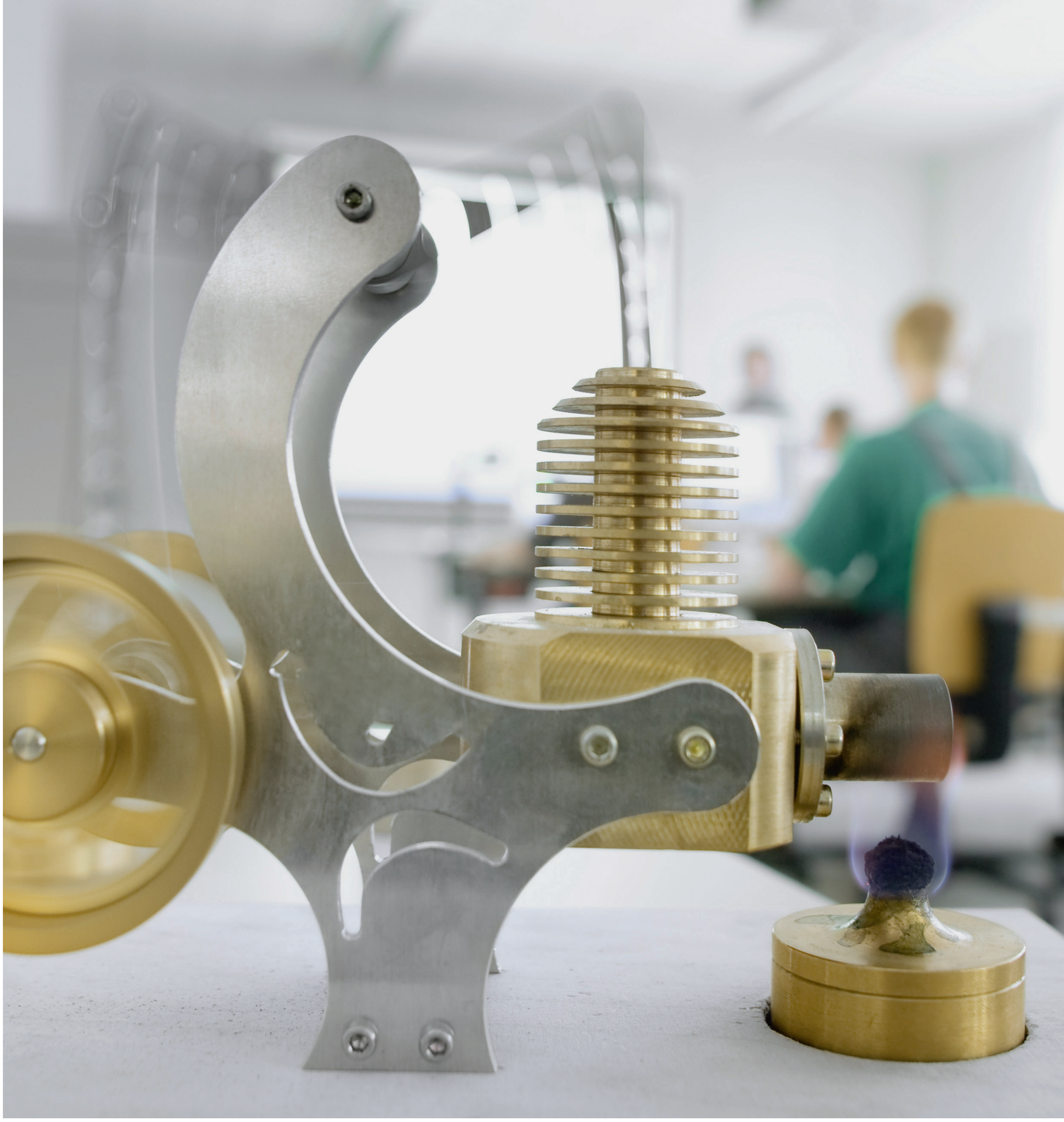
SinuTrain

Milling made easy with ShopMill

Training Documentation · 08/2006



SINUMERIK

SIEMENS

4th and revised edition 08/2006
Valid from software version V06.04

This Beginner’s Guide was produced in cooperation with Messrs.

SIEMENS AG
Automatisierungs- und Antriebstechnik
Motion Control Systems
Postfach 3180, D-91050 Erlangen

and

R. & S. KELLER GmbH
Siegfried Keller, Stefan Nover, Klaus Reckermann, Olaf Anders, Kai Schmitz
Postfach 131663, D-42043 Wuppertal.

Order No.: 6FC5095-0AA50-0BP2

Preface

Faster from the drawing to the workpiece - but how?

Up to now, NC production mainly involved complicated, abstract, coded NC programming. Work that only specialists were able to carry out. However, every technical worker learns his trade and is able to put the experience gained in the area of conventional machining to use to cope with the most difficult tasks - even if the cost/benefit ratio often suffered gravely. A way had to be found to let these technical experts apply their knowledge effectively using NC machine tools.

This is why SIEMENS took a new approach with ShopMill, which saved the need for any coding on the part of the operator. Instead, SIEMENS provides these technical experts with a new generation of SINUMERIK controls:

The solution here is to create a work plan rather than a program.

By creating a workplan with detailed operations of the kind a technician would carry out, the ShopMill user is able to apply his real expertise to the machining process, his actual know-how is not lost.

Even the most complicated of contours and workpieces can be produced easily with ShopMill thanks to the integrated, powerful traversing path creation function. The following therefore applies:

Move easier and faster from the drawing to the workpiece - with ShopMill!

Although ShopMill is really easy to learn, this ShopMill training course will introduce you to the new world even better. Before we start to work with ShopMill, we will address important fundamental issues in the first three chapters:

- First of all, we will outline the benefits of working with ShopMill.
- Then we shall demonstrate the basic operation to you.
- The geometrical and technological basics of production are then explained for newcomers in the chapter that follows.

Theory is followed by ShopMill practice:

- Five examples are used to explain the machining options offered by ShopMill; the complexity of the examples is increased continuously. At the outset, all the keys to be pressed are specified; later, you are prompted to act on your own.
- Then you are tought how to use ShopMill in automatic mode.
- If you wish, you can then test how fit you are in ShopMill.

Please note that the technology data used here can only be seen as examples, due to the numerous different conditions that apply in the workshop.

Just as ShopMill was produced with help from technicians, this training document was produced using input from practical users. In this vein. we wish you every success in your work with ShopMill.

The authors

Erlangen/Wuppertal, September 2003

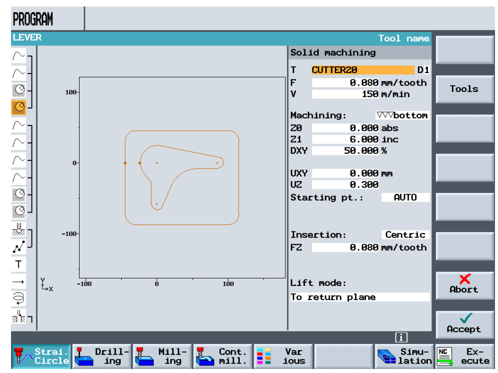
Table of contents

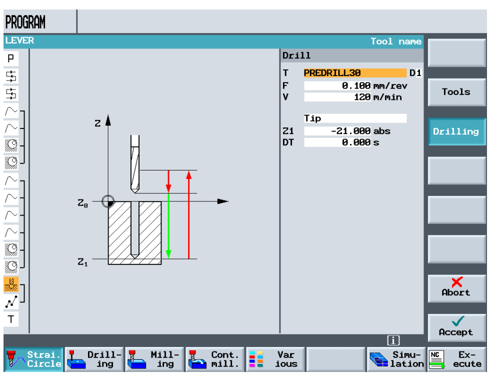
1 Benefits of working with ShopMill

This chapter states the special benefits of working with ShopMill.

1.1 You save training time …

… because there is no coding in ShopMill and no foreign-language terms that you must learn:
All necessary inputs are queried in plain text.

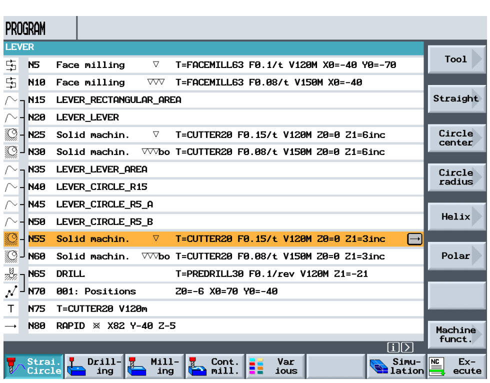


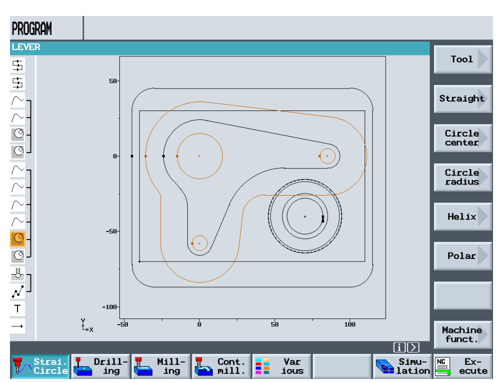


… because ShopMill provides colored help displays for your assistance.

…because you can also integrate DIN/ISO-SQL commands in the *graphic work plan* .

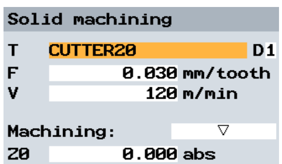
G	N25 G17 G54 G64 G90 G94
T	N30 T=EM16
G	N35 G0 X85 Y22.5
G	N40 G0 Z2 S500 M3 M8
G	N45 G0 Z-10
G	N50 G1 X-85 F200
G	N55 G0 Y-22.5
G	N60 G1 X85
G	N65 G0 Z100 M5 M9



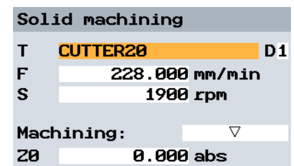


... because you can switch between the individual steps and the workpiece graphic at any time while producing a work plan.

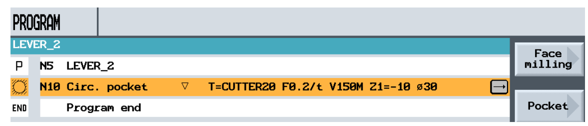
1.2 You save programming time …



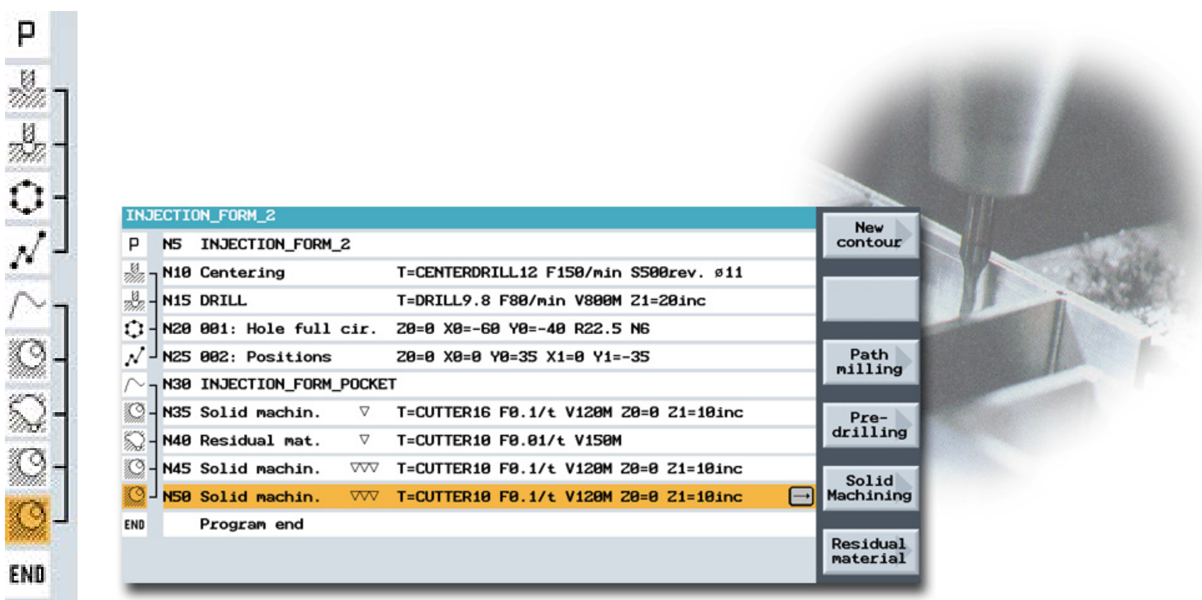
... because ShopMill provides optimum support while entering technology values: you only need to enter the following values from the book of tables: *Feedrate/tooth* and *cutting speed* - ShopMill automatically calculates the speed and the feedrate.



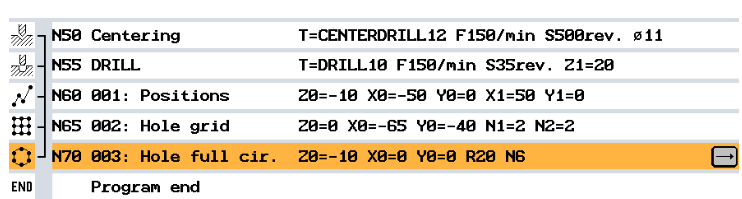
… because ShopMill can describe an entire machining step with one work step; and the necessary positioning movements (here from the tool change point to the workpiece and back) are generated automatically.



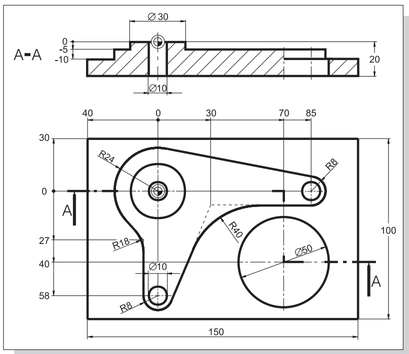
… because the *graphic work plan* in ShopMill represents all machining steps in a compact and concise manner. This gives you a complete overview and provides enhanced editing options, even in the case of extensive production sequences.

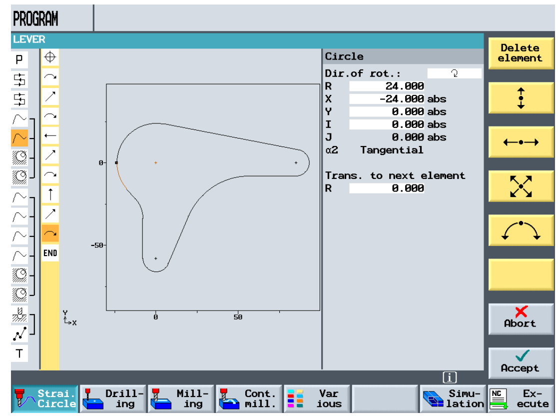


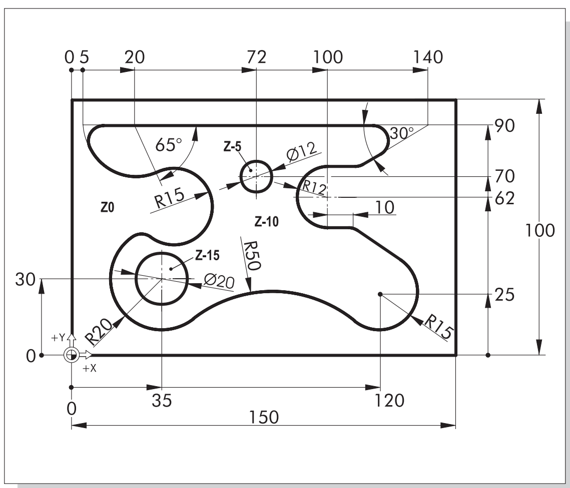
... because several machining operations with numerous position patterns can be linked during drilling and do not have to be called repeatedly.

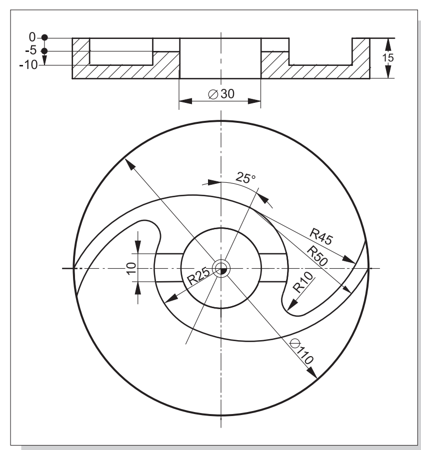


… because the integrated contour calculator can handle all conceivable dimensions and is still easy to operate - thanks to the general-language input and graphic support.

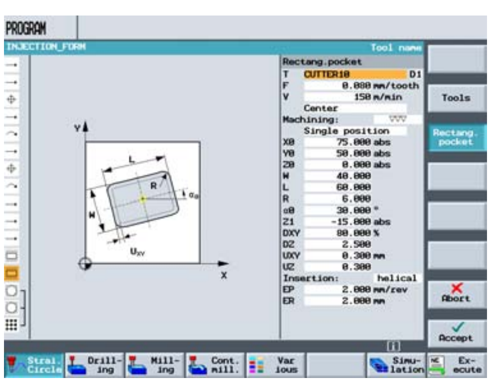


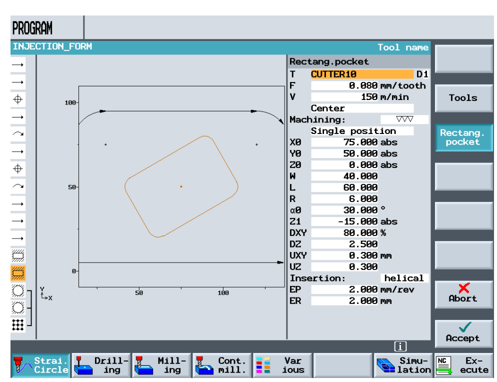






... because you can toggle between the static help displays and dynamic on-line graphics at any time with just one key-stroke. The on-line graphic provides you with a direct means of visually checking the entered values.

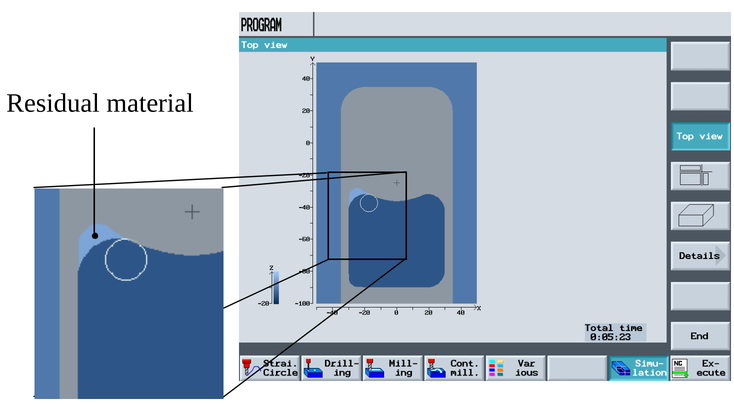




… because the work plans Extensions and Finish are not mutually exclusive: With ShopMill you can create a new work plan in parallel with your production.

1.3 You save production time …

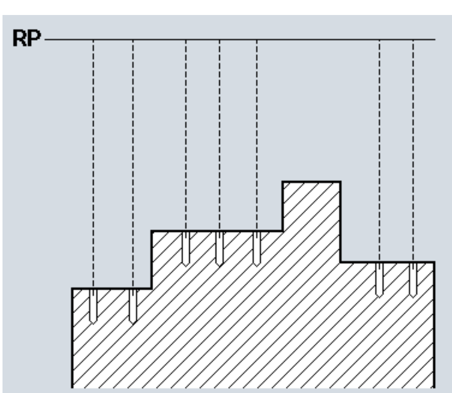
… because you are not restricted by the radius of the pocket in your selection of milling tools for machining contour pockets:
The remaining residual material is detected and automatically machined by a smaller milling tool.



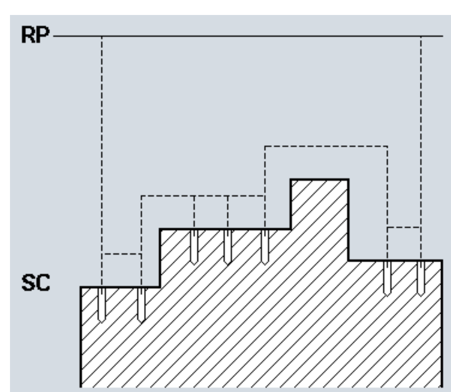
… because there are no superfluous infeed movements between the return and machining plane during positioning operations. This is made possible by the settings *Return on RP* or *Optimized return*.

Return on return plane (RP)

Return on machining plane
= time saving during production

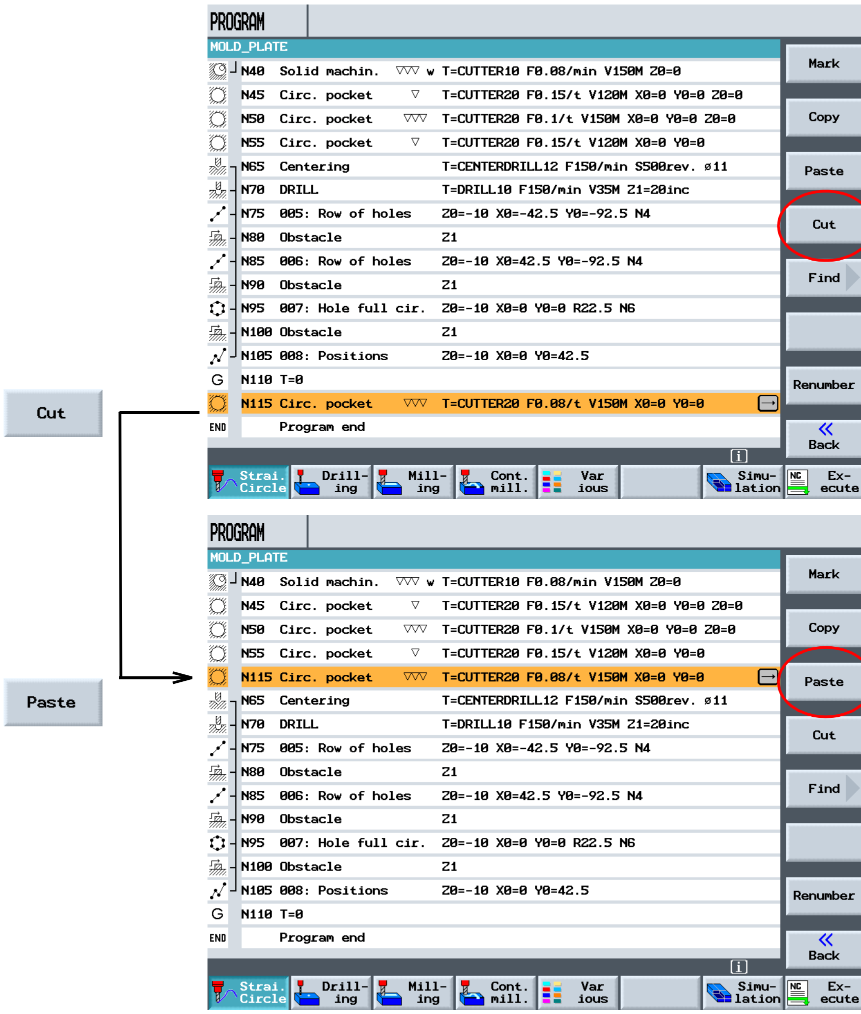


Help displays in ShopMill



The setting *Optimized return* must be made in the program header by a technical expert. He must consider such obstacles as Clamping elements.

... because you can utilize the compact structure of the work plan to optimize your machining sequence easily (here, for example, by saving tool change operations).



Original machining sequence

Optimized machining sequence through *Cut* and *Paste* for work steps

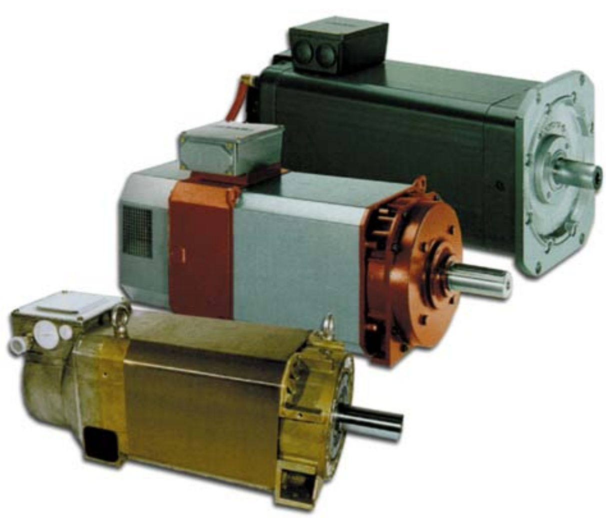
... because ShopMill makes full use of digital technology (SIMODRIVE drives, SINUMERIK controls) to achieve fastest feedrates and highest accuracy for repeated operations.

2 So that everything runs smoothly

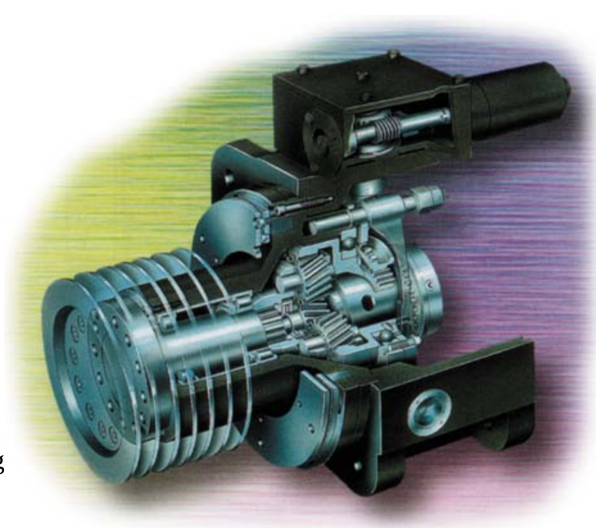
In this chapter, you learn the basics of how to operate ShopMill.

2.1 Tried-and-tested technology

The SINUMERIK 810D as the basis for ShopMill is the most cost-effective way to get started in the world of future-proof, digital CNC and drives for machine tools.



With the aid of the SIEMENS three-phase servo motors and ...



... SIEMENS gearbox technology, production is carried out at top speed, with the highest feedrates and with rapid traversing speeds where required.

2.2 The machine operator panel

It is okay having powerful software at hand; but it must be easy to operate.
The clearly laid out machine operator panel of ShopMill guarantees ease of operation. It is made up of three parts.

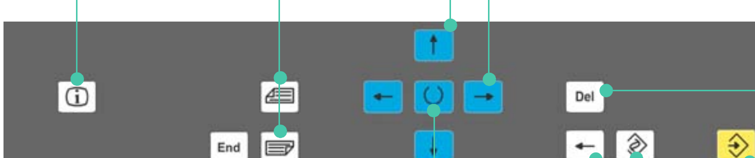
The most important navigation keys on the full CNC keyboard are shown here:

Pages up or down

The 4 arrow keys move the cursor ...

Info key

... this arrow key also opens existing work steps.



This key deletes the value in an input field.

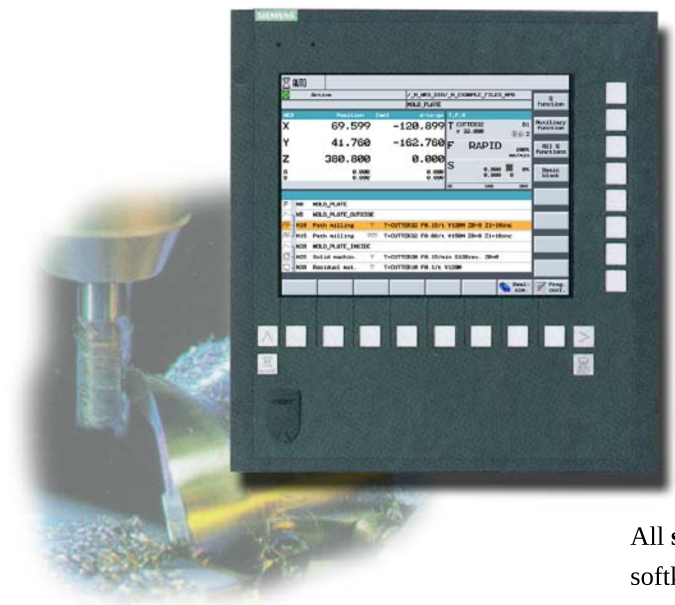
Alternative key (same function as Alternat.)

The input key is used to:
... accept a value in the input field.
... terminate the computation process.
... move the cursor down.

This key deletes the inputs to the left of the cursor.

This key starts the calculator function.

Take a look at the different groups of keys on the panel; they help you get used to ShopMill.

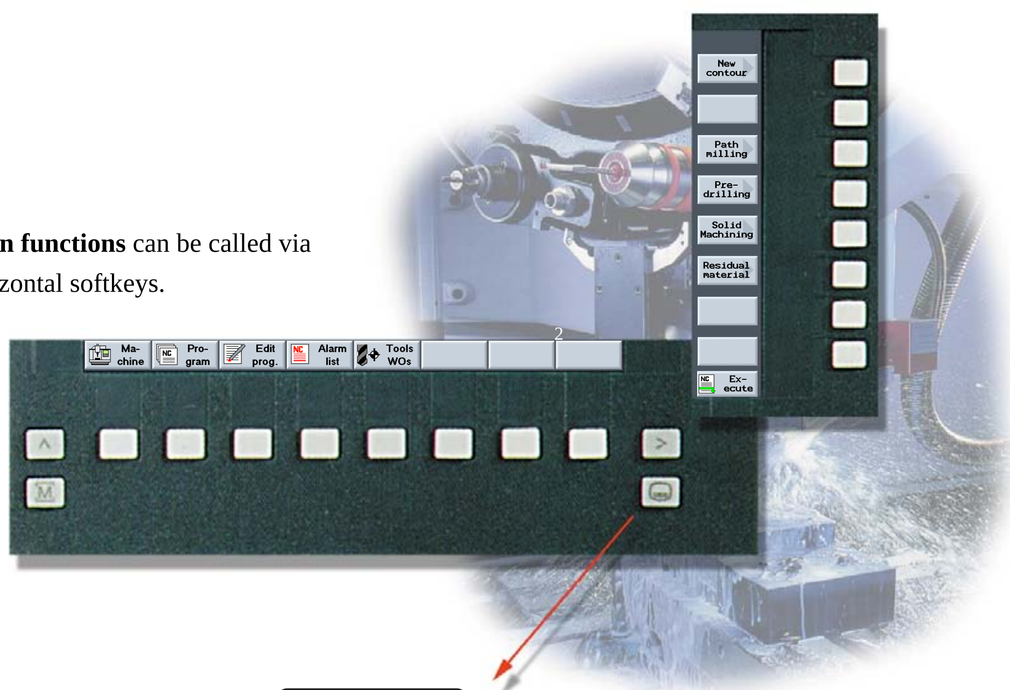


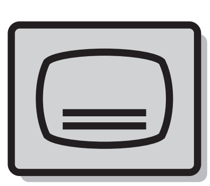
Softkeys

The actual functional selection in ShopMill is carried out with the keys located around the screen. These are generally assigned directly to the relevant menu items. Since the contents of the menus change depending on the situation, we speak of softkeys.

All **subfunctions** of ShopMill are reached via the vertical softkeys.

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





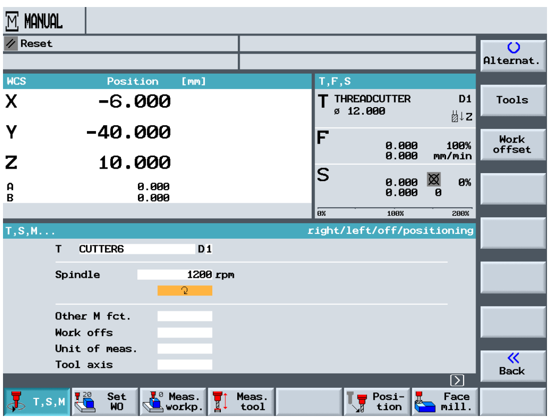
The basic menu can be called at any time - irrespective of the particular operating step where you happen to be.

Basic menu

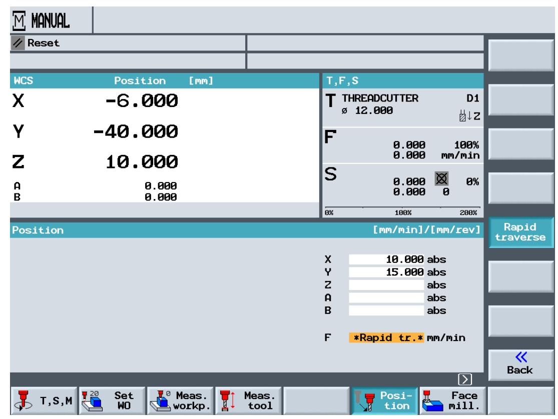
2.3 Contents of the basic menu

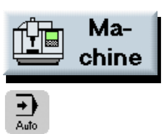
The machine is set up here, the tool traversed in manual mode, etc.
You can also calibrate the tools and set zero points.

Calling a tool and entering technology values

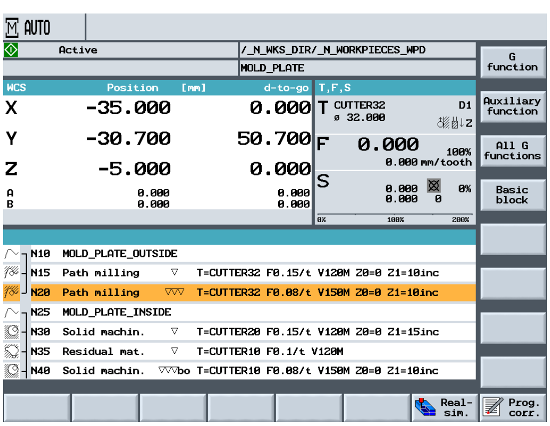


Enter a target position



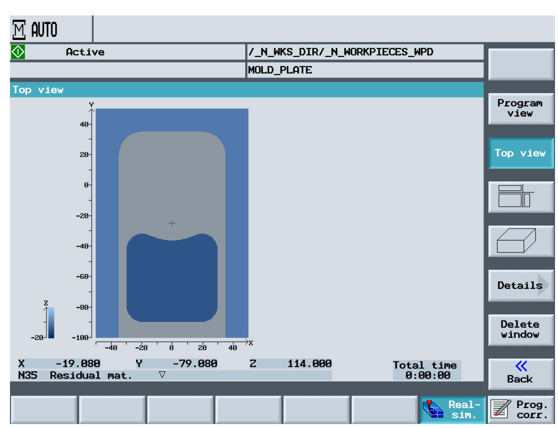


During production, the current work step is displayed. You can switch to a parallel simulation per keystroke. While processing a work plan, you can add work steps or start to create a new work plan.

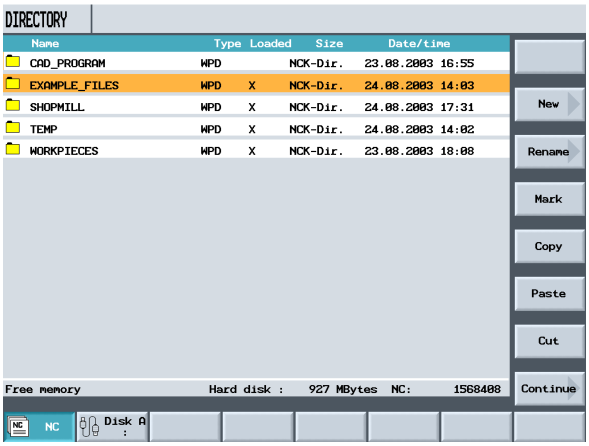


Display of work steps and current technology data ...

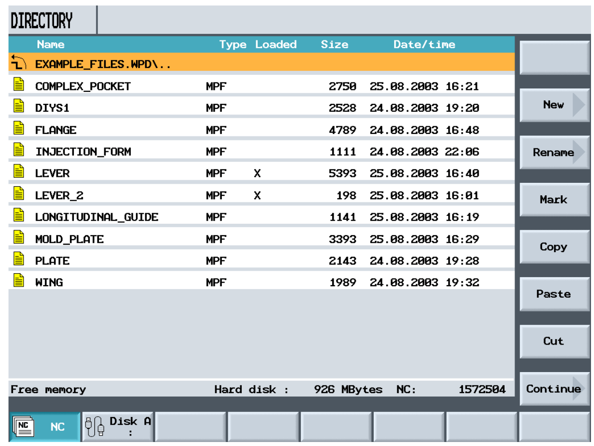
... or the simulation



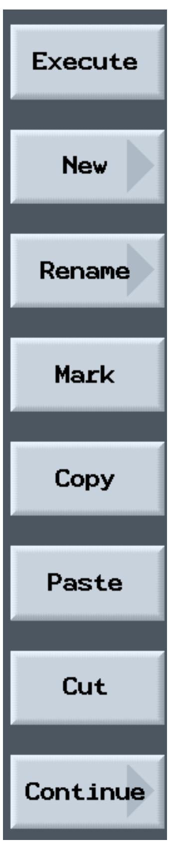
The work plans and contours are managed here. Furthermore, you can also input or output work plans.



To prevent a work plan list ecoming too long and difficult to handle, you can use the *Program Manager* to create as many directories as you like.



You can then save the various work plans in the different directories you have created.



The selected work plan is processed in the *Automatic machine* mode.

New folders and work plans are created.

Folders and work plans are renamed.

Work plans are grouped together for moving or copying.

The marked work plans are placed on a clipboard.

The contents of the clip board is added to another folder.

The marked work plans or work steps are removed here and placed on the clipboard.

Work plans are moved from the hard disk to the NC Kernel.

Work plans are moved from the NC Kernel to the hard disk.

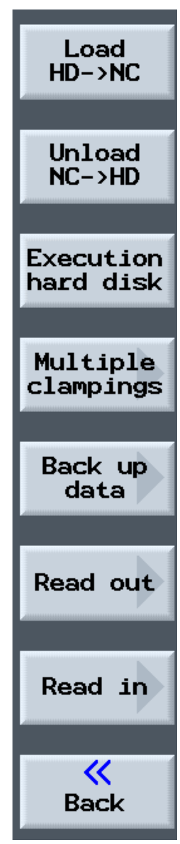
Block transmission is possible to execute long ISO programs.

More than one workpiece can be machined in parallel.

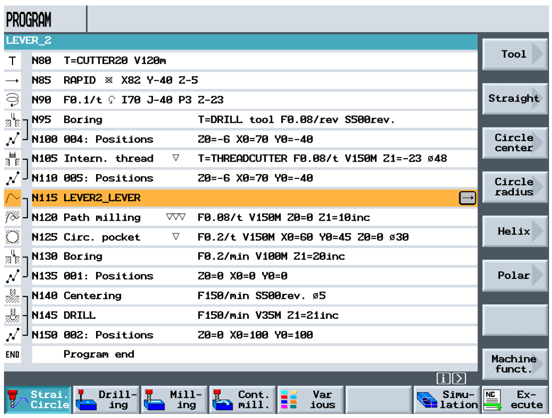
Existing work plans are renamed.

The work plans are exported to an external store.

The work plans are imported from an external store.

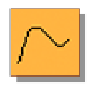


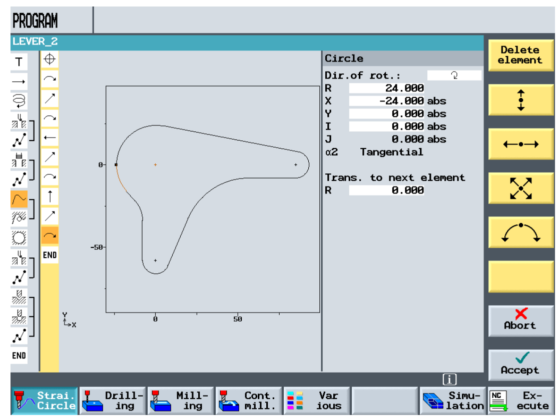
The softkeys *Continue* and *Back* can be used at any time to switch back and forward between the softkey bars.



The work plan is created for the relevant workpiece here along with its full machining sequence. Prerequisite for the optimum sequence is the experience of the technician.

Contour



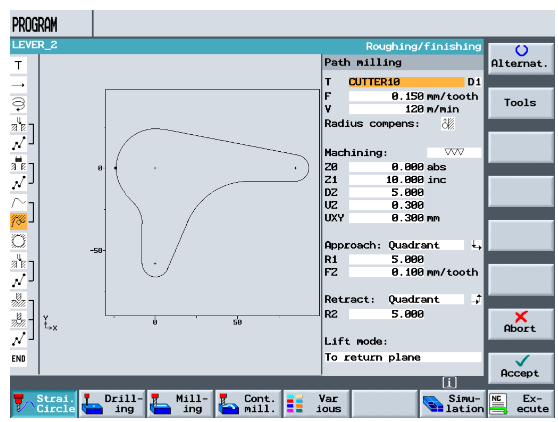


The contour to be machined is entered graphically...

... and then converted to swarf:

Geometry and technology are fully interlinked.

Machining path milling



Contour path milling

incl. approach and return strategies

Circular pocket incl. technology and position

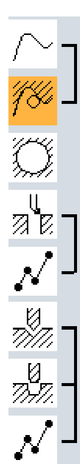
Boring technology

Position for boring

Centering technology

Drilling technology

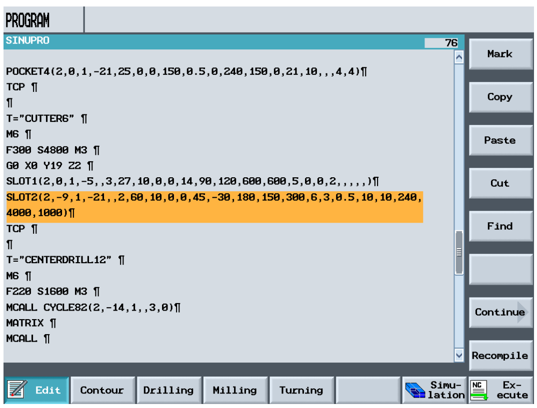
Positioning for centering and drilling



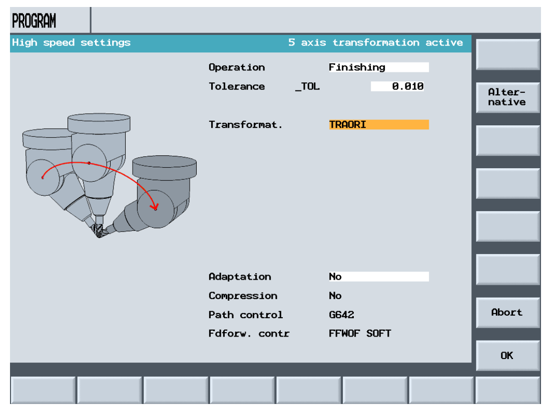
Example for the interlinking of geometry and technology

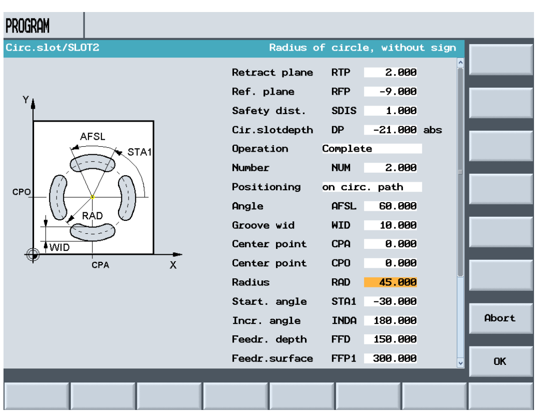
This geometrical/technological link is clearly demonstrated in the graphical display of the work steps in the form of a "grouping" of the relevant icons. The "grouping" refers to a geometry/ technology interlink.

