TwinCAM
TwinCAM Tutorial

Info

Company address:

Raabe Software
Kutscherweg 23
32312 Lübbecke
Deutschland

Telefon: 05741 / 310 304
Telefax: 05741 / 310 327

web: www.raabe-software.de
eMail: support@raabe.software.de

Table of Content

Chapter
I

Parametric drawings

Parametric drawings

1.1 Creating a parametric drawing

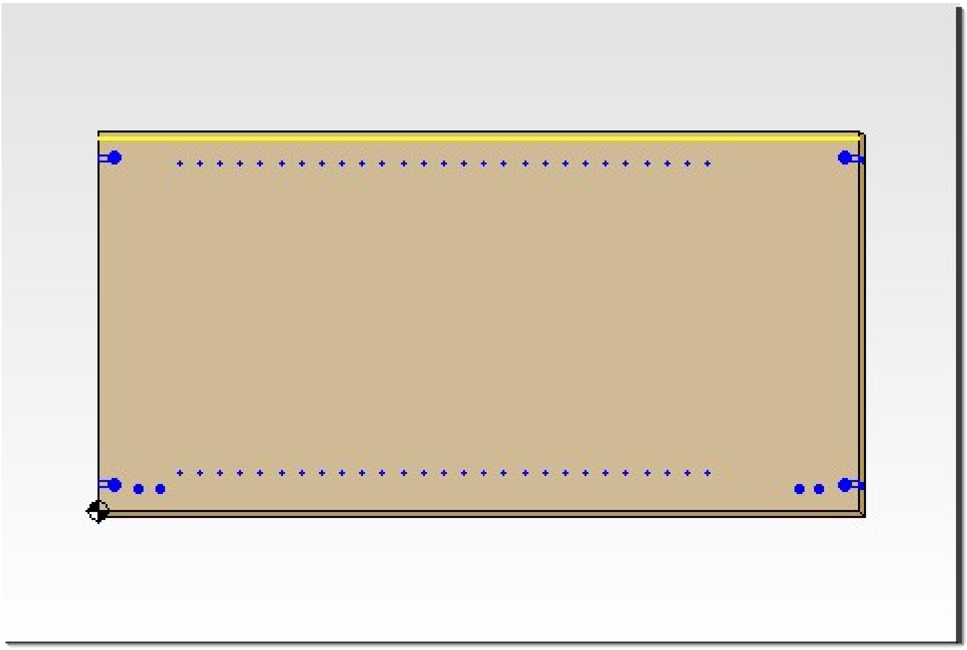
Creating a parametric drawing will be explained by the following example. The example is not completely realistic, but serves to convey a range of definitions for a variety of machining methods. The goal is to complete the side panel for a cabinet with drillings for camlock bolts, drawer runners at 32mm intervals and hinge mounts. The panel will need to be formatted on all sides to finished dimensions on completion. The following are the required dimensions for the various machining operations:

Panel size:	X = 1200; Y = 600; Z = 16
Shelf support:	Ø = 5 mm; Depth = 8 mm
Hinge mounts:	Ø = 10 mm; Depth = 12 mm
Back panel rebate:	Depth = 6 mm; Width = 3.4 mm

Camlock bolts location:

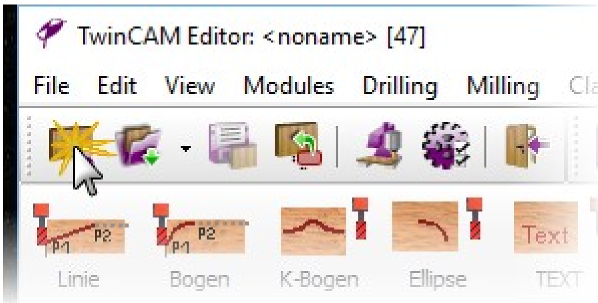
X axis:	25 mm
Y axis:	43 mm from outer edge
Depth:	12 mm
Z axis:	center of panel

The result should like something like this:



1.2 Creating a panel

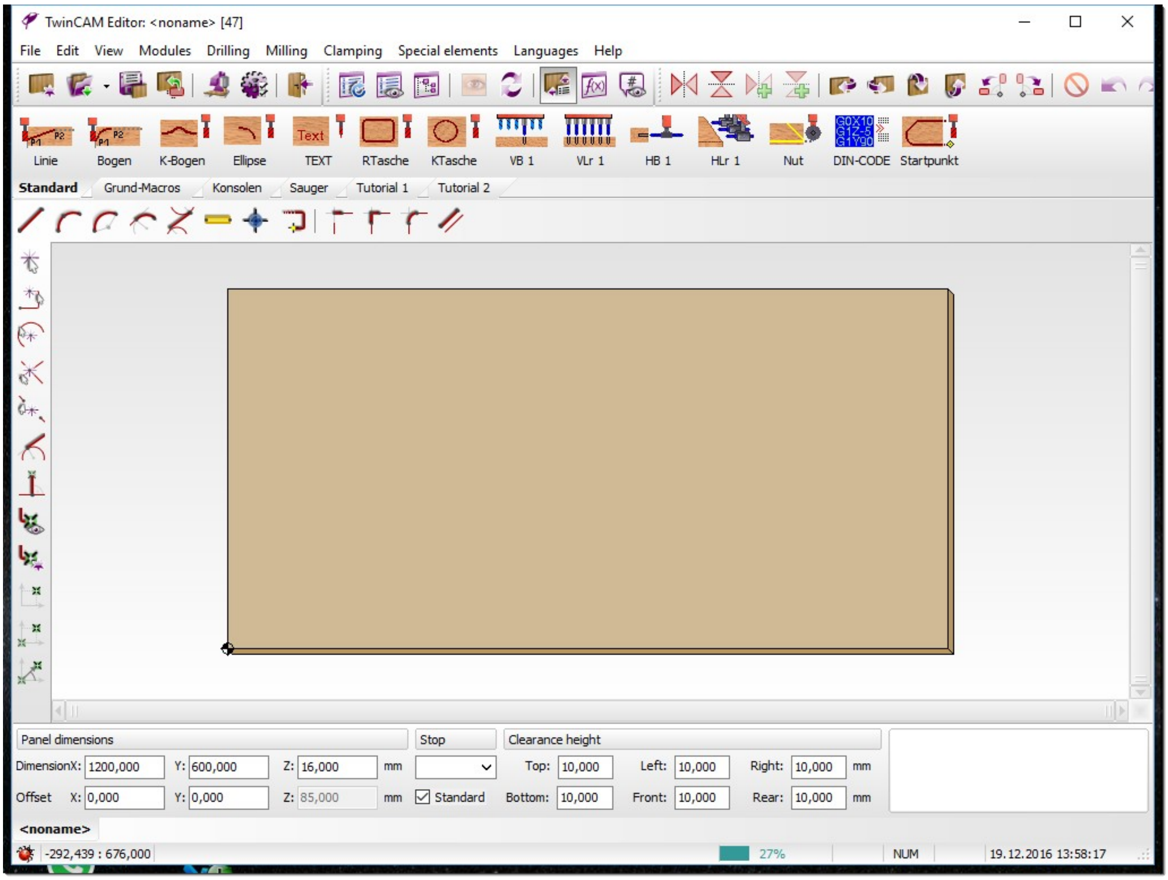
First we have to create a new workpiece. Select "New panel" from the multi-task bar at the top of the screen.



If any changes have not been saved, a respective question follows that we answer according to the corresponding situation.

Now enter the panel dimensions in the appropriate boxes on the base bar.

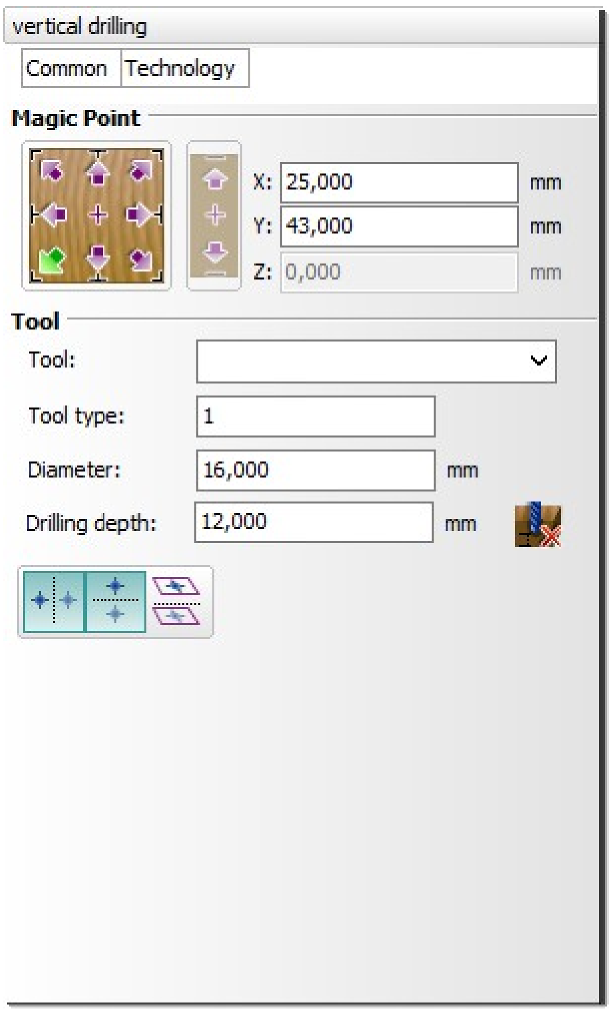
X	Y	Z
1200 mm	600 mm	16 mm



1.3 Camlock bolts - vertical element

Next, we create the drillings for the four Minifix-connectors. Each of them consists of a vertical and a horizontal drilling. We click on "Vertical drilling" in the Standard tab to define the vertical elements of the camlock bolts.

Select the bottom left corner of the panel as the "Magic point" by clicking on the bottom left arrow of the nine surrounding the grey square. This is found at the top right of the screen titled "Position". Now enter the co-ordinates for the drilling (X=25, Y=43) in the appropriate boxes. All the required drillings are laid out symmetrically so the remaining drillings can be created by selecting the mirror check boxes, both horizontal and vertical.



Once the locations for the drillings have been defined, the hole dimensions need to be entered. These are depth 12mm, diameter 16mm and the tool type is 1. Leave the cycle field blank. Confirm the entries by clicking on "OK".

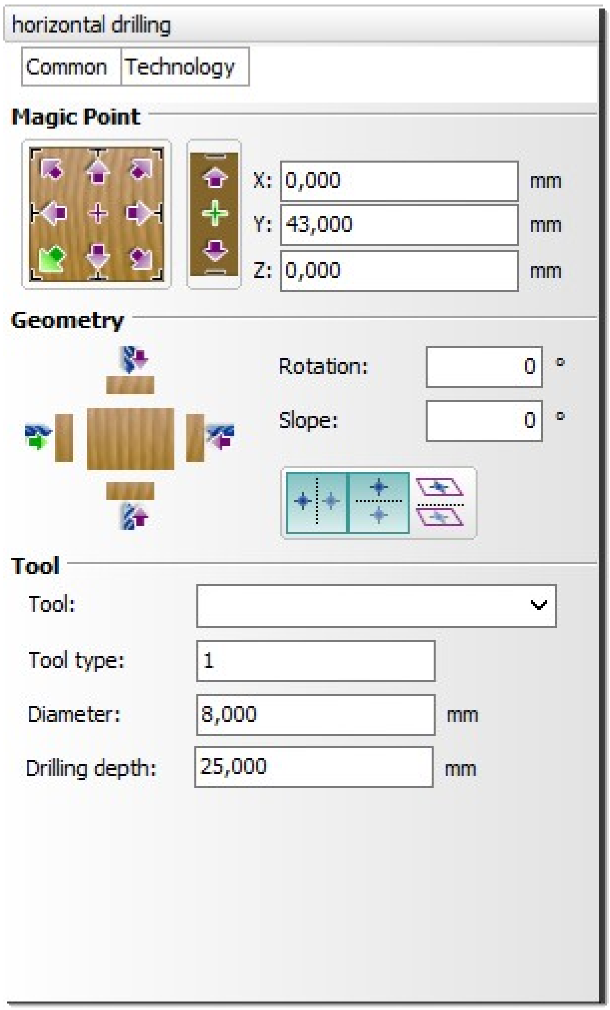
1.4 Horizontal drilling

Click on "Horizontal drilling" in the Standard tab to define the horizontal elements of the camlock bolts.

It is recommended to define the drilling direction for the horizontal drilling (here from left to right) before entering other data. After that we define the reference point via the "Magic Point" as we did before for vertical drillings. For logical reasons we also choose front left. The entry X=0 and Y=43 means that the horizontal drilling is exactly on the left hand side of the workpiece edge and 43 mm distant from the frontal workpiece edge.

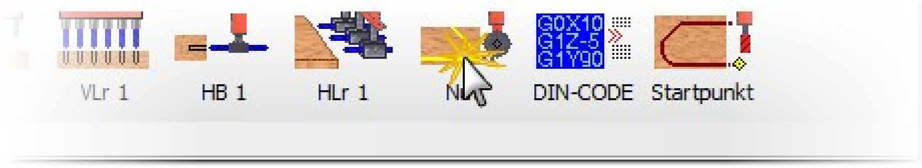
In contrast to a vertical drilling we also add a value for Z which defines the position of the drilling on the front side of the panel. We type in 0 in order to place the drilling in the middle of the panel thickness.

Now we add the values for tool type (1), the diameter of the drilling (8) and the depth (25) and we mirror the drilling in X and Y by clicking on the corresponding buttons. Confirm all entries by clicking on OK.



1.5 Back panel rebate

Now select the "Rebate" tab from the Standard menu.

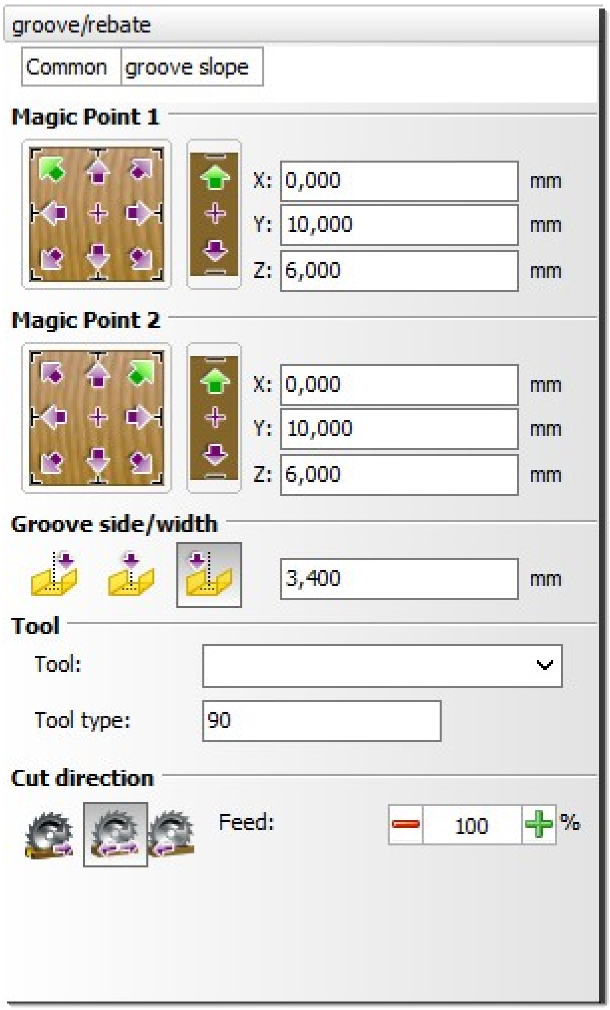


Set the "Magic points" using the "Magic points" selector. As the rebate is continuous, select the top left-hand corner as the starting point and the top right-hand corner as the finishing point. Now enter the co-ordinates for point one as X=0, Y=10, Z=6 and the same for point 2. The rebate is to run 10mm from the top edge of the panel, with a depth of 6mm.

So far we have just described a line from left to right that runs 10 mm from the top edge of the panel. Only by typing in slot width (3,4) and slot position we accomplish the desired rear panel groove. The slot position defines wether the slot width is on the right or the left side or in the middle of the line. The direction of the line is always viewed from point 1 to point 2.

We add the tool type (90) and the direction of cutting (random). If the machine provides this option both cutting directions can be selected.

Once all the entries have been made correctly, click on the "Save" button. This saves the current settings, and the next time the rebate tab is opened, the settings will be available. This is particularly useful when the same rebate is going to be cut in a large number of panels.



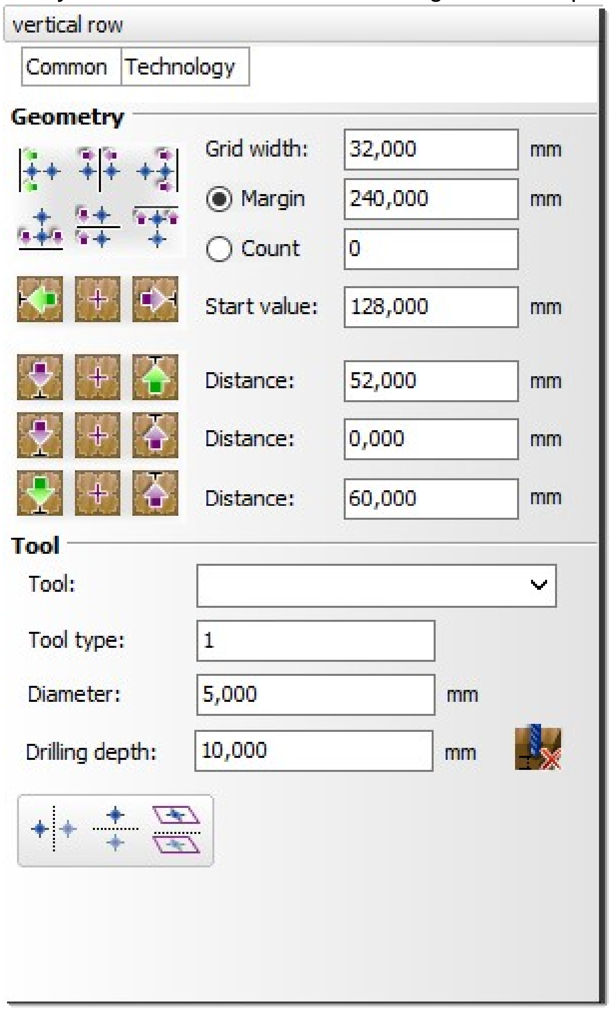
1.6 Shelf supports - a vertical drilling row

First open the vertical row tab in the Standard menu.

We picture a drilling row like a text line in Microsoft Word – it is either left-adjusted, centered or right-adjusted. If left-adjusted we define the position of the very left drilling; if right-adjusted the very right drilling and if centered the position of the middle of the drilling row (in the case of an odd number of drillings this is the middle drilling, otherwise it lies right between two drillings. If we also take drilling rows into consideration which run in Y-direction, they either run from front to back, from back to front or from the middle to front or back. These six buttons in Geometry stand for these six directions. In our case we select a left-adjusted direction.

The remaining Magic Point buttons conform to the selected direction. We define the Start value as 128 mm from the left edge. The length of the drilling row should leave a right margin of 128 mm – so we first click on Margin and type in 128 mm in the field to the right. For Grid width we enter 32 mm.

In the three Distance fields below we select for the first distance a reference from the back with a distance value of 52 mm and for the third a reference from front with a distance value of 60 mm. The middle distance is deselected by entering no reference. (Actually the array of the distance blocks do not matter. However, the array selected here is based on the geometrical position of the drilling row).

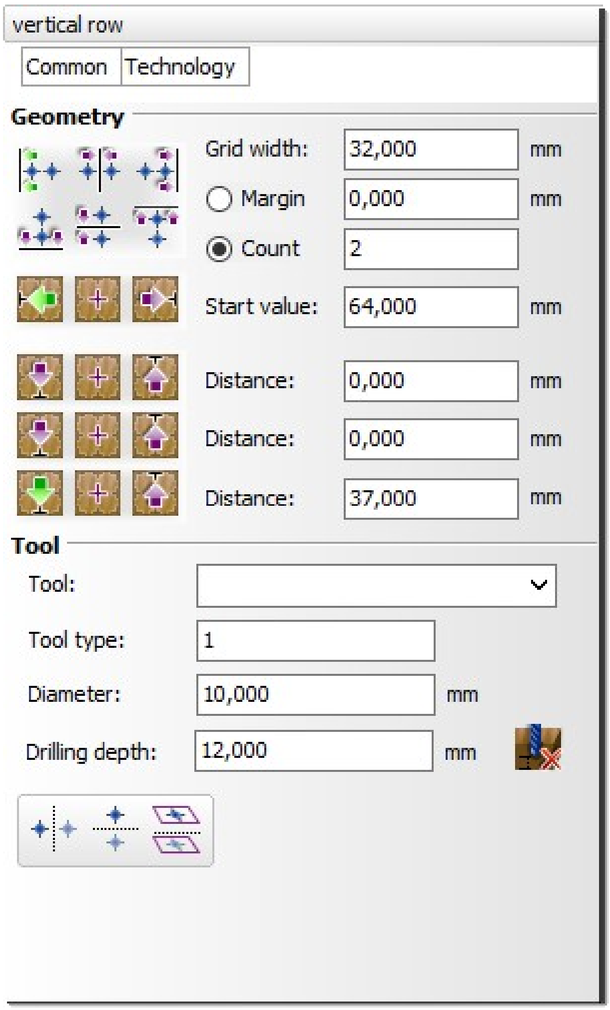


Now we add the remaining drill data: Tool type (1), Diameter (5) and Drilling depth (10). We skip mirroring the drilling row here and finish the entries by clicking on the OK button.

1.7 Hinge mounting plates

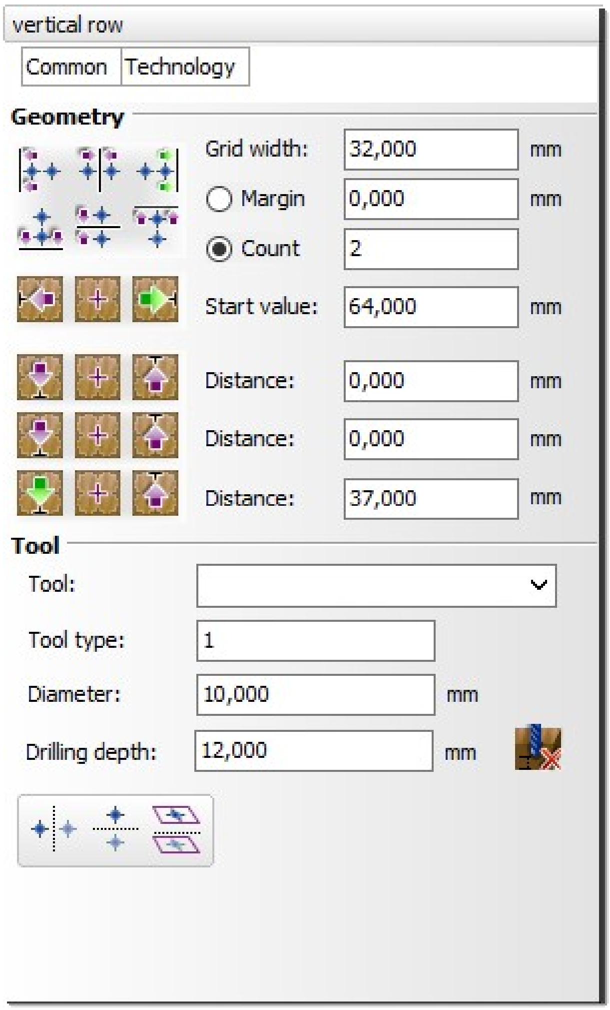
The two drillings for the left hinge mounting plate are also done using a vertical drilling row. But this time we limit the count of drillings to two. We select the option Count and enter 2 in the field to the right.

The direction of this drilling row is left-adjusted as well, the Grid width is 32 mm. The Start value also refers to the left edge, but here it only amounts to 64 mm. The rear drilling row is dropped and the front row amounts to a distance of 37 mm. After setting Tool type (1), Diameter (10) and Drilling Depth (12) we close the window by clicking on OK.



The right hinge mounting plate is created almost in the same way. It simply refers to the right edge. In order to type in the least possible, we double click on the recent drilling row (or one of its drillings) which opens the setting dialog again.

Now we change the direction to right-adjusted and starting point's reference on right panel edge. But we do not click on OK yet!



If we did that, we would only move the drillings to another place. In order to add a drilling row with changed data we click on Add instead.

With a few mouse clicks we can create further elements, using already existing elements with defined settings. Of course we could have also created a new drilling row.

We could have also mirrored the original drilling row on X axis because the drillings for the hinge mounting plates are arranged symmetrically in this case. Wouldnt that be a good exercise....?

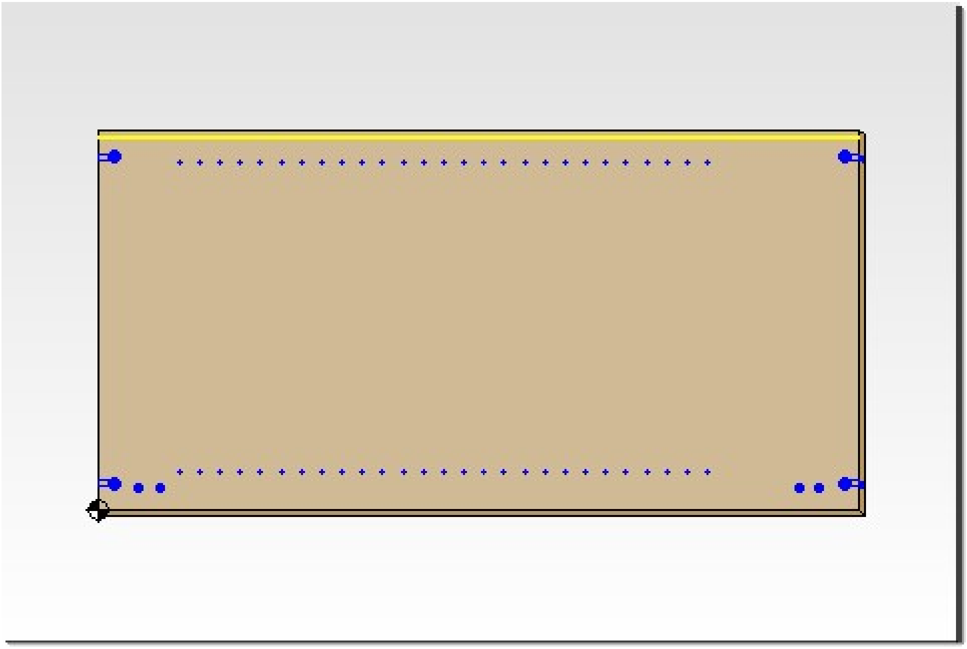
1.8 Formatting the panel

The last piece of machining that is required is to format the panel (umfräsen). We keep it simple here and use some of the provided basis macros.

With a click on the first symbol Umf-mitte-li (Umfräsen, mittig anfahren, links rum) the whole operation of formatting the panel is inserted.

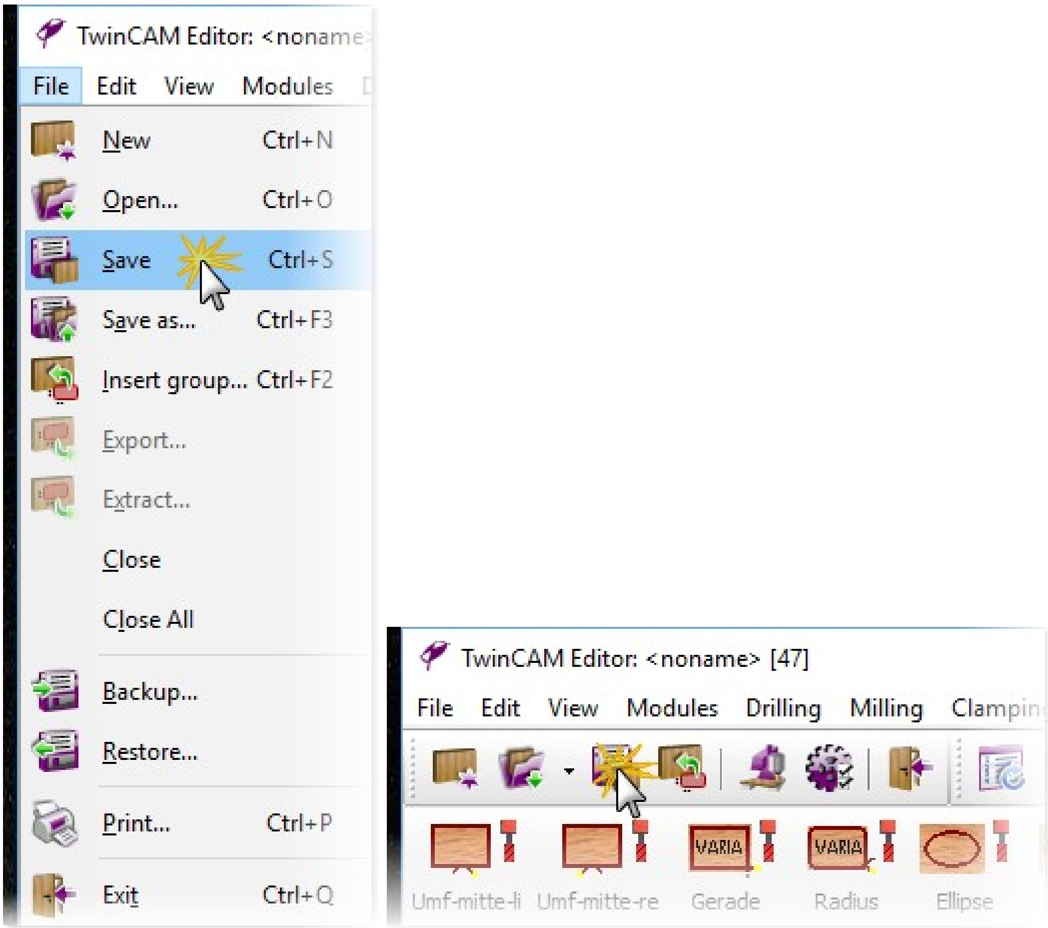
1.9 Viewing the finished drawing

This is how the finished drawing with all its created elements should look like.



