Gear Housing Tutorial

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Getting Started

Before starting this Gear Housing tutorial you should print the accompanying drawing from the **Learning Centre (http://www.sharpcam.co.uk/Training-Centre.aspx)**. Alternatively the drawing is also located at end of this tutorial.

Open a New Part

Create a new Part, depending on your settings SharpCam may have created a new Part on launch.

Menu: File -> New Part

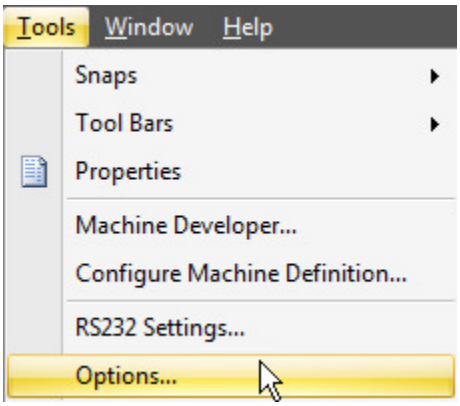
Toolbar button: 

Shortcut: Ctrl + N

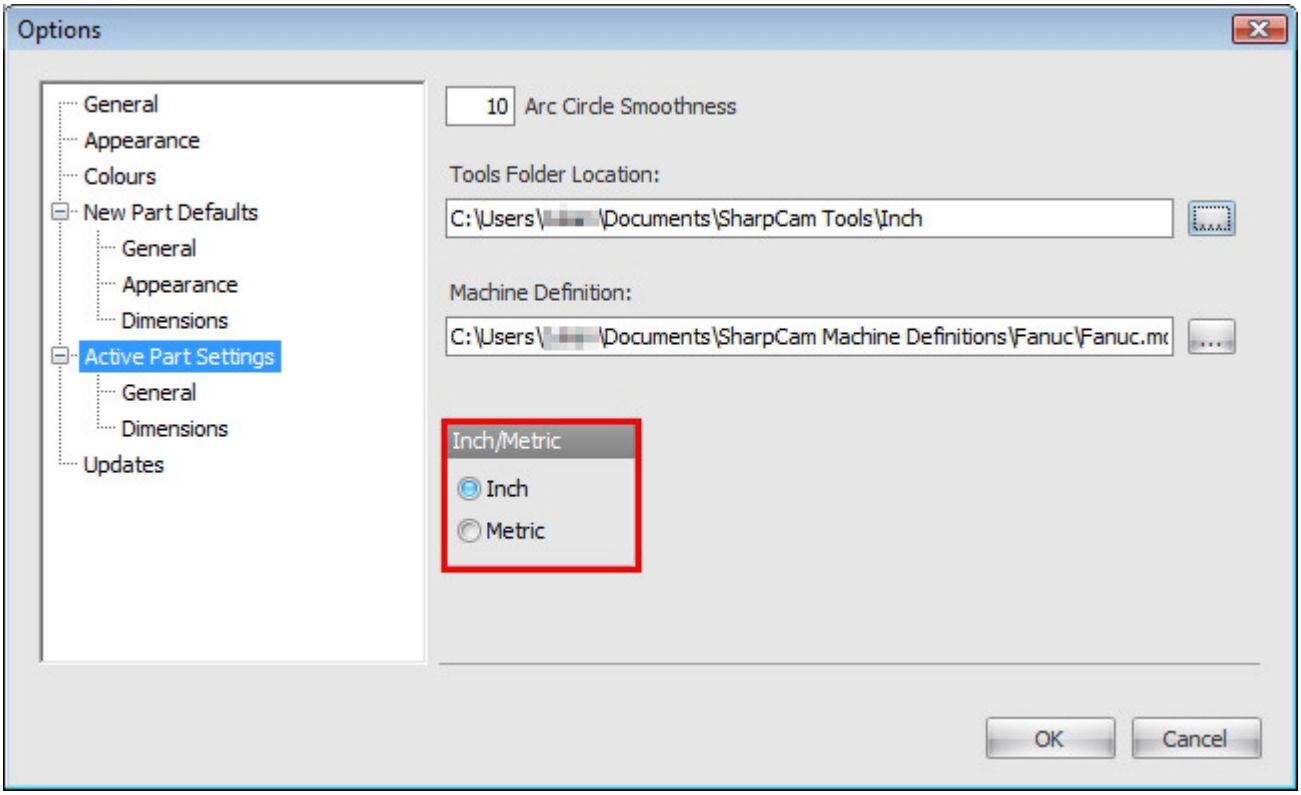
Units

Before starting it is important that the units are correct for the Part. When SharpCam was installed the default units would have been set for a new Part. It is also possible to set the units for an active Part. First check that the units are correct for the active Part:

Choose the Options... command from the Tools menu:



Select the Active Part Settings node and make sure that the Inch radio button is selected and click the OK button:



A confirmation message box will be shown if the units are changed, click OK to accept.

Other Settings

For this tutorial it is assumed that *Connection Markers* are displayed and *Axis Icon at Origin* is not active:

Snap Orthogonal and *Multi Snap* are being used:

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Create the Contours

Step 1 - Create three circles

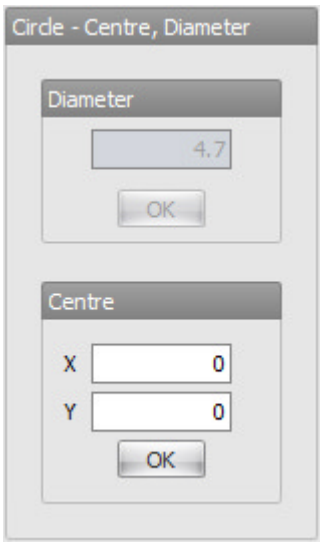
Before starting decide the origin for the Part. In SharpCam the origin is not actually set, the decision must be made where X0, Y0 is going to be located on the Part and create the Contours around this point.
A logical place for X0, Y0 is the centre of the Ø4.7 circle.

Create three circles using Circle - Centre, Diameter

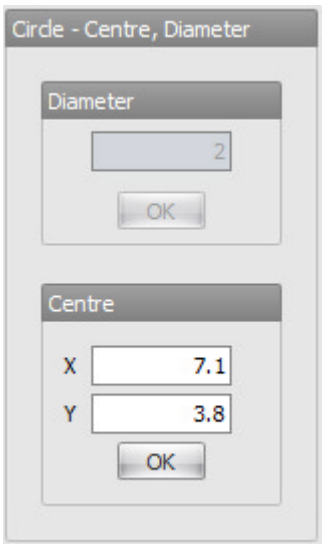
Menu: Draw -> Circle - Centre, Diameter

Toolbar button:

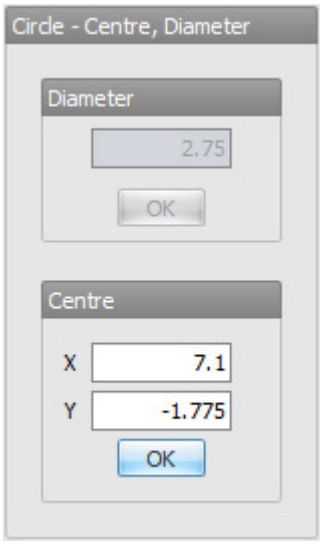
Choose the command and create a Ø4.7 circle, centred at X0, Y0:



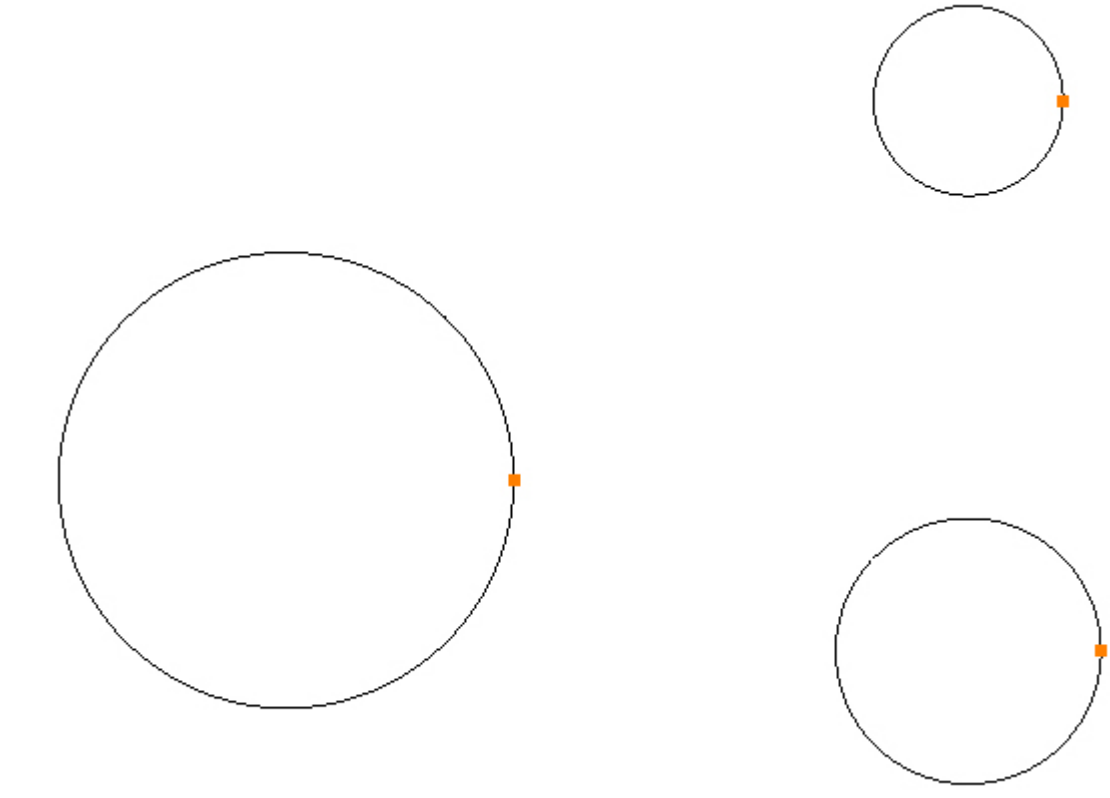
Create another circle Ø2.0 centred at X7.1, Y3.8:



Create another circle Ø2.75 centred at X7.1, Y-1.775:



The Part should now look like this:



Step 2 - Create offsets

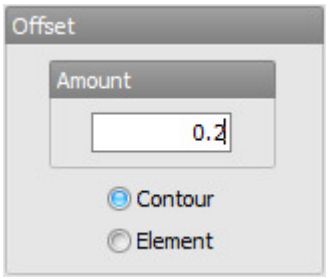
Create a 0.2 offset on the outside of the 3 circles using Offset

Menu: Modify -> Offset

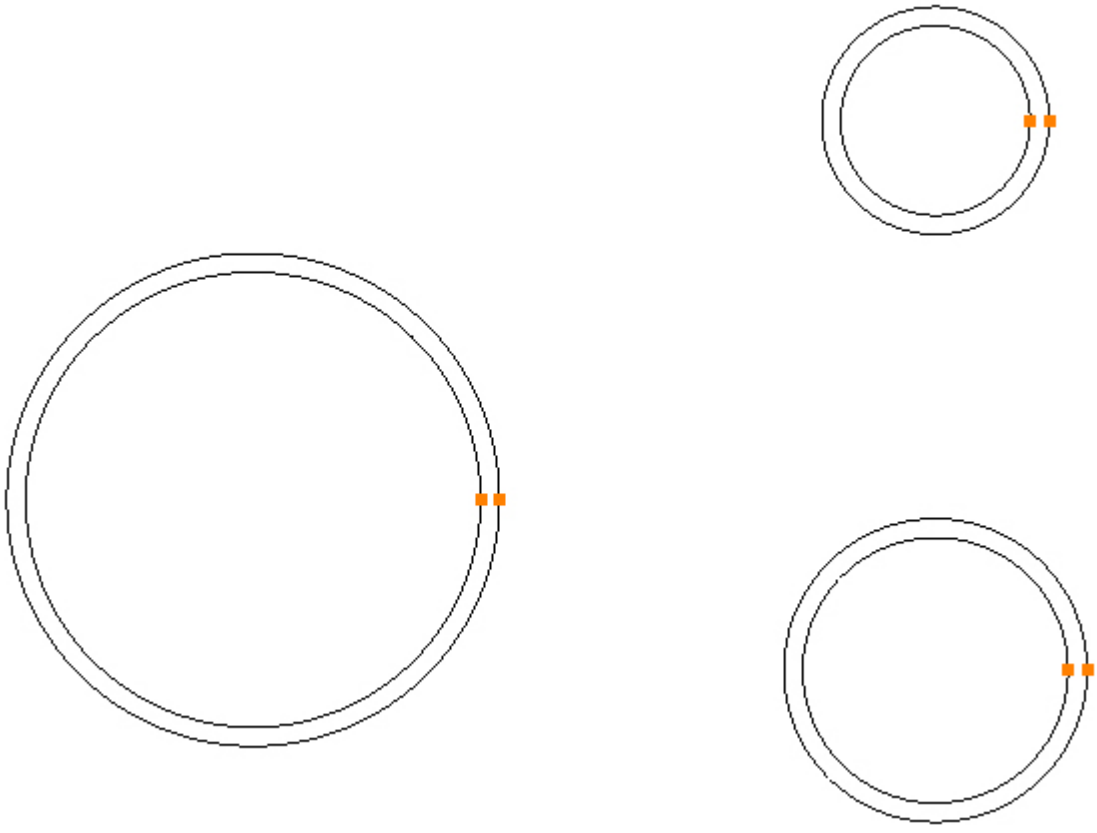
Toolbar button: 

Looking at the drawing it can be seen that there is a 0.2 typical offset that applies to the circles previously created.

Select the command and enter 0.2 into the Amount box. It does not matter if *Contour* or *Element* is selected when offsetting from a circle:



Move the cursor over the Ø4.7 circle and left mouse click to indicate that the offset is to be from this circle, then left click on the outside of the circle to create the offset; repeat for the Ø2.0 and Ø2.75 circles:



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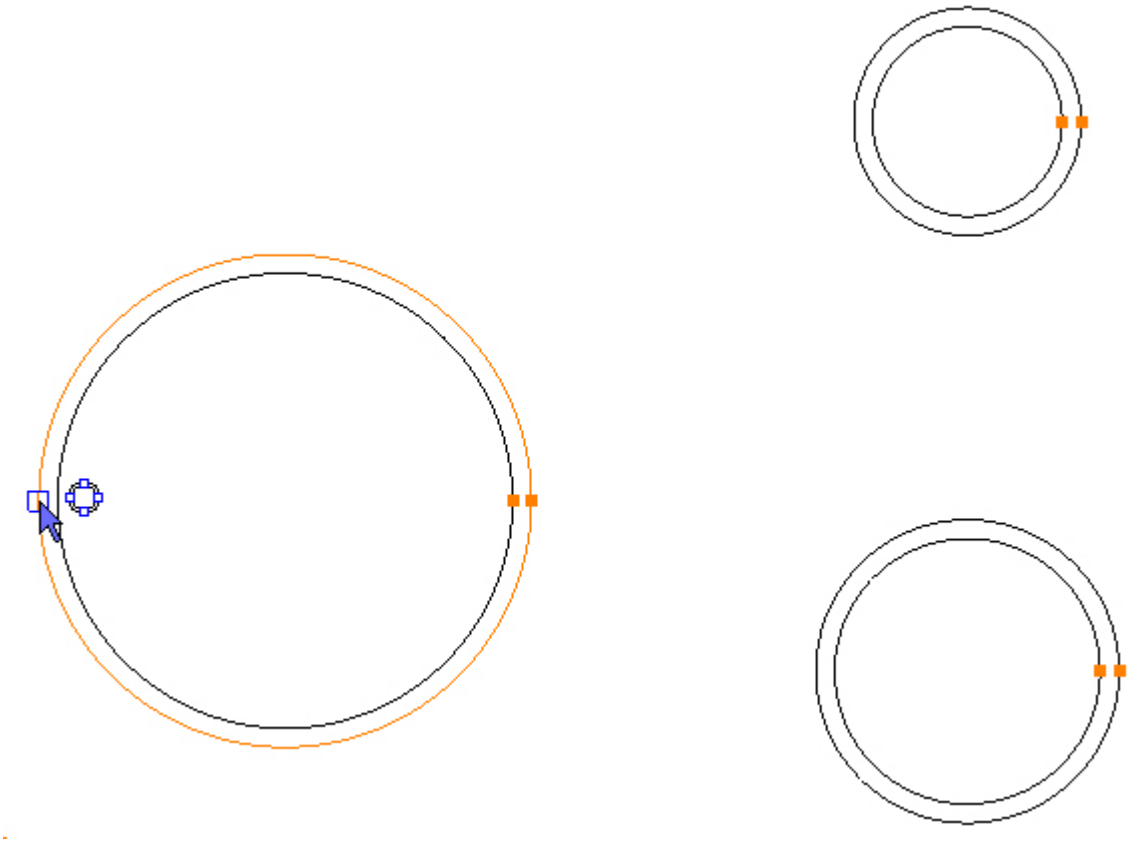
Step 3 - Create line

Create the left edge using Line

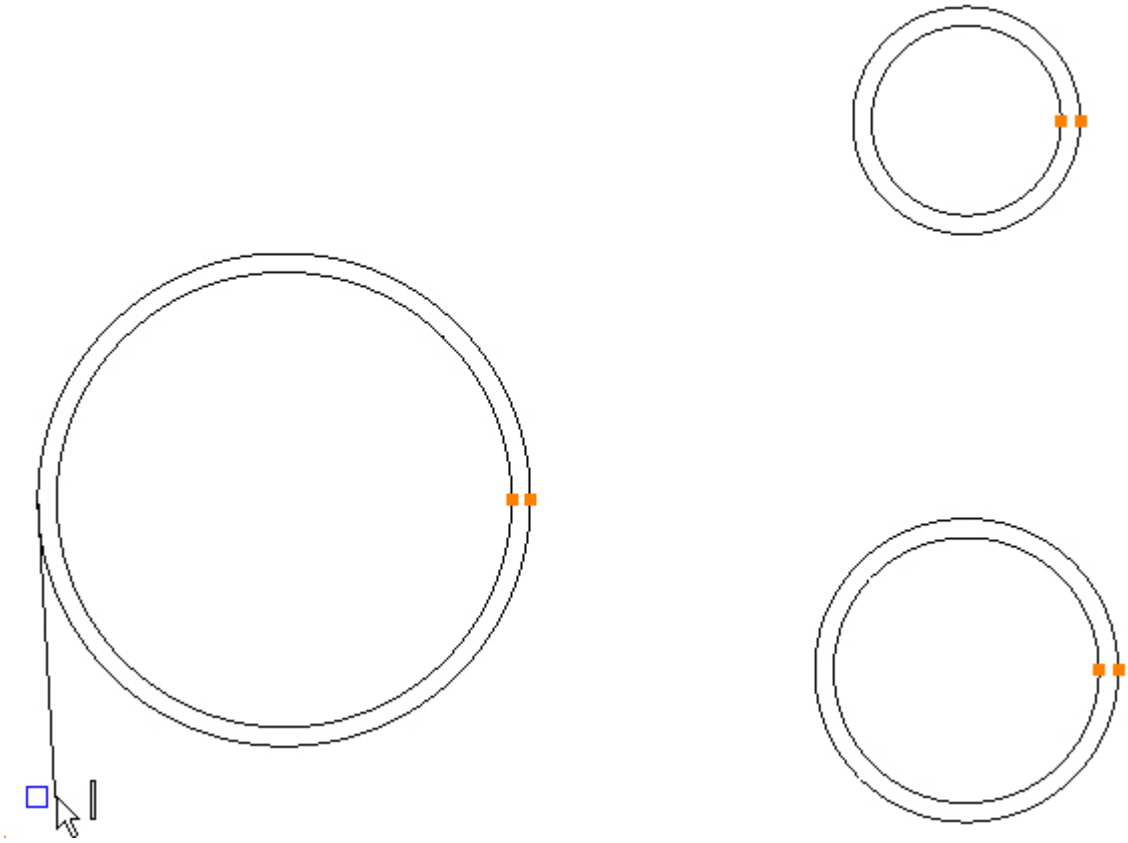
Menu: Draw -> Line

Toolbar button: 

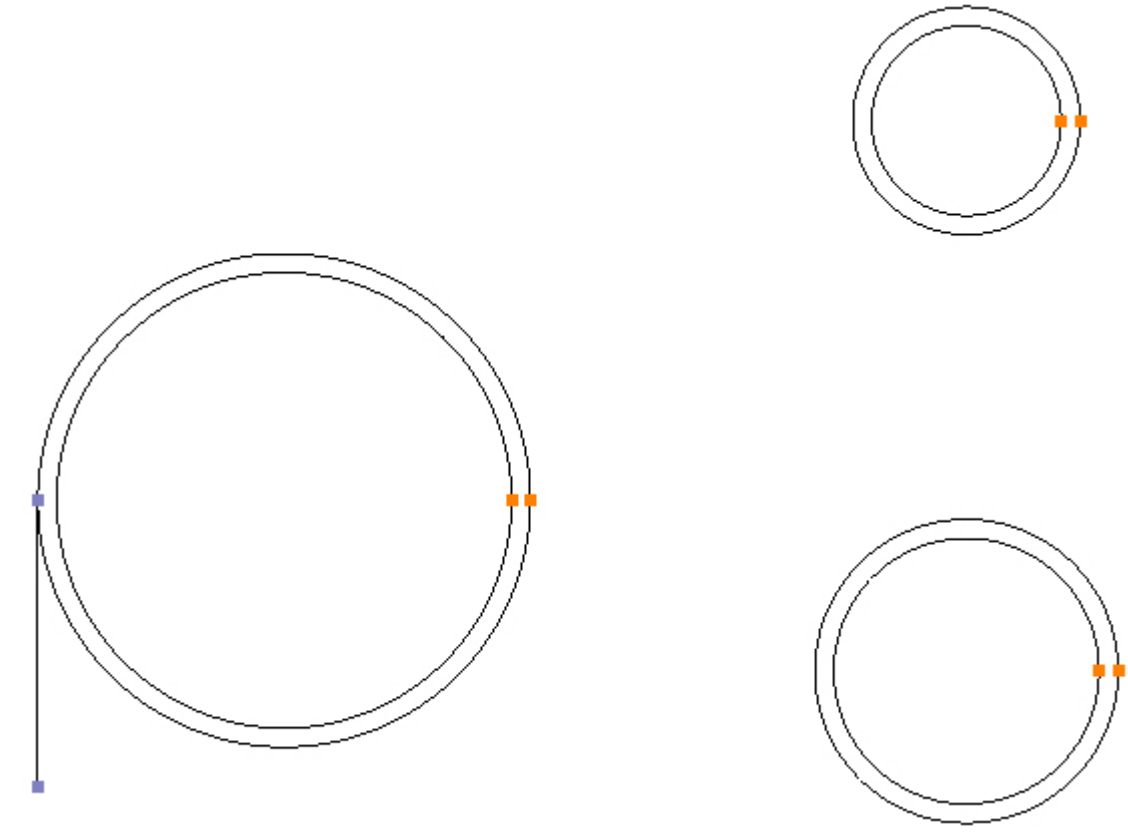
Choose the command and move the cursor near to the 9 o'clock position on the Ø5.1 circle:



Snap the start point of the line to the quadrant point, the cursor is showing the Quadrant snap symbol and a blue Snap Marker indicates the Snap Point. Left click the mouse to accept the Snap Point and create the start of the line:



Move the cursor so it is below the start of the line, the cursor is showing the Vertical Orthogonal snap symbol and a blue Snap Marker indicates the Snap Point. This will ensure that the line is created vertically. Left click the mouse to accept the Snap Point and create the end of the line:



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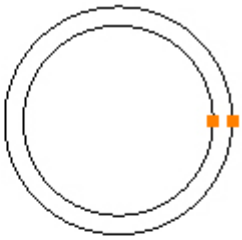
Step 4 - Create line

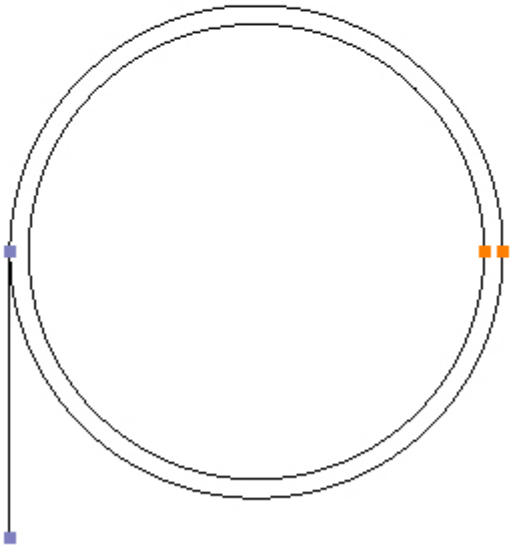
Create the bottom edge using Line

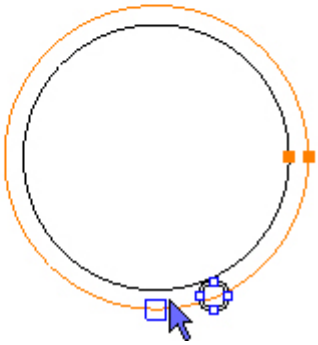
Menu: Draw -> Line

Toolbar button: 

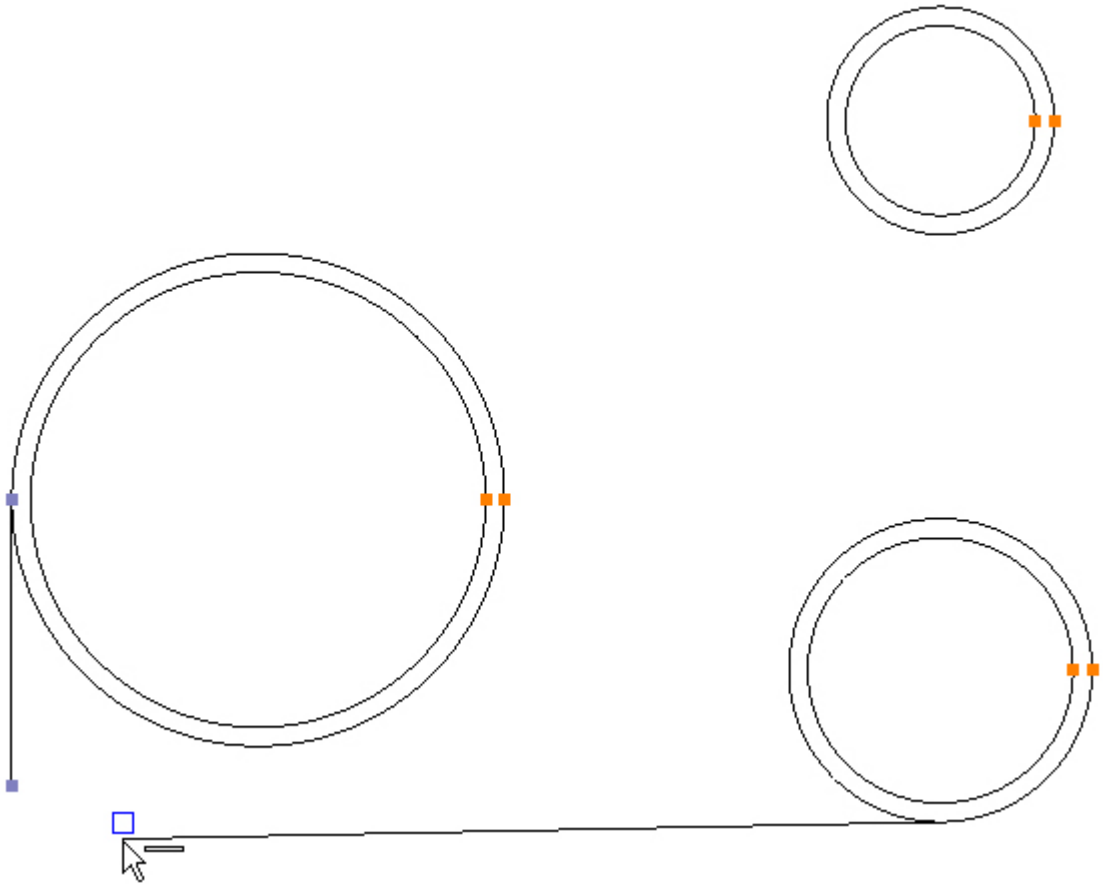
Choose the command and move the cursor near to the 6 o'clock position on the Ø3.15 circle:



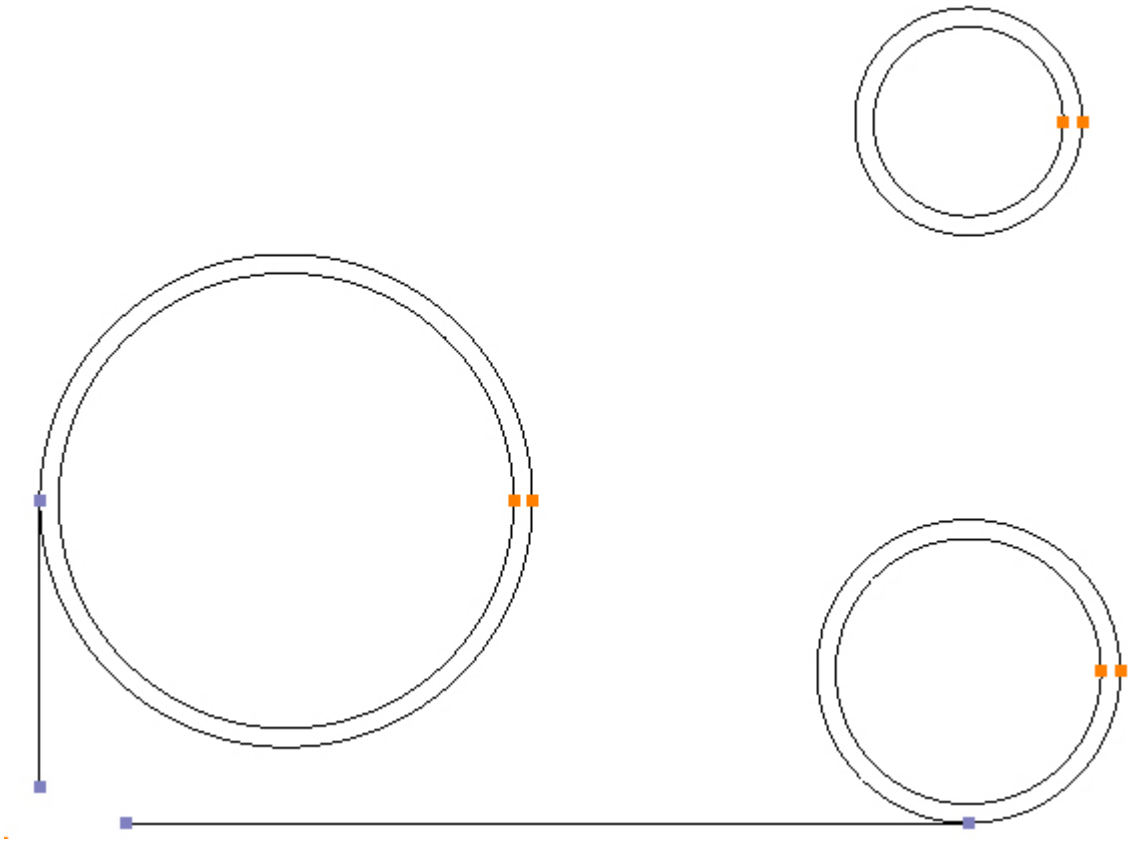




Snap the start point of the line to the quadrant point, the cursor is showing the Quadrant snap symbol and a blue Snap Marker indicates the Snap Point. Left click the mouse to accept the Snap Point and create the start of the line:



Move the cursor so it is to the left of the start of the line, the cursor is showing the Horizontal Orthogonal snap symbol and a blue Snap Marker indicates the Snap Point. This will ensure that the line is created horizontally. Left click the mouse to accept the Snap Point and create the end of the line:



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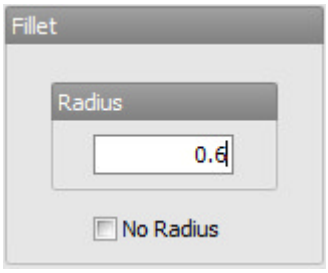
Step 5 - Create fillet

Create the R0.6 radius at the bottom lefthand corner using Fillet

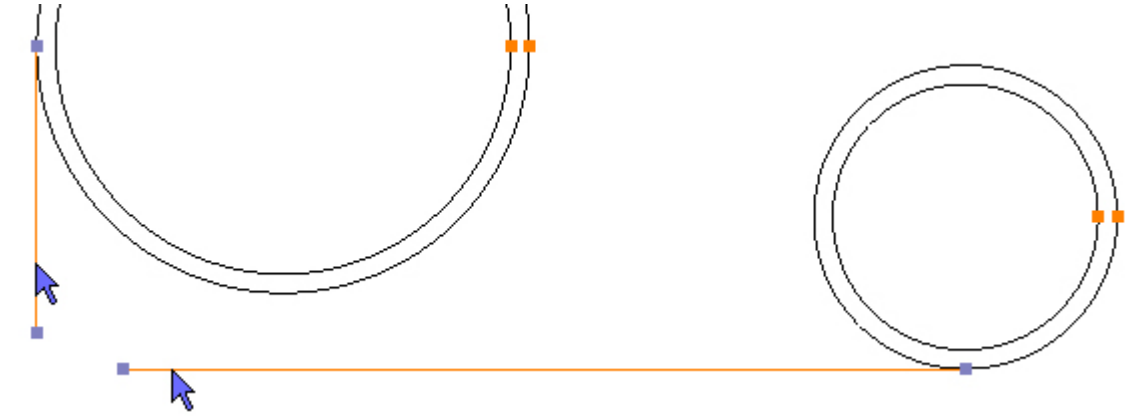
Menu: Modify -> Fillet

Toolbar button:

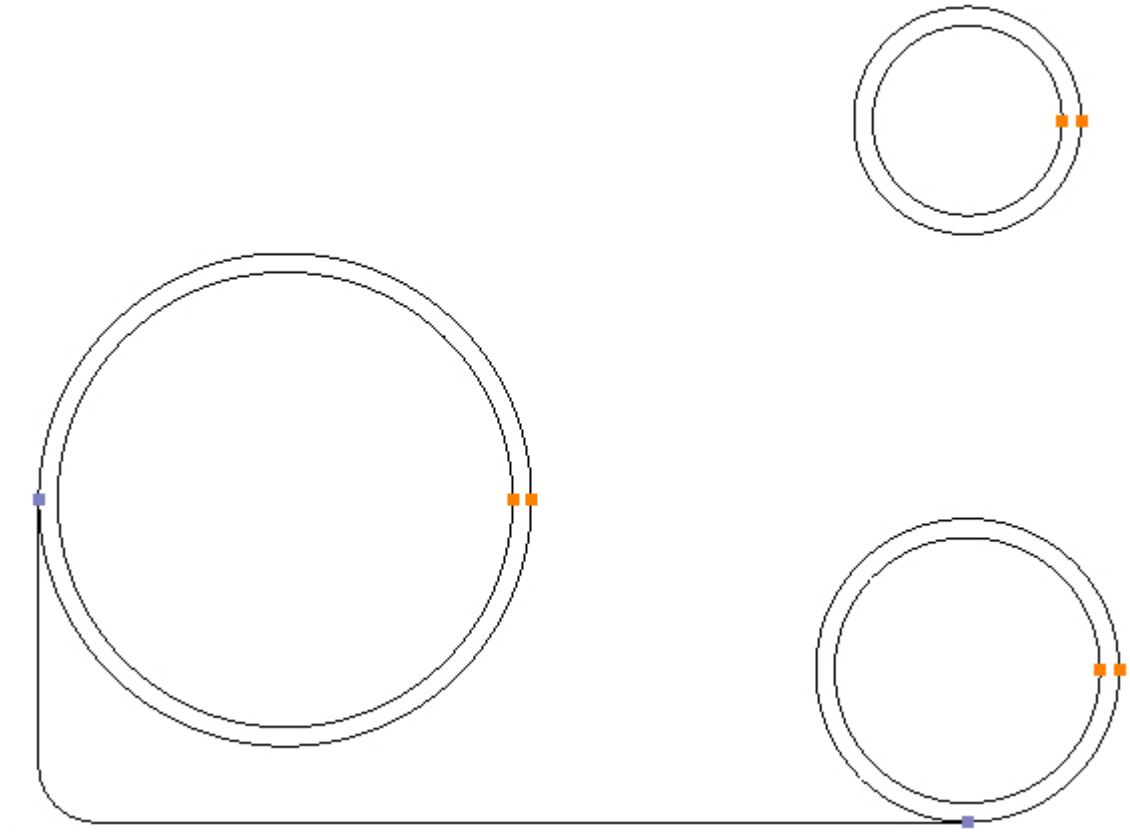
Choose the command and enter 0.6 in the radius box:



Highlight and left mouse click the two lines:



The fillet is created:



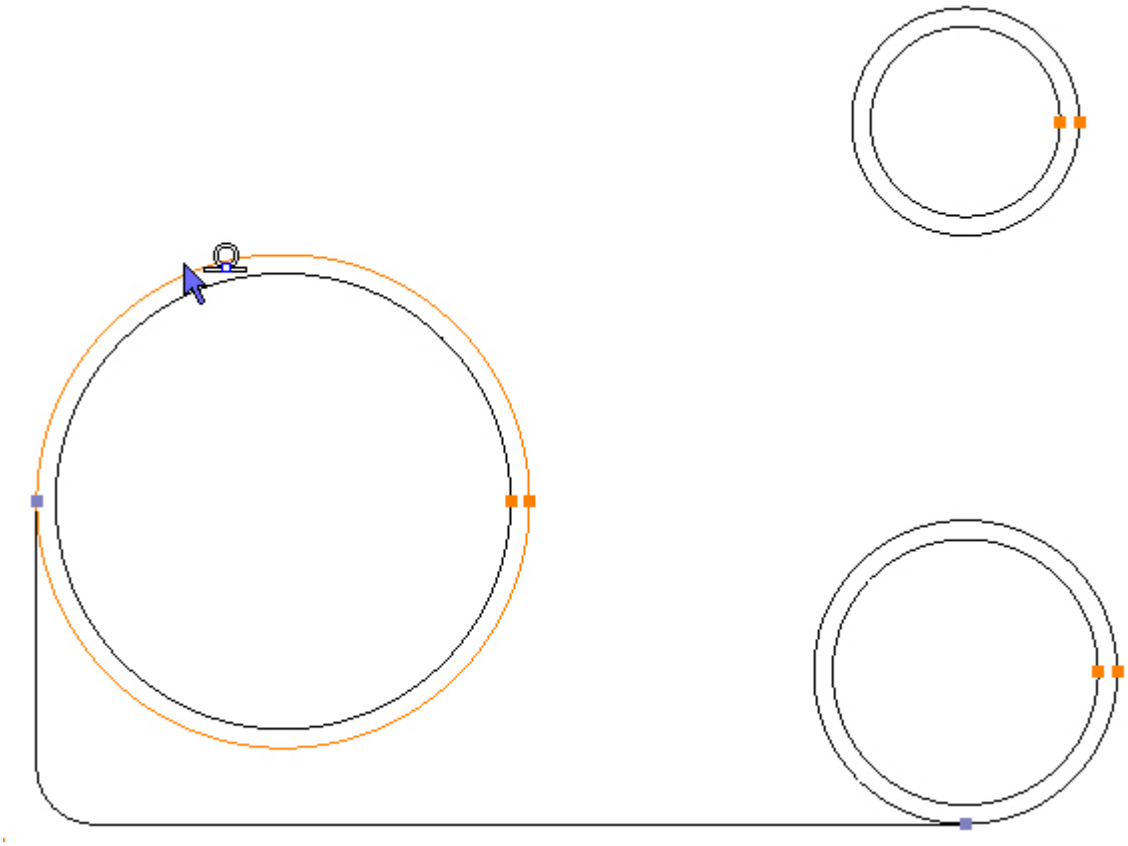
Step 6 - Create line

Create the line tangentially between the Ø5.1 and Ø2.4 circles using Line

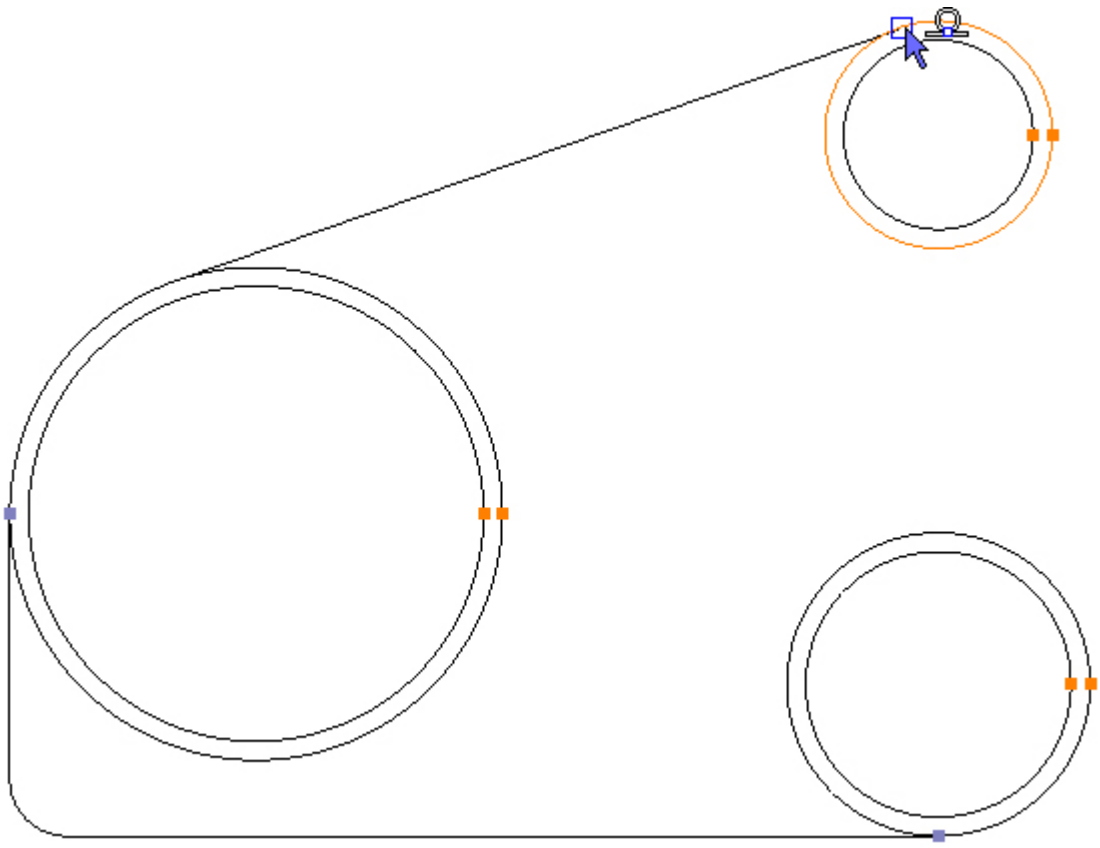
Menu: Draw -> Line

Toolbar button: 

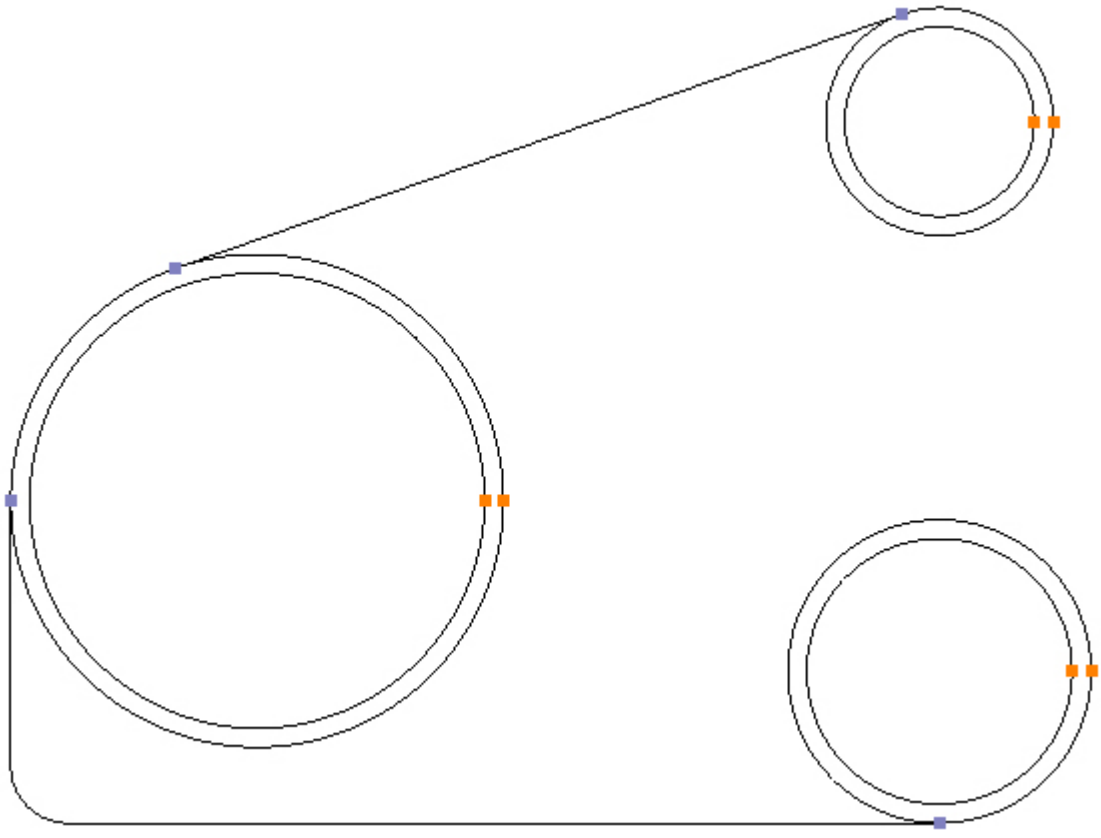
Move the cursor near to the expected start point of the line but on the outside of the circle, the cursor is showing the Tangent symbol. Left click the mouse to accept the Snap Point and create the start of the line:



Move the cursor over the Ø2.4 circle to the expected position of the tangent point, keep moving until the Tangent symbol is shown next to the cursor:



Left mouse click to create the end point of the line tangentially between the two circles:



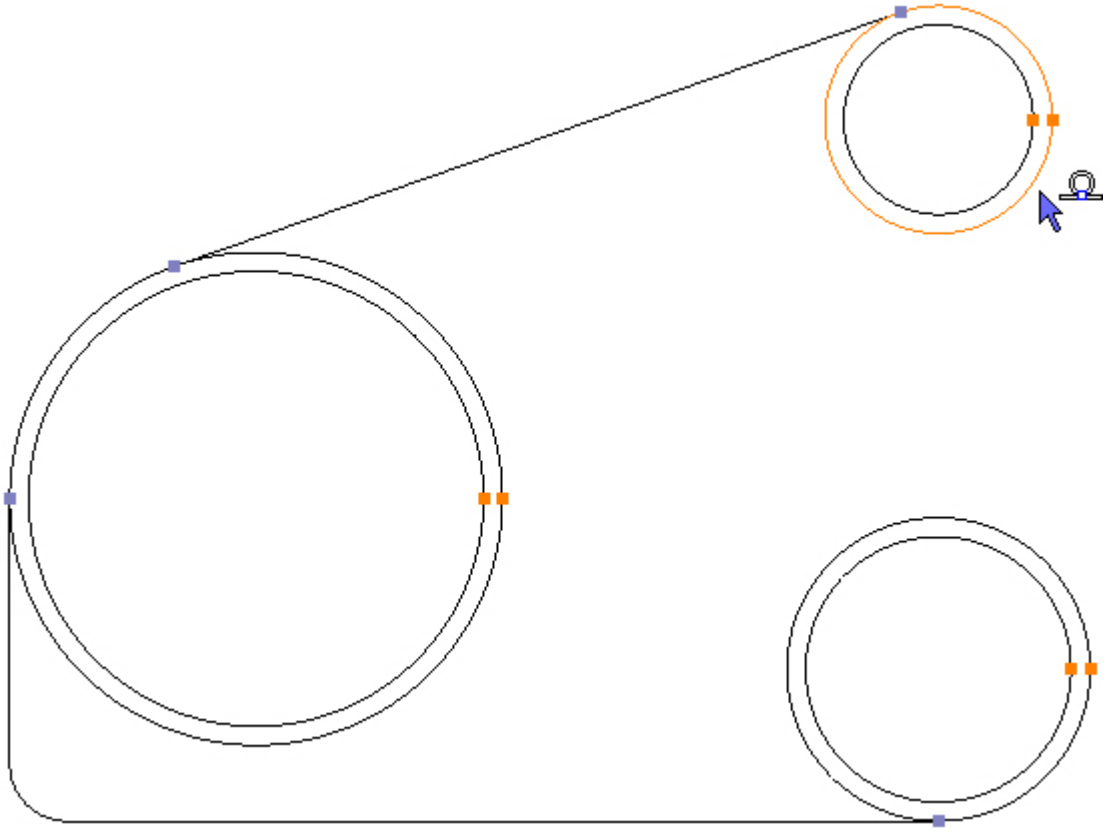
Step 7 - Create line

Create the 60° line tangent to the Ø2.4 circle using Line

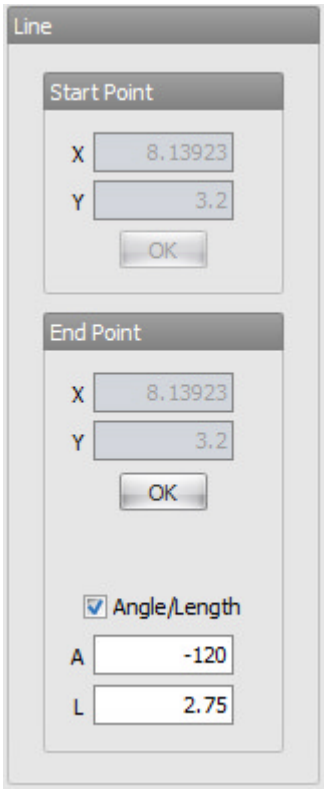
Menu: Draw -> Line

Toolbar button: 

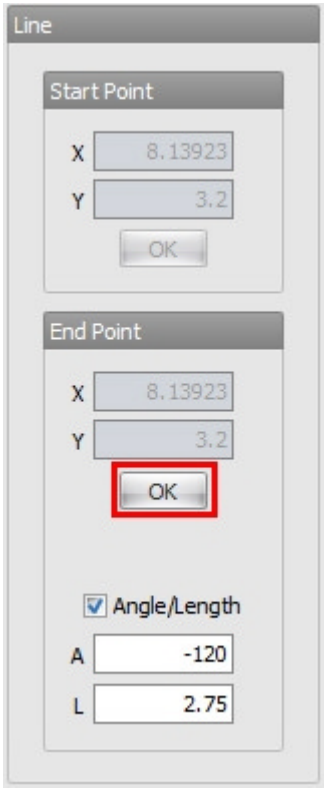
Move the cursor near to the expected start point of the line but on the outside of the circle, the cursor is showing the Tangent symbol. Left click the mouse to accept the Snap Point and create the start of the line:



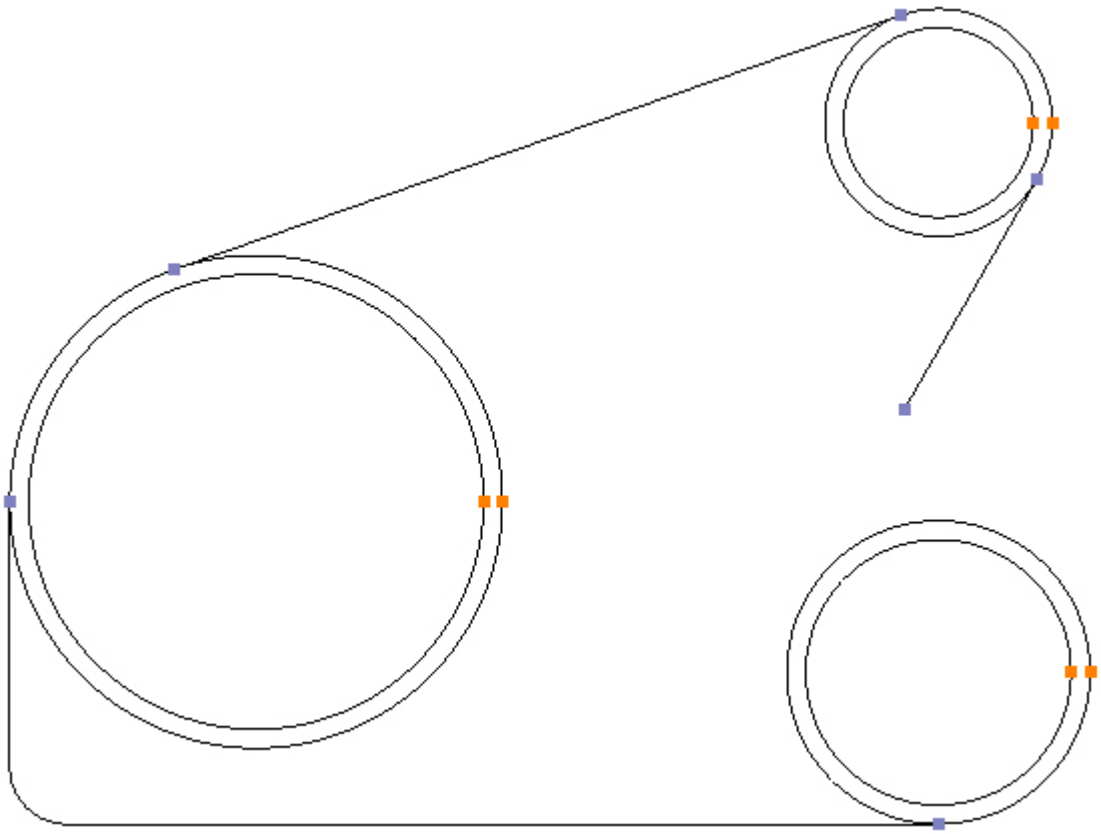
To create the end point of the line specify an angle and length. Check the Angle/Length check box and enter -120 into the Angle box and 2.75 in to the length box:



Click OK to create the end point of the line:



The Part should now look like this:



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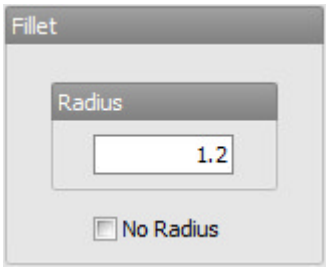
Step 8 - Create fillet

Create the R1.2 radius using Fillet

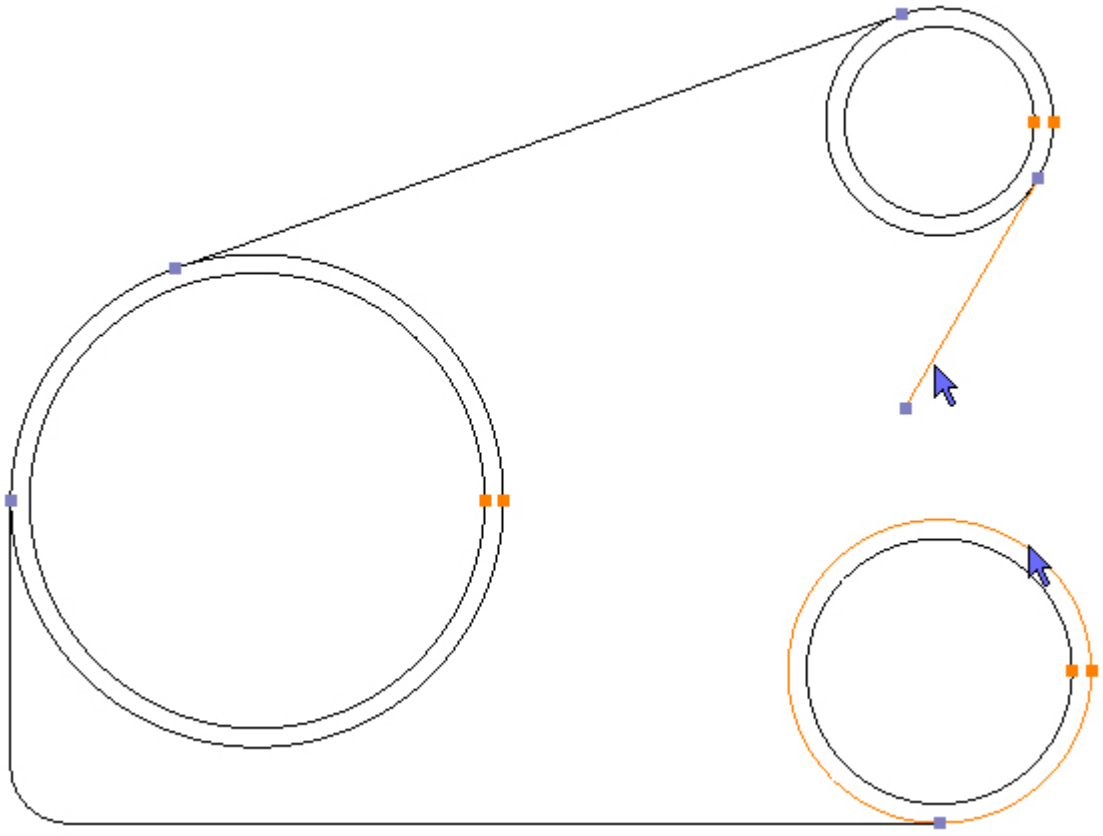
Menu: Modify -> Fillet

Toolbar button: 

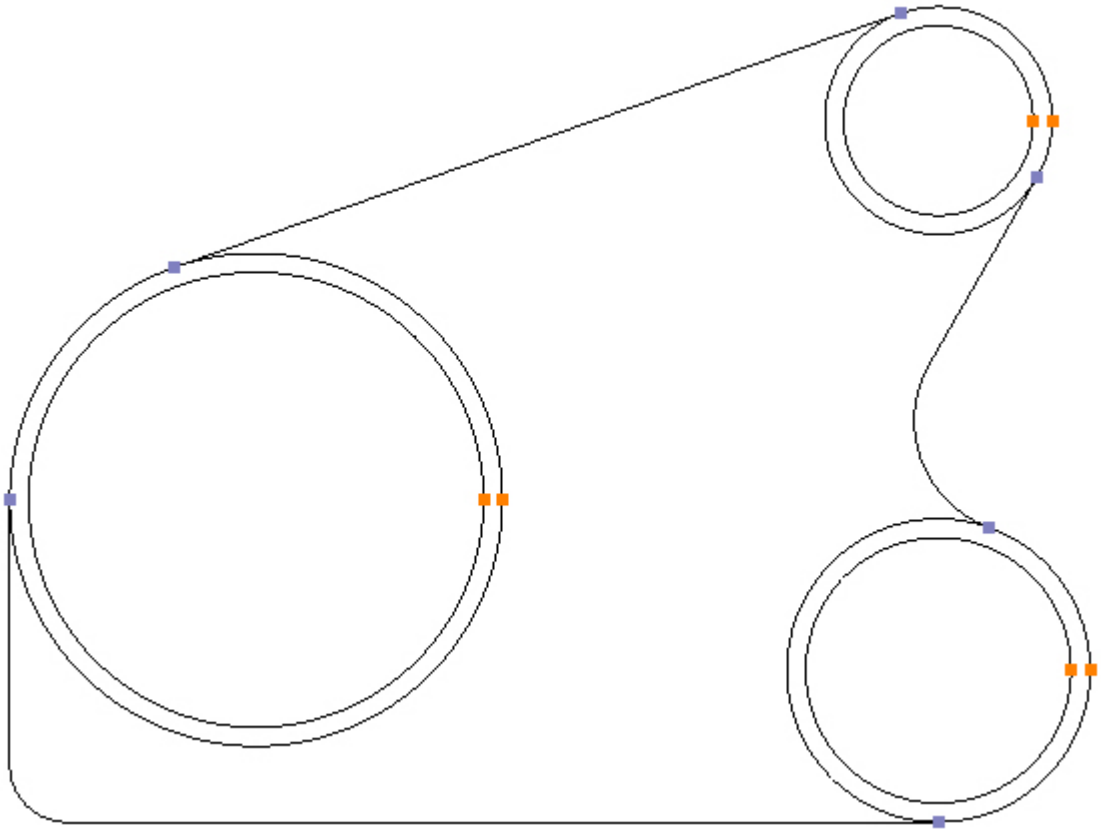
Choose the command and enter 1.2 in the radius box:



Highlight and left mouse click the Ø3.15 circle and the 60° line:



The fillet is created:



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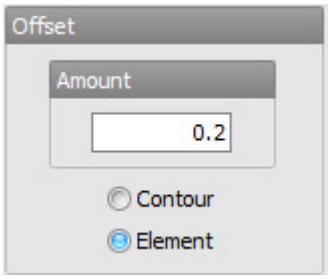
Step 9 - Create offsets

Create 0.2 offsets for internal pocket using Offset

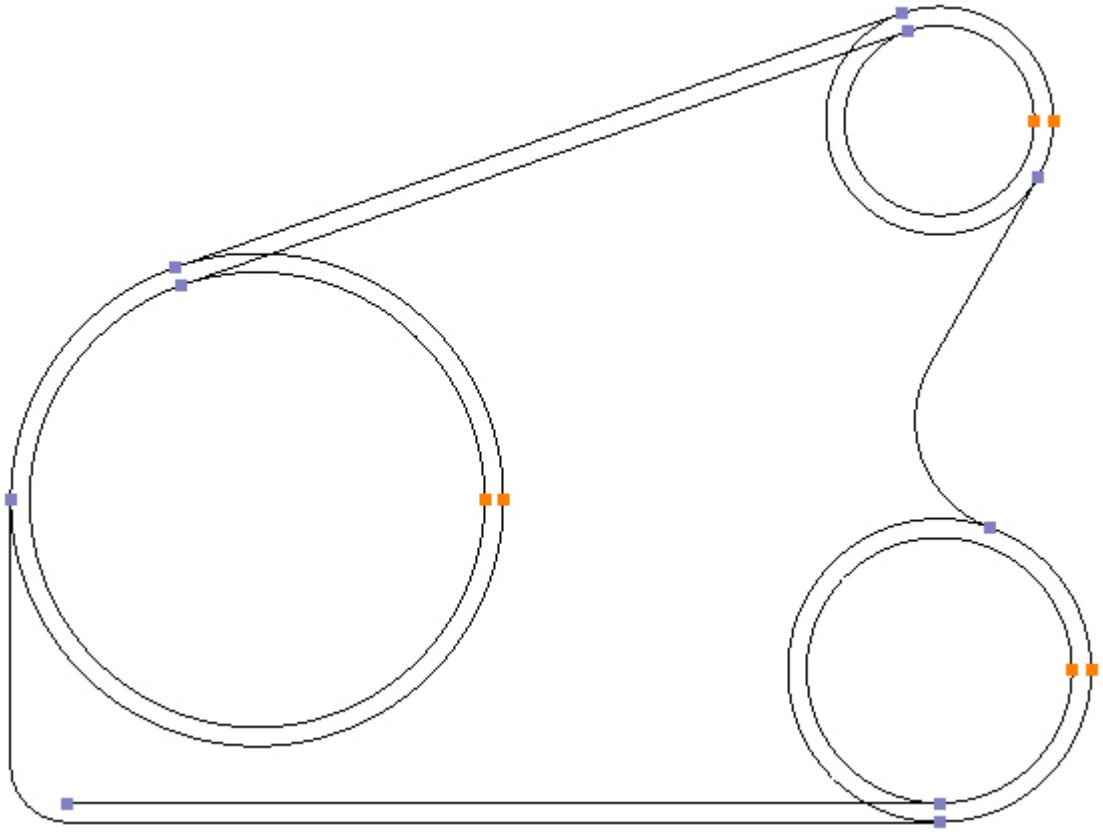
Menu: Modify -> Offset

Toolbar button: 

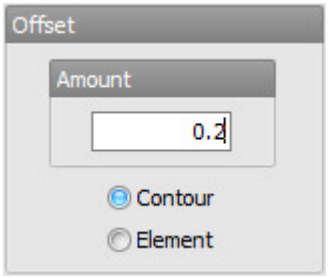
Choose the command and enter 0.2 into the Amount box, select the *Element* radio button:



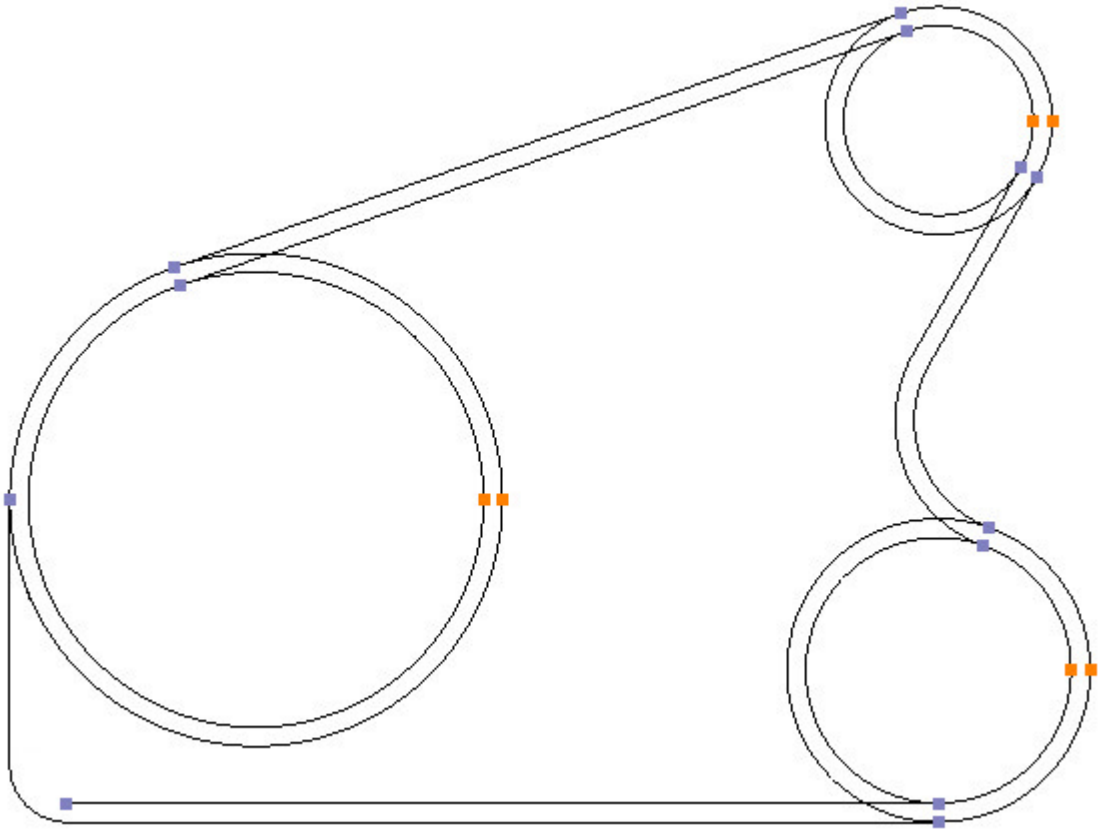
Move the cursor over the line at the bottom edge and left mouse click to indicate that the offset is to be from this line, then left click above the line to create the offset on the inside of the gear housing. Repeat for the line at the top edge to create an offset on the inside as well:



Select the Contour radio button:



Create an offset on the inside from the Contour formed by the 60° line:



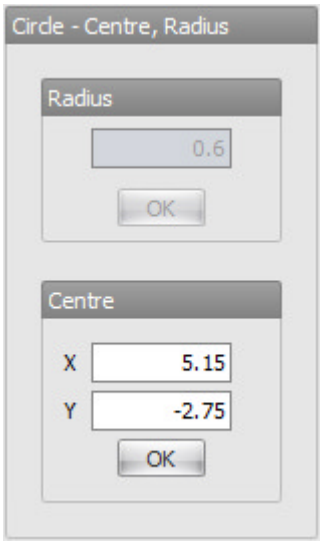
Step 10 - Create two circles

Create two circles using Circle - Centre, Radius

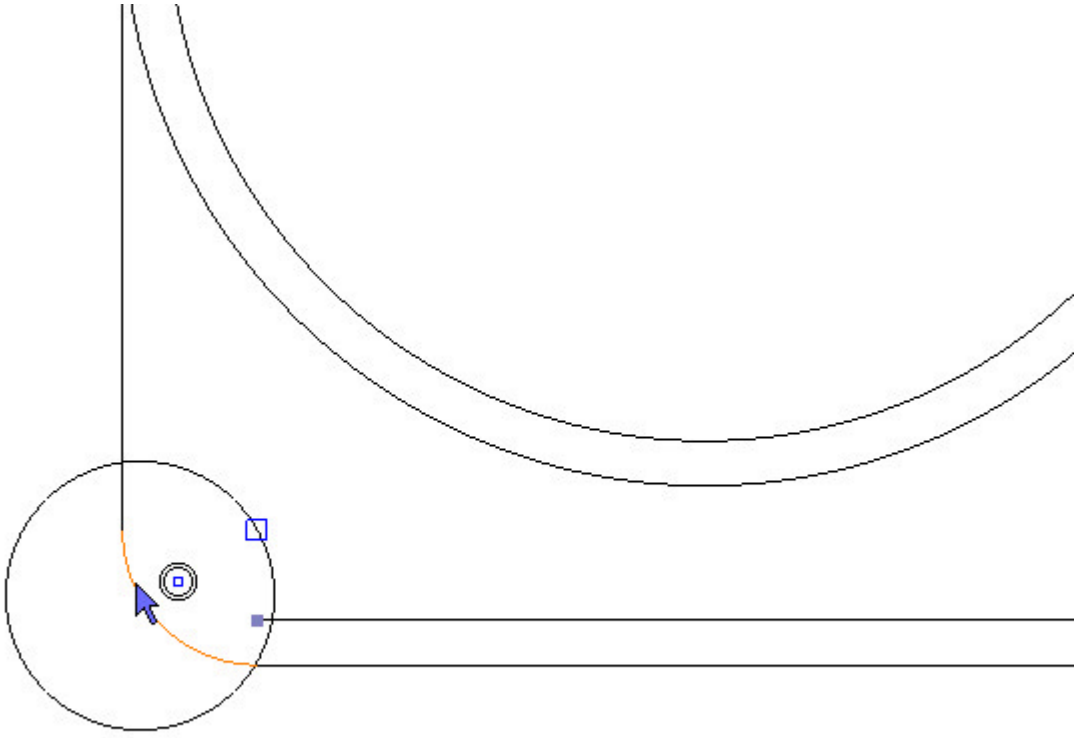
Menu: Draw -> Circle - Centre, Radius

Toolbar button:

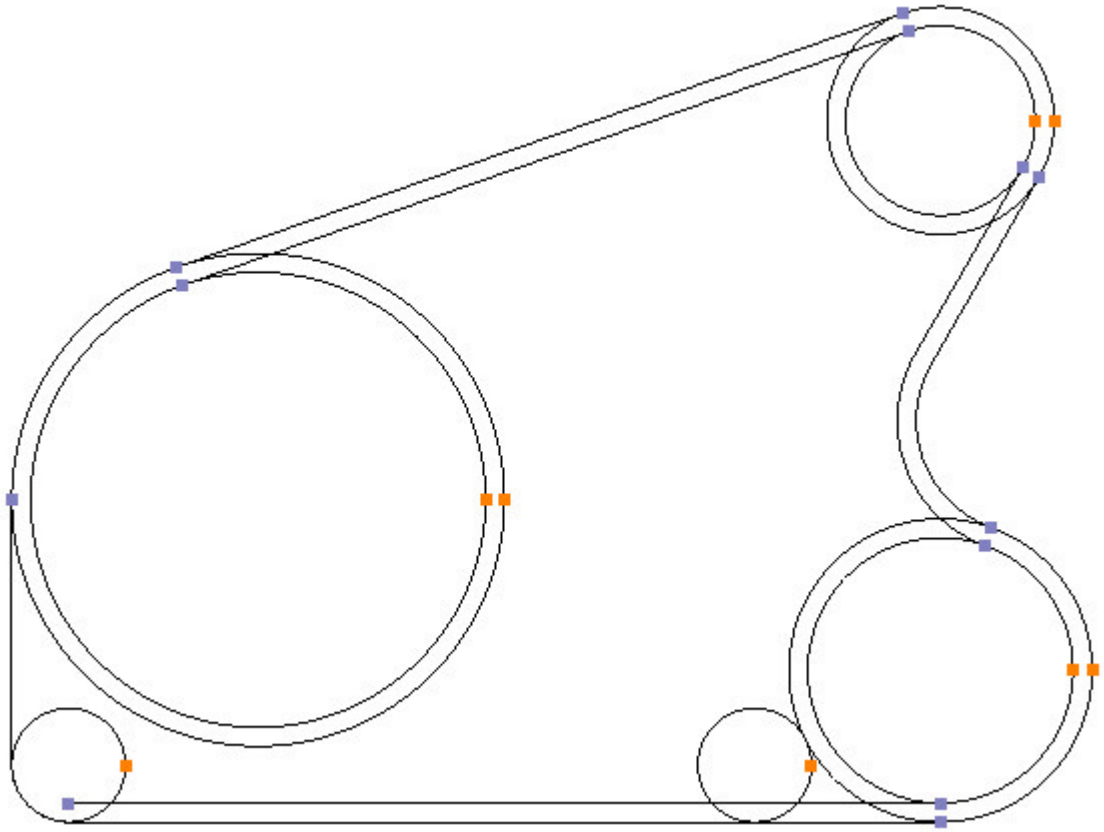
Choose the command and create a 0.6 radius circle, centred at X5.15, Y-2.75:



Create a second circle of the same radius but using snap to centre. Move the cursor over the 0.6 radius, not near the end or mid point, until the Center snap symbol is showing and a blue Snap Marker indicates the Snap Point. It may be necessary to zoom in on the 0.6 radius by pointing the cursor at it and rotating the middle mouse button:



Left click the mouse to accept the Snap Point and create the circle:



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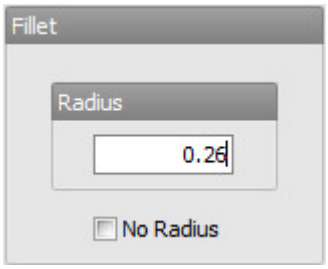
Step 11 - Create fillets

Create 8 off R0.26 Radii using Fillet

Menu: Modify -> Fillet

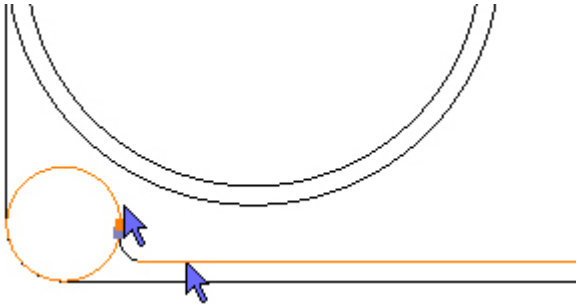
Toolbar button:

Choose the command and enter 0.26 in the radius box:

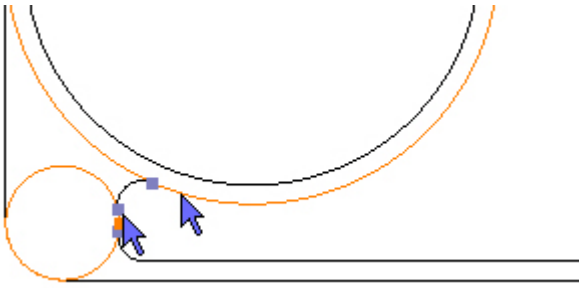


Fillet in 8 places, highlight and left mouse click the two elements that form the fillet:

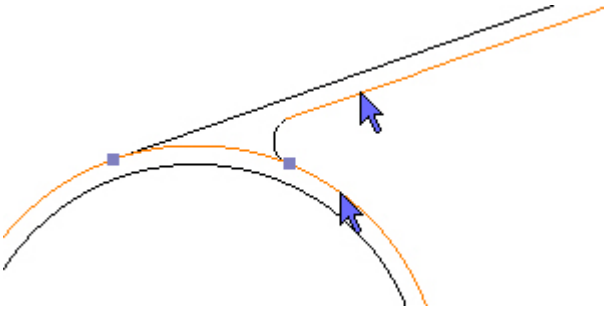
1.



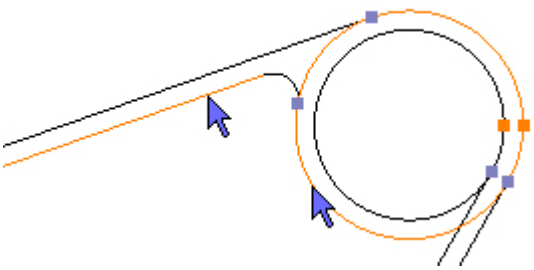
2.



3.



4.



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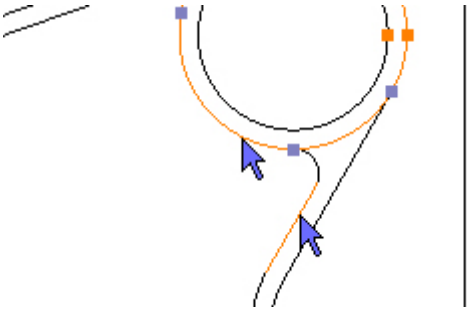
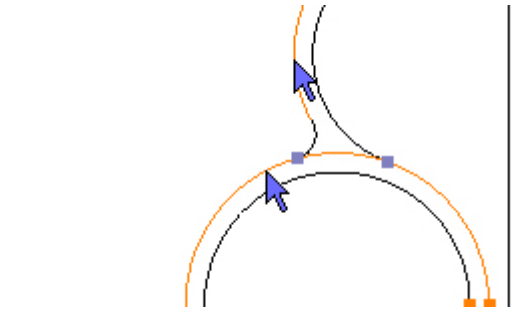
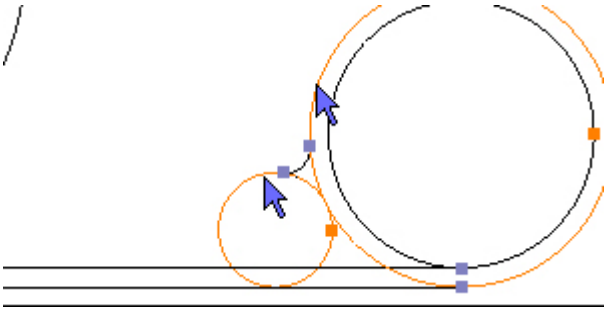


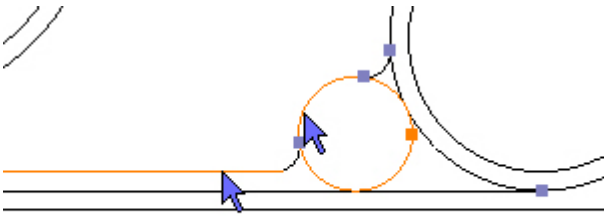
6.



7.



8.



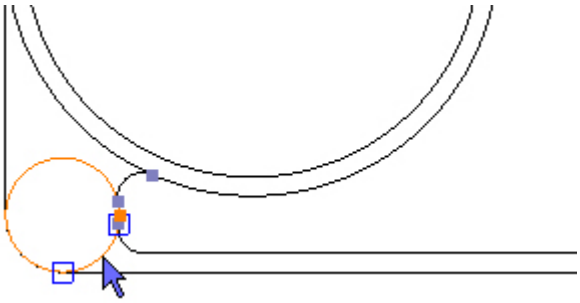
Step 12 - Trim contours

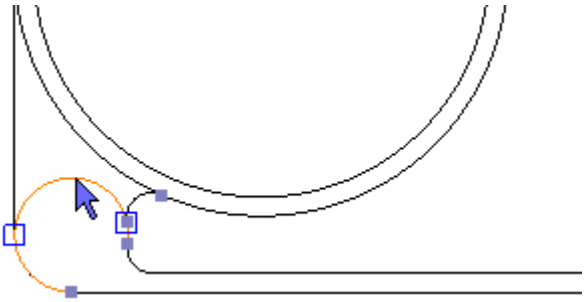
Trim unwanted sections of Contours using Trim

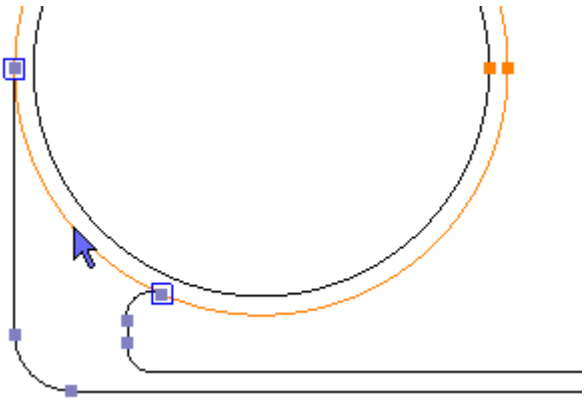
Menu: Modify -> Trim

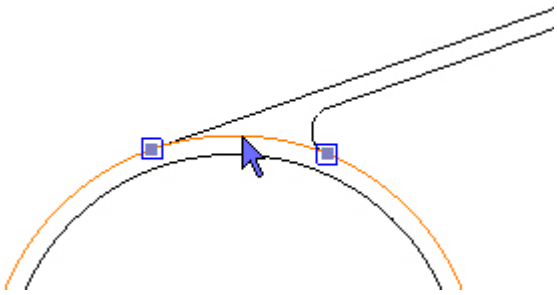
Toolbar button: 

Choose the command and Trim in 10 places, highlight the section of unwanted contours and left mouse click:

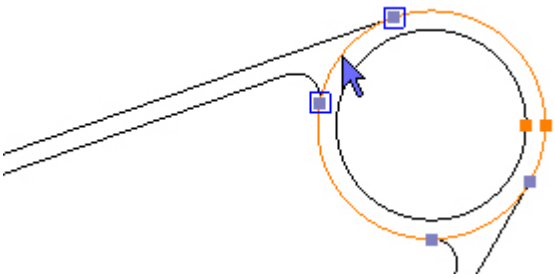
1. 

2. 

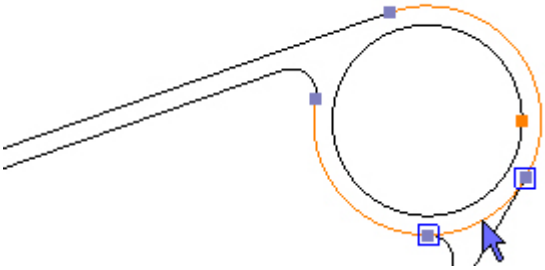
3. 

4. 

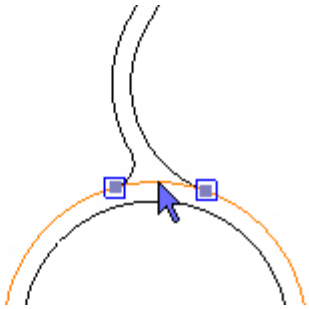
5.



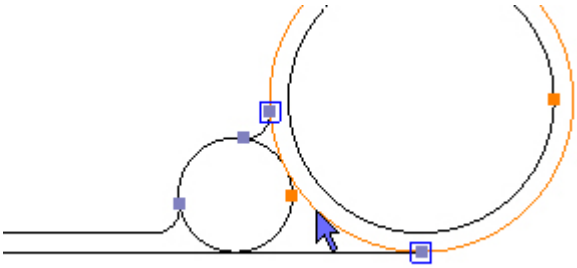
6.



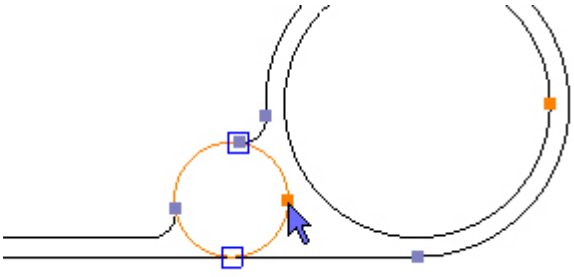
7.



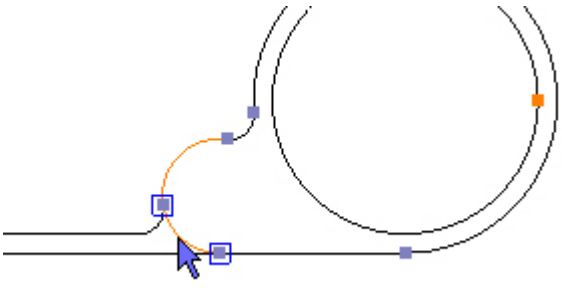
8.



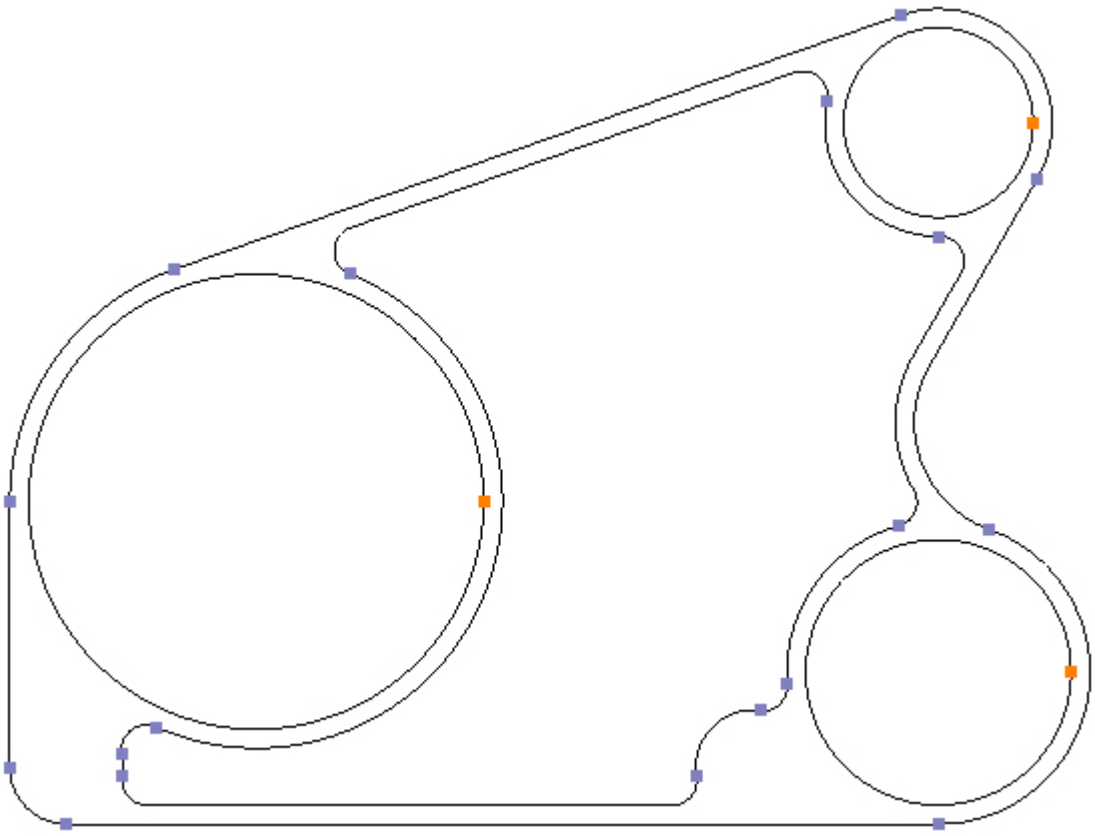
9.



10.



The Part should now look like this:



Step 13 - Connect contours

Connect Contours using Connect

Menu: Modify -> Connect

Toolbar button: 

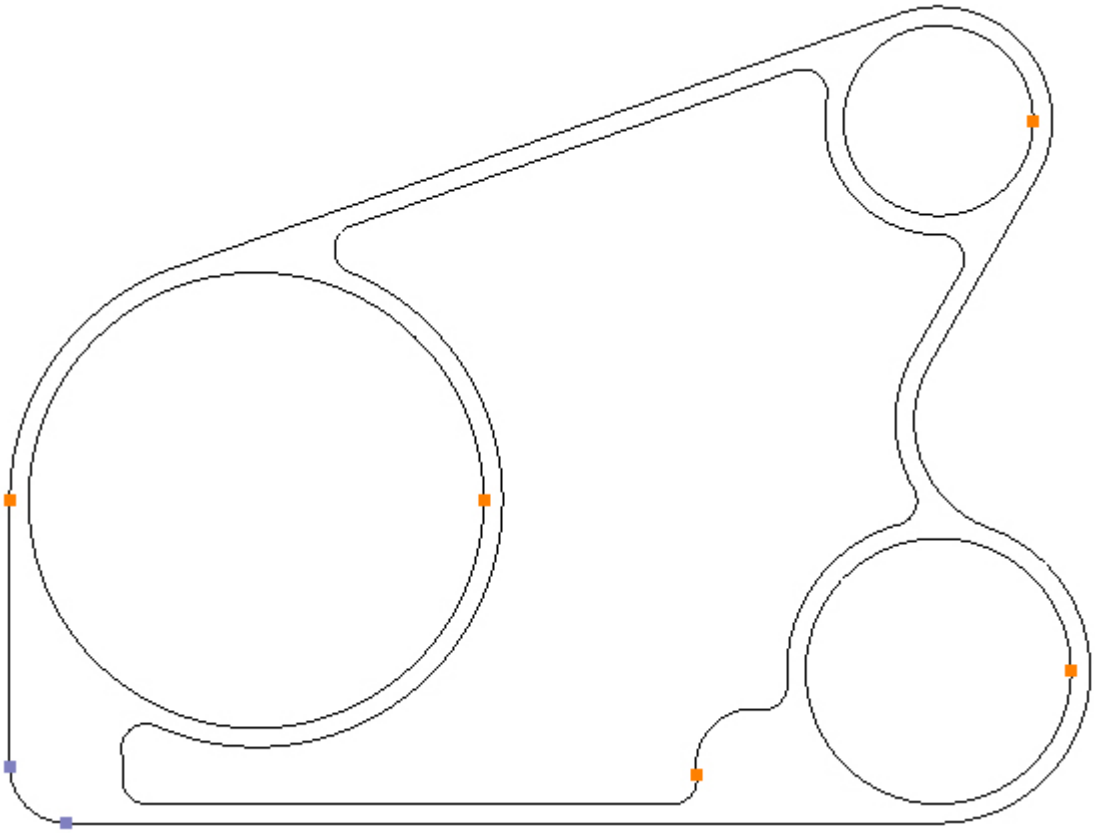
When the Connect command is chosen, it connects all selected Contours. It does not matter if a Contour is selected that does not need to be connected, as long as it cannot be connected. Therefore use the Select All command to select the Contours:

Menu: Edit -> Select All

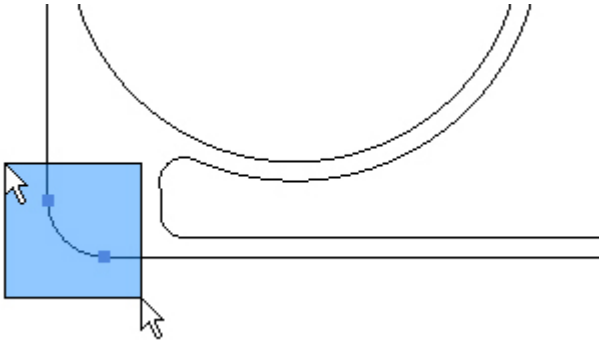
Toolbar button: 

Shortcut: Ctrl + A

Now all the Contours are selected choose the Connect command, the part should now look like this:



All the Contours are now connected and are closed, as indicated by the orange connection markers. There is still an open Contour in the bottom left hand corner, this is left over from the trimming and needs to be deleted.
Select the Contour. It may not be possible to select the Contour by directly clicking it, the outside Contour may be selected instead, in this case use a selection rectangle.
Right click over the Part to show the context menu and choose 'Select Mode '. Click away from any Contours and drag the rectangle over the open contour:

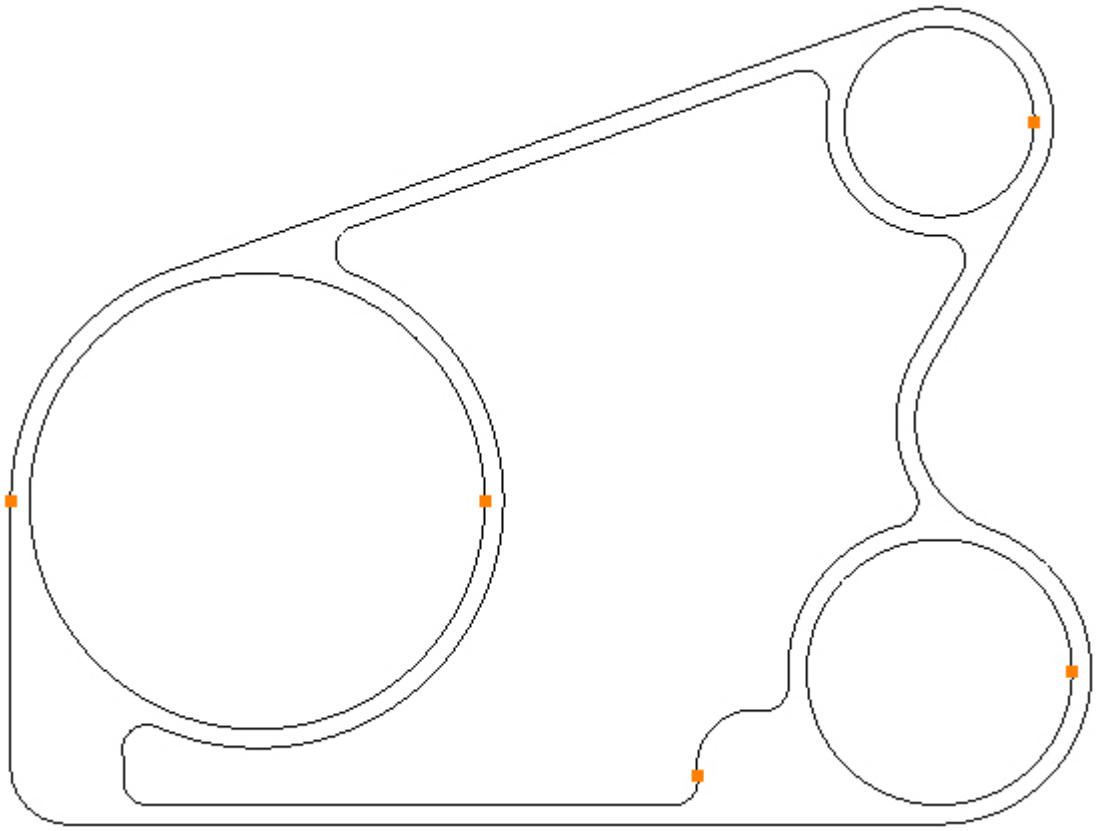


Now the Contour is selected use the Delete command to delete it:

Menu: Edit -> Delete

Toolbar button: 

The Part should now look like this:



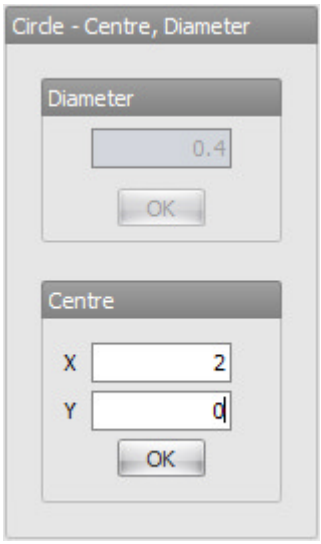
Step 14 - Create circle

Create a circle using Circle - Centre, Diameter

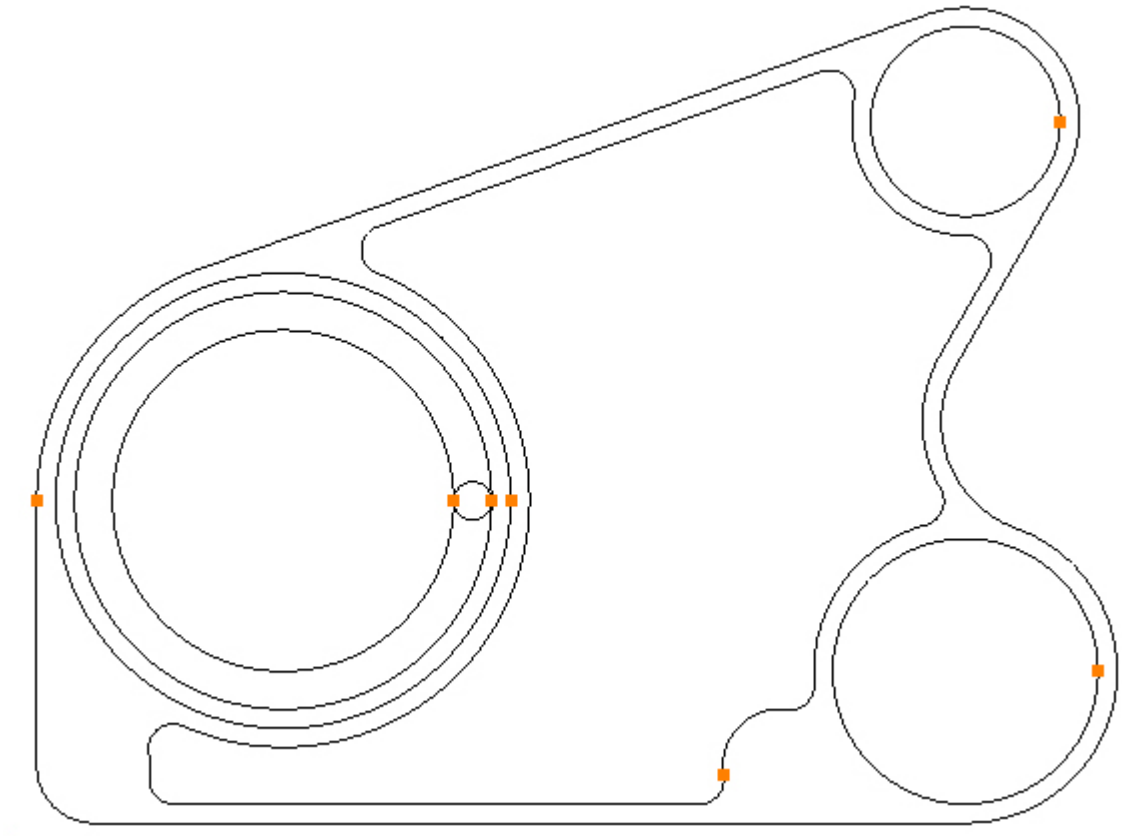
Menu: Draw -> Circle - Centre, Diameter

Toolbar button:

Choose the command and create a Ø0.4 circle, centred at X2.0, Y0:



Create another two circles of Ø3.6 and Ø4.4, centred at X0, Y0, the Part should now look like this:



Gear Housing Tutorial

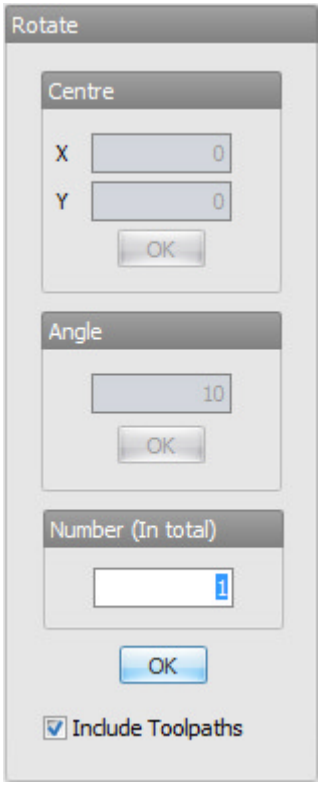
Step 15 - Rotate circle

Rotate the Ø0.4 circle using Rotate

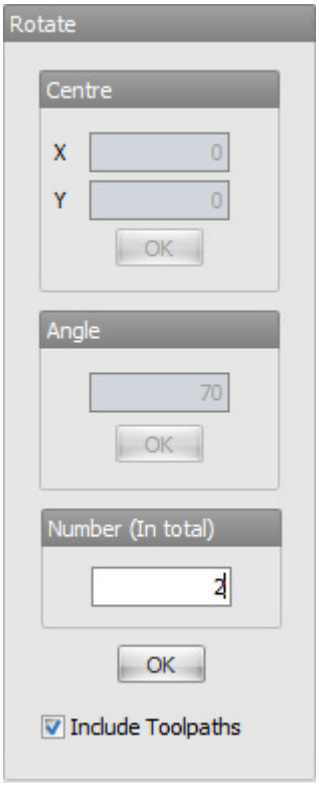
Menu: Modify -> Rotate

Toolbar button: 

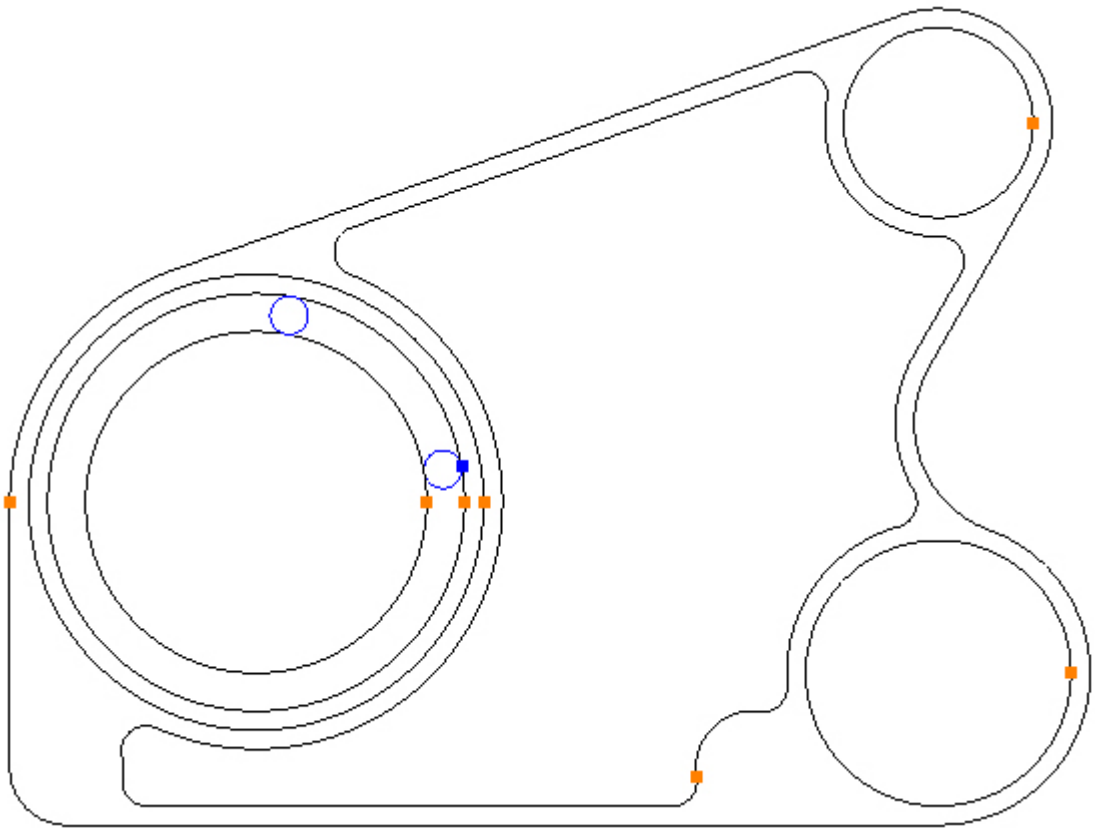
First select the Ø0.4 circle, right click over the Part to show the context menu and choose 'Select Mode' and directly click the circle. Then choose the Rotate command and rotate around a centre of X0, Y0 by 10°, once:



Leaving the circle selected rotate again by 70°, but keeping the original circle by specifying 2 for the *Number* box:



The Part should now look like this:



Step 16 - Trim contours

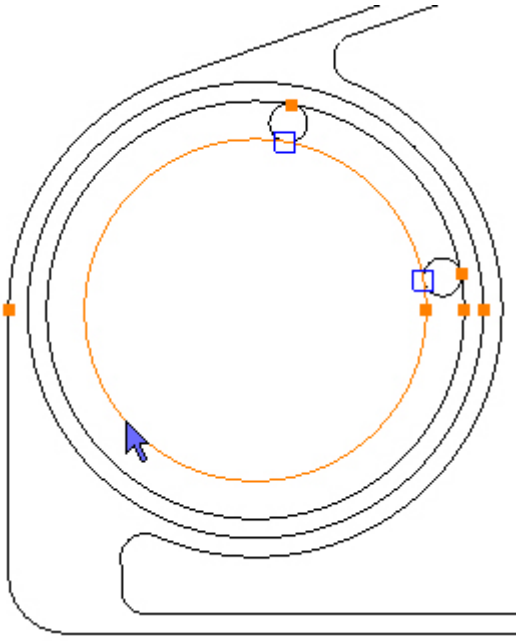
Trim unwanted section of Contours using Trim

Menu: Modify -> Trim

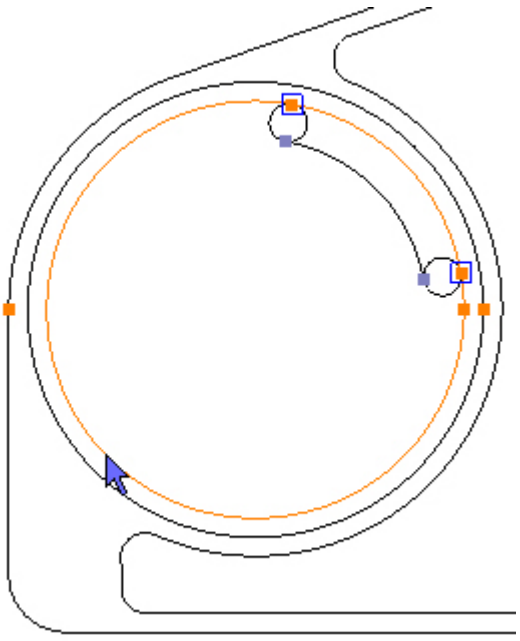
Toolbar button: 

Choose the command and Trim in 4 places, highlight the sections of unwanted contours and left mouse click:

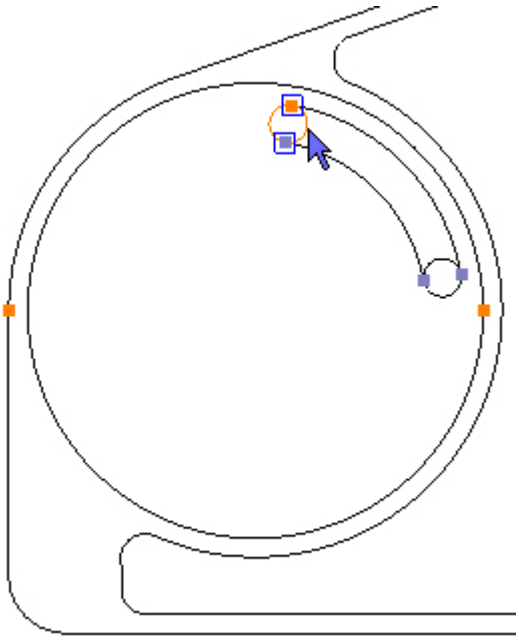
1.



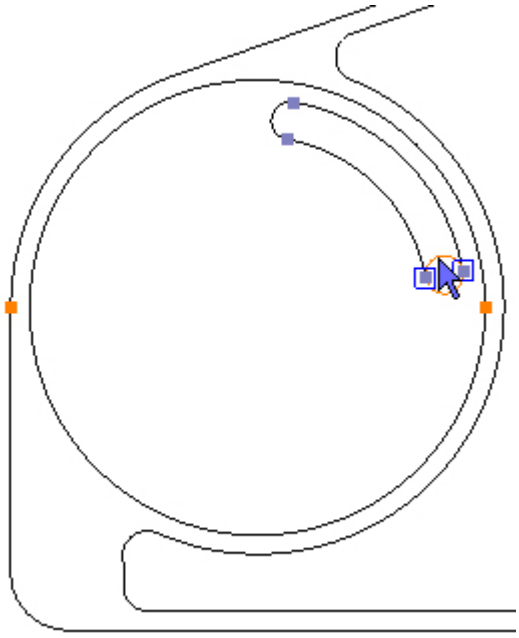
2.



3.

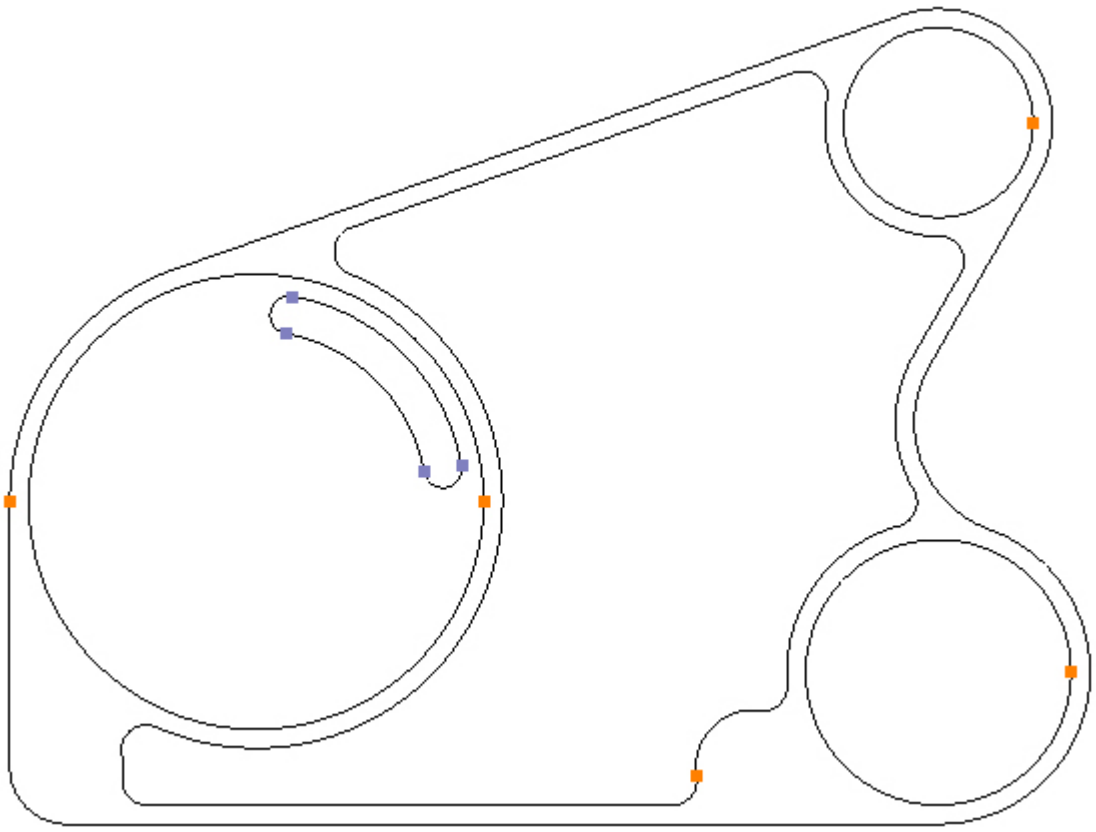


4.



The Part should now look like this:

Gear Housing Tutorial



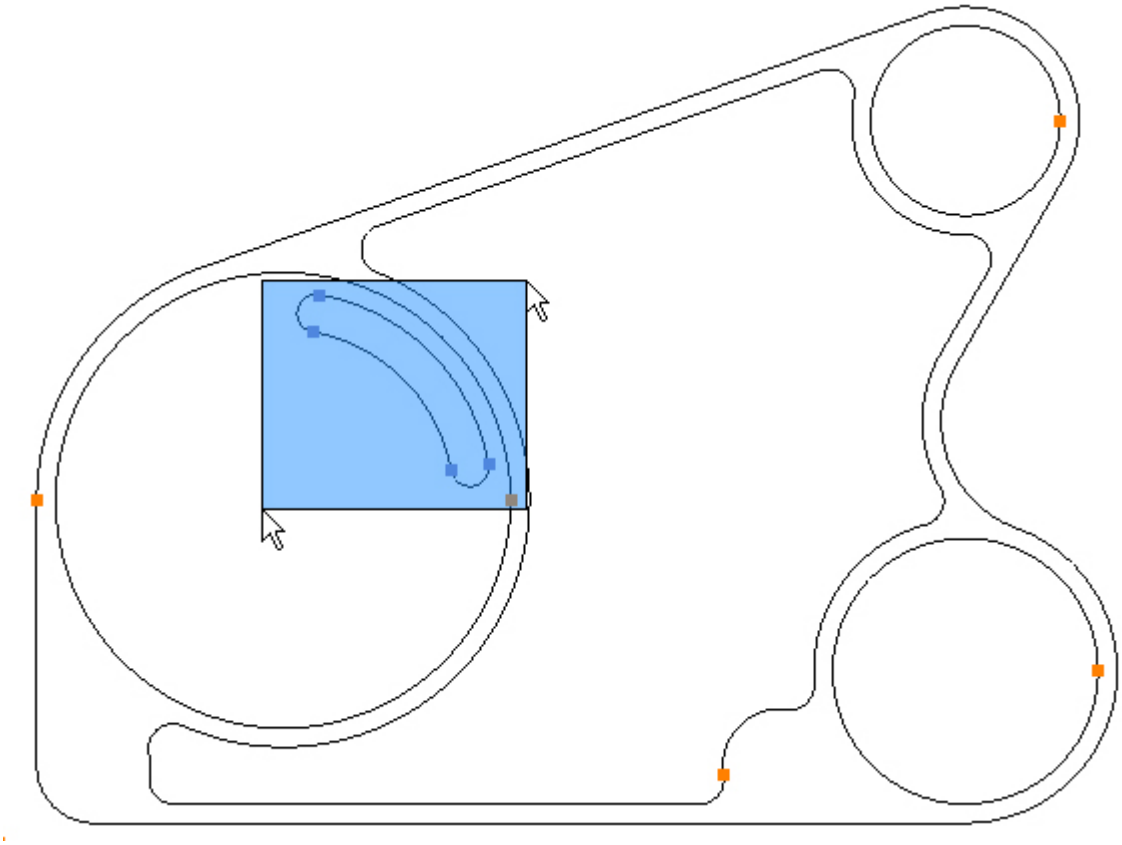
Step 17 - Connect contours

Connect slot using Connect

Menu: Modify -> Connect

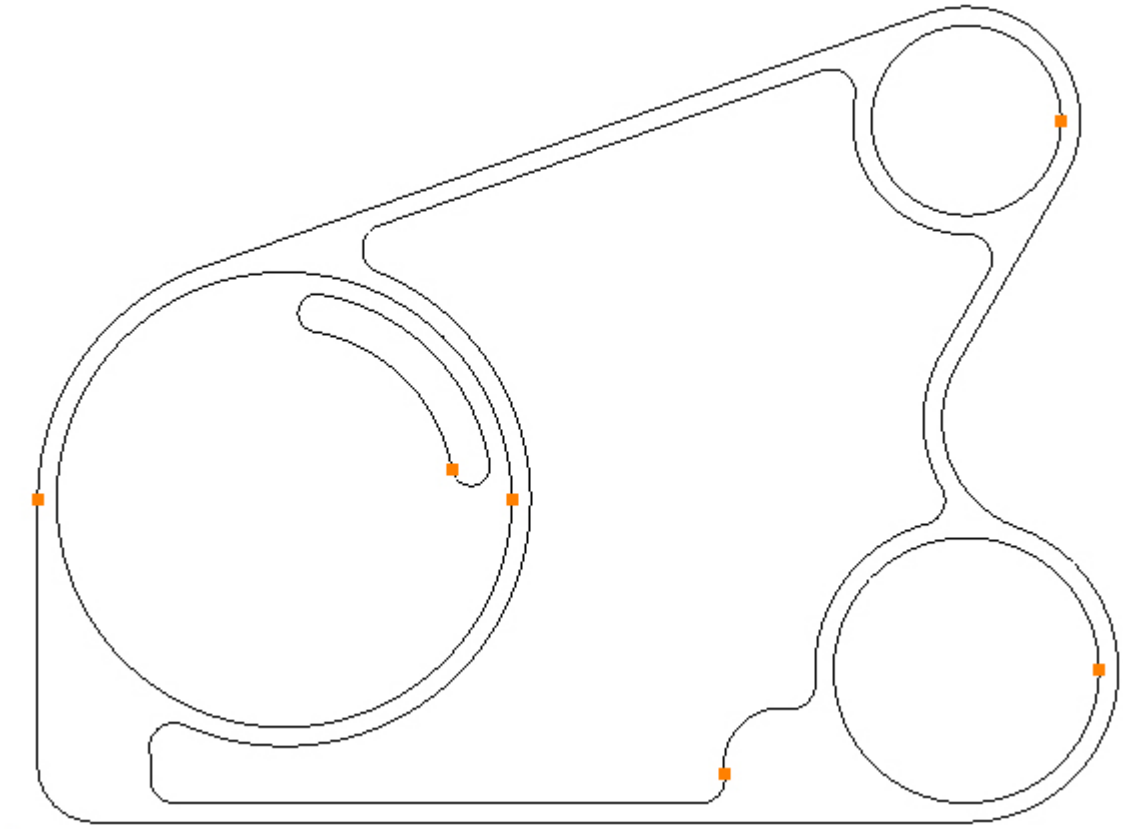
Toolbar button: 

First select the 4 arcs that make up the slot using a selection rectangle. Right click over the Part to show the context menu and choose 'Select Mode'. Click away from any Contours and drag the rectangle over the open contour:



Choose the Connect command, the Part should now look like this:

Gear Housing Tutorial



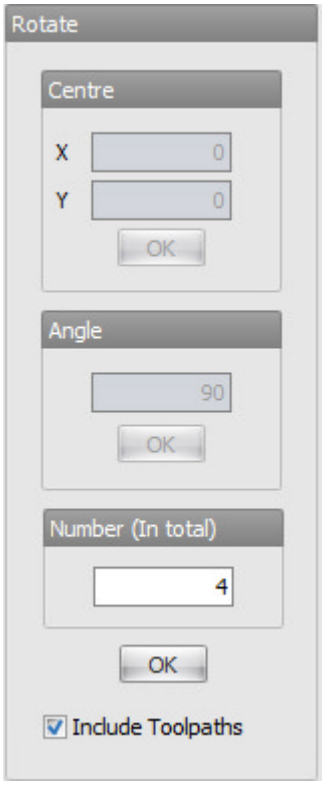
Step 18 - Rotate contour

Rotate the slot using Rotate

Menu: Modify -> Rotate

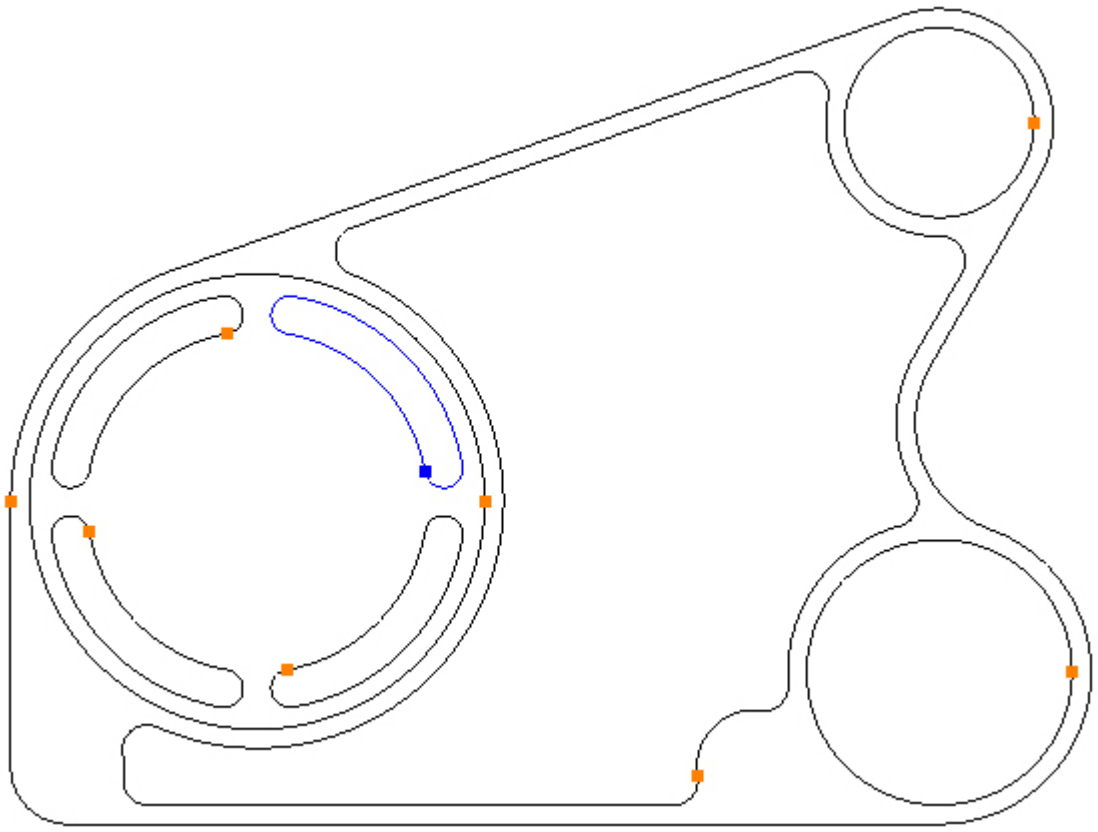
Toolbar button: 

First select the slot, right click over the Part to show the context menu and choose 'Select Mode' and directly click the slot. Then choose the Rotate command and rotate around a centre of X0, Y0 by 90°, in 4 places:



The Part should now look like this:

Gear Housing Tutorial



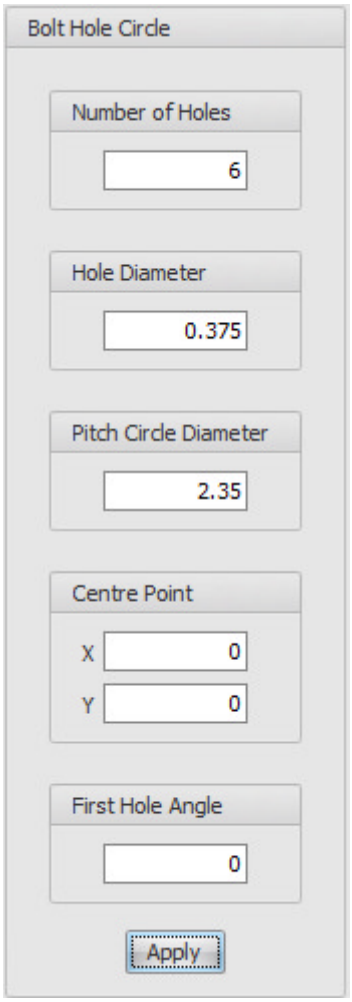
Step 19 - Create circles

Create circle using Bolt Hole Circle

Menu: Draw -> Bolt Hole Circle

Toolbar button:

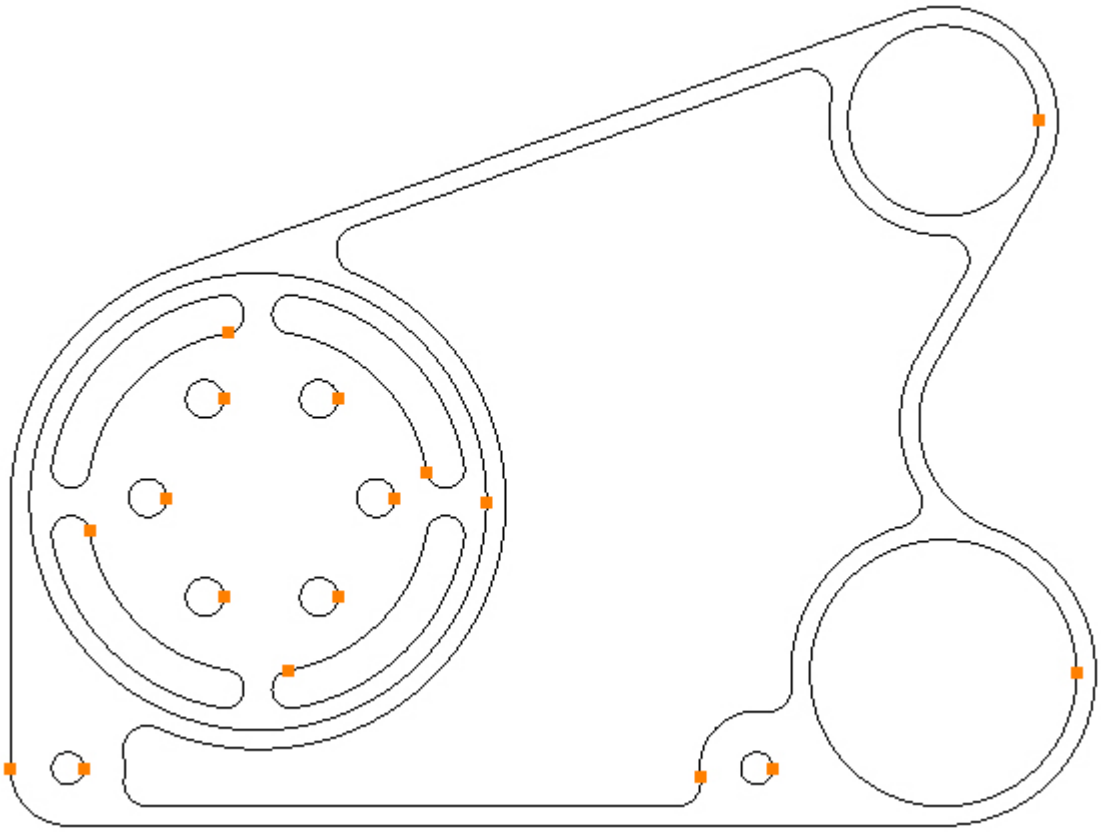
Choose the command and enter the following values:



Click the Apply button to action the command.

The Part should now look like this:

Gear Housing Tutorial



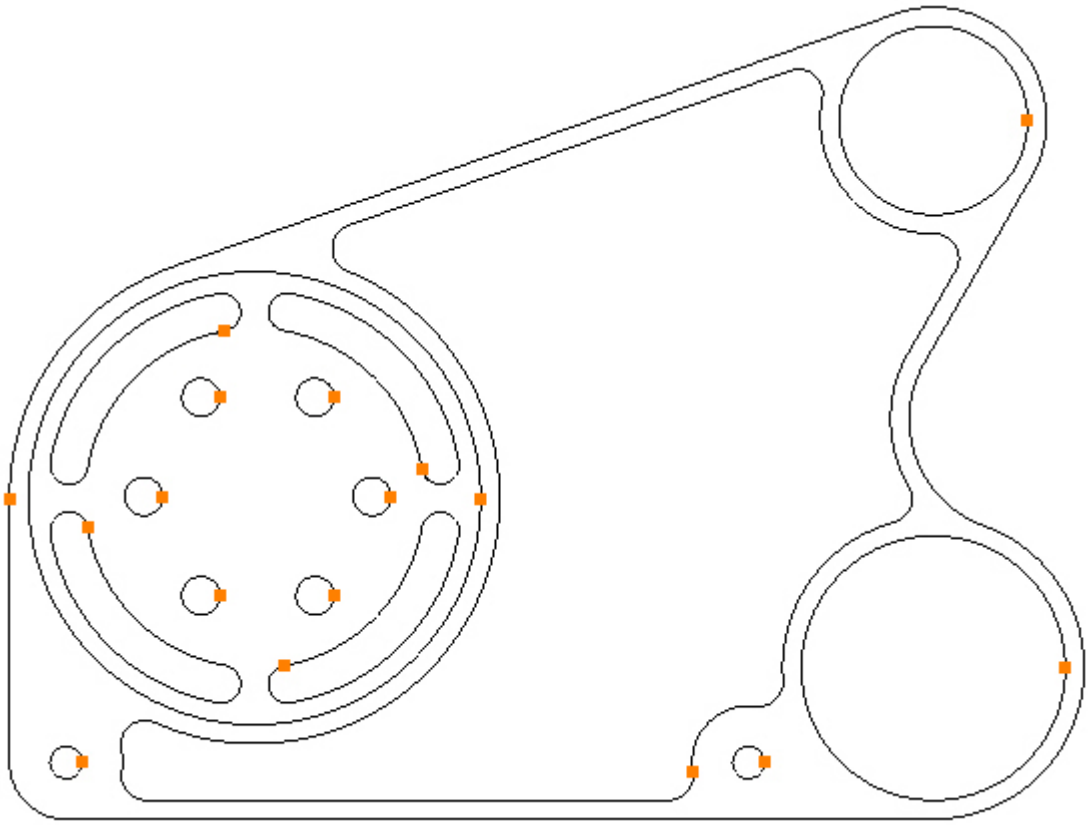
Step 20 - Create circles

Create two circles using Circle - Centre, Diameter

Menu: Draw -> Circle - Centre, Diameter

Toolbar button:

Choose the command and create two Ø0.3125 circles, centred at X5.15, Y-2.75 and X-1.95, Y-2.75:



The creation of the Contours is now complete.

Gear Housing Tutorial

Create the Toolpaths

Step 1 - Process plan

The Gear Housing is to be made from an aluminium rectangular billet 11.7" x 8.75" x 1.2". The billet is to be held in a machine vice, leaving 1.08" protruding above the top of the jaws. The Gear Housing is machined completley, leaving an operation to remove the excess material that it was held with. This operation will not be covered during the tutorial.
For the purpose of this tutorial assume that the top of the billet is at Z0 and does not require facing.

Before starting, a process plan is required:

Operation Description	Operation Type	Tool	Comments
Rough outside profile	Pocket	T01, 0.75inch End Mill	0.02 finish allowance
Rough Ø4.7 counterbore	Pocket	T01, 0.75inch End Mill	0.02 finish allowance
Rough Ø2.0 and Ø2.75 bores	Pocket	T01, 0.75inch End Mill	0.02 finish allowance
Rough inside pocket	Pocket	T02, 0.5inch End Mill	0.02 finish allowance
Finish outside profile	Profile	T03, 0.75inch End Mill	
Finish Ø2.0 and Ø2.75 bores	Profile	T03, 0.75inch End Mill	
Finish Ø4.7 counterbore	Pocket	T03, 0.75inch End Mill	
Finish inside pocket	Pocket	T04, 0.5inch End Mill (0.08 Corner Rad)	
Machine radial slots	Profile	T05, 0.3125inch Slot Drill	
Spot Drill holes	Drilling	T06, 0.5inch Spot Drill	
Drill Ø0.3125 holes	Drilling	T07, 0.3125inch Drill	
Drill Ø0.375 holes	Drilling	T08, 0.375inch Drill	

Gear Housing Tutorial

Step 2 - Set start point

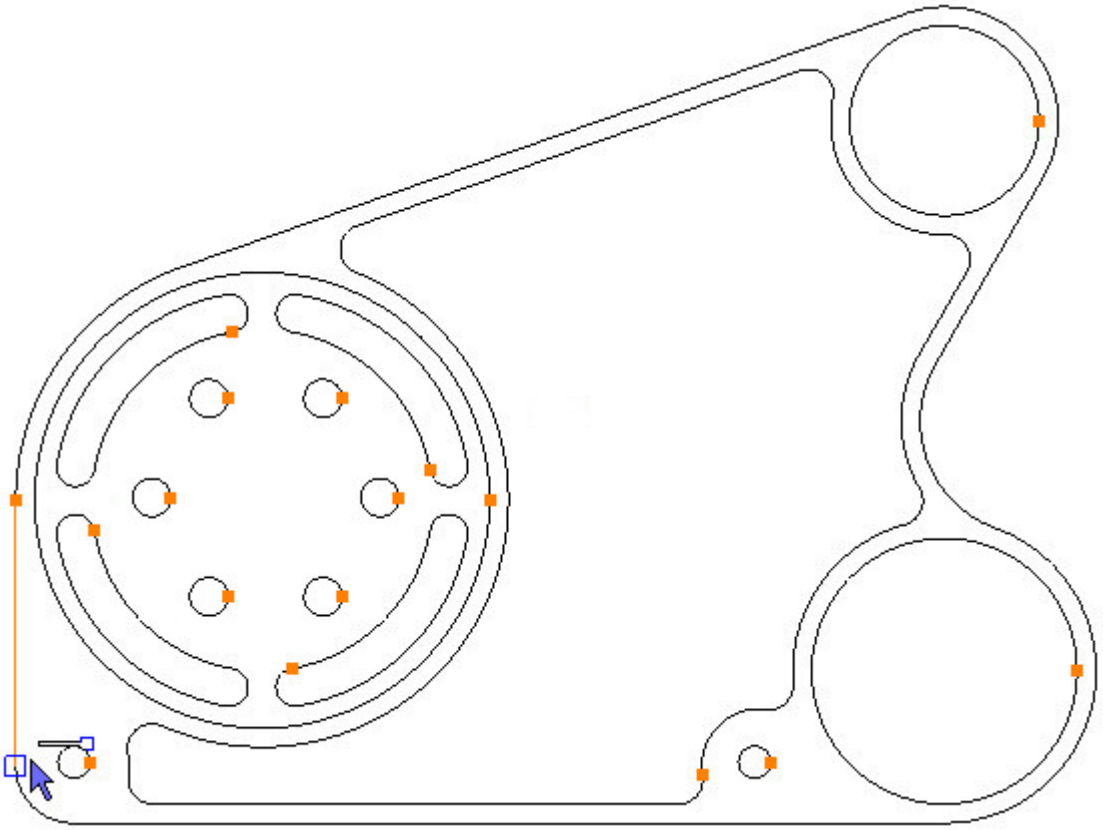
Set start point on closed Contours using Set Start Point

Before creating any operations, it is recommended that the start point on closed contours be set first.

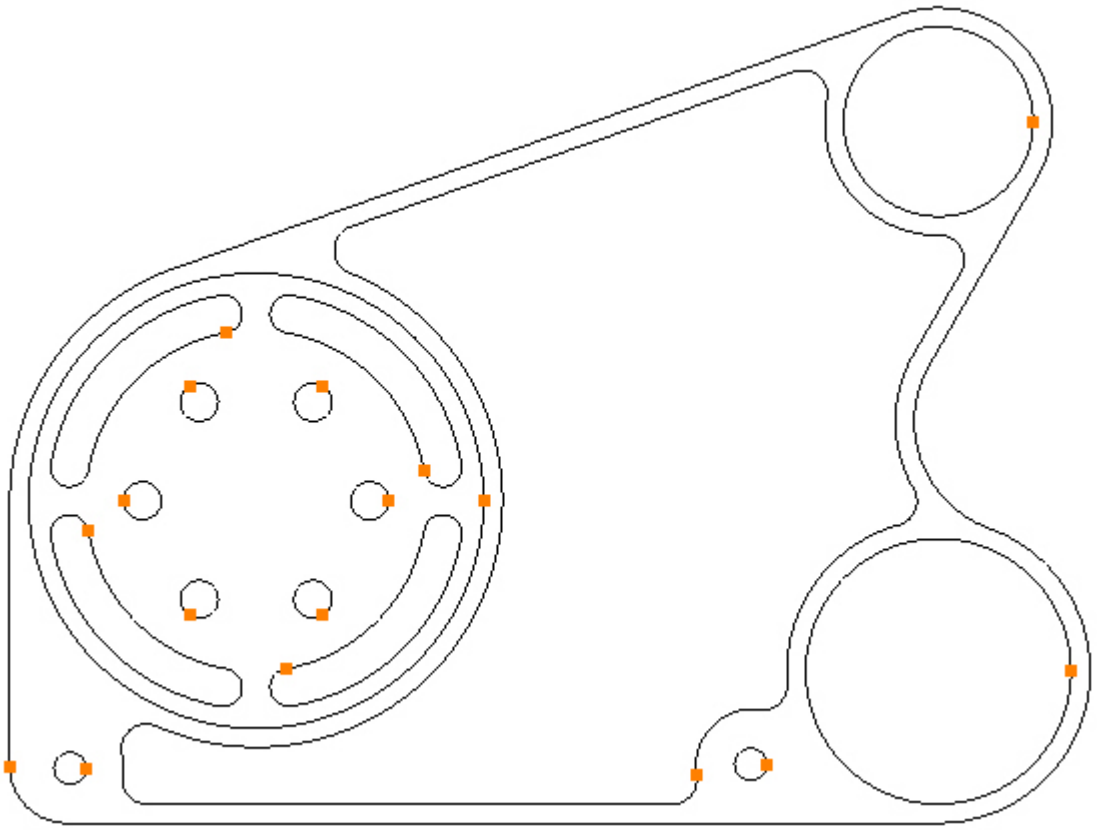
Menu: Modify -> Set Start Point

Toolbar button:

Choose the command and move the cursor near to the end of the line on the left:



Left mouse click to set the start point:



Step 3 - Prepare for roughing

Prepare for Roughing Outside Profile

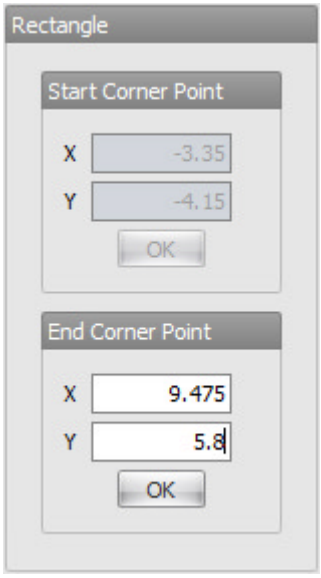
In order to rough the outside using a pocketing operation, a rectangle needs to be created that will contain the toolpaths.

Choose the Rectangle command:

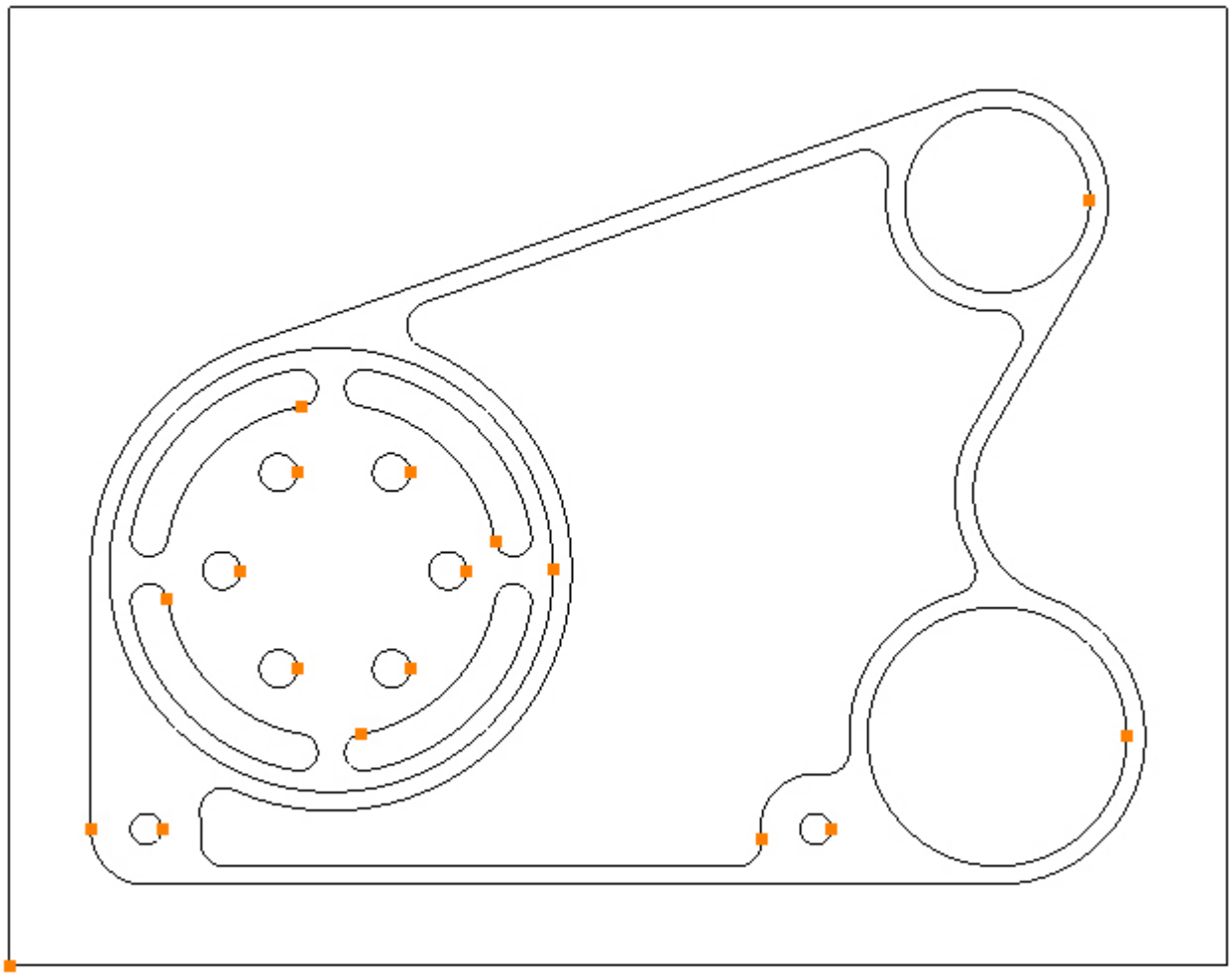
Menu: Draw -> Rectangle

Toolbar button: 

Create a rectangle with a start point value of X-3.35, Y-4.15 and an end point value of X9.475, Y5.8:



The Part should now look like this:



Step 4 - Rough outside profile

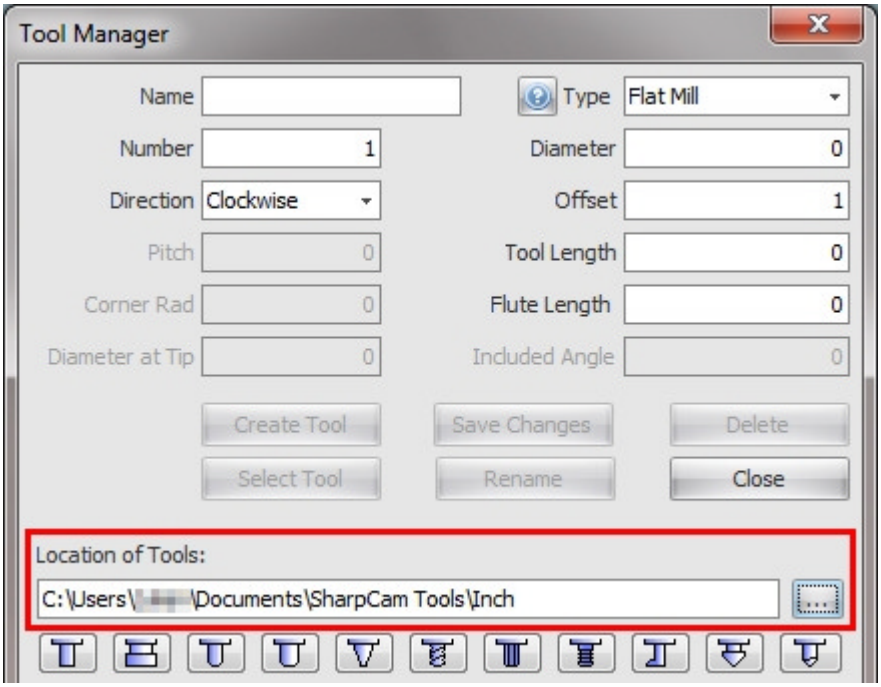
Rough Outside Profile using Pocket

Before choosing the pocket command, first select the cutting tool to be used for the operation. The Tool Manager is used to select a tool:

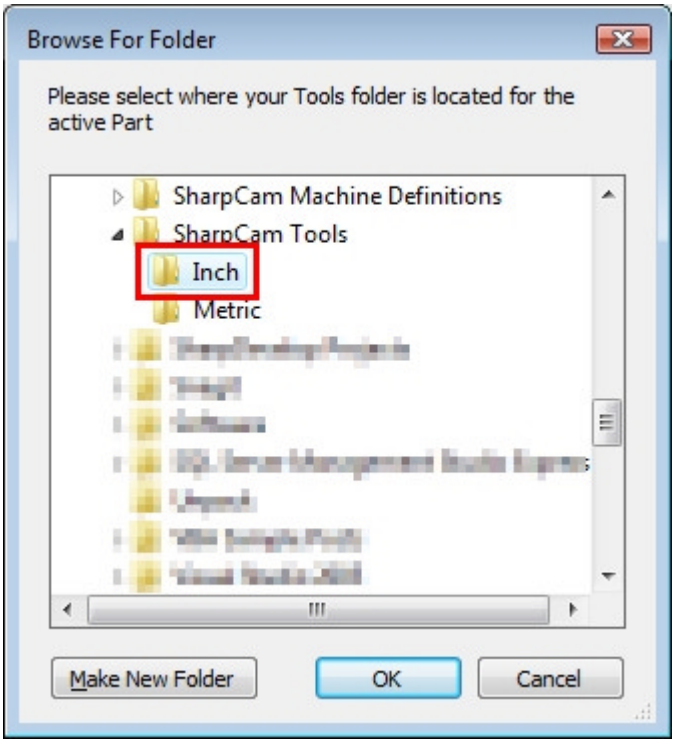
Menu: Machine-> Tool Manger

Toolbar button: 

Before selecting a tool, first check that the Tool Manager is pointing to the correct folder. The folder is indicated by the *Location of Tools box*:



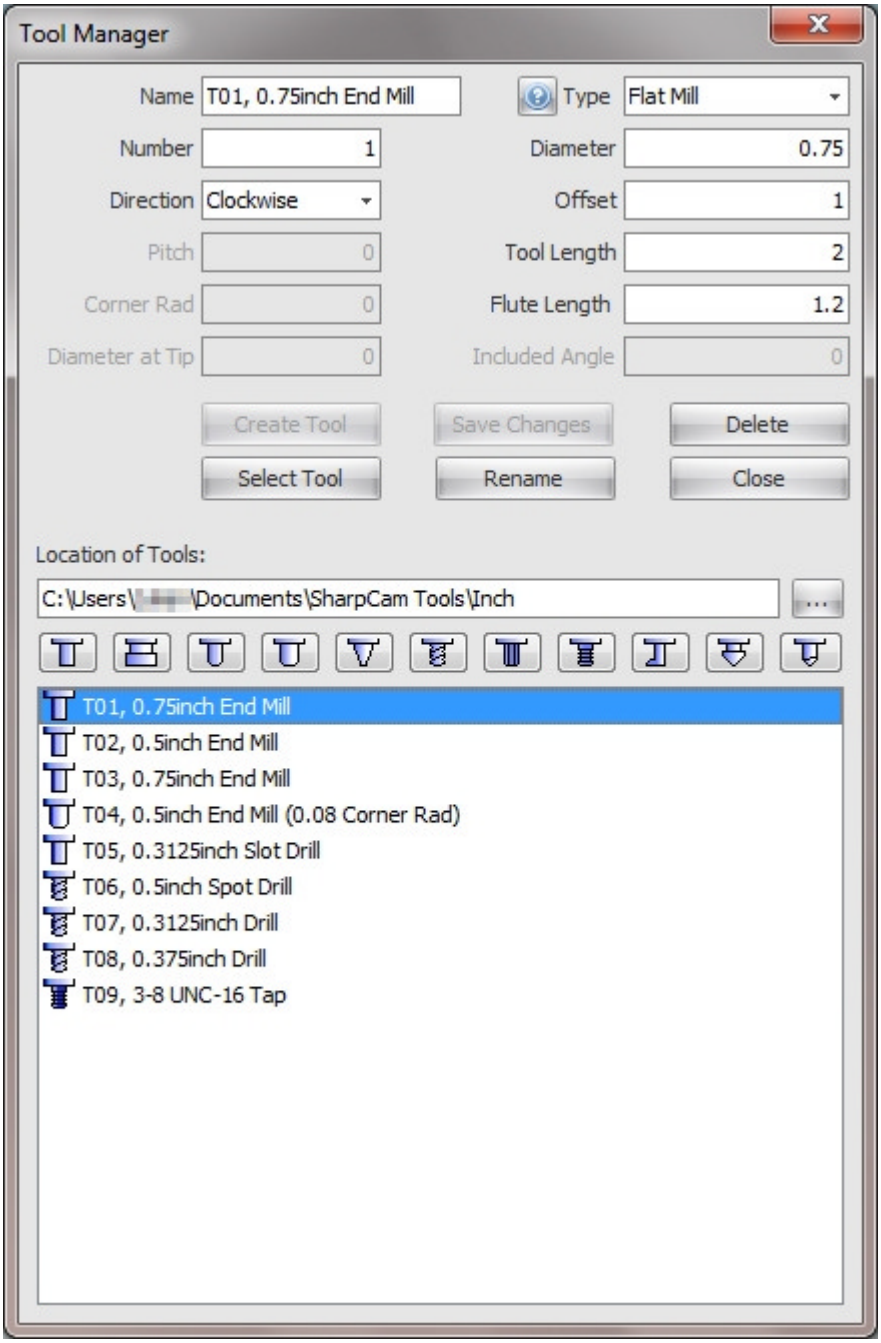
As this is an imperial Part the *Inch* tools folder, that was installed at the same time as SharpCam, should be used. This folder is located in My Documents (Windows XP)/Documents (Vista)/Documents Library (Windows 7), inside a folder called *SharpCam Tools*. If the location is wrong, click the browse button  and navigate to *'SharpCam Tools'* folder and select the *Inch* folder and click OK:



Choose the command to display the Tool Manager, click on 'T01, 0.75inch End Mill', then click the 'Select Tool' button. Alternatively double click a tool to select it.

All the tools required for this tutorial were installed when SharpCam was installed for the first time.

Gear Housing Tutorial



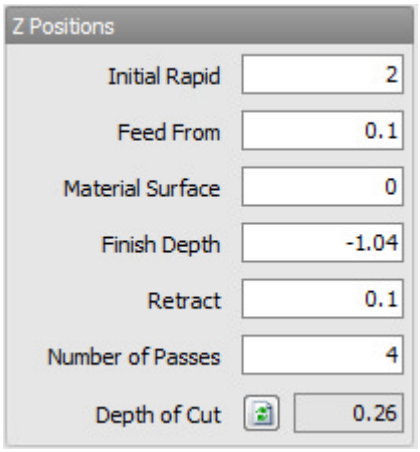
The tool currently selected is displayed in the Status Bar at the bottom:

The outside is now ready to pocket, choose the command:

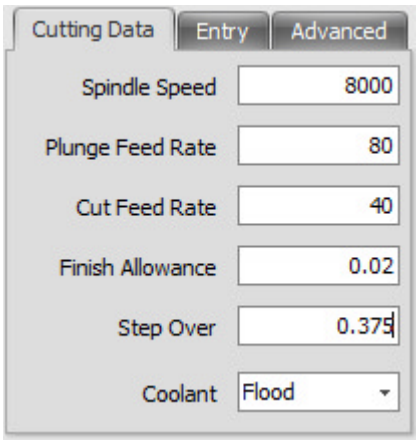
Menu: Machine-> Pocket

Toolbar button:

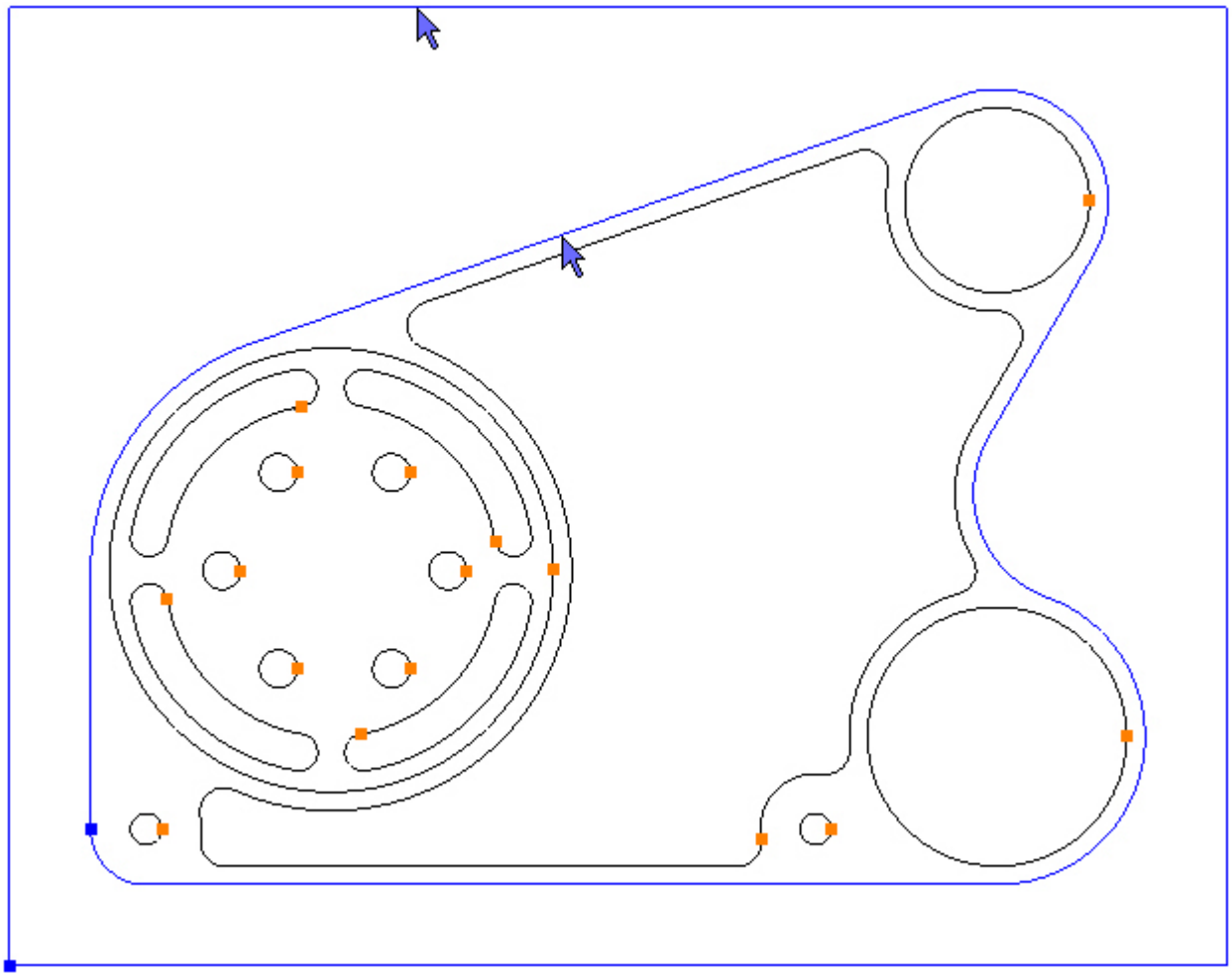
Enter the following values for the Z Positions:



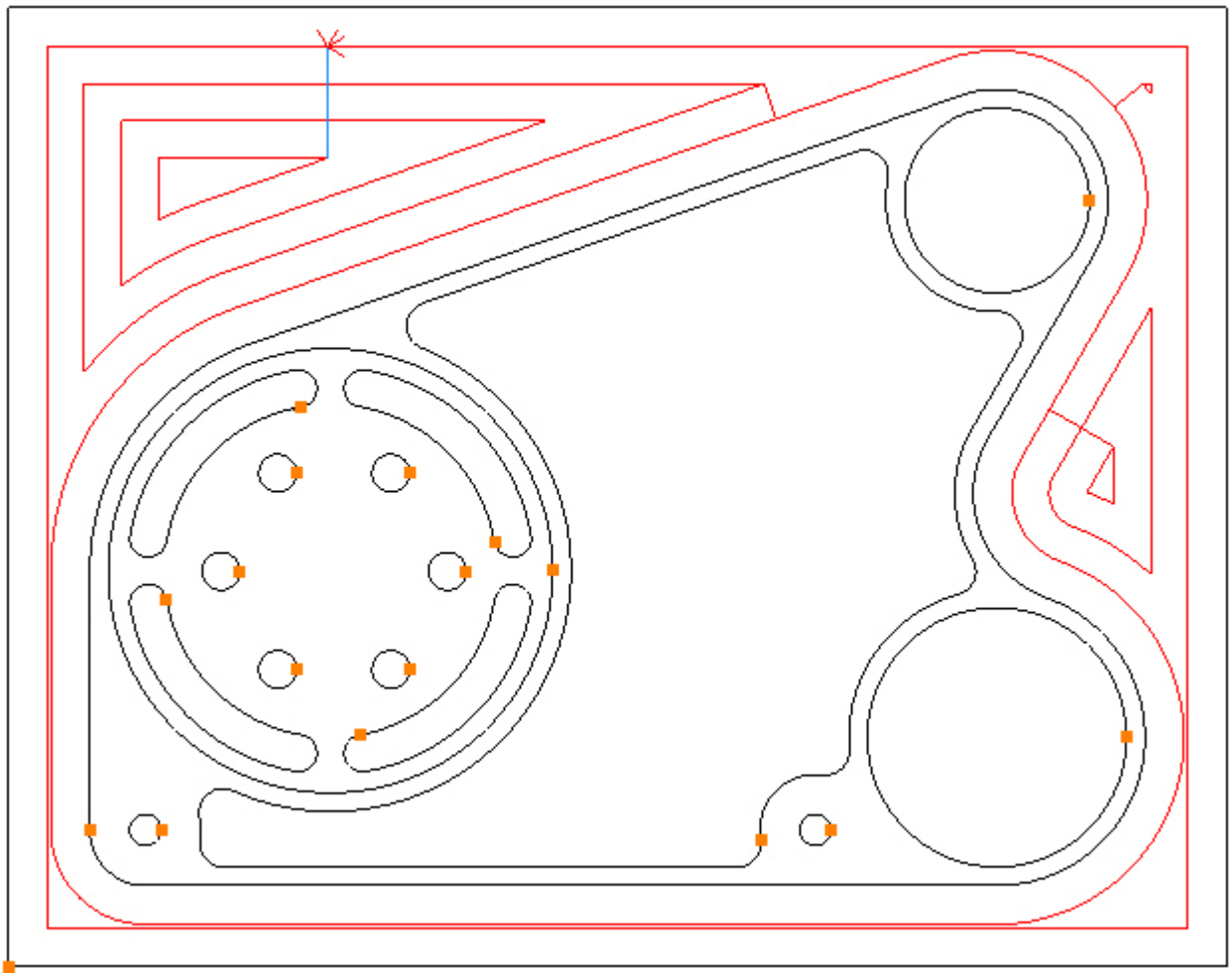
Enter the following values for the Cutting Data:



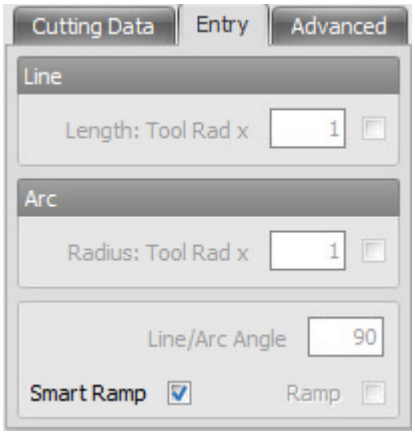
Select the outside profile and the outer rectangle by directly clicking them:



To pocket the selected Contours click the Add button  . The toolpaths are created:



Looking at the resulting toolpaths a blue line can be seen in the top left hand corner. Blue indicates an Entry, the entry is configured using the Entry tab:

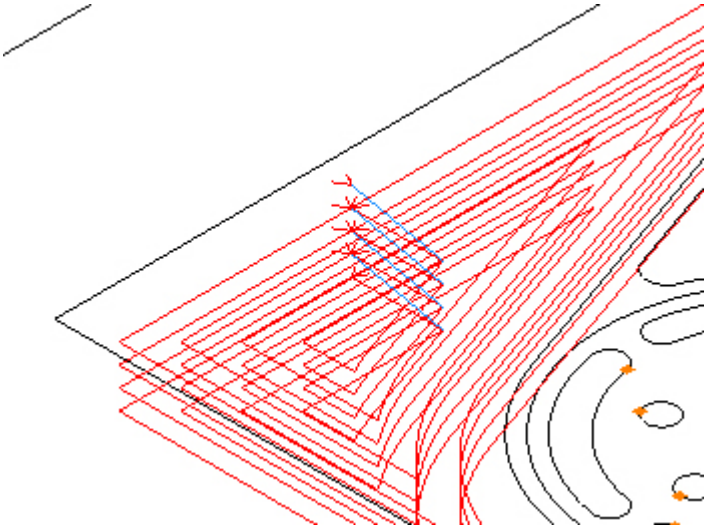


By default Smart Ramp is checked and the blue line is a result of this. To see what Smart Ramp is doing view the Part in an isometric view:

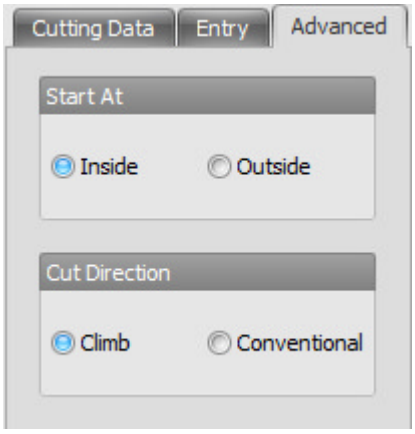
Menu: View -> Standard Views -> Isometric View

Toolbar button:

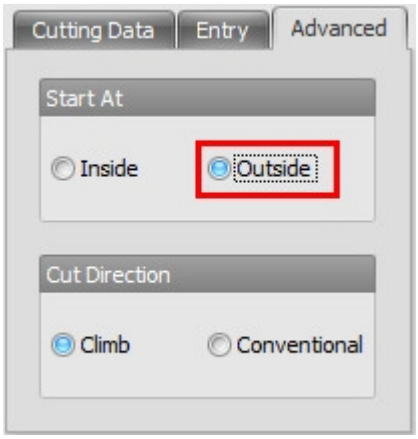
Choose the command and zoom in on the Smart Ramp by pointing the cursor at the Smart Ramp and rotating the middle mouse button:

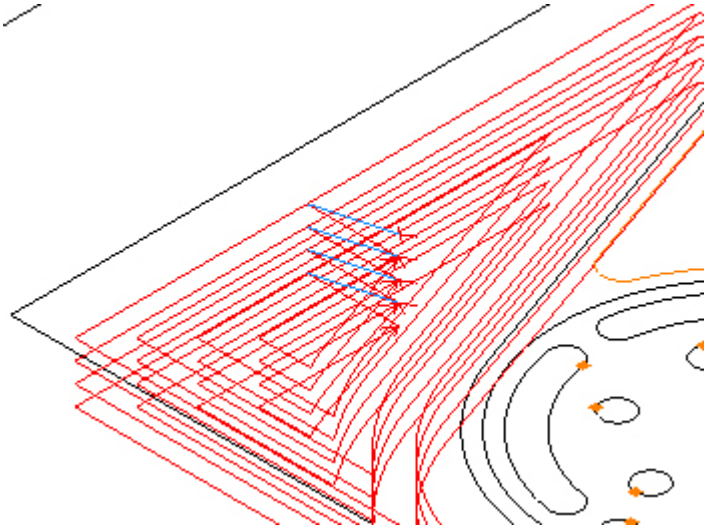


As can be seen the Smart Ramp, as the name implies, automatically ramps to depth at the shallowest angle possible. As it is the outside profile that is being machined Smart Ramp is not needed, because it is possible to plunge to depth outside of the billet.
First start from the outside, click the Advanced tab:

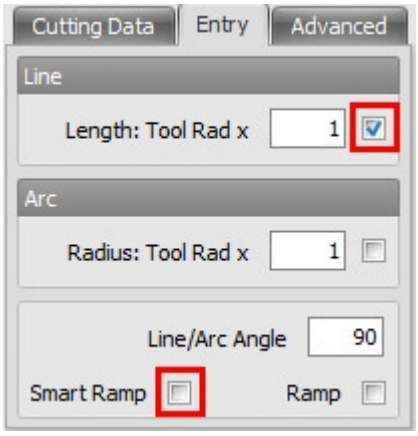


At the moment the pocketing starts from the inside, click the outside radio button and the toolpaths will be recalculated to start from the outside:

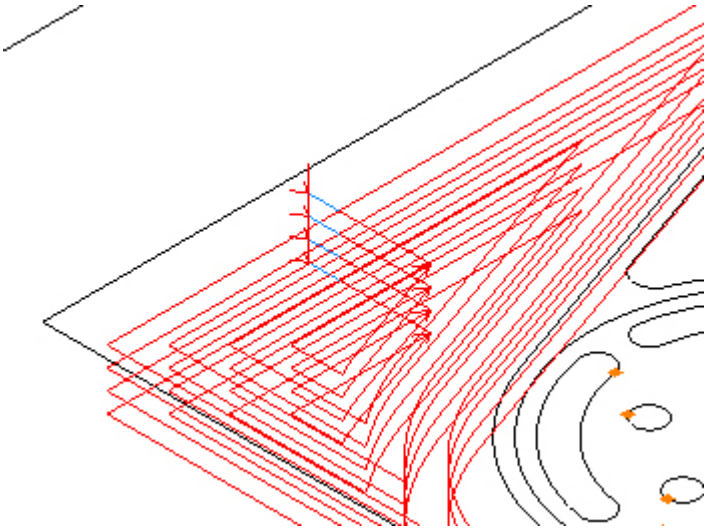




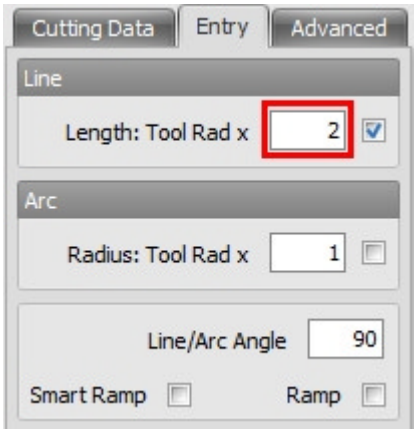
Choose the Entry tab and uncheck the Smart Ramp check box and check the Line check box:



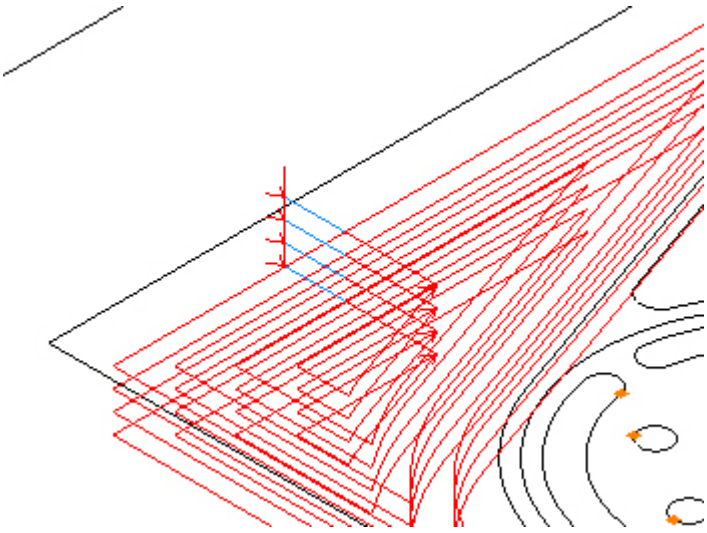
Now the tool is plunging in fresh air:



To start further outside of the billet increase the length of the entry line. Change the entry line length, for example, to 2 (2 x the tool radius = 0.75):



To update this change click the Refresh button  :



Changes to any value in a box are updated by clicking the Refresh button.

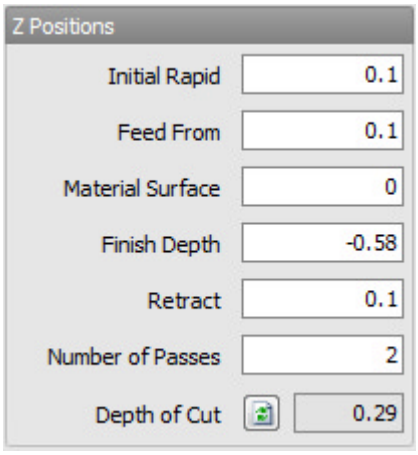
Step 5 - Rough counterbore

Rough Ø4.7 counterbore using Pocket

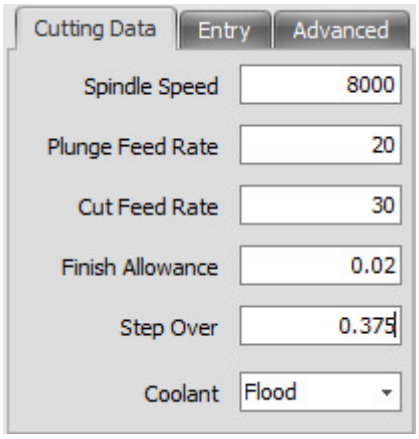
Menu: Machine-> Pocket

Toolbar button:

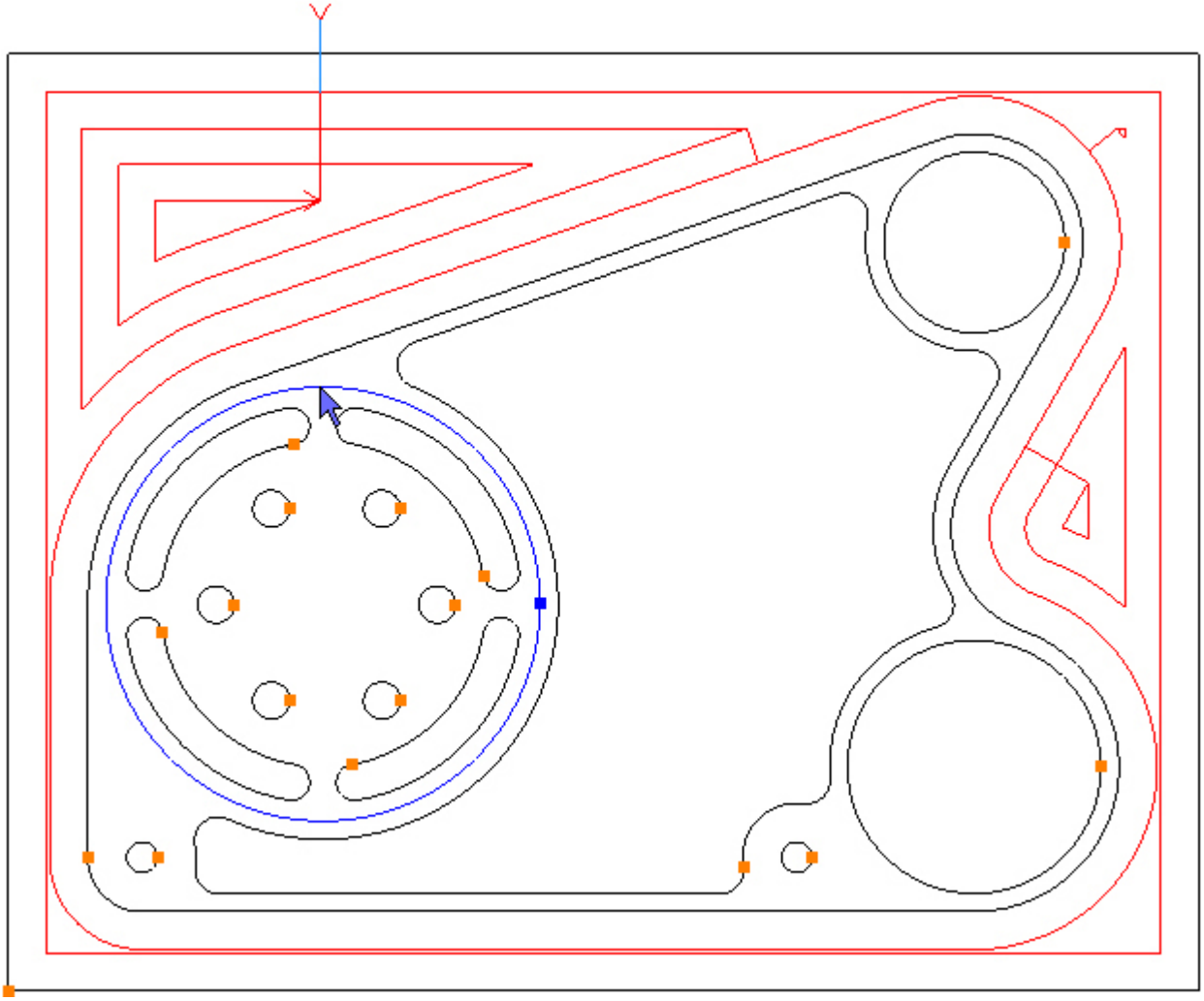
Choose the command and enter the following Z Positions:



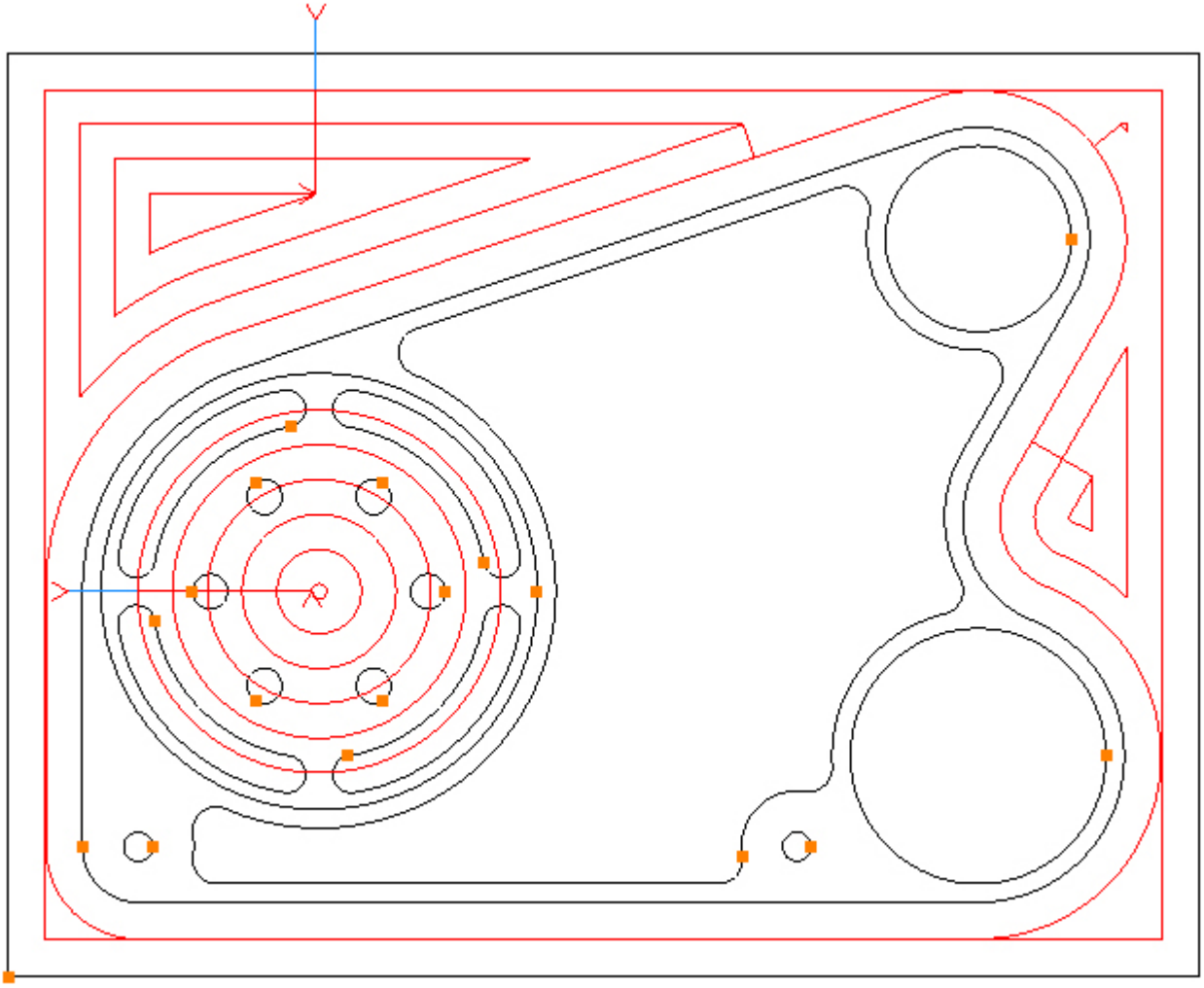
Enter the following values for the Cutting Data:



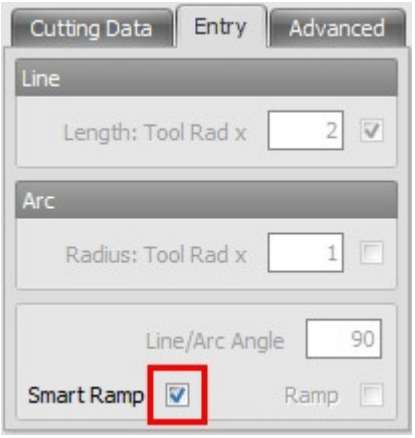
Select the Ø4.7 circle by directly clicking it:



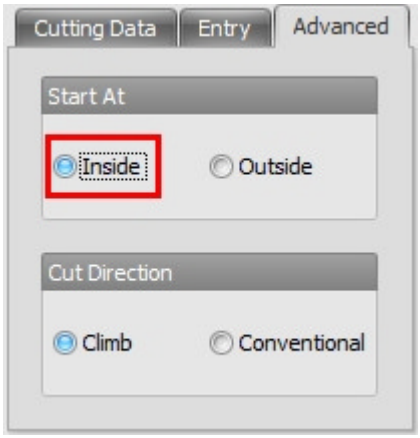
To pocket the selected Contours click the Add button . The toolpath is created:



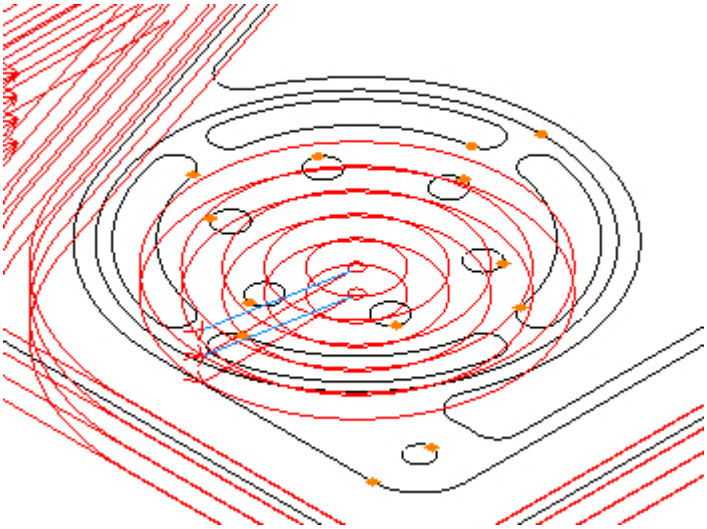
This time Smart Ramp is required, this is still unchecked from the previous pocketing operation. Choose the Entry tab and check Smart Ramp:



At the moment the pocketing starts from the outside, but it is preferable to start from the inside. Choose the Advanced tab and select the Inside Radio button:



The toolpath should look like this in isometric view:



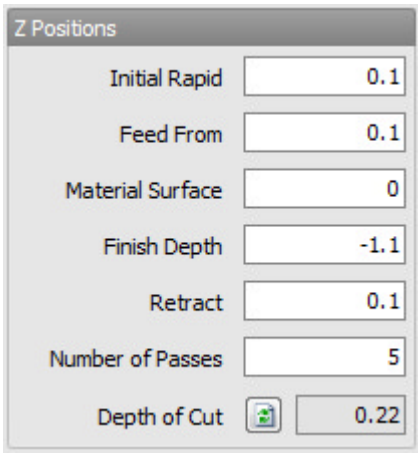
Step 6 - Rough bores

Rough Ø2.0 and Ø2.75 bores using Pocket

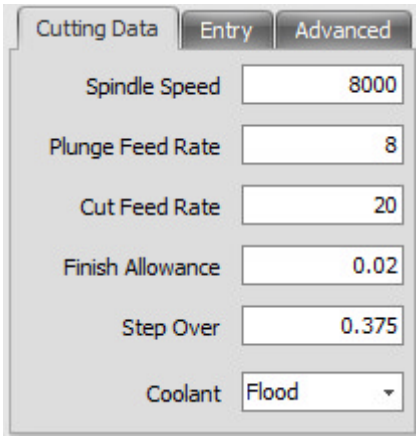
Menu: Machine-> Pocket

Toolbar button:

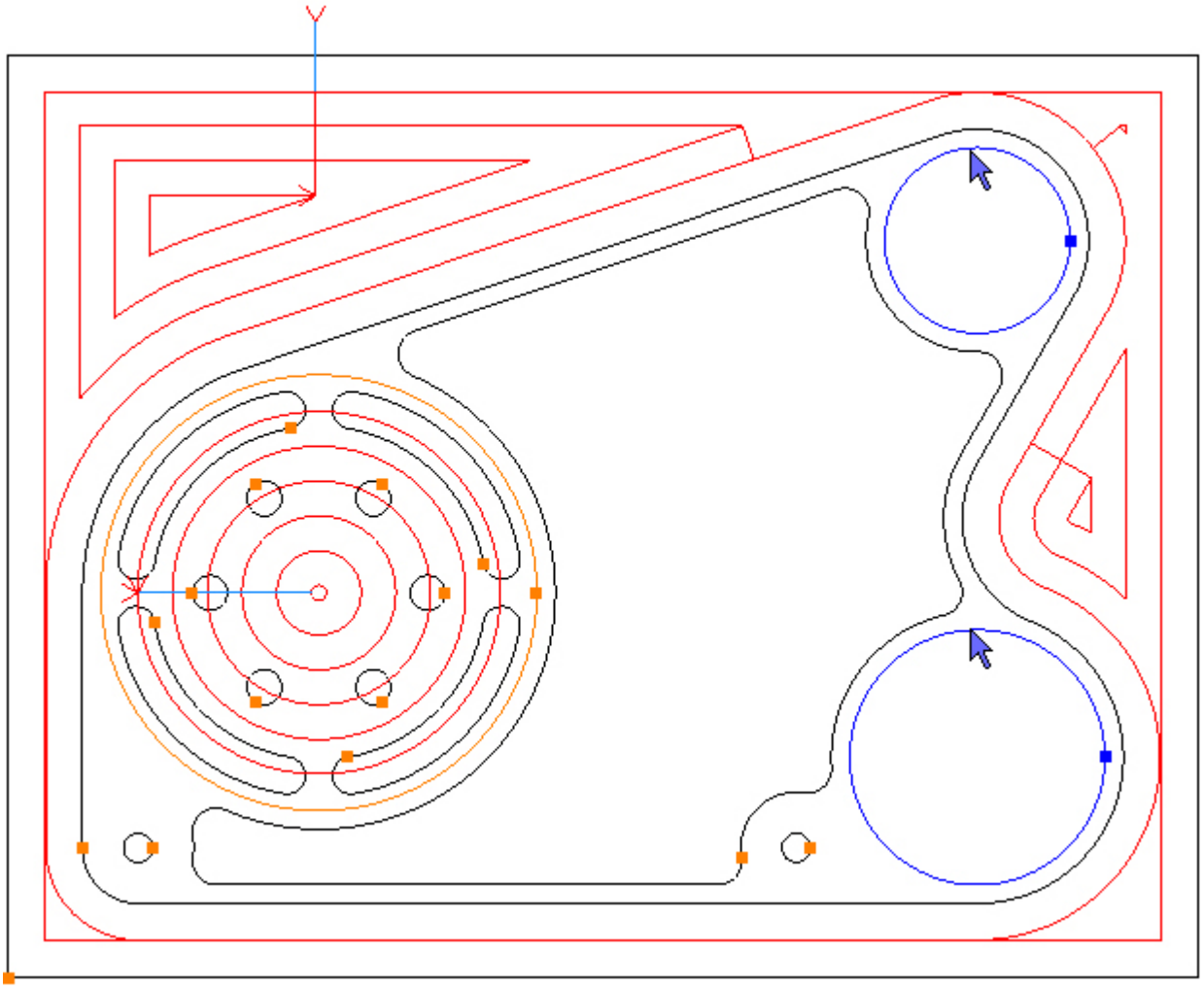
Choose the command and enter the following Z Positions:



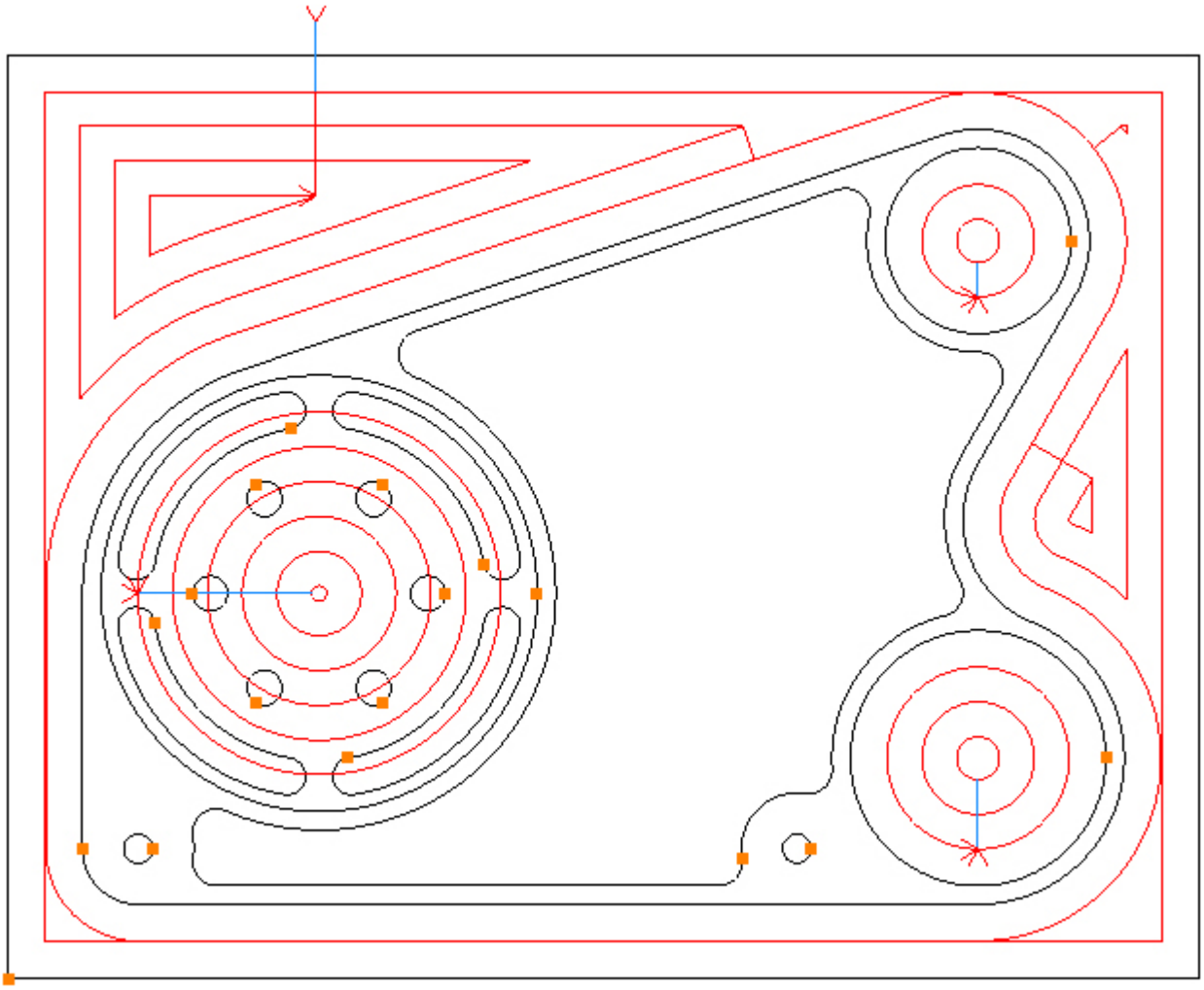
Enter the following values for the Cutting Data:



Select the Ø2.0 and Ø2.75 circles by directly clicking them:



To pocket the selected Contours click the Add button  . The toolpath is created:



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Step 7 - Rough inside

Rough inside pocket using Pocket

Menu: Machine-> Pocket

Toolbar button:

Choose the command and THEN select the tool for this operation.

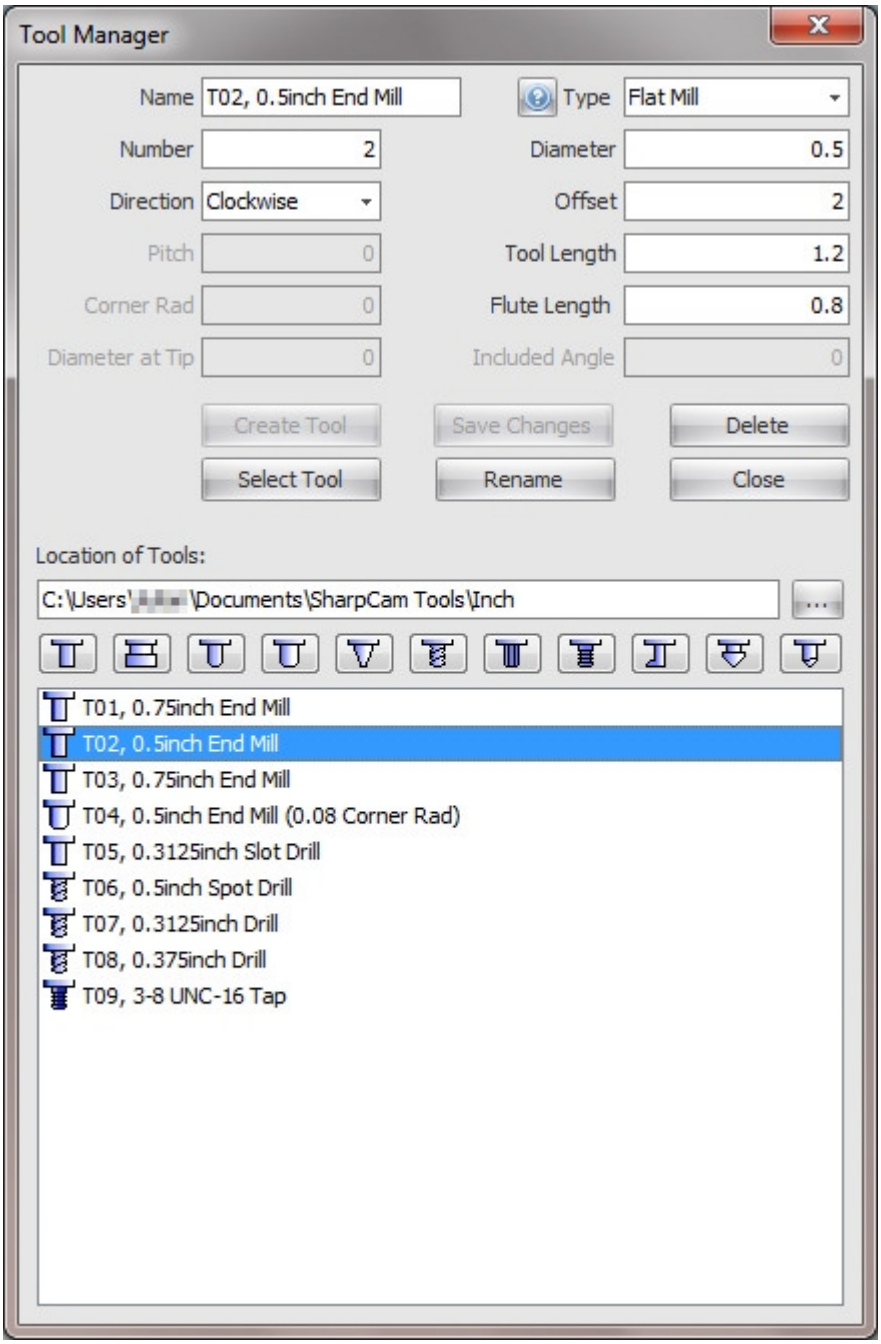
If a tool is selected, before choosing the Pocket command, SharpCam will assume that the current operation is to be created with a different tool and will regenerate the toolpath accordingly (do not forget that the Ø2.0 and Ø2.75 bores have just been roughed and are still editing the operation).

Choose the Tool Manager:

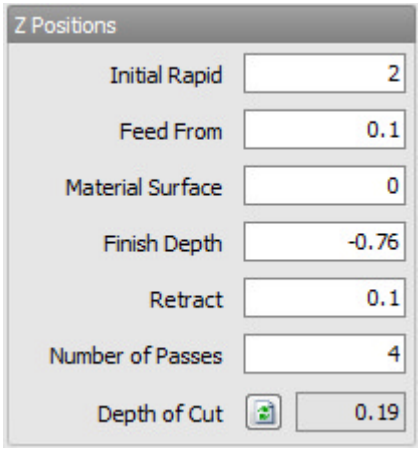
Menu: Machine-> Tool Manger

Toolbar button:

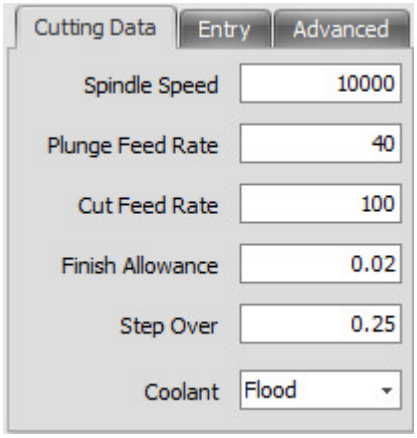
Choose the command to display the Tool Manager and click on 'T02, 0.5inch End Mill', then click the 'Select Tool' button. Alternatively double click a tool to select it:



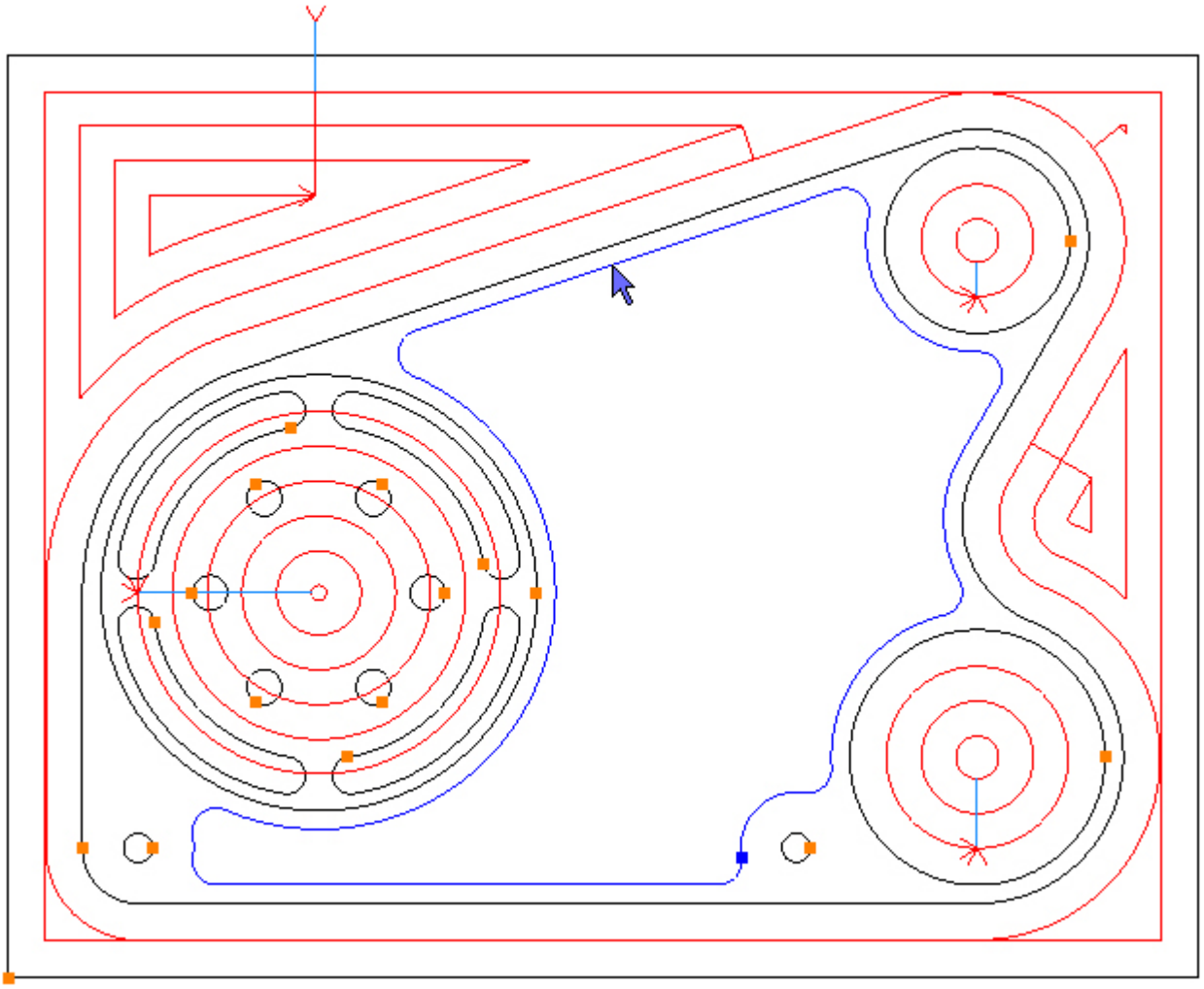
Enter the following Z Positions:



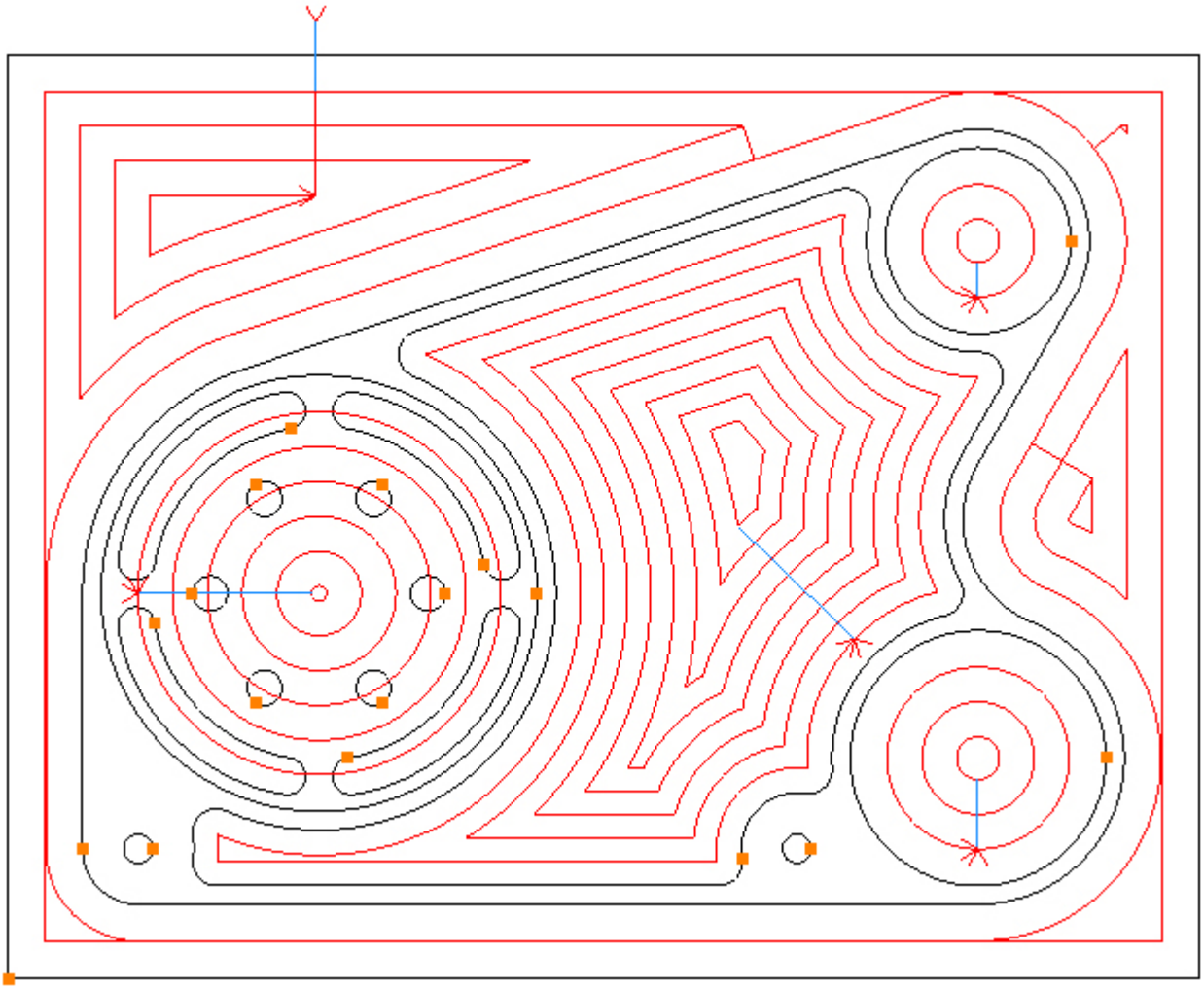
Enter the following values for the Cutting Data:



Select the Contour on the inside by directly clicking it:



To pocket the selected Contour click the Add button  . The toolpath is created:



Step 8 - Finish outside profile

Finish outside profile using Profile

Menu: Machine-> Profile

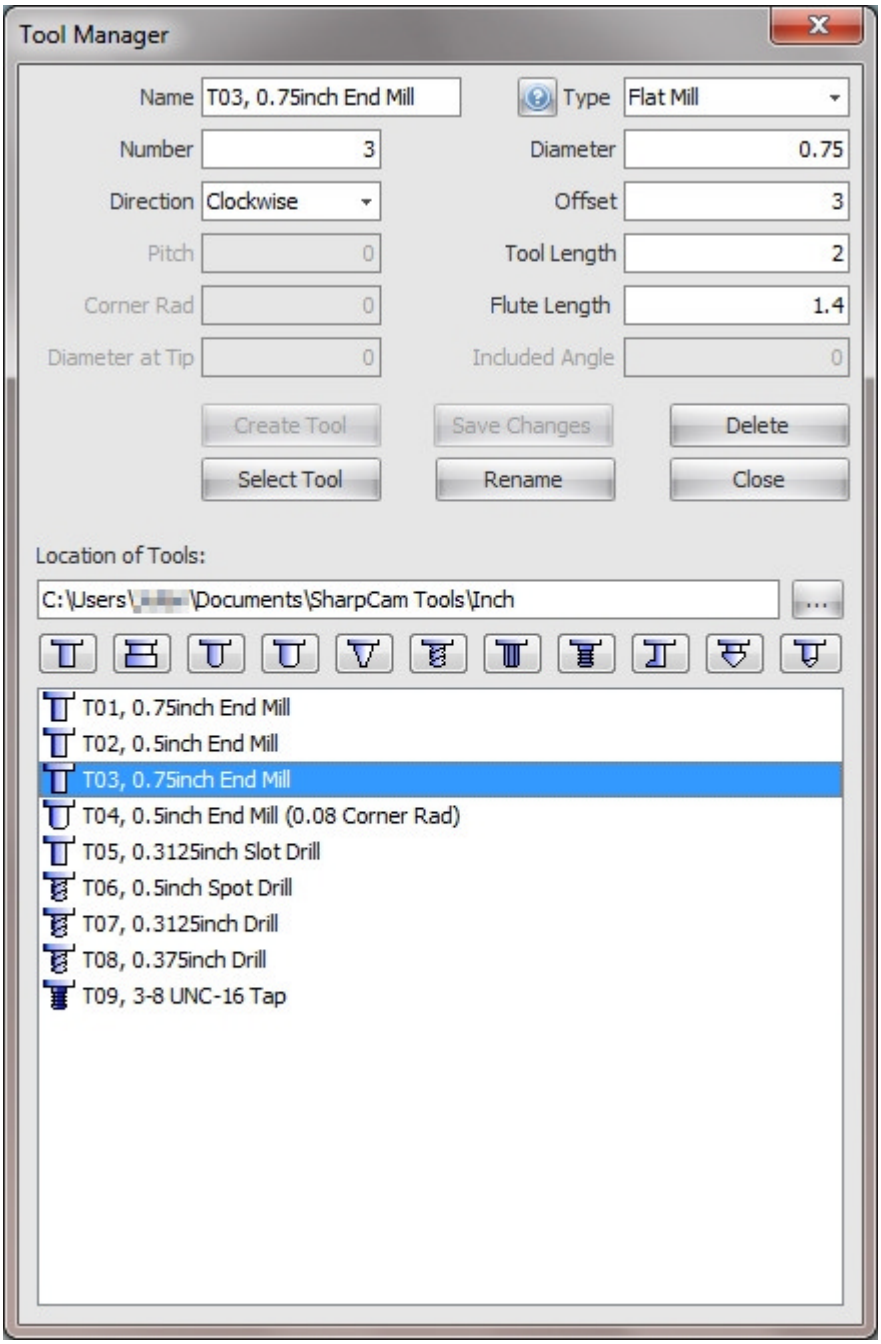
Toolbar button:

Choose the Profile command then select the tool for this operation:

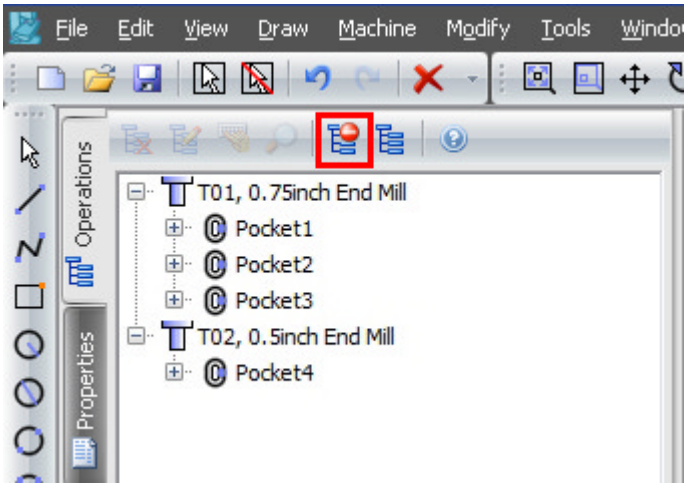
Menu: Machine-> Tool Manger

Toolbar button:

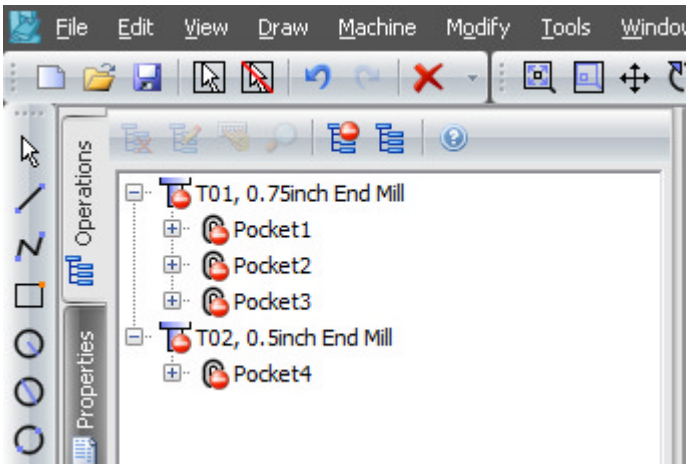
Choose the command to display the Tool Manager and click on 'T03, 0.75inch End Mill', then click the 'Select Tool' button. Alternatively double click a tool to select it:



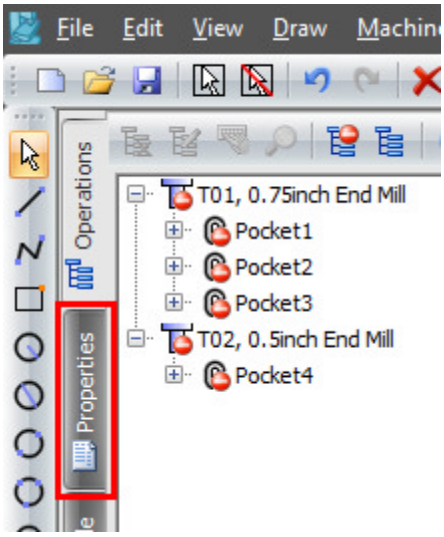
The Part now has a number of toolpaths. In order to prevent the view from being obscured, suppress them so they are hidden. Choose the Operations tab on the Part Manager and click the Suppress All button:



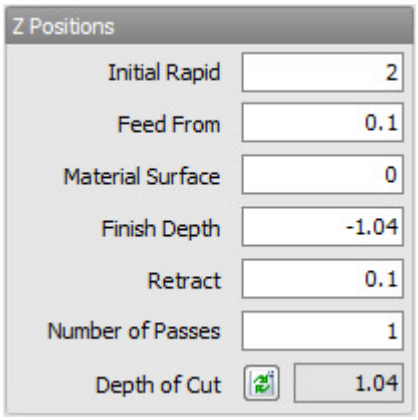
All operations now indicate that they are suppressed and the toolpaths are now hidden:



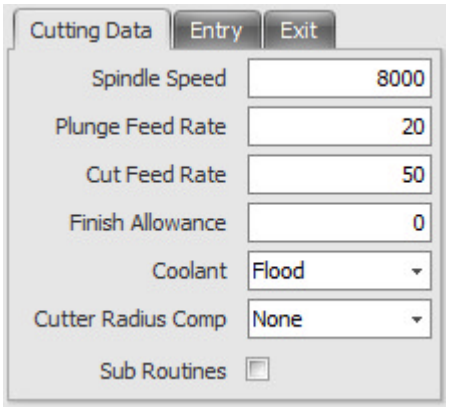
Click the Properties tab on the Part Manager to continue with the Profile command:



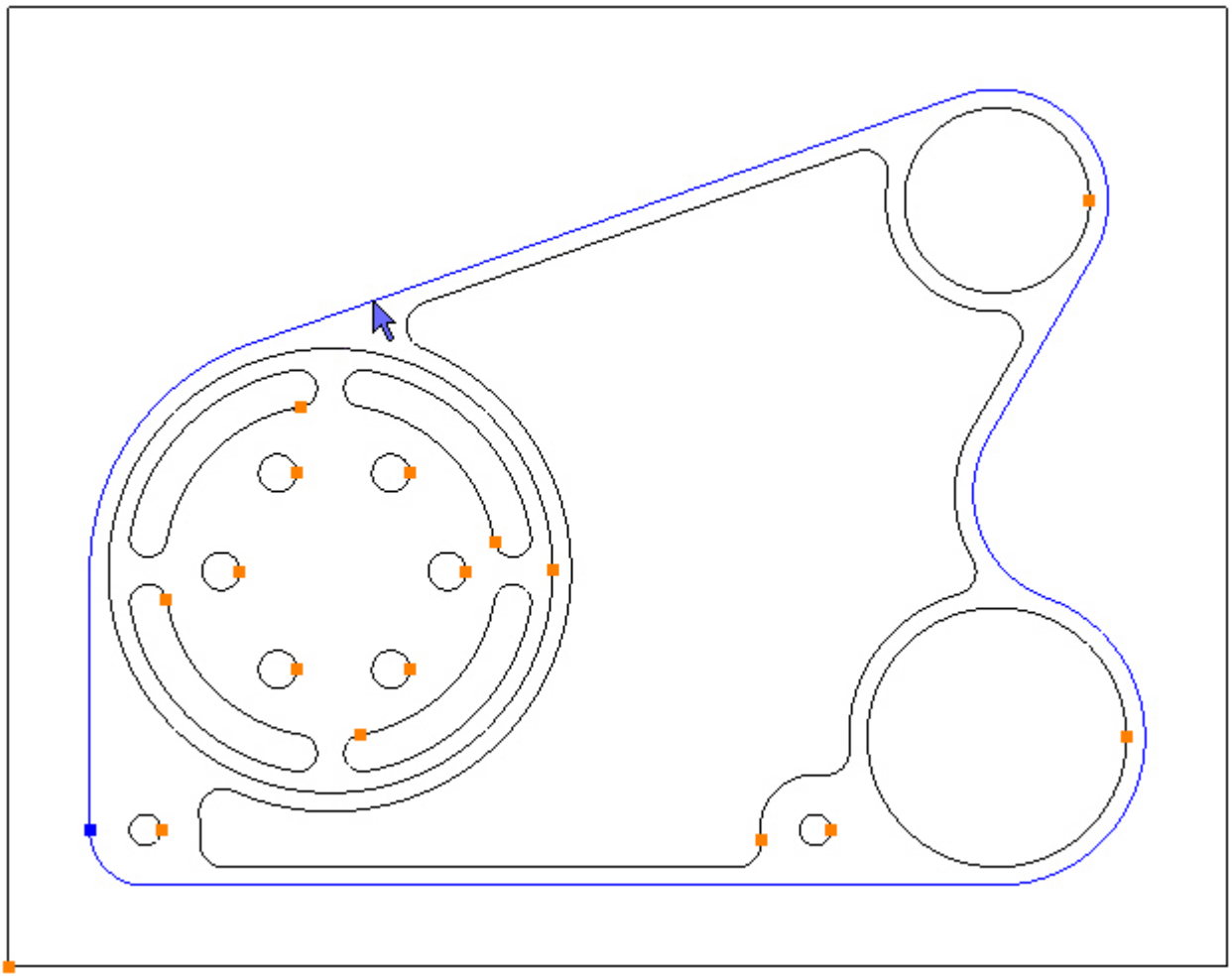
Enter the following Z Positions:



Enter the following values for the Cutting Data:



Select the outside Contour by directly clicking it:



To profile the selected Contour click the Add button  . The toolpath is created:

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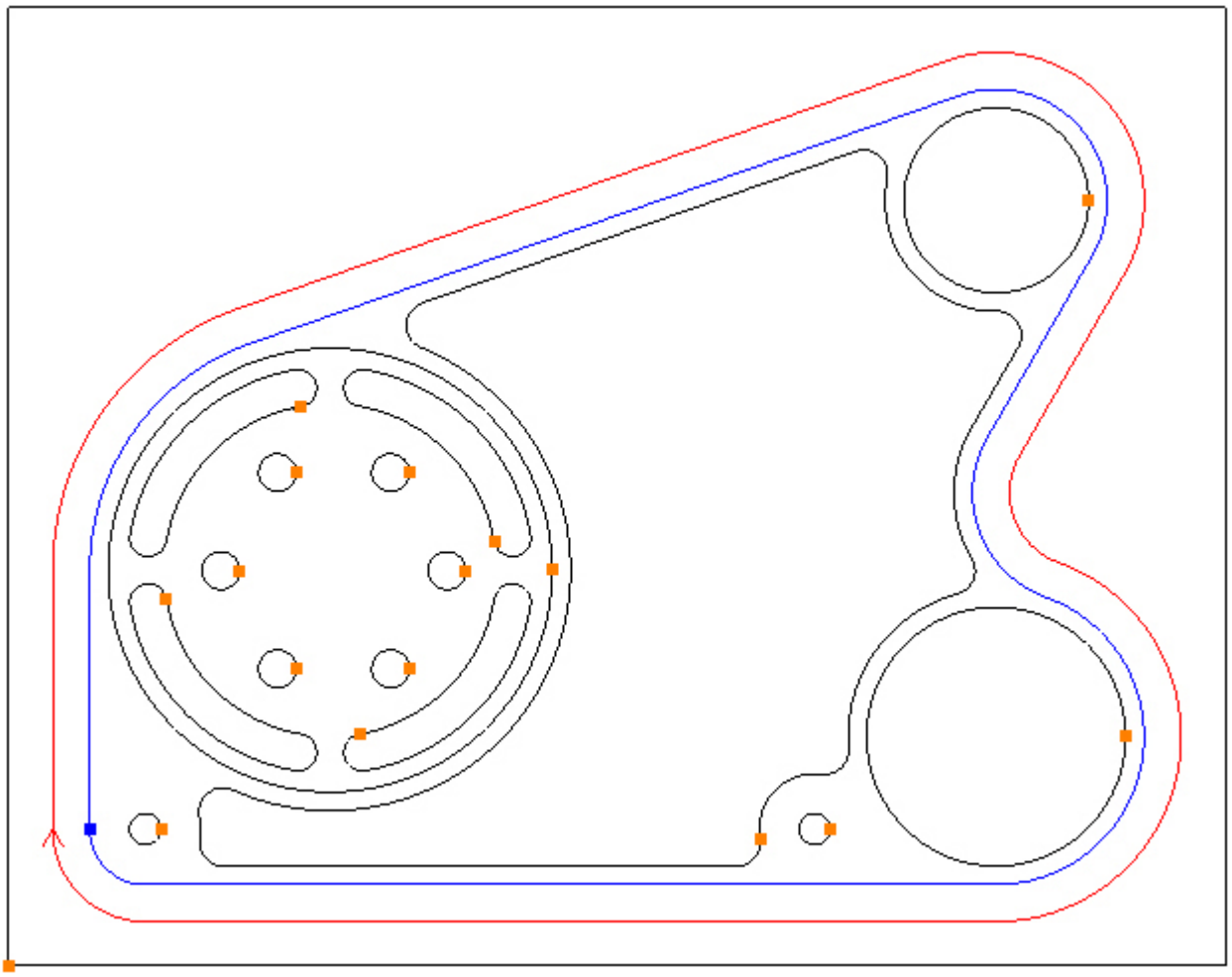
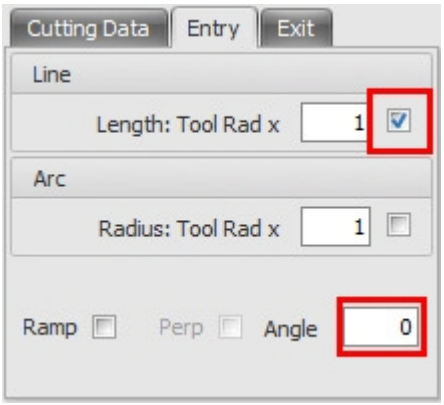
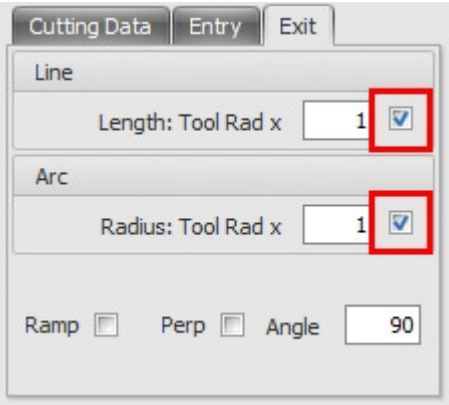


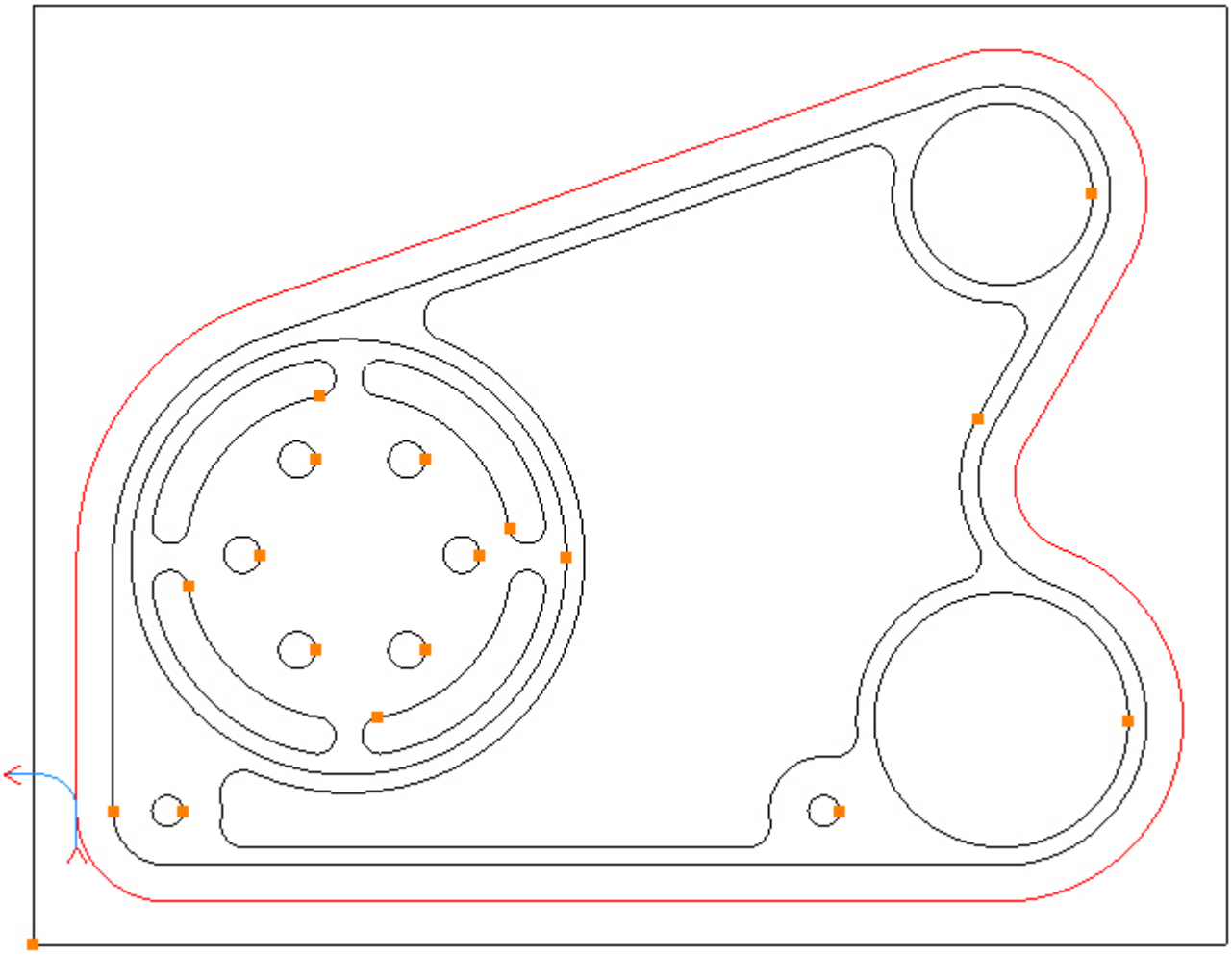
Add an entry and exit. Choose the Entry tab and change the Line/Arc Angle to 0 and check the Line check box:



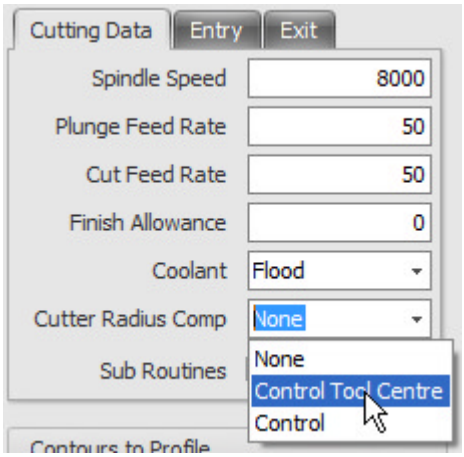
Choose the Exit tab and check the Line and Arc check box:



The entry and exit have been added:



Cutter radius compensation is required for this profile operation. Choose the Cutting Data tab and select *Control Tool Centre*:



When using *Control Tool Centre*, zero must be entered in the control offset table for the correct size tool. Adjust the size with small + or - values.
If you prefer to enter the full radius in the control offset table, then choose *Control*.

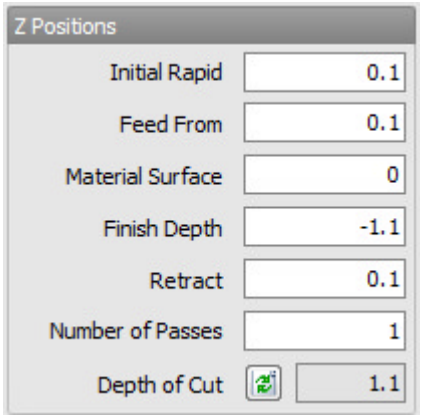
Step 9 - Finish bores

Finish Ø2.0 and Ø2.75 bores using Profile

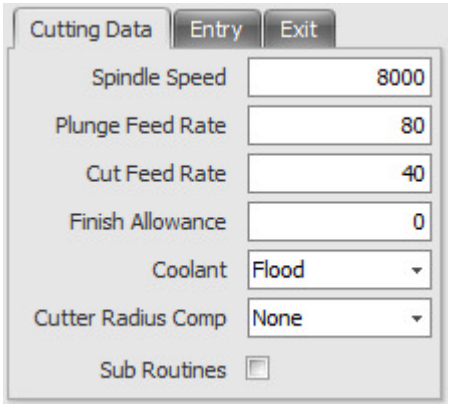
Menu: Machine-> Profile

Toolbar button:

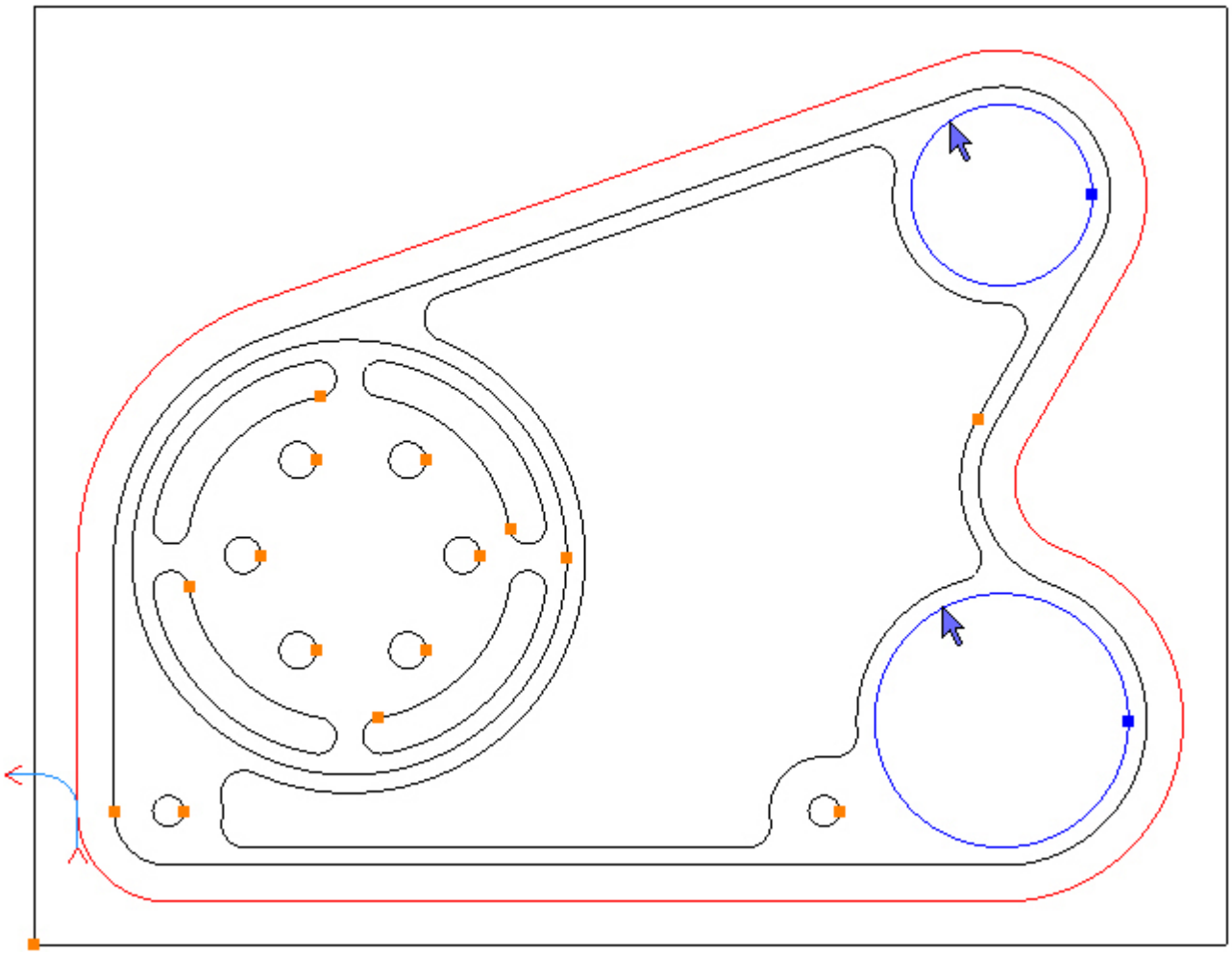
Choose the Profile command then select the tool and enter the following values for the Z Positions:



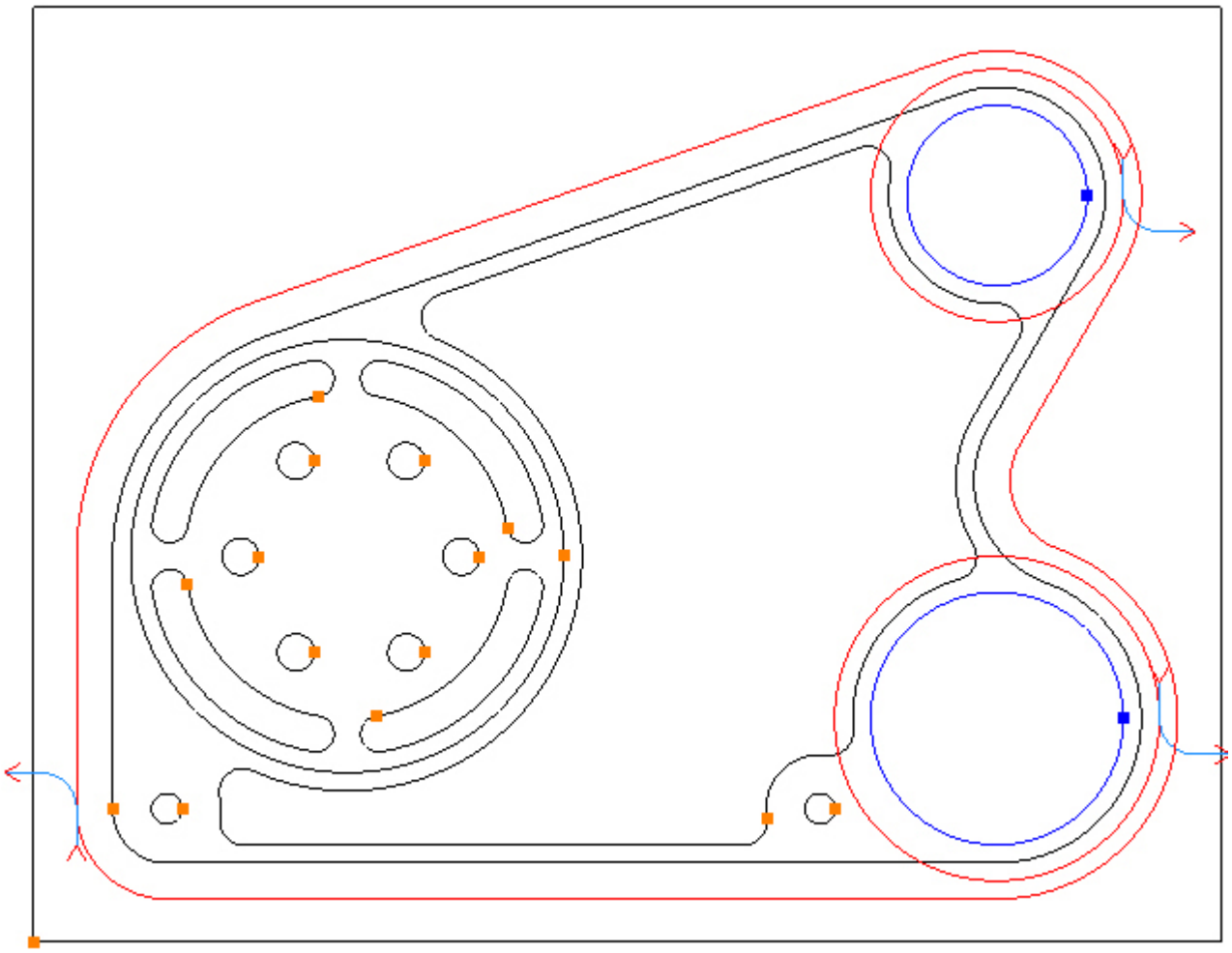
Enter the following values for the Cutting Data:



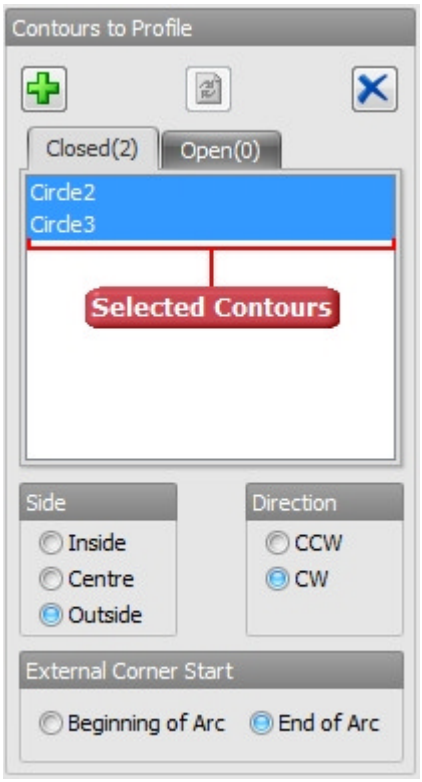
Select the Ø2.0 and Ø2.75 circles by directly clicking them:



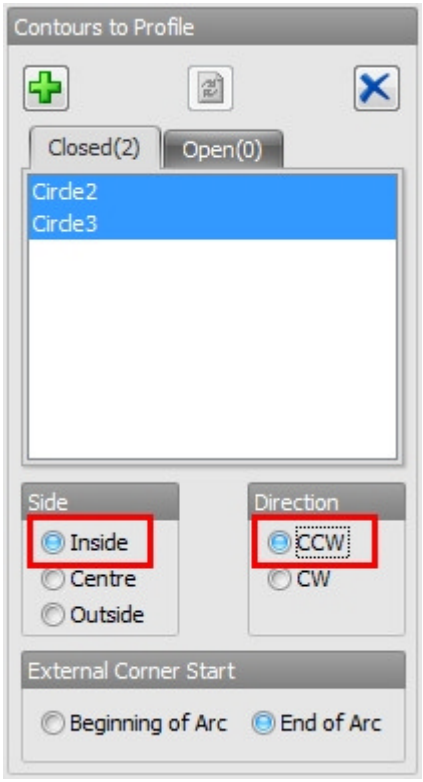
To profile the selected Contours click the Add button  . The toolpaths are created:



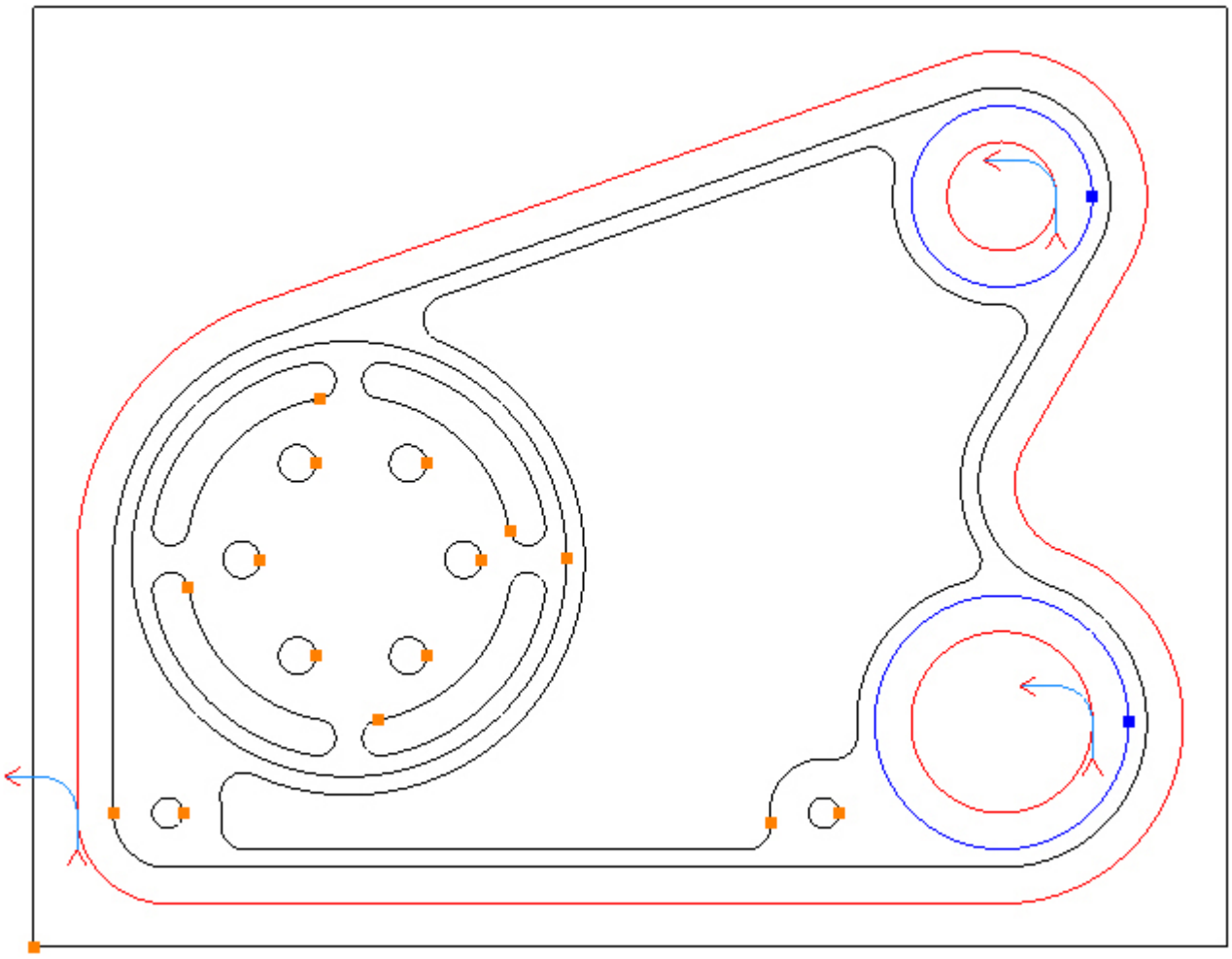
As can be seen the toolpaths are on the outside and in a clockwise direction, this is the default. This needs to be changed to the inside in a counter-clockwise direction.
The Contours must be selected in order to change the side and direction of the resulting toolpaths. When Contours are added to an operation they will be left selected. If the Contours are not selected they must be selected first, they can also be selected by clicking the name in *Contours to Profile* group box:



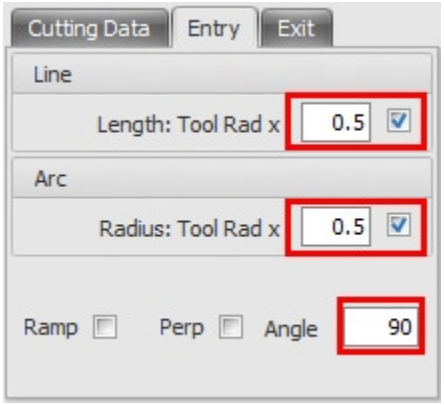
To change the side and direction click the appropriate radio button, in this case *Inside* and *CCW:*



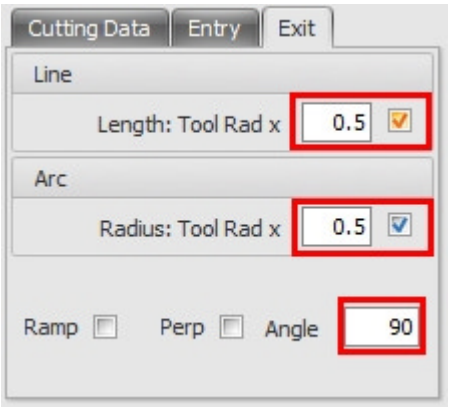
The toolpath is now on the inside and in a counter-clockwise direction:



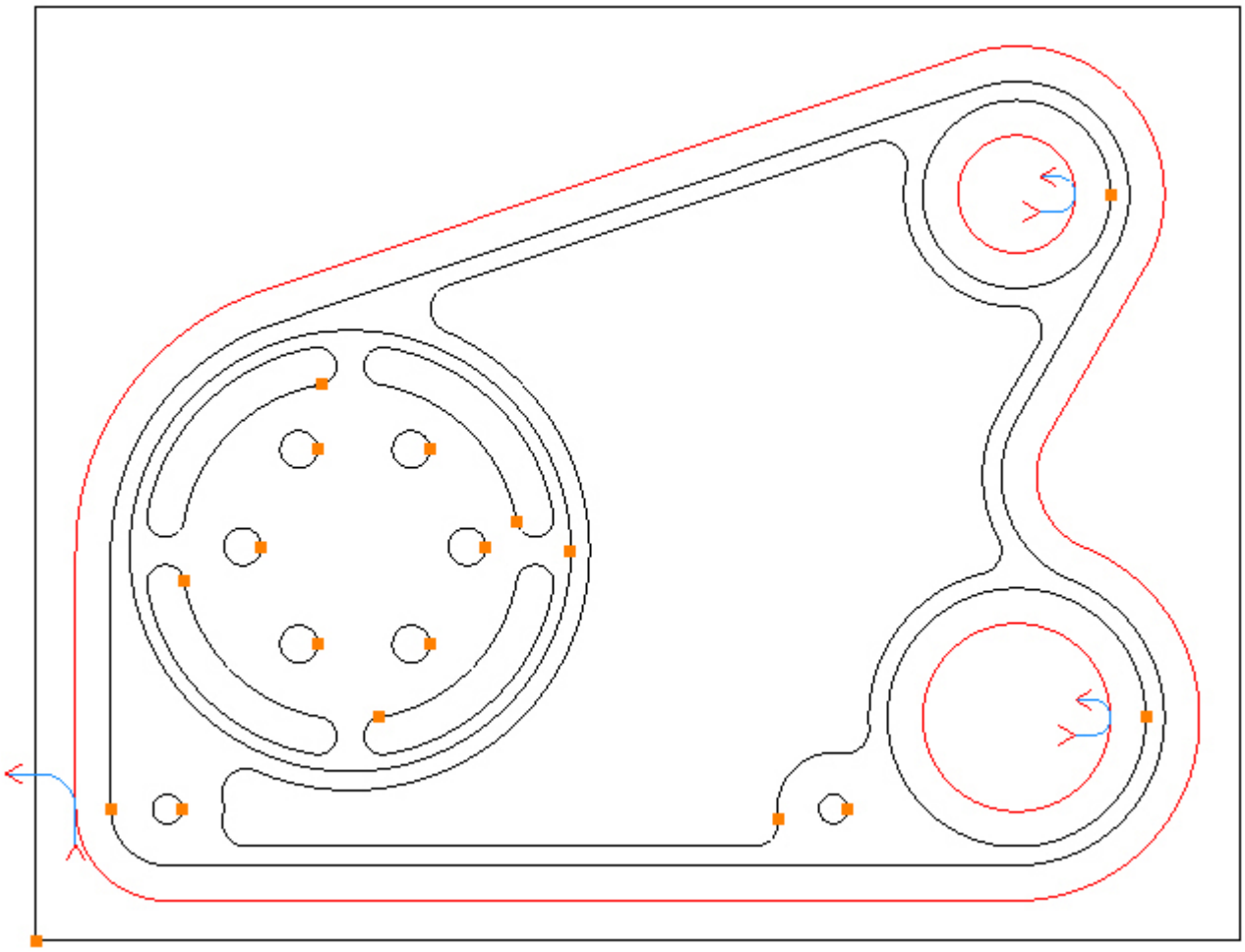
Change the entry and exit because the settings from the previous operation are still in effect. Choose the Entry tab and set the values (before checking the boxes) as below and check the line and arc check box:



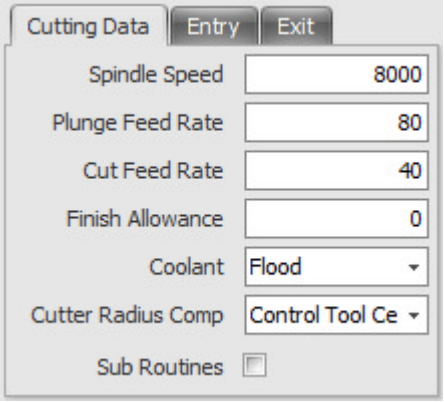
Choose the Exit tab and change the Length and Radius values and click the Refresh button:



The entry and exit have been added:



Cutter radius compensation is required for this profile operation. Choose the Cutting Data tab and select *Control Tool Centre*:



Deselect the Contours in readiness for the next operation:

Menu: Edit -> Deselect All

Toolbar button: 

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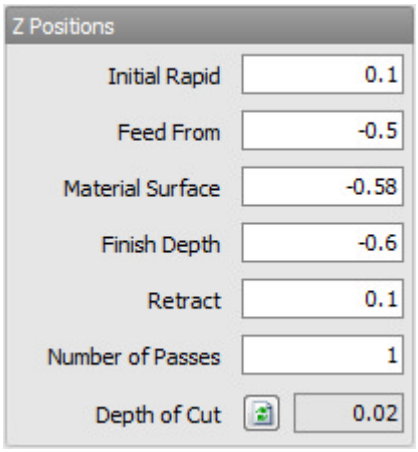
Step 10 - Finish counterbore

Finish Ø4.7 counterbore using Pocket

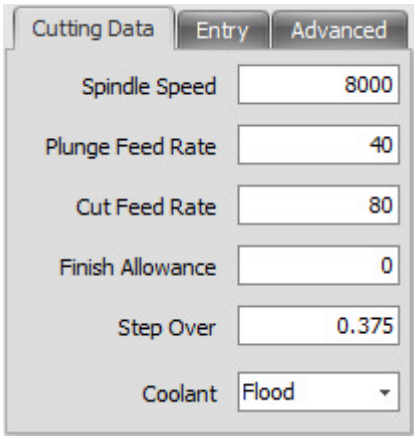
Menu: Machine-> Pocket

Toolbar button:

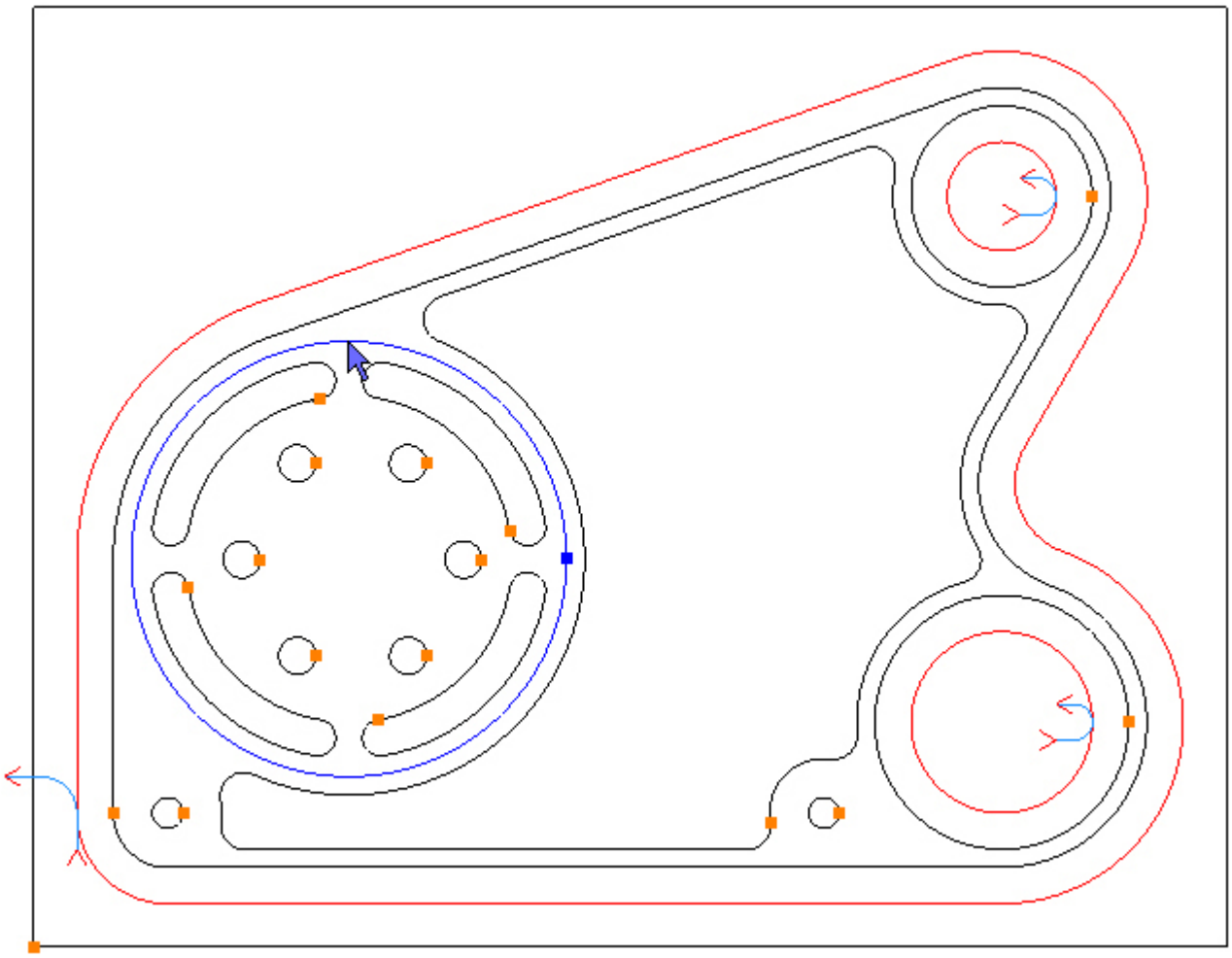
Choose the command and enter the following values for the Z Positions:



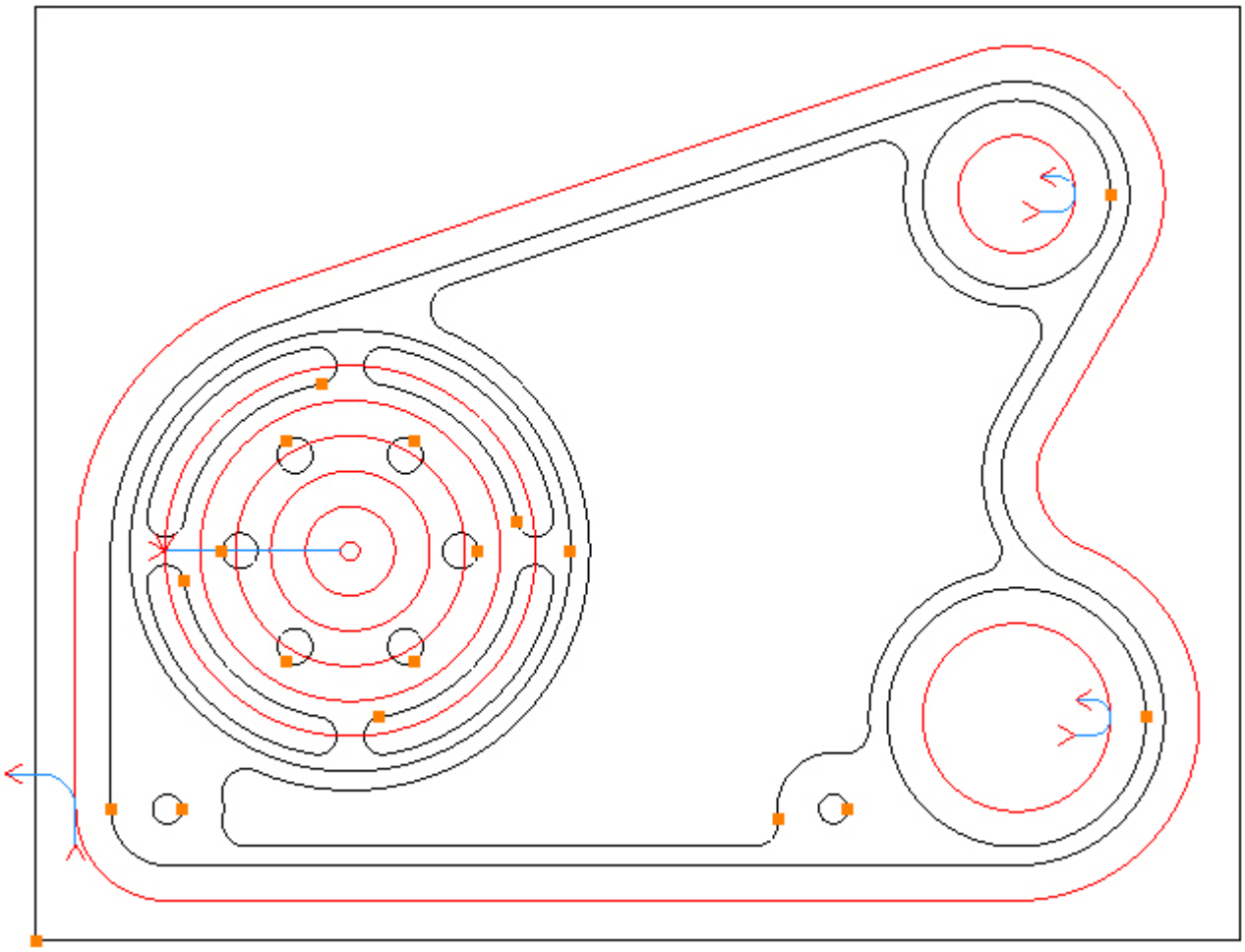
Enter the following values for the Cutting Data:



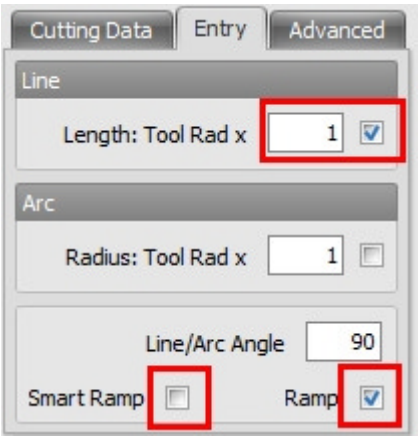
Select the Contour by directly clicking it:



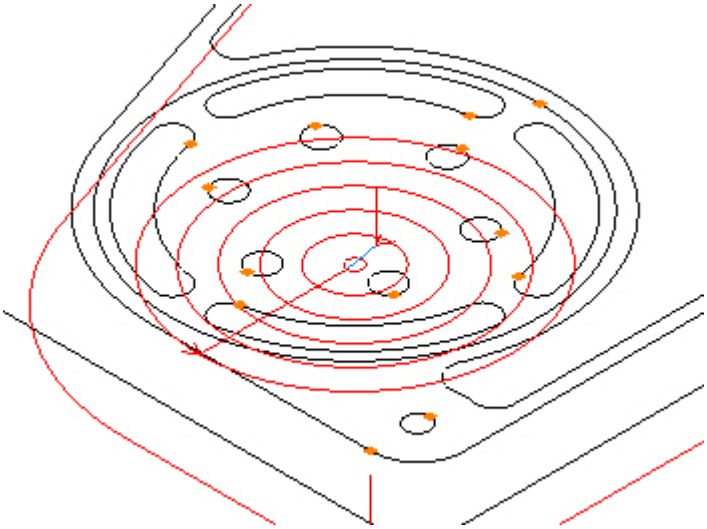
To profile the selected Contour click the Add button  . The toolpath is created:



Smart Ramp is applied, but is not particularly suitable as the cutter will plunge against the side wall. Choose the Entry tab, uncheck the Smart Ramp and change the line length to 1. Check the Line and then the Ramp check box:



This will create a smaller ramp from the centre of the counterbore:



Gear Housing Tutorial

Step 11 - Finish inside

Finish inside pocket using Pocket

Menu: Machine-> Pocket

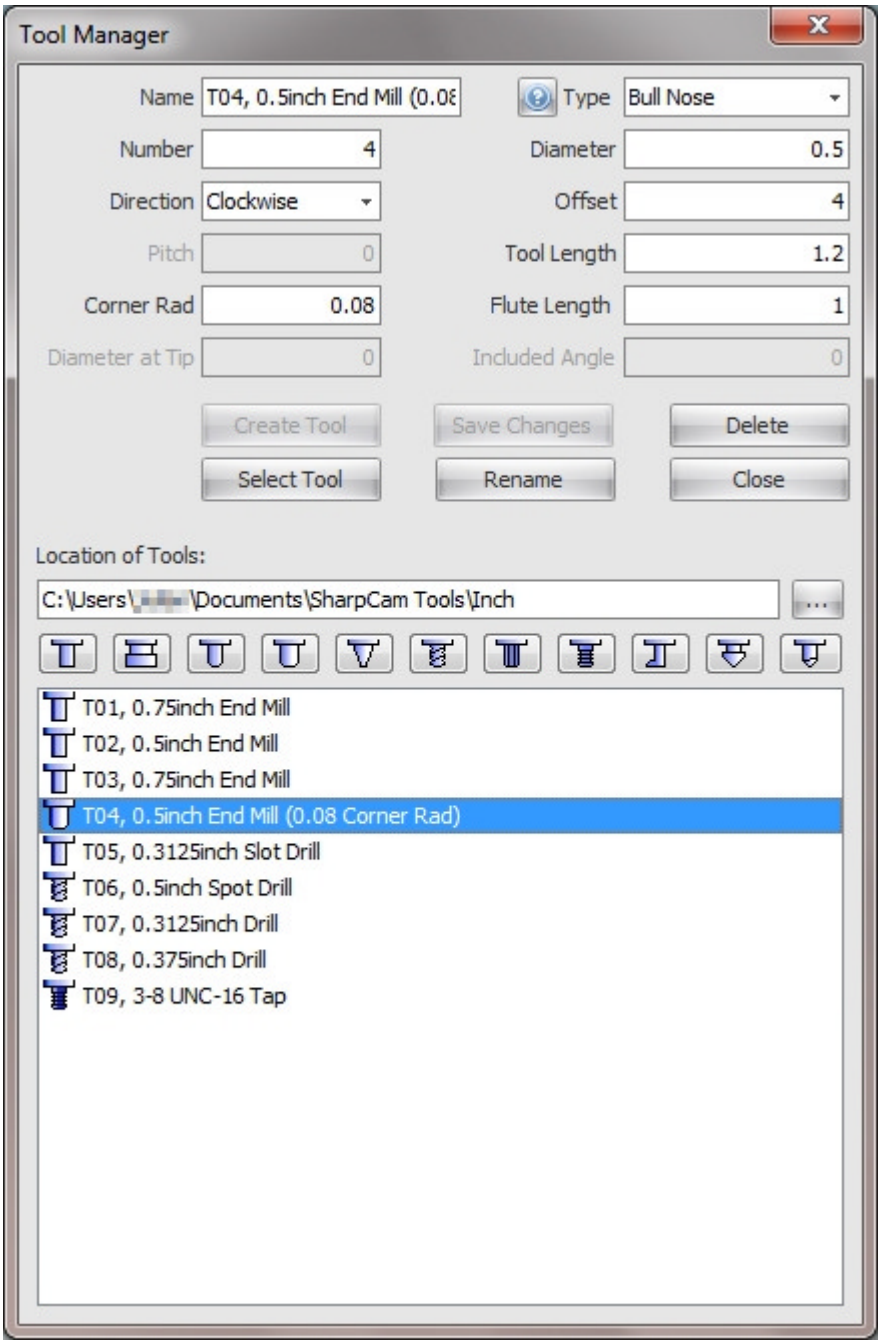
Toolbar button:

Choose the command and then select the tool for this operation:

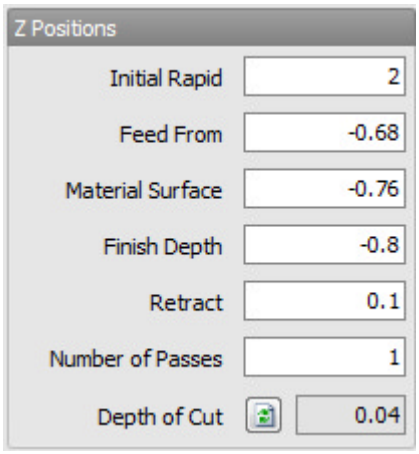
Menu: Machine-> Tool Manger

Toolbar button:

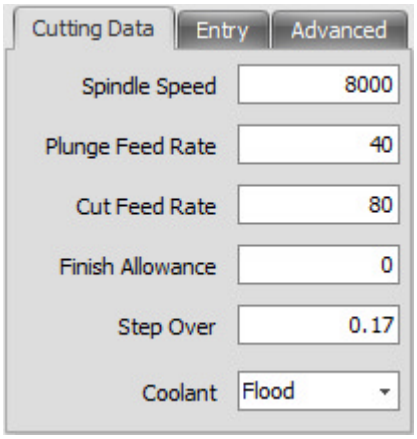
Choose the command to display the Tool Manager and click on 'T04, 0.5inch End Mill (0.08 Corner Rad)', then click the 'Select Tool' button. Alternatively double click a tool to select it:



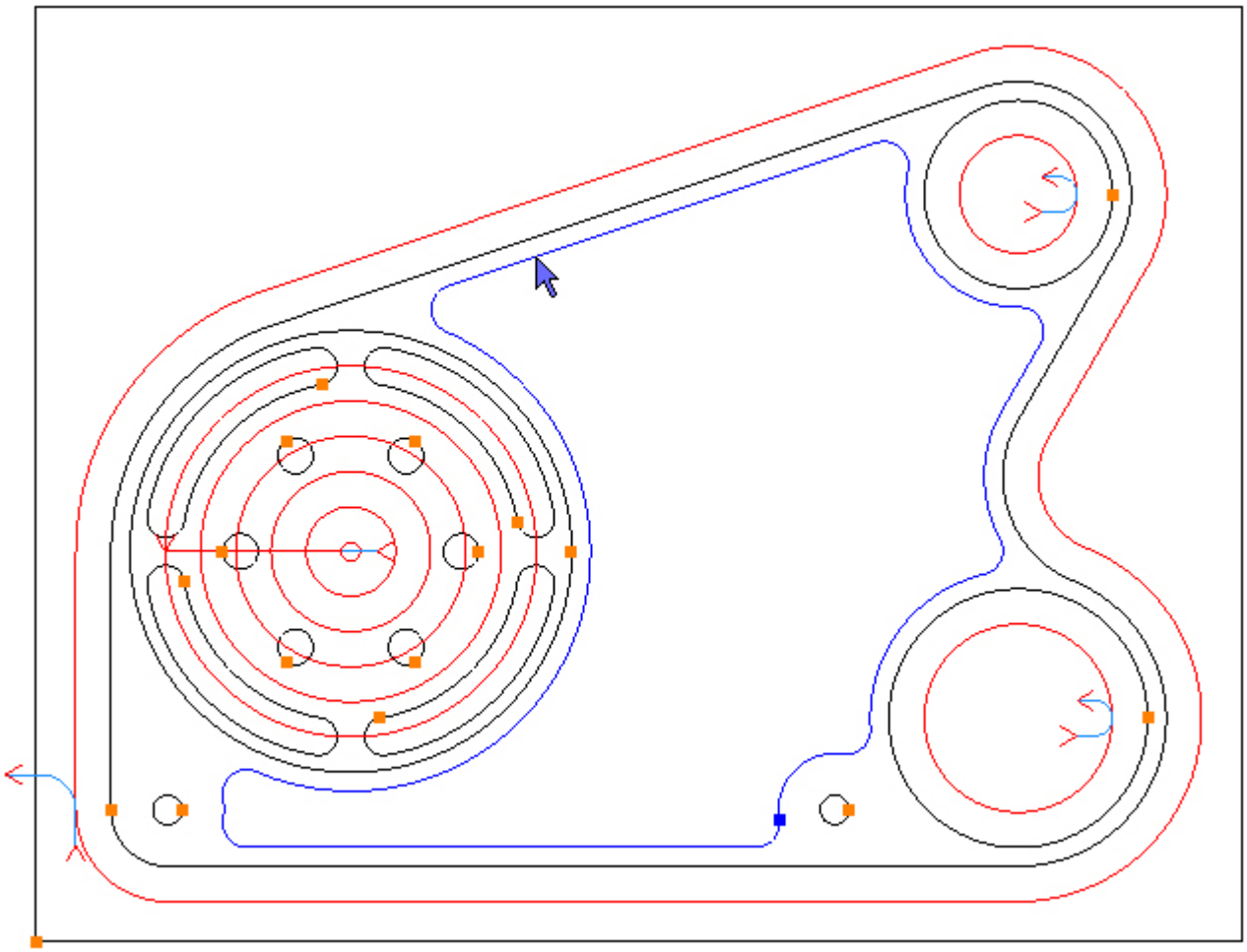
Enter the following values for the Z Positions:



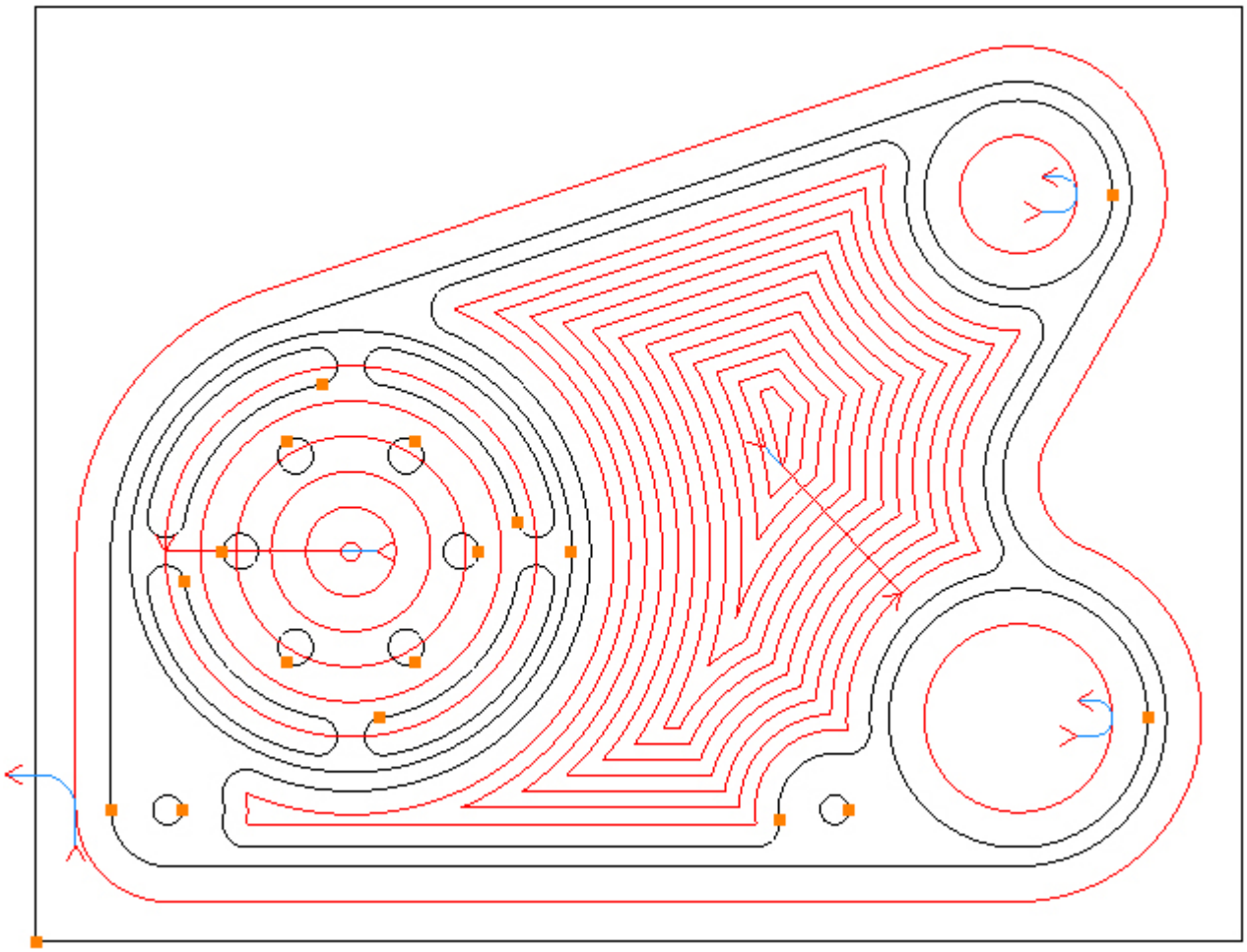
Enter the following values for the Cutting Data.
Note that the step over value has been changed from the default value of 0.25 to 0.17 to ensure that pocket fully cleans up on the bottom, don't forget that the cutter has a 0.08 corner radius:



Select the Contour on the inside by directly clicking it:



To pocket the selected Contour click the Add button . The toolpath is created:



The entry settings from the previous operation are acceptable.

Gear Housing Tutorial

Step 12 - Machine radial slots

Machine radial slots using Profile

Menu: Machine-> Profile

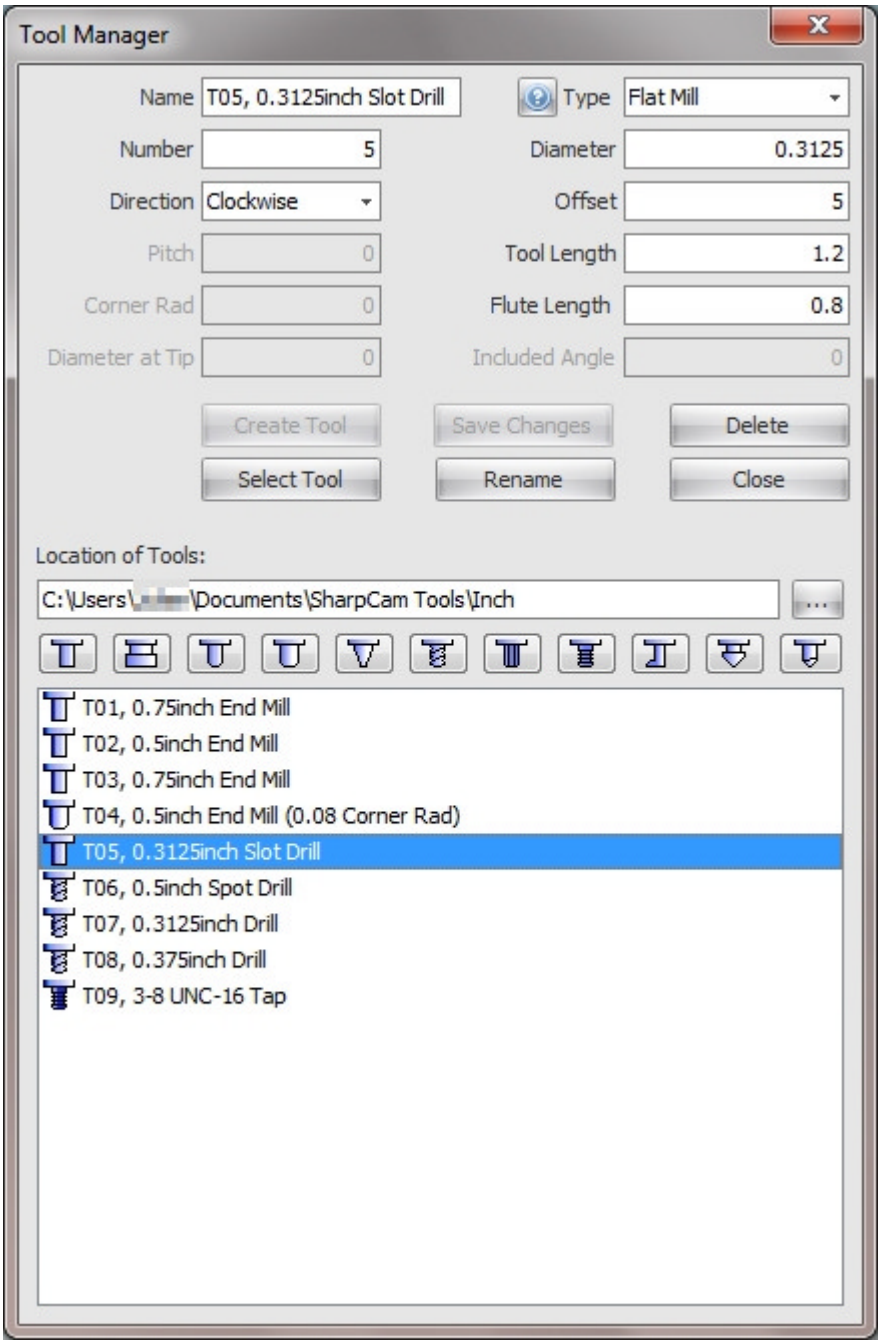
Toolbar button:

Choose the Profile command then select the tool for this operation:

Menu: Machine-> Tool Manger

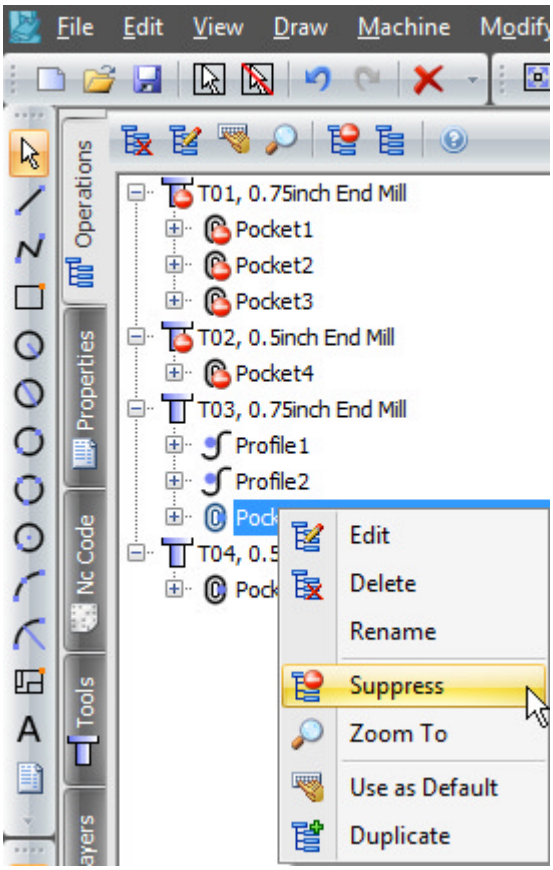
Toolbar button:

Choose the command to display the Tool Manager and click on 'T05, 0.3125inch Slot Drill', then click the 'Select Tool' button. Alternatively double click a tool to select it:

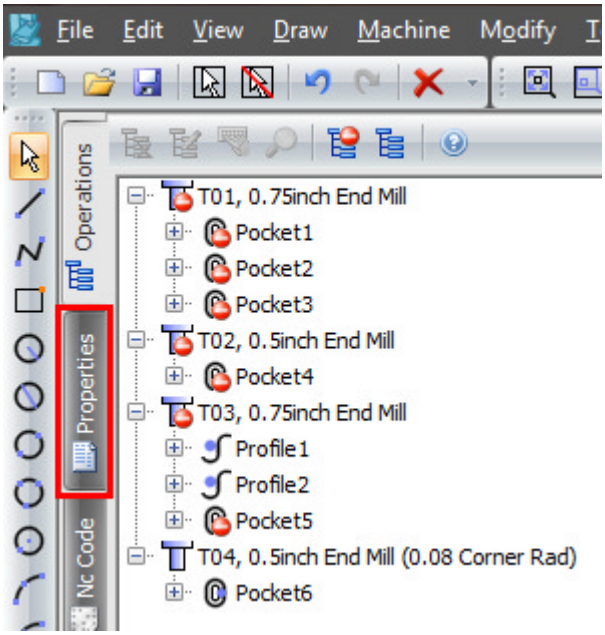


Before machining the slots suppress the operation that machined the Ø4.7 counterbore, so it does not obscure the view.
Choose the Operations tab on the Part Manager and right click on the last operation created by

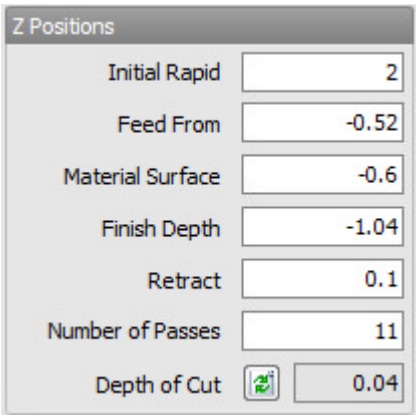
'T03, 0.75inch End Mill' and select Suppress:



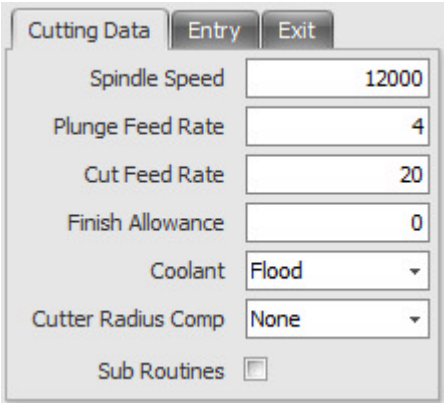
Click the Properties tab on the Part Manager to continue with the Profile command:



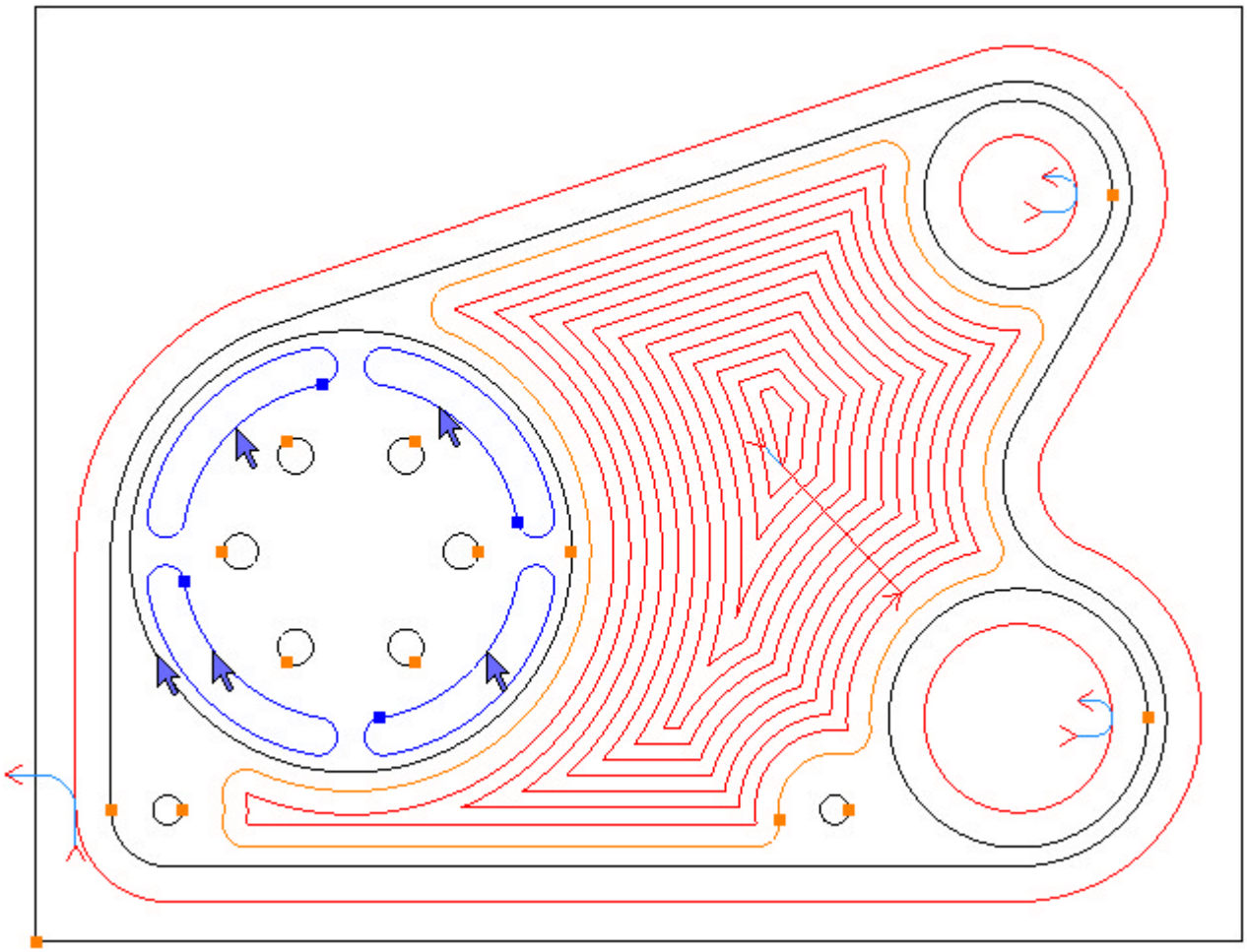
Enter the following values for the Z Positions:



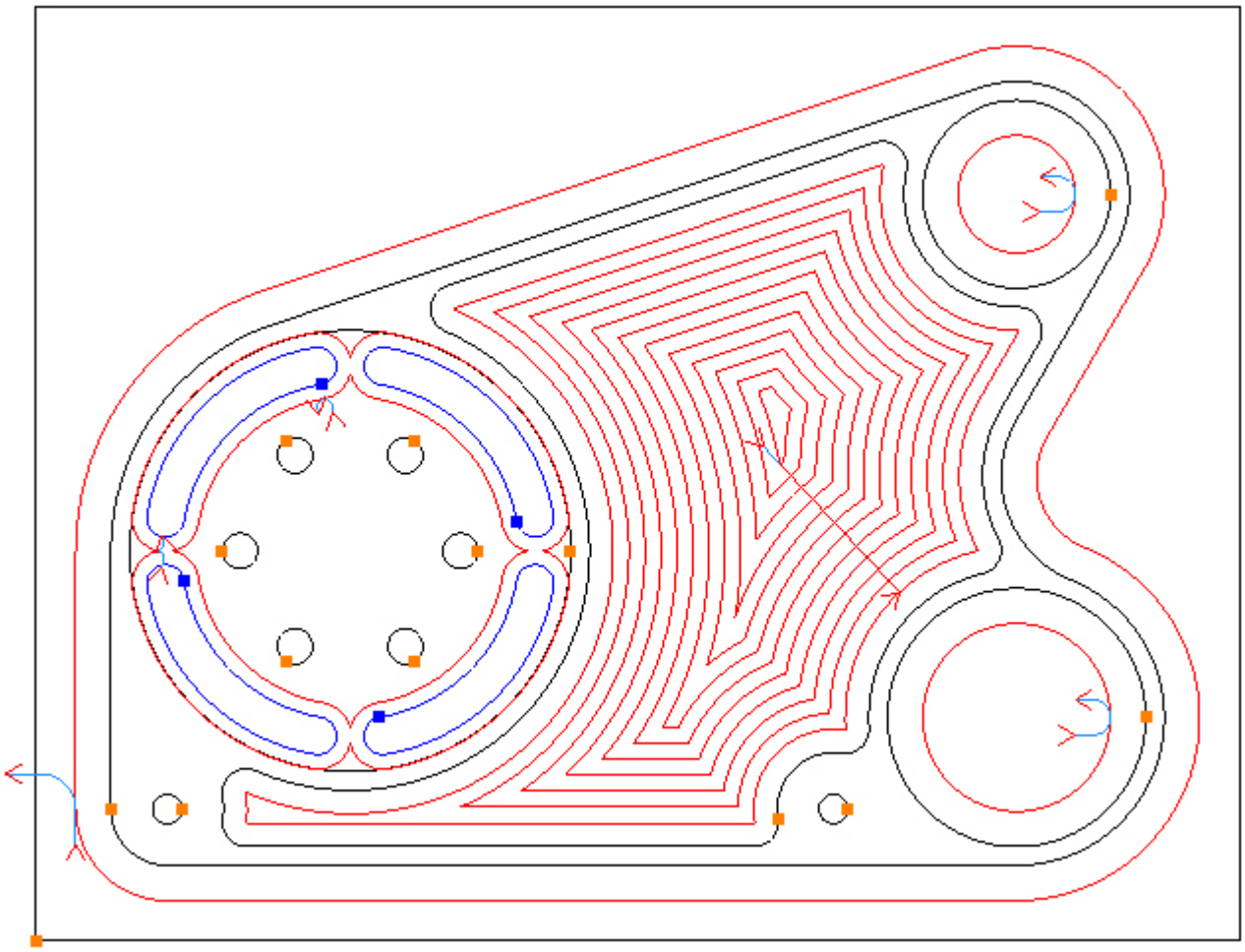
Enter the following values for the Cutting Data:



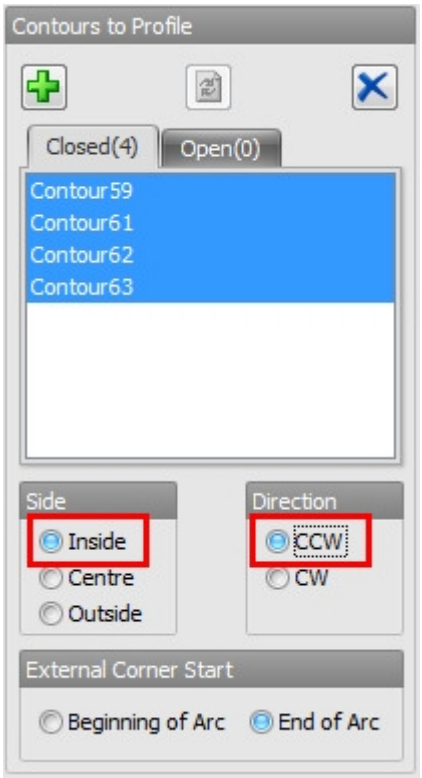
Select the slots by directly clicking them. Also unselect the Ø4.7 circle by directly clicking it, it was selected when it was suppressed:



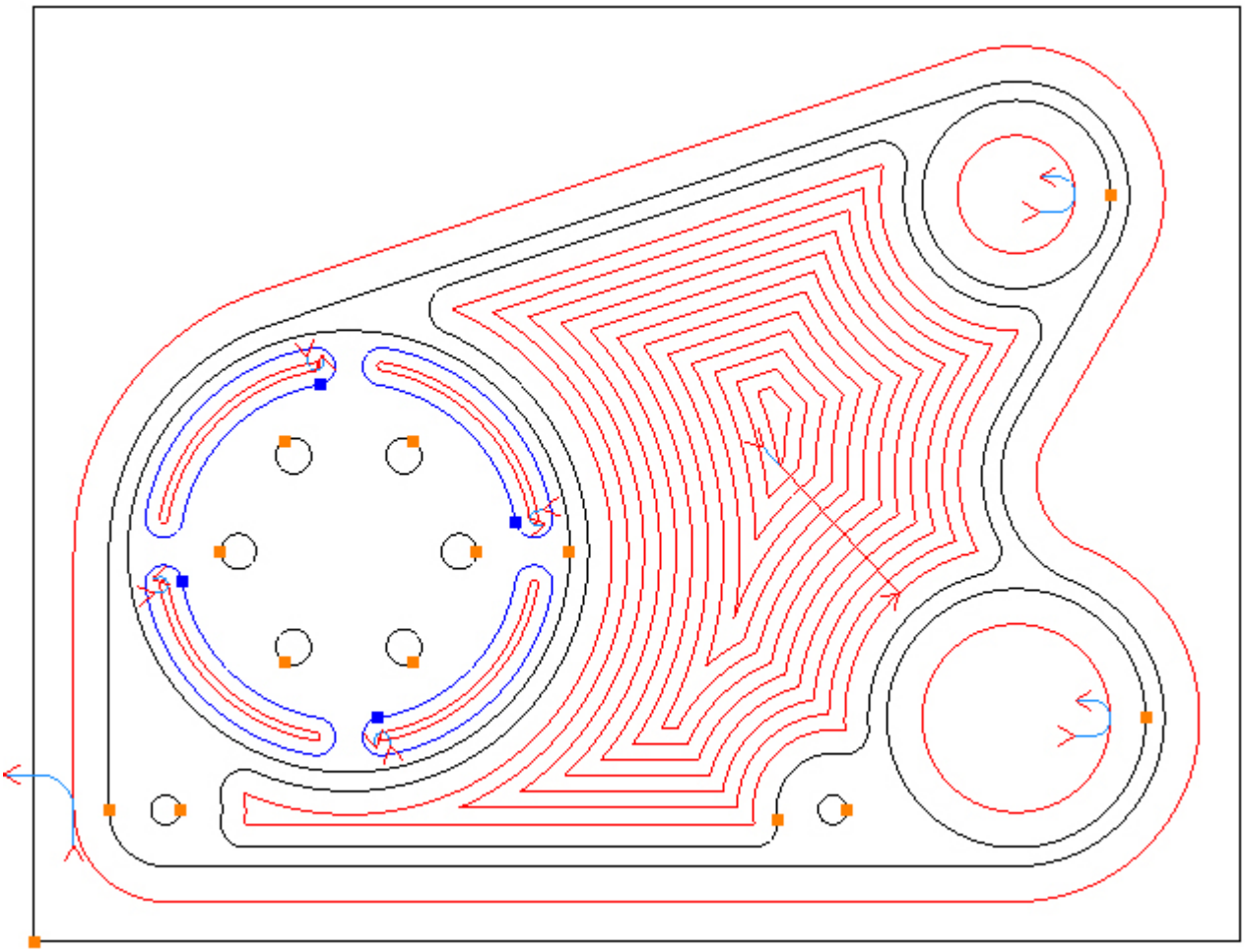
To profile the selected Contours click the Add button . The toolpaths are created:



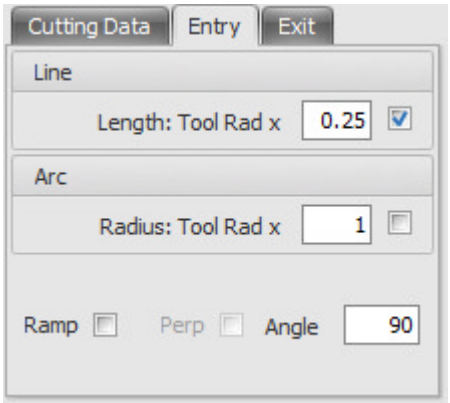
With the Contours still selected change the side to *Inside* and the direction to *CCW*:



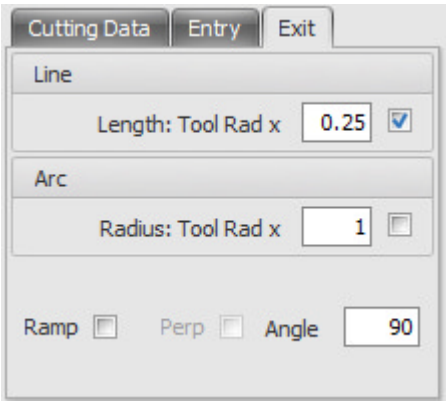
The toolpath is now on the inside and in counter-clockwise direction:



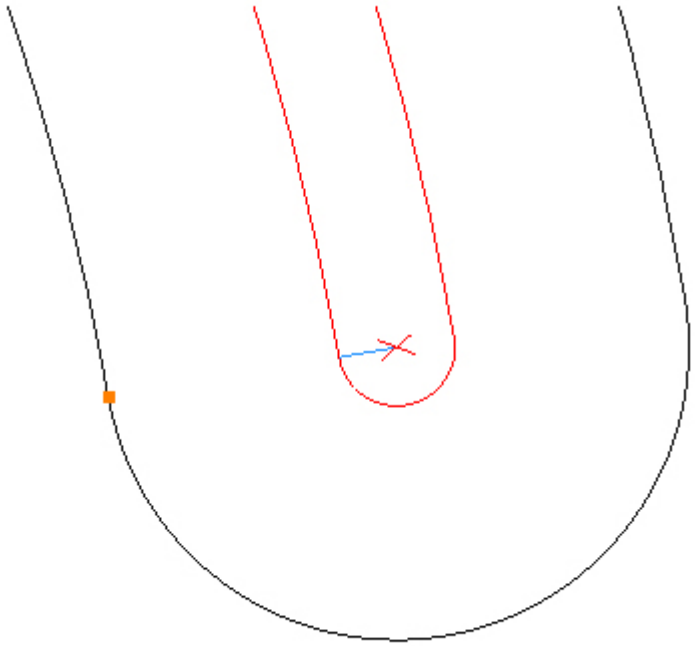
Choose the Entry tab and set as below, changes the values before checking/unchecking the boxes:



Choose the Exit tab and set the same as the entry:



The entry and exits have been added to all 4 slots:



Step 13 - Spot drill holes

Spot Drill holes using Drilling

Menu: Machine-> Drilling

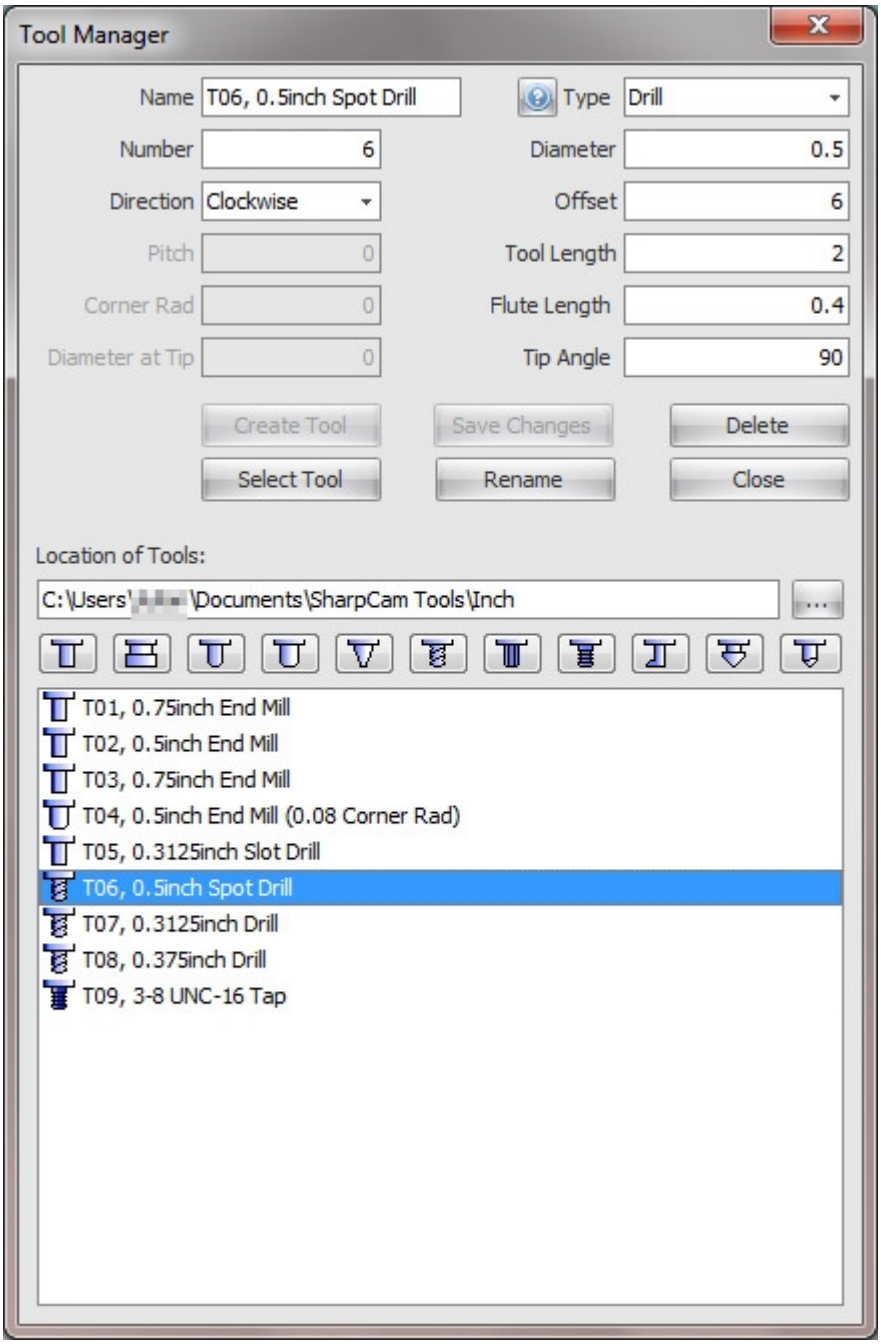
Toolbar button:

Choose the Drilling command then select the tool for this operation:

Menu: Machine-> Tool Manger

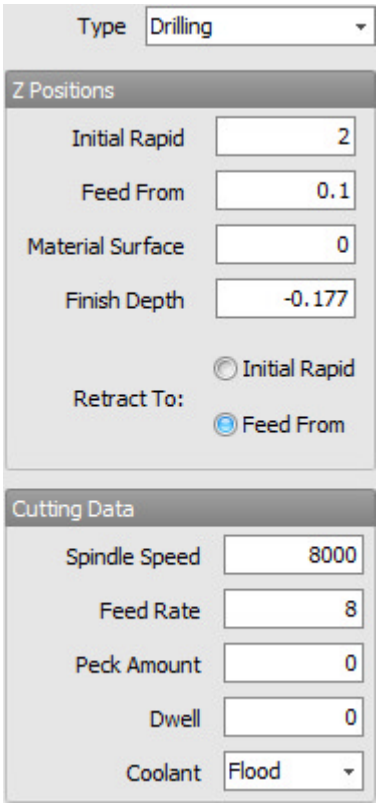
Toolbar button:

Choose the command to display the Tool Manager and click on 'T06, 0.5inch Spot Drill', then click the 'Select Tool' button. Alternatively double click a tool to select it:

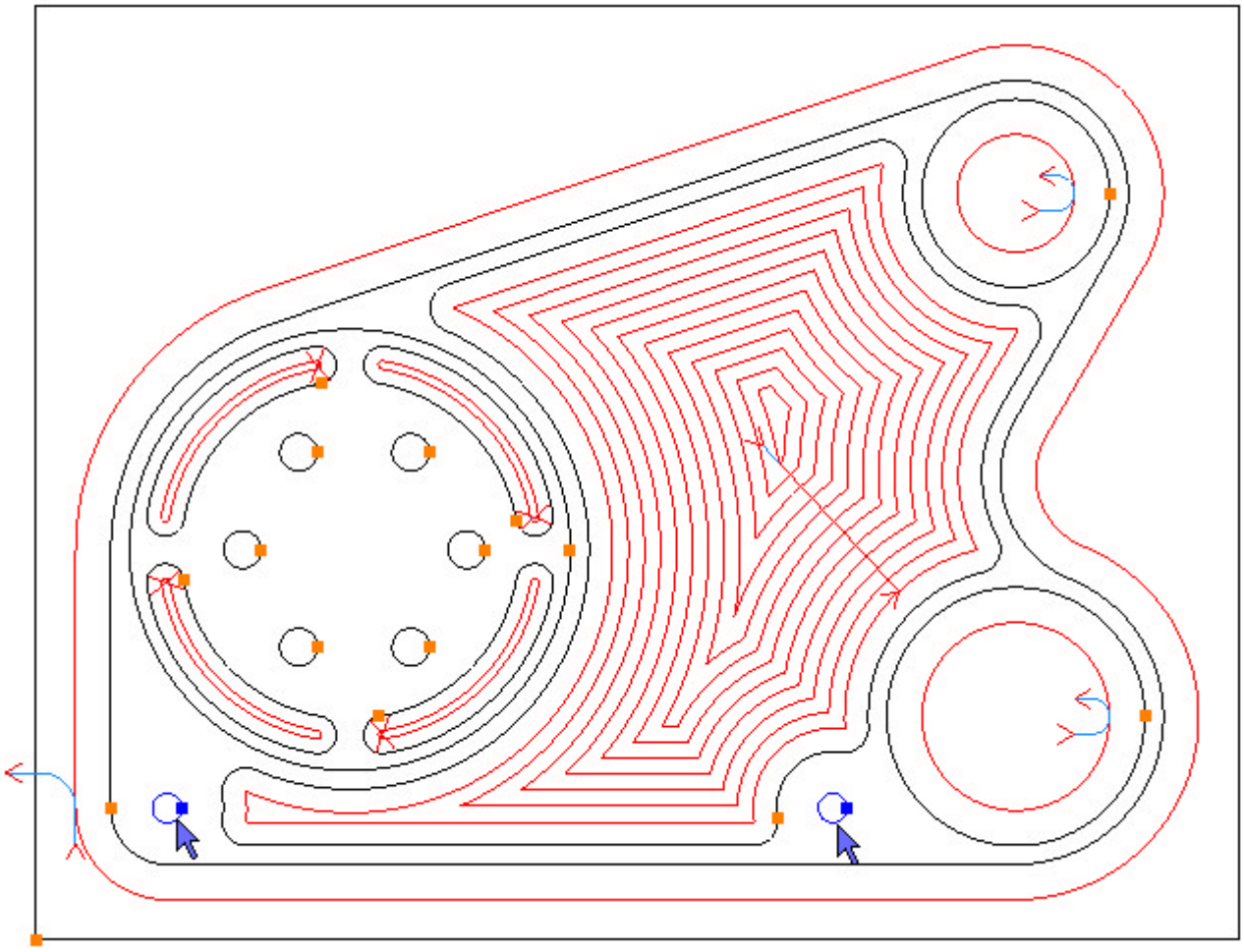


Enter the following Z Positions and Cutting Data as below:

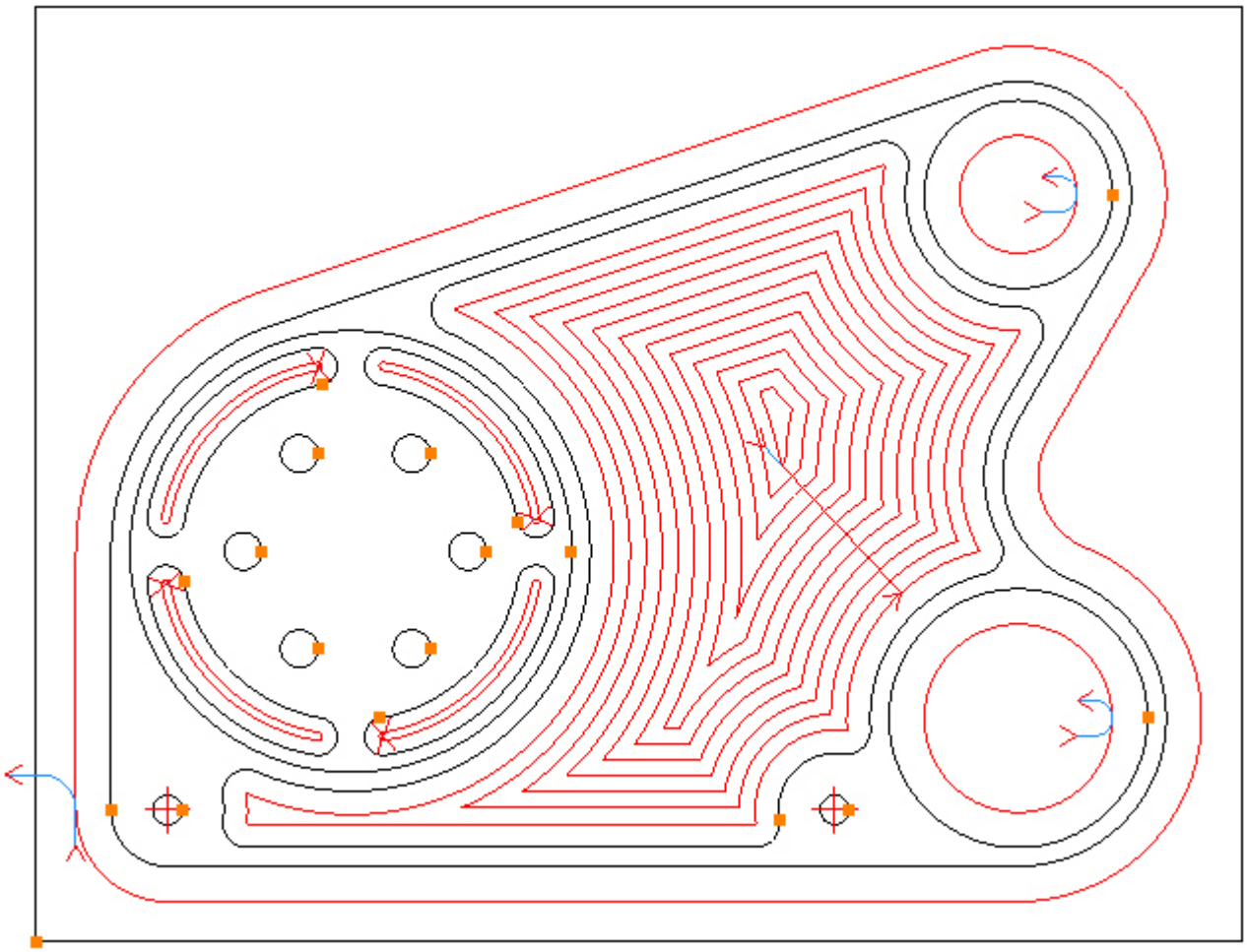
Gear Housing Tutorial



Select the Circles by directly clicking them:



To drill the selected Circles click the Add button . The toolpaths are created:



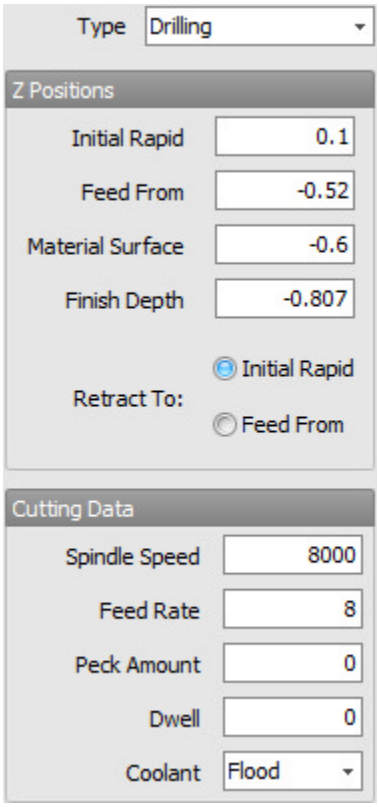
As the Z position of the Ø0.375 holes are different a new drilling operation is required.

Menu: Machine-> Drilling

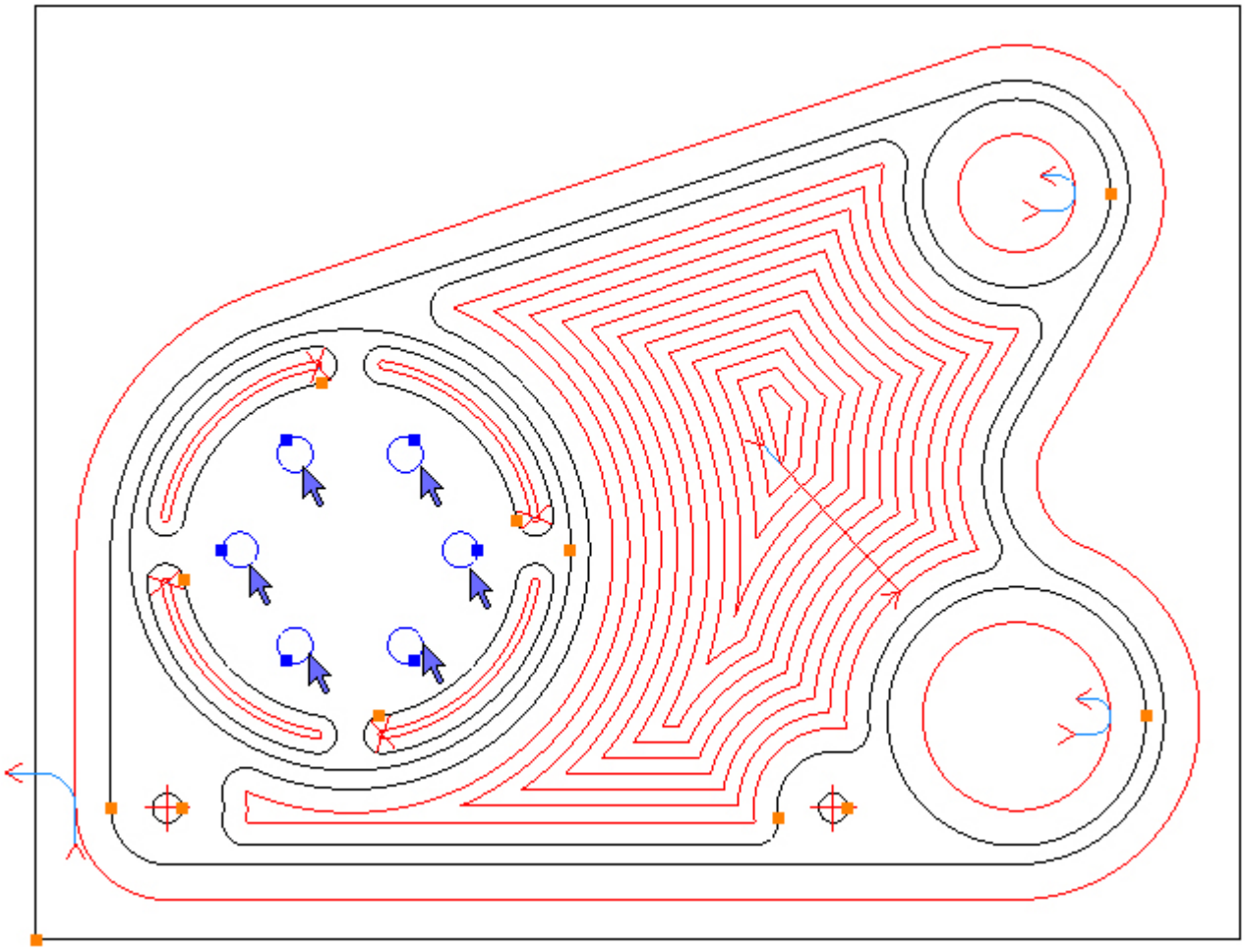
Toolbar button: 

Choose the Drilling command then enter the following values for the Z Positions:

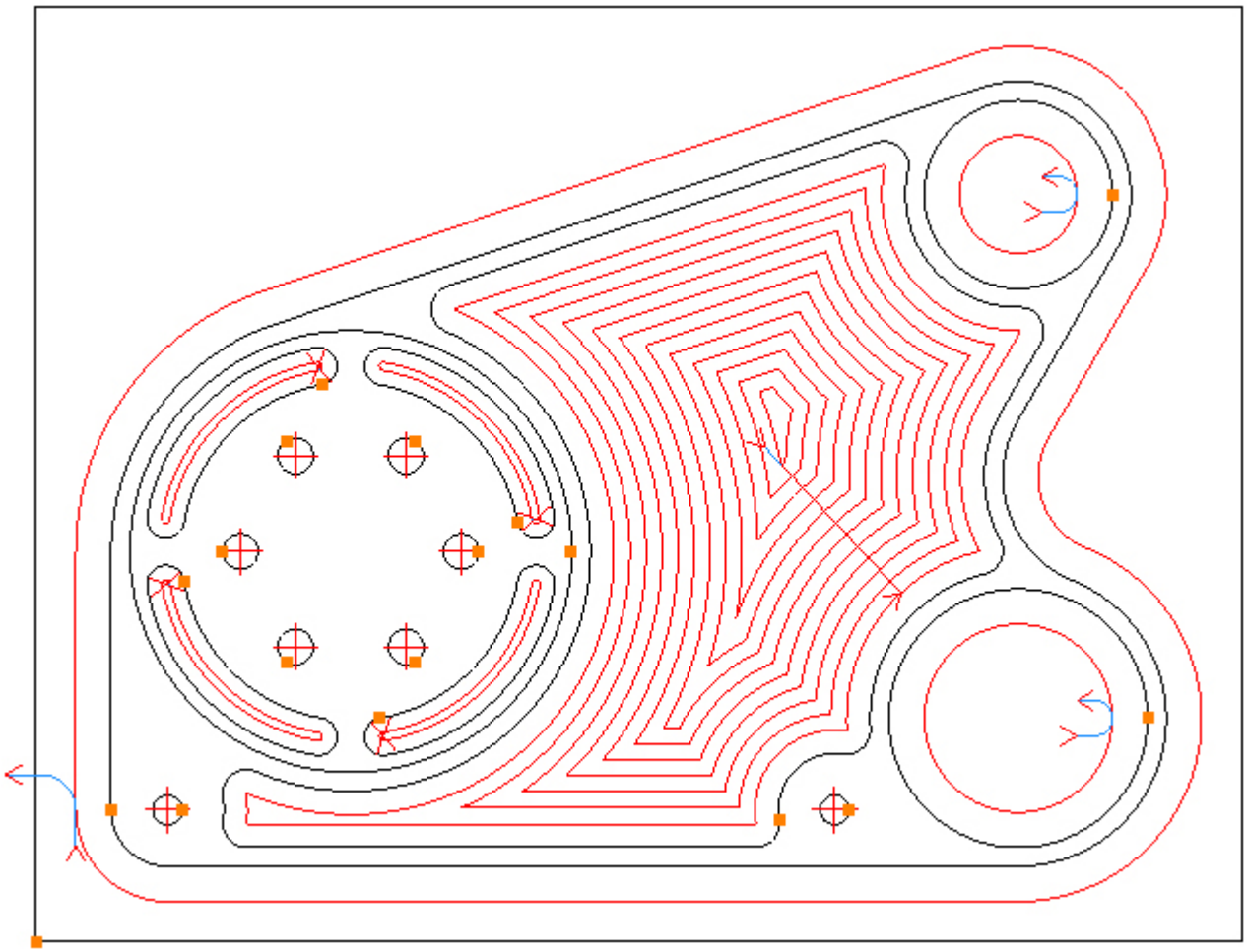
Gear Housing Tutorial



Select the Circles by directly clicking them:



To drill the selected Circles click the Add button  . The toolpaths are created:



Gear Housing Tutorial

Step 14 - Drill holes

Drill Ø0.3125 holes using Drilling

Menu: Machine-> Drilling

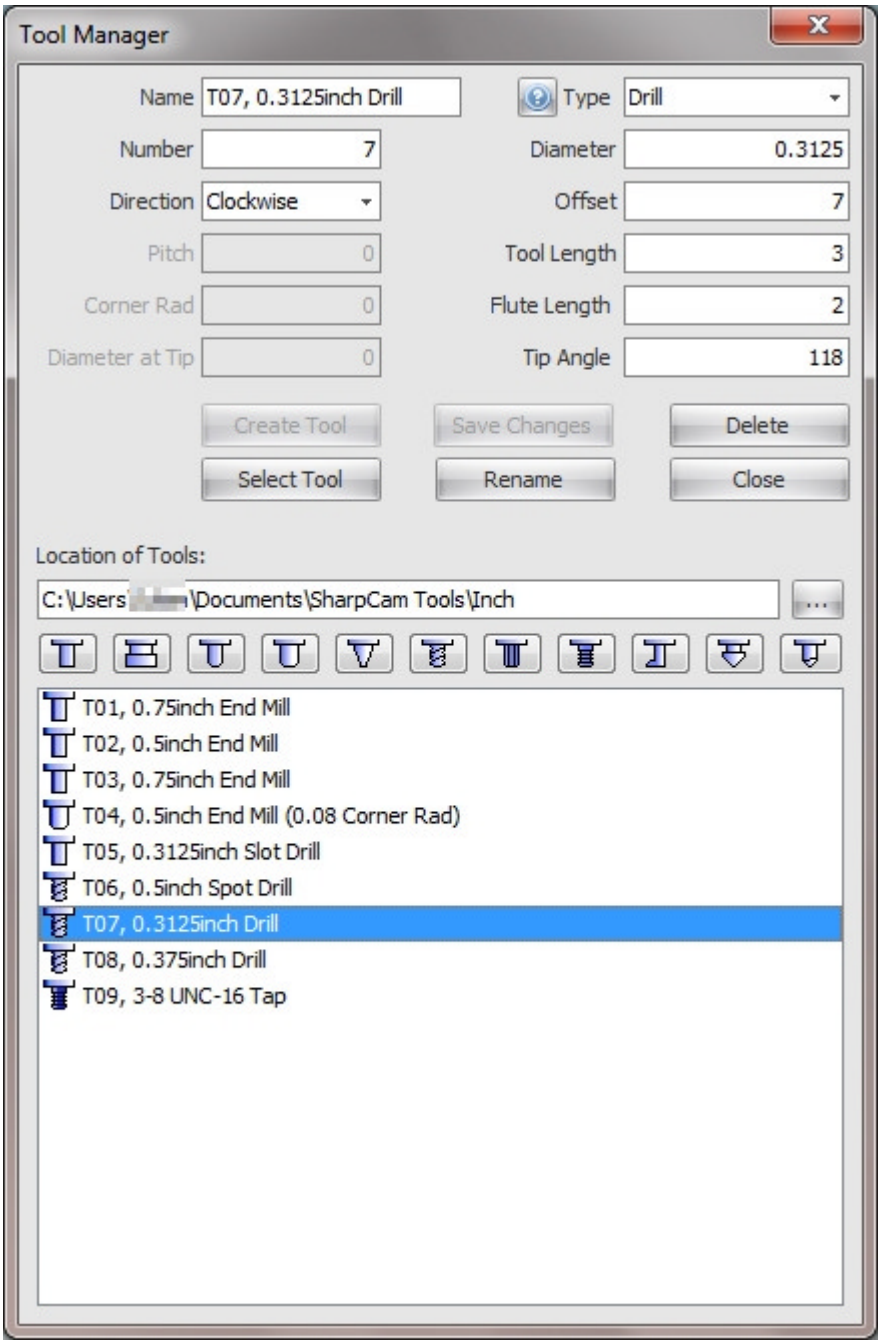
Toolbar button:

Choose the Drilling command then select the tool for this operation:

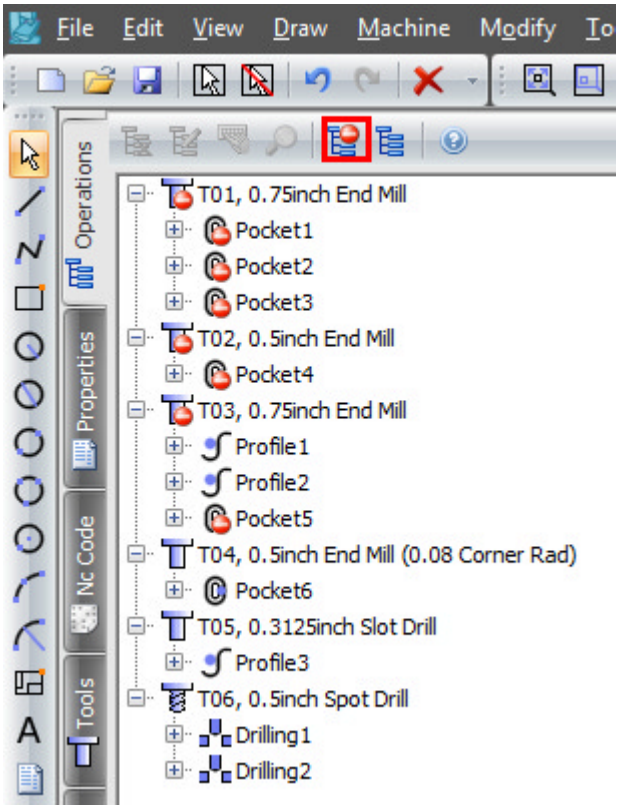
Menu: Machine-> Tool Manger

Toolbar button:

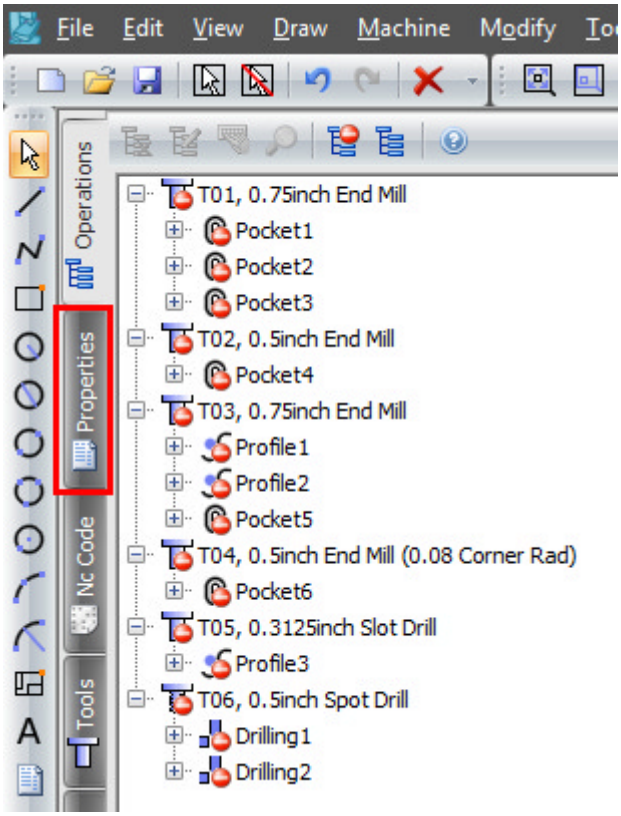
Choose the command to display the Tool Manager and click on 'T07, 0.3125inch Drill', then click the 'Select Tool' button. Alternatively double click a tool to select it:



The Part now has a number of toolpaths. In order to prevent the view from being obscured, suppress them so they are hidden. Choose the Operations tab on the Part Manager and click the Suppress All button:

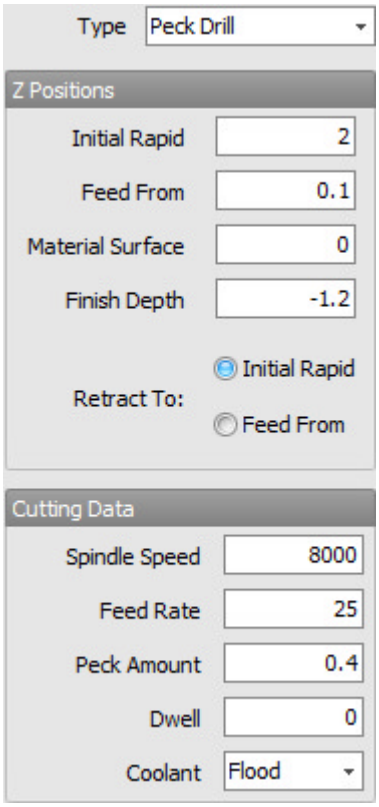


Click the Properties tab on the Part Manager to continue with the Drilling command:

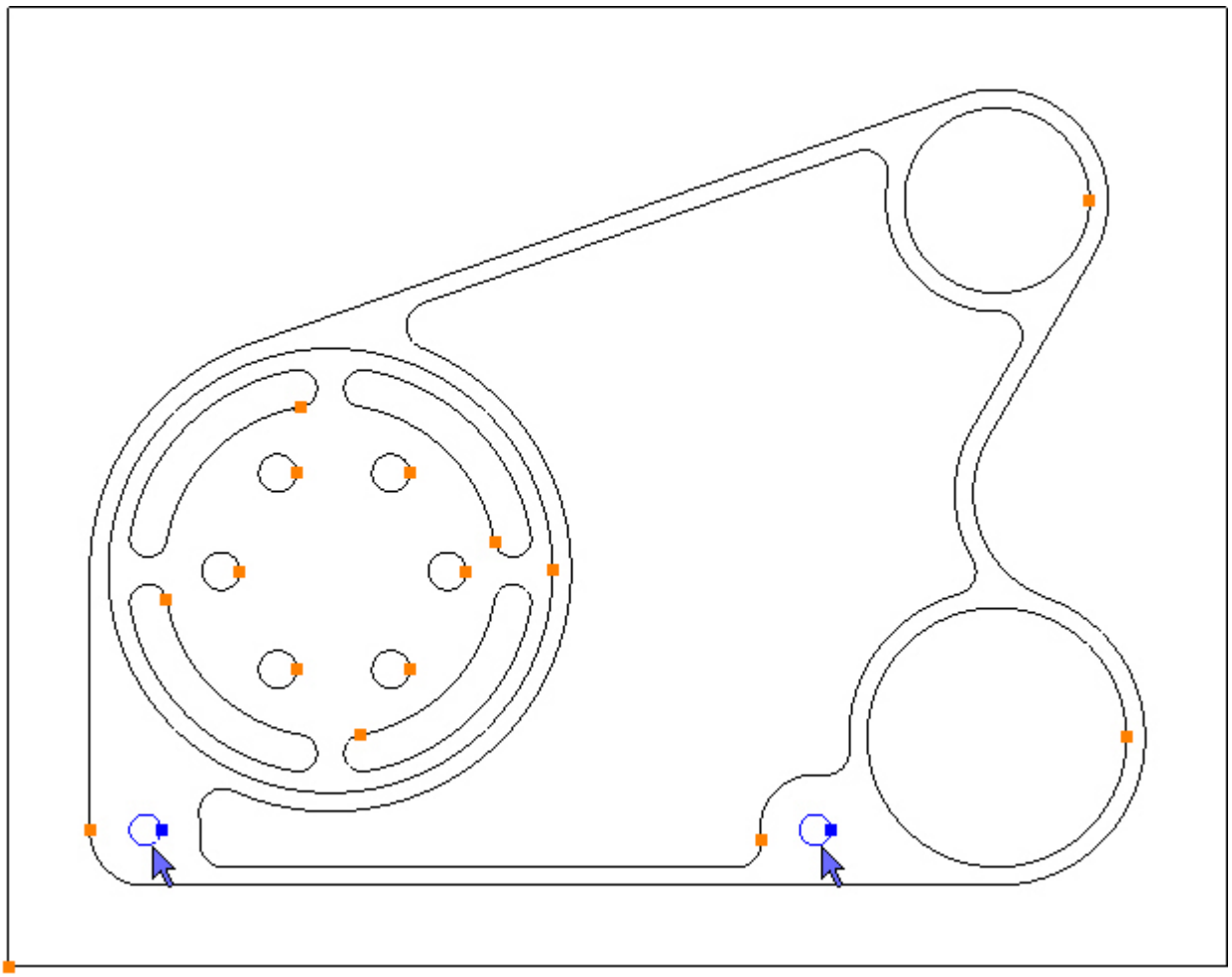


Enter the following Z Positions and Cutting Data as below, note that the type is Peck Drill:

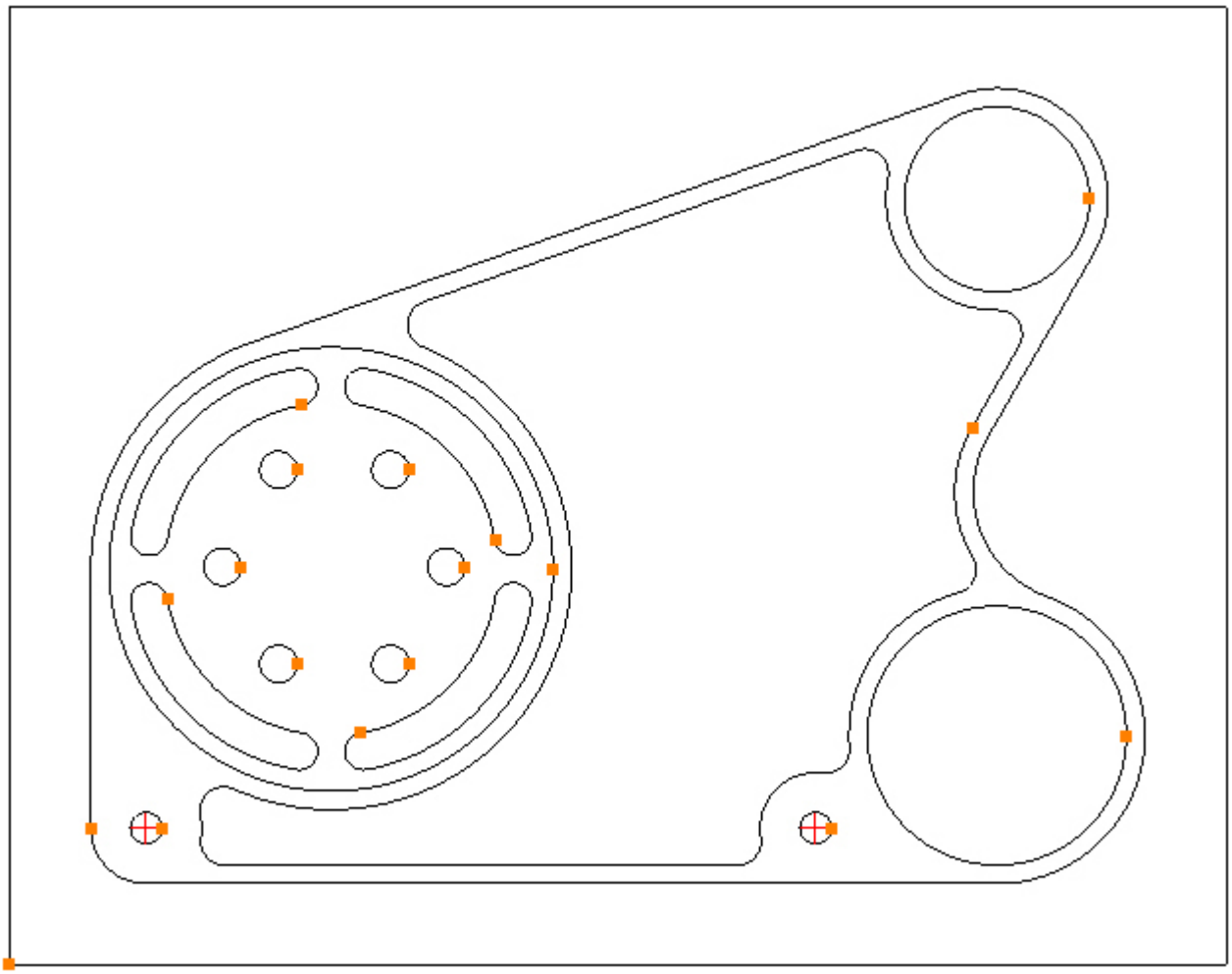
Gear Housing Tutorial



Select the Circles by directly clicking them:



To drill the selected Circles click the Add button  . The toolpaths are created:



Gear Housing Tutorial

Step 15 - Drill holes

Drill Ø0.375 holes using Drilling

Menu: Machine-> Drilling

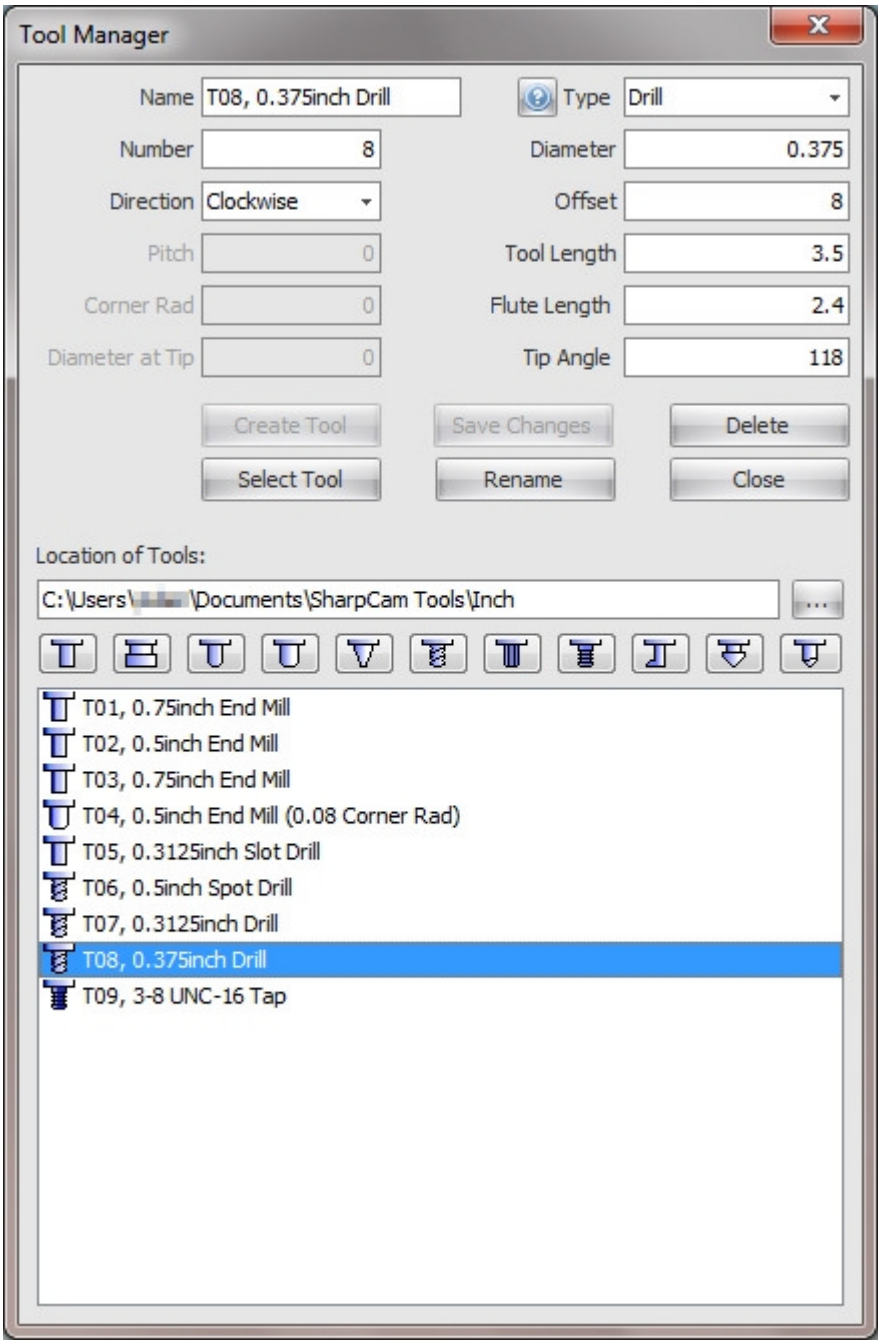
Toolbar button:

Choose the Drilling command then select the tool for this operation:

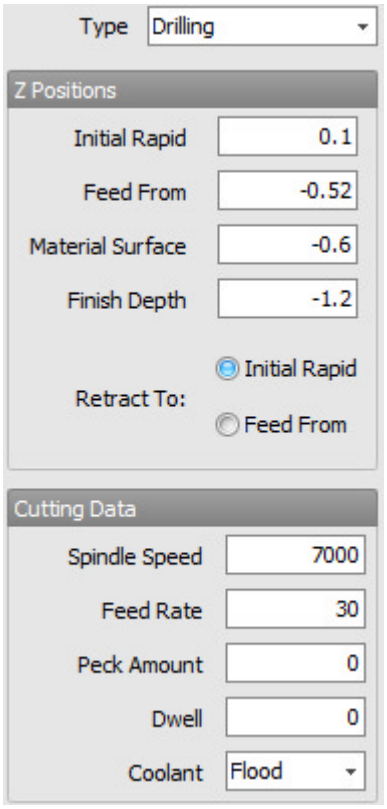
Menu: Machine-> Tool Manger

Toolbar button:

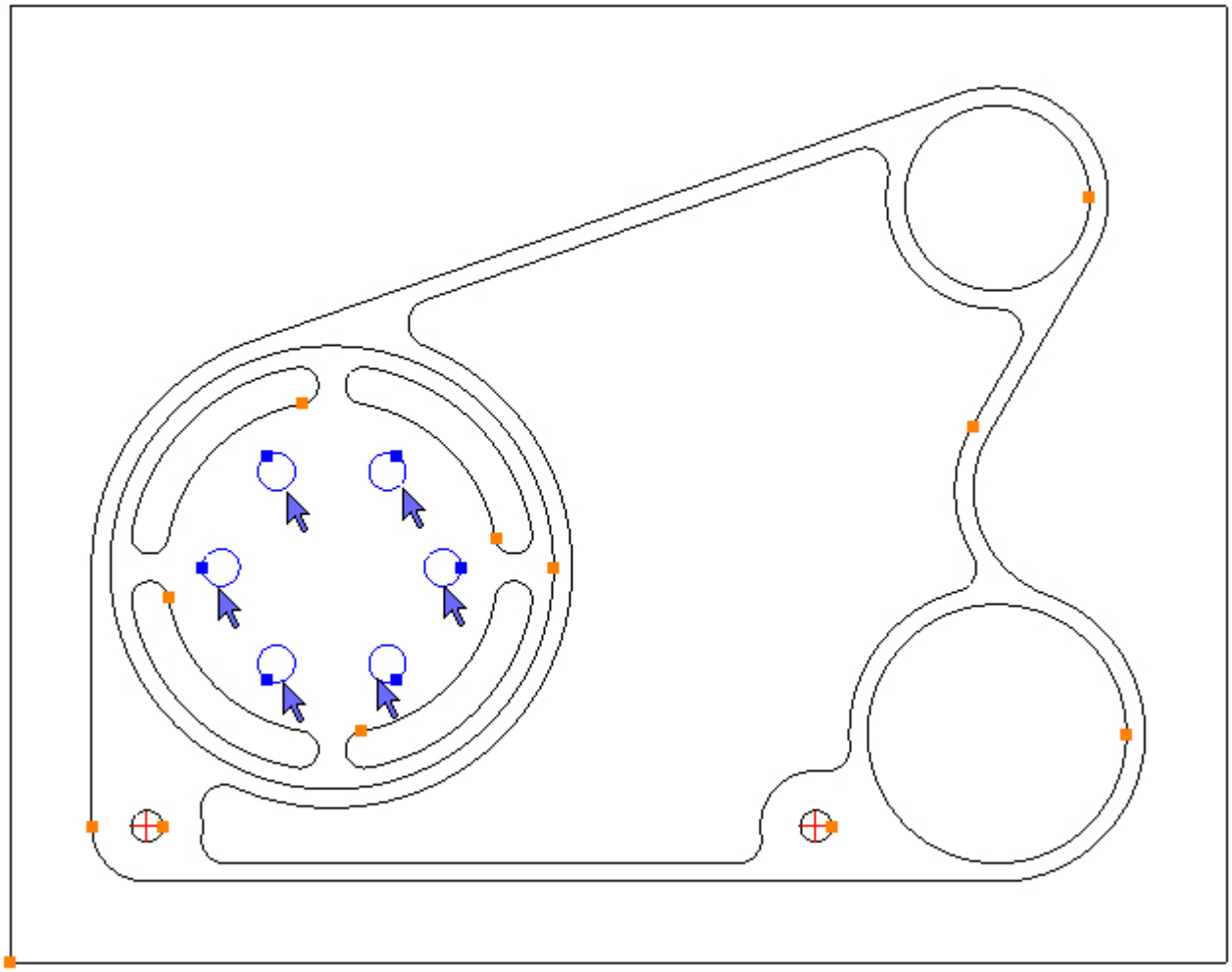
Choose the command to display the Tool Manager and click on 'T08, 0.375inch Drill', then click the 'Select Tool' button. Alternatively double click a tool to select it:



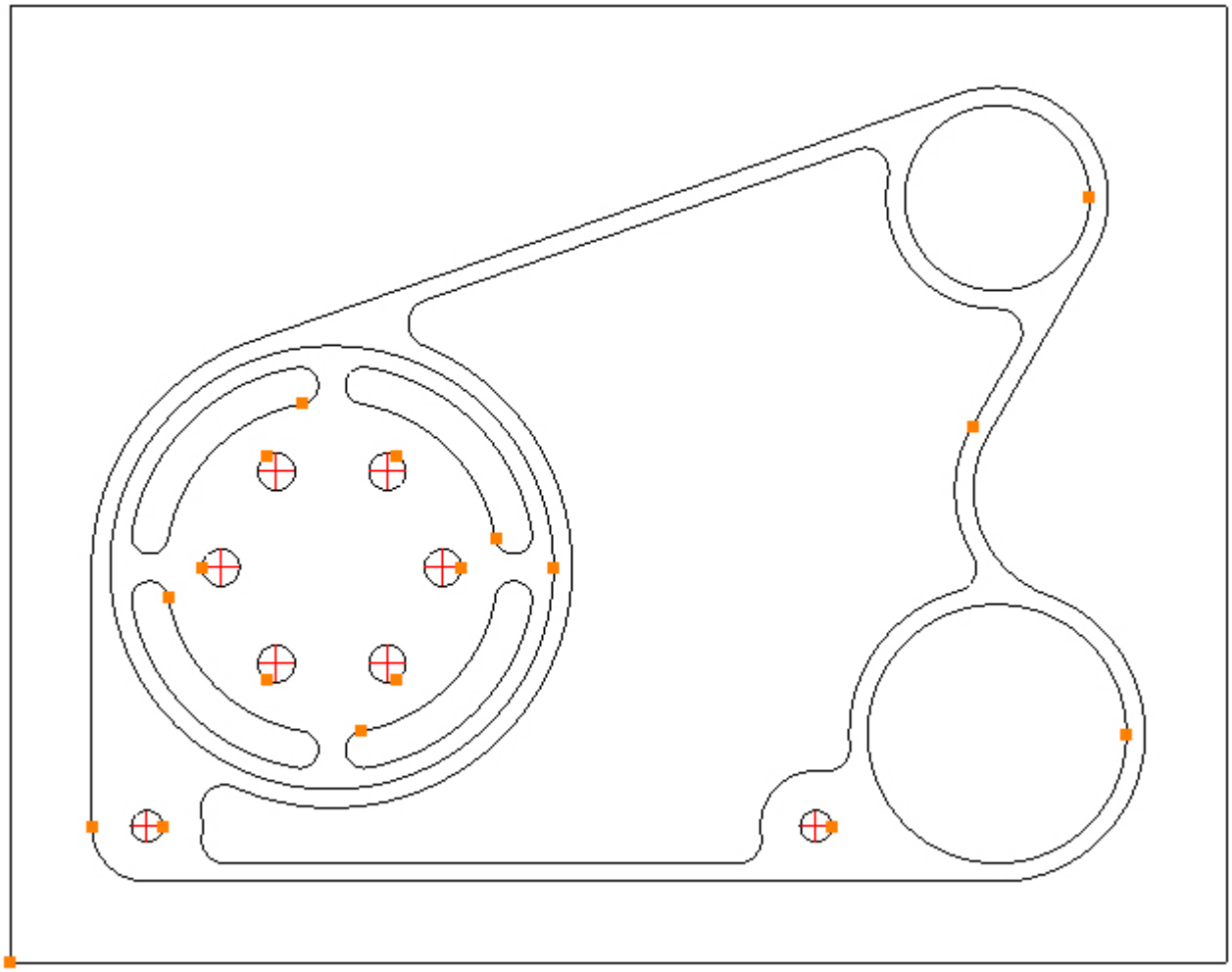
Enter the following Z Positions and Cutting Data as below, note that the type is Drilling:



Select the Circles by directly clicking them:



To drill the selected Circles click the Add button  . The toolpaths are created:



All toolpaths have now been created.

Step 16 - Create Material Stock

Create Material Stock for Solid Simulation

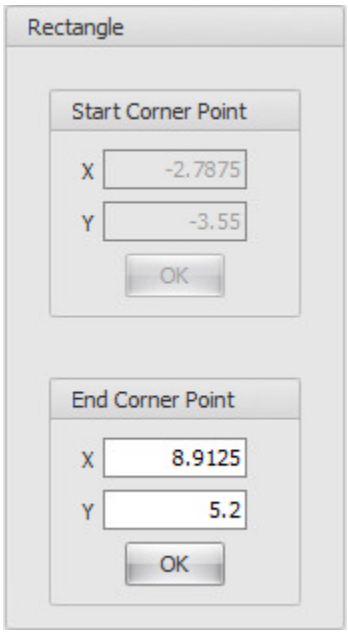
First a rectangle must be created that represents the Material Stock. As mentioned in Step 1 the Gear Housing is to be made from an aluminium rectangular billet 11.7" x 8.75" x 1.2".

Choose the Rectangle command:

Menu: Draw -> Rectangle

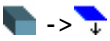
Toolbar button: 

Create a rectangle with a start point value of X-2.7875, Y-3.55 and an end point value of X8.9125, Y5.2:

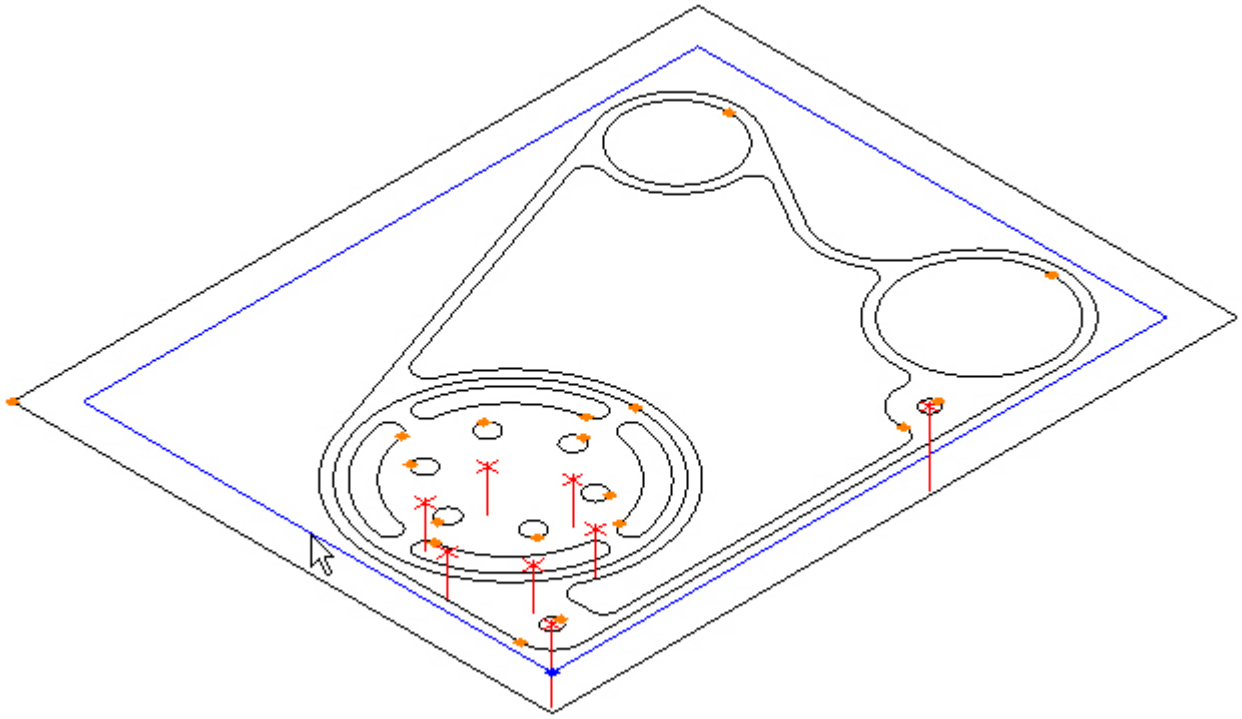


Extrude the rectangle

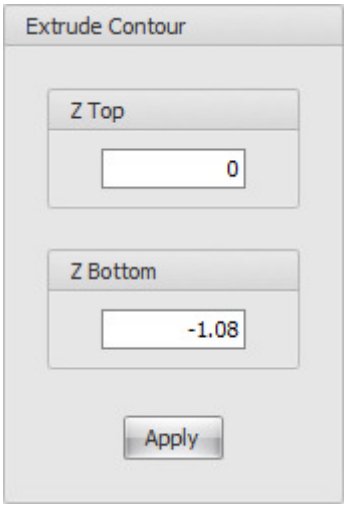
Menu: View-> Material Stock -> Extrude Contours

Toolbar button: 

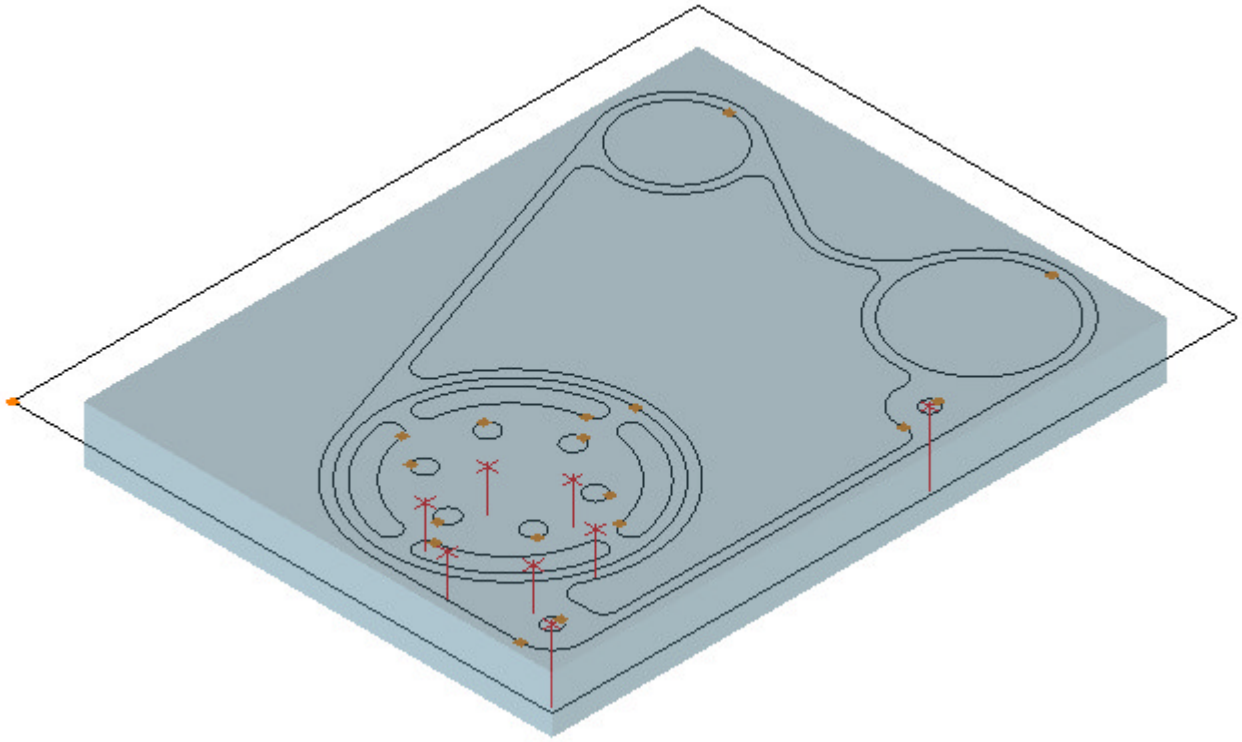
Select the Rectangle by directly clicking it:



Enter -1.08 in Z Bottom:



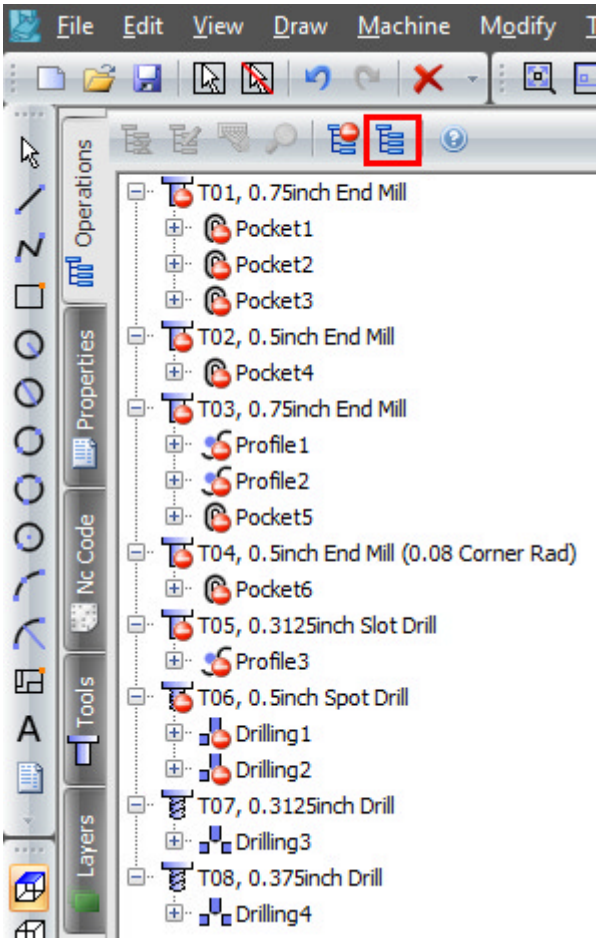
Click the Apply button, the Material Stock is created:



Gear Housing Tutorial

Step 17 - Solid Simulation

Before simulating the part the operations must be unsuppressed. Choose the Operations tab on the Part Manager and click the Unsuppress All button:

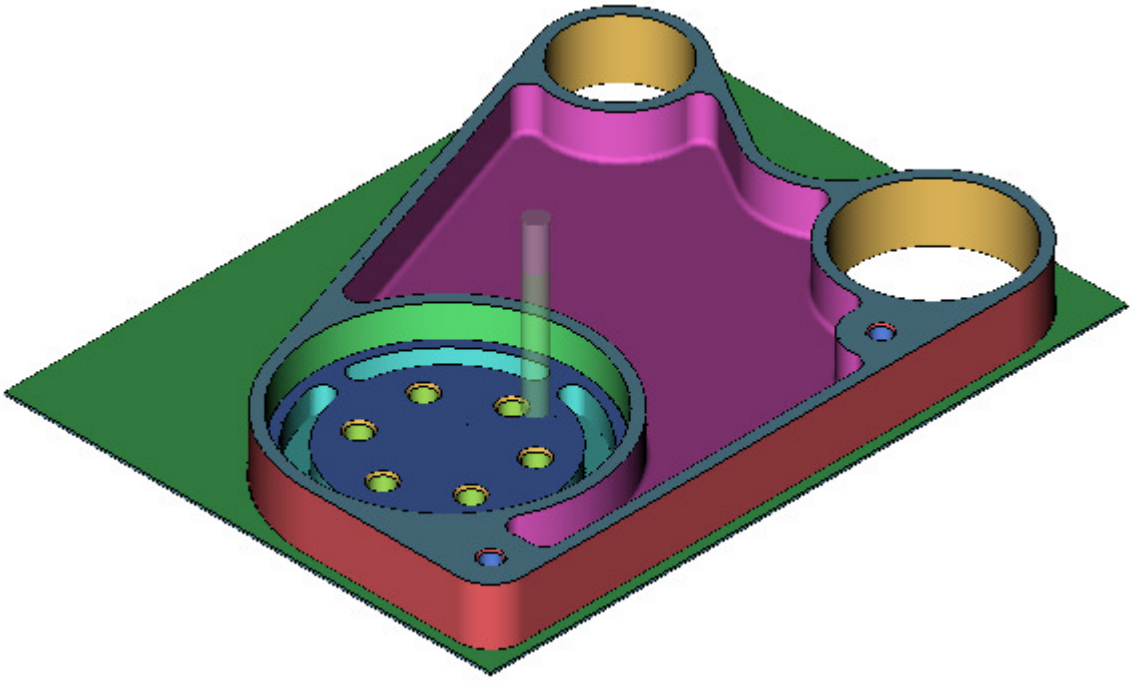


Simulate part being machined

Menu: View-> Solid Simulation

Toolbar button:

There are many options and you are encouraged to experiment, but for now you can press either the Forwards button or the Fast Forwards button. The part is simulated until the end:



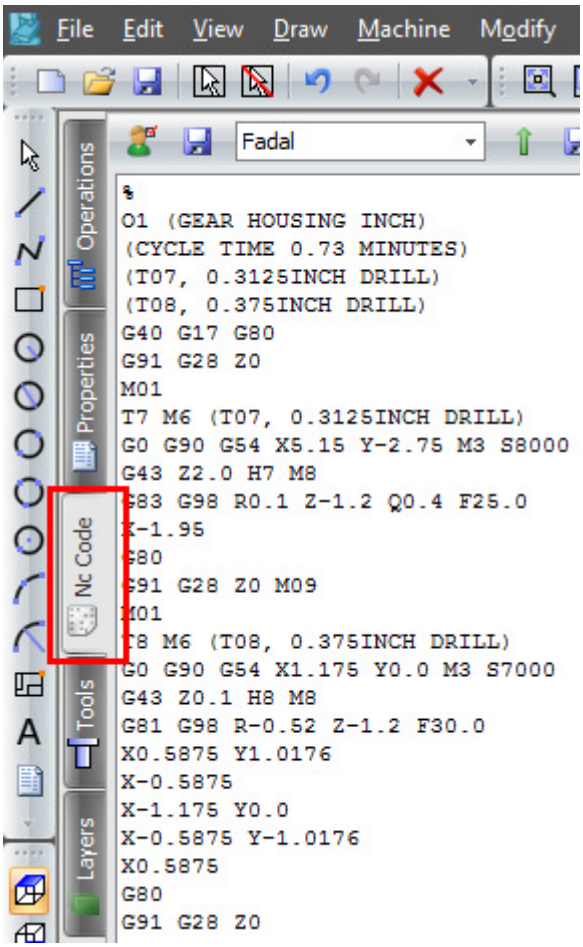
Simulating the Part is optional and does not need to be performed to create the Nc code.

Gear Housing Tutorial

Step 18 - Output Nc code

Outputting Nc Code

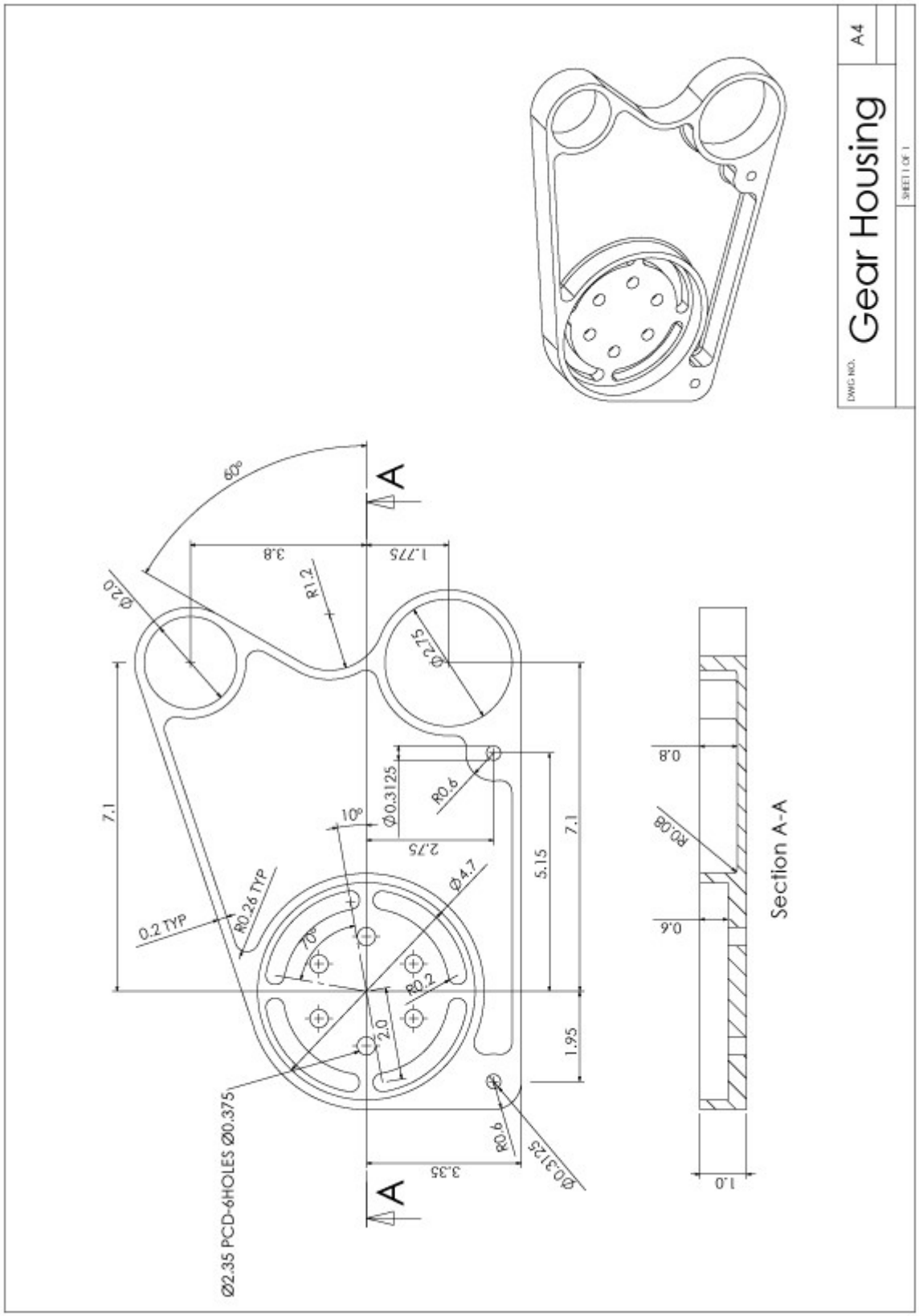
Choose the Nc Code tab on the Part Manager :



There are two choices:

- Click the button and save the Nc code to disk.
- Click the button and send directly to the machine, using RS232 communications.

Gear Housing Drawing



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