The ShopMill interface is based on the tried-and-tested Sinumerik 810D control. You can use the *CNC ISO* to switch to the Sinumerik plane. The production can now run in exactly the same way as the other 810D controls.



The combination of ShopMill with the Sinumerik 810D produces high flexibility in the CNC production.





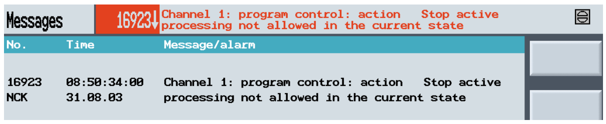
A dedicated Getting Started Guide (Order No. 6FC5095-0AB00-0BP1) with two sample programs for milling workpieces is available for the G code programming of the 810D/840D.

As explained in Chapter 1, you can also input NC programs in foreign control languages in addition to the standard SINUMERIK programs. These commands are "understood" by ShopMill and converted to chips.

N90 G291 (selection of the external language)

N100 G17 G54 Plane selection and zero point offset
N105 G90 G00 G43 X0 Y0 H1 Z100 ...
N110 G83 X10 Y11 Z-30 R10 F100 Q8 Drilling cycles with the control-related parameters
N120 X80 Y90 Drilling position
N130 G80 End of drilling cycles
N140 G53 X20 Y20...
N150 G55...

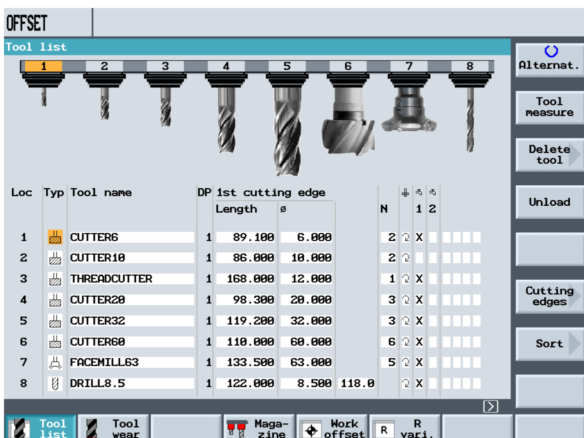
N160 G290 (back to SINUMERIK language)

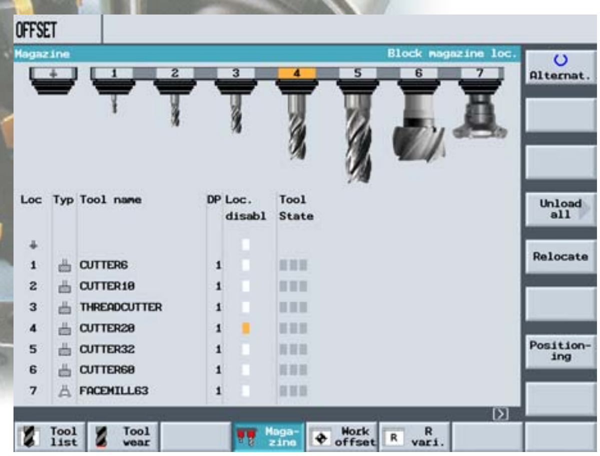


All currently present messages and alarms are displayed with the corresponding error number, the time at which the error occurred and further details of the particular error.

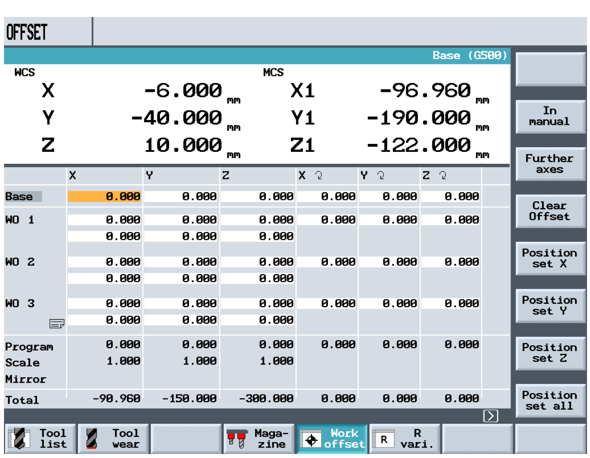
A list of messages and alarms is given in the ShopMill user documentation.

No stock removal without tools. You can manage these in a tool list ...





... and combine them in a magazine.



The zero points are saved in a clearly laid out table of zero points.

3 Fundamentals for newcomers

All the fundamentals of the geometry and technology for milling are explained in this chapter.
No entries have been made in ShopMill yet.

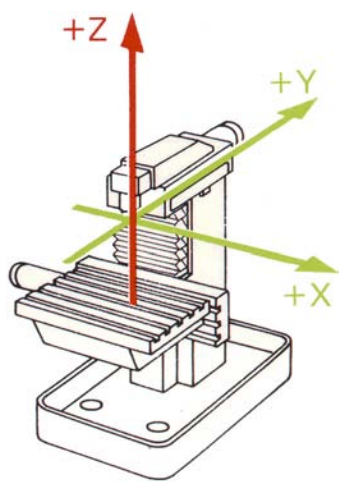
3.1 Geometry basics

3.1.1 Tool axes and work planes

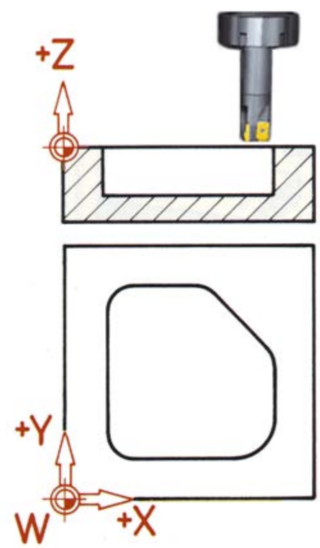
The tool can be installed in parallel to each of the three main axes on universal milling machines. These axes which stand at right angles to each other are oriented according to DIN 66217 or ISO 841 on the main guide ways of the machine.

The installation position of the tool produces a corresponding work plane. Z is usually the tool axis.

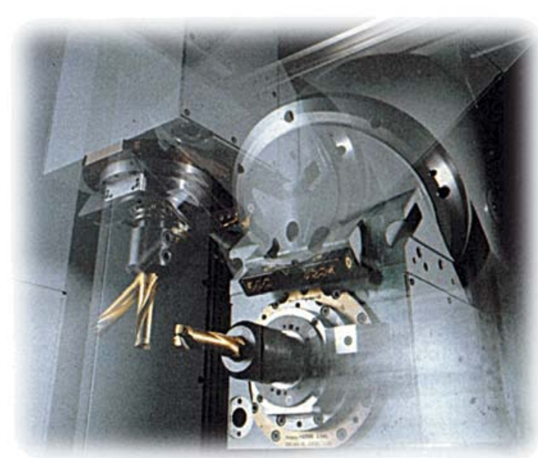
Tool axis Z



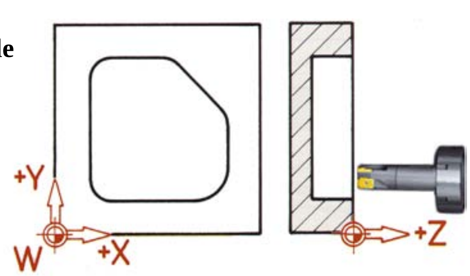
Vertical spindle



On modern machines, it only takes a few seconds to change the tool mounting position with a universal revolver and there is no need for conversion work.



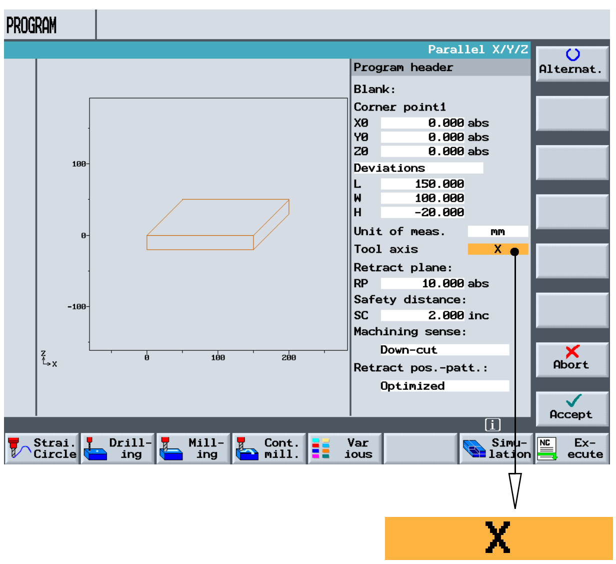
Horizontal spindle

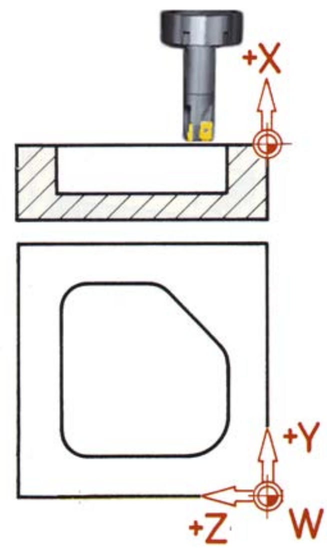


If the coordinate system on the previous page is rotated appropriately, the axes and their directions are changed in the corresponding work planes (DIN 66217).

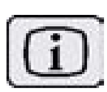
Tool axis X

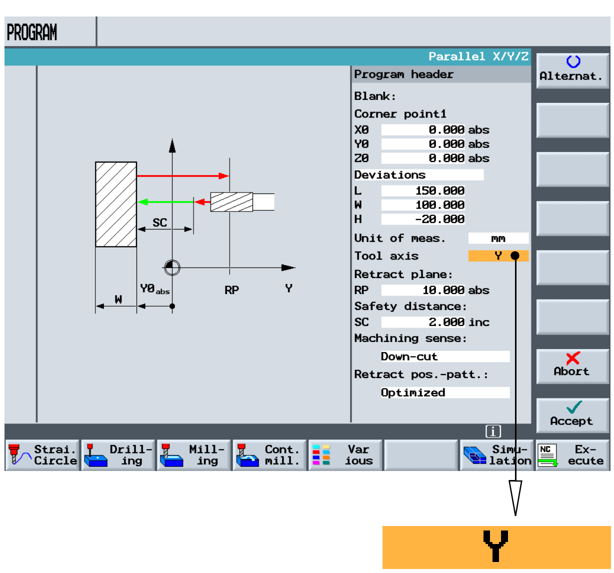
The figure shows the program header after switching to tool axis X.

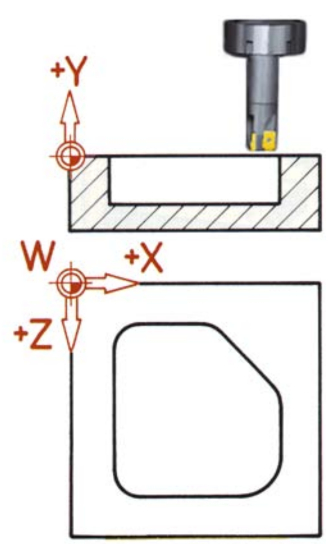




Tool axis Y

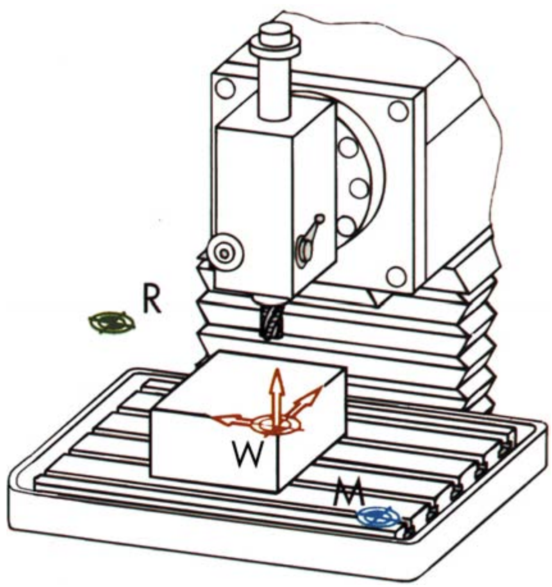
You can of course use the  key to call a help display to help you select the tool axis and enter the values in the program header.





3.1.2 Points in the work area

For orientation of a CNC control (like the SINUMERIK 810D with ShopMill) over the measuring system in the existing work area, important reference points must be defined.



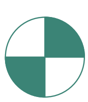
Machine zero M

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

Workpiece zero W

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

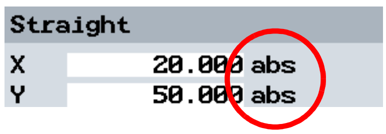
Reference point R



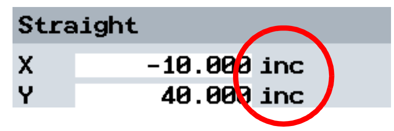
The reference point R is approached to set the measuring system to zero, since the machine zero generally cannot be approached. In this way, the control finds its starting point for counting in the linear measurement system.

3.1.3 Absolute and incremental dimensions

Absolute entry:
The input values refer to the workpiece zero.

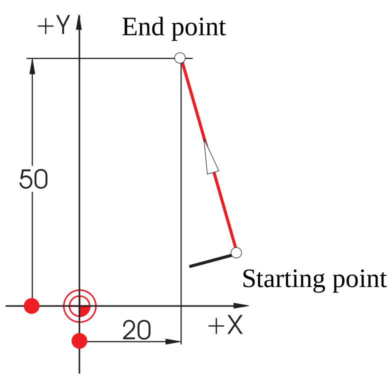


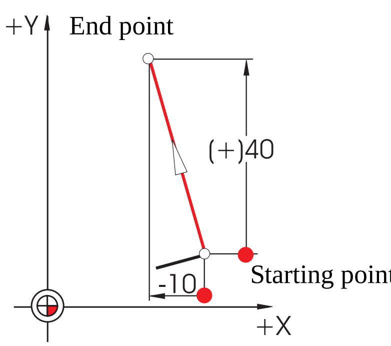
Incremental inputs:
The input values refer to the starting point.



You can use the 

key to switch over at any time.

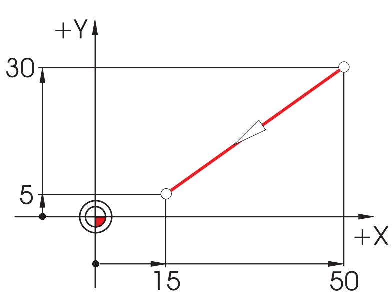




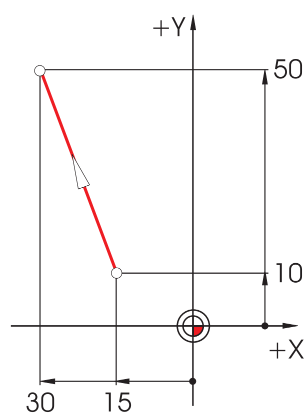
For absolute inputs, you must always enter the **absolute** coordinate values of the **end point** (the start point is not considered).

For incremental inputs, you must always consider the **direction** when entering the **difference** values between **start point** and **end point**.

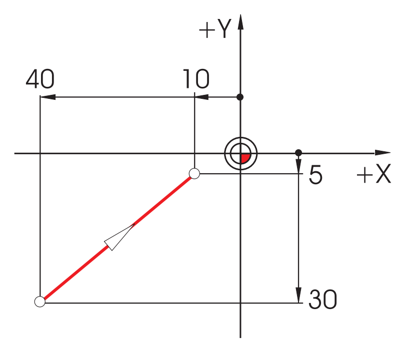
Here are some examples for the combination of absolute/incremental values:



Absolute: X15 Y5
Incremental: X-35 Y-25



Absolute: X-30 Y50
Incremental: X-15 Y40

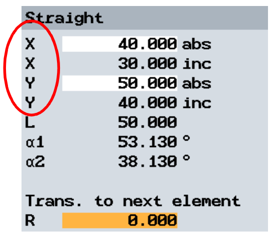


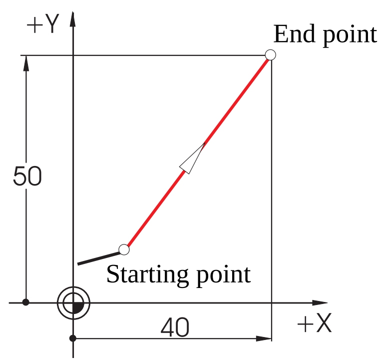
Absolute: X-10 Y-5
Incremental: X30 Y25

3.1.4 Movements on a straight line

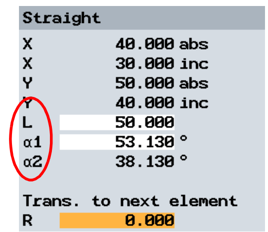
Two entries are required to precisely define the end point. The data could look like this:

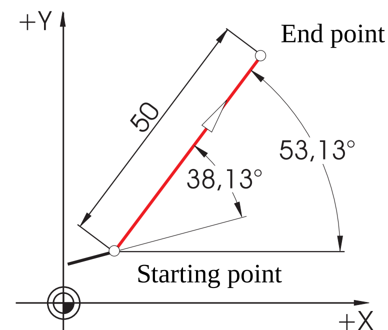
Cartesian: entry of X and Y coordinates





Polar: enter the length and an angle





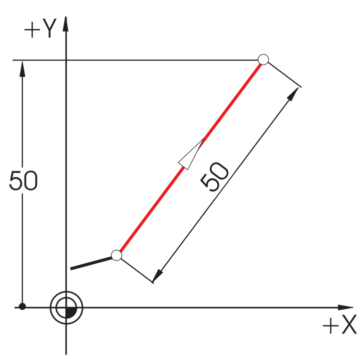
Angle 38.13° = angle to previous element

or

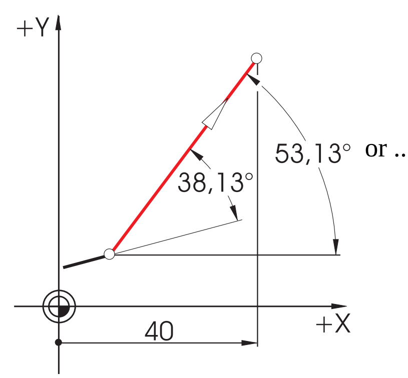
Angle 53.13° = start angle at positive X axis

You can combine Cartesian and polar inputs, e.g.:

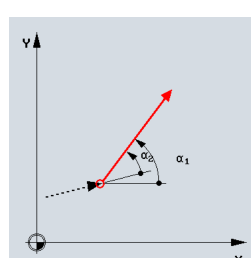
Input of the end point in Y and the length

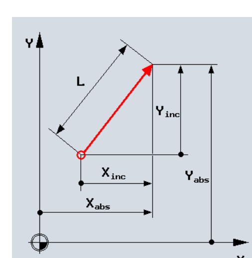


Input of the end point in X and an angle



The context-related ShopMill help displays can be called during entry of the values, and show the designations of the relevant input fields.





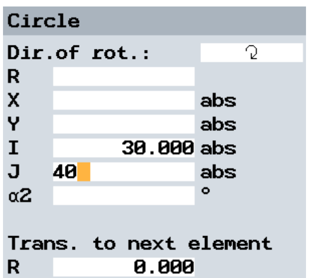
3.1.5 Circular movements

X and Y define the end point for the circular arc; the center point is entered with I and J. In ShopMill, you can enter these 4 values individually, either as **absolute** or **incremental values**.

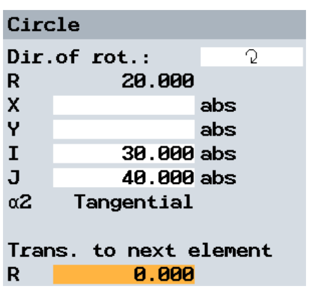
Whereas X and Y are entered as absolute, the center point I and J are entered as incremental for most controls. Here, it is essential not only to determine the difference from the starting point **A** to the center point **M** (often in combination with mathematical computation), but also the direction and thus the sign.

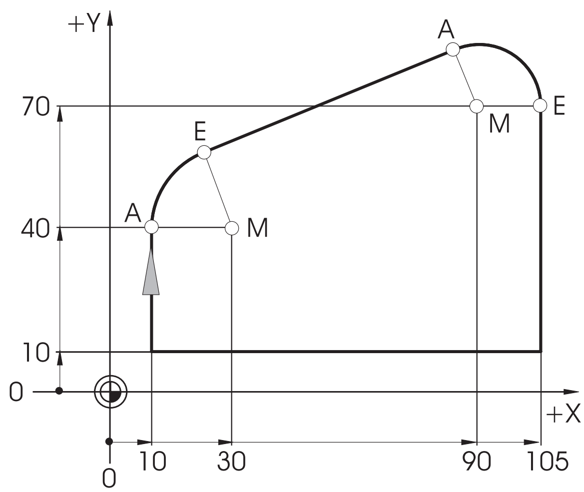
With ShopMill on the other hand, you do not have to perform any calculation because you can enter the absolute center point; you can use the contour calculator to determine even the most complex contours graphically.

Entering the center point (absolute):

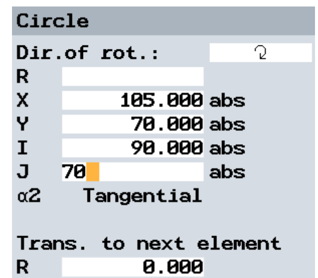


After input:

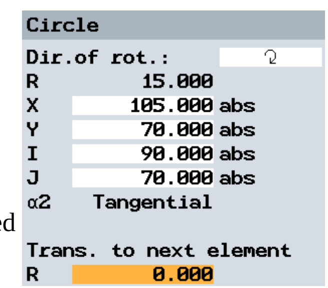




Values (in this case radii) that result from data already entered are computed automatically by ShopMill.

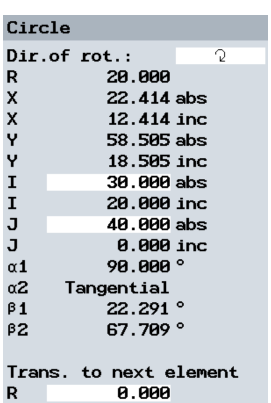


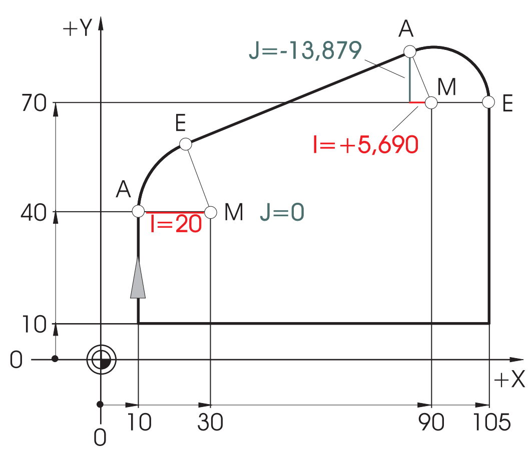
After input:

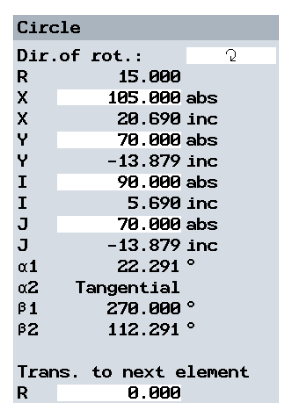


ShopMill also enables you to display **all** possible geometry values:

Display of all parameters:







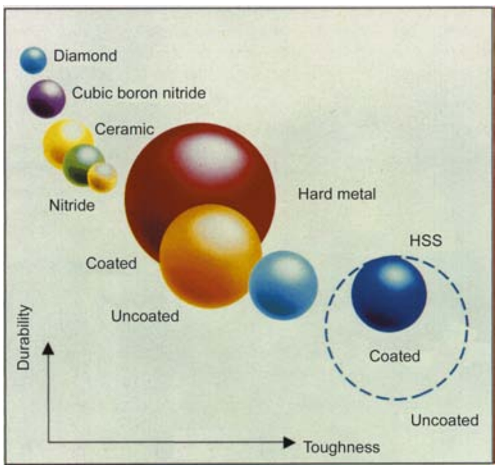
A further benefit of the absolute center point dimensioning: You do not have to recalculate the values for I and J when you reverse the milling direction.

3.2 Technology fundamentals

The basic requirements for optimized production are a sound knowledge of the tools (especially the cutting materials of the tools), the tool applications and the optimum cutting data.

3.2.1 Modern milling and drilling tools

Whereas HSS tool steels were dominant in the past, hard metals, ceramic plates, cubic bornitride (CBN) plates and polycrystalline diamond tools are used today. The following diagram shows the percentage distribution of the cutting materials and their properties, relative to their toughness and durability.



The diagram is taken from a SANDVIK tool catalog. The newly developed carbide materials which combine toughness and durability to produce high productivity values are also listed. Such cutting materials also bring the following benefits: longer tool life and better surface qualities.

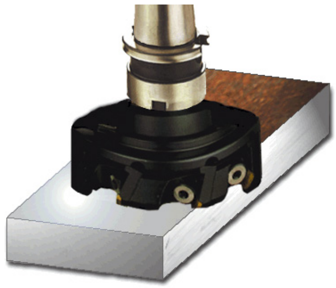
Non-coated tools made of HSS

Tools with sintered cutting plates

Titan nitride (TiN)-coated drilling and milling tools

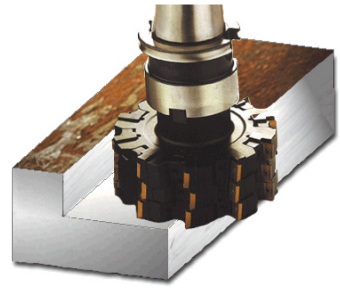
3.2.2 Tools used

Face mill



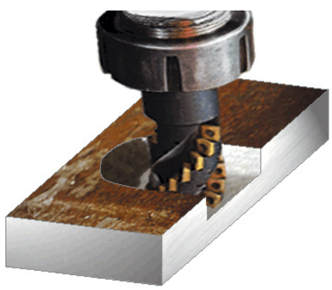
The face mill (also referred to as revolving blade) is used to remove large volumes.

Shell end mill



The shell end mill produces right-angled contour sections with vertical shoulders.

Shaft milling tool insert

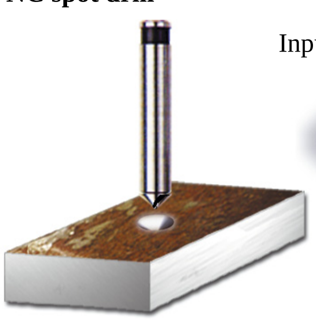


The shaft milling tool insert is a multi-cut tool, which uses a spiral-form arrangement of the cutters to produce an especially "smooth" machined result.

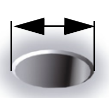
Long hole milling tool

The longitudinal hole mill (also referred to as a groove milling tool) cuts above the center and can also be inserted to the full depth. It generally has 2 or 3 cutting edges.

NC spot drill



Input value



NC spot drills are used for centering and to produce a chamfer for the subsequent drilling. ShopMill automatically calculates the depth when you specify the outside diameter of the chamfer.

Spiral drill

With ShopMill, you can choose between various types of drill (chip breakage, deep-hole drilling, etc.). The drill tip 1/3D is automatically taken into account in ShopMill.

Drill

Full drills are equipped with tool inserts and are only available for drills with a large diameter. The drilling process must always be made without interruption.

3.2.3 Cutting velocity and speeds

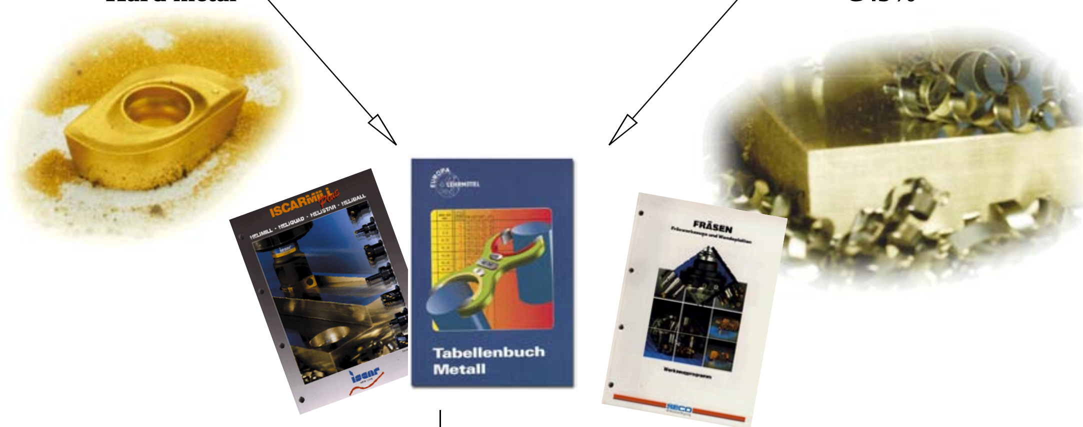
The optimum speed of the tool in each case depends on the cutter material and the workpiece material, as well as the workpiece diameter. You can often enter this speed on the basis of year-long experience, without calculation. However, it is better to calculate the speed from the cutting velocity given in the tables.

Determining the cutting velocity:

The manufacturer’s catalog or a book of tables helps you to determine the optimum cutting velocity initially.

Material of the **tool**: **Hard metal**

Material of the **workpiece**: **C45%**



v_c = 80 - 150 m/min:

The mean value $\mathbf{v_c}$ **= 115 m/min is selected**

This cutting velocity and the known tool diameter is used to compute the speed **n**.

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

In the example below, the speed is computed for two tools:

d_1 = 40mm **d_2 = 63mm**

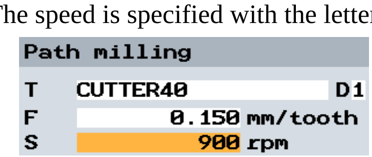
$$n_1 = \frac{115mm \cdot 1000}{40mm \cdot \pi \cdot min}$$

$$n_2 = \frac{115mm \cdot 1000}{63mm \cdot \pi \cdot min}$$

$$n_1 \approx 900\frac{1}{min}$$

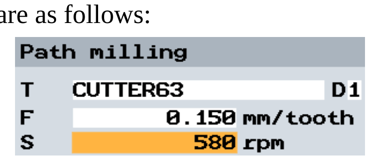
$$n_2 \approx 580\frac{1}{min}$$

The speed is specified with the letter **S** (for speed) in the NC coding. So the inputs are as follows:



S900

S580



3.2.4 Feed per tooth and feedrates

On the previous page, you learned how to calculate the cutting velocity and the speed. For the tool to start cutting, this cutting velocity or speed must be assigned a tool feed rate.

The basic value for computing the feedrate is the feedrate per tooth. Like the cutting velocity, the value for the feedrate per tooth is taken from the book of tables, the manufacturer documentation or from experience.

Determining the feedrate per tooth:

Tool **cutter**:

Hard metal

Workpiece **material**:

C45

f_z = 0,1 - 0,2 mm:

Select the average value $\mathbf{f_z = 0.15}$ **mm**

The feedrate per tooth, the number of teeth and the known speed is used to compute the feedrate $\mathbf{v_f}$.

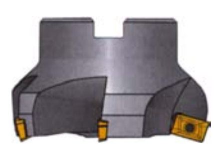
$$v_f = f_z \cdot z \cdot n$$

The feedrates for two tools with different numbers of teeth are computed in the example:

d_1 = 63mm, $\mathbf{z_1 = 4}$

d_2 = 63mm, $\mathbf{z_2 = 9}$

$$v_{f1} = 580\frac{1}{min} \cdot 0,15mm \cdot 4$$



$$v_{f2} = 580\frac{1}{min} \cdot 0,15mm \cdot 9$$

$$v_{f1} = 348\frac{mm}{min}$$

$$v_{f2} = 783\frac{mm}{min}$$

In the NC coding, the feedrate is specified as **F** for "feed". The entries are thus:

Path milling		
T	CUTTER63	D1
F	340.000 mm/min	
S	580 rpm	

F340

F780

Path milling		
T	CUTTER63	D1
F	780.000 mm/min	
S	580 rpm	

4 Well equipped

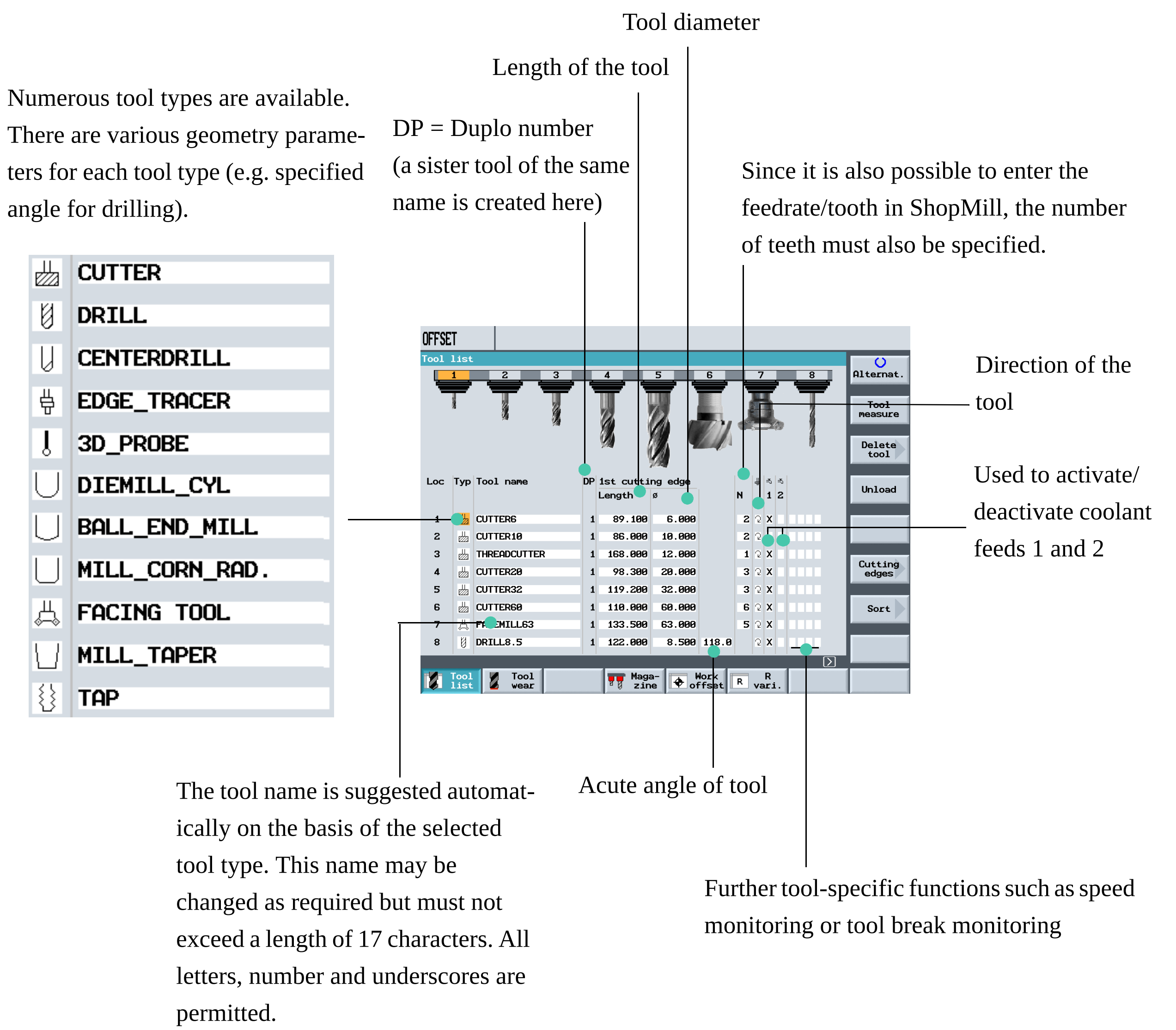
In this chapter, you learn how to create tools for the examples in the chapters that follow. An explanation is also given on how to compute typical workpiece lengths and how to set the workpiece zero.

4.1 Tool management

ShopMill offers three lists for tool management.

1. Tool list

All the tools and associated offset data in the NC are specified and displayed here, irrespective of whether the tools are assigned to a magazine location.

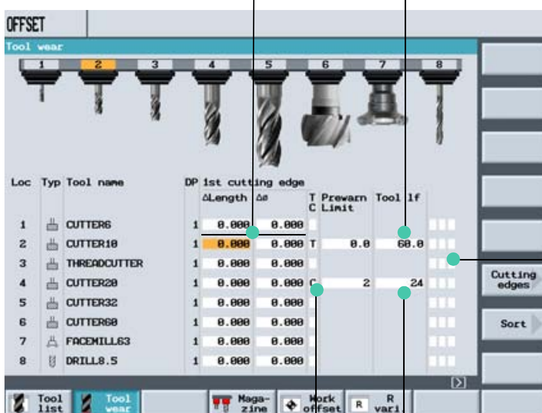


2. Tool wear list

You define the tool wear data for the relevant tools here.

You enter the tool wear here, relative to the difference values for the tool length and the tool diameter.

You specify the life in minutes here, provided that you have activated the function (T) previously.



You can use this toggle fields to define the following properties:

1. Lock tool
2. Oversize tool
3. Tool to fixed location

You enter the number of tool changes here, provided that you have activated this function (C) previously.

You define the tool monitoring here, relative to the tool life or the number of tool changes. T monitors the tool life, C the number of tool changes.

3. Magazine list

The magazine list contains all the tools that are assigned to one or more tool magazine(s). This list shows the status of each tool. Magazine positions can also be reserved or locked for particular tools.

The current tool status is shown here.

The location lock is activated here.

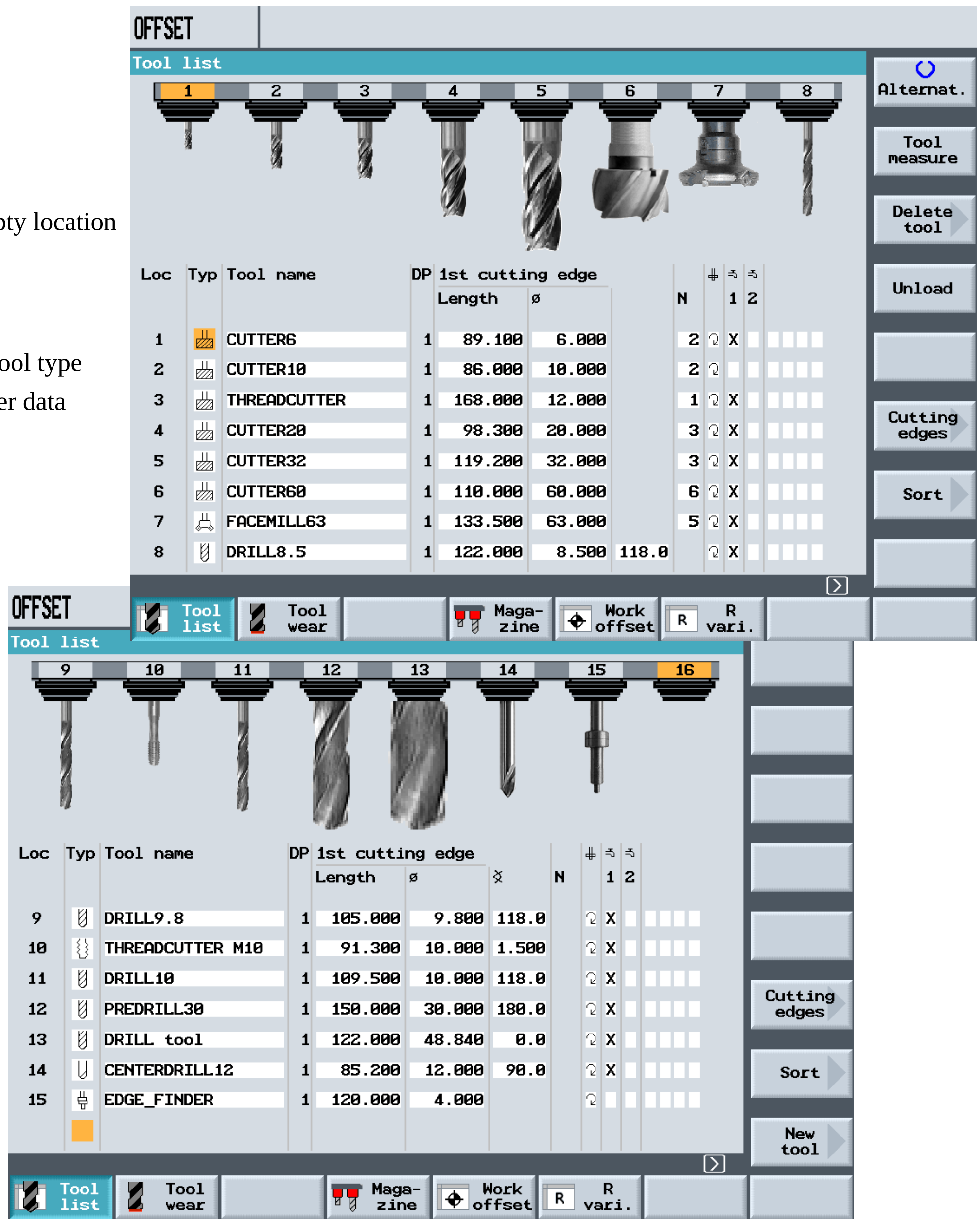
4.2 Tools used

In this chapter, you enter the tools required later for machining in the examples in the tool list.

Create tool

... find empty location

Select tool type and enter data



Note: The milling tools with diameters 6, 10, 20 and 32 must be capable of being inserted because they are also used to mill pockets in the following examples.

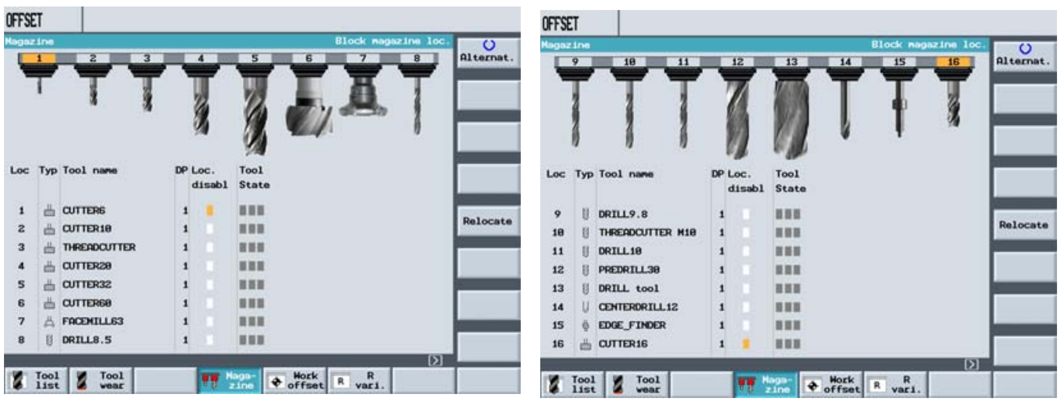
4.3 Tools in the magazine

In the following sections, you learn how to insert tools in the magazine.