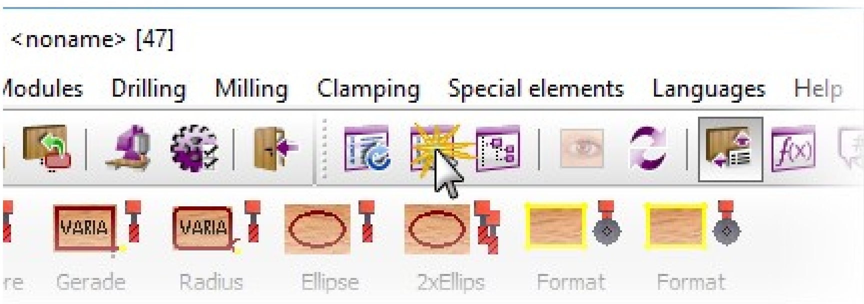
1.10 Saving the drawing

Now we still want to save the drawing. We either enter the File menu or we directly click on the symbol *Save panel*.

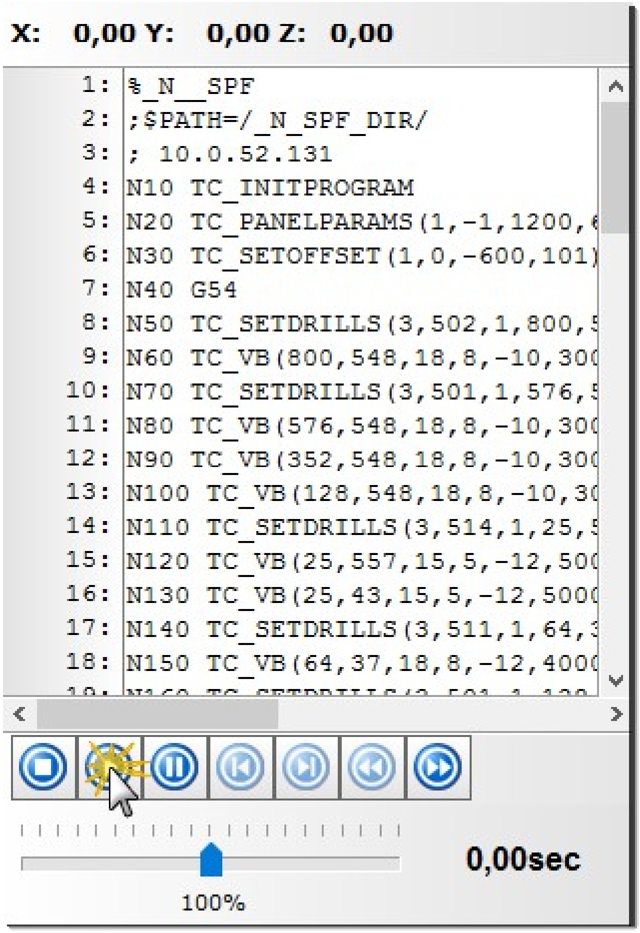


1.11 Generating the program

A click on the button *Show NC code* opens a new window with the generated NC code...



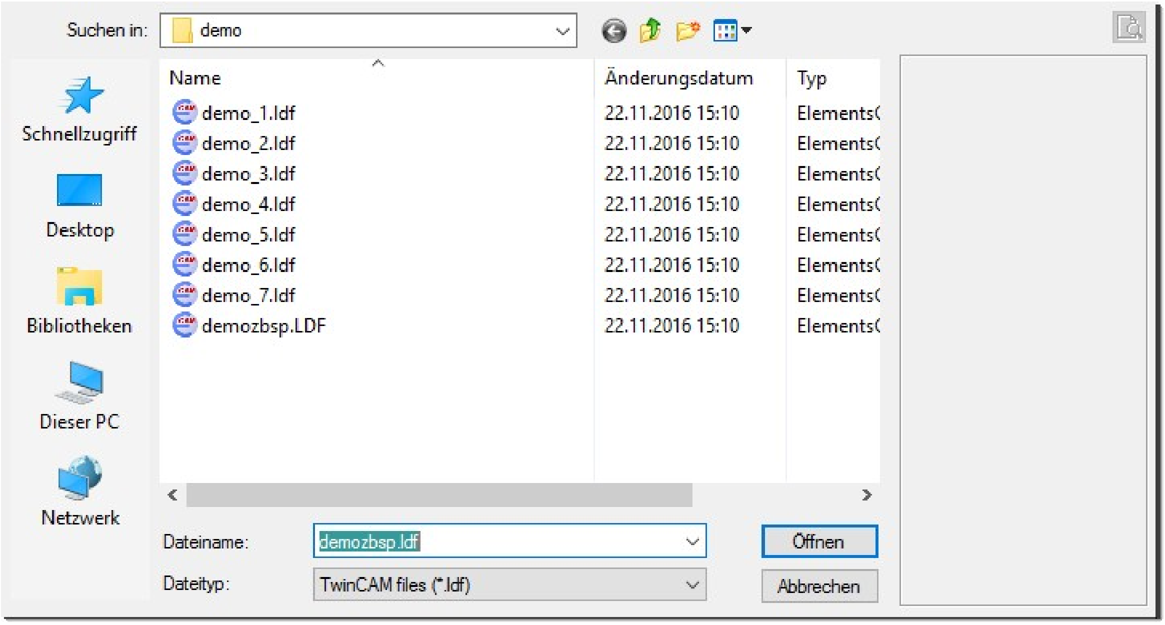
... and a click on the button *Start simulation* starts the simulation.



1.12 Opening drawings

To open a drawing, either select *File* and *Open* from the toolbar or click on the *open file* button.

In the subfolder DEMO we find some example files. If the files are not listed, we set the file type on *TwinCAM* Files (*.LDF). We select the file demozbsp and click on the *Open* button.



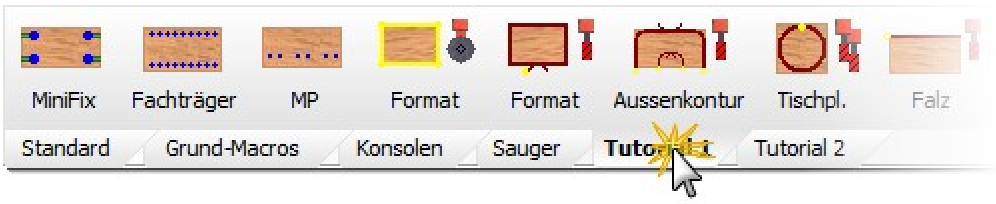
1.13 Combining macros to form a drawing

Of course, we can compile macros for frequently used program parts, for these macros are technically nothing else than normal TwinCAM files. However, one should observe the parametric aspect while creating macros.

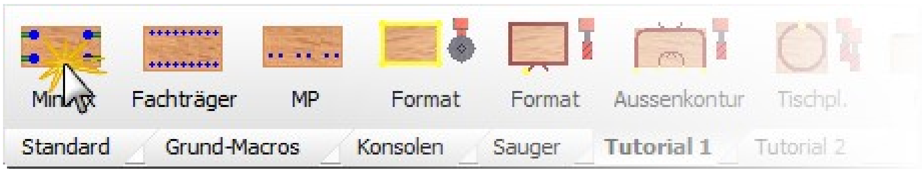
In this section we will get to know the efficient use of macros. For this tutorial we have already split the recently created side panel into single groups and saved it as macros for reason of demonstration.

At first we save the current drawing. Then click on *New panel* and type in the new panel measurements. This time, however, we choose different measurements. Which ones exactly is up to the user. (e.g. 1800x550x16).

We switch to the register tab *Tutorial1* on which the macros are created.



In order to insert the Minifix-connectors we click on the button *Minifix*.



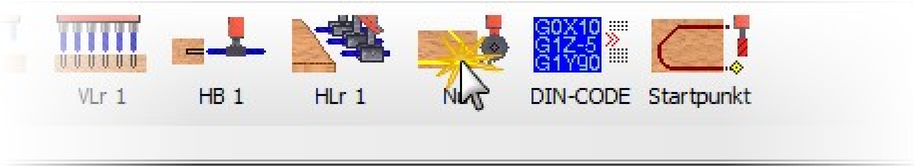
The elements of the Minifix-connectors are imported into the panel with its corresponding measurements and reference points. The modified panel measurements are automatically taken into account.

Now add the shelf mount drillings by selecting the *Shelf M* button.

Here too, the changed panel dimensions are taken account of, a shorter panel results in fewer drillings, a longer one, more. Nonetheless, the pre-set margins of 128mm and 240mm have been maintained, with the number of drillings between the two points varied according to the panel length and in accordance with the set grid.

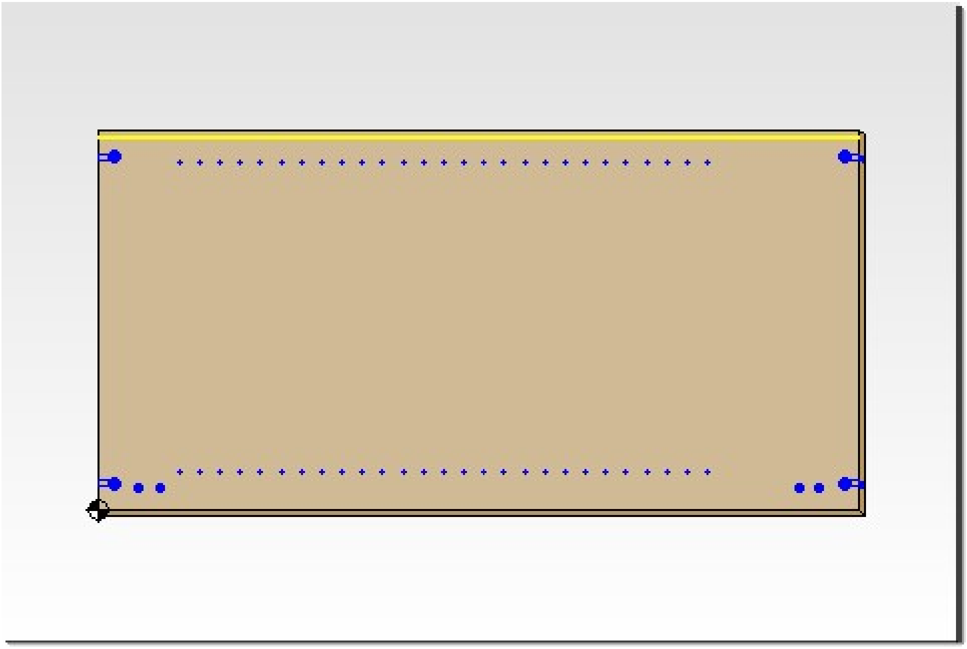
Now add the hinge mounts to your workpiece description. The hinge count is automatically adjusted to the width of your panel. (Admittedly, here we helped us out a little by using Variables and Formulae.)

This has already created the majority of elements for the new panel, to add the back panel rebate, select the rebate button from the standard tab. Clicking on save earlier in the demo, saved the settings required for the back panel rebate. .



To format the panel, select the basic macros tab and then the format button. Here to, the machining is adjusted to suit the panel dimensions.

The screenshots below show the drawing that you should now have before you (depending on the selected measurements).



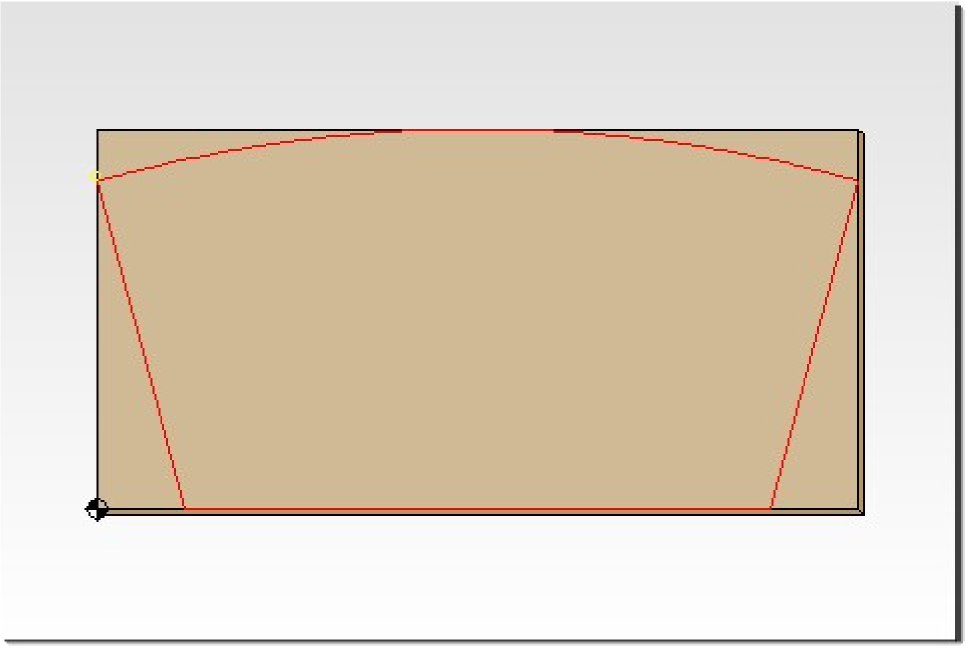
Chapter II

CAD functions

CAD functions

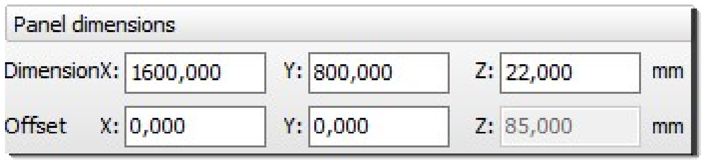
2.1 CAD usage example

In this section we will explain how to use ElementsCAM's CAD functions. The objective is to complete a simple routed arc.



Panel dimensions:	X = 1600; Y = 800; Z = 22
Arc height:	110 mm
Angle	both sides 15°

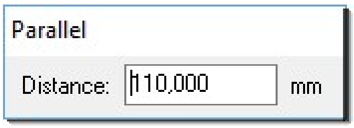
First, enter the panel dimensions.



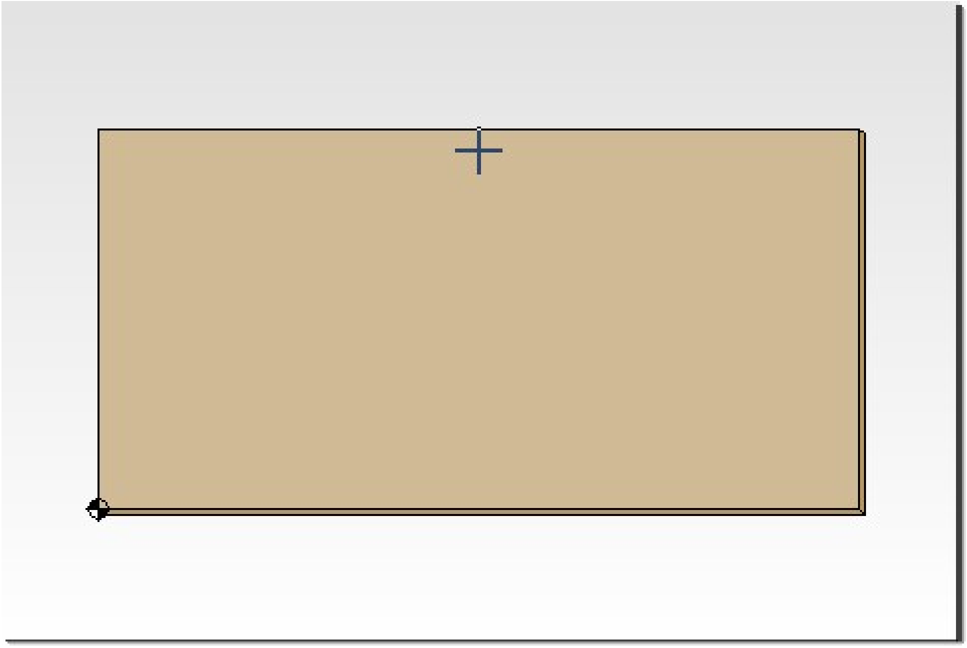
2.2 Drawing subsidiary lines

Before we construct an arc, we create a drawing subsidiary line that simplifies finding the endpoints of an arc. So select the *parallel element* button.

A small dialogue box will open. Enter 110mm in the box.



The cursor will now appear as a cross. Move the cross to a point below the top edge of the panel and click once. The position of click defines in which direction the parallel is created.

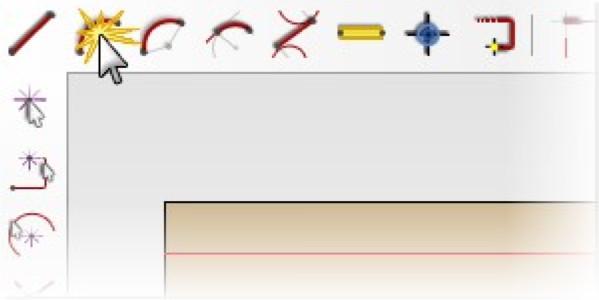


A routed line will now appear parallel to the edge of the panel; this is only for guidance purposes and will be removed at the end of the sequence. A right click or a click on the parallel element button will end the parallel function.

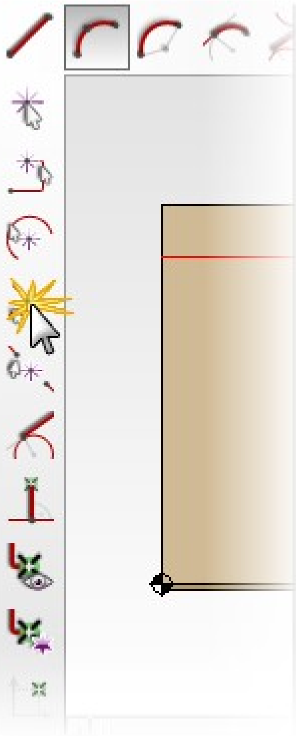
2.3 Drawing an arc

We want to construct the arc by 3 points. The starting point is the intersection of the guideline drawing and the left edge, the center of the arc is the center of the rear edge and the endpoint is the intersection of the guideline drawing and the right edge.

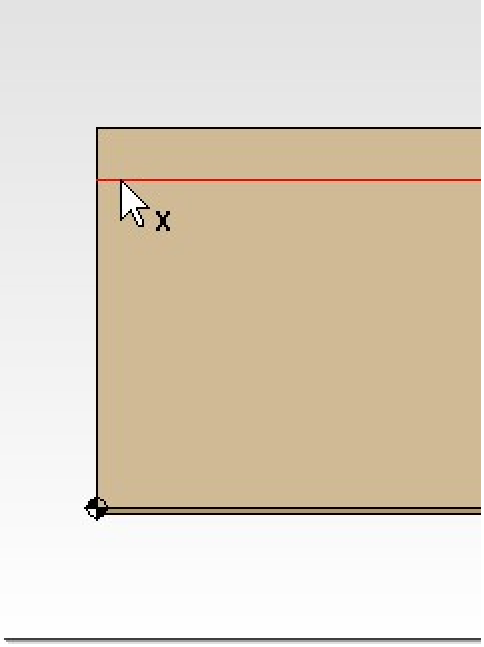
We select the button marked *arc by 3 points*.

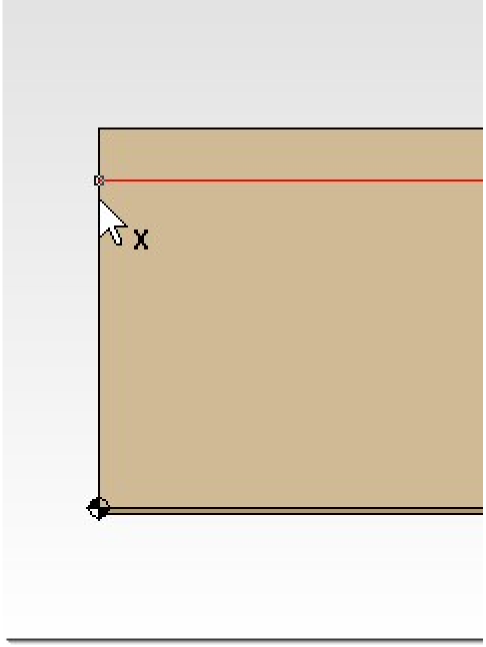


Now we select *Intersection point of two elements*. An X will now appear next to the cursor.



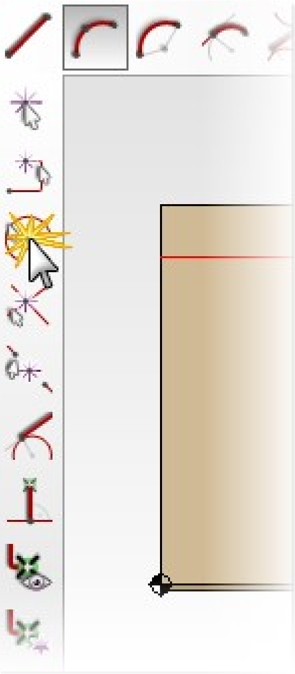
We need to select both elements to create their intersection point. First we click on the subsidiary line and then on the left edge.

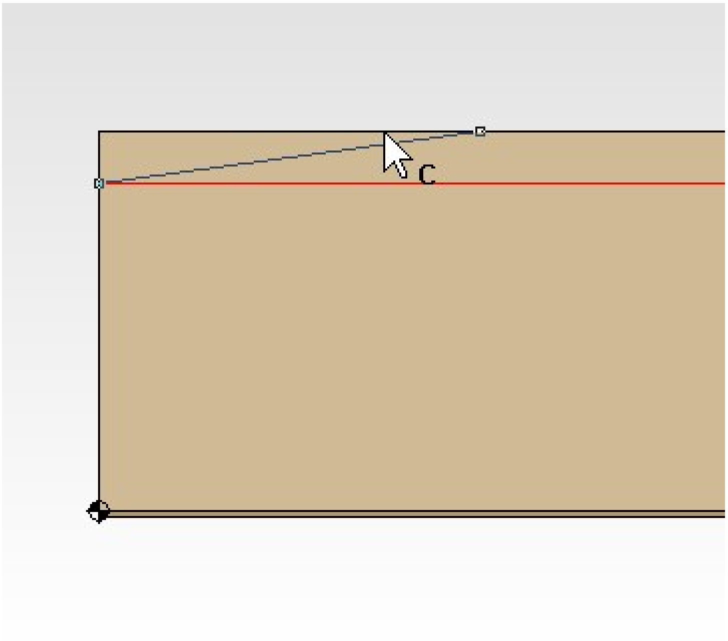




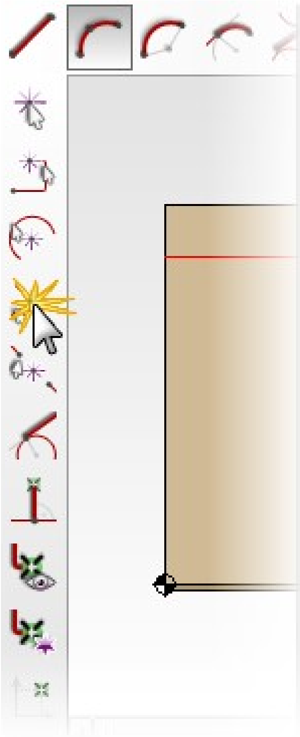
It does not matter if the two elements actually cross each other. If the guideline had been shorter, the intersection point would have been located as well.

Now select the Center of an element button; the cursor will have a C displayed next to it. Click on the top edge of the panel.

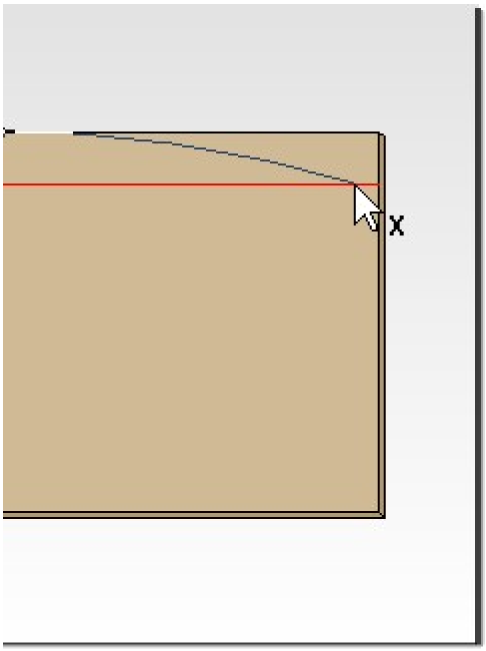


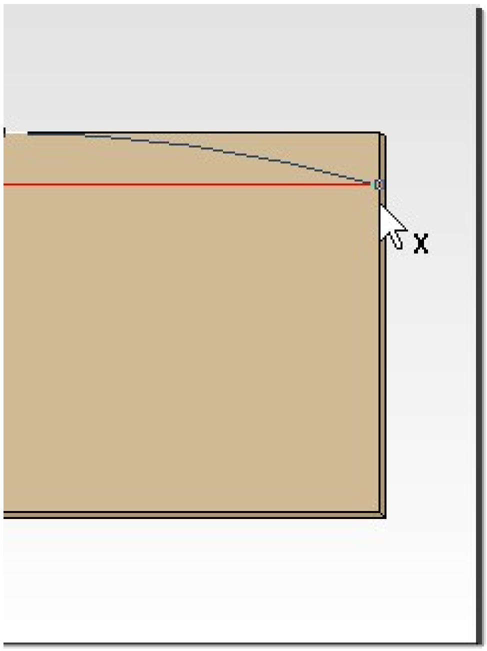


For the third point we just repeat point one, selecting *Intersection point of two elements*…

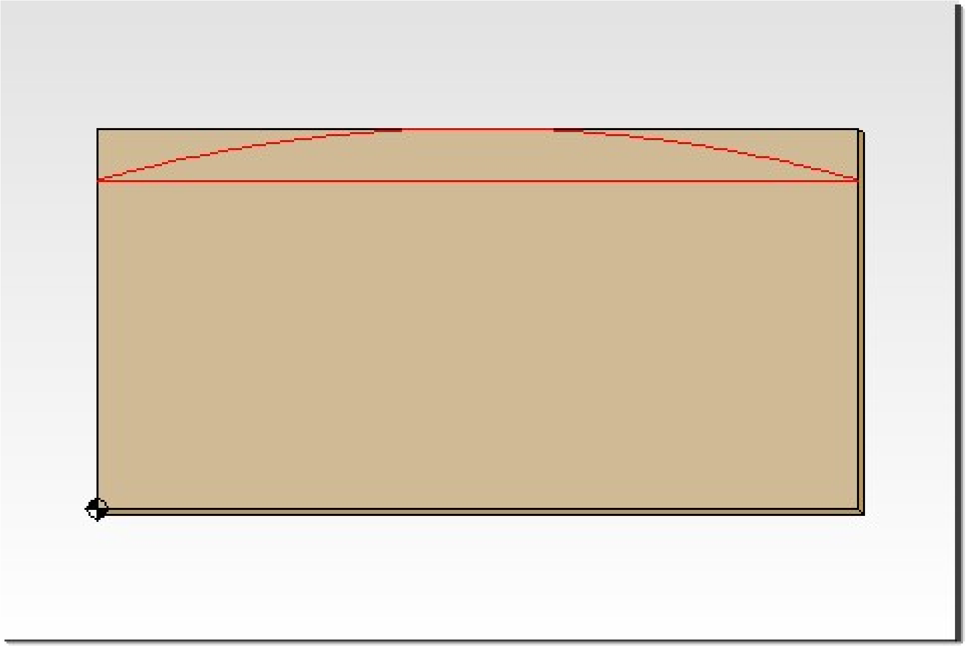


… and then click on the subsidiary line, followed by a click on the right edge.





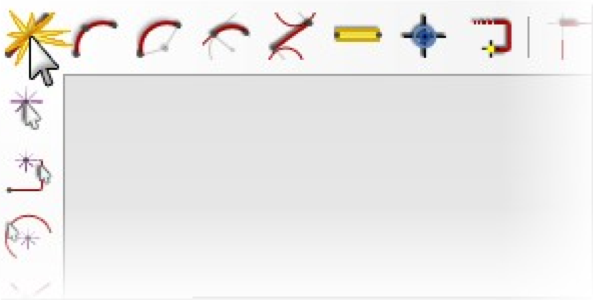
To complete the process, either right click in a free area or click on the *arc by three points* button again.



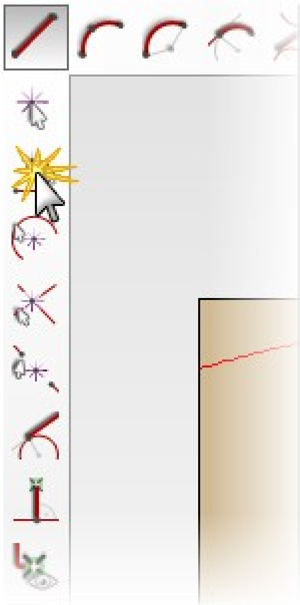
Finally, select the subsidiary line by clicking on it and then press the *delete* key on the keyboard to remove it.

2.4 Drawing diagonals

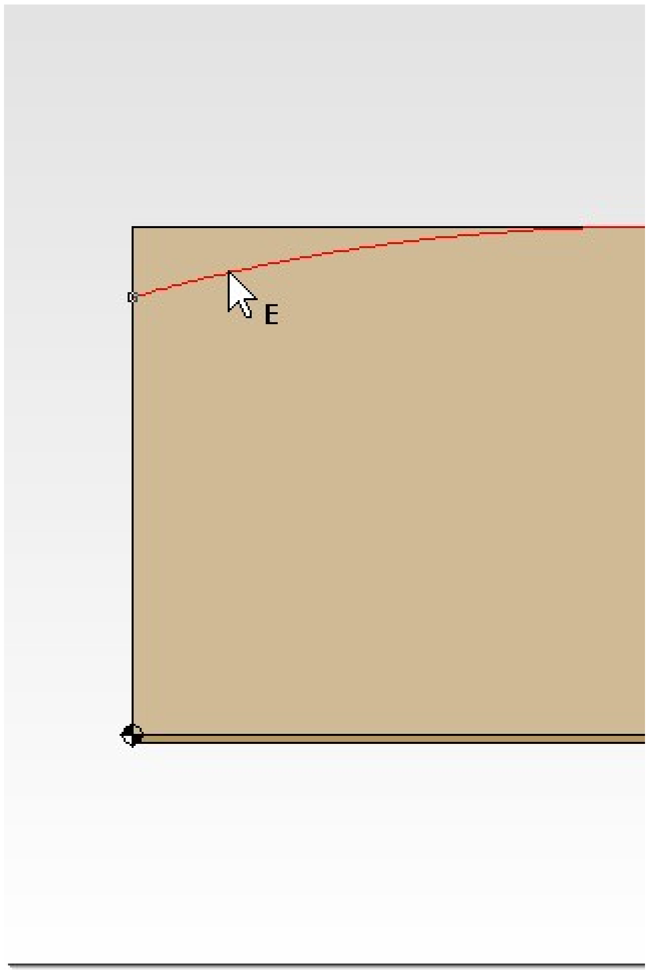
Both diagonals are created by using the CAD function *Line over 2 points*. It is the first button among the CAD functions.



