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

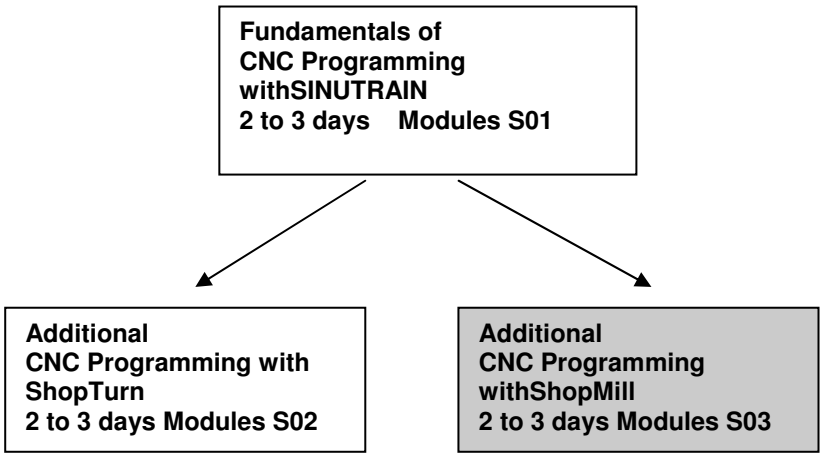
CNC Programming Milling
ShopMill
Module S03

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 ShopMill'** acquaints you with 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 carried out economically with the controllers ShopMill and ShopTurn.

.

Training Objective:
Module S03 shows you, step by step, how to program with ShopMill.
Subsequently, the reader is to solve the tasks provided.

Preconditions

To successfully work through this module, the following knowledge is assumed:

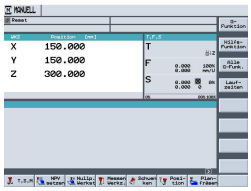
- Knowledge in handling Windows
- Fundamentals of CNC programming with Sinutrain (for example, Module S01)

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 when all products are installed, MS Internet Explorer starting with 6.0

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

1 PC



2. SINUTRAIN

2 Introduction

2.1 Development Phases of CNC Technology

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

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

3 Operator Components

In this chapter, the basic operator components are shown. These components are to be considered examples, and are not necessarily on the machine in the design described here.
The manufacturer's specifications have to be noted!

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

This operator panel consists of a scrren with horizontal

and
vertical

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

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

Here an example of a machine control panel.

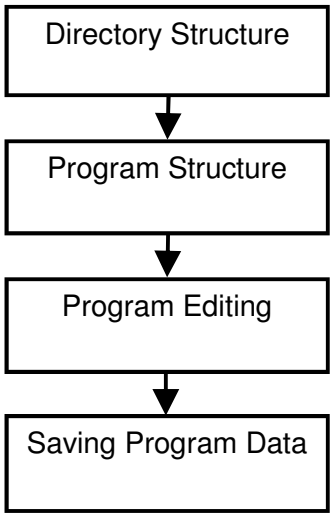
With the machine control panel the machining of a workpiece is started.

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

4 Program Management - Milling

The structure, the management, editing and saving programs under ShopMill is described in detail.

Content of the Module:



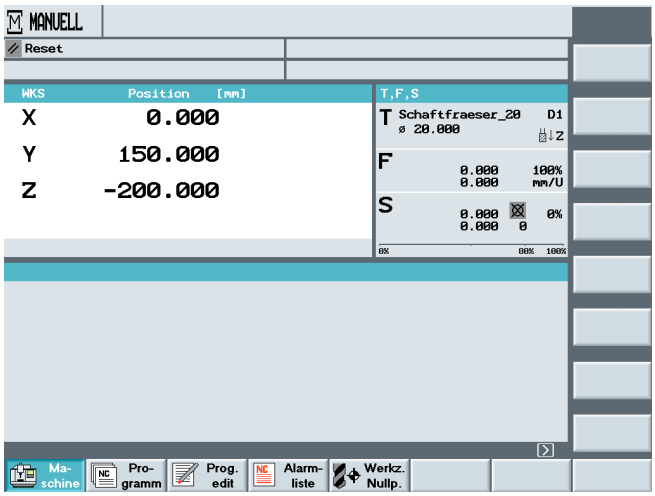
4.1 Directory

Programs can be stored in the directories. That keeps the program memory clearly arranged.

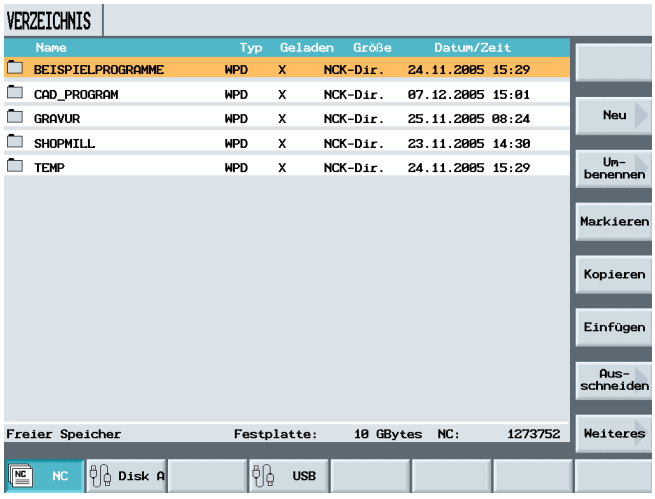
By pressing the softkey

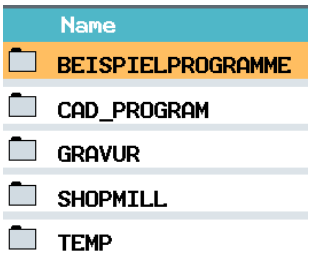
the available directories under ShopMill are opened in the operating area

.



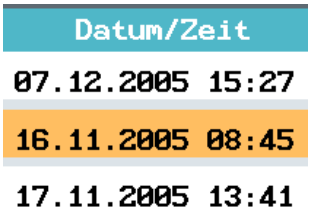
The names of the directories,





the directory type

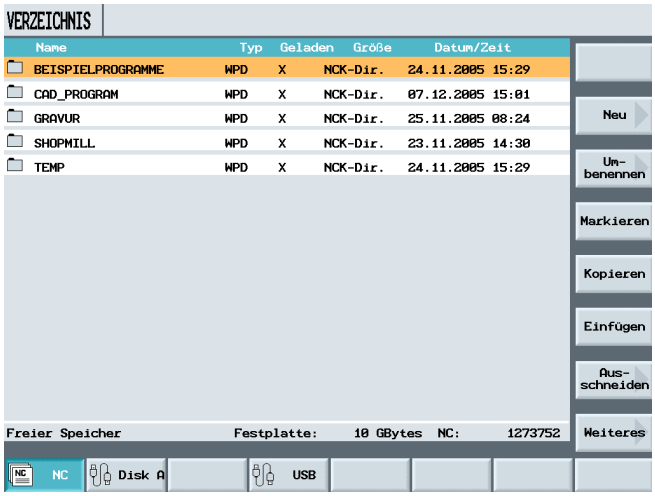
and the generation date and time are displayed.



Directory Structure

The size of the directory is not displayed.
The X in the area “Loaded“ tells us 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, we can access existing programs, or we can set up new ones.

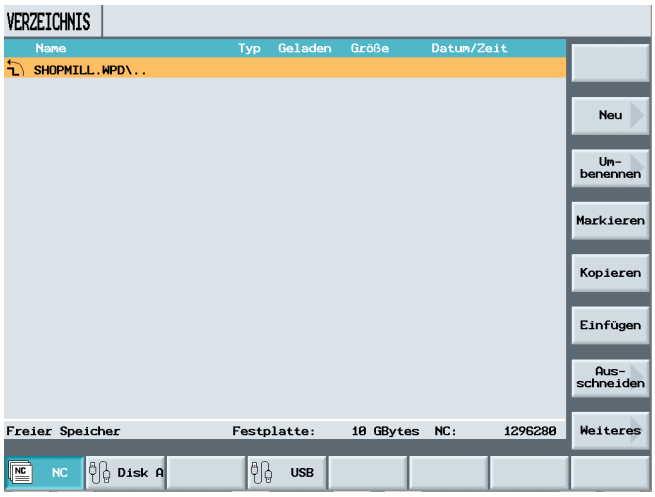
By pressing the arrow key

on the CNC keyboard,
the selected directory opens.

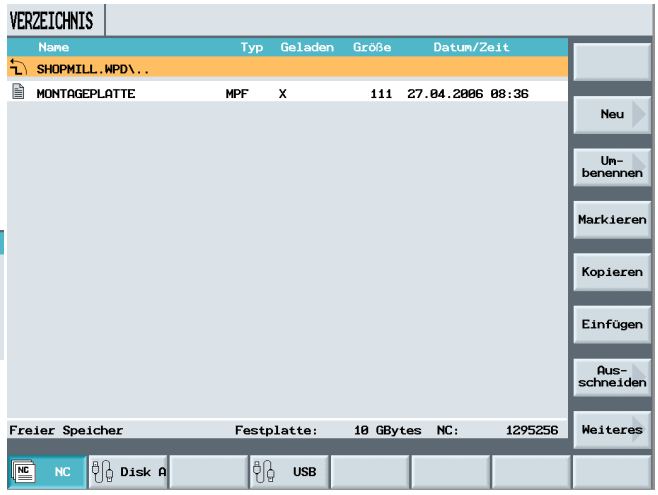
In ShopMill, only main directories can be generated - mpf - and no subprograms - spf -.

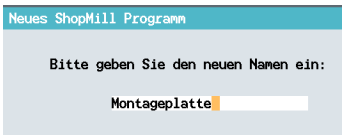
After opening the directory, available programs are displayed in the directory. New programs can be created.

By pressing the softkey



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





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

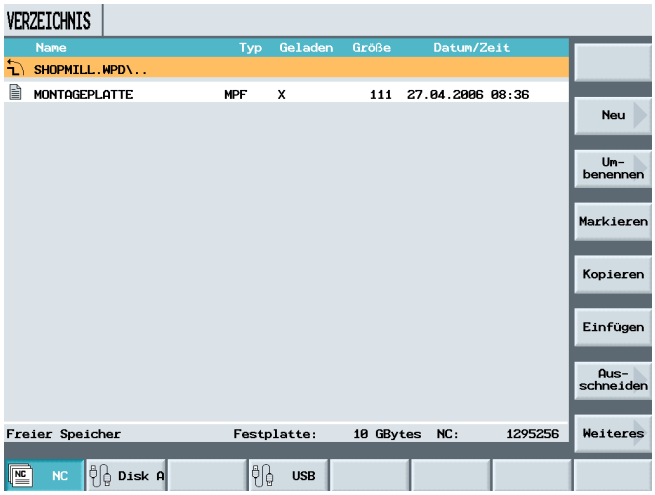
By pressing the arrow key,

the directory is closed. We are returned to the directory overview.

4.3 Editing Programs

The same functions are available as in MS Word.

With the softkeys below



directories, programs or individual program parts can be edited.

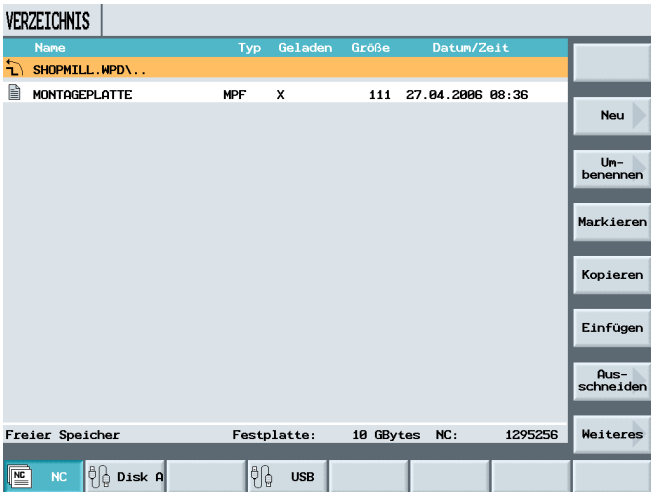
The individual softkeys are not described here in detail since they occur on the pages below in the exercises.

5 Saving Program Data

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

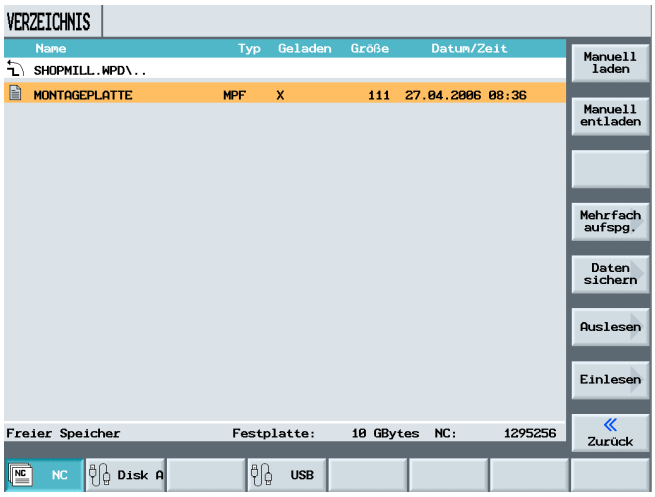
By pressing the softkey

in the operating area



the softkey

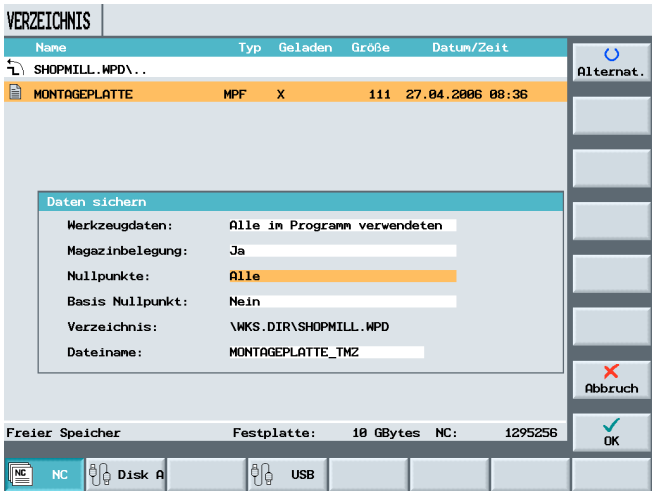
appears in the vertical softkey bar.
With “Save data“, the relevant machining data of the currently selected program can be saved.



By pressing this softkey,

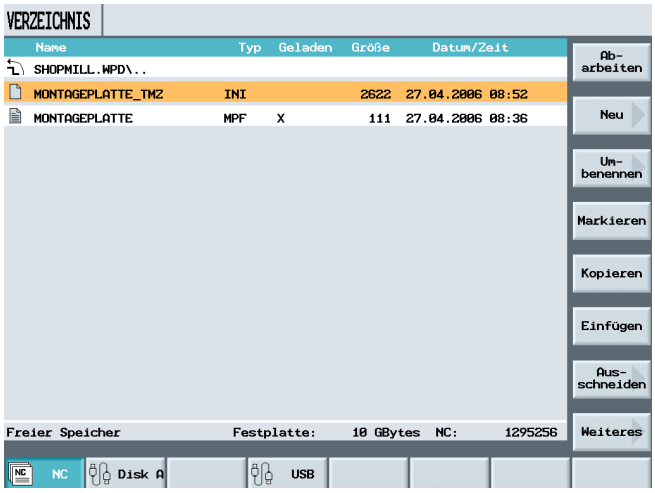
a dialog field opens.
With the key

the desired data is selected.



By pressing the softkey

an "ini“ file is generated in the operating area



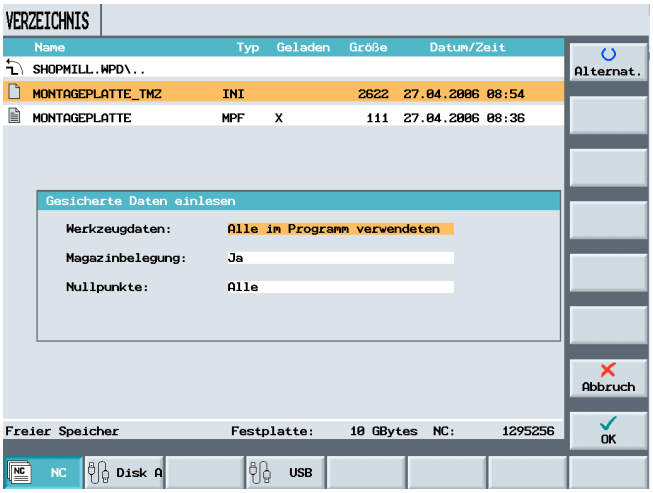
with the program name whose data was saved in the file.

The program with the corresponding “INI“ file can now be saved externally.

With the program and the INI file, all relevant data for manufacturing a workpiece is saved and can be called any time.

When an “INI file“ is selected, the stored data is read in again.

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



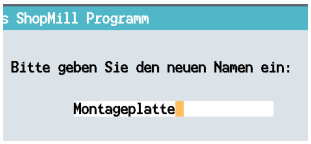
Here, the “complete tool list“ or the tools “used in the program“ can be saved.
If the “complete tool list“ is read in, all present tools are deleted, and replaced with those that were stored.

6 Program Structure - Milling

6.1 Program Header

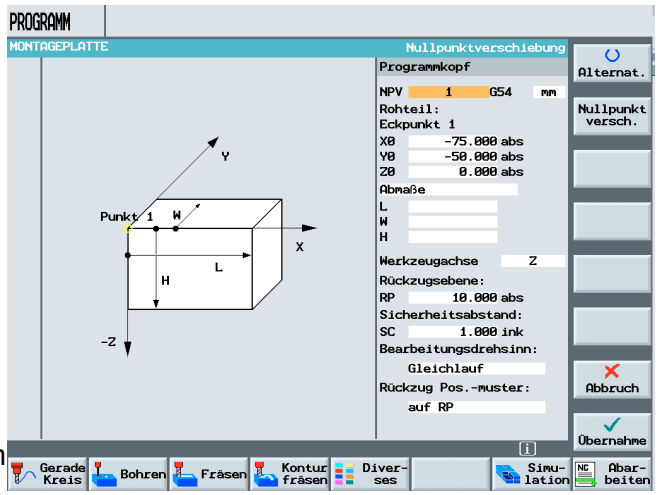
The basic settings are described in the program header.

After entering the program name



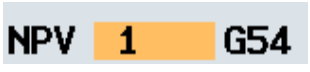
and confirming the input with

the program header of the new program is opened automatically.



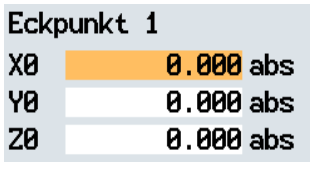
Here, the basic settings are made for the program sequence.

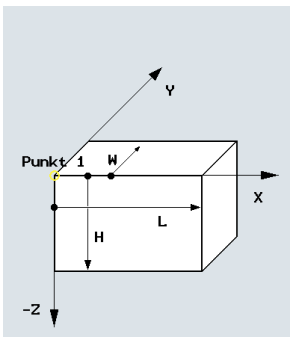
As the first input, a zero point shift for the program can be programmed directly in the program header.



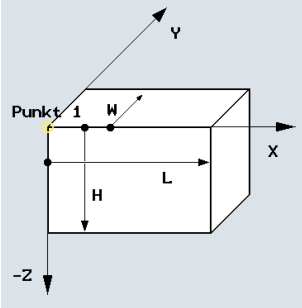
In the next input field, the raw part is defined.

As the first input field, the corner of the raw part is described.

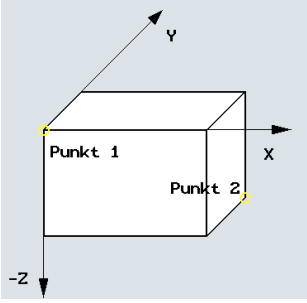




Next, the raw part can be described either by means of the deviations in reference to the first corner

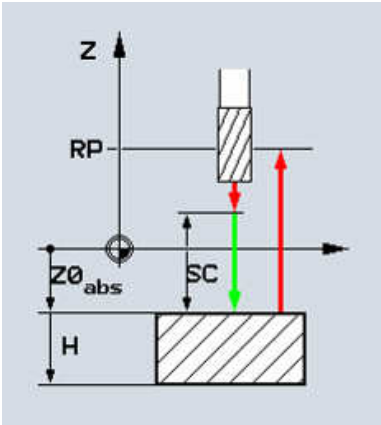


or the second corner of the raw part is described.



In the program header, only the raw part for simulation is defined. If no values for the raw part are entered, the milling center point path is displayed in the simulation.

In the next input fields, the following is defined: the tool axis, the return plane, and the safety distance.



Werkzeugachse	Z
Rückzugsebene:	
RP	10.000 abs
Sicherheitsabstand:	
SC	1.000 ink

A value for the safety distance has to be entered.

Regarding the machining rotational direction, we can select between synchronism

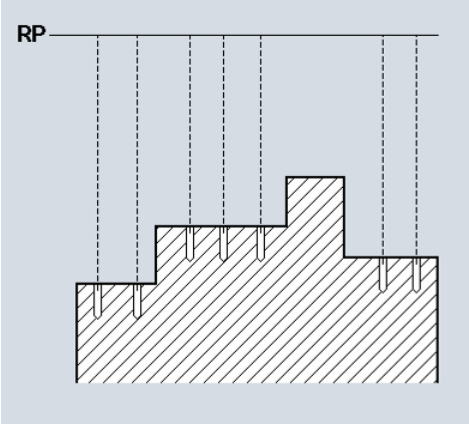
Gleichlauf

and reverse rotation

Gegenlauf

For the return motion, we can select between "return motion on the return plane"

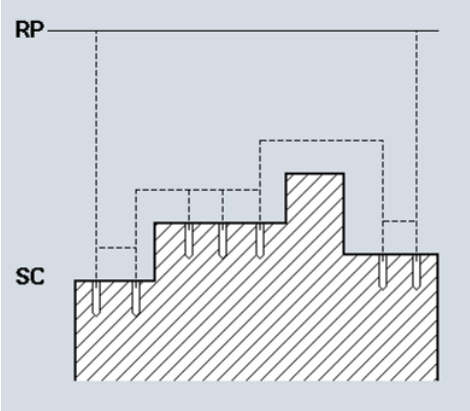
auf RP



and

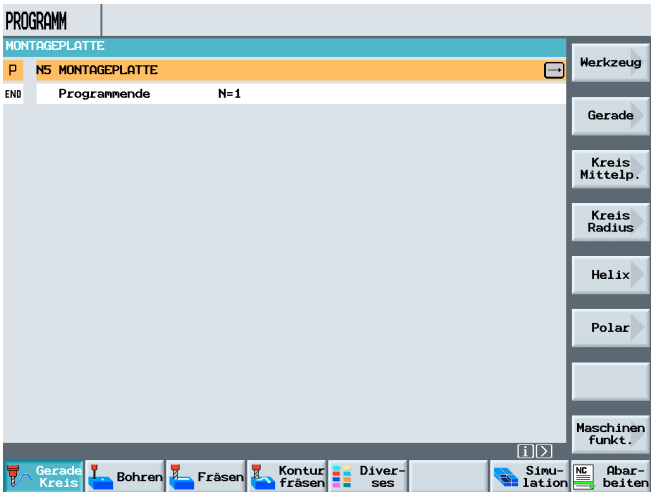
"return motion to safety distance".

optimiert

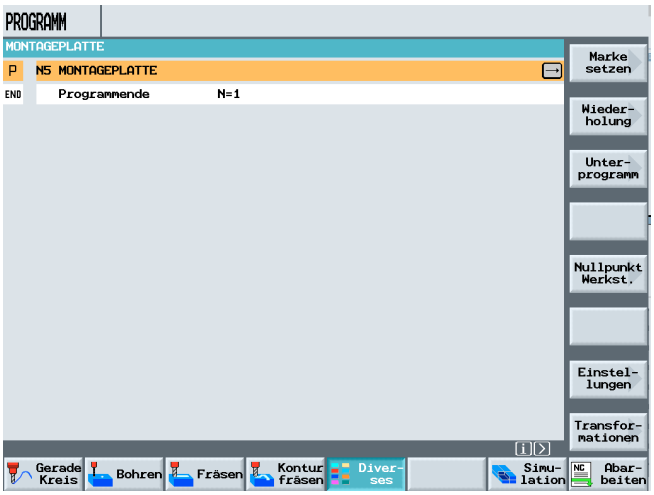


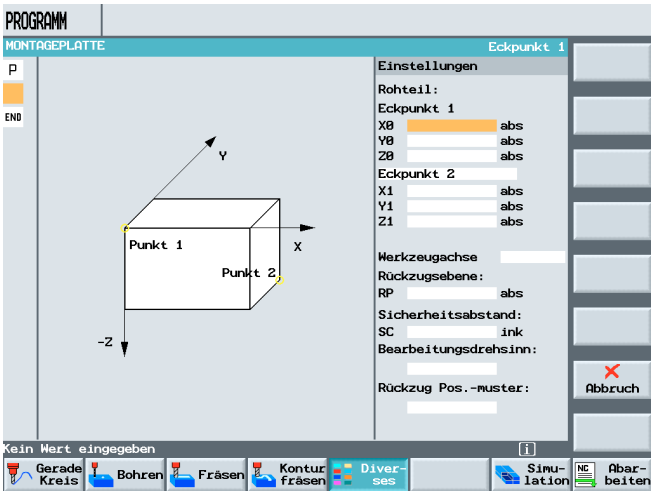
For each machining, a return motion and a safety distance have to be specified.

The data entered in the program header can be redefined any time in the program by pressing the softkey



and Settings

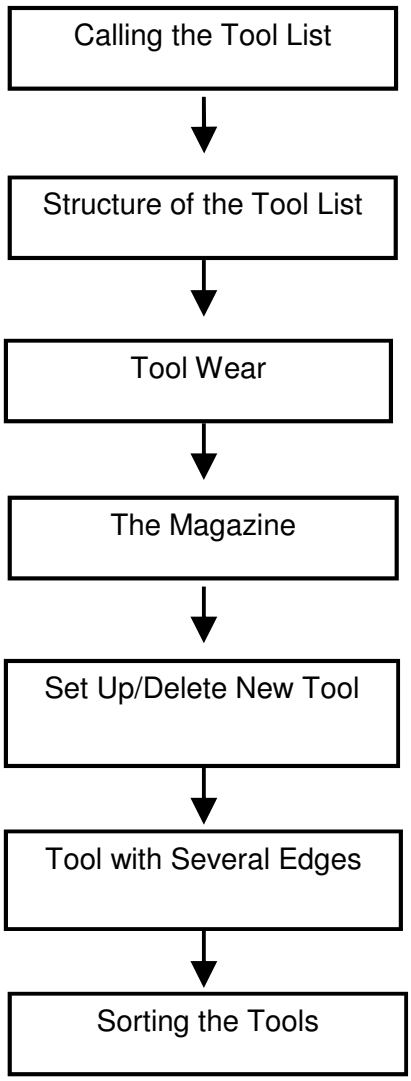




7 Tool Management – Milling

In this module, the structure of tool management is described with its individual operating and setting options.

Content of the Module:

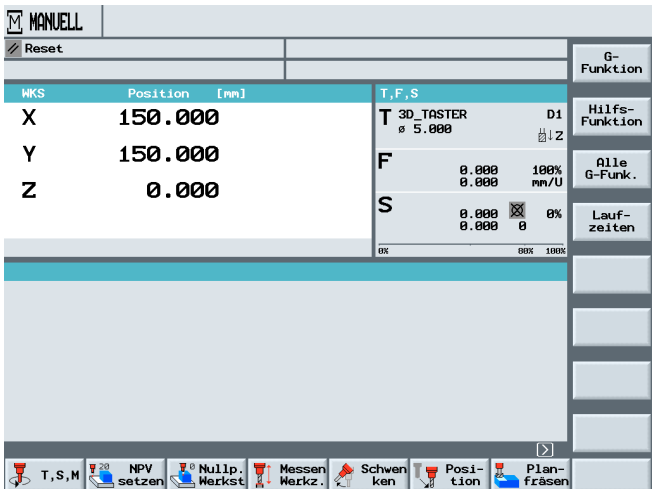


7.1 Calling the Tool List

After starting ShopMill, the operating area

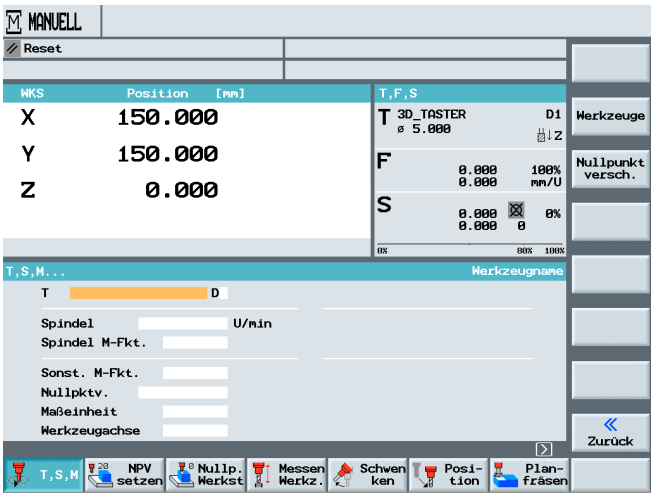
is active.

By pressing the softkey

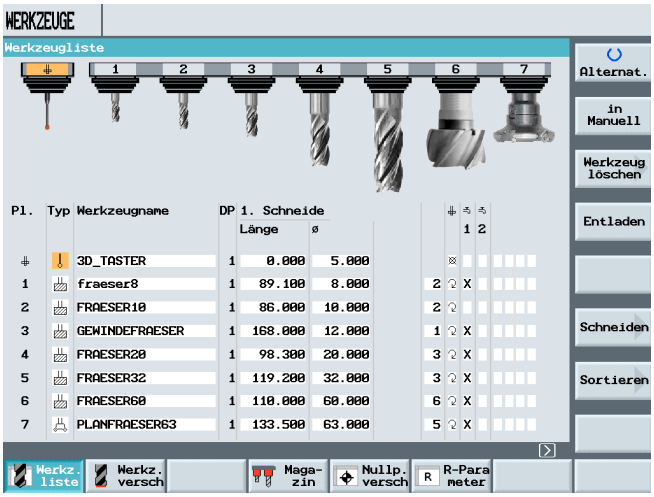


and Tools,

, in the area Tools

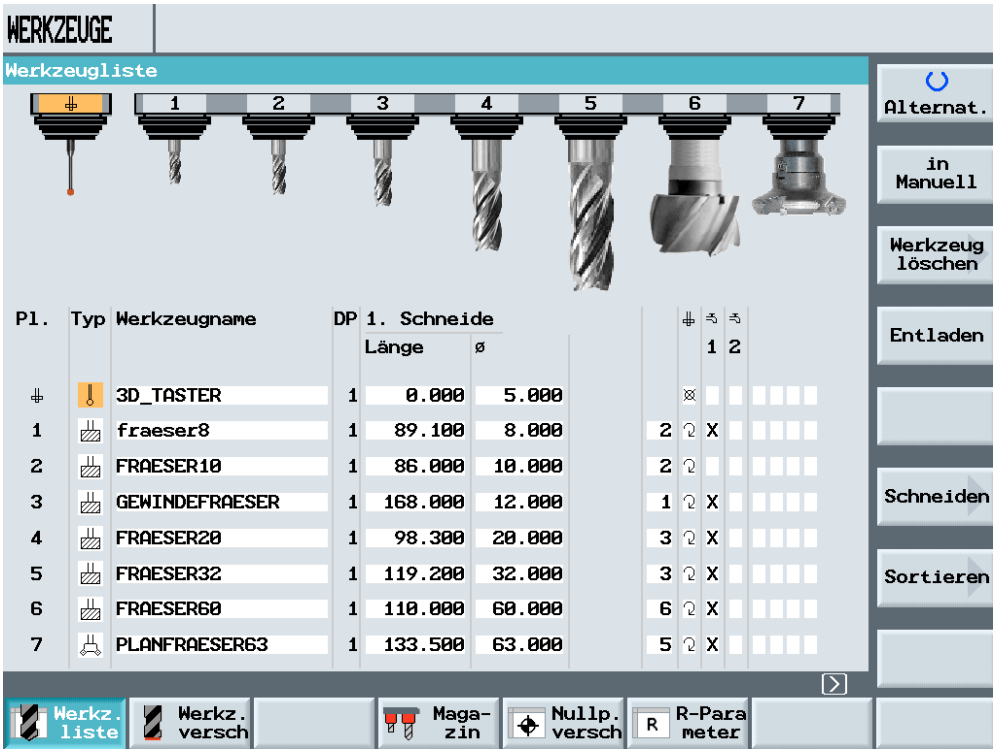


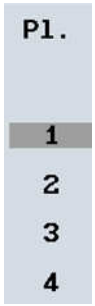
the Tool List



Is called.

7.2 Structure of the Tool List





The Location Number:

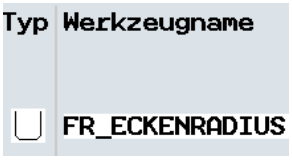
The location number describes the magazine location. If there is a tool listed behind the location number, it is active; that is, it is present in the magazine.

Tools that don't have a location number are not active in the magazine.
They are located in the "drawer", or in the manual magazine.

The Type:

Here, a symbol is assigned to the respective tool type.

The following tool types with their corresponding symbols are available:

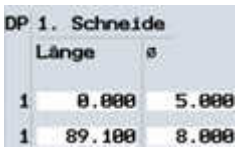


<u>The Tool Name:</u>

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

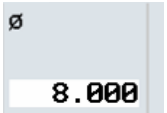
<u>The Duplo Number:</u>

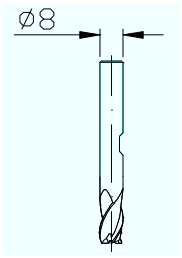
If an additinal tool is set up with a name that already exists, it becomes a duplo tool.



<u>The Edge Length:</u>

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





<u>The Diameter:</u>
Here, the tool diameter is entered into the list.

<u>Number of Edges:</u>
Here, the number of tool edges is entered.

The Spindle's Rotational Direction:

In the case of tools, the spindle's rotational direction refers to the tool spindle.

Right: 

Left: 

and spindle stop 

Cooling Water Supply:

Under ShopMill, the inside as well as the outside coolant supply can be activated.

7.3 Tool Wear

In the area Tools

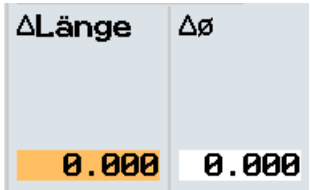
-by pressing the softkey

- the input fields for Tool Wear

are activated

Here, the edge of a tool

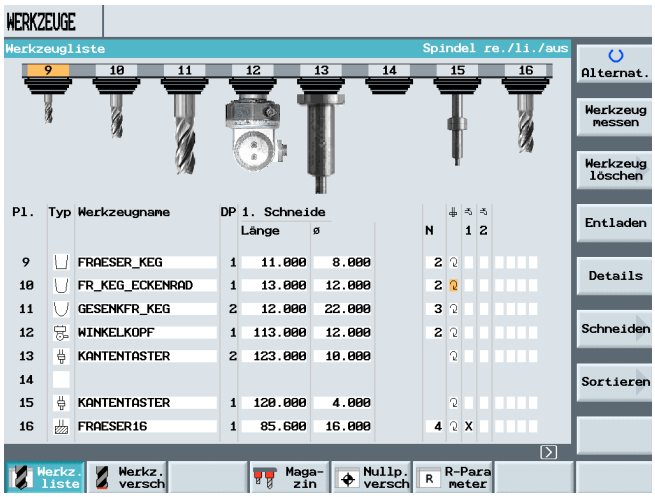
can be assigned a wear value for the length and the diameter

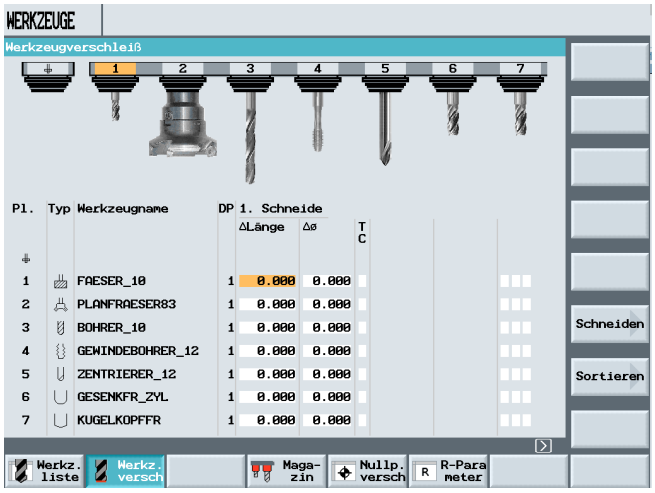


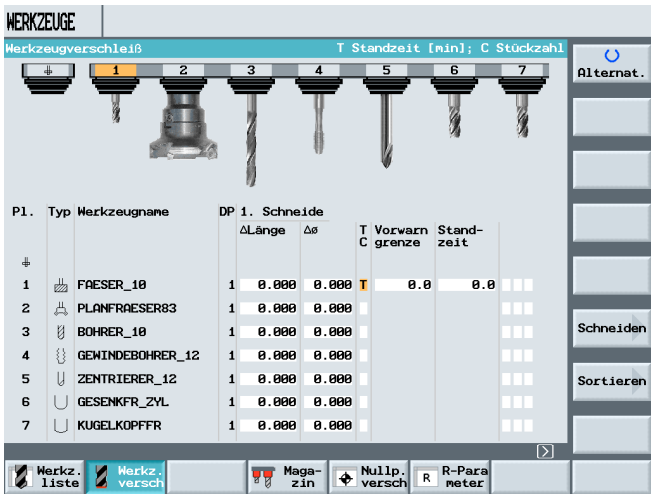
The duration of tool operation can be defined by means of its service life

or the

number of units.







After reaching the prewarning limit, a message is displayed that the end of service life or the number of loads will soon be reached.

Tools can be defined as “blocked“

or as “oversize“.

7.4 The Magazine

7.5

In the area

WERKZEUGE

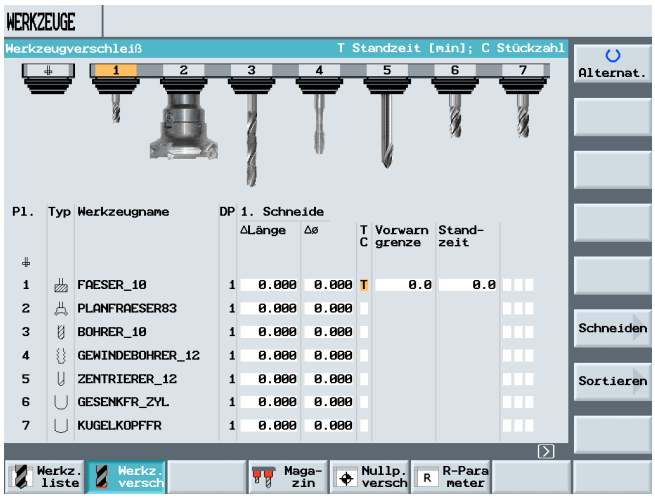
-by pressing the softkey

Maga-zin

- the iput fields for the

Magazin

are activated.



Here, a magazine location can be blocked.

G

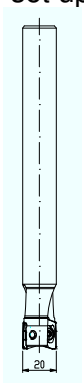
The tool status:

Werkz. zustand

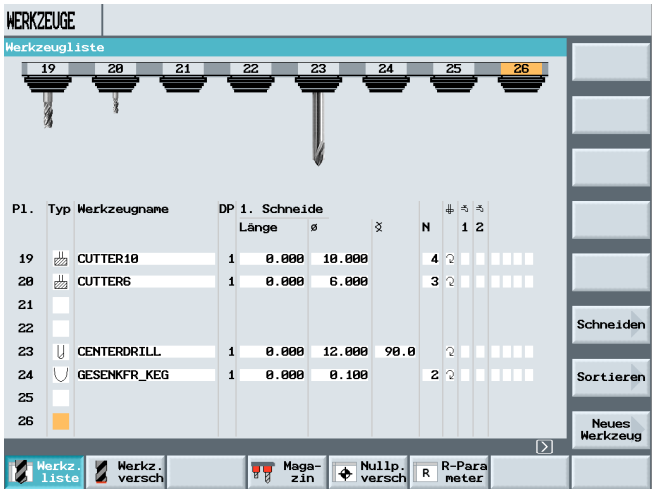
that is, when a tool is “blocked“ or “oversize“, is displayed here.

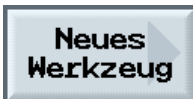
Setting Up/Deleting a New Tool

As an example, the following tool will be set up.



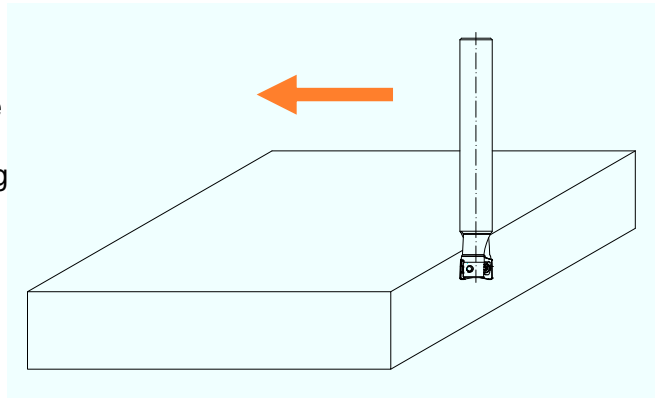
In this case, the cursor has to be on a free input field of the *tool list*.
By pressing the softkey





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

Example:

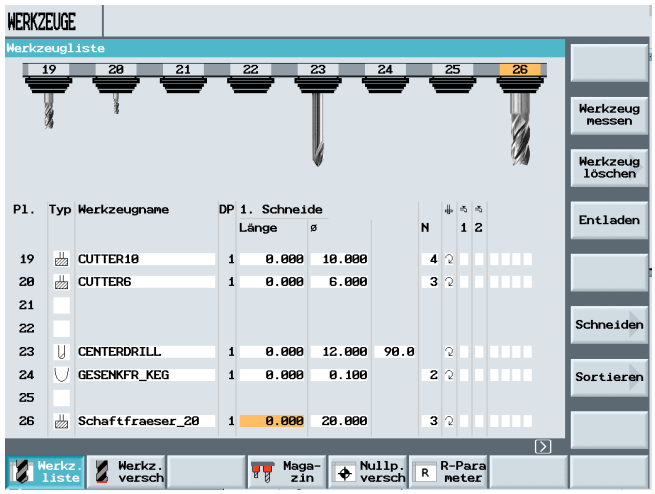


the name of the tool will be defined.

Next, the diameter

as well as the number of teeth and the rotation direction are specified

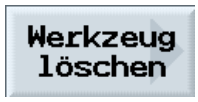
The tool can now be measured in its length.

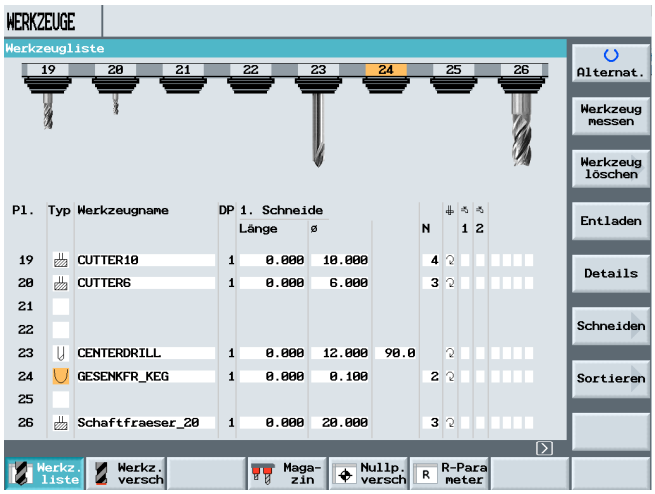


When machining with tools of the same type, unique names should be assigned, since duplo tools are generated from tools with the same name, without a warning.

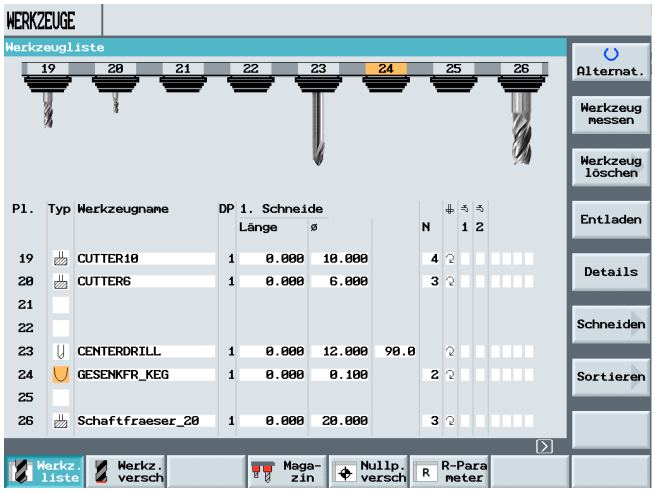
To delete a tool from the tool list, select the location with the tool.

By pressing the softkey





the following softkeys are diesplayed:



By pressing the softkey

the selected tool is deleted from the tool list.

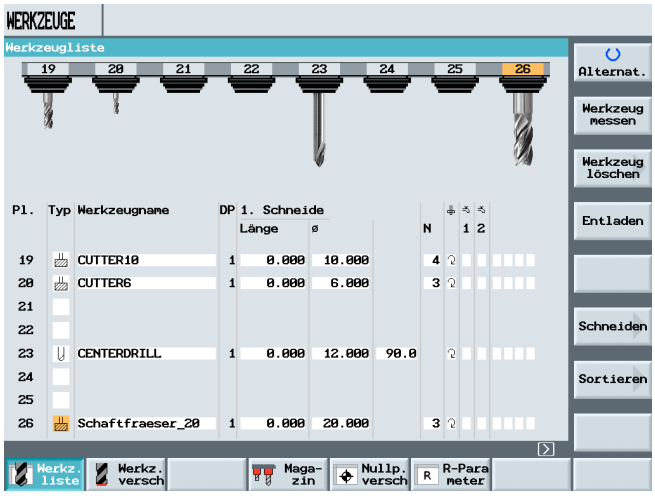
Take note of the machine manufacturer's data. In the case of some manufacturers, an active tool can not be deleted.

7.6 A Tool with Several Edges

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

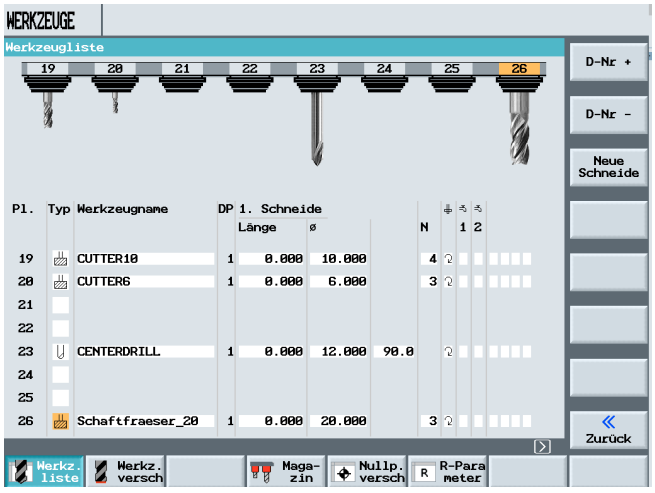
By pressing the softkey

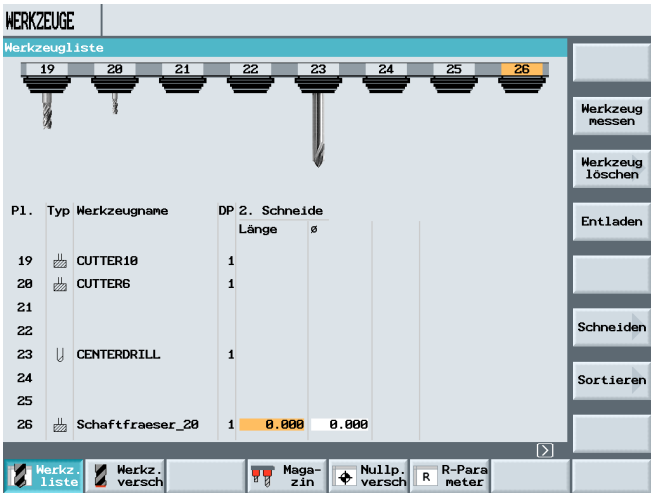
the input screen form is displayed where several edges can be defined.



By pressing the softkey

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

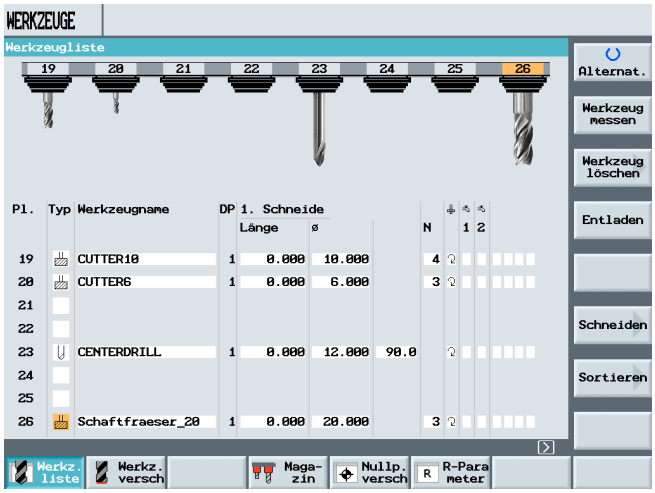




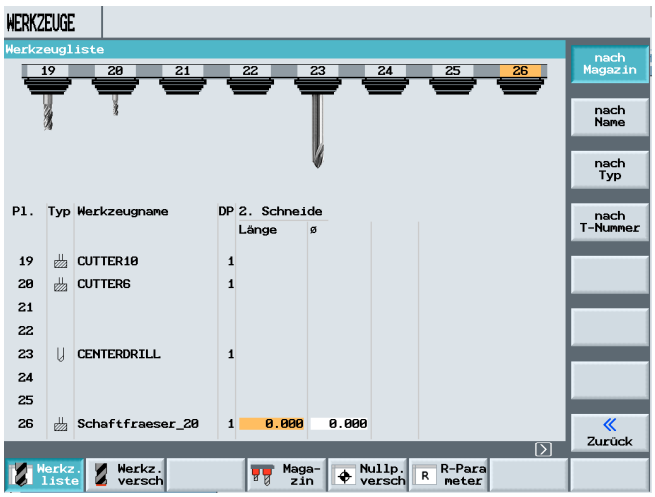
7.7 Sorting the Tools

When calling the tool list and pressing the softkey

the four options for sorting the tools are displayed.

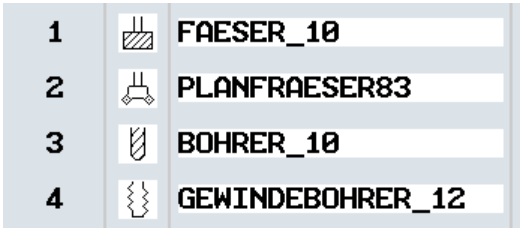


The sorting that is active is shown darkend.

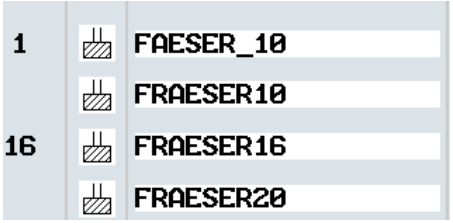


For sorting according to

the tools are shown in the ascending mode according to their magazine location.



When sorting accoding to

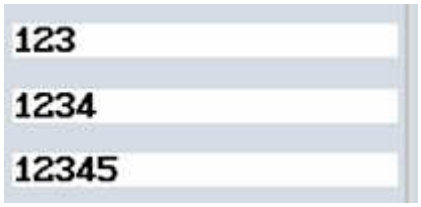


the tools are shown in the alphabetical order of the name.

When sorting

the tools are sorted according to the tool type.

When sorting



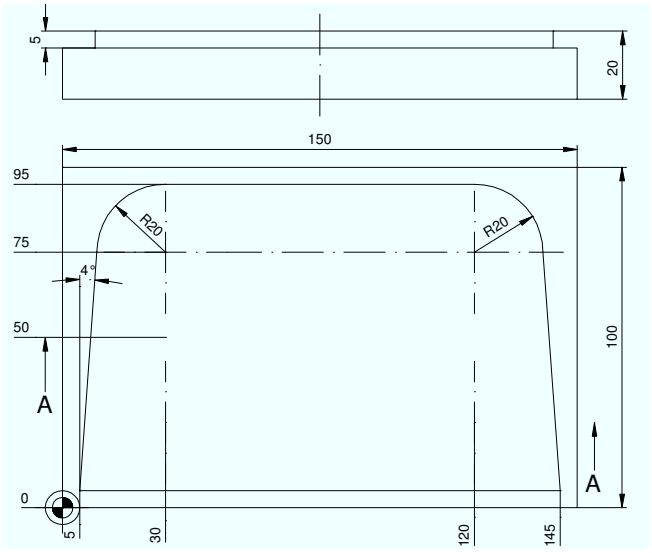
tool names that are defined by numbers are sorted in the ascending mode.

8 Programming Example: Contour Programming

The contour calculator and subsequent machining are described, based on an example.

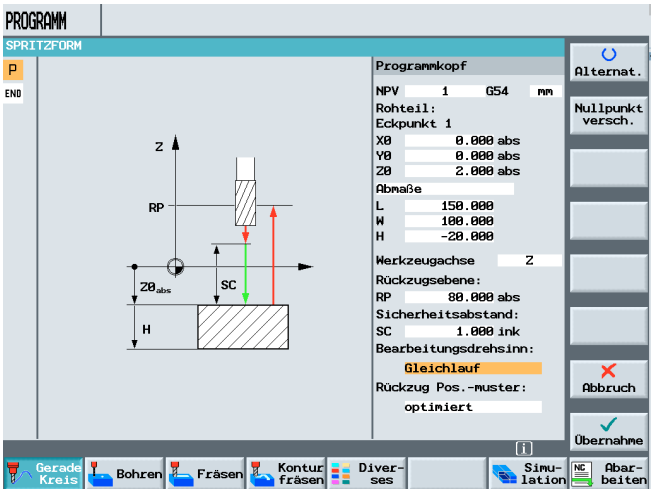
8.1 Example of Contour Programming

A ShopMill program is to be created for this workpiece.



After setting up a program with the name “Mounting Plate“,

the inputs are made in the program header.

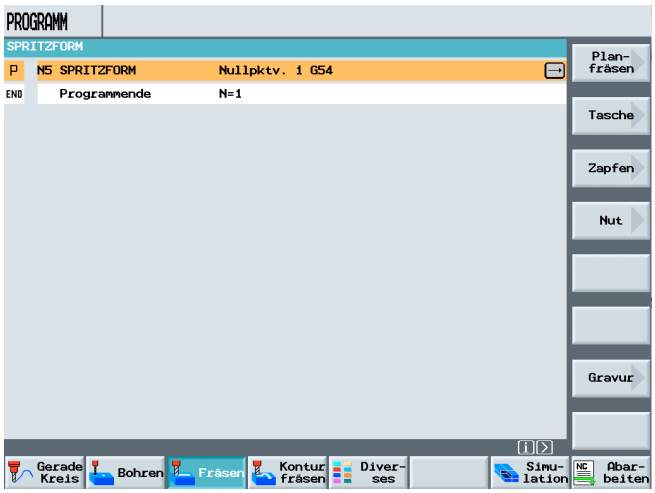


NPV 1 G54 mm
Rohteil:
Eckpunkt 1
X0 0.000 abs
Y0 0.000 abs
Z0 2.000 abs
Abmaße
L 150.000
W 100.000
H -20.000
Werkzeugachse Z
Rückzugsebene:
RP 80.000 abs
Sicherheitsabstand:
SC 1.000 ink
Bearbeitungsdrehsinn:
Gleichlauf
Rückzug Pos.-muster:
optimiert

8.2 Face Milling

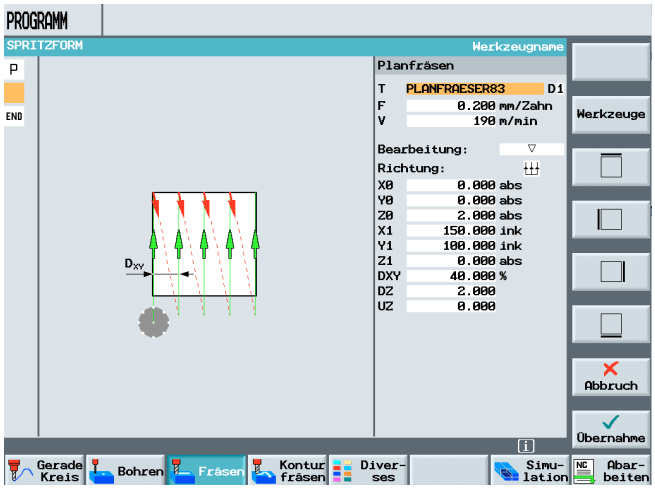
The workpiece surface is milled in order to obtain a level, clean surface. This is done with face milling.