Select a tool from the tool list without location number and press the key **Load** .

The following dialog offers the first free magazine location which you may change or accept as offered.

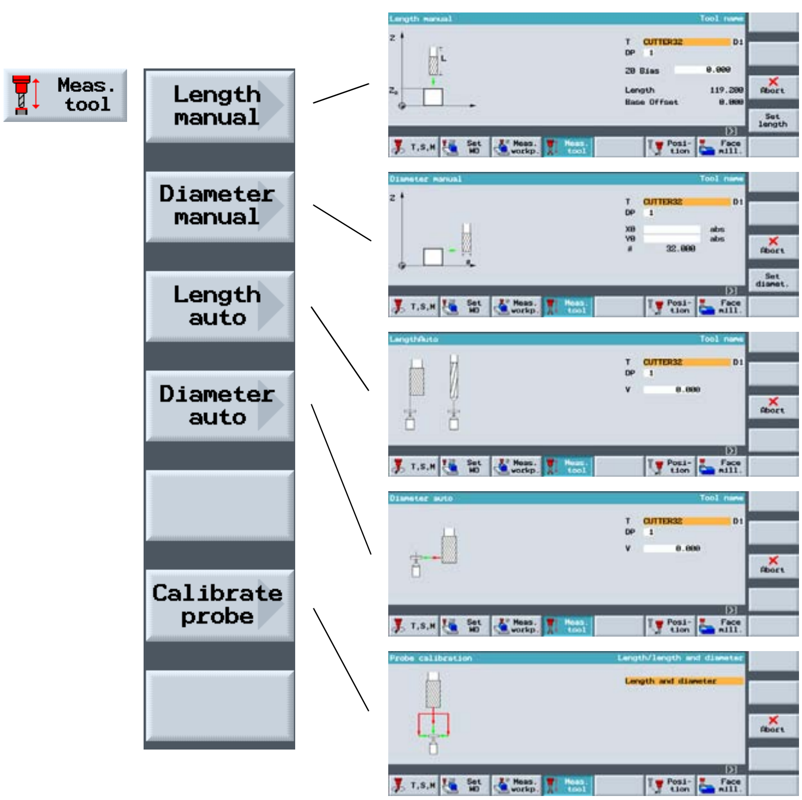
The magazine for the following exercises could look like this.



4.4 Measuring tools

In the following, you will learn how the tools are calculated

Load a tool into the spindle using softkey **T,S,M** . Change to the menu **Meas. tool** .



The tool is measured in the Z direction using the function *Length manual*.

The tool diameter is measured using the function *Diameter manual*.

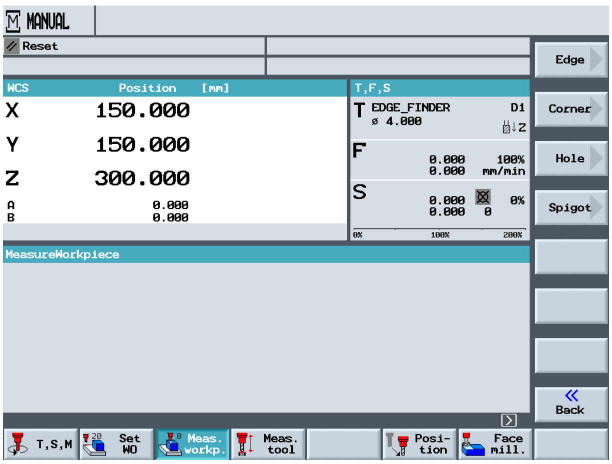
The tool is measured in the Z direction by means of a tool gauging device using the function *Length auto*.

The diameter of the tool is measured by means of a tool gauging device using the function *Diam. auto*.

The tool length and tool diameter are measured automatically using the function *Calibrate Probe*.

4.5 Set the workpiece zero

To set the workpiece zero, you must switch to the *Manual machine* mode in the basic menu.



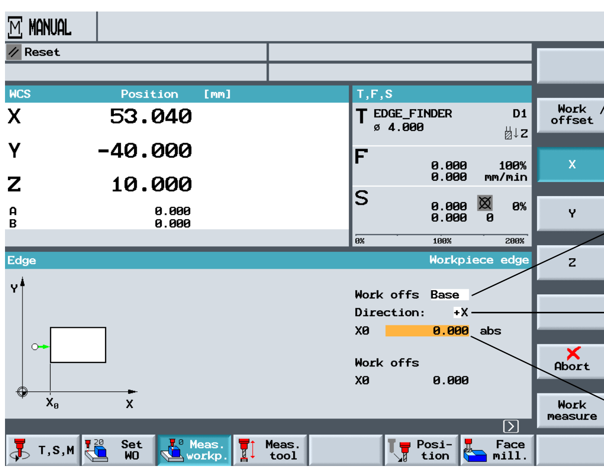
The option *Meas. workp.* in the submenu provides several options for setting the workpiece zero.

The example shows how to set the zero point of a workpiece edge (Edge) with an edge probe.

This key calls the list of zero offsets, which can then be set in the *Zero offset* field.

Procedure:

1. Edge



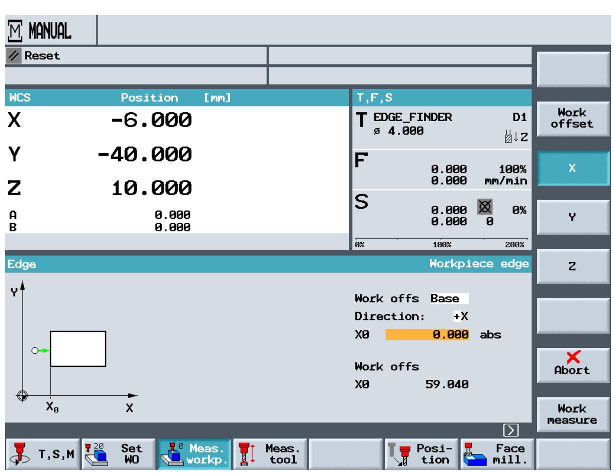
Enter a zero offset

Clicking direction left (+) or right (-)

Shift the workpiece zero offset if it is not to lie at the edge of the workpiece

2. Select the edge X (the help display shows the necessary clicking direction).

3. Click the workpiece edge

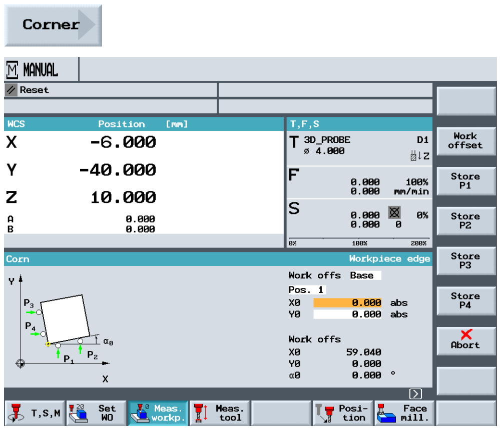


4. Work measure

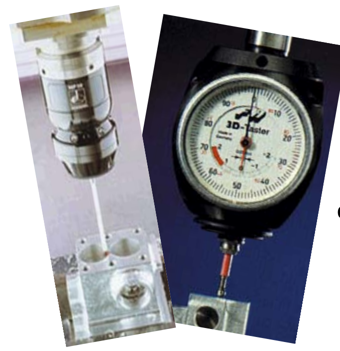
The workpiece zero is set, taking account of the edge probe diameter (4 mm).

This procedure must now be repeated for Y with the edge probe and for Z (usually with the milling tool).

Since the workpieces to be machined are not always present in the form of a cuboid or cannot be clamped in straight, further computation options are available:

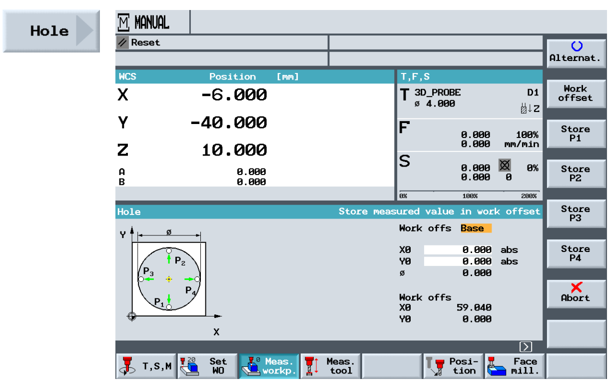


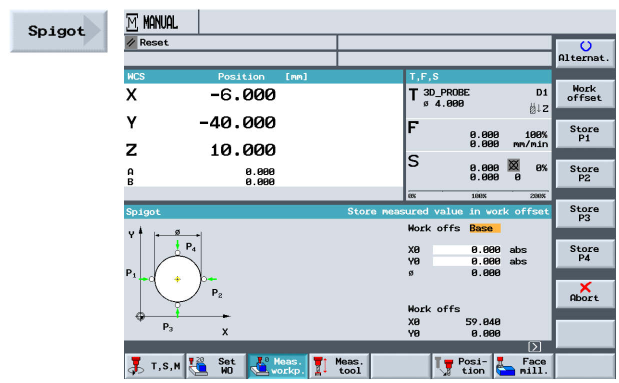
If such a workpiece position is the case, the workpiece position/ corner can be determined by approaching the four points.



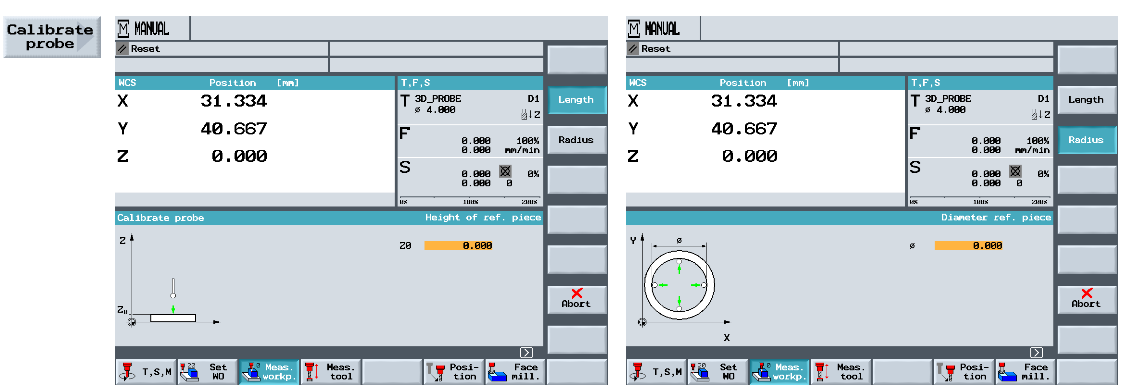
3D probes are available in electronic and mechanical designs. The signals of the electronic probe can be processed directly by the control.

Considering a hole or a spigot:





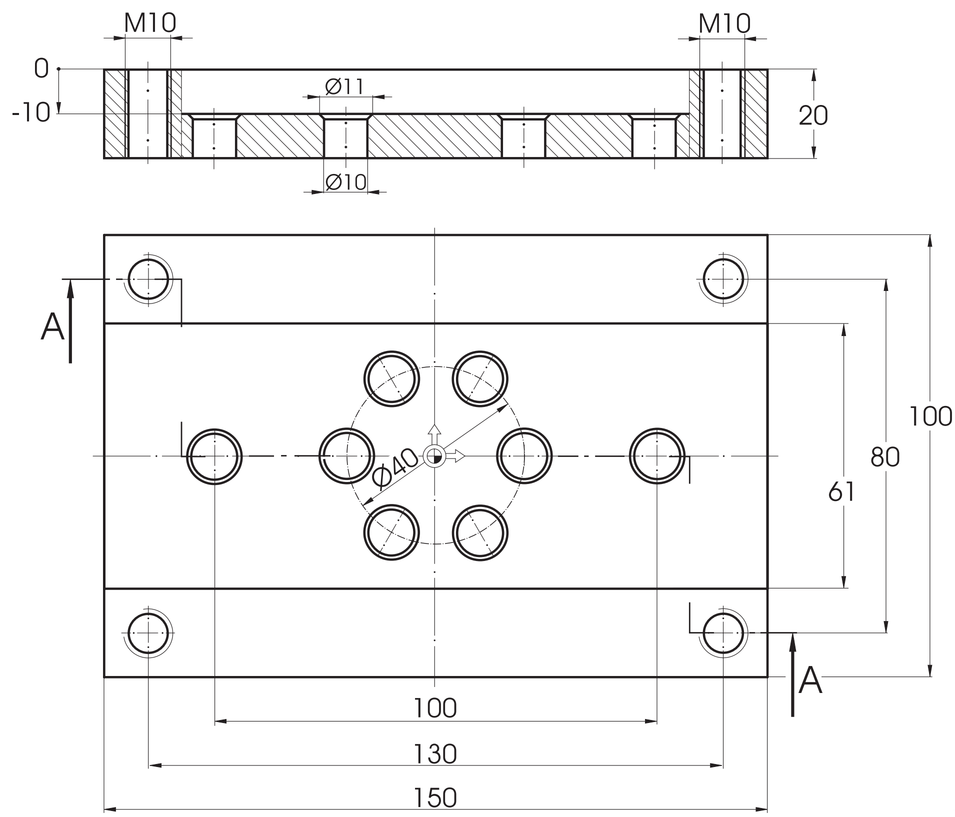
When you insert an electronic 3D probe from the tool magazine, clamping tolerances apply. These would falsify the results in further measurements. To prevent this happening, you can use the *Calibrate probe* cycle for the 3D probe on any reference surface or in any reference hole for calibration purposes.



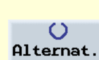
5 Example 1: Longitudinal guide

In this chapter, we will take a detailed look at the first steps required to create a workpiece:

- Program management and creating a program
- Calling the tool and chamfer radius offset
- Entering the traversing path
- Producing holes and position repetitions

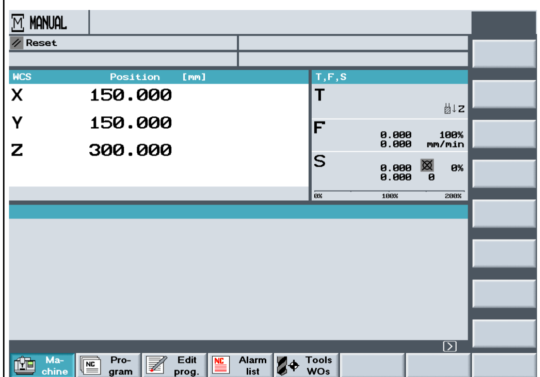
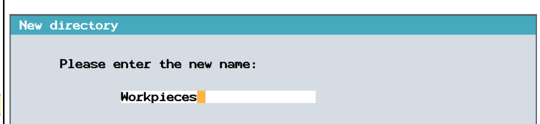
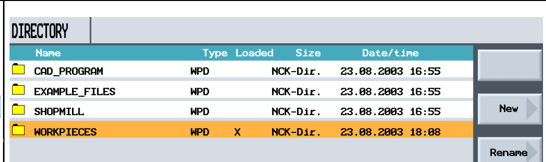
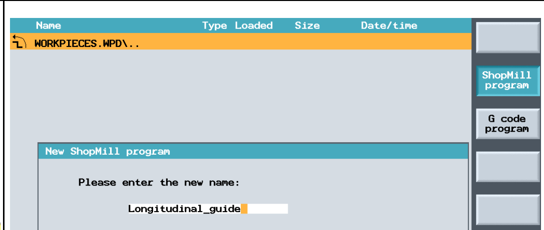
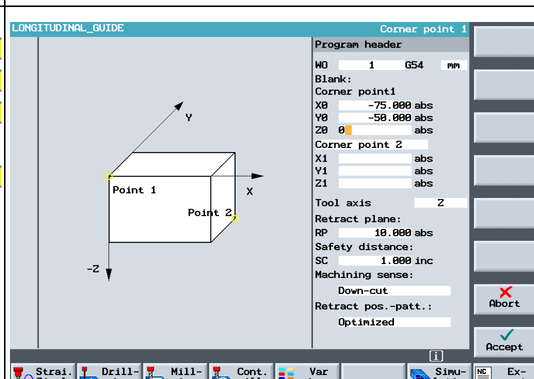


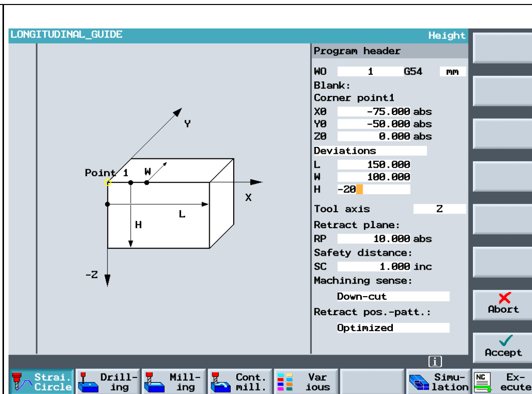
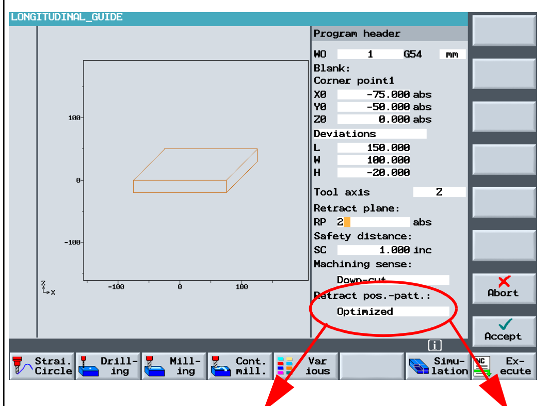
Note: Since ShopMill always saves the last setting set via the  key or the softkey 

, you must make sure that all the units, texts and symbols are set as displayed in the dialog boxes shown for all the examples both for numerous of the input fields and for all toggle fields.
The switchover option can always be identified by the 

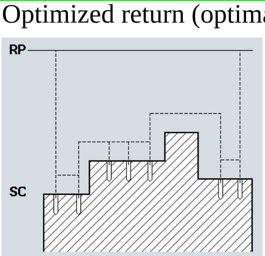
sofkey that is visible.

5.1 Program management and creating a program

Keys		Screen	Explanations
Pro-gram			• In the basic menu, you can call the various areas of ShopMill (see Chapter 2). • In the program manager a list of the available ShopMill directories is shown.
New	W...		• A new directory is created to save the work plans in the next chapter. It is given the name "Workpieces".
	...		• The work plan and contour management is organized in the program manager (e.g. *New*, *Open*, *Copy* ...). • You can use to move the cursor to the WORKPIECES directory and the key to open it.
New	L...		• The name of the work plan is entered here, in this case "Longitudinal guide". • You can use to accept the name. • The softkeys *ShopMill program* and *G code program* can also be used to select the input format.
	1 -75 -50 0		• The workpiece data and the general data about the program are entered in the program header. • Since the zero of the workpiece lies in the center of the workpiece surface, the coordinates of the left-hand workpiece corner have a negative value. • You can use the key to call the help displays at any time.

(i)	150 100 -20		• You can use the key to toggle between inputting the *Corner point 2* and *Deviations*. • The setting *Deviations* is selected here so that the dimensions of the unmachined part can be entered directly (when entering the height, you must pay attention to the sign). • With the key (i) switch back to on-line graphic.
Accept	2 2x		• You can also specify the *Retract plane*, the *Safety distance*, the *Machining sense* (synchronism or in opposite direction) and the *Return for position patterns* in the program header. • The position pattern can be set to *optimized* (= time-optimized travel paths) or *on return plane*. • The Accept key means that all values in the dialog window are accepted.

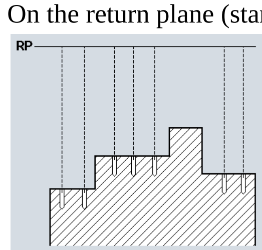
Optimized return (optimal)



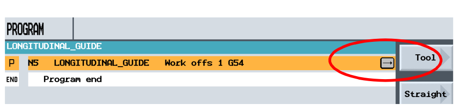
The tool returns over the workpiece at the safety clearance as appropriate to the contour.

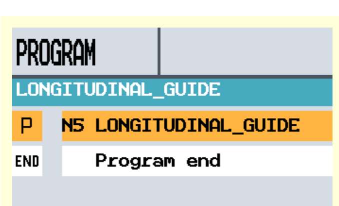
Help displays from ShopMill

On the return plane (standard)



The tool returns on the return plane and feeds at the new position.

			• The program header created is marked with the pictogram P. • You can use to re-call the program header to make a change, for example.

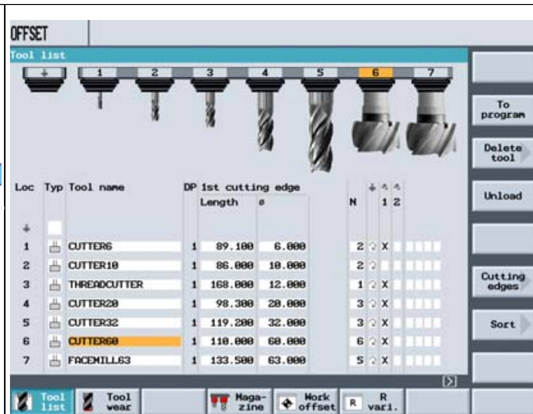
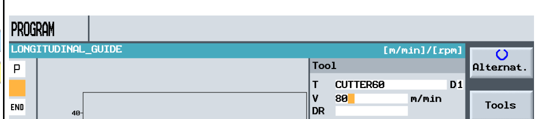
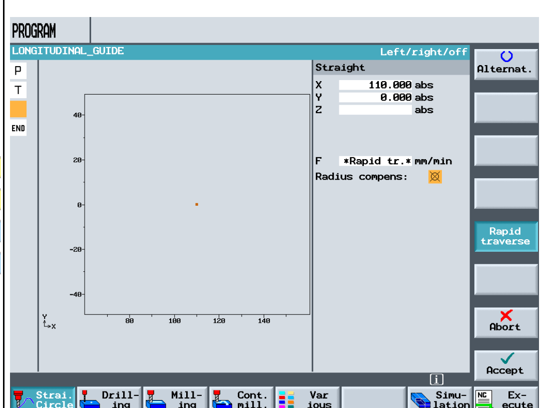


The program has now been created as the basis for further machining steps.

It has a name, a program header (abbreviated by the "P") and a program end (designated by the symbol "END").

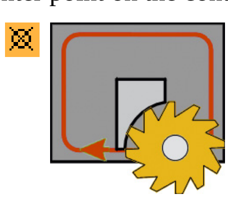
The relevant machining steps and contours are stored one below the other in the program. Processing later is carried out from top to bottom.

5.2 Calling the tool, cutter radius correction and travel path input

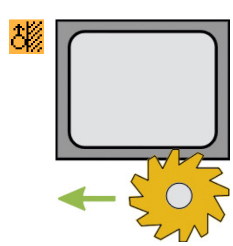
Tool Tools To program	...		• The size-60 milling tool is selected from the tool list and accepted. • You must press the key repeatedly until the red cursor is positioned at the relevant tool.
Accept	80		• When you have selected the tool you must change the cutting speed as necessary in the input field (80 m/min).
Straight Rapid traverse Accept	110 0 2x 3x		• The value X is 75 mm + 30 mm + clearance. • The radius offset is deactivated. Alternative settings in this field: - Previous offset setting (symbolizes an empty field) - To the left of the contour in the direction of milling - to the right of the contour in the direction of milling

Explanations for the topic radius offset:

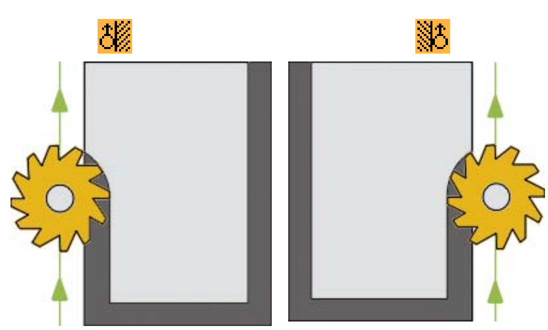
Just imagine that the milling tool were to approach the center point on the contour that has been created:



Tool **not corrected** = **scrap**

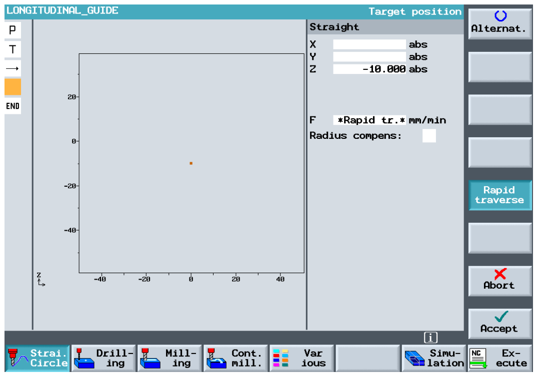
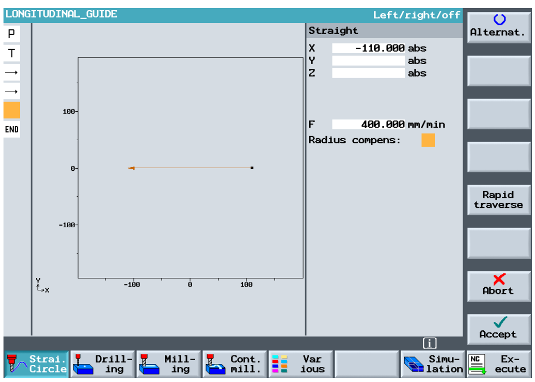
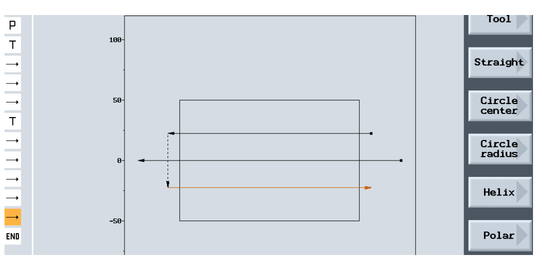
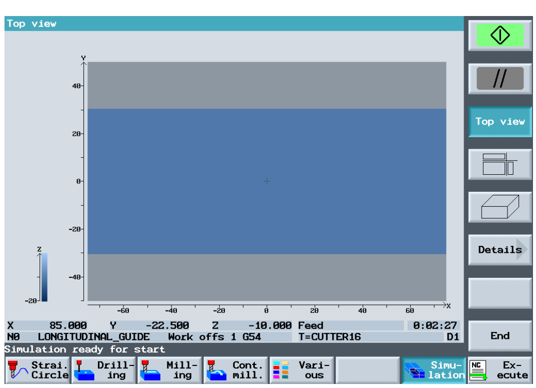


Tool **corrected**

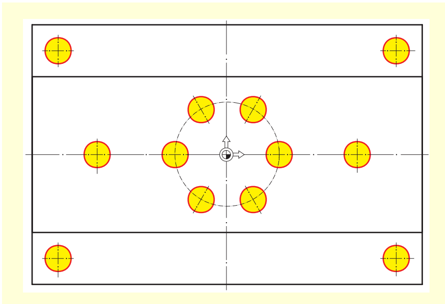


Tool **to the left of** the contour

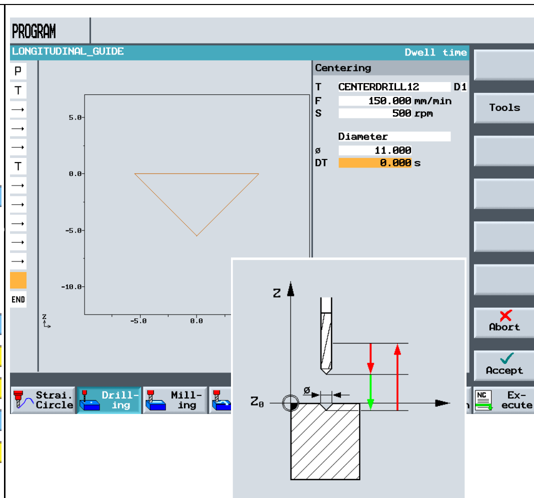
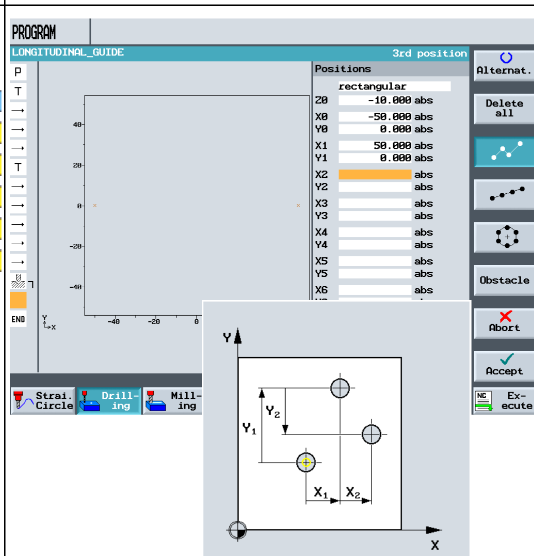
Tool **to the right** of the contour

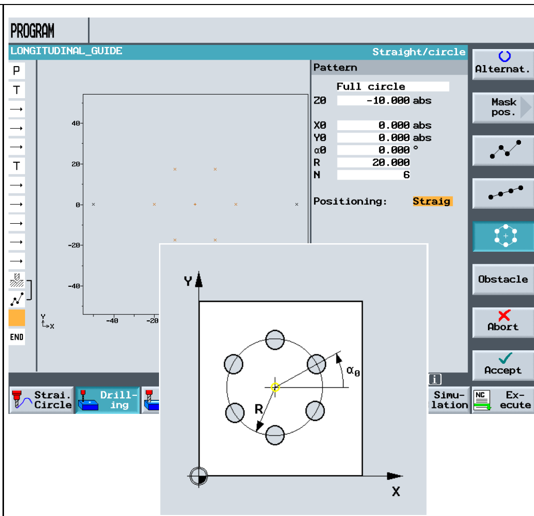
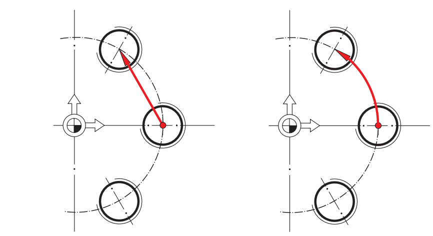
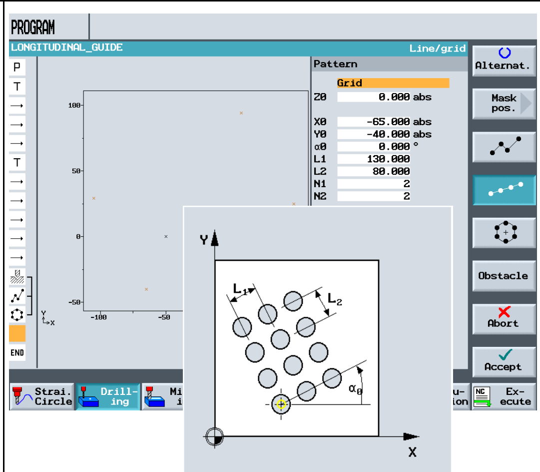
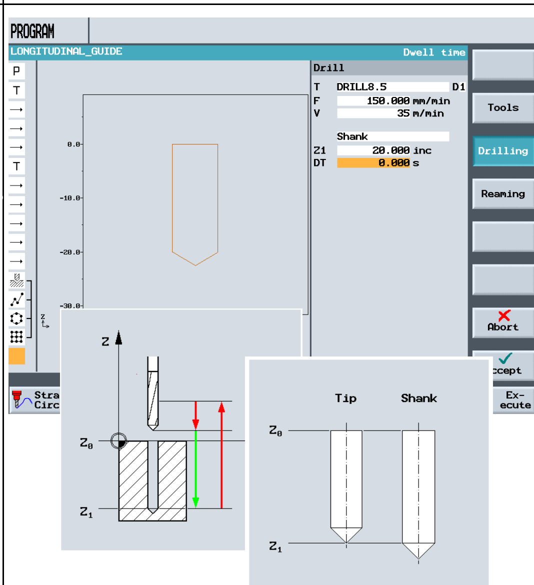
Straight Rapid traverse Accept	2x ↓ -10		• The tool is positioned in Z.
Straight Accept	-110 2x ↓ ↻ 400		• Enter the first machining path up to X -110 • At *F*, the system switches over to mm/min. • The workstep list looks like this after the dialog is accepted: T N10 T=CUTTER60 V80m → N15 RAPID ⌧ X110 Y0 → N20 RAPID Z-10 → N25 F400/min X-110
Tool •••	...		• Now change the next tool on your own (CUTTER16, V 100 m/min). • Then create the travel paths to be entered in the work plan below. T N30 T=CUTTER16 V100m → N35 RAPID X85 Y22.5 → N40 RAPID Z-10 → N45 F200/min X-85 → N50 RAPID Y-22.5 → N55 F200/min X85
Simu-lation Simu-lation			• The simulation is started with Simu-lation • In the following examples, the simulation can also be called when it is not shown explicitly. • Further information is given at the end of Chapter 7. • The simulation is terminated End

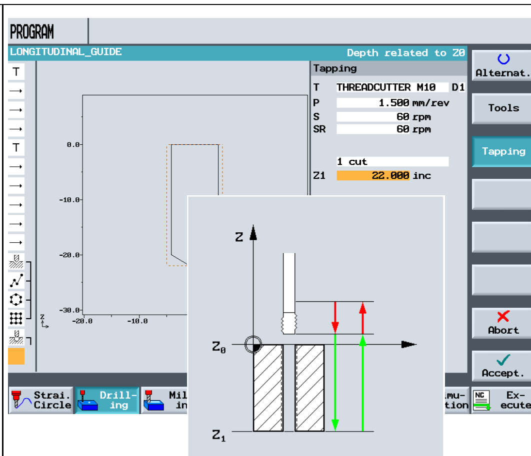
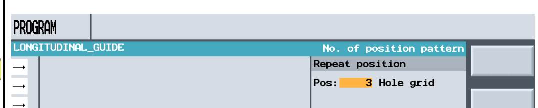
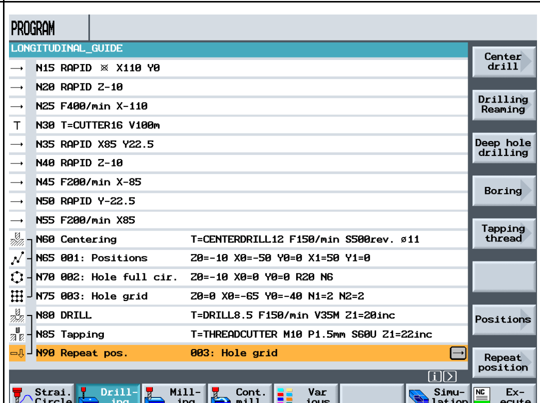
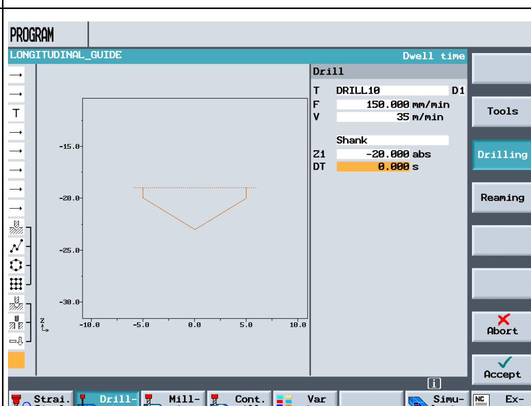
5.3 Creating holes and position repetitions



The following entries center the 12 holes, drill them and produce the thread.

Drilling Center drill Tools To program Accept	... 150 500 11		• The holes should be centered with the size-12 drill (F 150 mm/min and S 500 rpm). • Centering can be entered relative to the diameter or relative to the depth. Since holes have a 0.5 mm chamfer, it is all right to enter the diameter 11 here.
Positions Accept	-10 -50 0 50 0		• The *Positioning* option is used to enter the two single holes and link these to the previously specified cutting data. • The starting depth is -10 mm.

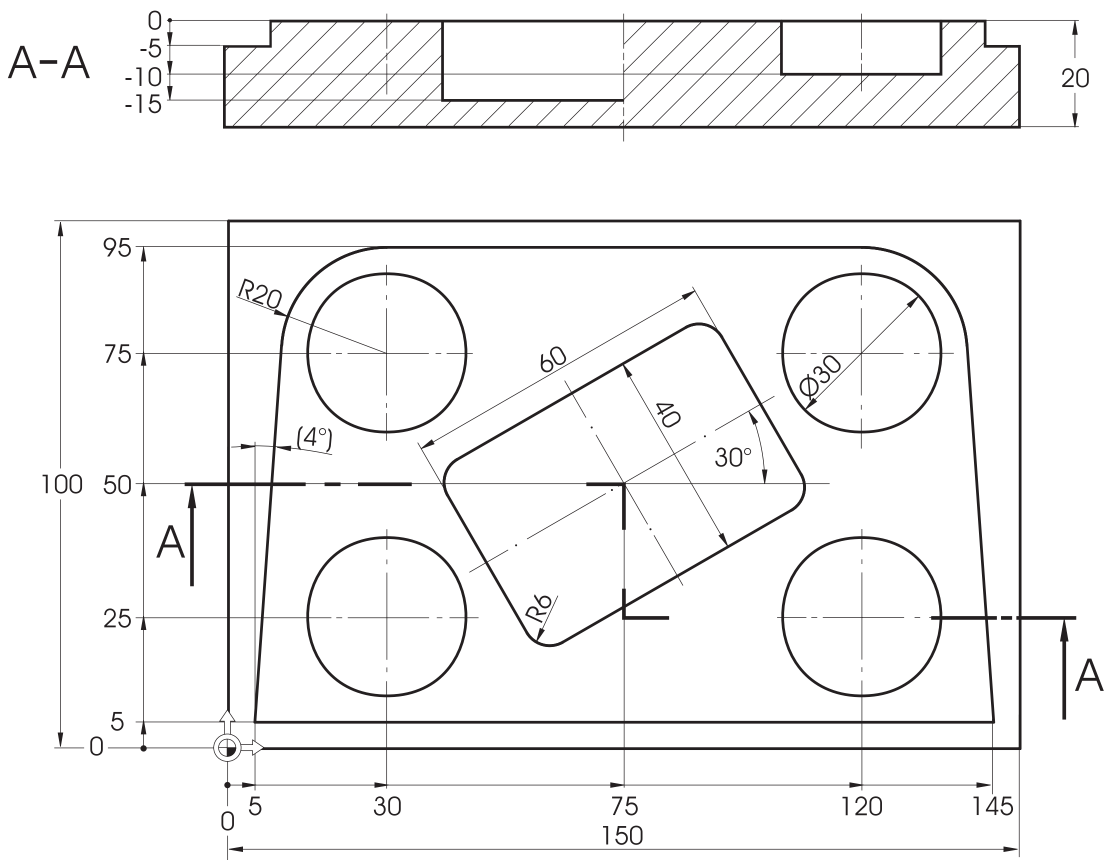
Positions Accept	-10 0 0 0 20 6		• The *Positioning* field specifies how the holes are to be approached within the drill image. If the holes lie in the circumferential groove, for example, the positioning *Straight line* must not be used since this would produce a contour violation. Positioning ... ... on a straight line ... on a circle 
Positions Accept	0 -65 -40 0 130 80 2 2		• The drilling positions are switched over from *Line* to *Grid*.
Drilling Reaming Tools To program Accept	... 150 35 20		• DRILL8.5 is used for drilling (F 150 mm/min and V 35 m/min). • The work steps *Centering*, *Drilling* and *Thread cutting* are interlinked automatically. • The depth is input incrementally here relative to the *shaft*, i.e.: The drill tip 1/3 D is considered automatically. • When entering the value, you must check whether the input field is set to *abs* or *inc*. • Drilling is carried out without *dwell time*.

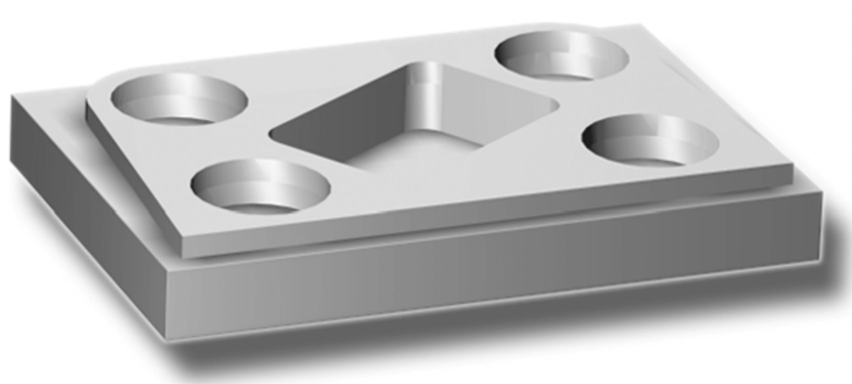
Tapping thread Tapping Tools To program Accept	... 1.5 60 60 22		• THREADCUTTER M10 is used to cut the thread (P 1.5 mm/rev. and S 60 rpm). • After calling the tool, you must enter the pitch, the speed and the depth of cut (incremental).
Repeat position Accept	3		• The drilling positions are numbered sequentially when created. The number is placed directly after the block number of the relevant position pattern (see N65-N75 in the figure below). It is then sufficient to specify this position, in our example, *Pos: 3 grid of holes*.
			• The very helpful chaining of work steps described previously then becomes very clear. N60 Centering N65 001: Positions N70 002: Hole full cir. N75 003: Hole grid N80 DRILL N85 Tapping N90 Repeat pos.
Drilling Reaming Tools To program Accept	... 150 35 -20		• The size-10 holes are produced using the DRILL10 tool. This is done by using a feedrate of F 150 mm/min and a cutting velocity of 35 m/min. • Set the depth reference to *shaft* to drill through. • The depth is entered as an absolute value.
		N95 DRILL T=DRILL10 F150/min V35M Z1=-20 N100 Repeat pos. 001: Positions N105 Repeat pos. 002: Hole full cir.	• Repeat the positions *001* and *002* for the size-10 drill last. • Call the simulation and check the result.

6 Example 2: Injection form

In this chapter, you learn the following new functions:

- Straight lines and circular paths via polar coordinates
- Rectangular pockets
- Circular pockets on a position pattern

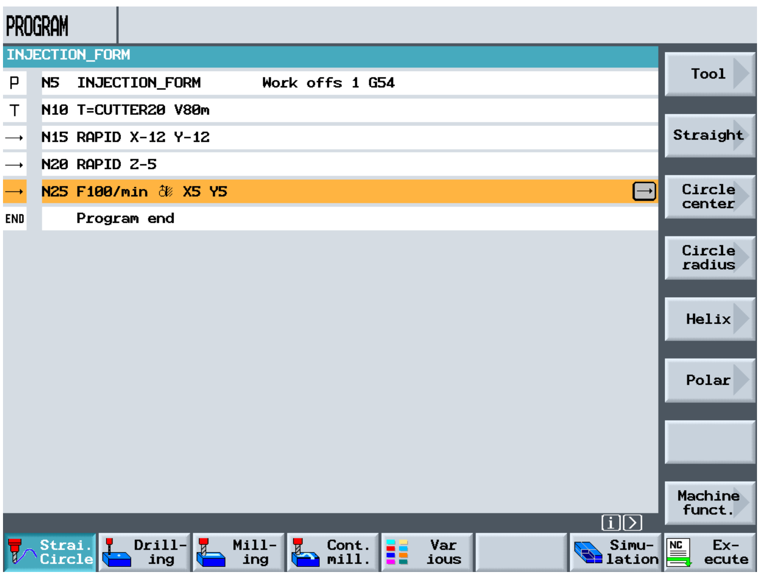




Creating a work plan and approaching the starting point

First create a new work plan with the name "Injection form" yourself. The dimensions of the unmachined part are entered simultaneously (cf. chapter "Longitudinal guide" for procedure). Note the new zero point.