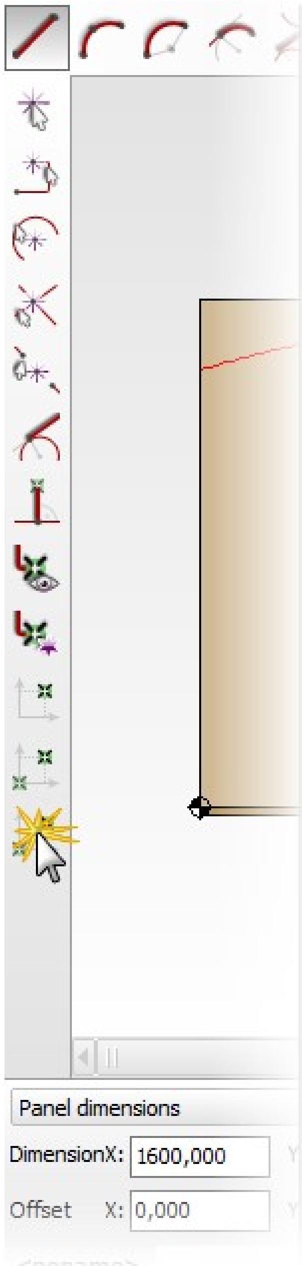
Now we have to select two points. The first one is the starting point of the arc. So we select the snap function *End point* of an element. An E will appear next to the cursor.



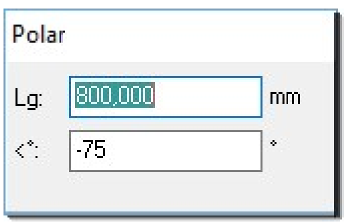
And then click on the left-hand end of the arc.



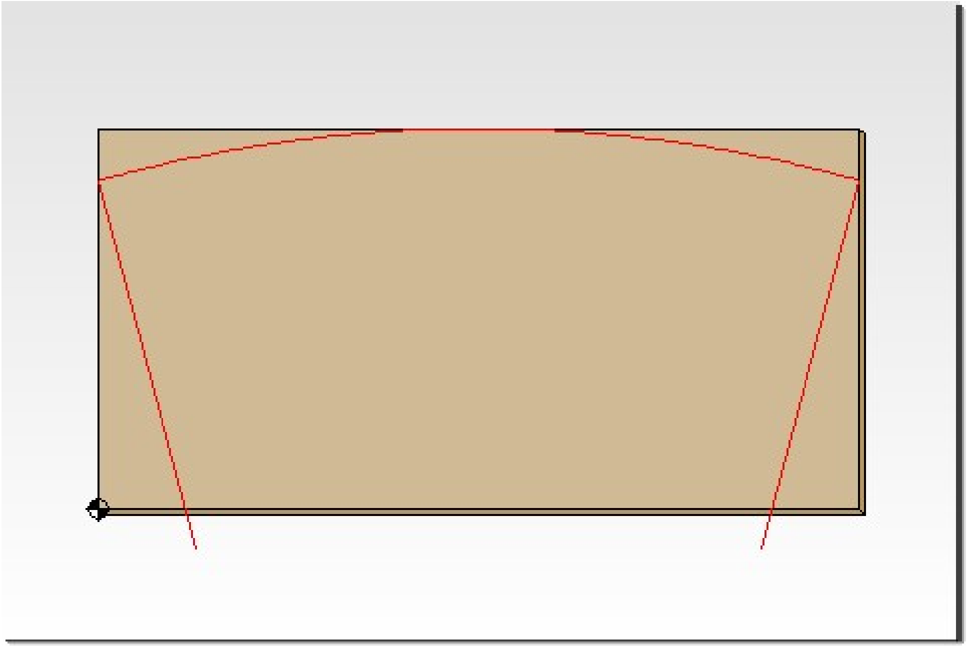
The line and the left edge are supposed to create an angle of 15 degree. We achieve this by clicking on Relative polar-coordinates to current point



and then enter 800 mm for the length and -75º (-90+15) for the angle. Press enter to confirm these entries.

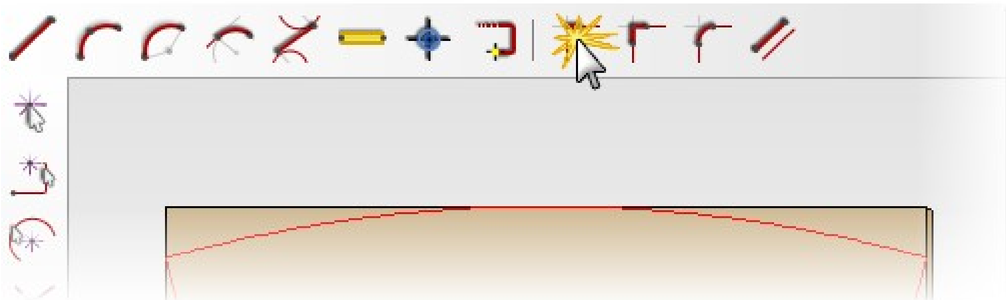


Repeat the same steps on the right hand end of the arc, this time entering -105º (-90-15) as the angle.

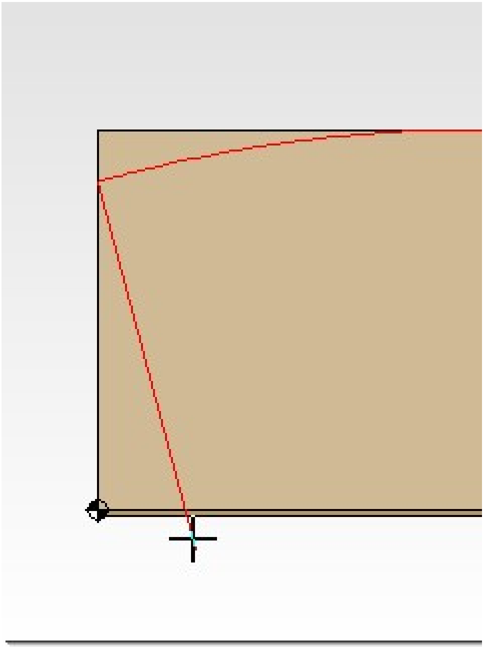


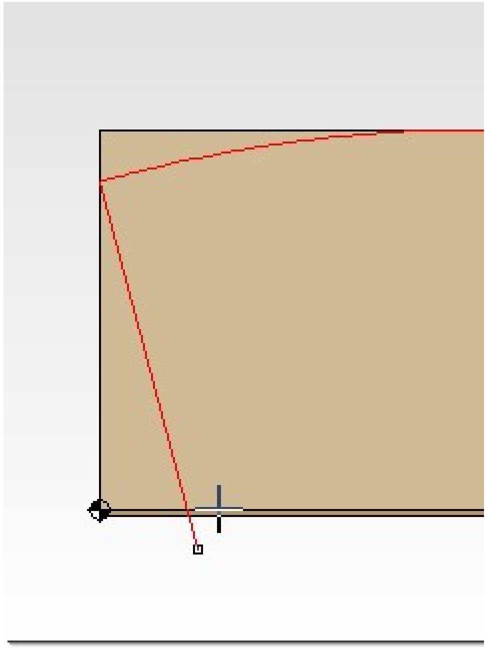
2.5 Removing the excess

In both cases, the diagonal lines extend beyond the edge of the panel. To remove the excess, first select the *trim 1 element* button.

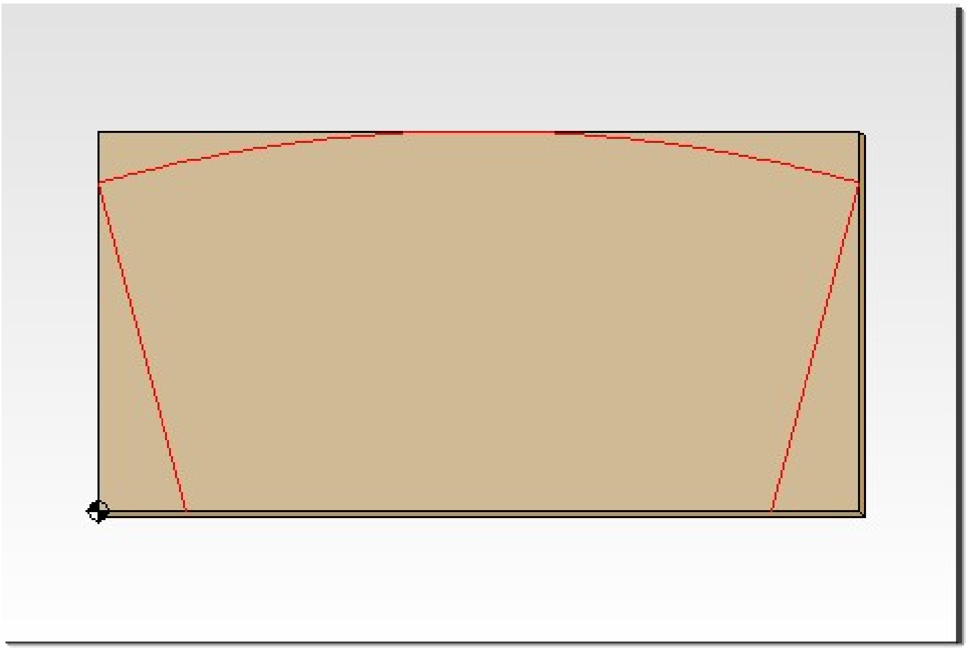


Now click on the end of the line and again where it intersects with the edge of the panel.



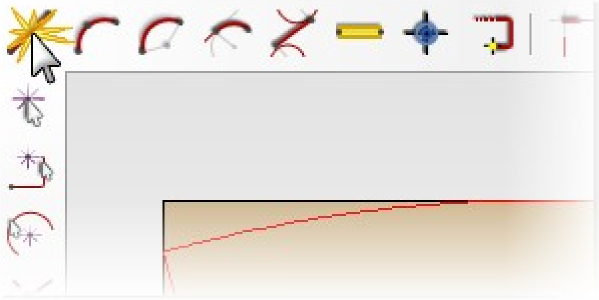


The function remains active, so we trim the right line in the exact same manner.

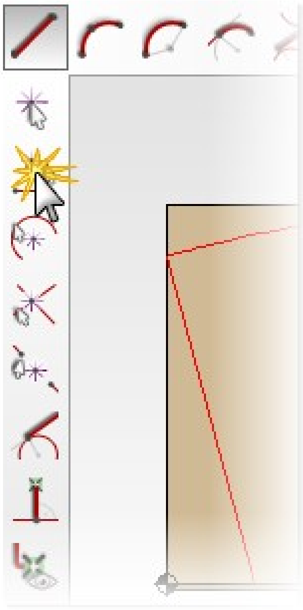


2.6 Drawing a line from point to point

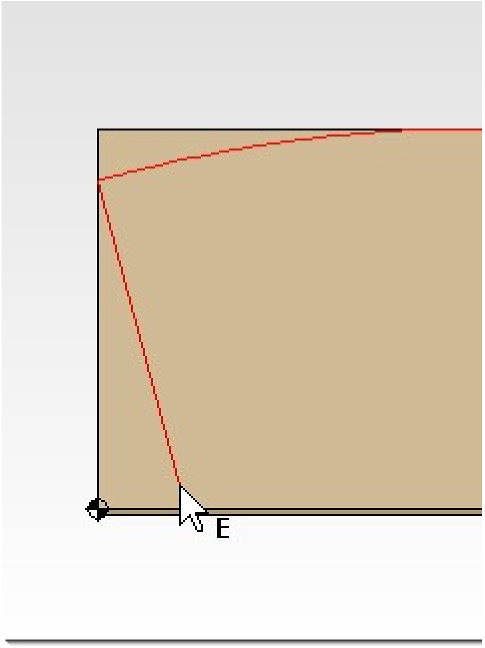
The line below is easily drawn from the endpoint of the left diagonal to the endpoint of the right diagonal. First select the *line over 2 points* button again.

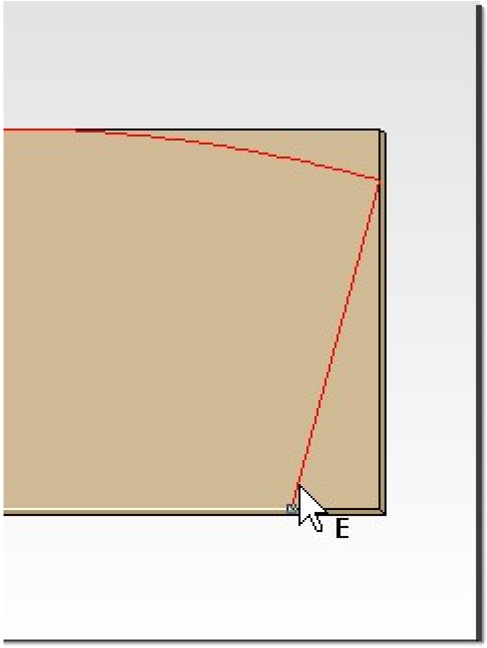


Select the snap function *End point of element*.

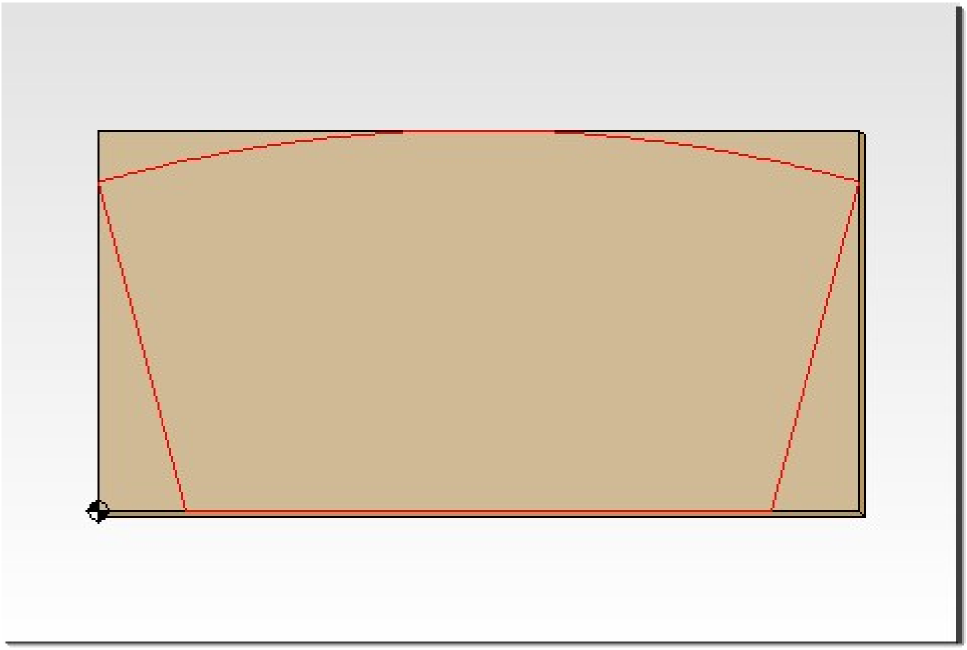


Now we click on the left diagonal somewhere near its lower endpoint, then click again on *End point of element* and repeat on the right side what we did on the left side.



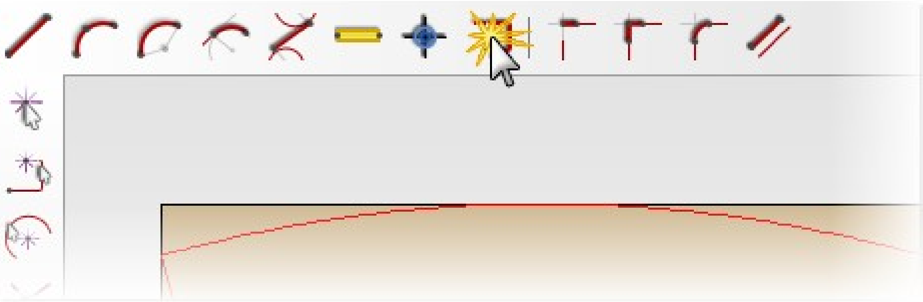


In so doing we have created all contour elements. End the process either with a right mouse click or by clicking on *line over 2 points* again.

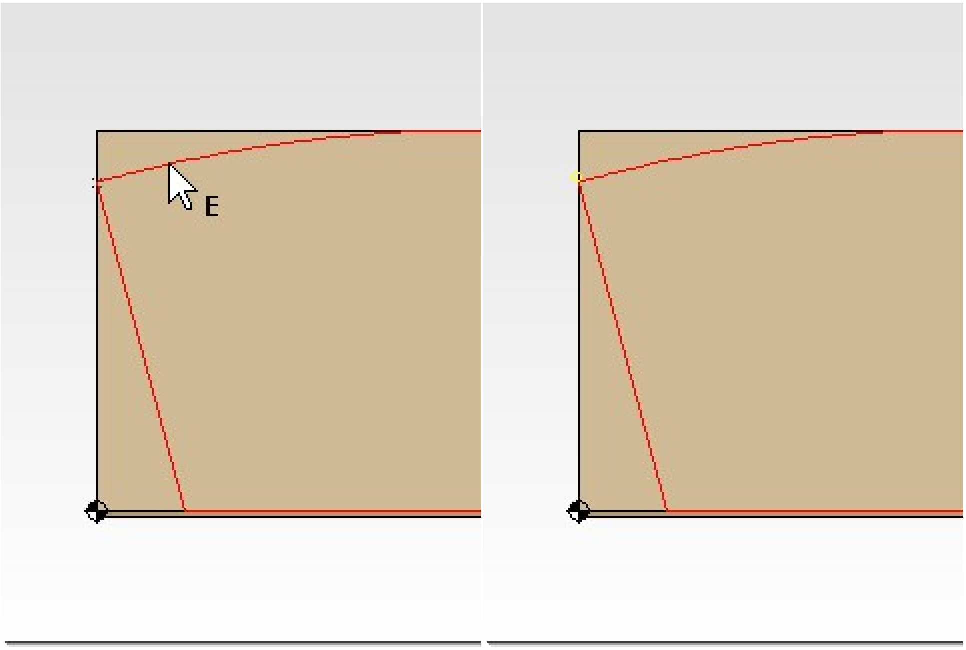


2.7 Setting the start point

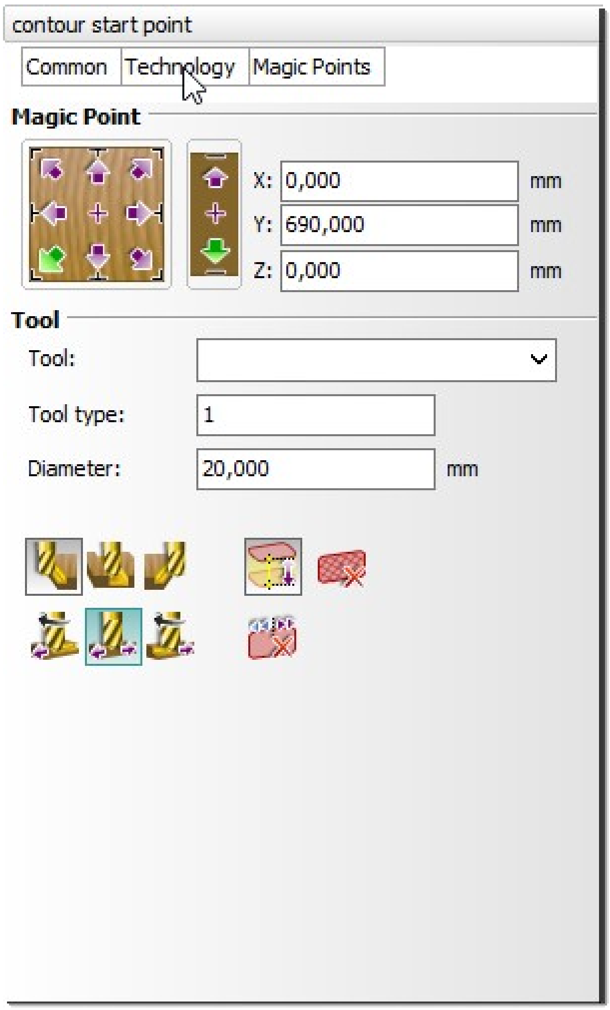
Now we have to set a starting point, that defines the router and technological data. We use the CAD function *Startpoint*.

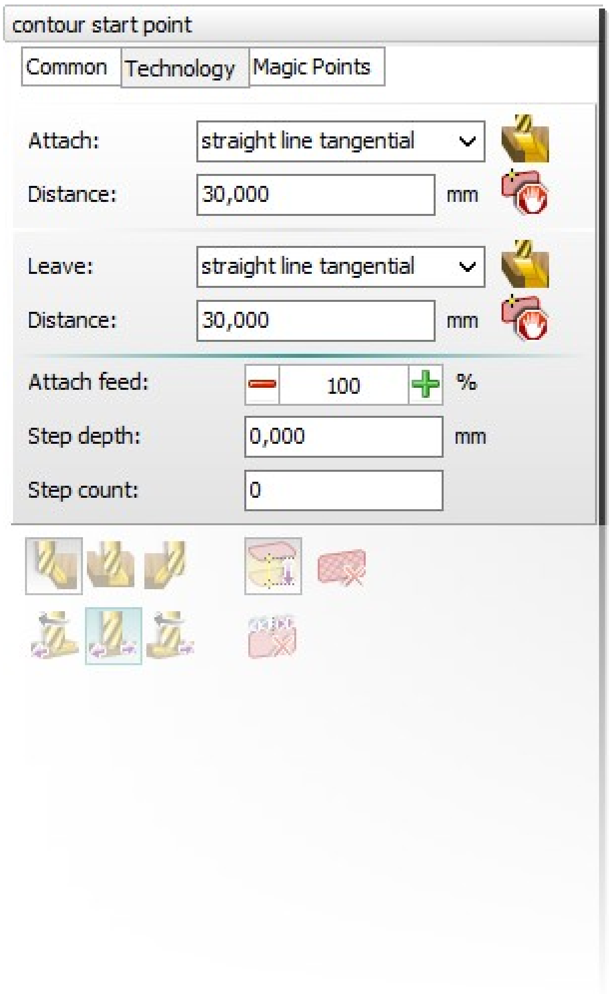


The snap function *End point of an element* is default because Startpoints only make sense on a contour element. We want the milling to start at the arc, so we click on the endpoint of the arc.



We doubleclick on the startpoint of the contour startpoint. A new input dialog pops up. We enter *Tooltype 1* and *20mm* for *Diameter* and we set the correction on left.
In the area *Technology* we set *straight line tangential* for *Attach* with *30mm* for *Distance*.


contour start point
Common
Technology
Magic Points
Magic Point
X: 0,000 mm
Y: 690,000 mm
Z: 0,000 mm
Tool
Tool:
Tool type: 1
Diameter: 20,000 mm



We close the input dialog by clicking on OK.

Done!

Chapter

III

Programming in practice

Programming in practice

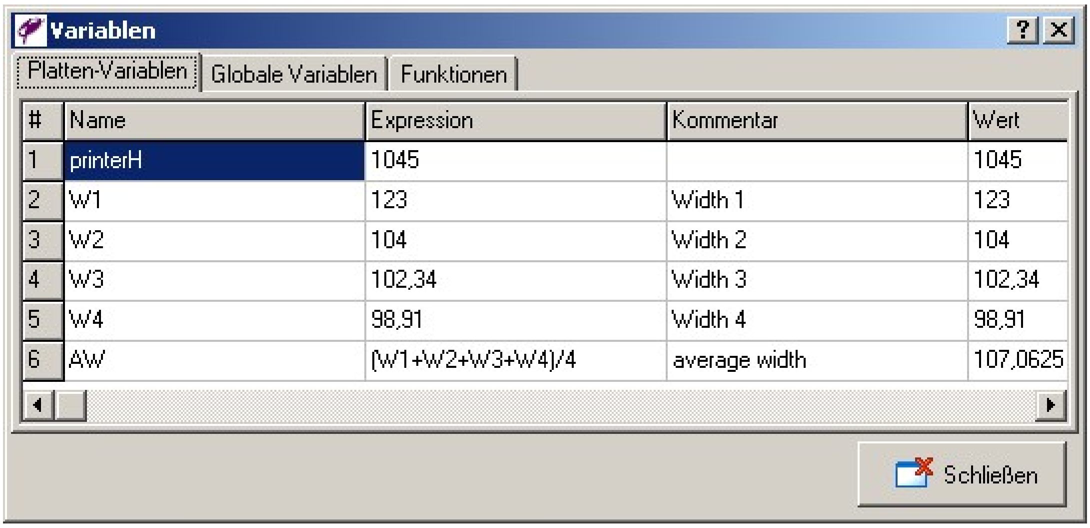
3.1 Digression

3.1.1 Parametric programming

The parametric programming allows a fast adaptation of existing workpieces to new requirements as measurement changes for example. A certain discipline while programming is basis for an efficient usage of parametric workpiece description. This also always affects some preliminary considerations about the references for individual parametric measurements and its application.

There is a large number of predefined parameters, all available in TwinCAM. So all panel measurements are accessible by the variables DX, DY, DZ and its corresponding offsets OX, OY, OZ. Furthermore TwinCAM also offers a powerful parser. Thus, TwinCAM is capable to interpret not only individual variables but also complex terms or even entire functions.

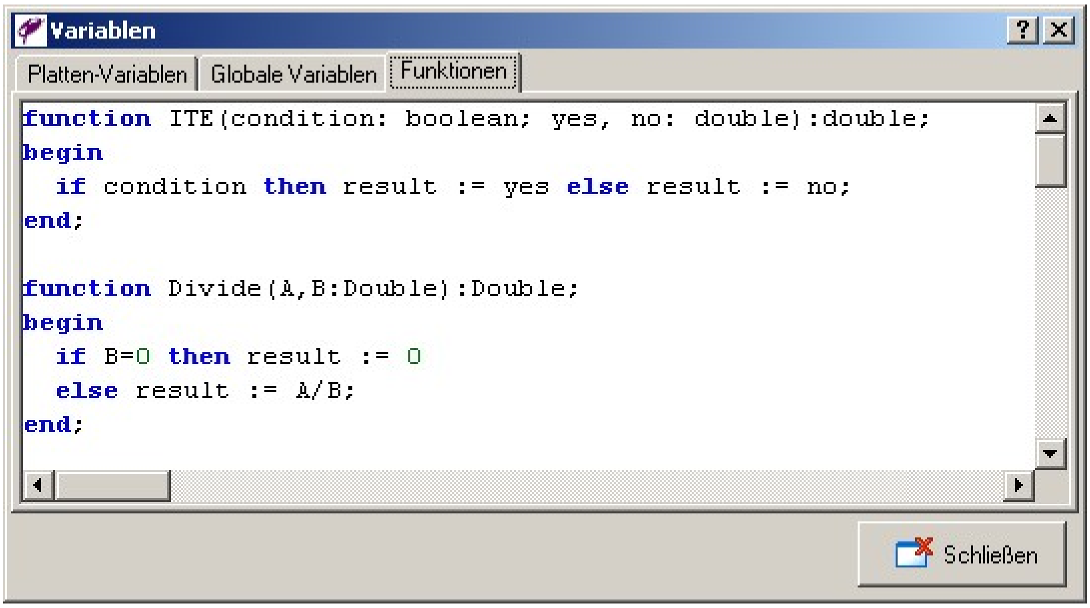

You enter the dialog via the button "Variables" where you can see and change your variables and functions.



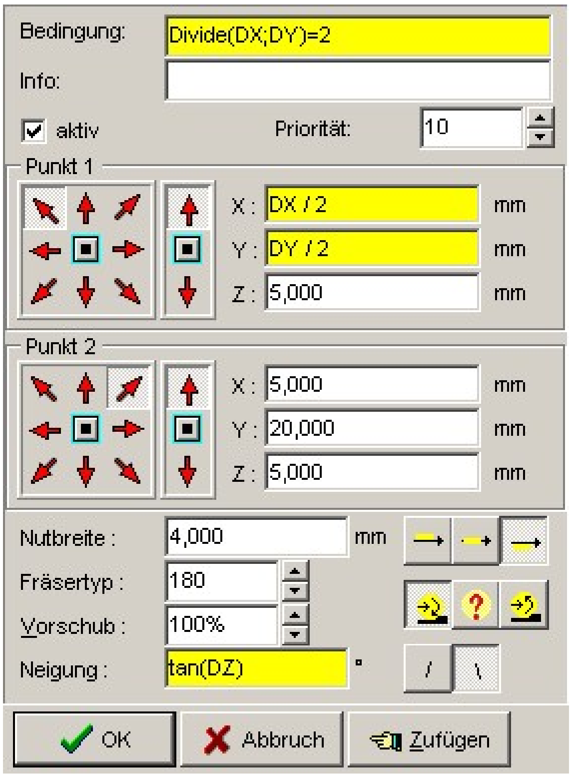
#	Name	Expression	Kommentar	Wert
1	printerH	1045		1045
2	W1	123	Width 1	123
3	W2	104	Width 2	104
4	W3	102,34	Width 3	102,34
5	W4	98,91	Width 4	98,91
6	AW	(W1+W2+W3+W4)/4	average width	107,0625

However, it is not necessary to switch to the dialog in order to create a variable. In case you use a variable, that does not exist yet, anywhere in an element, TwinCAM creates this variable automatically and attaches the value 0 to it.

Together with the possibility to write functions on your own, you are capable of solving problems in a smart way, may they be ever so complex.



You may use all variables and functions everywhere in TwinCAM. As you can see, the variables are very easy and useful to handle.