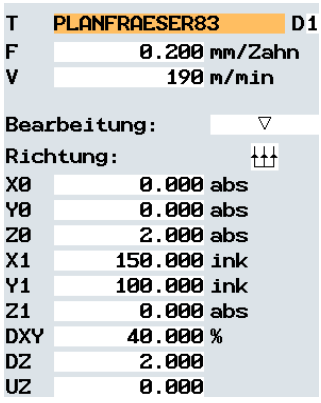
After accepting the program header,



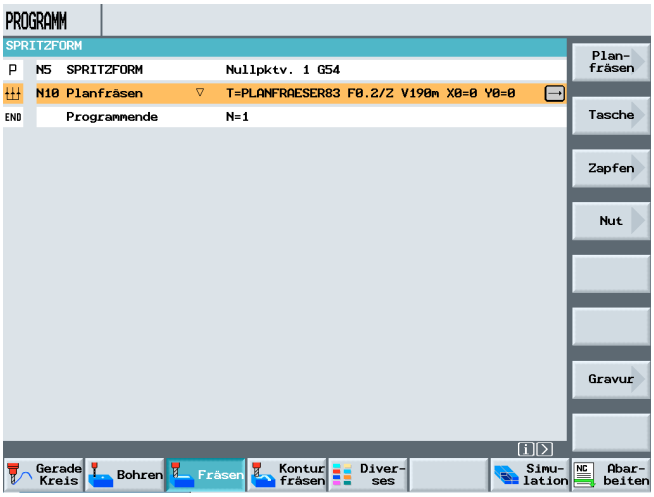
face milling is performed next.

After entering the values in the input screen form,





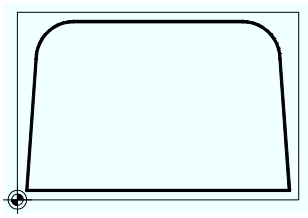
face milling is accepted into the machining plan.



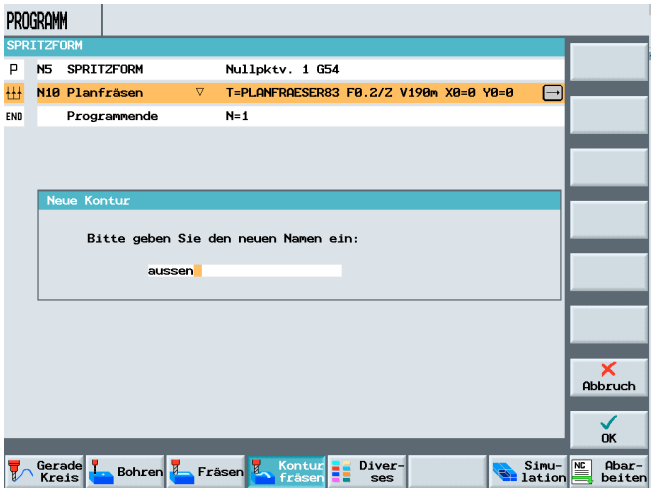
8.3 Contour Calculator

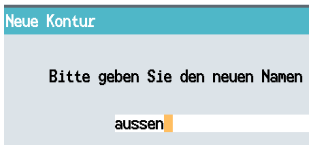
With the contour calculator, even difficult contours can be programmed relatively easily.

Next, the milling contour is described.

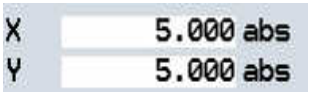


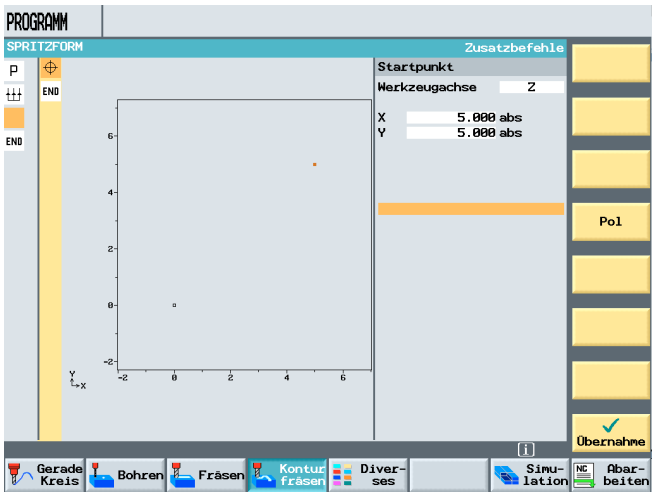
After setting up a new contour,



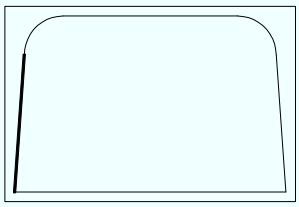


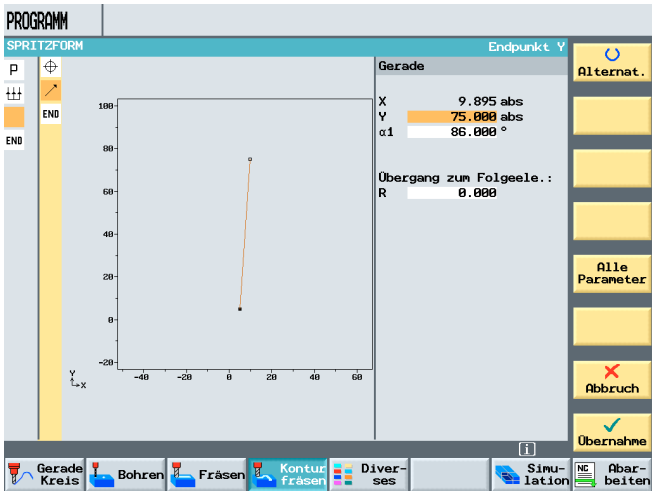
the starting point is defined in the contour calculator.

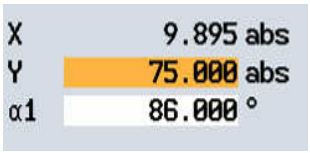




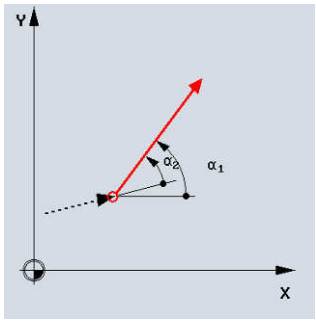
Then, the first contour element is programmed.



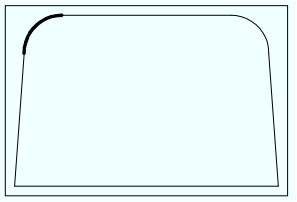




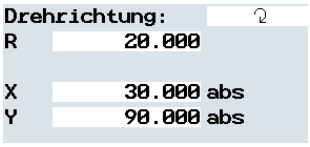
When entering the angle, the direction has to be noted.



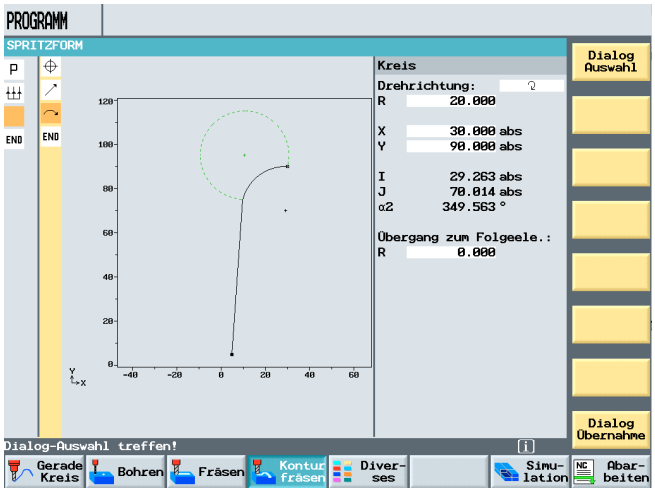
Next, the arc is programmed



After entering the values

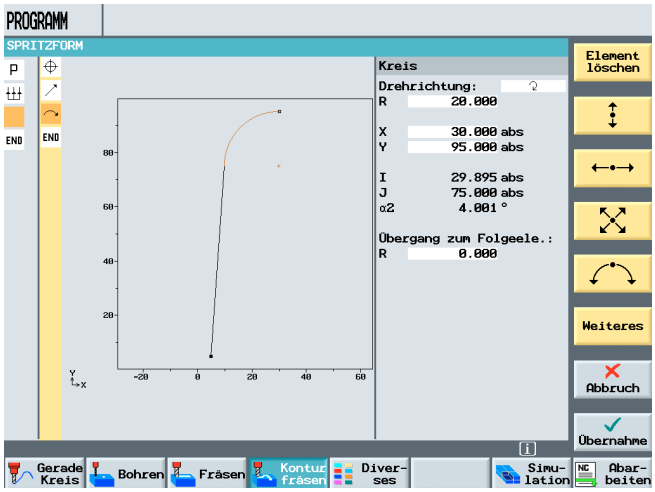


a dialog selection is displayed.



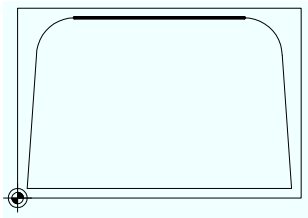
By pressing the softkey

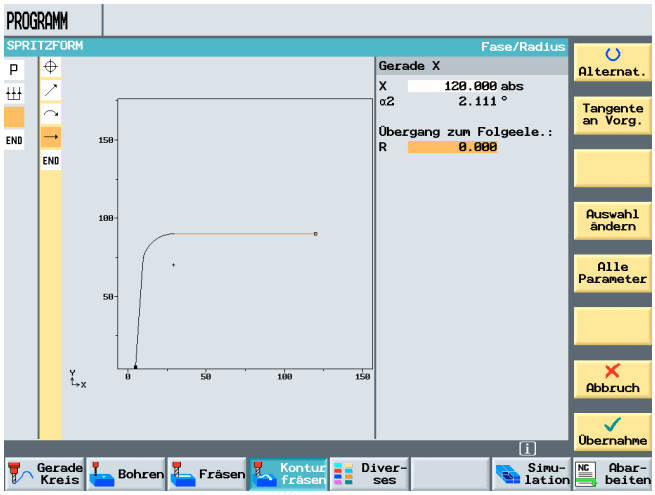
the selected element is accepted into the contour calculator.

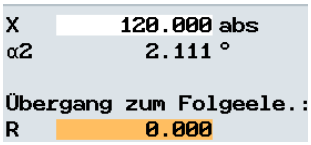


The direction of the element selected in the dialog can be changed any time by pressing the softkey

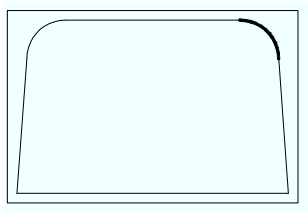
The next element is a straight line in X direction.

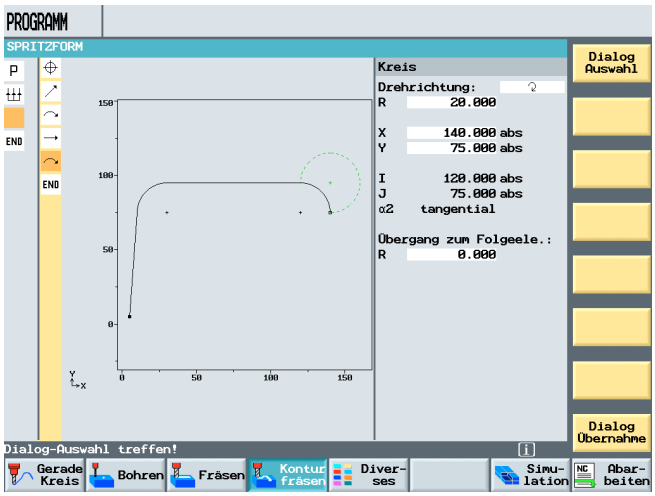






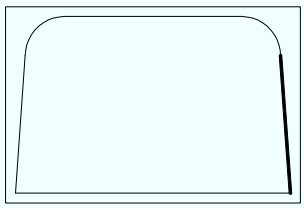
Then, an arc follows again.

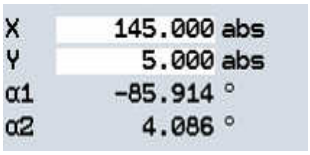


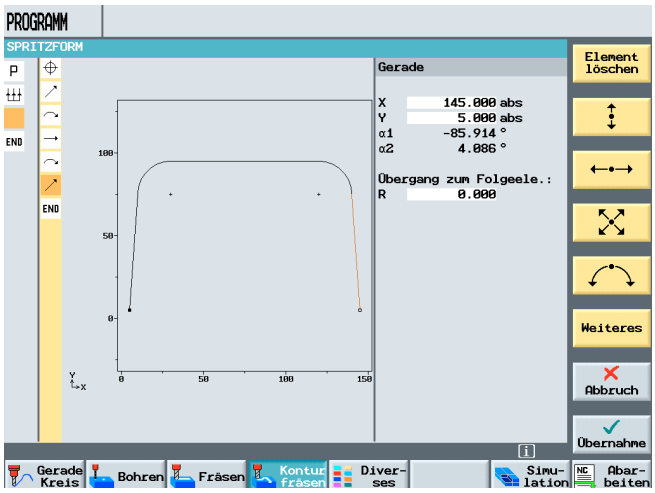


For this element also, a dialog selection is made which is accepted into the contour calculator by pressing the softkey

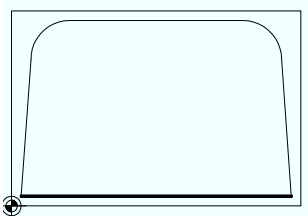
The next element is an oblique line whose end point is known.

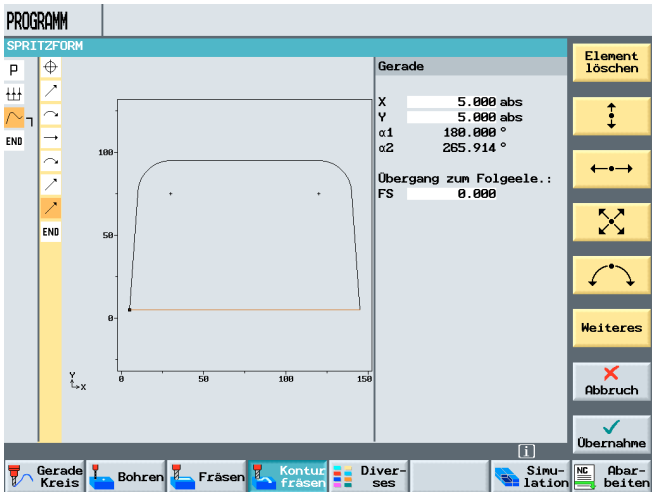






With the last contour element, a straight line, the contour is closed.





By pressing the softkey Additional

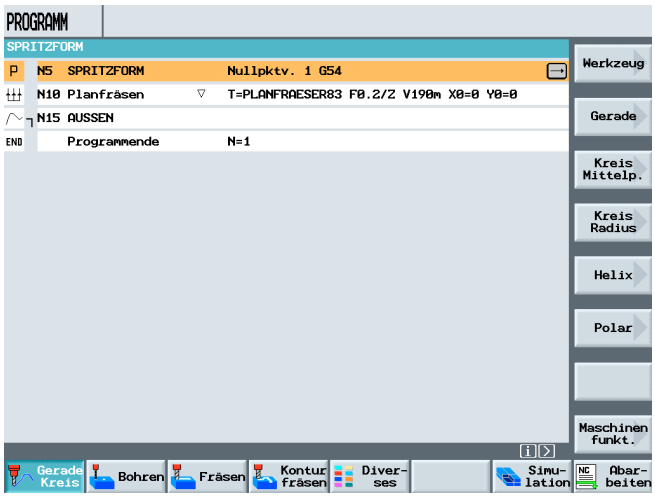
and Close Contour

the contour is completely generated.

8.4 Path Milling

After accepting the contour in the machining plan

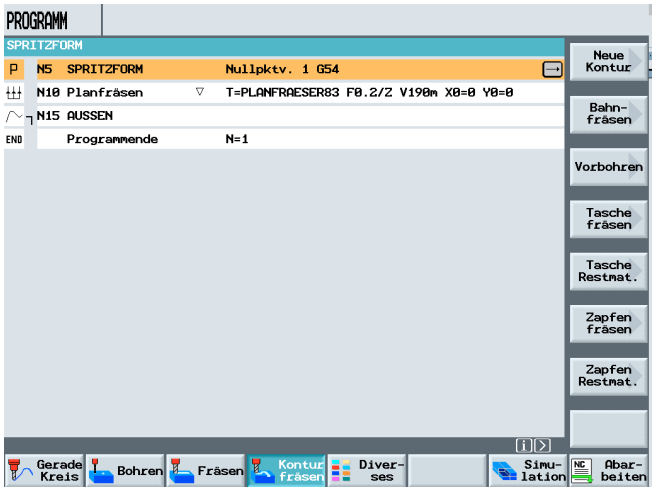
a bracket opens next to the symbol for the contour element.



Next, the machining process is described. Path milling is to be used for machining.

By pressing the softkey

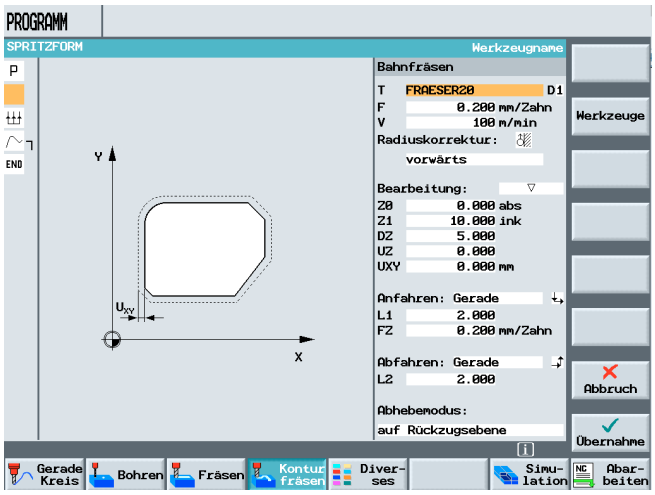
In the area "***Contour milling***", a corresponding input screen form is opened.

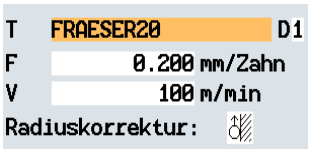


8.5 Forward-Backward

We can also machine against the programmed contour (backward).

After entering the tool, the technology and selecting the radius correction

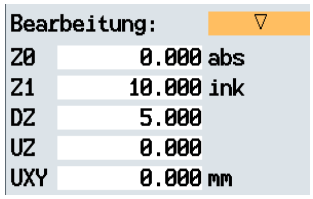




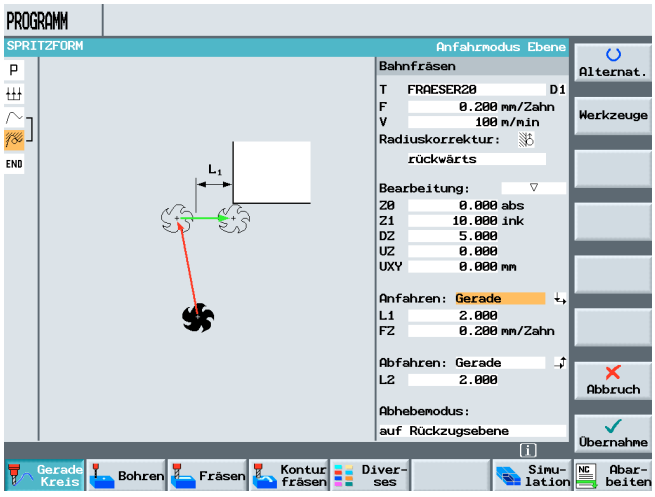
we can then specify the machining direction in reference to the contour.

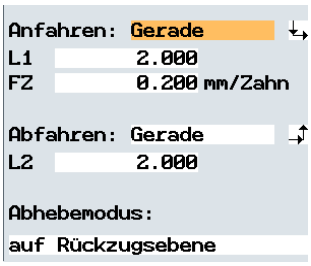
The contour is processed in the direction the contour was programmed.

The contour is processed against the contour's programmed direction. After entering the machining strategy,

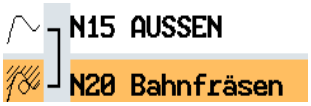


the ***approach and the return strategy as well as the retraction mode*** are defined.

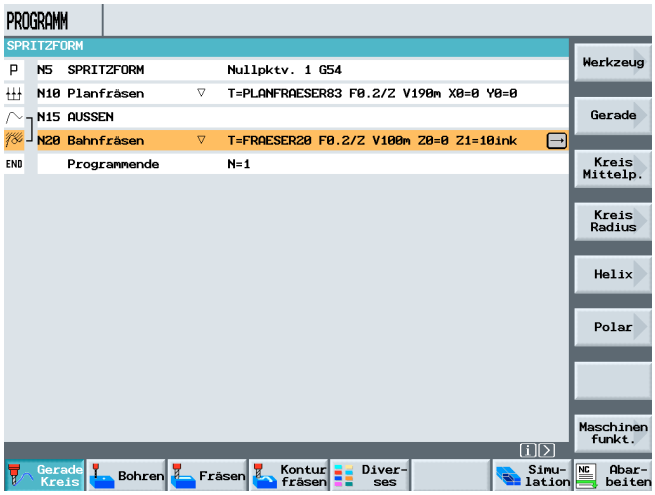




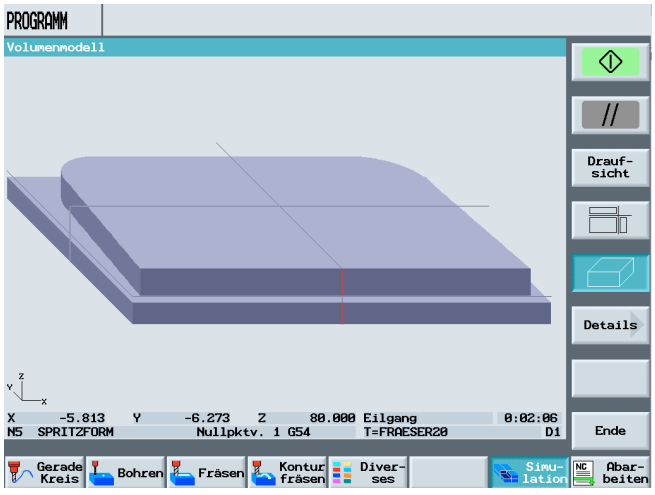
Accepting the machining into the machining plan



closes the bracket. A machining was added to the contour.



The program is done.



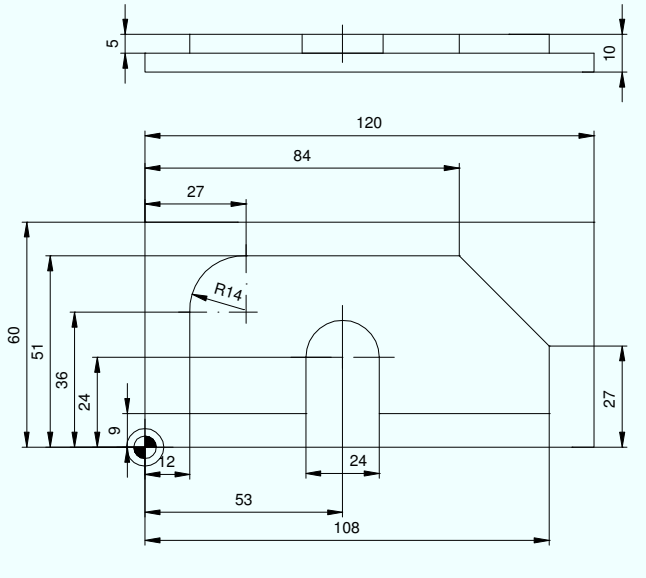
9 Programming Example Contour Spigot

Content of this module:

This module describes the standard milling cycle Contour Spigot, using an example. The contour remains, and the residual material is milled off.

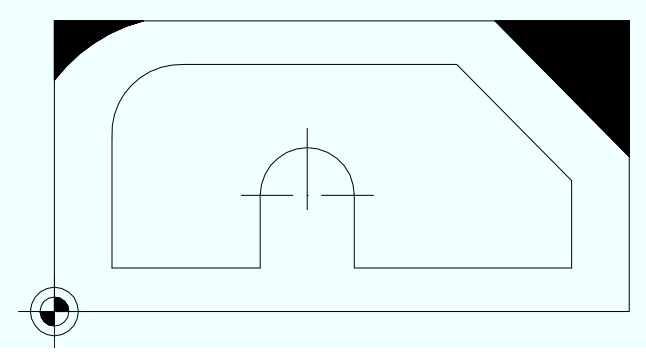
9.1 Contour Spigot Milling – Removing Residual Material

This example is programmed using ShopMill.

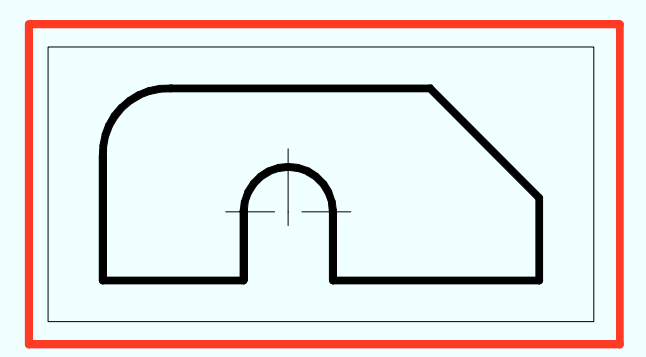


This is not possible with the path milling cycle, since as a maximum, a 10 milling tool can be used.

Residual corners remain.



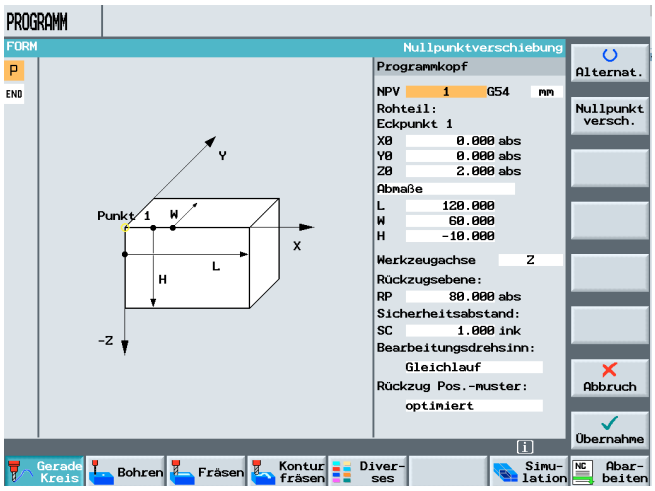
For this reason, two contours have to be described: one that surrounds the raw part, the socalled “limit contour“ and the actual contour that jumps out; that is remains as an “island“.

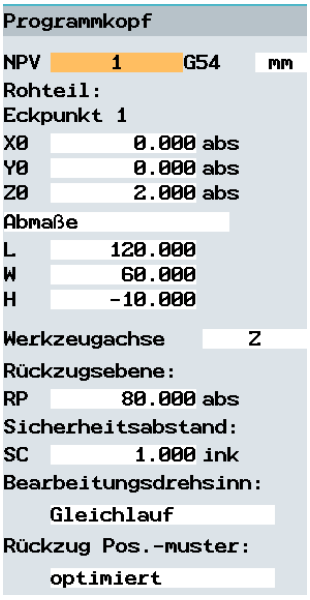


9.2 Face Milling

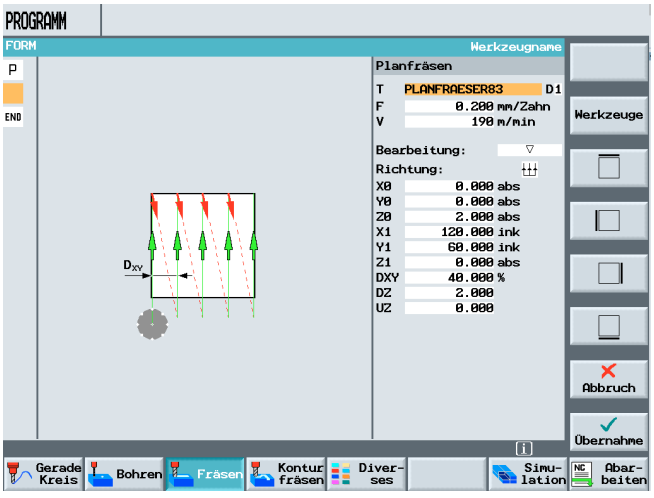
After setting up a new program

and making the input in the program header



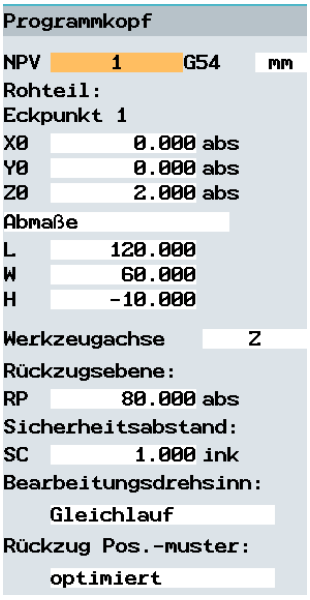


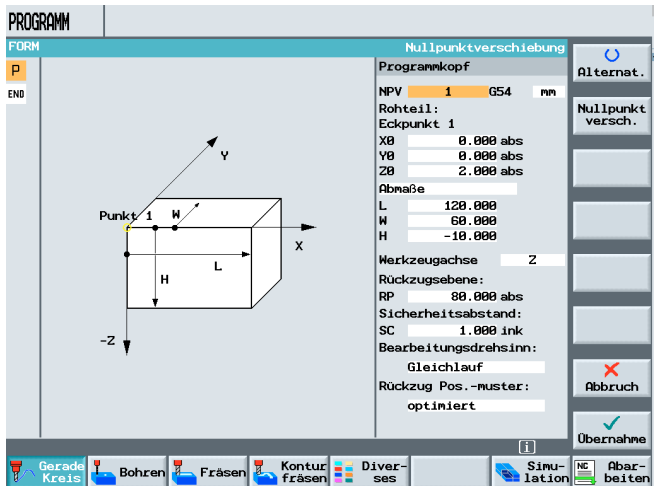
face-milling the workpiece surface
<<something missing like: a new contour is set up?>>



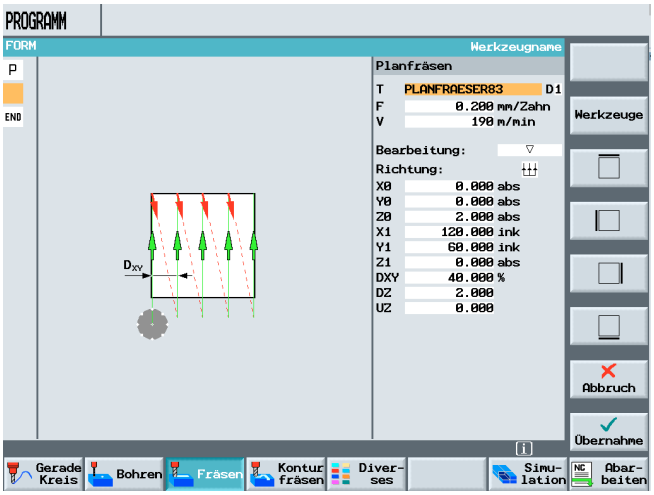
After setting up a new program

and making the inputs in the program header



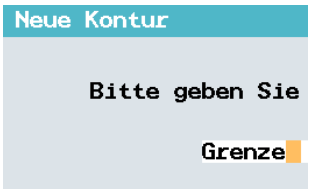


face-milling the workpiece surface,



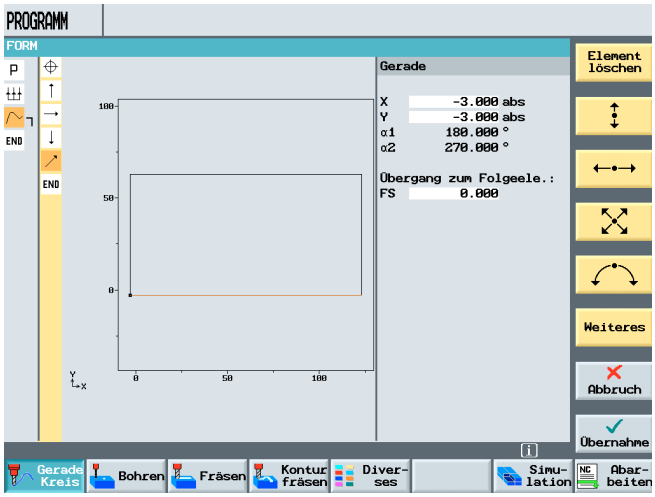
a new contour is set up <<same page twice?>>.

9.3 First Contour Limit-Contour



This contour, a little larger than the workpiece, will later describe the machining limit.

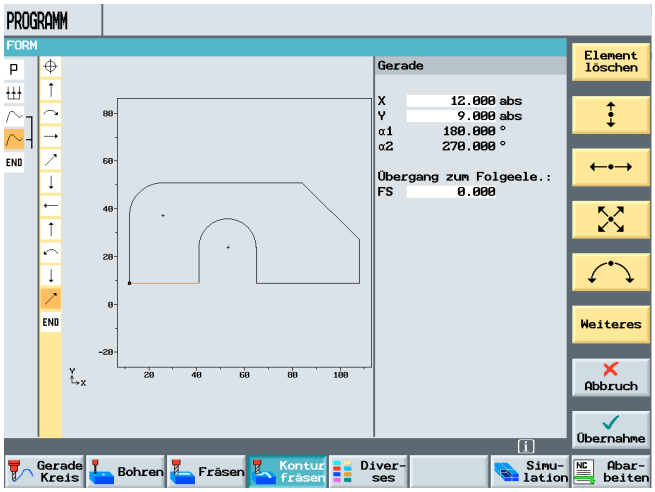
The initial machining continues to be done with the facing tool.



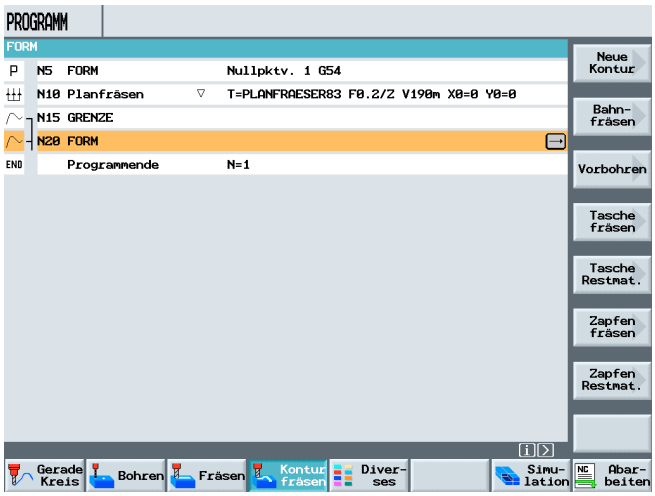
In the case of spigot milling, the “limit contourr“ can be angular. When describing a contour pocket, the contour corners should have a radius. The corner radius should be at least as large as the radius of the largest tool.

9.4 Second Contour: Actual Contour Spigots

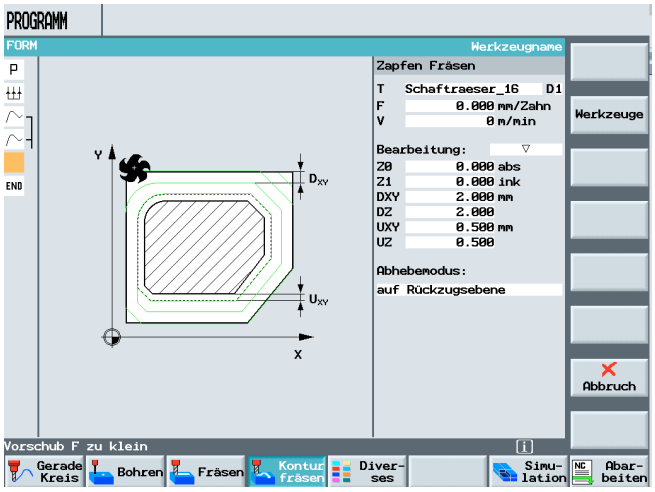
Next, a second contour is programmed.



After accepting the contour in the machining plan, the cycle for contour spigot milling is opened by pressing the softkey

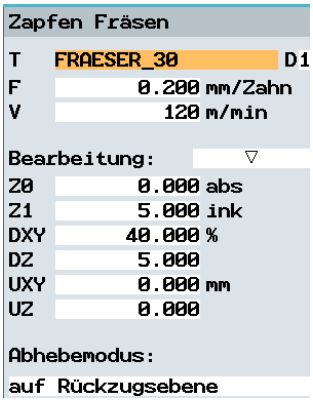


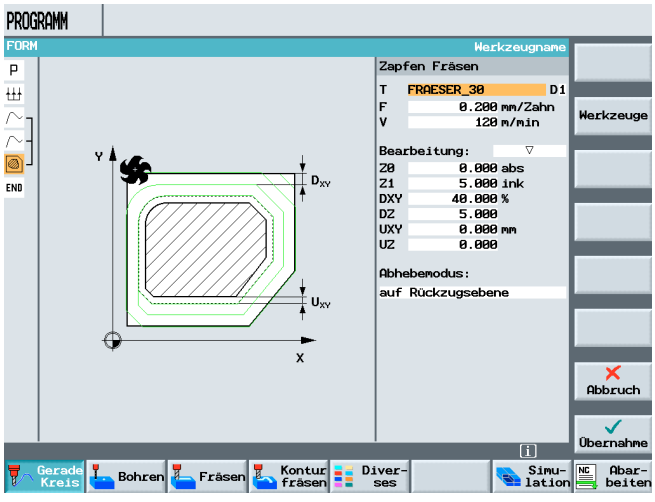
The strategy is that we always start outside of the workpiece when starting machining.



By machining from the outside to the inside, tough and hard materials can also be machined in this cycle.

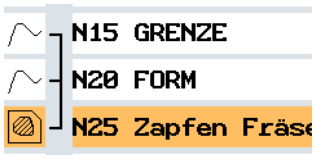
Then, the values are entered.

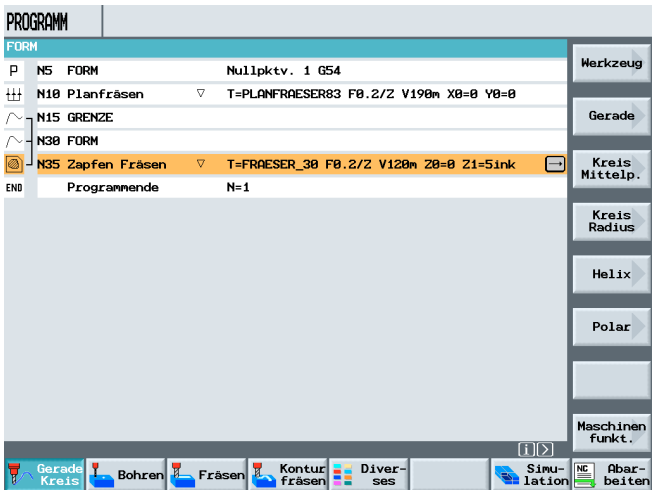




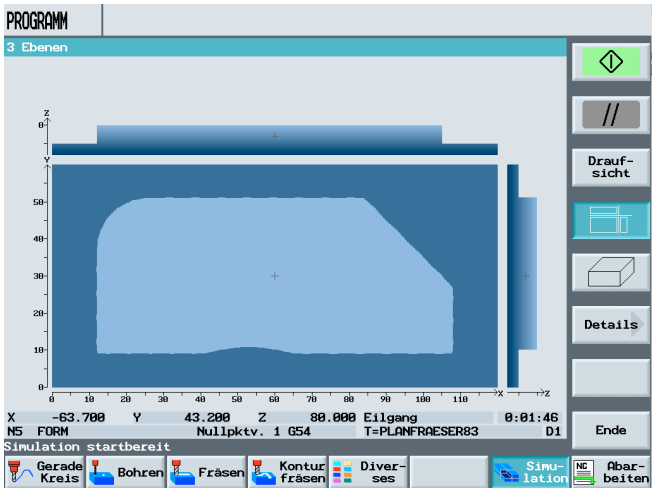
9.5 Spigot – Residual Material

The values that were entered are accepted into the machining plan. Thus, the bracket with the two contour elements is closed.



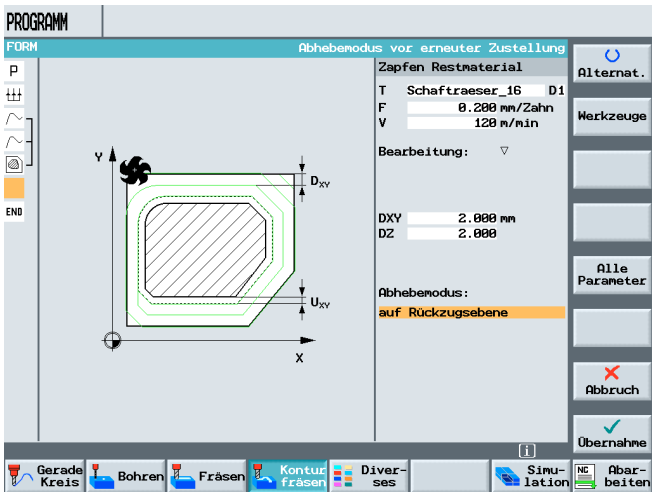


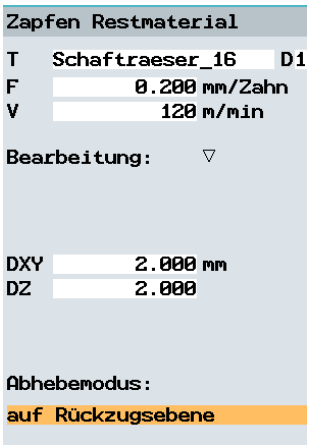
Since based on the milling tool diameter of 30mm the workpiece can not be completely manufactured, another machining cycle follows; socalled “residual material processing“.



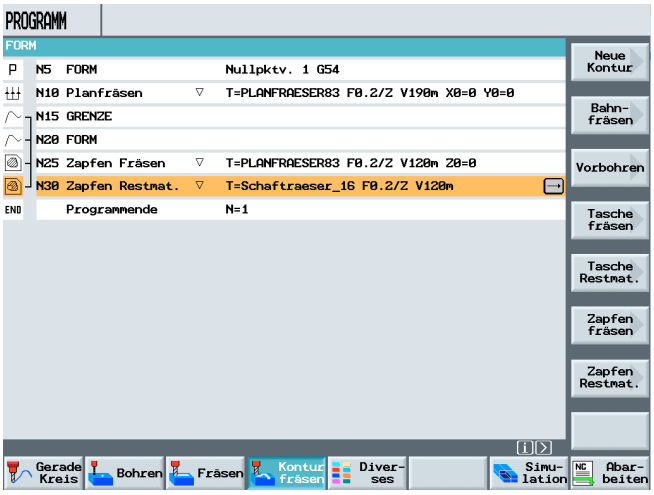
Pressing the softkey

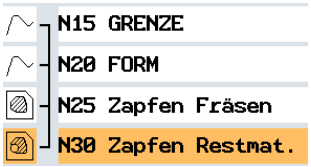
opens the cycle for residual material machining.
After selecting a smaller milling tool and inputting the values,



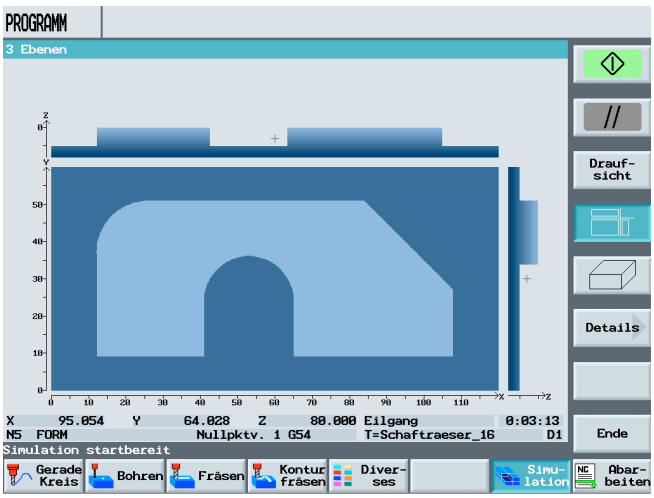


this cycle is accepted into the machining plan.
This cycle is added to the bracket.





The program is done.

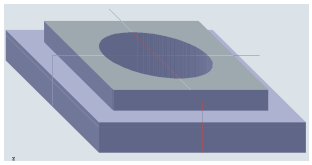


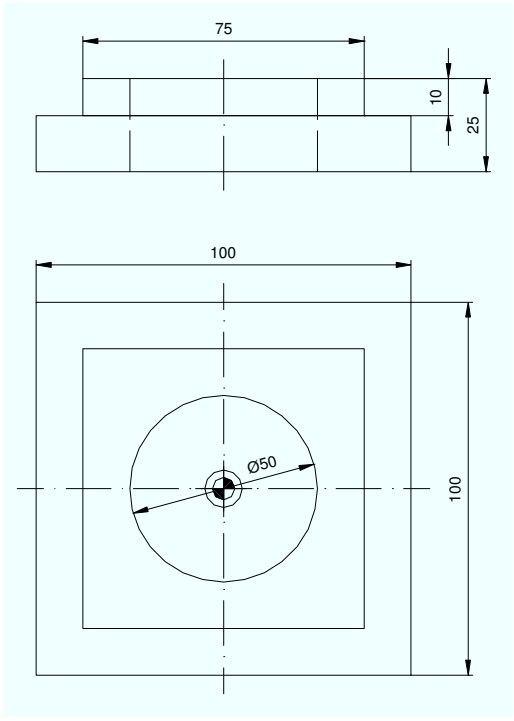
10 Programming Example Standard Milling Cycles

Machining with the standard milling cycles under ShopMill is described, using an example
.

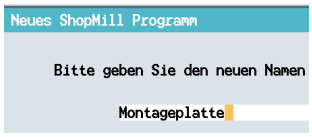
10.1 Programming Example for Milling Cycles (Rectangular Spigot, Circular Pocket)

This workpiece “Mounting Plate “ is to be generated using the milling cycles

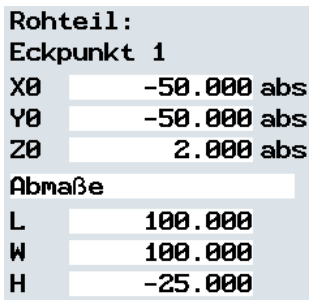




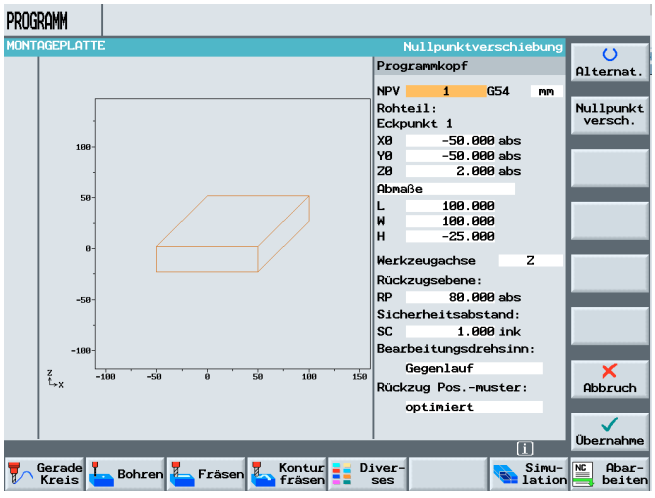
After setting up a program named "Mounting Plate“



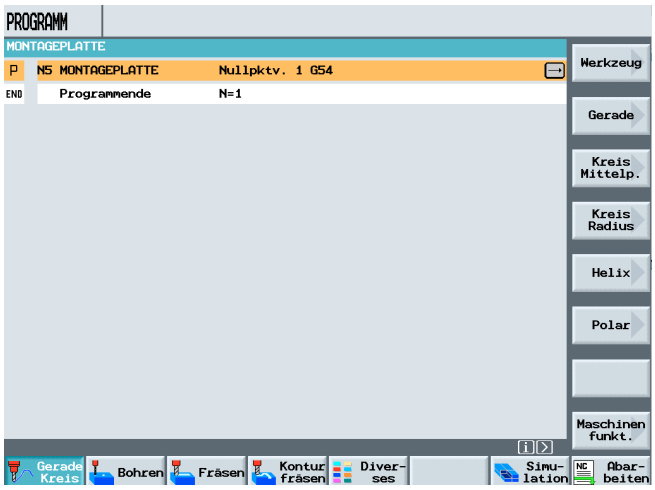
the raw part for the simulation is entered, after entering the zero point.



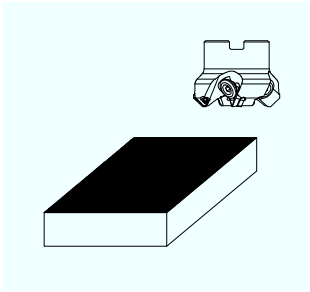
After entering the additional values in the program header



and accepting them into the machining plan,

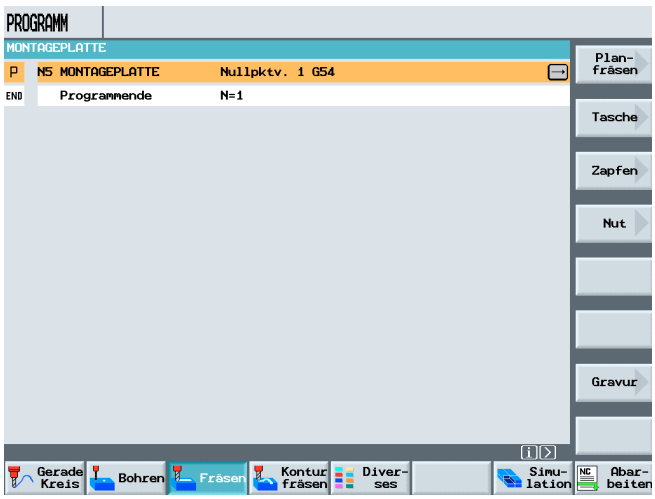


the next step consists of facing.



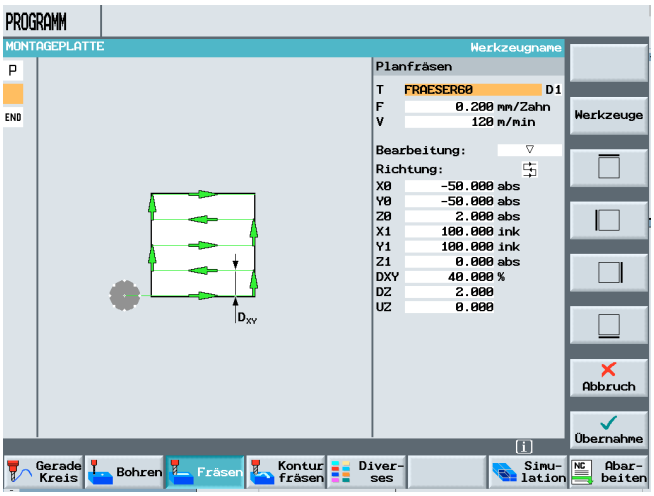
Pressing the softkeys

and

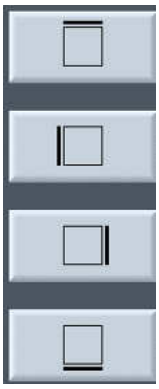


10.2 Face Milling

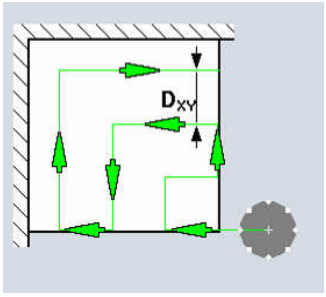
opens the input screen form for face milling.



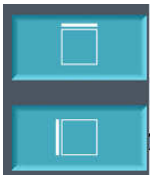
Using the four vertical softkeys



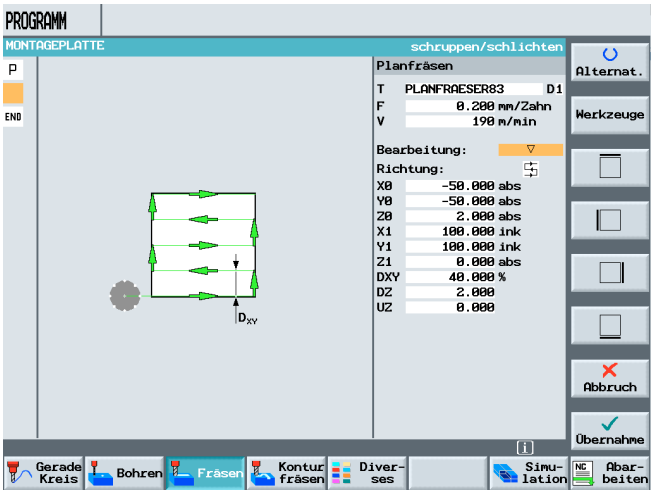
machining limits can be defined.

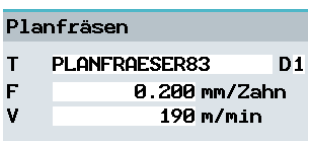


Example:
Activated limits



We don't have to define a machining limit in this example. After selecting the tool and the technological data,



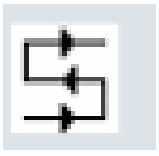


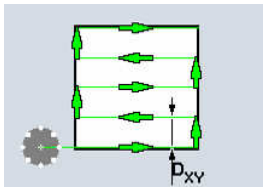
next Rouging,

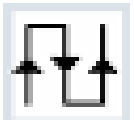
or Finishing

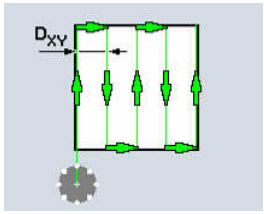
is selected.

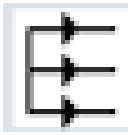
Then, we select the machining strategy.