Then change to the size-20 milling tool (V 80 m/min) and position it at point X-12/ Y-12/ Z-5 in rapid traverse. The starting point for X5 and Y5 is approached on a straight line (F 100 mm/min, cutter radius correction left).

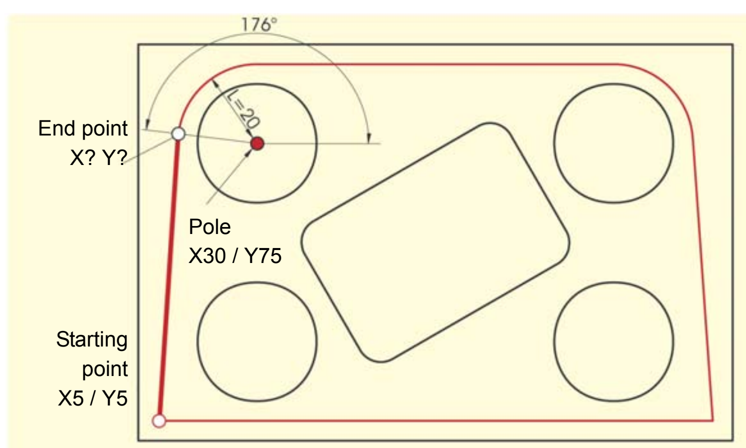


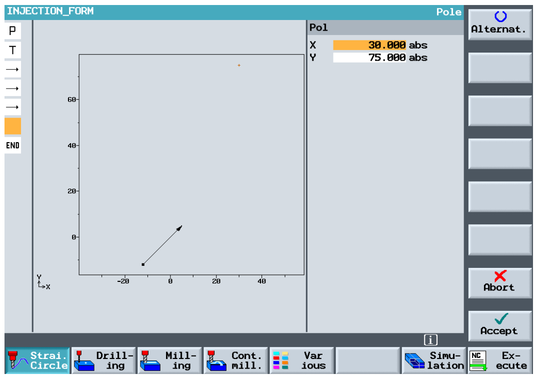
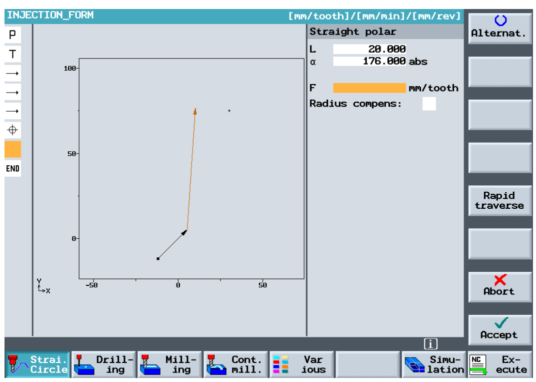
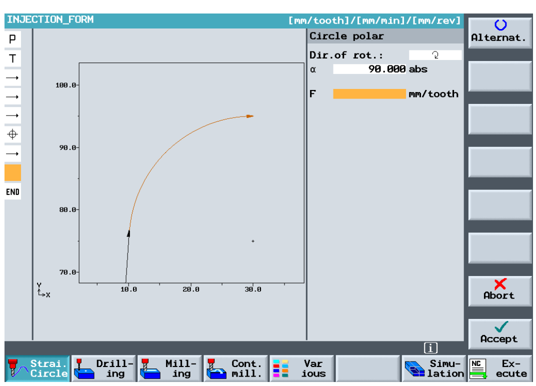
When you have entered the first traversing blocks, the work plan should look like this.

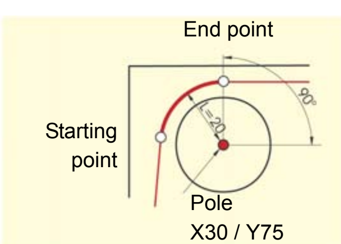
6.1 Straight lines and circular paths via polar coordinates

The end point of the traversing block can not only be described via its X and Y coordinates, but also via a polar reference point.

In this case, X and Y are unknown. You can also define the point indirectly: It lies 20 mm away from the center point of the circular pocket marked here behind the pole. The polar angle 176° results from the calculation 180° - 4° (see workshop drawing).

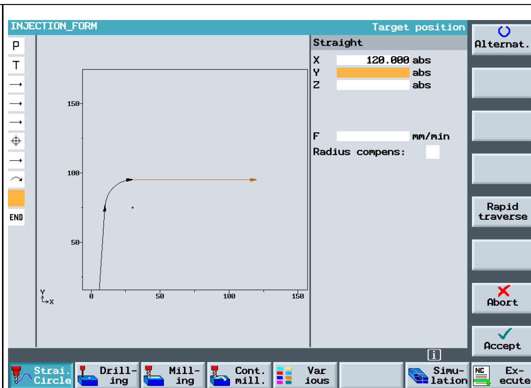
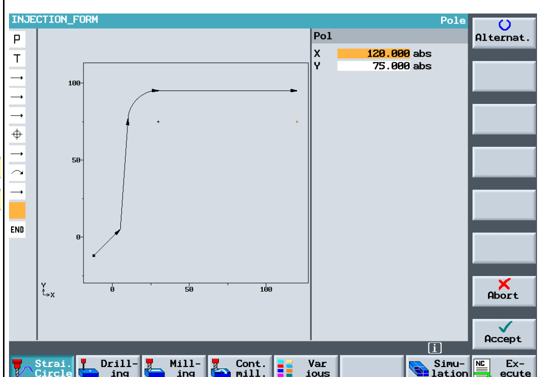
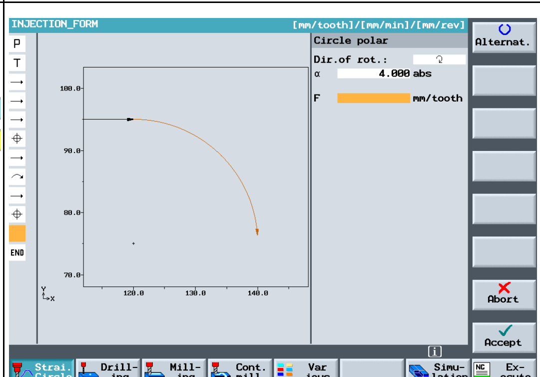
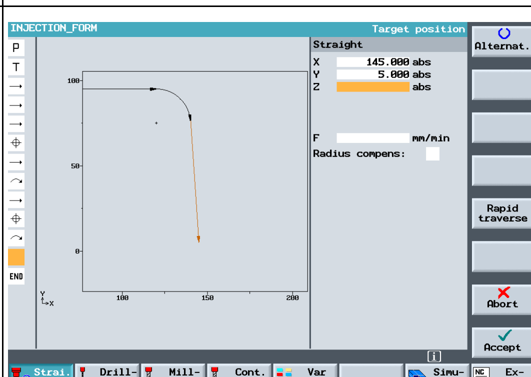


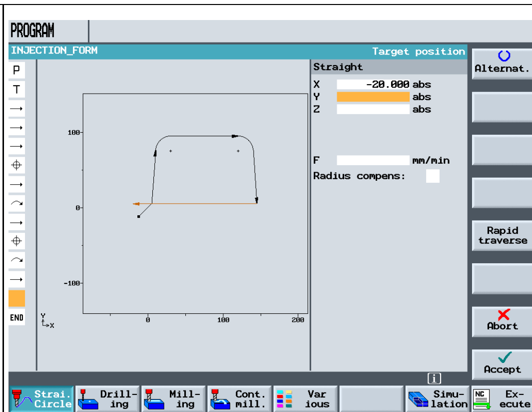
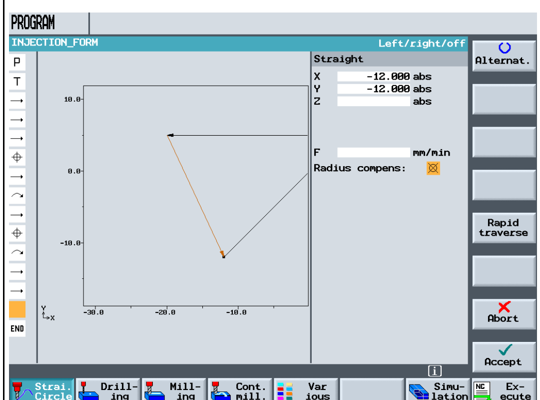
Keys		Screen	Explanations
Polar Polar Accept	30 75		• Inputting the poles
Straight polar Accept	20 176		• Length L defines the distance from the end point of the straight line from the pole. • The polar angle specifies how far the length L must be rotated around the pole to reach the end point of the straight line. • The polar angle can be rotated clockwise (176°) or counterclockwise (-184°).
Circle polar Accept	90		• A circular path can also be defined via polar coordinates.

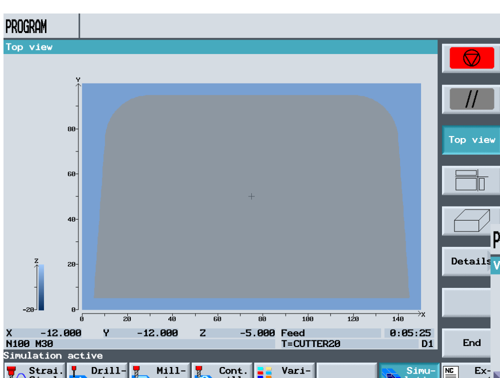


Since the pole applies both for the circular path and for the straight line, it need only be entered once.

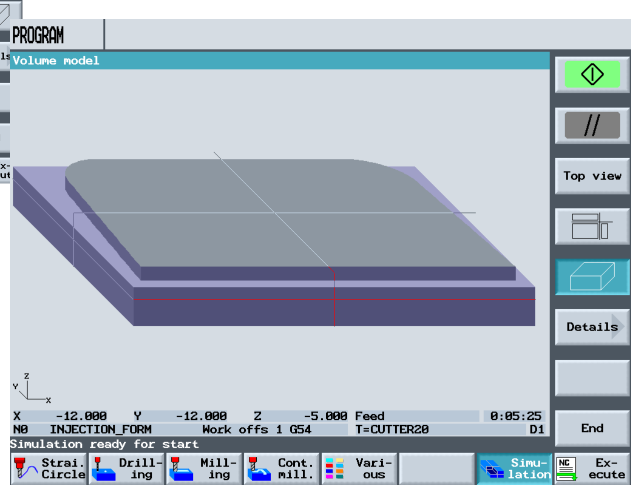
The polar angle is 90° in this case.

Back Straight Accept	120		• Since the end point of the straight line is uniquely identified, the *straight line* function can be used here.
Polar Polar Accept	120 75		• Since the end point of the next circular path is unknown, you must work with polar coordinates here again. • The pole of the circular path is known from the drawing.
Circle polar Accept	4		• The polar angle is also known on account of the symmetry.
Back Straight Accept	145 5		• The end point of the straight line is known and may therefore be entered directly.

Straight Accept	-20		• With the last straight line, the contour has been milled fully once.
Straight Accept	-12 -12 2x 3x		• The last traversing movement uses the specified safety clearance; the radius correction is deactivated in this case.



The following simulation shows the production sequence to let you check it before the workpiece is produced.

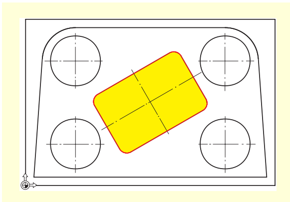


ShopMill also allows you to

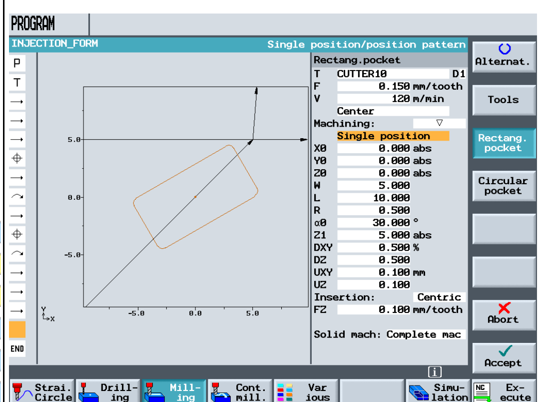
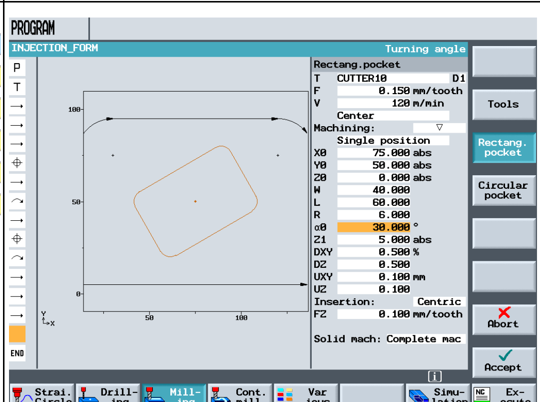
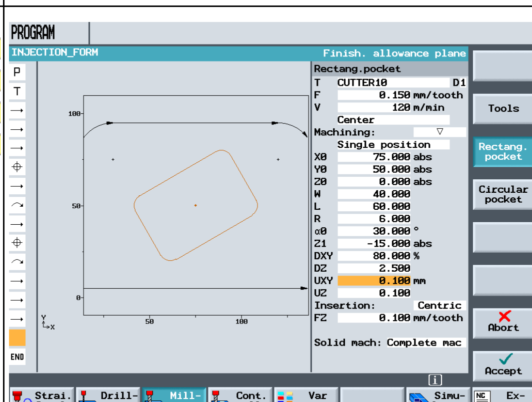
- cut,
- zoom,
- rotate the 3D view and
- view the workpiece in a three-side view.

Further information about these variations for the workpiece representation are given at the end of Chapter 7.

6.2 Rectangular pocket

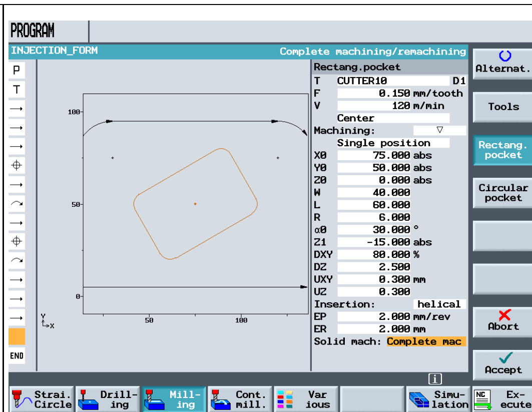


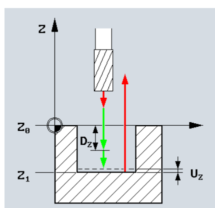
The rectangular pocket is created with the following inputs.

<table>
<tr><th colspan="2">Keys</th><th>Screen</th><th>Explanations</th></tr>
<tr><td>Mill-ing
Pocket
Tools
To program</td><td>...
0.15
120</td><td>
</td><td>• The tool for machining the pockets is the size-10 mill (F 0.15 mm/tooth and V 120 m/min).
• The pocket should be roughed first.
▽ - Roughing icon (coarse machining)
▽▽▽ - Finishing icon (fine machining)
Use the key to select the machining mode.
• Note that the switchover button is set to Single position.</td></tr>
<tr><td></td><td>75
50
0
40
60
6</td><td>
</td><td>• The geometrical data for the rectangular pocket are entered in these fields: Position, width and length, ...</td></tr>
<tr><td></td><td>30
-15
80
2.5</td><td>
</td><td>• The max. Infeed in the plane (DXY) indicates the width of stock removal for the material. This can be entered either as percentage of the cutter diameter or directly in mm (toggle with).
• The maximum infeed in the plane is specified in % here.</td></tr>
</table>

Accept

0.3
0.3
2
2

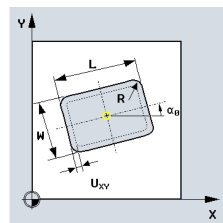




DZ =
Max. Infeed depth

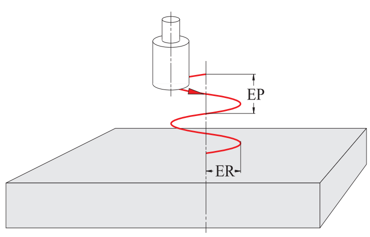
UZ =
Finishing allowance, depth

- Select helical insertion, if not already activated.
- If the pocket is already prefabricated, the *Solid mach:* field can be set to *Remaching*. Then enter the size of the pre-fabricated pocket in the input fields displayed. Roughing of this area is then omitted.



UXY =
Finishing allowance, plane

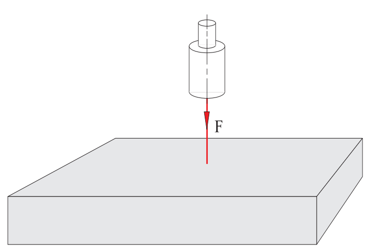
Helical insertion



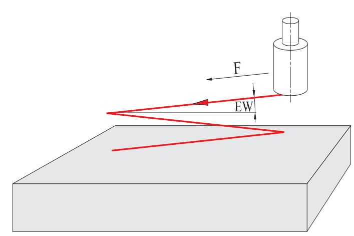
EP = Insertion pitch

ER = Insertion radius

Centered insertion



Oscillating insertion

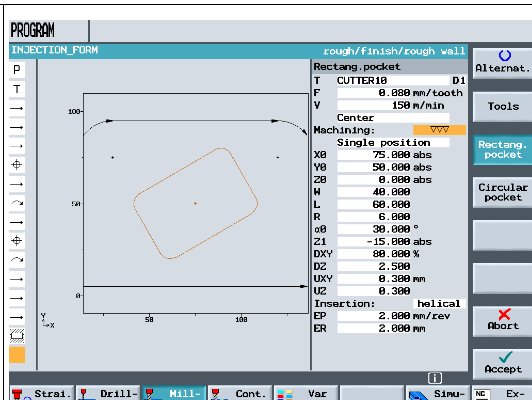


EW = Insertion angle

Pocket

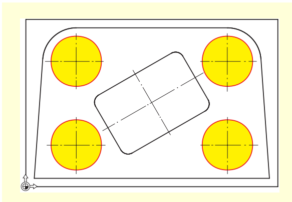
Accept

0.08
150

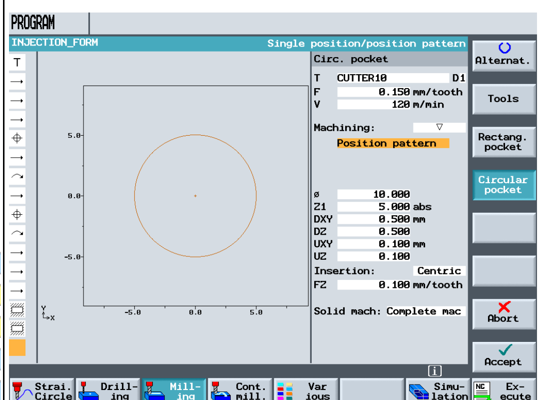
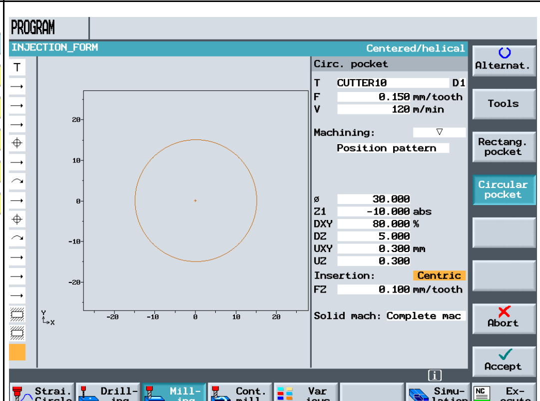
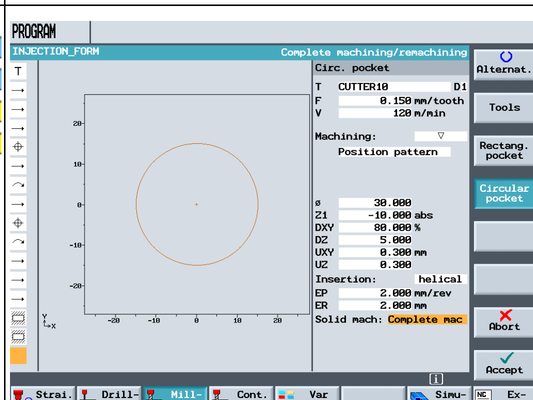


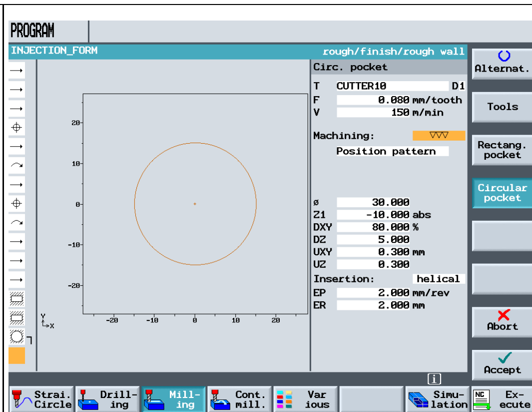
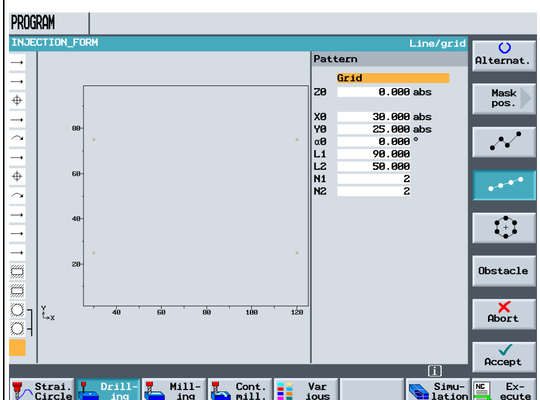
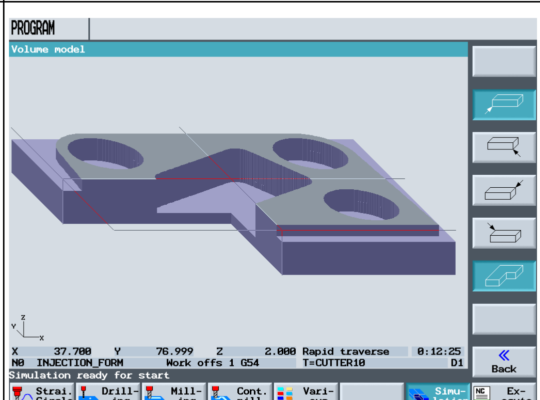
- The finishing work step is created next. Reduce the feedrate 0.08 mm/tooth, increase the cutting speed to 150 m/min and switch the machining mode from *Roughing* to *Finishing* (▽▽▽).
- With this setting, you finish edges and bottom. Alternatively, you may only finish edges (▽▽▽ wall) or chamfer the pocket (Chamfer).

6.3 Circular pockets on a position pattern



The following entries create the circular pockets.

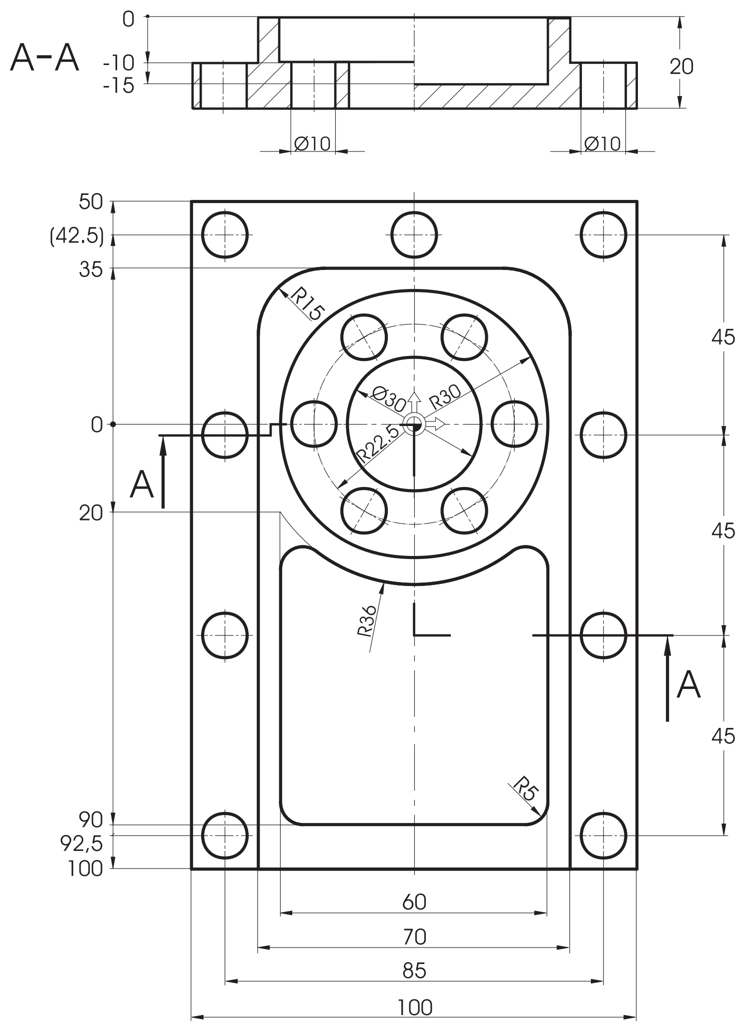
Keys		Screen	Explanations
Pocket Circular pocket Tools To program	0.15 120		• The size-10 milling tool (F 0.15 mm/tooth and V120 m/min) is used to machine the pockets. • Machining must be set to *Roughing*. • Analogous to drilling, you can create pockets of a position pattern. • In ShopMill, the last tool setting is stored. You must therefore switchover here if necessary.
	30 -10 80 5 0.3 0.3		• The maximum infeed in the plane is specified in % here.
Accept	2 2		• The insertion must be set to *helical* if required.

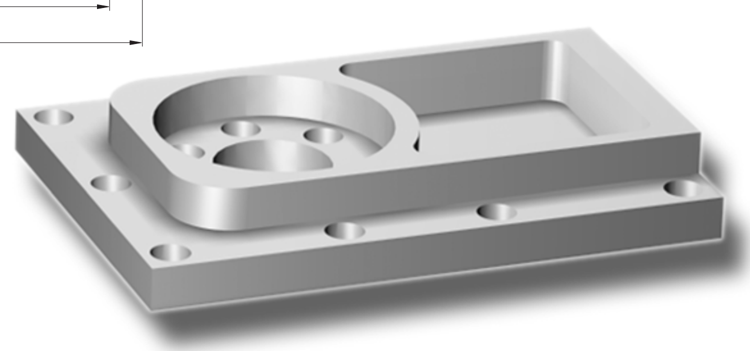
Pocket Circular pocket Accept	0.08 150		• The pockets should always be finished using the same milling tool (F 0.08 mm/tooth and V 150 m/min). • The machining must be set to *finishing*.
Drill-ing Positions Accept	2x 30 25 0 90 50 2 2		• Now you must enter the data to position the circular pocket. • The pattern type is set to *Grid*. • Note: The description of the position pattern is made in the *Drillng* menu with the submenu *Positioning* (irrespective of the type of machining).
Simu-lation ... Details ... Simu-lation			• Before [softkey] is pressed, the simulation must be fully completed in the top view or in the 3-plane view. • Before [softkey] is pressed, the desired cut must be set using the cursor keys. • The Refresh softkey can be used to display the new volume model during the course of the simulation and/or when the cutting path changes.

7 Example 3: Mold plate

In this chapter, you learn about other important functions, in particular the contour calculator:

- Path milling for open contours
- Stock removal, residual material and finishing contour pockets
- Machining on several planes
- Considering obstacles

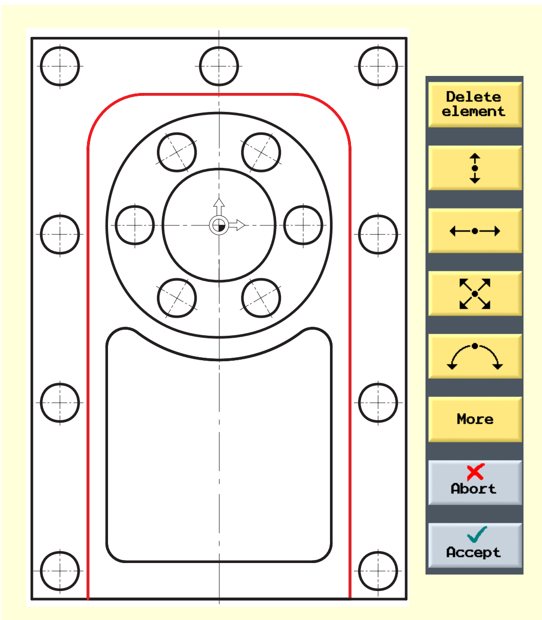




Creating a program

The workpiece dimensions must be taken from the drawing and entered in the program header of a new program. Observe the correct position of the zero point.

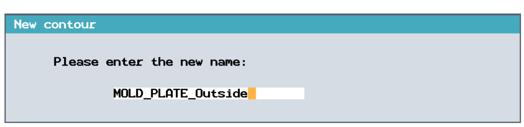
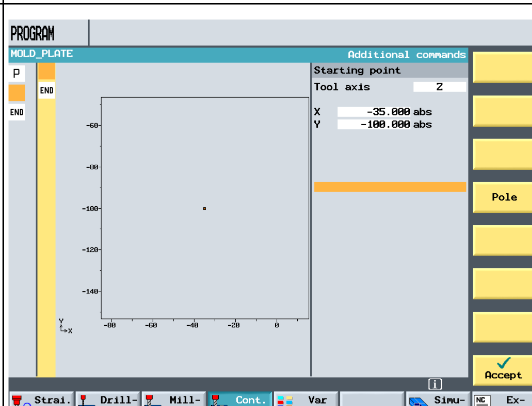
7.1 Path milling for open contours



To enter complex contours, ShopMill provides a contour calculator, which you can use to simplify the entry of highly complex contours.

- **Vertical route**
- **Horizontal route**
- **Diagonal route**
- **Arc**

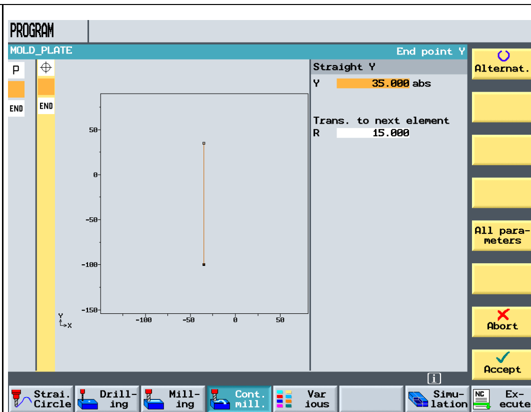
This graphic contour calculator lets you enter contours more easily and faster than with conventional programming - without the need of mathematics.

Keys		Screen	Explanations
Cont. mill. New contour	M...		• Each contour will get its own name. This makes reading the program easier.
Accept	-35 -100		• Enter the *Starting point* of the contour definition first. • The starting point of the structure is simultaneously the starting point for machining the contour later. • Note: You describe only the workpiece contour here, the approach and retraction paths are defined later.

↕

35
15

Accept

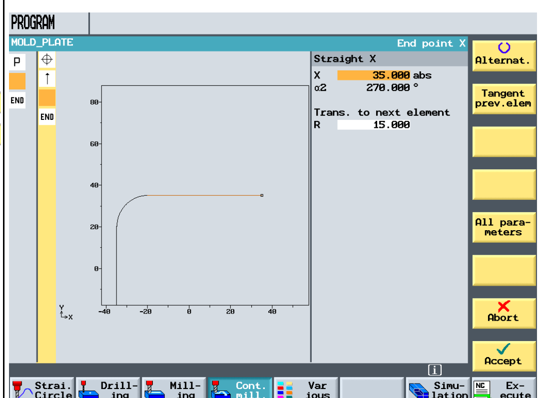


- The first contour element is a vertical line with end point at Y20.
  The following circular contour can be entered simply as a transition element up to the next straight line in this dialog. The theoretical end point of the straight lines therefore lies at Y35.
- With the *Alternative* key, you can also design a *chamfer* as a transition element.

←•→

35
15

Accept



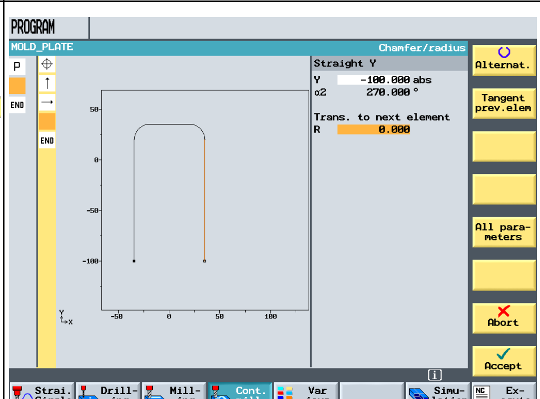
- Continue on the horizontal plane. The *Radius* is entered as a filleting.

↕

-100

Accept

Accept



- It is followed by a vertical path.
- The contour is thus fully defined and can be incorporated into the work plan.

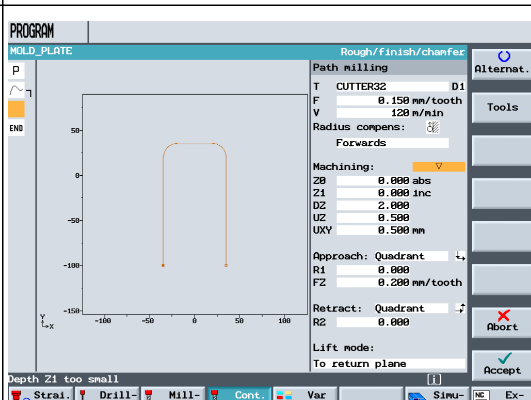
Path milling

Tools

To program

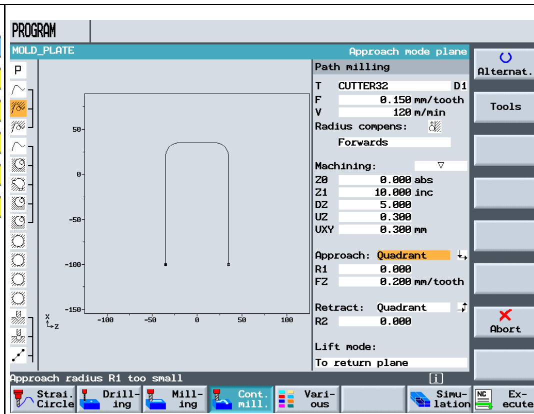
...

0.15
120



- To machine the contour you have created , you must now create the work step.
- The tool (CUTTER32) should move to the left of the contour. To do this, you must switch to in the *Radius comps.* input field.
- ShopMill V6.4 and higher also allows *backward* milling (against the constructional direction).
- The first machining step performs roughing (▽) .

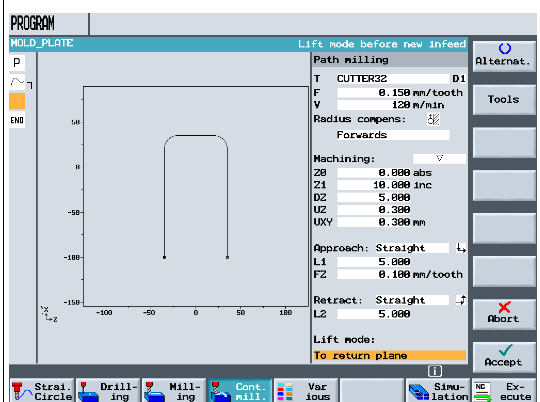
0
10
5
0.3
0.3



- In the fields that follow, enter the *start depth*, the *machining depth*, the *depth infeed* and the *final machining allowance*.
- Note: The *depth Z1* was switched to *inc*. This has the advantage that the actual depth of the pocket can always be entered without a sign. This simplifies the input of nested pockets.

Accept

3x
5
0.1
3x
5

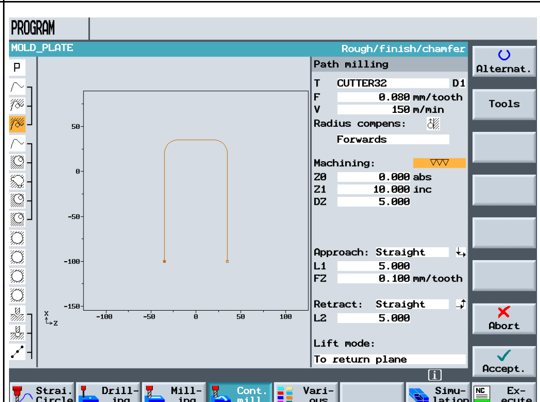


- You can approach in a *Quadrant*, a *Semi-circle*, *Vertical* or on a *Straight line*.
- It makes sense here to approach the contour at a tangent on a straight line.
- The mill radius does not have to be considered for the approach length *L1*. This is computed automatically by ShopMill.

Path milling

0.08
150

Accept

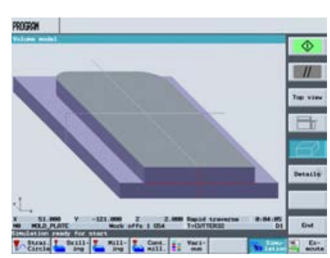


- The following work step is to be finished along the pre-roughed contour. This is done by reducing the feedrate to 0.08 mm/tooth, the cutting velocity to V 150 m/min increased and the machining to *finishing* (▽▽▽).

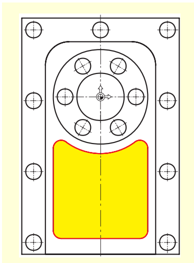
MOLD_PLATE		
P	N5 MOLD_PLATE	Work offs 1 G54
	N10 MOLD_PLATE_OUTSIDE	
	N15 Path milling ▽	T=CUTTER32 F0.15/t V120M Z0=0 Z1=10inc
	N20 Path milling ▽▽▽	T=CUTTER32 F0.08/t V150M Z0=0 Z1=10inc
END	Program end	

The two work steps are linked in the work plan.

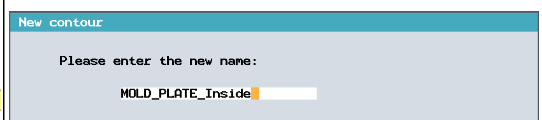
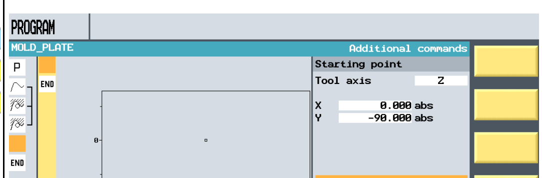
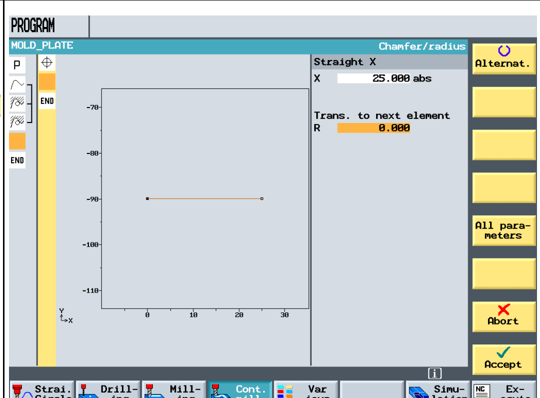
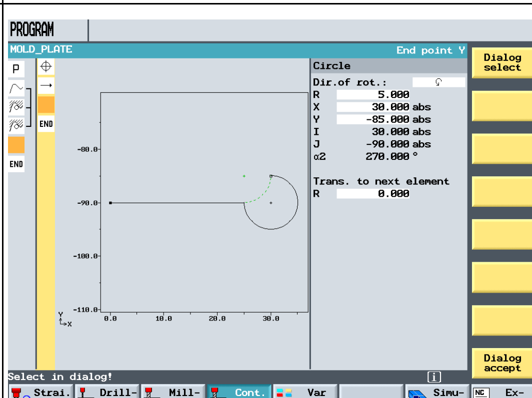
The simulation and subsequent 3D view show the correct production of the workpiece.

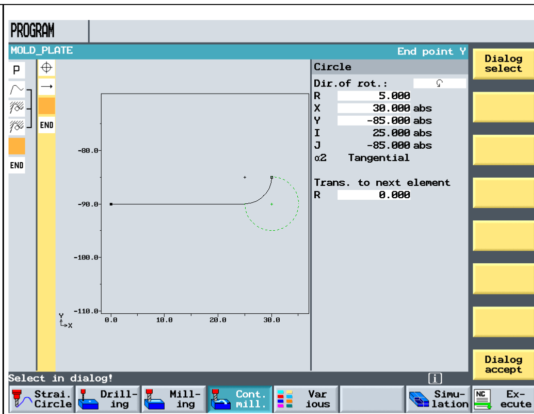
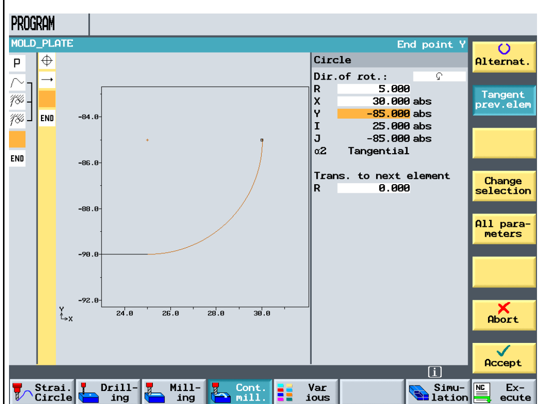
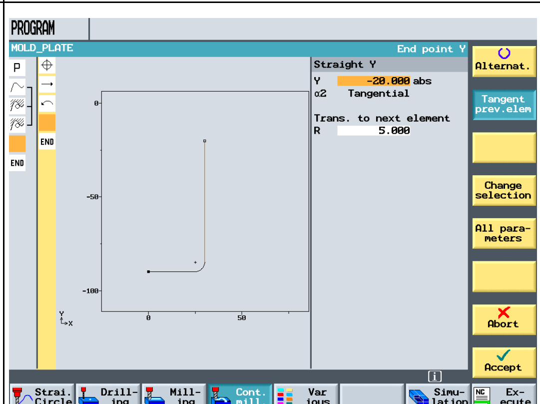
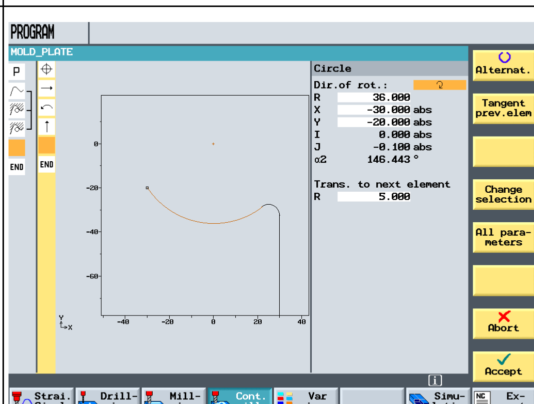


7.2 Stock removal, residual material and finishing of contour pockets



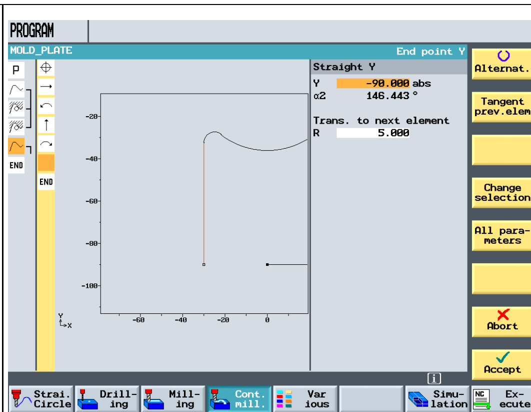
This contour pocket is created below. Then, the pocket is machined and finished.

