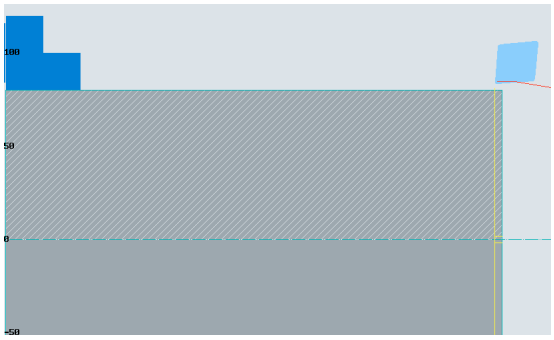






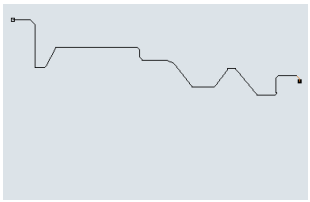
Machining is to be done with a roughing tool.

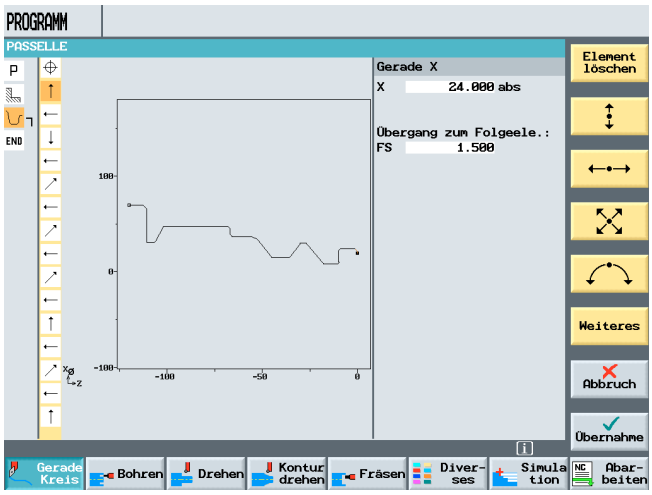




The input can be checked with a simulation.

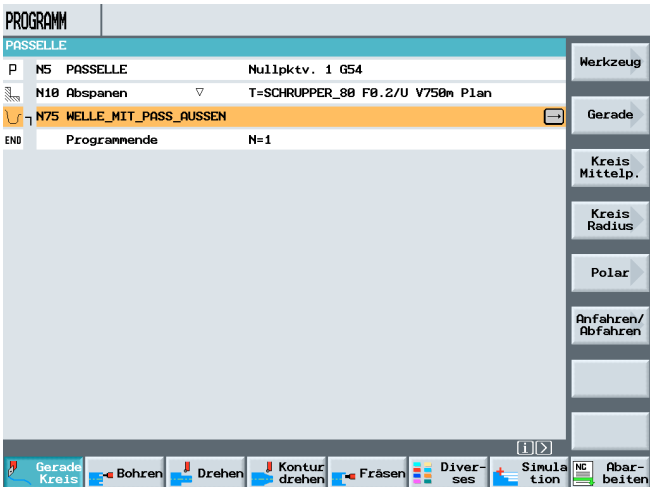
Next, the contour is described using the contour calculator.





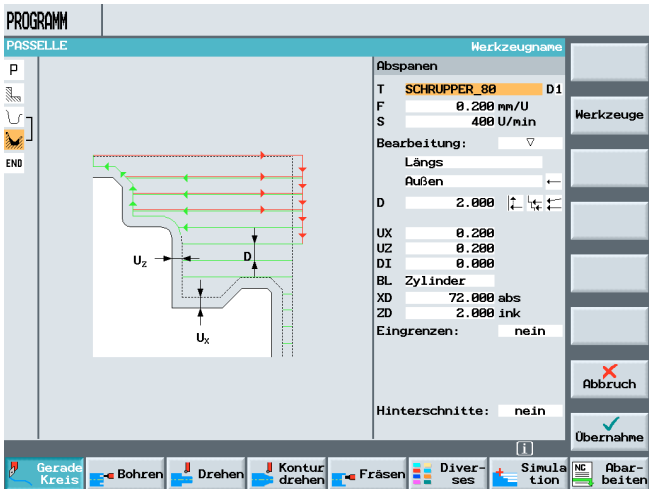
After entering the contour and accepting it into the machining plan, the bracket has opened.

For the bracket to be closed, the machining for the generated contour has to be specified next.



The machining cycle for contouring is selected.

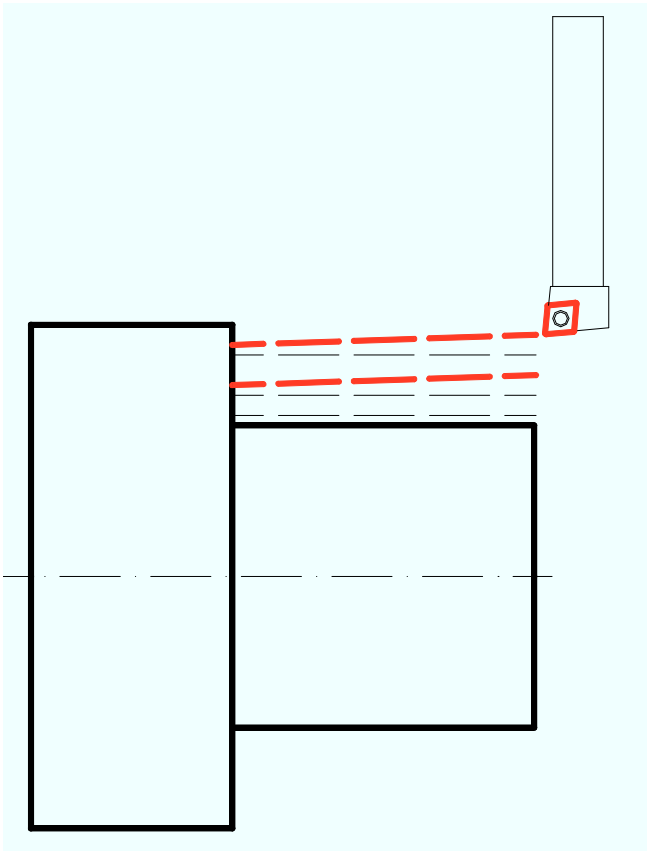
In this case, the changing cutting depth is selected.



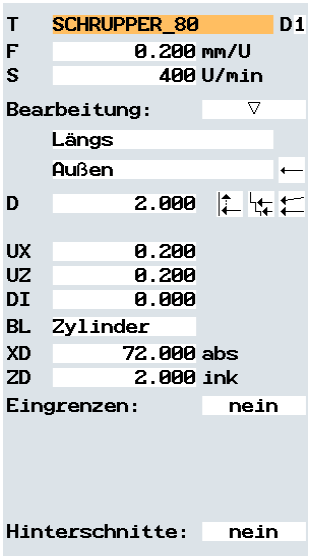
For changing cutting depths, the value for the adjustment is conical.

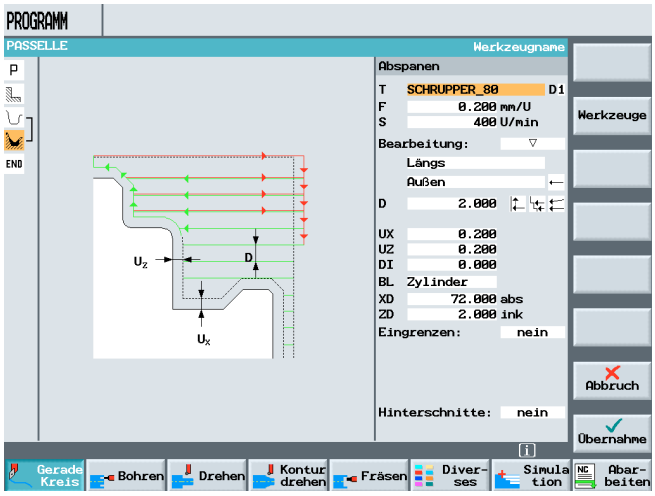
By selecting the changing cutting depth, the point on the tool tip where the greatest wear takes place is shifted.

The friction is distributed over a larger area of the tip. The result is a longer life for the tip.

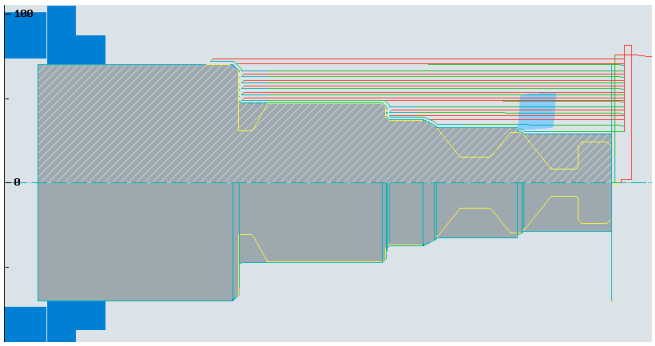


After entering the values in the screen form,

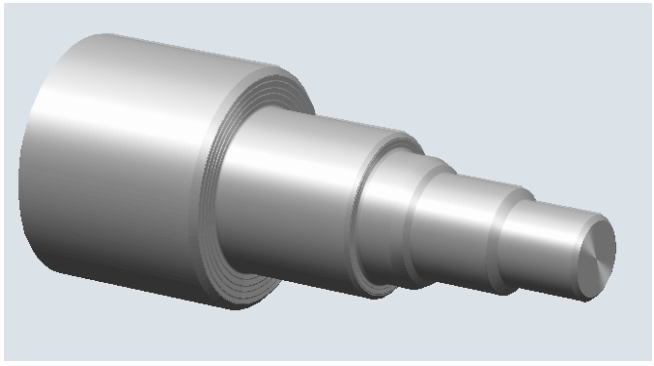




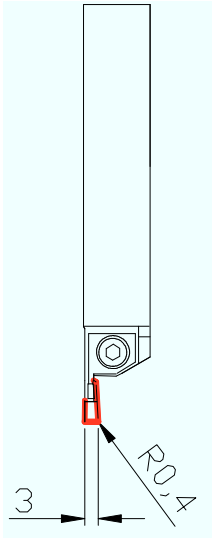
it can be checked with a simulation.



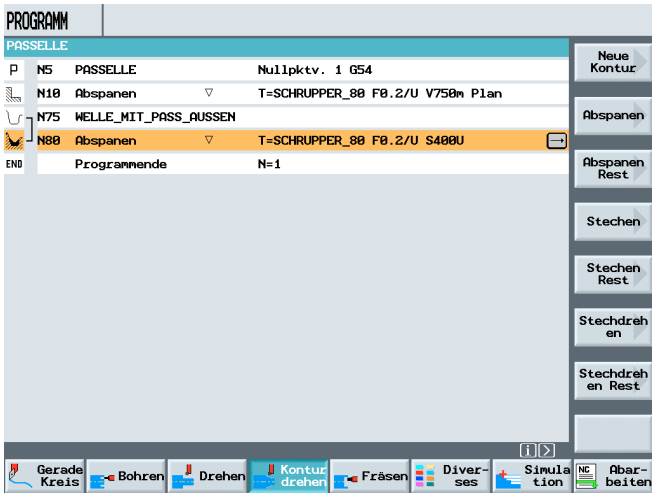
Since for roughing the workpiece an 80° insert is used, machining takes place without relief cutting.



The residual material is machined with a grooving lathe tool.

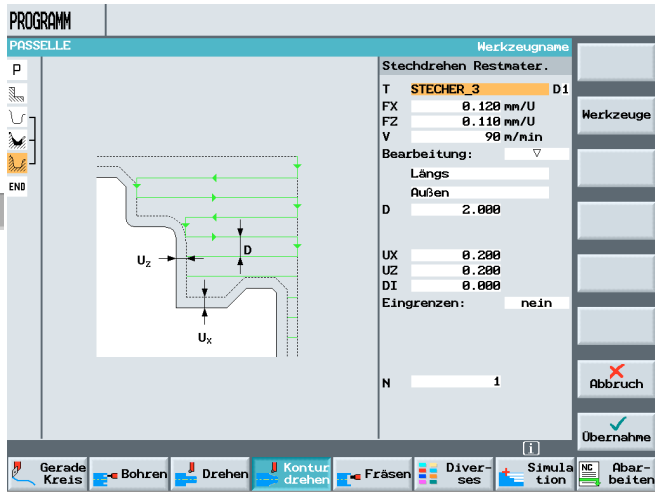


Pressing the softkey



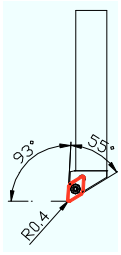
opens the screen form for

Stechdrehen Restmater.

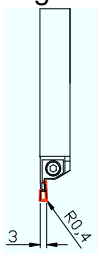


It has to be noted that during residual machining, the correct cycle is selected for the tool that is used respectively.

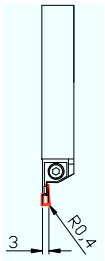
Example:
Roughing tool 55°



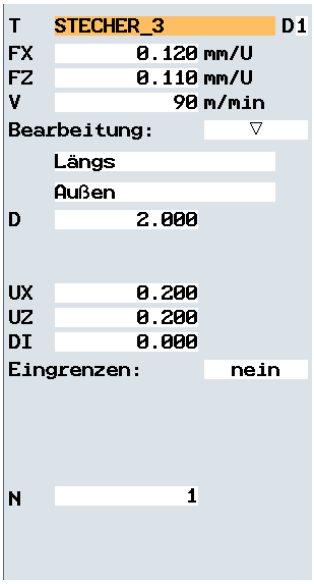
Grooving tool 3mm

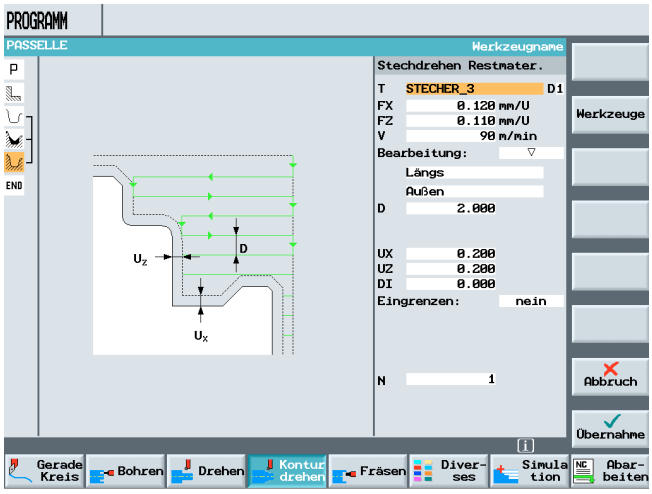


Grooving lathe tool

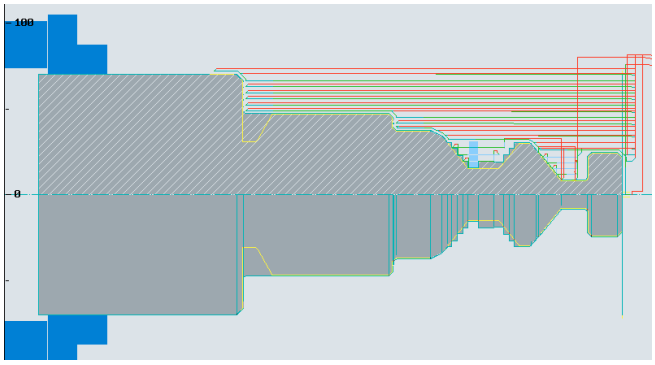


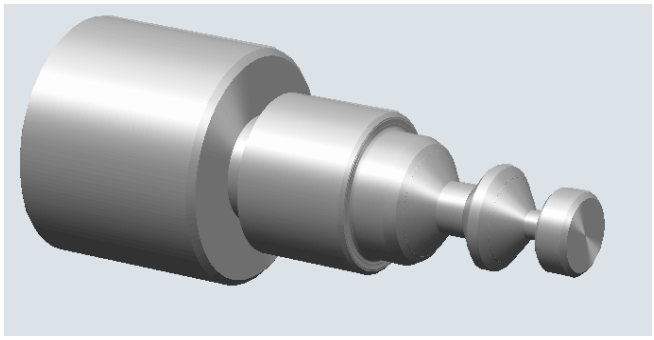
After entering the values,





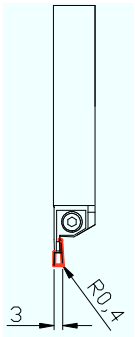
residual material machining is shown in the simulation.





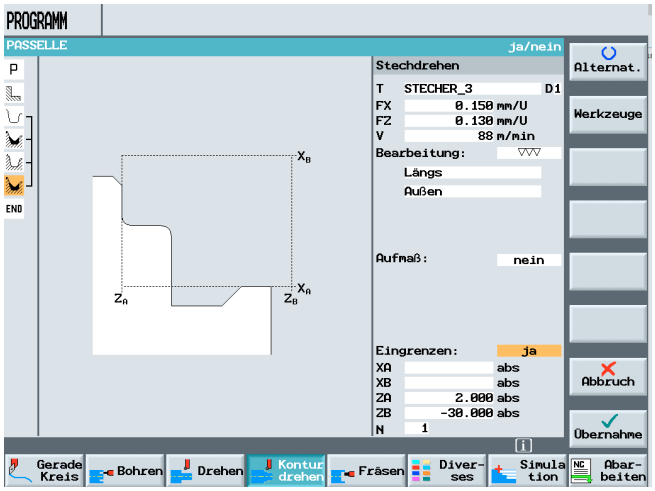
Next, finishing follows.

This machining is done with the active tool.

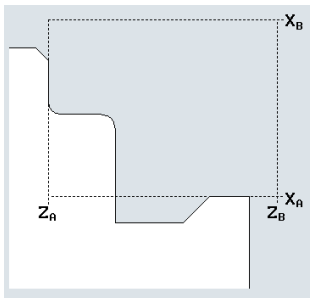


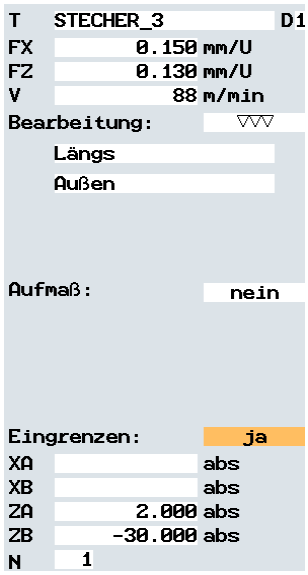
After calling the grooving lathe cycle

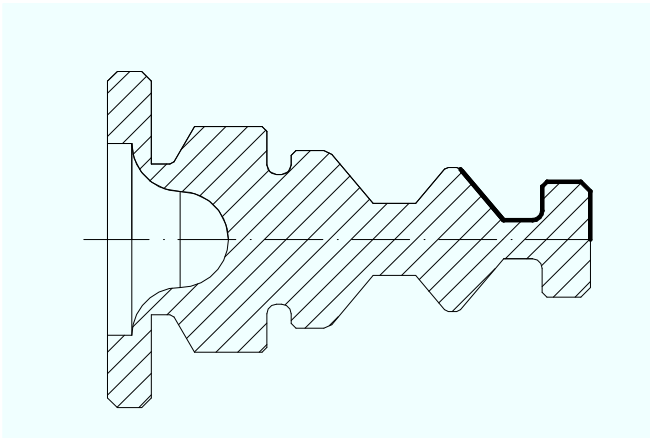
and pressing the softkey



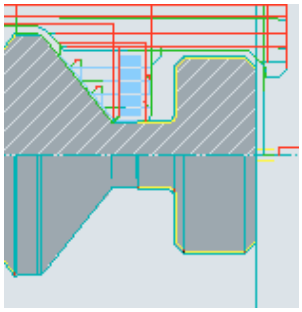
the function delimitation in this input screen form is used.





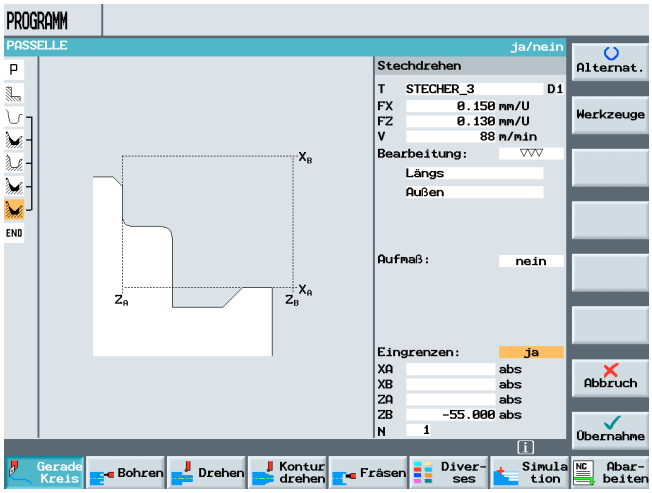


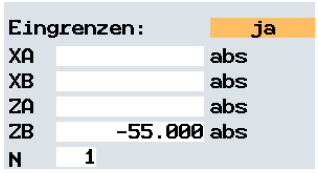
By starting the simulation, the travel movement is shown with the delimitation.

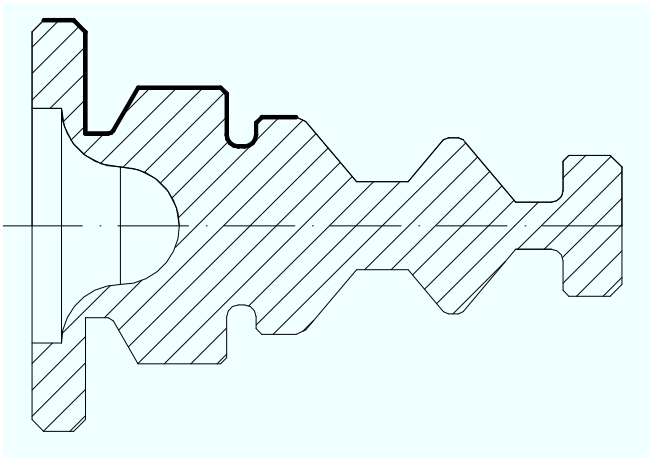


The same cycle is called again (this can also be done with the function “Copy“).

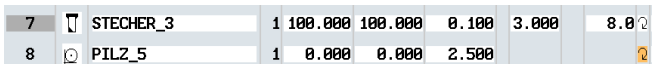
Only the values for delimiting are changed.

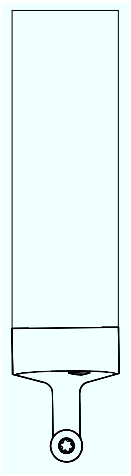


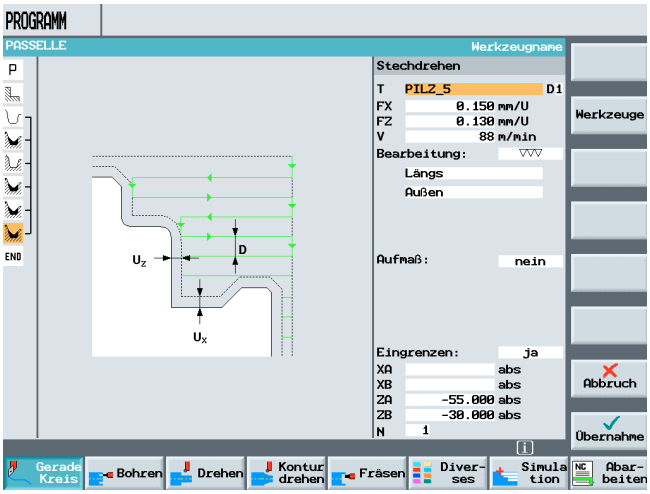




After setting up a mushroom tool

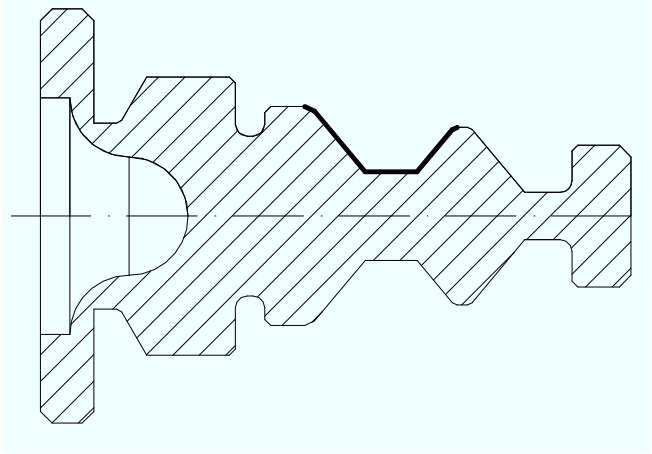


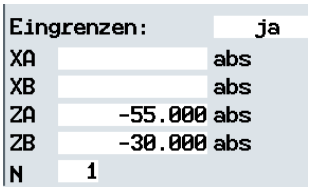


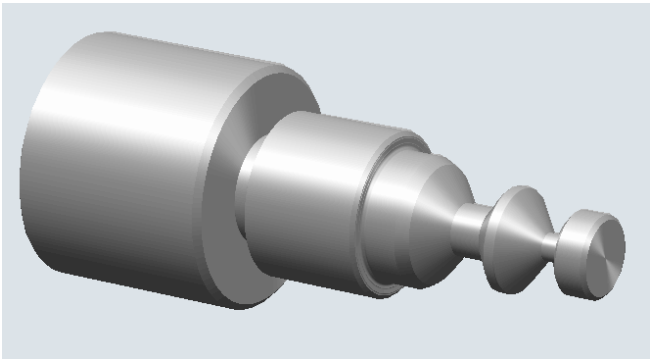


this tool is called in the screen form for machining.

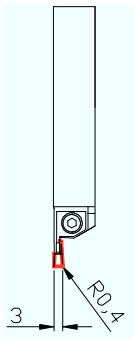
Here also, the machining area is delimited.



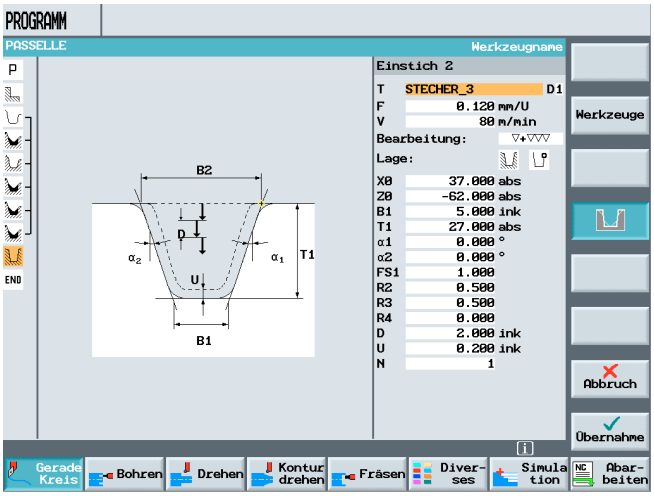




As the last machining step on the main spindle, a groove is made with the grooving tool.

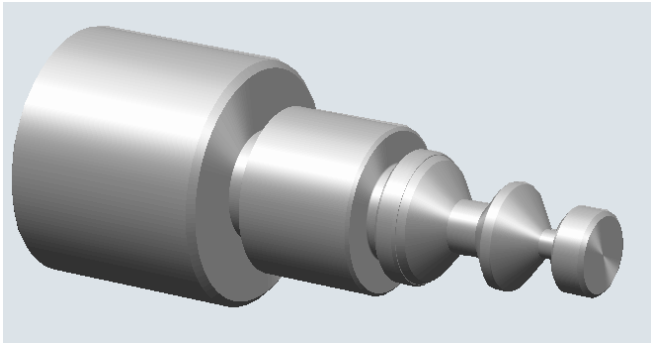


After entering the values



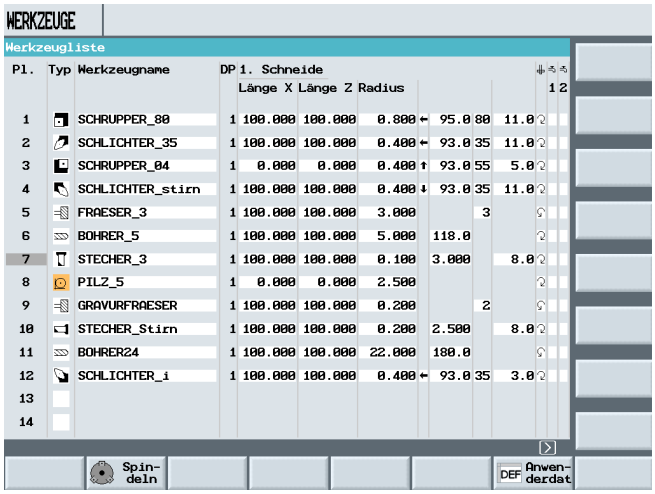
T	STECHER_3	D1
F	0.120	mm/U
V	80	m/min
Bearbeitung:		
Lage:		
X0	37.000	abs
Z0	-62.000	abs
B1	5.000	ink
T1	27.000	abs
α1	0.000	°
α2	0.000	°
FS1	1.000	
R2	0.500	
R3	0.500	
R4	0.000	
D	2.000	ink
U	0.200	ink
N	1	

and accepting them into the machining plan, machining on the main spindle is completed.

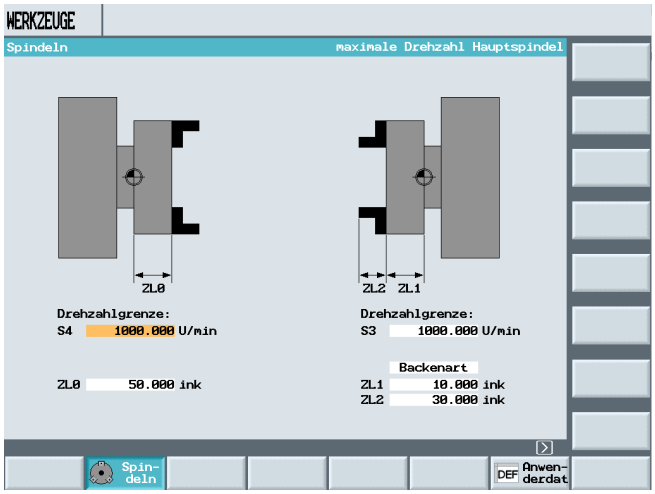


To machine with the counter spindle, first the chuck of the spndle and the jaw type have to be defined.