3.1.2 Conditions

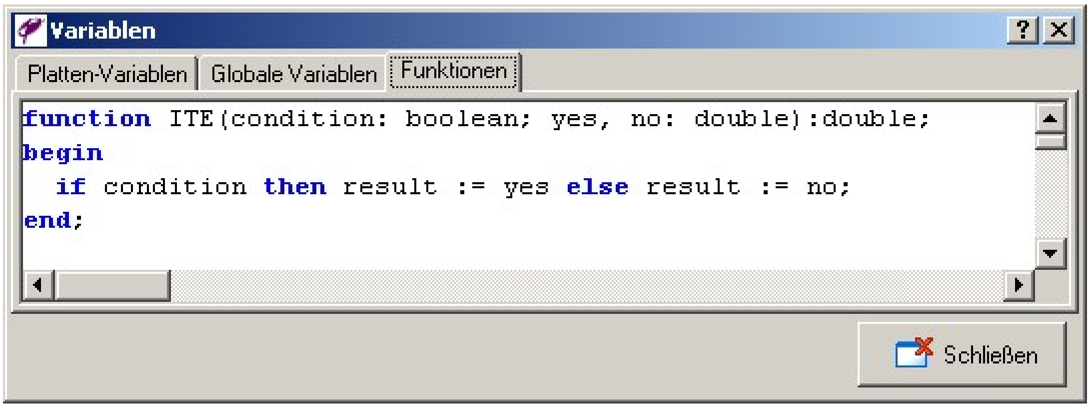
Parametric programming often goes along with so called rule-based programming, or called programming with conditions. Rule-based programming gives the cance to process company standards in CNC programs. Thus, a certain machine processing will be executed dependent on the condition being true or false. If false the processing will simply be skipped. Parameters play a crucial role in conditions. They are compared with other parameters or simply with numeric values.

Every element in TwinCAM has the field "Condition". As a condition you can enter variables or functions. The following rules are to be followed:
If the term in the condition is 0, the element is not considered when the program is generated.
For all terms uneven 0 the corresponding element will be considered when the program is generated.

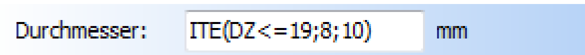
You can also use truth values. Thus, 0 stands for false and every other value stands for true. Of course, you can also combine all terms with the following logical operators:

TwinCAM Operator	Meaning
>	...greater than...
<	...less than...
=	...equal...
>=	...greater than or equal...
<=	...less than or equal...
<>	...not equal...
AND	...logical and...
OR	...logical or...
NOT	...logical not...

ITE is a very powerful and useful function (IF-THEN-ELSE). TwinCAM is equipped with this function from the start.



Using ITE, you can define exactly what is supposed to happen if a condition is *true* or *false*. ITE is usable in every dialogue. In this way, the diameter of a drilling, for example, can be defined dependent on the panel thickness: If a function with parameters is called, the parameters are separated with a comma or semicolon (depending on the regional settings of your system).



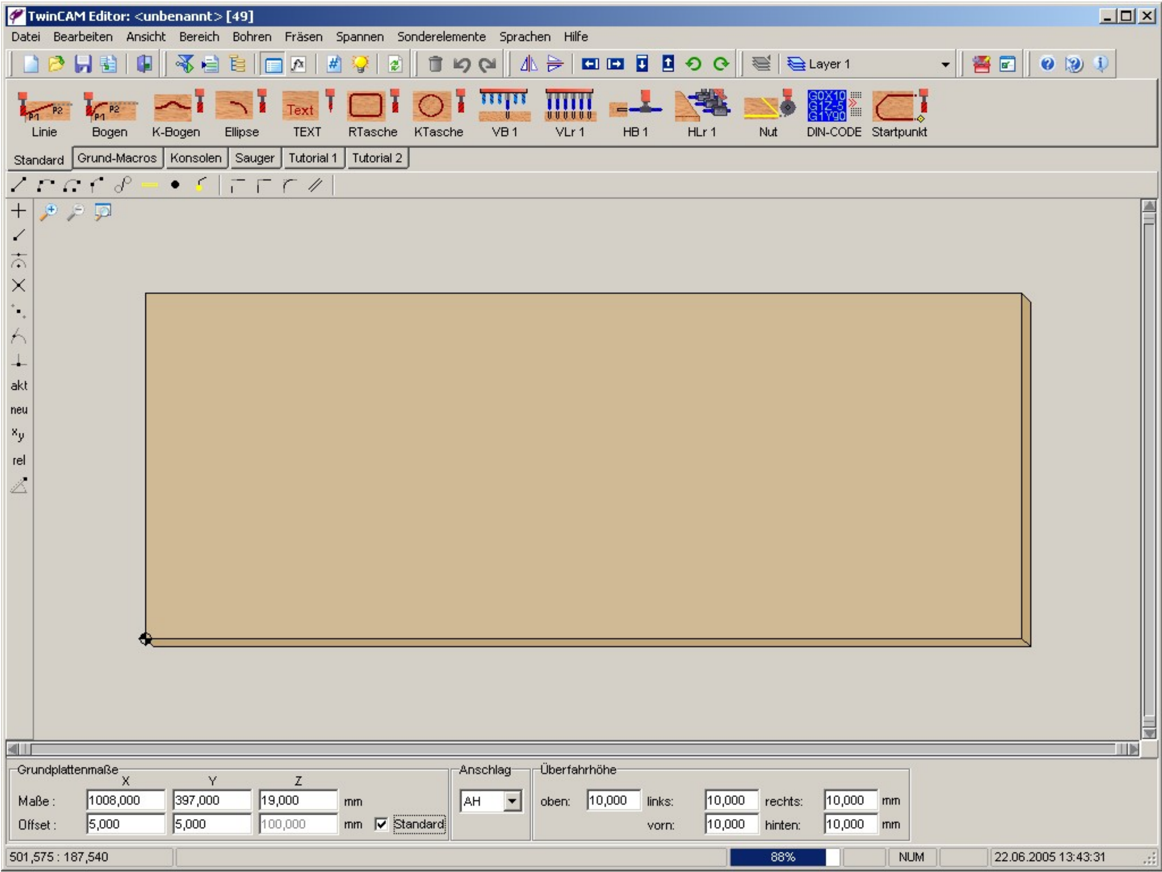
In this example, DZ<=19 is set as condition in the ITE function. If the panel thickness is smaller or equal 19mm, the function returns the value 8. However, if the panel thickness is greater than 19mm, the function returns 10 as result.

3.2 Defining workpiece measurements

First, there is the raw panel in the beginning of every workpiece description in TwinCAM. In order to start the machining process, you have to enter the measurements of the finished part. The size of the panel is displayed according to these measurements. Many machining processes take place on unformatted raw panels that are greater than the finished part. In this case the measurements of the finished part are to be entered first as well. In order to put the processing into execution with a certain blank allowance as distance to the end stop, the blank allowance is to be entered as offset on both sides of the end stop. The formatting takes place along the displayed edge of the finished part.

Excercise:

In our example, a cabinet door is to be programed. Its finished part measurements are 1008 x 397 x 19 mm. The blank allowance amounts to 10 mm in length and 10 mm in width. You can enter the measurements below the graphics area.



Please be aware that the blank allowance of 10mm is split up to both sides. So the offset only amounts to 5 mm distant to the endstop.

3.3 Formatting a solid wood panel

Basically, formatting a solid wood panel can be put into execution by a router or a saw. Using a router, different directions of rotating should be used due to the different fiber orientation. You achieve very good results, using a saw blade for solid wood. Additionally, there is the chance of scoring before sawing to produce clean edges. Both side edges (end grain) should be formatted first, then the long edges.

TwinCAM provides a set of parametrical programed macros from the start which can be used for such standard tasks.

Zur Übung:

Formatieren Sie die die gerade angezeigte Platte. groovezen Sie zuerst das Makro und versuchen Sie anschließend, die Schritte, die das Makro Ihnen abnimmt, einzeln nachzuvollziehen.

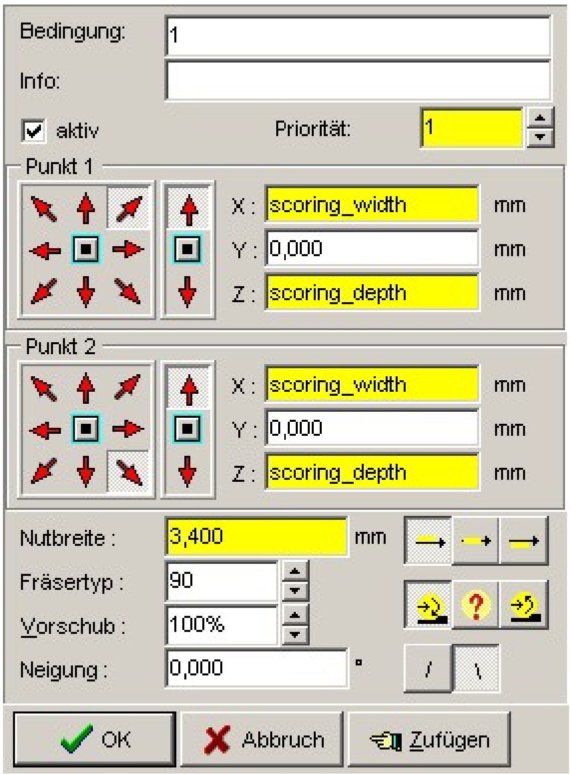
In the palette under the register "Basic macros" you can go back to ways of formatting of any kind. For our example we choose formatting with a saw, "Format". If there is no such element in your palette, you could easily generate it by yourself.

- Create the variables in the variable editor by clicking on the variable symbol  with the following values:

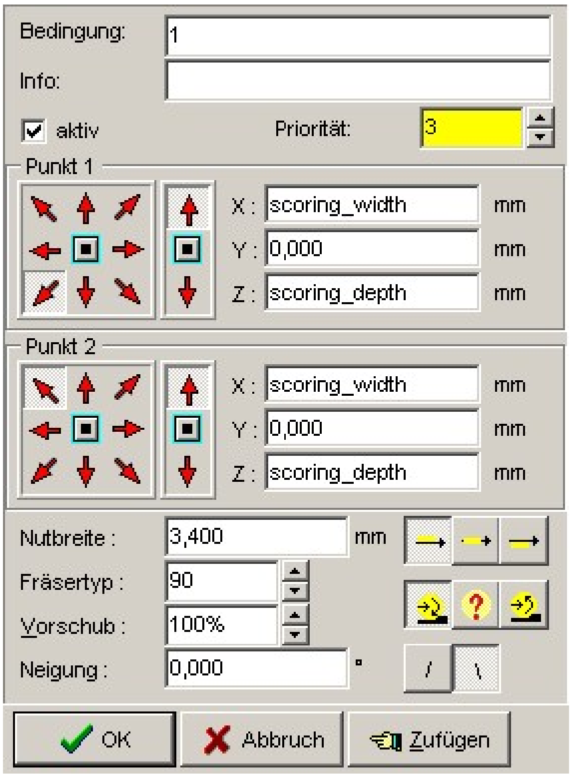
Variable name	Expression	Description
scoring_depth	0,5	scoring_depth stands for the scoring depth
scoring_width	0,1	scoring_width is a possible correction value that is to be kept while scoring

- Now you need 8 grooves in total that you attain over the pallet "Standard" or the menu "Milling". In order to insert a groove just click on the groove symbol. You can take the individual values of the grooves out of the images. You need:

1. Four grooves for scoring:



After you adjusted the dialogue, confirm with OK. Your groove will be placed on the base plate with the set parameters. This groove builds the starting situation for the next scoring groove. So you simply need to make some modifications on your already existing groove and click on "Add". The original groove will be unmodified and a new groove that contains your modified adjustments will be placed on the base plate. The necessary changes for the second groove can be taken out of the following image:



In this way, also the third groove can be created easier. However, you need to make some more adjustments here:

Third groove (scoring, bottom side)		
Priority		5
	Point 1	Point 2
Reference (X / Y)		
X	0	0
Y	scoring_width	scoring_width

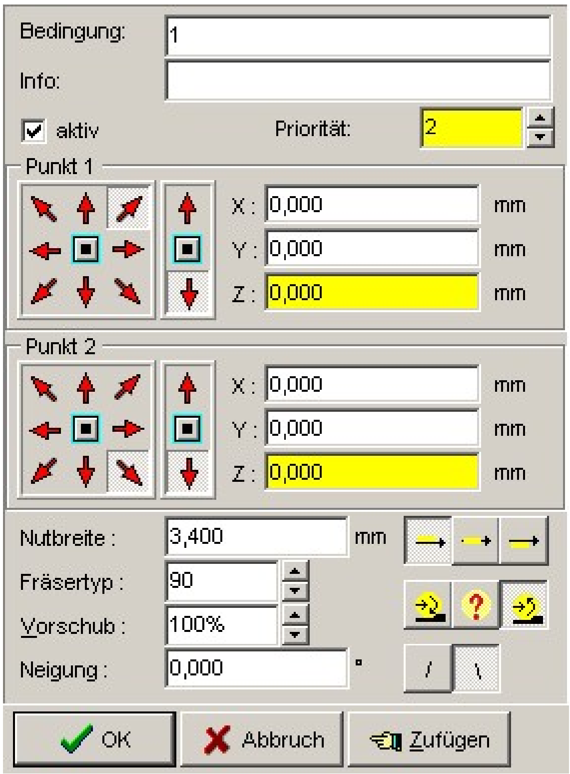
For the next groove you can save some work by taking the third groove as basis.

Fourth groove scoring, top side)		
Priority		7
	Point 1	Point 2
Reference (X / Y)		

Using these four grooves, you will score your workpiece with 0.5 mm in synchronism. It guarantees a clean surface. Now that you successfully mastered scoring the panel, we finally spend time on sawing.

Note:
To achieve elements that lie on top of each other easier, it is recommended to select those in the structure tree  .

2. Four grooves for saw cut:



Here, too, you can use this groove for creating the next groove:

Sixth groove (saw cut, left side)		
Priority		4
	Point 1	Point 2
Reference (X / Y)		

The seventh groove also is created easily the same way.

Seventh groove (saw cut, untere Seite)		
Priority		6
	Point 1	Point 2
Reference (X / Y)		

Eventually, you insert the eighth groove, following the usual pattern.

Eighth groove (saw cut, obere Seite)		
Priority		8
	Point 1	Point 2
Reference (X / Y)		

Looking at the definitions of these grooves you can see that the saw plunges into the workpiece up to the bottom of the panel (counter rotating). The Z-protrusion of the tool makes sure that the panel will be cut through. The Z-protrusion will be added if you refer to the panel bottom side and attach 0 to the Z-value. Indicating priorities makes sure that scoring takes place before sawing. With these last 4 grooves you completed a template for formatting with a saw. Now, select the group and then right-click on the graphic area. In this context menu you finally choose the entry "Deposit on pallet". After that, TwinCAM offers you to save this workpiece description and adds it subsequently as macro to the pallet. Now you have created a complete fomatting template and you can use it for all further formatting.

3.4 Formatting a laminated panel

Unlike in the previous chapter you achieve best results with a milling tool if you format plastic-coated panels. One should turn one's attention to the lead-in-movement of the milling cutter. Because of the danger of pulling-out during the lead-out-movement from the workpiece one does not approach the workpiece from one of its corners. Ideally, the lead-in/lead-out-movement takes place tangential on the center of a panel edge.

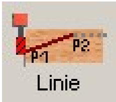
TwinCAM also provides a basic macro for this task.

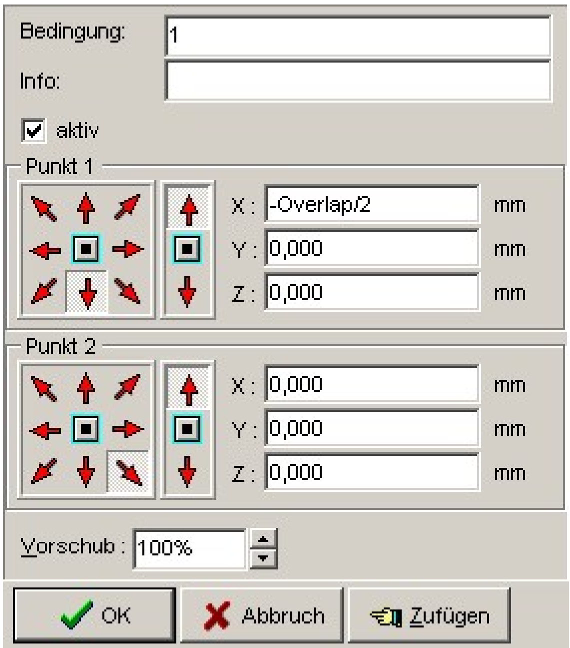
Exercise:

Format the workpiece, using a milling cutter. First, use the macro and then try to understand the steps the macro has taken off of your hands.

For this purpose, click on "Umf-Mitte-re" in the pallet in the register Basic macros. If you cannot find such element in your pallet, you can generate it yourself without any great effort.

In order to do this you only need 5 lines and a starting point. Both two basic elements are in the "Standard"-register of the TwinCAM pallet. Place the 5 lines so that there is a line on each side of your workpiece description. By using Magic-Points, creating these lines is very simple. In order that no corner is left standing, create a variable "Overlap" in the variable dialogue  and attach the value 4 to it. You can enter this variable for the X-value of the first and the last line.





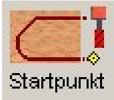
Line 2:		
	Point 1	Point 2
Reference (X / Y)		
Line 3:		
	Point 1	Point 2
Reference (X / Y)		

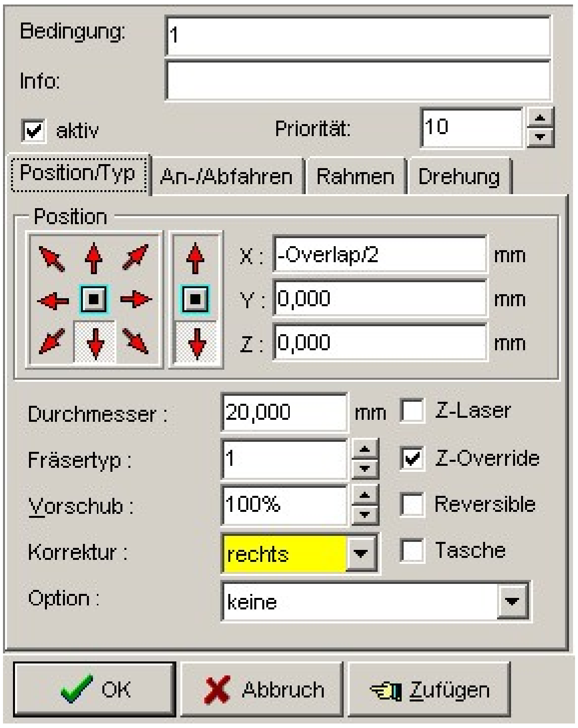
Line 4:		
	Point 1	Point 2
Reference (X / Y)		
Line 5:		
	Point 1	Point 2
Reference (X / Y)		
X	0	Overlap/2
Y	0	0
Z	0	0

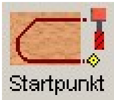
After having placed the 5 lines properly, select all lines with the help of the CTRL-key. Please choose the first line the very first. If you right-click in the graphical area, a context menu will pop up. Choose "group". By reason of the grouping elements TwinCAM adds a starting point to the elements. Only by this starting point the elements have all required technological data as "Correction" or "Type of milling cutter". Doubleclick on the group to edit these technological data.

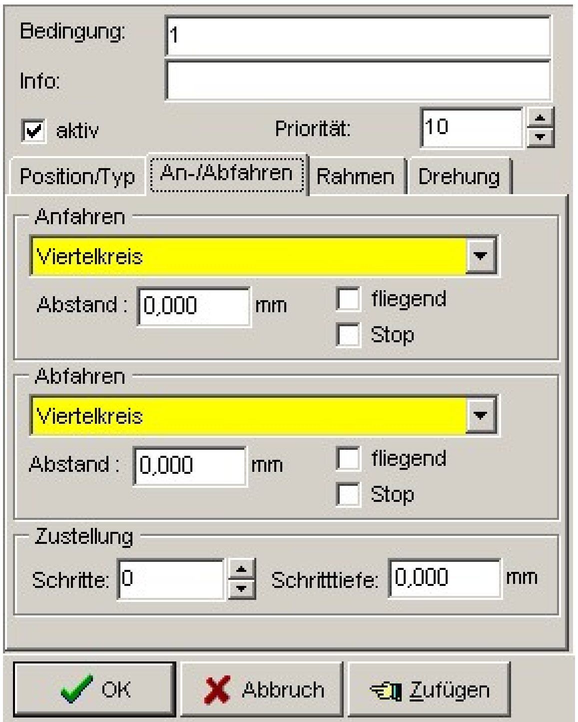
Hinweis:

Alternatively, selecting elements can also be made over the structure tree . Especially if elements lie on top of each other, this way is to be preferred.



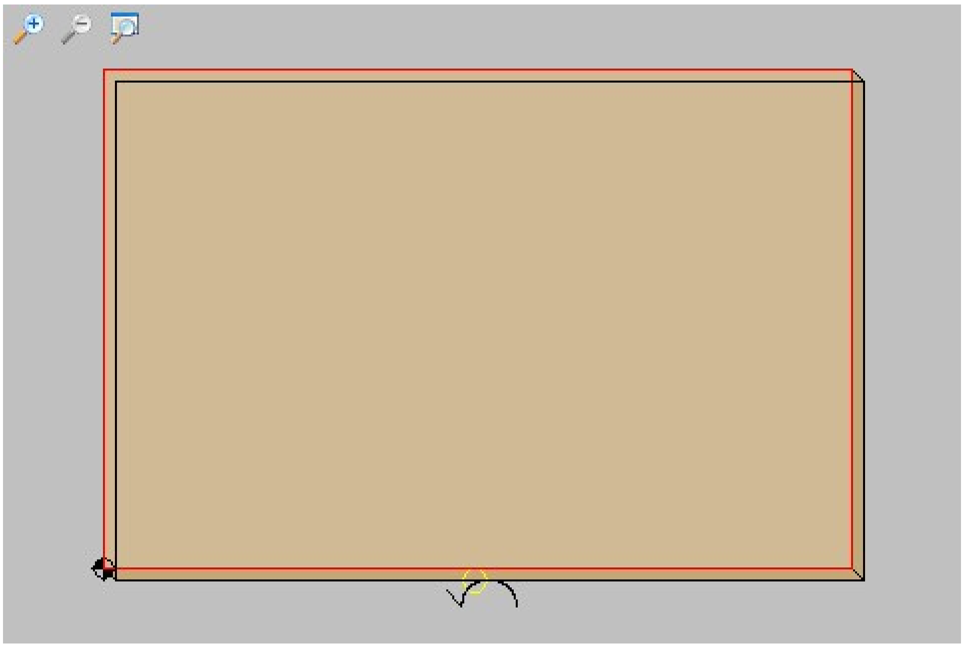






Now, select the group and then right-click on the graphical area. In this context menu choose "Deposit on pallet". Then TwinCAM offers to save this workpiece description and adds it to the pallet subsequently. Now we have created a complete formatting template, usable for all further formatting.

Your workpiece should like this at the moment:

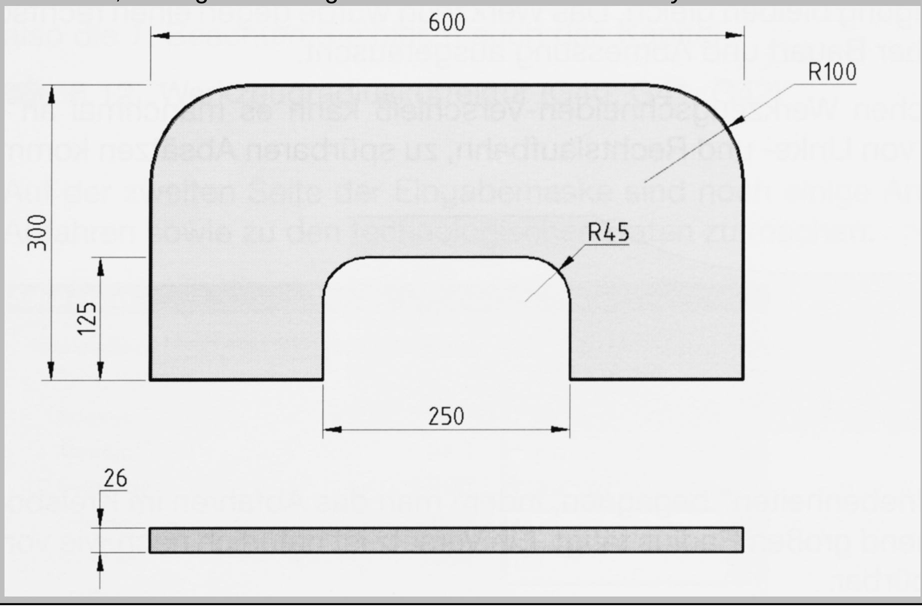


3.5 Milling outer contours

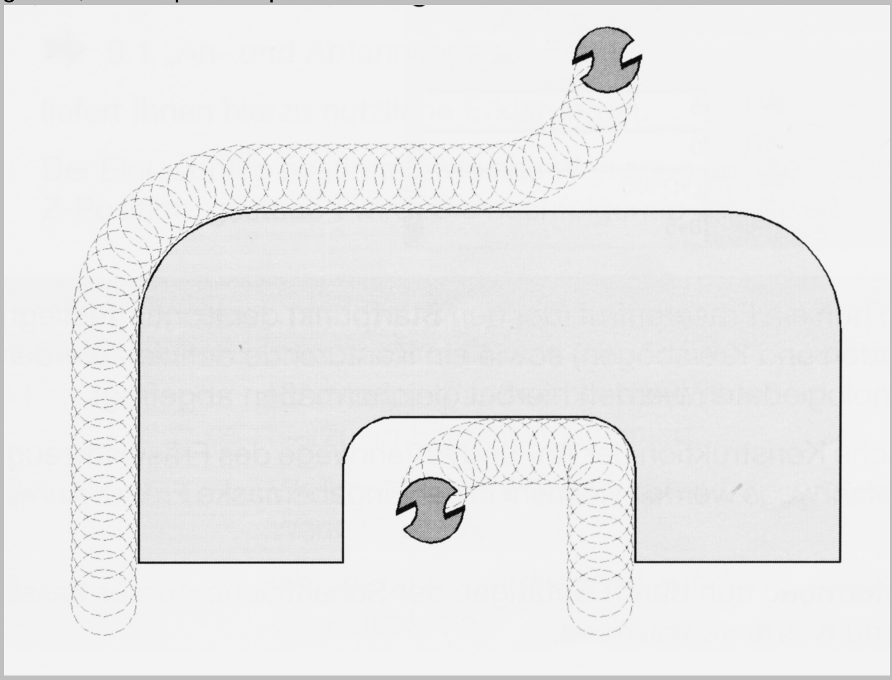
A machining center is downright ideal for milling contours. Without templates all kinds of forms can be created and processed. However, processing solid wood can imply some difficulties. There is an indivual order of processing to be created for each solid wood workpiece. Furthermore, lead-in/lead-out points and movements have to be set carefully, considering tearouts on the corners and edges. Just as processing with conventional woodworking machinery you have to turn your attention to the fiber orientation and the type of wood. With few exceptions, you usually mill across the grain first (counter rotation) and afterwards along the grain.

Exercise:

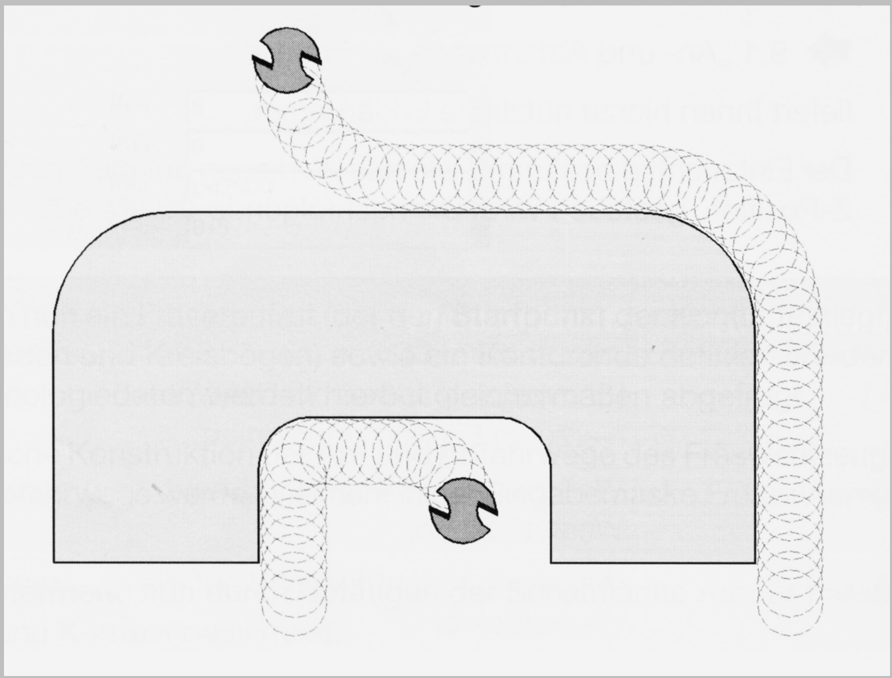
On a counter plate (veneered chipboard), a longitudinal edge has already been glued onto one length side with an edge banding machine. The remaining three sides feature the common blank allowance. Later on, an edge is to be glued onto the outer contour by hand.



Because of the danger of pulling-out, it is necessary to work with a clockwise rotation and an anti-clockwise rotation milling cutter. The first two operations are composed of a left-hand motion tool, approaching the workpiece tangentially in counter rotation. The lead-out movement is supposed to happen tangential, so to speak a quadrant.



The next two operations are to look like this:



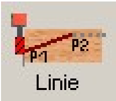
Lead-in/Lead-out movement remain the same. The tool is to be exchanged with a clockwise rotation milling cutter of the same type.