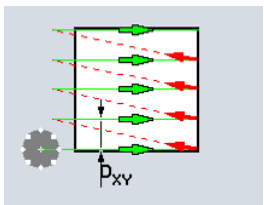


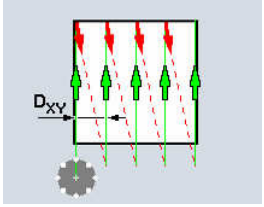




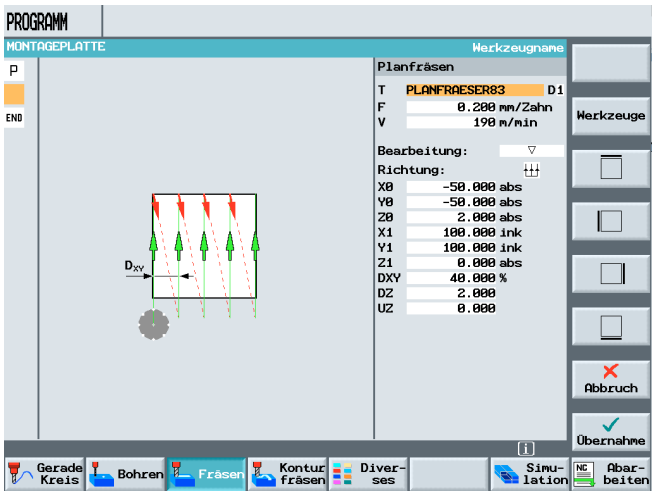




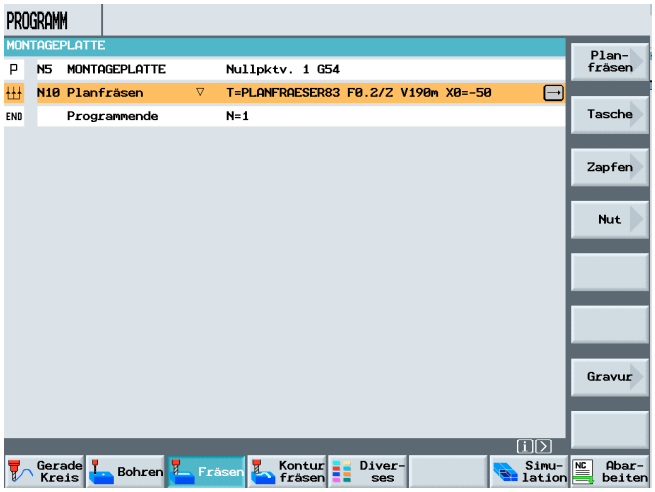




After entering the values, face milling is accepted into the machining plan

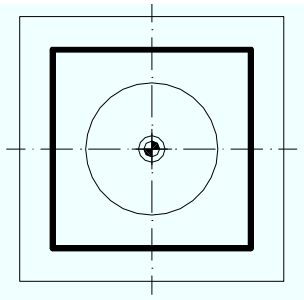


by pressing the softkey



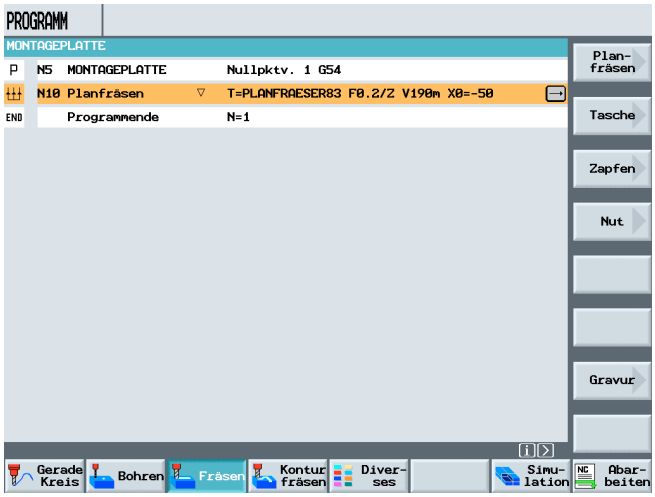
10.3 Rectangular Spigot

Next, the rectangular spigot is programmed.

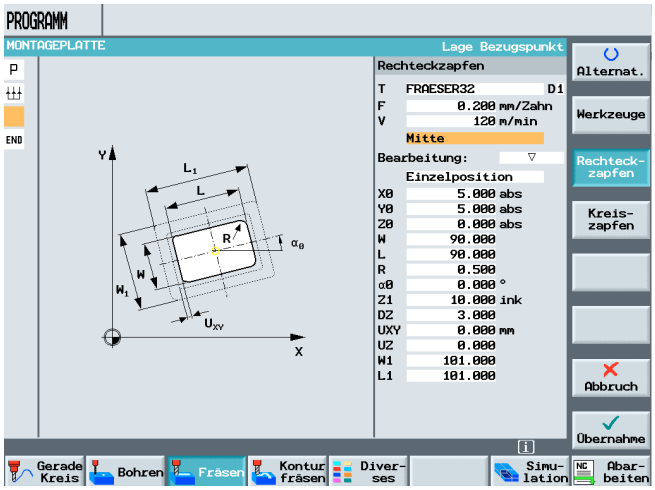


Pressing the softkey

in the area Milling and selecting



opens the input field.

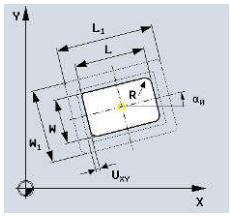


After selecting the tool with the corresponding technology,

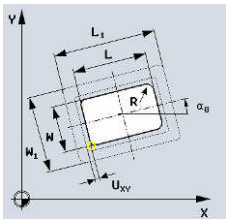
T	FRAESER32	D1
F	0.200	mm/Zahn
V	120	m/min

next the reference point of the reactangular spigot is defined.

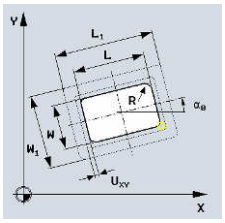
Mitte



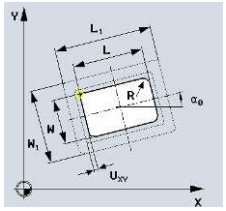
unten links



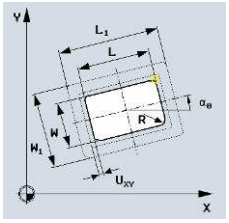
unten rechts



oben links



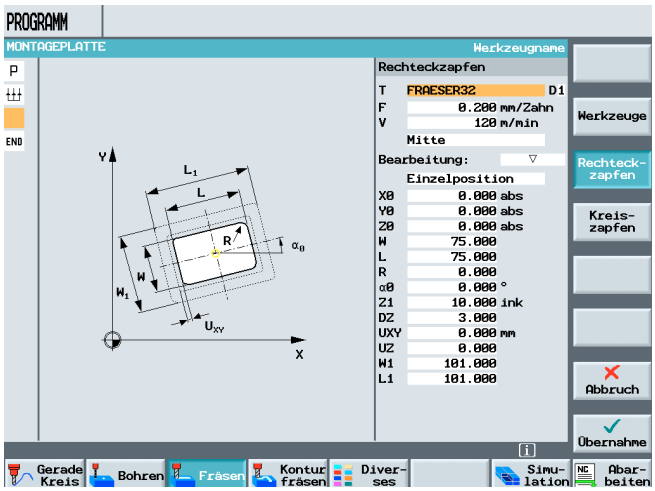
oben rechts



After selecting the position

Mitte

the spigot and the machining strategy are described.

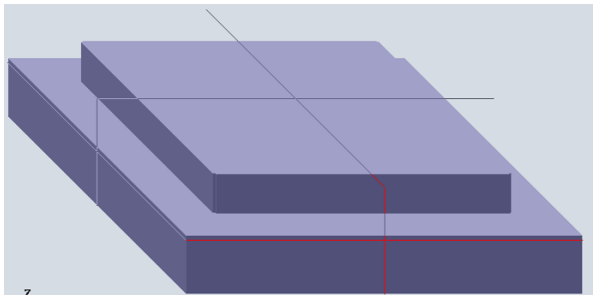


Mitte		
Bearbeitung:	▽	
Einzelposition		
X0	0.000	abs
Y0	0.000	abs
Z0	0.000	abs
W	75.000	
L	75.000	
R	0.000	
α0	0.000	°
Z1	10.000	ink
DZ	3.000	
UXY	0.000	mm
UZ	0.000	

Finally, the raw part spigot is specified that is to be machined.

W1	101.000
L1	101.000

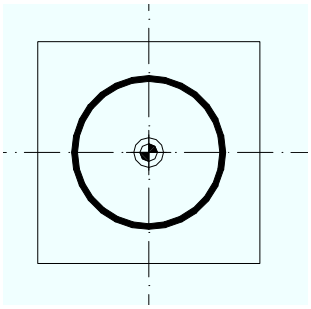
The spigot is programmed.

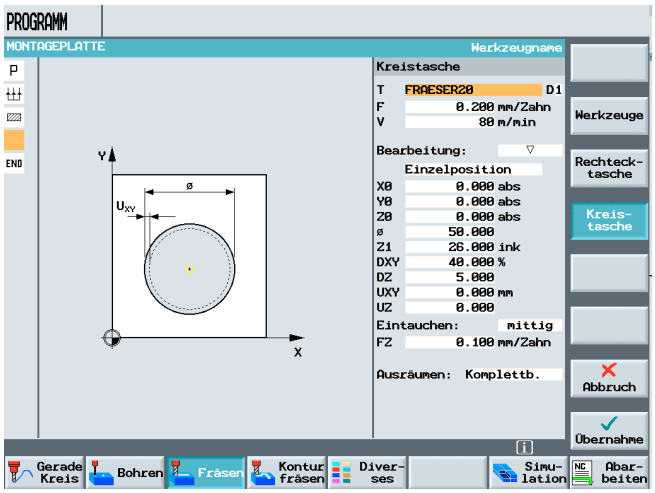


The material between the raw part spigot and the finished part spigot is machined in a lateral feed setting.
If the feed setting is too large, the completed spigot should be programmed in several steps.

10.4 Circular Pocket

Finally, the circular pocket is programmed.





By selecting the cycle “circular pocket“, the corresponding input field opens. After selecting the tool and entering the technology data

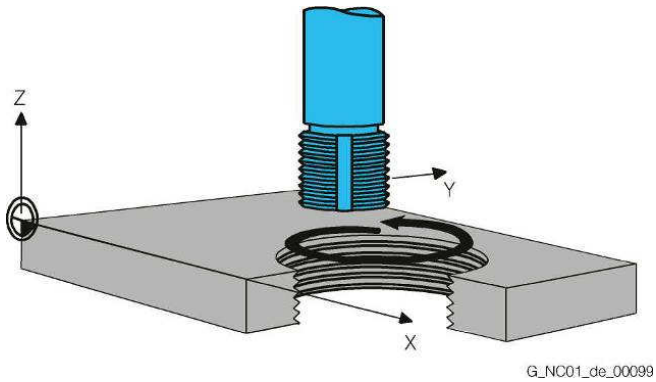
T	FRAESER20	D1
F	0.200	mm/Zahn
V	80	m/min

the circular pocket is programmed.

Bearbeitung:		
	Einzelposition	
X0	0.000	abs
Y0	0.000	abs
Z0	0.000	abs
ø	50.000	
Z1	26.000	ink
DXY	40.000	%
DZ	5.000	
UXY	0.000	mm
UZ	0.000	

Immersion in the material can be centered

Eintauchen:	mittig	
FZ	0.100	mm/Zahn



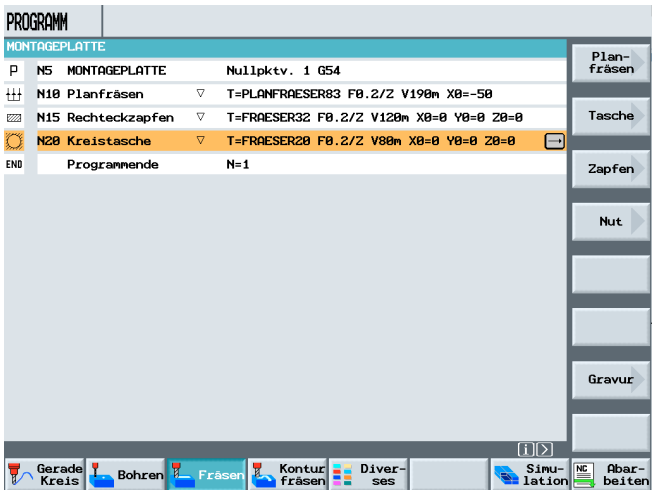
G_NC01_de_00099

or helical.

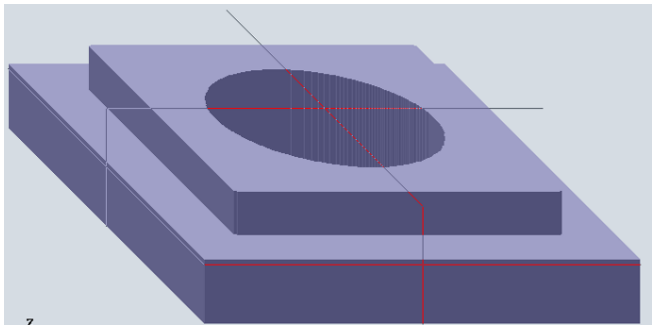
Eintauchen:	helikal	
EP	2.000	mm/U
ER	4.000	mm

For ***helical***“ immersion, a Z-motion is superimposed on the X-Y motion.
The milling tool backs off permanently because of this.

Accepting the cycle into the machining plan completes the program



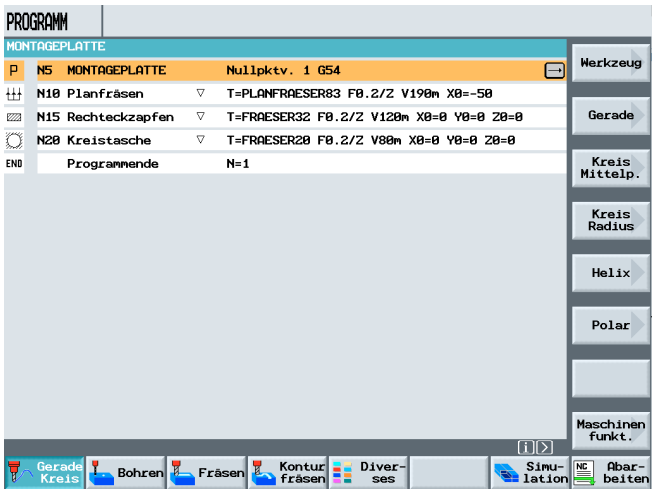
and can be simulated.



10.5 Processing (Basic Block)

By pressing the softkey

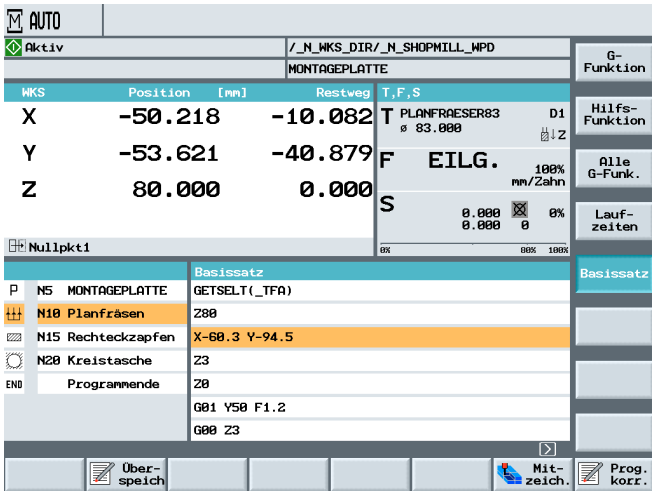
the program is loaded to the operating mode



and can be processed.

Pressing the softkey

displays the program during processing in an additional window in ***G-code***.



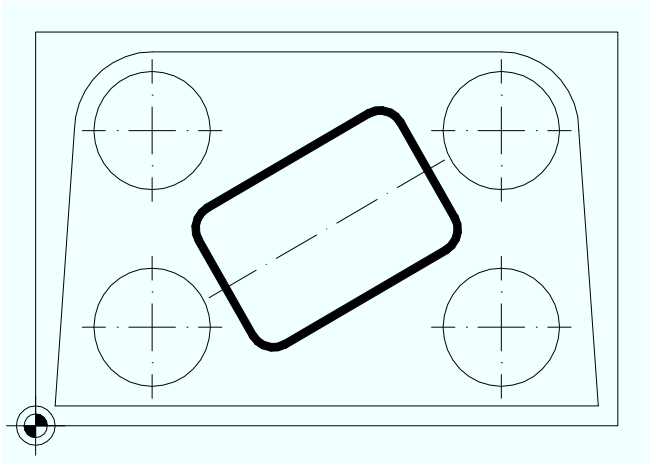
By activating the basic block, the next programmed travel movements are visible.
This makes a possible early intervention in program execution simpler for the operator.

11 Programming Example – Position Patterns for Drilling and Milling Cycles

Below, the position patterns for drilling and milling cycles under ShopMill are explained, using an example.

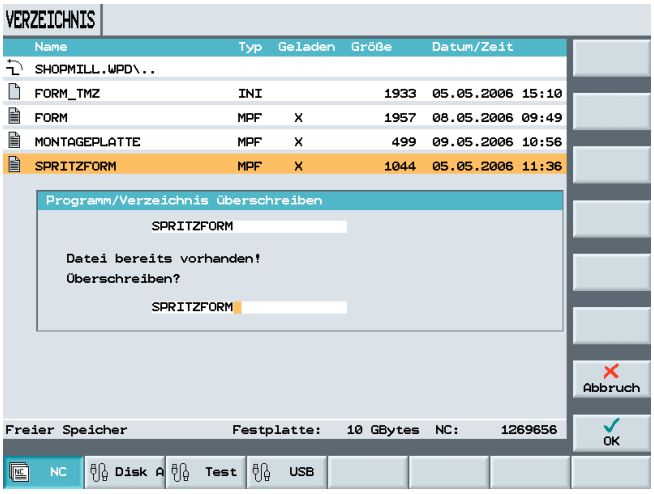
11.1 Example: Drilling and Milling Positions

A rectangle and the circular pockets are added to the example for contour programming.

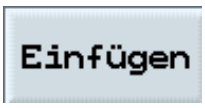


The program “Injection Mold“ is selected in the area

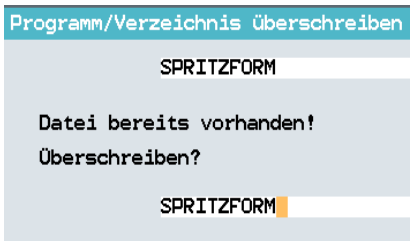
Pressing the softkeys



and

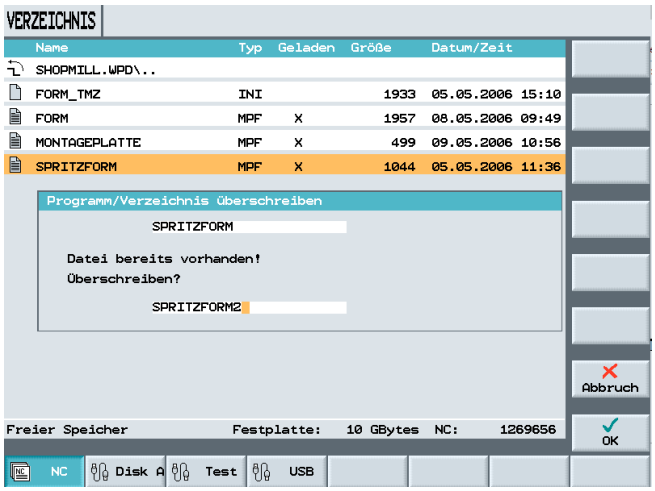


opens a dialog window.

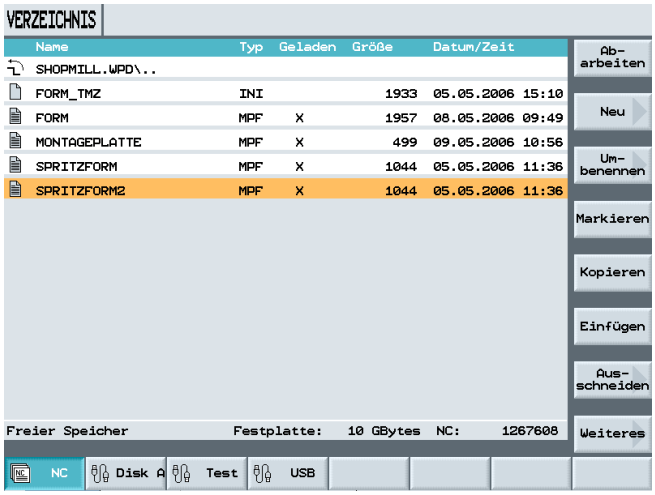


Since the existing program is not to be overwritten, a 2 is appeded to the name of the program.

By pressing the softkey

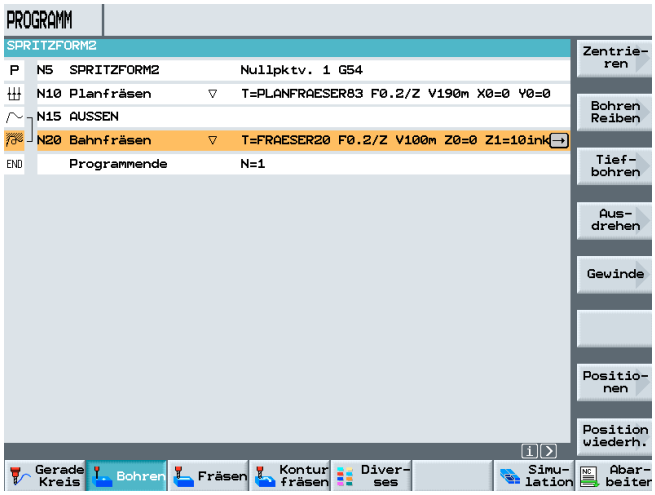


a new program named “InjectionMold2“ is stored in the selected directory.



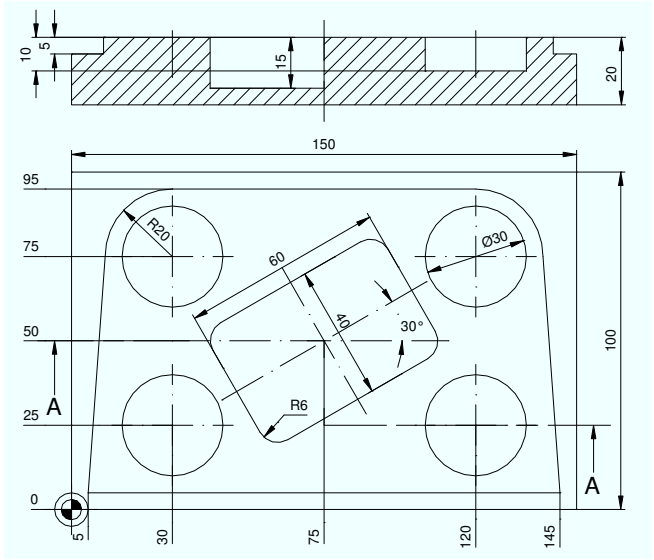
11.2 Rectangular Pocket

After opening the program, the corresponding cycle for machining the center pocket is called by pressing the softkey

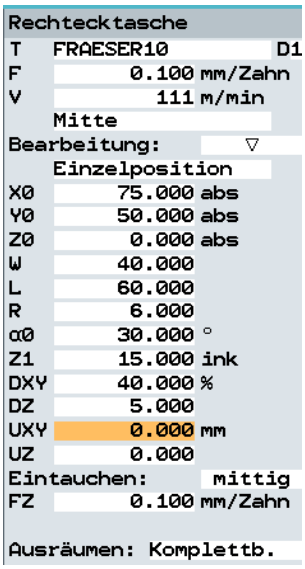


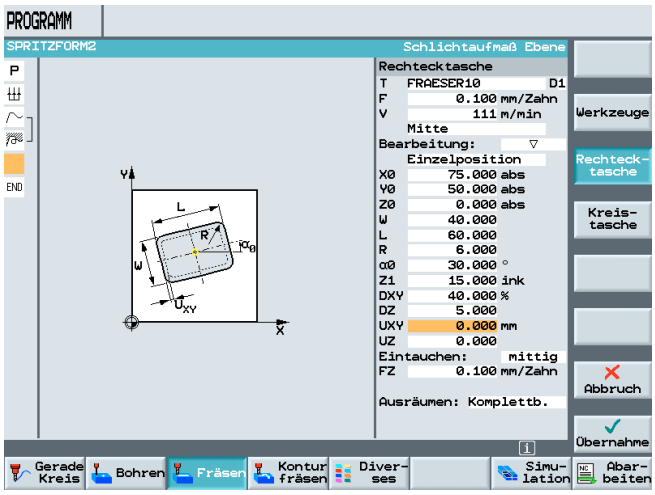
and

.

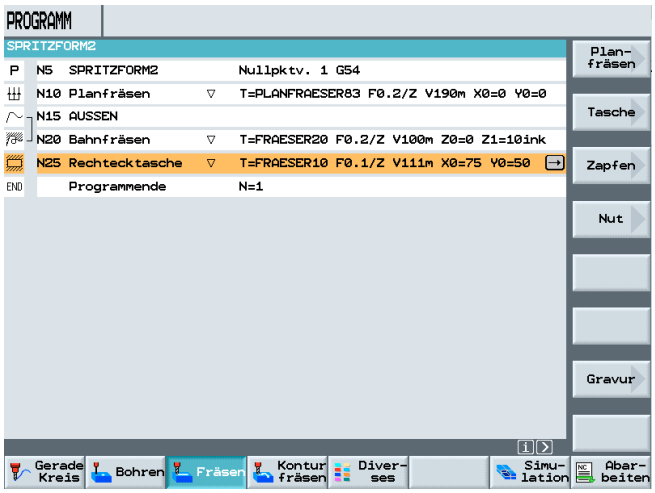


After entering the corresponding values in the input screen form

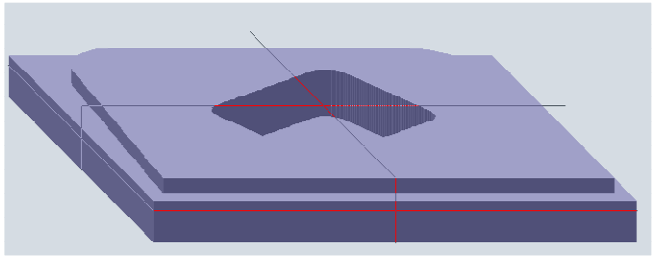




and acceptance into the machining plan,



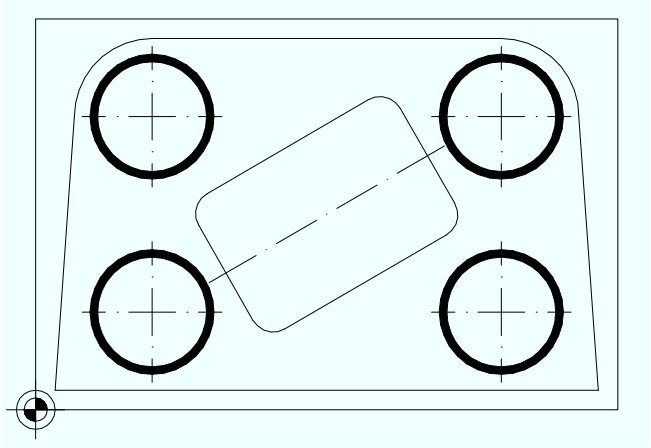
the rectangular pocket is programmed.



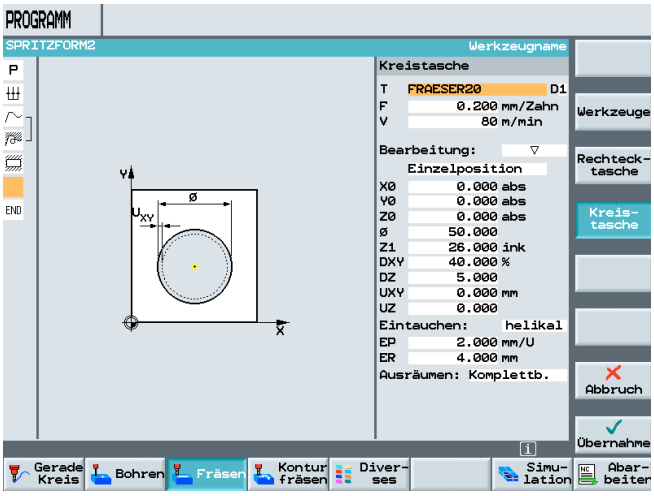
If necessary, any number of position patterns can be described one after the other.

11.3 Circular Pockets

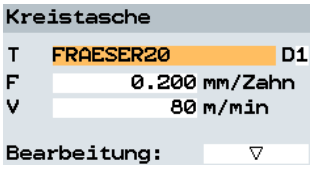
Next, the circular pockets are programmed.



After opening the corresponding cycle



and <<something missing?>>, the tool with the corresponding technology is entered.



11.4 Position Pattern

Now, not the Individual Position

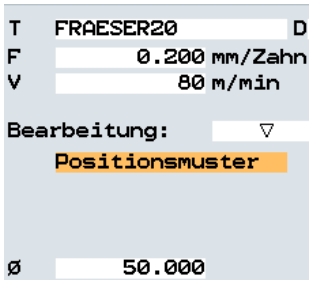
Einzelposition

as previously, but -since it is a question of several circular pockets of the same type- by pressing the softkey

the

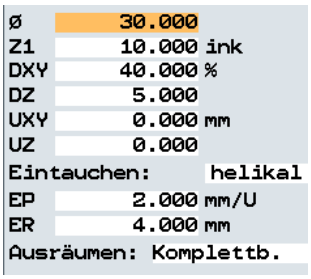
Positionsmuster

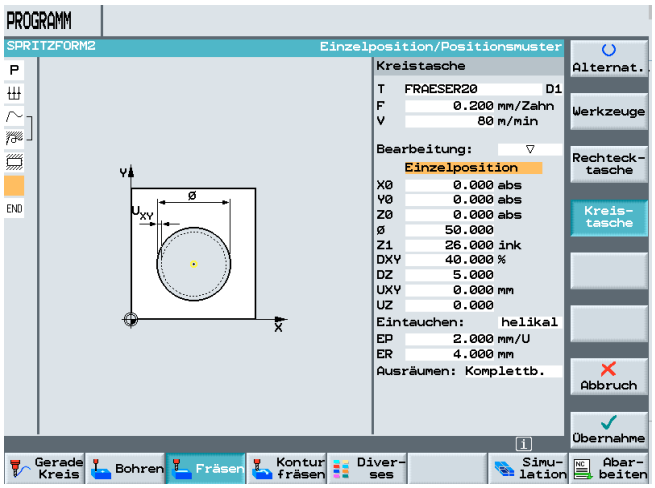
is selected. Based on this selection,

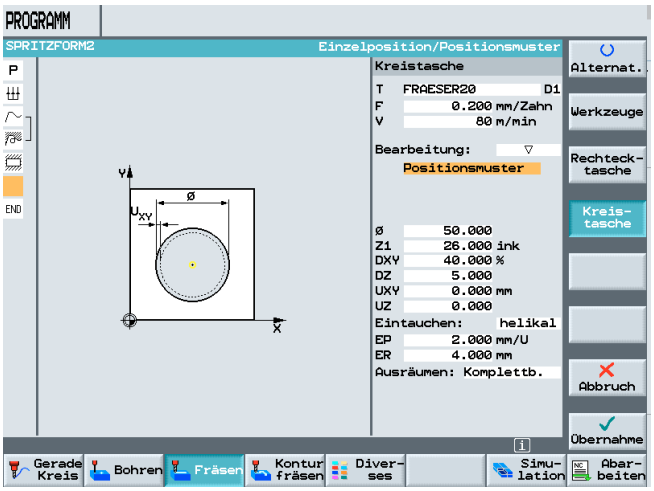


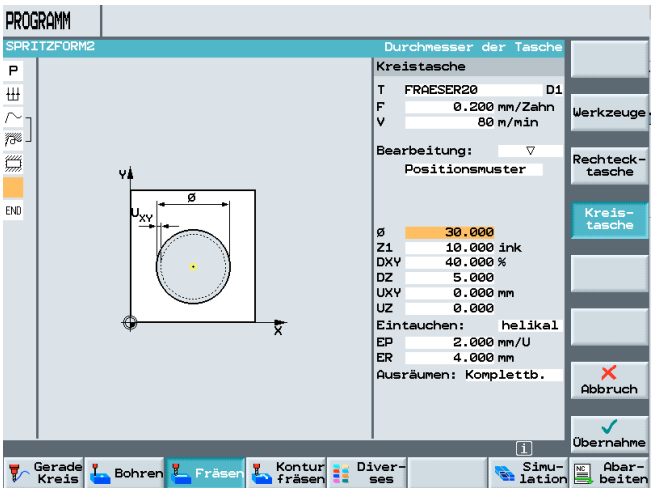
it is no longer possible to describe positions in the input screen form of the cycle.

Input of values in the screen form.



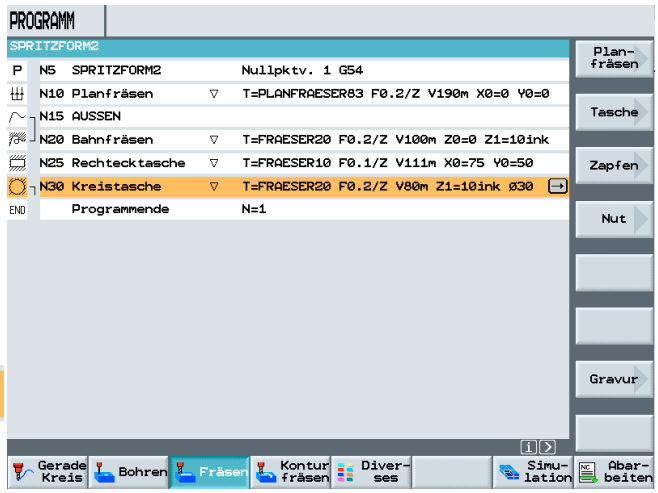






11.5 Drilling and Positions

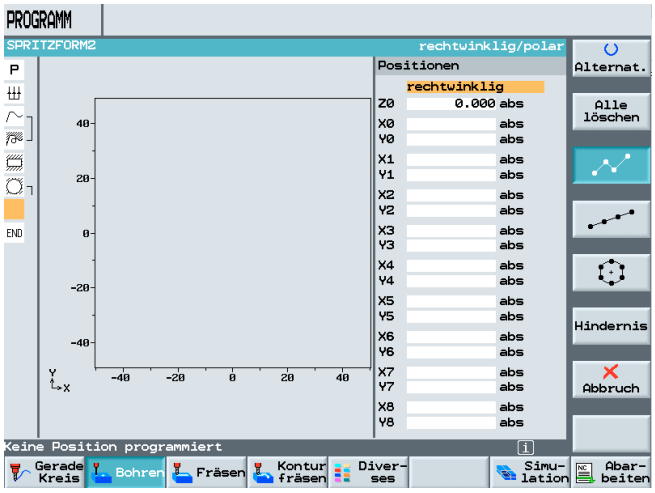
Through acceptance into the machining plan, an open bracket is displayed at the circular pocket.



Next, the positions of the circular pockets are programmed.

Pressing the softkey

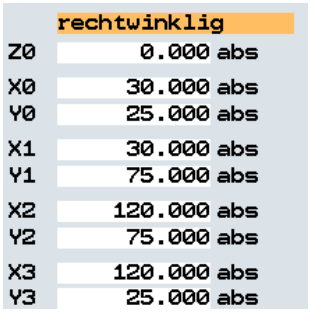
and

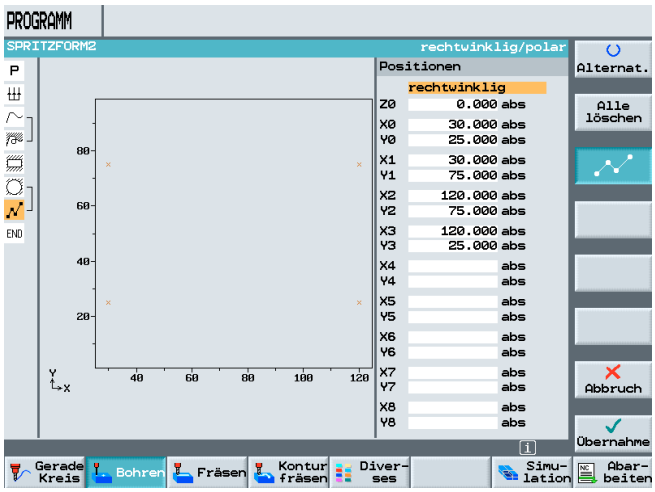


opens the corresponding input field.

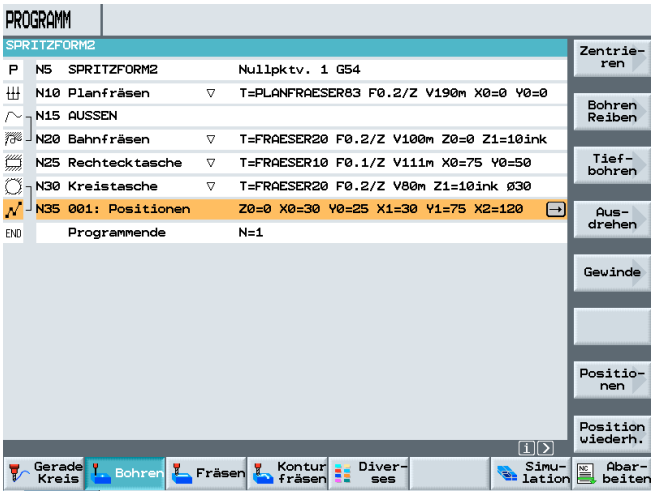
As an alternative, the circular pocket positions can also be entered with the position pattern “Frame“.

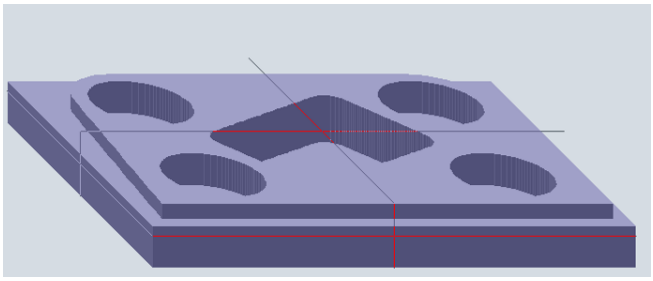
After entering the corresponding positions in the input screen form





and acceptance into the machining plan, the bracket is closed and the program is complete.



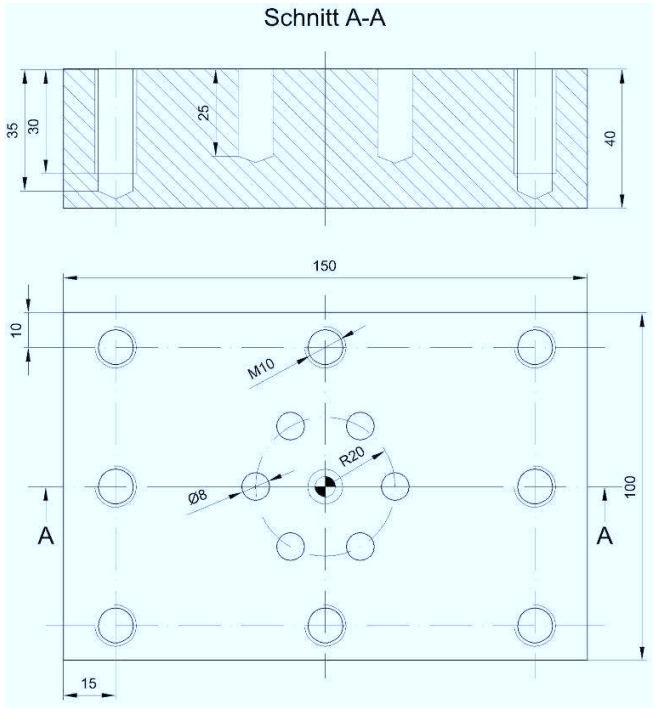


12 Program Example – Centering, Drilling, Threading

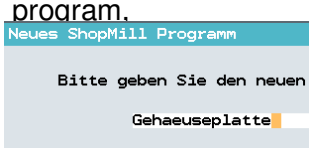
In this module, centering, drilling and threading under ShopMill is described.

12.1 Exercises for Centering, Drilling, and Threading

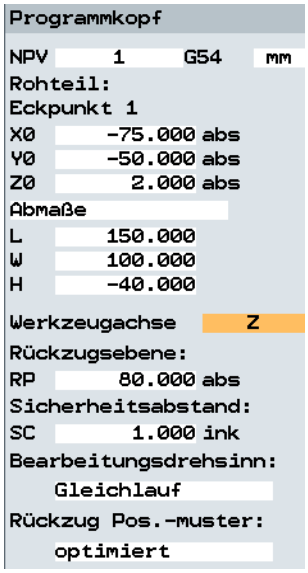
As an example for centering, drilling and threading, this drawing is to be programmed.



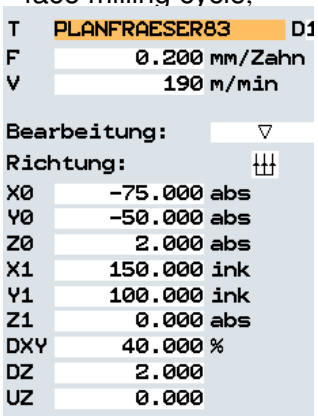
After setting up a new program,



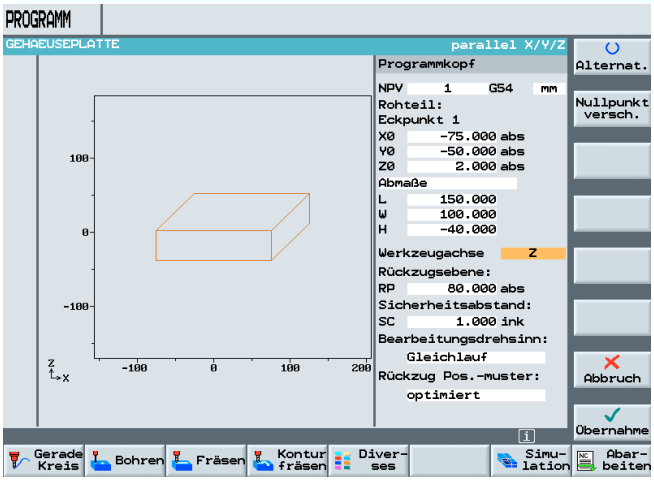
defining the program header,

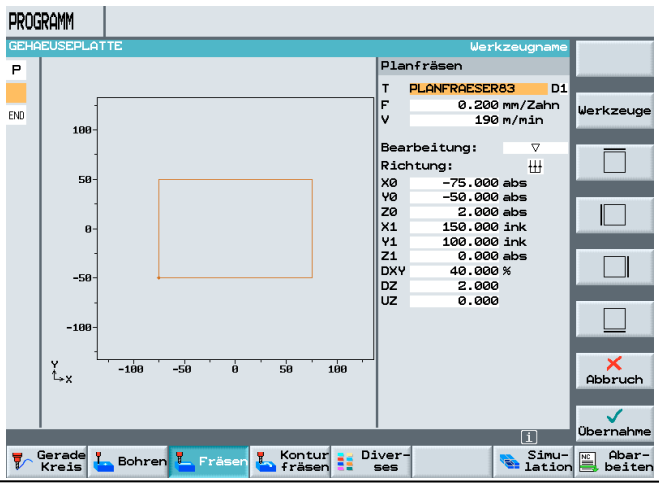


and programming the face milling cycle,



the next step consists of programming the centering.

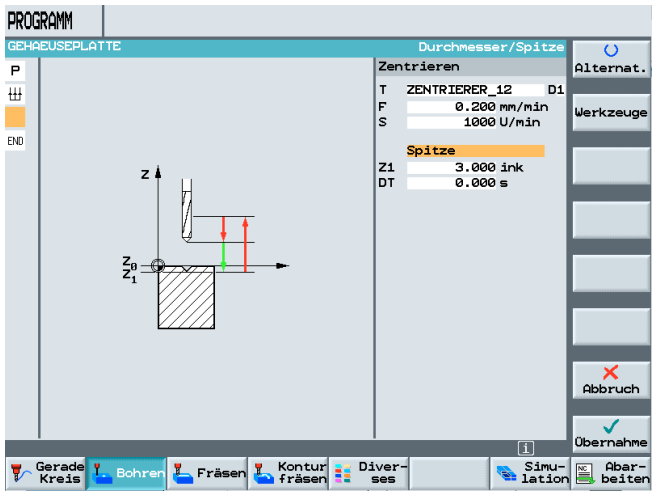




12.2 Centering the Frame and the Hole Circle

After opening the input screen form for the centerings (spot drilling), the values are entered in the input fields.

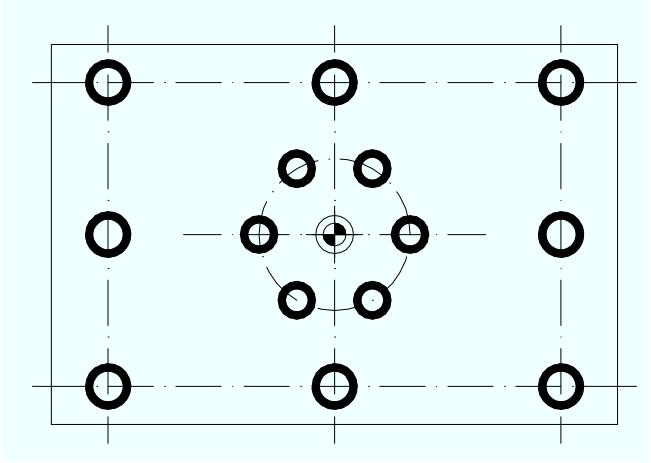
T	ZENTRIERER_12	D1
F	0.200	mm/min
S	1000	U/min
	Spitze	
Z1	3.000	ink
DT	0.000	s

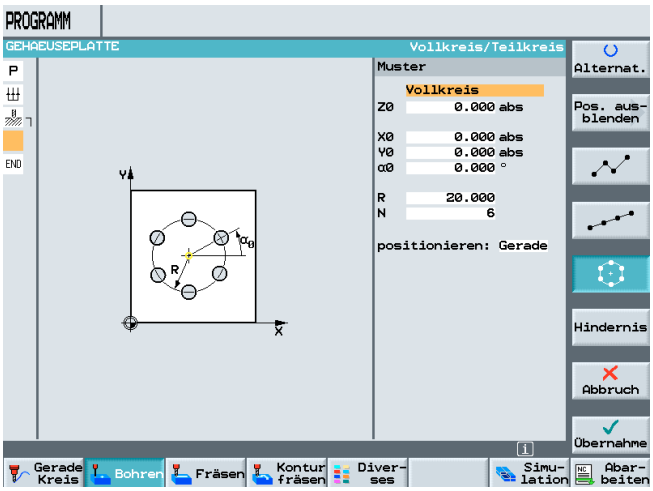


Since at none of the holes a chamfer is programmed, centering can be done for all holes.

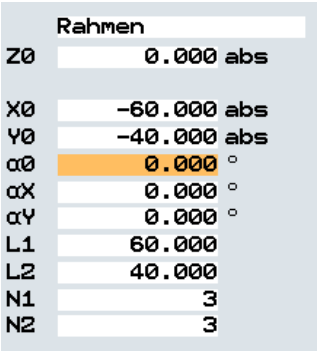
The positions of the center circle

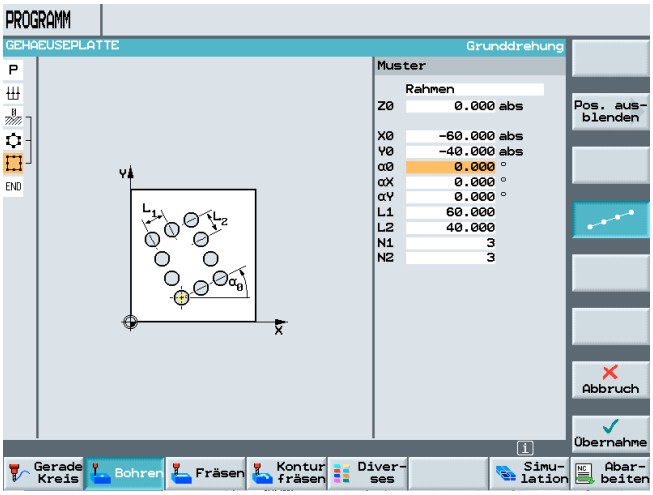
	Vollkreis	
Z0	0.000	abs
X0	0.000	abs
Y0	0.000	abs
α0	0.000	°
R	20.000	
N	6	
positionieren:	Gerade	





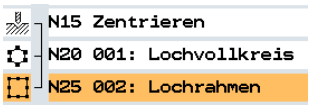
as well as the outside frame are programmed.

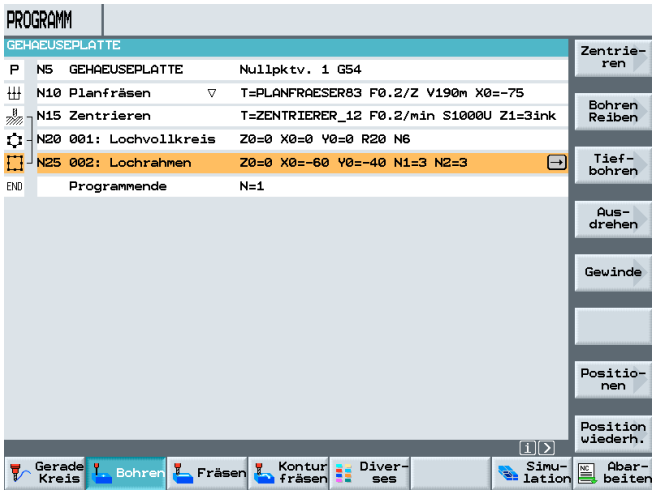


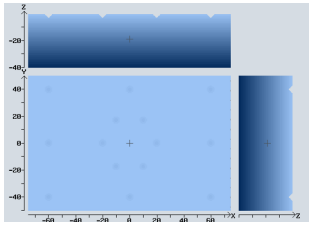


12.3 Drilling

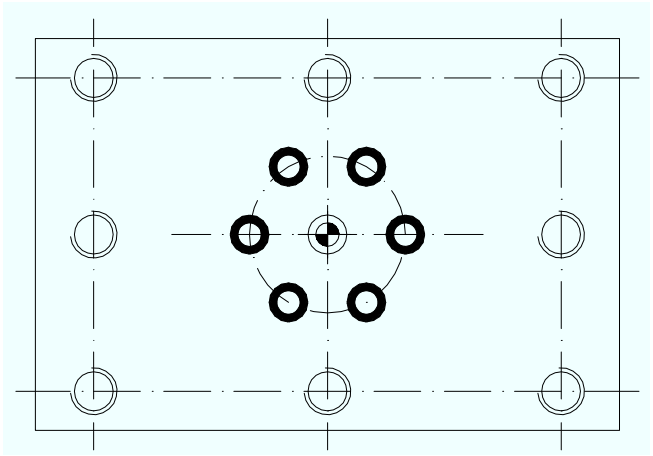
After accepting the position patterns into the machining plan, the centerings for the holes are completely programmed.





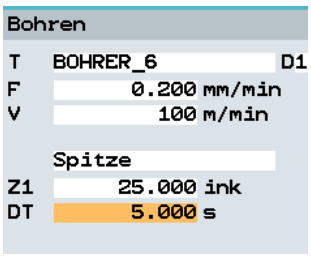


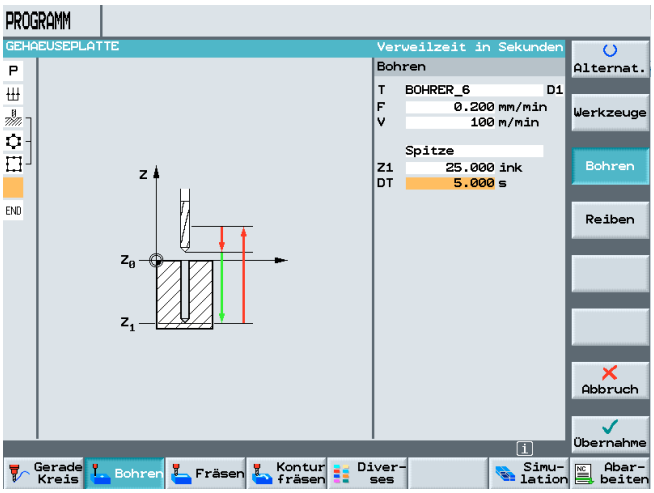
Next, the holes for the center hole circle are programmed.



12.4 Programming "Drilling the Hole Circle“ by Using Copy and Insert

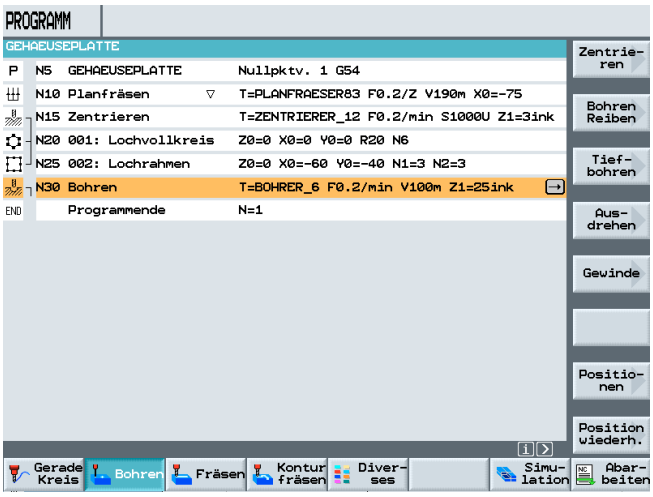
After opening the input screen form for drilling and entering the values,



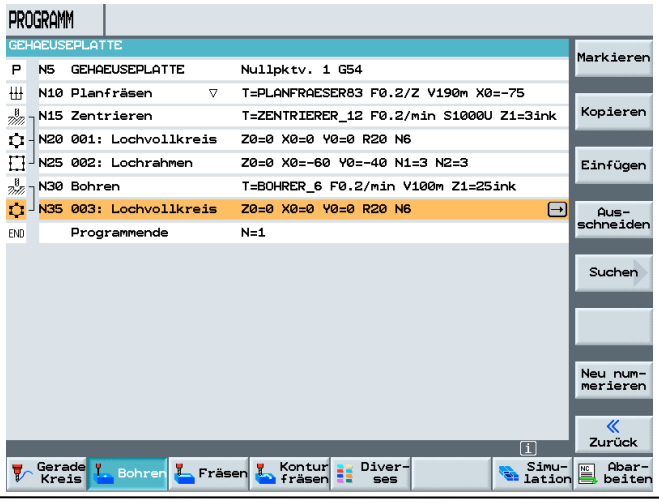


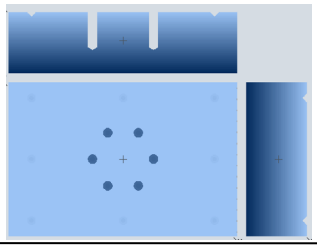
the input is acccepted into the machining plan.
Since the drilling positions were already programmed, they are appended to the boring by pressing the softkeys

and



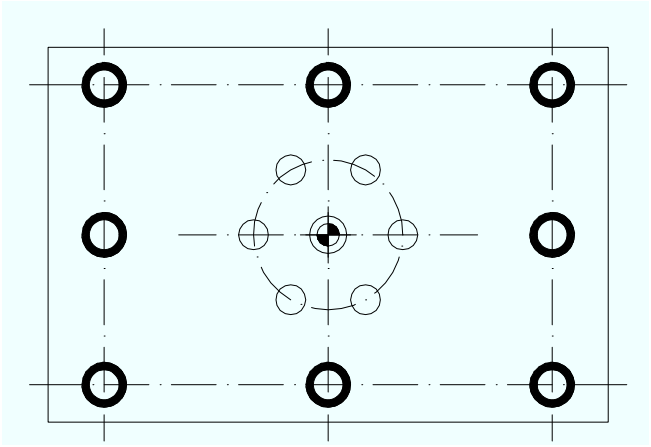
The center borings are completely generated.



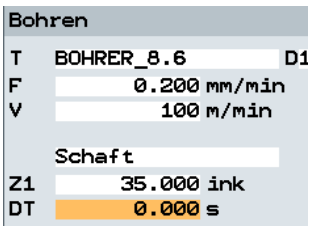


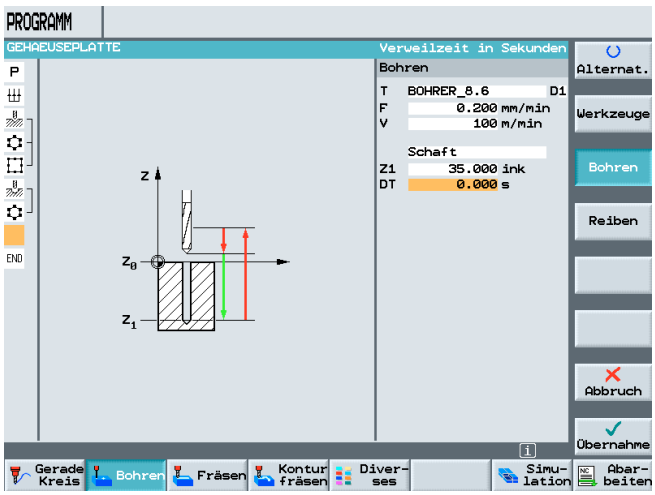
12.5 Borings Threads for Frames

Finally, the borings of the “frame“ are programmed.