Keys		Screen	Explanations
Cont. mill. New contour	M...		• The contour is assigned the name "MOLD_PLATE_Inside".
Accept	2x 0 -90		• The starting point should lie at X0 and Y-90.
←•→ Accept	25		• Because the pocket is to be machined in synchronism, the contour must be designed in the same direction. • As an exercise, the first arc should not be rounded but entered as a separate element. The straight line is therefore only designed up to X25.
Dialog select	5 30 -85		• When you enter the Y end point, you obtain two design solutions which can be called from the software via the softkey *Dialog select*. The solution selected turns black, the alternative green.

Dialog accept			• The *Dialog accept* softkey accepts the desired quadrant from the possible solutions.
Accept			• The geometry processor has automatically detected that the programmed arc connects tangential to the straight line. The corresponding softkey *Tangent prev. elem* is displayed in inverse mode (i.e. printed).
Accept	-20 5		• The end point of the of the straight lines is known. The transition to R36 is rounded with R5.
Accept	36 -30 -20 5		• A circular arc in clockwise direction follows.

Accept

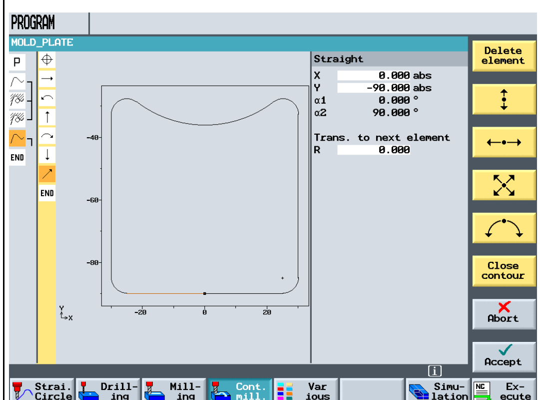
-90
5



• The radius R5 is specified as a filleting.

Close contour

Accept



• With the key *Close contour*, the contour is closed directly.

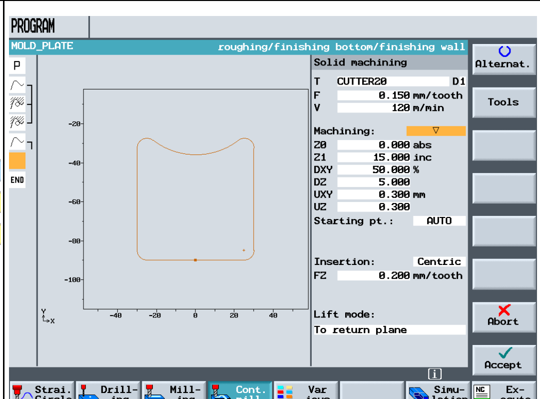
• The pocket contour is then fully defined and is incorporated in the work plan.

Solid Machining

Tools

To program

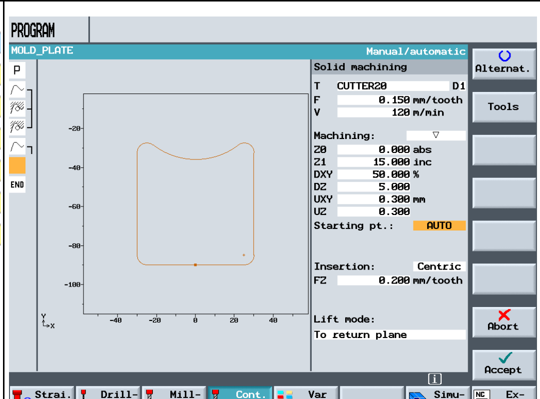
...
0.15
120



• The pocket is to be machined using the size-20 milling tool (F 0.15 mm/tooth and V 120 m/min).

• First of all, the pocket is roughed (▽).

0
15
50
5
0.3
0.3



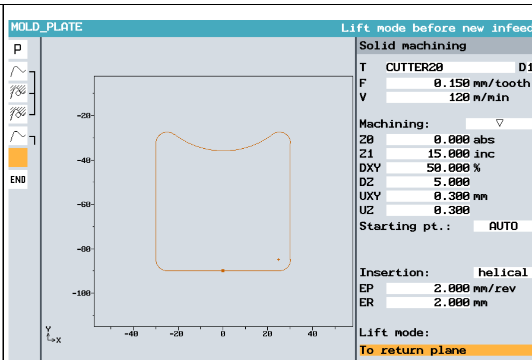
• The machining depth can also be entered as an incremental value. However, you must enter the depth as a positive value.

• The maximum infeed in the plane is specified in % here.

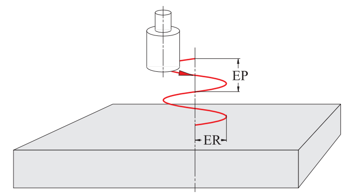
• The starting point (insertion position) is defined by ShopMill when *Auto* is selected.

Accept

2
2



- The *insertion* should be *helical* with a pitch and a radius of 2mm.



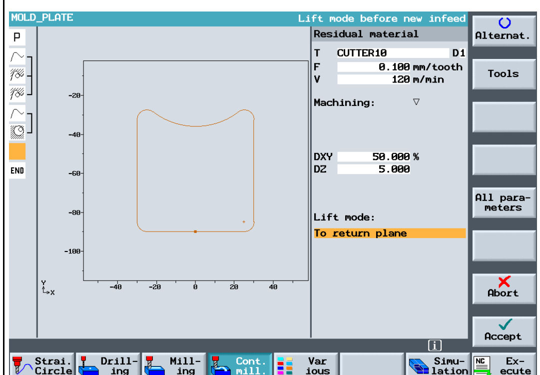
Residual material

Tools

To program

Accept

...
0.1
120
50
5



- Since the size-20 cutter cannot machine the R5 radius, the "corners" of the material are left over. The *Residual material* function and a smaller milling tool (CUTTER10 with F 0.1 mm/tooth and V120 m/min) are used to accurately rough off the areas that have not yet been machined.
- The maximum infeed in the plane should be 50%.

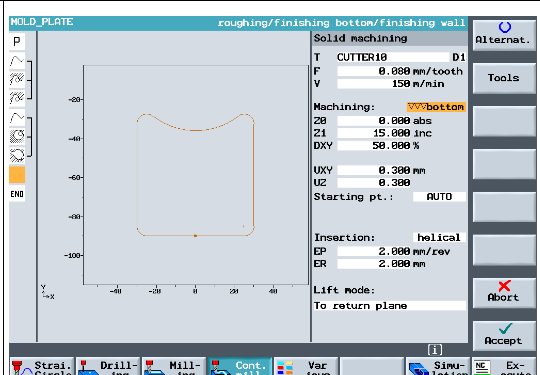
Solid Machining

Tools

To program

Accept

0.08
150

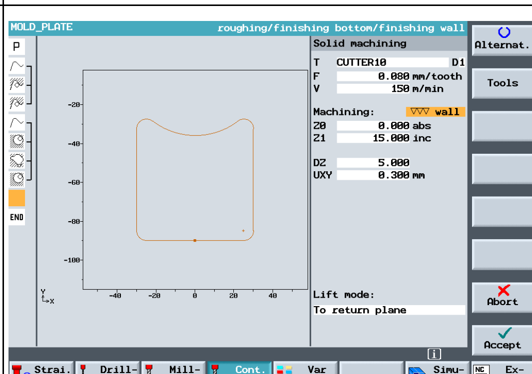


- You can also use the *Stock removal* function to post-machine the pocket. The machining must be switched to *Finish bottom* (**Base**).
- The allowance entered previously for roughing must be set once again for the values in the fields *Finishing allowance in the plane* (UXY) and *Finishing allowance in depth* (UZ). This value is relevant for the automatic computation of the traversing paths.

Solid Machining

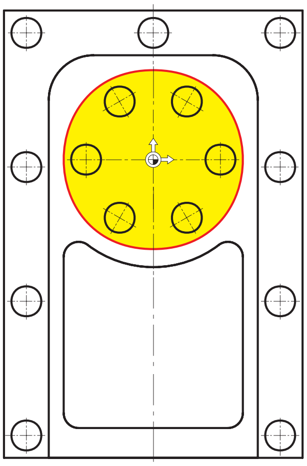
Accept

3x

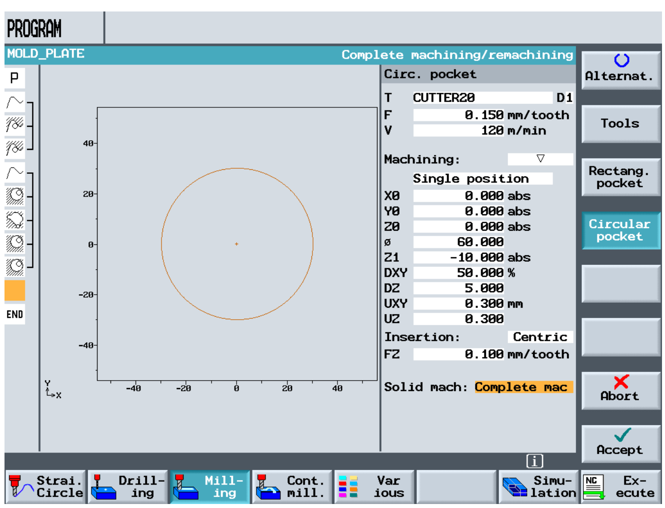


- The *Finish Wall* (**wall**) function machines the residual material at the contour.

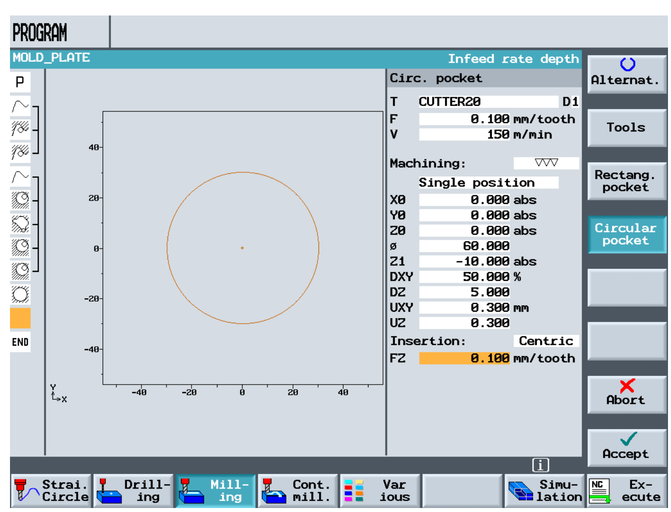
7.3 Machining on several planes



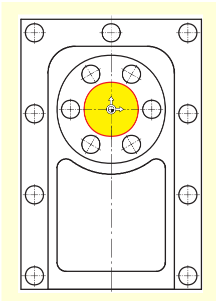
The size-60 circular pocket is milled in two work steps in exactly the same way as in the "Injection form" example.



The first step is to rough the pocket down to -9.7 mm using the size-20 milling tool.

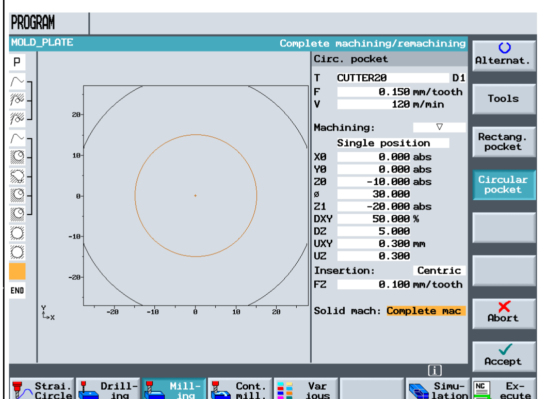
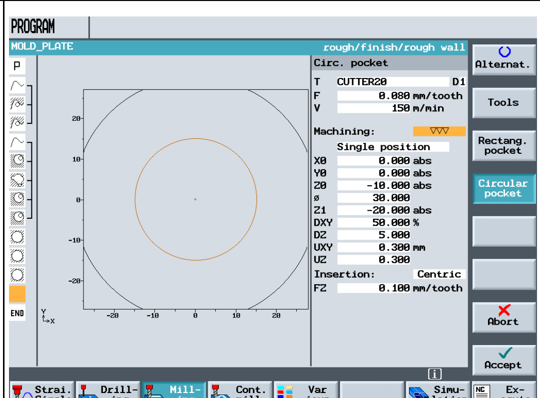
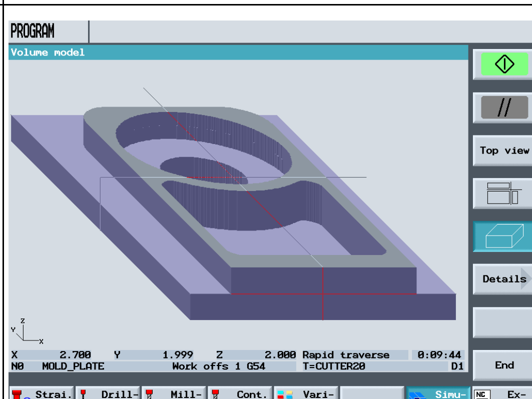


In the second step, the pocket is finished with the same tool.



Then, the inside circular pocket is machined down to the depth of -20 mm.

You must note here that the starting depth is -10 mm not 0 mm.

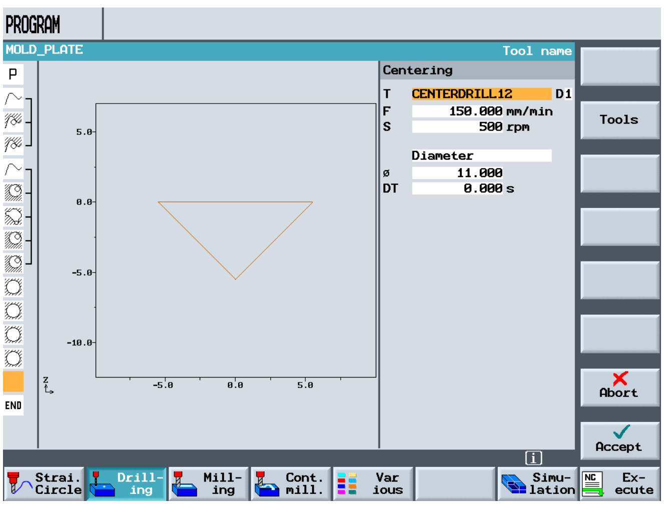
Keys		Screen	Explanations
Mill-ing Pocket Circular pocket Accept	0.15 120 …		• When you have entered the values as shown in the figure, you can accept the dialog box.
Mill-ing Pocket Circular pocket Accept	0.08 150		• The second step is to finish the pocket. • The position, size and dimensions are taken automatically from the roughing step performed previously. So you only have to enter the technology values. • The value *Z0* (= *High workpiece*) indicates the starting depth for machining.
Simu-lation Simu-lation			• The more complex the workpiece, the greater the significance of the 3D image in the preliminary production steps.

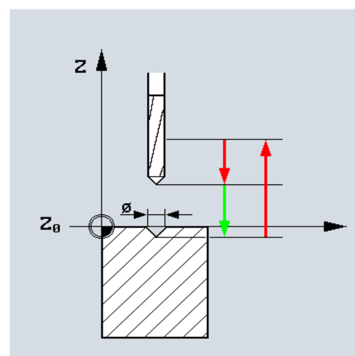
7.4 Considering obstacles

Just as for "Longitudinal guide", you can also chain various drilling patterns for this workpiece. But you must remember that one or more "obstacles" have to be traversed, depending on the order of machining operations. Traversing between the holes is carried out with the *safety distance* or on the *retract plane*, as appropriate to the settings you have defined.

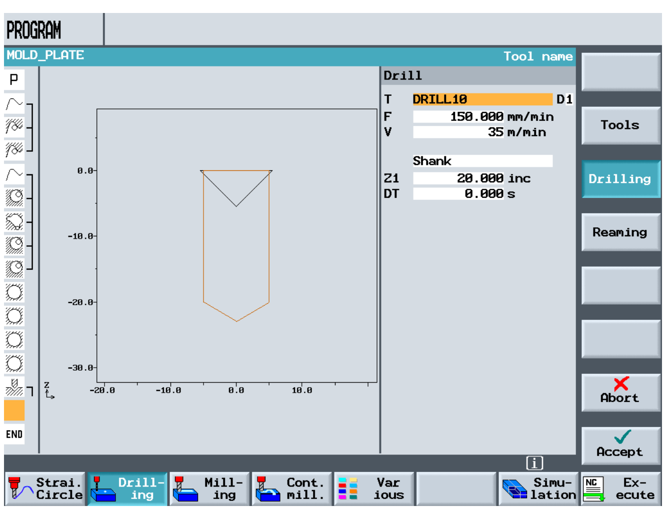
First, create the work steps: *Center* and *Drill* in the manner you were taught in Chapter 5.

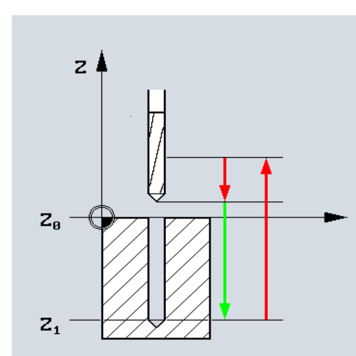
1. Work step *Centering*



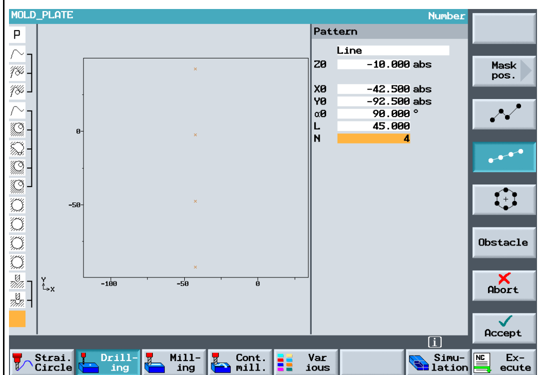
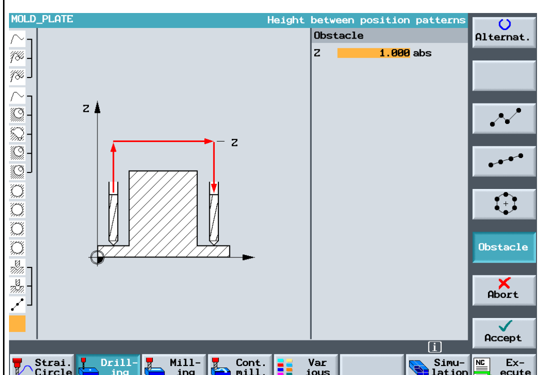
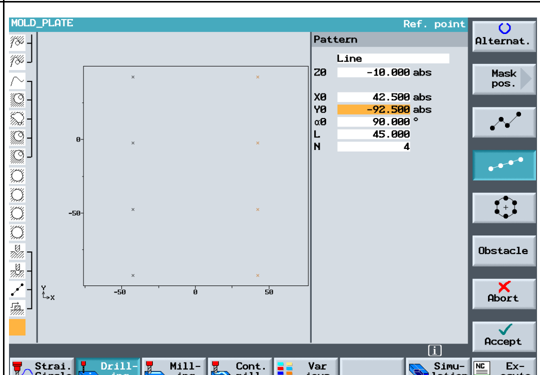
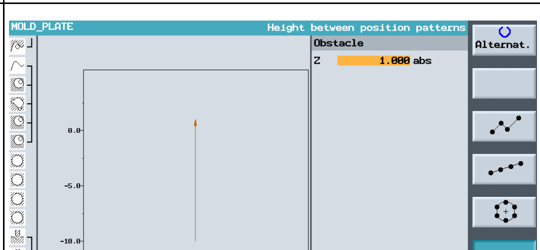


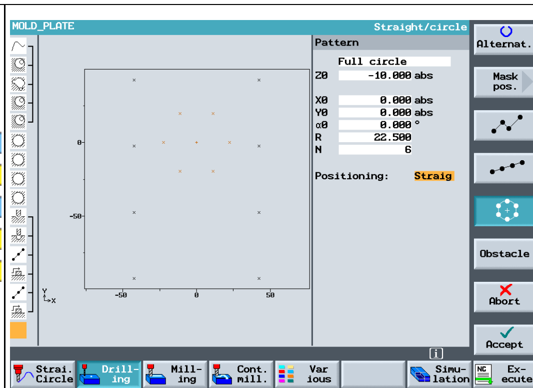
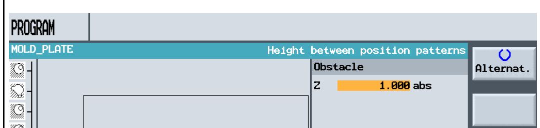
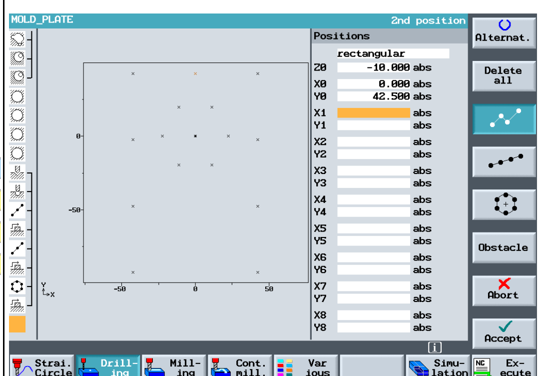
2. Work step *Drilling*





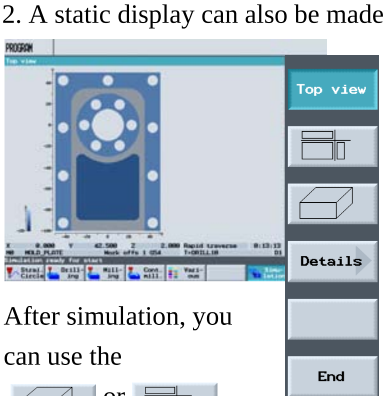
After you have created these two work steps, enter the associated drilling positions on the next page.

Keys		Screen	Explanations
Positions [line pattern key] Accept	↓ -10 -42.5 -92.5 90 45 4		• First, create the left-hand row of holes in the sequence from bottom to top.
Positions Obstacle Accept	1		• The *Obstacle* function is used to enter a travel path at a height of 1 mm, since the next step is to practice drilling the right-hand row of holes also from bottom to top.
Positions [line pattern key] Accept	2x ↓ 42.5		• Enter the second drilling route here.
Positions Obstacle Accept			• To obtain the next drilling pattern, the circle of holes, you must also navigate around an obstacle.

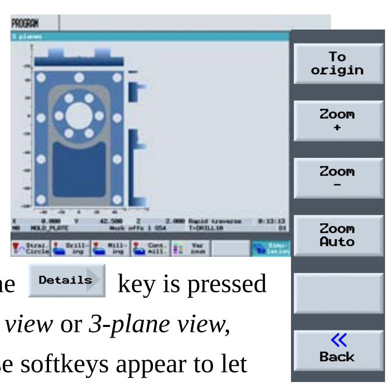
Positions Accept	-10 3x 22.5 6		• The six holes form a full circle.
Positions Obstacle Accept			• To produce the last hole, you must again navigate around an obstacle.
Positions Accept	-10 0 42.5		• Enter the last drilling position • Delete any positions that already exist with Del . • Note: This programming example should help you become acquainted with the *Obstacle* function. There are of course more elegant ways to program the drilling positions that have only one obstacle to overcome. Try out various strategies yourself

Further information about the display of the workpiece:

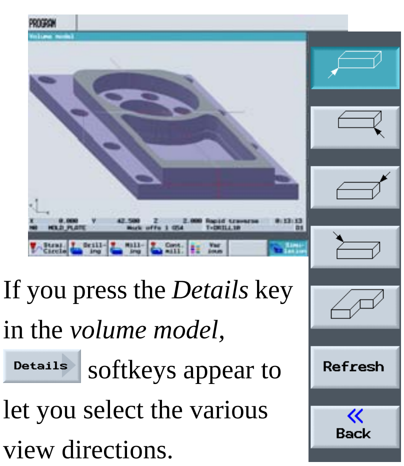
1. The simulation can only run in the *Top view* or in the *3-plane view*. The last setting remains active.
2. A static display can also be made in the *volume model*.



After simulation, you can use the [key] or [key] keys to switch to other display.



If the Details key is pressed *Top view* or *3-plane view*, these softkeys appear to let you increase the view zoom factor.



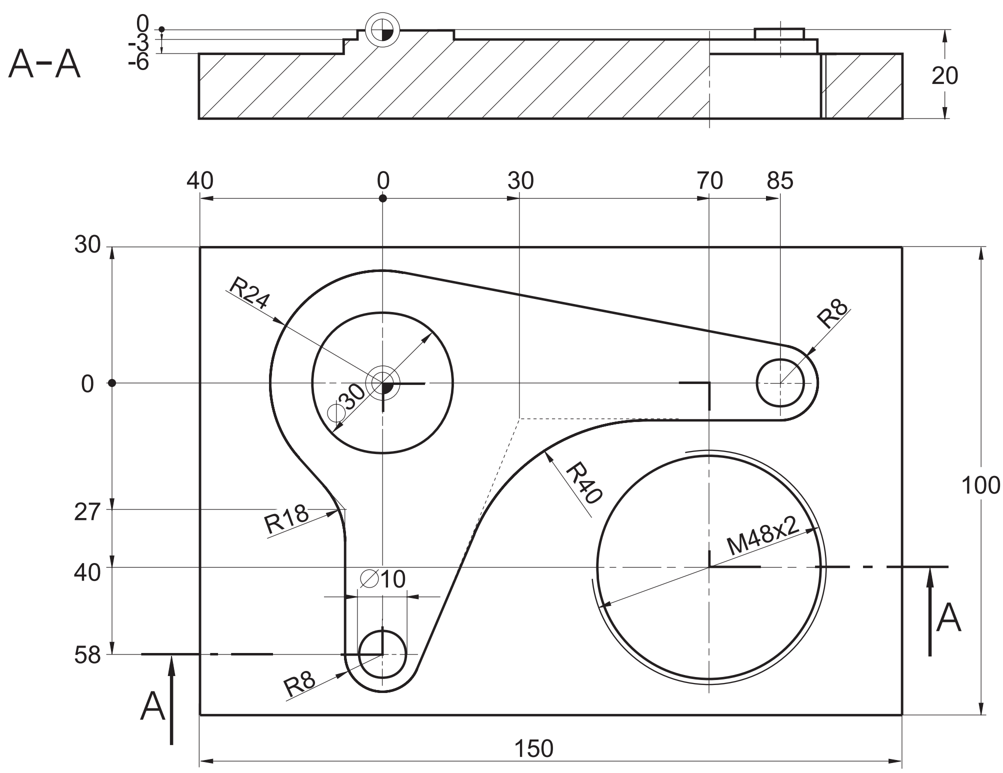
If you press the *Details* key in the *volume model*, Details softkeys appear to let you select the various view directions.

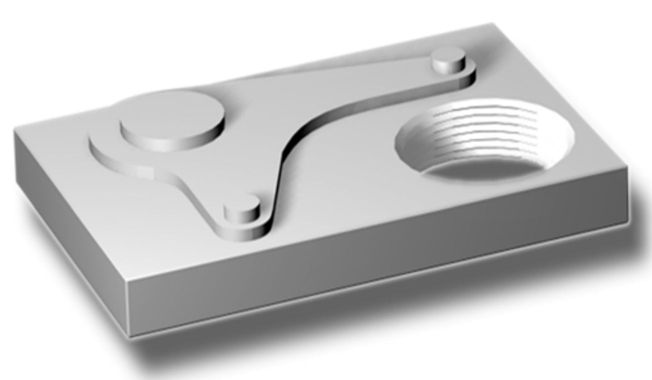
You can use the arrow keys to preset the cutting path execute this path with the [key] key.

8 Example 4: Lever

In this chapter, you become acquainted with the further important functions of ShopMill:

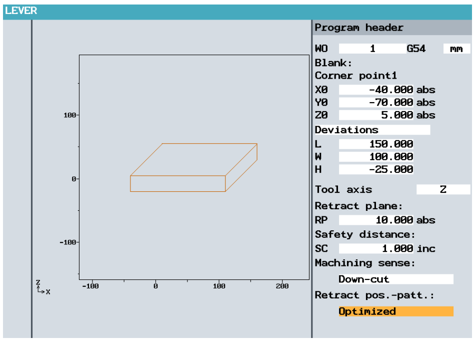
- Face milling
- Creating borders (auxiliary pockets) for solid machining around islands
- Creating circular islands by copying
- Extended editor and producing the islands
- Deep-hole drilling, helical milling, boring and thread cutting
- Programming contours with polar coordinates (new with ShopMill V 6.4 and higher)





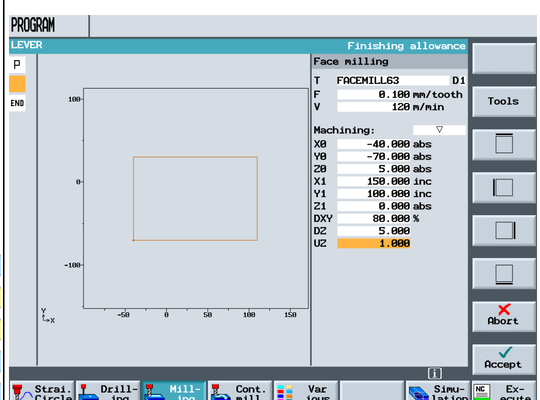
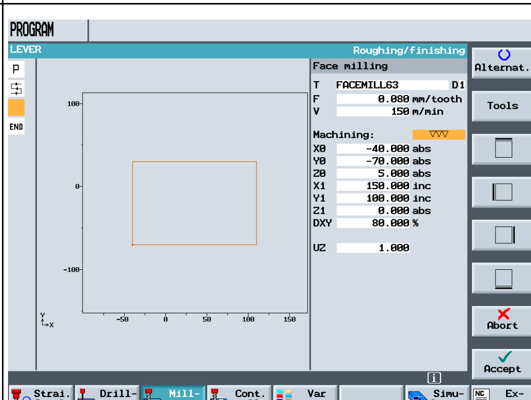
Creating a work plan

The workpiece dimensions must be taken from the drawing and entered in the program header. Here, you must observe that the unmachined part is to be 25 mm thick and that corner point 1 must therefore be set to 5 mm in Z.



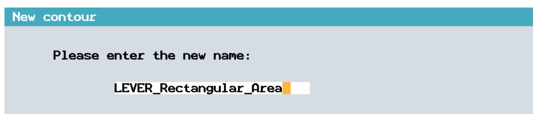
When you have entered the data, the input window should look like this.

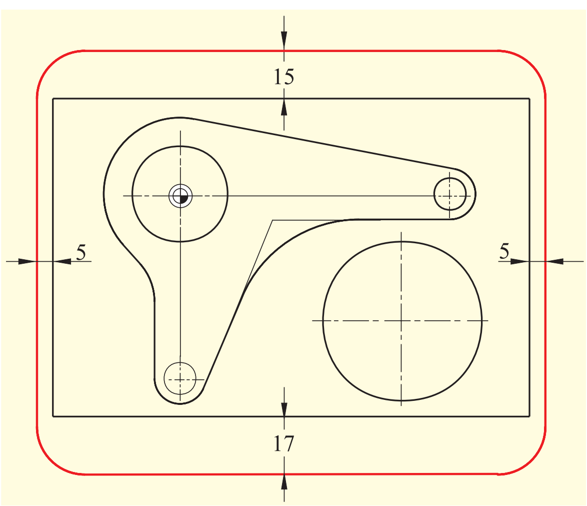
8.1 Face milling

Keys		Screen	Explanations
Mill-ing Face milling Tools To program Accept	... 0.1 120 ...		• When the function is called, you can choose from various machining directions, which are selected via the vertical softkey bar. • FACEMILL63 is used (F 0.1 mm/tooth and V 120 m/min). • The surface is roughed first. To do this, you must switch the Machine field to ▽ . • The dimensions of the unmachined part and the insertion depth and finishing allowance still have to be defined (see input window).
Face milling Accept	0.08 150		• To finish the surface, you must adapt the technology values (F 0.08 mm/tooth and V 150 m/min) and switch over the machining mode from *roughing* to *finishing* (▽▽▽). • The final allowance must have the same value as for roughing and finishing because the allowance for the subsequent finishing operating, and during finishing, refers to the material thickness still to be machined.

8.2 Creating a border for the lever island

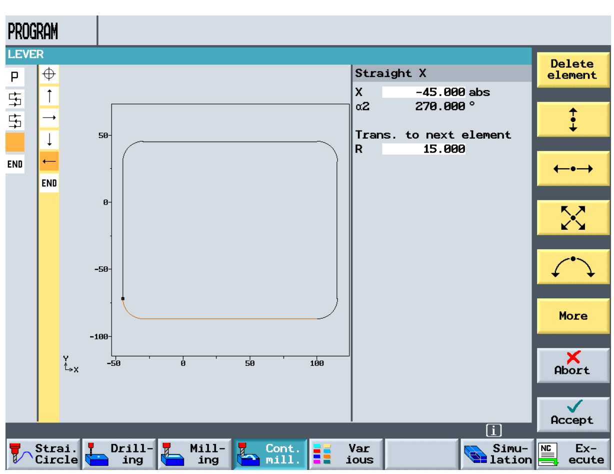
Islands are described as a contour in the graphic contour calculator in exactly the same manner as pockets. They do not become islands until they are linked in the work plan: The first contour always describes the pocket. One or more subsequent contours are interpreted as islands. Since there is no pocket in the "lever" example, a theoretical auxiliary pocket is applied to the outside contour. This is used as the required outside boundary for the traversing paths and thus defines the framework in which the tool movements are carried out.

Keys		Screen	Explanations
 	R...		• The outside contour is given the name "LEVER_Rectangular_Area".



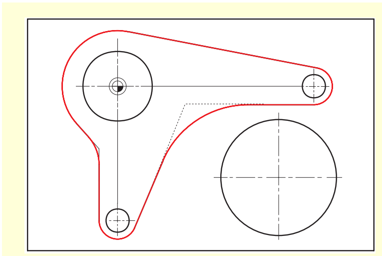
Design the pocket with the distances shown on the left (variable values) around the unmachined part.
The corners are rounded with R15.

Always make sure that the values you select cover the workpiece edges of the "Pocket"

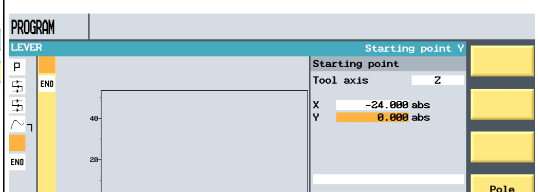
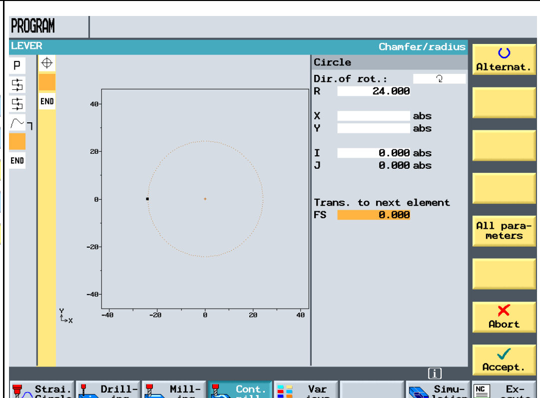
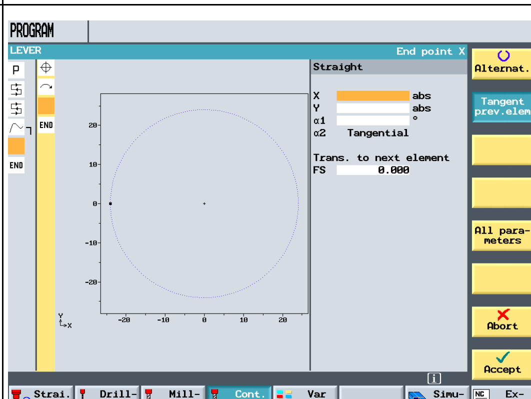


When the contour is finished, the screen looks like this.

8.3 Producing the lever

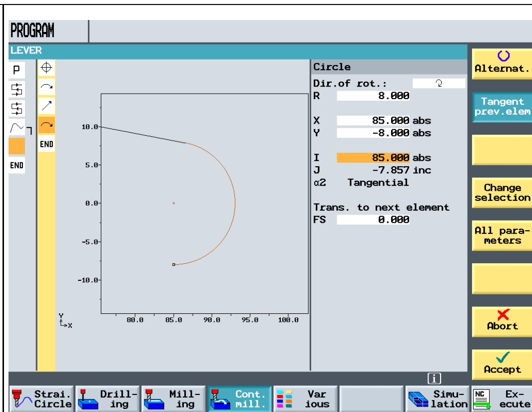


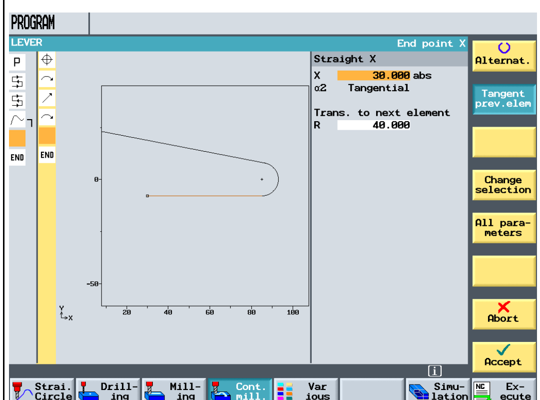
When you have added the outside contour after the last work steps, the next step is to create the following island. To give you practice in creating geometries, this example is explained step-by-step.

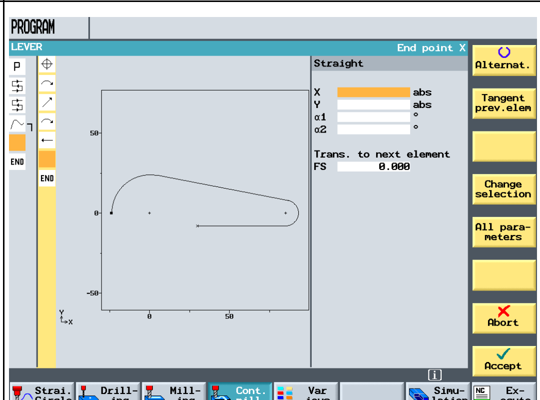
Keys		Screen	Explanations
Cont. mill. New contour	L...		• The island is given the name "LEVER_Lever".
Accept	1x -24		• The starting point of the contour should lie at X-24 and Y0.
Accept	24 2x 0		• The first arc runs counterclockwise, the radius and the center point are known.
Tangent prev.elem Accept			• A sloped tangent to the previous element follows.

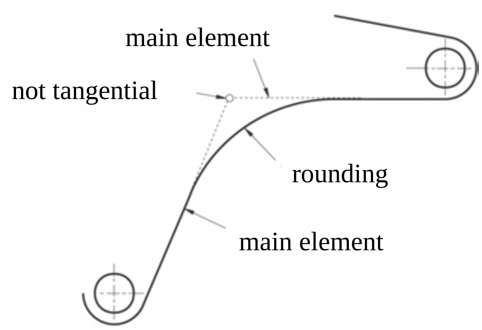
<table>
<tr>
<td>Tangent prev.elem

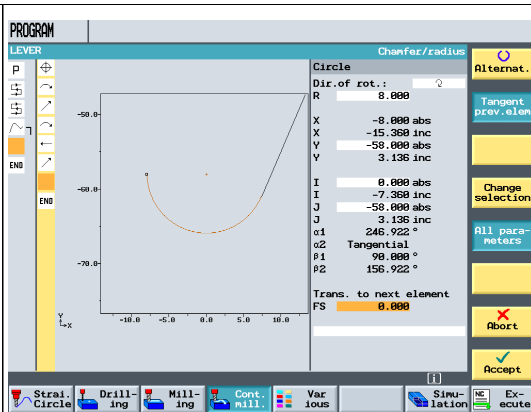
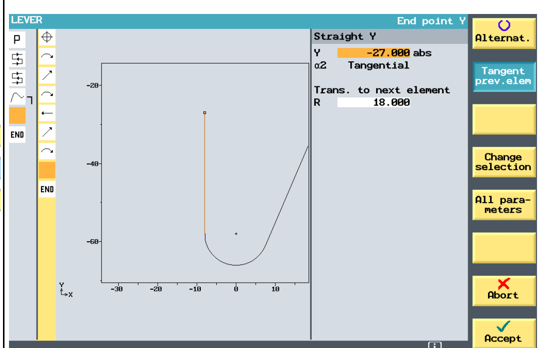
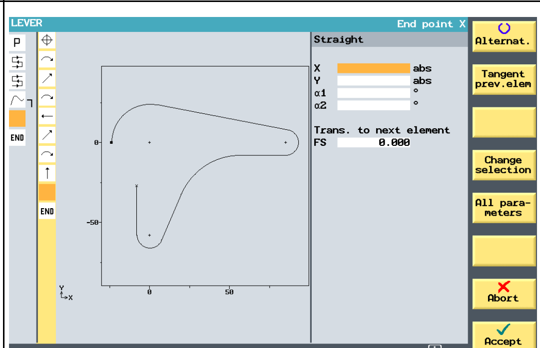
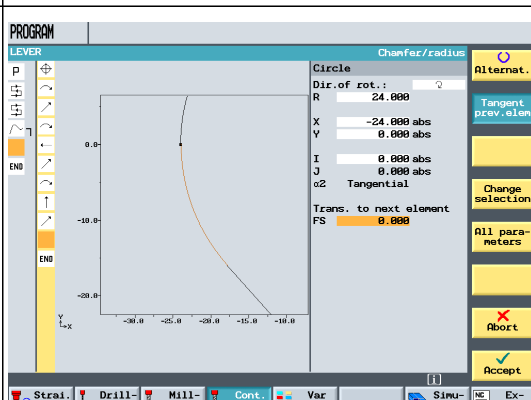
Dialog accept

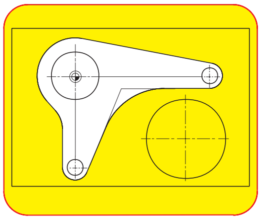
Accept</td>
<td>8
85
-8
85</td>
<td>
</td>
<td>• A tangential circular path follows.