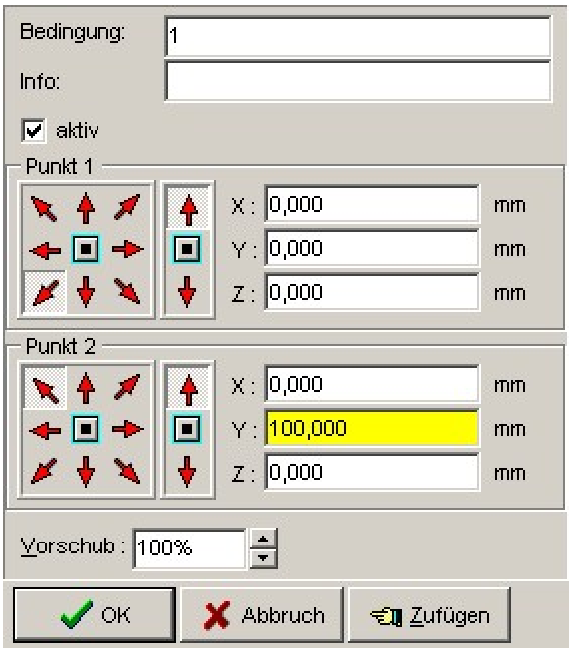
In TwinCAM contours consist of one or more geometric elements and a starting point, containing technology data.

The approach of programming a contour looks like this:

1. Generating the single geometric elements of a contour.
2. Mark all elements that belong to one contour.
   The first element is selected by left mouse click. After that, all further elements can be marked by a left mouse click while pressing CTRL on the keyboard.
3. Right-click on the grouped contour and a context menu will pop up. Select "group".
4. A doubleclick on the generated group opens a dialogue for the starting point.
   Here, all technological information can be entered for processing the contours that were generated a moment ago.

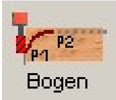
Step 1:
For our example we are going to program 2 lines and one arc of both contours.
1. Contour:
Here the first line:

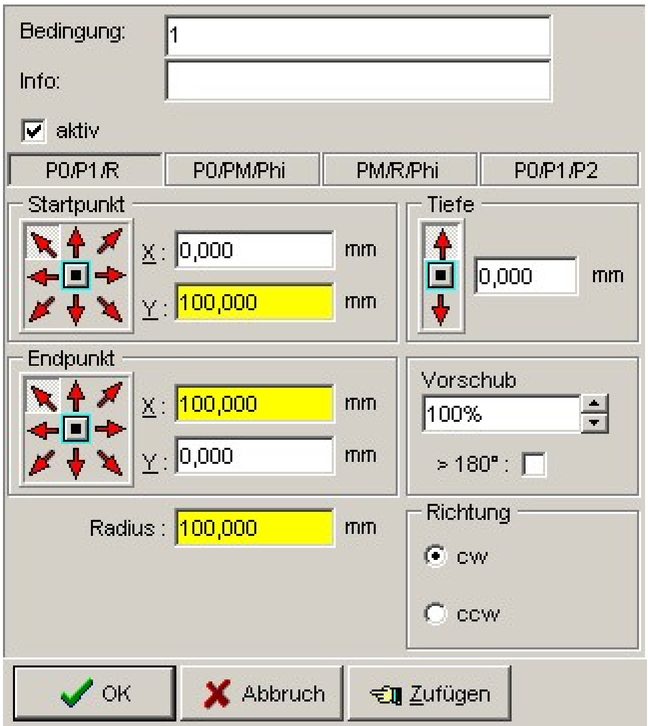




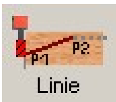
After that, create the arc clockwise.

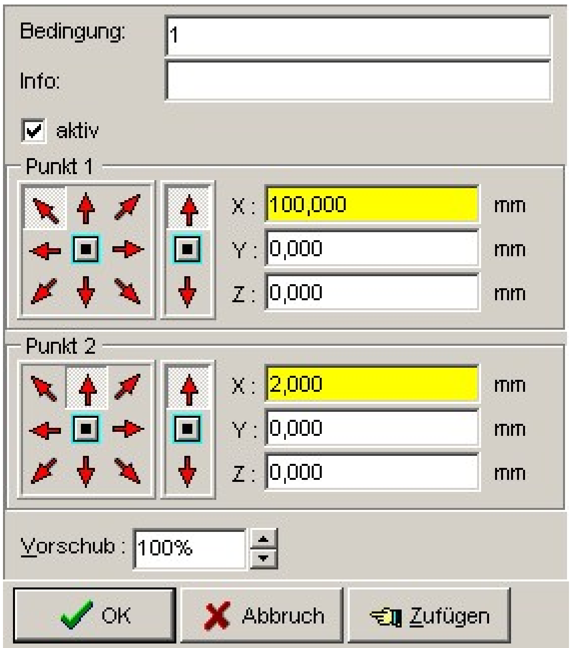
Note:
If you doubleclick on "starting point", the starting point obtains the values of the line’s endpoint. You can use this procedure anywhere. As soon as you doubleclick on a magic point, it will always obtain the values of the last endpoint.



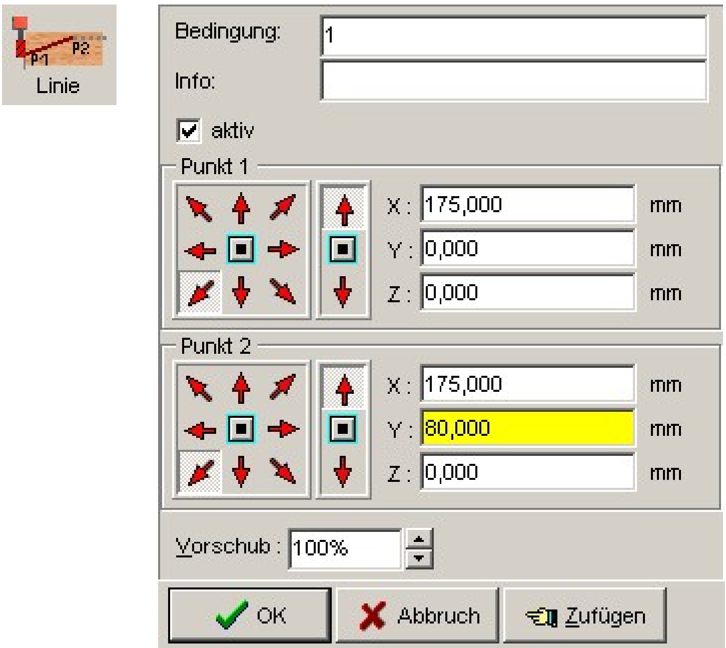


At the end, define the second line.

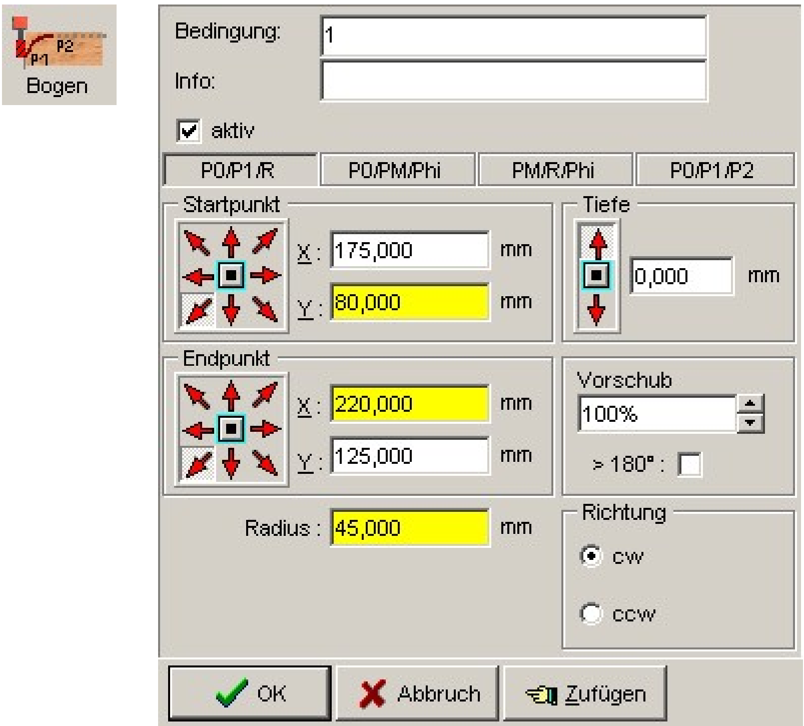




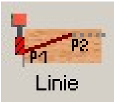
2. Contour:
Now, create the first line of the second contour.

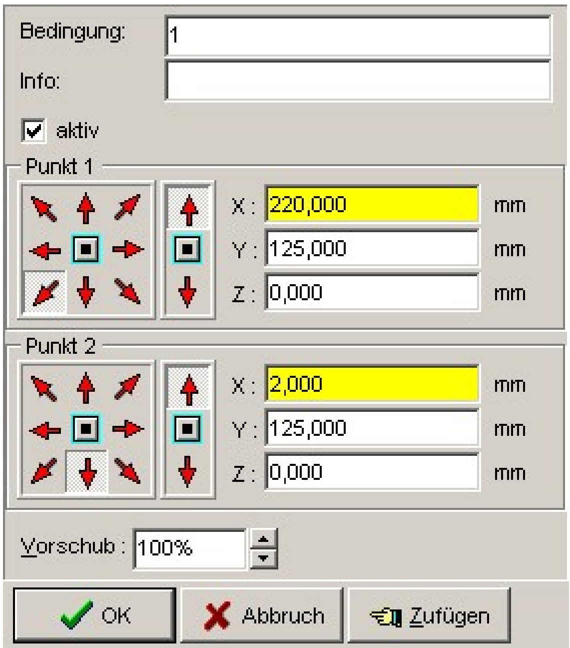


The arc follows.



Now, you only need the second line.

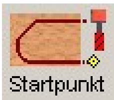


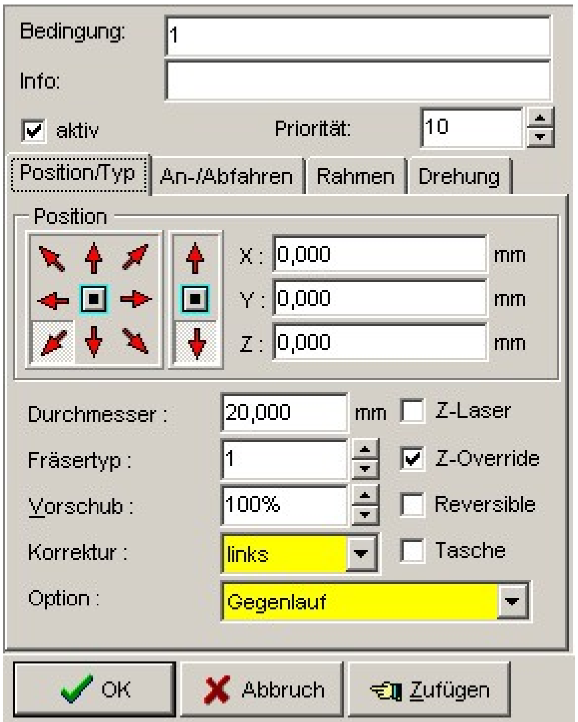


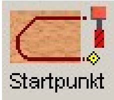
This completes step 1. Follow step 2 – 4 in order to provide all necessary technology data for your contour.

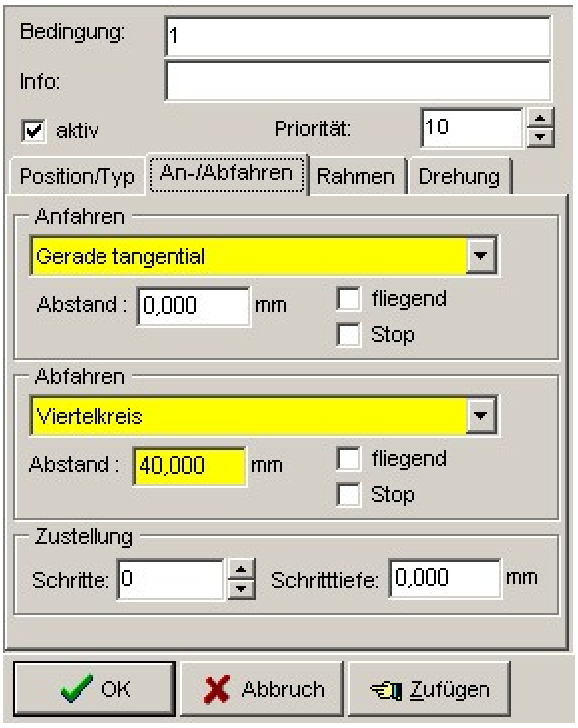
Step 4:

1. Contour:





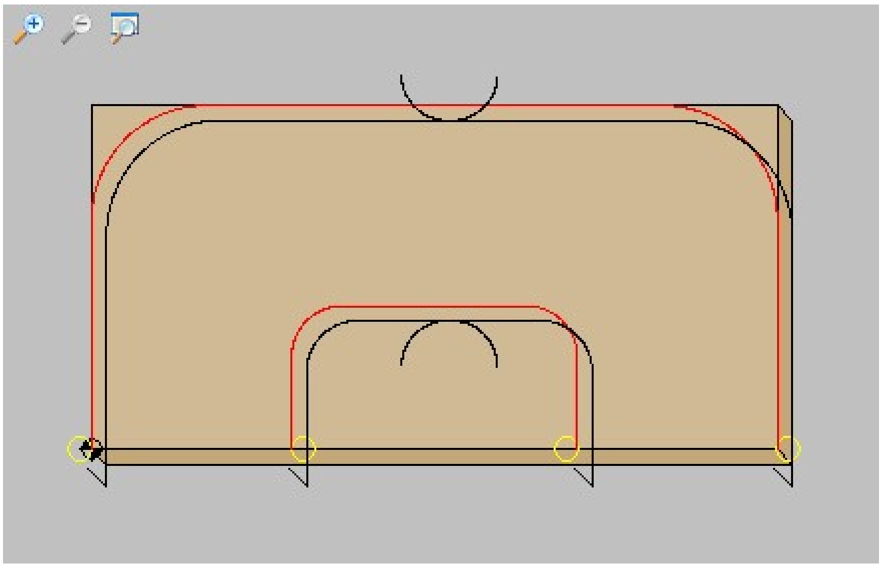




The technological data for the second contour are entered analogously.

Please be aware that TwinCAM is capable of selecting the appropriate tool according to each option (counter rotation/ synchronization). Concerning the lead-in/ out distance TwinCAM always observes at least the tool's safety distance. In order to create the second subcontour, edit the group (doubleclick) and click on „Add“. This way you have duplicated the existing group and you only need to mirror it. You can reach the mirror-functions over the icon bar .

If you generate the final workpiece and let the NC code be viewed, the access and departure paths are shown in the graphical area.

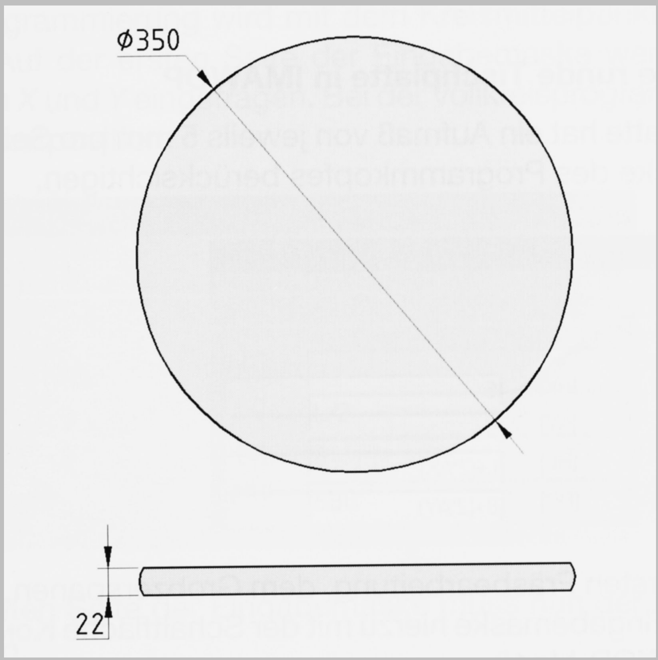


3.6 Milling a round tabletop

A special form of using an outer contour mill is milling a round tabletop. The same rules and recommendations as for milling an outer contour can be applied.

Exercise:

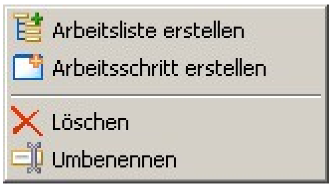
In this example, the following MDF panel is to be milled for a flower stand:



On the edges, a profile milling cutter is to mill a drum profile. First a roughing cutter is to pre-mill the edges in order to mill the contour to the desired dimensions subsequently with the actual profile milling cutter. The rough machining is to take place in counter rotation and a defined clearance to the final contour.

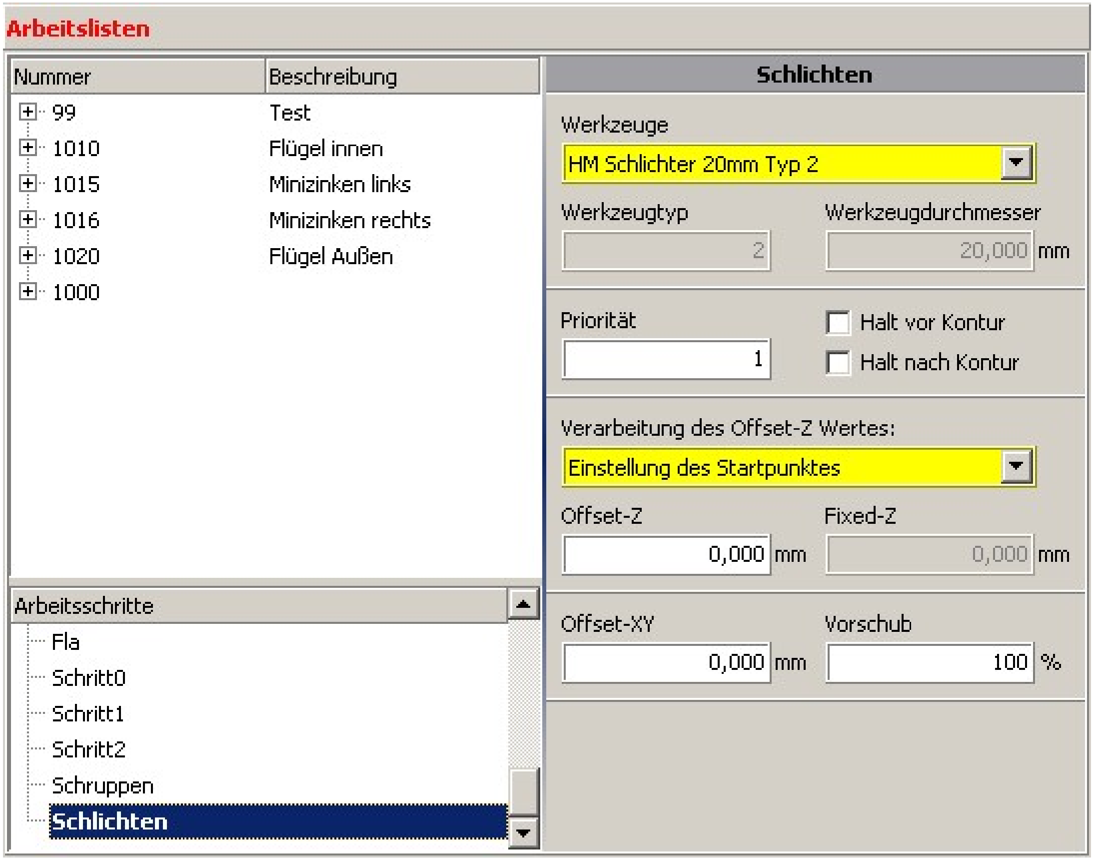
Of course, you could create two contours for both roughing and finishing. However, TwinCAM offers a far more elegant way to manufacture such a workpiece. For these kinds of tasks use the work lists. Then, for the starting point you only need to enter the number of the work list as the type of milling cutter. TwinCAM will then automatically conduct all work steps contained in the work list. This way you could create a work list with the work steps "roughing" and "finishing" and use it in each of your workpiece description.

1. Go to the settings dialog  and select "Work lists".
2. Right-click in the field "Work lists". A context menu pops up from which you select the option "Create work list".



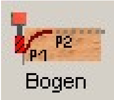
3. Afterwards, right-click in the field "Work steps" and select "Create work step".

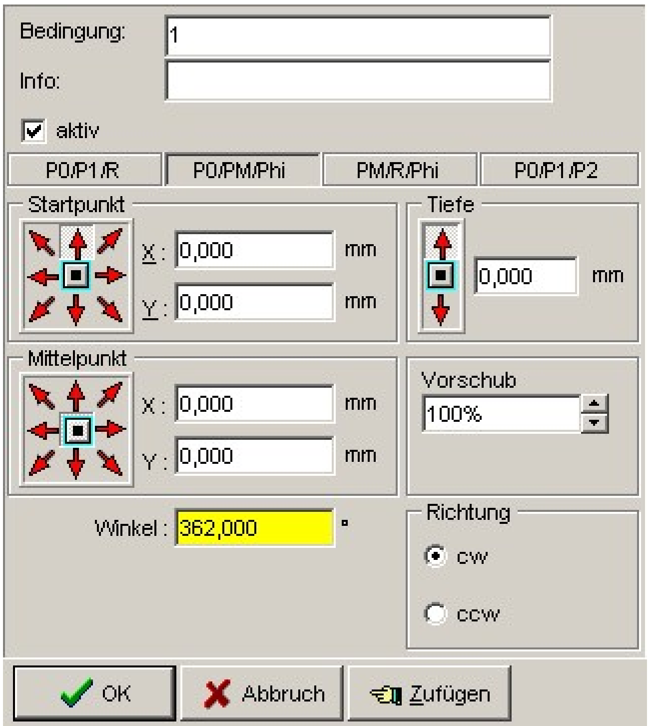
Name this work step "roughing" and attach reasonable values on the listed attributes on the right side. Now, you can create the work step "finishing". As soon as you have created both work steps, place them on your previously created work list using drag and drop. Now, for each following processing in that you use the work list number as tool type, both work steps are conducted automatically.



In order to manufacture a round tabletop in TwinCAM, you need an arc contour. In our example, we want to use a full circle for our tabletop. In TwinCAM you can only define a full circle by selecting an arc with a startpoint, center, and angle (P0/PM/Phi). All other varieties of creating a full circle are unfit.

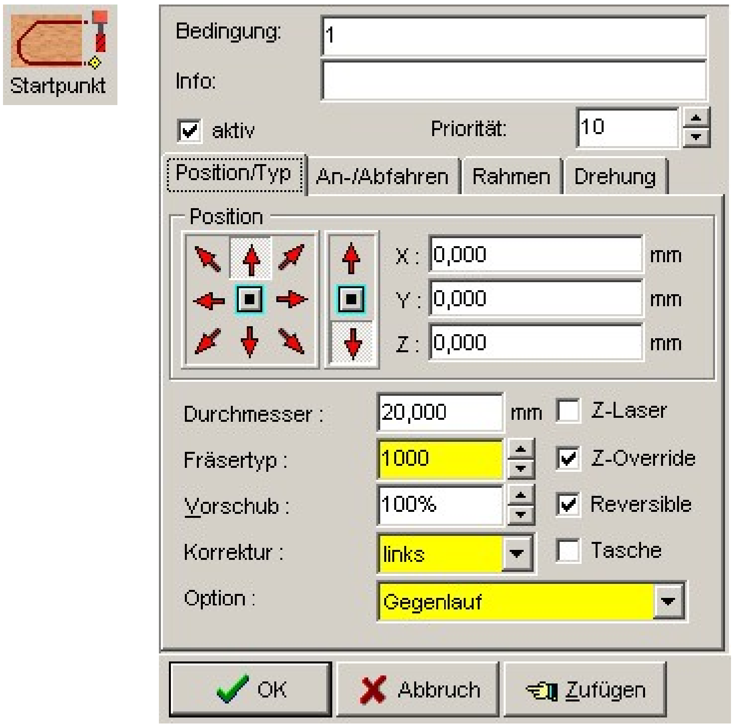
1. At first, place an arc on your panel and revise the attributes in the following way:

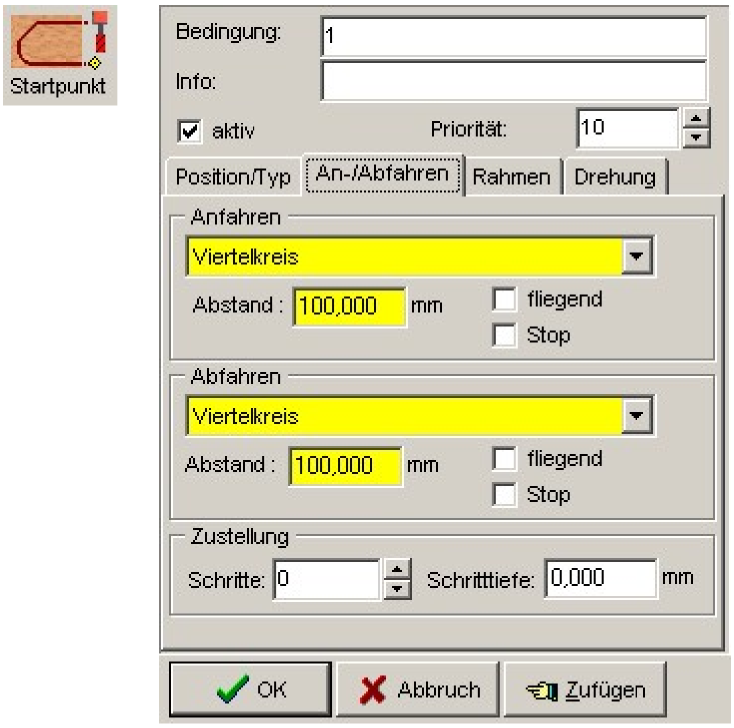




Note:
Create as much automations and standardizations as possible in order to save valuable time.

2. Now, you only need to add the technological data to the arc to accomplish this example successfully. For that purpose, select the arc contour and right-click on the base panel. The context menu that pops up offers you the option to group the contour. Thereby, a starting point is added to the group automatically which provides all technological data. As type of milling cutter we enter the work list we have just created, of course. To avoid tearouts, we want our contour to be milled in counter rotation. As not for both rotation directions a milling cutter is always existent, we define our contour as reversible. This ensures that the contour is always processed in counter rotation.

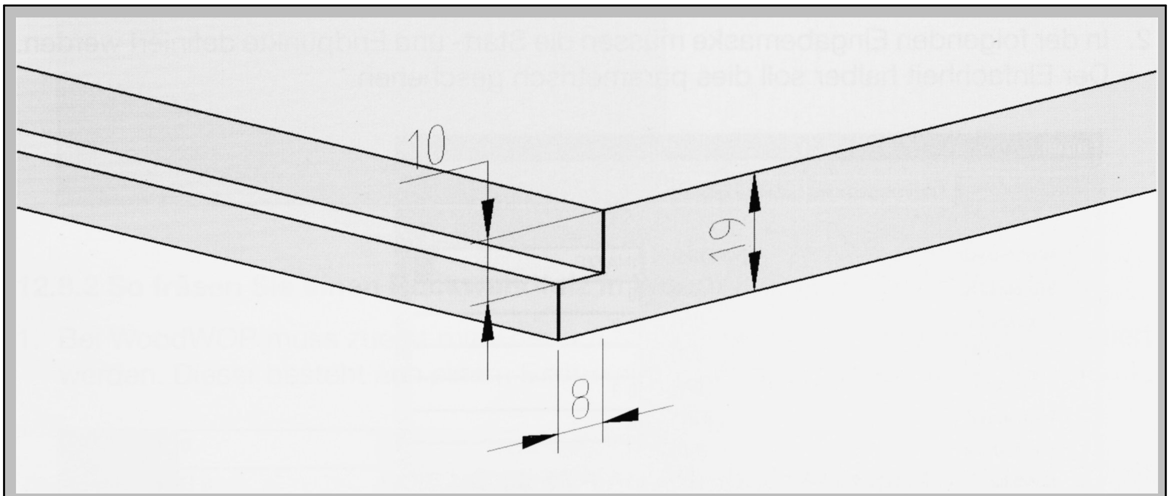




3.7 Milling a back panel rebate

Exercise:

In this example, a continuous back panel rebate is to be milled-out at the bottom panel of a cabinet.

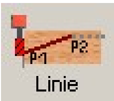


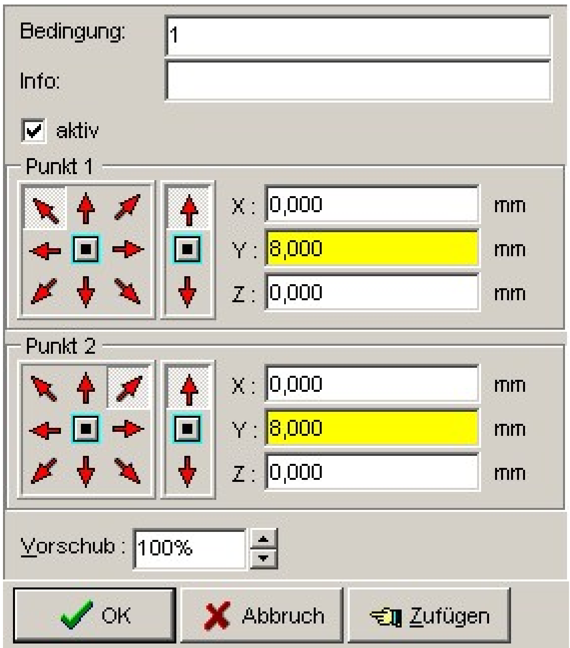
A line is to be used as lead-in/lead-out movement.