We access the corresponding screen form by pressing the softkey



expanding the softkey bar and pressing the softkey



After entering the speed limit for the chuck of the main spindle

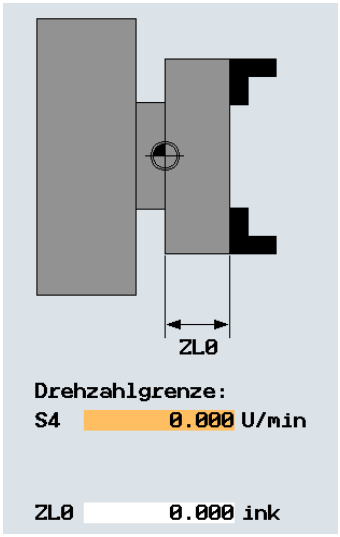
Drehzahlgrenze:
S4 1000.000 U/min

and the chuck of the counter spindle,

Drehzahlgrenze:
S3 1000.000 U/min

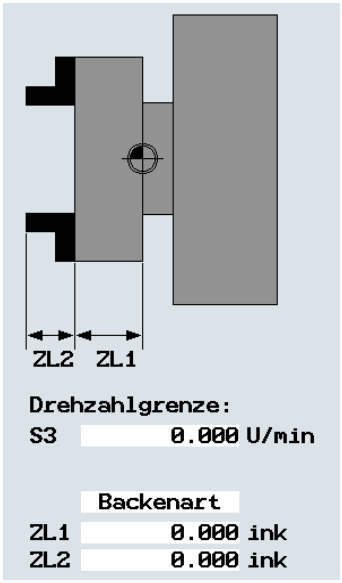
we select between two types of chuck jaws.

The type of jaw is defined by means of the respective input fields



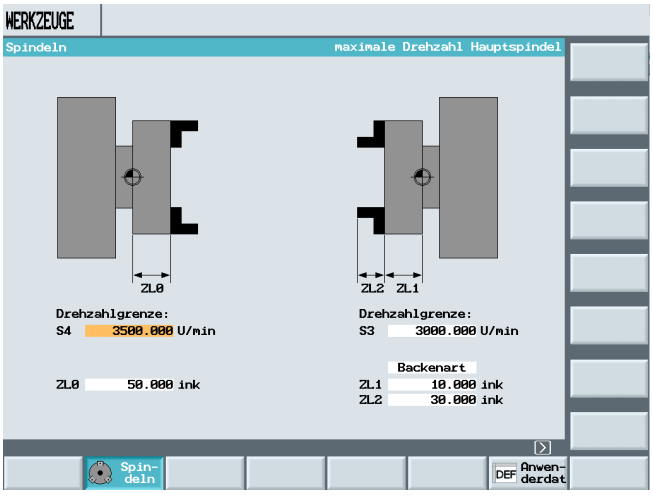
Backenart
ZL1 0.000 inkr
ZL2 0.000 inkr

After entering the values, the corresponding input screen form is opened

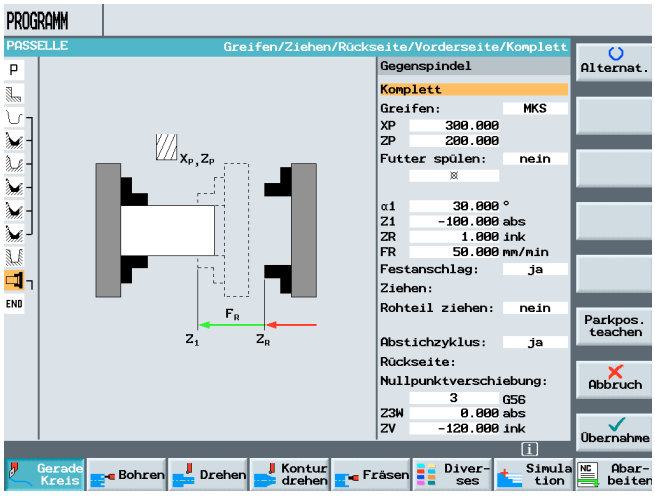


by pressing the softkey

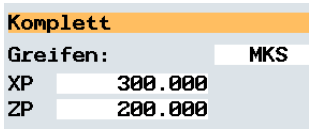
Backenart
ZL1 10.000 inkr
ZL2 30.000 inkr



and pressing the softkey

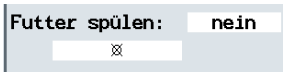


After the complete machining is selected, the tool's park position is defined.

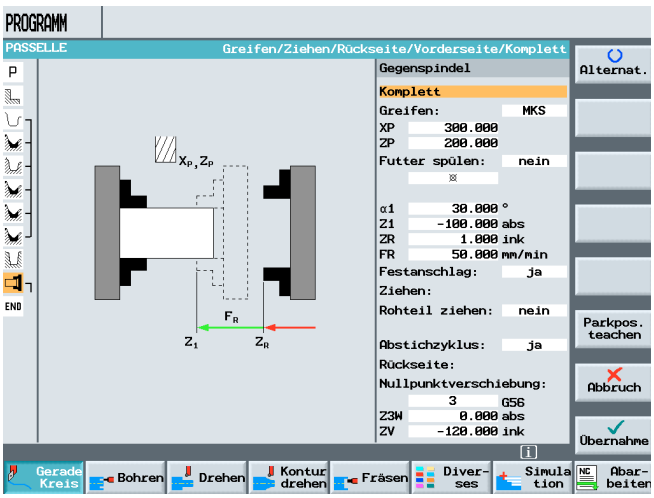


When positioning the workpiece, it is advisable to use the coordinates of the tool change point in the program header for the park position, in order to obtain a defined travel movement.

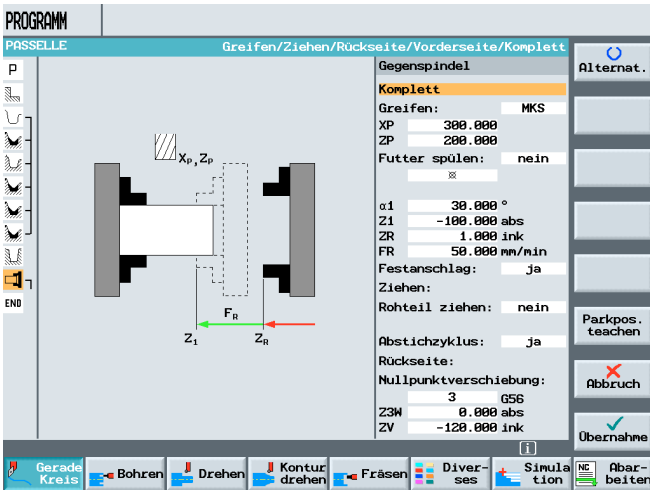
The workpiece is to be transfered without flushing the chuck



and when the spindle is not moving.



After entering the remaining values



Komplett

Greifen:	MKS	
XP	300.000	
ZP	200.000	
Futter spülen:	nein	
	☒	
α1	30.000	°
Z1	-100.000	abs
ZR	1.000	ink
FR	50.000	mm/min
Festanschlag:	ja	
Ziehen:		
Rohteil ziehen:	nein	
Abstichzyklus:	ja	
Rückseite:		
Nullpunktverschiebung:		
3	G56	
Z3W	0.000	abs
ZV	-120.000	ink

the input is accepted into the machining plan.

The zero point shift can be selected freely.

Nullpunktverschiebung:

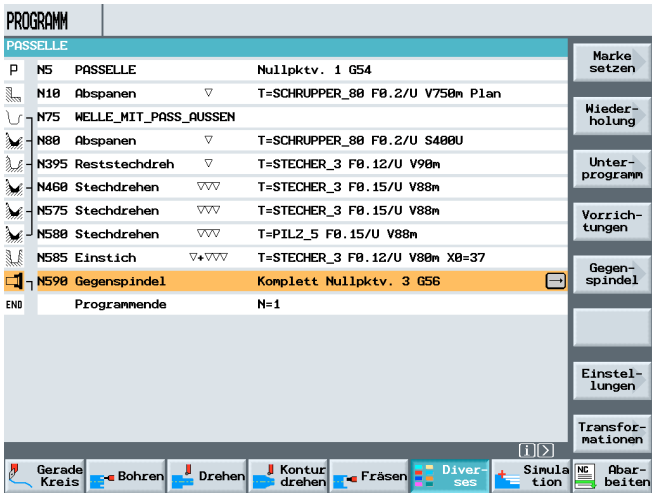
3	G56	
Z3W	0.000	abs
ZV	-120.000	ink

The value Z3W defines the machining position of the counter spindle in reference to the machine zero point. If no value is entered, machining takes place with the maximum possible distance to the machine zero point.

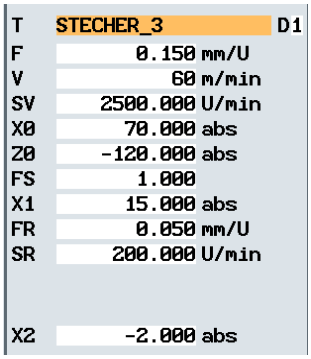
Since in the input screen form, the question about a cut-off cycle was answered with yes

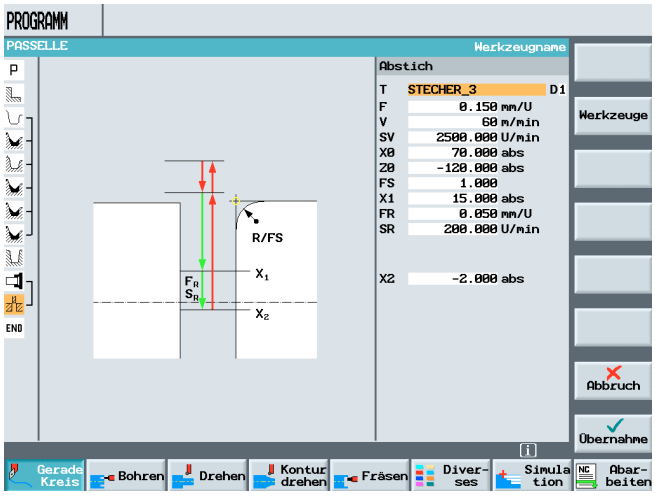
Abstichzyklus: ja

a bracket opens in the machining plan.

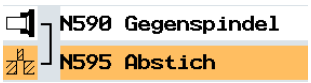


After opening the input screen form for the cut-off cycle and entering the values,

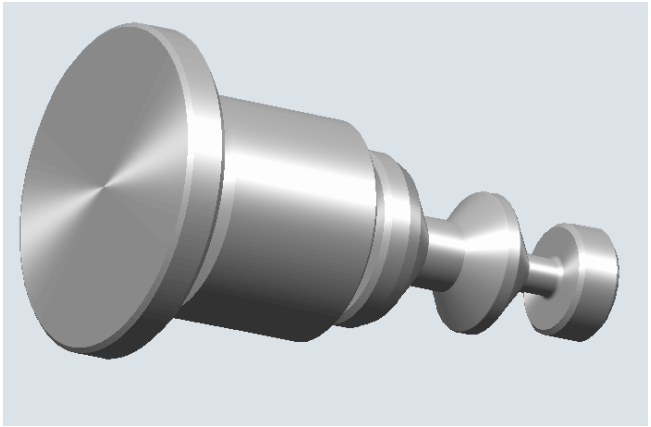




the bracket is closed through acceptance in the machining plan.



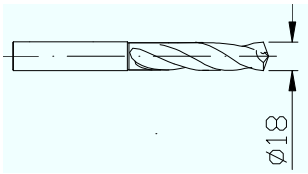
The machining can be tested in the simulation.



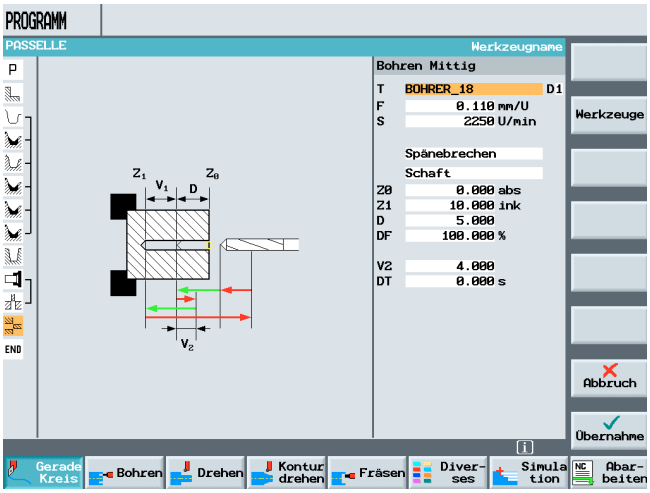
First, a hole is made on the back side, so that in the next machining step, the inside contour can be machined.

A drill with a diameter of 18mm is set up.

10		STECHER_Stirn	1	100.000	100.000	0.200	2.500	8.0
11		BOHRER_18	1	100.000	100.000	18.000	180.0	

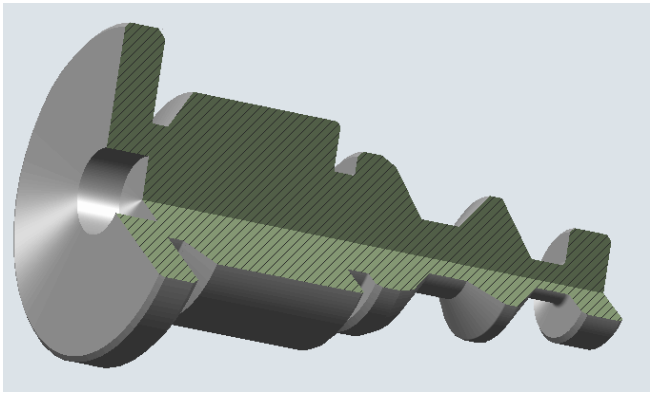


After the drill is set up, the screen form for drilling in the center is opened and the following values are entered.

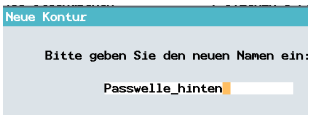


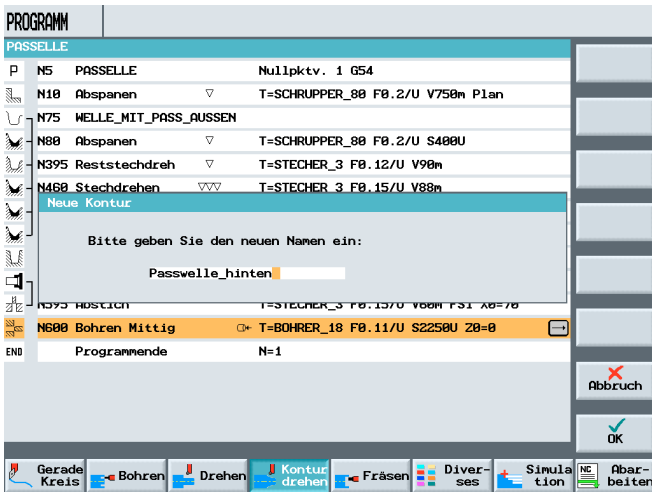
Bohren Mittig		
T	BOHRER_18	D1
F	0.110	mm/U
S	2250	U/min
	Spänebrechen	
	Schaft	
Z0	0.000	abs
Z1	10.000	ink
D	5.000	
DF	100.000	%
V2	4.000	
DT	0.000	s

The input can be checked with the simulation.



Next, the contour for rear machining is set up.

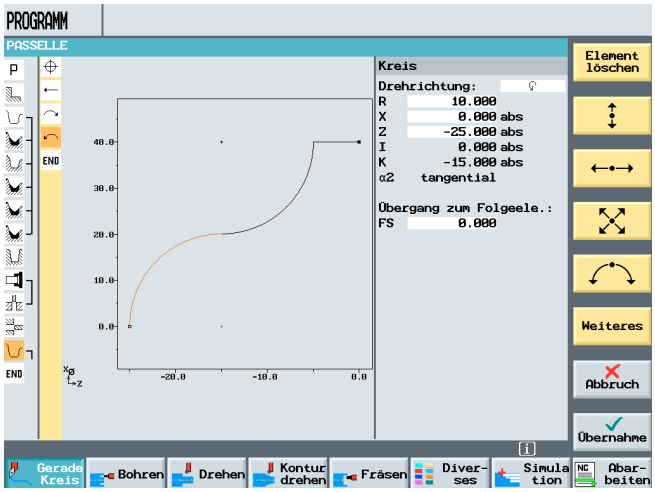




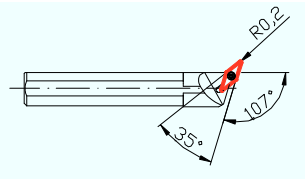
The contour is described in the usual direction.

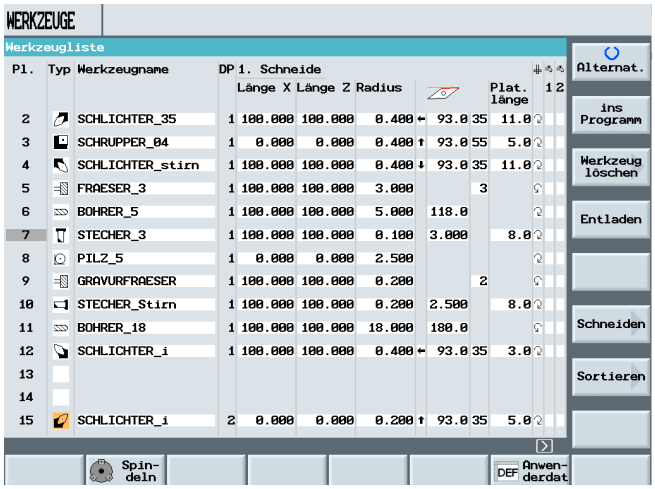
Working on the counter spindle is not taken into account for contour programming.

After the contour is described,

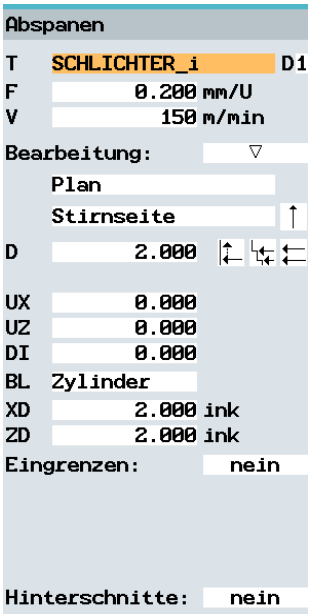


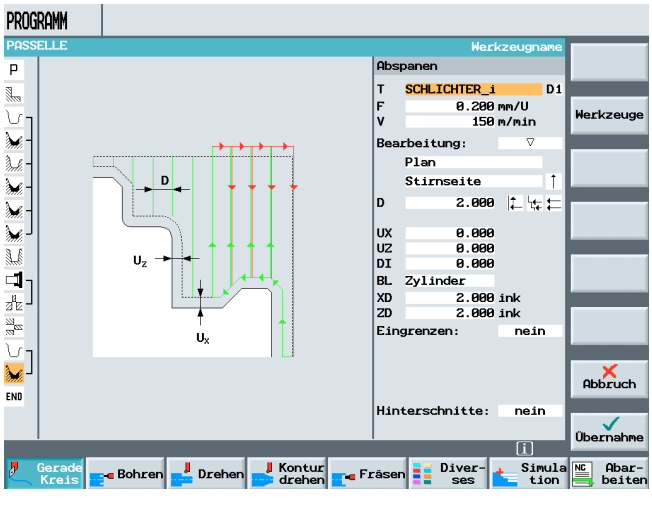
the following tool is set up for internal machining



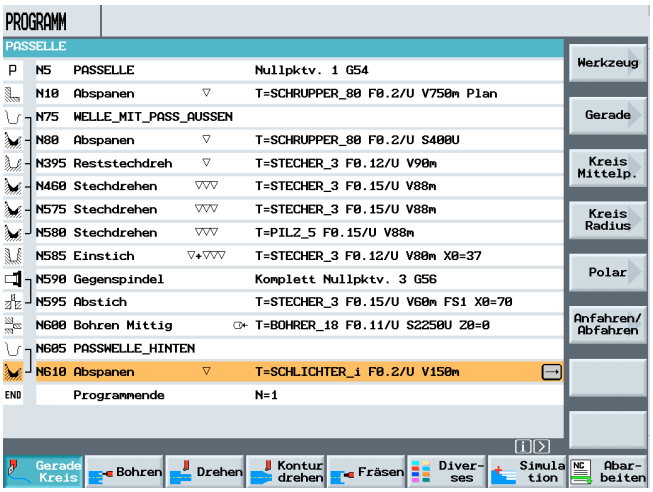


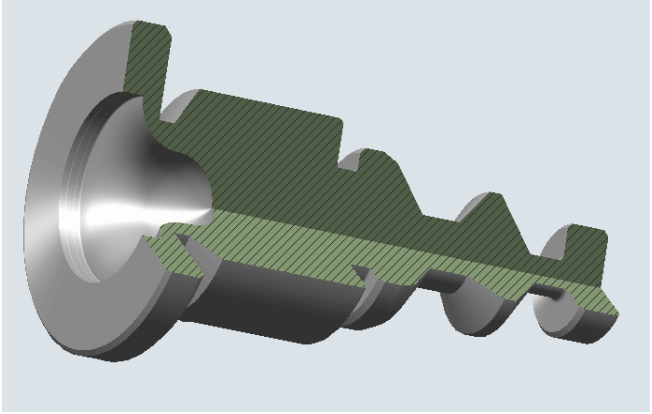
After entering the values in the screen form





the program is done.

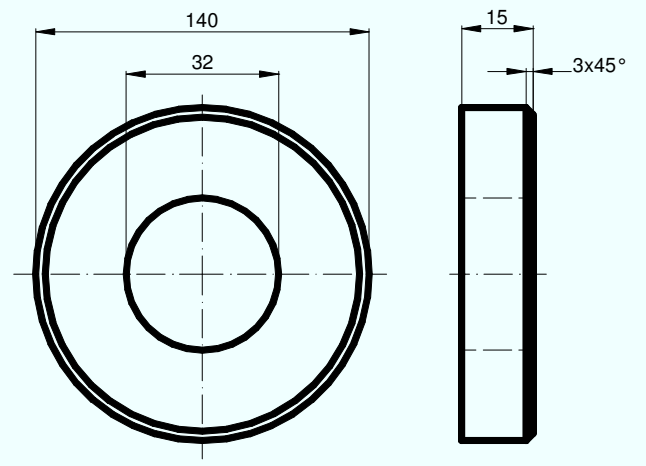




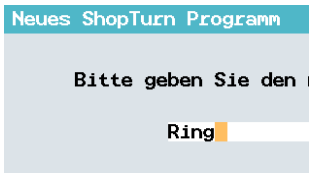
14 Program Loops, Shifts

Based on an example, the following is described in this module: programming loops and shifts as well as the setup of a workpiece counter with message texts, and the setup of a sub-program in ShopTurn.

The ring shown here is to be manufactured 6 times.

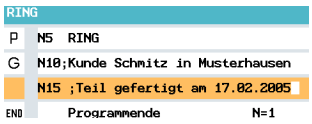


A program with the name



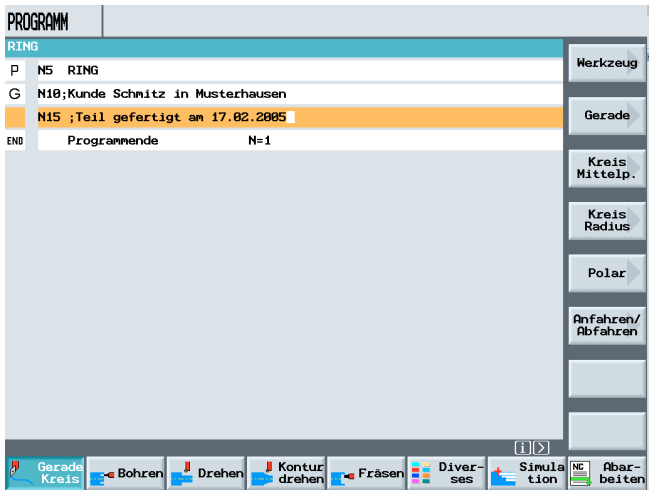
is created.

Under ShopTurn, it is possible to insert G-code lines in a program.
These lines can also be used as header and information lines.

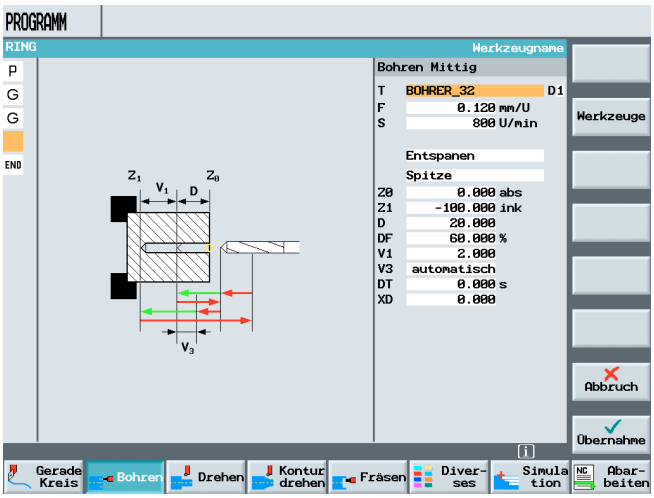


By commenting out the respective line

any text that is not processed in the program can be inserted after the semicolon.

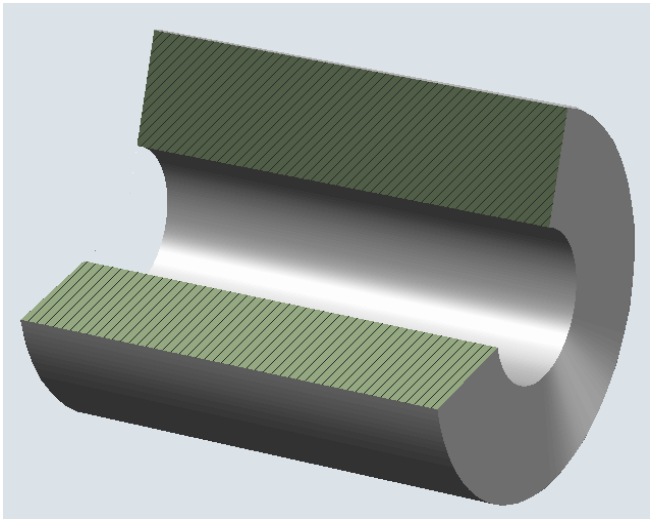


In the first machining step, a through hole of 32mm is programmed.



BOHRER_32	D1
0.120	mm/U
800	U/min
Entspanen	
Spitze	
0.000	abs
-100.000	ink
20.000	
60.000	%
2.000	
automatisch	
0.000	s
0.000	

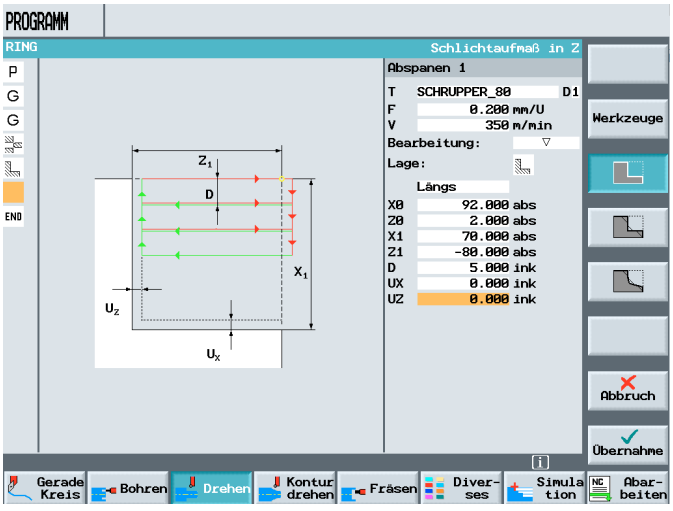
After the subsequent facing,



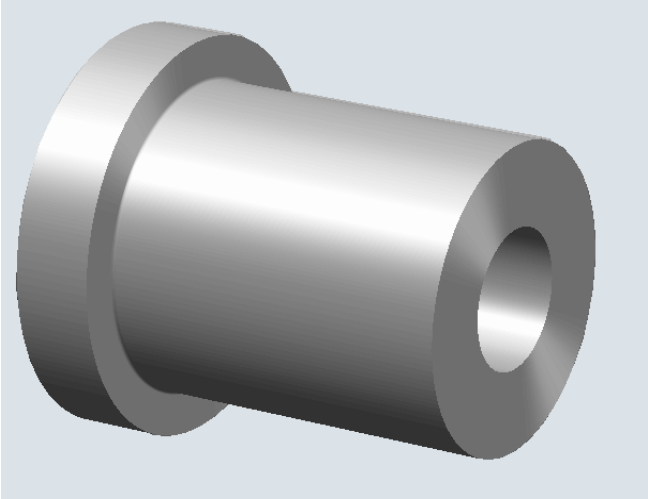
longitudinal machining to a diameter of 70mm is the next step.

.

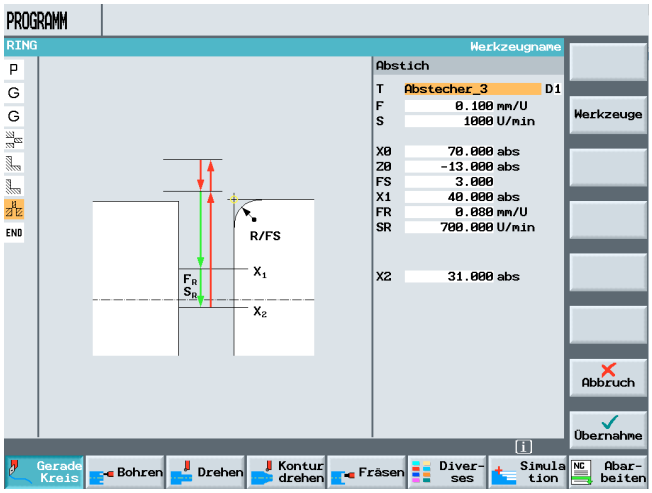
T	SCHRUPPER_80	D1
F	0.200	mm/U
V	350	m/min
Bearbeitung:		
Lage:		
	Längs	
X0	92.000	abs
Z0	2.000	abs
X1	70.000	abs
Z1	-80.000	abs
D	5.000	ink
UX	0.000	ink
UZ	0.000	ink



Finally, to manufacture a workpiece, the cut-off is made.