• Radius, center point and corner points are known.</td>
</tr>
<tr>
<td>Accept</td>
<td>30
40</td>
<td>
</td>
<td>• A horizontal route to end point X30 follows.

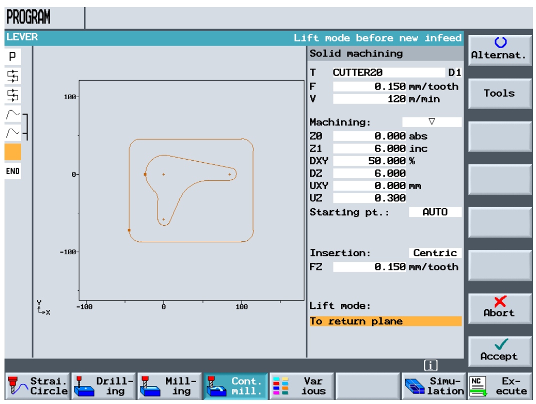
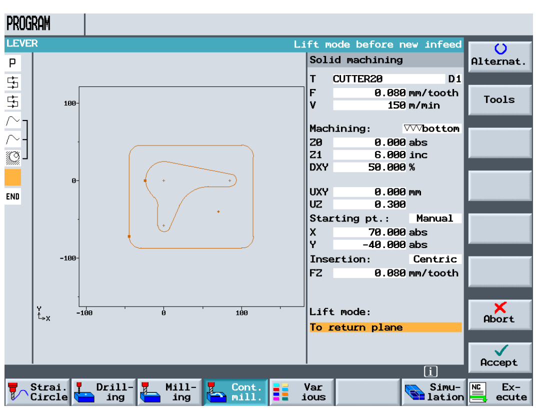
• The transition to the next element is to have a radius of 40 mm.</td>
</tr>
<tr>
<td>Accept</td>
<td></td>
<td>
</td>
<td>• A sloped path follows.

• Note: The tangential transition is always relative to the main element only, i.e. in this case the straight line does not lie at a tangent.

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

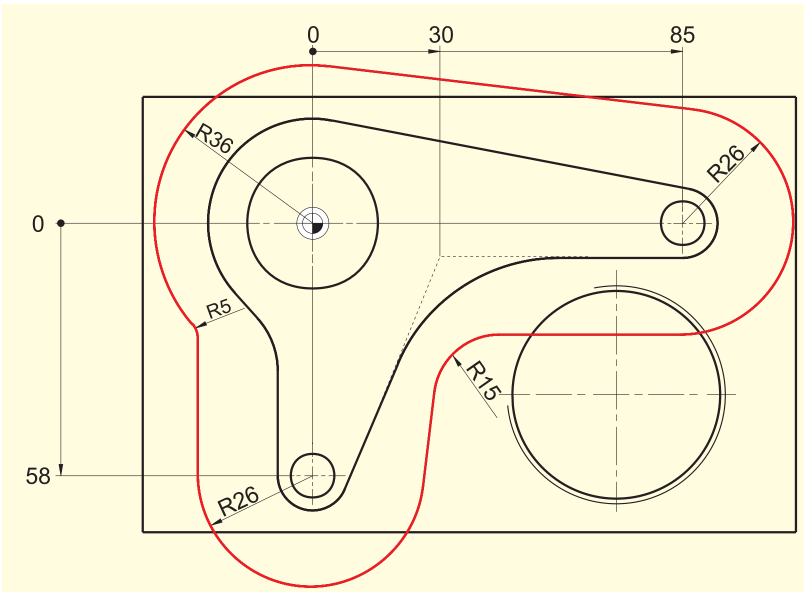
Tangent prev.elem All para-meters Dialog select Dialog accept Accept	8 2x -58 0 -58		• A tangential arc follows with a center point and end point that are known. • The *All parameters* function provides detailed information about the arc. This can be used to check the input values (for example: Does the arc end vertically ...?).
Tangent prev.elem Accept	-27 18		• A vertical path (automatically at a tangent) follows to end point Y-27. • The transition to the next straight line is to be rounded with R18.
Accept			• This is followed by an slope.
Tangent prev.elem Accept Accept	24 -24 0 0		• The contour is closed with an arc to the starting point.



The materials around the lever are first roughed and then finished to a depth of -6.

Keys		Screen	Explanations
Solid Machining Tools To program Accept	... 0.15 120 0 6 50 6 0 ...		• The pocket is machined and the lever contour taken into account. The CUTTER20 tool is used for roughing (F 0.15 mm/tooth and V 120 m/min). • The maximum infeed in the plane is specified in % here.
Solid Machining Tools To program Accept	... 0.08 150 0 6 50 0 70 -40 ...		• The base of the pocket is finished (F 0.08 mm/tooth and V 150 m/min).

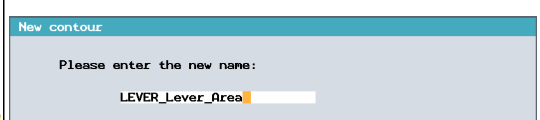
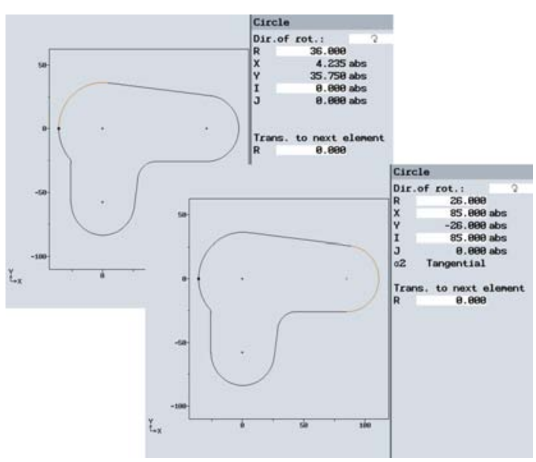
8.4 Creating a border for the circular islands



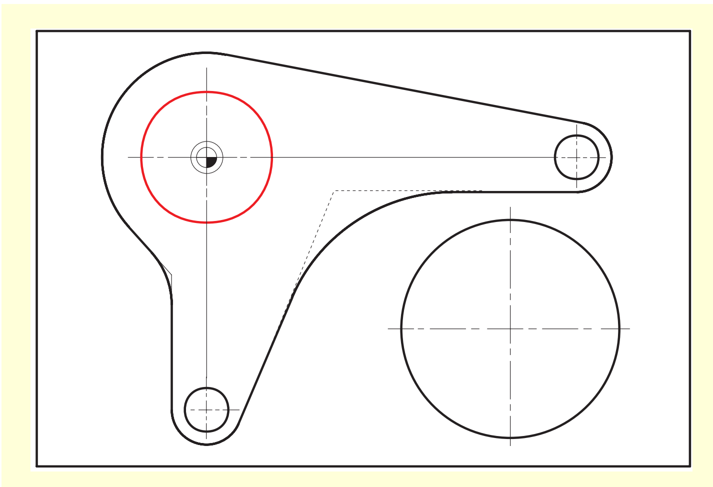
A border is created below as a traversing limit for milling to depth -3.

The values R36 and R26 are derived from the relevant
Island radius + cutter diameter (here 20 mm + 1 mm allowance).

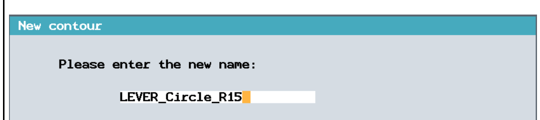
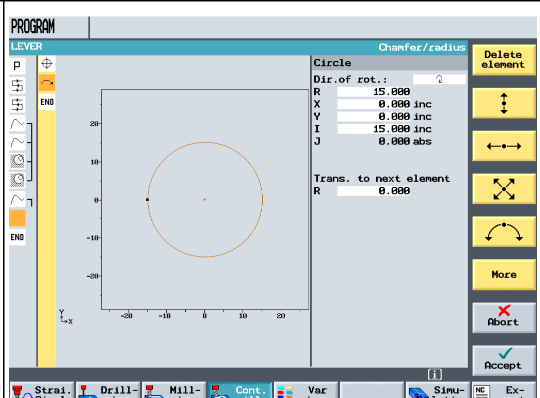
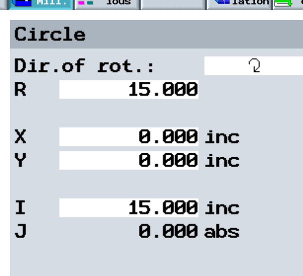
The radii R5 and R15 can be selected freely.

Keys	Screen	Explanations
Cont. mill. New contour L...		• The contour is assigned the name "LEVER_Lever_Area"
		• The limit for the traversing paths is (as described above) designed around the workpiece contour in such a manner that the size-20 cutter fits between the limitation and the islands. • Enter this limiting contour in the same way as the lever contour.

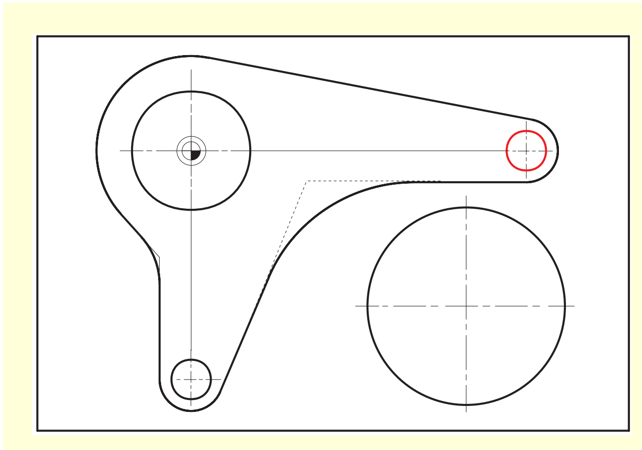
8.5 Creating a size-30 circular island



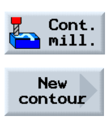
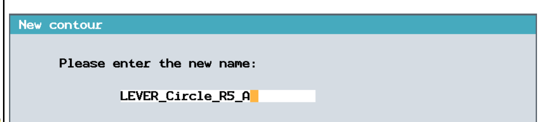
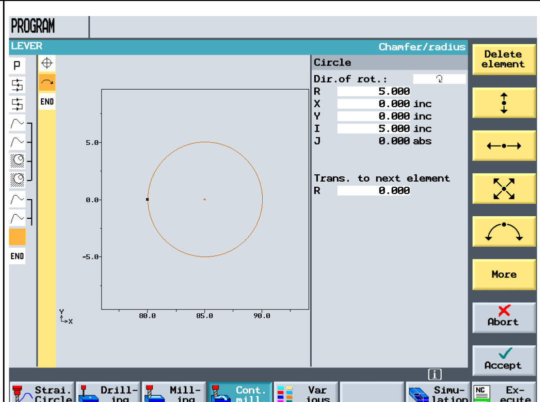
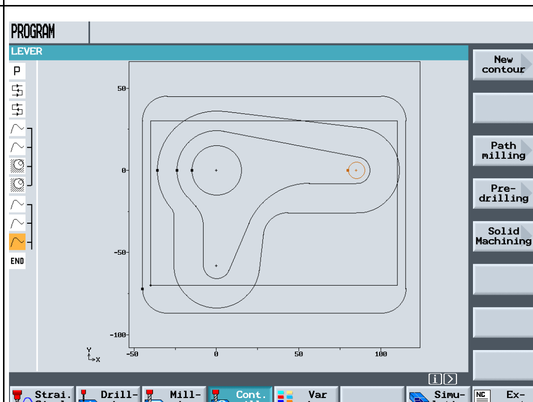
Now create the size-30 circular island.

Keys		Screen	Explanations
	C... 		• The contour is assigned the name "LEVER_Circle_R15"
			• The starting point of the circular structure lies at X-15 and Y0. • Complete the entries for the circular contour on your own according to the values below. Note that several values have incremental dimensions.

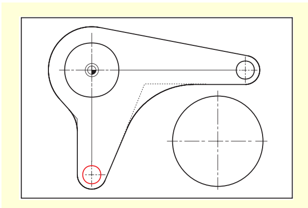
8.6 Creating a size-10 circular island



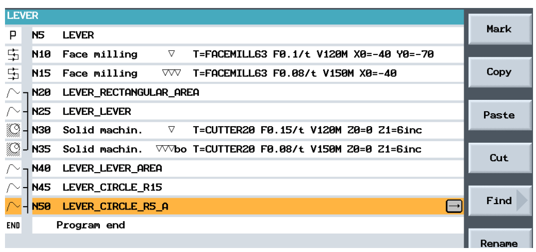
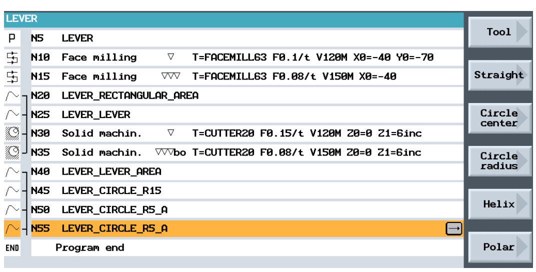
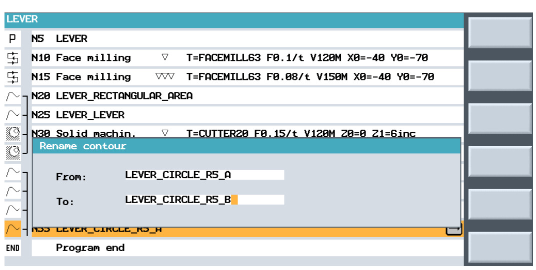
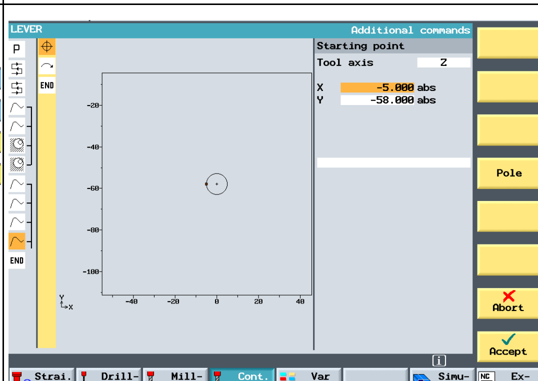
Now create the first size-10 circular island.

Keys		Screen	Explanations
	C... 		• The contour is assigned the name "LEVER_Circle_R5_A"
			• The starting point of the circular island lies at X80 and Y0. • Since these circular islands are copied below, the contour must be input as incremental so that only the starting point has to be changed after copying.
			• When you have entered the circle, the work plan graphic looks like the one shown here, if you have activated the work plan graphic with .

8.7 Copying the size-10 circular island



In the section below, you learn how to copy in ShopMill.

Keys		Screen	Explanations
> Copy			• Click on the > key to open the extended editor and then copy the contour.
Paste			• Insert the copied contour. • Because changes to the contours affect other contours that have the same name, the contour must be renamed.
Rename	← B ⇔		• Only the name of the contour needs to be changed to "LEVER_CIRCLE_R5_B" in the information dialog. You have now created a copy of the first circular island.
Accept Accept	↑ → 2x ↓ -5 ⇔ -58 ⇔		• After selecting the "LEVER_CIRCLE_R5_B" contour, click on the → key to call up the contour so that you can make changes. • Because the contour was previously entered incrementally, only the start point needs to be changed. • Click on the → key to open all geometry elements to allow changes to be made.

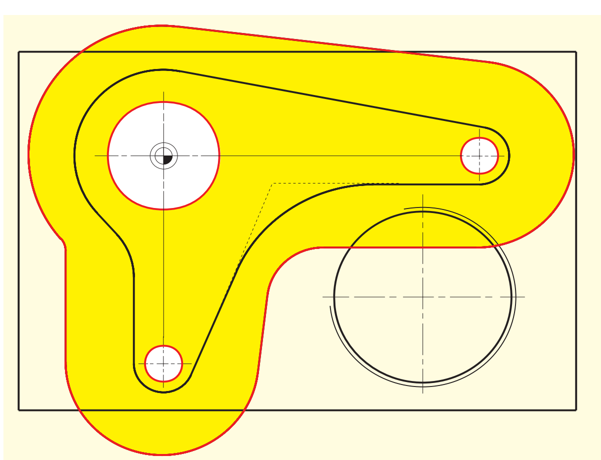
8.8 Production of the circular island using the extended editor

ShopMill offers a series of special functions that allow multiple use and management of sections of the work plan. These special functions can be reached at any time via the > key on the flat panel.

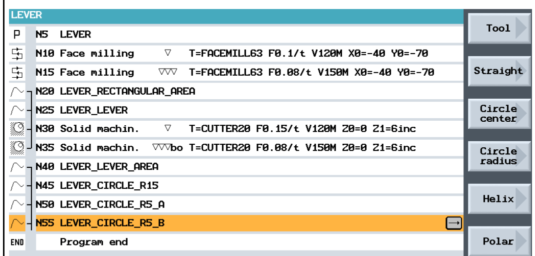
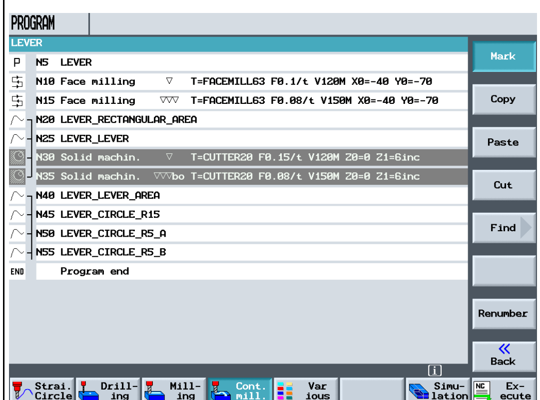
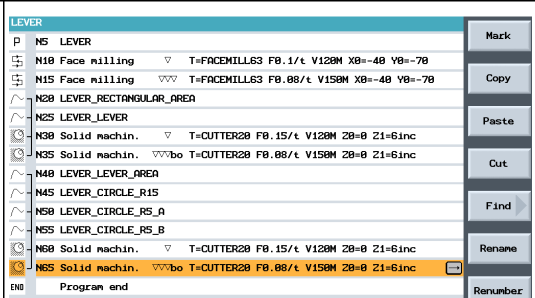
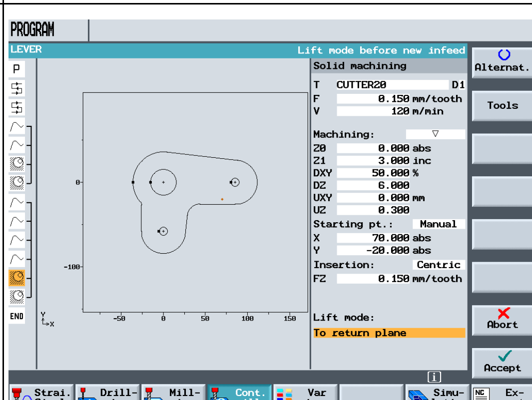
These functions are explained below:

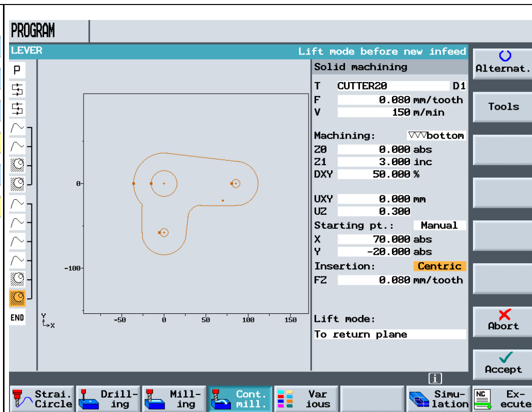
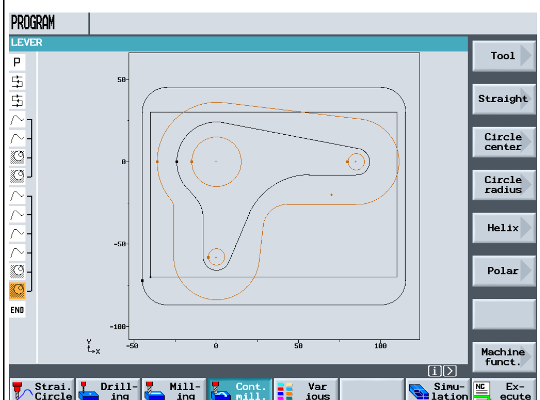
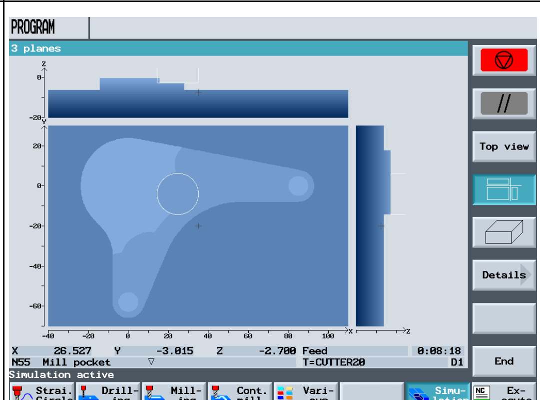
Mark	You can use the *Mark* function to select several work steps for further processing (e.g. *Copy* or *Cut*).
Copy	The *Copy* function copies the work steps to the clipboard.
Paste	The *Paste* function adds work steps to the work plan from the clipboard. Pasting is always performed behind the marked work step.
Cut	The *Cut* function copies work steps to the clipboard and at the same time deletes them from their original location. The softkey is used purely for deletion purposes.
Find	You can use the *Find* function to look for texts in the program.
Rename	The *Rename* function can be used to change the names of the contours, directories and workplans.
Renumber	The *Renumber* function renumbers the work steps.
« Back	The *Back* function returns you to the previous menu.

Some of the functions described initially are used below to produce 3 circular islands effectively. The efficiency is obtained by copying the existing work steps.

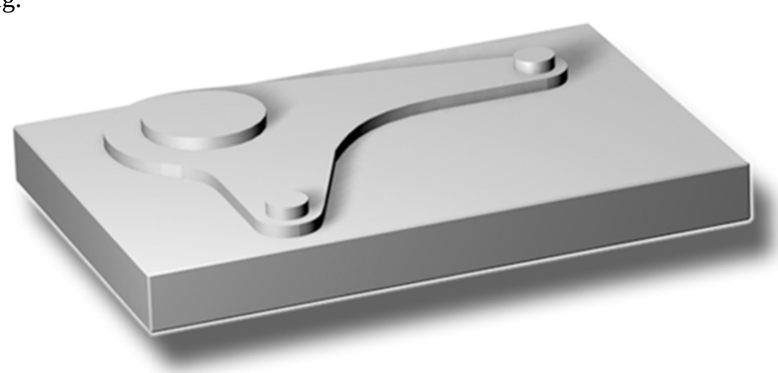


The border highlighted red in section 8.4 is used as the traversing path limitation here.

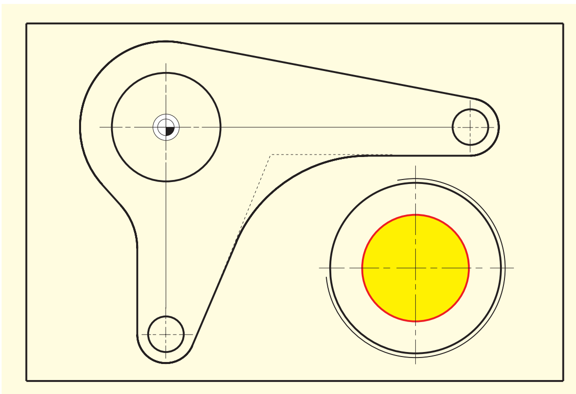
Keys		Screen	Explanations
			• The work plan should now look like this.
Mark Copy Paste Back	5x ↑ > ↓ 4x ↓		• The two previous stock removal technologies are added to the chained contours with the *Copy* function.
	↑ →		• The two stock removal technologies must now be adapted to the new machining depth.
Accept	5x ↓ 3 ◇ 4x ↓ ↻ ↓ 70 ◇ -20 ◇		• The roughing depth is set with the value *Z1* to 3 mm and a starting point selected outside the residual material.

	5x  3 5x -20		• The finishing depth is also adapted to suit.
 			• The geometries that belong to the finishing technology are displayed here (work plan graphic).
 			• As before: The simulation ...

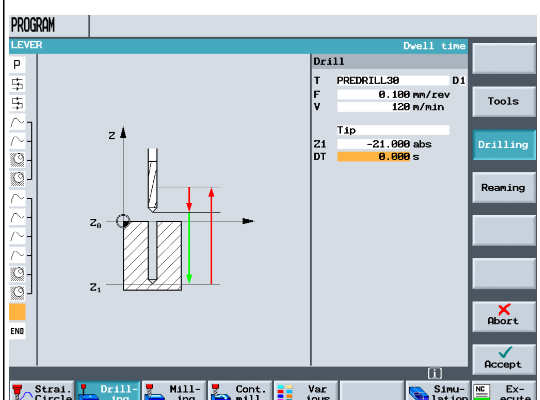
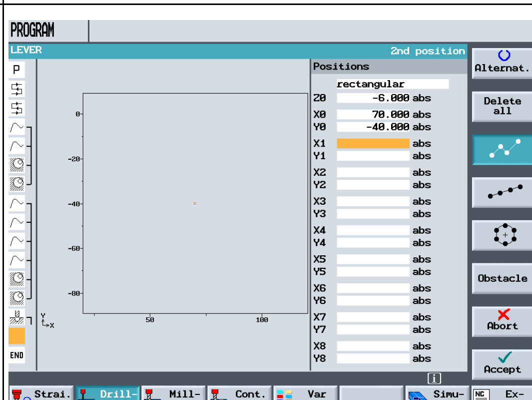
... is shown for checking.



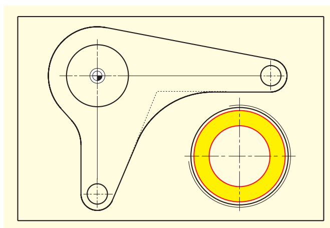
8.9 Deep-hole drilling



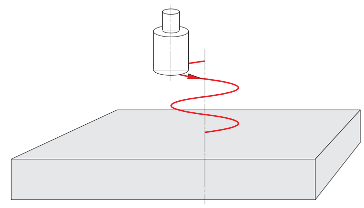
A drill is used below.

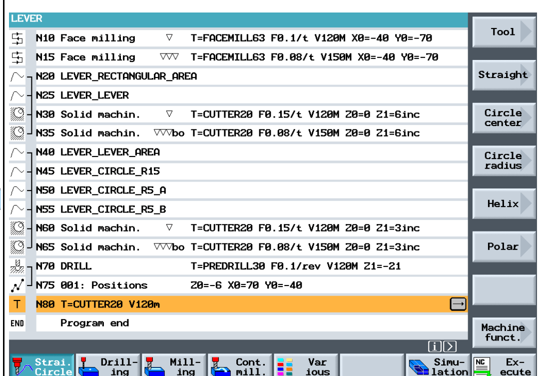
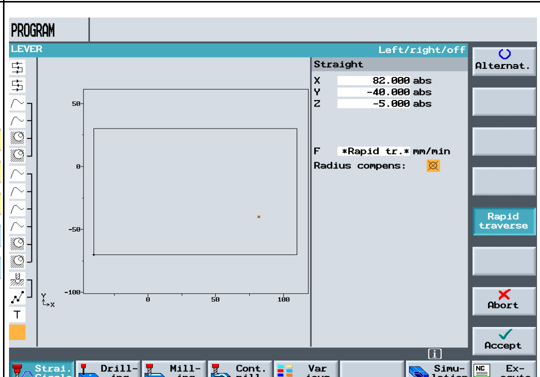
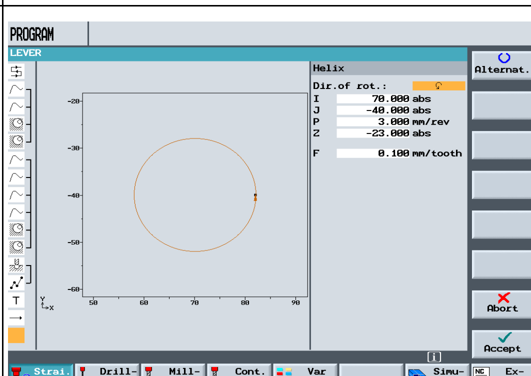
Keys	Screen	Explanations
Drill-ing Drilling Reaming Tools ... To program 0.1 120 -21 Accept		• PREDRILL30 is used for predrilling (F 0.1 mm/rev and V 120 m/min). • The depth reference point is set to *tip* with the setting *abs*.
Positions -6 70 -40 Accept		• The drilling position is entered here.

8.10 Helical milling

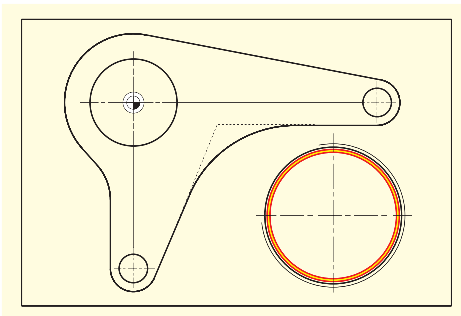


Below, a milling tool is used to remove the residual material in a spiral motion, referred to as a *helix*.



Keys		Screen	Explanations
Strai. Circle Tool Tools To program Accept	↓ ...		• The helix is used to remove the remaining circular ring after drilling. The CUTTER20 is used to do this (V 120 m/min).
Straight Rapid traverse Accept	82 -40 -5 ↓ 3x		• Since you are milling without cutter radius correction here, the milling tool must be positioned on the core hole diameter (here 45.84 mm) minus the finishing allowance.
Helix Accept	↓ 70 -40 3 -23 0.1		• The helix is milled in synchronism. • The pitch of the helix is 3 mm. • Since the tool travels over a sloped path, 6 revolutions are created here to prevent any residual material being left over (although the final depth is reached after five).

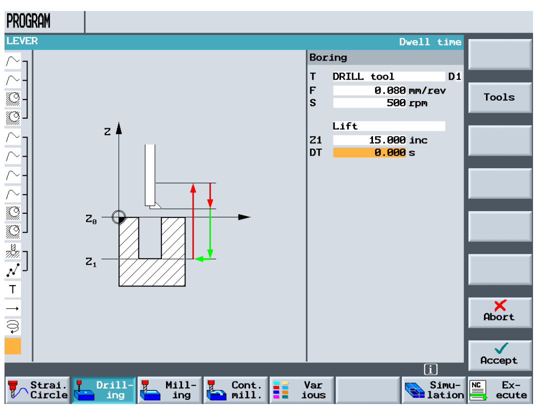
8.11 Boring



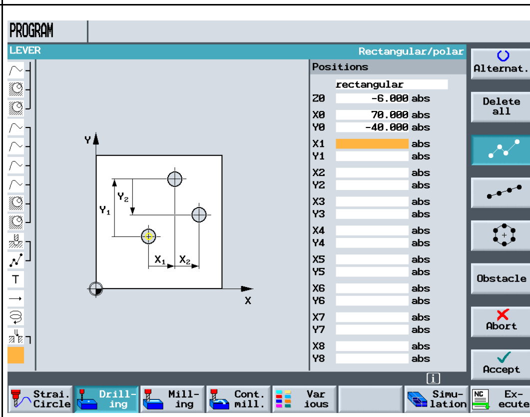
The pre-fabricated circular pocket is machined to dimension using a boring tool in the section below.

<table>
<tr><th colspan="2">Keys</th><th>Screen</th><th>Explanations</th></tr>
<tr><td>Drill-ing
Boring
Tools
To program

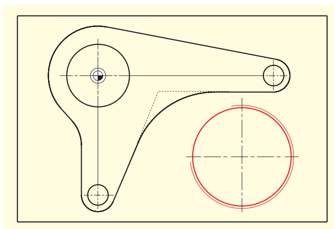
Accept</td><td>...
0.08
500

15
0</td><td>
</td><td>• The hole for the thread core is drilled as appropriate using DRILL tool (F 0.08 mm/rev and S 500 rpm).
• The Lift option retracts the tool in front of the contour before it emerges from the hole. This option may only be used for mono-cut tools.

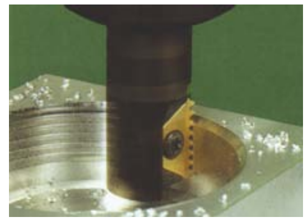
Note: The angular position is specified by the machine manufacturer.</td></tr>
<tr><td>Positions

Accept</td><td>-6
70
-40</td><td>
</td><td>• Position the tool in the center of the hole. The dimension 45.84 mm is defined by the diameter of the tool selected.
• Rather than enter the position, you could also work with the Repeat position function.</td></tr>
</table>

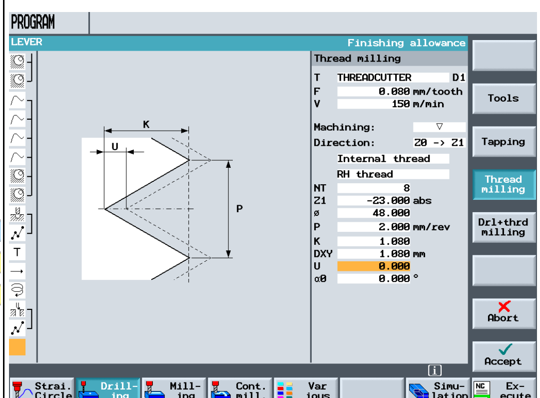
8.12 Thread cutting



The thread is produced with a thread cutter below.

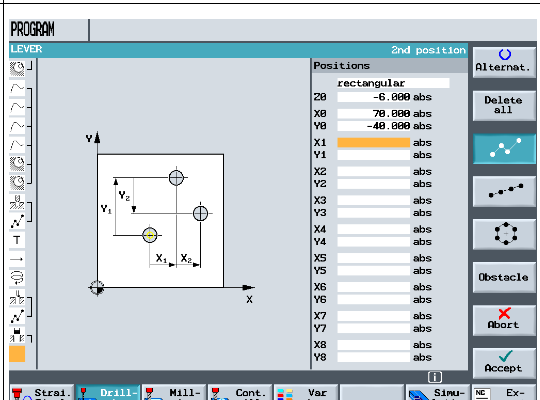


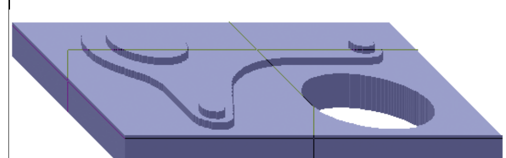
<table>
<tr><th>Keys</th><th></th><th>Screen</th><th>Explanations</th></tr>
<tr>
<td>Drill-ing
Tapping thread
Thread milling
Tools
To program

Accept</td>
<td>0.08
150
...</td>
<td>
</td>
<td>• The thread is milled from top to bottom. Use the THREADCUTTER to do this (F 0.08 mm/tooth, V 150 m/min and a pitch of 2 mm).

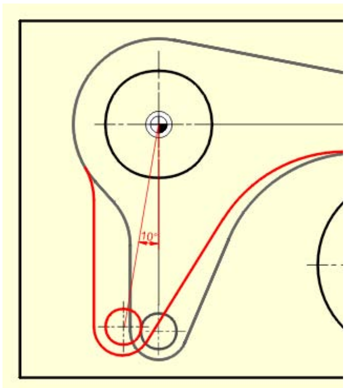
• A right-hand thread is to be milled to Z-23 absolute. The overlap of 3 mm means that the thread is definitely milled properly up to the workpiece, even if the bottom tooth is worn slightly.

• The help displays are very useful when entering values.</td>
</tr>
<tr>
<td>Positions

Accept</td>
<td>-6
70
-40</td>
<td>
</td>
<td>• The position of the thread is specified here.

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

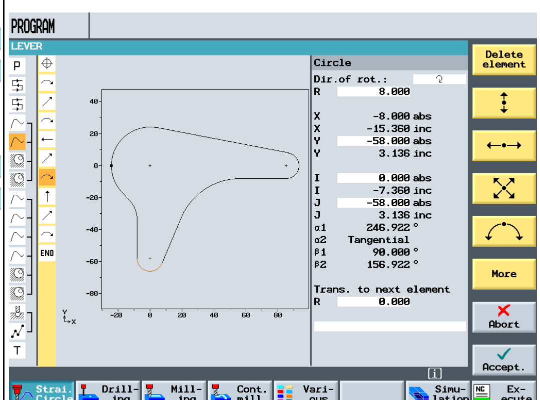
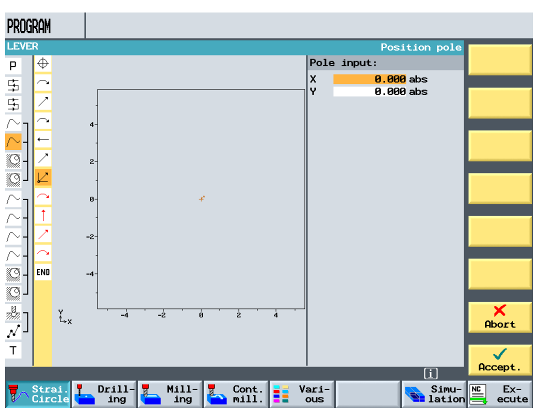
8.13 Programming contours with polar coordinates

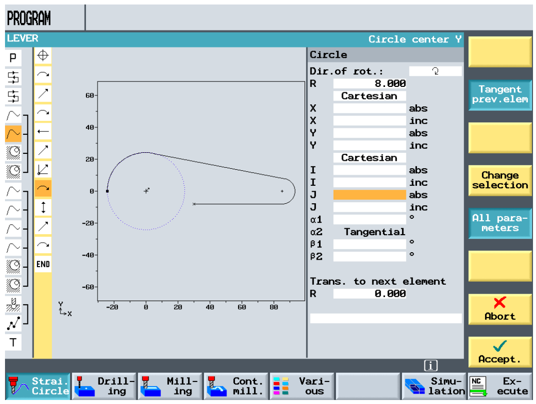
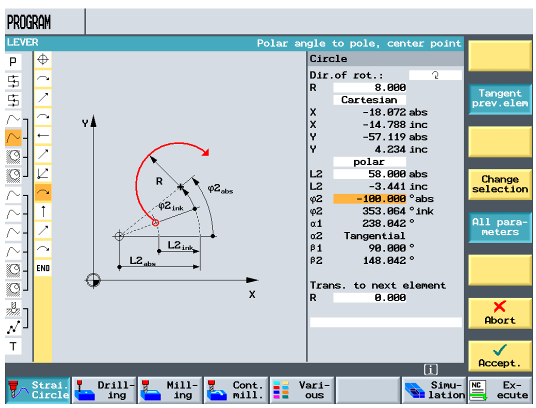
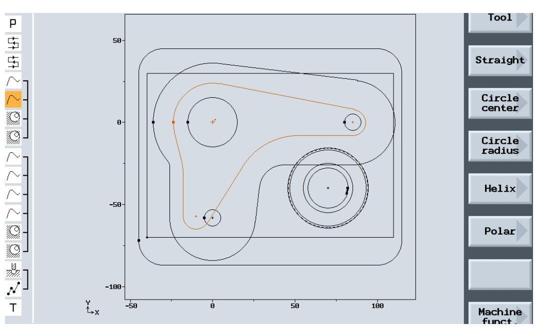
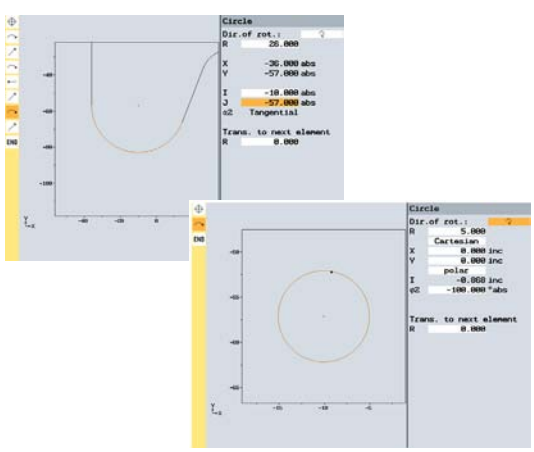


It is not uncommon that contour elements in workpiece drawings refer to a pole point. If so, you do not know the Cartesian coordinates (X/Y), but the polar coordinates, i.e. the distance (L) and the angle (φ) to this pole.

With ShopMill V 6.4 and higher, also such cases can easily be programmed graphically without pocket calculator or auxiliary construction.

You can understand this by means of a small change of the lever: The lower "lever arm" is then no longer perpendicular to zero at X0 but rotated around 10° in clockwise direction.

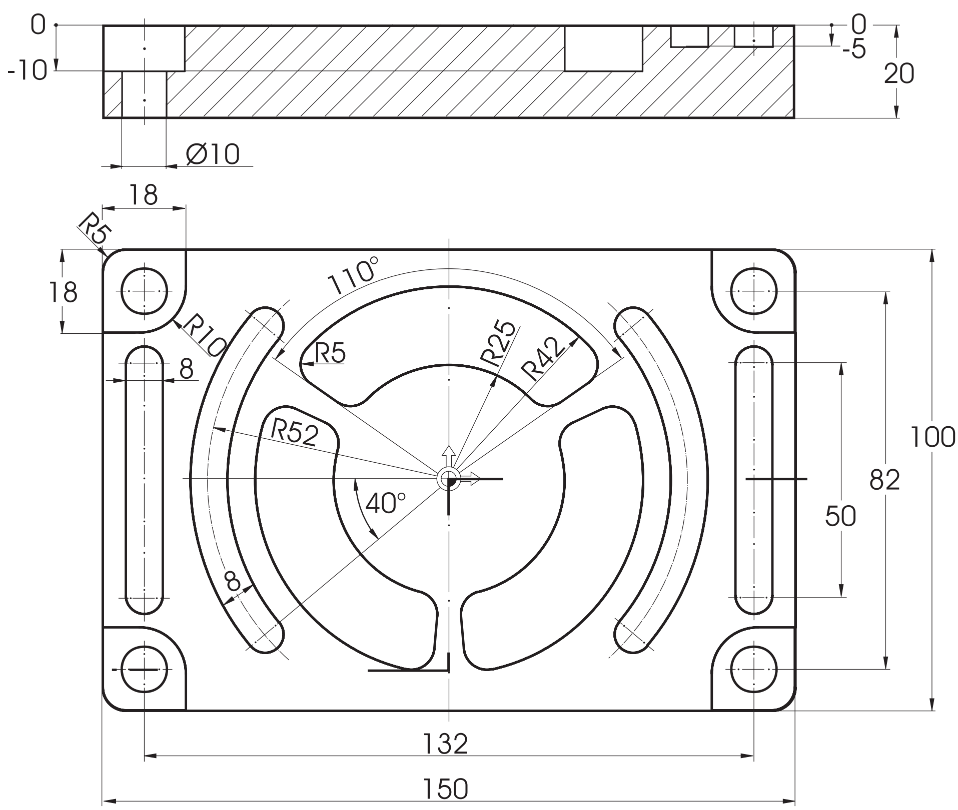
Keys	Screen	Explanations
↑ ↑ … → 3x ↑		• First move the cursor to this arc, for which new center-point dimensions are to be entered.
↑ More Pole Accept		• First of all, the pole must be set in the zero point before the arc. • Therefore, place the cursor on the element before the arc and then add the pole at this position.