In order to program a back panel rebate in TwinCAM, follow the steps below.
The following panel dimensions are taken as a basis:

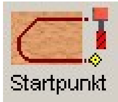
X	Y	Z
800 mm	594 mm	19 mm

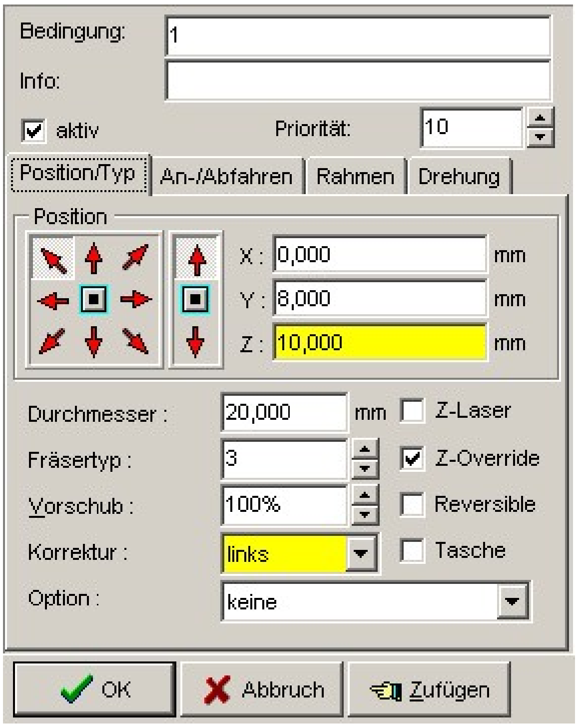
1. Place a line on your base panel, using the following attributes:



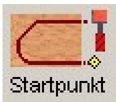


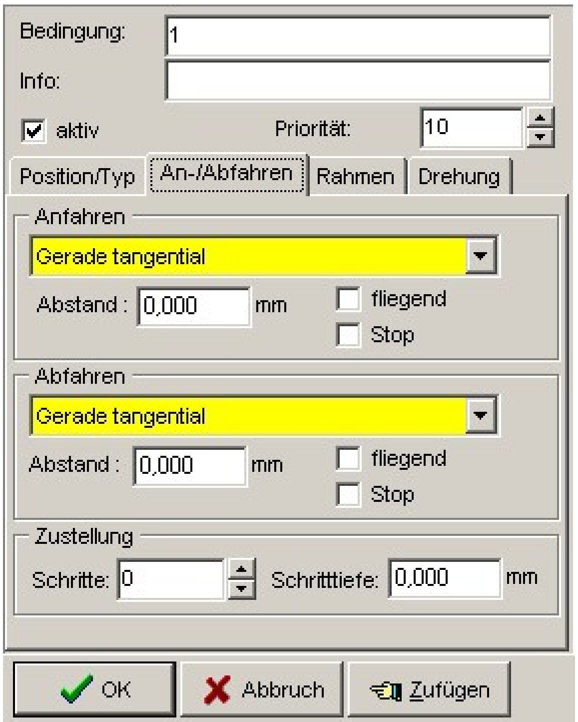
2. As the line alone does not yet provide any technological data, you have to add a starting point to it. Right-click on the line and select "group" from the context menu. If you doubleclick on the contour, you will come to the technological data. Enter the following values in the starting point.





3. Now, you still have to set the lead-in/lead-out movement.

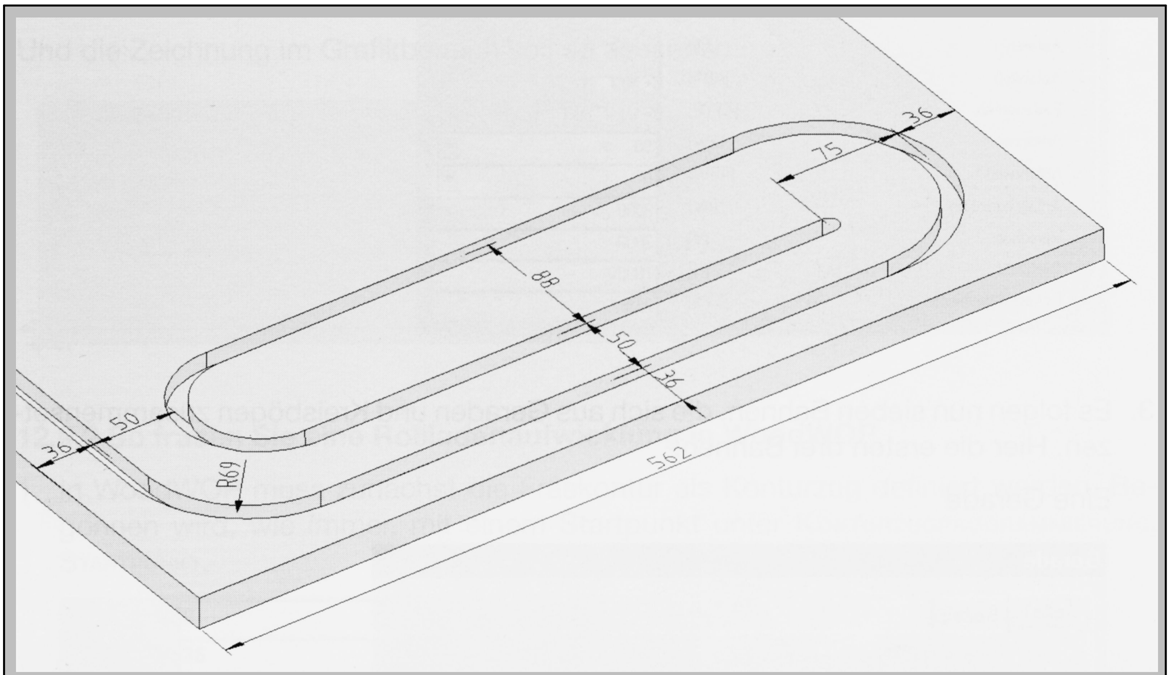




3.8 Milling a roller shutter winding

Exercise:

In the following example, a winding (12mm) for a roller shutter is to be milled out in a furniture part (19mm):

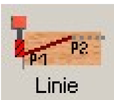


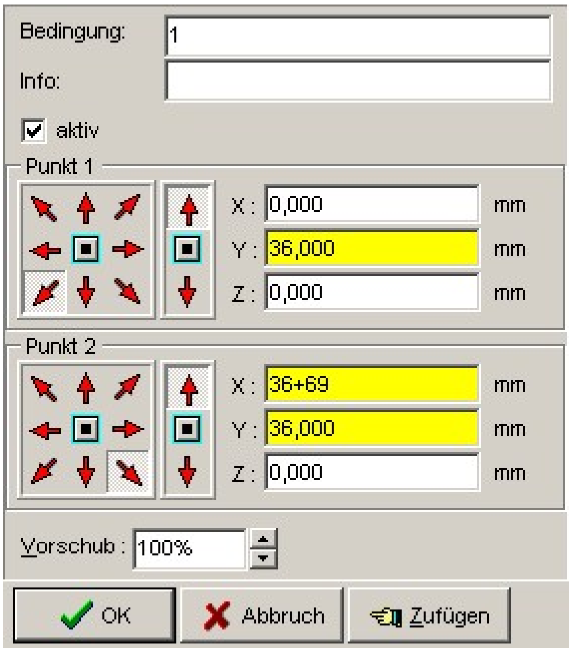
Thereby the milling cutter is to go straight into the workpiece. The lead-out movement has to happen vertically upwards.

1. The drawing consists of 8 parts: 4 lines, 3 arcs and one starting point. In our example, we want to create the first 3 elements. But first of all, you need to attach the following values to your base panel:

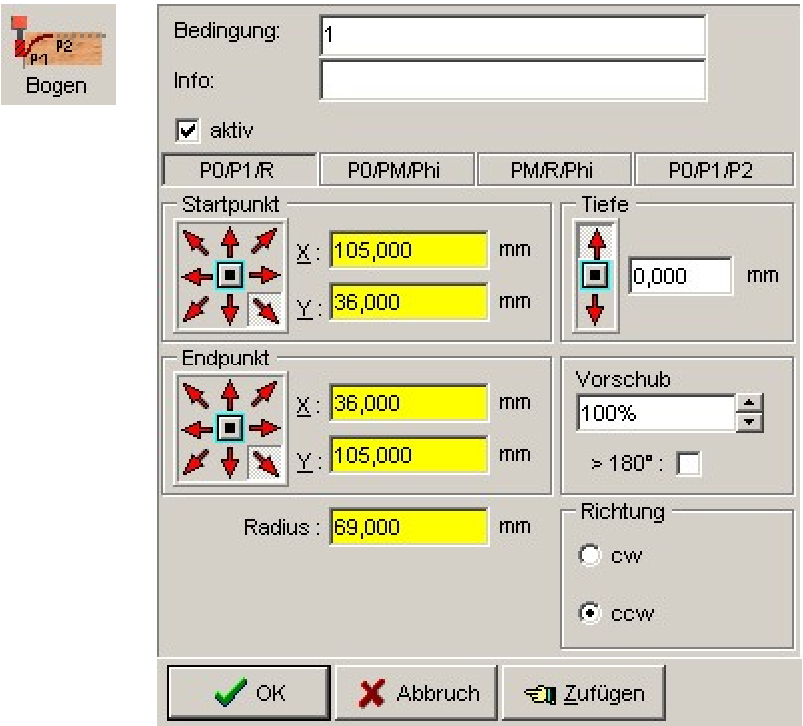
X	Y	Z
600 mm	562 mm	19 mm

2. Now, place the first line on the base panel with the following values:

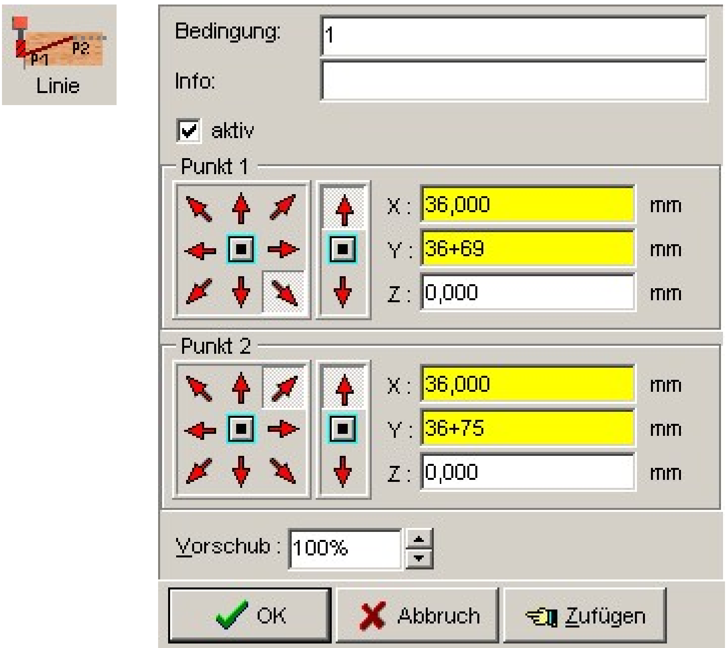




3. Now you need an arc:
   You can doubleclick on "Starting point" to continue at the line's endpoint.



4. Eventually, the second line:
   Here, too, the starting point can be set by doubleclicking on "Point 1".



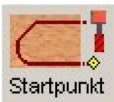
Now, place the missing elements on the base panel. Extract the element's values of the tables below. We will turn towards the starting point in the next step separately.

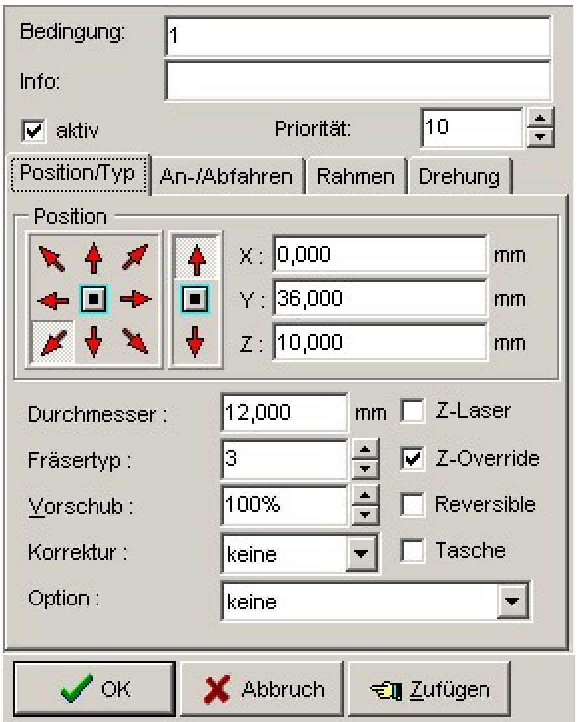
Arc 2 (P0/P1/R)		
	Starting point	End point
Reference (X / Y)		
X	36	36+69*2

Y	36+75	36+75
Radius	69	
Line 3:		
	Point 1	Point 2
Reference (X / Y)		
X	36+50+88	36+50+88
Y	36+75	36+50+44
Z	0	0

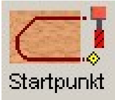
Arc 3 (P0/P1/R)		
	Starting point	End point
Reference (X / Y)		
X	174	86
Y	130	130
Radius	44	
Line 4:		
	Point 1	Point 2
Reference (X / Y)		
X	36+50	36+50
Y	36+75	36+50+44
Z	0	0

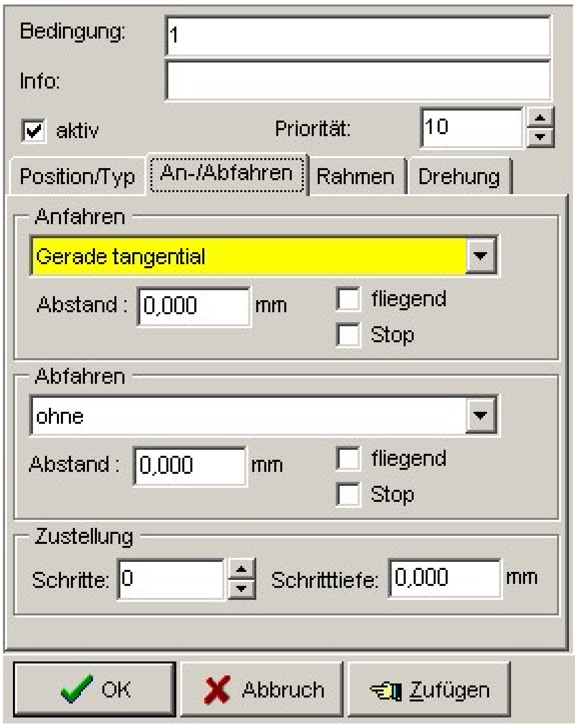
5. In order to add a starting point to your contour, select all elements with the CTRL-key pressed and right-click on the base panel. Select "group" from the context menu which popped up. Having doubleclicked on the generated starting point you should now adjust the appearing dialog.



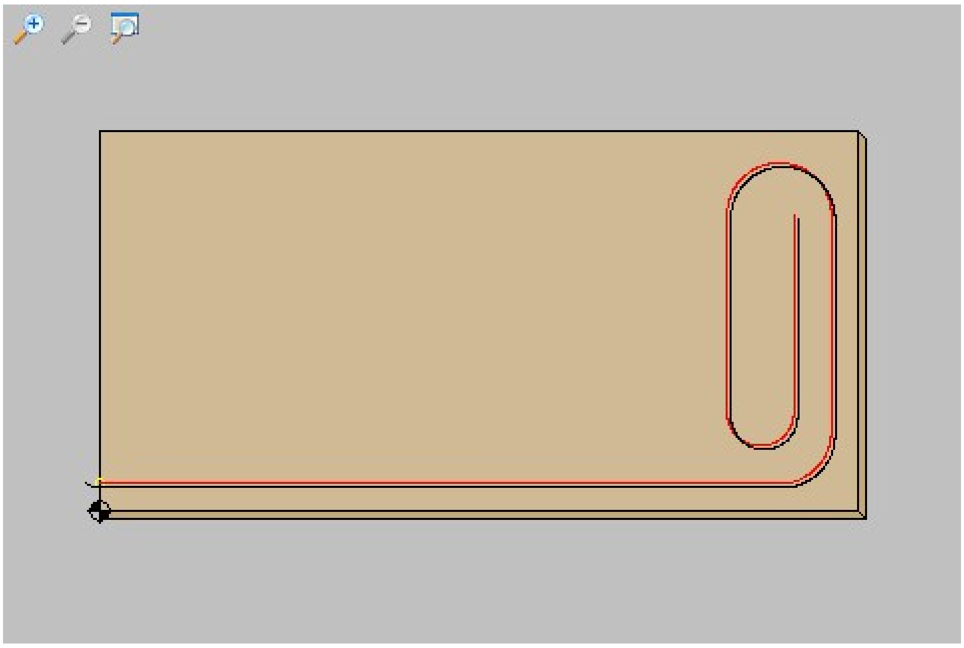


6. Now, you are only missing instructions for the lead-in and lead-out movement.





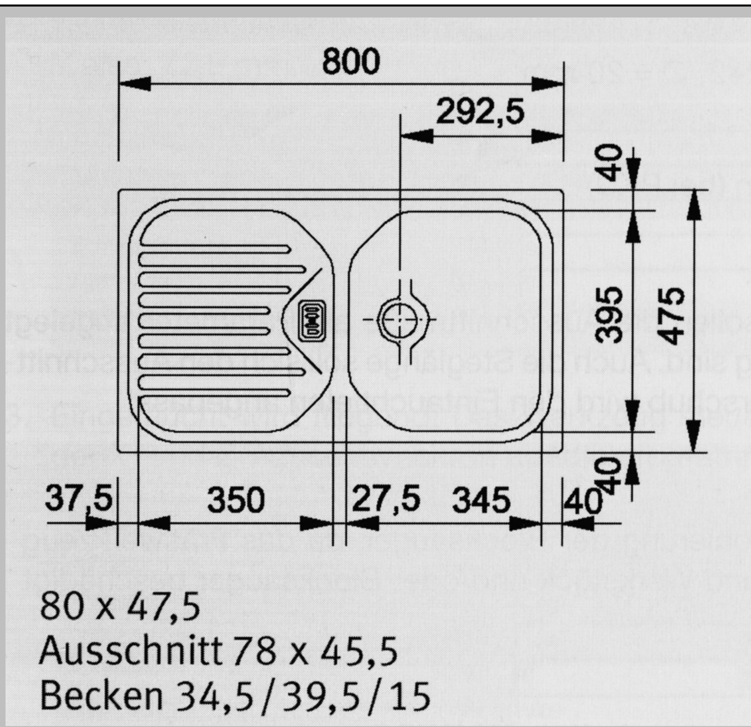
You have now completed your workpiece successfully. Your drawing should look just about like this:



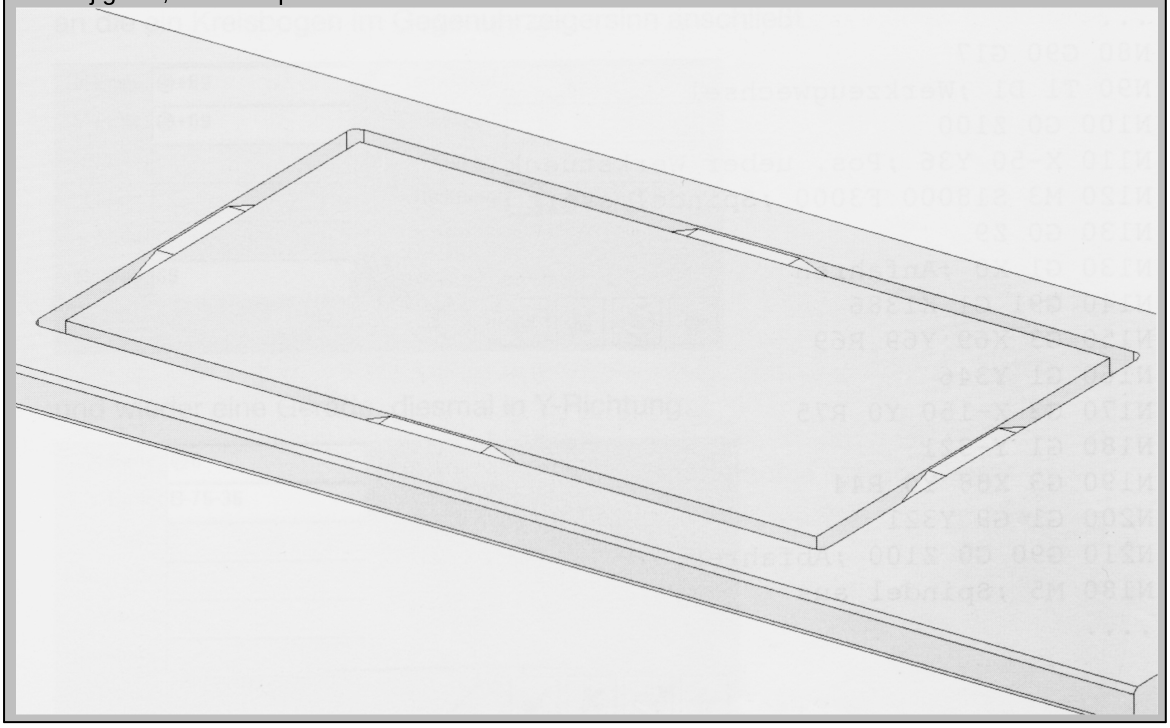
3.9 Milling a sink cut-out

Exercise:

In this example, a sink cut-out for a kitchen countertop is to be milled out. The dimensions are to be extracted from the drawing below:



The rectangular cut-out is not to be milled out completely. The edges are to be milled out because they are formative. The sides of the cut-out are not be milled continuously. The tool is rather to be moved in z-axis while it moves along the contour. This way, four crosspieces remain as "ramps" that will guarantee a safe transportation to the construction site. On site, the cut-out can be made with a jigsaw, for example.



The generation of this workpiece happens parametrically. This creates the possibility of scaling the drawing randomly in order to use it for similar workpiece descriptions. For this, enter the base panel dimensions so that it coincides exactly with size of the cut-out (780x 455 x 38).

1. For this workpiece description you will need 21 lines.
   In our example, however, the first 4 lines shall be enough. The crosspieces are to be 100mm and the ramps are to be 50mm long.

Line 1: (Crosspiece)		
	Point 1	Point 2
Reference (X / Y)	↓	↓
Reference (Z)	↑	↑
X	-50	50
Y	0	0
Z	2	2
Line 2: (Ramp)		
	Point 1	Point 2
Reference (X / Y)	↓	↓
Reference (Z)	↑	↓
X	50	100
Y	0	0
Z	2	0

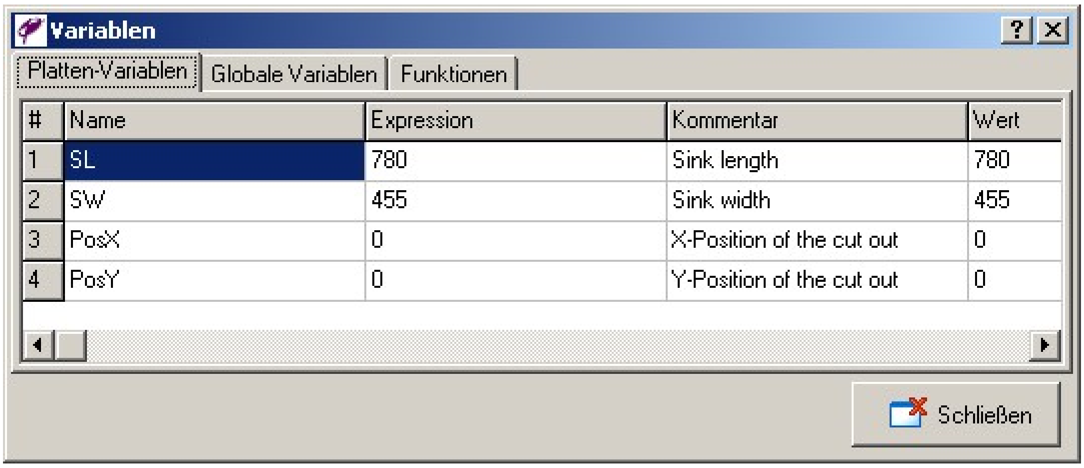
Referencing on the bottom of the panel makes sure that the panel is milled through because the tool's Z-protrusion is added.

Line 3: (milled through)		
	Point 1	Point 2
Reference (X / Y)	↓	↘
Reference (Z)	↓	↓
X	100	0
Y	0	0
Z	0	0

Line 4: (milled through)		
	Point 1	Point 2
Reference (X / Y)	↘	→
Reference (Z)	↓	↓
X	0	0
Y	0	-100
Z	0	0

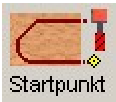
Now, create the missing lines likewise. Please note that starting points and endpoints of the crosspieces are always referenced on top of the panel and are milled with a Z-depth of 2. Ramps go from [top edge; Z=2] to [lower edge; Z=0] and vice versa. For milling through, the start/endpoints are simply referenced to [lower edge; Z=0]. As the lines are created successively, the endpoint of the previous line can serve as the starting point of the new line by doubleclicking on the word "Point 1" in the dialog.

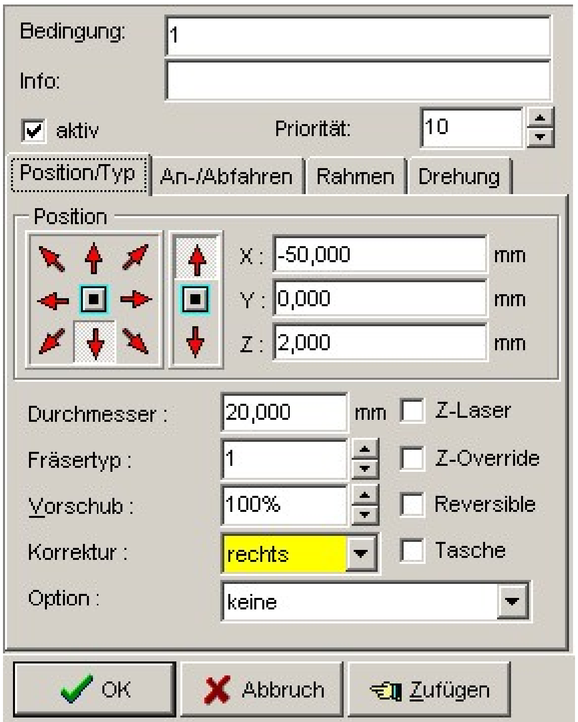
2. Now go into the variable dialog  and create the variables below.

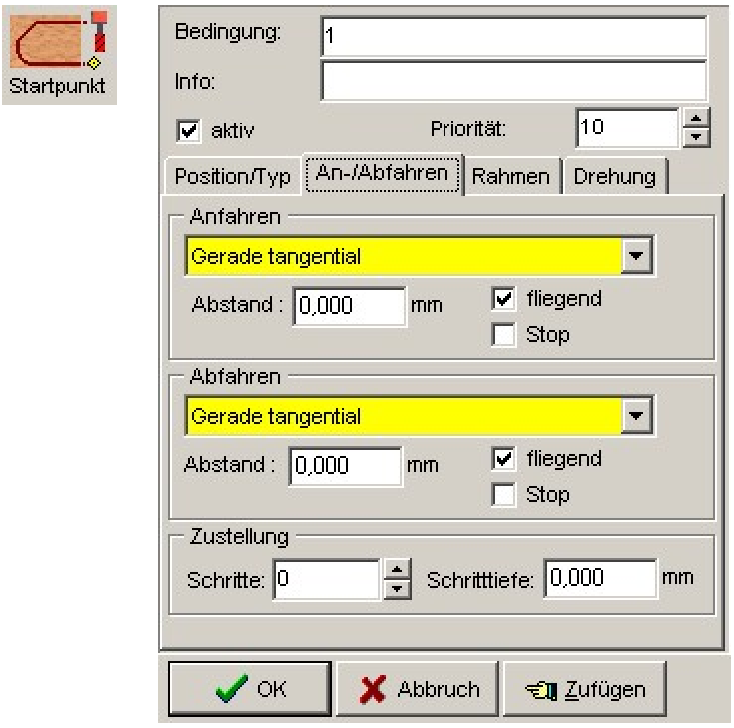


3. Next, select all elements while holding down CTRL. You should be mindful of selecting the first line before all others by all means. The selection can also happen in the structure tree. Now, right-click on the base panel and select "group" from the context menu that popped up. TwinCAM connects a starting point automatically to the first element you have selected before.

4. Additionally, you only need to set the parameters of the starting point. For this, doubleclick on the contour, set the correction on left and disable the function Z-Override.Especially the last change is very important because otherwise TwinCAM will overwrite all Z-values of the contours with Z-values of the starting point.

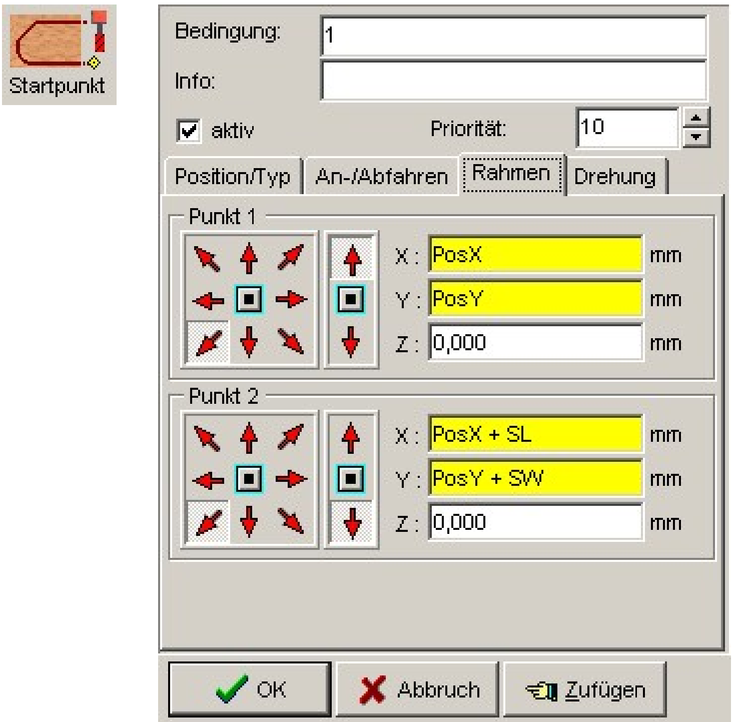




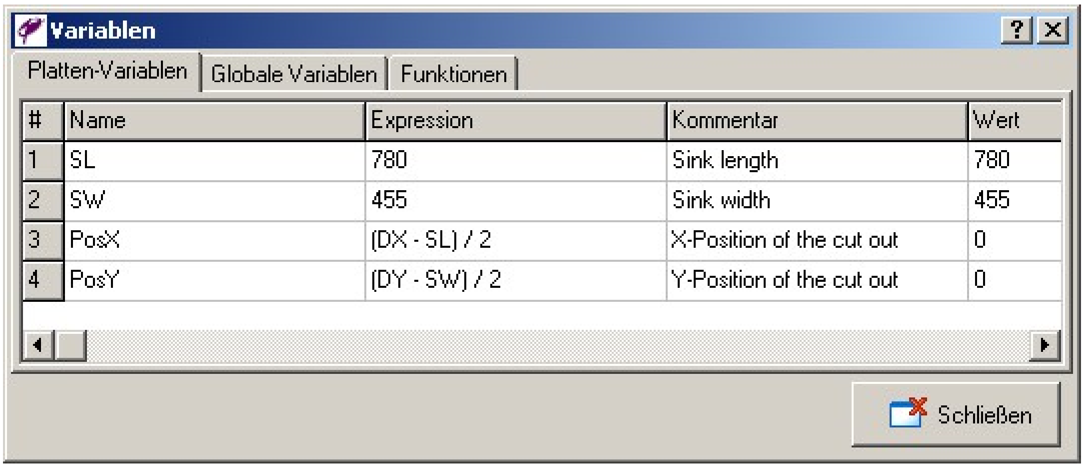


In the register Lead-in/lead out movement, set "straight tangential" and "flying" both for lead-in movement and for lead-out movement.

