Cambridge Numerical Control

Driving Productivity Forward

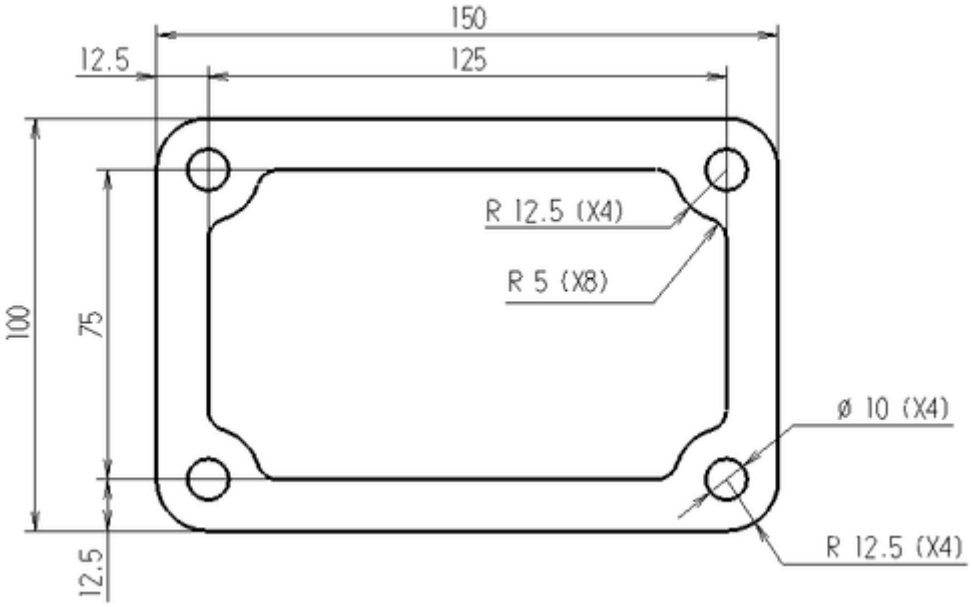
CNC Calc 7

Entry Level CAD/CAM for the Shopfloor

CNC-Calc Tutorial

2D Construction of a Part for Milling & Creating Toolpaths

Tutorial Mill 1 - 2D Construction of a Part for Milling



Sample 2D part.

This tutorial demonstrates one of many ways in which the 2-dimensional part above can be drawn in CNC-Calc v7. Since the part consists of a number of similar elements and since its part-elements are symmetrical, only a subsection of the part needs to be drawn. The rest emerges from mirroring. Finally, joining the mirrored elements with straight lines completes the part.

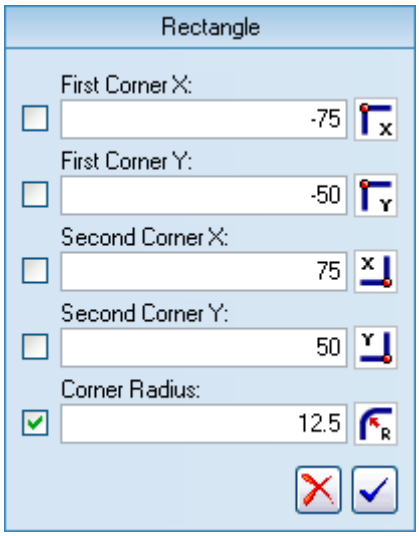
This tutorial demonstrates the use of the following functions:

- Draw a rectangle with a corner radius.
- Draw a circle with known center and radius.
- Draw vertical and horizontal lines from known points.
- Offset a circle.
- Make curves between elements.
- Delete elements.
- Mirror elements about lines.
- Join end points with straight lines.

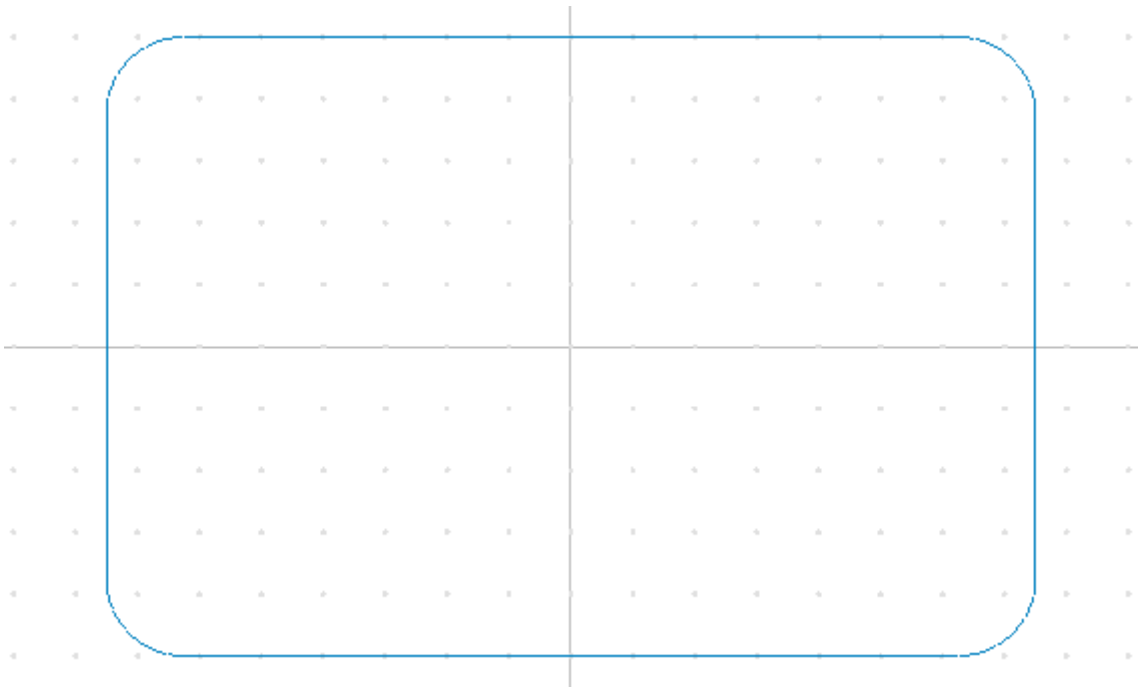
Draw a rectangle with round corners

- Click on the **Rectangle** icon in the *Draw Points / Lines* toolbar.

- Enter the values shown in the figure below to draw a rectangle with sides = 150, height = 100 and corner radius = 12.5, and centered in the drawing area:

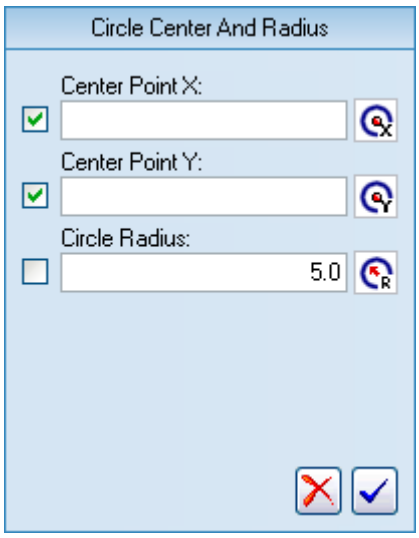


- Click on to approve the command and draw the rectangle.

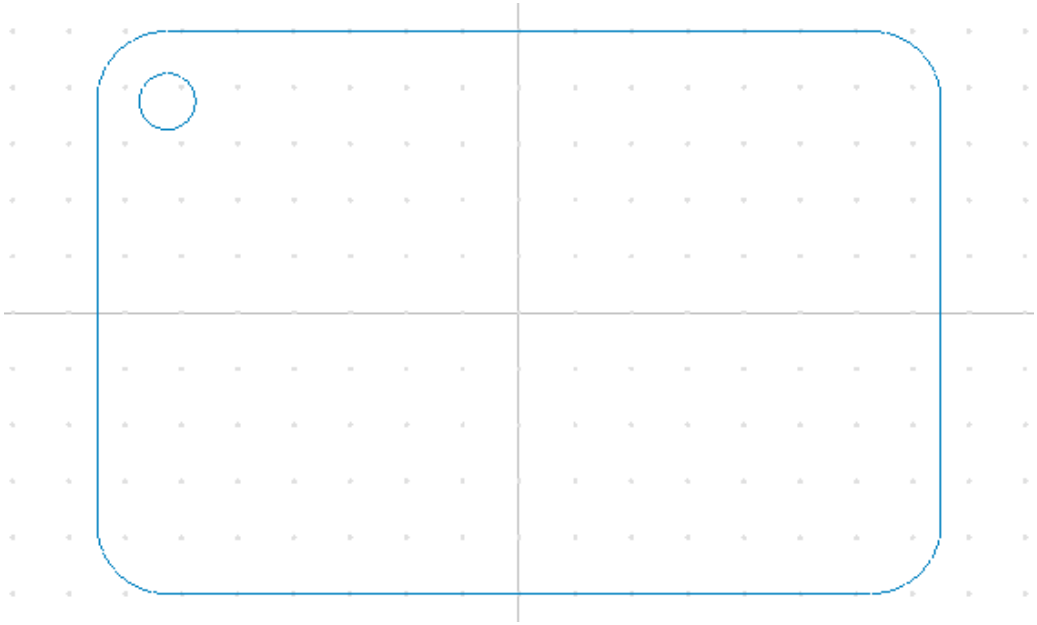


Draw a circle defined by its center and radius

- Click on the **Center Radius** icon in the *Draw Arcs / Circles* toolbar.
- Enter the value **Circle Radius = 5** in the CNC-Calc pane.



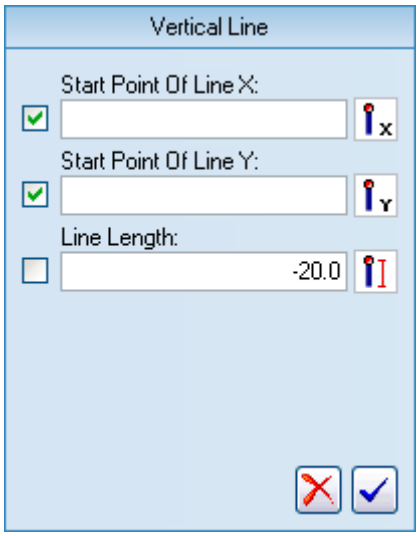
- Activate the function **Snap to Center Points** in the *Snap* toolbar.
- Snap to the center of the left topmost corner arc. The circle will automatically move to the center point when the cursor is close to it. The pointer also changes indicating snap to arc/circle center.
- Left-click to add the circle.



Draw vertical and horizontal lines

Vertical Lines

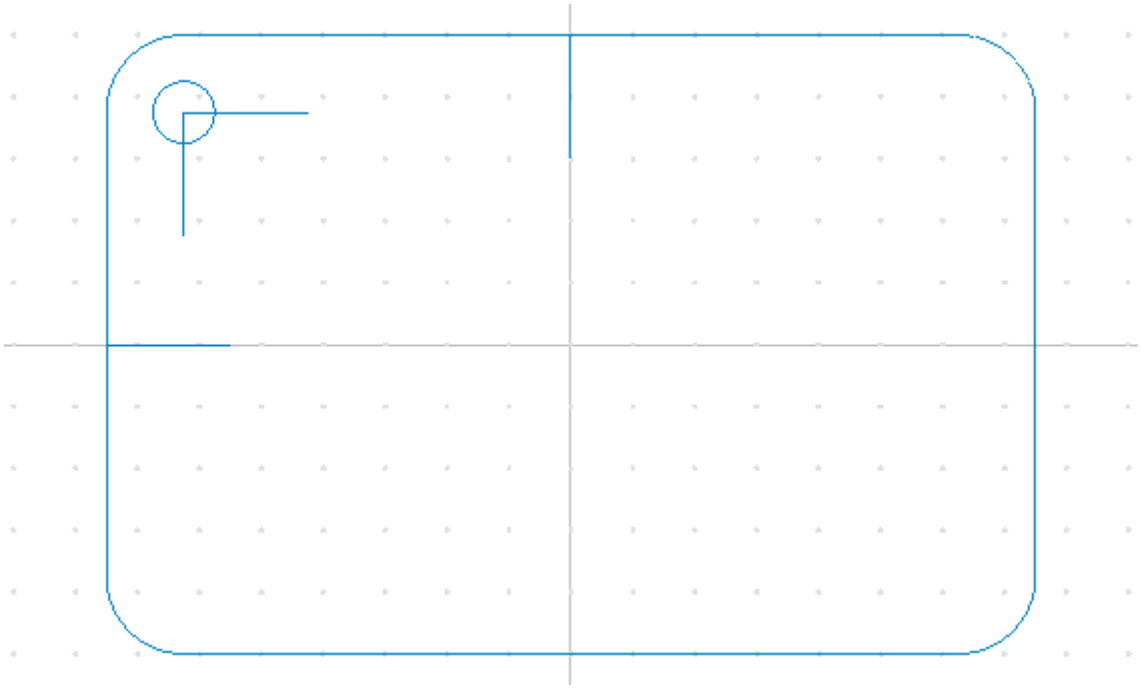
- Activate the functions **Snap to Center Points** and **Snap to Mid Points**
- Click on **Vertical** in the *Draw Points / Lines* toolbar to draw a vertical line.
- Enter the value **Line Length = -20.0** into the CNC-Calc pane. The negative value indicates the vertical line is in the -Y direction from the start point of line.



- Snap to the center of the circle drawn in the previous step.
- Click to add the vertical line.
- Snap to the midpoint of the topmost horizontal line (match up with the Y-axis).
- Click to add another vertical line with the same values. This one will serve as a mirror line for the mirroring of our part about the Y-axis.

Horizontal Lines

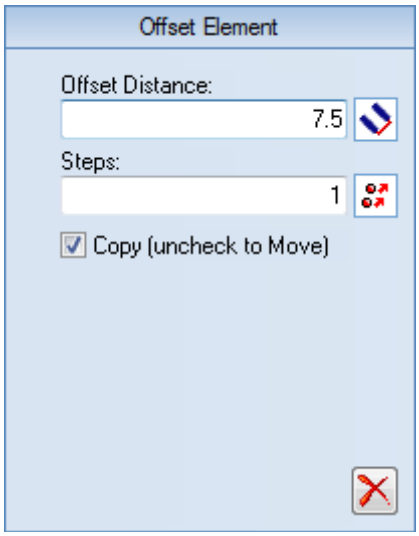
- Click on **Horizontal** in the *Draw Points / Lines* toolbar to draw a horizontal line.
- Similar to the above, draw a line with **Line Length = 20** from the same circle center. This time Line Length is set to a positive value to indicate the horizontal line is in the +X direction.
- Add then a horizontal mirror line from the center of the left vertical line (match up with the X-axis).



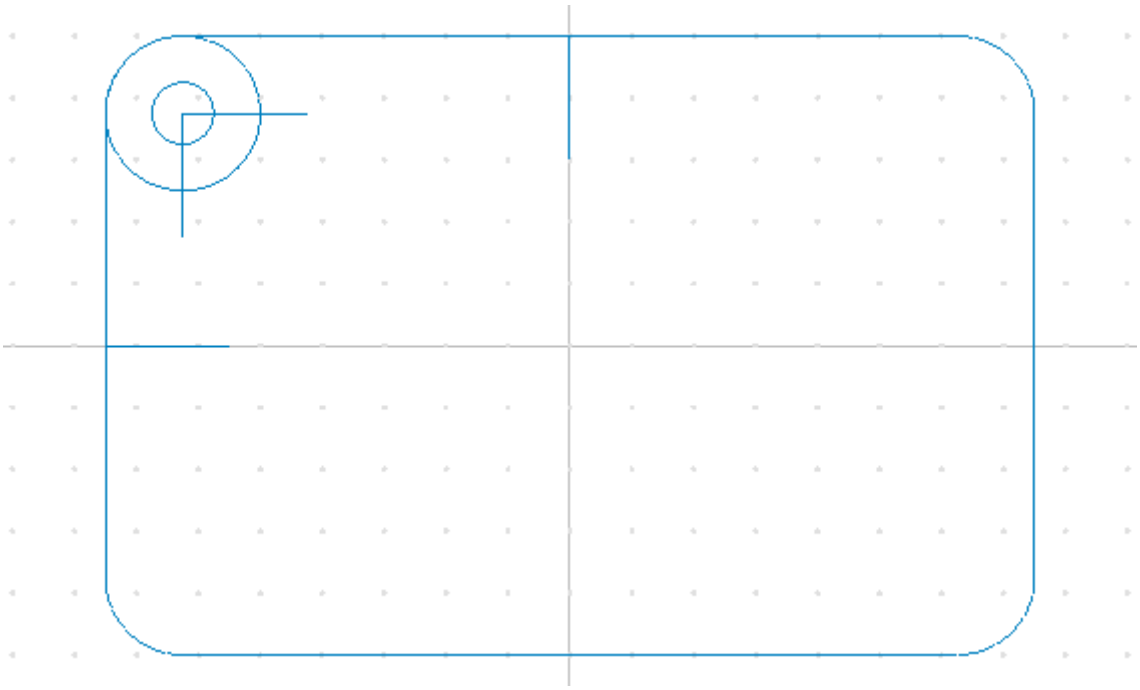
Offset a circle

- Click on the **Offset Element** icon  in the *Modify* toolbar.

- Enter the value **Offset Distance = 7.5** (12.5 - 5 = 7.5) into the CNC-Calc pane. Select the option *Copy (uncheck to Move)*.



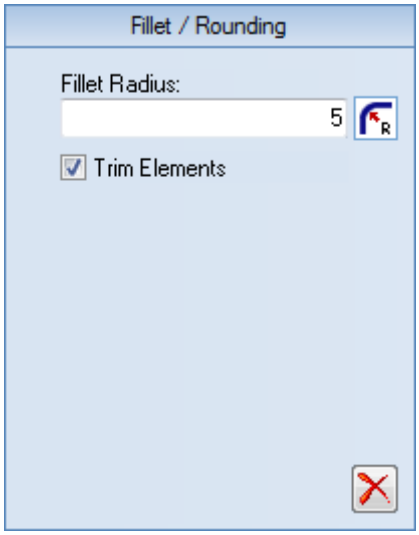
- Click on the circle and select the outermost of the appearing circles.



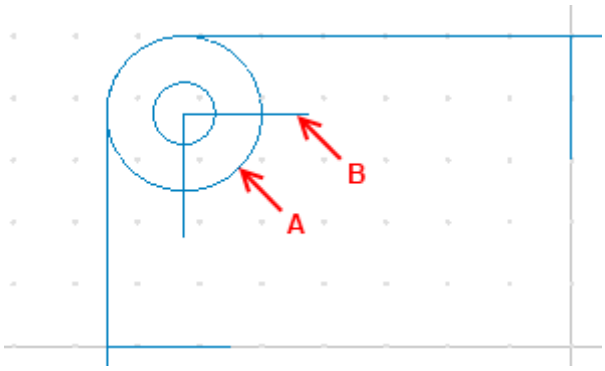
Create a fillet between elements

- Click on the **Fillet Elements** icon  in the *Modify* toolbar.

- Enter the value **Fillet Radius = 5**, and leave the option *Trim Elements* checked.



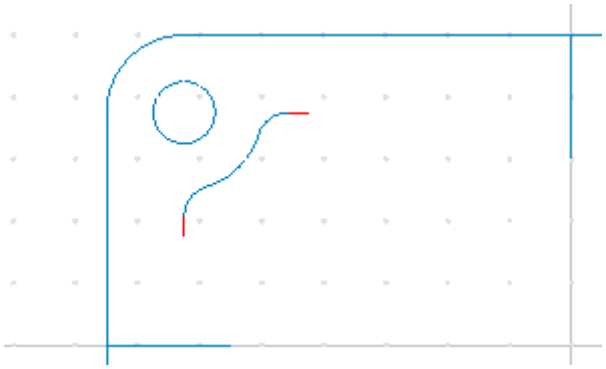
- To select the elements for the fillet operation, click on the circle by A and on the line by B.

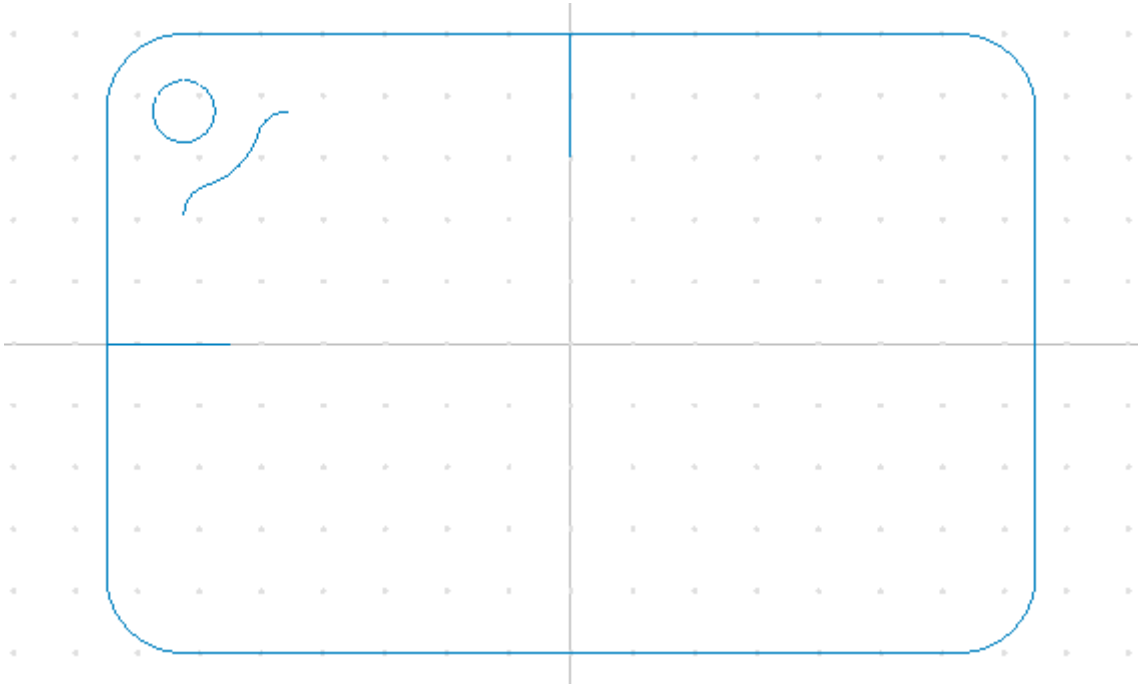


- From the possible solutions select the part of the circle which makes the right fillet. In the picture below you can see how it should look.

- Do the same by the vertical line.

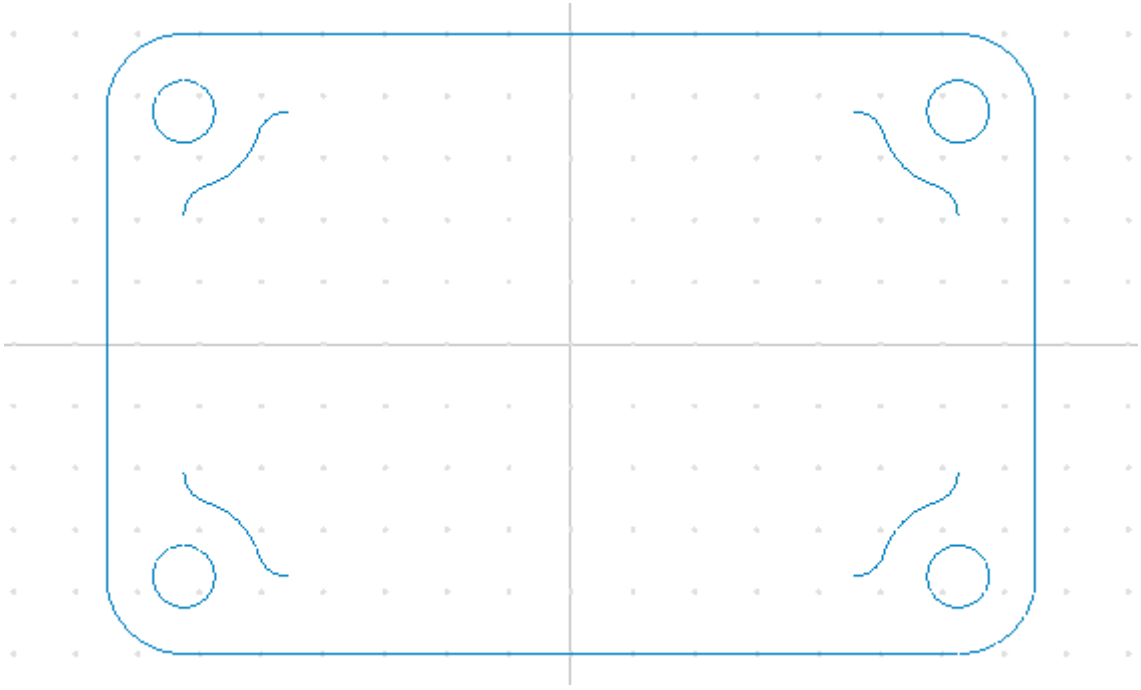
- Click on the **Delete** icon  and delete the two residual lines (the ones pointing away from the center of the circle).





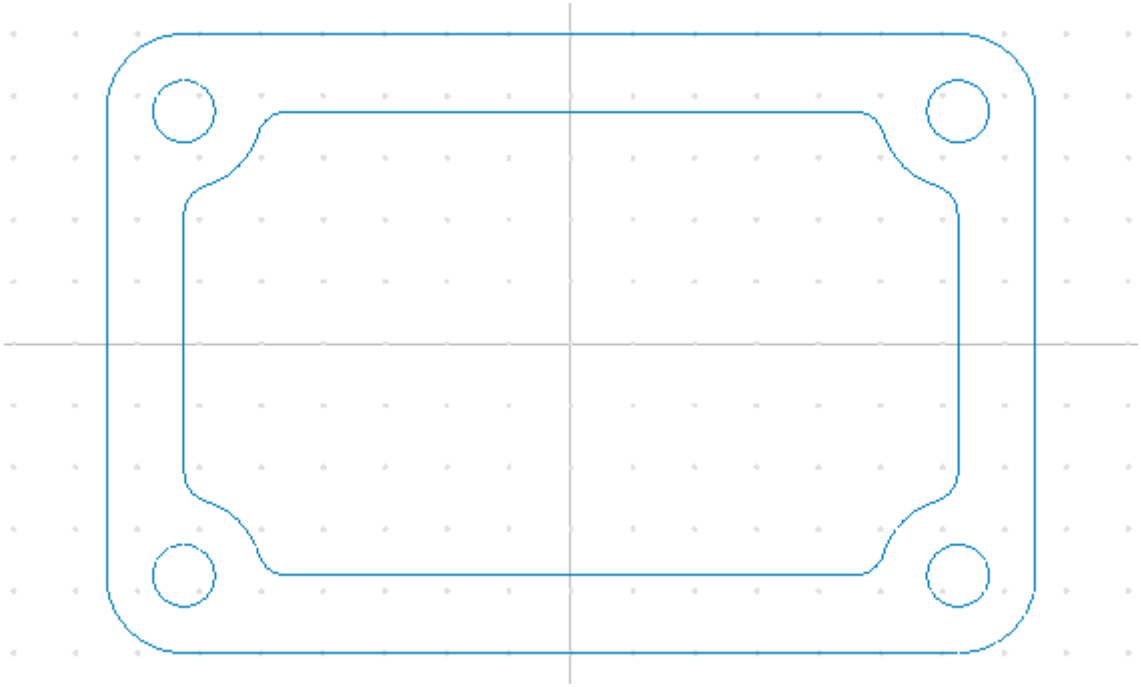
Mirror elements

- Click on the **Mirror Elements** icon in the *Modify* toolbar.
- First, click on the vertical mirror line.
- Then click on all the elements which should be mirrored (the circle and the inner corner). You can hold down the left mouse button while dragging out a window around the elements.
- Now do the same and mirror along the horizontal mirror line. Continue mirroring until your drawing looks similar to the one below.
- Click on the **Delete** icon and delete the mirror lines.



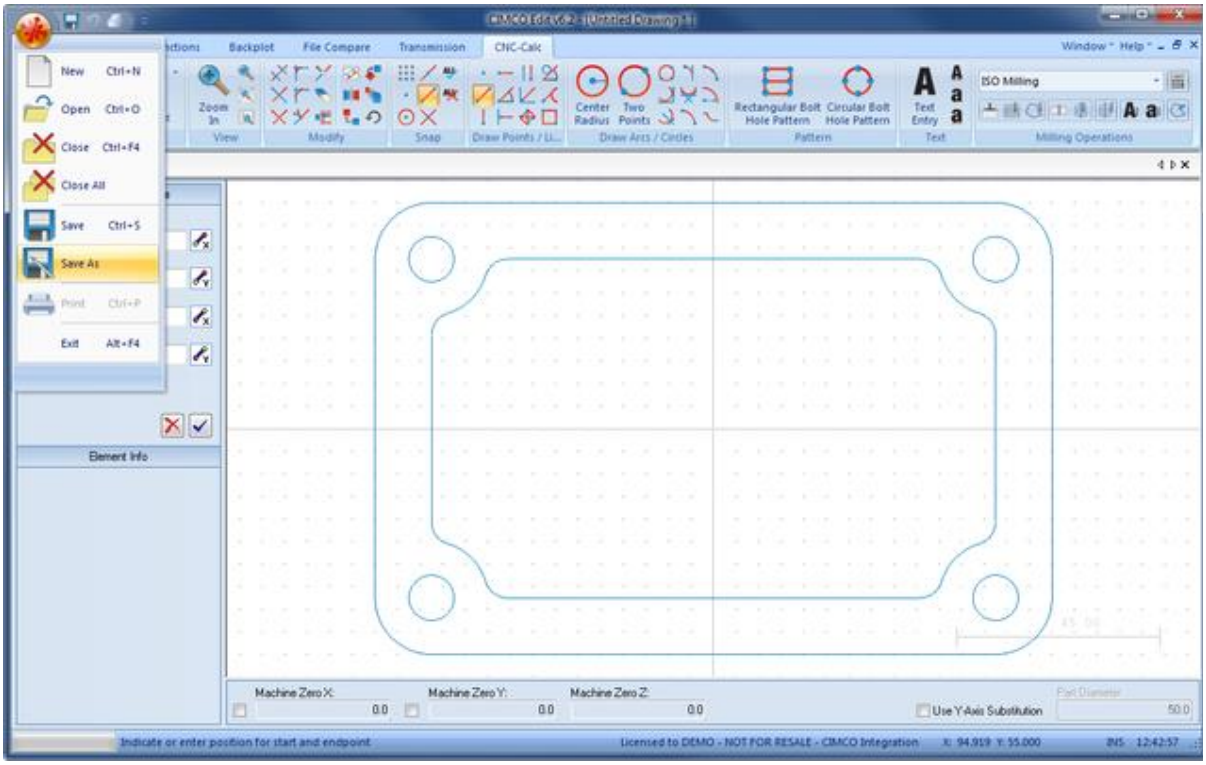
Connect the inner elements

- Activate the snap function **Snap to End Points** .
- Click on **Between 2 Points** icon in the *Draw Points / Lines* toolbar.
- Snap to the two arcs' end points and add the remaining horizontal and vertical lines to finish the part.



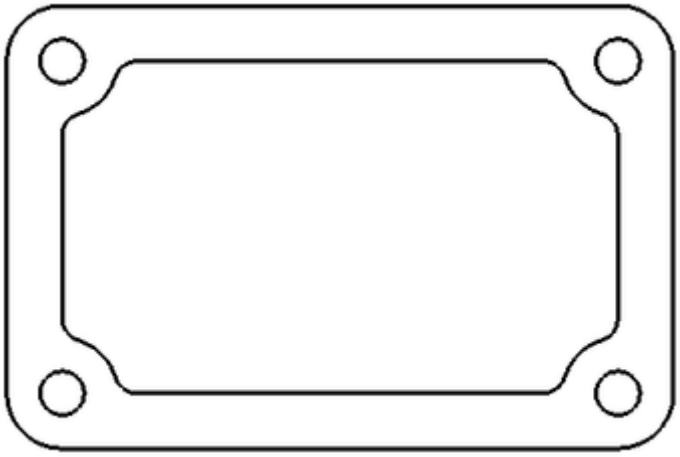
Save the File

Click on the **Main Menu** icon and then select **Save As** from the drop-down menu. Give the file the name *CNC-Calc Milling Tutorial 1* and save it (the file name extension is added automatically).



Machining the Part

With CNC-Calc v7 it is possible to create toolpaths directly from the program's geometrical drawings. Thereby, calculations become more secure and programming becomes much faster compared to doing it manually. At the same time you get a big advantage since it is possible to move, copy, rotate, scale, and mirror elements with the result of instant NC-code generation. There are several machine types in CNC-Calc, but the most commonly used are ISO G-code programming and Heidenhain plain text.

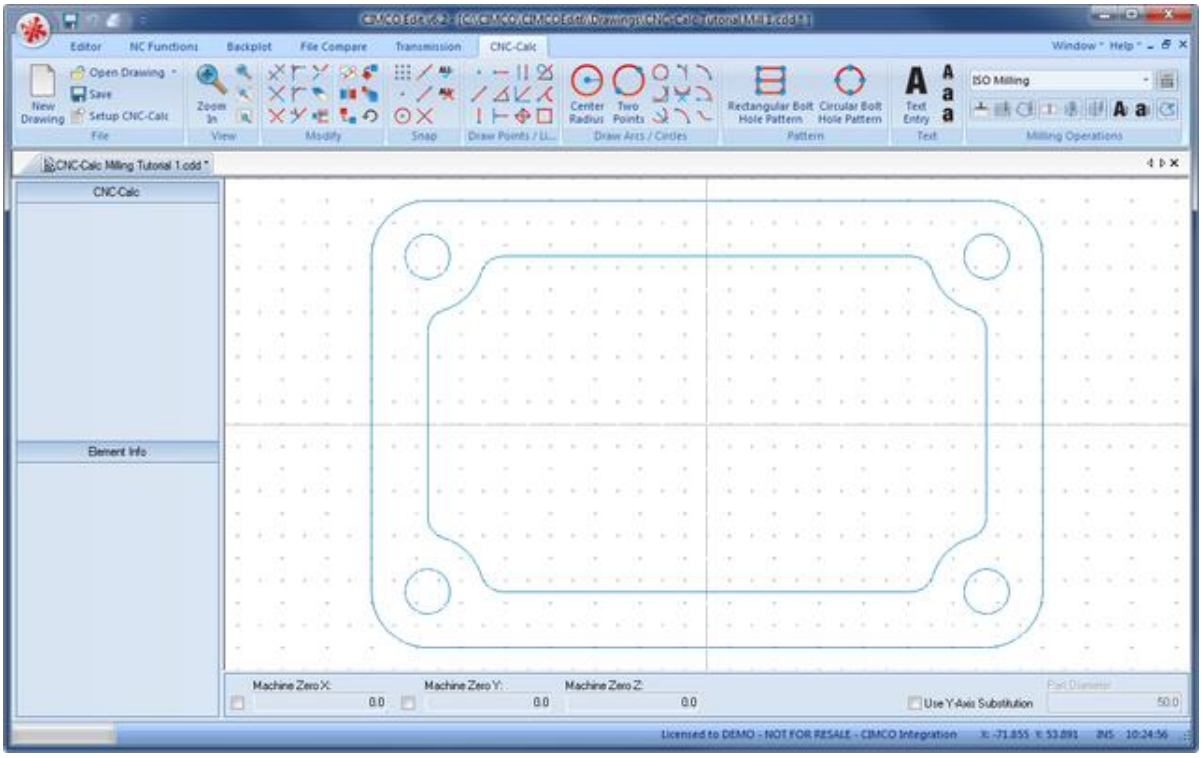


Sample 2D part.

This tutorial demonstrates how the 2-dimensional part above can form the basis for NC-codes for various types of machining.

In order to produce the final part we will use the following operations:

- Face Milling
- Contour Milling
- Pocket Milling
- Drilling



If you hold the cursor over an icon, a short description of the icon's functionality will appear.

You can change the colors of the drawing area by selecting **Setup CNC-Calc** and then **Global Colors** from the tree menu.

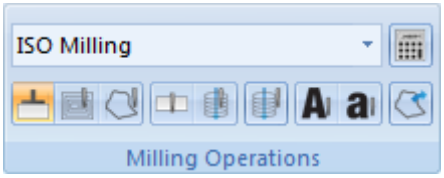
Face Milling

CNC-Calc v7 can generate CNC toolpaths for face milling, with or without finishing passes and using different cutting strategies.

Creation of Facing Toolpaths

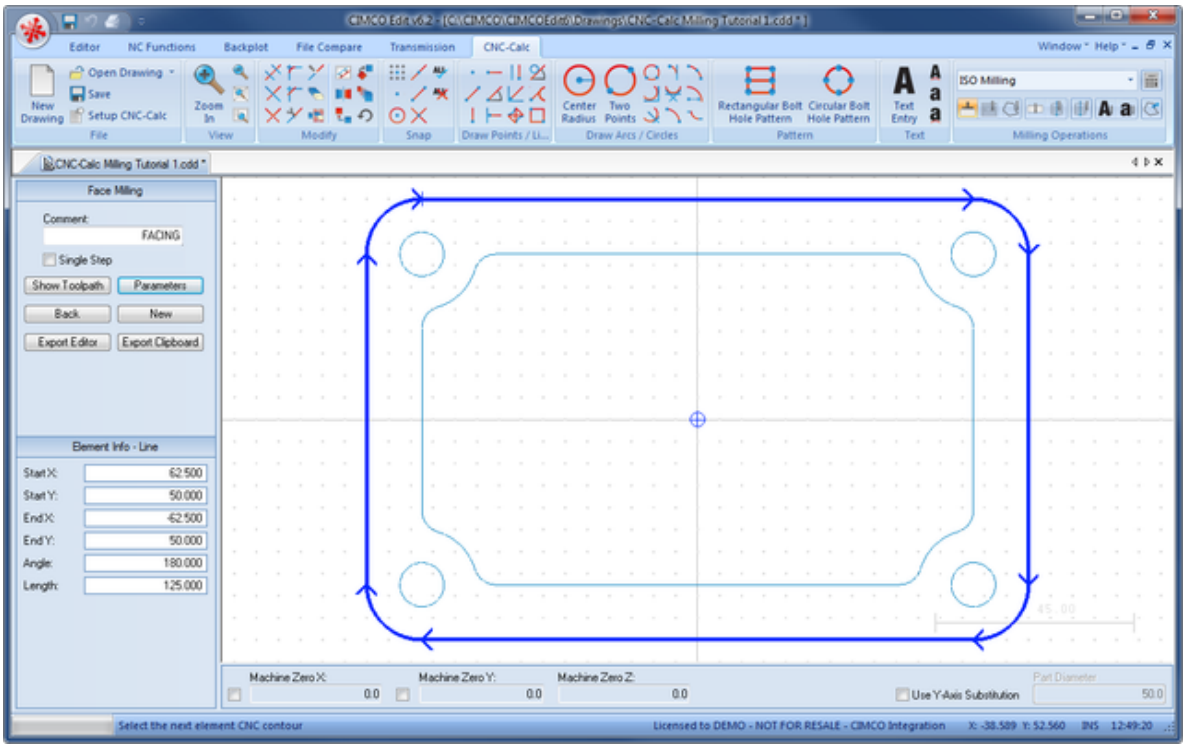
First, in the *Operations* toolbar, select the programming format of the NC-program in the *File Type* drop-down menu. Select **ISO Milling**.

Then click on the icon  **Face Milling** to generate a CNC-Toolpath for face milling.



Write the text FACING in the *Comment* field of the CNC-Calc pane Face Milling. This text will be included at the start of the final NC code for this operation. When multiple operations exist in the same NC program, the comments will help to locate and identify the start of each operation.

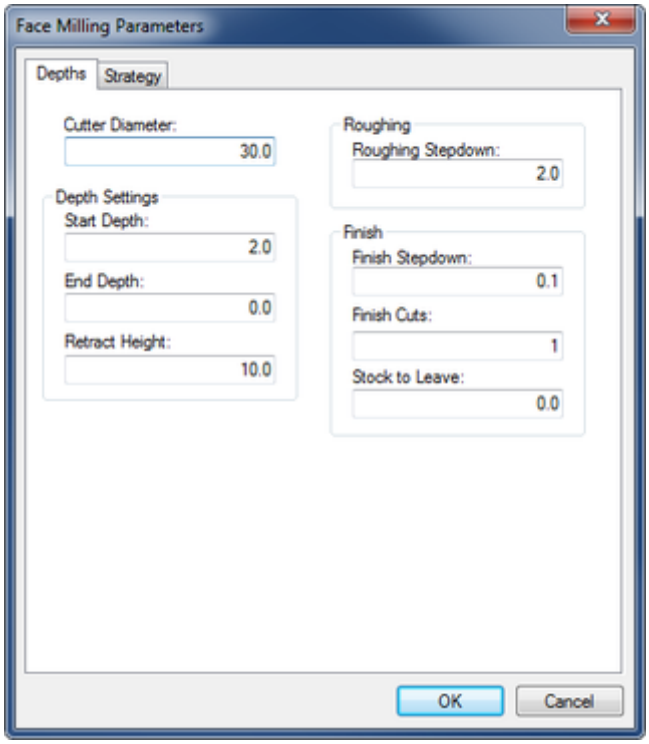
Click on the outlining contour of the drawing. This will select the bounding contour that the facing operation will operate on.



Click on the button **Parameters** in the CNC-Calc pane Face Milling. This will open the configuration dialog for setting the face milling parameters.

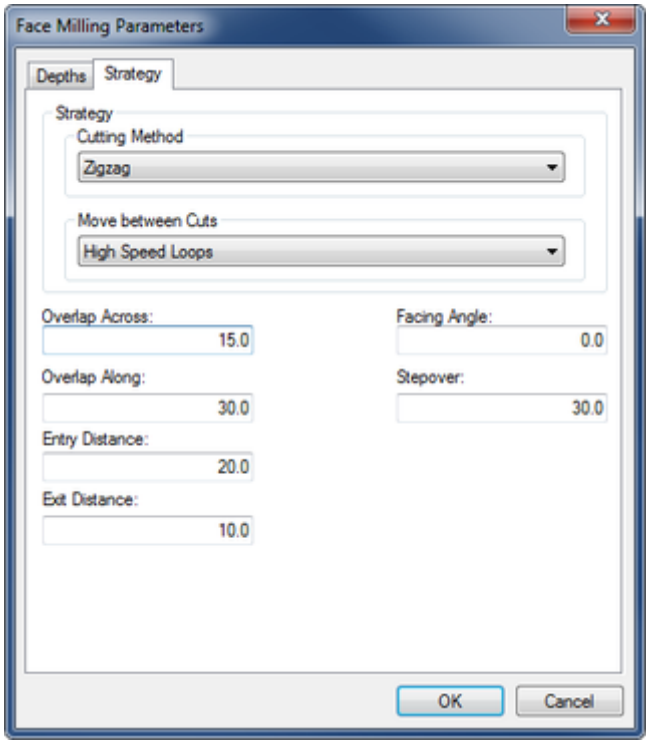
Enter the values into the dialogs as shown in the pictures below.

Depths Tab



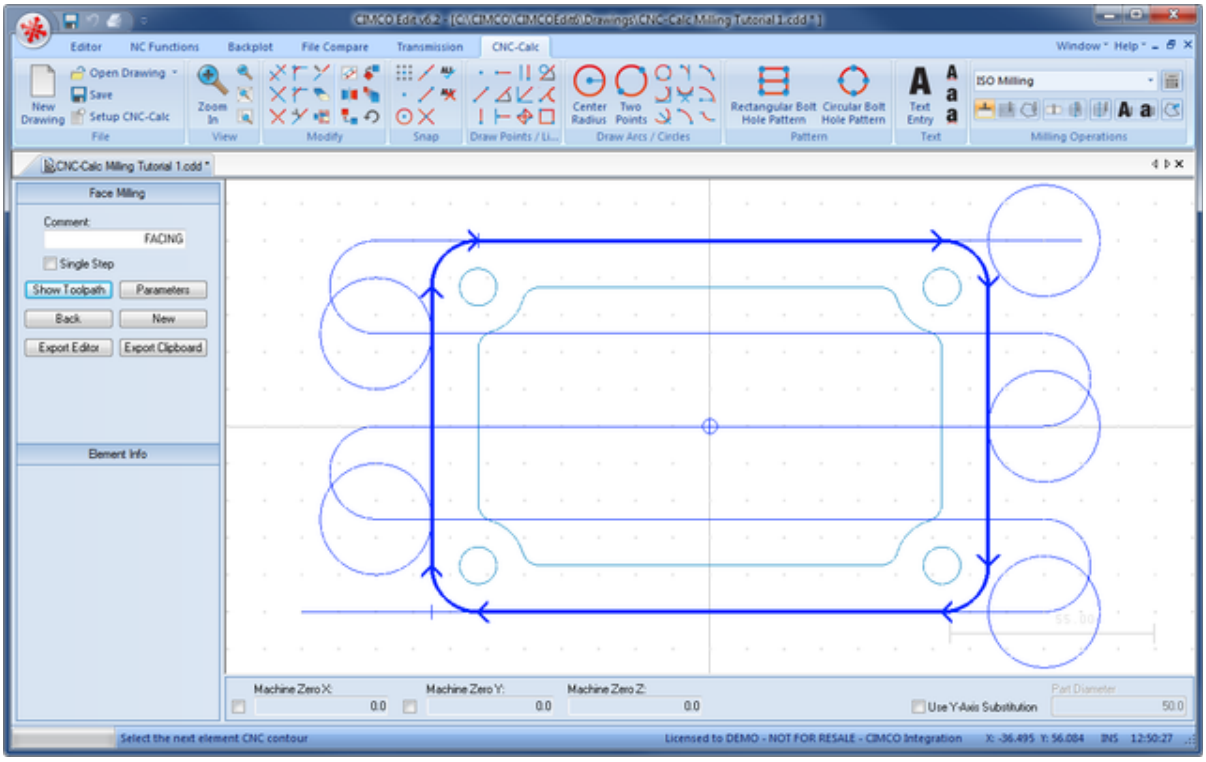
- **Cutter Diameter**: This is the diameter of the cutter. Here it is a 30 mm Face Mill.
- **Start Depth**: This is the top of the part.
- **End Depth**: The final depth (will be corrected by *Stock to Leave*).
- **Retract Height**: When the operation is finished, this is the height that the tool will retract to.
- **Roughing Stepdown**: The maximum roughing cuts that the operation will take.
- **Finish Stepdown**: If Finish Cuts is larger than zero, this is the cut that will be taken in each finishing cut.
- **Finish Cuts**: The number of finishing cuts that the operation will perform. If the value is left at zero, only roughing cuts will be made.
- **Stock to Leave**: The amount of stock that is left at the end of the operation (after both roughing and finishing cuts).

Strategy Tab

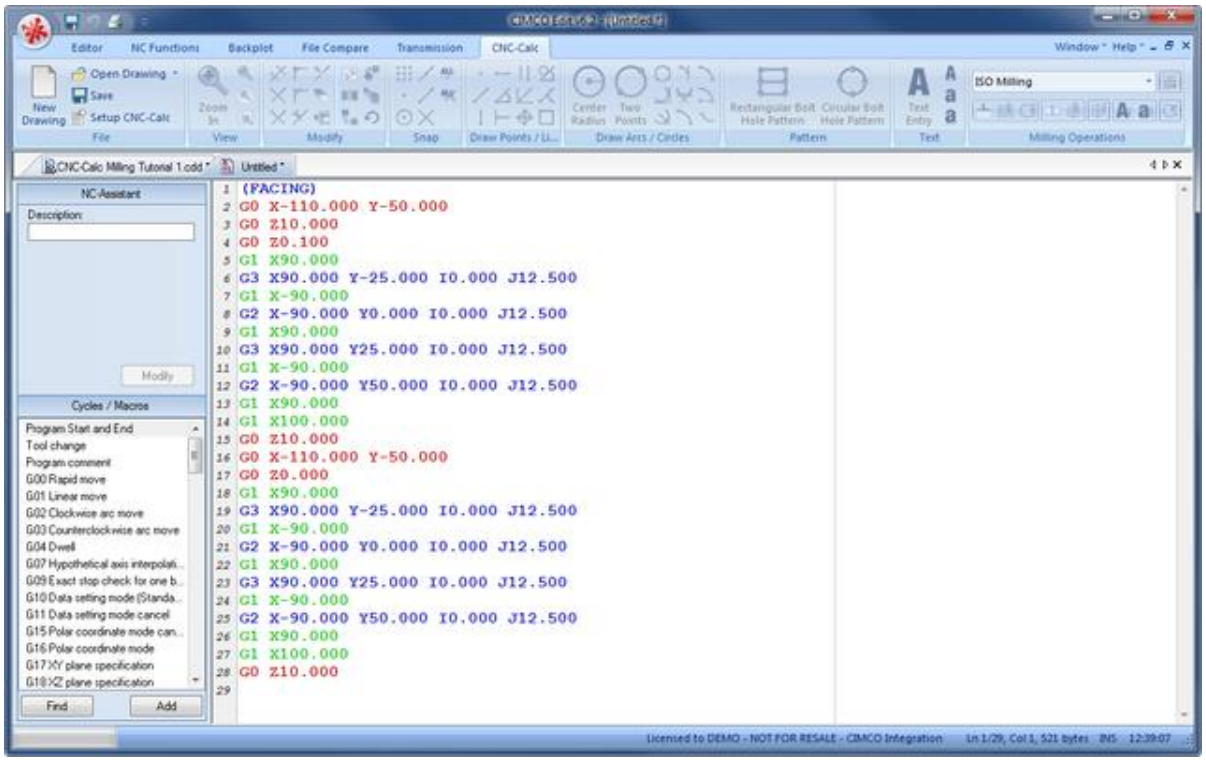


- **Cutting Method**: The method used to perform the face operation. It is possible to select *Zigzag*, *Climb*, or *Conventional*.
- **Move Between Cuts**: Is only used for the Zigzag Cutting Method, since the other methods will move free between cuts.
- **Overlap Across**: The amount that the mill will hang out over the side diagonal to the cutting direction.
- **Overlap Along**: The distance that the tool will move out over the end before the high speed loops are taken.
- **Entry Distance**: The distance that the tool will start out at before the actual cut is taken.
- **Exit Distance**: The distance the tool moves out after the final cut is taken.
- **Facing Angle**: The angle at which the operation is performed. An angle of zero is along the X-axis, and an angle of 90 is along the Y-axis.
- **Stepover**: The distance between each of the parallel cuts of the facing operation.

After entering the values, close the parameters dialog with **OK**. To show the generated toolpath click on **Show Toolpath** button in the Face Milling pane.



Now click on the button **Export Editor** to export the generated NC code to a new window in the Editor. The following screen should now be displayed.

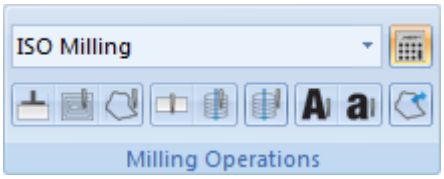


Inserting a Tool with Feed and Speed Calculator

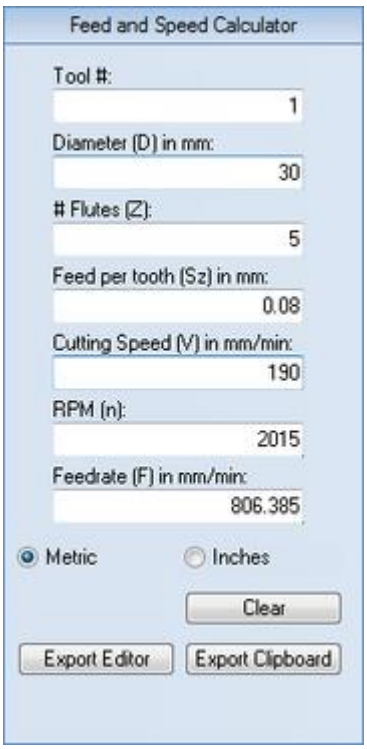
The Feed and Speed calculator built into CNC-Calc is used to insert feed and speed data into the NC program. All the data used in the calculations can normally be found in the reference material supplied by the manufacturer.

In the facing example, we use a face mill that we give the following characteristics: diameter is 30 mm, it has 5 flutes, a cutting feed of 0.08 mm per tooth, and a cutting speed of 190 mm/min.

To use the feed and speed calculator for milling operations, select the icon **Calculator** in the *Milling Operations* toolbar.



Fill in the following values into the Feed and Speed Calculator pane:



- **Tool #**: Number assigned to the tool, let's say that the face mill have a tool number of 1.
- **Diameter (D)**: Diameter is 30 mm.
- **# Flutes (Z)**: The number of flutes is 5.
- **Feed per tooth (Sz)**: In this example it is set to 0.08 mm.
- **Cutting Speed (V)**: Is set to 190.

The fields are linked together, so as entries are made in the *Cutting Speed* field, the fields *RPM* and *Feedrate* will be automatically updated.

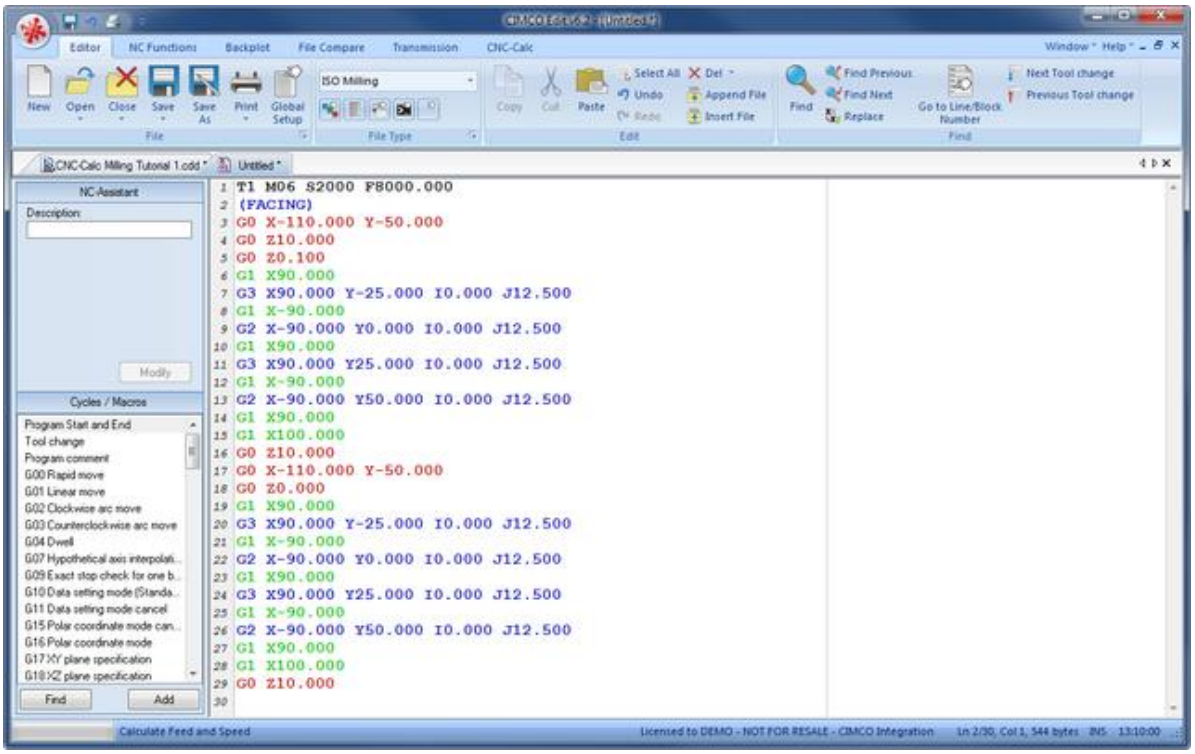
If we then want to have 2000 rpm and a feedrate of 800 mm/min instead of the calculated 2015 and 836.385, the value for the cutting speed will be updated to 188.5 mm/min.

Change the RPM to 2000 rpm and the feedrate to 800 mm/min.

Click on the button **Export Clipboard** on the Feed and Speed Calculator pane. The line for the NC program is now in the clipboard, and it is ready for insertion.

Change the window to that of the NC program, and press *Ctrl+Home* to move to the very start. Insert the text from the clipboard, either by pressing *Ctrl+V*, or selecting the icon **Paste** from the *Edit* toolbar in the Editor tab.

The NC program should now look similar to the one below.

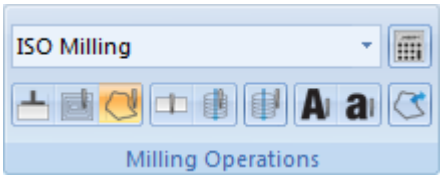


Contour Milling

CNC-Calc v7 can generate contour milling toolpaths - with or without radius compensation.

Creation of Contour Toolpaths

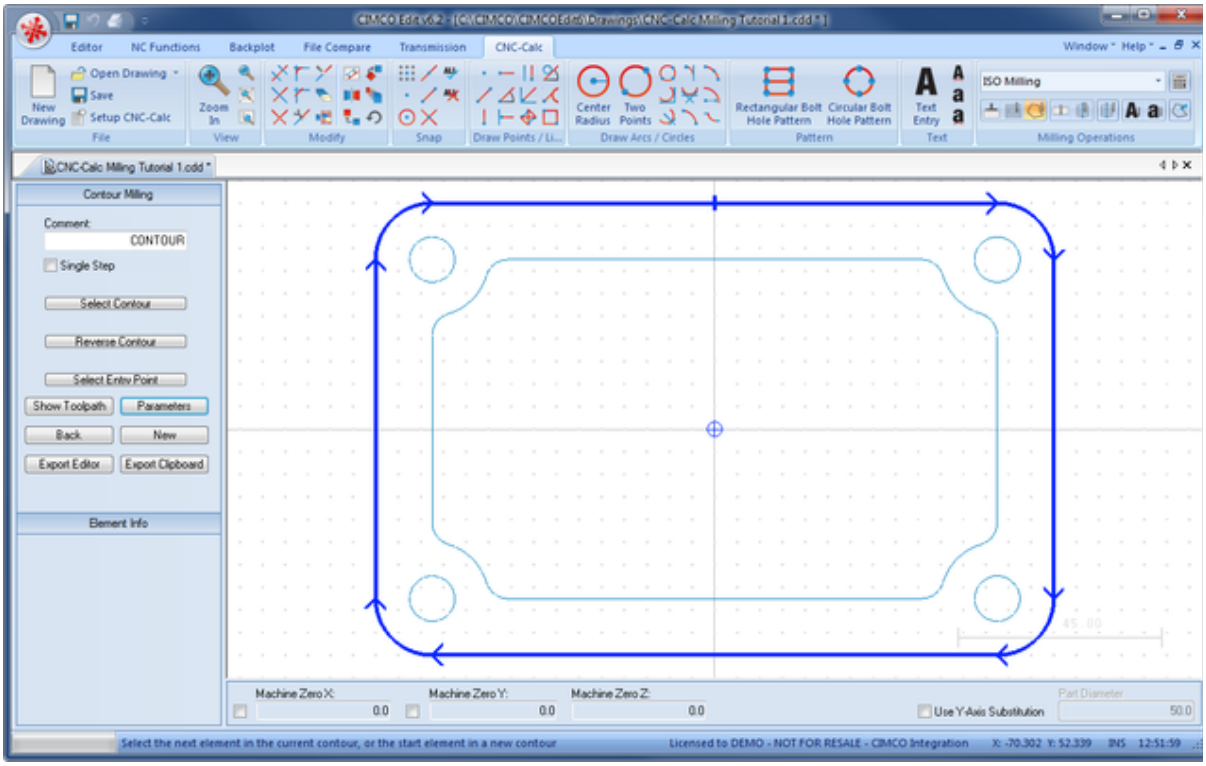
To begin the creation of an NC program for the contour operation, select **Contour Milling** to generate a CNC-toolpath for contour milling (ensure that **ISO Milling** is selected in the field *File Type*).



Write the text CONTOUR in the *Comment* field of the CNC-Calc pane Contour Milling. This text will be included at the start of the final NC code for this operation. When multiple operations exist in the same NC program, the comments will help to locate and identify the start of each operation.

Move the pointer over the outlining contour of the drawing. This highlights the contour element; the arrows indicate the direction the tool will travel. Click on the part of the element that makes the contour direction clockwise like in the picture below.

What side the tool will machine is controlled by the **Work Side** drop-down box on the *General* tab in the parameters dialog.

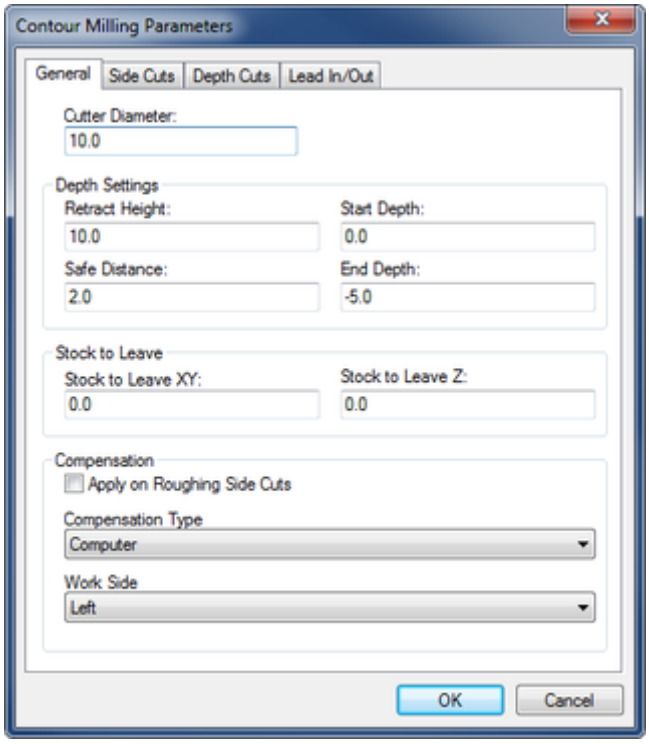


Click on the button **Parameters** in the CNC-Calc pane Contour Milling. This will open the configuration dialog for setting the contour milling parameters.

Enter the values into the Parameters dialogs as shown in the pictures below.

General Tab

This tab contains all the general parameters that are used for roughing and finishing in both depth and side cuts.

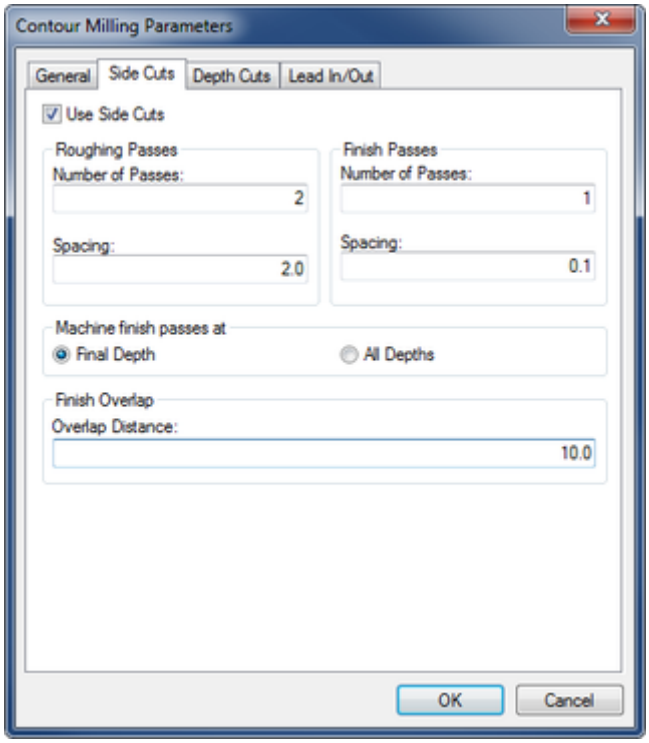


- **Cutter Diameter**: The diameter of the tool in use.
- **Retract Height**: The height to which the tool will move between contours, and where it will stop at the end of the operation.
- **Safe Distance**: The distance above the part, where the feedrate will change from rapid to cutting speed.
- **Start Depth**: This is the top of the stock.
- **End Depth**: The depth at which the last cut will be taken. This value is corrected by the *Stock to Leave Z* value.
- **Stock to Leave XY**: The amount of stock that is left in the XY/side direction at the end of the operation (after both Roughing and Finishing).
- **Stock to Leave Z**: The amount of stock that is left in the Z/depth direction at the end of the operation (after both Roughing and Finishing).
- **Apply on Roughing Sidecuts**: If this option is checked, the compensation type will be applied to both roughing and finishing side cuts. Otherwise computer compensation is used for roughing cuts, and the selected compensation type for finishing cuts.

- **Compensation Type**: This is the compensation type used for the operation.
- **Work Side**: This field determines on which side of the contour the tool will pass. Together with the selected direction of the contour it determines if the milling type will be climb or conventional.

Side Cuts Tab

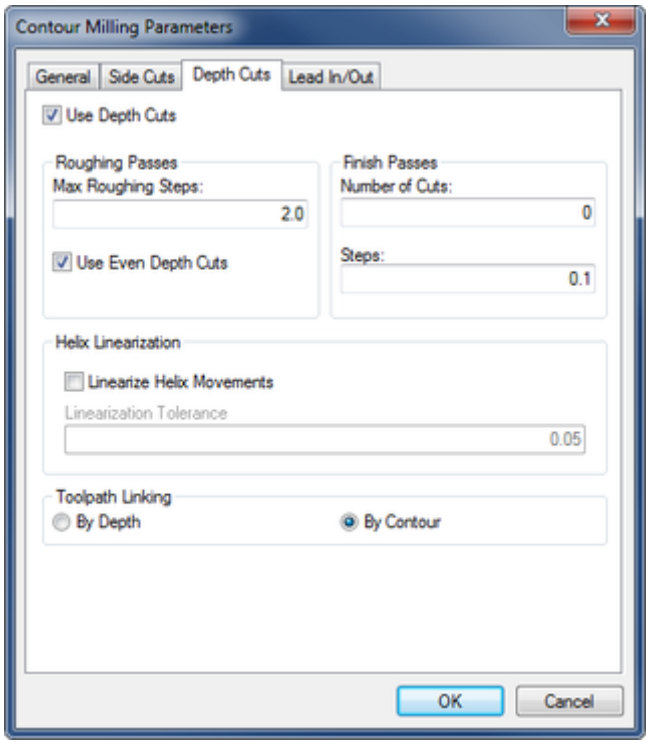
Configures the cuts taken in the XY direction.



- **Use Side Cuts**: If this option is checked, the operation will perform the cuts defined by the parameters. Otherwise, only one cut at the final contour will be performed.
- **Number of Passes** (Roughing Passes): The number of roughing side cuts in the operation.
- **Spacing** (Roughing Passes): If more than one roughing pass is taken, this is the distance between them.
- **Number of Passes** (Finish Passes): The number of finishing side cuts in the operation.
- **Spacing** (Finish Passes): The distance of each finishing pass.
- **Final Depth**: If this radio button is checked, the finishing passes will only be taken at the final depth.
- **All Depths**: If this radio button is checked, the finishing passes will be taken at every depth.
- **Overlap Distance**: The distance that all the finishing laps will overlap, in order to smooth the surface.

Depth Cuts Tab

Configures the cuts taken in the Z direction.

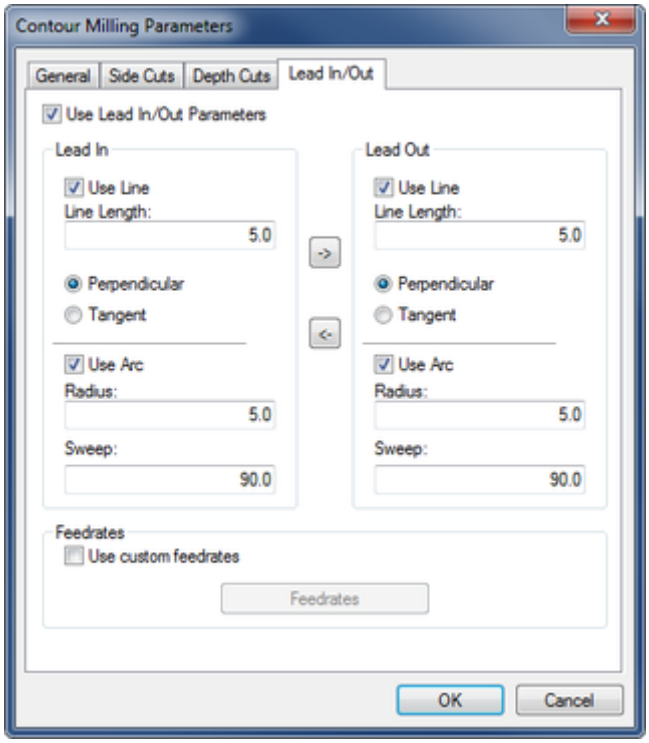


- **Use Depth Cuts**: If this option is checked, the operation will perform the cuts defined by the parameters. Otherwise, only one cut at the final depth will be performed.
- **Max Roughing Steps**: The maximum cut that will be taken in a roughing cut.
- **Use Even Depth Cuts**: If this option is checked, all the roughing passes will have the same distance. If it is left unchecked, cuts will be taken at the *Max Roughing Steps* distance, and any rest material will be taken with the last cut.
- **Number of Cuts**: The number of finishing depth cuts in the operation.
- **Steps**: The distance of each finishing pass.
- **Linearize Helix Movements**: Some machines cannot make helix movements, and if this option is checked, all helix movements will be converted to lines in the NC operation.
- **Linearization Tolerance**: When the helix is converted to lines, this will be the maximum error for the final lines.
- **By Depth**: This is only used if multiple contours are milled in the same operation. If selected, the cut on each depth will be performed on all contours, before any cuts are made at a new depth.
- **By Contour**: If selected, one contour will be milled from start to finish, before the next contour is worked upon.

Lead In/Out Tab

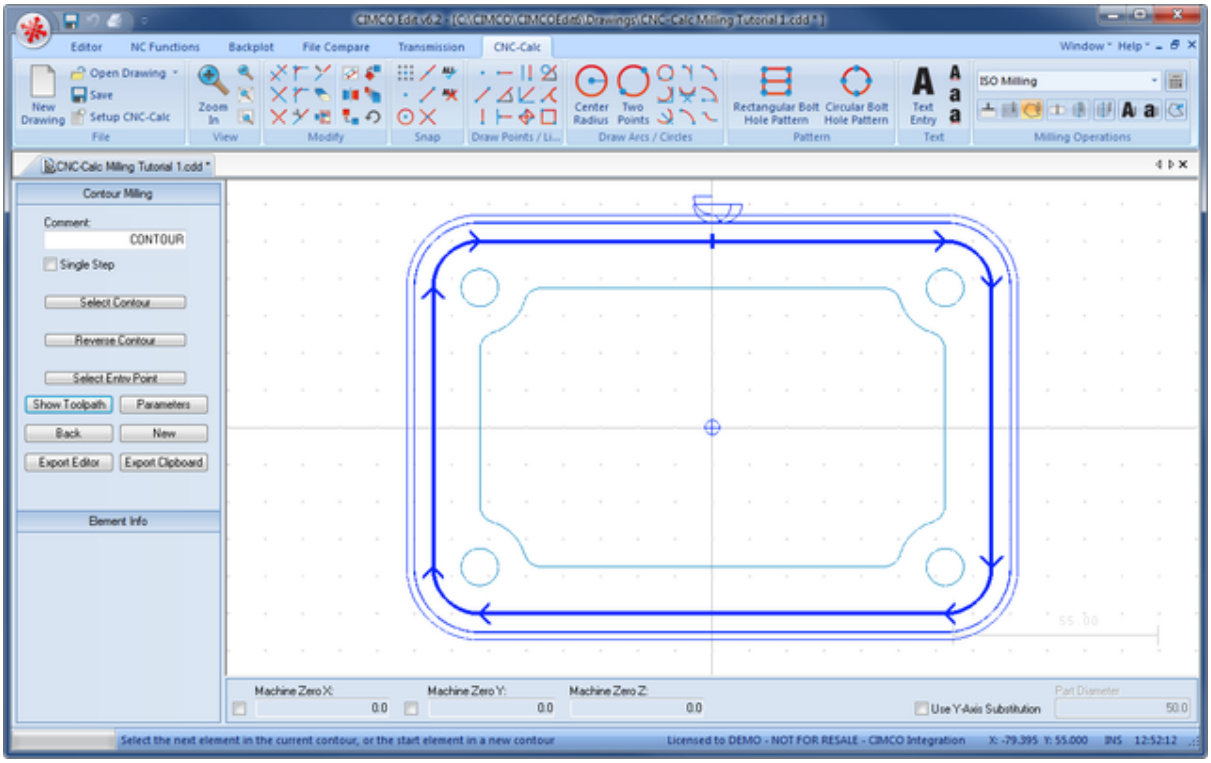
Configures the way the tool will approach the contour at the start/end of the roughing, and for each finishing pass.

The use of lead in/out is optional, when the compensation is set to computer or none. It is however mandatory, when any compensation is performed by the controller.



- **Use Lead In/Out Parameters**: Enables or disables the lead in and out.
- **Use Line**: Enables or disables the lead in/out lines.
- **Line Length**: The length of the lead in/out line.
- **Perpendicular**: If this option is selected, the line will be perpendicular to the following element for lead in, and the previous element for lead out.
- **Tangent**: If this is selected, the line will be tangent to the following element for lead in, and the previous element for lead out.
- **Use Arc**: Enables or disables the lead in/out arcs.
- **Radius**: The radius of the lead in/out arc.
- **Sweep**: The sweep angle of the lead in/out arc.
- The two arrows in the middle of the dialog are used to copy all values from lead in to lead out, and vice versa.
- **Use custom feedrates**: Check this option to enable using custom feedrates for the milling operation.
- **Feedrates**: Click this button to open a new window to enter custom values for *Cutting (XY)*, *Helix/Ramp*, and *Plunging (Z)* feedrates.

Now, close the parameters dialog with **OK**. To show the generated toolpath click on **Show Toolpath** button in the Contour Milling pane.



Click on the button **Export Clipboard**. The NC operation is now in the clipboard, and it is ready for insertion.

Change the window to that of the NC program and press *Ctrl+End* to move to the very end of the file. Insert the text from the clipboard, either by *pressing Ctrl+V*, or selecting the icon  **Paste** from the *Edit* toolbar in the Editor tab.

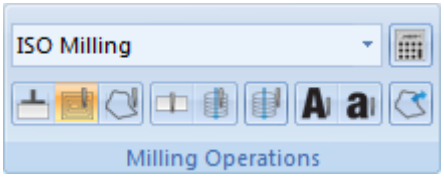
The NC program in the Editor now consists of two operations, and currently they are both made with the same tool. Now we need to insert a new tool for the contour operation. See section "Inserting a Tool with Feed and Speed Calculator" for information on how to insert a tool using the Feed and Speed Calculator.

Pocket Milling

CNC-Calc v7 can generate pocket milling toolpaths.

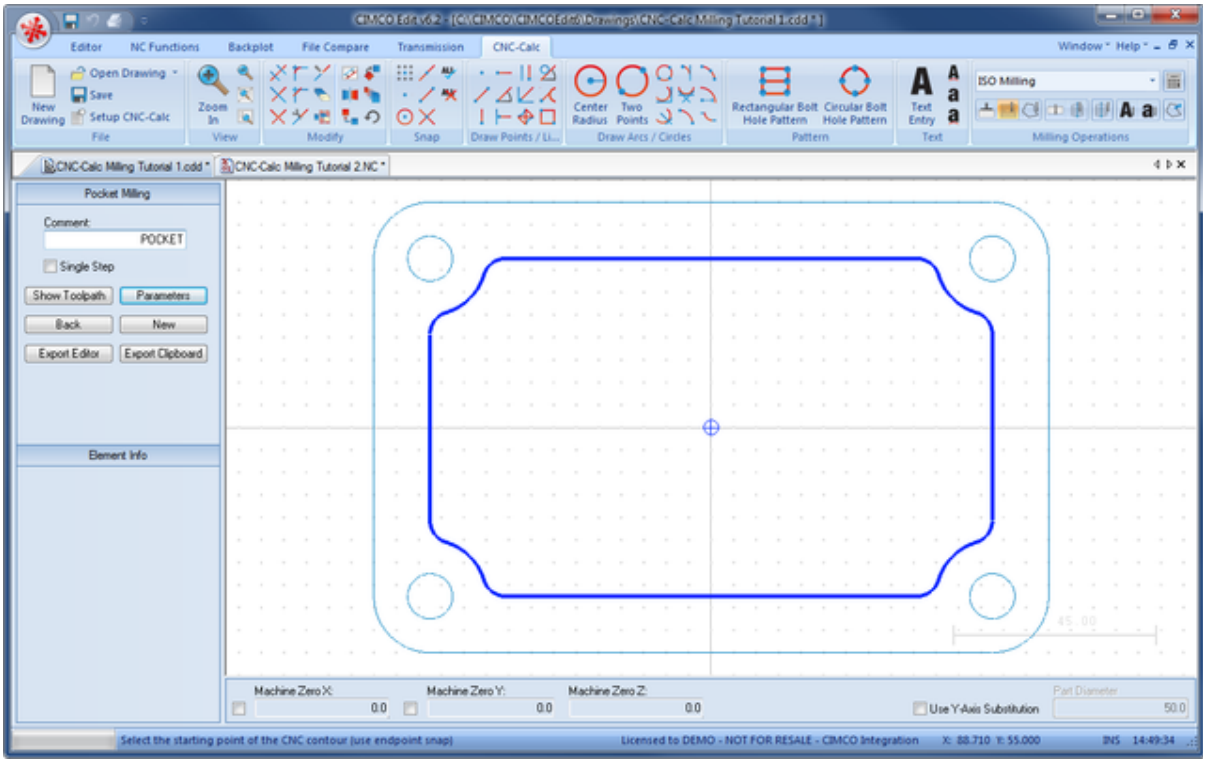
Creation of Pocket Toolpaths

To start creating an NC program for the pocket operation, select the function **Pocket Milling** to generate a CNC-toolpath for pocket milling (ensure that **ISO Milling** is selected in the field *File Type*).



Write the text POCKET in the *Comment* field of the CNC-Calc pane Pocket Milling. This text will be included at the start of the final NC code for this operation. When multiple operations exist in the same NC program, the comments will help to locate and identify the start of each operation.

Click on the inner contour of the drawing. This will highlight the inner contour that will be used as boundary for the pocket operation.

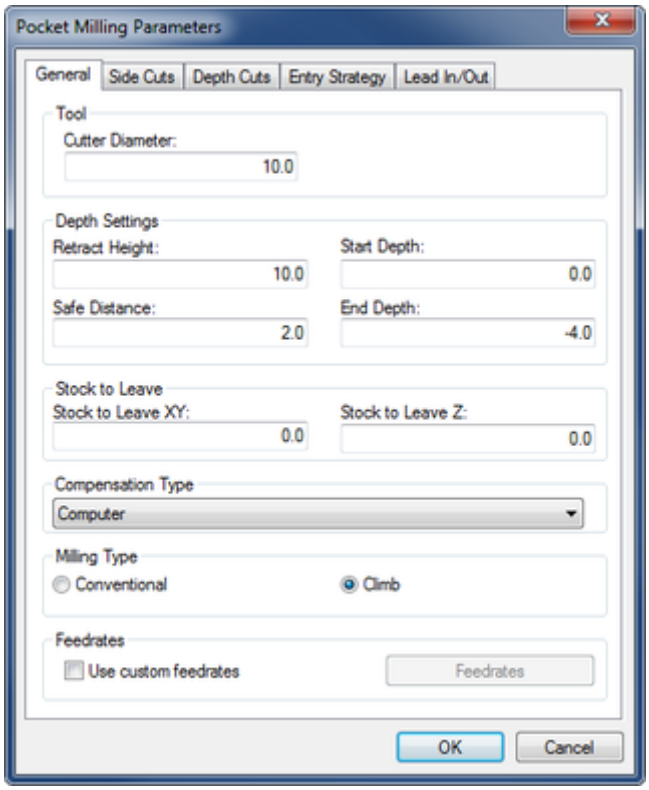


Click on the button **Parameters** in the CNC-Calc pane Pocket Milling. This will open the configuration dialog for setting the pocket milling parameters.

Enter the values into the Parameters dialogs as shown in the pictures below.

General Tab

This tab contains all the general parameters that are used for roughing and finishing in both depth and side cuts.

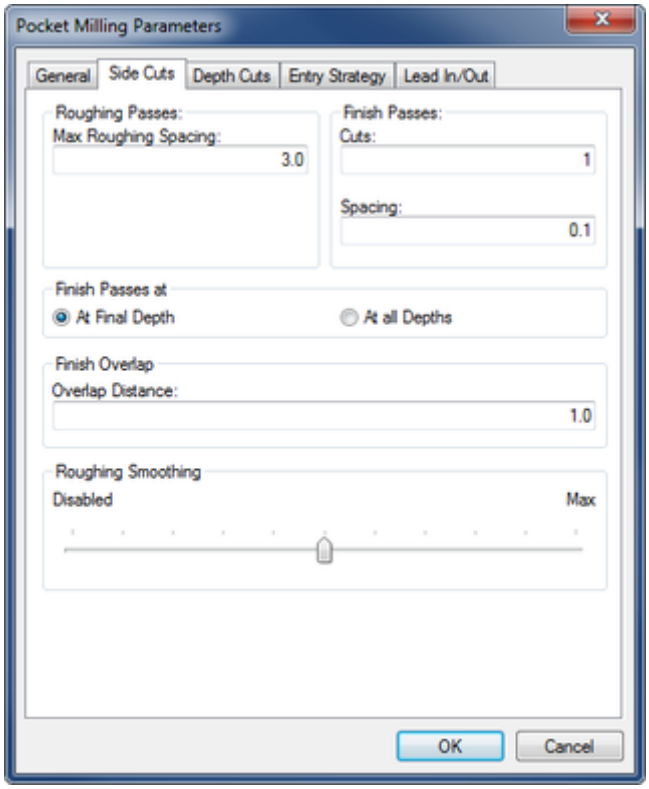


- **Cutter Diameter**: The diameter of the used tool.
- **Retract Height**: The height to which the tool will move between contours, and where it will stop at the end of the operation.
- **Safe Distance**: The distance above the part, where the feedrate will change from rapid to cutting speed.
- **Start Depth**: This is the top of the stock.
- **End Depth**: The depth at which the last cut will be taken. This value is corrected by the *Stock to Leave Z* value.
- **Stock to Leave XY**: The amount of stock that is left in the XY/side direction at the end of the operation (after both Roughing and Finishing).
- **Stock to Leave Z**: The amount of stock that is left in the Z/depth direction at the end of the operation (after both Roughing and Finishing).

- **Compensation Type**: This is the compensation type used for the operation.
- **Conventional**: When checked, the operation will be generated using conventional milling.
- **Climb**: When checked, the operation will be generated using climb milling.
- **Use custom feedrates**: Check this option to enable using custom feedrates for the milling operation.
- **Feedrates**: Click this button to open a new window to enter custom values for *Cutting (XY)*, *Helix/Ramp*, and *Plunging (Z)* feedrates.

Side Cuts Tab

Configures the cuts taken in the XY direction.

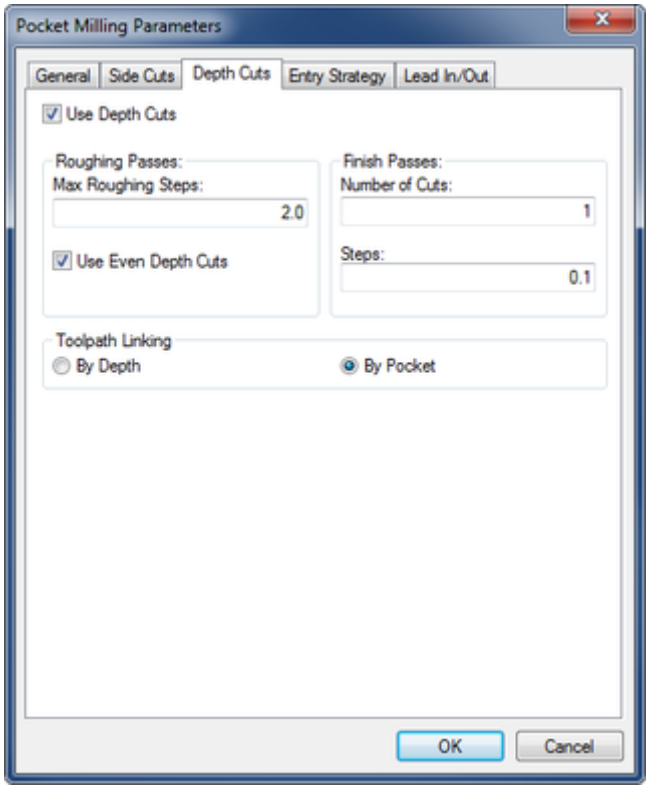


- **Max Roughing Spacing**: The maximum side stepover used in the roughing of the part.
- **Cuts**: The number of finishing side cuts in the operation.
- **Spacing**: The distance of each finishing pass.
- **At Final Depth**: If this radio button is checked, the finishing passes will only be taken at the final depth.
- **At All Depths**: If this radio button is checked, the finishing passes will be taken at every depth.
- **Overlap Distance**: The distance that all the finishing laps will overlap, in order to smooth the surface.

- **Roughing Smoothing**: This slider controls the amount of smoothing used. The higher the value (rightmost), the smoother the resulting toolpath will be.

Depth Cuts Tab

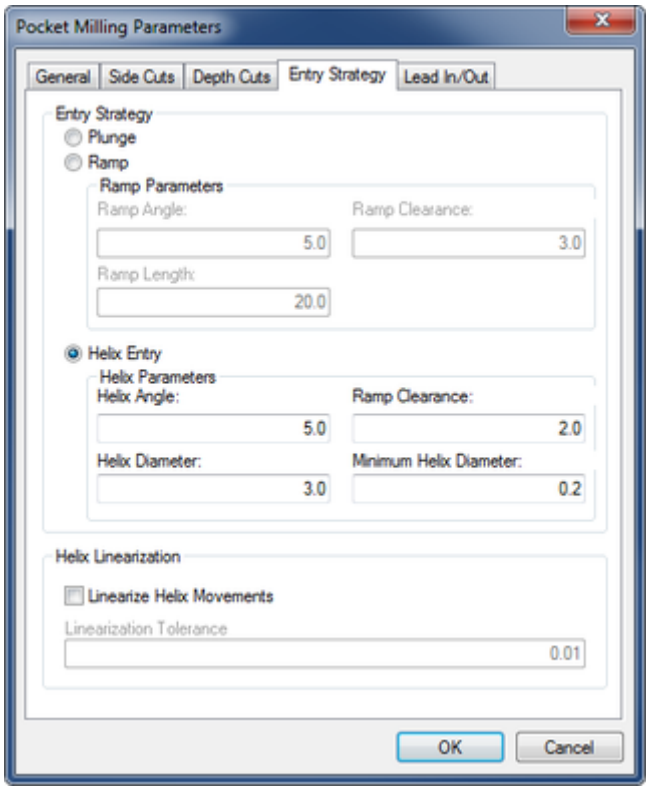
Configures the cuts taken in the Z direction.



- **Use Depth Cuts**: If this option is checked, the operation will perform the cuts defined by the parameters. Otherwise, only one cut at the final depth will be performed.
- **Max Roughing Steps**: The maximum cut that will be taken in a roughing cut.
- **Use Even Depth Cuts**: If this option is checked, all the roughing passes will have the same distance. If it is left unchecked, cuts will be taken at the *Max Roughing Steps* distance, and any rest material will be taken with the last cut.
- **Number of Cuts**: The number of finishing depth cuts in the operation.
- **Steps**: The distance of each finishing pass.
- **By Depth**: This is only used if multiple pockets are milled in the same operation. If selected, the cut on each depth will be performed on all pockets before any cuts are made at a new depth.
- **By Pocket**: If selected, one Pocket will be milled from start to finish before the next pocket is worked upon.

Entry Strategy Tab

Configures how the tool cuts from one Z level to the next.

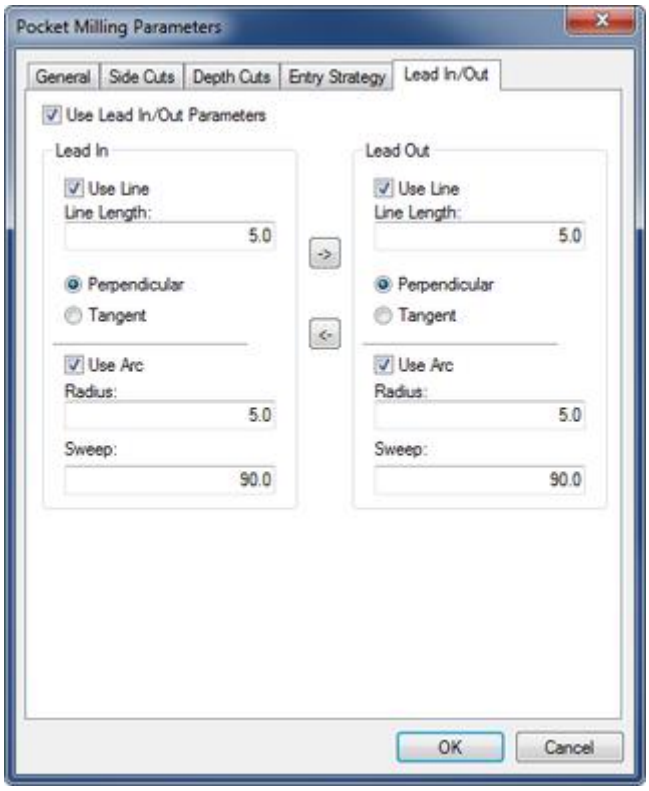


- **Plunge**: When this is selected, the tool will move straight down.
- **Ramp**: With the ramp entry, the tool moves down to the **Ramp Clearance** above the part. Then it makes a ramp movement with the length **Ramp Length** and the angle **Ramp Angle**.
- **Helix Entry**: Moves down to **Helix Clearance** above the part. Then it will spiral down with the angle **Helix Angle** in a circular movement with a diameter between **Helix Diameter** and **Minimum Helix Diameter**. How big the actual diameter will be depends on the geometry.
- **Linearize Helix Movements**: Some machines cannot make helix movements, and if this option is checked, all helix movements will be converted to lines in the NC operation.
- **Linearization Tolerance**: When the helix is converted to lines, this will be the maximum error for the final lines.

Lead In/Out Tab

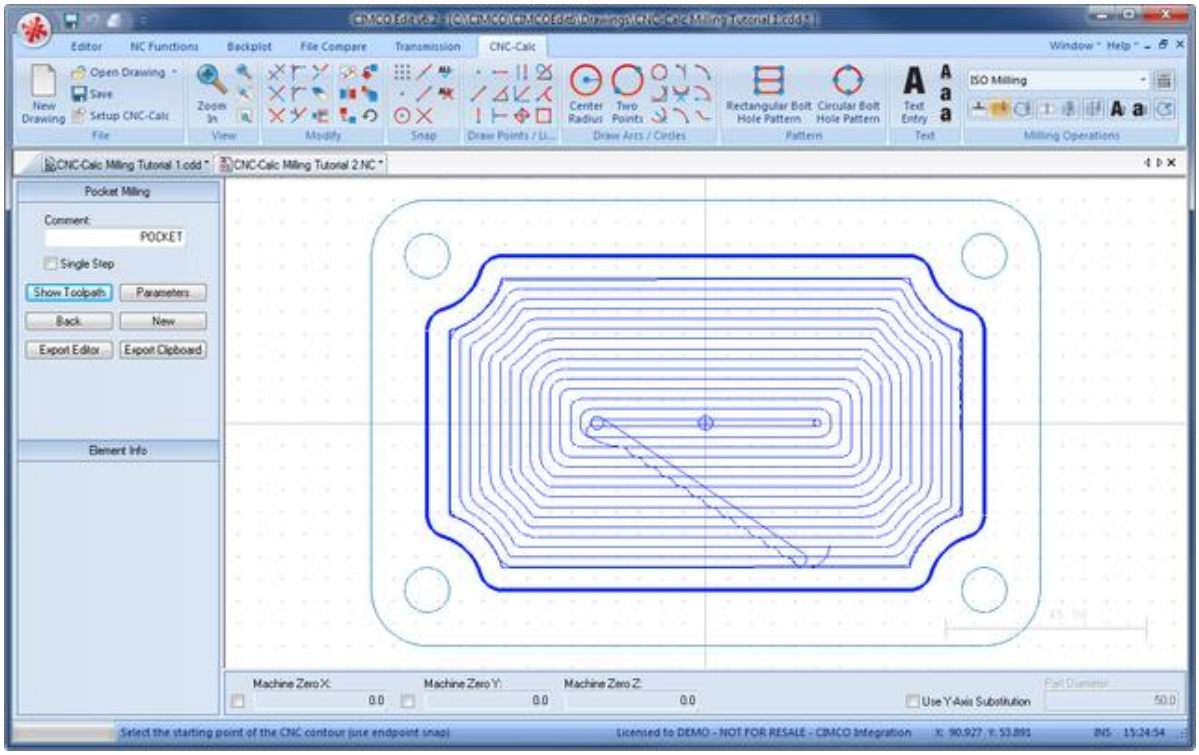
Configures the way the tool will approach the pocket at the start/end of the roughing, and for each finishing pass.

The use of lead in/out is optional, when the compensation is set to computer or none. It is however mandatory, when any compensation is performed by the controller.



- **Use Lead In/Out Parameters**: Enables or disables the lead in and out.
- **Use Line**: Enables or disables the lead in/out lines.
- **Line Length**: The length of the lead in/out line.
- **Perpendicular**: If this option is selected, the line will be perpendicular to the following element for lead in, and the previous element for lead out.
- **Tangent**: If this is selected, the line will be tangent to the following element for lead in, and the previous element for lead out.
- **Use Arc**: Enables or disables the lead in/out arcs.
- **Radius**: The radius of the lead in/out arc.
- **Sweep**: The sweep angle of the lead in/out arc.
- The two arrows in the middle of the dialog are used to copy all values from lead in to lead out, and vice versa.

Now, close the parameters dialog with **OK**. To show the generated toolpath click on **Show Toolpath** button in the Pocket Milling pane.



Click on the button **Export Clipboard**. The NC operation is now in the clipboard, and it is ready for insertion.

Change the window to that of the NC program and press *Ctrl+End* to move to the very end of the file. Insert the text from the clipboard, either by *pressing Ctrl+V*, or selecting the icon  **Paste** from the *Edit* toolbar in the Editor tab.

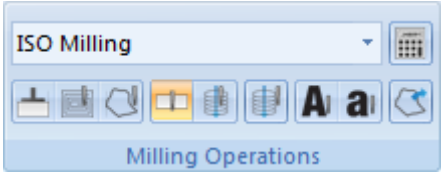
The NC program in the Editor now consists of three operations, and since we use the same tool for the contour and pocket operations we will not insert a tool before the pocket operation.

Drilling

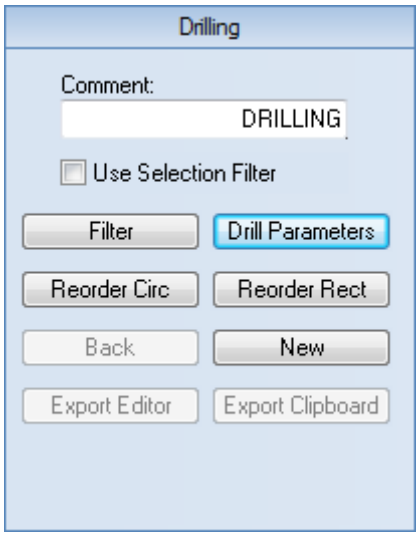
CNC-Calc v7 can generate codes for drilling in either canned cycles or as longhand.

Generate a Drill Cycle

To start creating a NC-program for the drilling operation, select the function **Drill Holes** in the *Milling Operations* toolbar to generate a drill cycle (ensure that **ISO Milling** is selected in the field *File Type*).

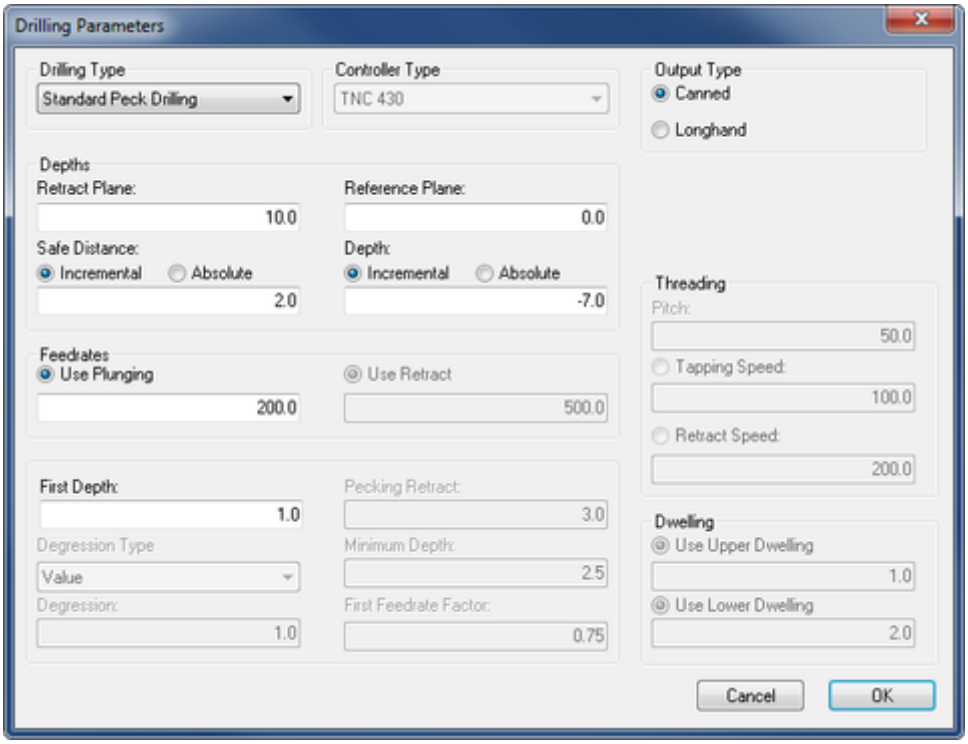


Write the comment DRILLING in the *Comment* field of the CNC-Calc pane Drilling. This text will be included at the start of the final NC code for this operation. When multiple operations exist in the same NC program, the comments will help to locate and identify the start of each operation.



Click on the button **Drill Parameters** to open the parameter dialog window shown below. For this drilling operation, please enter the parameters shown.

Drilling Parameters



- **Drilling Type**: This drop-down box is used to select the operation type. The possible parameters depend on the type selected.
- **Canned** (Output Type): Select this radio button to use a canned cycle. The canned cycle depends on the selected machine, and the possible parameters reflect this canned cycle.
- **Retract Plane**: The retract plane is the height that the tool is moved to before it traverses between holes.
- **Reference Plane**: This is the height of the material. For some machines like Maho, this is also the height that the operation is calculated around.
- **Safe Distance**: The safe distance is the distance above the reference plane where all moves toggle between feed and rapid.
- **Depth**: This field is used to enter the final depth of the operation.
- **Use Plunging**: This radio button is used to indicate if plunging moves should be performed with the entered plunging feedrate.
- **First Depth**: This field is used to enter the first depth for a pecking operation. The following pecks will be calculated based on degression and minimum depth.

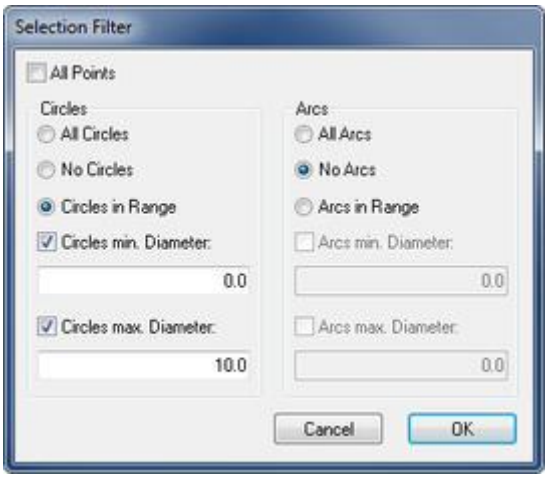
  Notice that in this example it makes no difference if **Incremental** or **Absolute** is selected for *Safe Distance* and *Depth*, since these incremental values refer to the *Reference Plane*, which is 0.

For the selection of the location of the holes, several options are available:

1. Select each hole location with the cursor. In order to get the correct hole center for circles and arcs, the **Snap to Center Points** function should be used.
2. Select the actual circle or arc. This will create a new hole location at the center of the circle/arc.
3. Use window selection with or without filter. If the filter is used, it is possible to limit the selection to circles or arcs in different ranges.

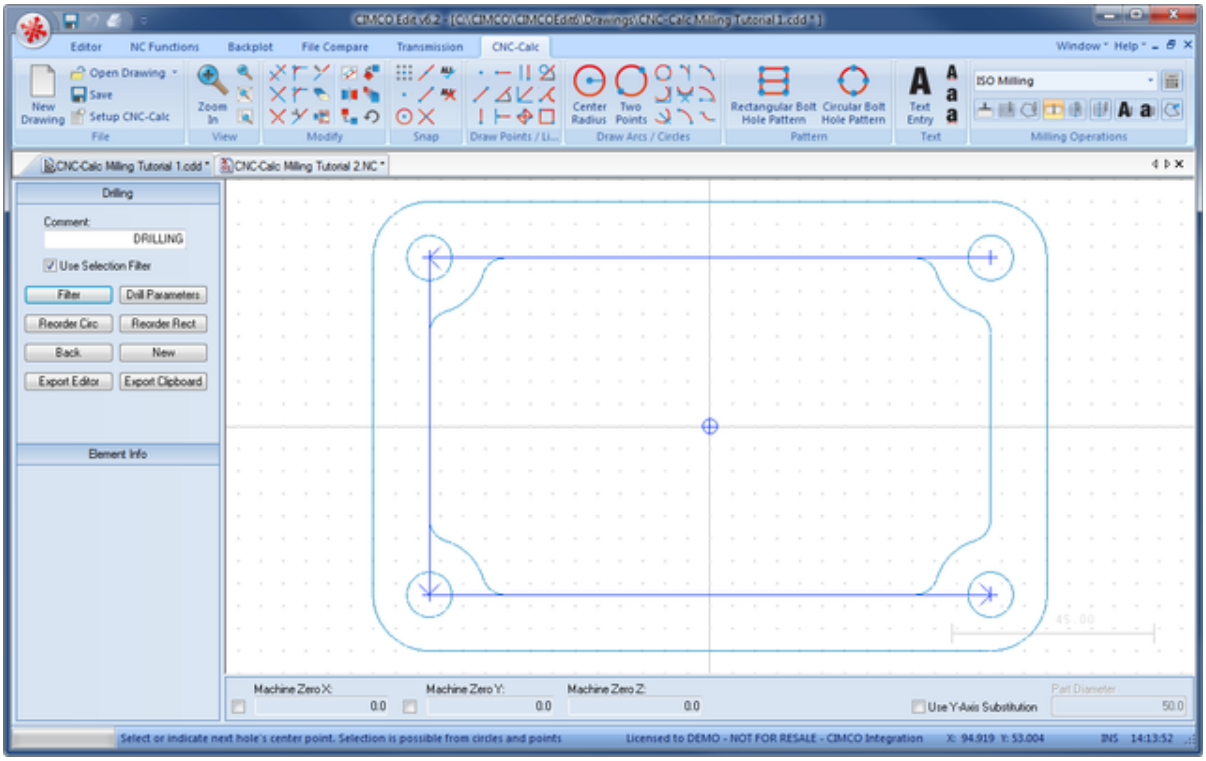
In the following we will use the filter to select the corner holes, but not any of the arcs.

Click on the button **Filter** in the CNC-Calc pane Drilling. By setting up the filter as shown, we will limit the window selection to include only circles in the range from 0 to 10 in diameter. Click **OK** after entering the values shown.



Now enable the option **Use Selection Filter** in the left hand pane, and then make a window selection that includes the entire drawing.

When this selection is made, only the four corner holes should be selected.

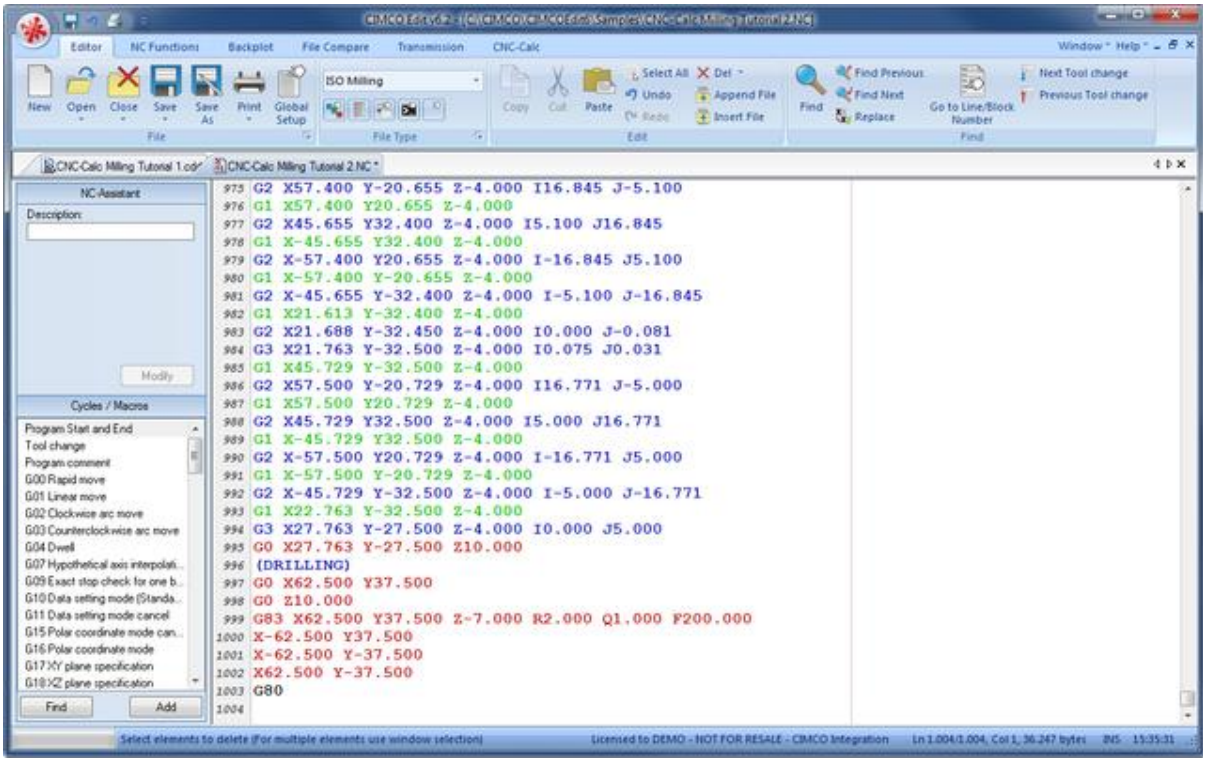


The order of operation can then be changed by clicking on **Reorder Circ** and **Reorder Rect** in the Drilling pane.

Click on the button **Export Clipboard**. The drilling operation is now in the clipboard, and it is ready for insertion.

Change the window to that of the NC program and press *Ctrl+End* to move to the very end of the file. Insert the text from the clipboard, either by pressing *Ctrl+V*, or selecting the icon **Paste** from the *Edit* toolbar in the Editor tab.

The NC program should look like the following.



Since the feedrate for the operation is defined in the canned cycle, we will enter manually the tool change. Write the following line just before the DRILLING comment:

T3 M06 S1200

This will assign the tool no. 3 with a spindle speed of 1200 rpm to the drilling operation.