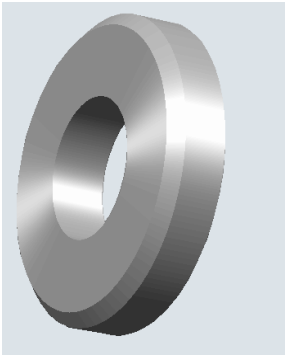


T	Abstecher_3	
F	0.100	mm/U
S	1000	U/min
X0	70.000	abs
Z0	-13.000	abs
FS	3.000	
X1	40.000	abs
FR	0.080	mm/U
SR	700.000	U/min
X2	31.000	abs

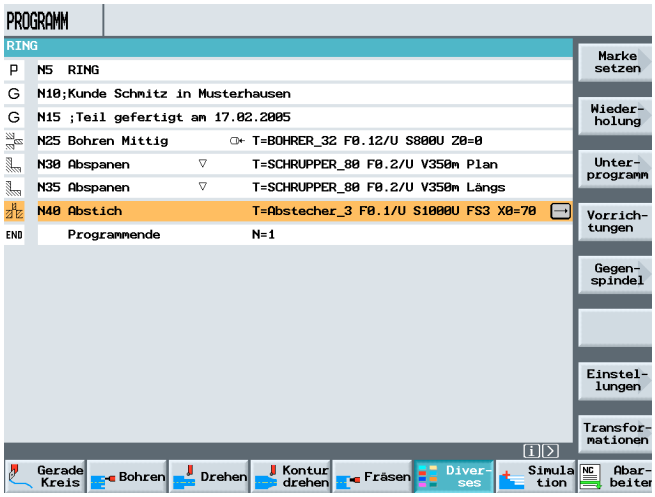


The ring is done.

Five more will follow.



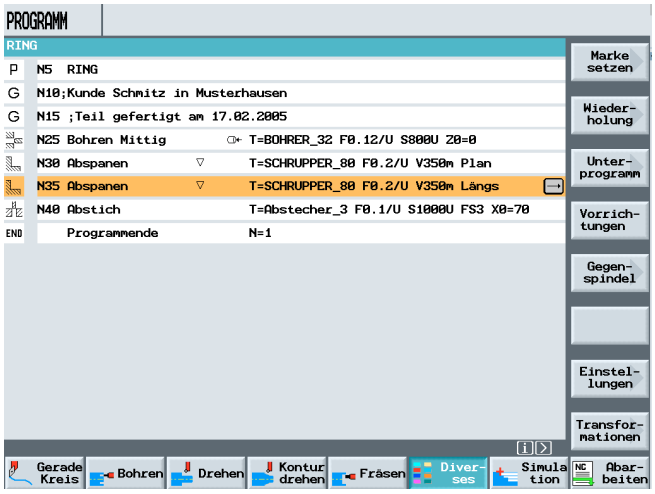
First, the area of the program that is to be repeated is delimited.



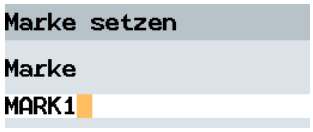
To this end, the cursor has to be above the program block to be delimited; in this example, above the machining.

By pressing the softkey

and

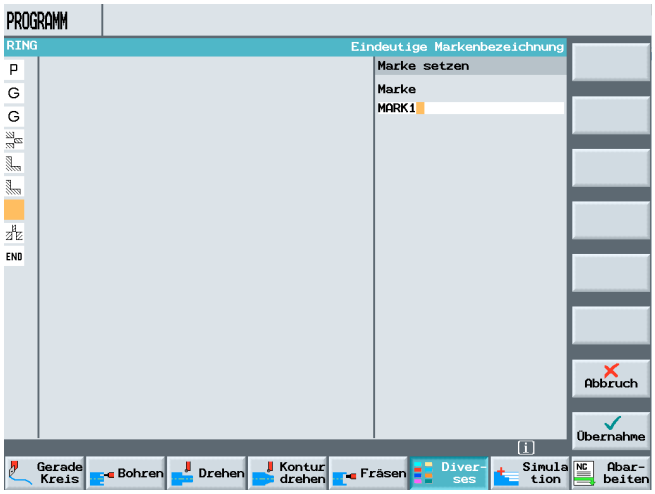


the name of the mark

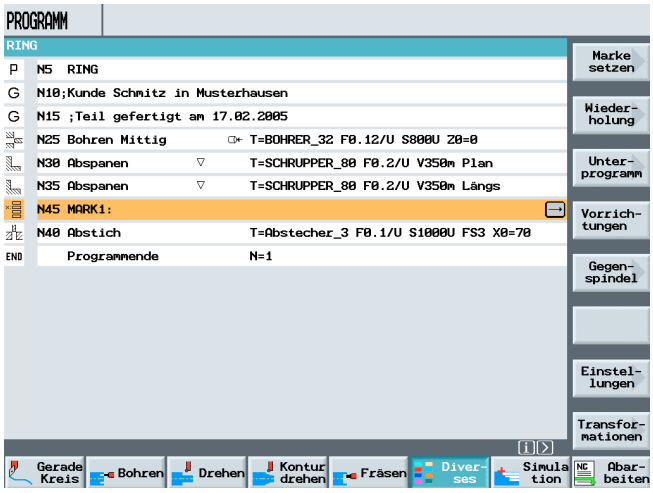


(any name) can be entered

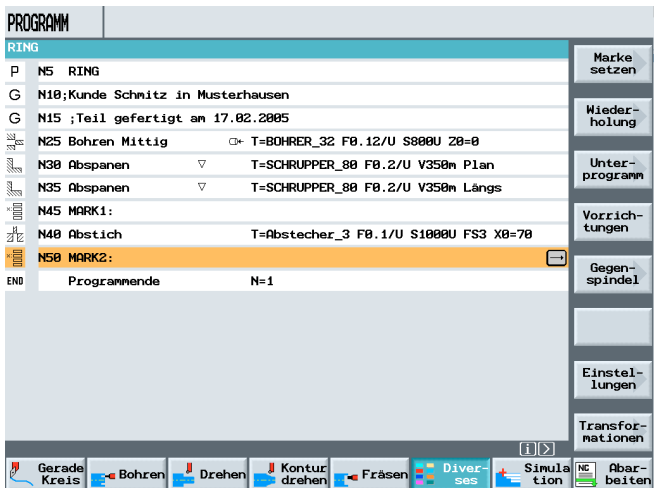
Through acceptance,



the mark is inserted above the cut-off.

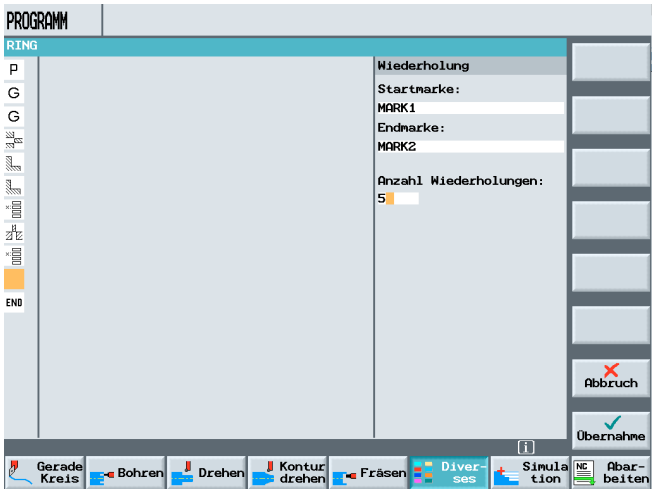


In this way, another mark (with a different name) is inserted below the cut-off.

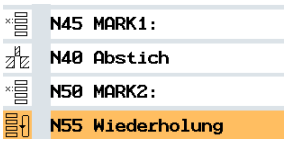


Next, after pressing the softkey Repeat

the number of repeats is entered in the input field

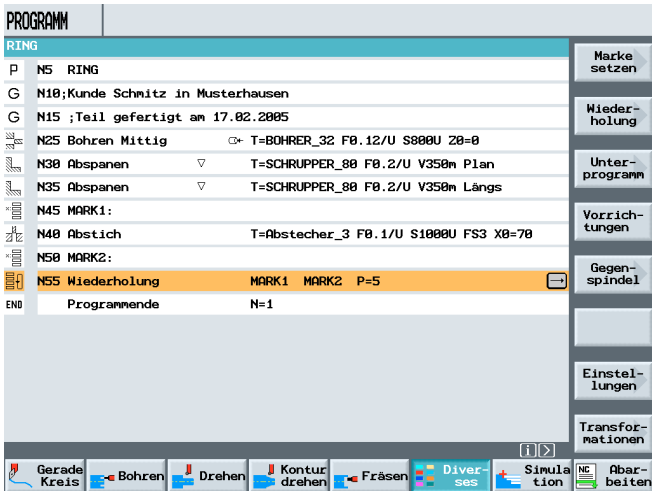


and accepted into the machining plan.



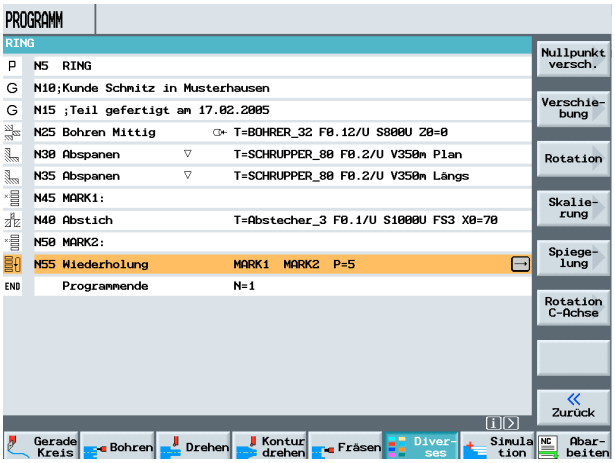
Finally, a shift has to be defined within the program loop.

To this end, a shift is inserted below the cut-off.

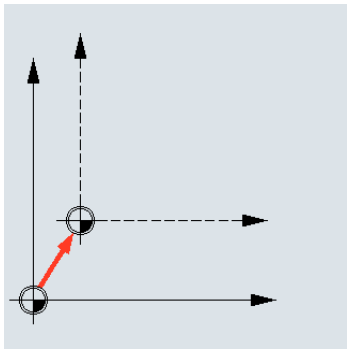


By pressing the softkey

and

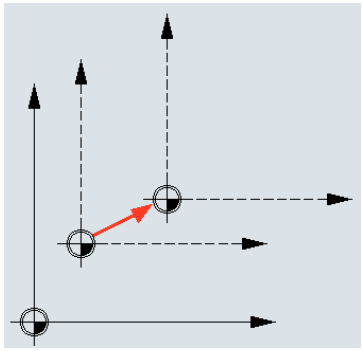


shifts can be made in this screen form



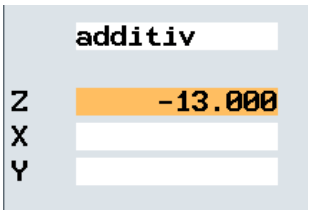
in reference to the workpiece zero point

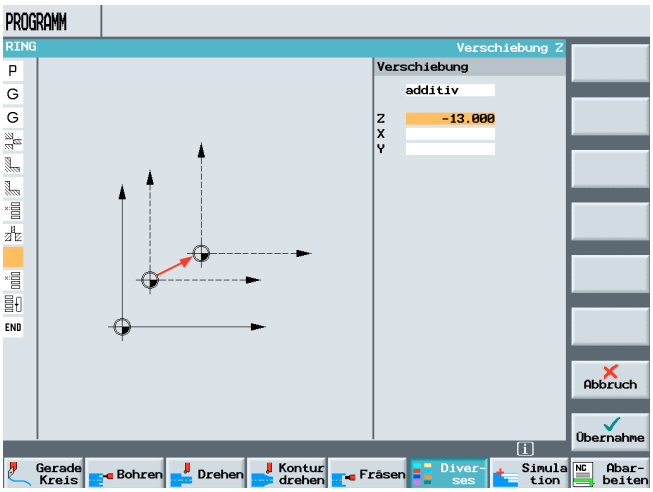
or



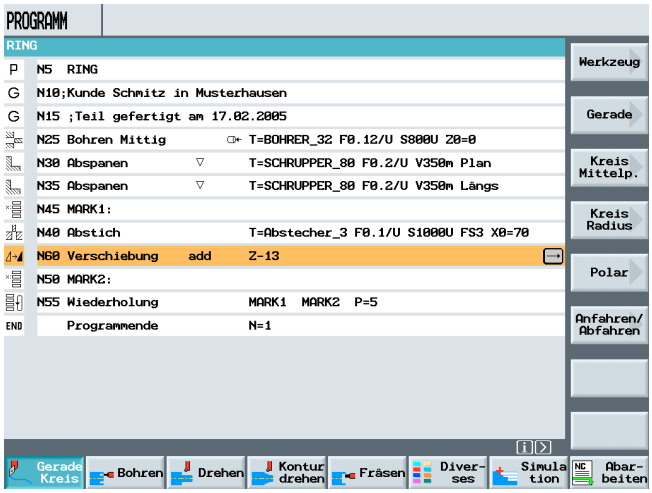
in reference to the last shift.

After entering the value



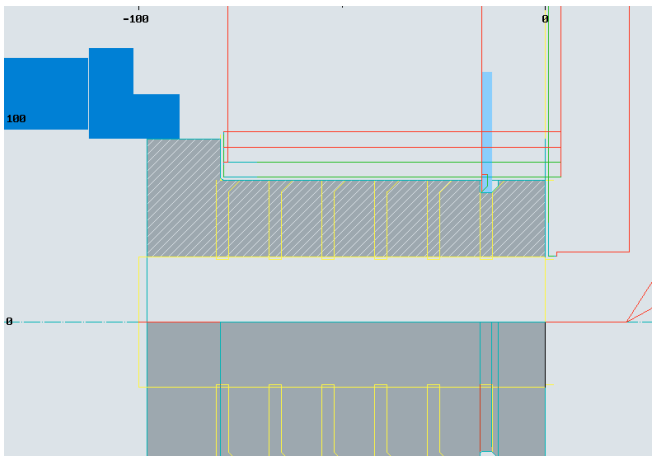


and accepting it into the machining plan,



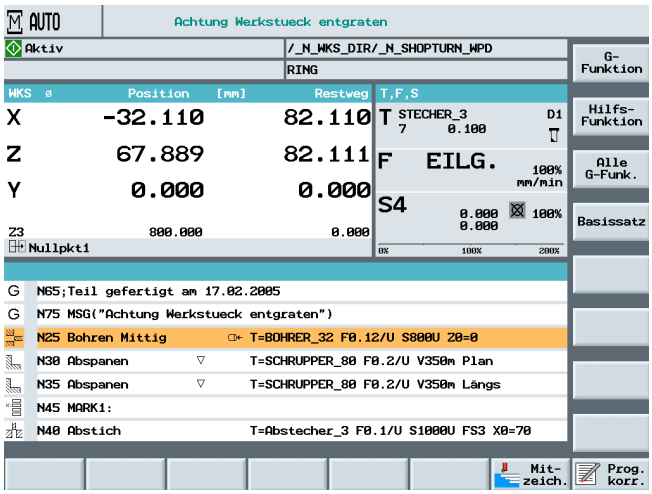
the program is done.

Six rings are manufactured.



In reference to the program example “Ring“, a workpiece counter is programmed using R-parameters.

From the software perspective, it may be advisable to program message texts that are displayed automatically when processing a program in the automation area.



Message texts are programmed as follows.

The text is displayed during machining until the end of the program.

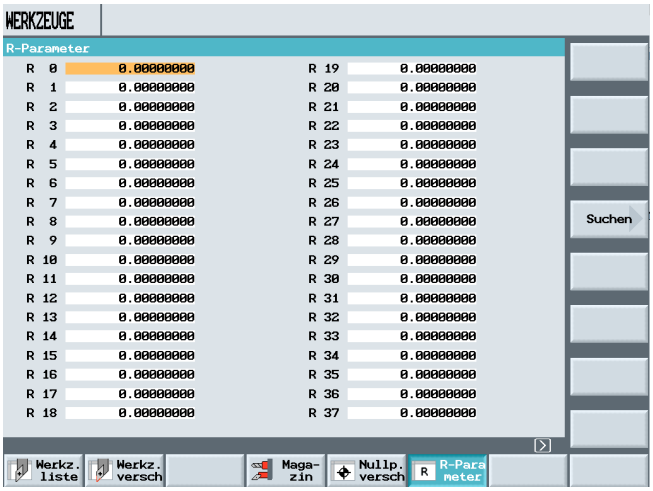
If the text is not to be displayed permanently during machining, it is deleted as follows.

R-parameters are variables that are available to the programmer.

By pressing the softkey

and

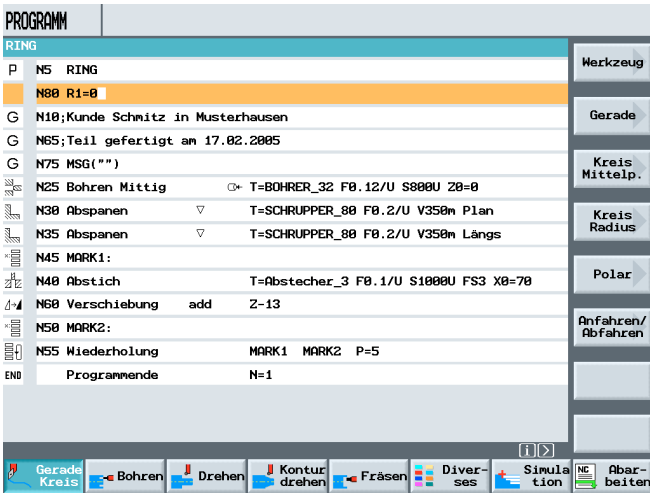
, the list of the available parameters is displayed.

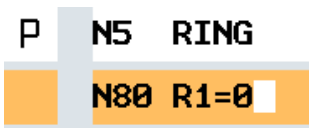


Under ShopTurn, it is not possible to replace the parameter values with R-parameters. This is possible only for standard cycles under standard programming.

In the program example “Ring“, the current machining operation is to be displayed as message text during machining.

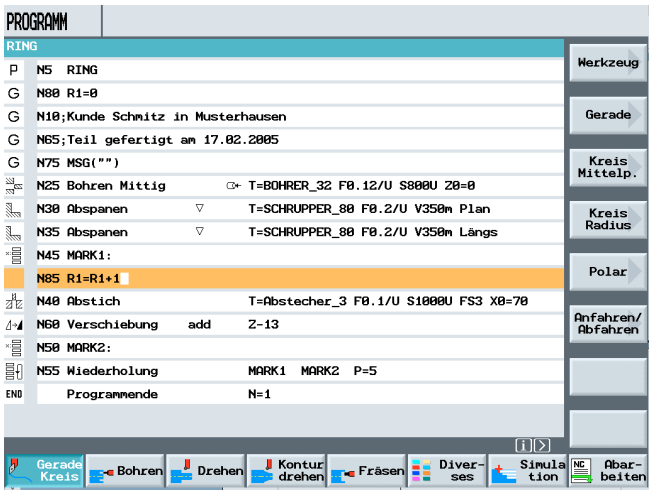
First, we set the variable R1 to zero at the program start.





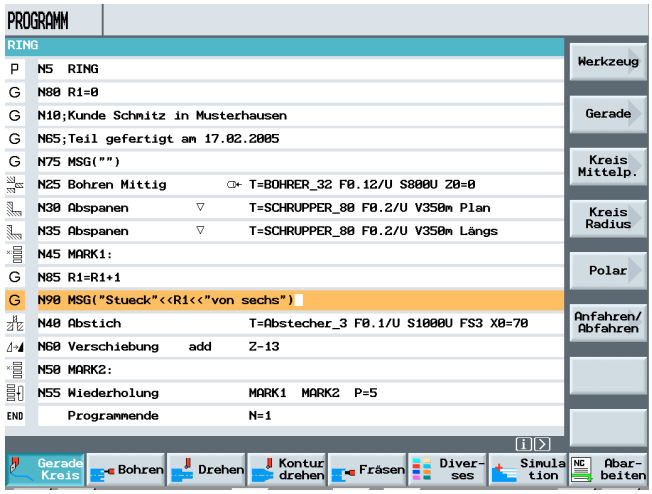
Within the loop, the number “1“ is added to the variable R1 for each repetition.

```
N85 R1=R1+1
```



With the following program instruction, the value of variable R1 is visible during machining.

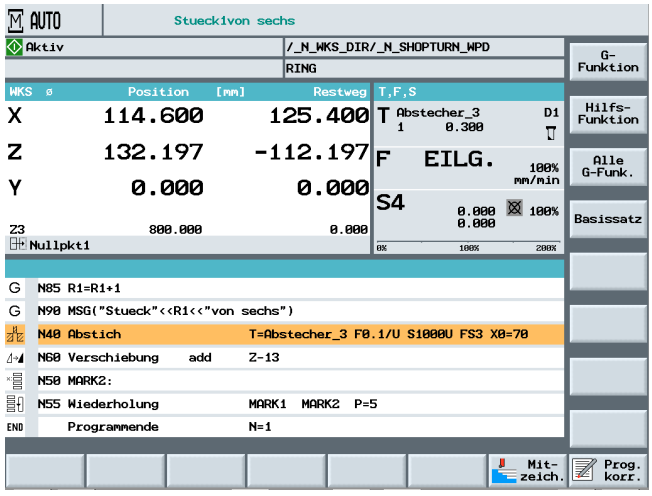
```
N90 MSG("Stueck"<<R1<<"von sechs")
```



In this case, the correct notation has to be adhered to for displaying the variable.

```
<<R1<<
```

When processing the program, only the current run of the loop is displayed as message text.



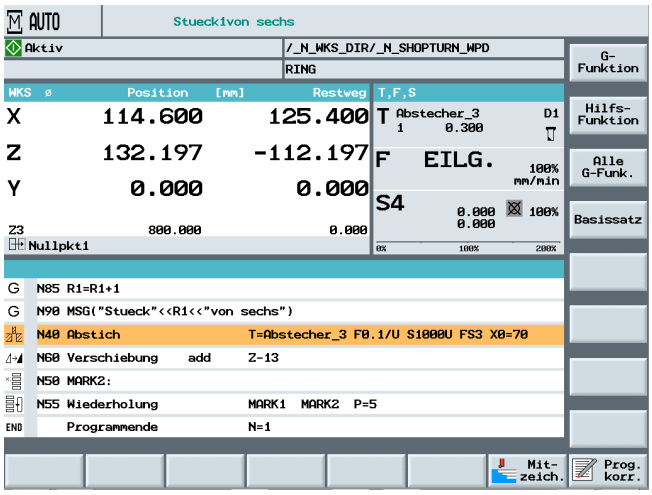

```
Stueck1von sechs
```

In ShopTurn, every subprogram is a main program and every main program is a subprogram.

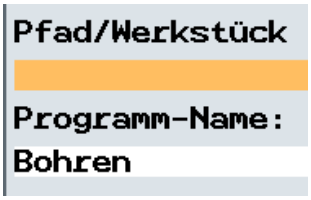
No differentiation is made regarding programming.

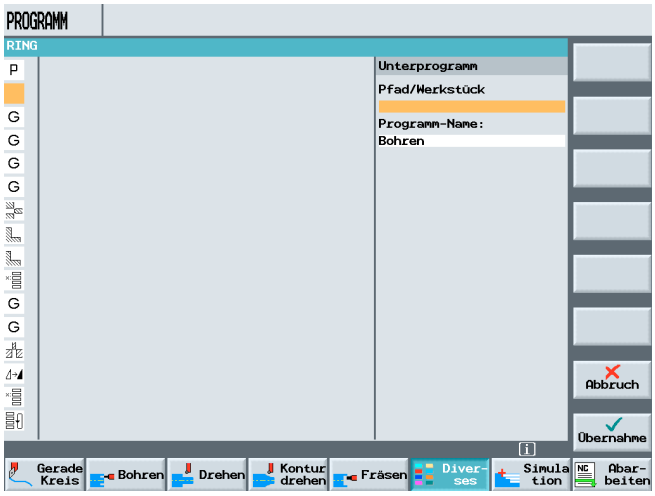
In our example, hole machining is to be performed in a subpogram.

By pressing the softkey



and

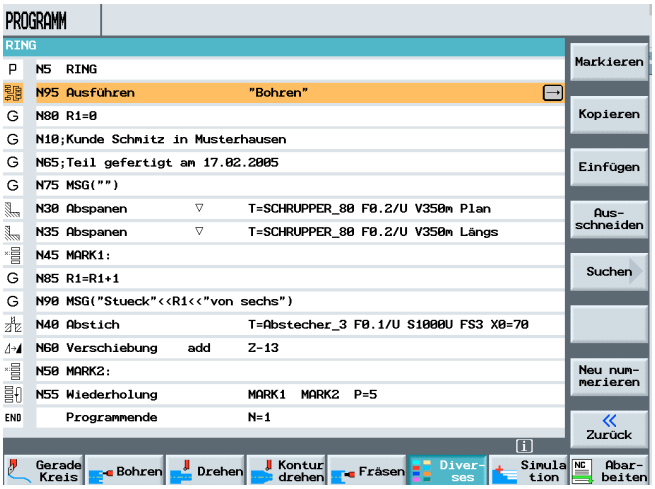




the program name that is to be called as subprogram is entered.
If the program is located in the same path as the main program, the name of the path does not have to be entered. The program extension, such as. “.mpf“ does not have to be entered either.

The program

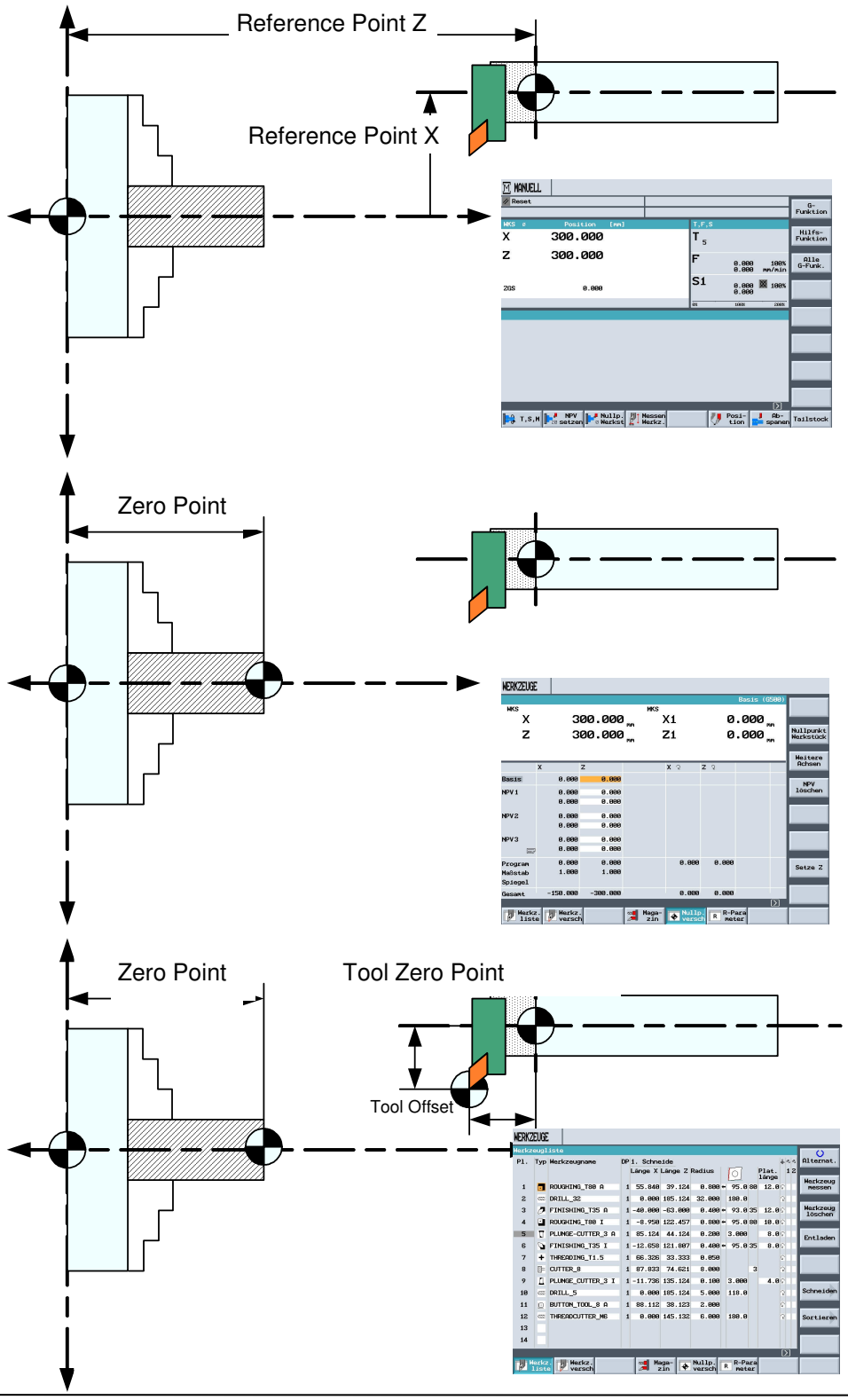
is now executed as subprogram in the main program “Ring“.