Now comes the most brilliant thing about this procedure. Go to the register "Frame" and place values in all fields as in the image below.



Now you can change your base panel data randomly. The group frame makes sure that the cut-out is always positioned correctly. Even the size or position of the cut-out can be canged in the variable dialog without any great effort.



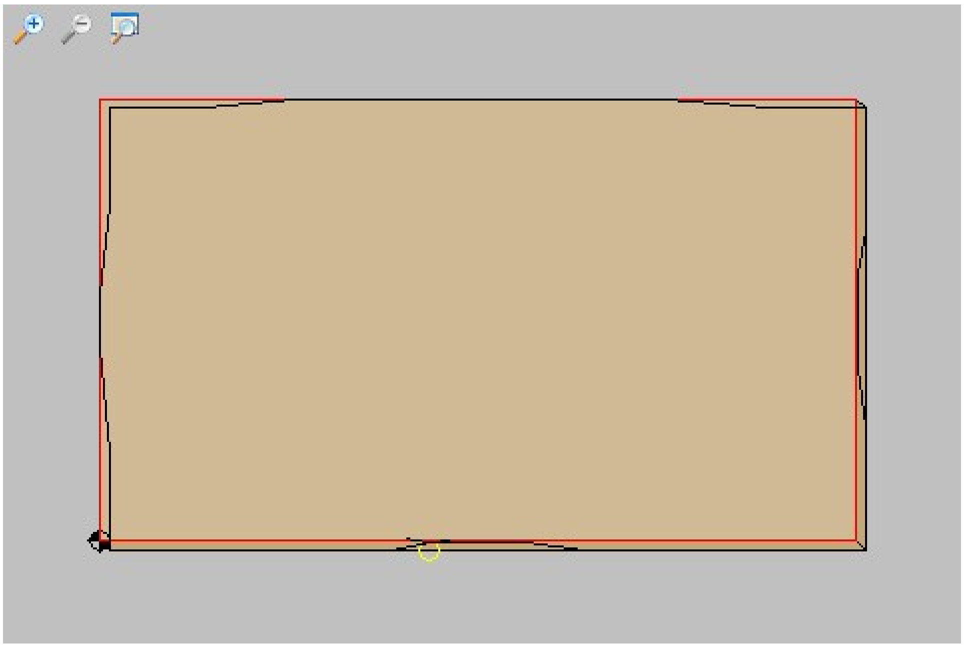
#	Name	Expression	Kommentar	Wert
1	SL	780	Sink length	780
2	SW	455	Sink width	455
3	PosX	(DX - SL) / 2	X-Position of the cut out	0
4	PosY	(DY - SW) / 2	Y-Position of the cut out	0

If the variables PosX and PosY are configured like this, the cut-out will always be made in the center of the panel.

With a little more effort you now have created a flexible solution that is easily adjusted to new customer requests. For another sink cut-out, it would not be necessary to start from scratch. Instead, you can use this workpiece description from time to time.

Note:
You can work even more efficiently if you place this group on your pallet. For this, select the group and right-click on the base panel. Then, select "Save to pallet" from the context menu.

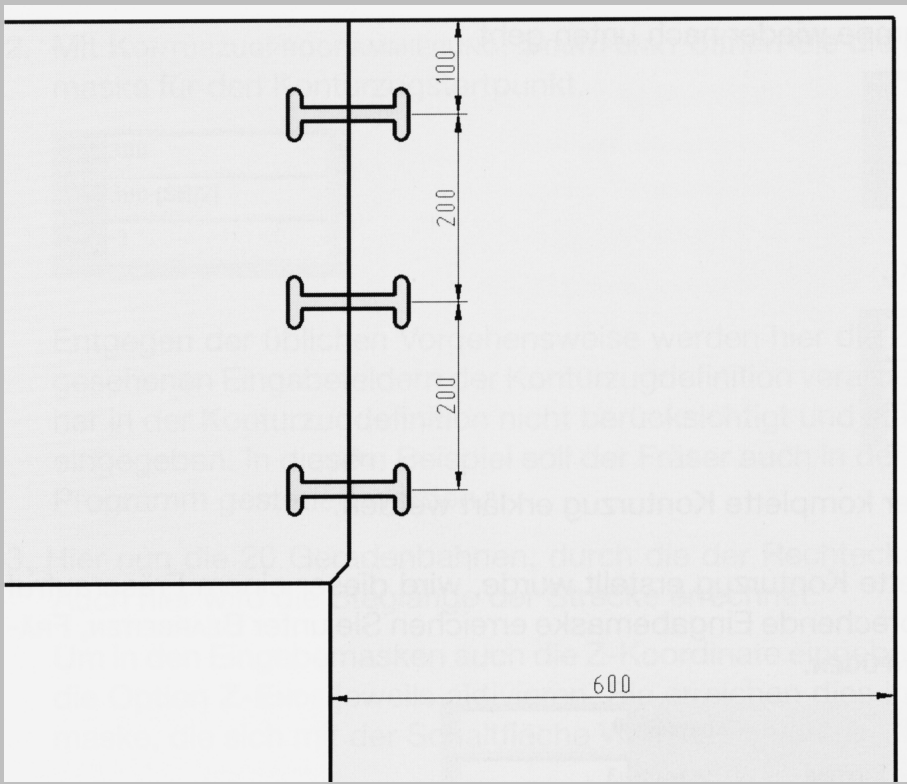
Finally, your workpiece description should look like this:

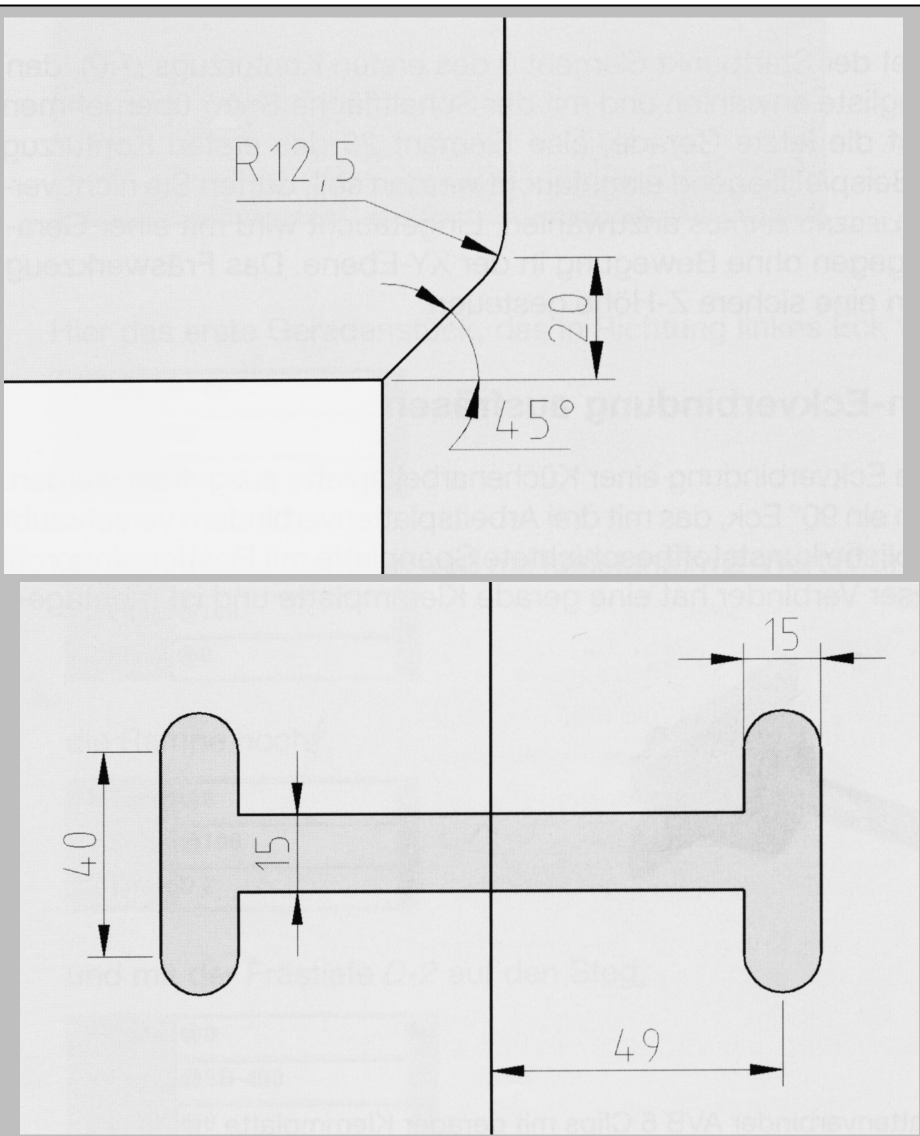


3.10 Milling cut-outs for worktop connectors

Exercise:

In this example, a corner connection for a kitchen worktop is to be milled out. It is a 90° worktop that is bolt together with three worktop connectors. The details of the contour cutting and the cuttings for the connector fitting stand below:


100
200
200
600



The depth of milling is supposed to be 20mm.

1. Adjust the panel dimensions to the desired size (1200 x 600 x 38).
2. We split the first contour into two equally sized halfs. Therefore, create the following three lines:

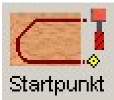
Line 1:		
	Point 1	Point 2
Reference (X / Y)		
Reference (Z)		
X	0	300
Y	20	20
Z	0	0
Line 2:		
	Point 1	Point 2
Reference (X / Y)		
Reference (Z)		

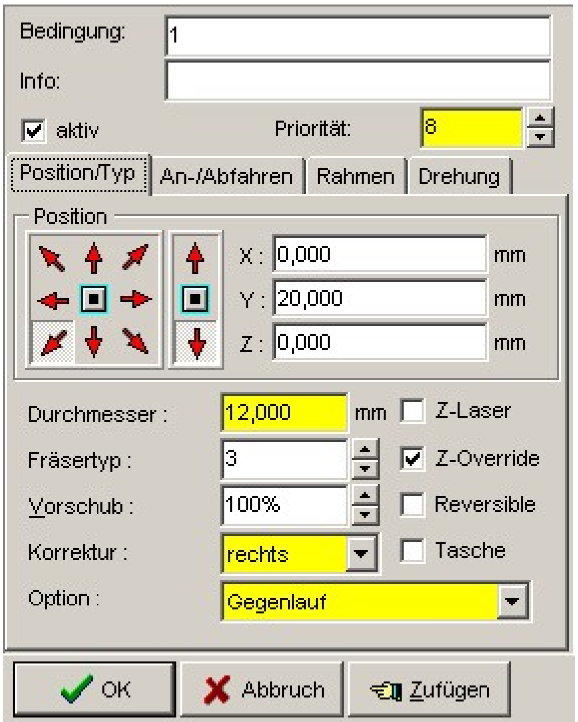
X	600	580
Y	0	20
Z	0	0

Line 3:		
	Point 1	Point 2
Reference (X / Y)		
Reference (Z)		
X	580	300
Y	20	20
Z	0	0

3. Now choose from the CAD functions "Fillet" and enter 12,5mm in the dialog. After that, select the second line's endpoint and then the third line's starting point. The CAD function is found in the right upper corner. The result can be checked with the zoom function .

4. Now you only need to add the technological data to your contours. For this, group the first line olely, then group the second line, the arc and the third line. As you group the second group, make sure that you select the second line first. Then the starting point will be placed on the right spot automatically. Do not forget: Give both groups a lower priority than for the groups of the connector fittings. Up next, alter the features of the first starting point in the following way.





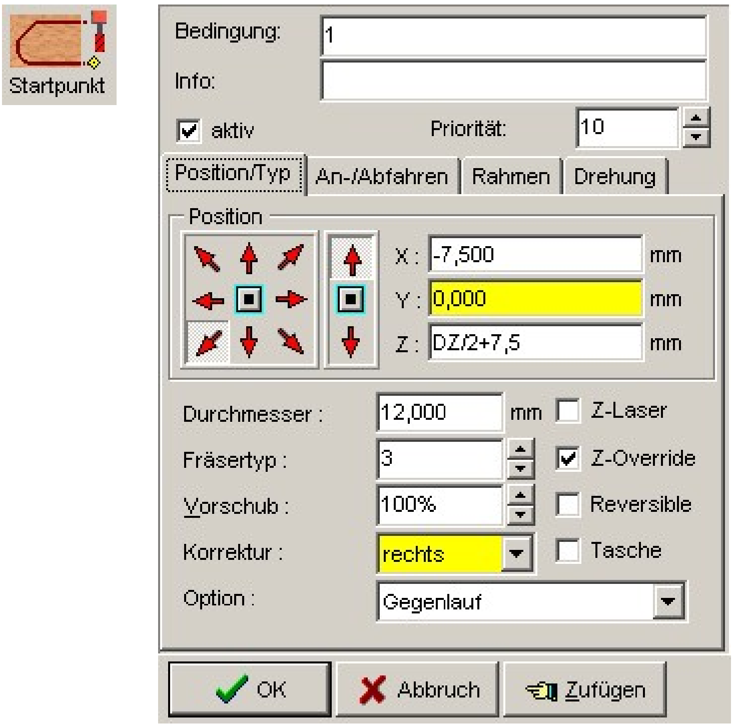
In the register Lead-in / Lead-out movement adjust the attach movement to "straight line tangential" and the leave movement to "quadrant" with the distance 30 in order to ensure retracting from the material. For the second group, the same adjustments are to be set, except the correction has to be set on "left" and the priority set on 9.

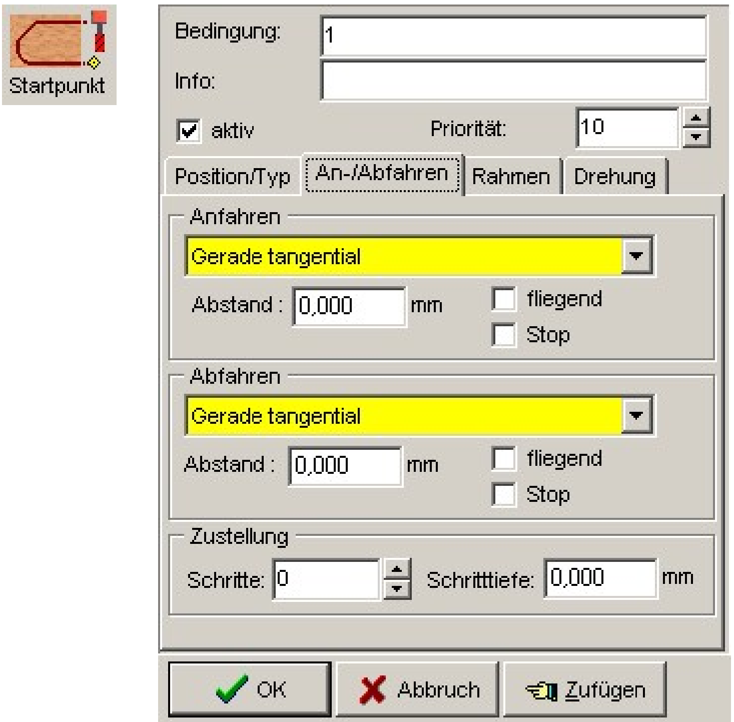
5. The first connector fitting follows: being grouped afterwards, it is very simple to multiply the remaining connector fittings.

Line 1:		
	Point 1	Point 2
Reference (X / Y)		
Reference (Z)		
X	-7,5	-7,5
Y	0	49-7,5
Z	0	0
Line 2:		
	Point 1	Point 2
Reference (X / Y)		
Reference (Z)		
X	-7,5	-7,5-12,5
Y	49-7,5	49-7,5
Z	0	0

Bogen (P0/P1/R)		
	Point 1	Point 2
Reference (X / Y)		
X	-7,5-12,5	-7,5-12,5
Y	49-7,5	49-7,5+15
Radius	7,5	
Line 3:		
	Point 1	Point 2
Reference (X / Y)		
Reference (Z)		
X	-7,5-12,5	7,5+12,5
Y	49-7,5+15	49-7,5+15
Z	0	0

One more arc as well as two more lines follow which are not displayed here for the sake of saving space. These elements are to be grouped, also. In doing so, the first line should be the first element of the group. Adjust the group's features as following and press OK to confirm.





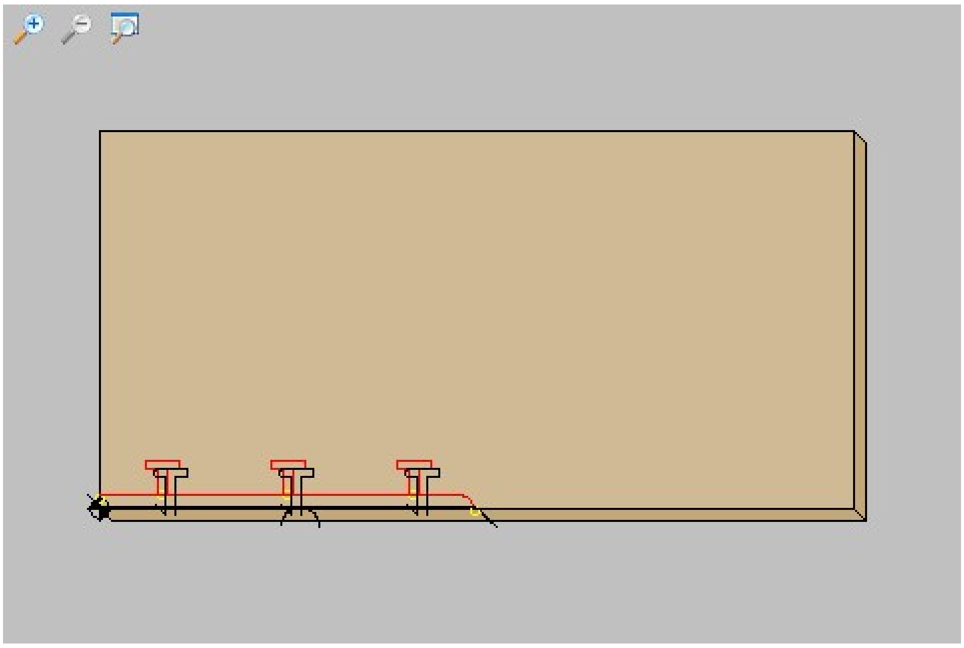
6. In order to add the remaining connector fittings now, edit the previously created group and go to the register "Frame". Here, you have the option to shift the entire group frame. Enter the shifts given in the tables and click on OK for the first group. Open the edit dialog for this group again, enter the shift for the second fitting, but this time click on "Add". Proceed in the same manner for the third fitting.

1. Connector fitting		
	Point 1	Point 2

Reference (X / Y)		
Reference (Z)		
X	100	0
Y	20	0
Z	0	0
2. Connector fitting		
	Point 1	Point 2
Reference (X / Y)		
Reference (Z)		
X	300	0
Y	20	0
Z	0	0

3. Connector fitting		
	Point 1	Point 2
Reference (X / Y)		
Reference (Z)		
X	500	0
Y	20	0
Z	0	0

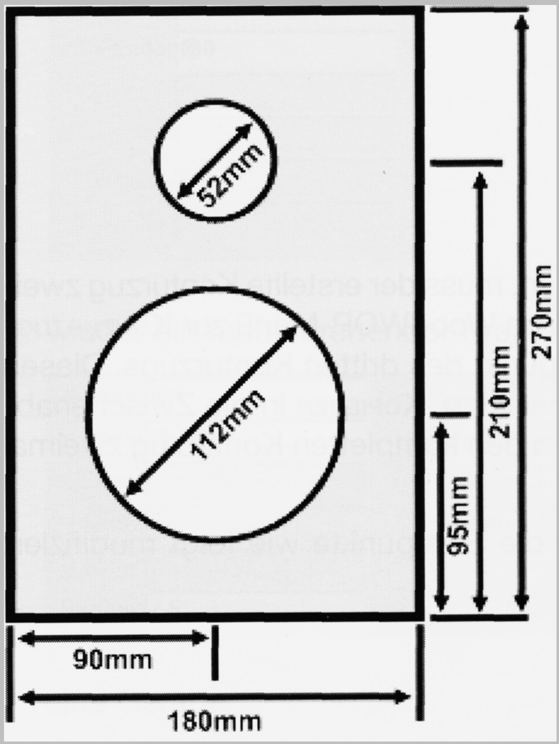
You have created all elements, now. Having enabled NC code generation and displayed the NC Code, your workpiece should look like this:



3.11 Milling speaker holes

Exercise:

In this example two speaker holes are milled out:



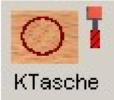
In the lead-in and lead-out movements the tool is supposed to have a flying immersion and removal.

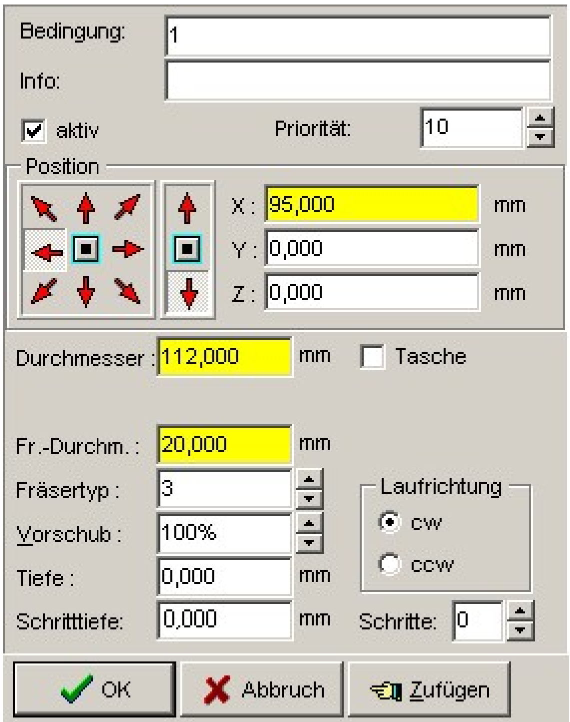
Note: Since the scrap pieces are not machined, they should be held by a suction pad in practice.

This workpiece turns out to be easily programmed in TwinCAM because a basic element for this is already provided.

This basic element can create a circular pocket or a circle sector. In contrast to the previous elements, an additional starting point is not needed, here. All technological data for programming a pocket are recorded in the input dialog for this element.

1. Having set the dimensions of the base panel (270 x 180 x 19), place two pockets on the base panel.
   The first speaker hole:



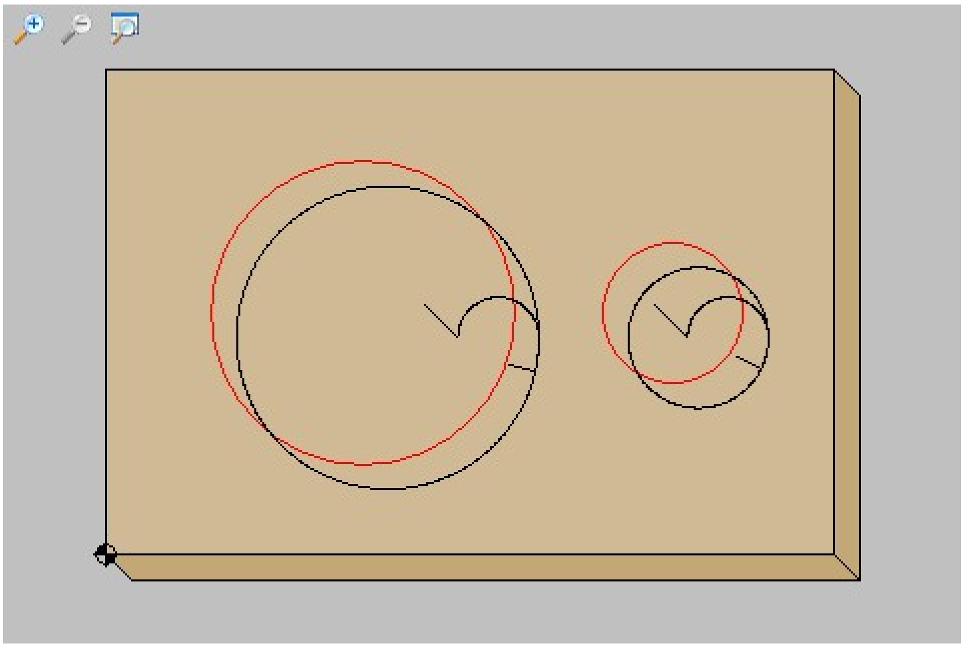


Note the button "Pocket", please. Here you decide wether the machining is a pocket or a cut-out. A pocket is always cleared out whereas a cut-out is only milled out. In both cases, TwinCAM calculates the lead-in and lead-out movement automatically. A pocket is displayed with a hatching whereas a cut-out is simply displayed as a circle.

For the second speaker hole you only have to change position, diameter and the tool diameter of the first hole according to the second hole in the drawing. Click on "Add".

Note:
If a circle sector is supposed to be created without leaving a scrap piece, a pocket with a depth greater or equal the panel thickness can be inserted. In this case, the scrap piece is completely machined.

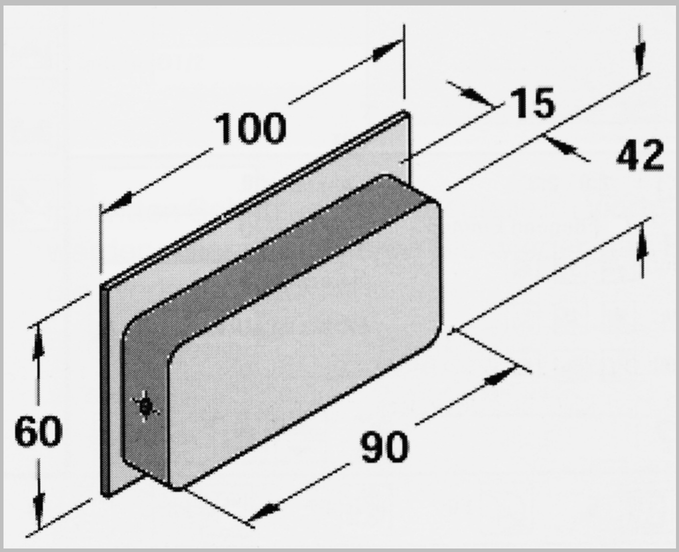
Having enabled Show NC code and NC code generation, your workpiece should look like this:



3.12 Milling a pocket

Exercise:

In this example, a rectangular pocket for a shell grip is milled out. The installation dimensions are found in the drawing below:



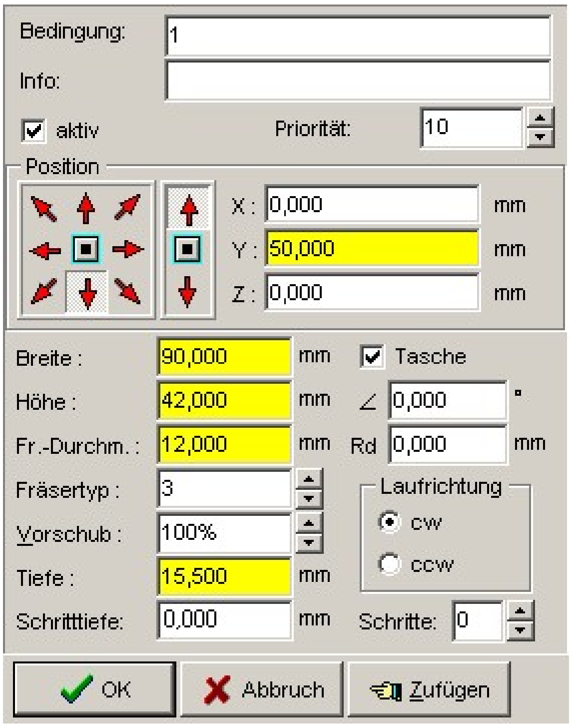
The workpiece, a cabinet door, is a 19mm multiplex board.

For our example, we want to use a rectangular pocket. This element supports the programming of rectangular pockets or cut-outs. Many details are equivalent to the circular pocket. Just as a circular pocket, a rectangular pocket does not need a separate starting point because all required technological data for milling are already recorded in its dialog.