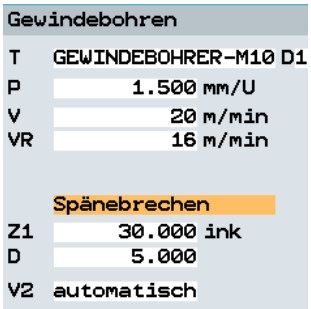


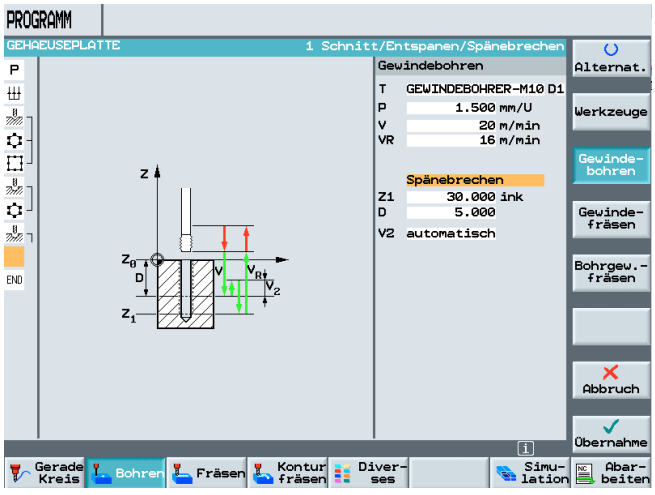
After entering the values for the borings





and entering the values for the thread,

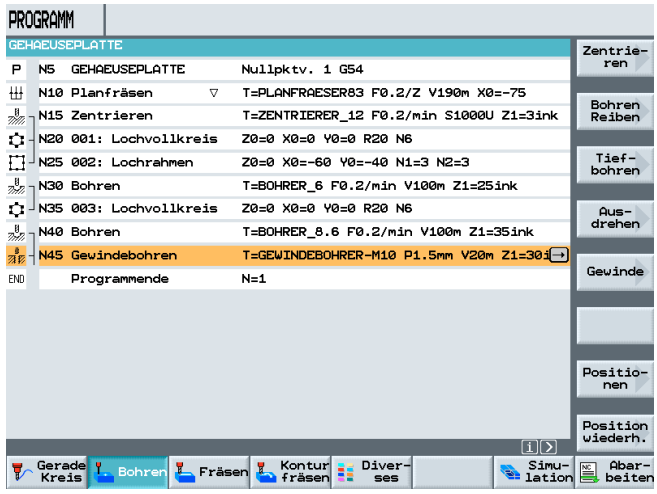


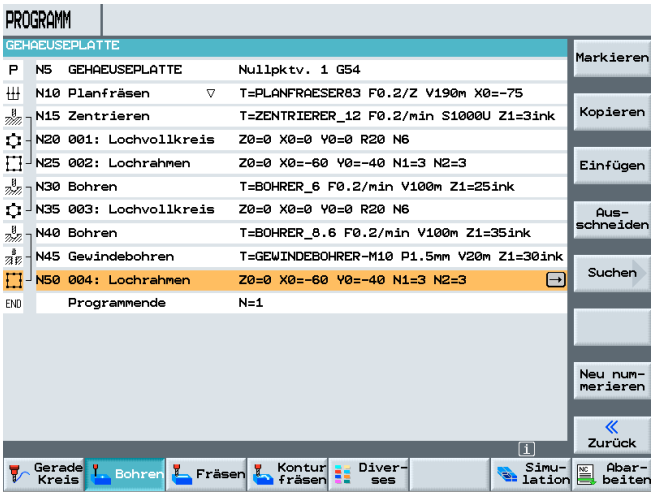


the inputs are accepted into the machining plan.

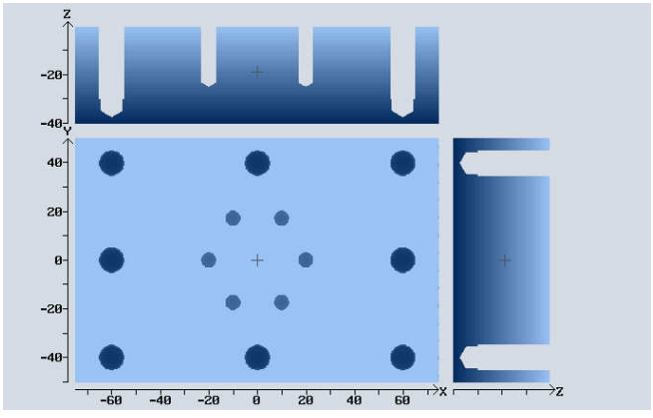
Since the positions for the borings were already programmed also, they are again appended to the boring by pressing the softkevs

and





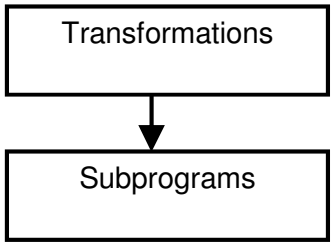
The program is completely generated.



13 Programming Example – Programmable Transformations, Subprogram Technology

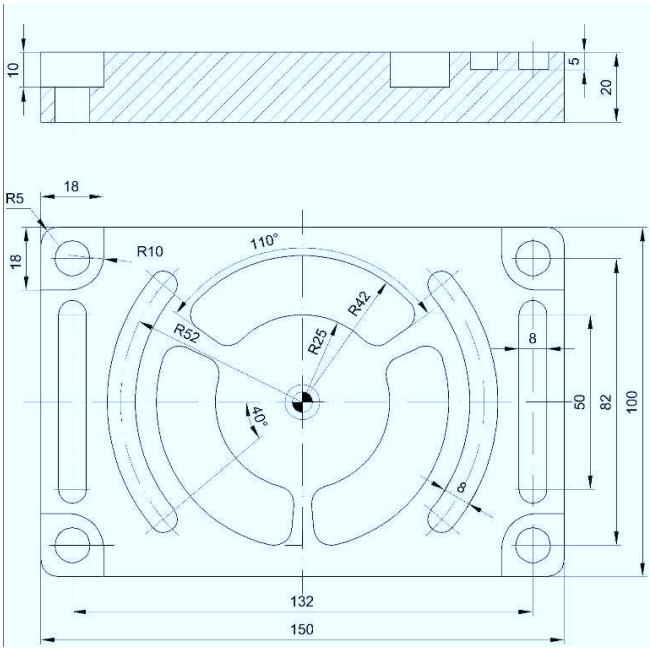
Content of the Module:

This module describes how ShopMill transformations and subprograms are programmed, using an example.



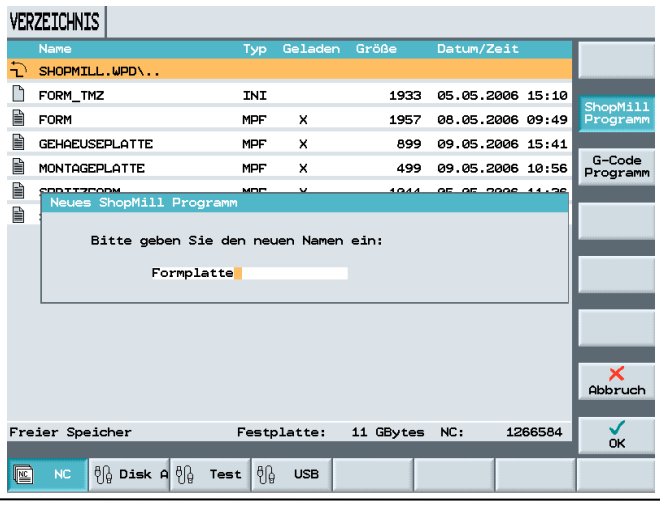
Mirroring/rotating transformations

Based on this example, the functions “Program Loop“ - “Shift“ are explained in greater detail..



All non-dimensioned radii R=5

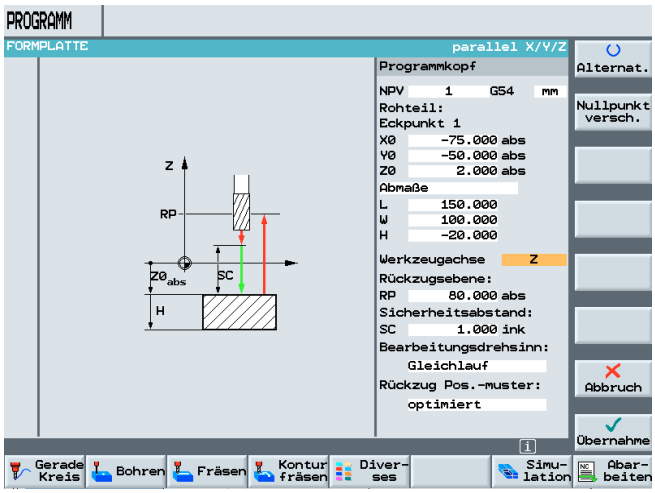
Setting up the program



13.1 Program Header

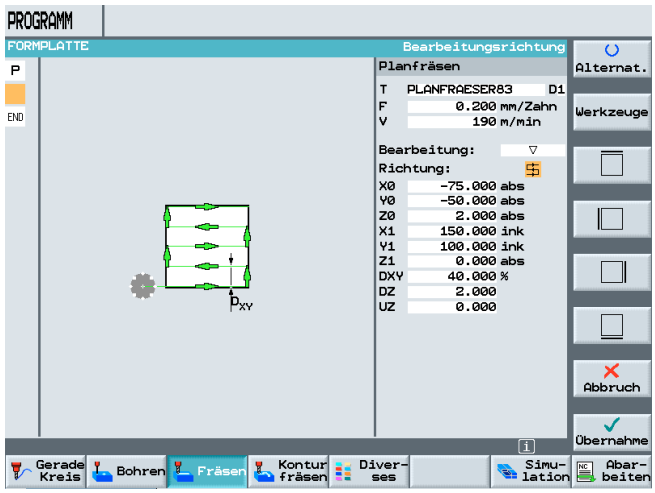
After entering the values in the program header

Programmkopf			
NPV	1	G54	mm
Rohteil:			
Eckpunkt 1			
X0	-75.000	abs	
Y0	-50.000	abs	
Z0	2.000	abs	
Abmaße			
L	150.000		
W	100.000		
H	-20.000		
Werkzeugachse	Z		
Rückzugsebene:			
RP	80.000	abs	
Sicherheitsabstand:			
SC	1.000	ink	
Bearbeitungsdrehsinn:			
Gleichlauf			
Rückzug Pos.-muster:			
optimiert			

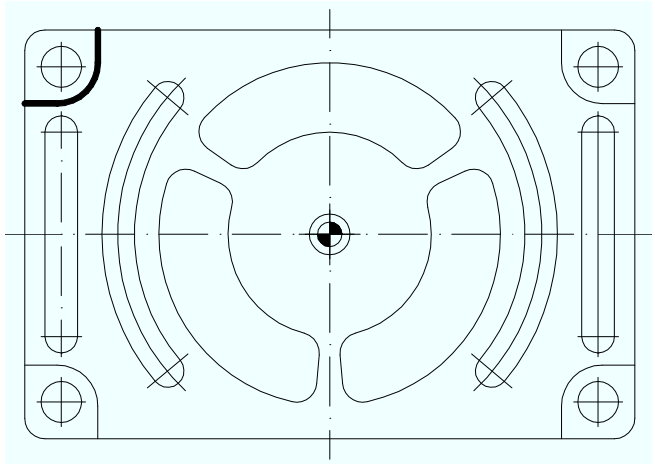


and entering the parameters for face milling,

Planfräsen		
T	PLANFRAESER83	D1
F	0.200	mm/Zahn
V	190	m/min
Bearbeitung:	▽	
Richtung:		
X0	-75.000	abs
Y0	-50.000	abs
Z0	2.000	abs
X1	150.000	ink
Y1	100.000	ink
Z1	0.000	abs
DXY	40.000	%
DZ	2.000	
UZ	0.000	

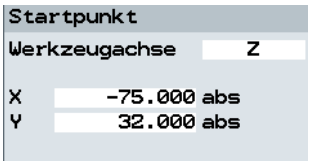


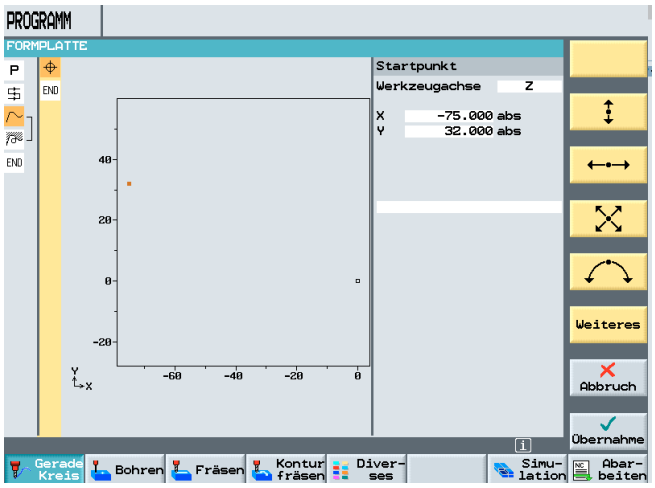
next the corner contour is programmed.



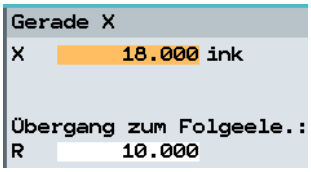
13.2 Contour Calculator Left Upper Corner

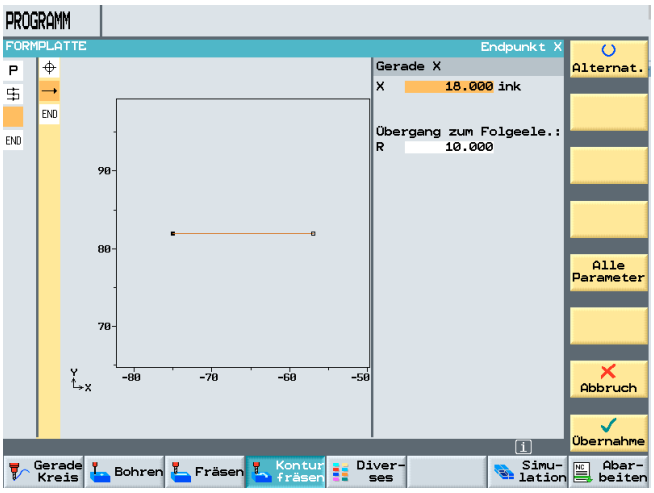
After opening the contour calculator and defining the starting point for the left upper corner,

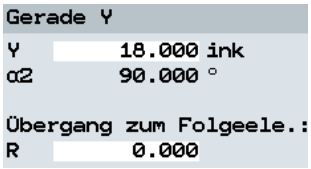


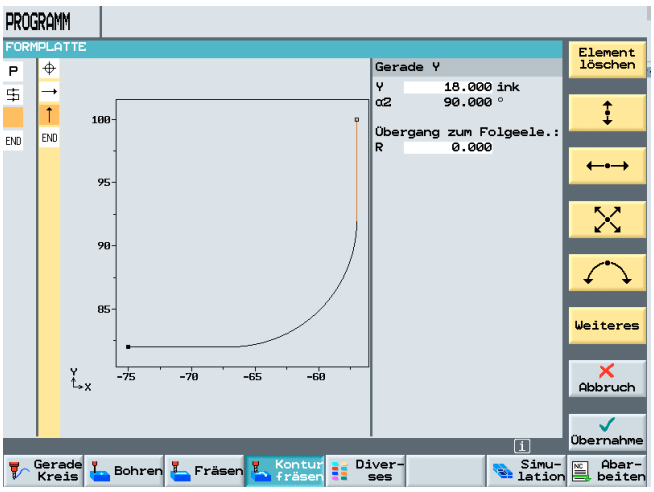


the contour is programmed incrementally.



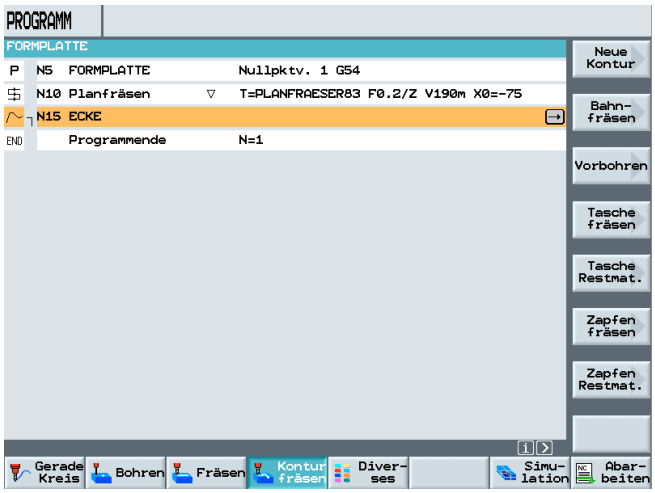




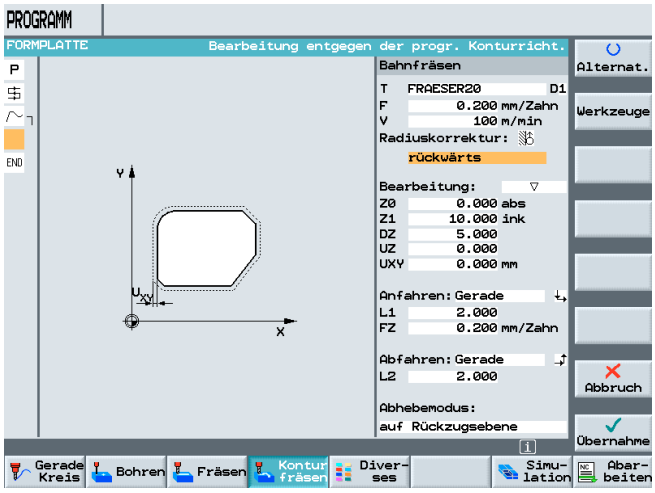


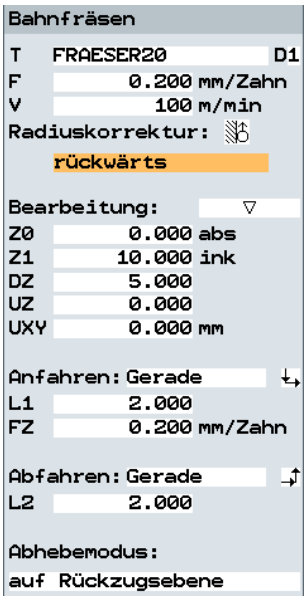
13.3 Path Milling

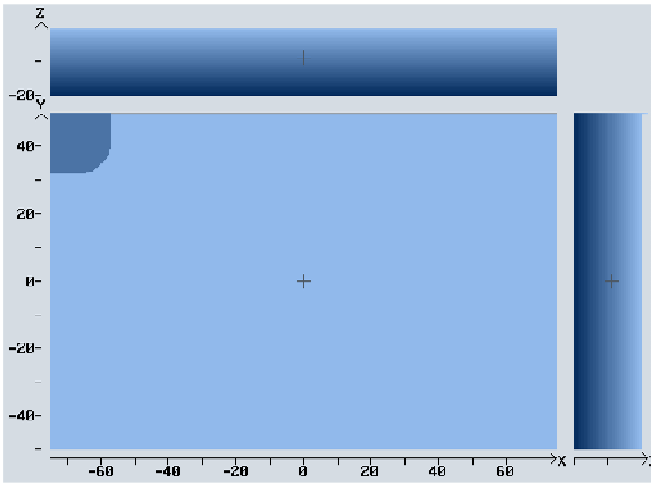
After accepting the contour into the machining plan,



the cycle “Path Milling“ is opened.







After accepting the values into the machining plan, corner machining is programmed.

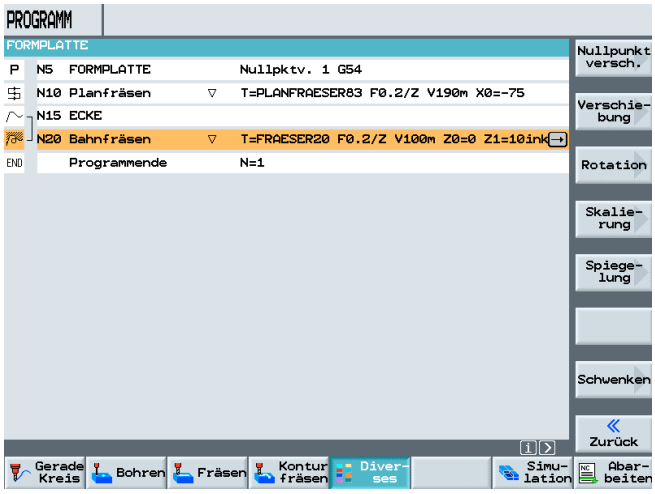
13.4 Mirroring

We don't want to program this corner three more times, but mirror it under ShopMill.

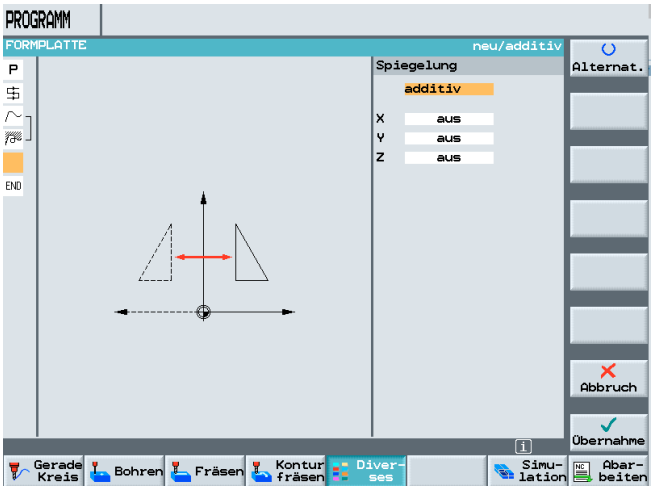
After pressing the soft key

and

the possible transformations under ShopMill are shown.



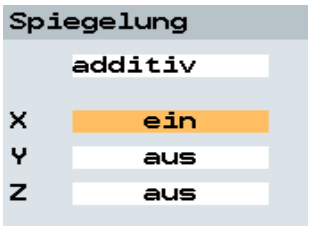
Pressing the softkey Mirroring

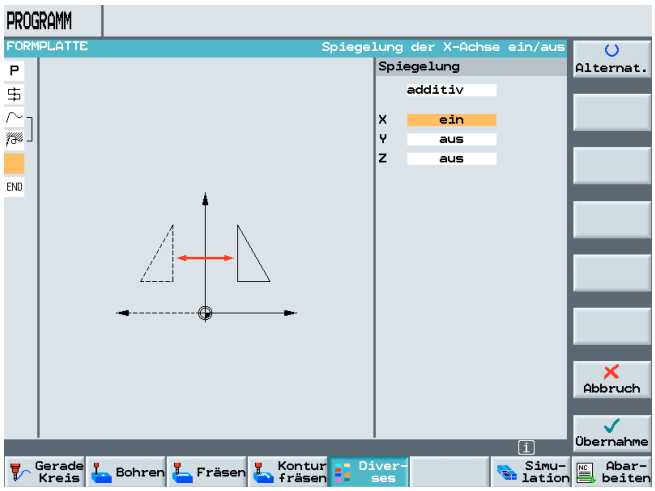


opens the corresponding input screen form.

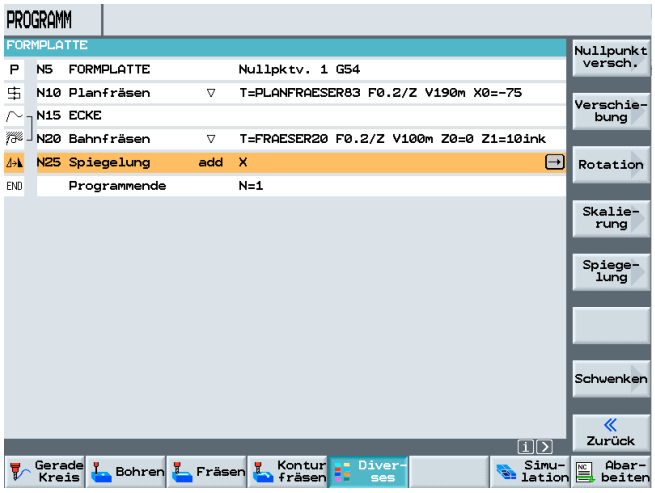
The corners are mirrored “additively“; that is, always in reference to the reference point that was mirrored last.

After activating the corresponding axis

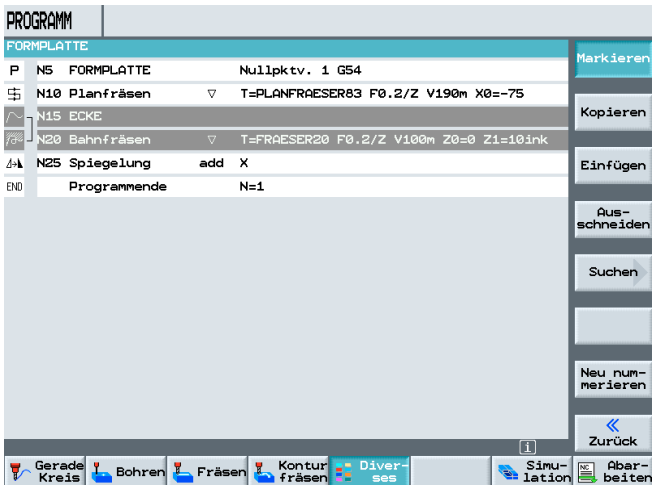




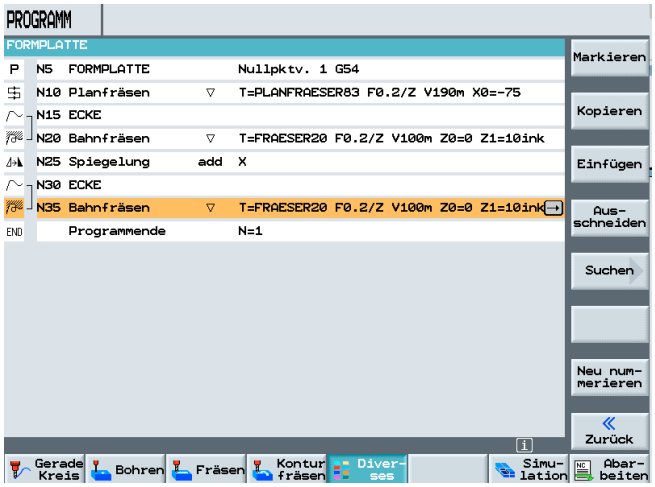
and accepting it into the machining plan, all additional program steps are inserted behind the mirror image around the X-axis.



After highlighting and copying the contour with the associated processing of Copy

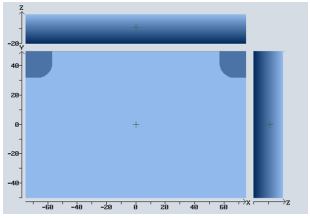


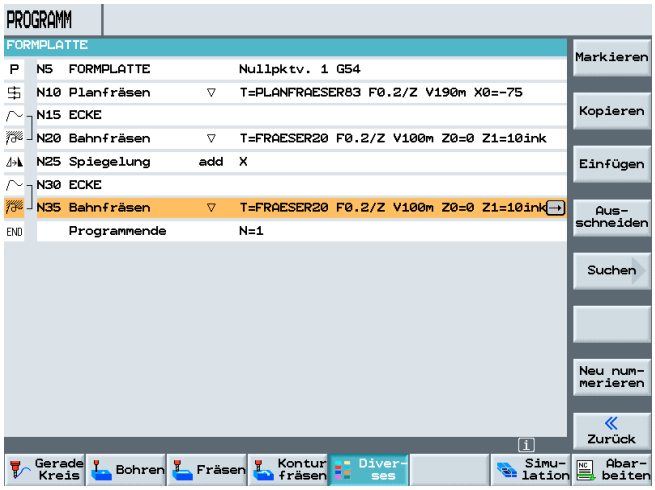
the contour is inserted behind the mirror image by pressing the softkey



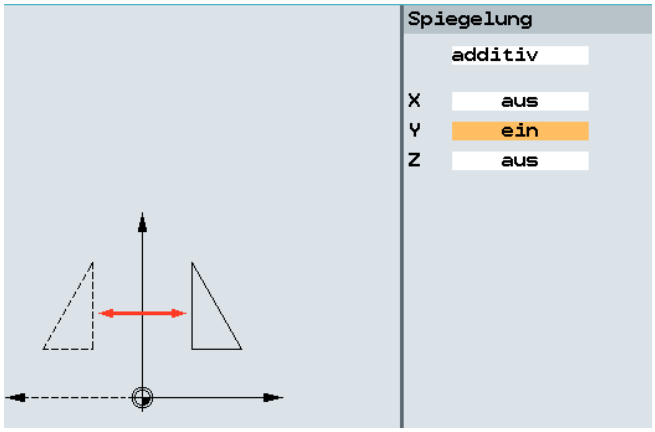
Copied program parts are inserted below the current position.

The second corner has been programmed



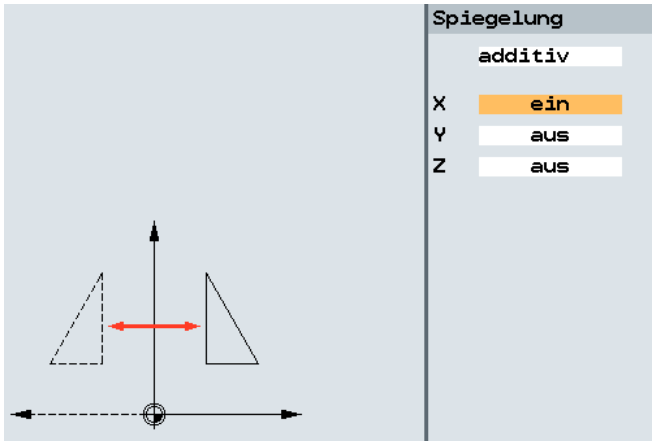


With another additive mirroring -this time around theY-axis-



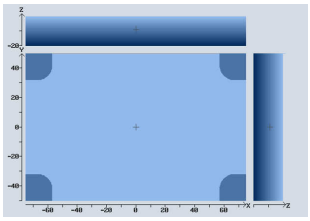
and subsequent additive mirroring around the X-axis,

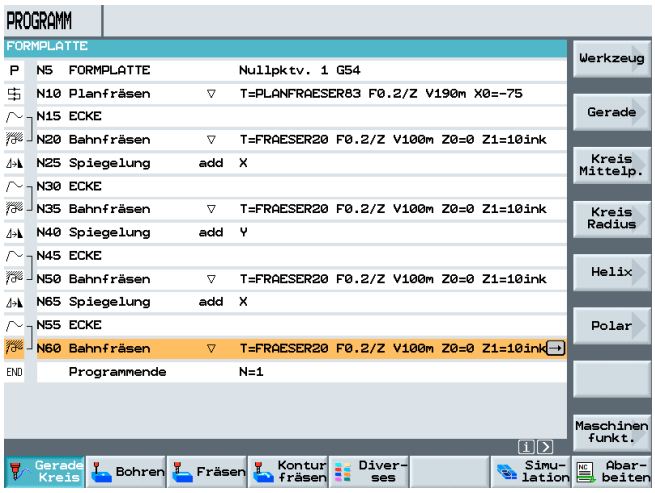
and additionally inserting the contour including the processing below the mirrorings



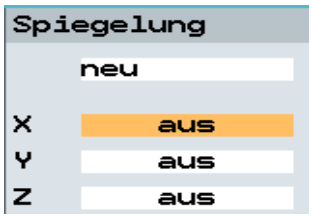
13.5 Longitudinal Grooves

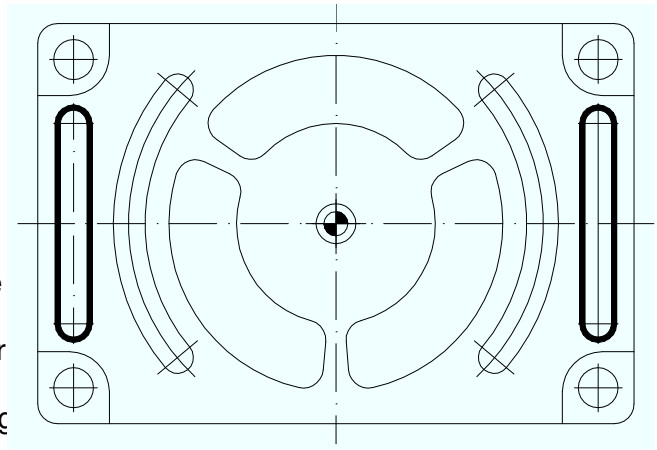
the program for the 4 corner machinings is completed



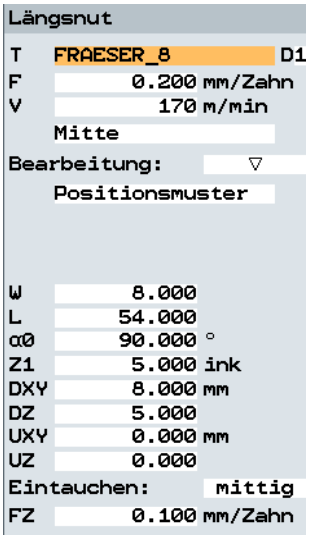


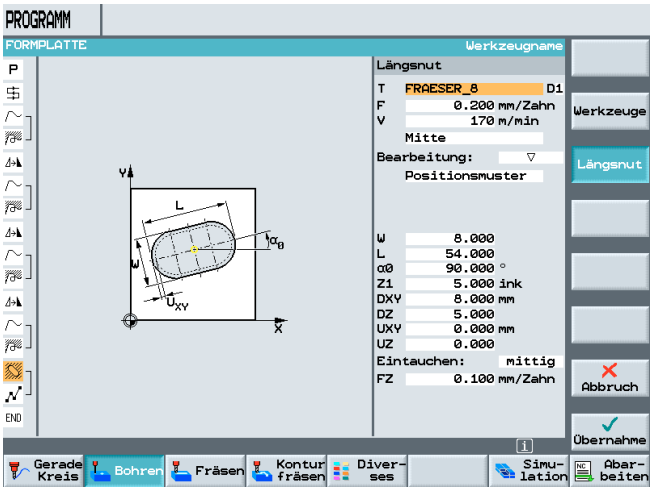
After entering the last corner, mirroring is still active and has to be switched off with



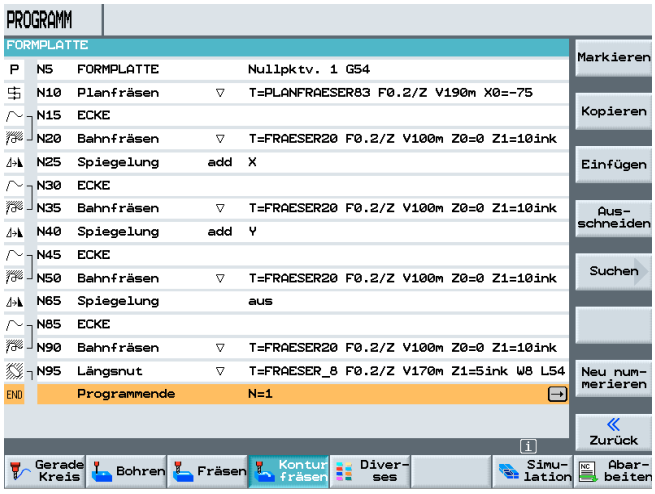


Next, the longitudinal grooves on both sides are milled.
After opening the cycle for longitudinal grooves and entering the corresponding values in the input screen form,

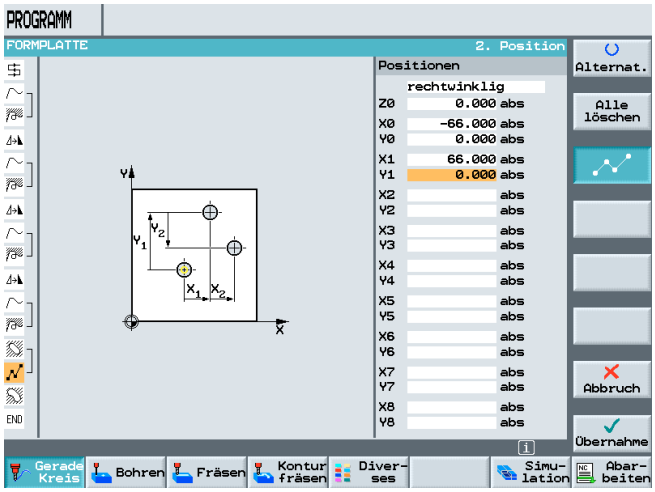




the cycle is accepted into the machining plan.

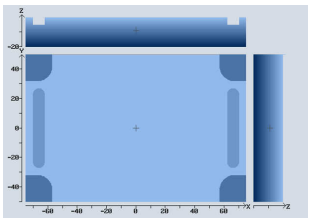


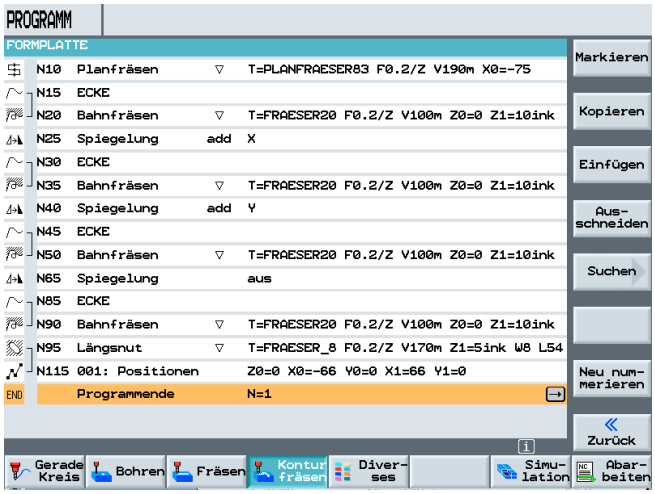
After opening the input screen form for the position pattern, entering the corresponding values



and acceptance into the machining plan,

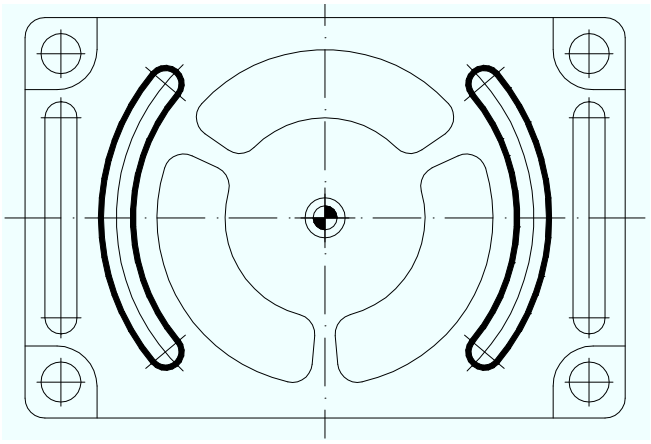
the longitudinal grooves are completely programmed.



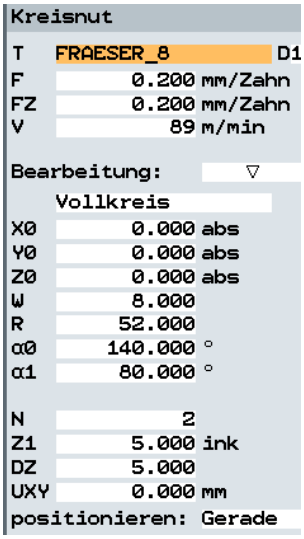


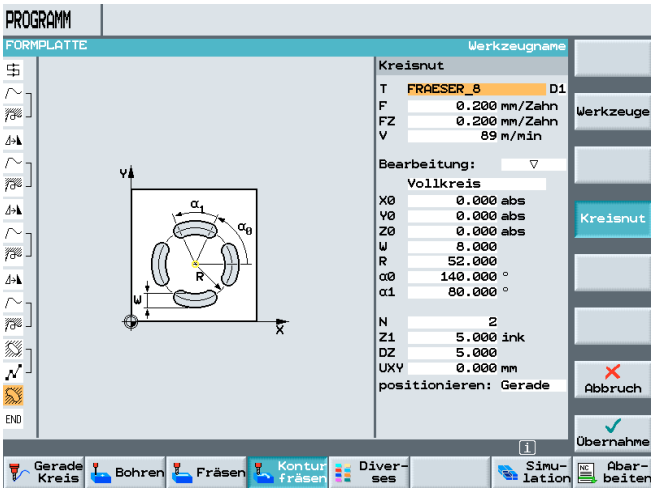
13.6 Circumferential Groove

Next, the circumferential grooves are programmed.



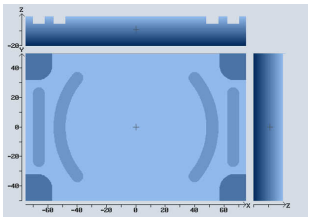
After opening the cycle for circumferential grooves and entering the corresponding values in the input screen form,

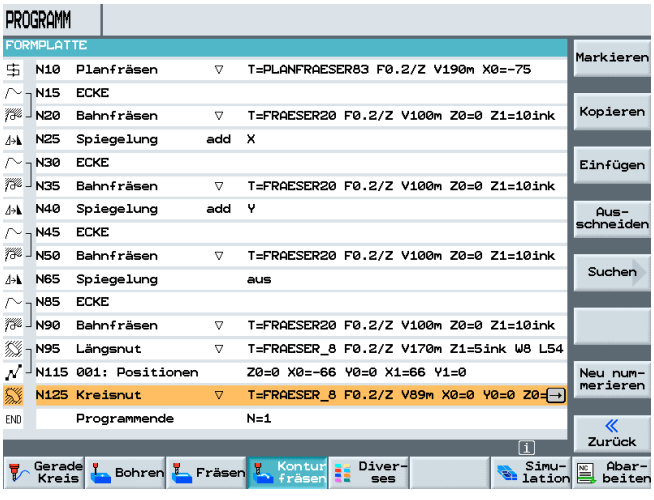




the cycle is accepted into the machining plan.

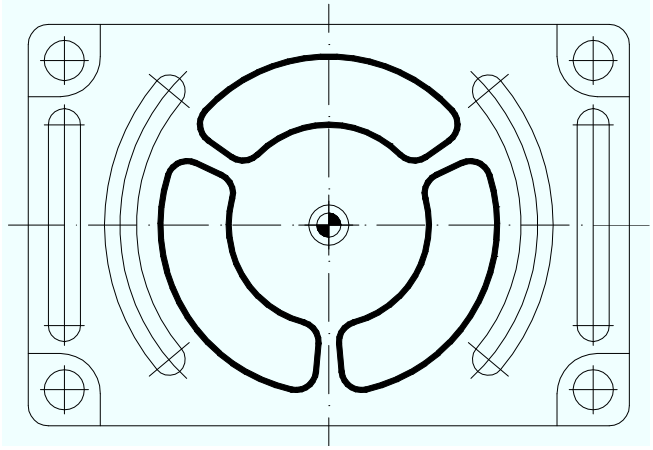
The circumferential grooves are completely programmed.



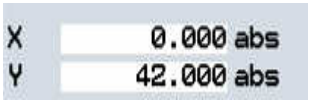


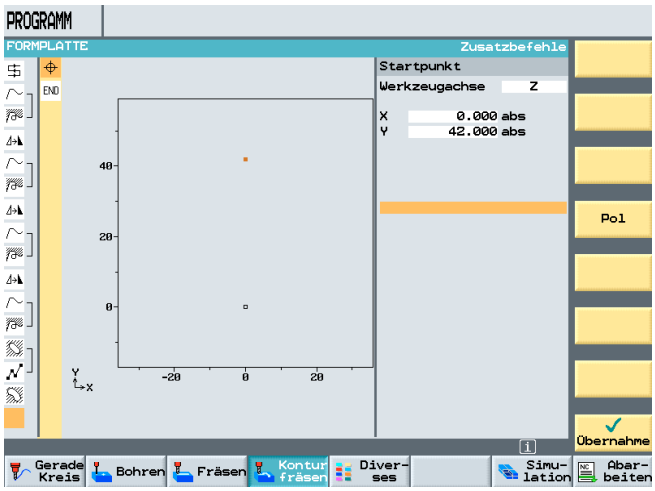
13.7 Contour Pockets with Contour Calculator

Next, the contour pockets are programmed.

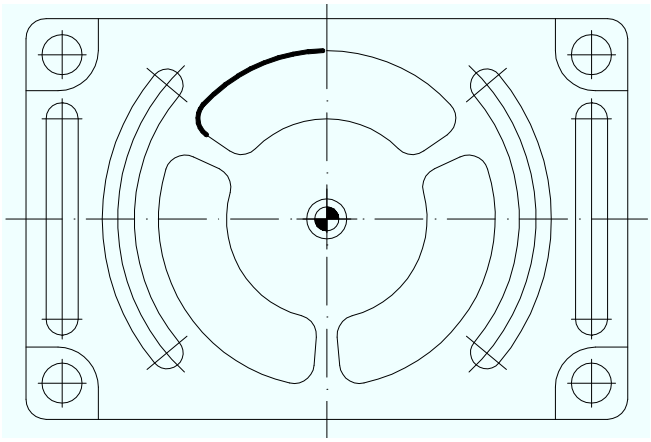


After opening the contour calculator, that pocket's starting point is defined.

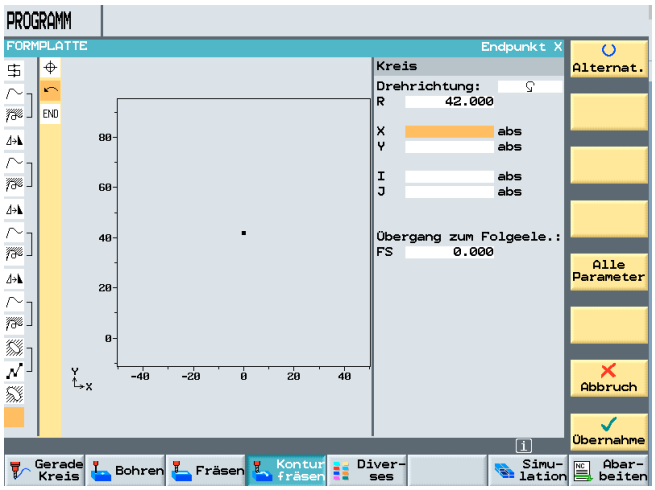




Then, half the arc is programmed.

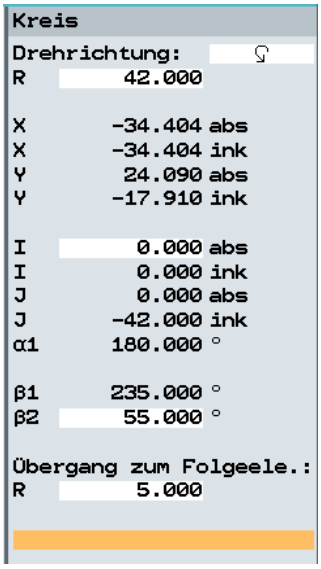


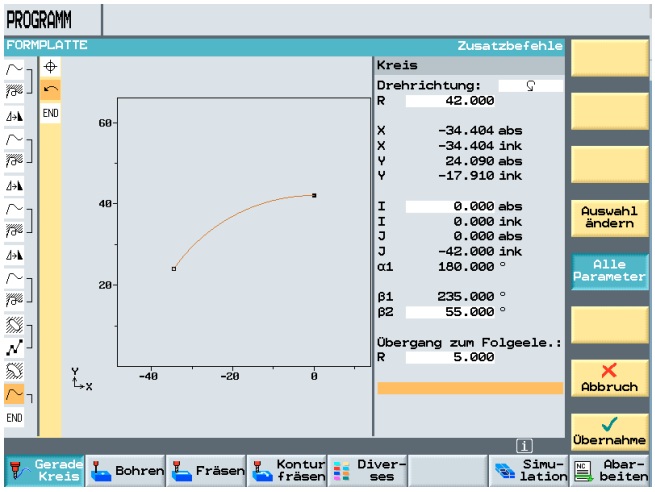
By pressing the softkey



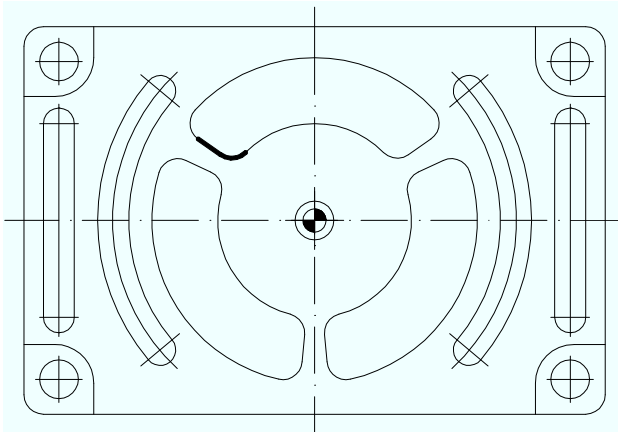
additional input options are available.

The angle of opening that was dimensioned in this way is entered in the extended input screen form.

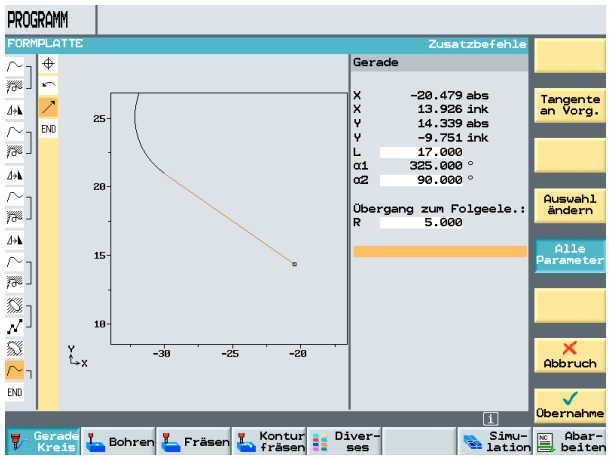




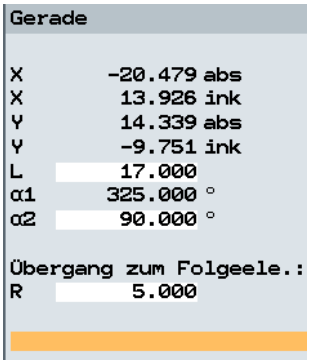
Then, the oblique line is programmed.



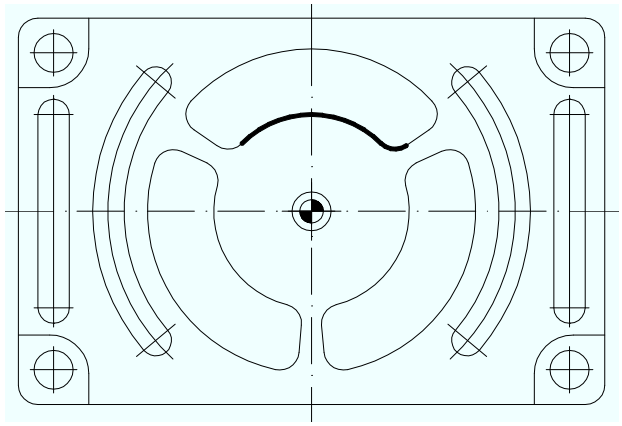
After opening the input screen form and pressing the softkey



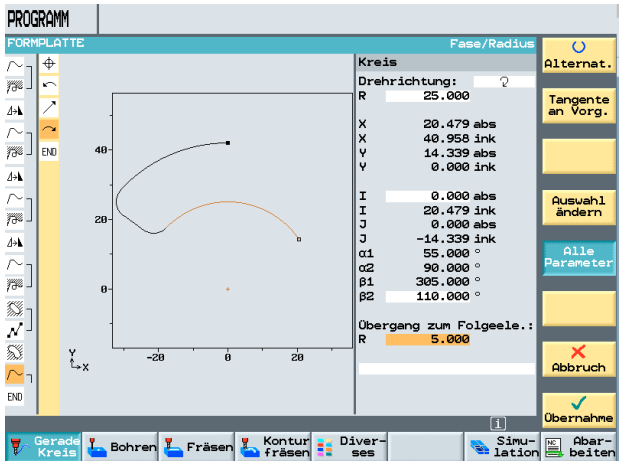
the length of the oblique line as well as the angle to the predecessor element is entered.



Next, the lower arc is programmed.

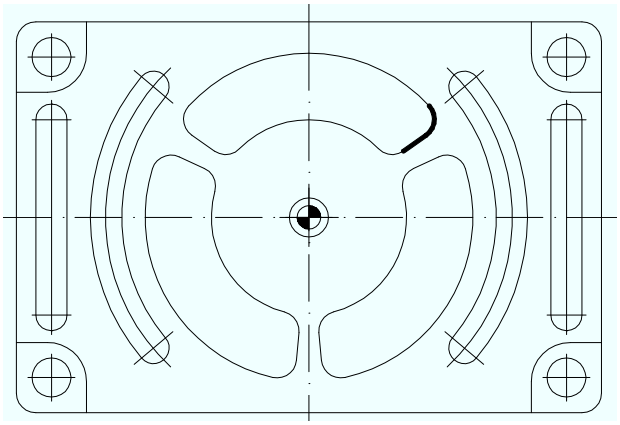


After opening the input screen form and pressing the softkey

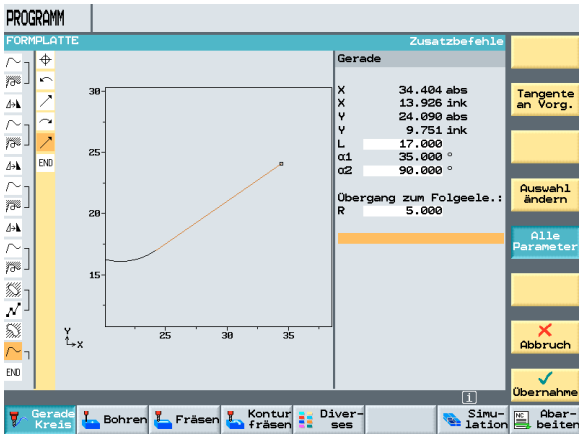


the arc is programmed again by means of the angle of opening.

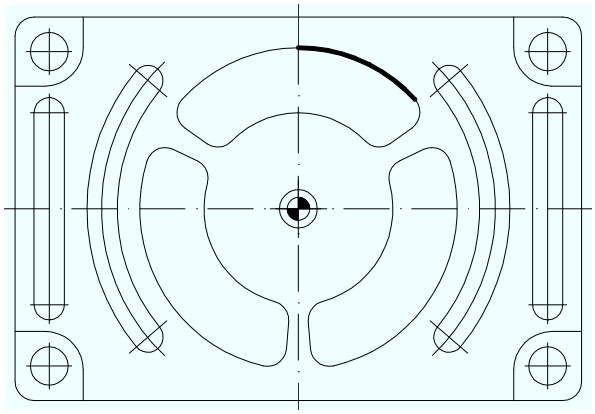
Now the oblique line is programmed again.



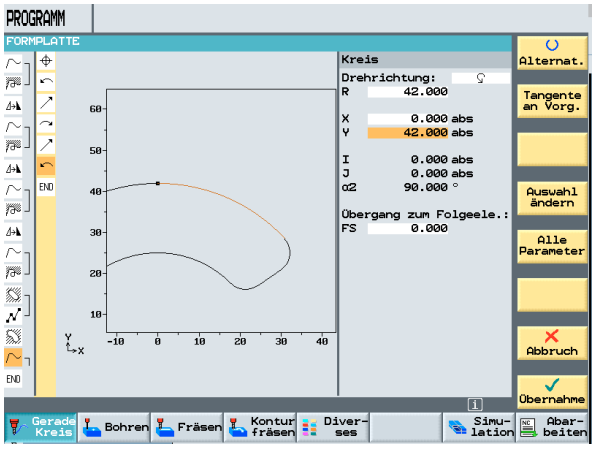
After opening the input screen form and pressing the softkey



the length of the oblique line as well as the angle to the predecessor element is entered.