15 CNC Basics Turning

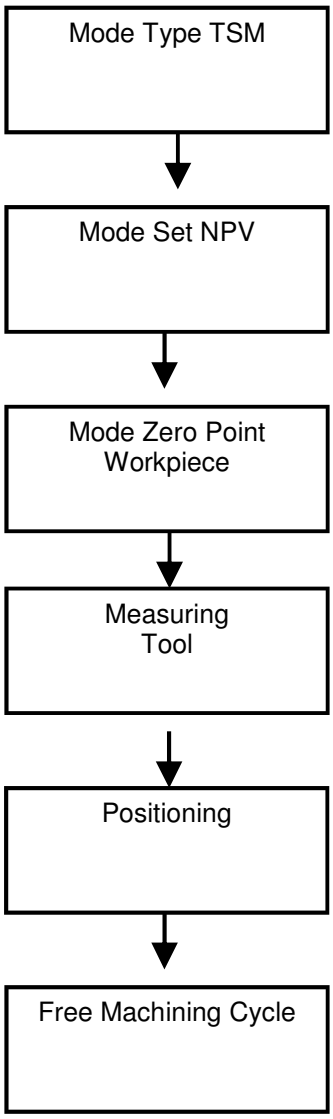
Regarding CNC machines, the required sequences for manufacturing a workpiece are stored in a CNC program. The calculations (compensations) between tool, workpiece and machine required for this are taken into account when processing an NC program.

The difference between a manual and a CNC machine consists of the logic operation of numerical values.



16 Manual Operating Area

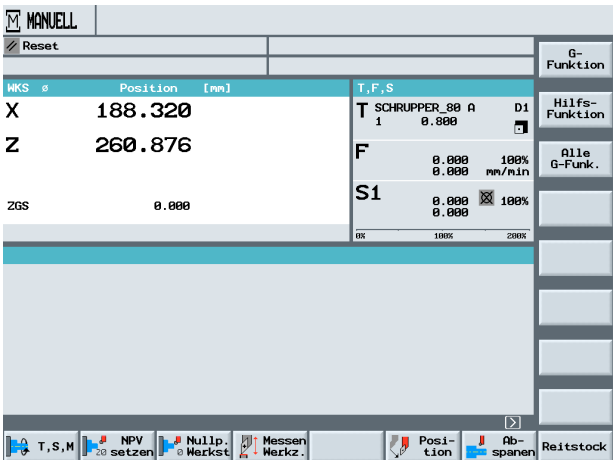
In this module, the individual function areas in the manual operating area are shown, and explained based on examples.



16.1 Mode TSM

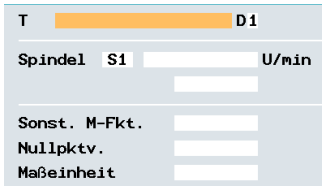
After starting Shop Turn, the operating area

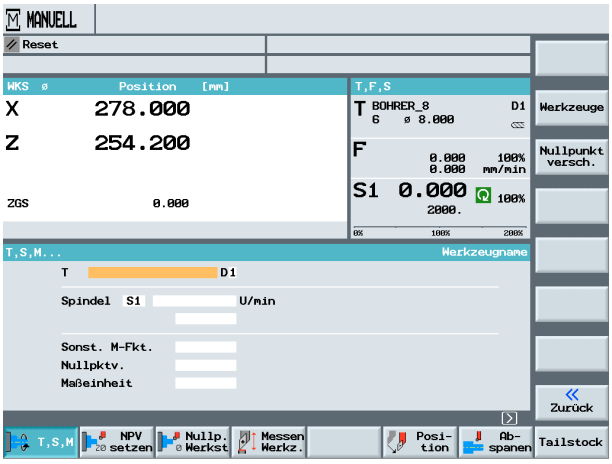
is active.



By pressing the softkey

the input window for manually operating the machine is displayed.

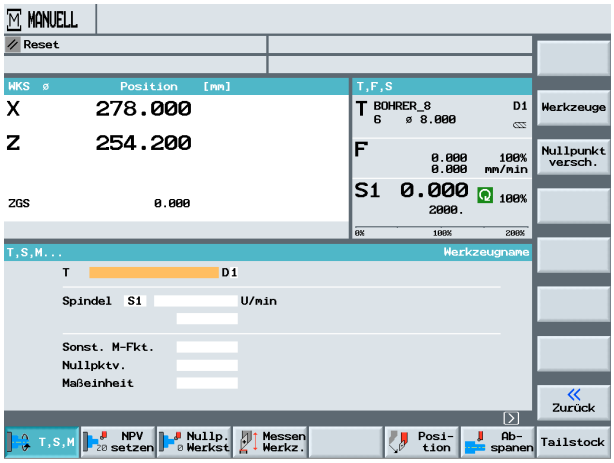




<<During spell check, “Freier Abspanzyklus Abschnitt 7 = free machining cycle, Section 7” appeared for this page, although this phrase is not visible on the page>>

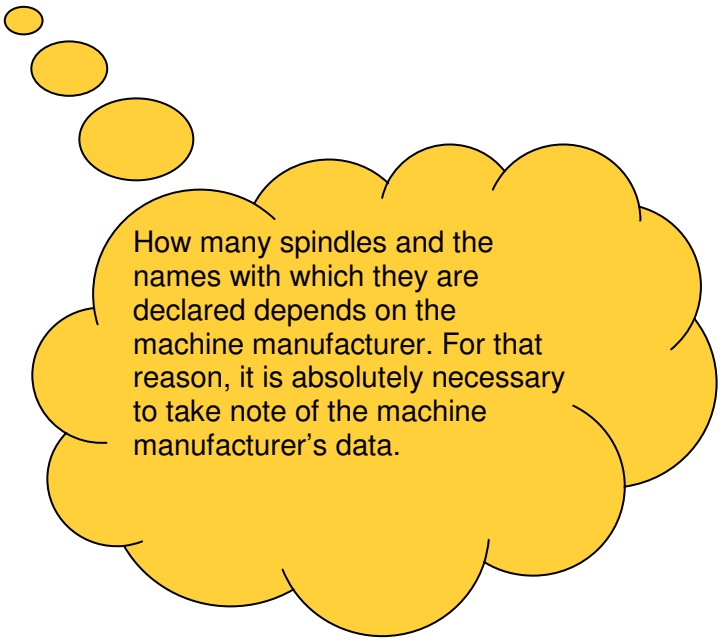
The tool for manual machining is called in the first input field.

Then, the spindle is selected.



Several spindles can be configured.

Example:



This page is blank

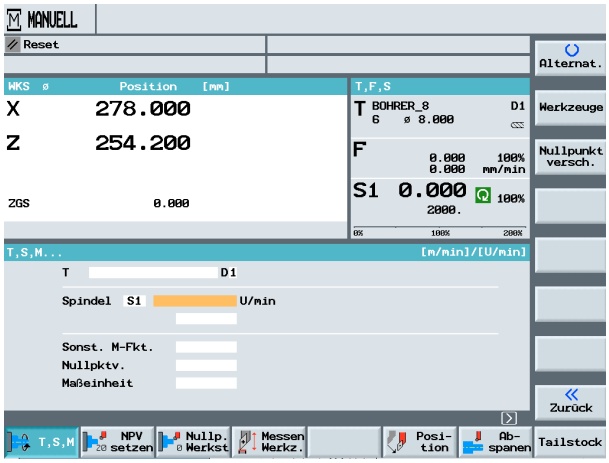
This page is blank

The spindle speed can be entered as

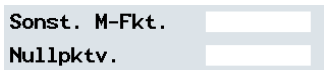
U/min

or as a constant cutting speed

m/min



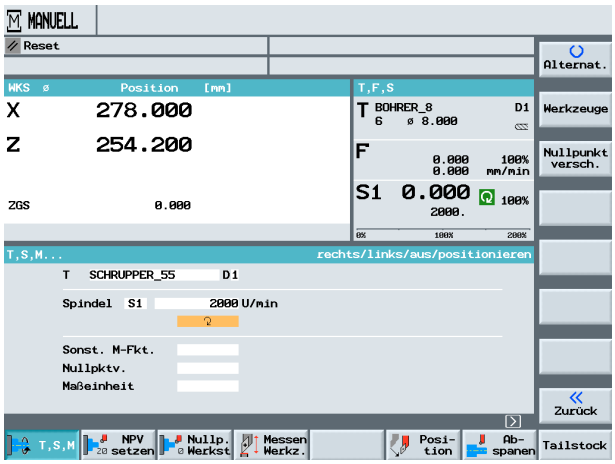
In the additional input fields, we can enter M-functions or zero shifts,



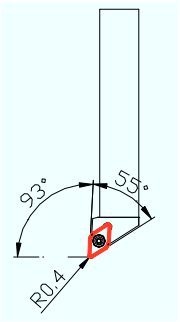
or we can select between the measurement units

mm

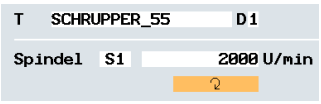
in



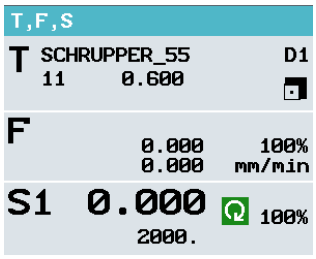
Example:

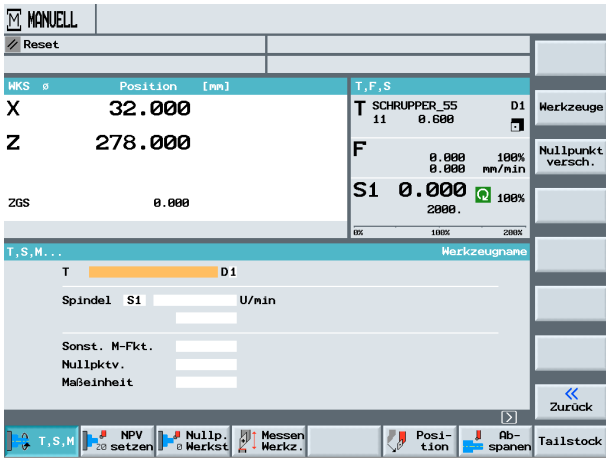


After calling the tool with the corresponding technology



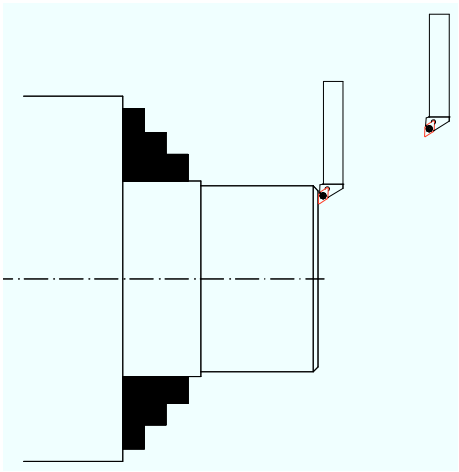
the tool with the entered technology data is activated by pressing the NC start button.





16.2 Mode Set ZO

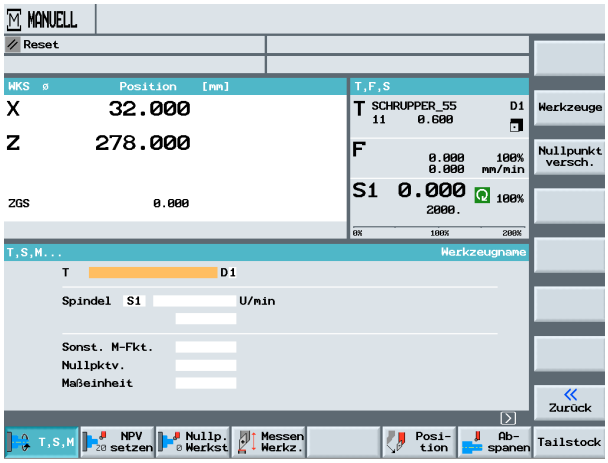
The mode "Set ZO“ is needed to compare the Z-axis with the workpiece.



Example:

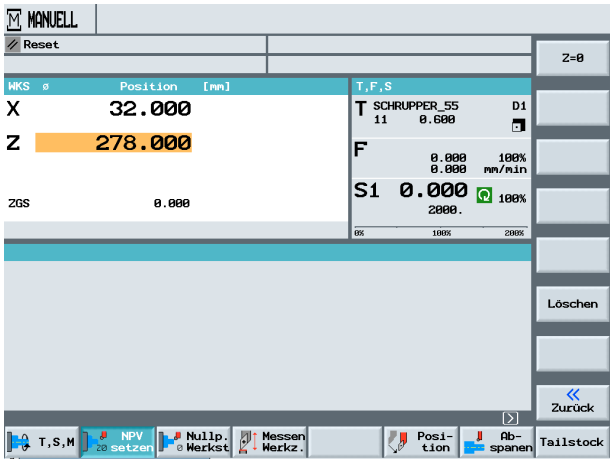
With a turning tool, the frontal face of a workpiece is scratched.

By pressing the softkey Set ZO



the current value for the Z-axis is colored

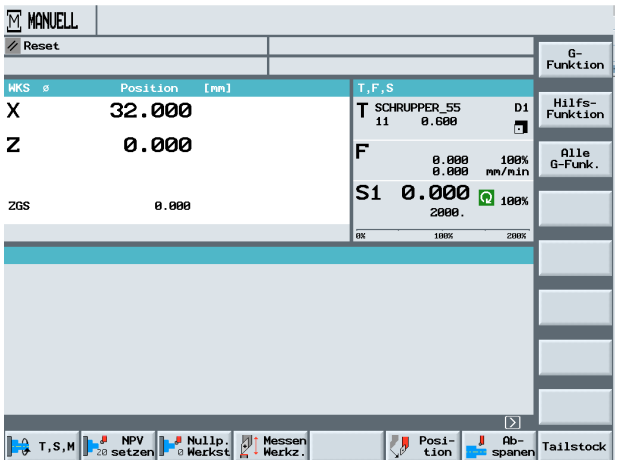
Z 278.000



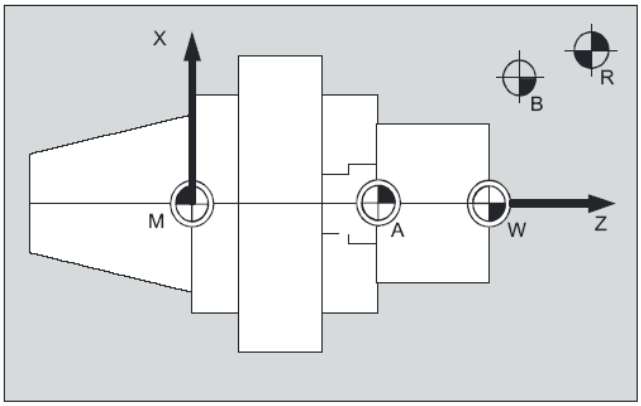
Here, by using the machine keyboard, any value can be entered, or the axis is set to zero by pressing the softkey

The basic shift G500 is now active.

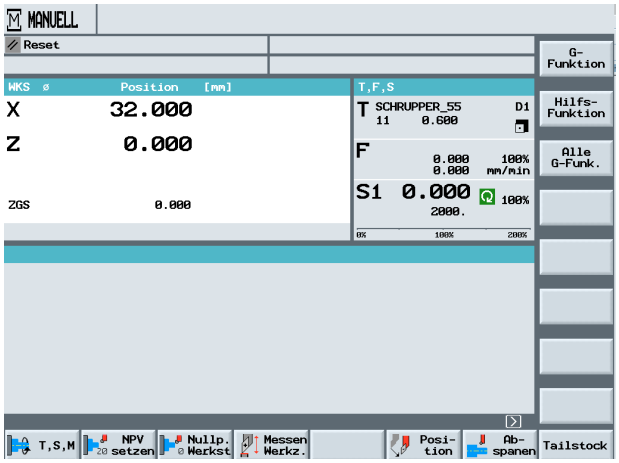
X	32.000
Z	0.000



16.3 Zero Point Workpiece



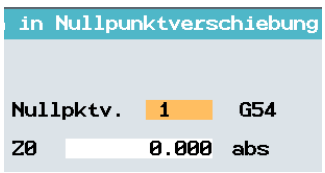
In this mode, the value for workpiece gauging can be written directly to the desired zero point shift .

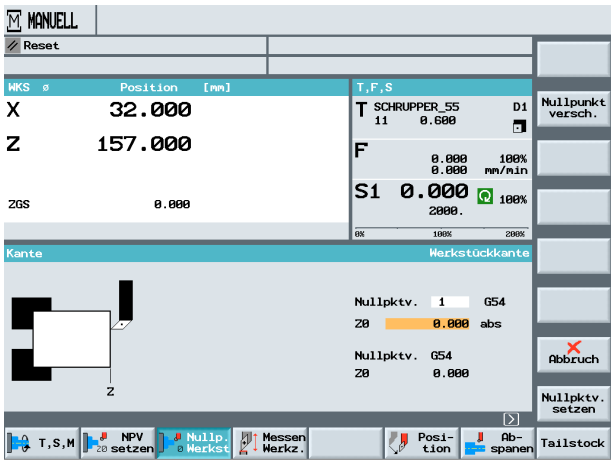


By pressing the softkey

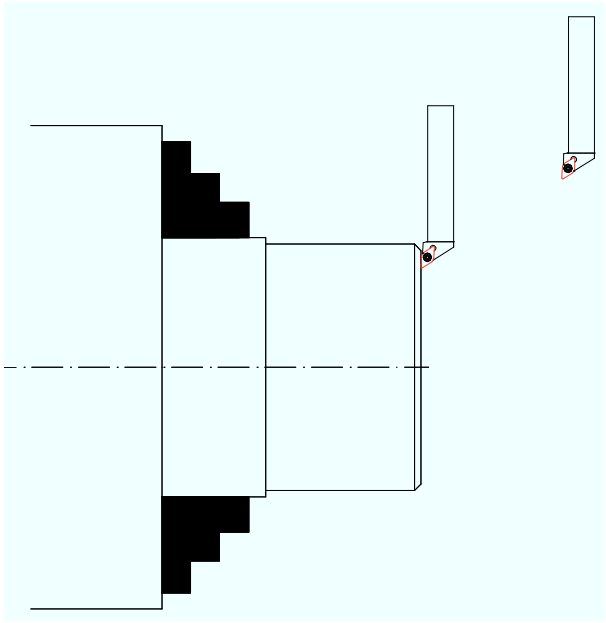
the corresponding input screen form is opened.

After selecting the zero point shift

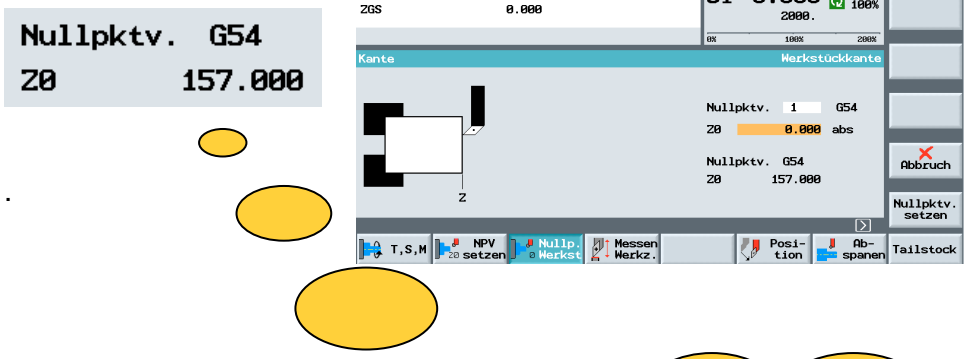




and “scratching“ the workpiece,



the workpiece zero point can be accepted directly into the selected zero point shift by pressing the softkey



Nullpktv. G54
Z0 157.000

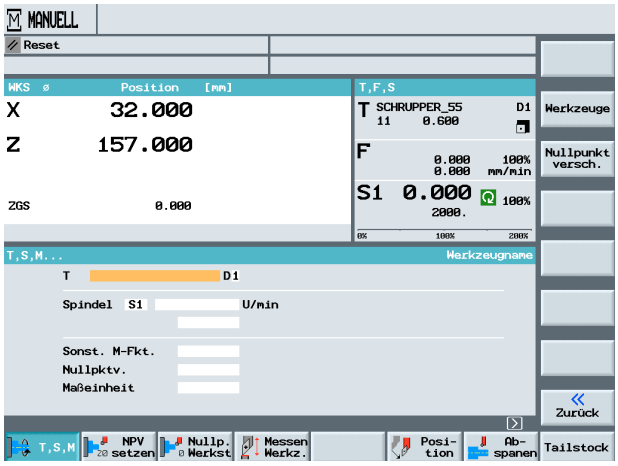
.

As a rule, the basic shift G500 at the three jaw chuck is located at the location that serves as limit stop for the raw parts. When “scratching“ the workpiece at the face, the value of the zero point shift thus corresponds to the length of the workpiece.

16.4 Measuring the Tool

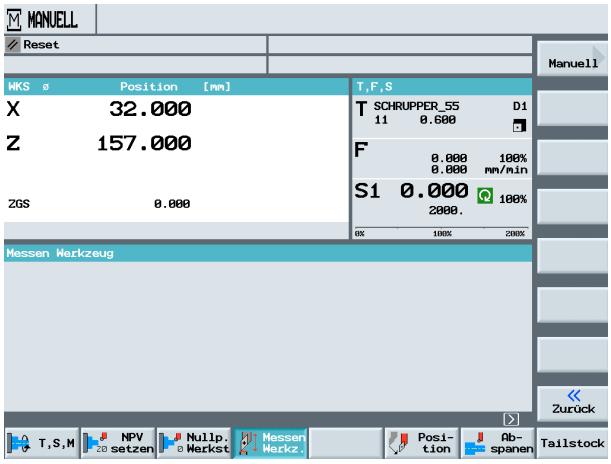
By pressing the softkey

in the operating area



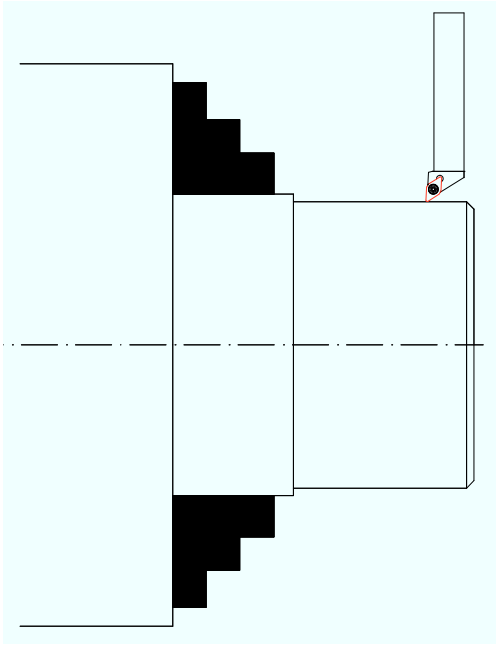
and pressing the softkey

,



This page is blank

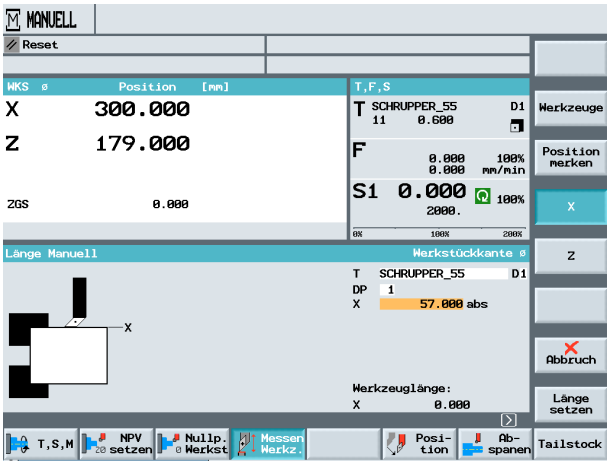
the tool can be measured manually.



After measuring has been selected in

,

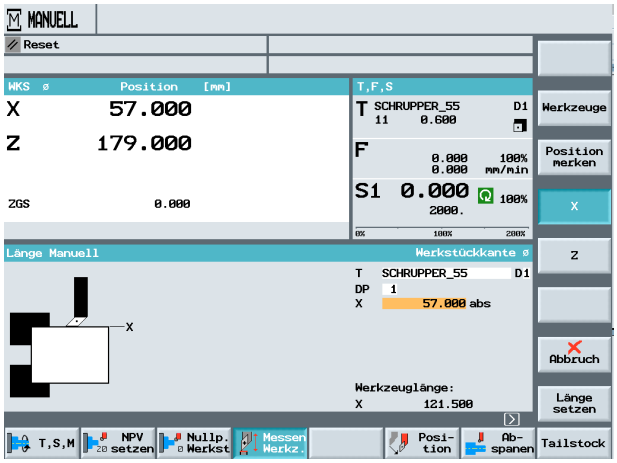
the screen form is displayed with the corresponding display.



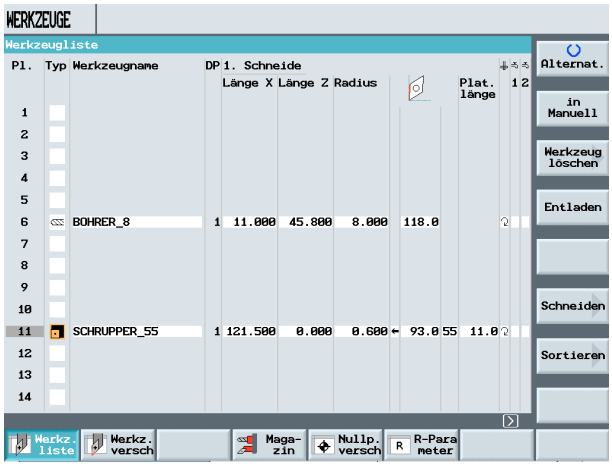
The workpiece diameter is entered in the input field.

By pressing the softkey Set Length

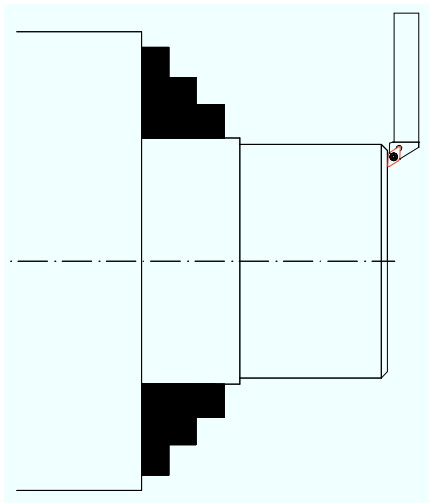
the tool is dimensioned in the X-direction.



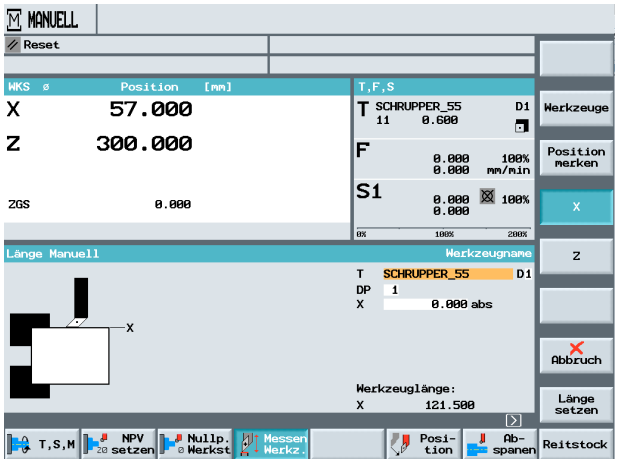
This value for the tool length is assigned to the measured tool in the tool list.



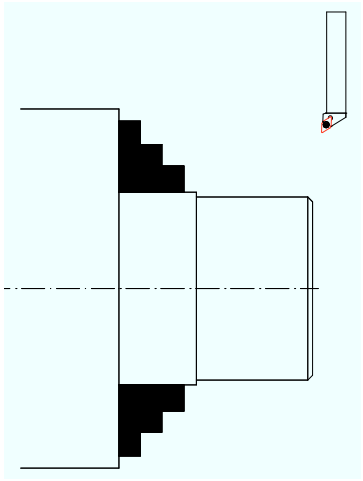
The process is repeated for tool gauging in the Z-direction.



By pressing the softkey



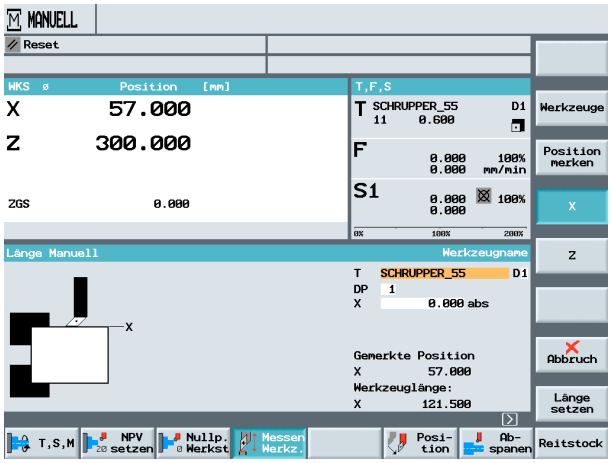
, after “scratching“ the tool can be moved as required in the working area.



The measured position is saved

Gemerkte Position	
X	57.000

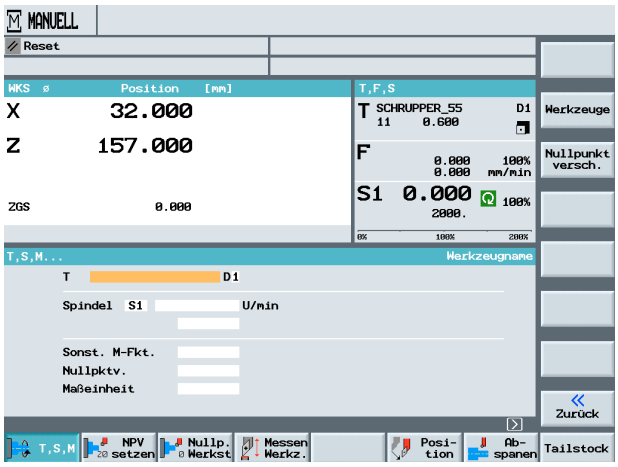
and is automatically balanced against the workpiece diameter that was entered.



16.5 Positioning with Feed or Rapid Feed

By pressing the softkey

in the operating area



the target position is entered in the input fields.