The following panel dimensions are taken as a basis:

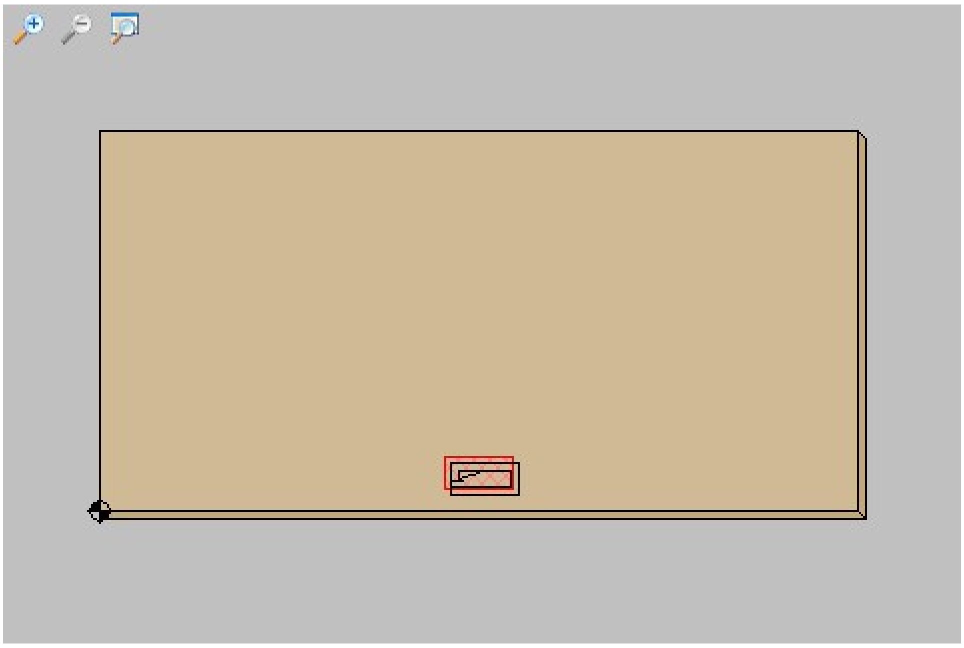
X	Y	Z
1000 mm	500 mm	19 mm

1. Create a rectangular pocket with the following features:



Note:
The corner radius is at least as big as the tool radius. Only if a bigger corner radius is given, the tool is moved with an appropriate modified track.

After inserting this pocket, the generated workpiece should look like this:



3.13 Sawing a sliding door groove

Basically, grooves can be milled. In many cases, however, it is more efficient to cut them with a saw. Many machining centers have an own saw unit for that, but often it can only move in two directions, either in X-axis or in Y-axis. If a saw unit from the tool magazine is put in the main spindle, it is mostly rotatable over the C-axis in any random direction. This way, it is applicable in a more flexible way.

Exercise:

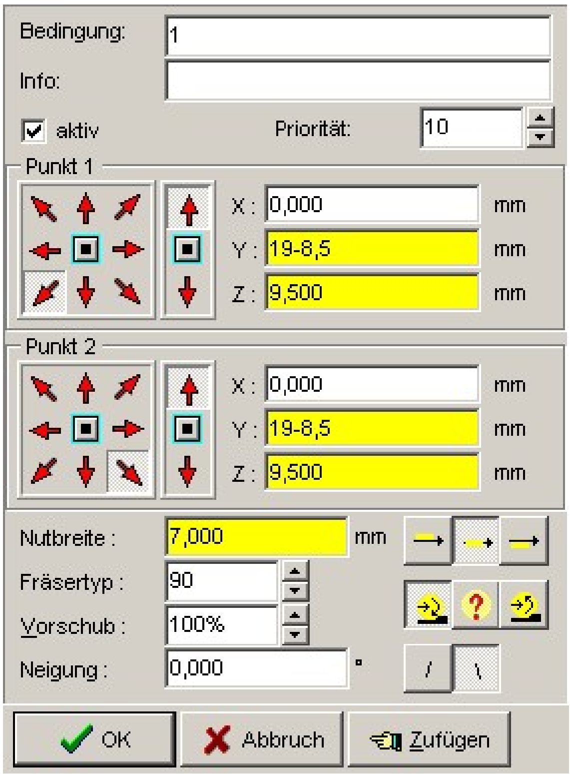
In this example, a groove for a running rail in a cabinet bottom is to be sawed with the help of a saw unit. The groove must have a width of 7 mm and a depth of 9.5 mm.

The following panel dimensions are taken as a basis:

X	Y	Z
600 mm	400 mm	19 mm

The basic element "Groove" is provided for grooves or saw cuts.

For our example, you only have to change the groove in the following way:



Note:
If a groove is drawn unto the outer edge of the panel, the travel starts outside of the panel. Other than that, the travel is calculated so that the contour is not violated.

3.14 Drilling dowel holes

Most modern machines are able to drill horizontally and vertically into workpieces.

Exercise:

In the following example a vertical row of dowel holes is to be drilled in a cabinet side.

In the example, the required dimensions of the cabinet are 1000 x 600 x 2000 mm.
Resulting from the cabinet's design, the cabinet sides are 594 mm. The material is 19 mm thick.

6 dowel holes are to be drilled. The positions on the y-axis are 37, +32, +96, +128, +128, +128.

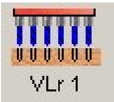
Even rows of holes can be easily expressed parametically in ElemtsCAM. This applies to both horizontal and vertical rows of holes.

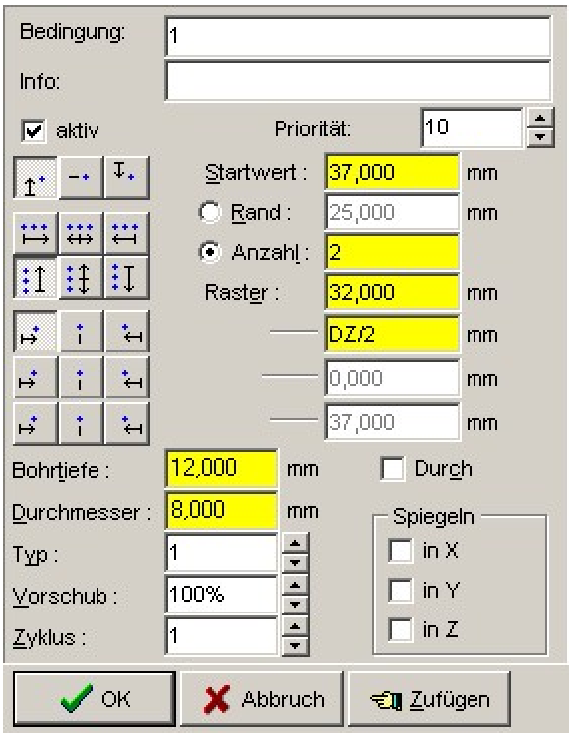
The dialogue for describing rows of holes is very powerful. Therefore the user has great flexibility while programming rows of holes. On the other hand this dialogue requires a little more effort to learn. The fastest way to learn is by doing. Once the dialogue is understood, even complex rows of holes are managed directly and fast.

The following panel dimensions are taken as a basis:

X	Y	Z
1000 mm	594 mm	19 mm

1. To comprehend this example, first create the first vertical row of holes according to the following values:





The basic element for vertical rows of holes allows the programming of up to 3 parallel rows of holes in one step. However, for our first row of holes we only need one.

a. Determination of the movement direction and of the reference point of the row of holes:
In our example we need a row of holes that moves in y-axis and that has its front hole as reference point.
b. Indicating the start position.
The reference point which was defined in step 1 is positioned now. For our example, the reference point has to be placed at the front panel edge.
c. Defining the position of the row of holes.
Up to 3 rows of holes can be defined at the same time. There is the combination of buttons and indicating measurements for that for three times. Parallel to determining the start position, the position of the row of holes is determined, by again first selecting a reference and then giving an appropriate offset. If the selected reference is clicked on again, the relating row of holes is shut down.

2. Now we just need the second row of holes. Modify the first row of holes and then click on “Add”.

Drilling row 2:	
Starting value	37+32+96
Count	4
Grid	128
Drilling depth	12
Diameter	8

If NC code display and NC code generation is enabled, your workpiece should look like this.

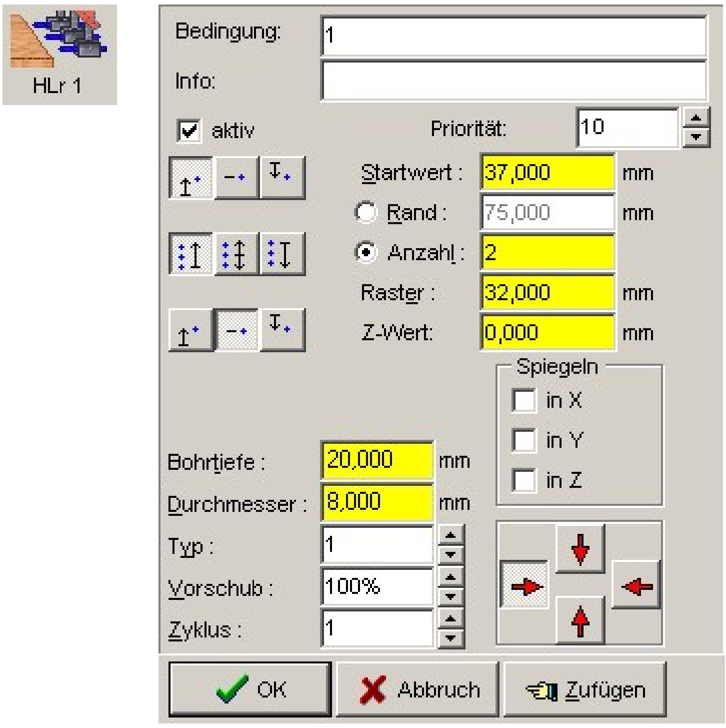
3.15 Drilling front-end dowel holes

Exercise:

In this example, we learn to do front-end drillings for dowel holes on a bottom panel. Therefore we use horizontal drillings. The bottom panel has the following dimensions according to the side panel in the previous example: 962 x 594 x 19 mm. The corresponding positions in y-axis for the drillings are: 37, +32, +96, +128, +128, +128.

For generating horizontal drilling rows in TwinCAM there is a basic element provided as well. In our example we need again two more elements of rows of holes.

1. Create the first horizontal row of holes with the following attributes:



a. Determination of the movement direction
First, we select the panel side the horizontal drilling is to be made in. By doing that, the symbols change for the upcoming adjustments.
b. Determination of the reference point within the row of holes.
Via 3 buttons the reference point within a row of holes is determined. This way, either the first or the last drilling or the middle of the row of holes can be determined as reference. In our example, the first drilling is the reference.
c. Indicating the start position
The reference point which was defined in step 2 is positioned now. For our example, the reference point has to be placed at the front panel edge.
d. Defining the position of the row of holes
Parallel to determining the start position, the position of the row of holes is determined, by again first selecting a reference (top edge, center of the panel or bottom edge) and then giving an appropriate z-value. If the selected reference is clicked on again, the relating row of holes is shut down.

2. Now we need one more horizontal row of holes, that you can create by altering the values of the first row of holes to the following:

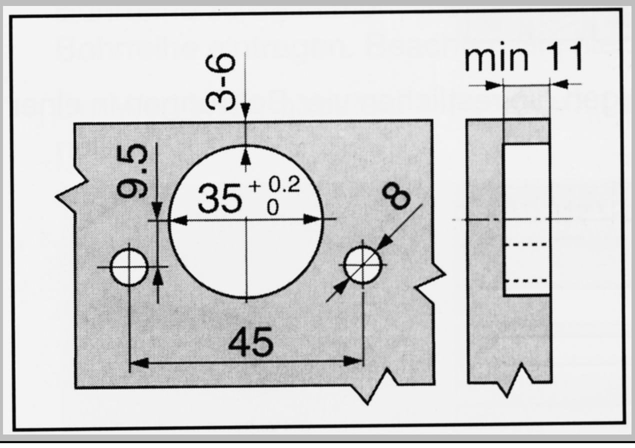
Drilling row 2:	
Starting value	37+32+96
Count	4
Grid	128

Now your workpiece should look like this:

3.16 Drilling concealed hinges

Exercise:

This example is to show how drillings for conclealed hinges are programmed. This example is supposed to match the cabinet from the previous task. The dimensions of the door are 1901 x 494 x 20 mm. The drilling pattern for a hinge is supposed to look like this:



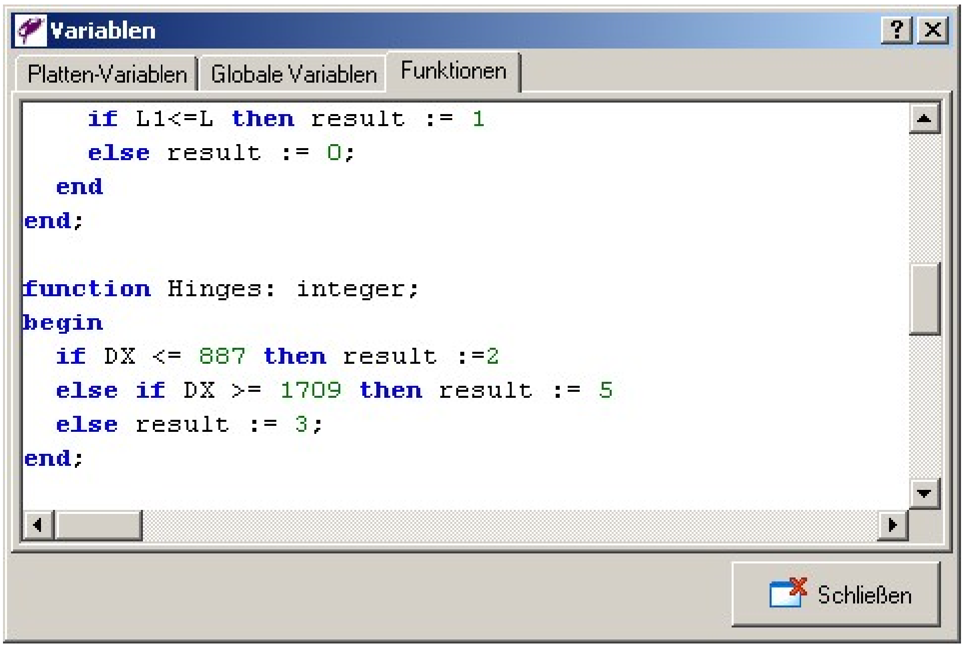
We want to give you another way of doing this task. This way is especially interesting for creating company standards. For that we accept the following rule for the amount of concealed hinges dependant on the door height:

Door height	Numer of hinges
301 - 887 mm	2
941 - 1.645 mm	3
1709 - 2477 mm	5

Of course, TwinCAM offers also the plain programming of this workpiece, but is way more efficient to solve this task with ElementsCAM's script programming. In this example we write a little function that always provides the correct amount of hinges dependant on the size of the panel.

Note:
Standards are inalienable for a faster, optimized production.

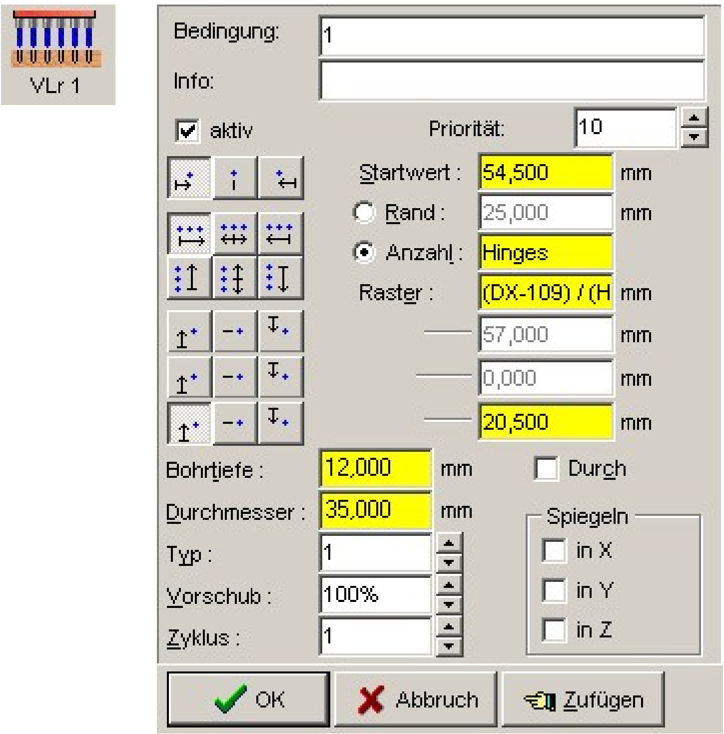
1. Go into the variable dialog and click on the register "Functions".



Copy the whole function "Hinges".

In the third line the function checks if the variable DX is less or equal 887. If the condition is met, the return value of the function Hinge is set on 2, which is activated with "result". With the command "else if" in the fourth line you determine a second condition if the first condition is not met. This time, however, it is checked if DX is greater or equal 1709. If the condition is met, the value 5 is given out by the command "result". Eventually in the fifth line you determine what will happen if none of the previous conditions was met. In this case the return value is 3. So this last command gives out the result 3 for all panel sizes between 887 and 1709.

2. Now you can give yourself to the proper workpiece description. First we need a drilling for a concealed hinge. For that, place a vertical row of holes according to the following data:

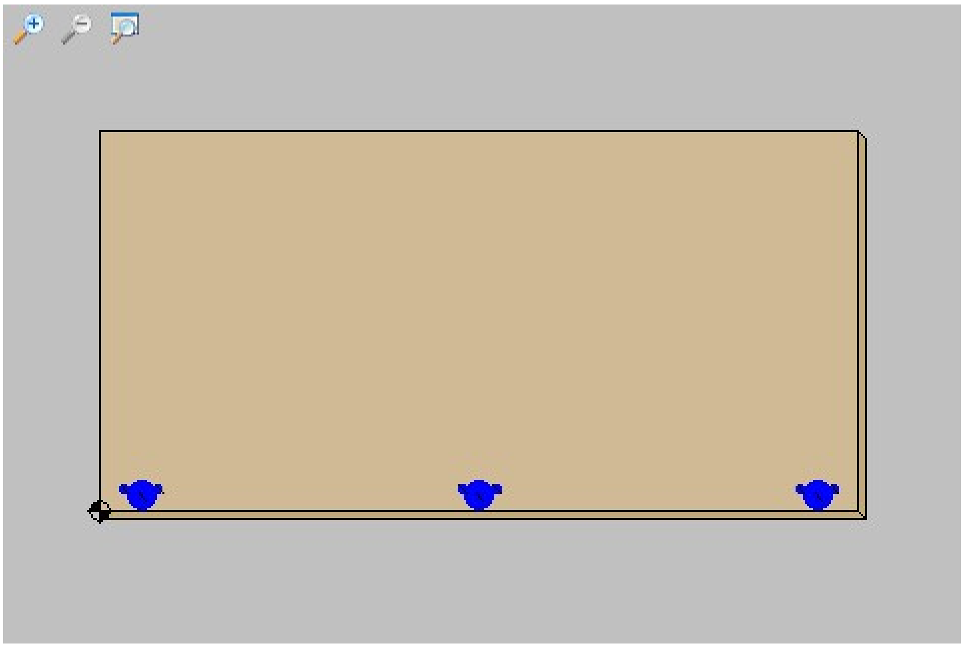


3. Again, edit the row of holes. Alter it as the following and eventually click on "Add".

Drilling row 2:	
Starting value	54,5-17,5-5
First row	20,5+9,5
Diameter	8

4. Now the last drilling of 8 mm is only missing. Edit now the second row of holes and change the starting value to "54,5+17,5+5". Click again on "Add". Now you have created all three row of holes. Additionally, they have the attribute that they adjust to the panel size.

Your simulated workpiece description should look like this by now:



3.17 Drilling 32 mm raster holes

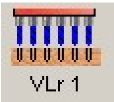
Exercise:

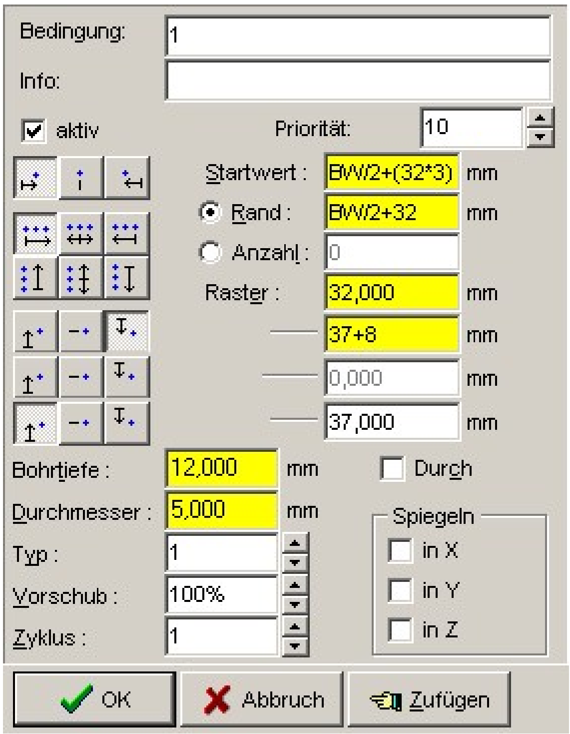
In a cabinet side there are rows of raster-holes to be drilled (32 mm raster) with 5 mm as diameter. The workpiece dimensions are 1.971 x 594 x 19 mm. Two of three rows (front and rear) are to be drilled without condition. An additional middle row of holes should be only inserted if the cabinet side exceeds a width of 562 mm.

Creating raster-holes in TwinCAM is also no problem. For our example we use 2 elements of rows of holes.

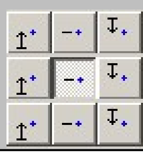
1. In order to create row of holes more flexible, it seems natural to define a variable for the thickness of the panel. For that, go in to the variable editor and create the variable "BD" with 19 as value.

2. Place the first vertical row of holes on the base panel and copy the following attributes:

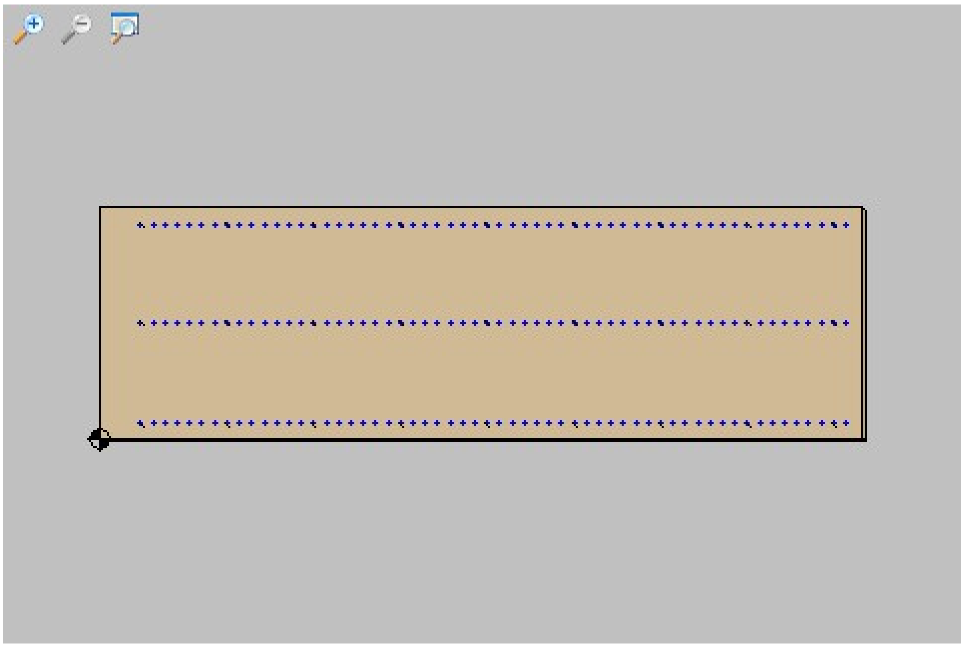




3. The middle row of holes is to be activated by a condition. Please note the term in the field "Condition". Other than simple expressions, this field can contain complex terms or even functions. Create one more row of holes. You can use the previously created row of holes as basis.

Drilling row 2: (Panel thickness (3))	
Condition	DY > 562
Position (Y)	

Now your base panel should look like this.



3.18 Drilling a central panel

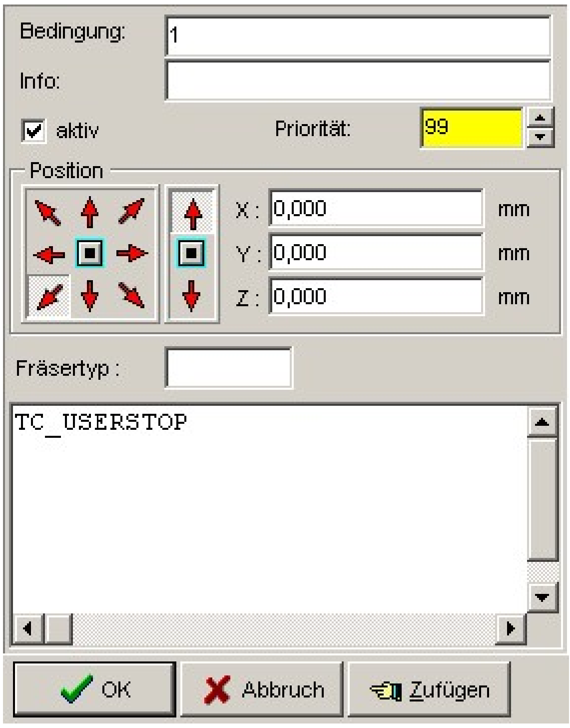
Exercise:

Drilling central panels is a special form of drilling rows of holes. The drill holes of the raster-hole are through bores. In order to avoid tear-outs, the workpiece will be drilled from both sides with a depth of 12 mm. The central panel has the dimensions 1.869 x 586 x 19 mm.

In order to flip the workpiece after its first processing, a programmed stop is to be inserted.

In order to insert a programmed stop in TwinCAM, you need the DIN-element "Stop". If you do not have a macro in your pallet, you can use the basic element DIN and follow step 1 below.

1. Add the DIN element with the values below to your workpiece description.



2. Now you only need to program the rows of holes that are on the bottom side of the panel. Select the already existing rows of holes and confirm the dialog with "Add". Because our panel is symmetrical, you do not have to make alterations to these rows of holes.

3. Now, add a new layer. This function is found in the toolbar.

4. Select the rows of holes that are to be drilled on the rear side of the panel and click on the button "Set selected object on current layer" in the symbol area.

3.19 Placing suction pads

The placing of clamping elements is mainly dependant on the machine in use. As long as each of the processes on the surface of the panel do not restrict the way the suction pads are placed (e.g. through bores), TwinCAM will automatically place the suction pads in the job list. Only as soon as a process requires the explicit placing of suction pads one should use this option in the workpiece description. Since the workpiece descriptions in TwinCAM are to be basically independant of machining centers, the explicit placing of suction pads and if necessary traverses means a definite restriction of this independance. However, the specifical placing of clamping elements is inevitable for some workpieces and is therefore supported by TwinCAM.

You can select the different standard clamping elements over the menu "Clamp". Usually this should not be necessary because the producers of machining centers provide a wide range of suction pads and consoles. Suction pads can be parametically placed as every other process in a workpiece description. Furthermore, suction pads can also be moved on the panel by drag & drop.

3.20 Creating templates for suction pads

Exercise:

In this example, a template for suction pads is supposed to be created according to a given workpiece contour (breakfast board). In the data directory of ElementCAM's demo version you find the following workpiece description in the file [...].

[Picture]

The template will be created by modifying the contour. For that, the unmachined template part is to be grooved (6x6 mm). And in the groove, a foam rubber strip is to be inserted. The groove is to be milled with a fixed distance parallel to the outer contour of the workpiece.

In TwinCAM, generating templates for suction pads is pretty easy. Basically you need the CAD function  "Parallel element" and  "Fillet".

1. First, apply the CAD function "Parallel element" on all contours in the workpiece description. If you click on the CAD function, you will be asked to give a distance to the original contour.

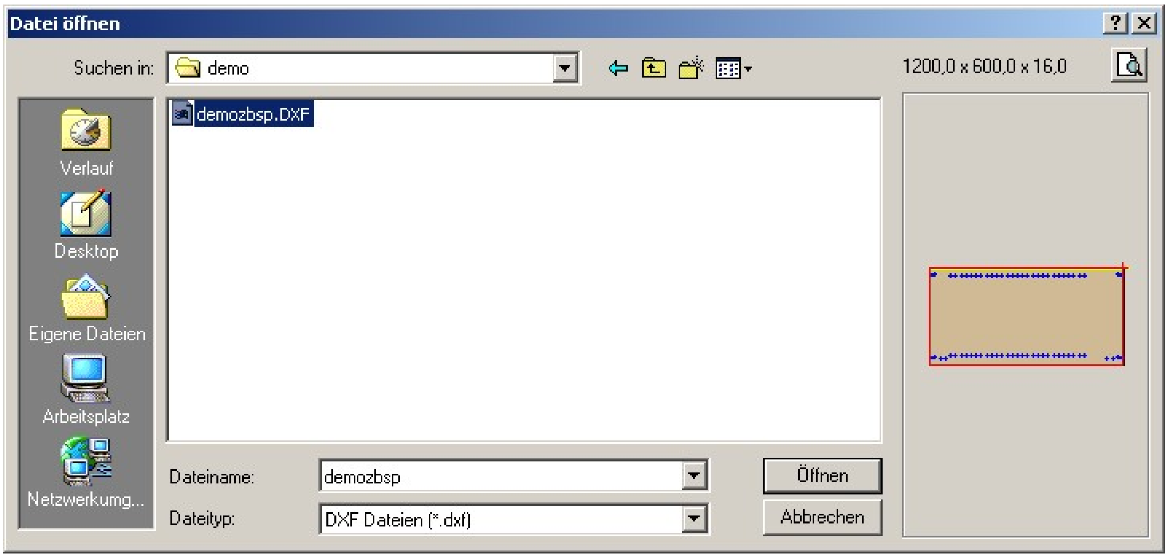
2. After having created all parallel contours, you might have to "smooth" sharp edges or connect incoherent contours with the CAD function "smooth". If you click on the CAD function, you will be asked to give a radius for smoothing.

3. Now you only need to add a starting point to the newly created contour. For that, just group the desired and first element of your contour. Type in 6 mm as diameter for the starting point and select "right" for the running side.

3.21 Exchanging CAD data

The basis for any data exchange via DXF are conventions that regulate how geometric elements of the DXF file are to be interpreted in the WOP system. TwinCAM supports 2 different DXF conventions as standard.

In order to use a DXF file in TwinCAM, click on "Menu" and select the subentry "Open". Alternatively, you can select this function from the toolbar, also. After selecting it, the following dialog appears.



As type of file "DXF" must be chosen.

Alternatively, you can also use the script "DXF-Importer" in order to load DXF files into TwinCAM. This script is standardly located in the register "Scripts". Then, you have the full control over the interpretation of each DXF element and can decide for yourself how it is to be imported into TwinCAM. However, adjusting the script requires programming skills in a programming language as Pascal, VBScript or JavaScript. Please be aware that the provided DXF-Importer merely serves as a motivation and basis for your own extensions.