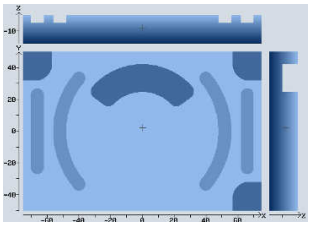


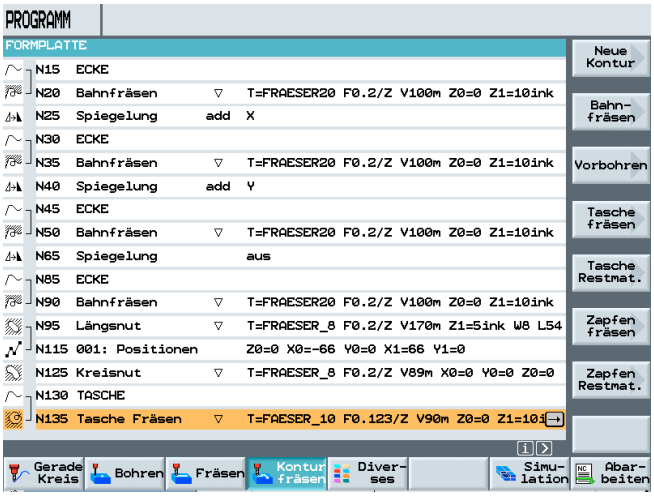
With the last element, the contour is closed.



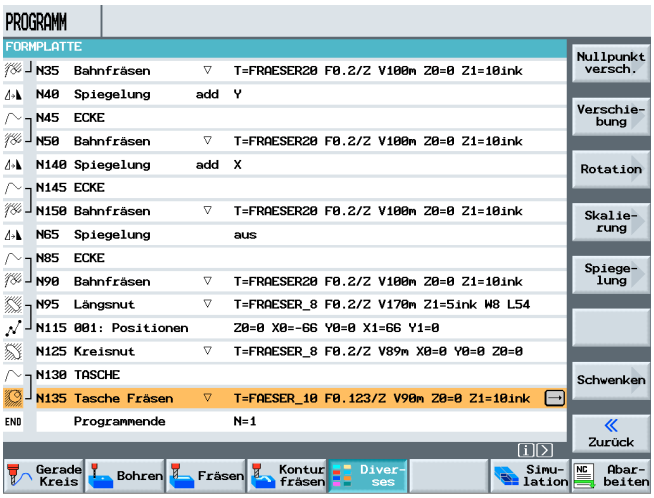
14 Rotation Contour Pockets

After accepting the contour into the machining plan and adding the processing “Contour pocket“, the first pocket is completely programmed.



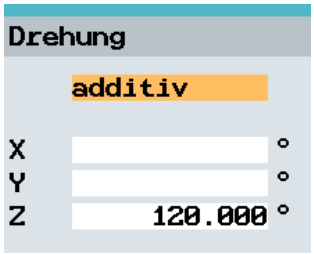


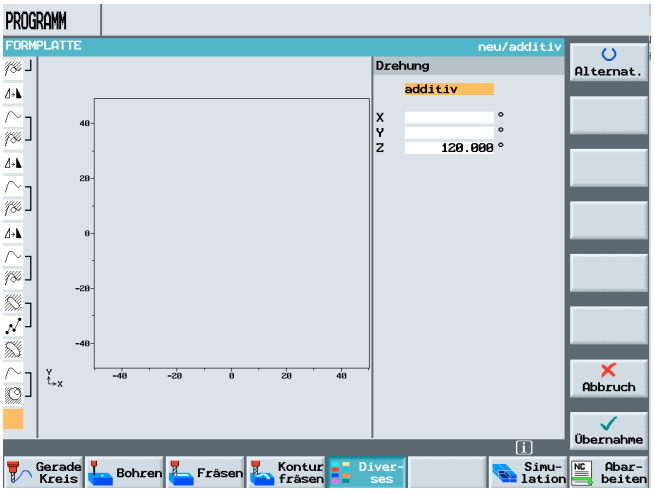
By pressing the softkey



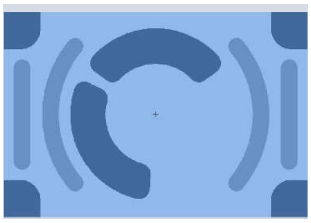
in the area “Diverses“, the contour pocket is rotated around the Z-axis.

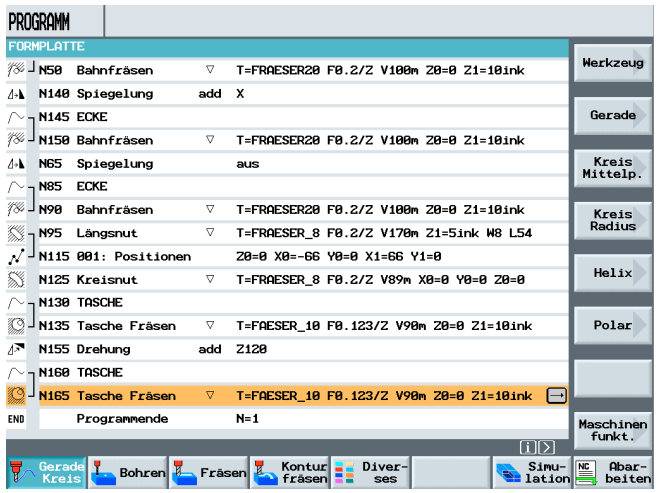
After entering the rotation



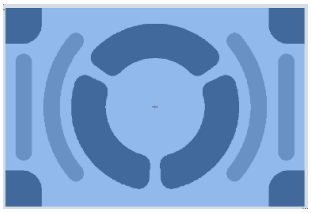


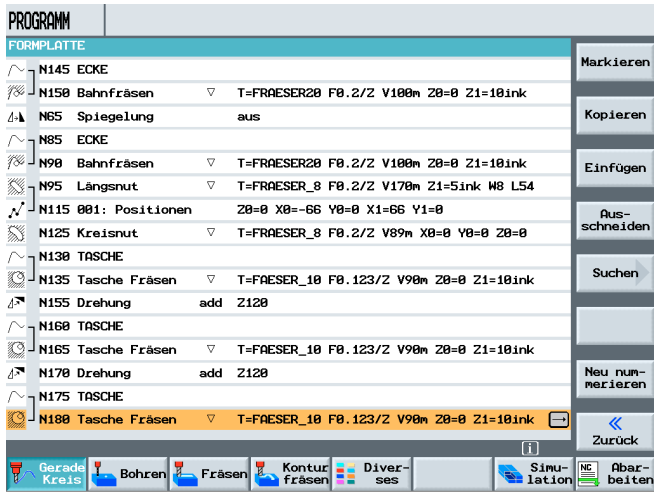
and copying and inserting the contour with the processing below the rotation, the second contour pocket is complete.



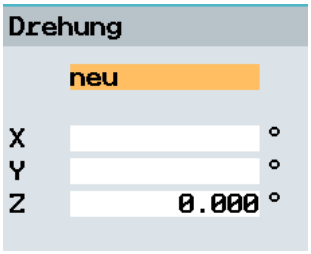


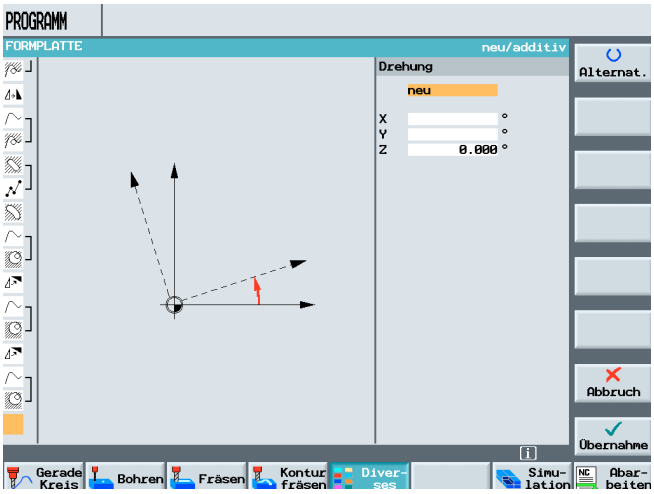
The last contour pocket is generated in the same way.





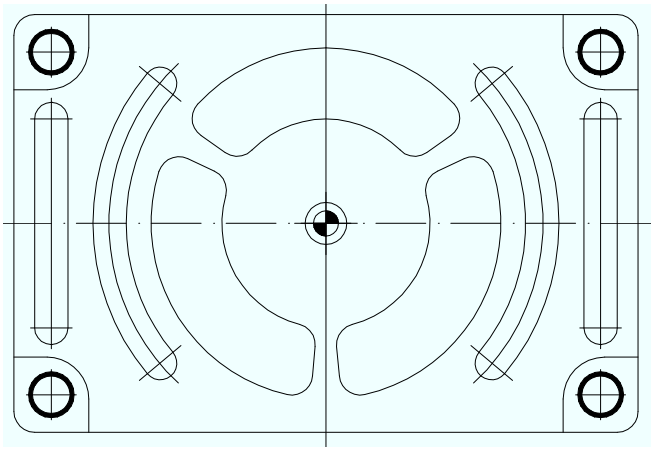
Delete the rotation





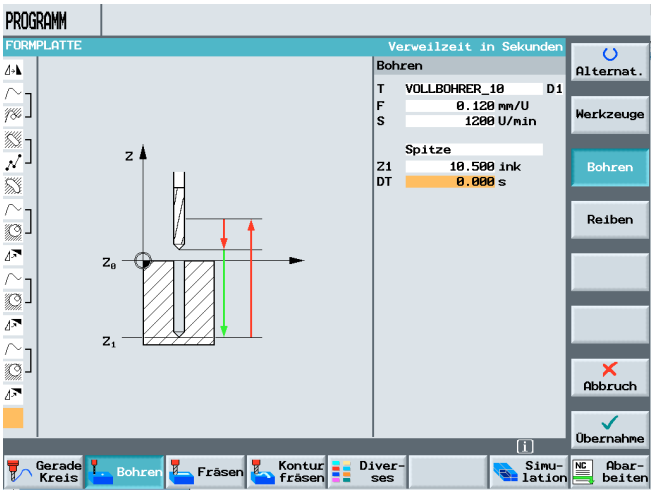
15 Making Holes with a Drill

As the last step, the holes are programmed.



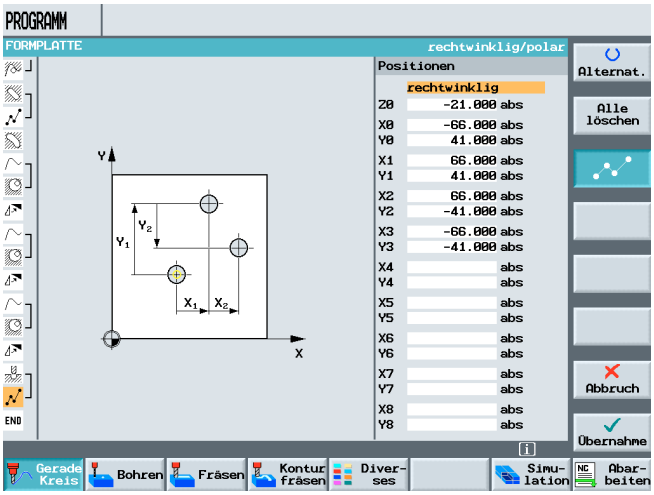
After entering the values

Verweilzeit in Sekunden		
Bohren		
T	VOLLBOHRER_10	D1
F	0.120 mm/U	
S	1200 U/min	
	Spitze	
Z1	10.500 ink	
DT	0.000 s	

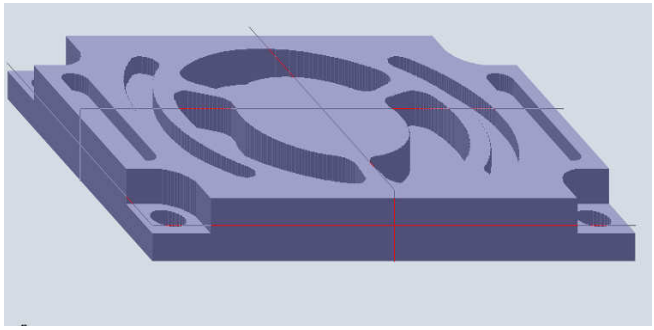


and the positions

rechtwinklig/polar		
Positionen		
	rechtwinklig	
Z0	-21.000	abs
X0	-66.000	abs
Y0	41.000	abs
X1	66.000	abs
Y1	41.000	abs
X2	66.000	abs
Y2	-41.000	abs
X3	-66.000	abs
Y3	-41.000	abs
X4		abs
Y4		abs
X5		abs
Y5		abs
X6		abs
Y6		abs
X7		abs
Y7		abs
X8		abs
Y8		abs

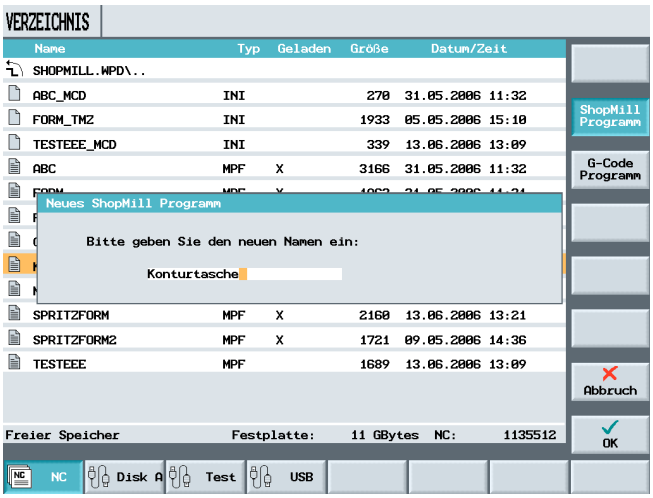


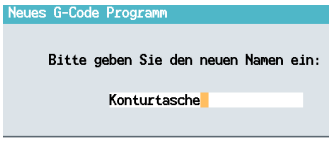
the program is complete.



16 Subprograms

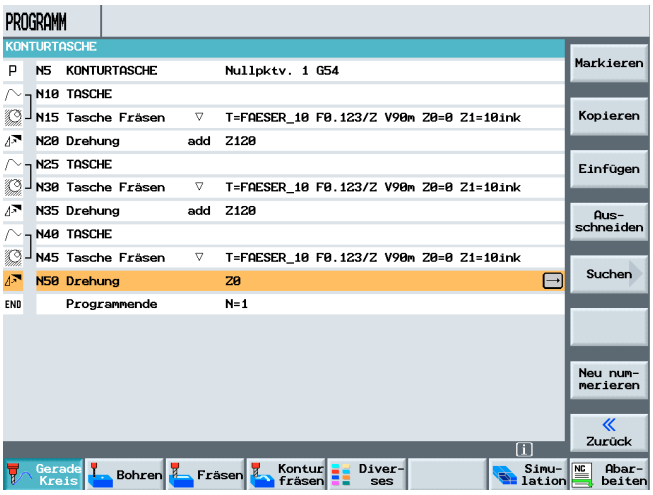
In this program, subprograms are inserted as repetitions. To make the program more transparent, the contour pockets are moved to a subprogram.





by cutting out the contour pockets from the program "Pattern Plate“ and inserting them into the new program “Contour Pocket“.

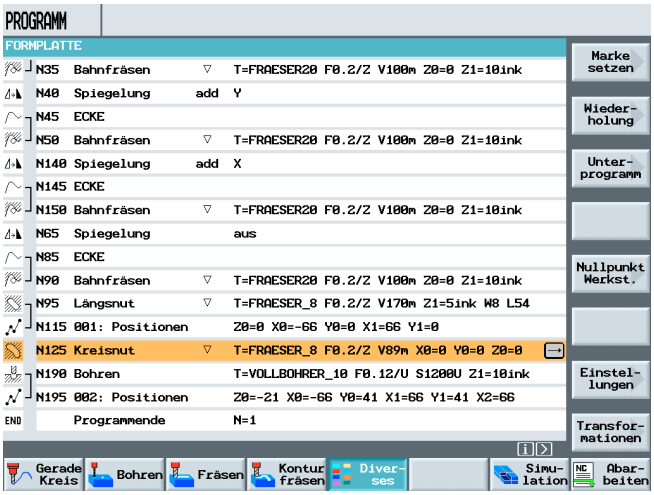
Then, the program is complete.



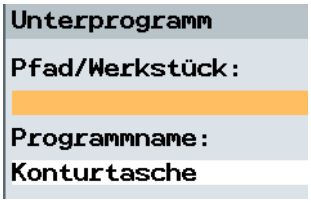
Now, the program has to be simulated once. This action calculates the program, and can be used as a subprogram .

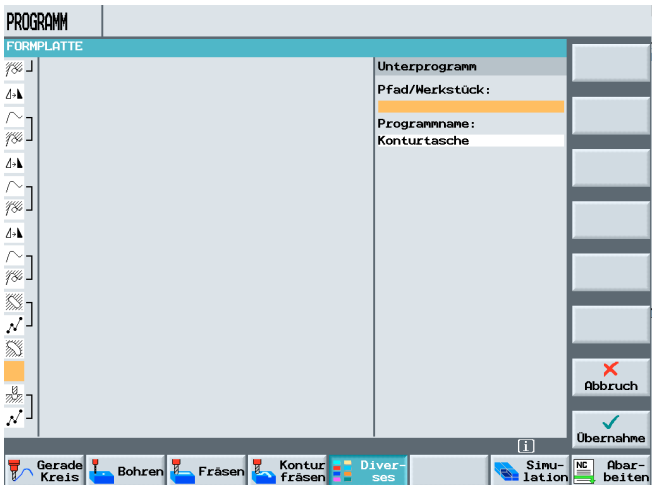
By pressing the softkey

in the area “Diverses“



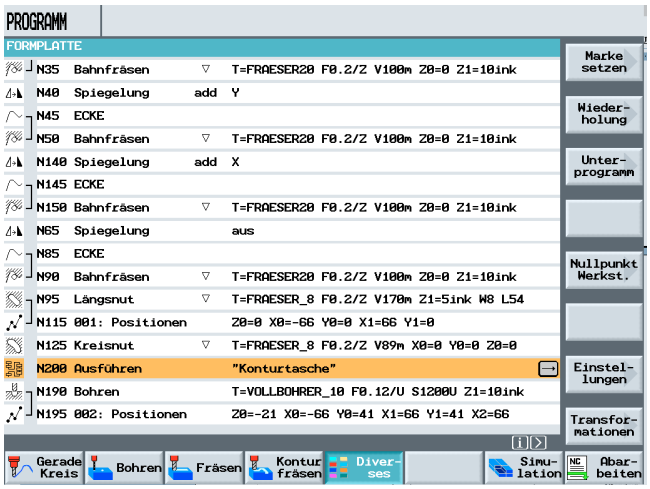
the name of the subprogram is entered.





If the subprogram is located in the same path, no input is necessary under Path/Workpiece.

The program name is entered without an extension such as *mpf* and accepted into the machining plan.



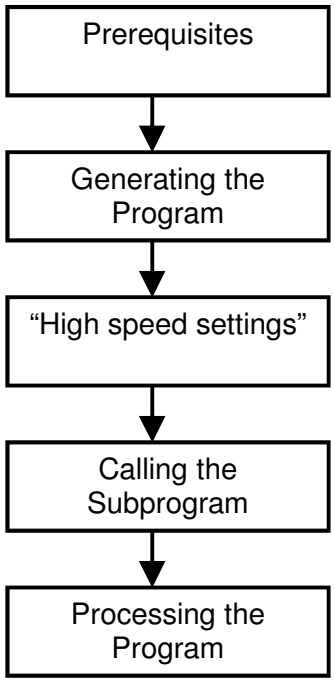
The subprogram is complete.

Any main program can also be used as subprogram!

17 Mould Making - Milling

A mould making program is generated, using an example.

Sequence



17.1 Prerequisites

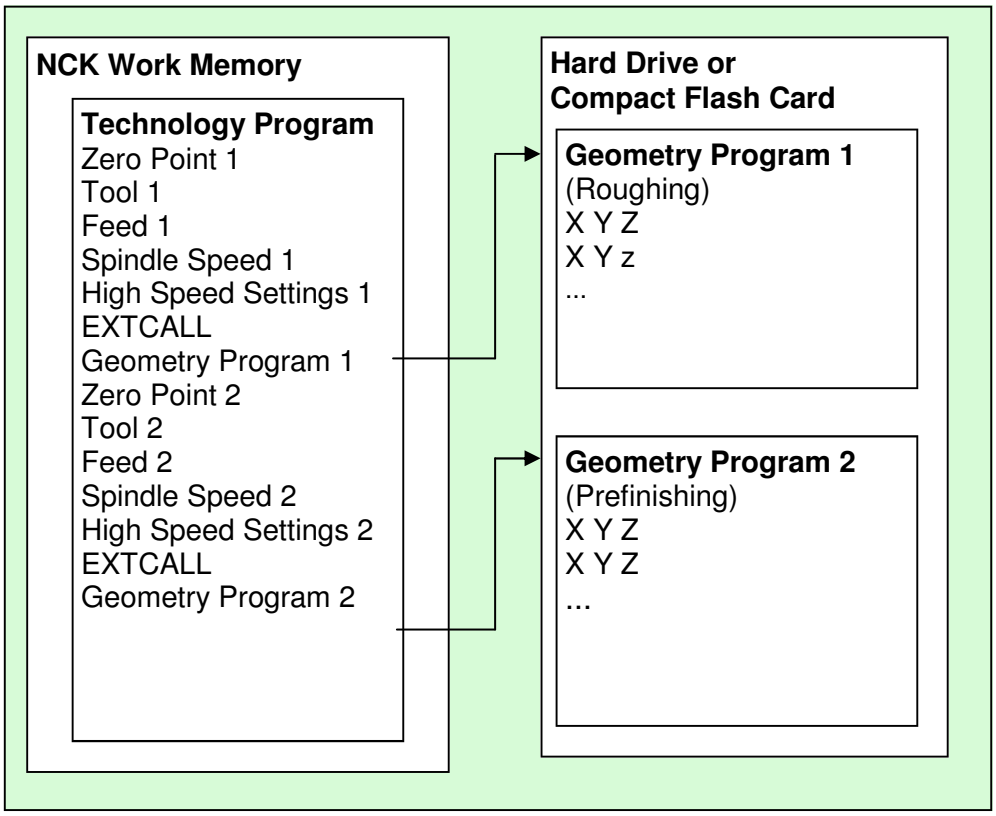
In addition to machining step programs, ShopMill can also process G-code mould making programs.
The prerequisite for this are optimized drives.

Program Structure
In order to attain the optimum velocity control for the mould making programs, you should divide the mould making program into a central technology program and separate geometry programs, and not generate one complete program.

Technology Program
The technology program includes the basic settings such as zero point shift, tool calls, feed values, spindle speed, and control commands for velocity control.
In addition, the technology program calls the geometry programs as subprograms.
The technology program can be generated in the ShopMill's G-code editor..

Geometry Program
The geometry programs of the individual machining modes (roughing, prefinishing and finishing) exclusively contain the geometry values for the free form surface to be machined. The geometry programs are generated on an external CAM system in the form of G01 blocks. Depending on their application, the geometry programs have a size of 500KB up to 100MB. Programs of this size can no longer be processed directly in the NCK work memory, but have to be processed externally by means of EXTCALL. That means, the geometry programs have to be stored either on the hard drive of the PCU 50.3 (HMI Advanced) or on a Compact Flash Card at ShopMill on NCU (HMI Embedded). For both ShopMill variants you also have the option to store the geometry programs on a network drive.

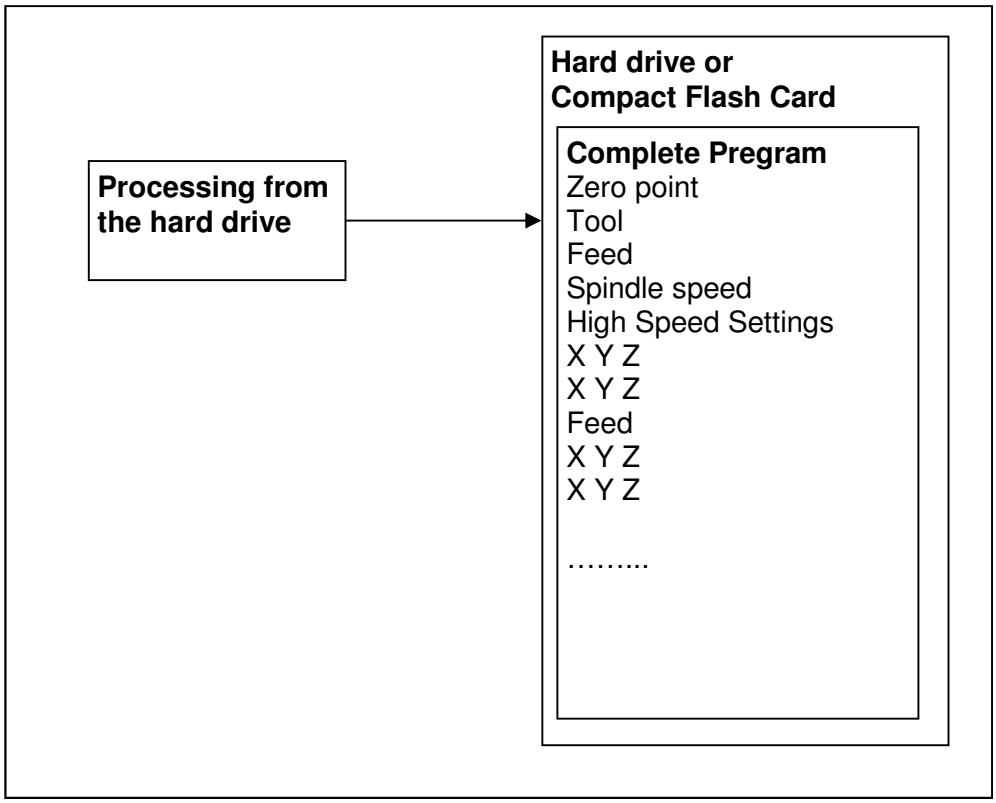
17.2 Program Structure Technology Program with Geometry Program



Complete Program

Complete programs include the basic settings such as the zero point shift, tool function <<T word>> etc. as well as the geometriy values of the free form surface to be machined. However, programming the optimized velocity control for a complete program is very complicated.
Complete programs are also generated on enternal CAM systems. Because of their size, the complete programs are located on the hard drive of the PCU 50.3 (HMI Advanced) or on the CompactFlash Card at ShopMill on the NCU (HMI Embedded).
Here also you have the option to store the complete programs on a network drive.

17.3 Program Structure Complete Program



Data Transmission

A mould making program can be copied directly to the controller from a network drive or a USB drive.

• ShopMill on NCU (HMI Embedded)
• The programs are copied to the user memory of the CompactFlash card.
• PCU 50.3 (HMI Advanced)
The programs are copied to the hard disk drive.

Measuring the Tool

When generating the geometry program, the CAM system takes the tool geometry into account. The calculated tool path refers either to the tool tip or the tool center point. That means, when you specify the length of your tools, you have to use the same reference point (tool tip or tool center point) as the CAM system.
If you are using a ShopMill function for measuring your tools, the tool length refers to the tool tip. If, on the other hand, in the CAM system the tool center point was taken into account when calculating the tool path, you have to deduct in the tool list the radius of the tool from the length of the tool. To process mould making programs, the entry of the tool diameter in the too list is not relevant. However, to have a better overview, you should enter the tool diameter in the tool list nevertheless.

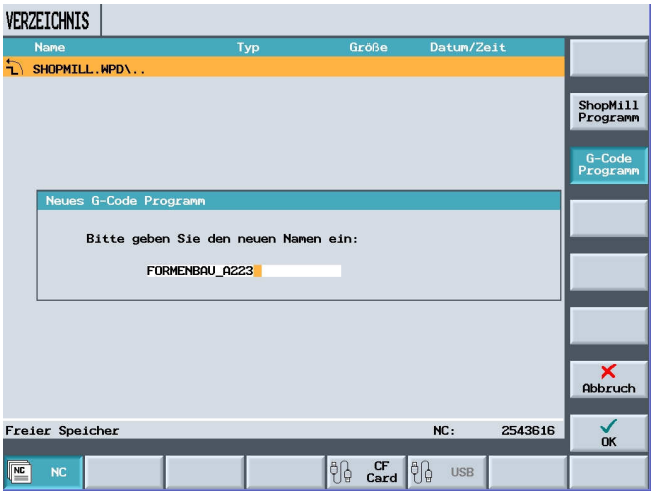
17.4 Creating the Program

Setting Up the Program

For the technology program, set up a new G-code program in the program manager and then edit it there.
Editor. *A machining step program is not suitable as a technology program.*

Create the geometry program or the complete program with an external CAM system. If afterwards you would like to add comments, for example, to the geometry program, or change the tool name in the complete program, you can use also the ShopMill G-code editor for this.

Use the arrow keys to open a directory.



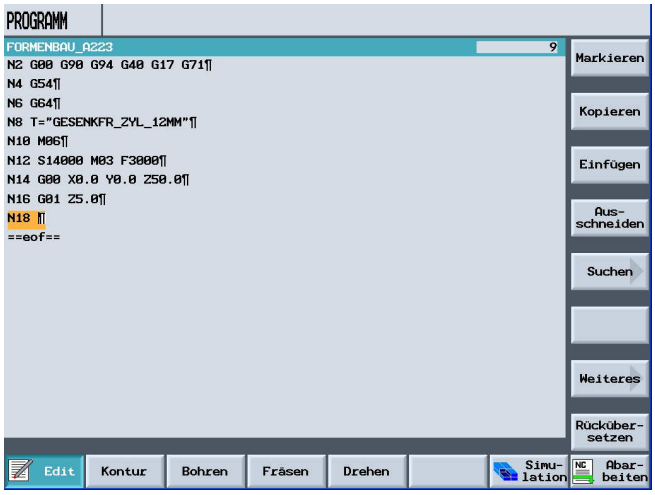
After entering a program name,

the G-code Editor under ShopMill is opened.

Programming the Tool

If you program a tool in the technology program, you have to take note of the following:
The geometry of the programmed tool has to agree with the tool geometry that the CAM system took into account when the geometry program was generated.

First, the tool, the spindle speed and the spindle direction are programmed. In addition, the following is programmed: feed, switching on the coolant, and the zero point shift with the starting point.

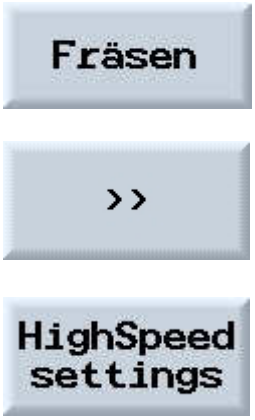


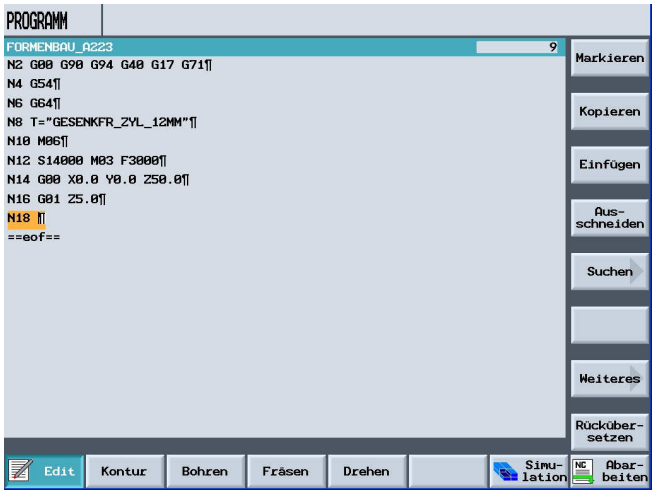
17.5 High Speed Settings

Programming the Cycle "High Speed Settings"

When machining free form surfaces, great demands are made on the speed as well as the accuracy and surface quality. You can attain the optimum velocity rate in dependence on the machining mode (roughing, pre-finishing, finishing) very simply by using the cycle "High Speed Settings". You can call the cycle by means of the cycle support in the G-code editor. As a rule, the output tolerance of the post processor of the CAM system is entered in the parameter "Tolerance". Program the cycle in the technology program prior to calling the geometry program.

After pressing the softkeys





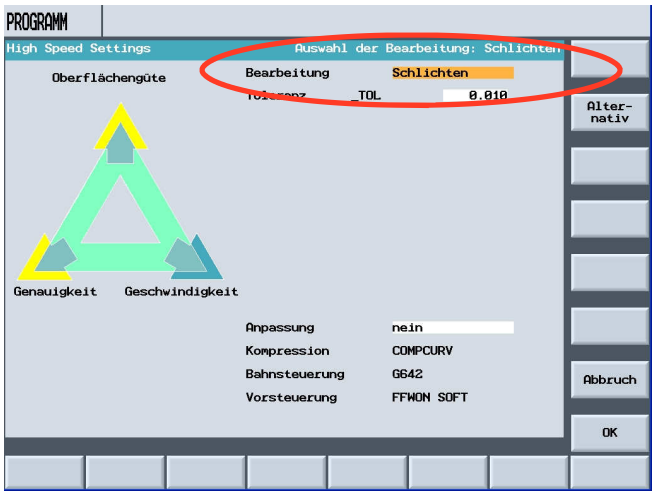
High speed settings

the following input screen form opens, where we can select among the machining modes

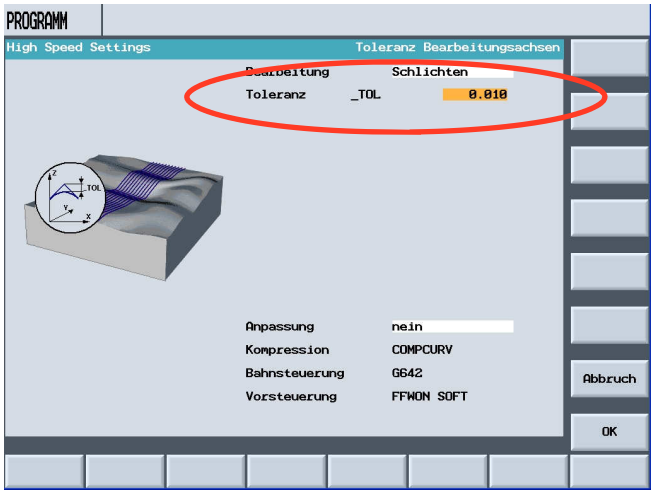
Schlichten

Vorschlichten

Schruppen



Tolerance
The required tolerance can be entered in the input screen form in the cycle for the program generated from the CAM system.



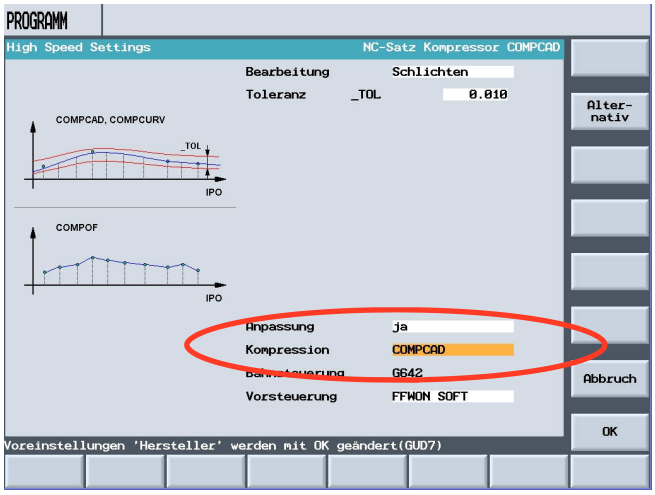
If the compressor is active, it can be parameterized by means of the input screen form.

By pressing the softkey

we can select between

COMPCAD

COMPCURV



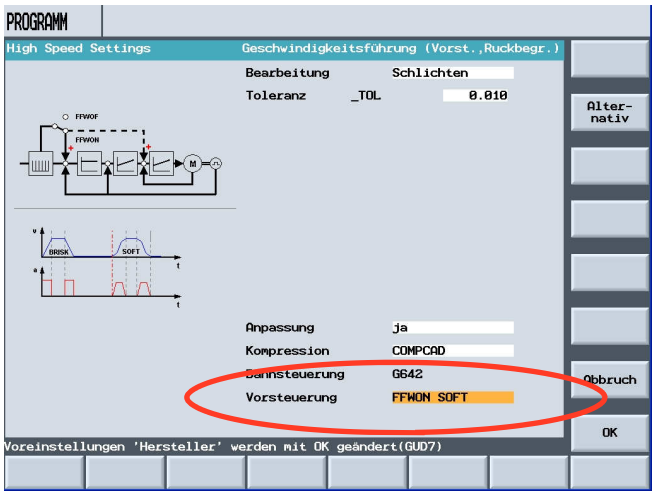
By pressing the softkey

we can select among

FFWOF SOFT

FFWON SOFT

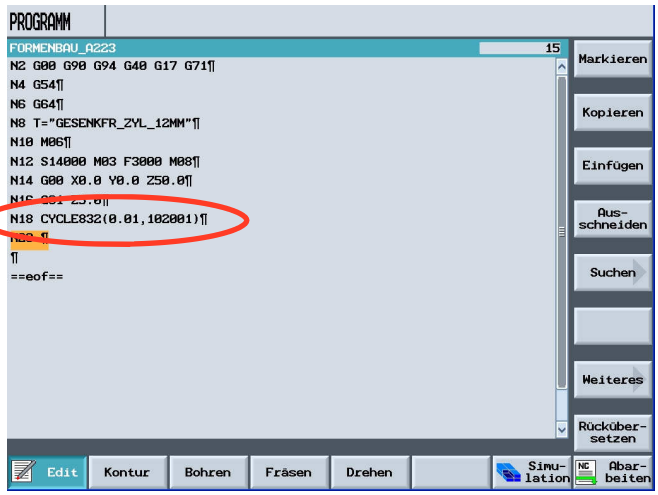
FFWOF BRISK



Pressing the softkey

closes the input screen form.

“CYCLE832” was accepted into the program.



17.6 Calling the Subprogram

Call the geometry program as subprogram from the technology program. Since the geometry programs are not stored in the NC work memory but on the hard disk drive of the PCU 50.3, or on the Compact Flash Card of theTCU, or on a network, the subprogram has to be called with the G-code command "EXTCALL".

PCU 50.3

The technology program and the geometry programs are located in the same directory on the hard disk drive.
EXTCALL "Geometry program"
Example: EXTCALL "SCHRUPPEN"
<<ROUGHING>>

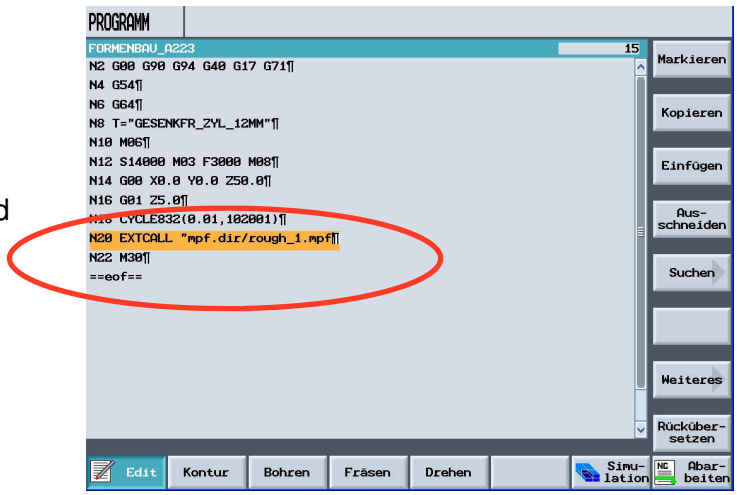
NCU HMI Embedded

Depending on the storage location of the geometry program on the Compact Flash Card, the programming syntax differs somewhat.
• The geometry program is located directly on the Compact Flash Card
EXTCALL ("C:\Geometrieprogramm.mpf")
Example: EXTCALL ("C:\Schruppen.mpf")
• The geometry program is located in a directory on the Compact Flash Card
EXTCALL ("C:\Verzeichnis\Geometrieprogramm.mpf")
Example: EXTCALL ("C:\Mold\Schruppen.mpf")

Network drive

If the geometry program is located on a network drive connected by means of the Ethernet, the programming syntax is as follows.
EXTCALL ("Pfad\Geometrieprogramm.mpf")
Example: EXTCALL ("H:\Mold\Schruppen.mpf")

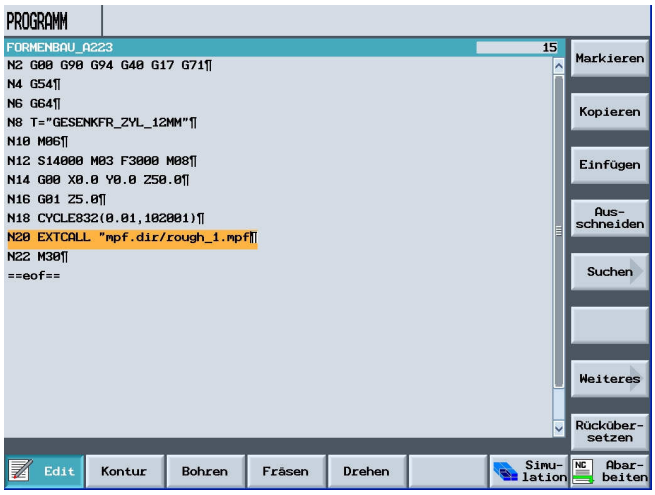
Here, a program is called that is stored on the CFCard as mpf– program.

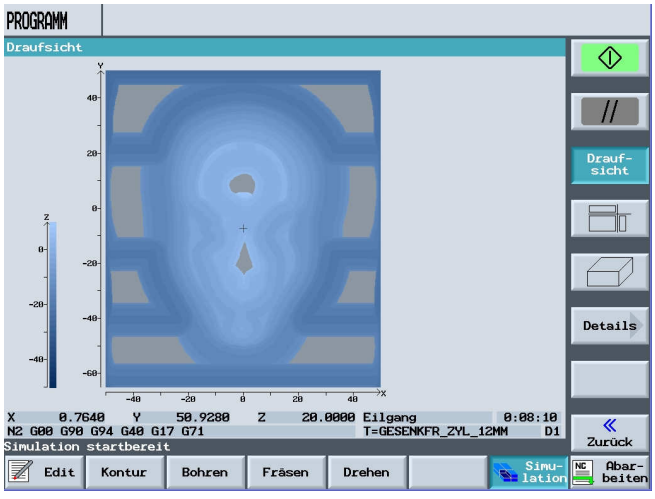


17.7 Processing the Program

Select the technology program that is located in the NCK work memory like a standard G-code program for processing. The geometry program is then selected automatically with the command "EXTCALL".
A complete program that is located either on the hard disk drive of the PCU 50.3 (HMI Advanced), or on the Compact Flash Card at ShopMill on NCU (HMI Embedded), or on a USB /network drive, is selected with the softkey "Execute hard drive" in the program manager.

Prior to executing the program, the program is simulated graphically by pressing the softkey

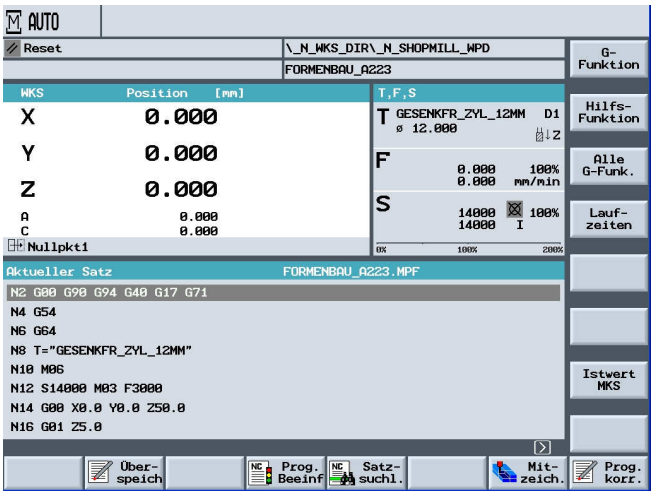




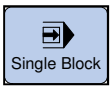
Pressing the softkey

and

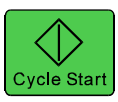
the program is selected in the operating area “Auto”.



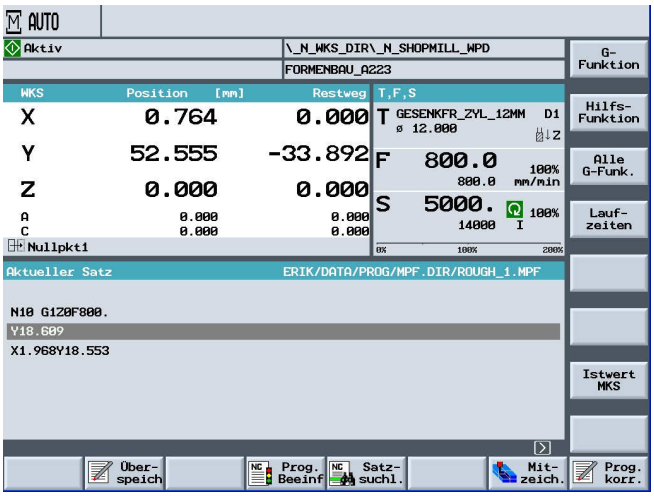
The program can be executed in the single block.



With “Cycle Start” the program can be executed step by step.



After positioning, the program can be executed without a single block.



17.8 Starting Processing at a Certain Program Location

Executing the Program

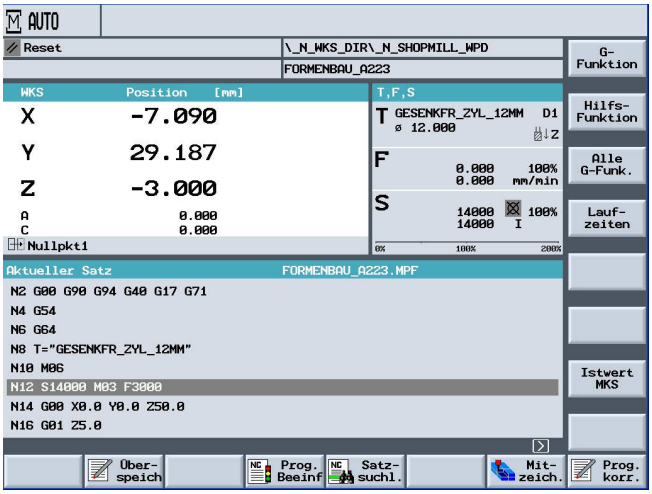
Starting Processing at a Certain Program Location

In order to start in a geometry program the execution of a certain program segment, enter the destination in the search pointer.
Layer 1 (technology program):
Program line with the call of the desired geometry program

Layer 2 (geometry program):
Program line for starting processing.
If the geometry program is located on the Compact Flash Card, you not only have to specify in Layer 2 in the input field "Program" the program name, but also the path.
Select the accelerated calculation variant "External – without calculation". The block search in the technology program is performed with calculation. In that case, all EXTCALL commands before the desired geometry program are skipped. The block search in the desired geometry program is performed without calculation. However, this calculation variant presupposes that all machine functions such as tool function, spindle speed etc. are located in the technology program. The geometry program must only contain geometry values for the free mold surface.

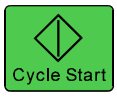
The line for starting the program can be selected with the arrow keys.

After pressing the two additional softkeys

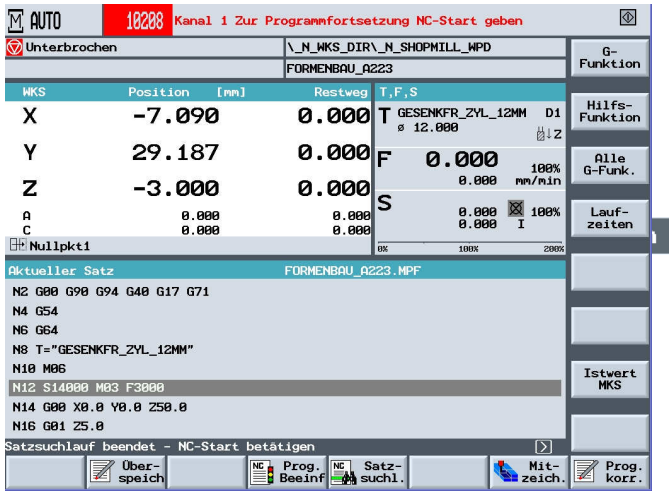


the selected program block is active.

By then pressing the key



the message 10208 Channel 1 for contunuing the program appears. Press NC Start

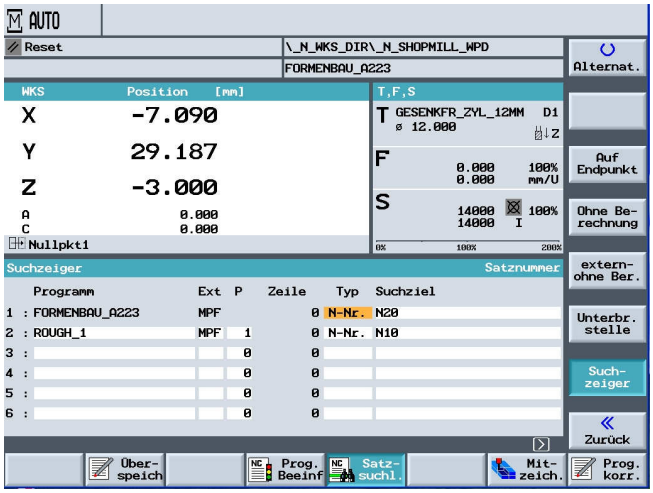


If we want to start from a certain program point, the program is started by pressing

and

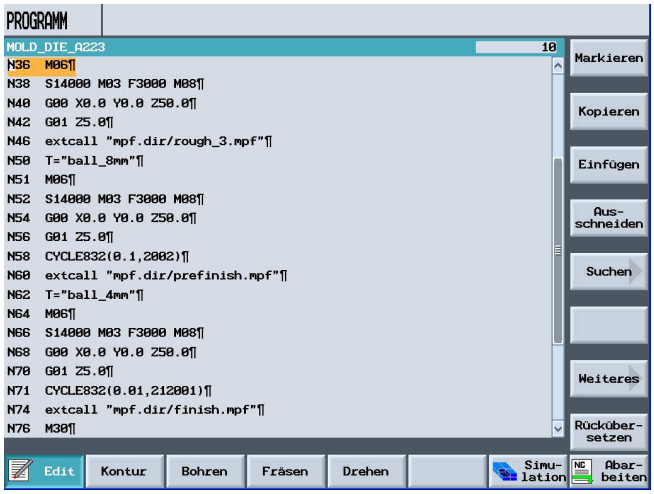
and entering the program number

and pressing the softkey

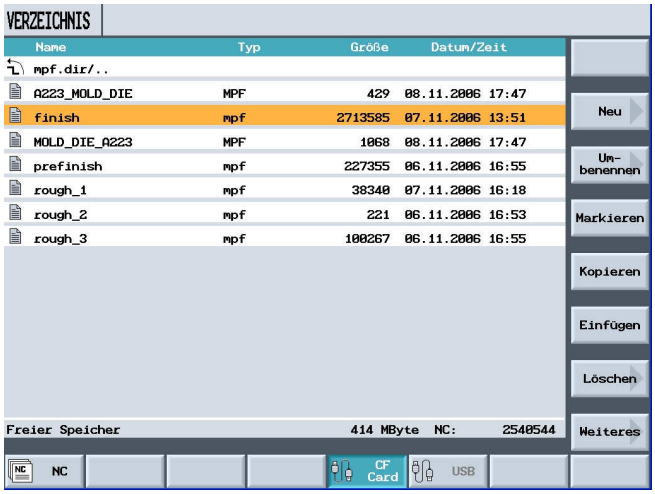


Sample Program

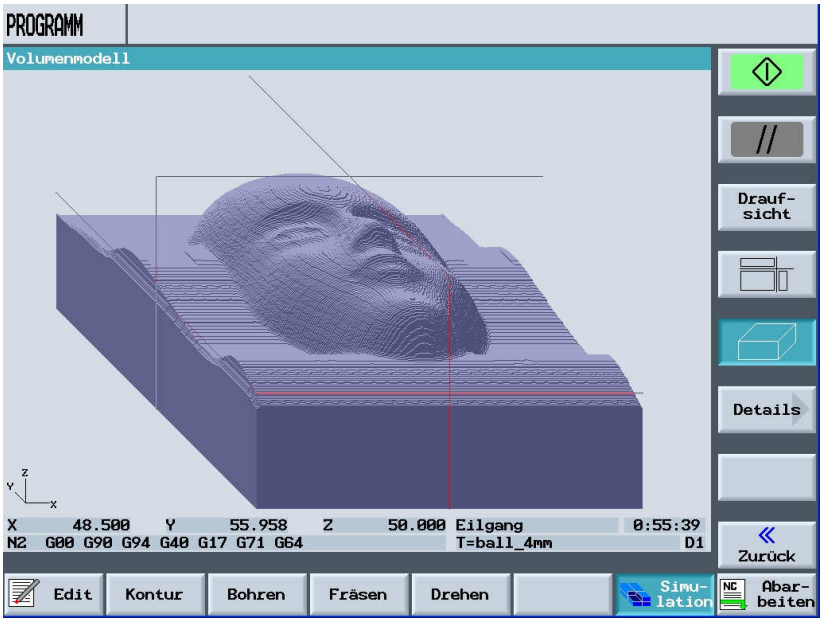
In the program area “NC”, a mould making program is opened.



The corresponding technology data is located on the “CF Card“. With a mouse click, it was loaded to the “CF Card“ with a USB stick.