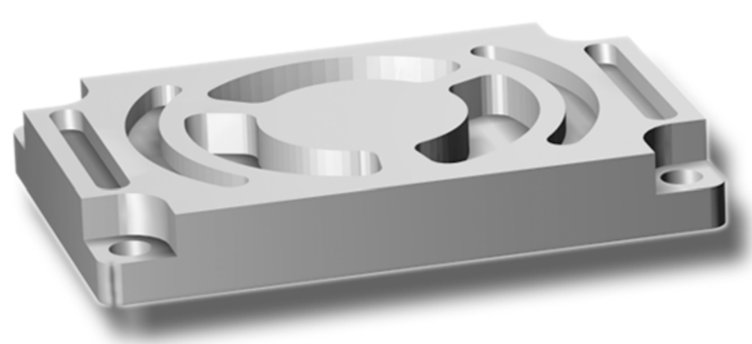
	3x Del 4x Del 2x Del		• In the dialog window of the arc, delete the values Y-58, I0 and J-58 which are no longer valid.
Accept Accept	3x 58 -100		• To enter the center-point, switch the coordinates from Cartesian to polar, and enter the distance to the pole (L2) and the polar angle (φ2). • Where required, the auxiliary graphics (ⓘ) illustrates the meaning of the input values.
	ⓘ		• The work plan graphics show that the auxiliary pocket LEVER_LEVER_AREA in line N40 and the circular island LEVER_CIRCLE_R5_B in line N55 will have to be adjusted in a similar manner .
	...		• Change these two contours yourself. • With the auxiliary pocket, you can, of course, have a rather "rough" approach, i.e. approximate the center-point of arc R26 (with polar dimensions) with Cartesian coordinates (X-10/Y-57). The contour can then be terminated directly with a vertical line. • In the case of the circular island, the starting point is already defined by polar coordinates. You then still have to change the center-point of the full circular arc.

9 Example 5: Flange

87This chapter addresses the following new contents:

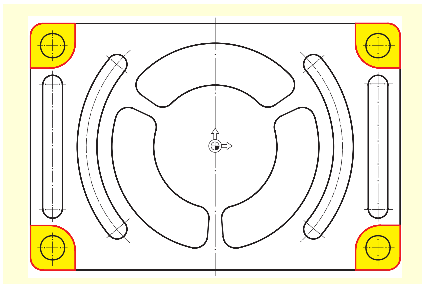
- Creating a subroutine
- Mirroring work steps
- Rotation of pockets
- Chamfering any contours
- Longitudinal and circumferential grooves



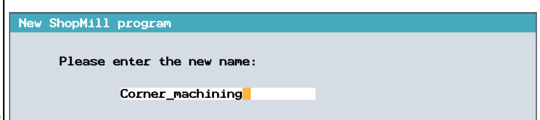
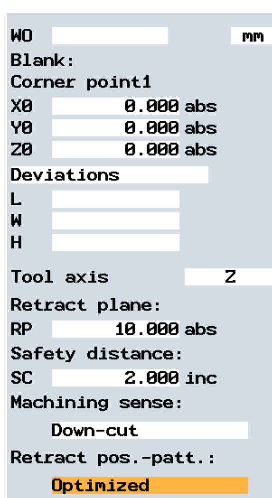
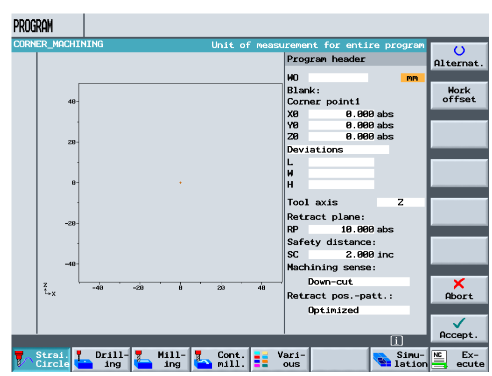
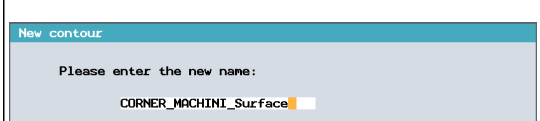
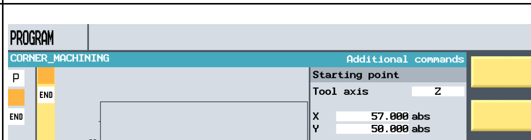


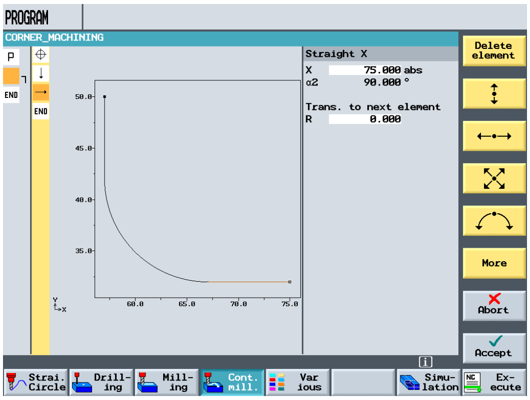
Remarks: Up to now, almost all keys that you pressed were displayed. In this example, the entries are no longer specified, only the main keys. Since the values in the dialogs are very important, however, these dialogs are shown in large format. The result is shown as an overall display in the right-hand column.

9.1 Creating a subroutine



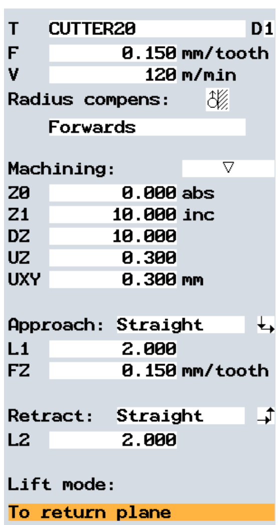
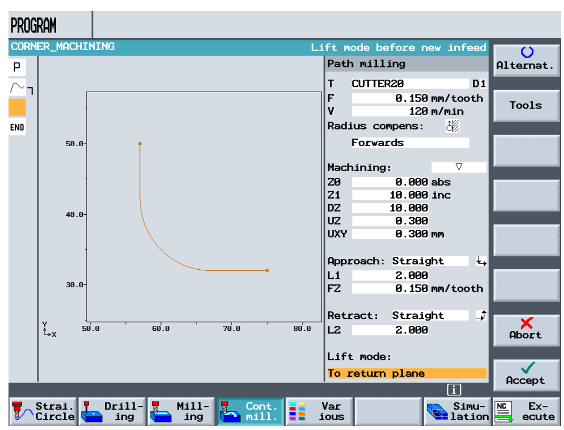
The example demonstrates the creation and mode of operation of the subroutines for the "flange" workpiece. The four corners are machined using a subroutine and the *mirroring* function below.

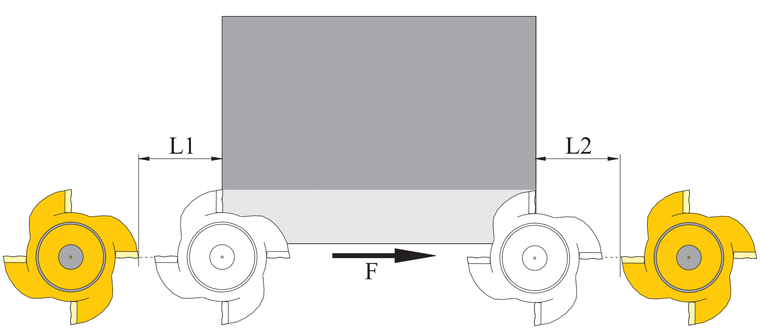
Keys		Screen	Explanations
Pro-gram ... New	C...		• The subroutine, which does not differ formally from the main program, is given the name "Corner_machining".
Accept			• Enter these data for the program header. Zero and blank dimensions are determined later centrally in the main program. 
Cont. mill. New contour	C...		• The contour is assigned the name "CORNER_MACHINI_Surface".
Accept	57 50		• For example, the above right corner should be constructed. • Enter a suitable starting point.



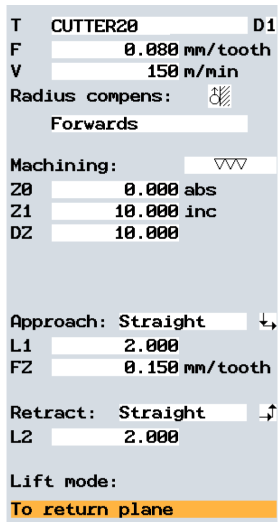
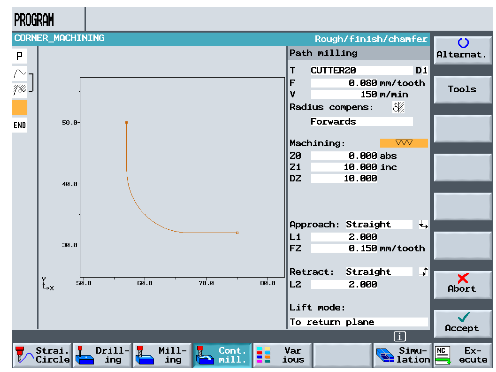
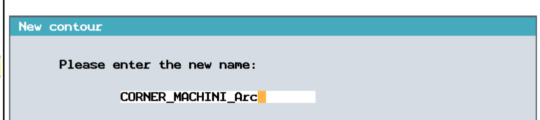
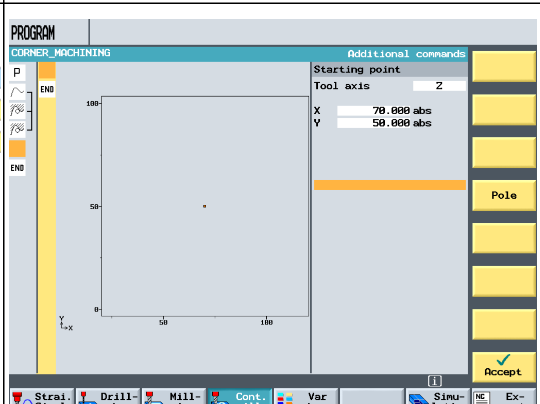
When you have entered the two contour elements, the screen should look like this.

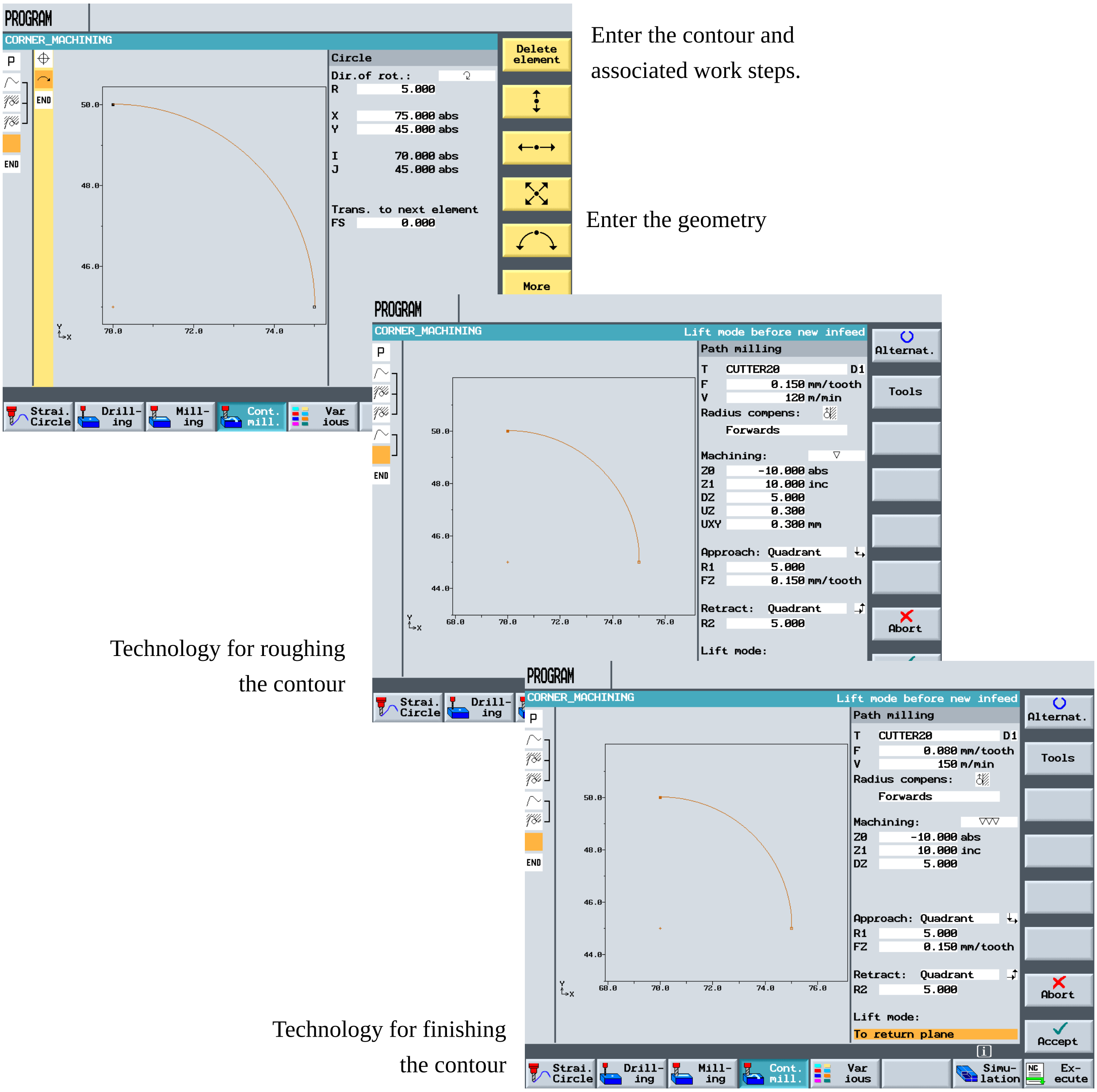
Incorporate the contour in the work plan.

Keys		Screen	Explanations
Path milling ... Accept	...		• The contour is to be roughed with a size-20 cutter (F 0.15 mm/tooth and V 120 m/min). 



The approach and return paths are approached here on a straight line. The length values are the distances between the edge of the milling tool and the workpiece.

Keys		Screen	Explanations
Path milling ✓ Accept			• The contour is to be finished with the same cutter (F 0.08 mm/tooth and V 150 m/min). 
Cont. mill. New contour	C...		• Next, the corner of the unmachined cuboid should be rounded with R5. The contour is assigned the name "CORNER_MACHINI_Arc".
✓ Accept	70 50		• Enter the starting point

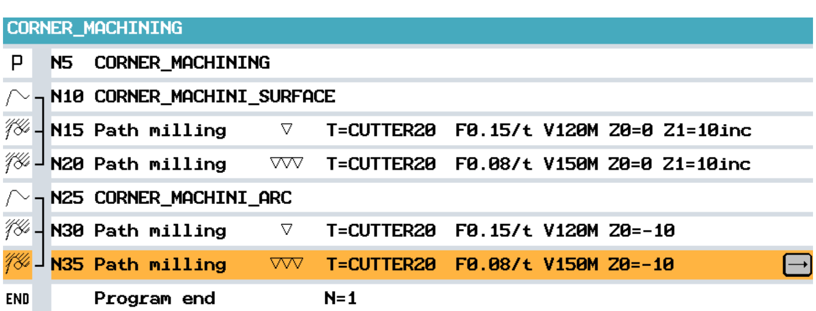


Enter the contour and associated work steps.

Enter the geometry

Technology for roughing the contour

Technology for finishing the contour



Complete subroutine consisting of the geometric and technology data

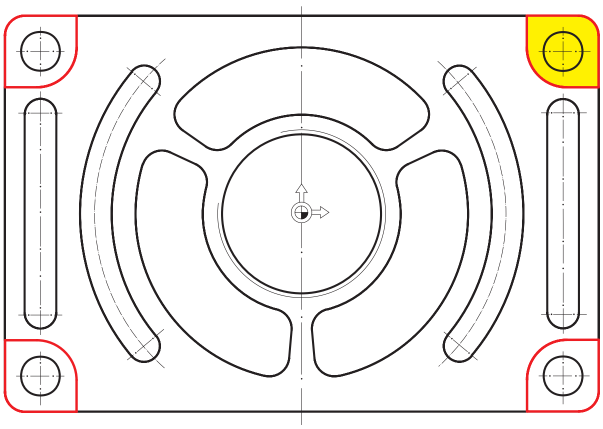
9.2 Mirroring work steps

When the subprogram is completed, the main program is then created. The *mirroring* function from the *Transformation* menu can be used for all four workpiece corners.

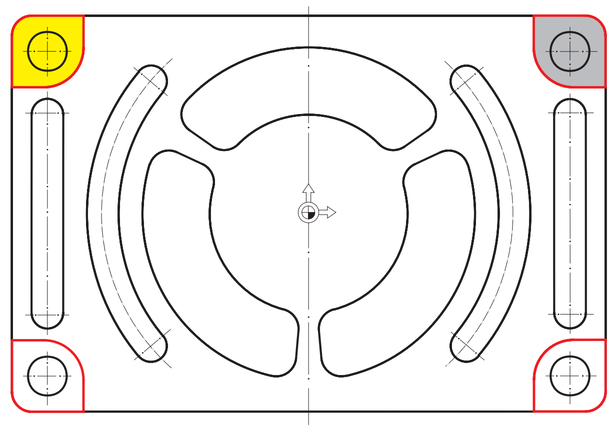
Mirroring can be performed in two different ways: *new* and *additive*
new means: mirroring is carried out from the location where the 1st machining step has been carried out.
additive means: mirroring is carried out from the location machined last.

The order of machining is outlined in the schematic below with the setting *new*:

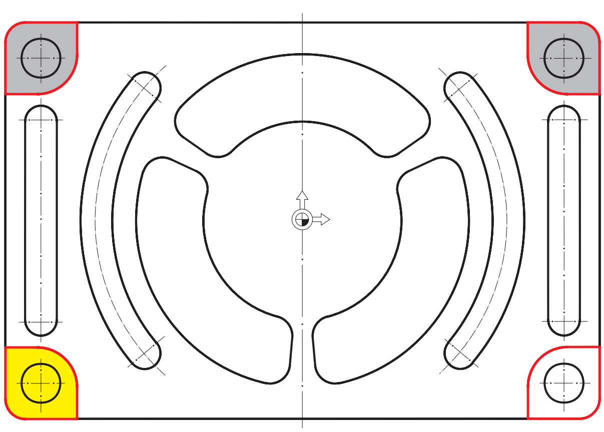
1. Machining (see subprogram)



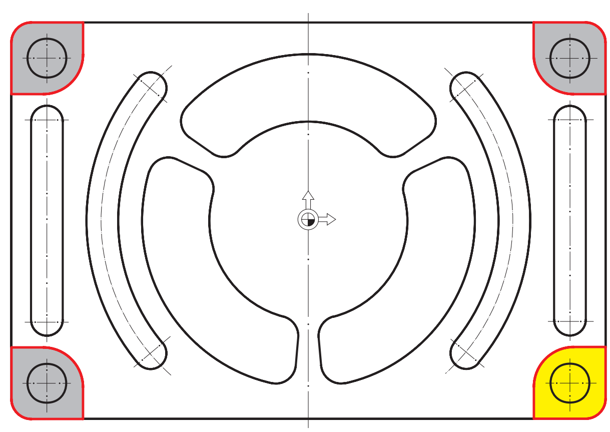
2. Machining: *Mirroring of the X axis*
(the X values are mirrored here)

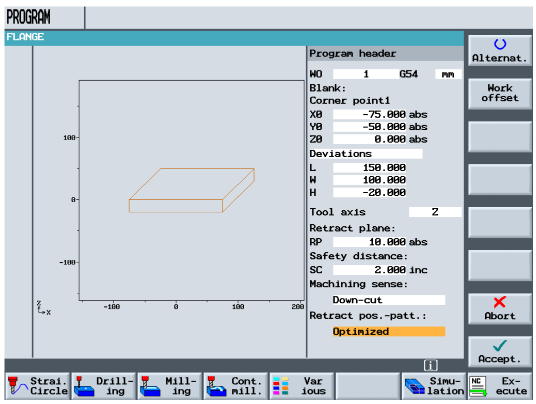
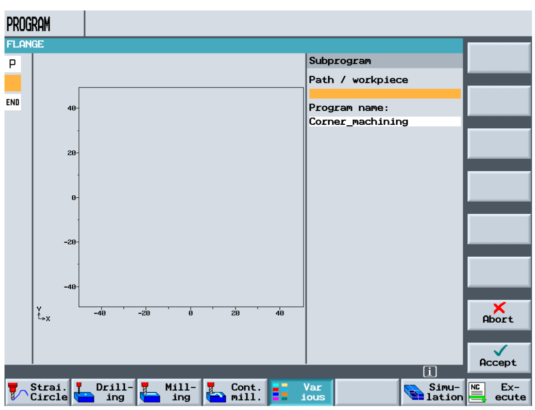
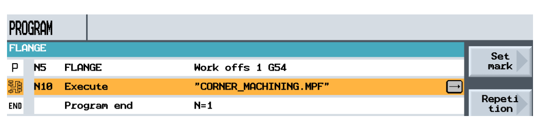
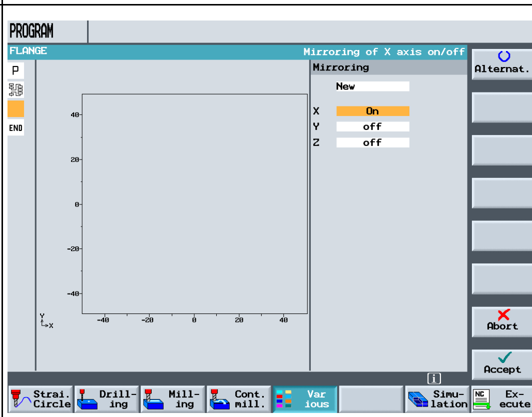


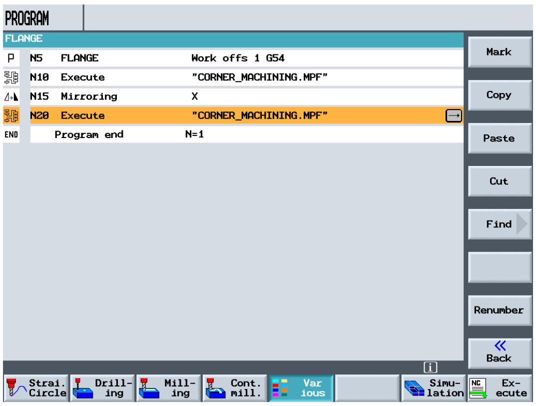
3. Machining: *mirroring of the X and Y axes*
(the X and Y values are mirrored here)



4. Machining: *mirroring of the Y axis*
(the Y values are mirrored here)

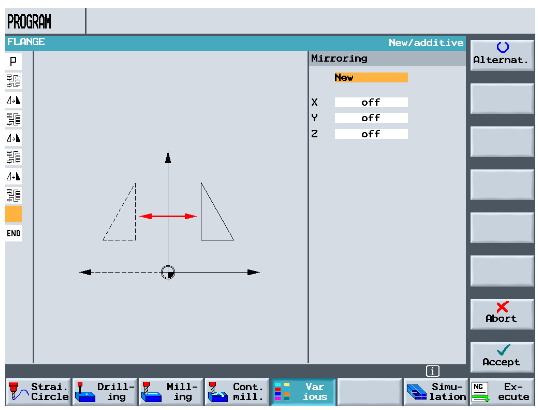


Keys		Screen	Explanations
Pro-gram New	F...		• The main program gets the name "Flange".
Accept			• The program header is entered.
Var ious Sub-program Accept	C...		• The *Various* key allows you to call the subroutine. • If the subroutine was created in the same directory as the main program, the input field "Path/workpiece" may remain empty. • Enter the name of the subroutine in the input field. ("Corner_machining").
Transfor-mations Mirror Accept			• The *Transformations* function can be used to shift, rotate and perform similar operations on the axes. • Preparation of the 2nd machining: Mirroring the X values



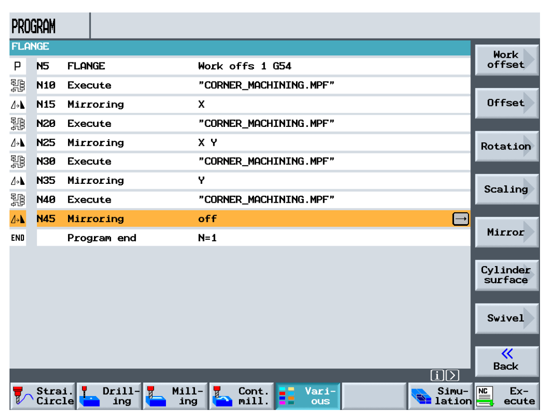
Then the subprogram behind the *mirroring* function is copied: The 2nd machining step.

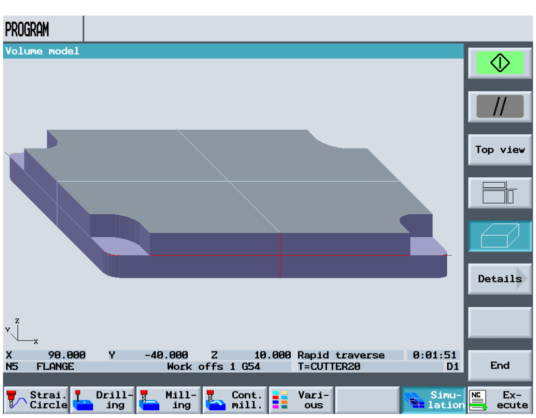
These processes *mirroring* and *subprogram call* are then repeated for the two other corners.



Auxilary display for *mirroring*

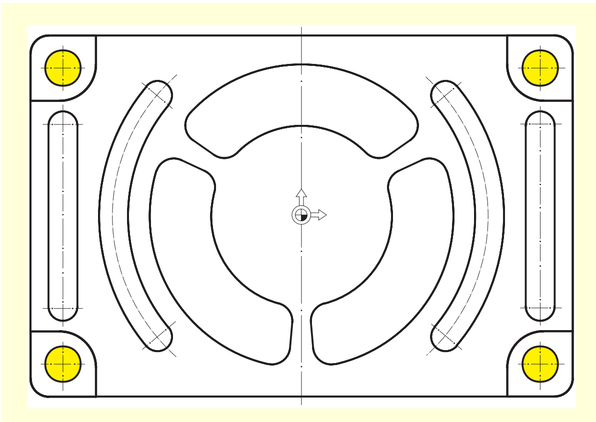
After the 4th machining step, the mirroring function is deactivated in all three axes (see line N45).

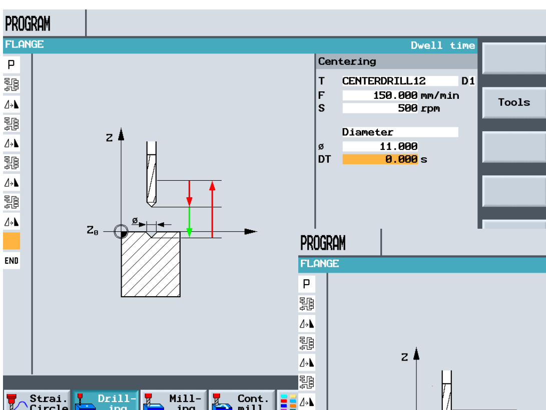




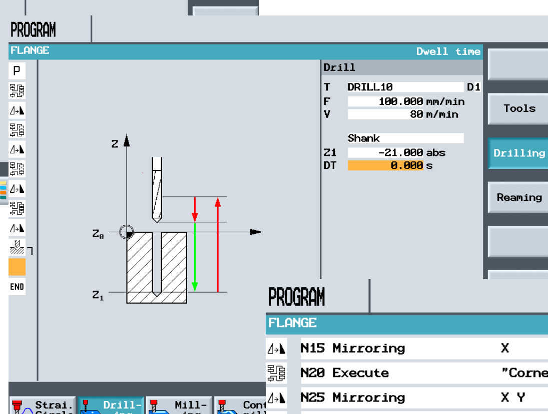
9.3 Holes

The next work steps create four holes at the corners. Since there is an obstacle between the individual holes, these must be entered between the positions.

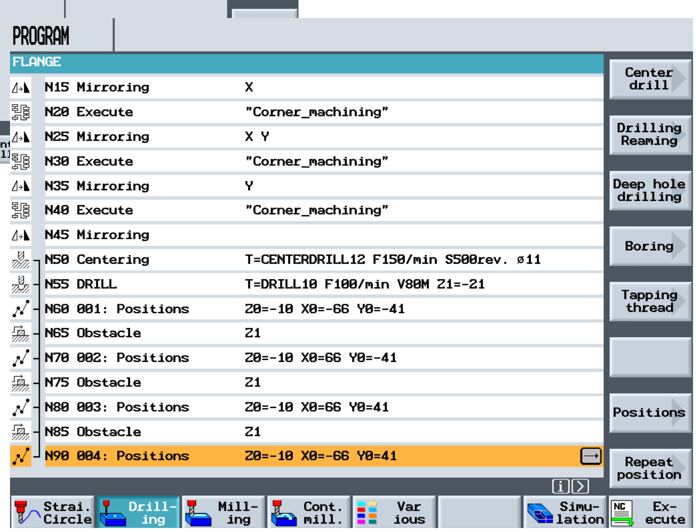




Technology for *Centering*

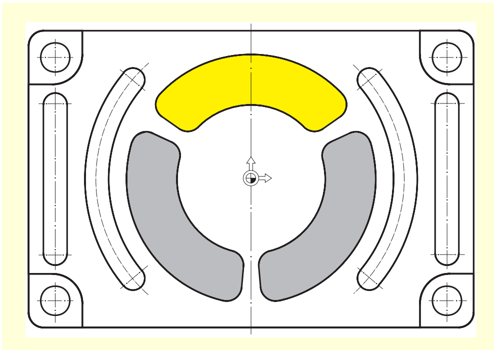


Technology for *Drilling*



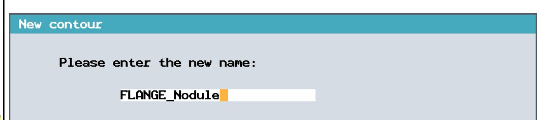
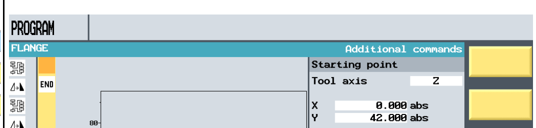
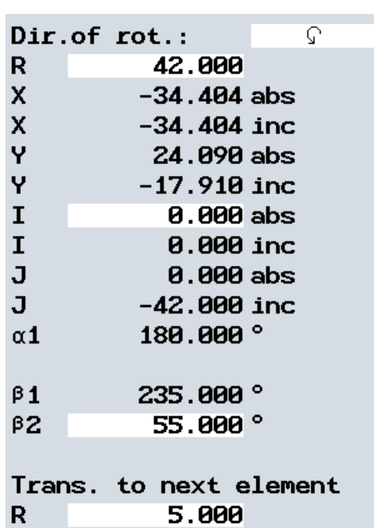
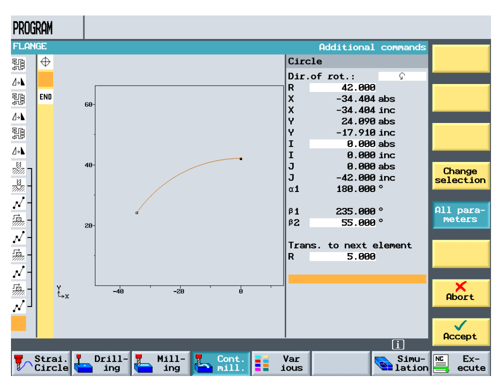
Enter the positions and obstacles.

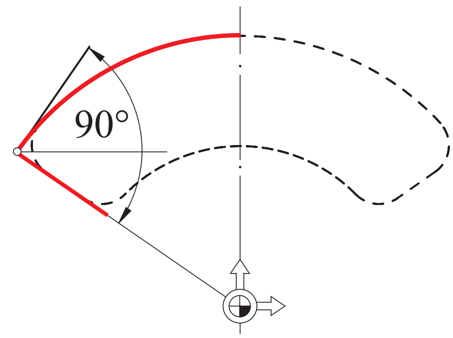
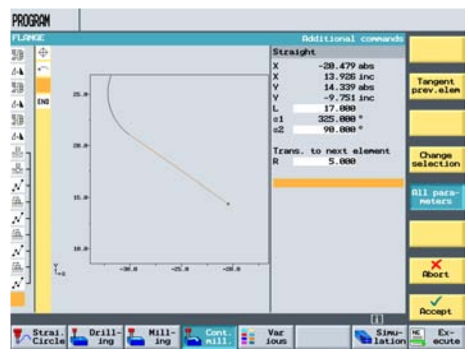
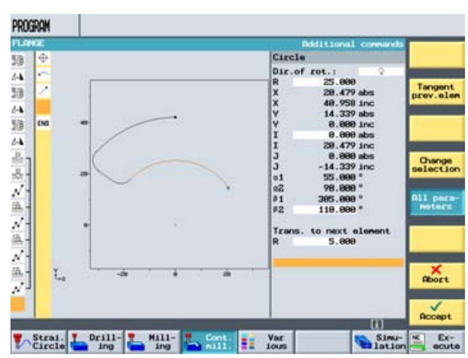
9.4 Rotation of pockets

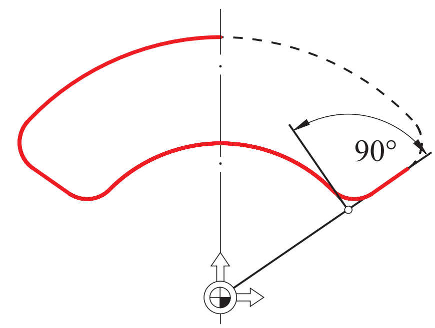
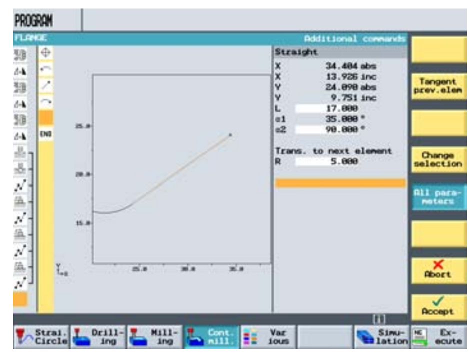
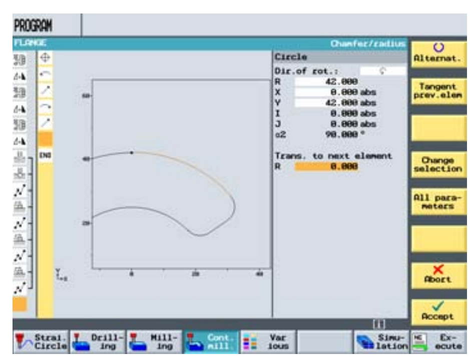


The contour and machining for the pocket highlighted in yellow are programmed below.

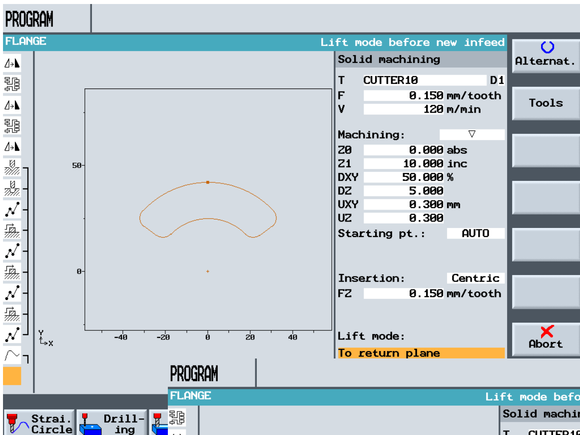
The two other pockets are created by rotating the coordinate system.

Keys		Screen	Explanations
Cont. mill. New contour	N...		• The contour is assigned the name "FLANGE_Nodule".
Accept	0 42		• Enter the starting point.
All para-meters Accept			• The arc R42 is described uniquely, for example via the radius, the center point in X and the run out angle. • The design is carried out counterclockwise so that the pocket can also be finished in synchronism. 

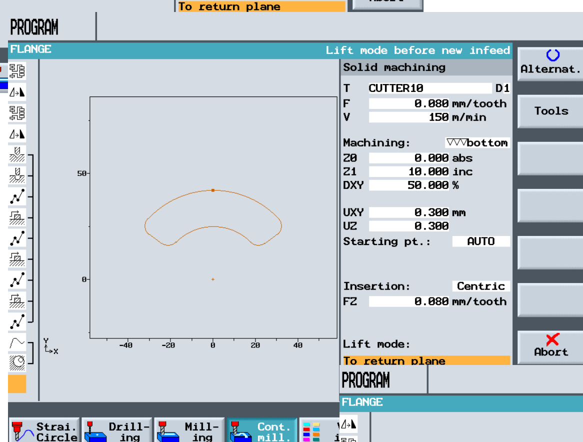
Keys	Screen	Explanations
 All para- meters Accept	X -20.479 abs X 13.926 inc Y 14.339 abs Y -9.751 inc L 17.000 α1 325.000 ° α2 90.000 ° Trans. to next element R 5.000	• Creating the diagonal route  
 All para- meters Accept	Dir.of rot.: R 25.000 X 20.479 abs X 40.958 inc Y 14.339 abs Y 0.000 inc I 0.000 abs I 20.479 inc J 0.000 abs J -14.339 inc α1 55.000 ° α2 90.000 ° β1 305.000 ° β2 110.000 ° Trans. to next element R 5.000	• Creating the 2nd arc 

 All para-meters Accept		X 34.404 abs X 13.926 inc Y 24.090 abs Y 9.751 inc L 17.000 α1 35.000 ° α2 90.000 ° Trans. to next element R 5.000	• Creating the 2nd diagonal route  
 Accept Accept		Dir.of rot.: R 42.000 X 0.000 abs Y 42.000 abs I 0.000 abs J 0.000 abs α2 90.000 ° Trans. to next element R 0.000	• Creating the final arc  • Take the contour pocket from the work plan.

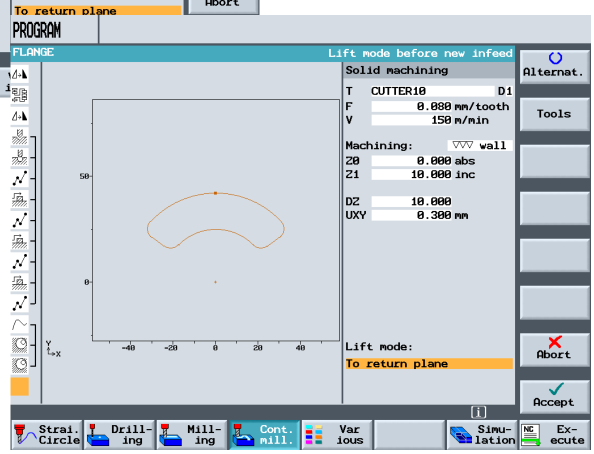
Create the following work steps on your own:



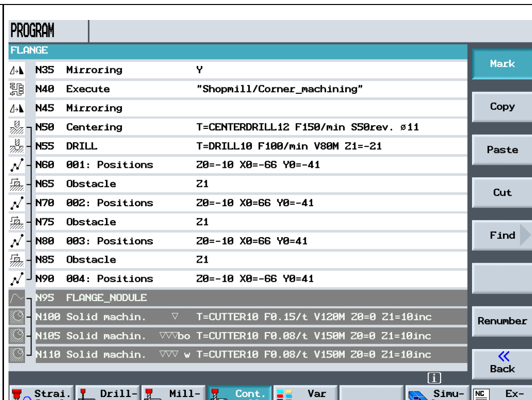
Rough the pocket



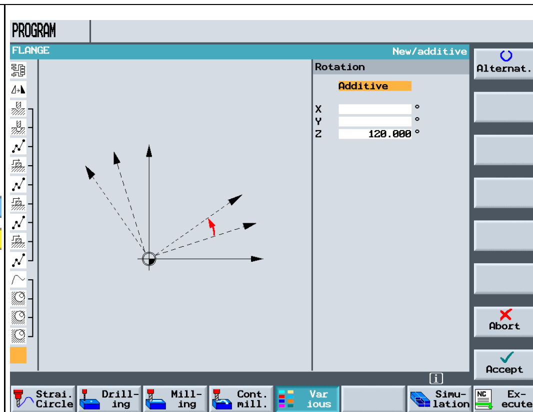
Finish the pocket base



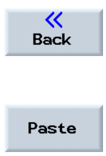
Finish the pocket edge

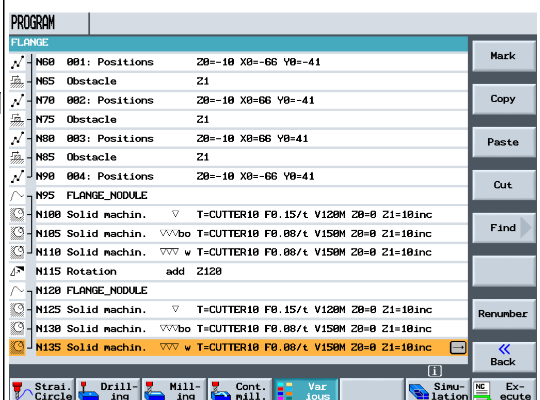
Mark Copy Back	> 3x ↑ 3x ↓		• Now mark and copy the complete work step to define the pocket machining to the clip-board.

3x 
120 

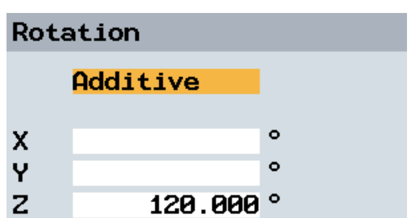


- Rotate the coordinate system 120° around the Z axis.

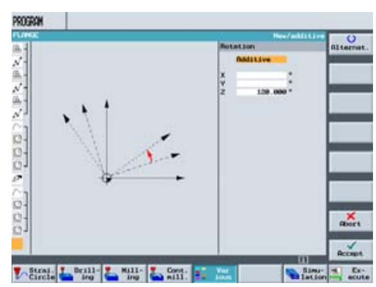


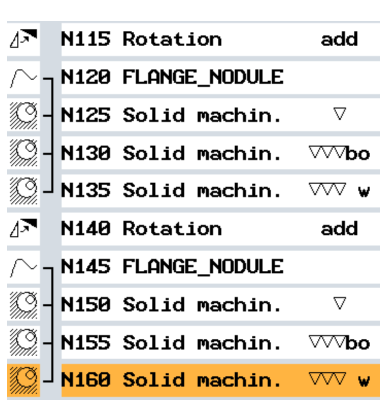


- Paste the copied work steps.

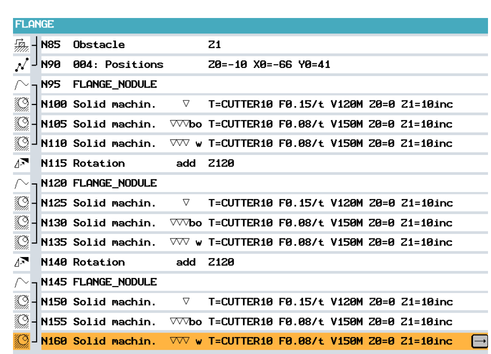


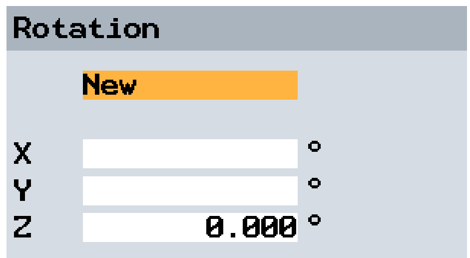
- Enter a further rotation of 120°.