X	100.000 abs
Z	100.000 abs

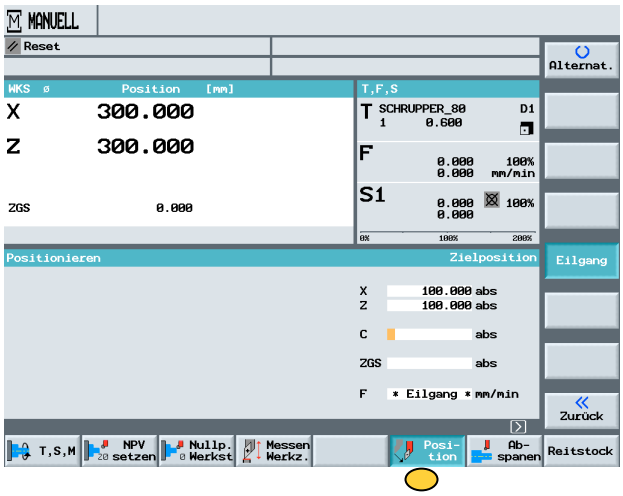
The travel movement can be in feed, or

F	0.200 mm/U

in rapid feed

F	* Eilgang * mm/min

.



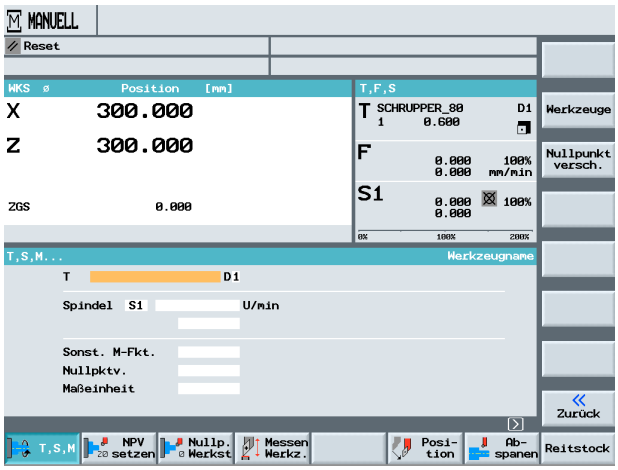
There is no collision check during travel to the target position.

16.6 Free Machining Cycle

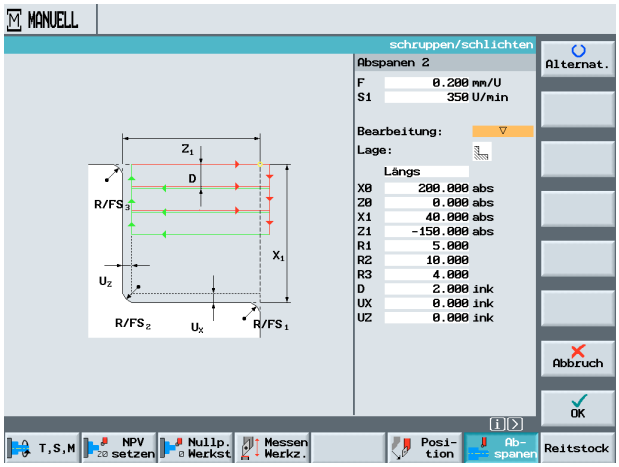
By pressing the softkey

in the operating area

the input field for the free machining cycle is selected.



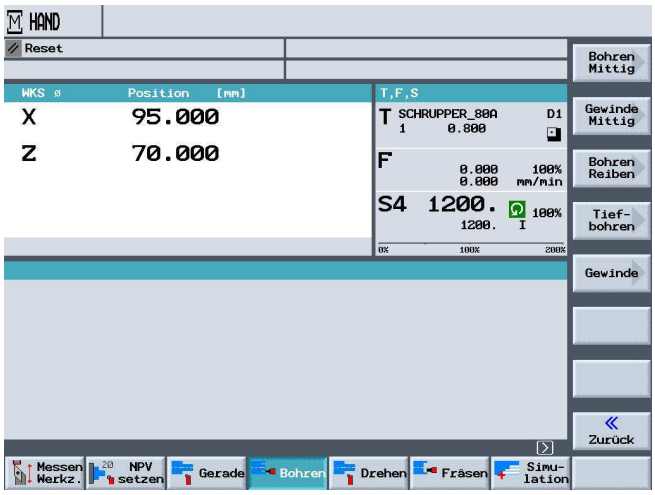
This machining cycle is described in greater detail in the “Programming Example for the Machining Cycle“.



17 Manual Functions ShopTurn

In this module, the manual functions are described.

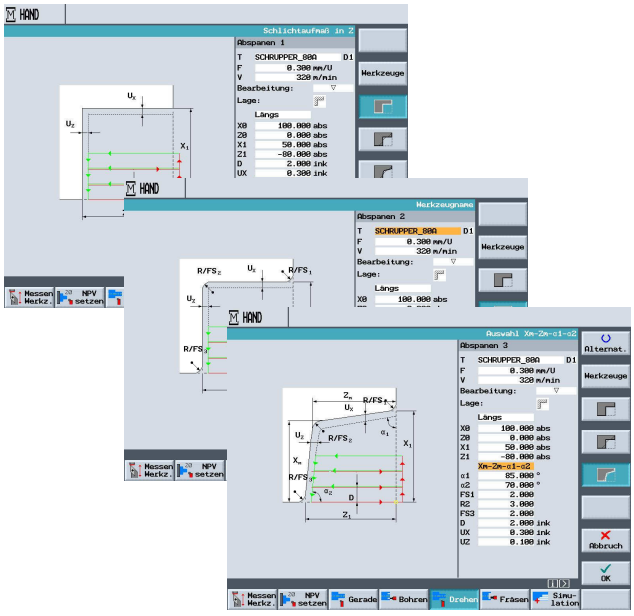
Under the “Manual“ mode in ShopTurn, the following cycles can be utilized.



By pressing the softkey Turning

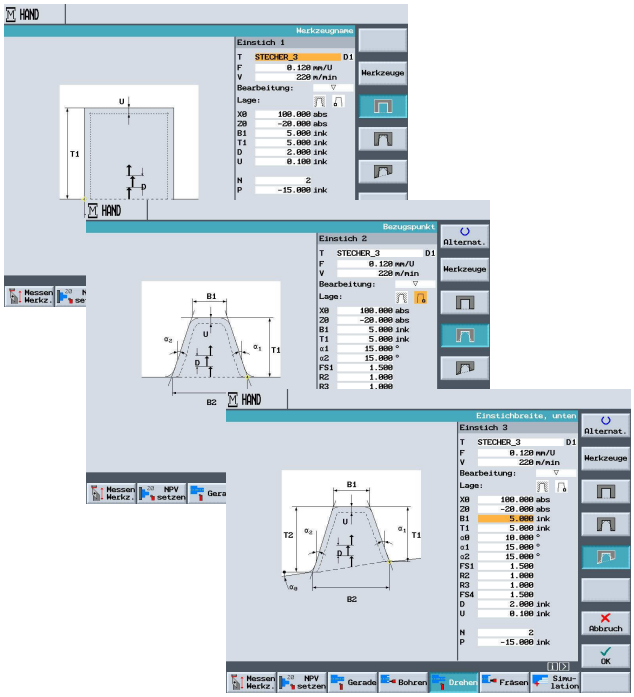
, the following cycles are available.

The machining cycles for face turning and longitudinal turning.

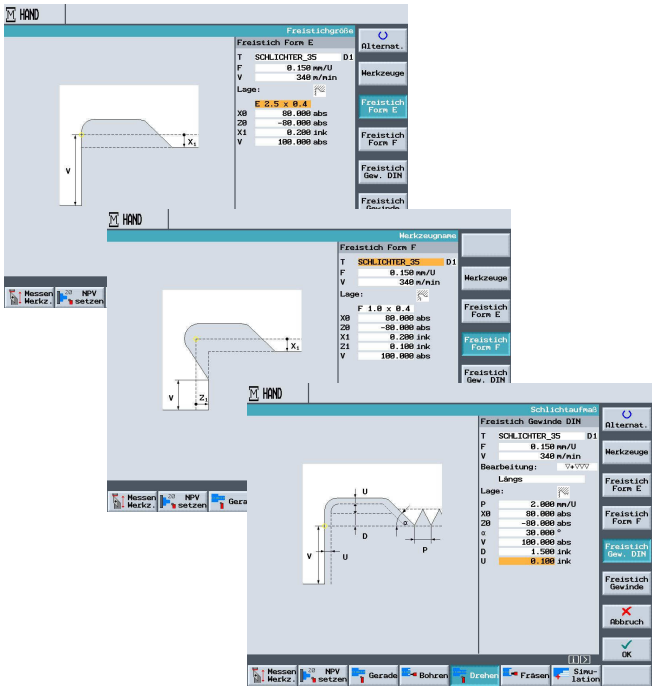


Each of the cycles described here can be processed directly after entering the values. No program need to be generated.

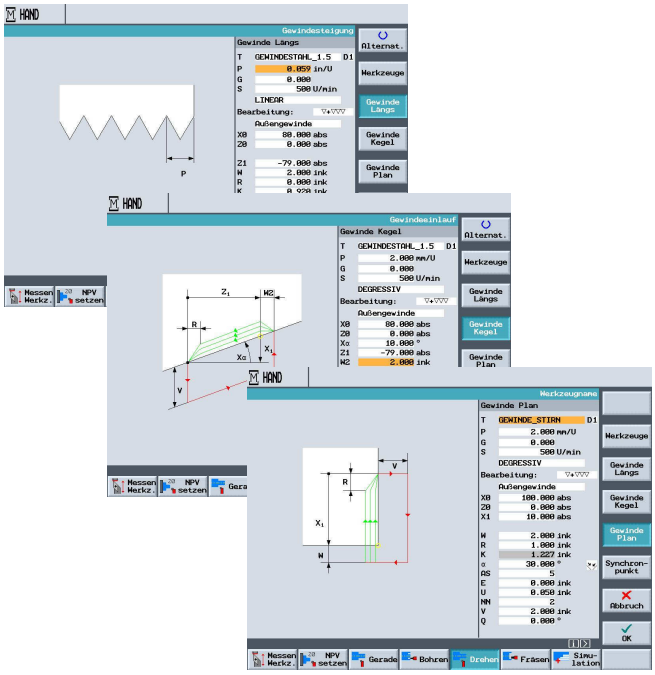
The grooving cycles,



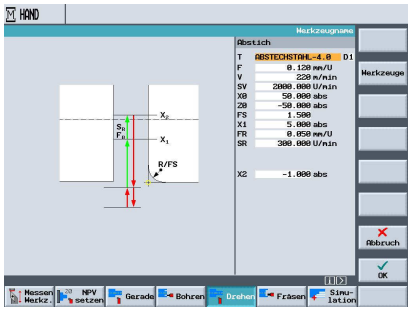
the undercut cycles,



the thread cycles,

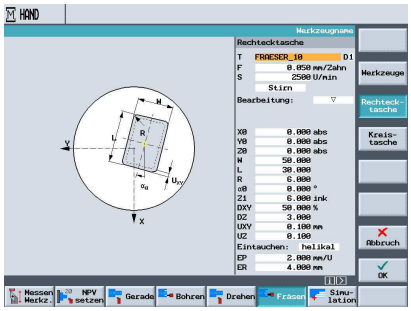


the cut-off cycle.

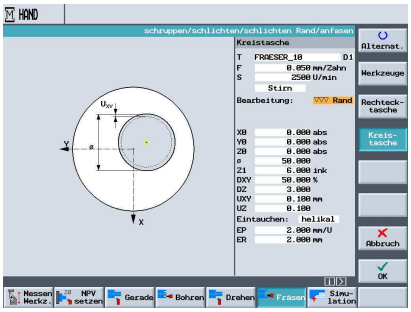


In the “manual“ mode, the following milling cycles are provided:

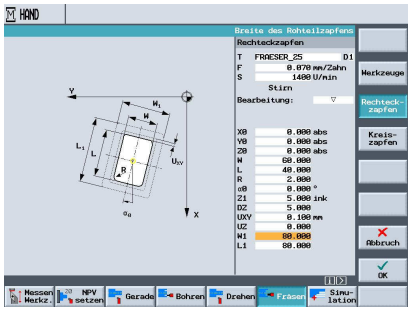
Reactangular pocket



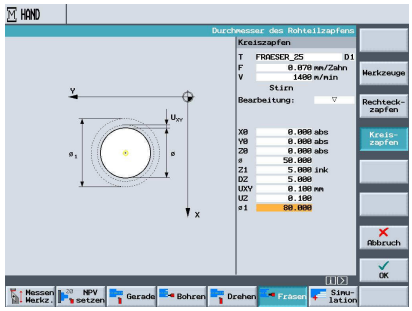
Circular pocket



Rectangular spigot



Circular spigot,

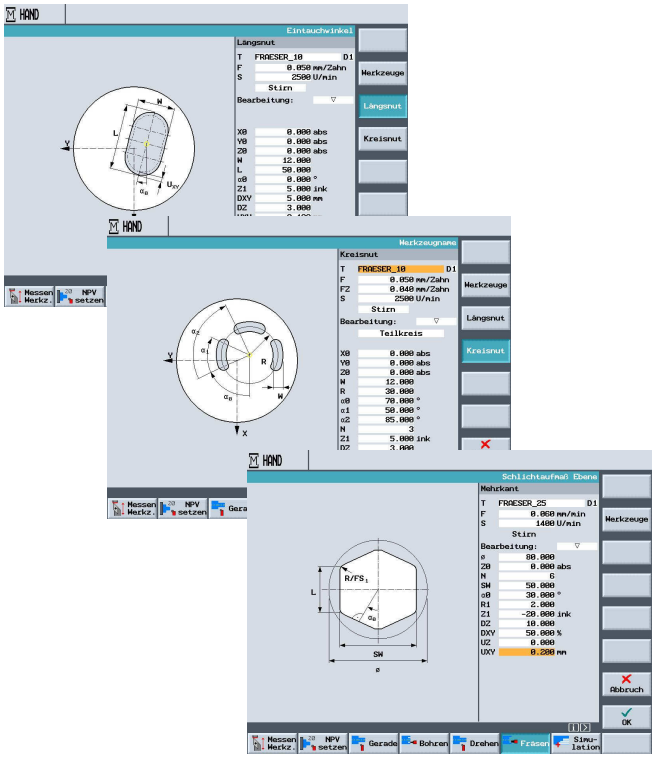


Longitudinal groove

Circular groove

and

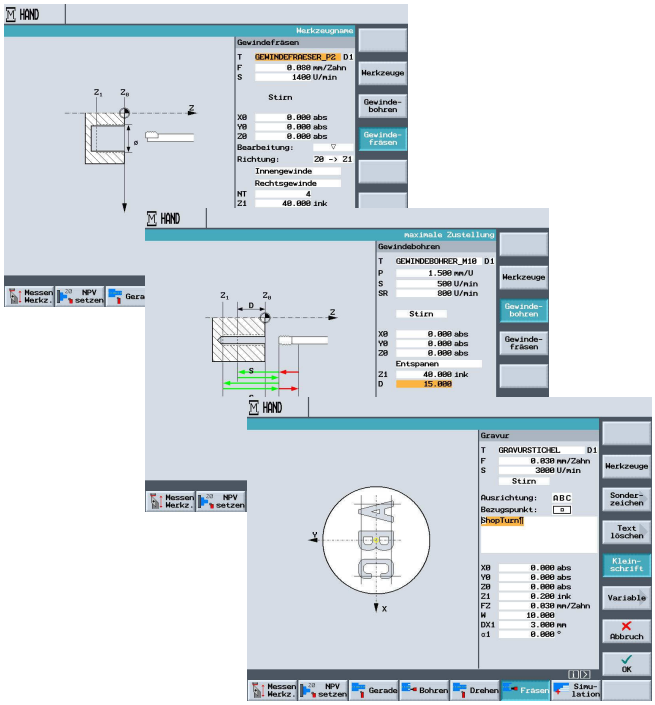
Multi-edge possible.



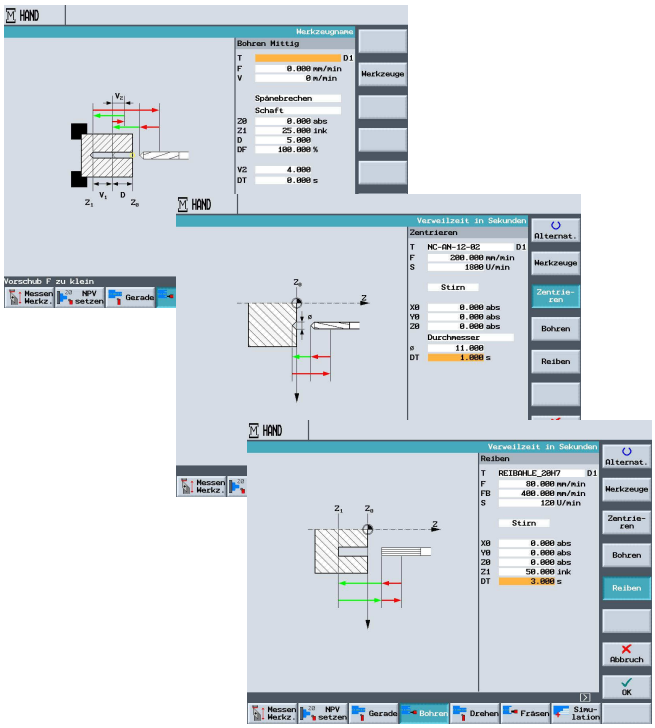
Also the thread milling cycles,

tapping

and the engraving cycle is available in the “manual“ mode.
.



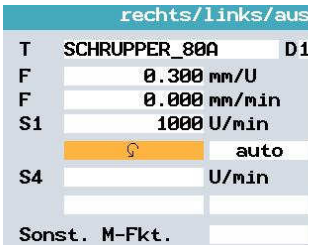
In addition, centered – off centered drilling and reaming is possible.

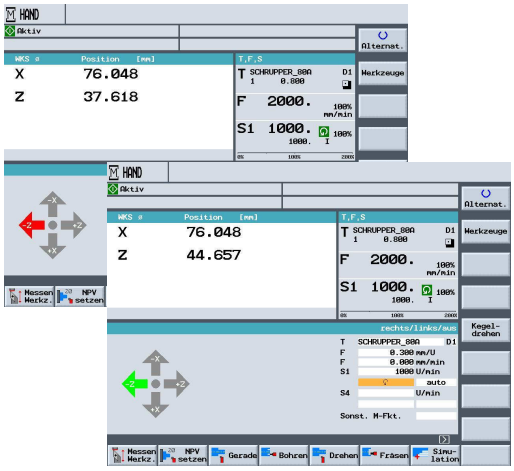


All cycles described here are identical with the cycles used in a ShopTurn program.

In the “manual“ mode, it is also possible to operate the machine manually.

After selecting the tool and the technology,





the machine runs with the technology entered.

The motion of individual or several axes -for example, for taper turning- is shown graphically. Feed “green“; rapid feed “red“.

In this module, DIN programming under ShopTurn is explained.

Example DIN
Programming

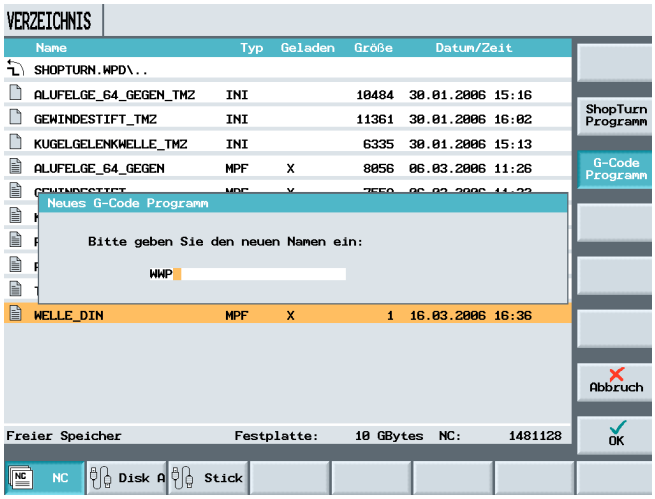
18 DIN Programming under ShopTurn

In this module, programming in DIN under ShopTurn is explained, using on an example.

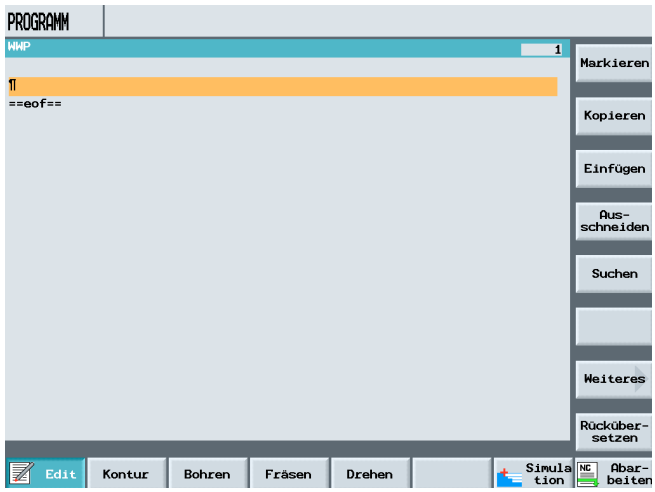
Under ShopTurn, we can program according to DIN (DIN 66025)
as well as with the cycles of the 840D.

When setting up a new program, the DIN Editor is opened under ShopTurn

by pressing the softkey



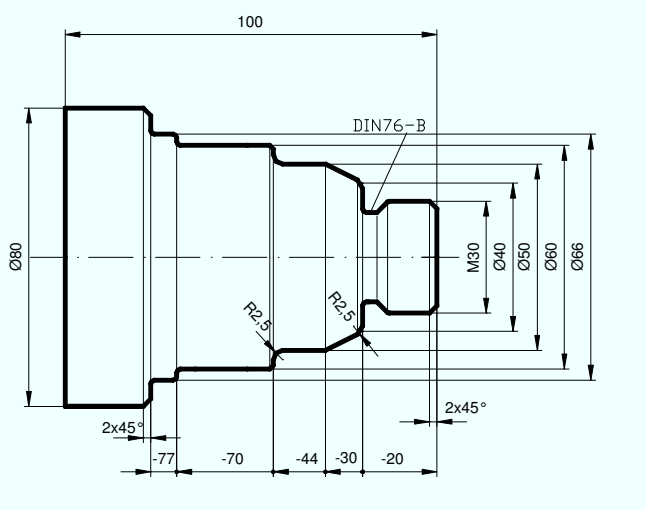
The program example is generated by using the standard cycles.



The following G-functions are used to generate the program.

G18- Plane selection XZ plane (turning plane)
G41- Tool radius correction from the left
G54- Activation of the first zero point shift
G90- Programming absolute dimensions
G95- Feed rate in mm/rev
G96- Constant cutting speed

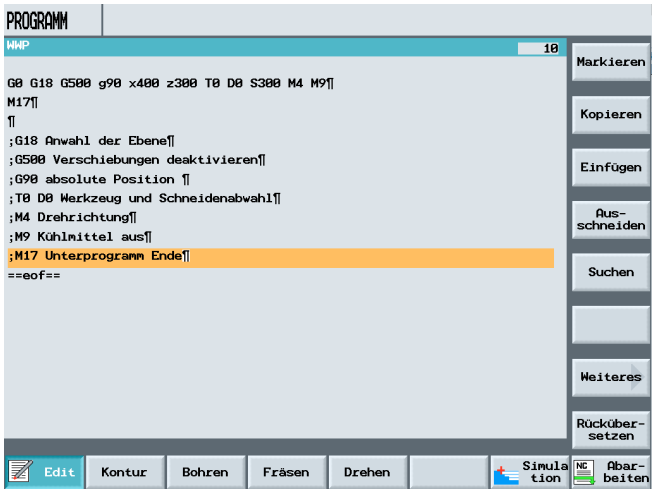
The following workpiece is to be programmed.



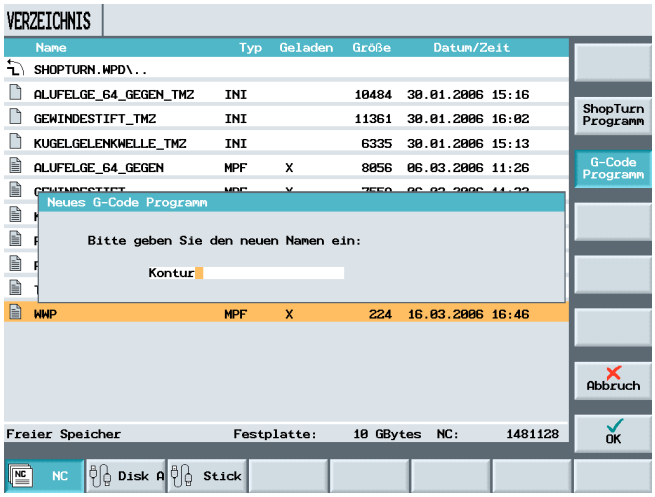
With the program “TCP“, the tool change point is defined that is approached after machining.

The approach and return cycles are contained in ShopTurn cycles, not in the standard cycles.

After entering the TCP



a new program with the name Contour is generated.

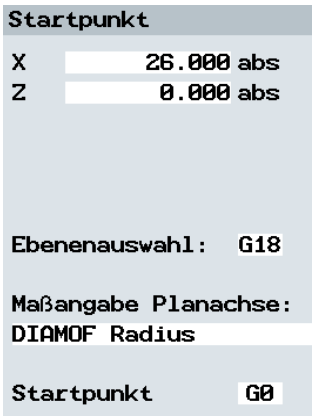


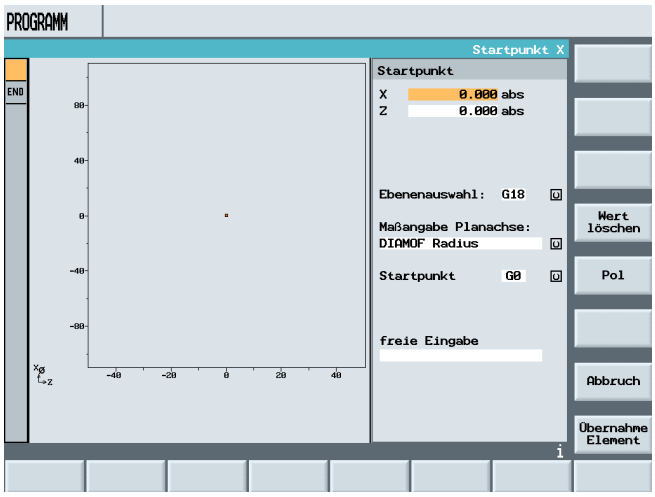
By pressing the softkey

and

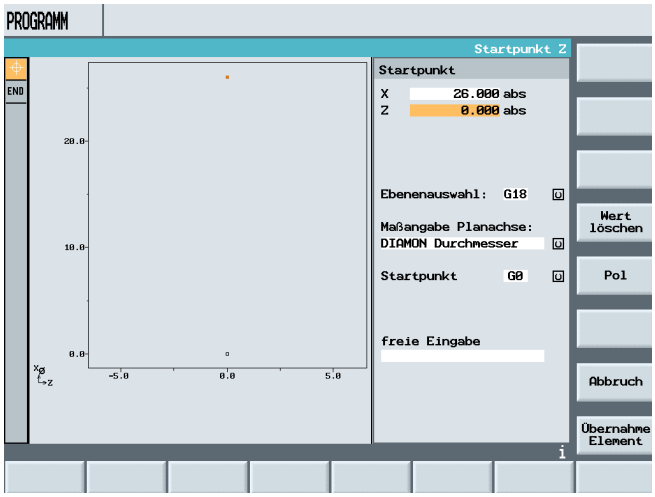
the contour calculator for standard programming is opened.

After selecting the starting point

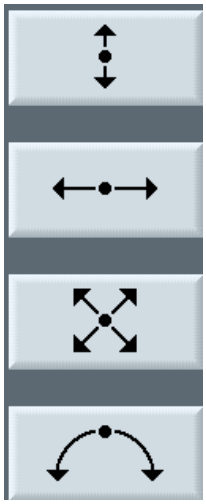




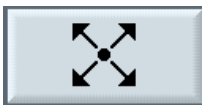
and pressing the softkey

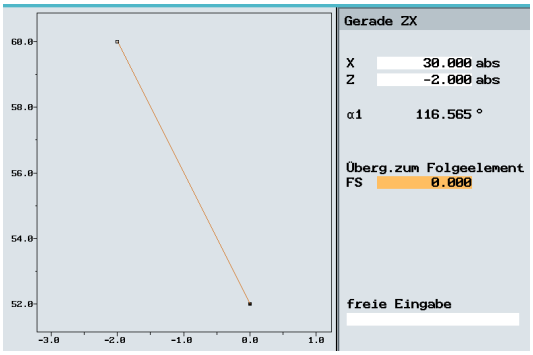


the workpiece contour (refer to the drawing) is described by using -as in the case of the contour calculator of ShopTurn- the softkeys below.

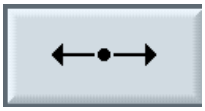


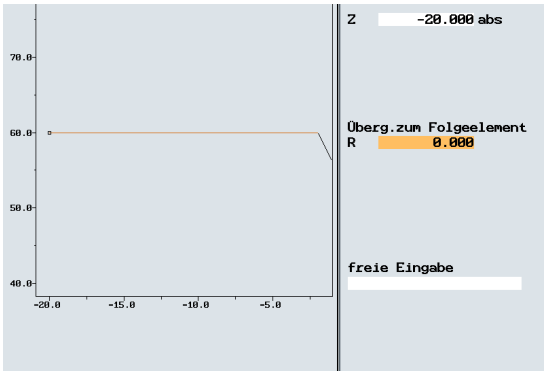
Element 1



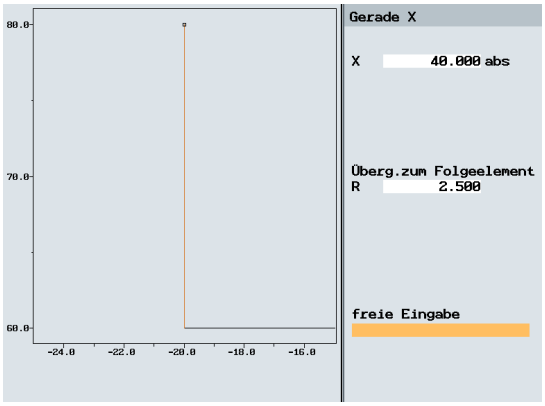


Element2

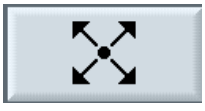


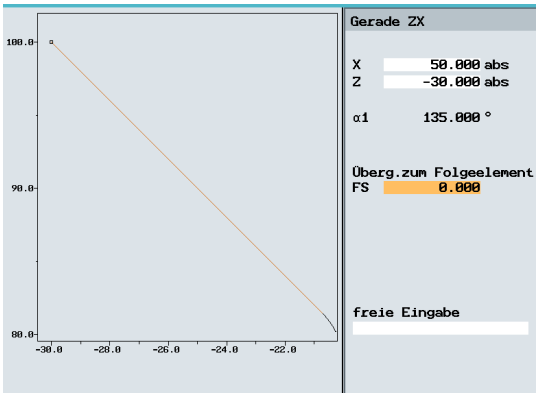


Element 3

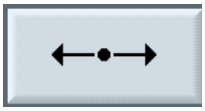


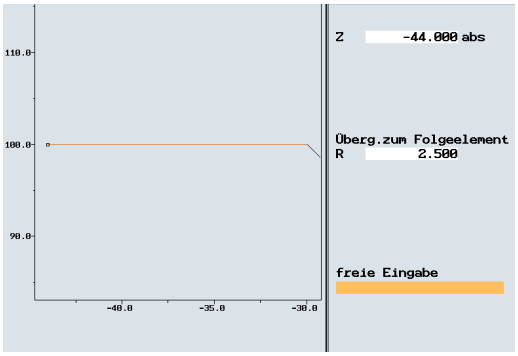
Element 4



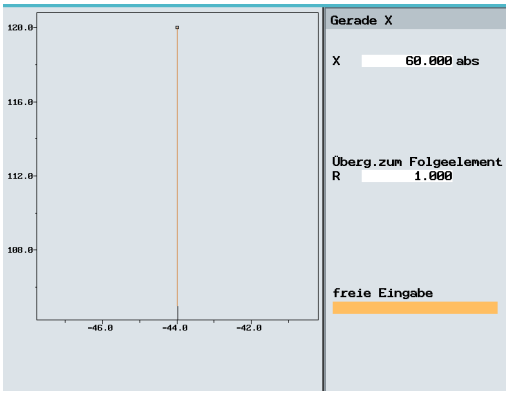


Element 5

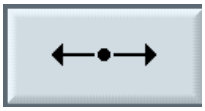


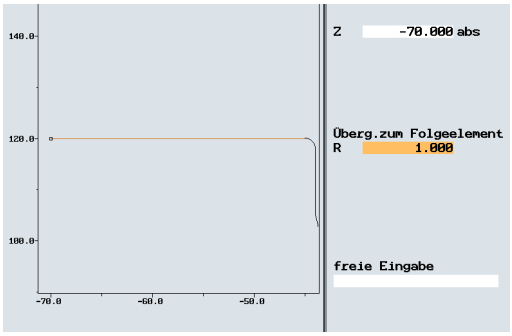


Element 6

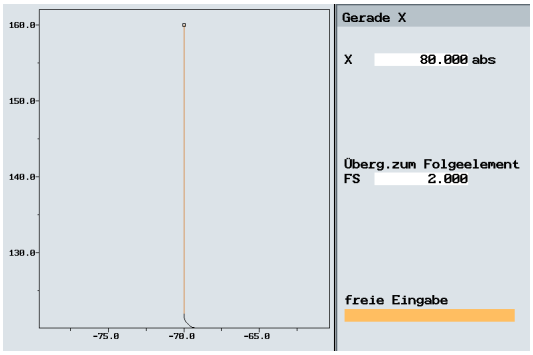


Element 7

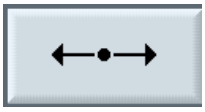


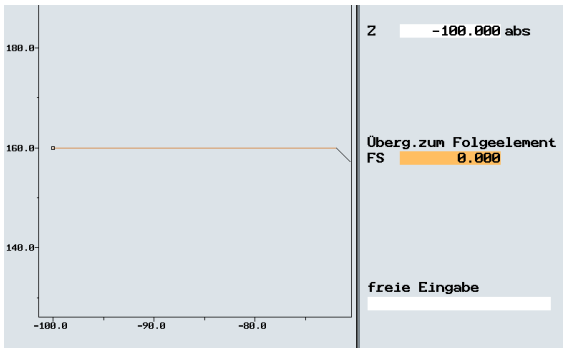


Element 8

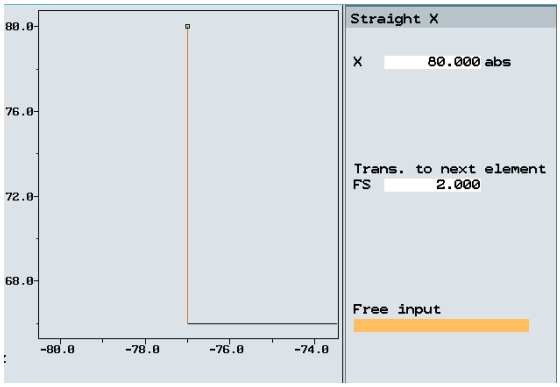


Element 9

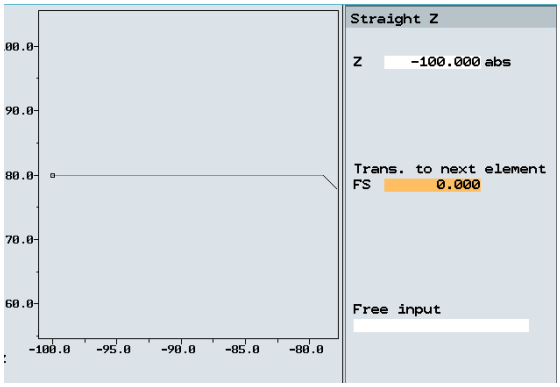




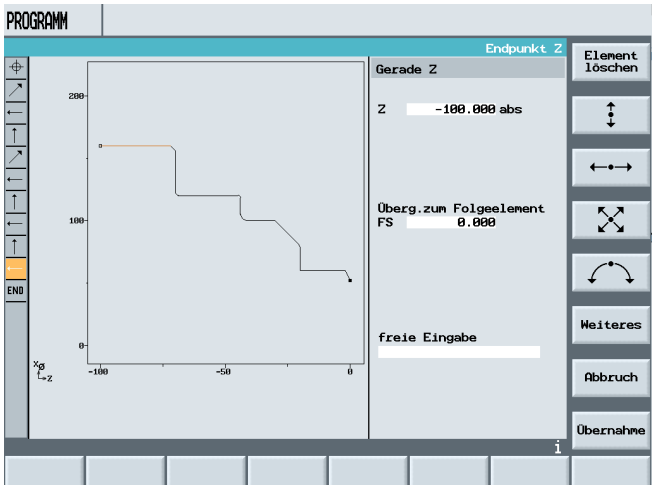
Element 10



Element 11

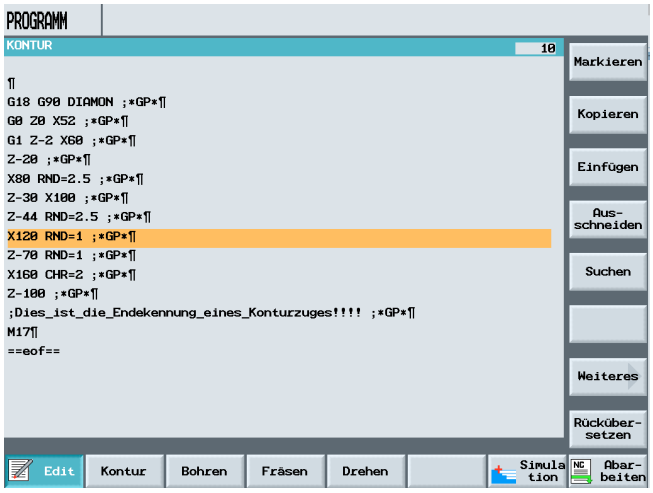


The contour is completed. Since this contour is called later as a subprogram in another program, the instruction M17 is used at the program end for “End of subprogram“.

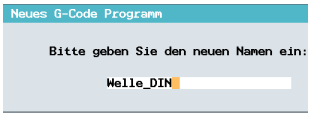


By pressing the softkey

it is accepted in the editor.



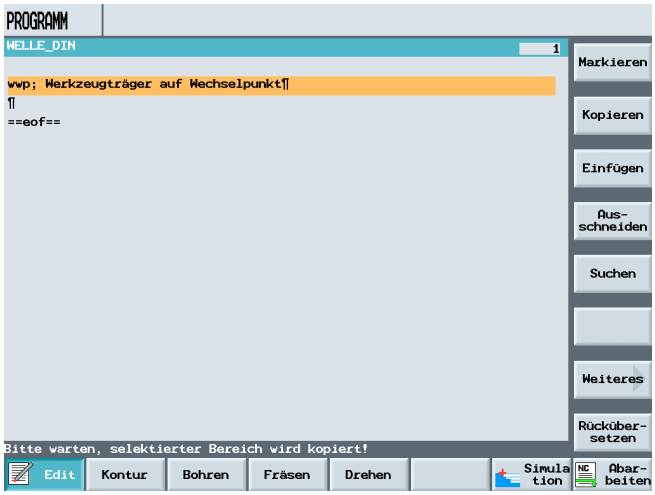
Now, another program with the name



is generated.

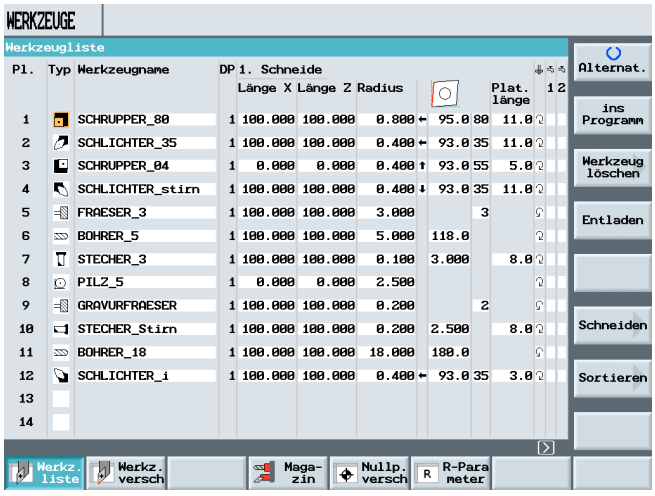
Next,

we travel to the tool change point by calling the program "TCP“..



The tool is called by pressing the softkey

and



For programming the tools, ShopTurn tool management is provided.

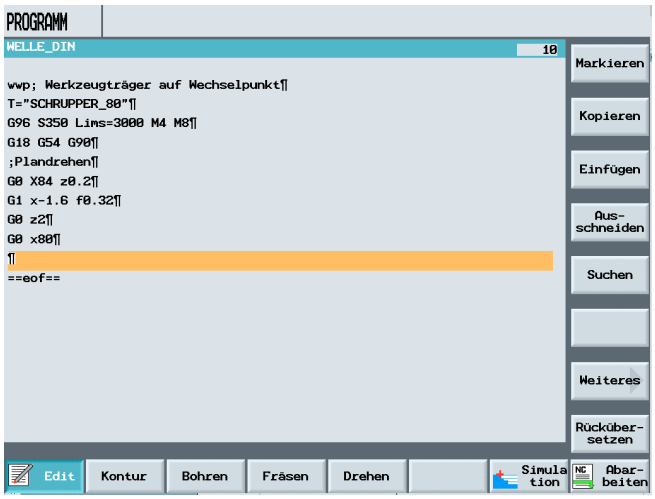
After selecting a tool,

facing machining is programmed.

By pressing the softkey

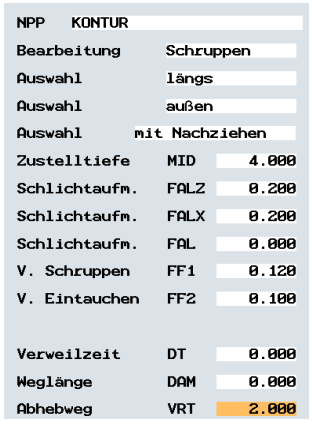
and

the machining cycle is opened.

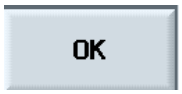


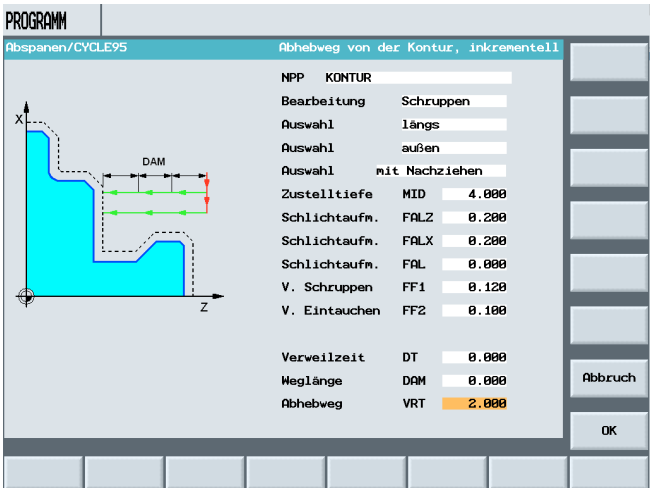
First, the program name of the contour is entered.

Then, the additional entries are made.



These are accepted by pressing

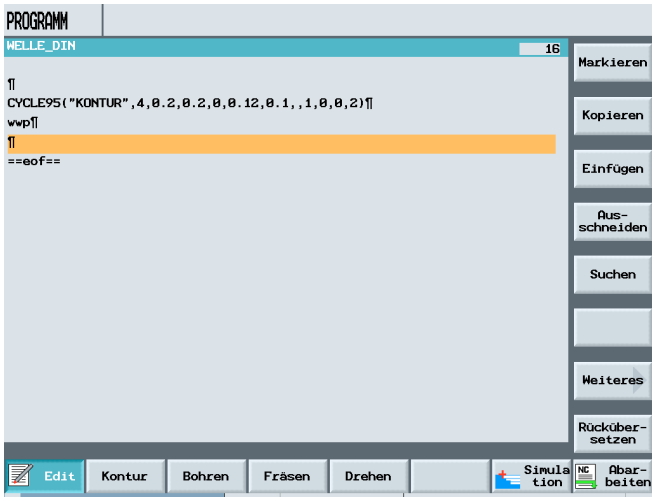




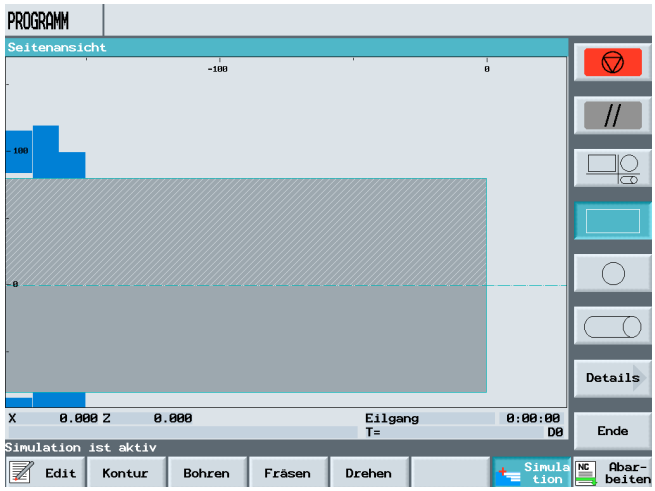
Then the tool change point is called.

To simulate the program, M30 has to be programmed at the end of the program.

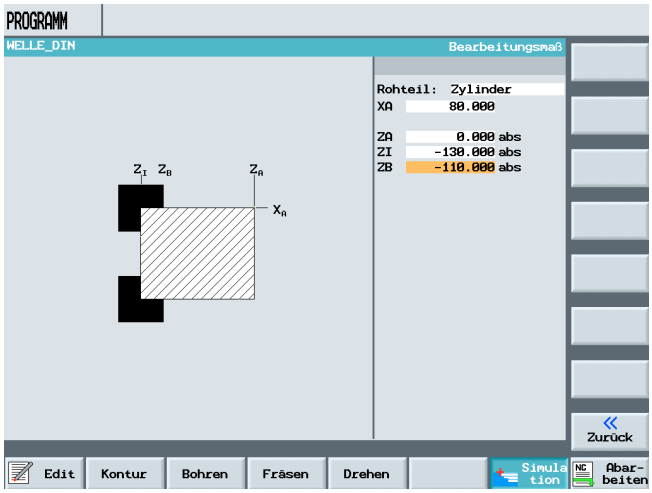
By pressing the softkey



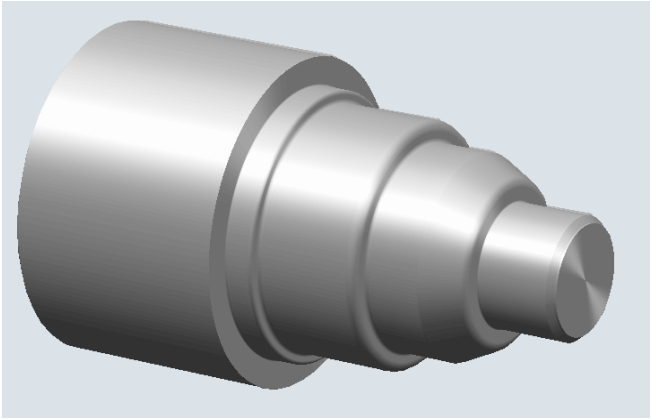
and



the raw part for the simulation can be defined.



Then, the simulation as under ShopTurn can take place.

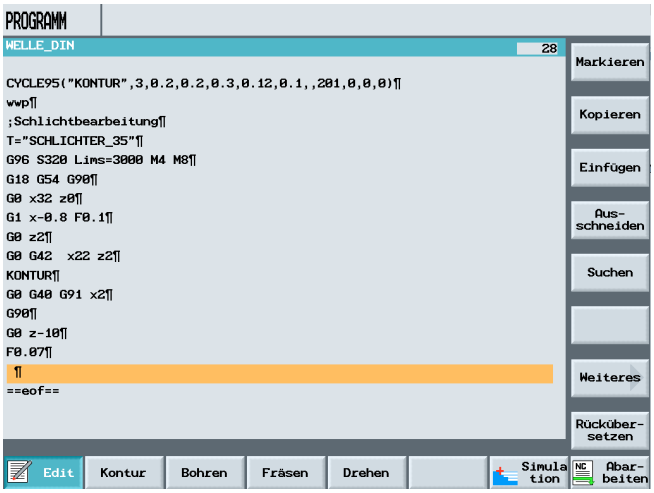


Next, the thread undercut is generated.

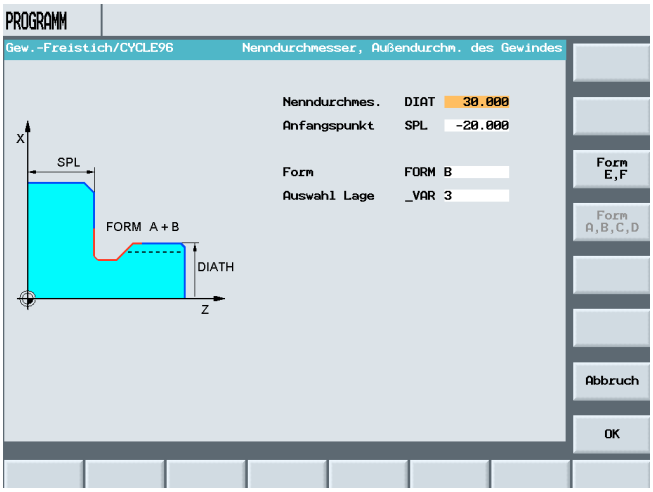
After entering the starting point

```
G90¶
G0 z-10¶
F0.07¶
```

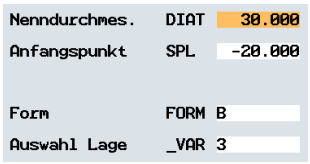
and pressing the softkey



and Undercut

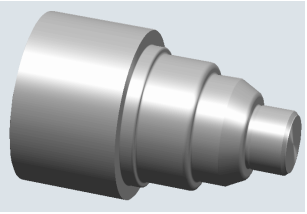


the screen form for the undercut is opened.
After entering the values



and accepting them into program, the retraction is programmed.

```
G0 x82 z2¶
wwp¶
```



Then, the thread is programmed.

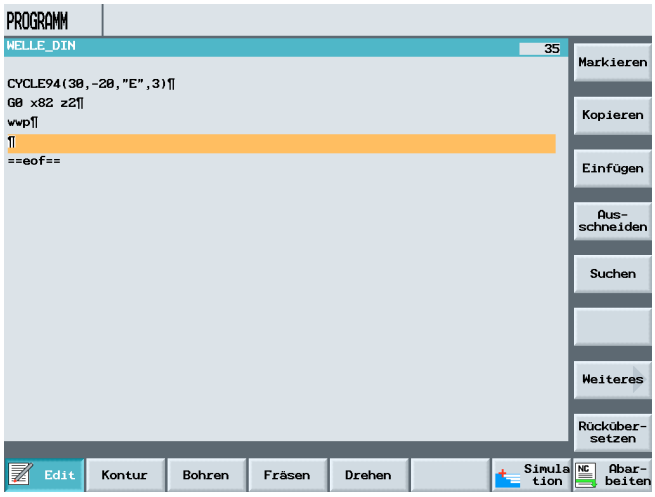
After calling the tool

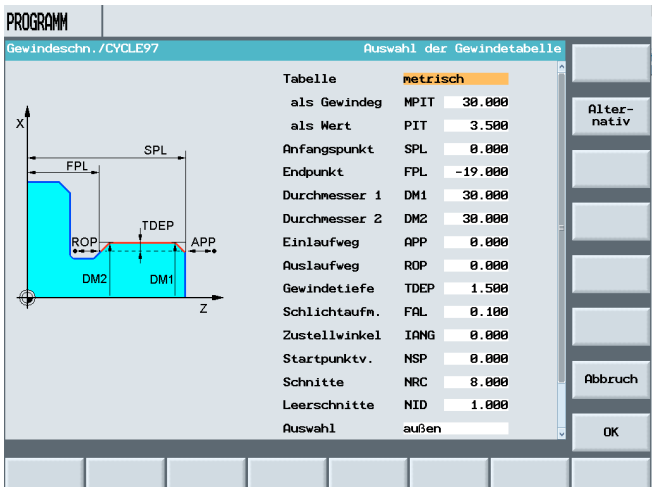
```
T="Gewinde"¶
G96 s200 lims=3000 M3 M8¶
G18 G54 G90¶
G0 X40 z7¶
```

und pressing the softkey

and

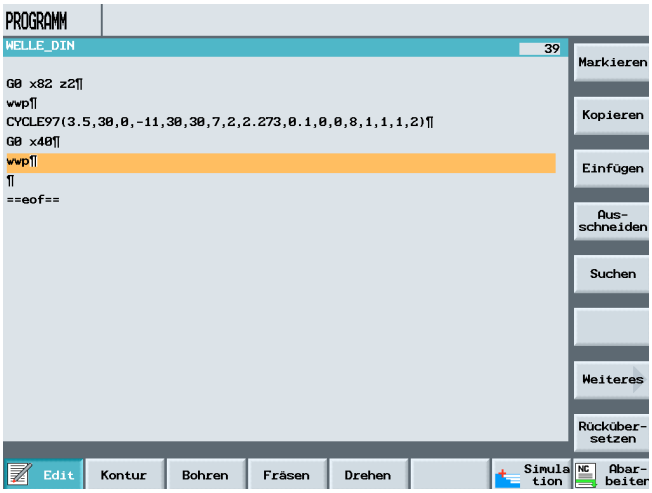
the input screen form opens.





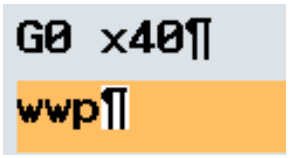
After entering the values

Tabelle		metrisch
als Gewindeg	MPIT	30.000
als Wert	PIT	3.500
Anfangspunkt	SPL	0.000
Endpunkt	FPL	-19.000
Durchmesser 1	DM1	30.000
Durchmesser 2	DM2	30.000
Einlaufweg	APP	0.000
Auslaufweg	ROP	0.000
Gewindetiefe	TDEP	1.500
Schlichtaufm.	FAL	0.100
Zustellwinkel	IANG	0.000
Startpunktv.	NSP	0.000
Schnitte	NRC	8.000
Leerschnitte	NID	1.000
Auswahl		außen

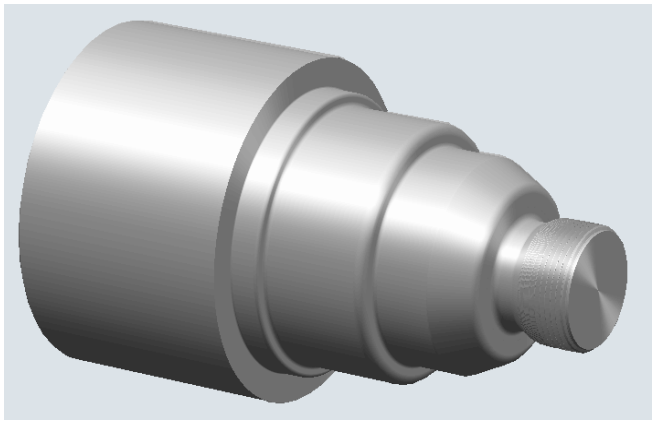


Auswahl	Konst. Zust.	
Anzahl Gänge	NUMT	1.000
Rückzug	VRT	2.000

and programming the tool change point



the thread is completely programmed and the program done.



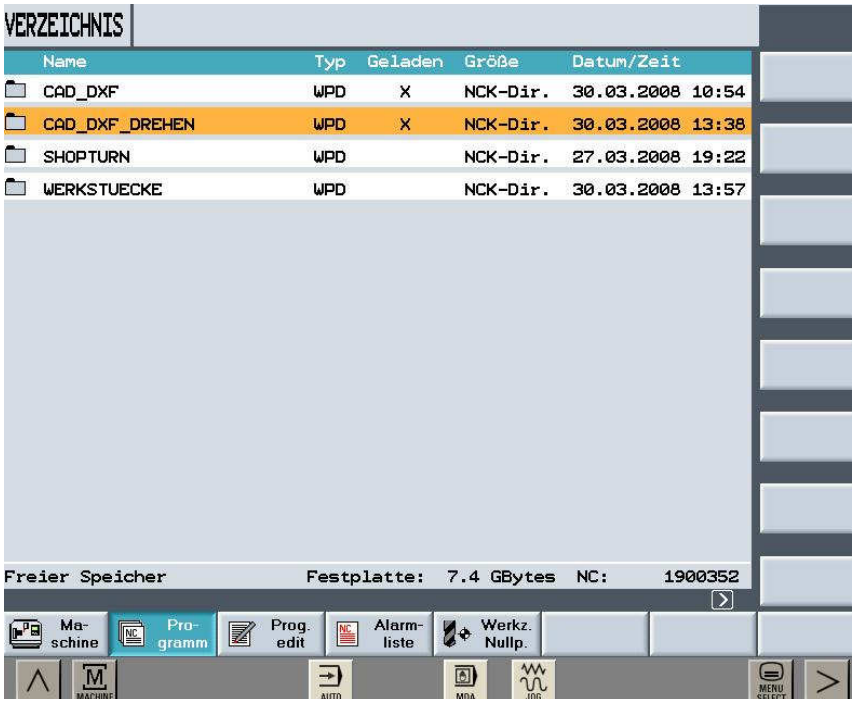
19 CAD Reader

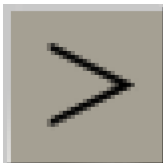
19.1 General Function

The CAD Reader is used for the following: to further edit drawings with the SINUMERIK controller that were constructed with a CAD system. As the format, a DXF file (**D**rawing e**X**change **F**ormat) is entered, and contours or drilling points are filtered out.
Parts that are not necessary for editing (such as dimensions, hatching, labeling, frames, etc.) can be removed. The generated contours or drilling patterns are implemented in a way that the geometry processor or cycle support understand them.
.

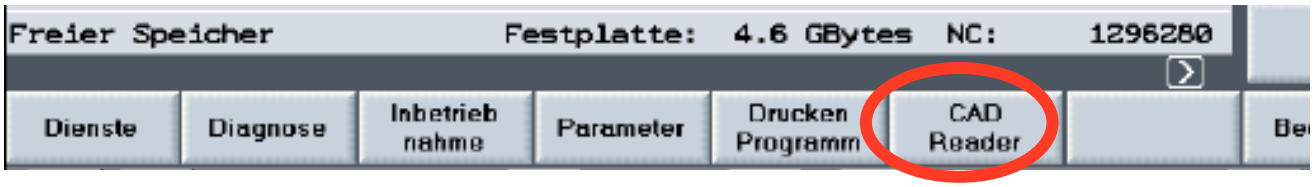
19.2 Opening the CAD READER

We are in the basic directory with the following softkeys:
- Machine
- Programs
- Program edit
- Alarm list
- Tool zero point



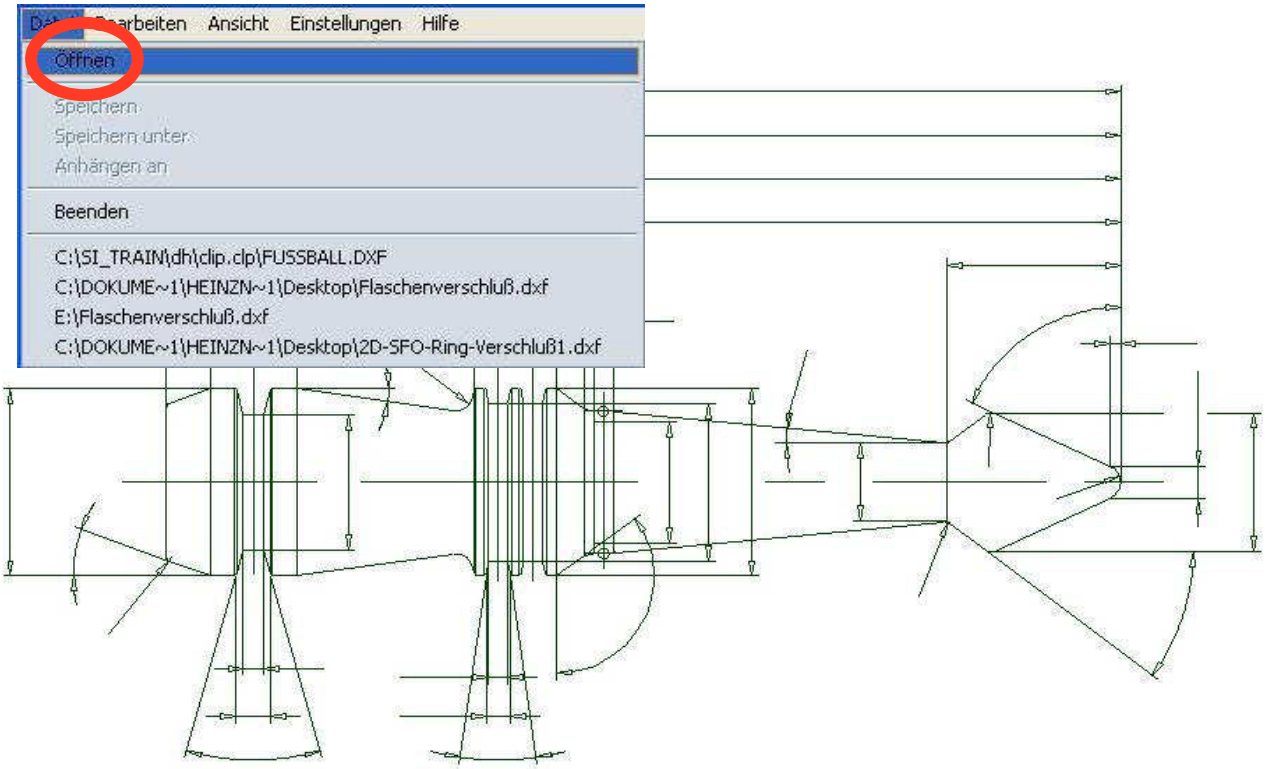


With this arrow key toward the right, the softkey bar opens.



Press the softkey "CAD Reader“

19.3 Open the DXF Drawing from a File

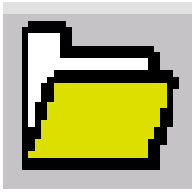


„Datei öffnen“.

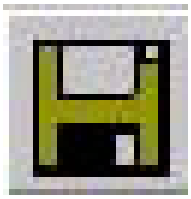
19.4 Toolbar

The toolbar can be selected by means of the global header "View → Toolbar Display“.

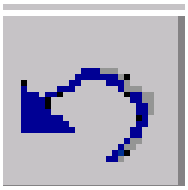
Operation in General
All CAD Reader functions can be accessed with the keyboard as well as with the mouse. The right mouse key corresponds to the "ESCAPE“ function that can be used to reset activated menus or functions.



Operating Sequence
Open DXF files
With **Open**, the selected CAD drawing is selected.

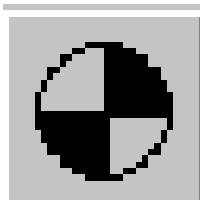


Saving the Generated Program
Generated sequences can be saved as data type in the
• MPF format (□.mpf)
• SPF format (□.spf)
• ARC format (□.arc) (SINUMERIK archive)



Return
When tracking contours, the last action is reset element by element, or the last intersection, depending on the selection.

19.5 Specifying the Zero Point



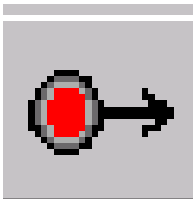
To read out the contour as an NC program, it is necessary to define a zero point of the drawing, since in most cases it will deviate from the zero point of the DXF file.

The following options are provided for defining the zero point:

Element Center	• Automatically on Element Center
Element Start	• Automatically on Element Start
Element End	• Automatically on Element End
Free input	• Direct input of the coordinates; for example. X100, Y100
Mouse position	• Any position by making the selection with the mouse

19.6 Contour Tracking

Setting the Contour Starting Point

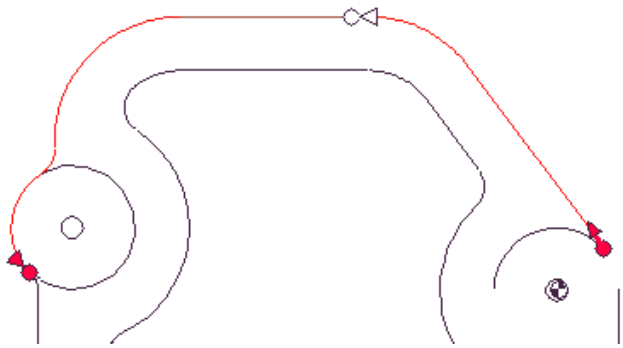


Contour Tracking with Starting Point and End Point

The starting and end point of the contour to be generated is selected depending on the initial position of the technology used:

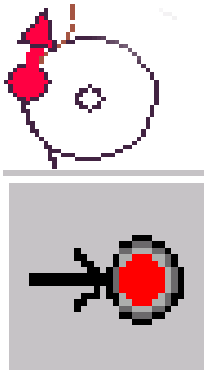
Element center	• Automatically on Element Center
Start/End point	• Automatically on Element Start/End point
Mouse position	• Directly through selection with the mouse

Example



1. Contour direction:
Results from specifying the starting point •
and further contour selection. When tracking contours, the attempt is made to select the contour automatically to the greatest extent.

2. Selection if there is a conflict
If automatic contour tracking can no longer uniquely determine a subsequent element, we switch to the interactive mode. The user is asked to specify the next element that continues the contour.

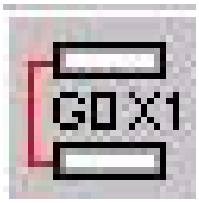


3. Full circle as contour
A full circle can be accepted with contour tracking in both directions.

4. Setting the end point
The end point can be set to any selected contour tracking element and accepted.

Setting the Contour Endpoint
Element center
Element endpoint
Mouse position
Current position

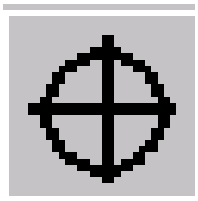
Additional Notes
• Full circles can be accepted as contour or as drill points.
• We can cancel contour tracking either by using the keyboard with the “Esc“ key, or with the mouse using the right mouse key.



Setting the Contour Label
Prior to contour tracking, labels can be set by entering start and end labels.
If labels are assigned twice, the CAD reader becomes interactive if the label was assigned once before
• Regarding contour tracking in contours that are already selected
• In the case of appendixes to files if the label already occurs in the file.

Setting Drilling Points

1. Full circle as hole
A full circle can be selected with the function drilling points. The output of the generated G-code corresponds to the cycle format.

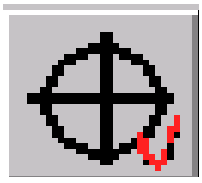


Drilling Points Start

2. By selecting Drilling Pattern, drilling points can be parameterized as

Any position
Row of holes
Hole circle
Hole spacing

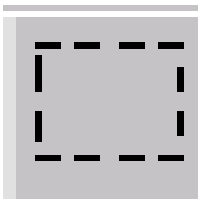
- Any drill position
- Corresponding to the cycle
- Corresponding to the cycle
- Corresponding to the cycle



Drilling Points End
3. Selected drilling points from the selection Drill pattern are accepted.

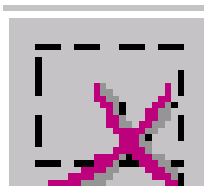
19.7 Influencing the Graph

Selecting the Machining Area:

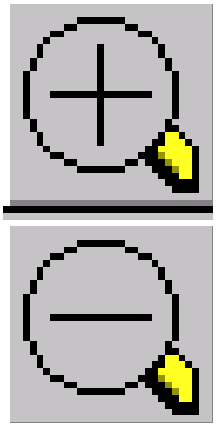


If a file includes many additional drawings such as cuts, dimensions, hatched area, labeling, detail representations, frames, etc., the selection of the machining area and the number of elements can be reduced by using a “lasso“.

Deselecting the Machining Area

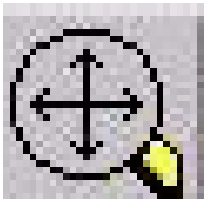


The selected machining area can be canceled with this deselection.



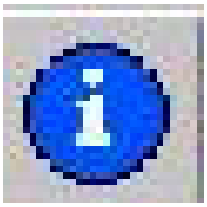
Zoom/Keys "+" and "– "
With the mouse key, it is possible to specify a zoom area within a drawing. By clicking on the symbol and using a "lasso" or the "+" and "–" keys, the graph area is centrically enlarged or reduced step by step.
The drawing area can be shifted by using the cursor keys.

19.8 Processing the File that was Entered



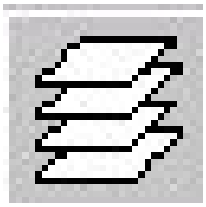
New Drawing/Spacer Key
Reads out the current drawing anew optimized according to the layer selection.

Geometry
With a mouse click, the coordinates are read out for the selected element corresponding to the current zero point. If the button Edit is displayed in the display screen form, this element can be edited by selecting this button.



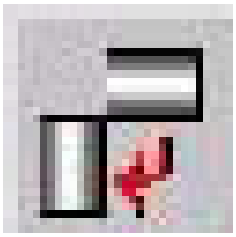
Note
This function is suitable for minor changes in the geometry, in order to remedy inadequacies (particularly missing intersections) in the CAD drawing.

For larger changes, use the geometry processor. Once a change is made, it **can not** be reset.

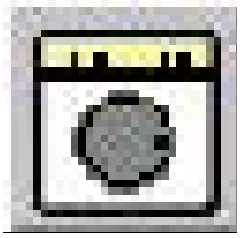


Layer Selection
Initially, the selected DXF file is always shown with all its layers. If the file contains several layers, all are shown in the basic view. However, layers without data that is relevant to the contour can be hidden. Likewise, contours ranging over several layers can be selected by means of a selection screen form for contour tracking.

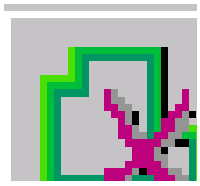
Layer selection can not be canceled.



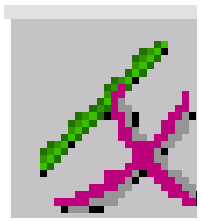
Contouring
By activating this icon, the drawing is rotated around the specified zero point by 90 degrees respectively, corresponding to the presetting. Contour characteristics that were already generated are not rotated.



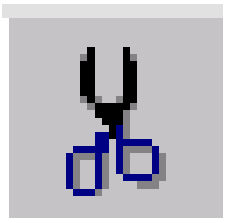
Insert Hatched Areas and Measurements
Is used to remove or insert hatched areas and measurements in the CAD drawings.
With another mouse click, this function is reset.



Deleting Contour Tracking
Defined contours can be selected or completely deleted. The function "Delete contour“ is activated with the initial operation of this icon, and deactivated by operating the icon again.
Deleting finished contours:
Select icon: activate Delete contour
Select contour: contour is deleted

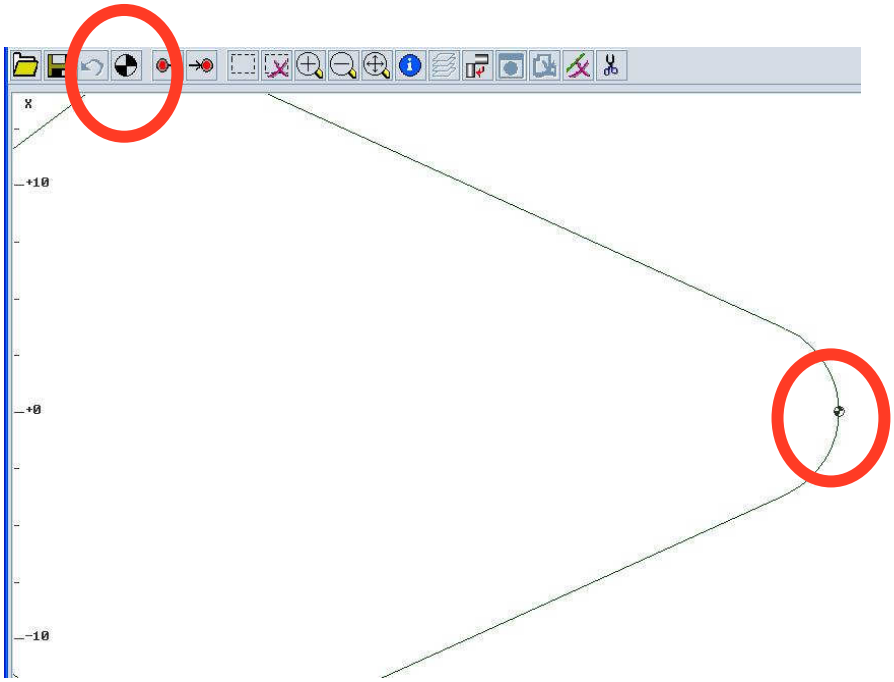


Deleting the Geometry Element
With this function, individual geometry elements can be deleted. The function “Delete geometry element“ is activated with the initial operation of this icon and deactivated by operating the icon again.
Deleting the geometry element:
Select icon: activate Delete geometry element
Select elements: geometry element is deleted

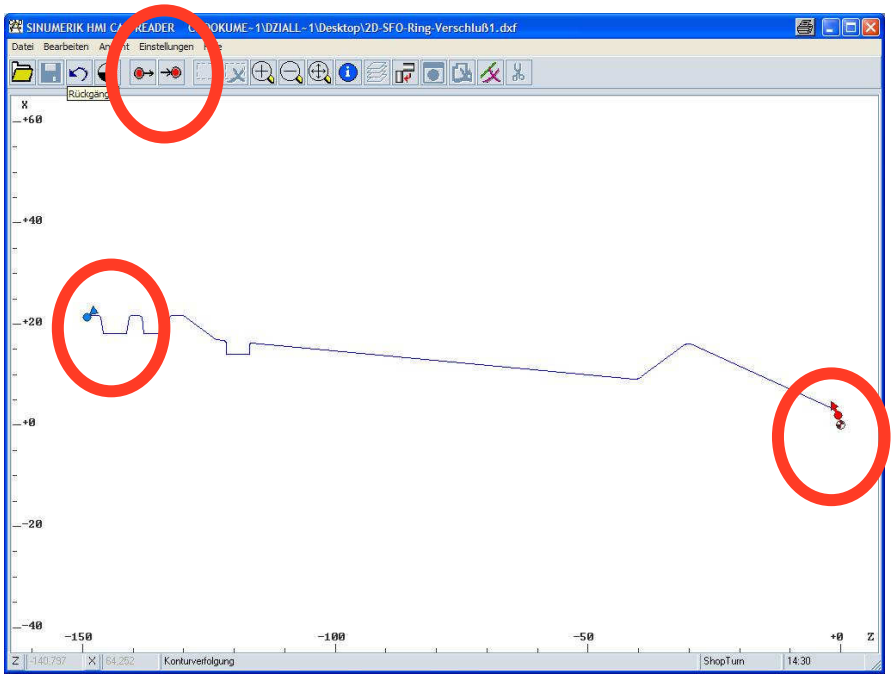


Deleting the Geometry Area
By means of a rectangle set up with the mouse (corresponds to the area to be deleted) a whole area can be deleted from the geometry.
This function is deactivated autonomously with each deletion und has to be reactivated each time with this icon.
Select icon: activate Delete the geometry area
Select the area: geometry area is deleted

19.9 Setting the Workpiece Zero Point

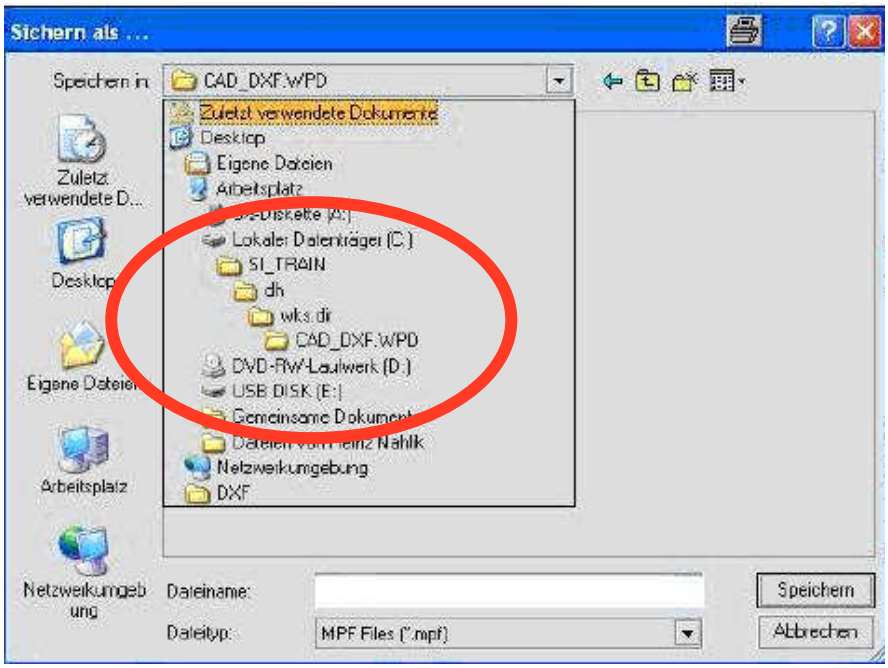


Contour tracking



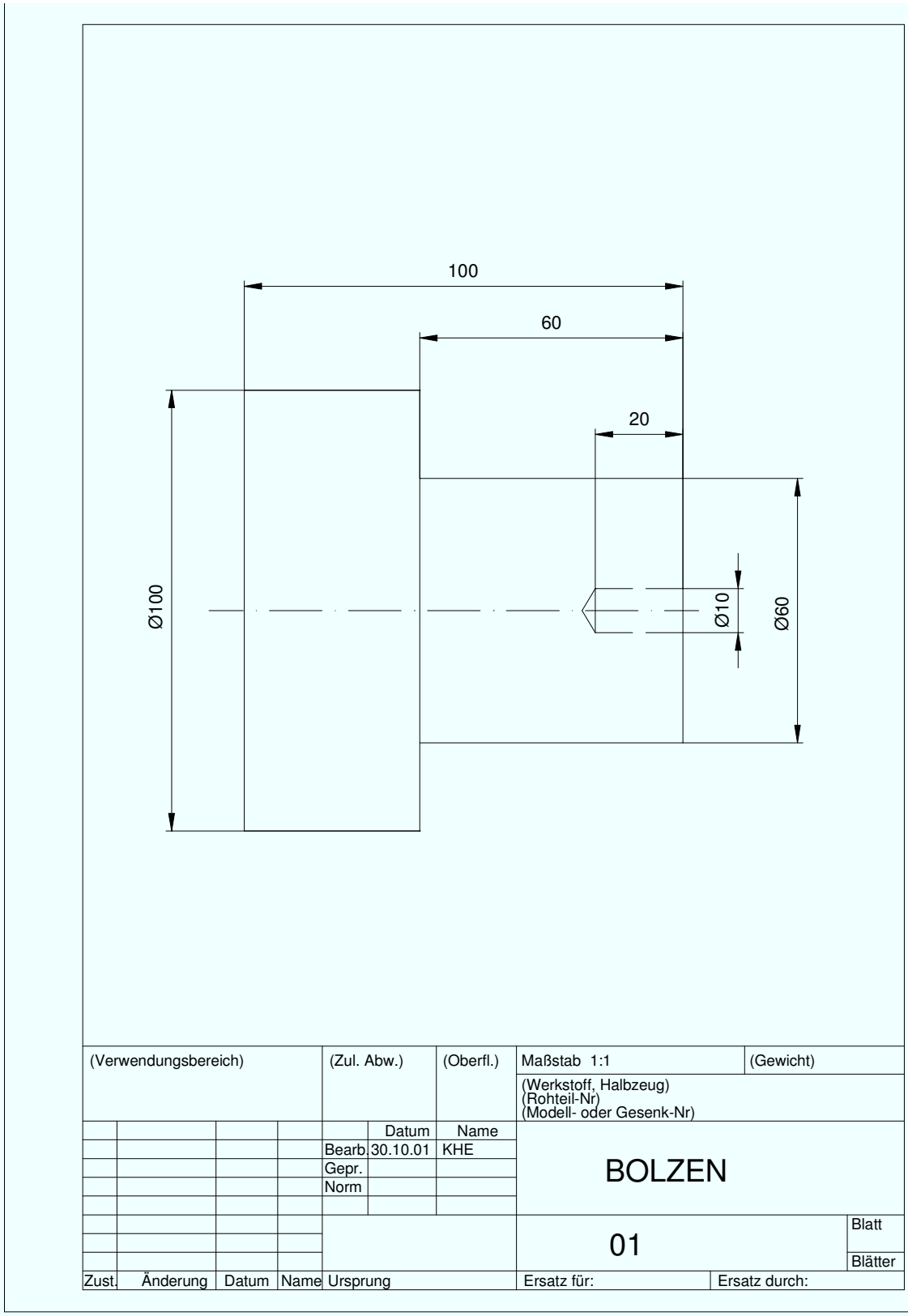
19.10 Transferring the Contour Elements to the Directory and the Contour Calculator

-Save as
-SI_Train
-dh
-wks.dir
-Select the directory where the contour is to be stored.

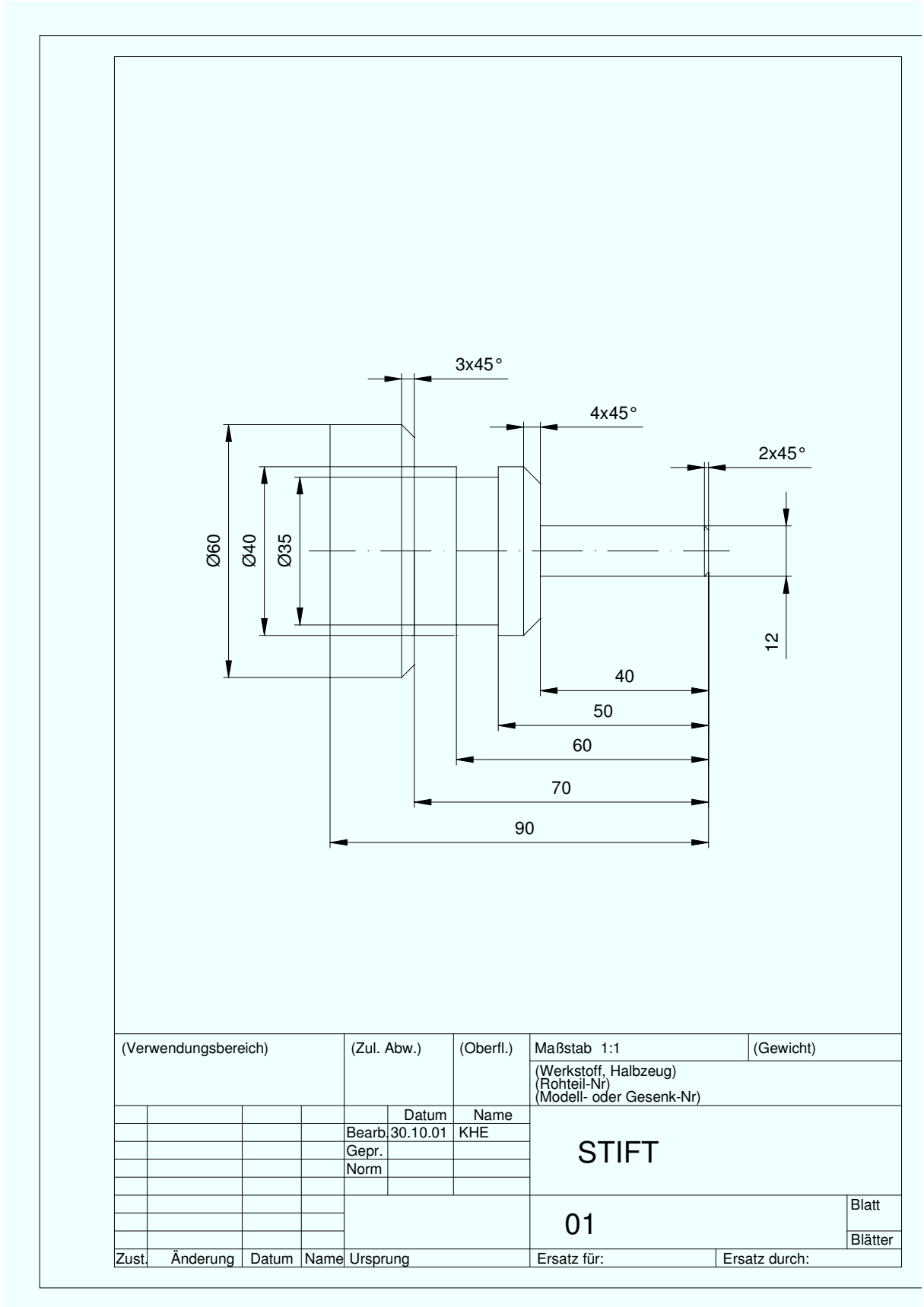


20 Sample Drawings - Turning

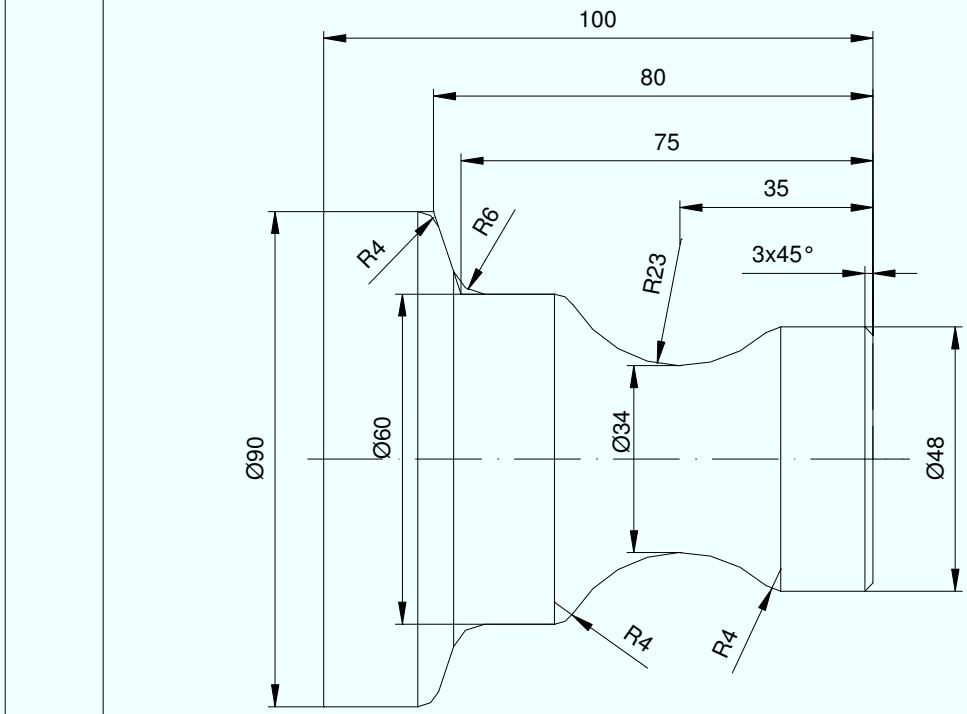
20.1 Bolt



20.2 Pin

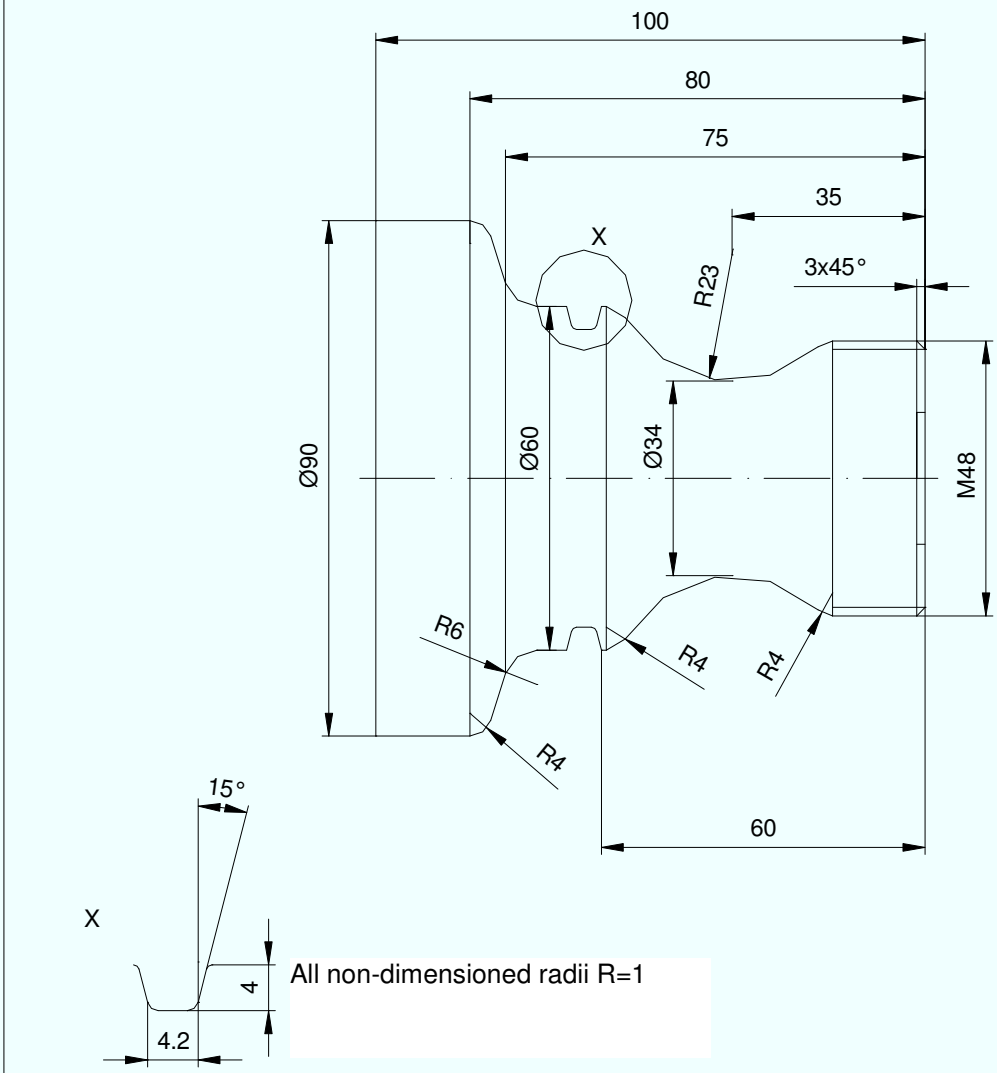

3x45°
4x45°
2x45°
Ø60
Ø40
Ø35
12
40
50
60
70
90
(Verwendungsbereich)
(Zul. Abw.)
(Oberfl.)
Maßstab 1:1
(Gewicht)
(Werkstoff, Halbzeug)
(Rohteil-Nr)
(Modell- oder Gesenk-Nr)
Datum
Name
Bearb.
30.10.01
KHE
Gepr.
Norm
STIFT
01
Blatt
Blätter
Zust.
Änderung
Datum
Name
Ursprung
Ersatz für:
Ersatz durch:

20.3 Shaft Outside-Contour Contour Calculator



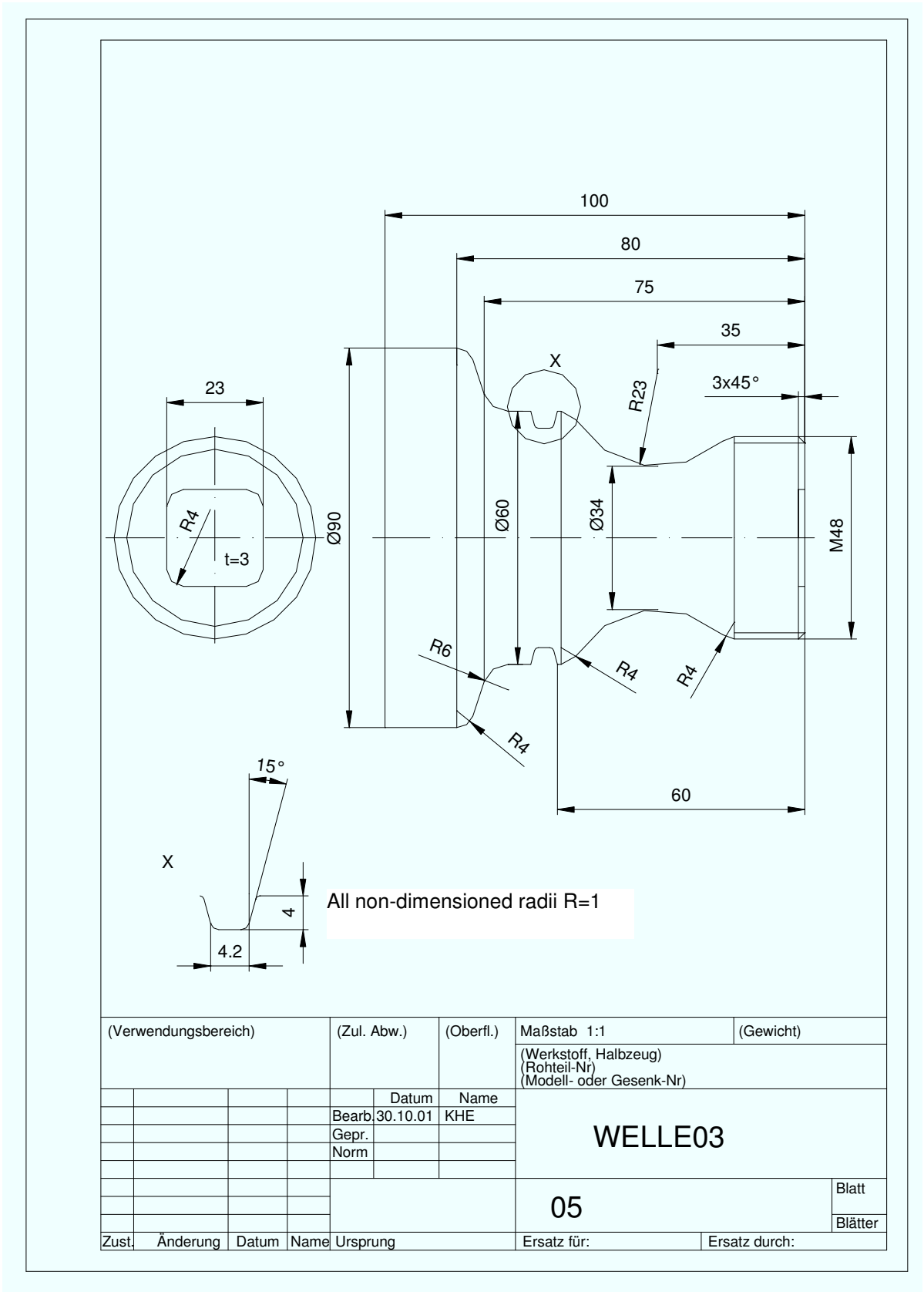
(Verwendungsbereich)				(Zul. Abw.)		(Oberfl.)	Maßstab 1:1	(Gewicht)
							(Werkstoff, Halbzeug) (Rohteil-Nr) (Modell- oder Gesenk-Nr)	
					Datum	Name	WELLE1	
				Bearb.	30.10.01	KHE		
				Gepr.				
				Norm				
							04	Blatt
								Blätter
Zust.	Änderung	Datum	Name	Ursprung			Ersatz für:	Ersatz durch:

20.4 Shaft Groove

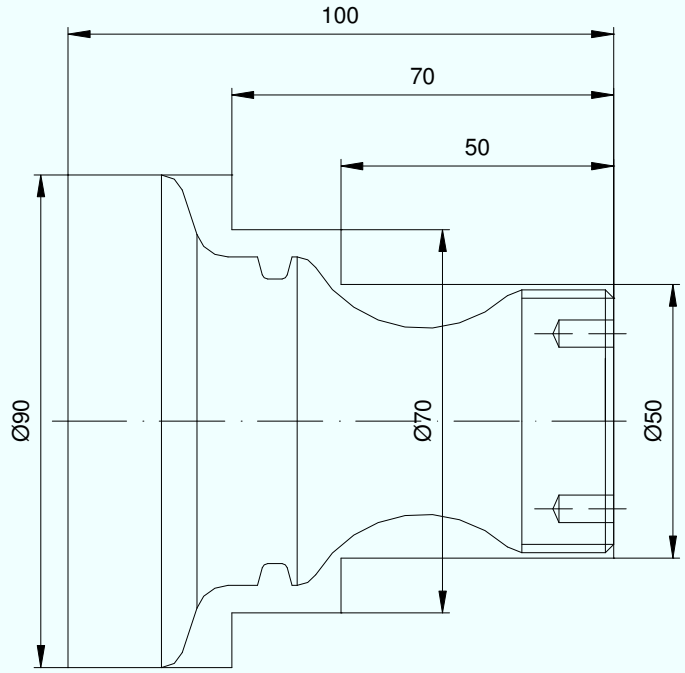


(Verwendungsbereich)				(Zul. Abw.)		(Oberfl.)	Maßstab 1:1	(Gewicht)
							(Werkstoff, Halbzeug) (Rohteil-Nr) (Modell- oder Gesenk-Nr)	
					Datum	Name	WELLE02	
				Bearb.	30.10.01	KHE		
				Gepr.				
				Norm				
							05	Blatt
								Blätter
Zust.	Änderung	Datum	Name	Ursprung			Ersatz für:	Ersatz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

20.5 Shaft Rectangular Pocket Frontal Face

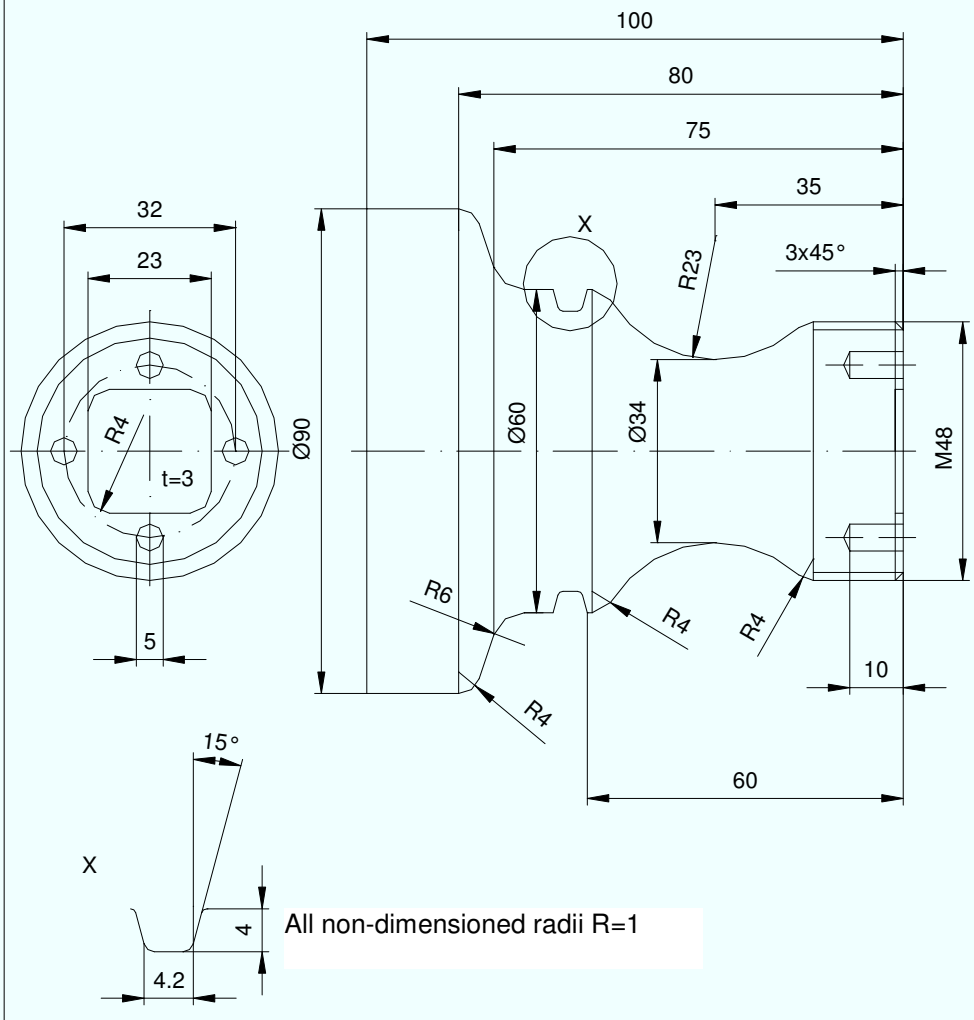


20.6 Shaft Raw Part Contour Available



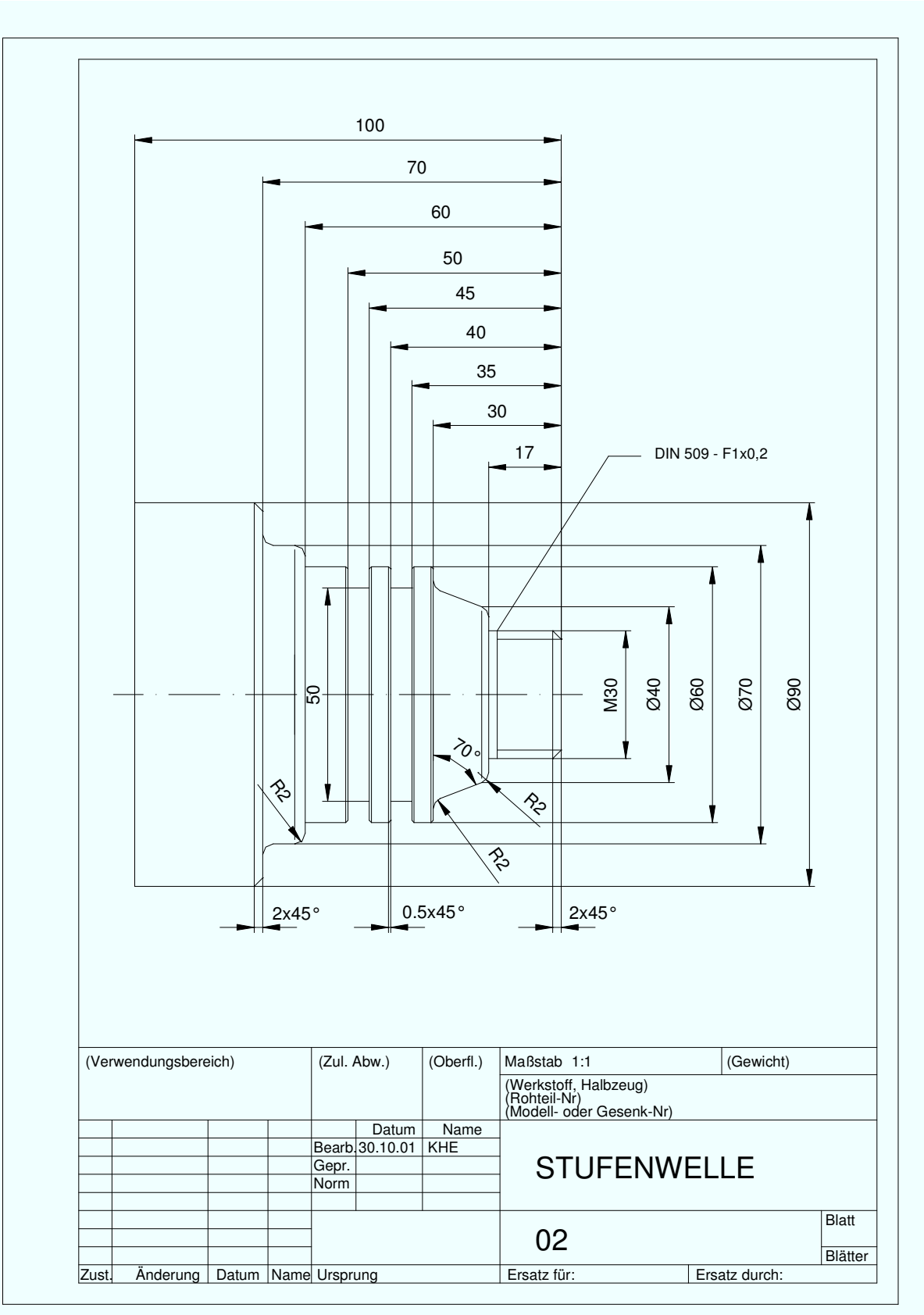
(Verwendungsbereich)				(Zul. Abw.)		(Oberfl.)	Maßstab 1:1	(Gewicht)
							(Werkstoff, Halbzeug) (Rohteil-Nr) (Modell- oder Gesenk-Nr)	
					Datum	Name	WELLE04	
				Bearb.	30.10.01	KHE		
				Gepr.				
				Norm				
							05	Blatt
								Blätter
Zust.	Änderung	Datum	Name	Ursprung			Ersatz für:	Ersatz durch:

20.7 Shaft with Holes on the Frontal Face

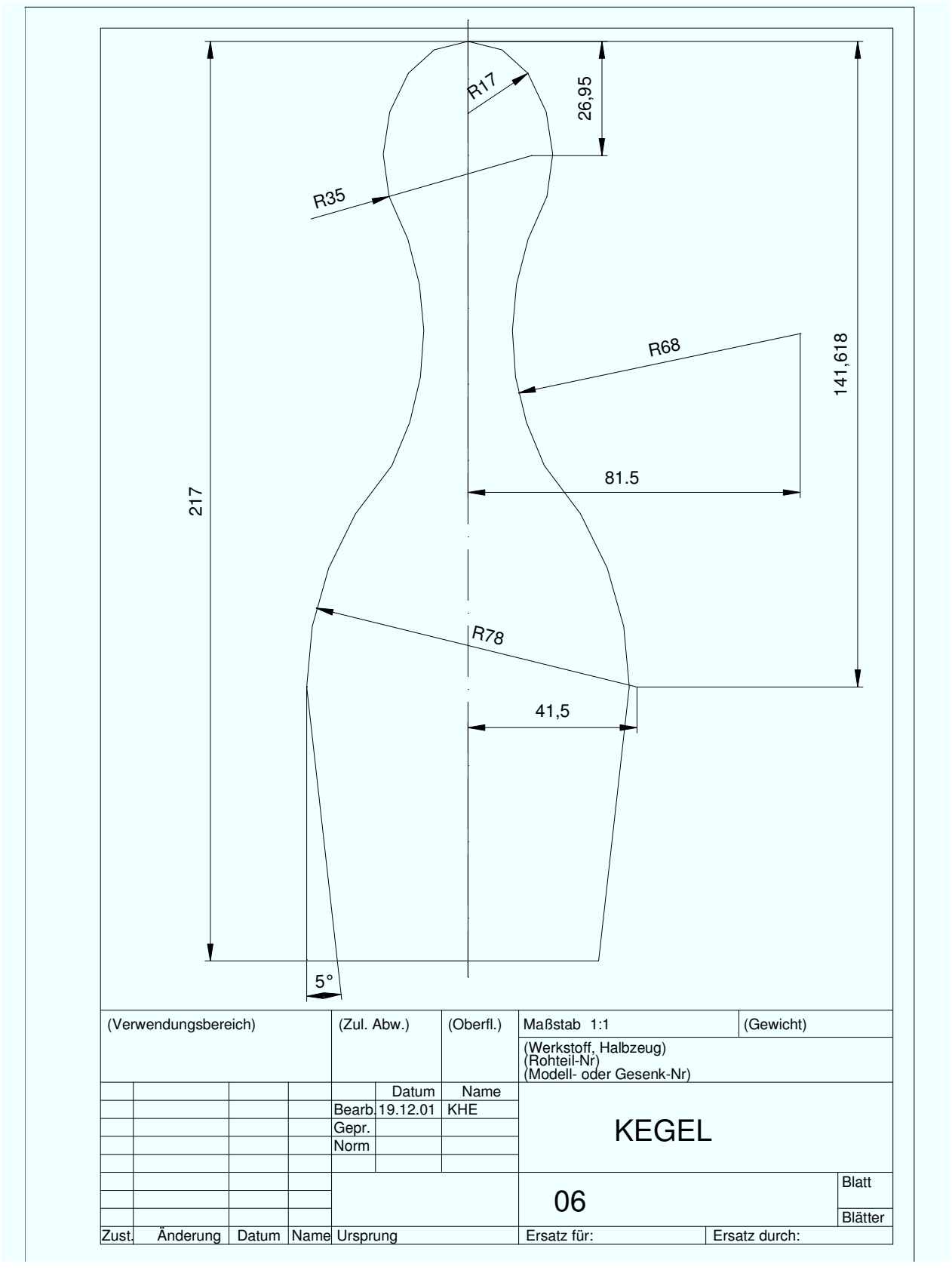


(Verwendungsbereich)				(Zul. Abw.)		(Oberfl.)	Maßstab 1:1	(Gewicht)
							(Werkstoff, Halbzeug) (Rohteil-Nr) (Modell- oder Gesenk-Nr)	
					Datum	Name	WELLE04	
				Bearb.	30.10.01	KHE		
				Gepr.				
				Norm				
							05	Blatt
								Blätter
Zust.	Änderung	Datum	Name	Ursprung			Ersatz für:	Ersatz durch:

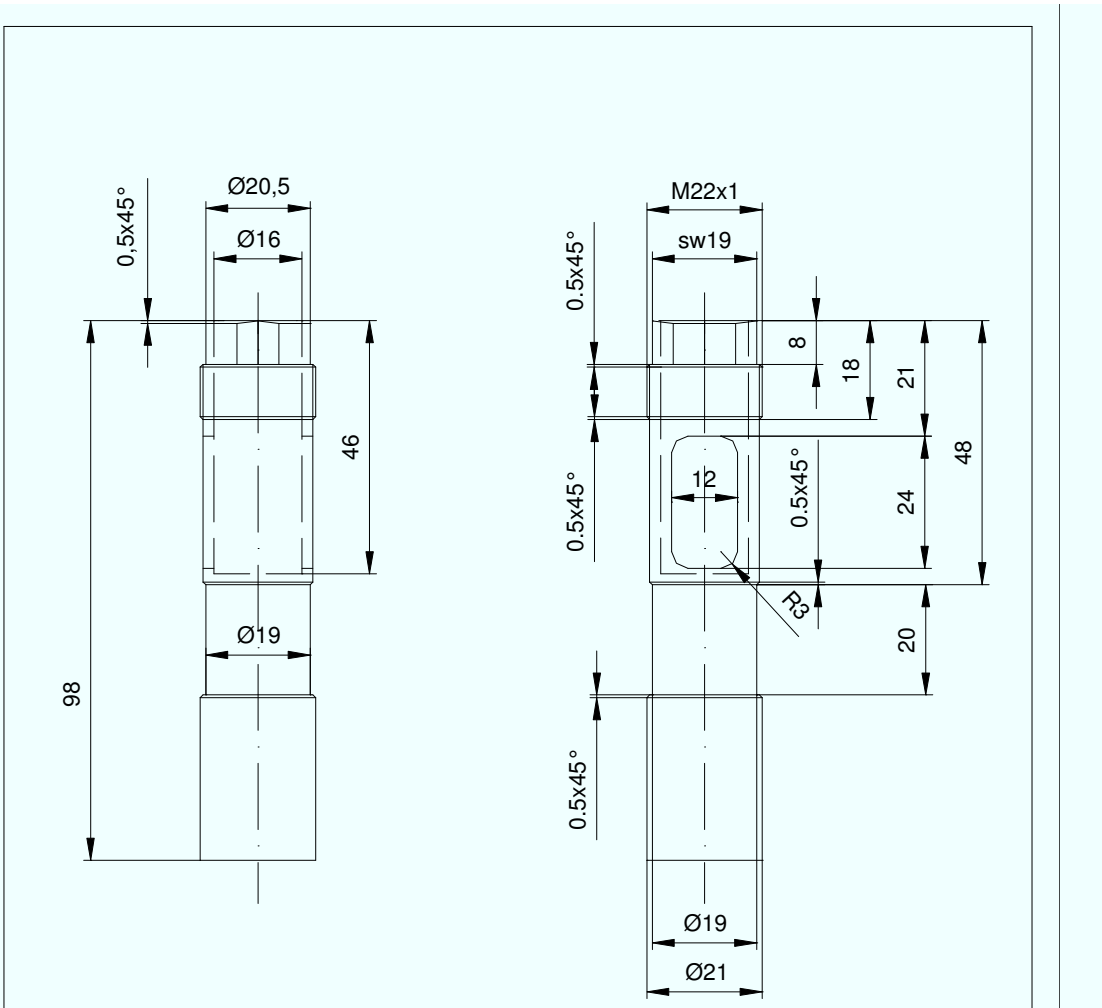
20.8 Step Shaft Grooves Threats



20.9 Cone

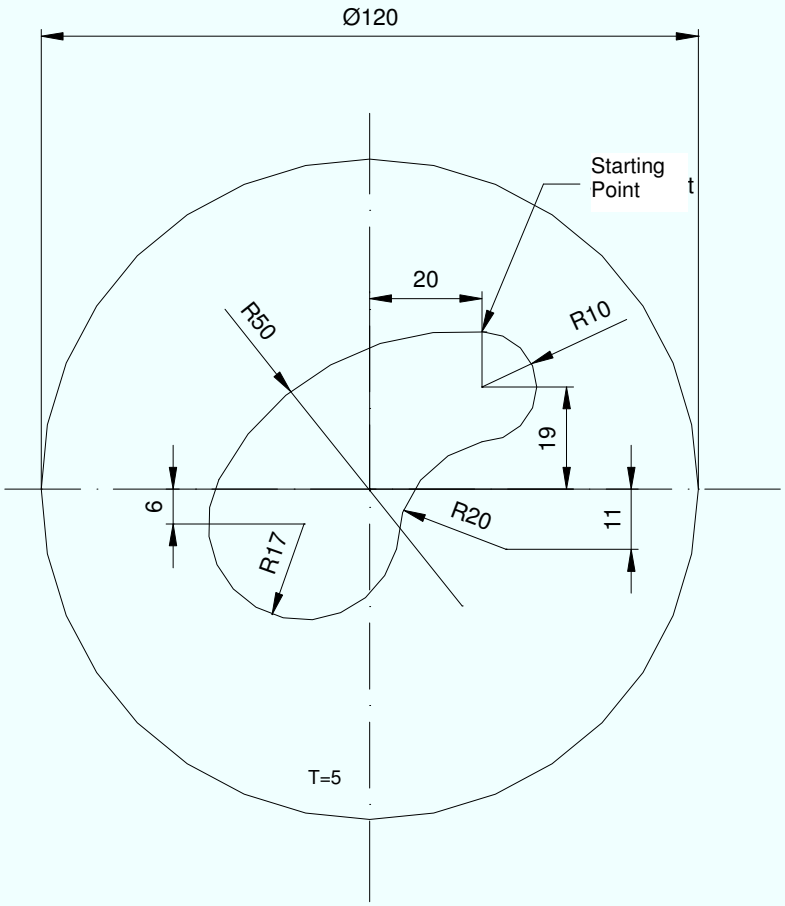

R17
26,95
R35
R68
141,618
81.5
217
R78
41,5
5°
(Verwendungsbereich)
(Zul. Abw.)
(Oberfl.)
Maßstab 1:1
(Gewicht)
(Werkstoff, Halbzeug)
(Rohteil-Nr)
(Modell- oder Gesenk-Nr)
Datum
Name
Bearb. 19.12.01 KHE
Gepr.
Norm
KEGEL
06
Blatt
Blätter
Zust. Änderung Datum Name Ursprung
Ersatz für:
Ersatz durch:

20.10 Cardan Shaft



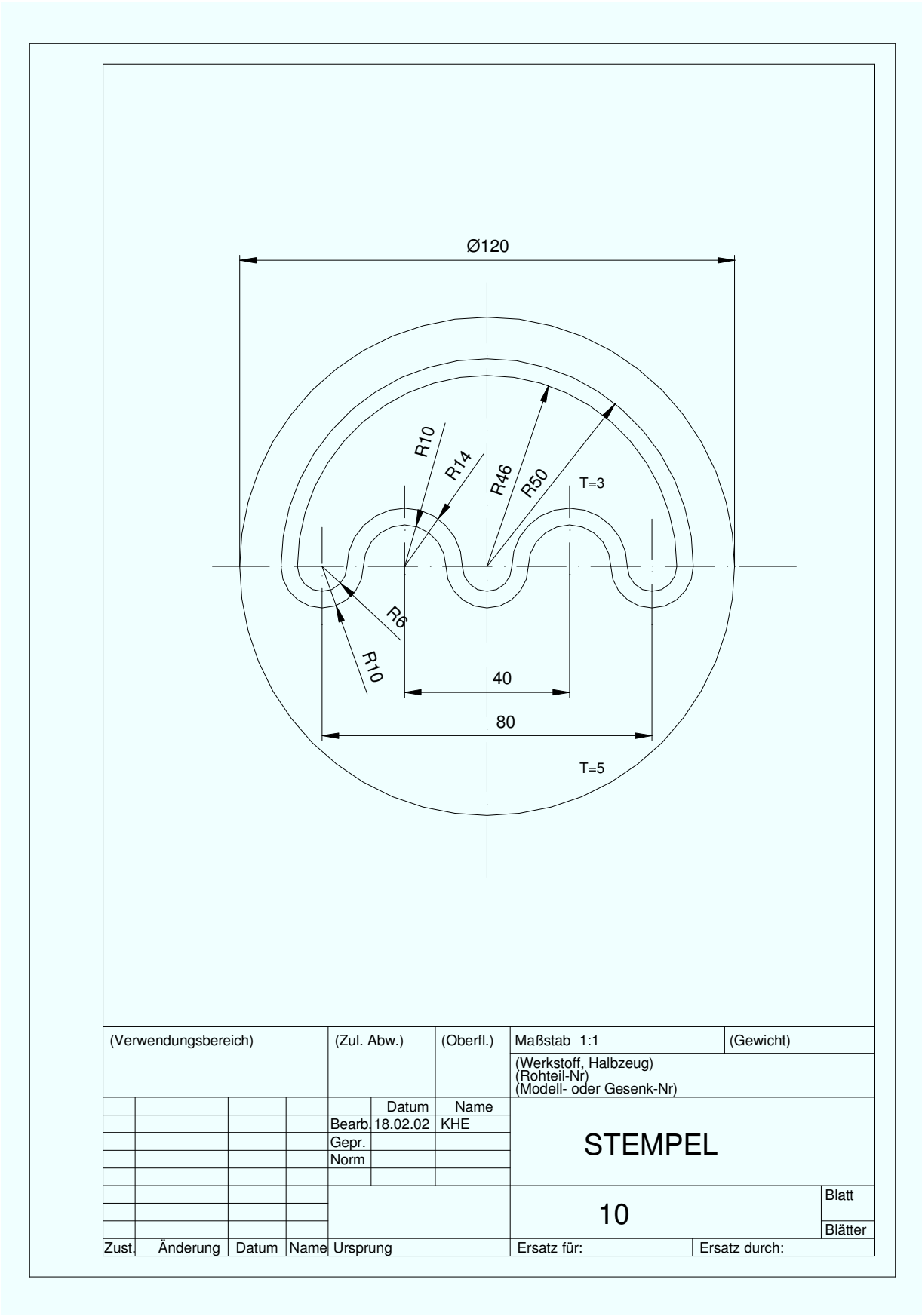
(Verwendungsbereich)				(Zul. Abw.)		(Oberfl.)	Maßstab 1:1	(Gewicht)
							(Werkstoff, Halbzeug) (Rohteil-Nr) (Modell- oder Gesenk-Nr)	
					Datum	Name	GELENKWELLE	
				Bearb.	02.02.02	KHE		
				Gepr.				
				Norm				
							08	Blatt
								Blätter
Zust.	Änderung	Datum	Name	Ursprung			Ersatz für:	Ersatz durch:

20.11 Kidney/Face End



(Verwendungsbereich)				(Zul. Abw.)		(Oberfl.)	Maßstab 1:1	(Gewicht)
							(Werkstoff, Halbzeug) (Rohteil-Nr) (Modell- oder Gesenk-Nr)	
					Datum	Name	NIERE	
				Bearb.	16.02.02	KHE		
				Gepr.				
				Norm				
							09	Blatt
								Blätter
Zust.	Änderung	Datum	Name	Ursprung			Ersatz für:	Ersatz durch:

20.12 Stamp/Face End



20.13 Fitted Shaft