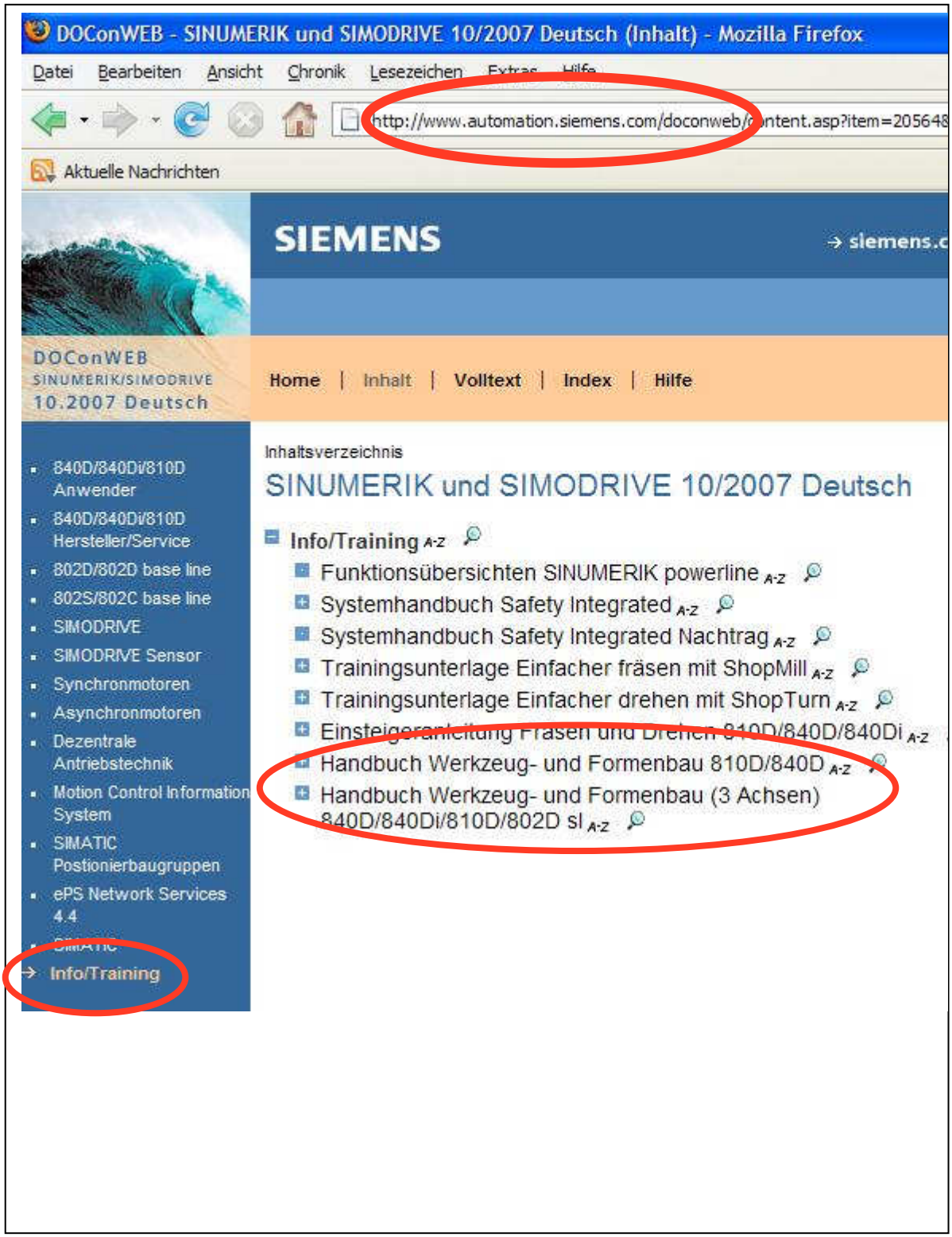


17.9 Simulation of a Volume Model



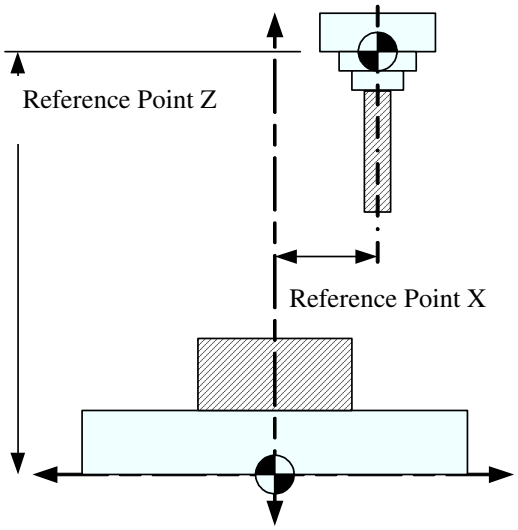
18 Information about Mould Making

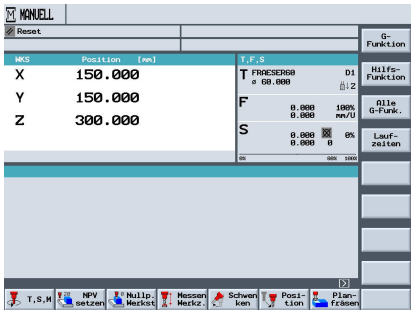
Information about mould making is provided on the Internet under
www.automation.siemens.com/doconweb/

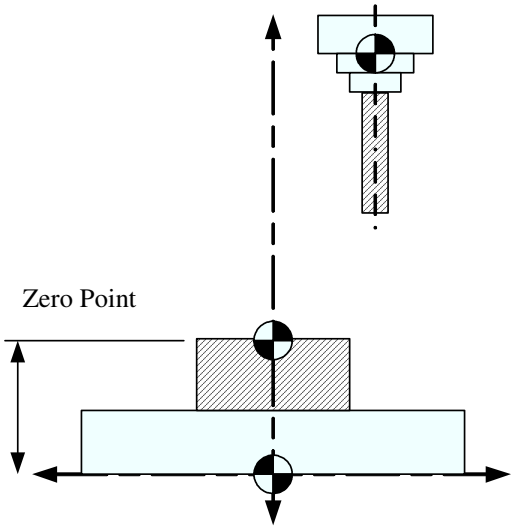


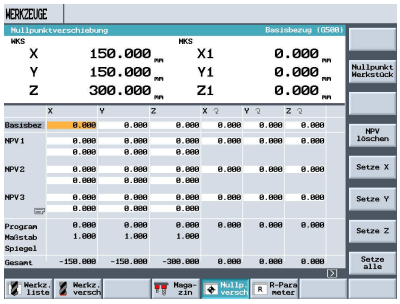
19 Fundamentals of CNC Machines

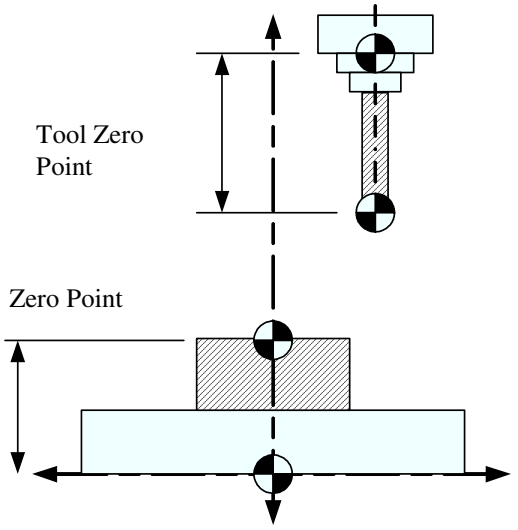
The difference between a manual and a CNC machine consists of the logic operation of numerical values. This module about the basics of CNC technology is to aid you regarding the CNC myth.

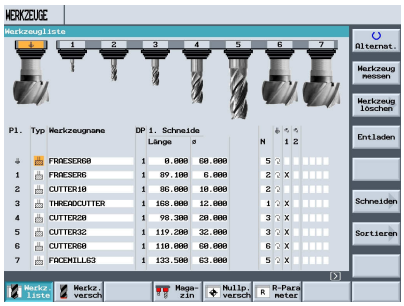






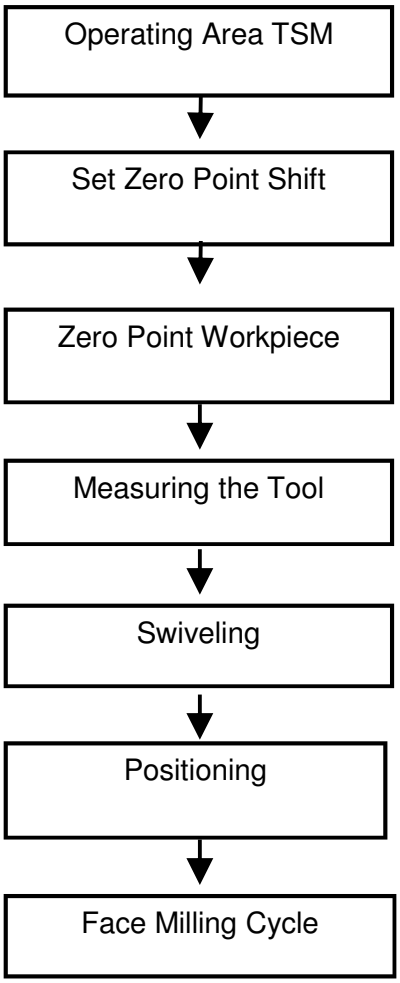






20 Manual Operating Area - Milling

In this module, the individual function areas in the manual operating area are presented and if necessary, explained by using an example.

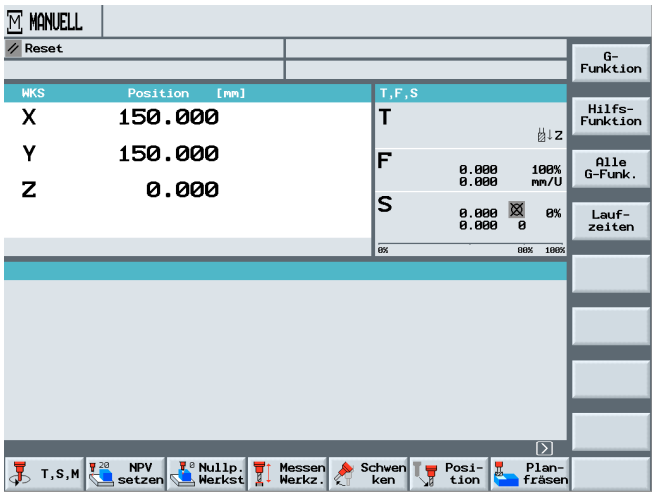


20.1 Operating Area TSM

Tool Spindle Speed Machine Functions

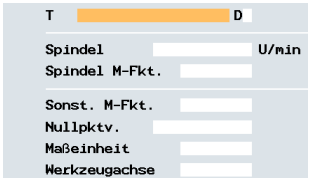
After starting ShopMill, the operating area

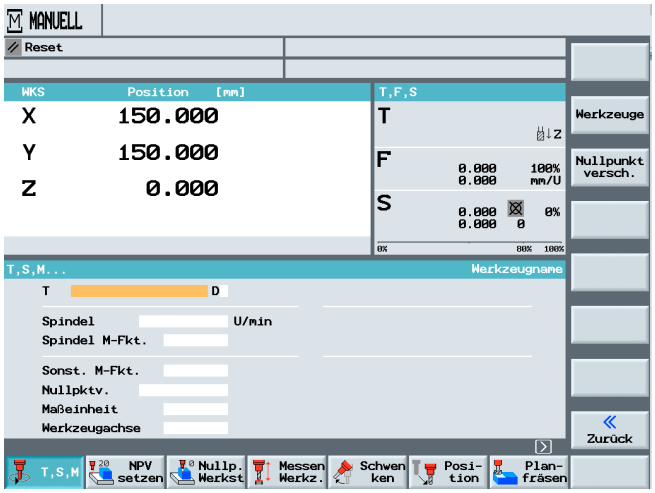
is active.



By pressing the softkey

the input window for operating the machine manually is displayed.

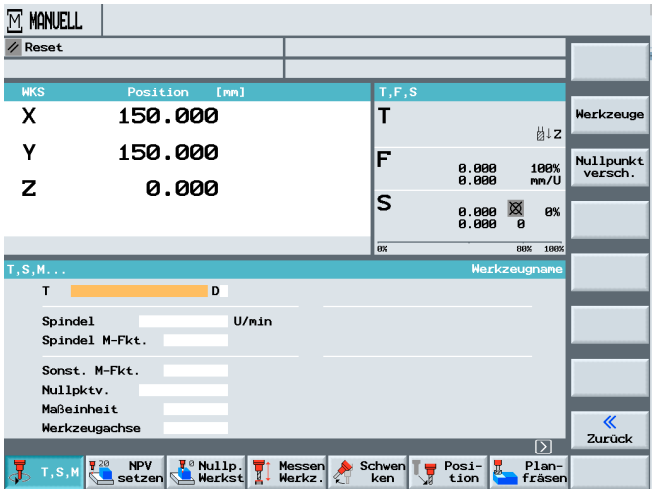




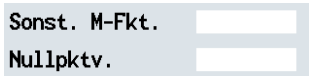
In the first input screen form, the tool is defined.

The spindle speed can be entered only in

.



In the additinal input fields, M-functions and zero point shifts can be input.

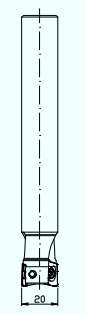


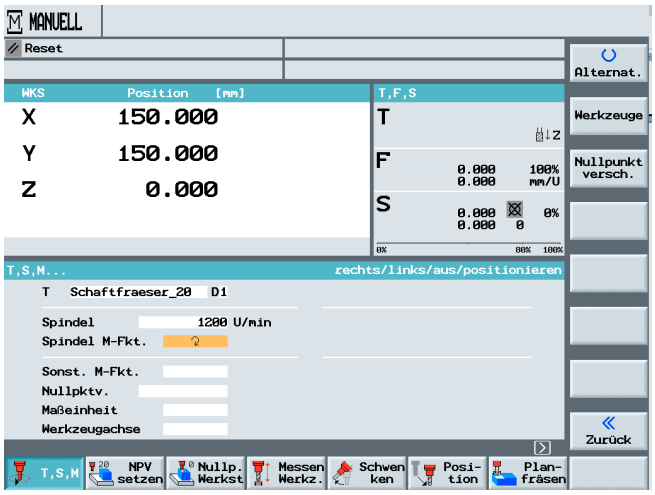
We can select between the measurement units

The tool axis

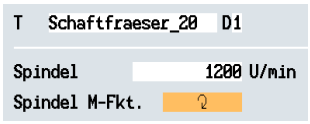
can also be selected.

Example:

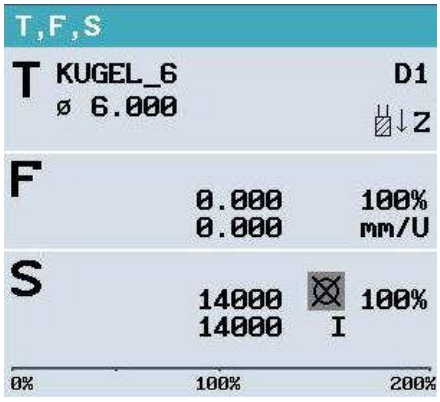




After calling the tool with the corresponding technology

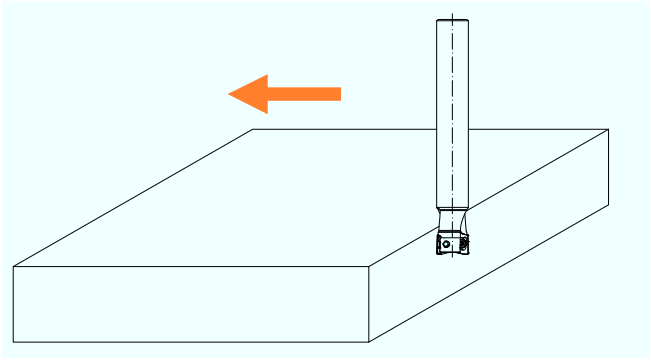


the tool with the input technology data is activated by pressing the NC start key



20.2 Operating Area Set NPV

The operating area "Set NPV“ is needed for synchronizing the axes and the workpiece.

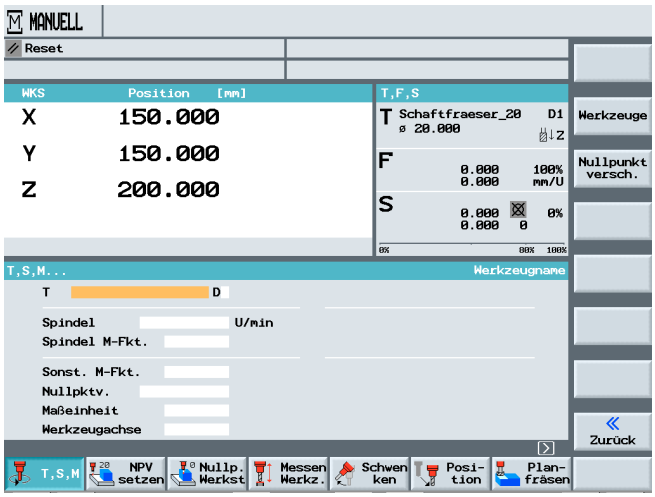


Example:

The edge of the workpiece is scratched.with a milling tool.

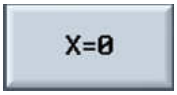
By pressing the softkey

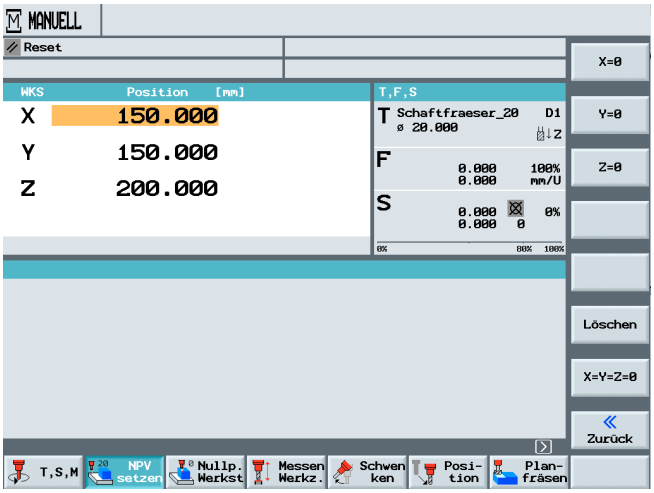
the value for the current axis is colored.



Using the following softkeys, each axis can be “zeroed“..

Also, using the machine keyboard, any value can be entered in the field that is currently selected, or with



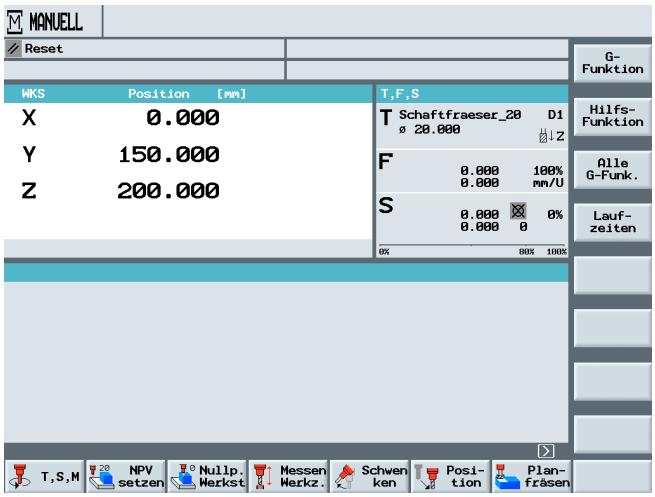


the value of the axis can be set to zero.

20.3 Operating Area Zero Point Workpiece

The value is taken into account in the **active** zero point shift and entered in the zero point table

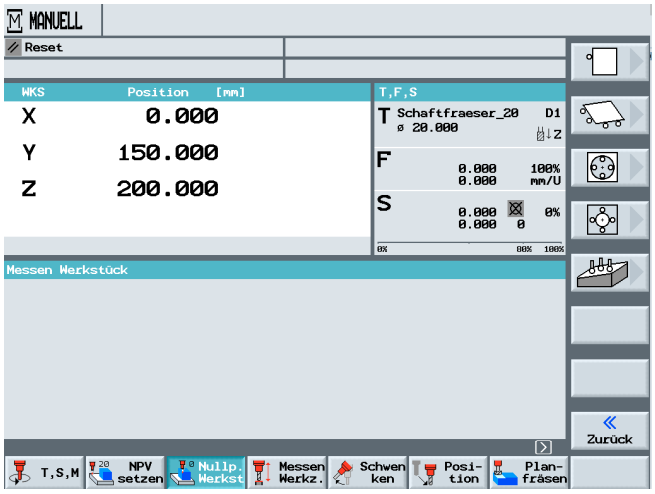
X	0.000
Y	150.000
Z	200.000



In this operating mode, the value for the workpiece measurement can be written directly into the desired zero point shift.

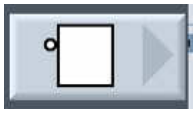
ShopMill Version 6.4 offers many options for this.

Pressing the softkey



offers five options for determining the zero point.

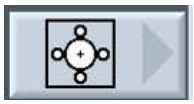
Edge



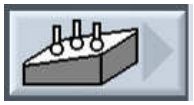
Corner

Drilling rectangular pocket

Spigot Rectangle

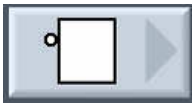


Aligning the plane

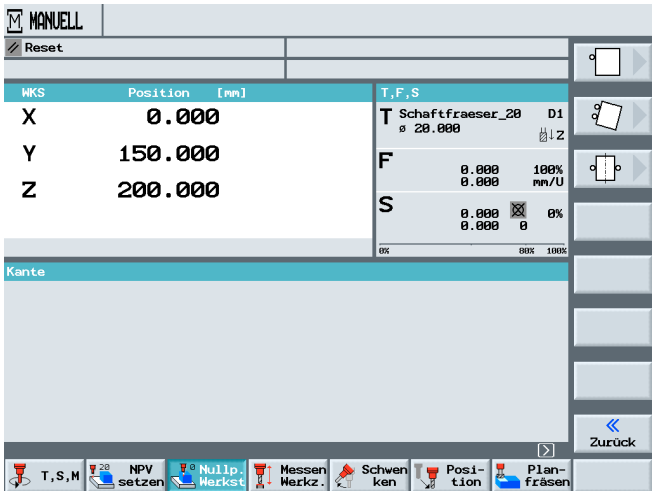


The described cycles can be utilized by using a manual touch probe or an automatic touch probe.

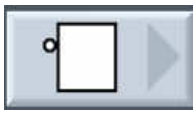
By pressing the softkey



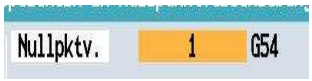
the options for “Edge measurement“ under ShopMill are displayed.



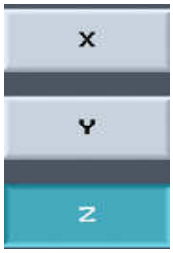
Pressing the softkey



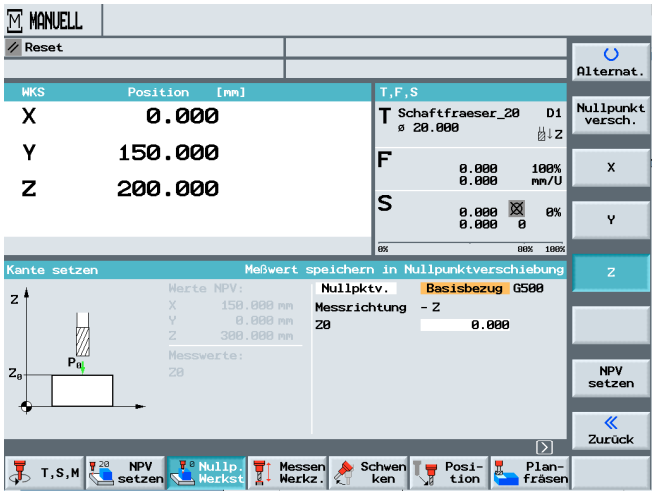
opens the input screen form.
After selecting the zero point shift



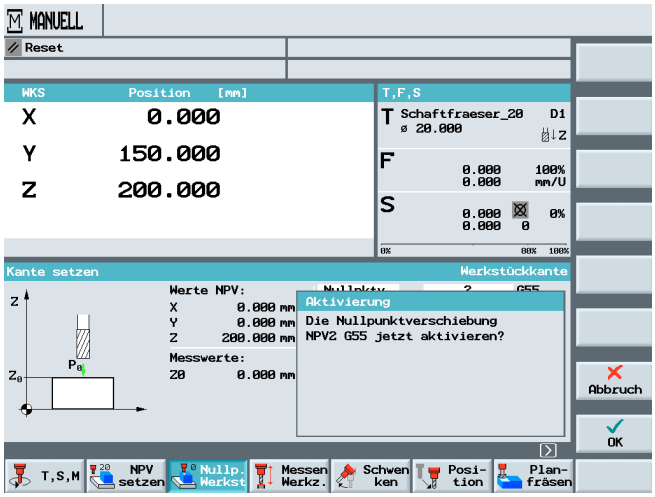
and the axis



measurements can be made.



After the measurement is performed into <<?>> a non-active zero point, a query is displayed whether the zero point is to be activated.



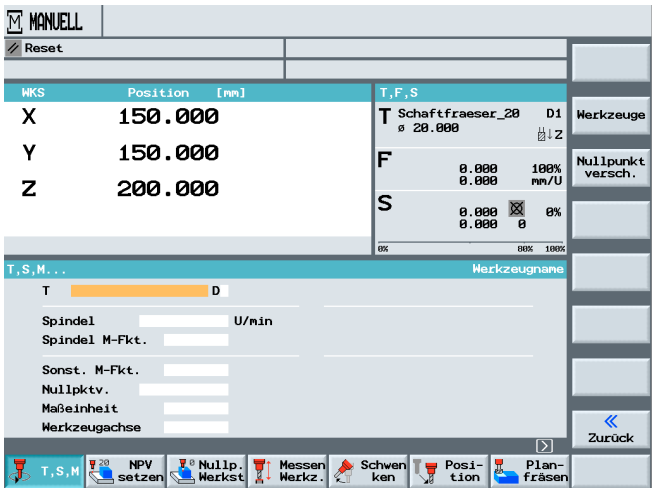
By pressing the softkey

the input is accepted and the zero point is activated.

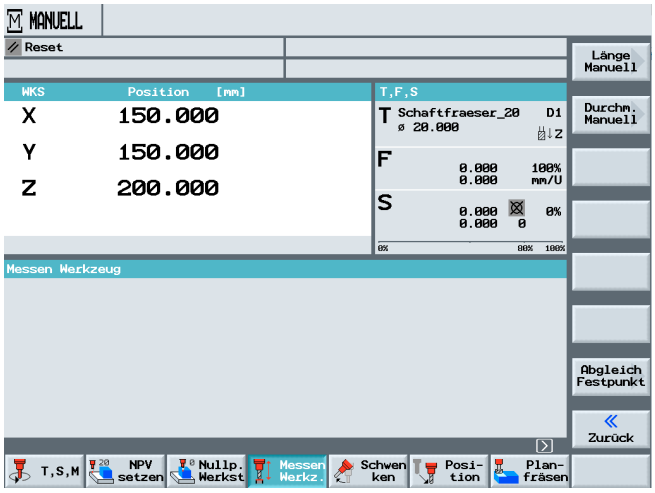
20.4 Measuring the Tool Length

By pressing the softky

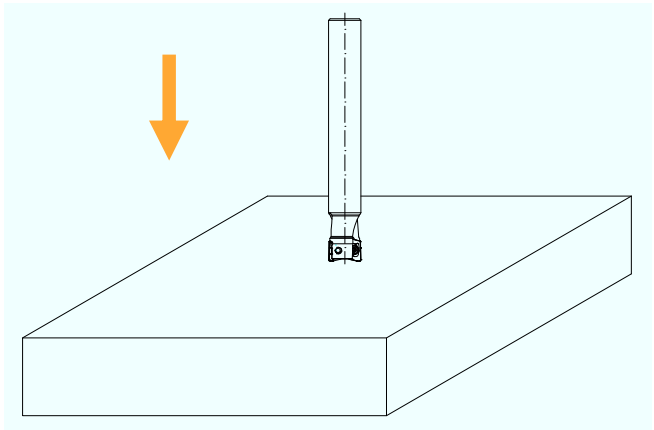
in the operating area



and pressing the softkey

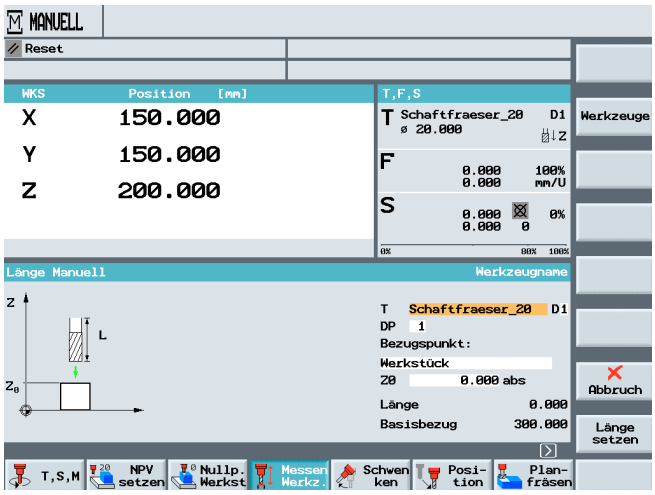


the tool is measured by starting the workpiece in the length - Z – direction.



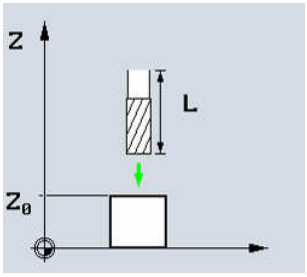
After selecting the tool

T Schaftfraeser_20 D1



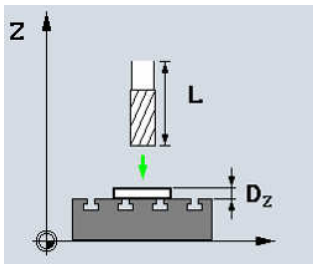
selecting the reference point whether

Werkstück



or

Festpunkt

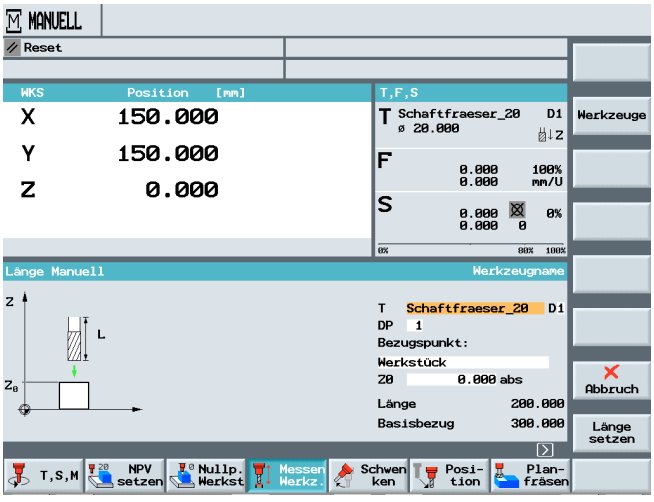


a tolerance can be entered after “scratching“ the surface of the workpiece.

Z0 0.000 abs

By pressing the softkey

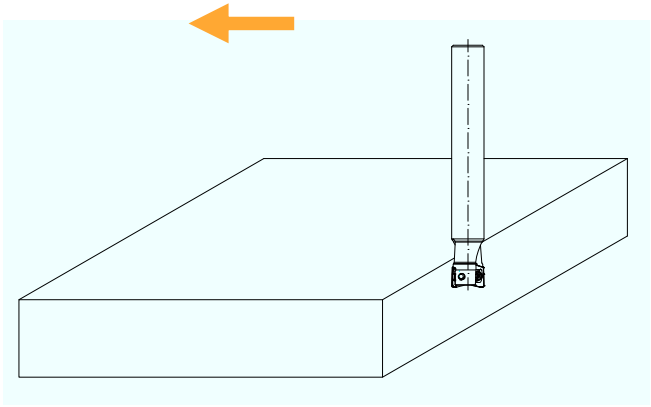
Länge setzen



the measured tool length is activated and accepted into the tool list.

20.5 Measuring the Tool Radius

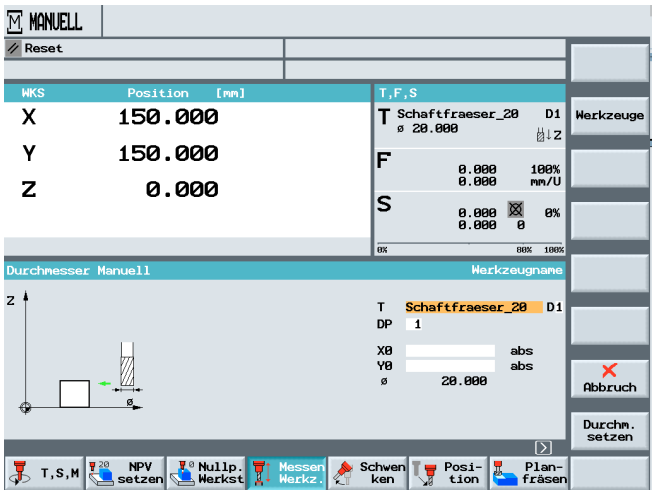
Next, the tool's diameter is ascertained.



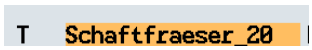
By pressing the softkey

in the area "Measure tool",

the corresponding input screen form opens.



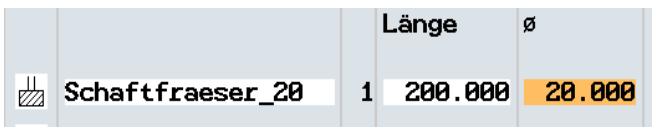
After selecting the tool



a tolerance can be entered in reference to the tool edge in X and in Y.

After approaching the workpiece edge with the tool, the measured diameter is activated by pressing the softkey

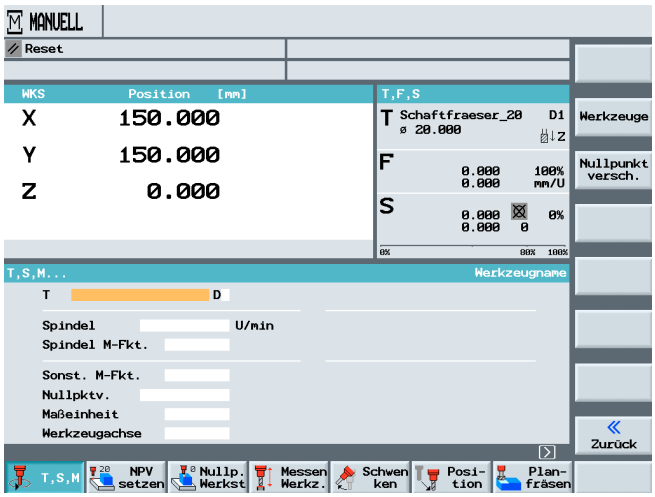
and accepted into the tool list.



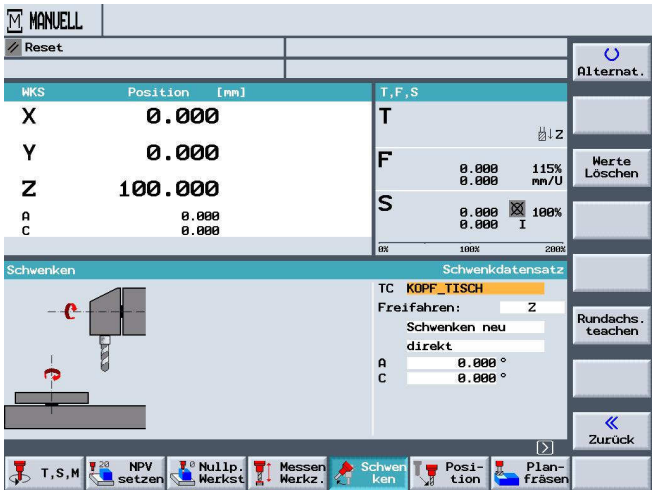
20.6 Swiveling

By pressing the softkey

in the operating area Manual



the area for manual swiveling under ShopMill opens.

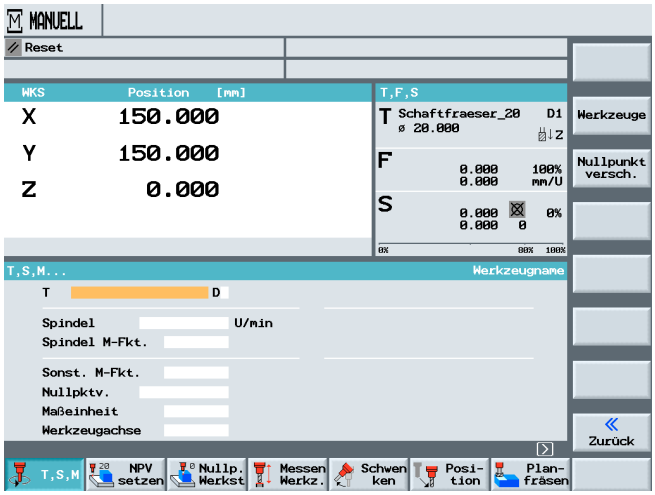


Manual swiveling under ShopMill is not part of the training document.

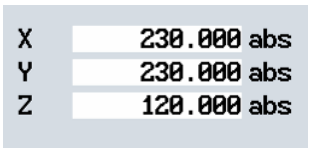
20.7 Manual Positioning

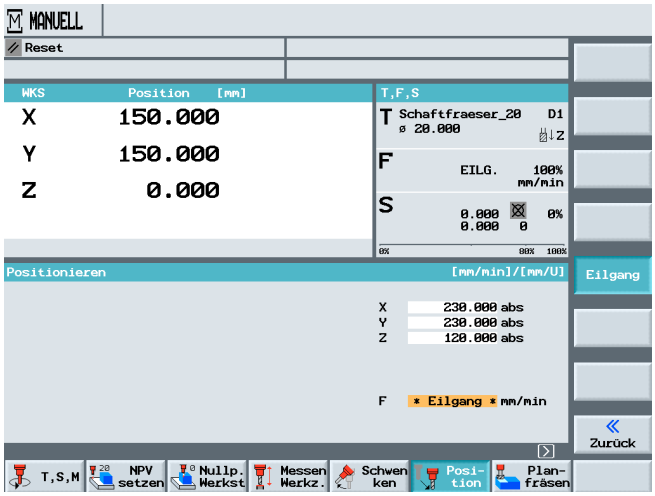
By pressing the softkey

in the operating area

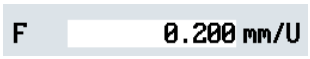


the target position is entered in the input fields.





The travel motion can be entered in feed or



rapid feed

.

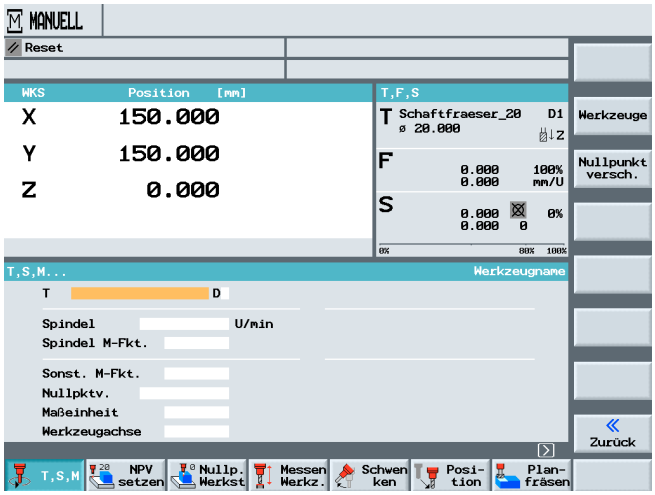
There is no collision monitoring when the target position is approached.

20.8 Manual Face Milling

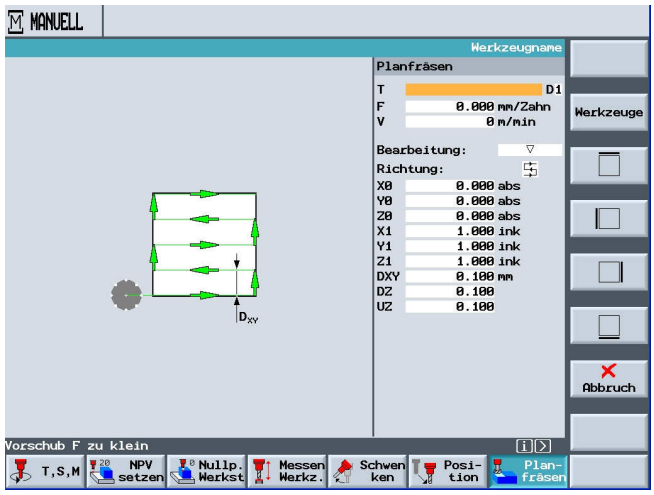
Pressing the softkey

in the operating area

opens the input field for the face milling cycle.



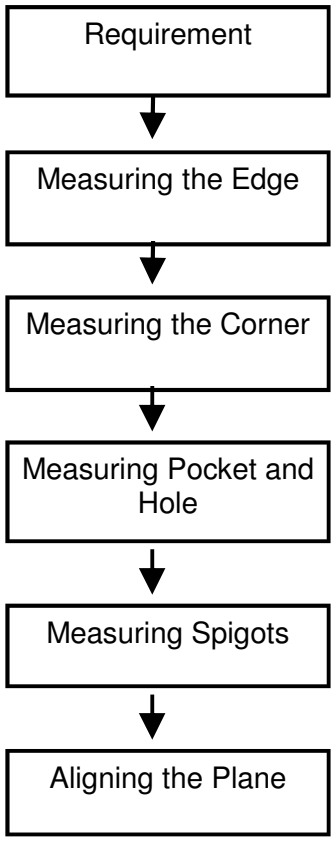
This face milling cycle is described in greater detail in the chapter “Programming Example Standard Milling Cycle“.



The face milling in the operating area Manual described above offers the only possibility to utilize a cycle under ShopMill 6.4 without having to create a program.

21 Measuring the Workpieces in the Setup Mode JOG - Milling

This module describes how to work in the setup mode with measuring cycles under ShopMill, using an example.



21.1 Measuring Manually – Measuring Automatically

Measuring Manually
When measuring the zero point manually, you have to move your tool manually to the workpiece. You can use edge probes, touch probes, or clock gauges whose radius and length is known. As an alternative, you can also use any tool with a known radius and a known length. However, the tools used for measuring must not be of the type 3DProbe.

Measuring Automatically
For automatic measuring, use exclusively electronic touch probes of the type 3D probes, or mono probes. The electronic touch probes have to be calibrated beforehand. For automatic measuring, preposition the probe first manually. After starting with the key “Cycle-Start“, the probe is moved automatically with measuring feed to the workpiece and back again to the start position with rapid feed.
To be able to measure the workpiece zero point automatically, the machine manufacturer has to set up the measuring cycles beforehand.

Please take note of the machine manufacturer’s data.

To obtain the desired measuring results, as a rule the sequence of the measuring points has to be noted that is shown in the help displays. Measuring points can be undone and then measured repeatedly. This is done by operating the softkey that is shown as being active (measuring value). For manual measuring, the reset can be made in any sequence; however, for automatic measuring only in the reverse measuring sequence.

Measuring Only
If you want to "only measure" the workpiece zero point, the measured values are displayed without the coordinate system being changed.

21.2 Measuring the Edge

When measuring an edge, you have the following options:

• Setting the edge
The workpiece is located on the work table parallel to the coordinate system. Measure one reference point in one of the axes (X, Y, Z).

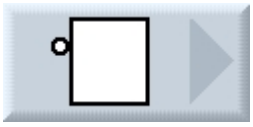
• Setting up the edge
The workpiece is located randomly on the work table; that is, not parallel to the coordinate system. By measuring two points at the edge of the workpiece, you determine the angle to the coordinate system.
.

Distance 2 edges
The workpiece is loacted on the work table parallel to the coordinate system. Measure the distance L of two parallel workpiece edges in one of the axis (X, Y or Z) and determine its center.
Below, an example is used to describe how to measure the edge of a workpiece.

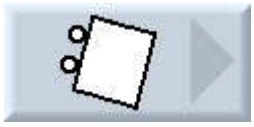
For “Measure Edge,” press the softkey below

and

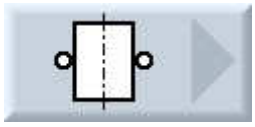
and



or



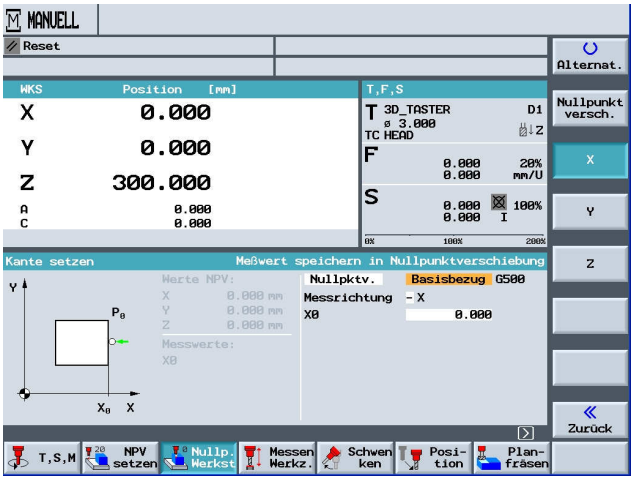
or

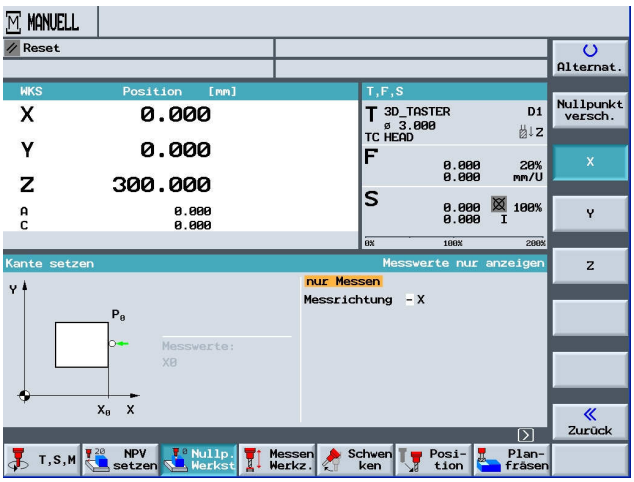


With the

selection of

By pressing the softkey





Pressing the arrow ey

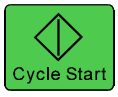
and

To change the axes, press

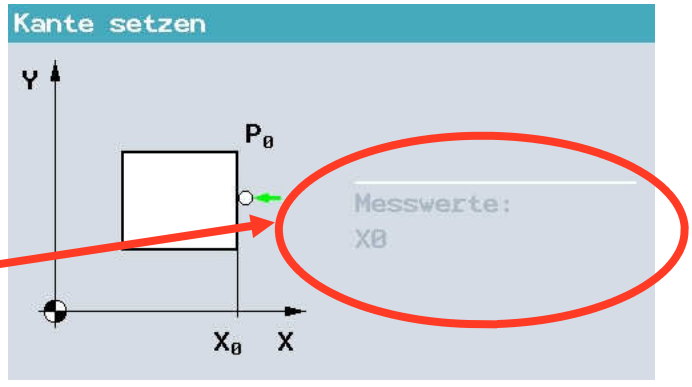
or

For manual measuring, approach the workpiece with the probe.

For automatic measuring, approach the workpiece closely and press ”CYCLE START”.



The position of the workpiece edge is calculated and displayed.



21.3 Measuring the Corner

You have the option of measuring workpieces with an angle of 90° as well as with any angles.

• Measuring a right-angled corner
The workpiece has a 90° corner and is located randomly on the work table. By measuring 3 points, the corner point in the working plane (X/Y plane) is ascertained and the angle α between the reference edge at the workpiece (line through P1 and P2) and the reference axis (always the 1st axis of the work plane).

• Measuring any corner
The workpiece has any type of corner (not a rectangular one) and is located randomly on the work table. By measuring 4 points, we determine the following: the corner point in the work plane (X/Y plane), angle α between the reference edge at the workpiece (line through P1 and P2) and the reference axis (always the 1st axis of the work plane), and angle β of the corner.

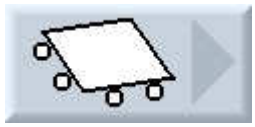
Below, measuring a workpiece under one angle is described.

To measure the edge,
press

and

and

or



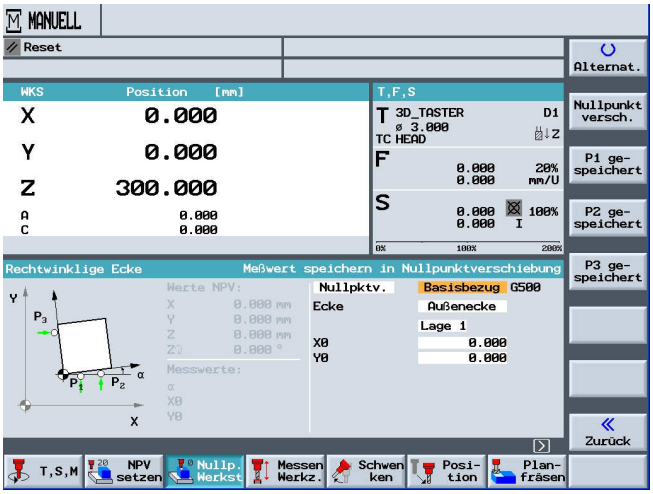
Press

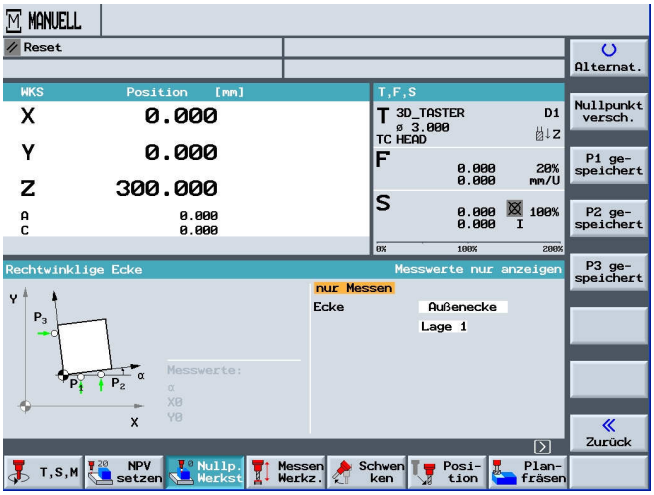
and go to

Nullpktv.

Press

nur Messen



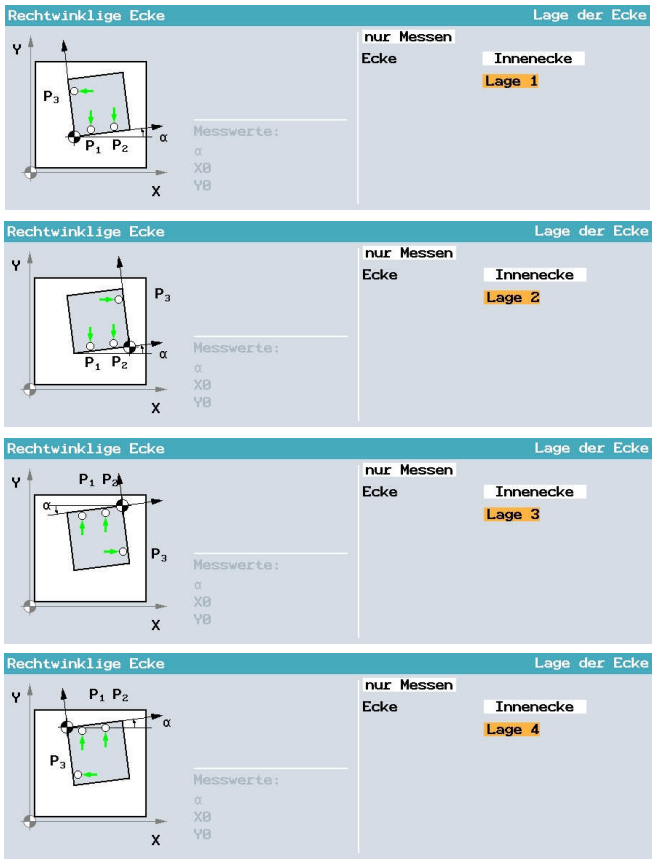


Measuring a Corner

To select “outside corner” “inside corner” press

To select the corner, press

and

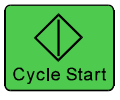


To measure “P1” approach it and press

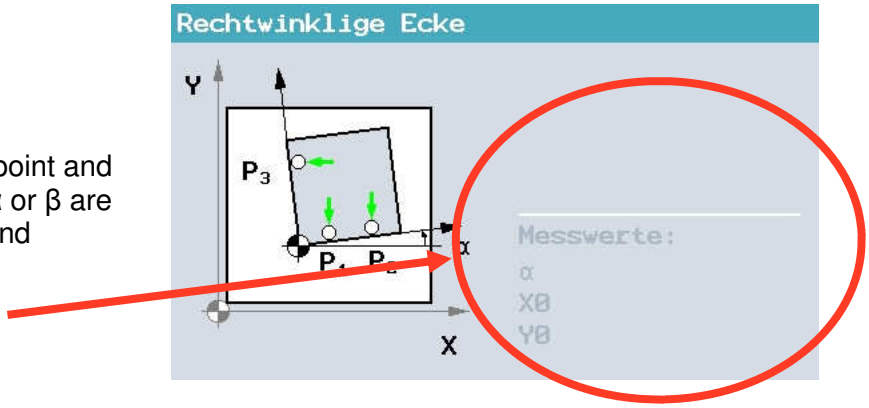
P1 stored

Repeat this for “P2” and “P3”

For automatic measurements, move in front of the measuring point and press “CYCLE START”



The corner point and the angles α or β are calculated and displayed.



21.4 Measuring the Pocket and Hole

You have the option to measure rectangular pockets as well as several holes, and then align the workpiece.

• Measuring the reactangular pocket
The rectangular pocket has to be set up at a right angle to the coordinate system. By measuring 4 points within the pocket, the length, width and center of the pocket is ascertained.

• Measuring 1 hole
The workpiece is located randomly on the work table and has 1 hole. By means of 4 measuring points, we determine the diameter and the center of the hole.

• Measuring 2 holes
The workpiece is located randomly on the work table and has 2 holes. In both holes, 4 points respectively are measured automatically and from that the center of the holes is calculated. From the connection line between the two center points and the reference axis, angle α is calculated as well as the new zero point is determined that corresponds to the center of the 1st hole.

• Measuring 3 holes
The workpiece is located randomly on the work table and has 3 holes. In the 3 holes, 4 points respectively are measured automatically and from that, the center of the holes is calculated. A circle is placed through the three center points. From that, the center of the circle and the diameter of the circle are ascertained. When selecting a phase angle correction, the basic rotation α can be determined in addition.

• Measuring 4 holes
The workpiece is located randomly on the work table and has 4 holes. In the 4 holes, 4 points respectively are measured automatically and from that the center of the holes is calculated. Two centers respectively of a hole are connected diagonally. From that, the intersection of the two lines is determined. When selecting a phase angle correction, the basic rotation α can be determined in addition.

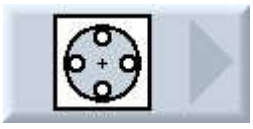
2, 3 and 4 holes can only be measured automatically.

Below, “Measuring a hole“ s described.

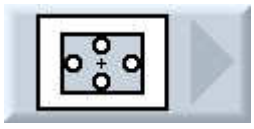
To measure the diameter and the center point of a hole, press

and

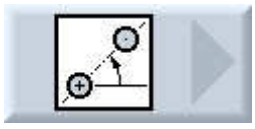
and



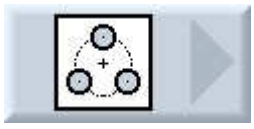
or



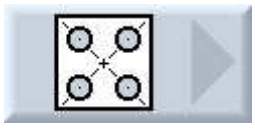
or



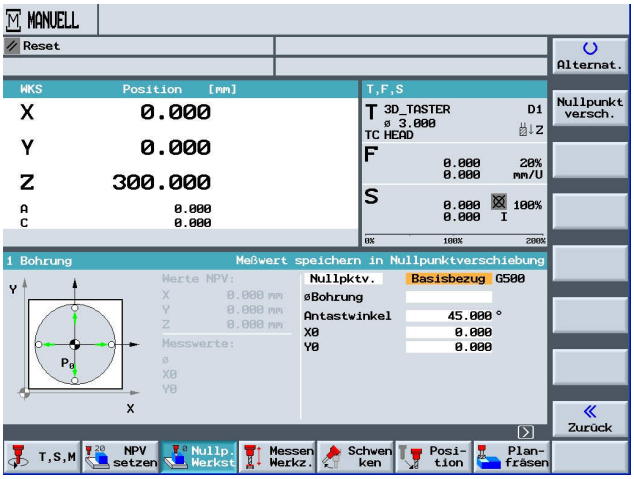
or



or

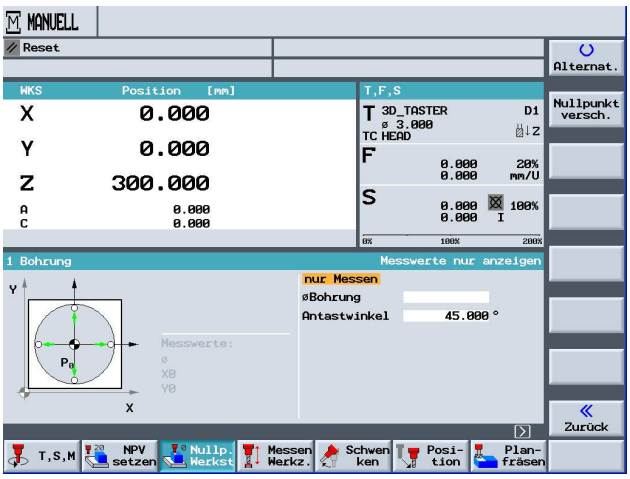


Press



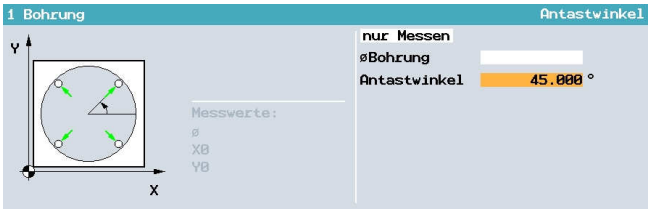
and select

Press



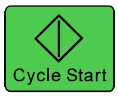
Go to “Hole diameter”

Select the “contact angle”.