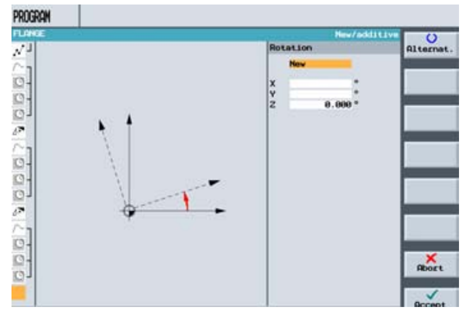


- Paste the work steps stored on the clipboard.





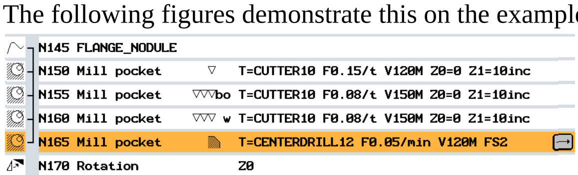
• Press *new* and the value 0° to cancel the rotation.



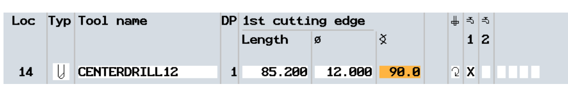
9.5 Chamfering contours

ShopMill version V6.4 and higher supports chamfering of contours. The selection field Machining - which is used for selecting roughing (▽), finishing (▽▽▽) etc. - has therefore been supplemented with the "Chamfering" option (Chamfer).

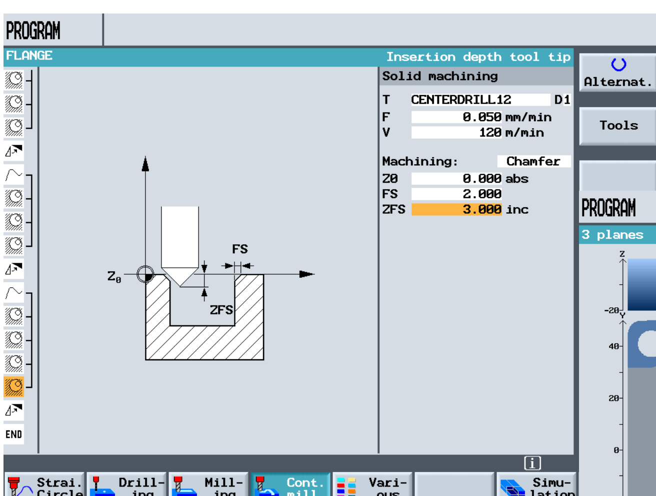
The following figures demonstrate this on the example of the last milled "nodule".



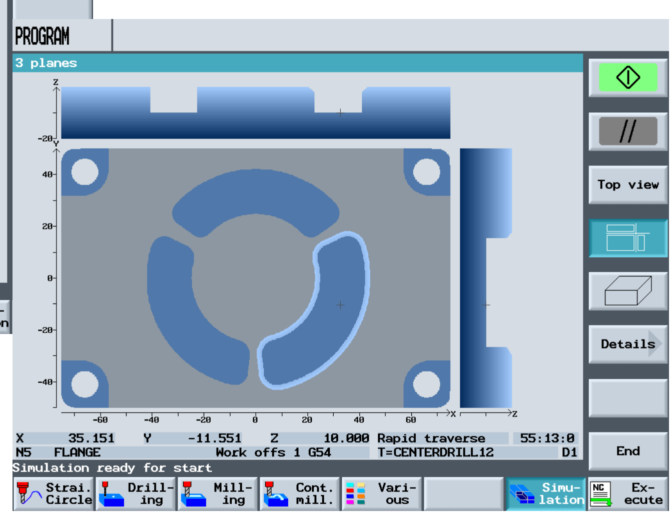
Work step "Chamfering" chained with the contour



A tool type is used which allows the input of a nose angle (here a center drill).

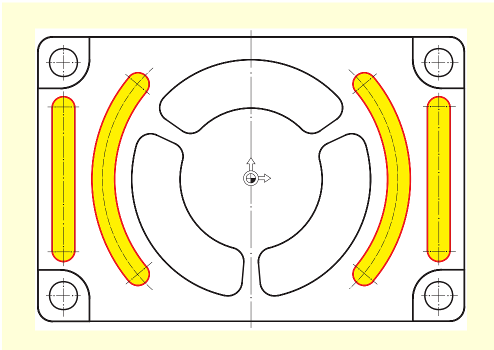


Machining of the chamfer is programmed via the chamfer width (FS) and the insertion depth of the tool tip (ZFS).

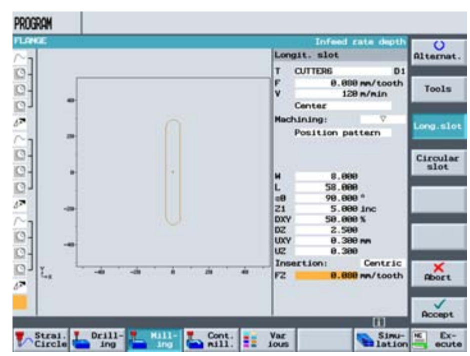
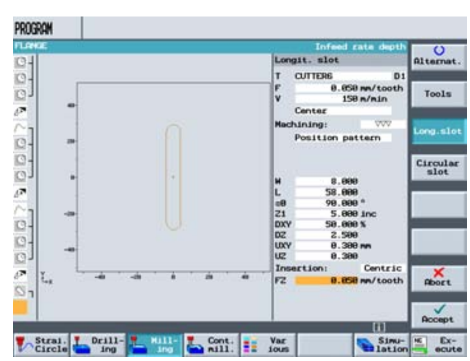


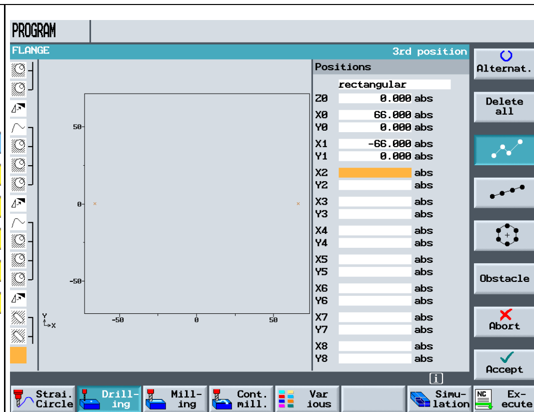
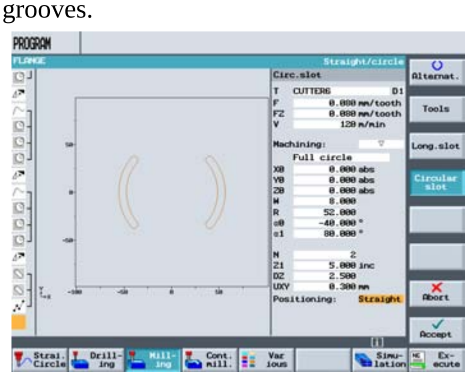
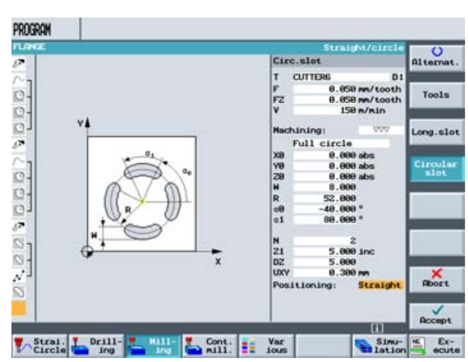
Chamfered contour in the 3-plane simulation

9.6 Longitudinal groove and circumferential groove

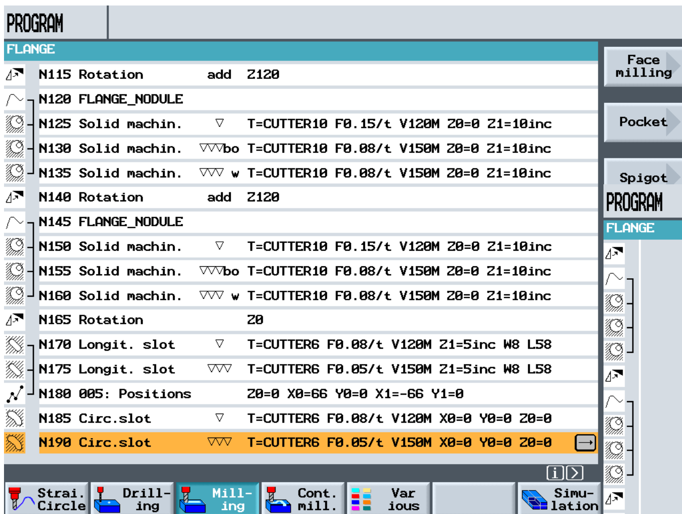


The grooves are programmed at the end. They must then be brought to the correct position via *position pattern* and positioning on a *full circle*.

Keys	Screen	Explanations
Mill-ing Slot Tool Accept	T CUTTER6 D1 F 0.080 mm/tooth V 120 m/min Center Machining: ▽ Position pattern W 8.000 L 58.000 α0 90.000 ° Z1 5.000 inc DXY 50.000 % DZ 2.500 UXY 0.300 mm UZ 0.300 Insertion: Centric FZ 0.080 mm/tooth	• The longitudinal grooves are roughed with the CUTTER6 tool (F 0.08 mm/tooth and V 120 m/min). 
Slot Accept	T CUTTER6 D1 F 0.050 mm/tooth V 150 m/min Center Machining: ▽▽▽ Position pattern W 8.000 L 58.000 α0 90.000 ° Z1 5.000 inc DXY 50.000 % DZ 2.500 UXY 0.300 mm UZ 0.300 Insertion: Centric FZ 0.050 mm/tooth	• The longitudinal grooves are finished with the same tool (F 0.05 mm/tooth and V 150 m/min). 

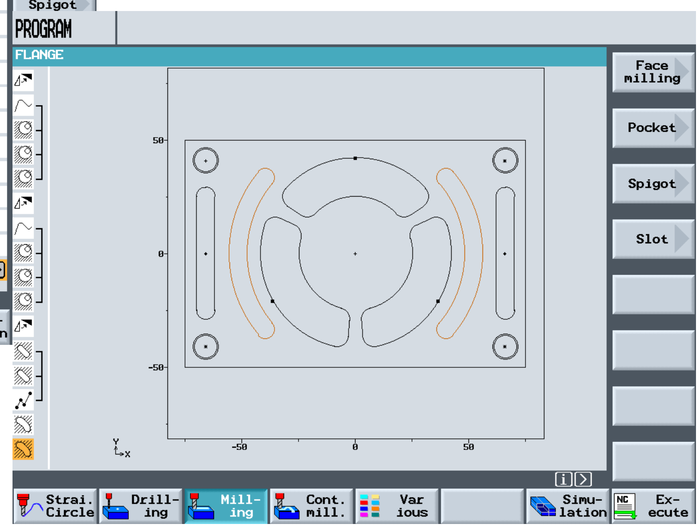
Drill-ing Positions Accept	0 66 0 -66 0		• Enter the two longitudinal groove positions • The reference point lies in the center of the groove.
Mill-ing Slot Circular slot Accept		T CUTTER6 D1 F 0.080 mm/tooth FZ 0.080 mm/tooth V 120 m/min Machining: ▽ Full circle X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 8.000 R 52.000 α0 -40.000 ° α1 80.000 ° N 2 Z1 5.000 inc DZ 2.500 UXY 0.300 mm Positioning: Straight	• The circumferential grooves are roughed using the CUTTER6 tool (F 0.08 mm/tooth and FZ 0.08 mm/gear tooth and V 120 m/min). • The *Full circle* option positions the circle automatically at a constant distance. • The reference point in X/Y/Z refers to the center point of the circle of circumferential grooves. 
Slot Circular slot Accept		T CUTTER6 D1 F 0.050 mm/tooth FZ 0.050 mm/tooth V 150 m/min Machining: ▽▽▽ Full circle X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 8.000 R 52.000 α0 -40.000 ° α1 80.000 ° N 2 Z1 5.000 inc DZ 5.000 UXY 0.300 mm Positioning: Straight	• The circumferential grooves are finished with the same tool (F 0.05 mm/tooth, FZ 0.05 mm/tooth and V 150 m/min). 

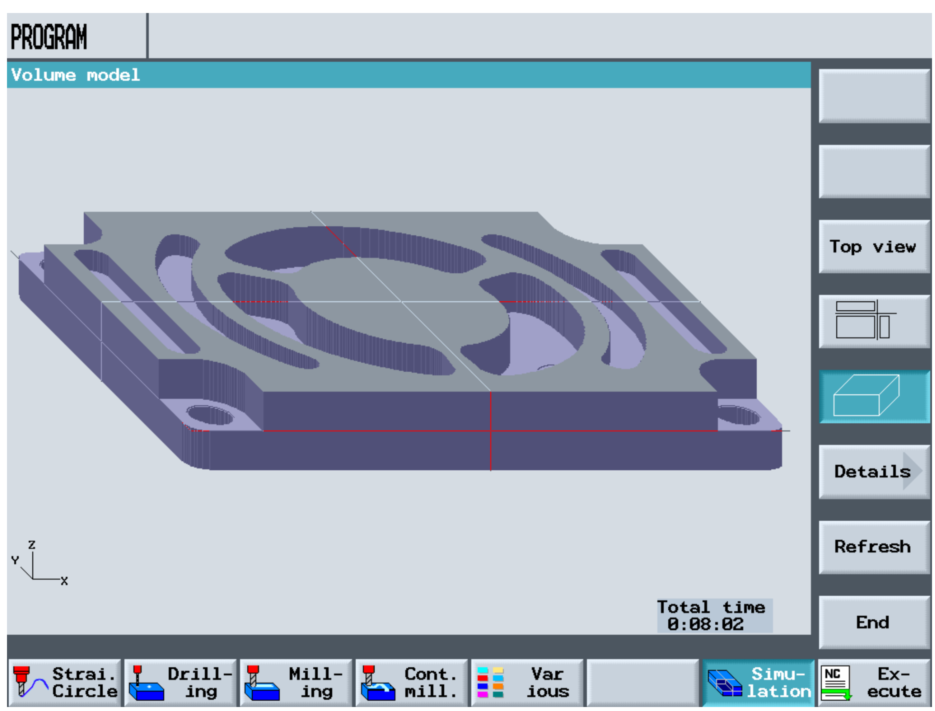
And finally: work plan, online graphique and 3D view



Excerpt from the work plan

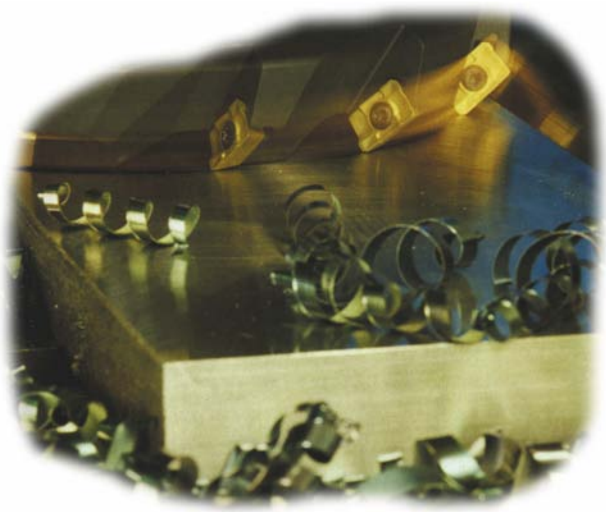
The online graphique





3D view

10 So now we can start



When you have acquired a sound knowledge of how to create a work plan with ShopMill by working through the examples, you can move on to produce workpieces.

10.1 Approach reference point

When you activate the control, you must approach the reference point before you run work plans or before you traverse manually. This enables ShopMill to find the counter starting point for the linear measurement system in the machine.

Since approaching the reference point may vary depending on the machine type and manufacturer, we can only provide a rough guide here:

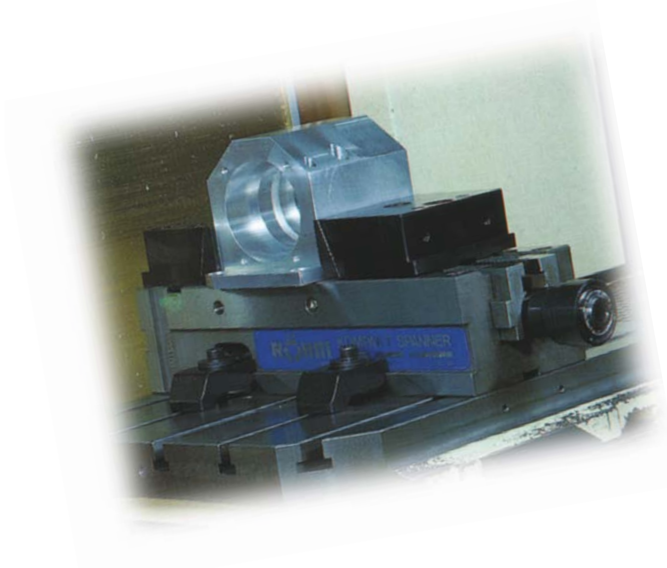
1. Move the tool to a free location in the work space, from which you can move in all directions without collision. When you do this make sure that the tool does not then lie behind the reference point of the relevant axis (since the reference point of each axis is only approached in one direction, it is otherwise not possible to reach this point).

2. Approach the reference point exactly according to the specifications of the machine manufacturer.

10.2 Clamp the workpiece

In order to ensure production true-to-dimension, and also for your safety, make sure that the workpiece is clamped firmly.

Normally, bolted machine blocks ...



... or metal clamps are used for this

10.3 Set the workpiece zero

Since ShopMill cannot guess where the workpiece is in the work area, you must determine the workpiece zero.

In the plane, the workpiece is usually set

- using the 3D key or
- with the edge key.

Symbol for workpiece zero W

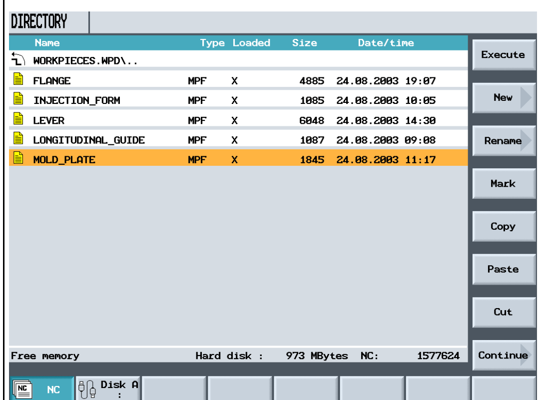
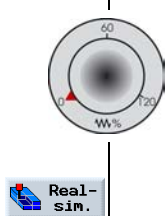
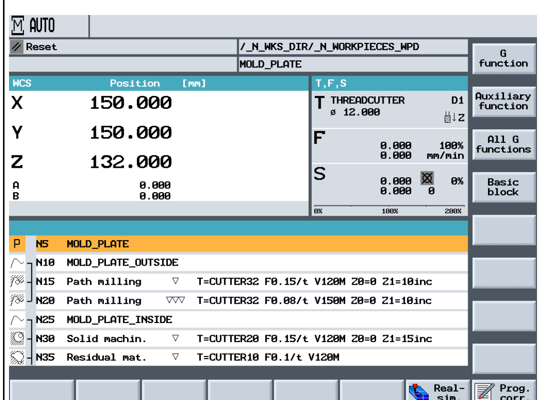
In the tool axis, the workpiece zero is usually set

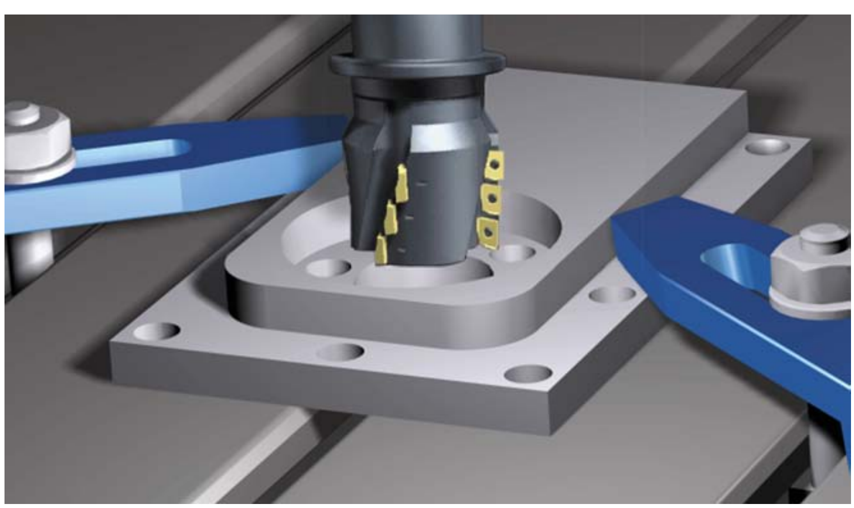
- by clicking the 3D key
- by scratching with a tool.

Please observe the instructions of the manufacturer when using measuring instruments or measuring cycles.

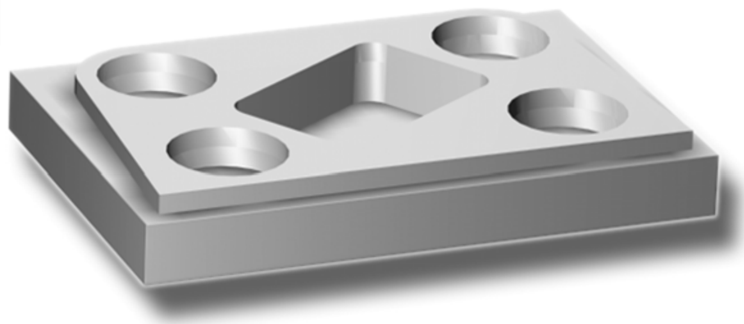
10.4 Edit work plan

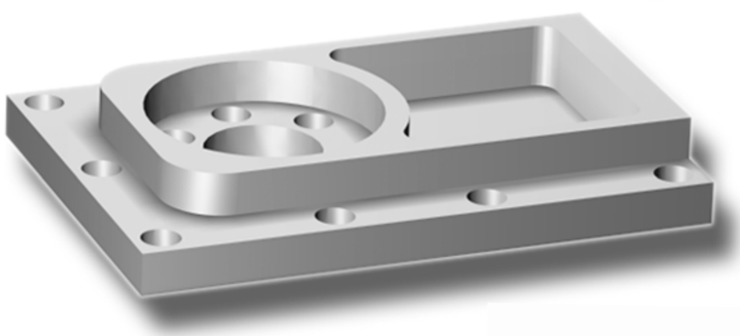
The machine is now ready, the workpiece set up and the tools calibrated (see Chapter 4). Now you can get started:

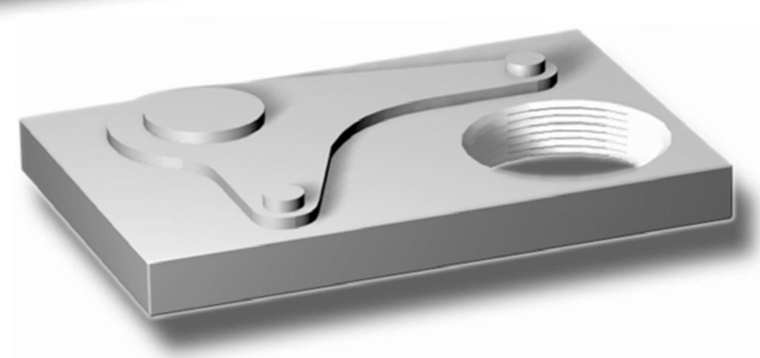
Keys	Screen	Explanations
		• Select the directory that contains the work plan you wish to use. The directory for the examples from this manual is WORKPIECES. • The *Execute* key loads the work plan in *AUTO* mode and switches to it.
		• If you want to see a simulation during production, you must select the *Real-sim.* function before you start. Only then all traversing movements and their effects are displayed. • Since the work plan has not yet been run and checked, set the feedrate override to zero so that you are "in control" from the start. • Start production with the  key and check the speed of the tool movements using the feedrate override.

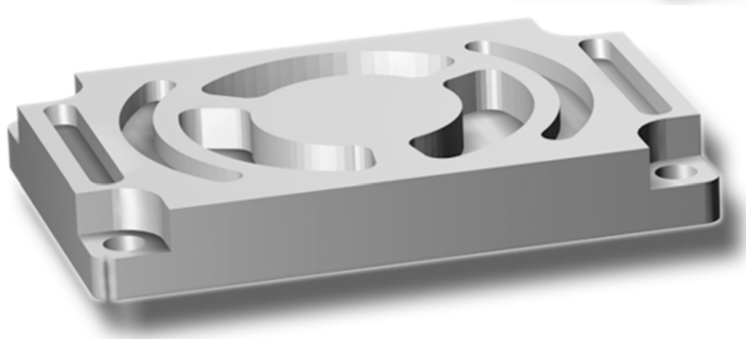


The speed and simplicity with which ShopMill has produced these workpieces...







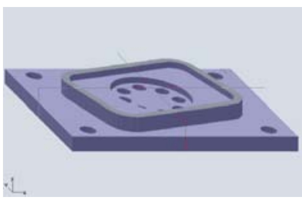


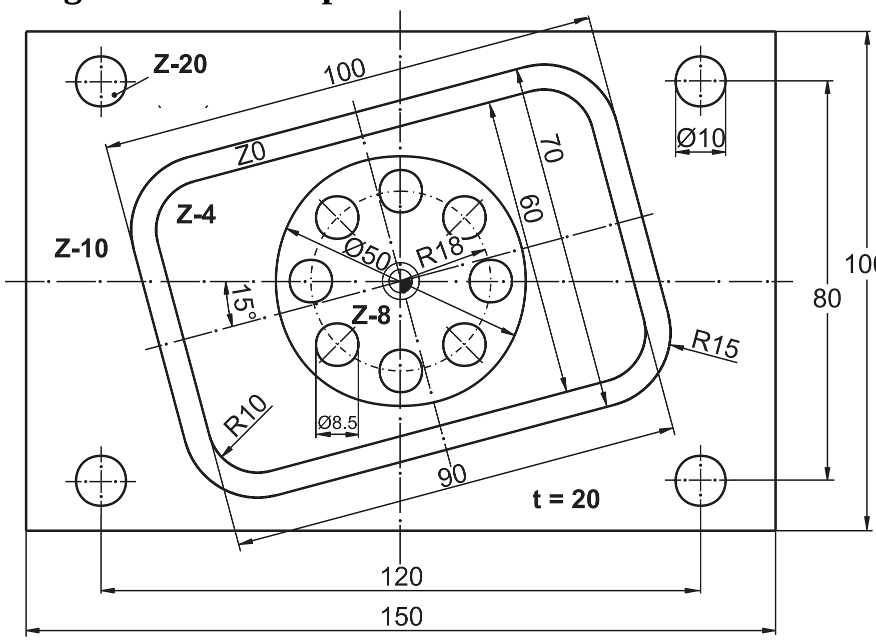
... will now apply to the workpieces YOU produce with ShopMill.

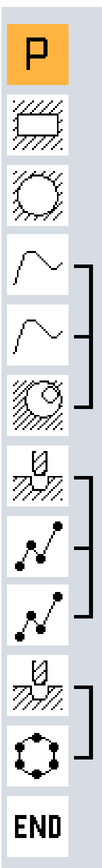
11 How fit are you with ShopMill?

The following 4 exercises form the base for your personal test in your work with ShopMill. A possible work plan is displayed to assist you in each case. The times stated are based on the procedure defined in the work plan. Please regard the times stated as a rough estimate for your answer to the question above.

Exercise 1: Can you manage that with ShopMill in 10 Minutes?





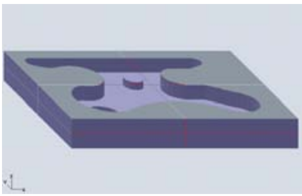


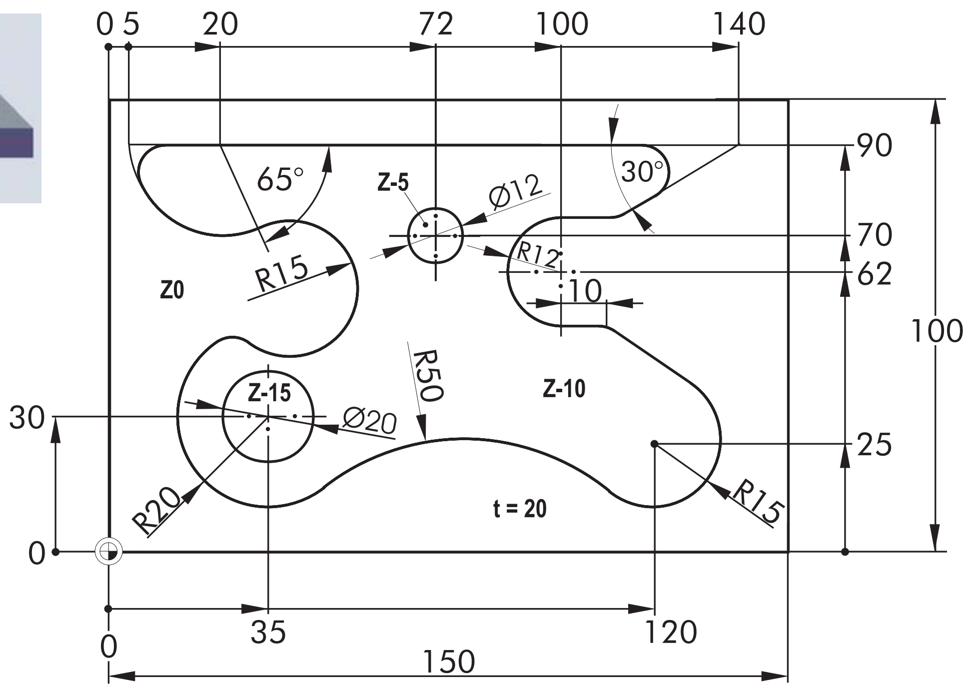

```
DIYS1
P    N0  DIYS1
     N5  Rectang.pocket   ▽   T=CUTTER20 F0.08/t V150M X0=0 Y0=0 Z0=0
     N10 Circ. pocket     ▽   T=CUTTER20 F0.08/t V150M X0=0 Y0=0 Z0=-4
     N15 DIYS1_AR_150_100_5_15_15
     N20 DIYS1_IR_100_70_15
     N25 Solid machin.    ▽   T=CUTTER20 F0.08/t V150M Z0=0 Z1=10inc
     N30 DRILL                T=DRILL10 F150/min V35M Z1=-20
     N35 001: Positions       Z0=-10 X0=60 Y0=40 X1=60 Y1=-40
     N40 002: Positions       Z0=-10 X0=-60 Y0=-40 X1=-60 Y1=40
     N45 DRILL                T=DRILL8.5 F150/min V35M Z1=-20
     N50 003: Hole full cir.  Z0=-8 X0=0 Y0=0 R18 N8
END      Program end
```

The rotated rectangular pocket has been constructed in the original coordinate system here. The start point initially lies on the zero point. An auxiliary straight line at 15° up to the edge of the pocket. The coordinates of this end point are the starting point for the actual construction. The auxiliary straight line must be deleted.

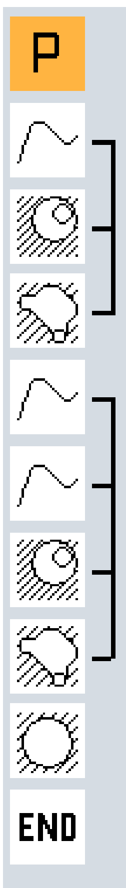
ShopMill also provides other ways to achieve this goal, e.g. with *Rotation* function or with the cycle *rectangular spigot* (see Exercise 3). Test which way is quickest for you and this procedure brings you the shortest production time.

Exercise 2: Can you manage that with ShopMill in 15 minutes?





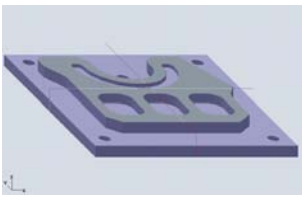
All non-dimensioned radii R6

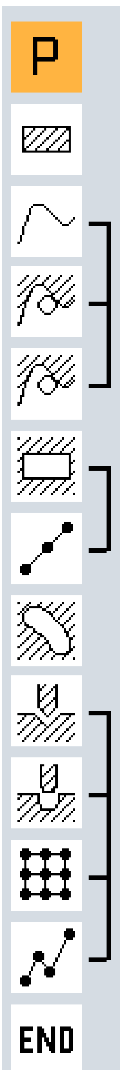


COMPLEX_POCKET			
P	N0	COMPLEX_POCKET	
	N5	COMPLEX_POCKET_1	
	N10	Solid machin. ▽	T=CUTTER20 F0.08/t V150M Z0=0 Z1=5inc
	N15	Residual mat. ▽	T=CUTTER10 F0.08/t V120M
	N20	COMPLEX_POCKET_1	
	N25	COMPLEX_POCKET_D12	
	N30	Solid machin. ▽	T=CUTTER20 F0.08/t V150M Z0=-5 Z1=5inc
	N35	Residual mat. ▽	T=CUTTER10 F0.08/t V120M
	N40	Circ. pocket ▽	T=CUTTER10 F0.08/t V120M X0=35 Y0=30
END		Program end	

Even if it looks complicated, this contour presents no problem to ShopMill. And the automatic stock removal for residual material can be applied with optimum results here. Compare the production times if you were to remove all that with CUTTER10.

Exercise 3: Can you manage that with ShopMill in 20 minutes?





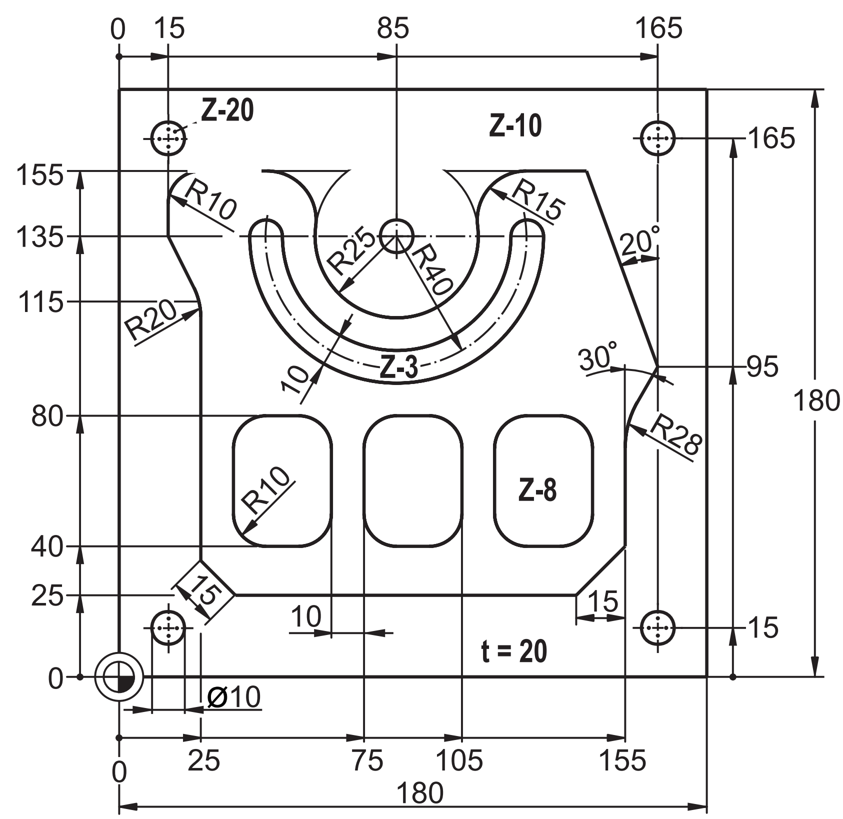
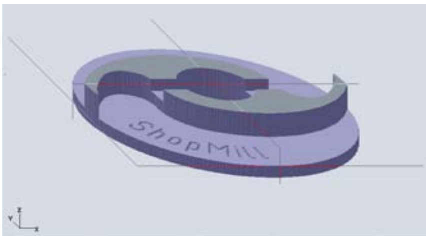
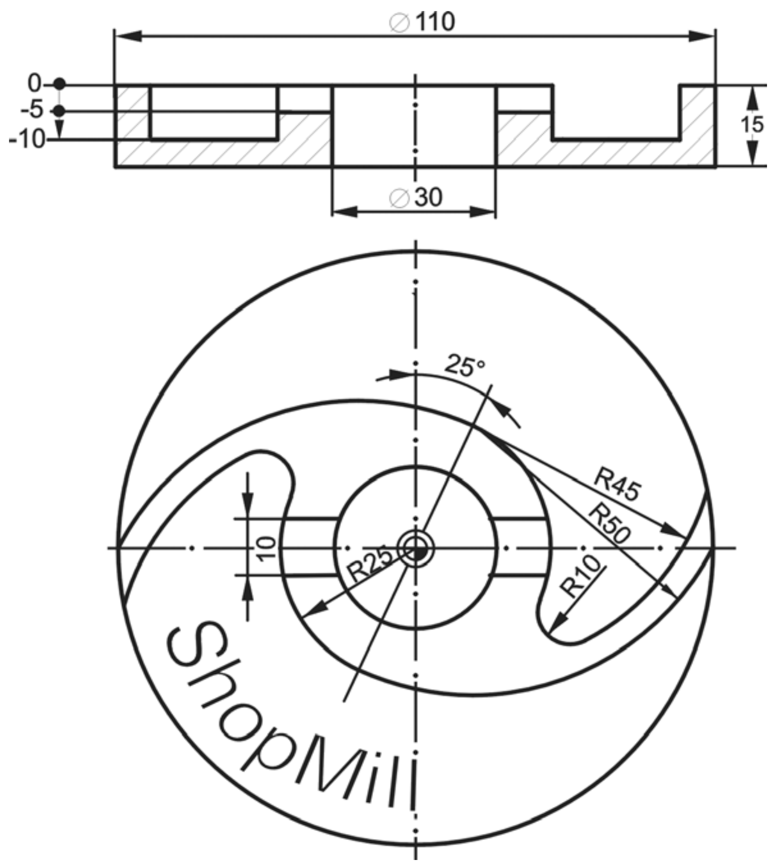


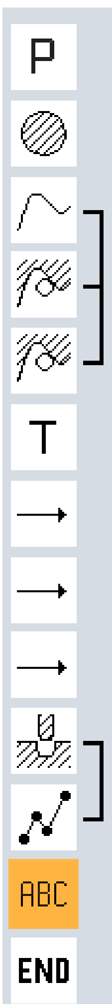
PLATE			
P	N0	PLATE	
	N5	Rectang.spigot ▽	T=CUTTER32 F0.08/t V150M X0=90 Y0=90
	N10	PLATE_ISLAND	
	N15	Path milling ▽	T=CUTTER32 F0.08/t V150M Z0=0 Z1=10inc
	N20	Path milling ▽▽▽	T=CUTTER20 F0.06/t V150M Z0=0 Z1=10inc
	N25	Rectang.pocket ▽	T=CUTTER20 F0.08/t V150M Z1=8inc W40 L30
	N30	001: Row of holes	Z0=0 X0=50 Y0=60 N3
	N35	Circ.slot ▽	T=CUTTER6 F0.08/t V150M X0=85 Y0=135
	N40	Centering	T=CENTERDRILL12 F150/min S550rev.
	N45	DRILL	T=DRILL10 F150/min V35M Z1=12inc
	N50	002: Hole grid	Z0=-10 X0=15 Y0=15 N1=2 N2=2
	N55	003: Positions	Z0=-10 X0=85 Y0=135
END		Program end	

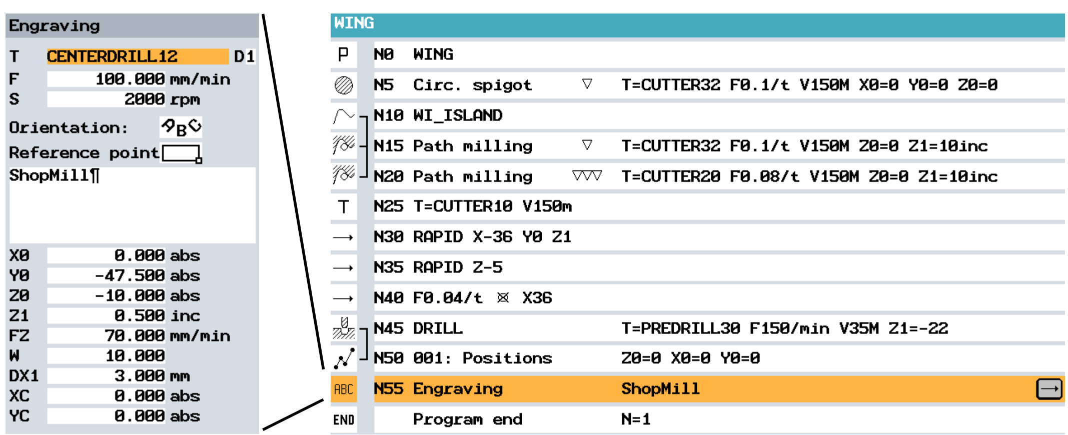
In this sample work plan, the surface around the island is first pre-milled roughly with the *rectangular spigot* cycle from the *Milling* menu. The rectangle described in this cycle is approached in circular motion and reaches the contour at the point described by the *length* and *angle of rotation*. The tool travels around the island once and exits at the same point again in a circle. The approach radius and return radius are obtained from the geometry of the remaining spigot.

Exercise 4: Can you manage that with ShopMill in 20 minutes?









In this sample work plan, the circular outside contour has been milled using circular outside contour and the *Circular spigot* cycle. The functional operation corresponds essentially to the rectangular spigots (see sample work plan for Exercise 3). The common center-point of the two arcs R45 and R50 (= starting point for the actual construction) is determined via polar coordinates (25 mm under 65°, relative to the pole point at X0/Y0, cf. Section 8.13).

From software version V6.4, a flexibly usable *Engraving* cycle is available under the *Milling* menu.

Index

Index of figures

We are grateful to

AMF

DMG

Verlag Europa-Lehrmittel

Haimer

Iscar

Krupp-Widia

Neumo

Reckermann

Renishaw

Röhm

Sandvik

Seco

for the provision of graphic material on pages 17, 18, 24, 26, 27, 29, 33, 81, 102 and 103.

Further information

More details on JobShop are available under:
www.siemens.com/jobshop

For detailed technical documentation refer to our Service&Support portal:
www.siemens.com/automation/support

For contact persons near you refer to:
www.siemens.com/automation/partner

Direct online ordering is possible in our mall:
www.siemens.com/automation/mall

Siemens AG
Industry Sector
Drive Technologies
Motion Control
Postfach 3180
91050 Erlangen
DEUTSCHLAND

www.siemens.com/sinutrain

Subject to change without prior notice
6FC5095-0AA50-0BP2