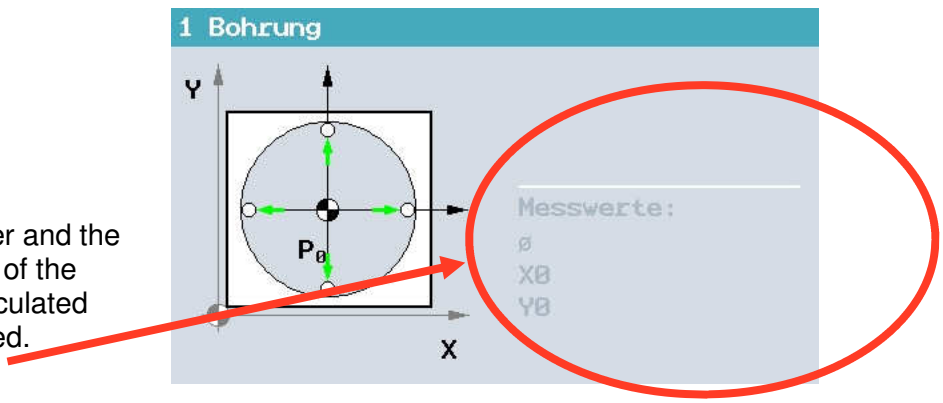


Select the first point that is described in the menu as the first, and approach it.

For automatic measurement, move into the hole and press “CYCLE START”.



The diameter and the center point of the hole are calculated and displayed.



21.5 Measuring Spigots

You have the option to measure and align rectangular spigots, as well as one and several circular spigots:

• Measuring the rectangular spigot
The rectangular spigot is to be aligned right-angled to the coordinate system. By measuring 4 points at the spigot, we ascertain the length, width and the center point of the spigot.

• Measuring 1 circular spigot
The workpiece is located randomly on the work table and has 1 spigot. With 4 measuring points, we determine the diameter and the center point of the spigot.

• Measuring 2 circular spigots
The workpiece is located randomly on the work table and has 2 spigots. At the two spigots, 4 points respectively are measured automatically, and from that the center points of the spigots are calculated. From the connection line between the two center points and the reference axis, angle α is calculated and the new zero point is determined that corresponds to the center point of the first spigot.

• Measuring 3 circular spigots
The workpiece is located randomly on the work table and has 3 spigots. At the three spigots, 4 points respectively are measured automatically and from that, the center points of the spigots are calculated. A circle is placed through the three center points, and the center point of the circle and the diameter of the circle are ascertained. When selecting a phase angle correction, the basic rotation α can be determined in addition.

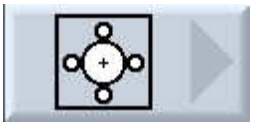
• Measuring 4 circular spigots
The workpiece is located randomly on the work table and has 4 spigots. At the four spigots, 4 points respectively are measured automatically and from that, the center points of the spigots are calculated. Two center points of the spigots respectively are connected diagonally, and then the intersection of both lines is determined. When selecting a phase angle correction, the basic rotaton α can be determined in addition.

2, 3 and 4 circular spigots can only be measured automatically.

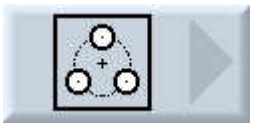
Below, measuring a spigot is described.

To measure the diameter or the center of a spigot, press

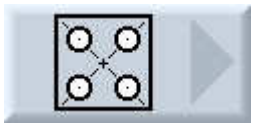
and



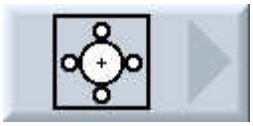
or



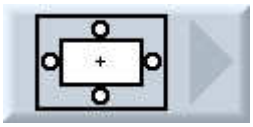
or



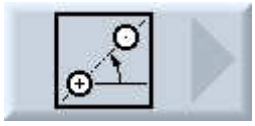
and



or



or



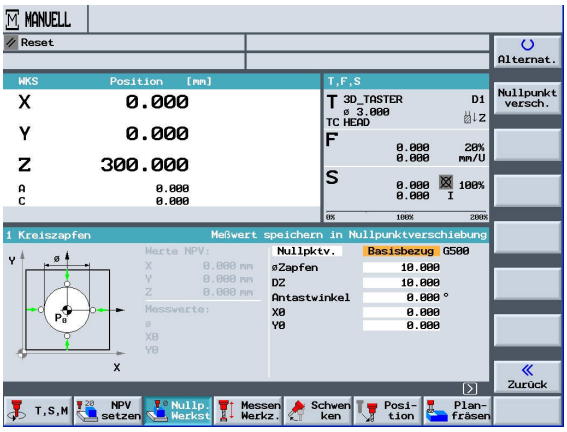
With the

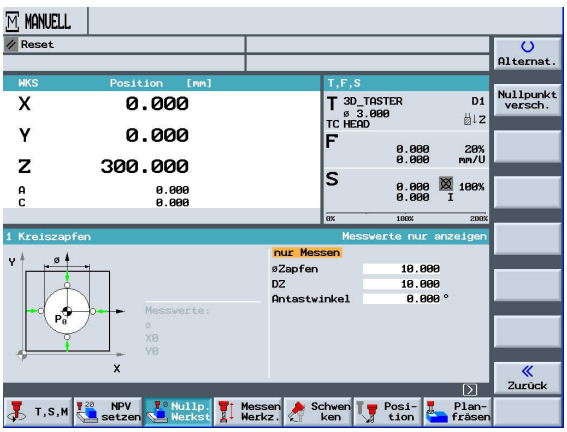
go to the

Nullpktv.

and press

nur Messen





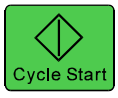
With the cursor, go to Diameter Spigot and press

Go to the value “DZ” and press

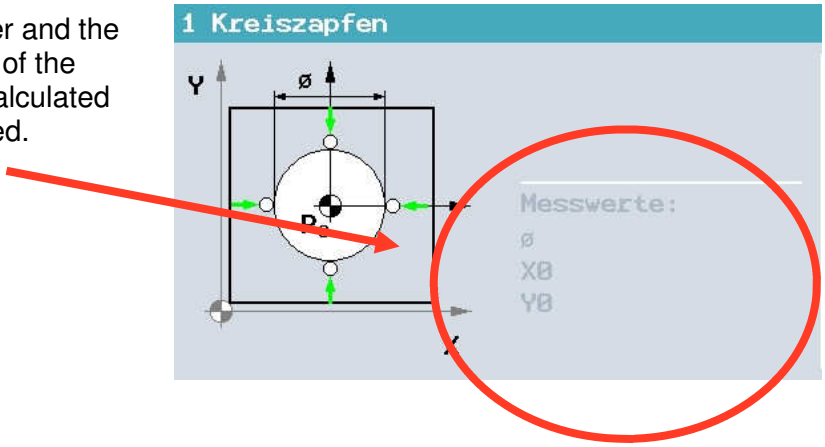
Go the the contact angle and press

Select the first point that is described first in the menu, and approach it.

When measuring automatically, approach the workpiece and press “CYCLE START”.



The diameter and the center point of the spigot are calculated and displayed.



21.6 Aligning the Plane

You can measure a spacially oblique plane of a workpiece, and ascertain the rotations α and β. With a subsequent coordinate rotation, the vertical alignment of the tool axis to the workpiece plane is possible. To determine the position of the plane in space, we measure in the tool axis at three different points. To align the tool axis vertically requires a tilting table or an inclinable head.

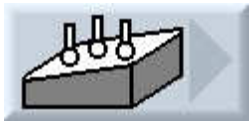
For the plane to be measured, the surface has to be even.

Below, measuring the oblique plane is described.

Aligning the Plane

To align and measure an oblique plane, press

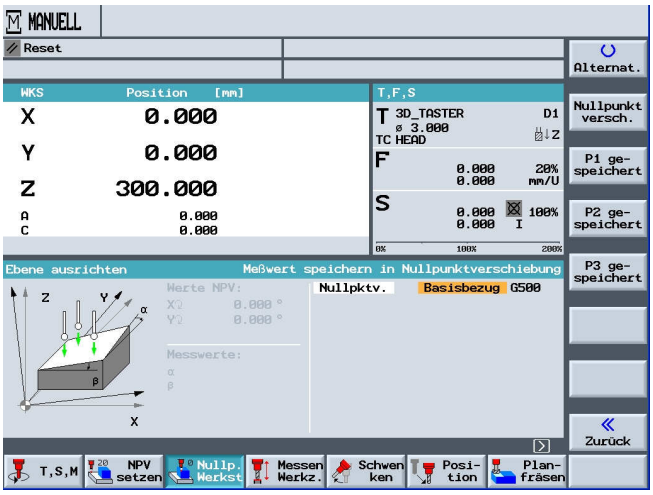
and

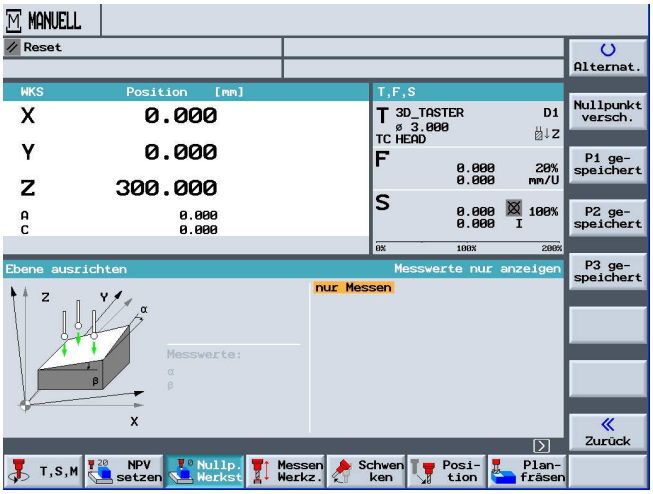


. Press

. Go to

and press

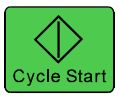




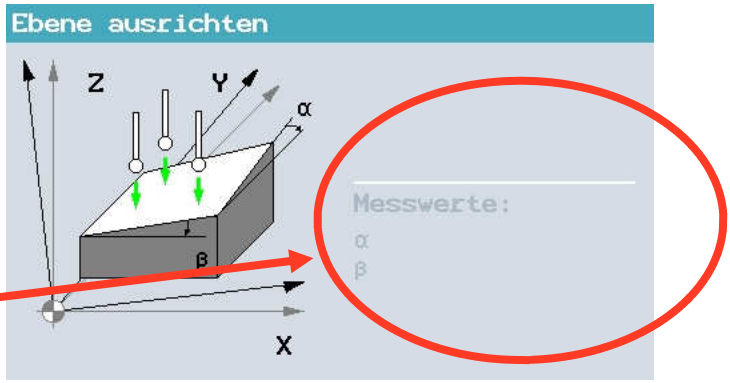
Approach the first measuring point that is shown in the cycle, and save it.

Then, approach the other points.

For automatic measuring, approach the tool and press



Angles α and β are calculated and displayed.



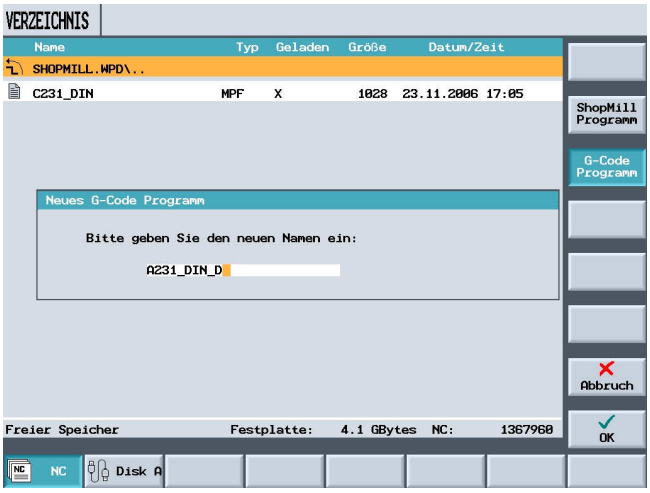
22 DIN/G-Code – Programming under ShopMill

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

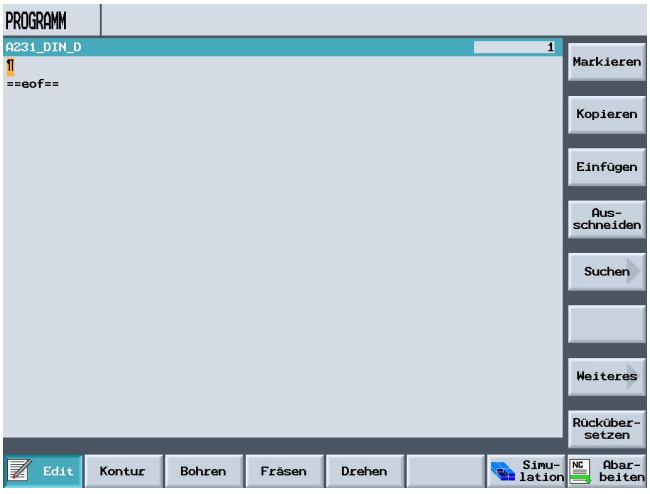
Under ShopMill, we can program in the socalled ISO G-code according to DIN 66025, and also with the cycles of the 840D.

When setting up a new program, the DIN editor under ShopMill is opened

by pressing the softkey



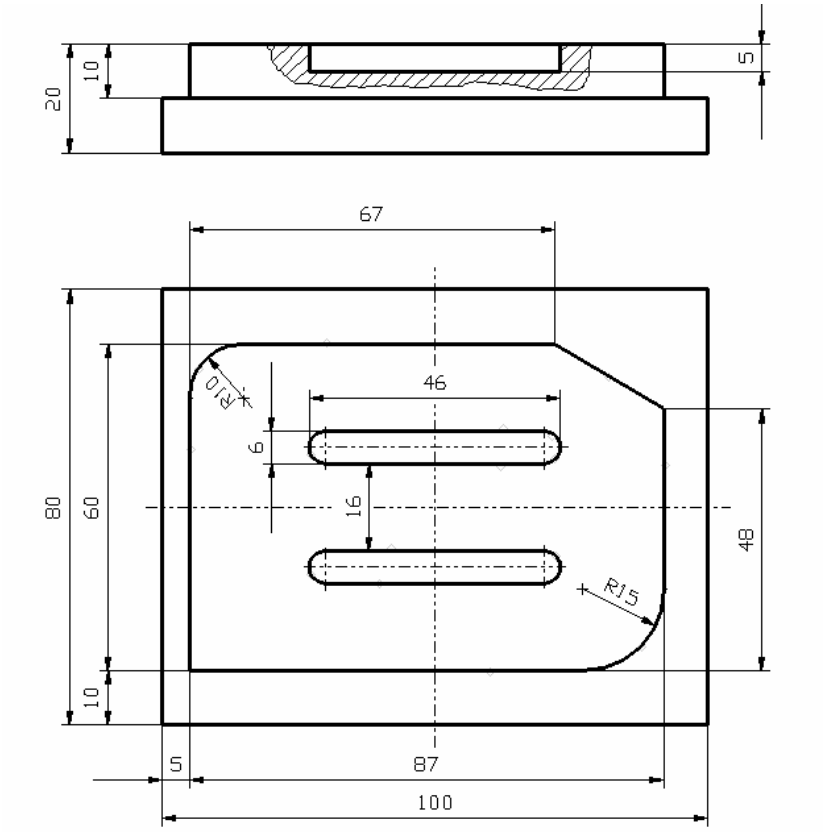
The proram example is generated by using the standard cycles.



G17– Machining plane XY plane
G41– Tool radius compensation (left)
G54– Active zero point shift
G90– Absolute measurement programming
G94– Feed in mm/min

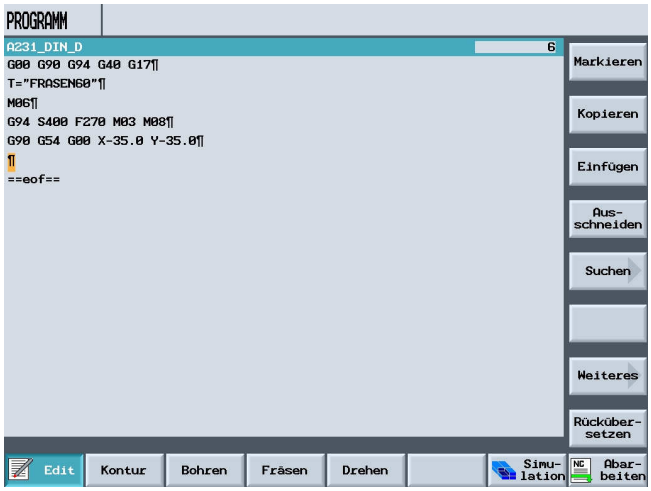
The follwing G-functions are used for generating the program.

The workpiece below is to be programmed in G-code with cycle support.

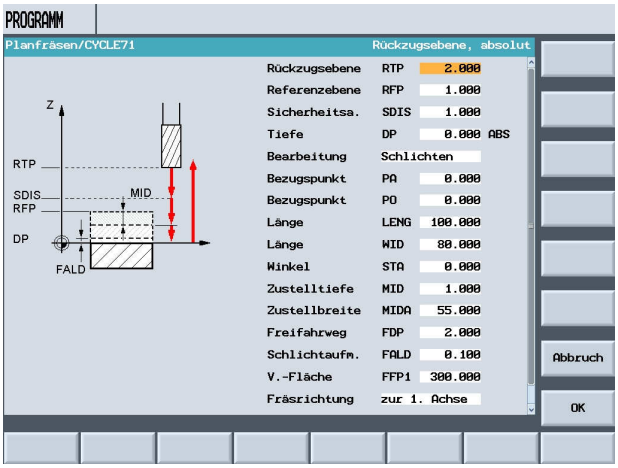


In contrast to the standard cycles, the ShopMill cycles include the approach and retract strategy to the contour.

After entering the basic commands, calling the tool and after the approach position prior to processing, the corresponding cycle is opened by pressing the softkey



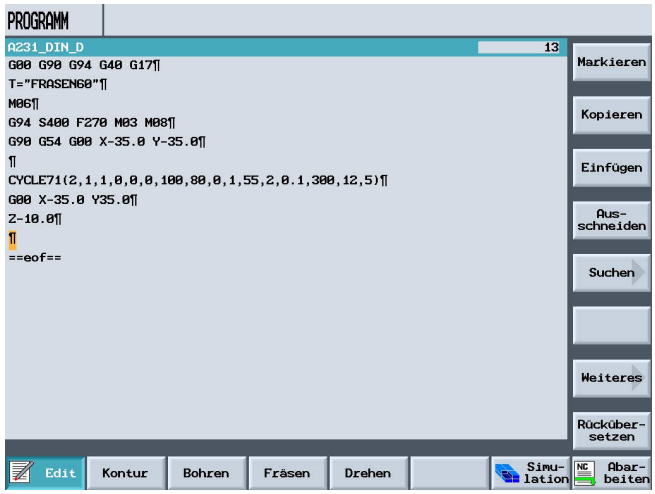
and



After entering the values and pressing the softkey

the cycle is closed and accepted.

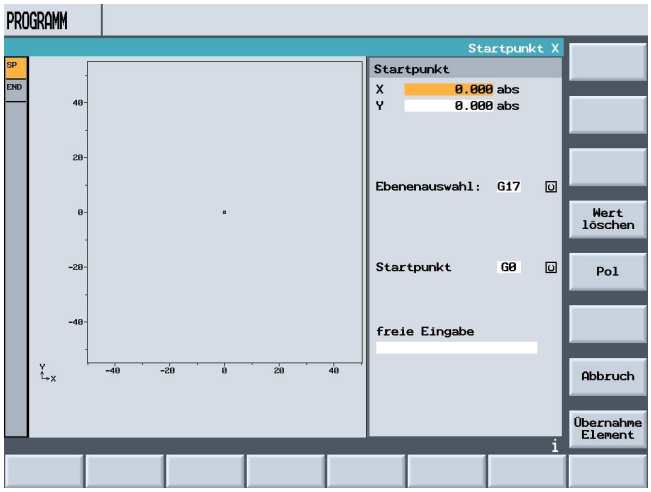
Next, the contour is described.



22.1 Generating the Contour with the Contour Calculator

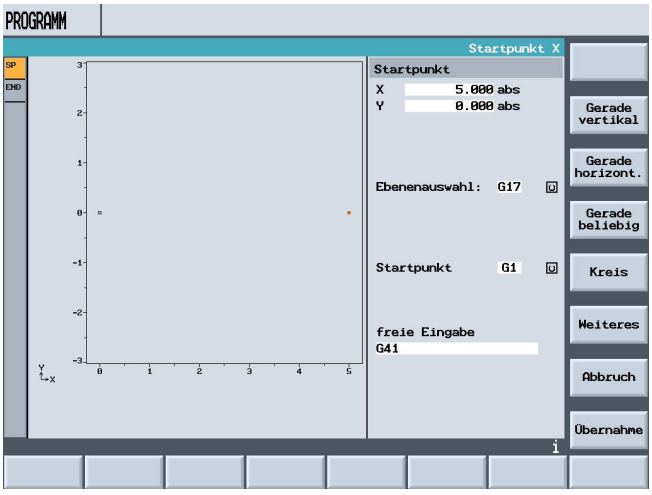
Pressing

opens the standard contour calculator.



After entering the values

and pressing the



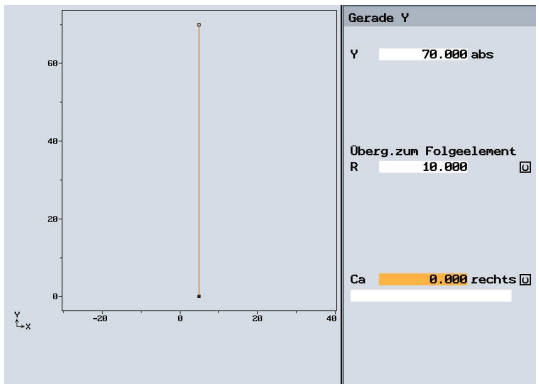
the contour element is accepted as for ShopMill.

Element 1

Gerade vertikal

To accept, press

Übernahme Element

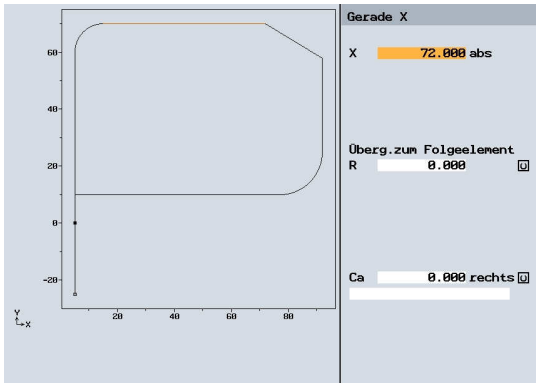


Element2

Gerade horizont.

To accept, press

Übernahme Element

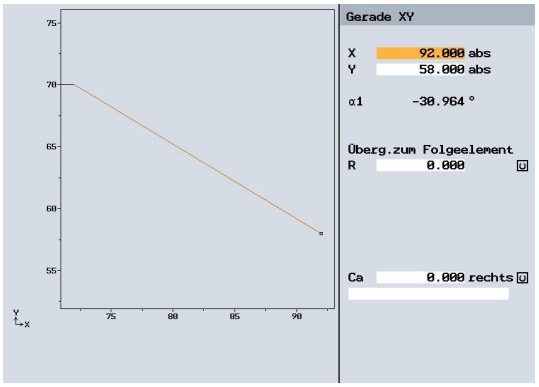


Element 3

Gerade beliebig

To accept, press

Übernahme Element

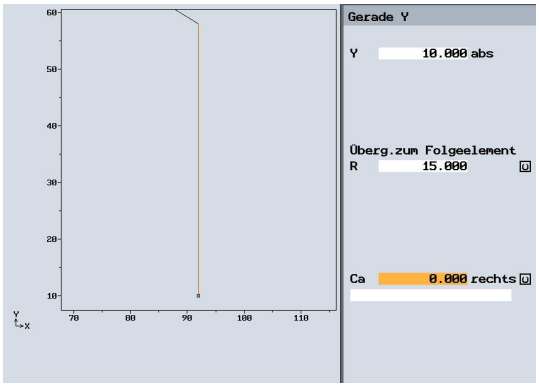


Element 4

Gerade vertikal

To accept, press

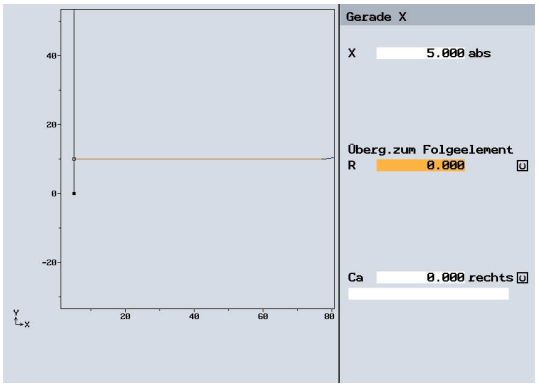
Übernahme Element



Element 5

Gerade horizont.

To accept, press

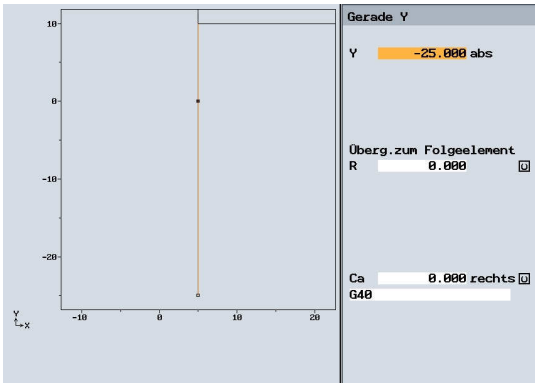


Übernahme Element

Element 6

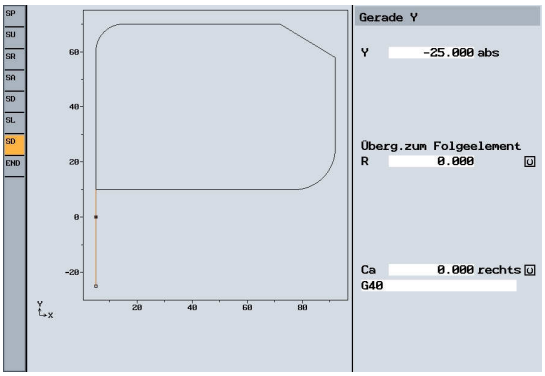
Gerade vertikal

To accept, press



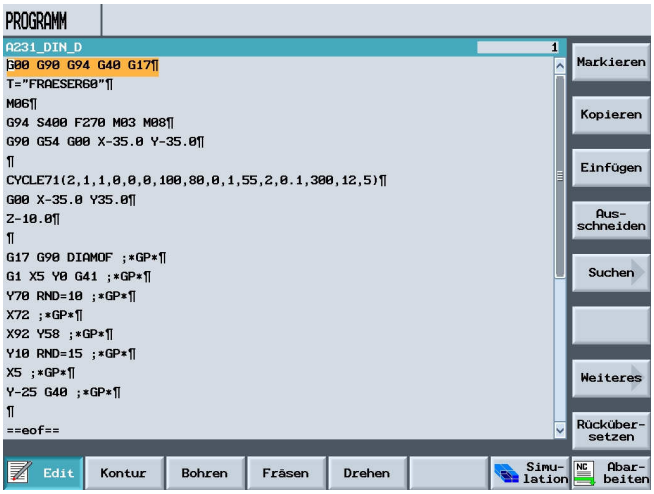
Übernahme Element

To accept the contour, press



Übernahme

The contour is shown as G-code.



With the second tool, the grooves are made.

Go to the end of the program

```
A231_DIN_D                                    37
M06¶
G94 S400 F270 M03 M08¶
G90 G54 G00 X-35.0 Y-35.0¶
¶
CYCLE71(2,1,1,0,0,0,100,80,0,1,55,2,0.1,300,12,5)¶
G00 X-35.0 Y35.0¶
Z-10.0¶
¶
G17 G90 DIAMOF ;*GP*¶
G1 X5 Y0 G41 ;*GP*¶
Y70 RND=10 ;*GP*¶
X72 ;*GP*¶
X92 Y58 ;*GP*¶
Y10 RND=15 ;*GP*¶
X5 ;*GP*¶
Y-25 G40 ;*GP*¶
¶
G00 Z2.0¶
Z100.0¶
¶
```

and set up a new tool with the technology data.

```
A231_DIN_D                                    41
G00 X-35.0 Y35.0¶
Z-10.0¶
¶
G17 G90 DIAMOF ;*GP*¶
G1 X5 Y0 G41 ;*GP*¶
Y70 RND=10 ;*GP*¶
X72 ;*GP*¶
X92 Y58 ;*GP*¶
Y10 RND=15 ;*GP*¶
X5 ;*GP*¶
Y-25 G40 ;*GP*¶
¶
G00 Z2.0¶
Z100.0¶
T="FRAESER6"¶
M06¶
G94 S4200 F150 M03 M08¶
G90 G54 G17 G00 X30.0 Y29.0¶
G00 Z2.0¶
==eof==
```

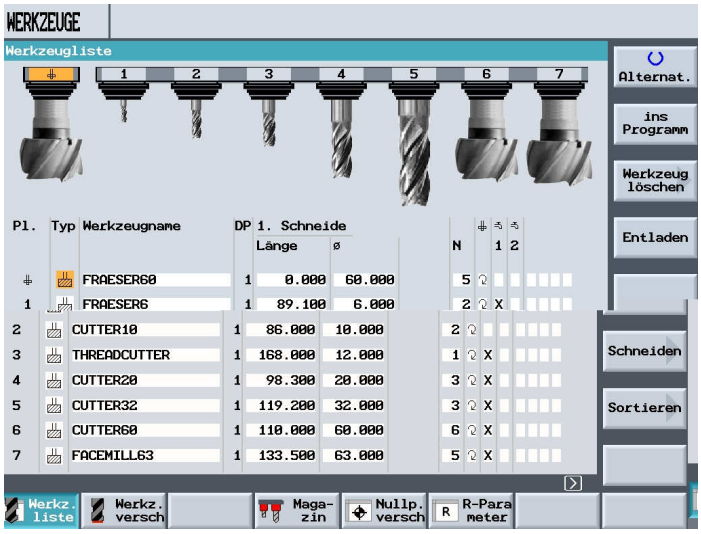
Then, the start position is defined.

```
A231_DIN_D                                    50
X5 ;*GP*¶
Y-25 G40 ;*GP*¶
¶
G00 Z2.0¶
Z100.0¶
T="FRAESER6"¶
G94 S4200 F150 M03 M08¶
G90 G54 G17 G00 X30.0 Y29.0¶
G00 Z2.0¶
G01 Z-5.0¶
X70.0 F560¶
G00 Z2.0¶
X70.0 Y51.0¶
G01 Z-5.0 F150¶
X30.0 F560¶
G00 Z2.0¶
Z100.0 M05 M09¶
X-200.0 Y150.0¶
M30¶
==eof==
```

Pressing

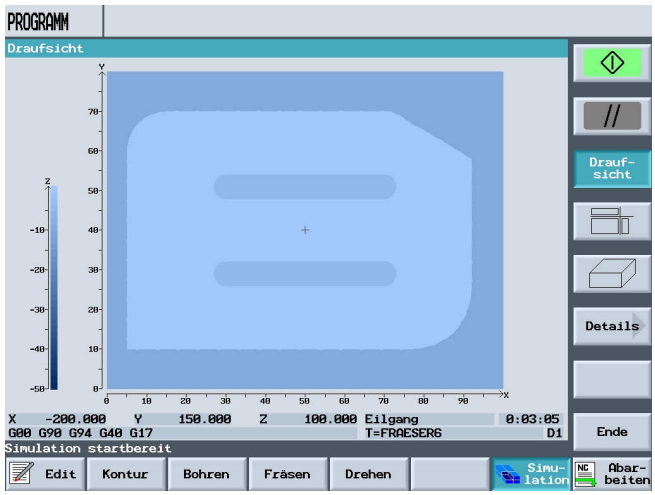
opens the tool list of ShopMill.

After selecting the tool and pressing the softkey “into the program”, the tool is included in the program.

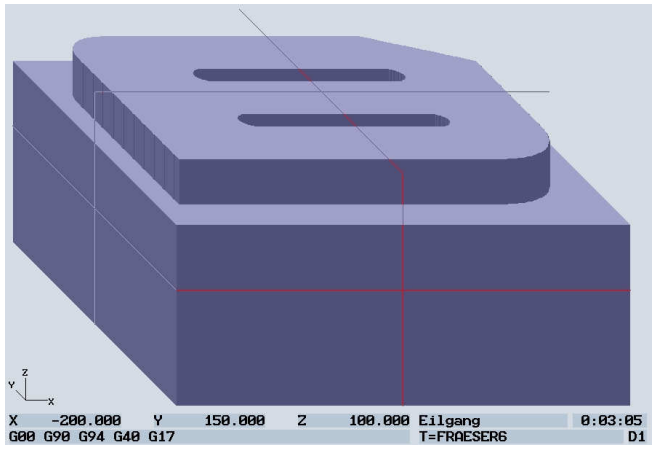


By pressing the softkey

the familiar simulation can also be started for a DIN program.



You have the option to change the view using different softkeys.



23 Multiple Clamping

The function "Multiple Clamping" optimizes the tool changes by means of several workpiece setups. This shortens the idle time since first, all machining of a tool is executed on all clampings prior to the next tool change being activated.

In addition to two-dimensional clamping, you can use the function "Multiple clamping" also for rotating clamping bridges. To this end, the machine has to have an additional rotary axis (for example, an A axis) or a dividing unit.

Please note the manufacturer's data regarding this.

Not only identical, but also differing workpieces can be processed with this function.

The function "*Multiple Clamping for Different Programs"* is a software option.

ShopMill generates automatically a single program from several programs.

The tool sequence within a program is retained.
Cycles and subprograms are not separated <<?>>
Position patterns are processed as a unit <<?>>.
The individual programs have to satisfy the following requirements:

- Only step sequence programs (no G-code programs)
- Programs have to be runnable
- Program of the 1st clamping has to be positioned
- No tags/repetitions; that is, no jumps in the program
- No inch/metric switching
- No zero point shifts
- No coordinate transformations (shifting, scaling, etc.)
- Contours have to have unique names; that means, the same contour name must not be called in several programs
- In the removal cycle (contour milling) the parameter "Starting point" must not be set to "manual".
- No modal settings; that means, settings that have an effect on all subsequent data blocks (only for multiple clamping for different programs)
- Max. 50 contours for each clamping
- Max. 99 clampings

Tags or repetitions that are not to be used in programs for multiple clamping can be circumvented by using subprograms.
Open the Program Manager.
Weiteres Mehrfachaufspg. <<additional, multiple clamping>>
-Press the softkey "Weiteres" and "Mehrfachaufspg.".
-Enter the number of clampings and the number of the first zero point shift that is to be used.
The clampings are processed in an ascending sequence starting with the **Start-Zero point shift**. The zero point shifts are defined in the menu "Werkzeuge/Nullpunktverschiebungen" <<tools/zero point shifts>>
(refer to chapter "zero point shifts").
-Enter a **name for the new overall program**
(XYZ.MPF).
-Press the softkey **"OK".**
A list is displayed in which the different programs have to be assigned to the zero point shifts. Programs don't have to be assigned to all zero point shifts -that is, clampings- but at least to two.
Press the softkey **"Program selection"**.
The program overview is displayed.
-Place the cursor on the desired program.
-Press the softkey **"OK".**
The program is accepted into the assignment list.
-Repeat this process, until a program is assigned to each desired zero point shift.
-Press the softkey **"Auf alle Aufspg.",** if you want to process the same program on all clampings.
You also can assign different programs to individual zero point shifts first, and then assign the same program to the remaining zero point shifts by using the softkey
"Auf alle Aufspg." <<to all clampings>>
.-Press the softkey **"Auswahl löschen"** <<clear selection>> or **"Alles löschen"**,
if you want to remove individual or all programs from the assignment list.
-Press the softkey **"Programm berechnen"**, <<calculate program>> when the assignment list is complete.
The tool changes are optimized.
Then, the overall program is renumbered consecutively, and when changing between different clampings, the number of the current clamping is specified.
In addition to the overall program (XYZ.MPF), the file XYZ_MCD.INI is set up where the assignment between zero point shifts and programs is stored. The two programs are stored in the directory that was previously selected in the program manager.
If you change from the assignment list (without "Cancel" or "Generate program") to another function and later you call the function "Multiple clamping" again, the same assignment list is displayed again.

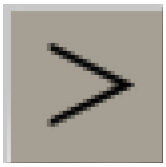
24 CAD Reader

24.1 General Function

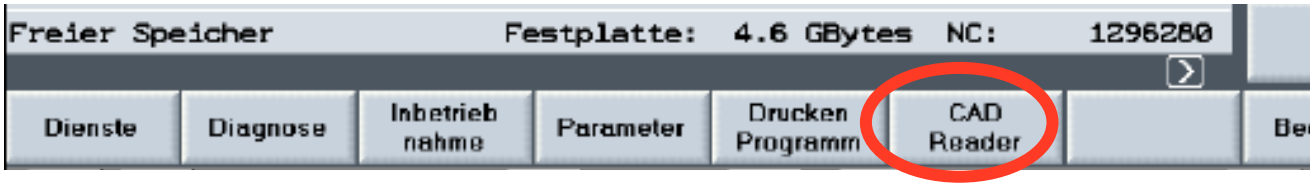
The CAD Reader is used for the following: to further edit drawings with the SINUMERIK controller that were constructed with a CAD system. As the format, a DXF file (**D**rawing e**X**change **F**ormat) is entered, and contours or drilling points are filtered out.
Parts that are not necessary for editing (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.

24.2 Opening the CAD READER

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

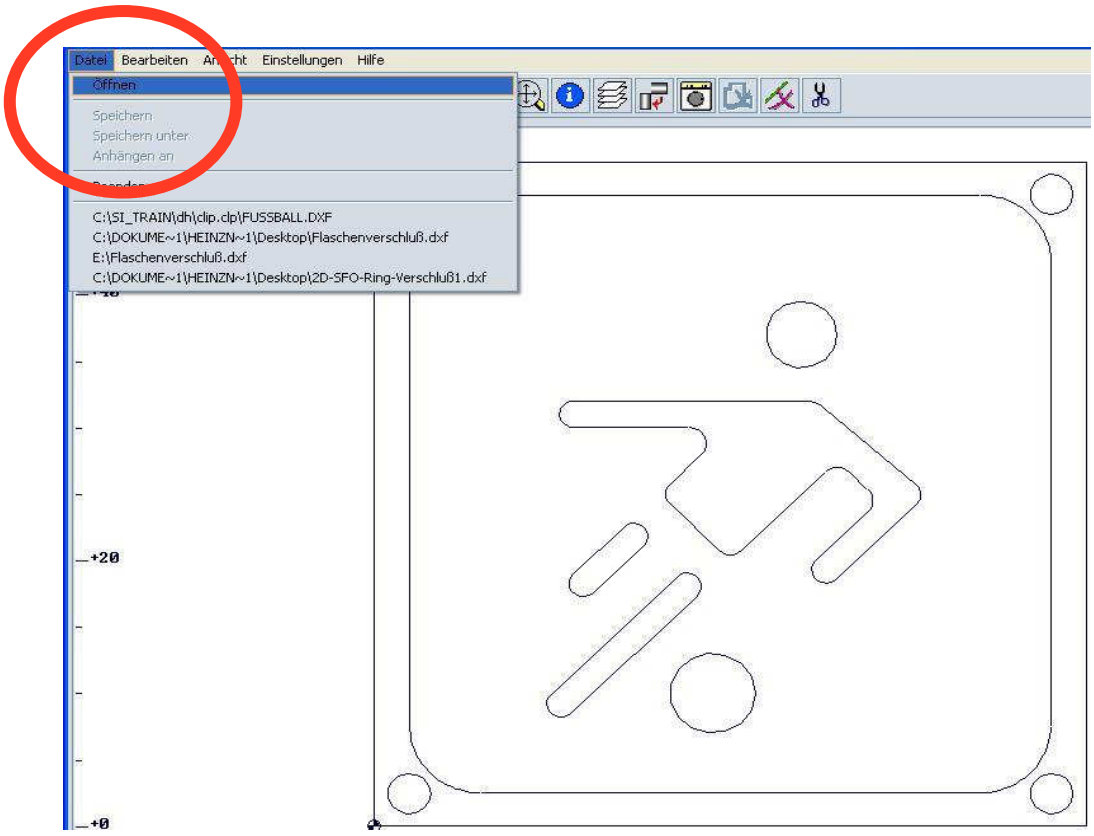


With this arrow key toward the right, the softkey bar opens.



Press the softkey "CAD Reader“

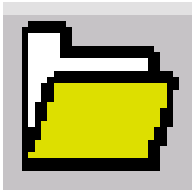
24.3 Opening a DXF Drawing from a File.



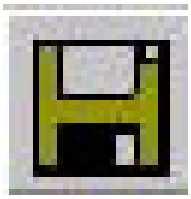
24.4 Tool Bar

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

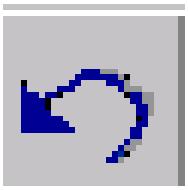
Operation in General
All functions of the CAD Reader can be operated with the keyboard as well as with the mouse. The right mouse key corresponds to the "ESCAPE“ function that can be used for resetting activated menus or functions.



Operating sequence
Open DXFfiles
With **Open**, the selected CAD drawing is selected.

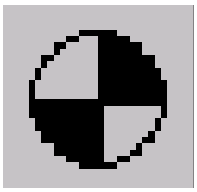


Saving the generated program
Generated contours can be saved as data type in the
• MPF format (□.mpf)
• SPF format (□.spf)
• ARC format (□.arc) (SINUMERIK archive)



Back
For contour tracking, the last action is reset corresponding to the selection element by element, or the last intersection is reset.
.

24.5 Specifying the Zero Point



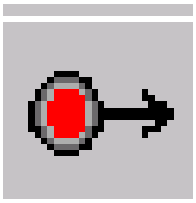
To output the contour as NC program, it is necessary to specify a zero point of the drawing because in most cases, it will deviate from the zero point of the DXF file.

The following options are available to define the zero point:

Element Center	• Automatically to Element Center
Element Start	• Automatically to Element Start
Element End	• Automatically to Element End
Free Input	• Direct input of the coordinates; for example X100, Y100
Mouse Position	• Any position by selecting with the mouse

24.6 Contour Tracking

Setting the Contour Starting Point

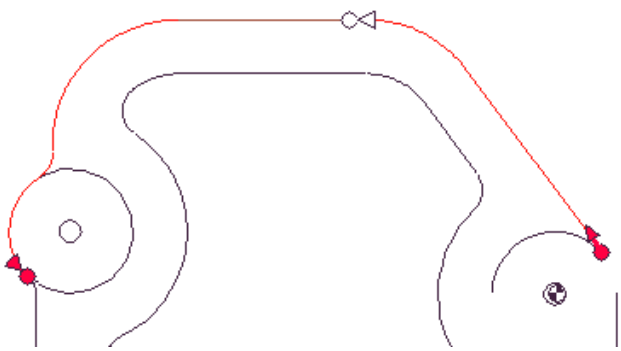


Contour Tracking with starting point and end point

The starting and end point of the contour to be generated is selected in dependence of the initial position of the technology applied:

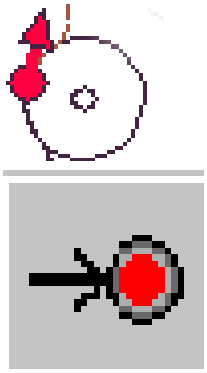
Element center	• Automatically to Element Center
Start/End point	• Automatically to Element Starting/Endpoint
Mouse position	• Directly with mouse selection

Example



1. Contour direction
Results from specifying the starting point •
and additional contour selection. Regarding contour tracking the attempt is made to select the contour automatically to the greatest extent possible.

2. Selection if there is a conflict
If automatic contour tracking can no longer clearly determine a successor element, we switch to the interactive mode.
The user is prompted to specify the next element with which the contour continues.

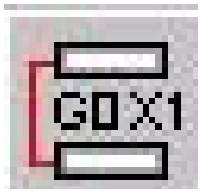


3. Full circle as contour
With contour tracking, a full circle can be accepted in both directions.

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

Setting the contour end point
Element center
Element end point
Mouse position
Current position

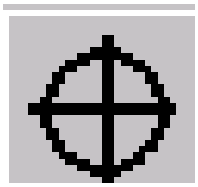
Additional Notes
• Full circles can be accepted as contour or as drilling points.
• Contour tracking is canceled either with the keyboard with the "Esc“ key, or with the right mouse key.



Setting the Contour Label
Prior to contour tracking, labels can be set by entering starting labels and end labels.
If a label is assigned twice, the CAD reader becomes interactive if the label was assigned once before.
• For contour tracking in contours already selected
• For appendixes to files if the label occurs already 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.

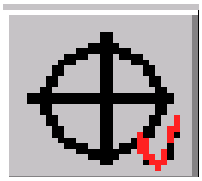


Drilling Points Start

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

Any position
Row of holes
Hole circle
Grid of holes

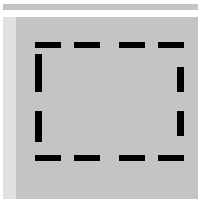
- Ay drilling position
- According to the cycle
- According to the cycle
- According to the cycle



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

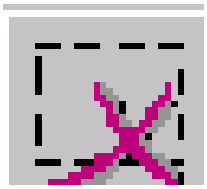
24.7 Influencing the Graphic Representation

Selecting the Processing Area:

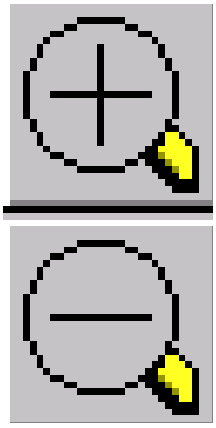


If the file includes many additional drawings -such as profiles, measurements, hatching, labelings, detail displays, frames etc.- the selection of a processing area, the number of elements can be reduced with a "Lasso“.

Deselecting the Processing Area:

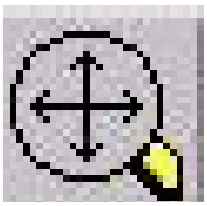


The selections made in this processing area can be deselected.



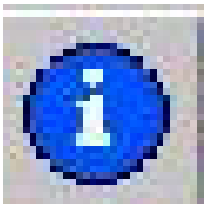
Zoom/Keys “+“ and **“– “**
You have the option to specify within a drawing a zoom area with the mouse key. By clicking on the symbol and using a "Lasso“ or with the keys “+“ and “–“ the display area is enlarged or reduced step by step.
The display area can be moved using the cursor keys.

24.8 Editing Input Data

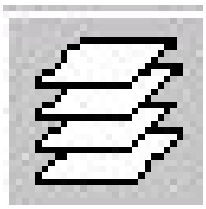


New Drawing/Spacer Bar
Reads out the current drawing anew and optimized corresponding to the layer selection.

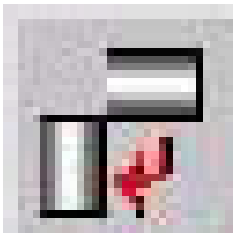
Geometry
With a mouse click, the coordinates for the selected element are read out according to the current zero point. If the button Edit appears in the display screen form, this element can be edited by selecting this button.



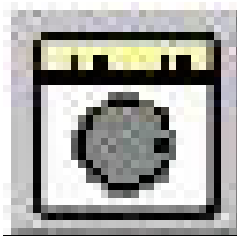
Note
This function is suitable for minor changes in the geometry to remedy inadequacies (particularly missing intersections) in the CAD drawing.
For larger changes, the geometry processor is used. Once a change is made, it **can not** be reset.



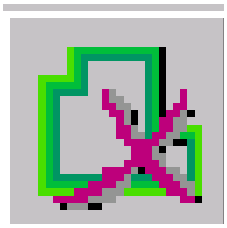
Layer Selection
Initially, the selected DXF file is always shown with all its layers. If the file contains several layers, they are all shown in the basic view.
However, layers that don’t contain data relevant to the contour can be hidden. Likewise, contours that are included in several layers can be selected by means of a selection screen form for contour tracking.
The layer selection can not be undone.



Rotating the Contour
By selecting this icon, the drawing is rotated by 90 degrees around the specified zero point, corresponding to the presettings. Contour characteristics that have already been set are not rotated.

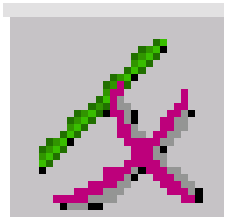


Displaying Hatching and Measurements
Hatching and measurements in the CAD drawings can be removed or displayed. Another click resets this function.



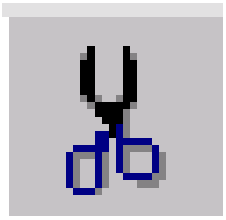
Deleting Contour Tracking
Defined contours can be selected and completely deleted. The function “Delete contour“ is activated by selecting this icon and deactivated by selecting it again. Deleting finished contours:

Select icon: activate Delete contour
Select contour: contour is deleted



Deleting a Geometry Element
With this function, individual geometry elements can be deleted. The function “Delete geometry element“ is activated by selecting this icon and deactivated by selecting it again. Deleting a geometry element:

Select: icon: activate Delete geometry element
Select elements: geometry elements are deleted

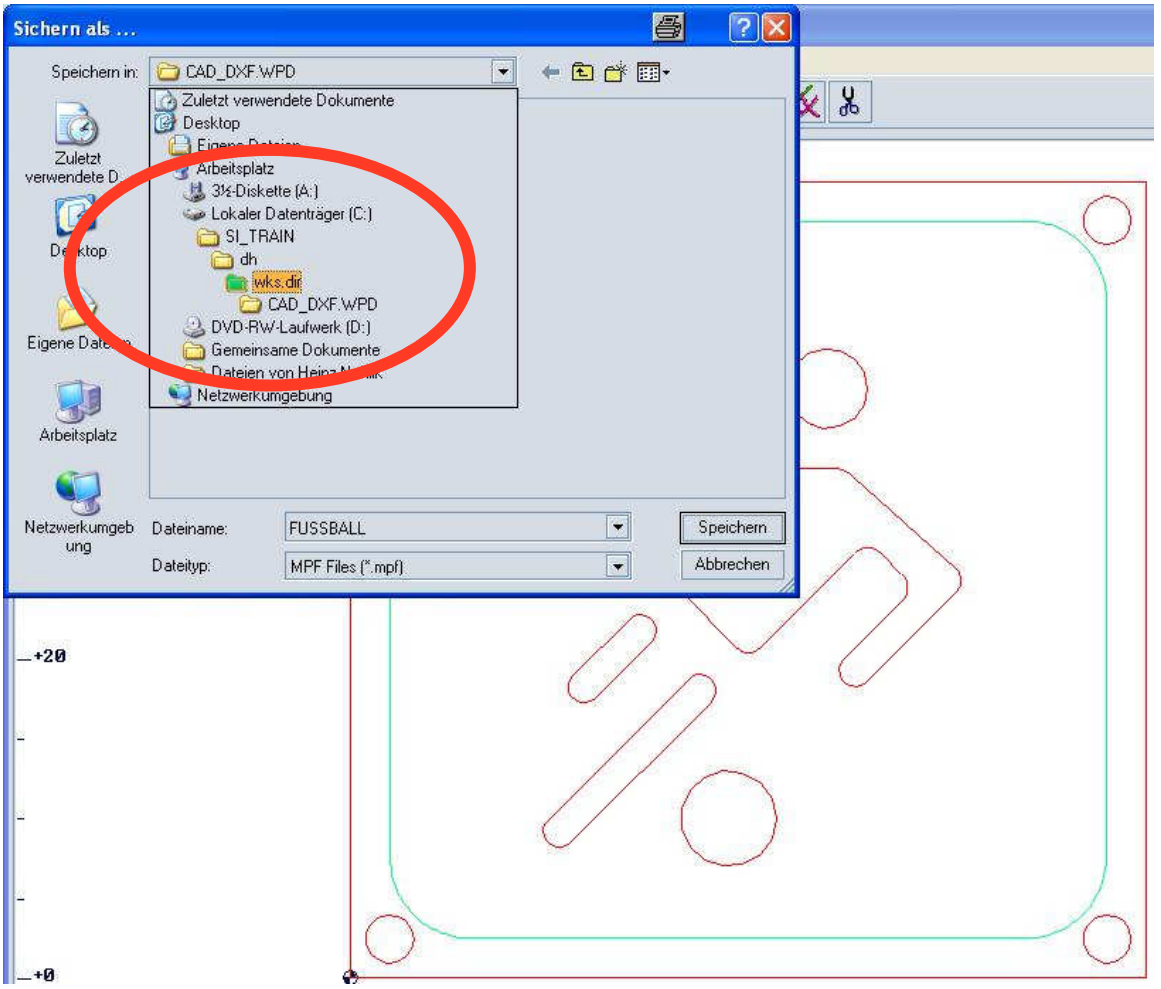


Deleting a Geometry Area
With a rectangle drawn with the mouse (corresponds to the area to be deleted) an entire area can be deleted from the geometry. This function is deactivated autonomously with each deletion, and has to be reactivated each time with this icon.

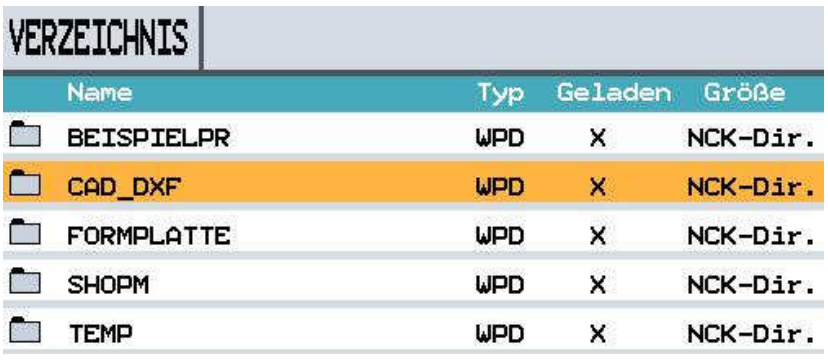
Select icon: Activate Delete geometry area
Select area: geometry area is deleted

24.9 Transferring Contour Elements to the Directory

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

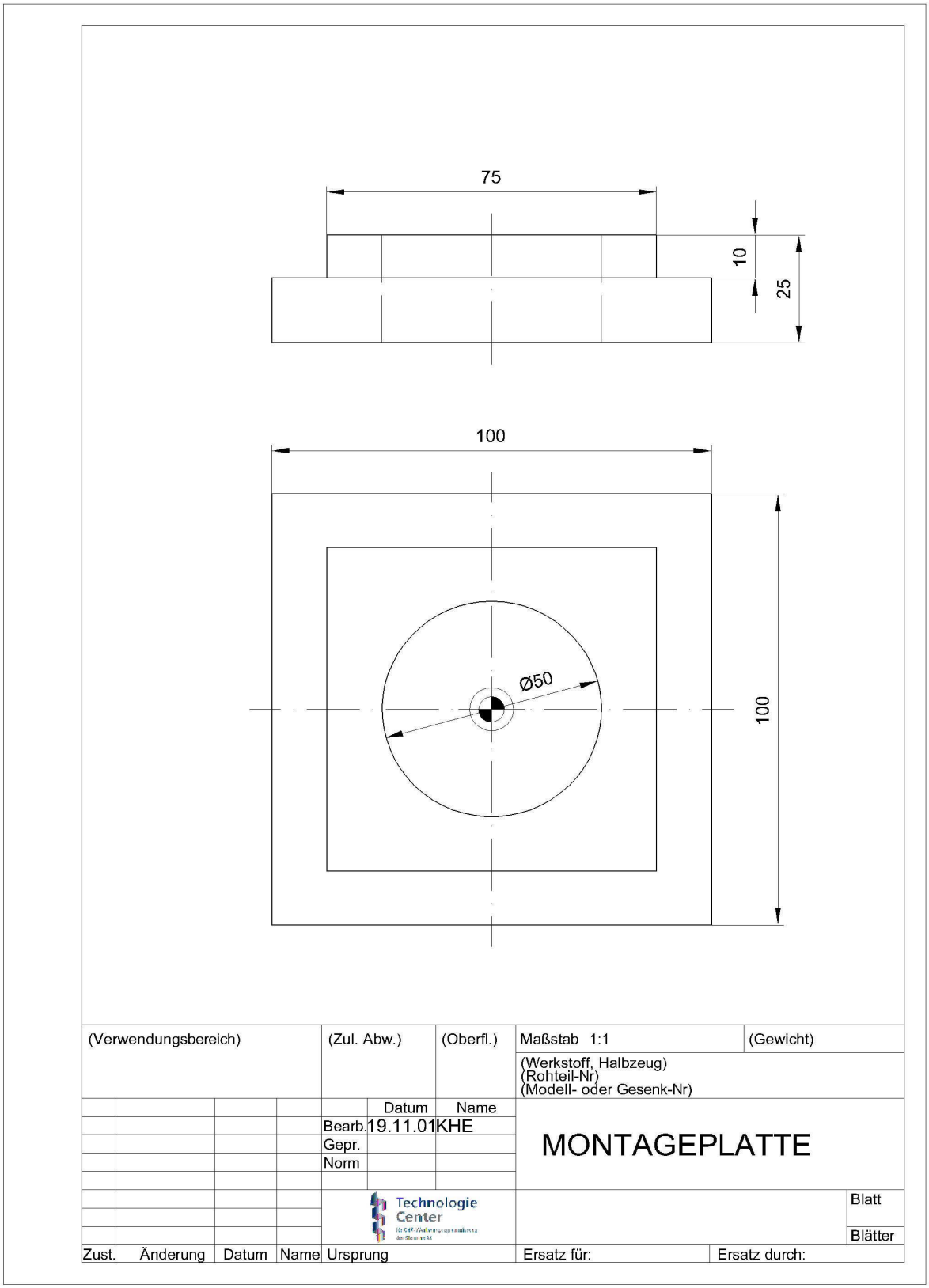


All contour elements are stored in the directory; for example, CAD_DXF.

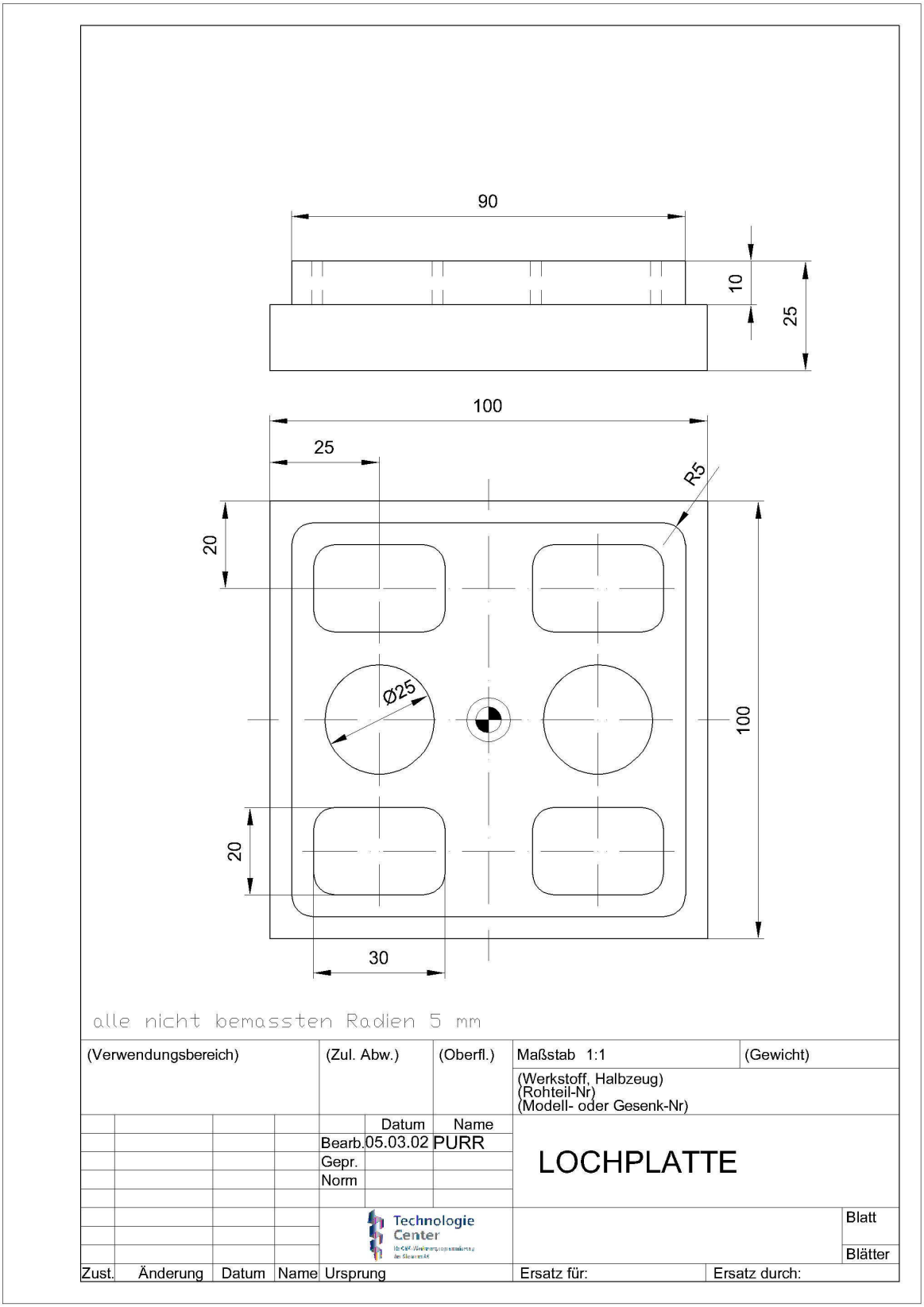


25 Sample Drawings - Milling

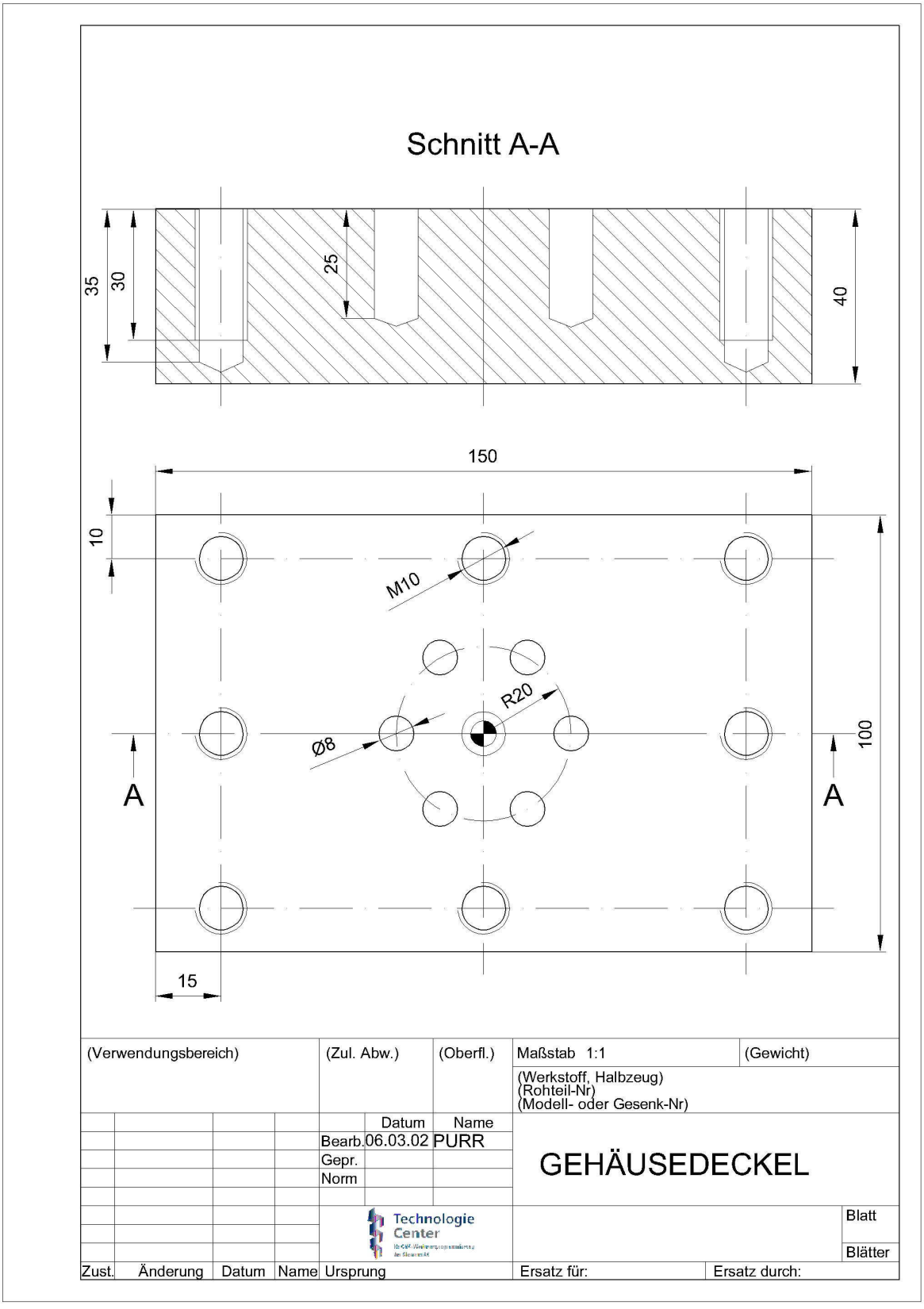
25.1 Mounting Plate



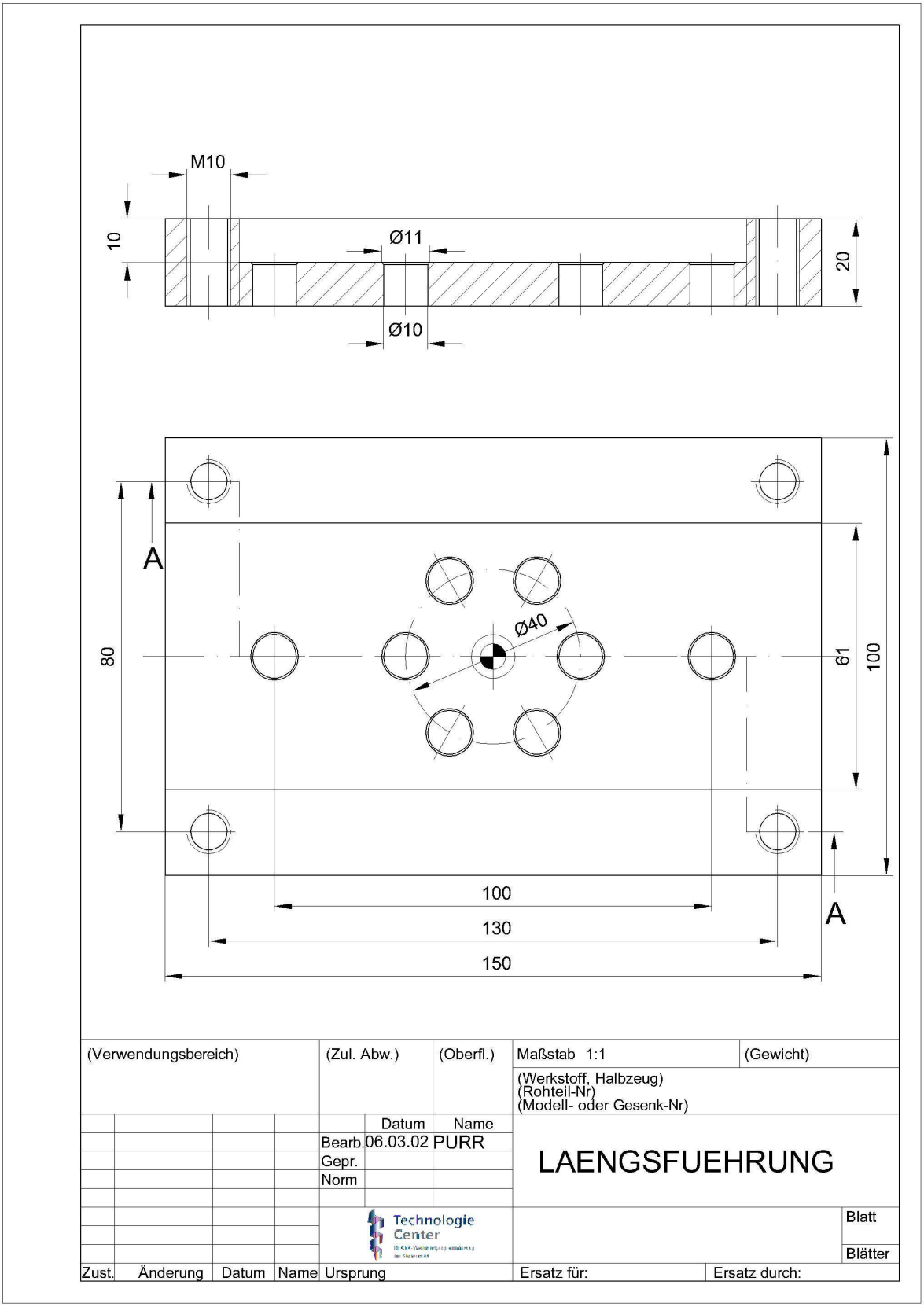
25.2 Hole Plate



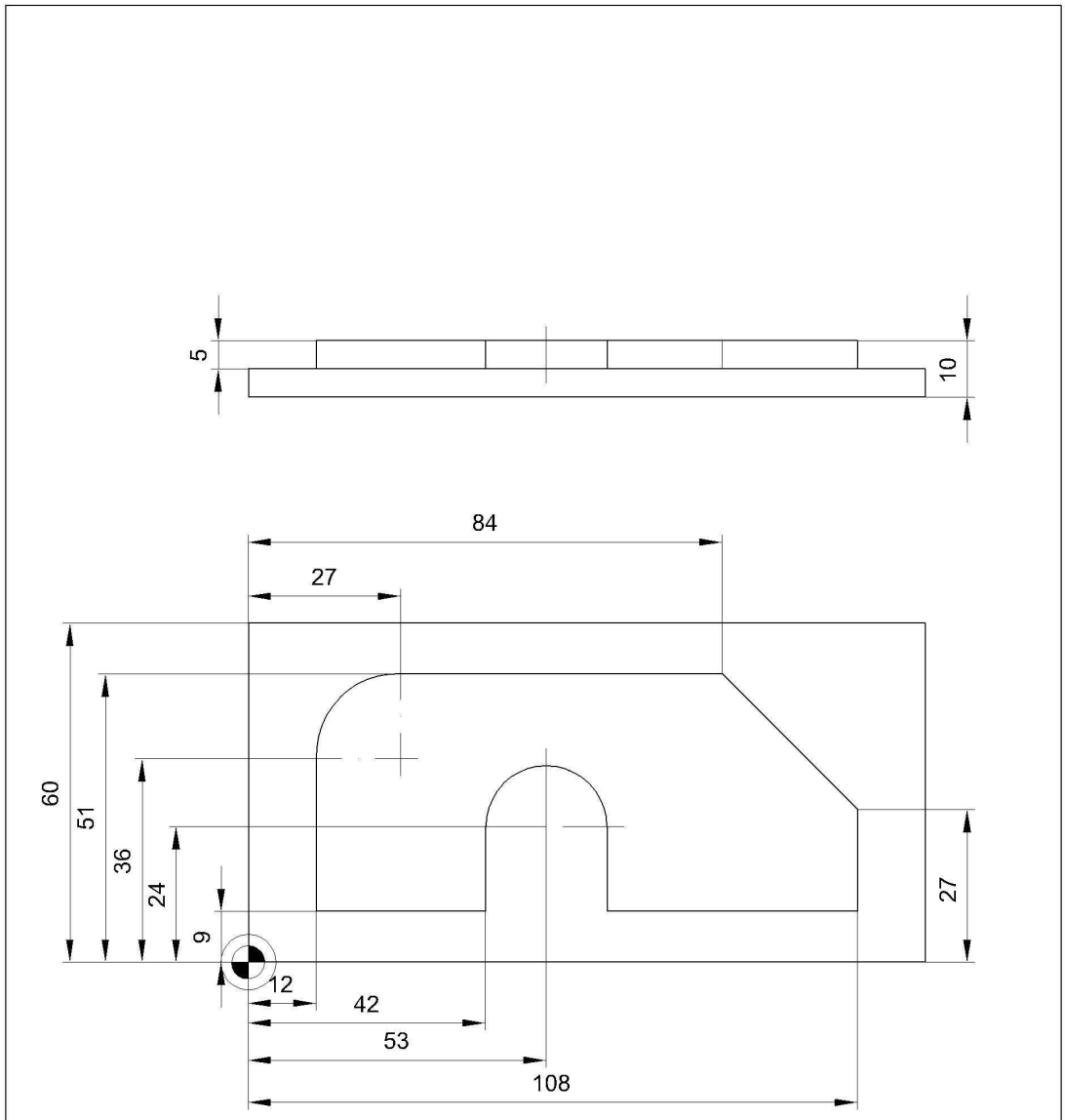
25.3 Housing Lid



25.4 Longitudinal Guide

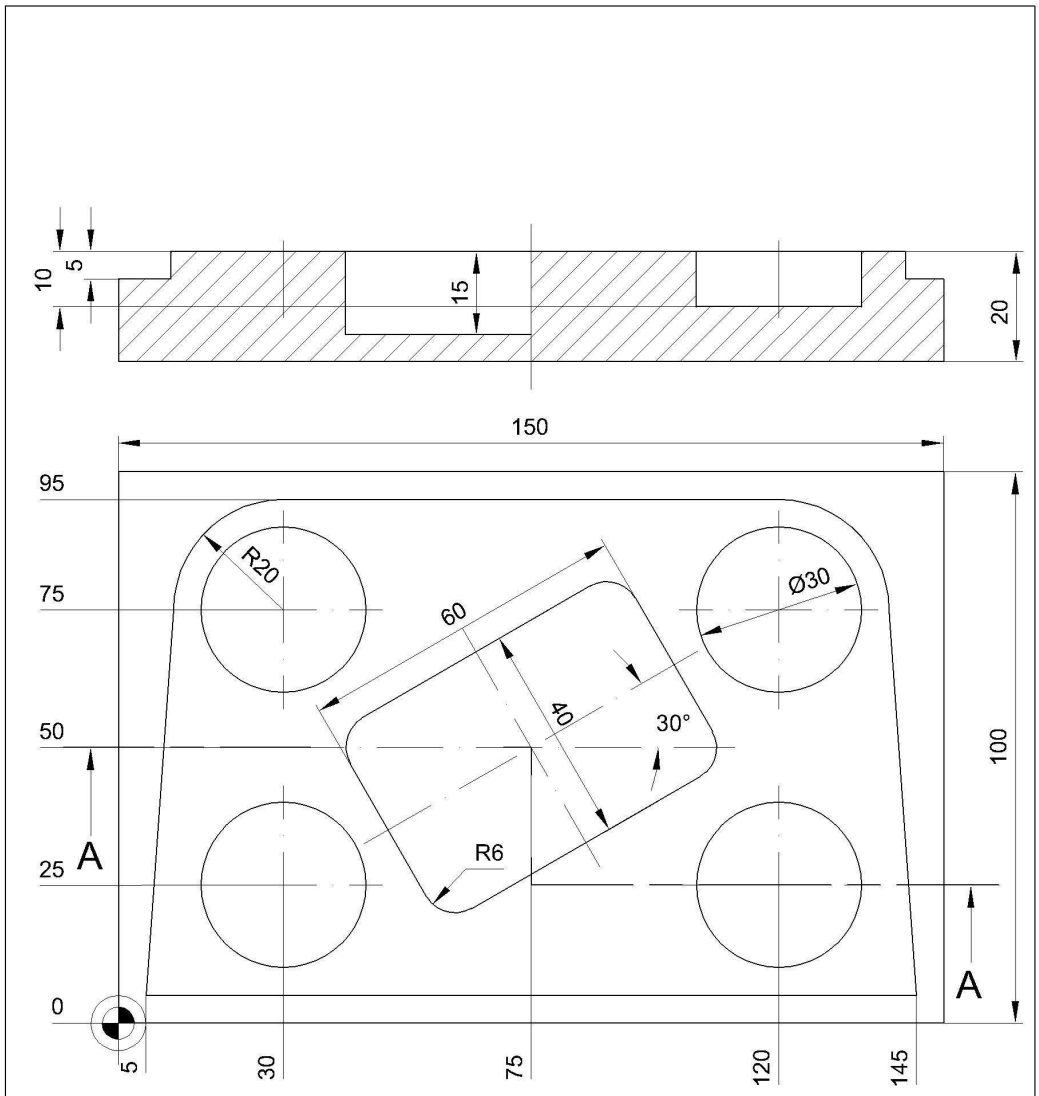


25.5 Example 1



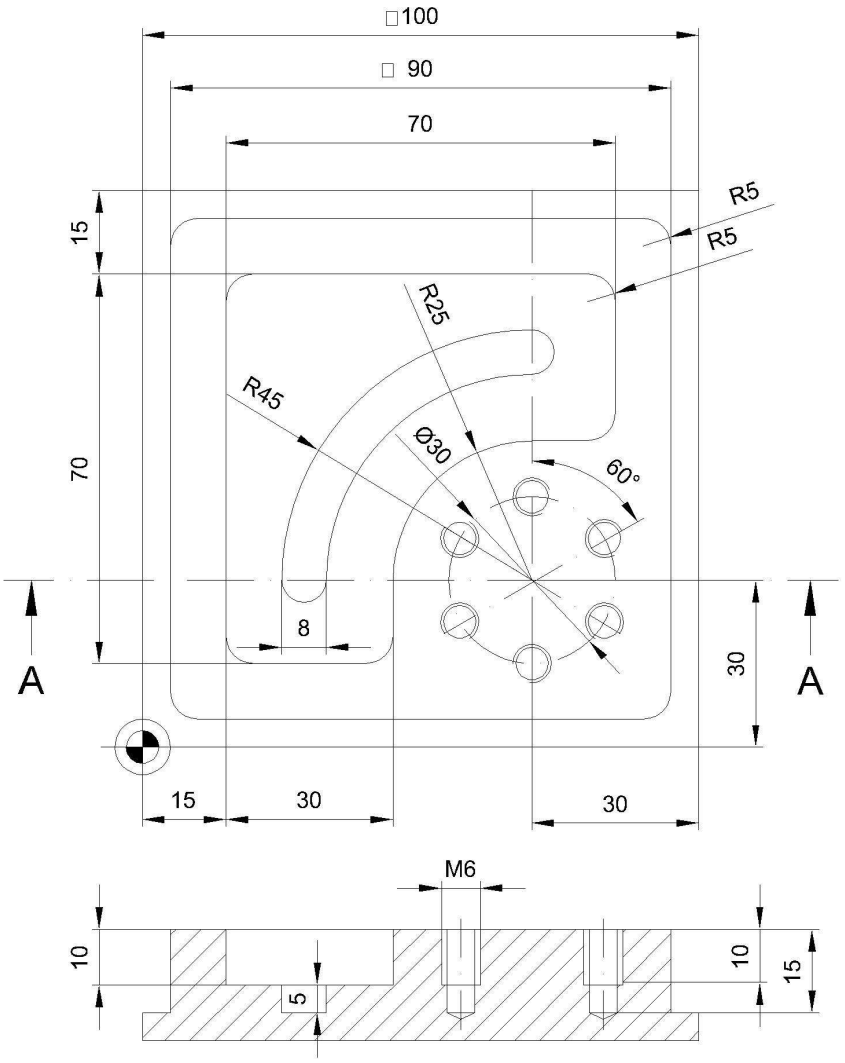
(Verwendungsbereich)				(Zul. Abw.)		(Oberfl.)	Maßstab 1:1		(Gewicht)
							(Werkstoff, Halbzeug) (Rohteil-Nr) (Modell- oder Gesenk-Nr)		
					Datum	Name	BEISPIEL 1		
				Bearb.	06.03.02	PURR			
				Gepr.					
				Norm					
				Technologie Center					Blatt
									Blätter
Zust.	Änderung	Datum	Name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

25.6 Injection Mould



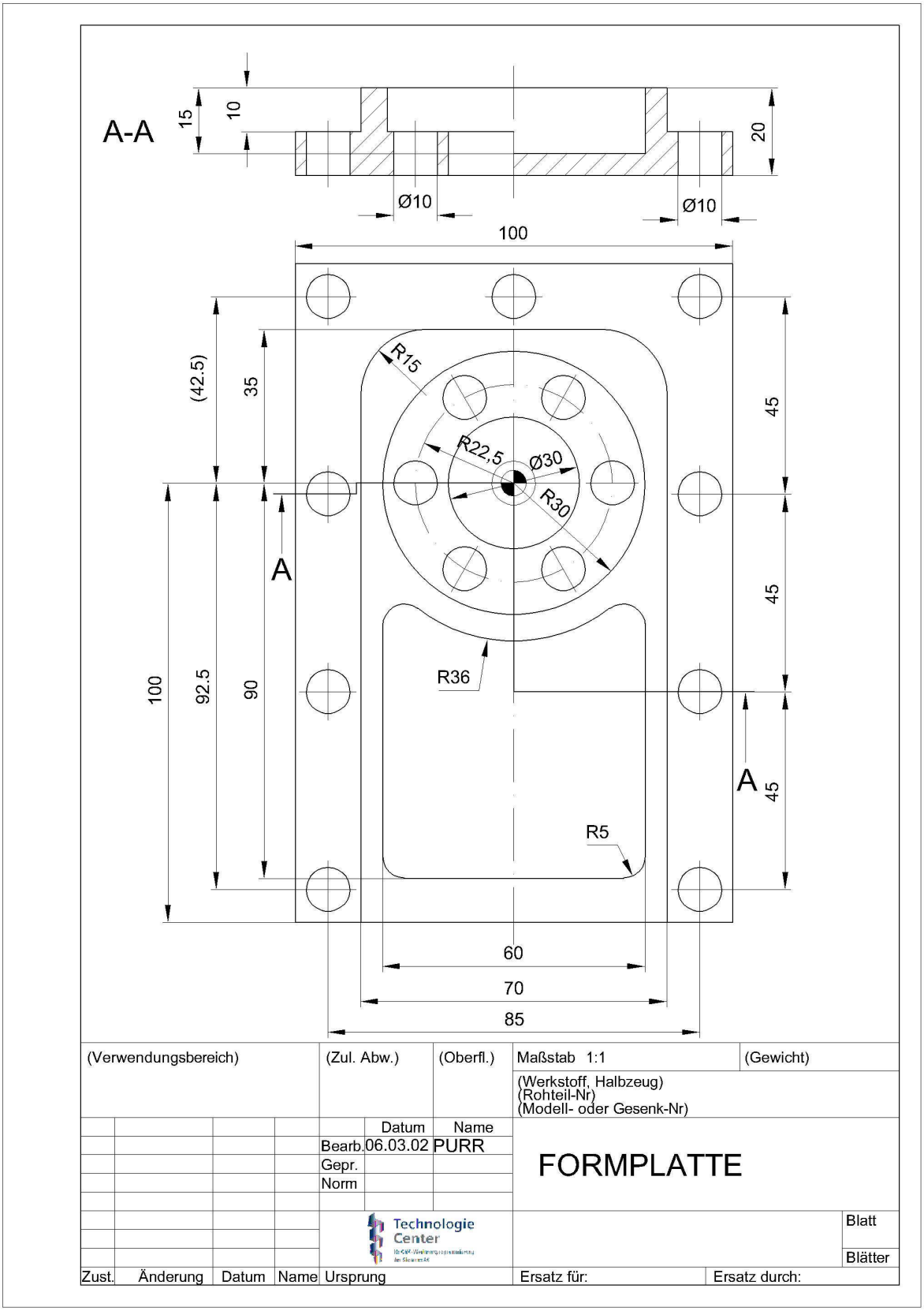
(Verwendungsbereich)				(Zul. Abw.)		(Oberfl.)	Maßstab 1:1		(Gewicht)	
							(Werkstoff, Halbzeug) (Rohteil-Nr) (Modell- oder Gesenk-Nr)			
					Datum	Name	SPRITZFORM			
				Bearb.	06.03.02	PURR				
				Gepr.						
				Norm						
				Technologie Center					Blatt	
									Blätter	
Zust.	Änderung	Datum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

25.7 Measuring Part

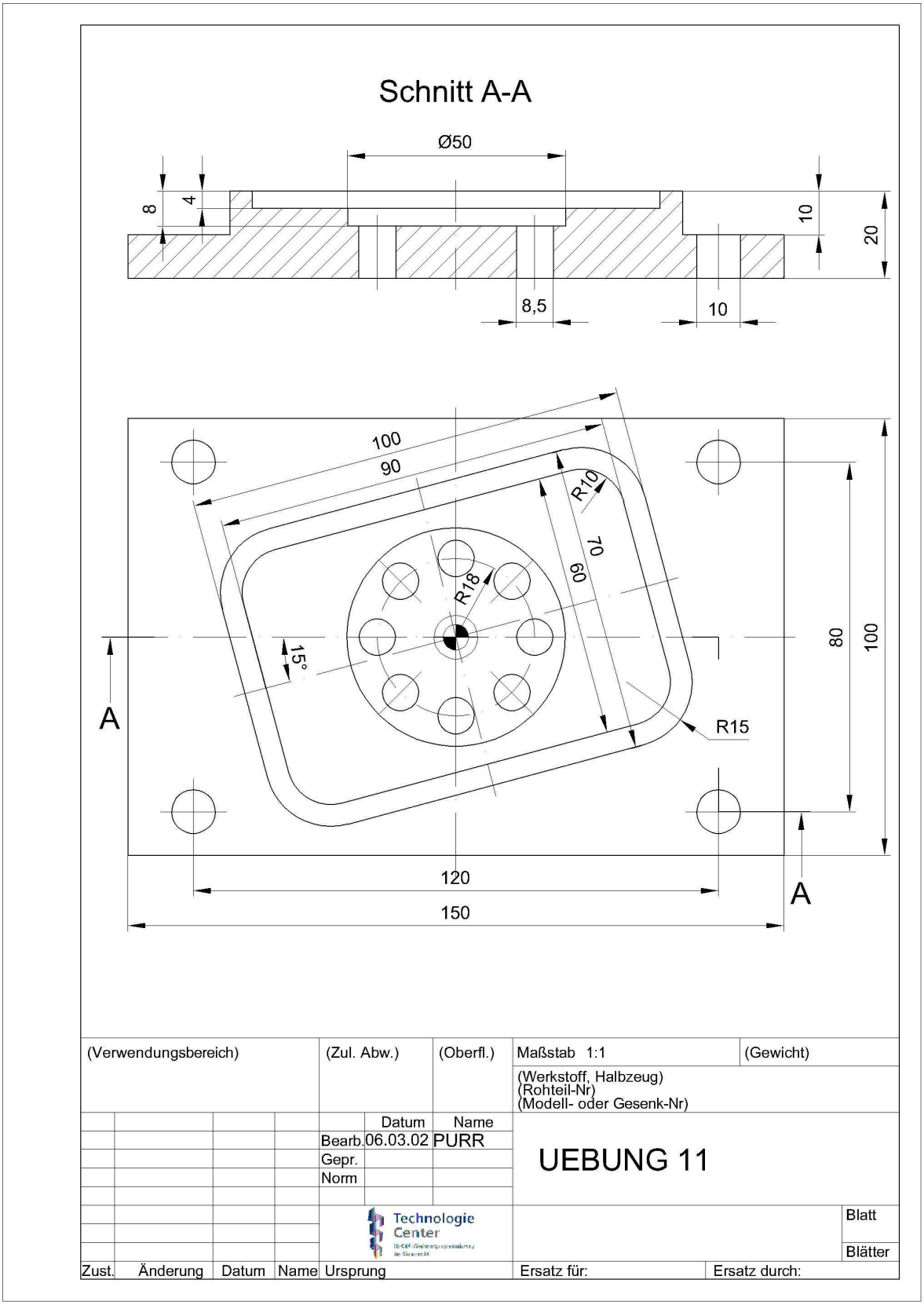


(Verwendungsbereich)				(Zul. Abw.)		(Oberfl.)	Maßstab 1:1	(Gewicht)
							(Werkstoff, Halbzeug) (Rohteil-Nr) (Modell- oder Gesenk-Nr)	
					Datum	Name	MESSE-TEIL	
				Bearb.	21.02.02	KHE		
				Gepr.				
				Norm				
				Technologie Center				Blatt
								Blätter
Zust.	Änderung	Datum	Name	Ursprung			Ersatz für:	Ersatz durch:

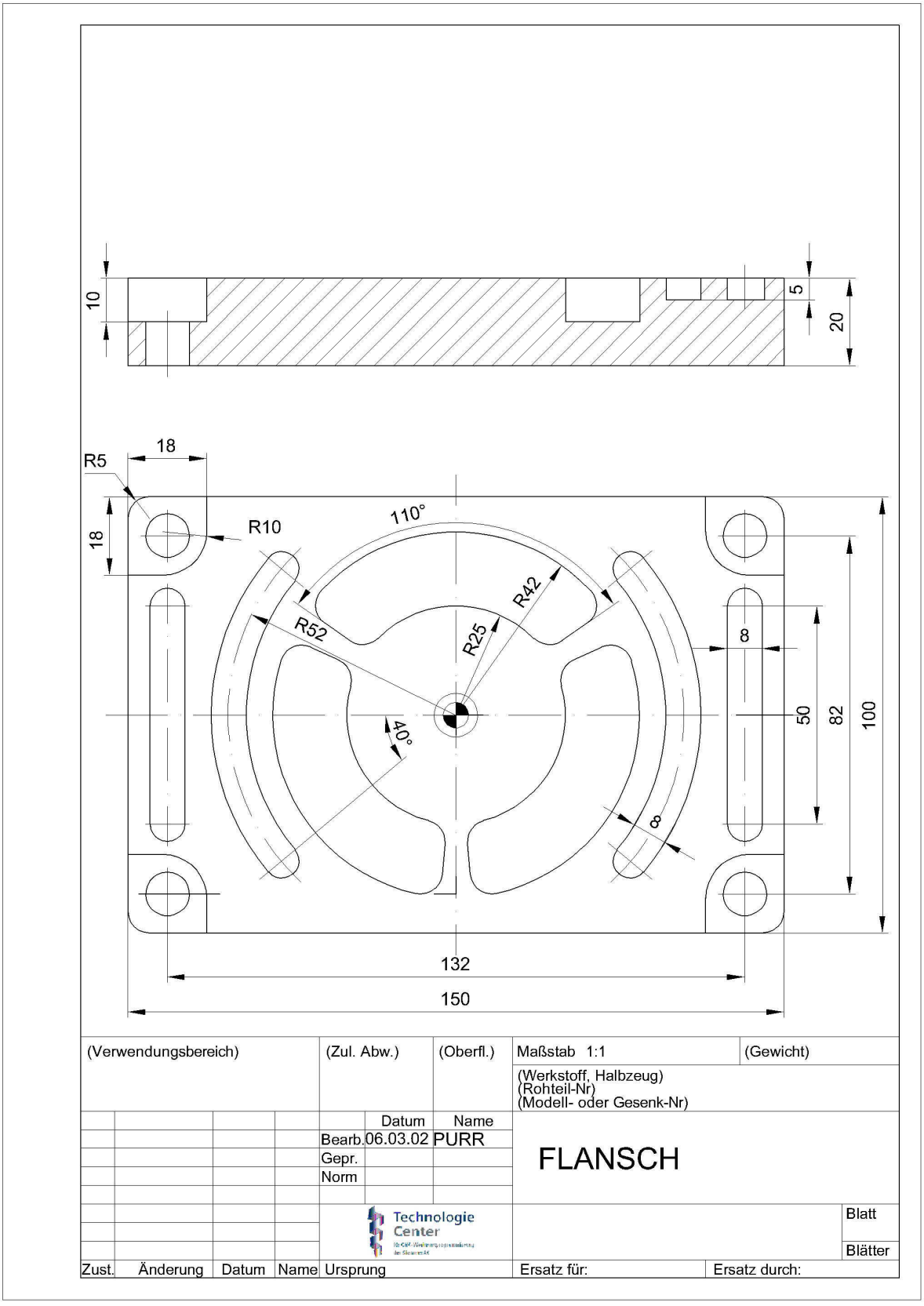
25.8 Pattern Plate



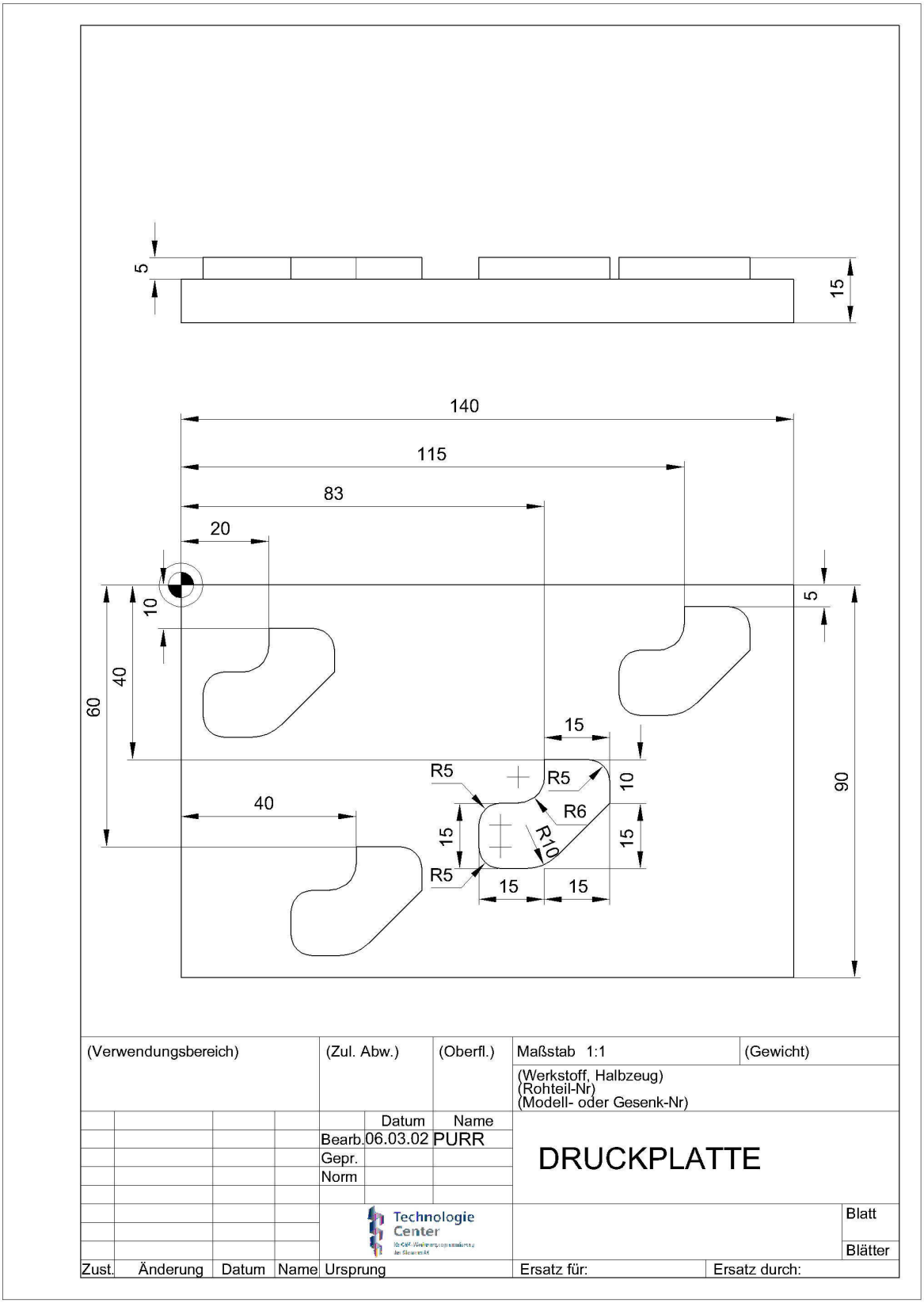
25.9 Exercise 11



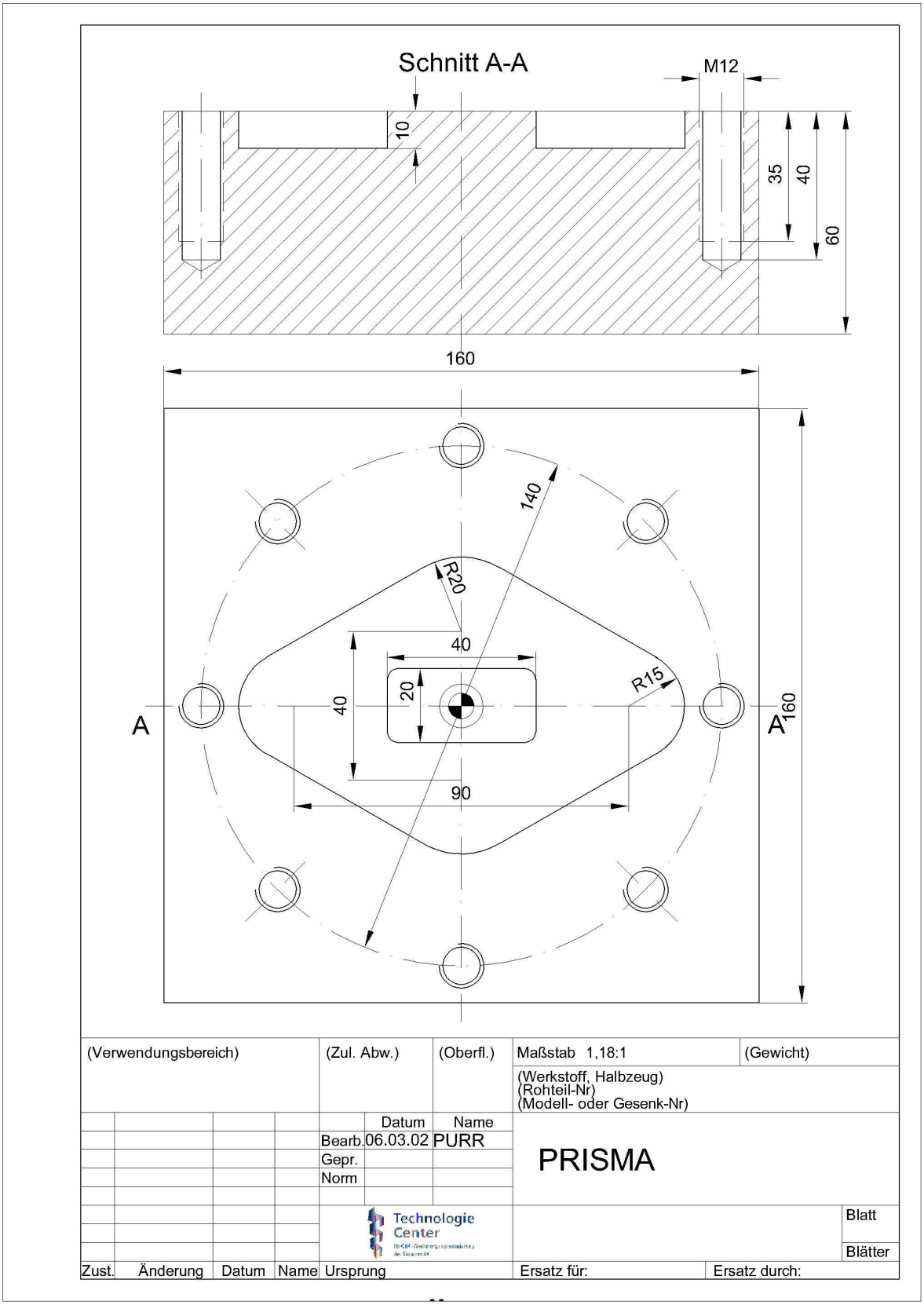
26 Flange


10
5
20
R5
18
18
R10
110°
R42
R52
R25
8
50
82
100
40°
8
132
150
(Verwendungsbereich)
(Zul. Abw.)
(Oberfl.)
Maßstab 1:1
(Gewicht)
(Werkstoff, Halbzeug)
(Rohteil-Nr)
(Modell- oder Gesenk-Nr)
Datum
Name
Bearb. 06.03.02 PURR
Gepr.
Norm
FLANSCH
Technologie Center
Blatt
Blätter
Zust.
Änderung
Datum
Name
Ursprung
Ersatz für:
Ersatz durch:

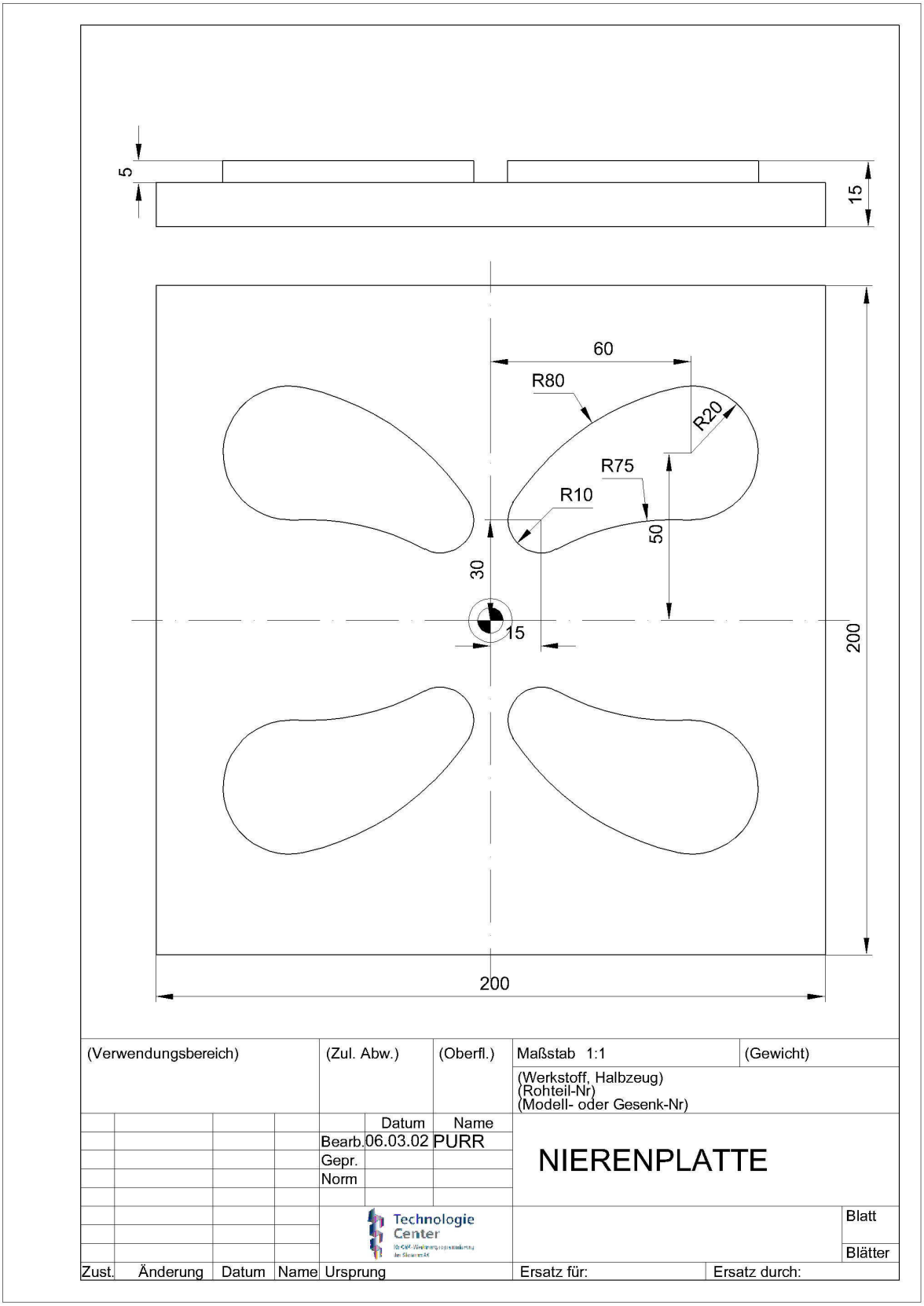
26.1 Clamping Plate



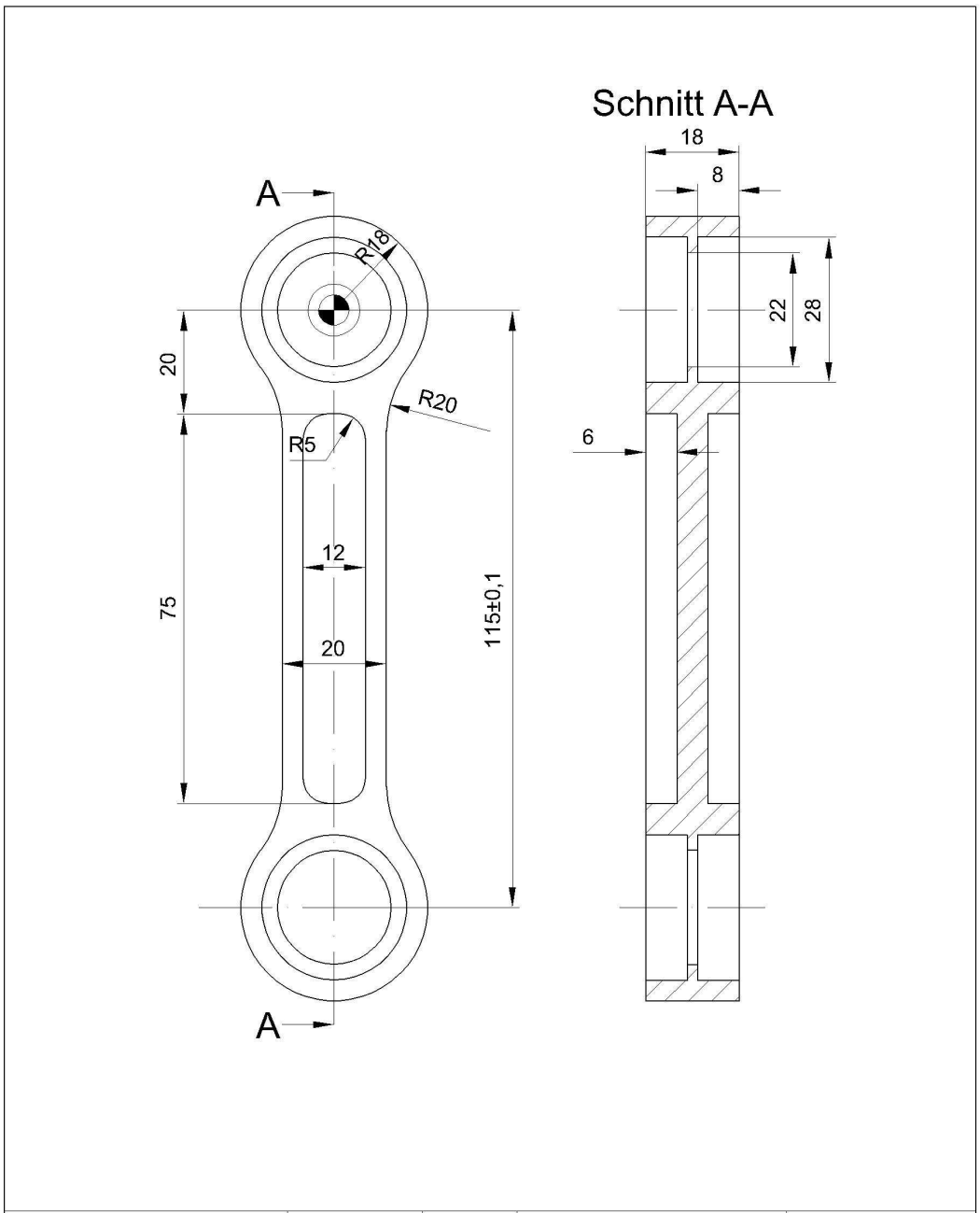
26.2 Prism


Schnitt A-A
M12
10
35
40
60
160
140
R20
40
20
40
R15
90
A
A
160
(Verwendungsbereich)
(Zul. Abw.)
(Oberfl.)
Maßstab 1,18:1
(Gewicht)
(Werkstoff, Halbzeug)
(Rohteil-Nr)
(Modell- oder Gesenk-Nr)
Datum
Name
Bearb. 06.03.02 PURR
Gepr.
Norm
PRISMA
Technologie Center
Blatt
Blätter
Zust.
Änderung
Datum
Name
Ursprung
Ersatz für:
Ersatz durch:

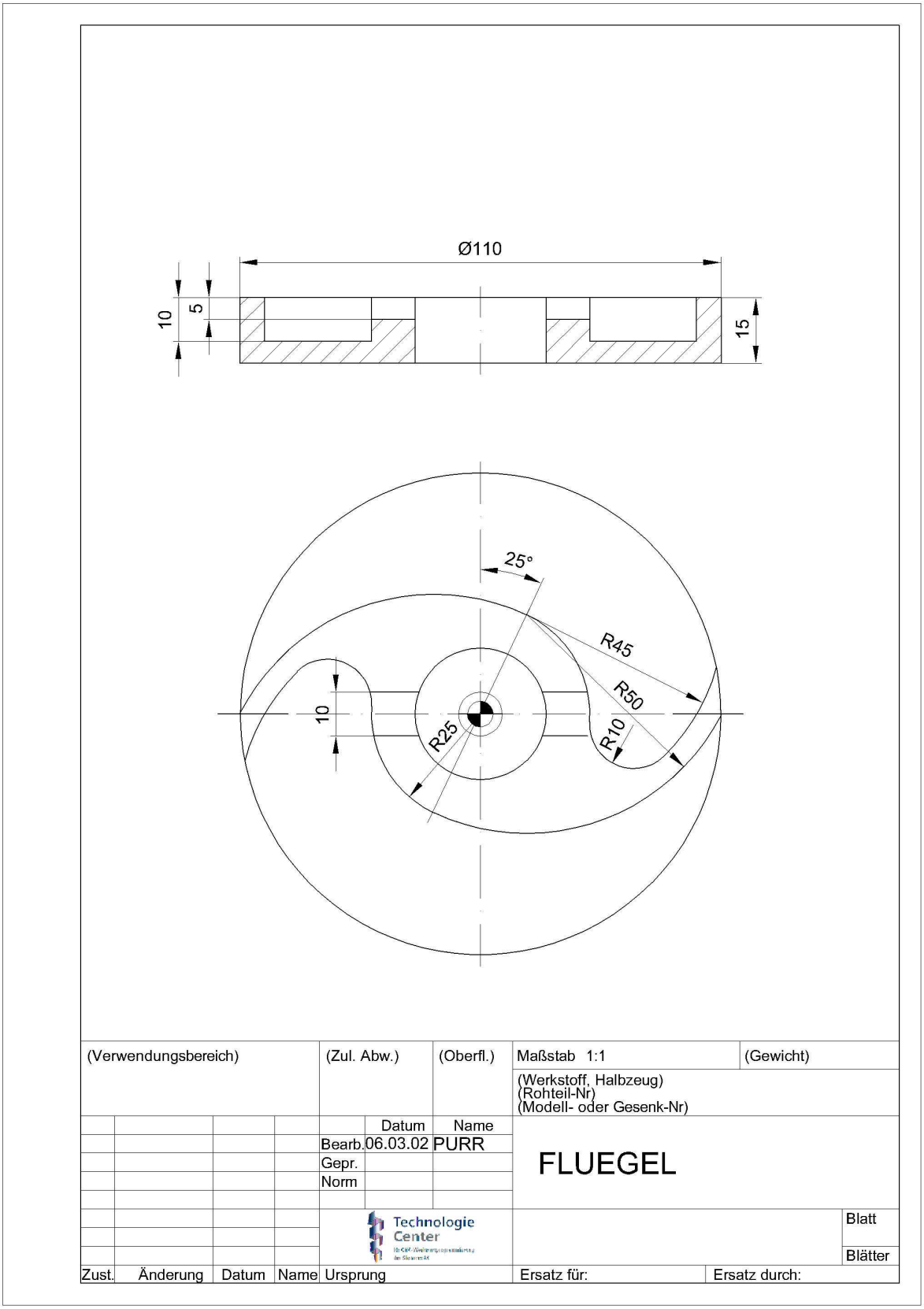
26.3 Kidney Plate



26.4 Connecting Rod



26.5 Wing


Ø110
10
5
15
25°
R45
R50
R10
R25
10
(Verwendungsbereich)
(Zul. Abw.)
(Oberfl.)
Maßstab 1:1
(Gewicht)
(Werkstoff, Halbzeug)
(Rohteil-Nr)
(Modell- oder Gesenk-Nr)
Datum
Name
Bearb. 06.03.02 PURR
Gepr.
Norm
FLUEGEL
Technologie Center
Blatt
Blätter
Zust.
Änderung
Datum
Name
Ursprung
Ersatz für:
Ersatz durch: