

Table of Contents


Opening Solid Model File	2-4
Create the Contours	5
Step 1 - Extract Contours from Model	5-9
Step 2 - Set Contour Heights	9-23
Create the Toolpaths	24
Step 1 - Process plan	24
Step 2 - Set start point	24-27
Step 3 - Prepare for roughing	27-30
Step 4 - Rough outside profile	30-38
Step 5 - Rough counterbore	38-41
Step 6 - Rough bores	41-44
Step 7 - Rough inside	44-47
Step 8 - Finish outside profile	47-54
Step 9 - Finish bores	54-60
Step 10 - Finish counterbore	60-63
Step 11 - Finish inside	63-66
Step 12 - Machine radial slots	66-73
Step 13 - Spot drill holes	73-78
Step 14 - Drill holes	78-83
Step 15 - Drill holes	83-87
Step 16 - Create Material Stock	87-89
Step 17 - Solid Simulation	89-91
Step 18 - Output Nc code	91-92
Gear Housing Drawing	93
1. Index	94-95

Gear Housing 3D Tutorial

Opening Solid Model File

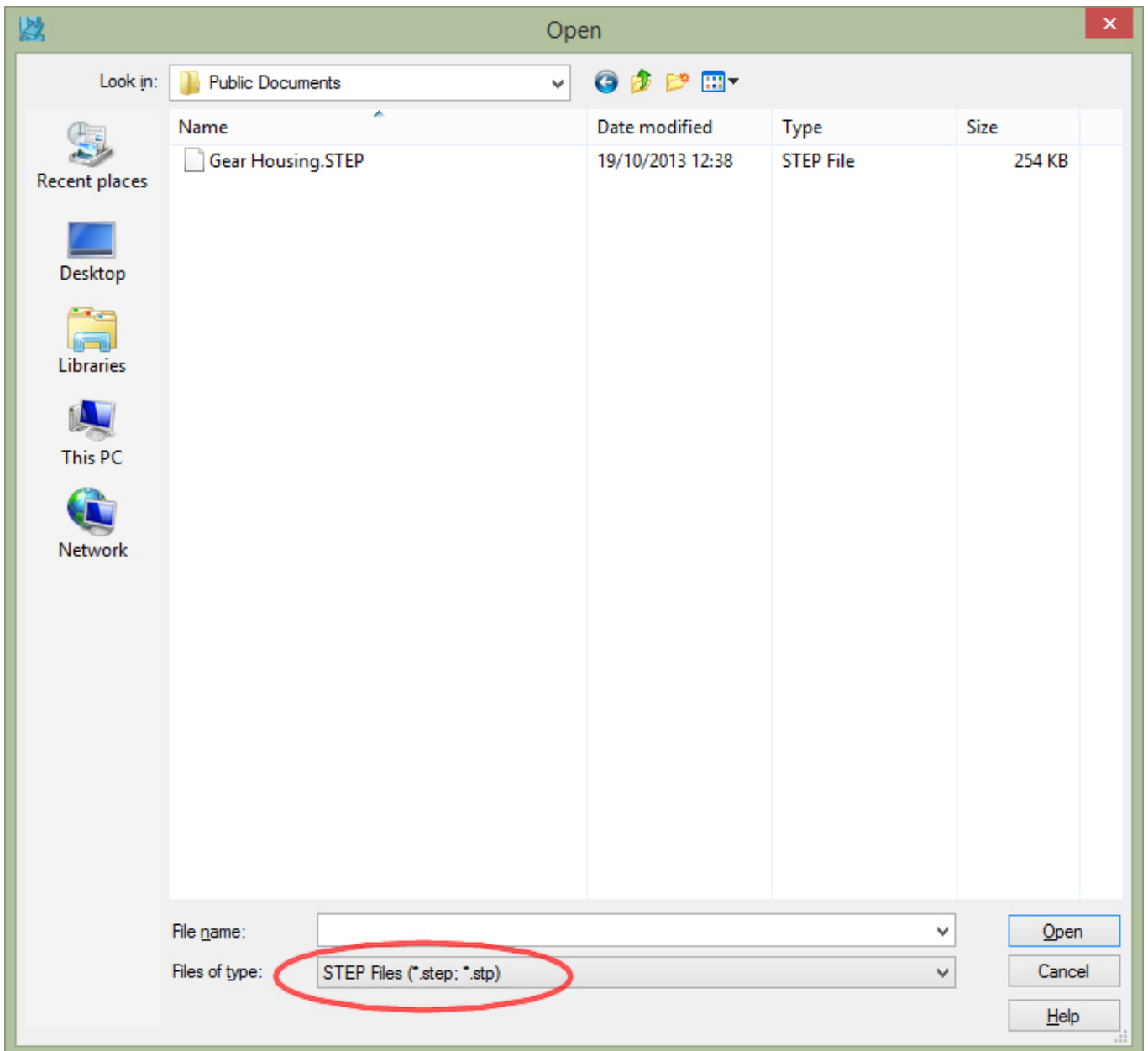
Download the accompanying STEP file - Gear Housing.STEP to a location of your choice.

Menu: File -> Open

Toolbar button: 

Shortcut: Ctrl + O

Choose the File type to open - STEP Files, as shown below:



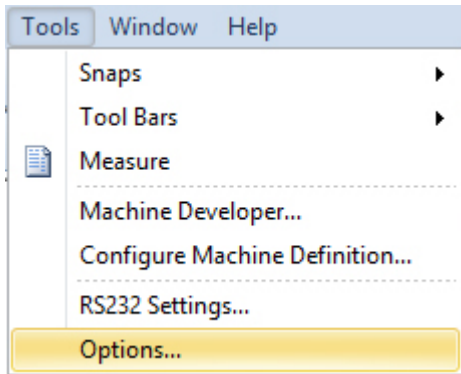
Navigate to the downloaded file, then select the *Gear Housing.STEP* file and click the Open button.

SharpCam will read the units from the STEP file and set the units for the opened file as required.

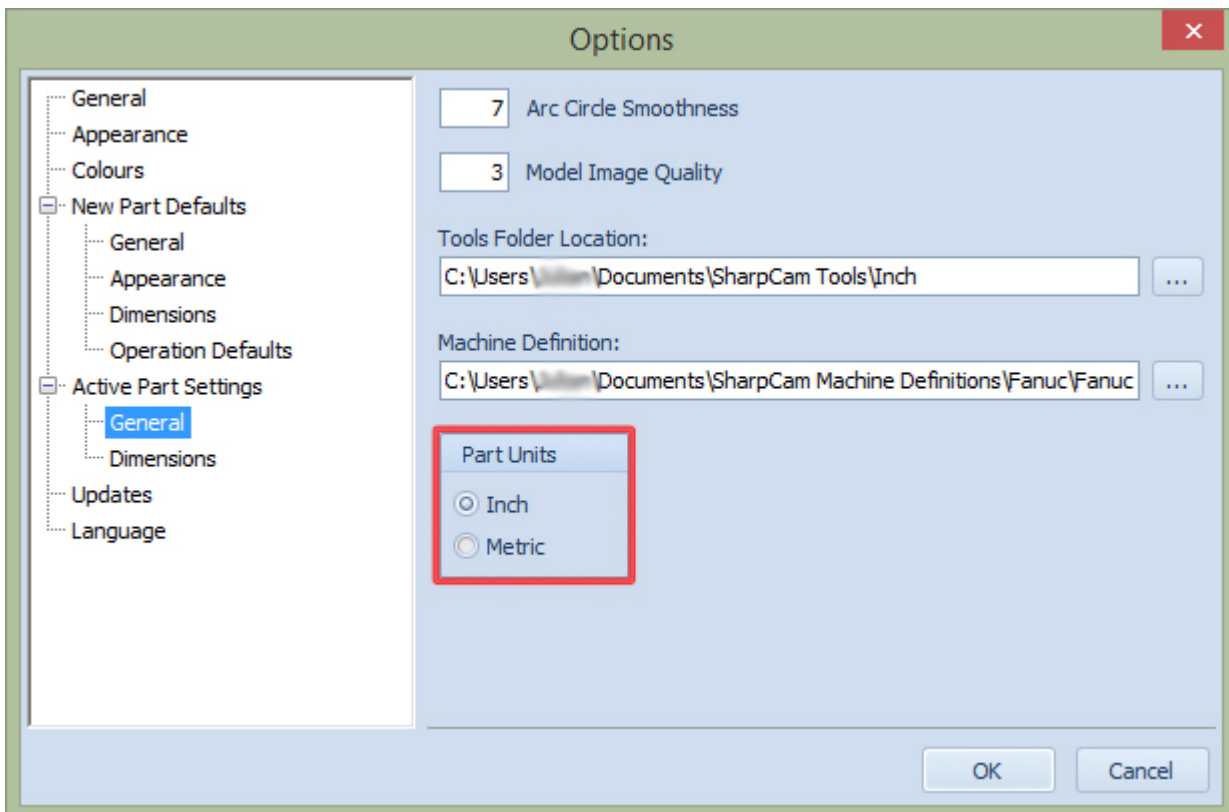
Units

The units stored in any STEP file may not be correct and are arbitrary. It is always best to check the unit settings for the opened file:

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



Select the Active Part Settings node and make sure that the correct units are selected and click the OK button:



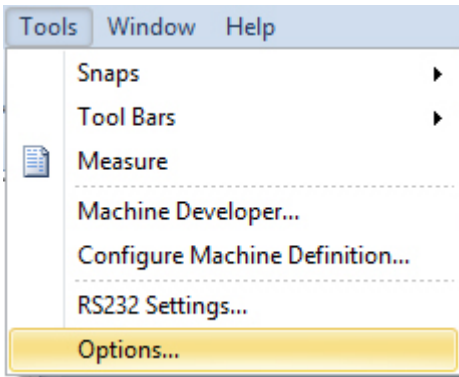
A confirmation message box will be shown if the units are changed, click OK to accept.

Opening other types of CAD files

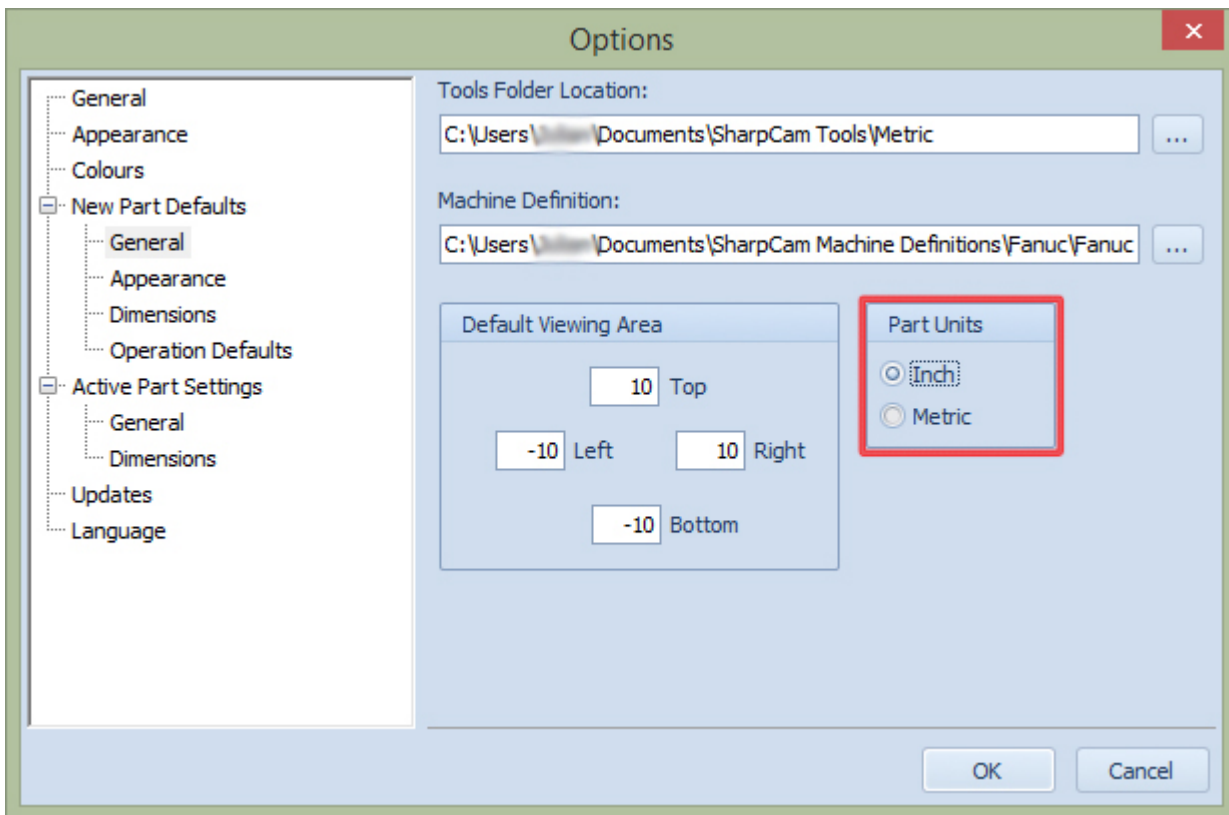
Dxf/dwg/STL file units cannot be read, the opened file will adopt the units set for a new Part, see below. Be sure to change the units if required, see above.

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

Gear Housing 3D Tutorial



Select the New Part Defaults node. The Part Units group indicates the units that other CAD files will adopt by default:



Other Settings

For this tutorial it is assumed that the following buttons are depressed - as indicated by the orange appearance:



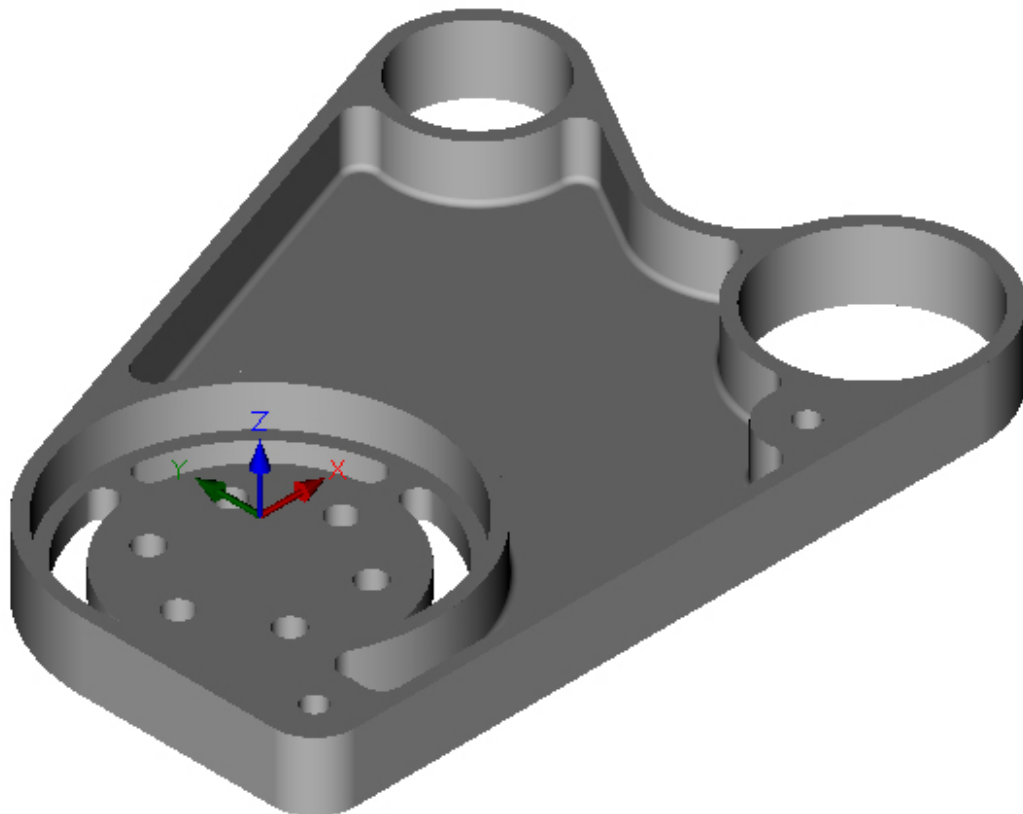
Create the Contours


Step 1 - Extract Contours from Model

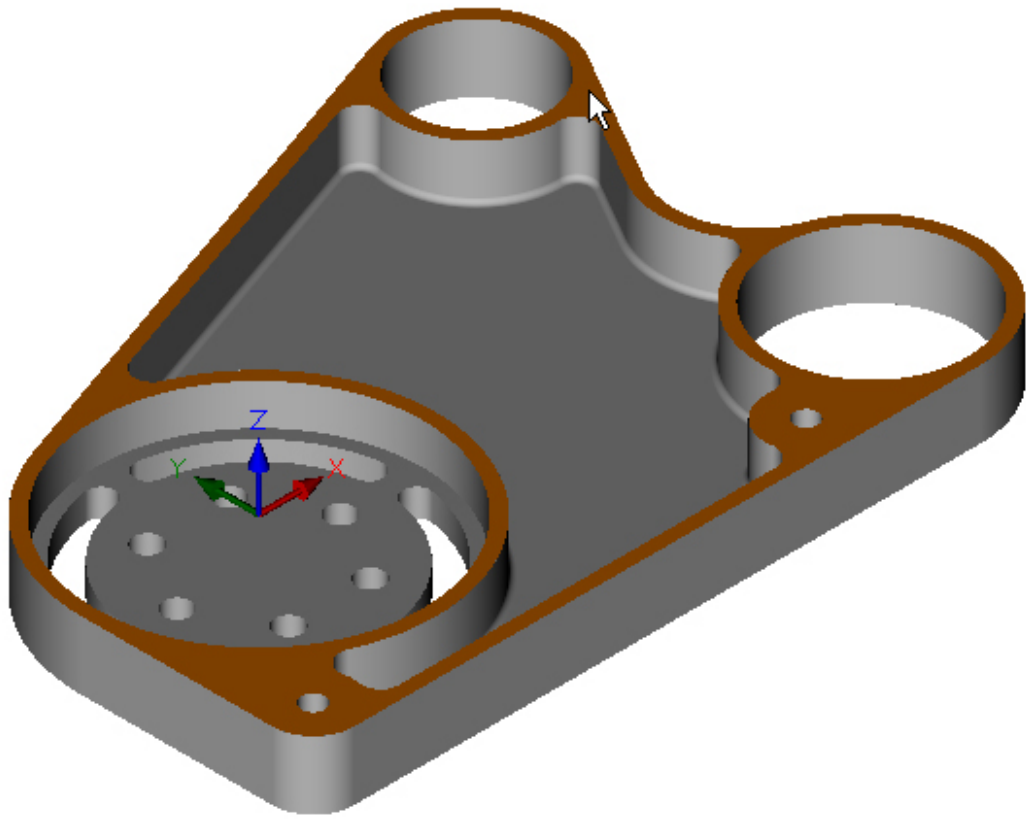
Rather than drawing the Contours, we will extract them from the Solid Model

Extract Contours using Extract Contours from Face

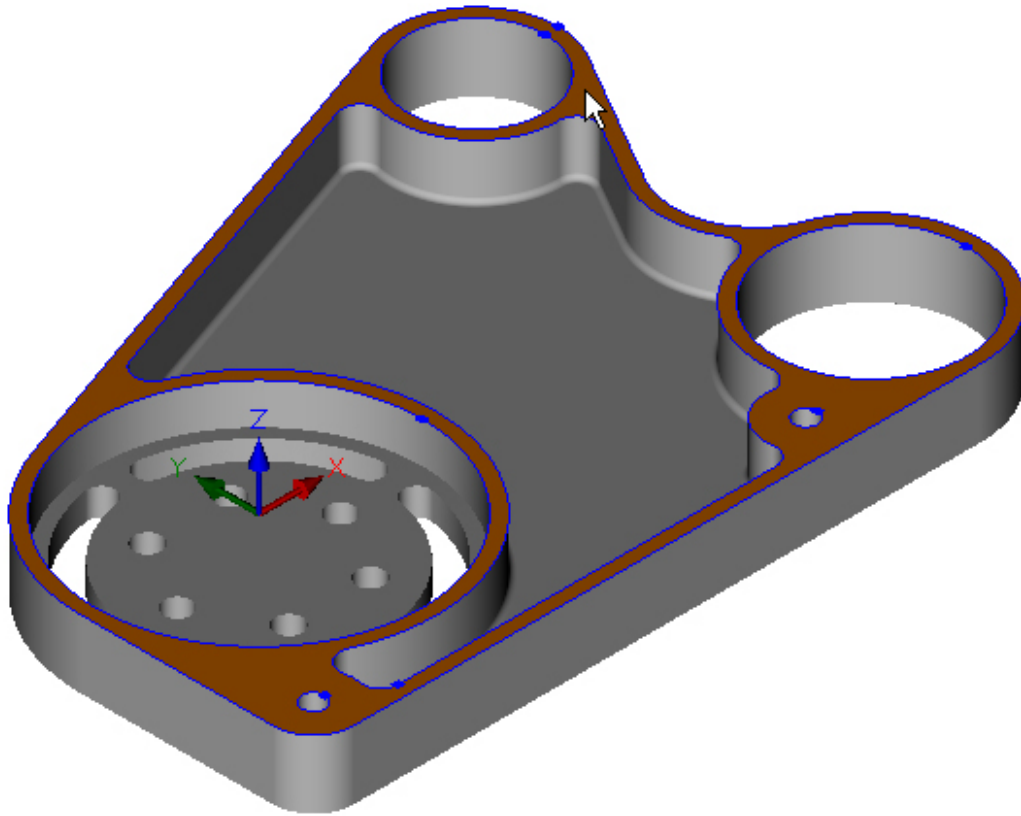
- Set the model view to isometric:
Menu: View -> Standard View -> Isometric View
Toolbar button: 



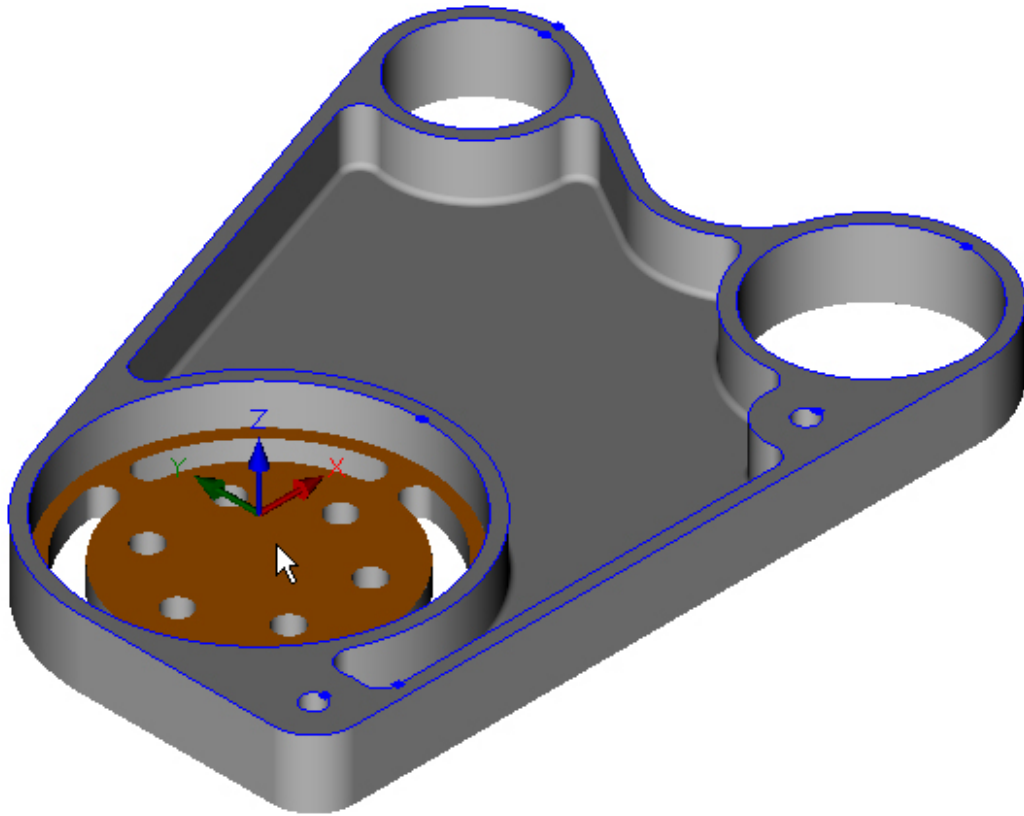
- Extract Contours from top face:
Menu: View -> Solids -> Extract Contours from Face
Toolbar button: 
Highlight top face:



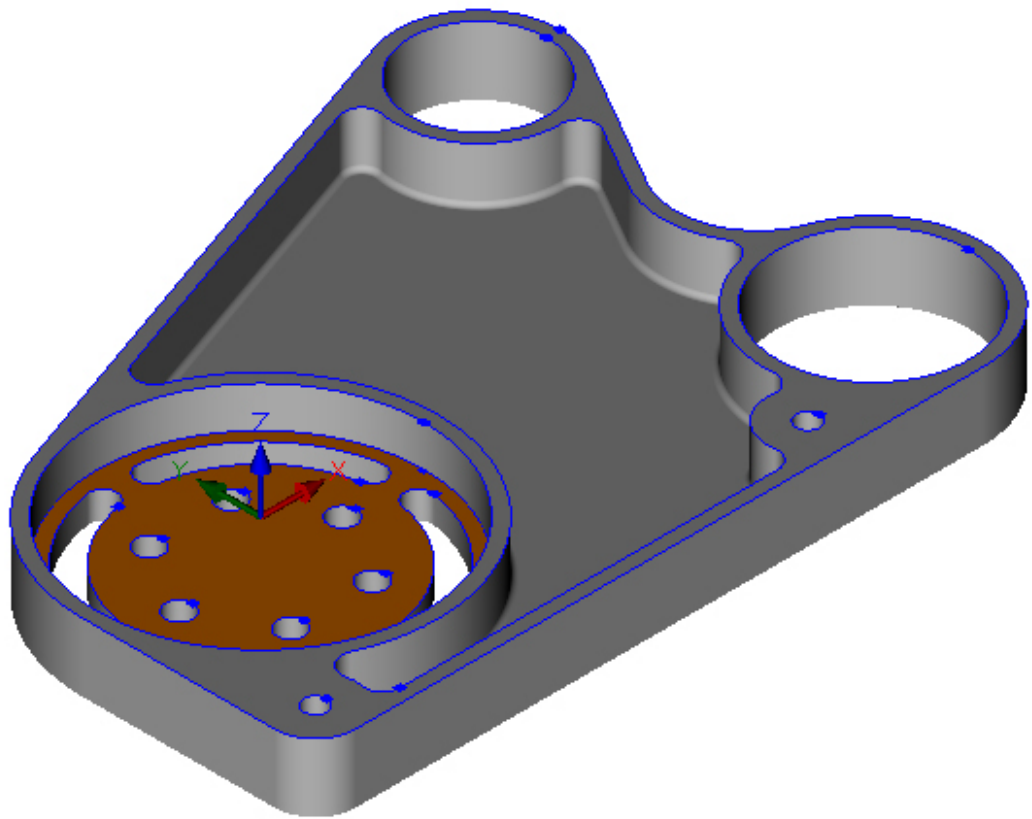
Left click mouse to extract Contours from the top face:



- Extract Contours from counter bore face - highlight face:



Left click mouse to extract Contours from the counter bore face:

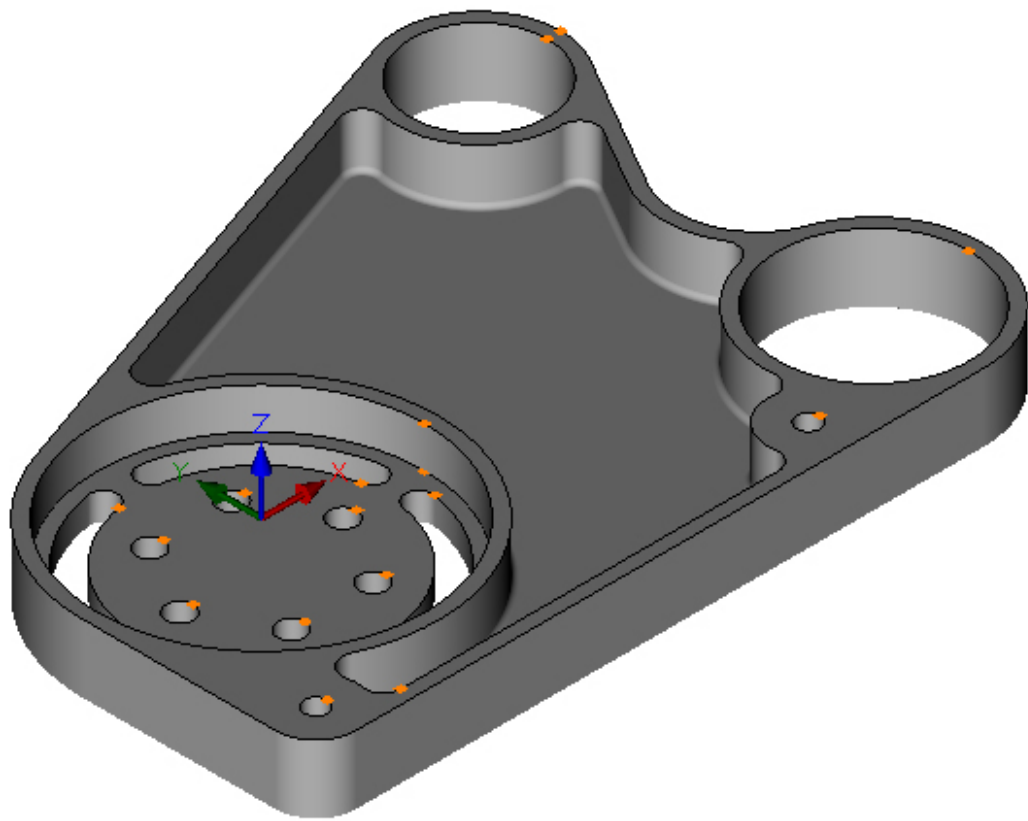


The Contours have now been extracted.

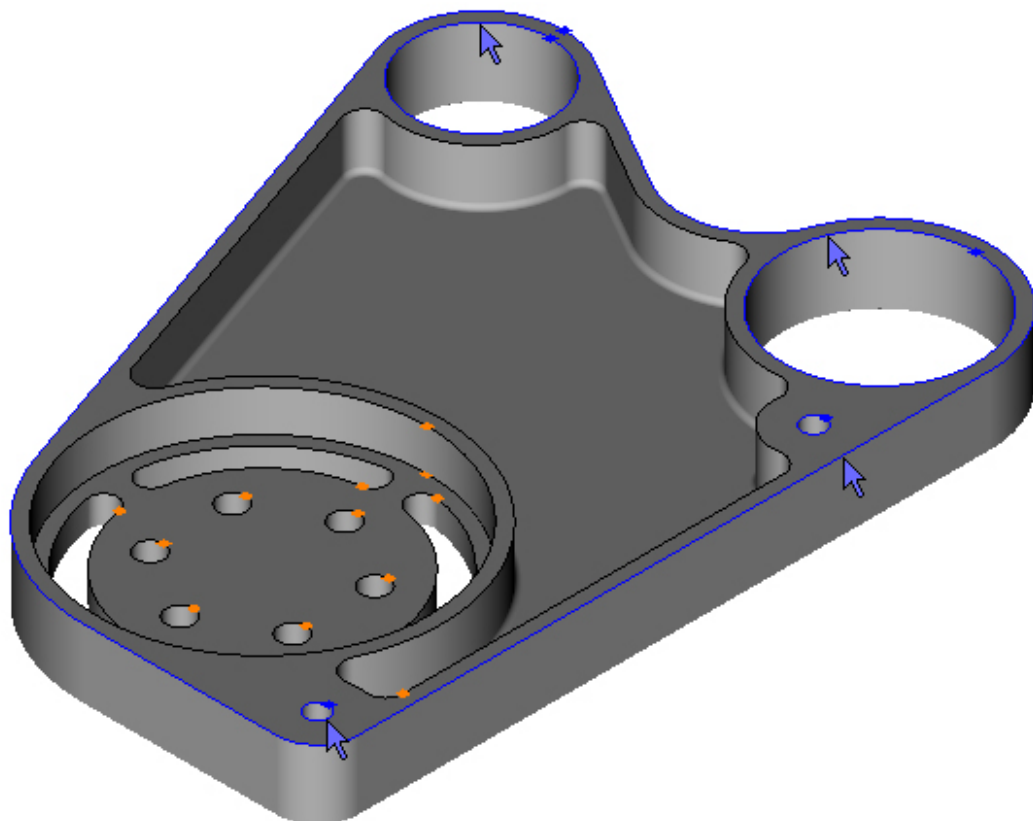
Step 2 - Set Contour Heights


Set Contour Heights for machining depths

- Deselect all Contours - Press the Escape Key:

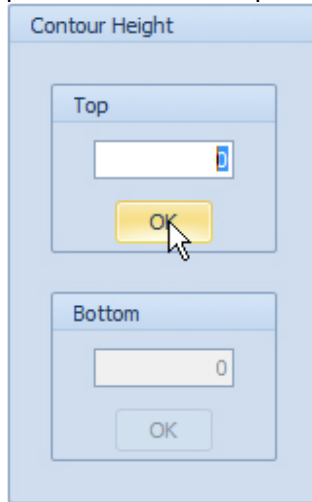


- Highlight outer Contour on top face, left click to select, repeat for four circles on top face also:

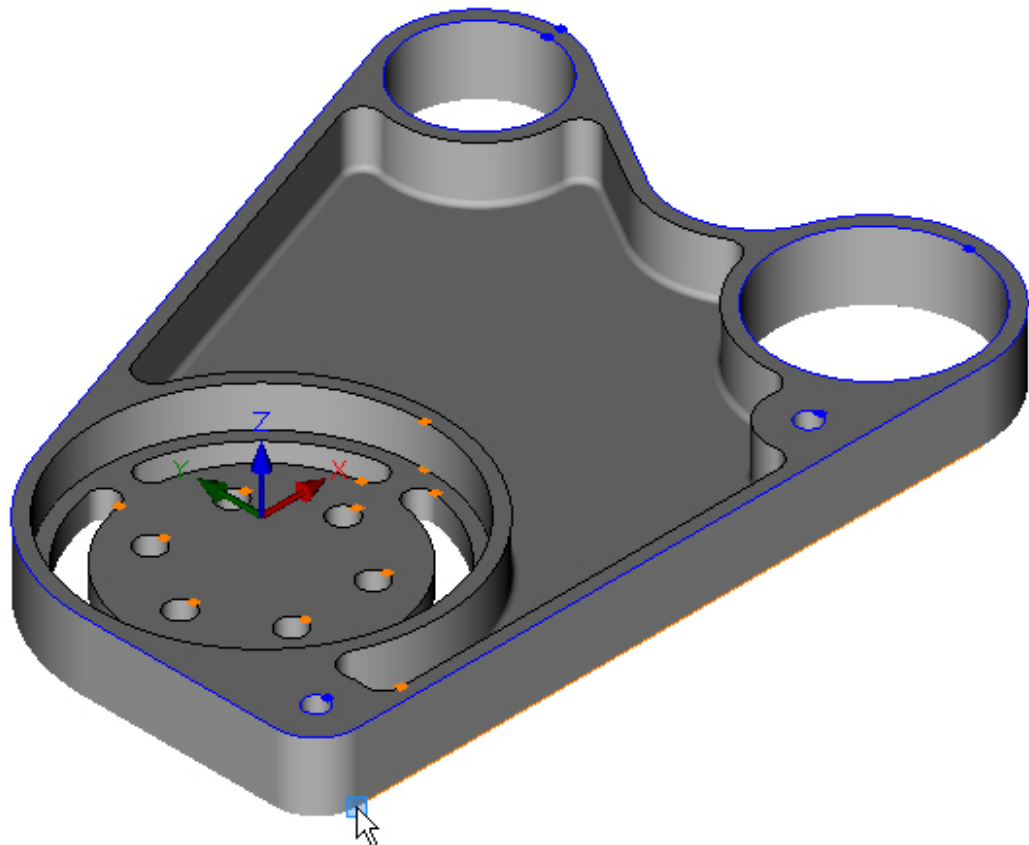


- Set the Contour Heights:
Menu: Modify -> Set Contour Height
Toolbar button: 

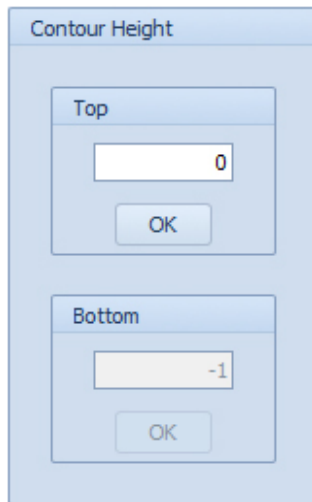
The default value for the Top of the selected Contours is zero, this is correct, we want Z0 on the top of the part. Click OK to accept:



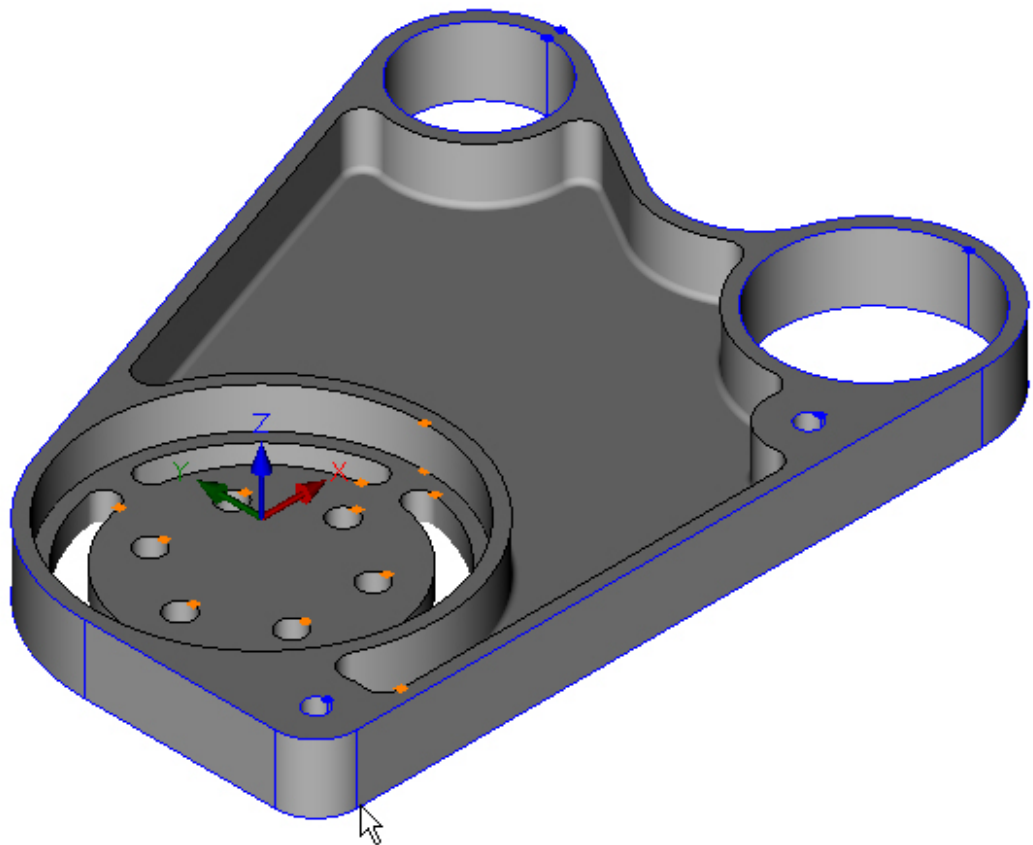
We could fill in the Bottom value, but it is easier to pick the value from the model. Highlight a point on the bottom edge:



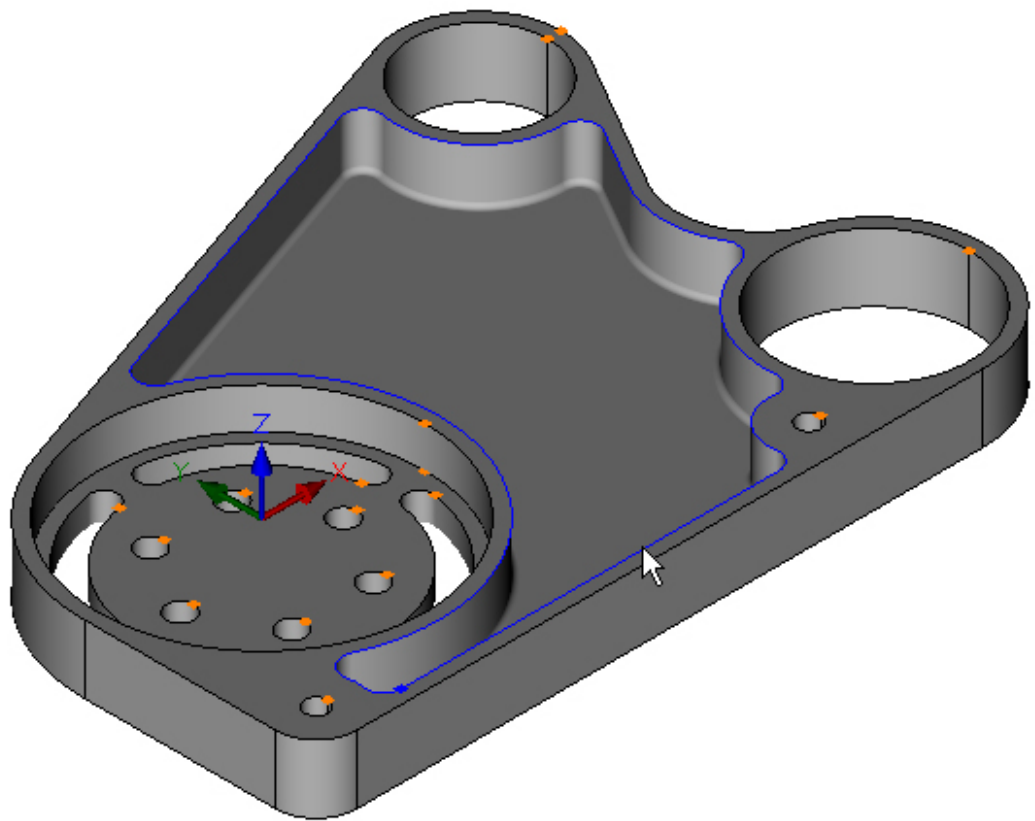
Left click to enter value:



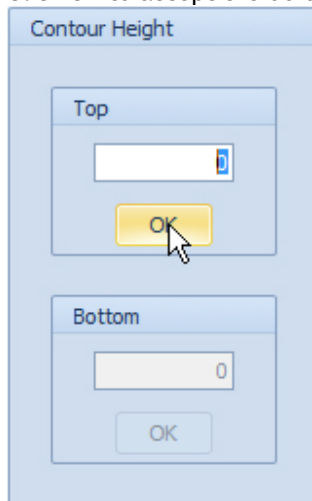
The Contour heights will be set at the same time:



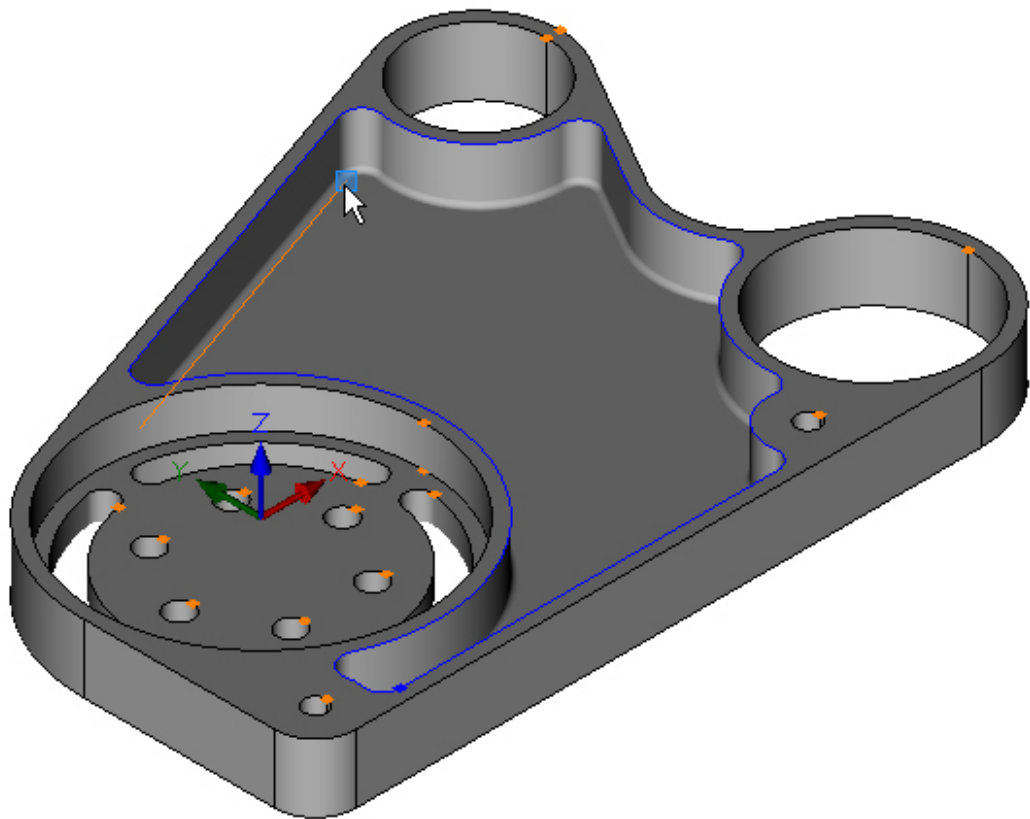
- Press the escape key to deselect the Contours and exit the *Set Contour Height* command. Highlight and select the inner pocket Contour:



Choose the *Set the Contour Heights* command.
Click OK to accept the default Top value:



Pick the Bottom value from the model. Highlight a point on the bottom edge of the pocket:



Left click to enter value:

Contour Height

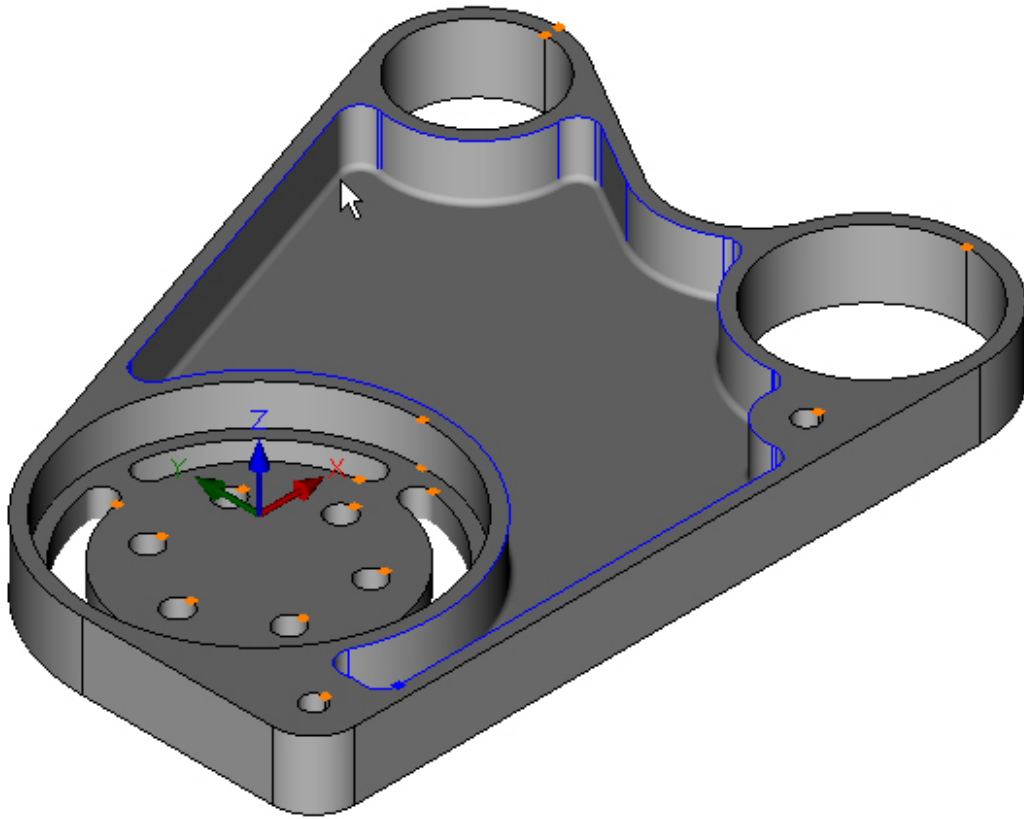
Top

OK

Bottom


OK

The Contour heights will be set at the same time:




The bottom of the Contour is hidden by the radius in the corner, you can set the view to Wire Frame to see the bottom of the Contour:

Menu: View -> Wire Frame View

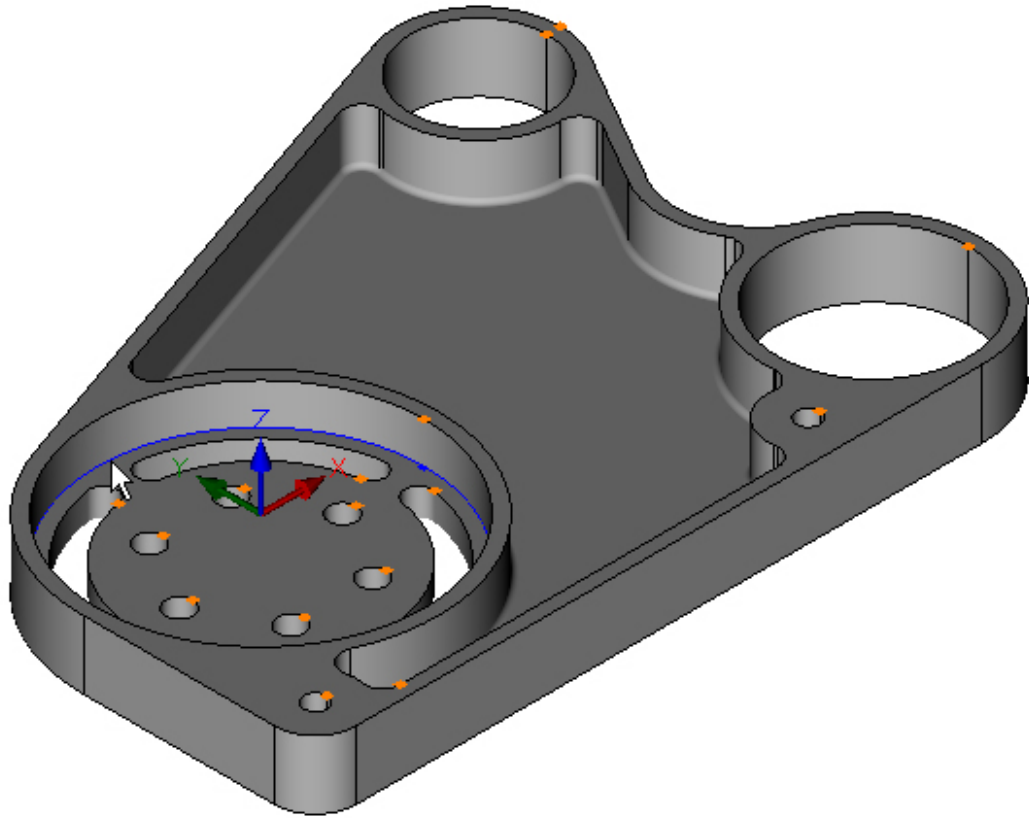
Toolbar button: 

Put the view back to Shaded View

Menu: View -> Shaded View


Toolbar button: 

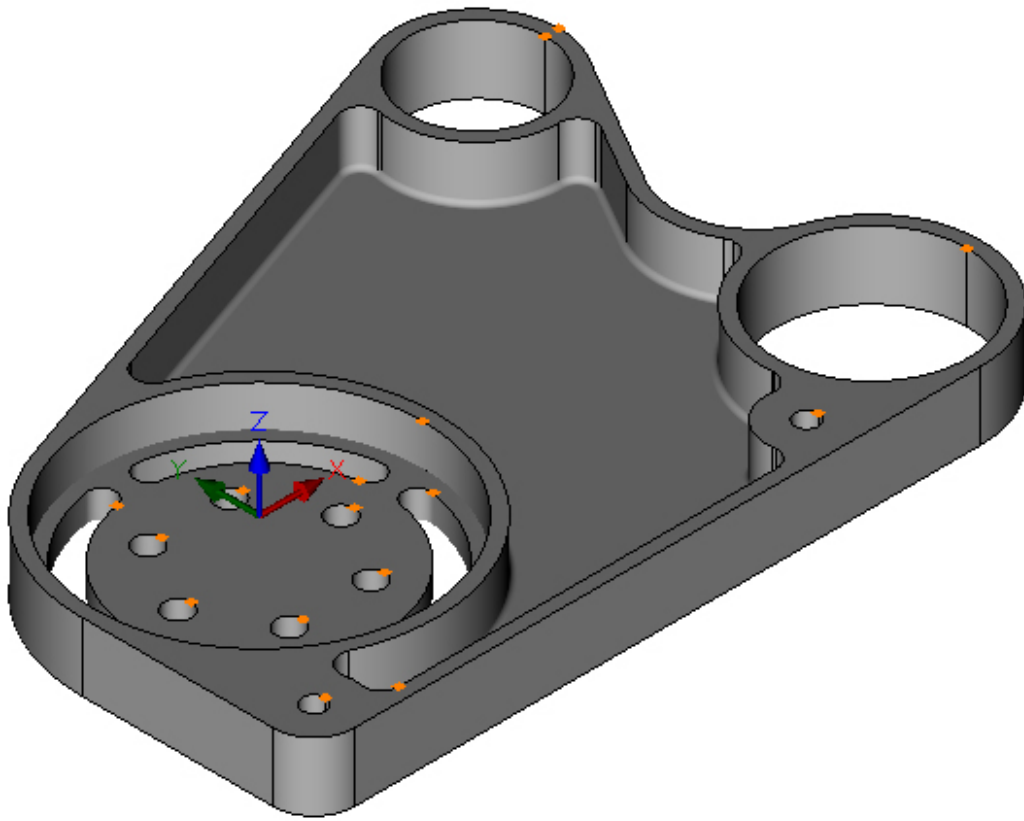
- Next delete the unwanted Contour in the bore.
Press the escape key to deselect the Contours.
Highlight and select the unwanted Contour:



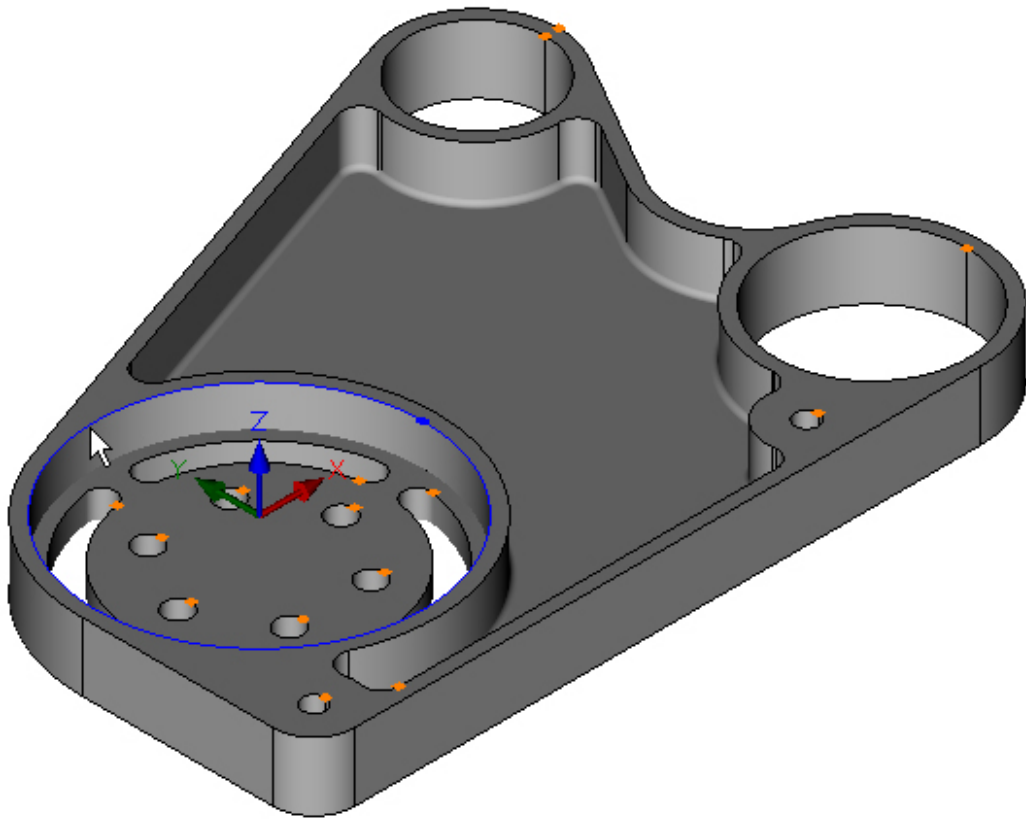
Choose the Delete command to remove unwanted Contour:

Menu: Edit -> Delete

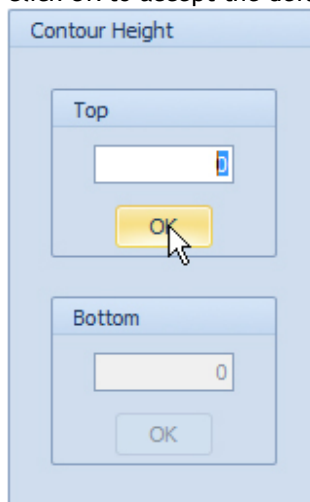
Toolbar button: 



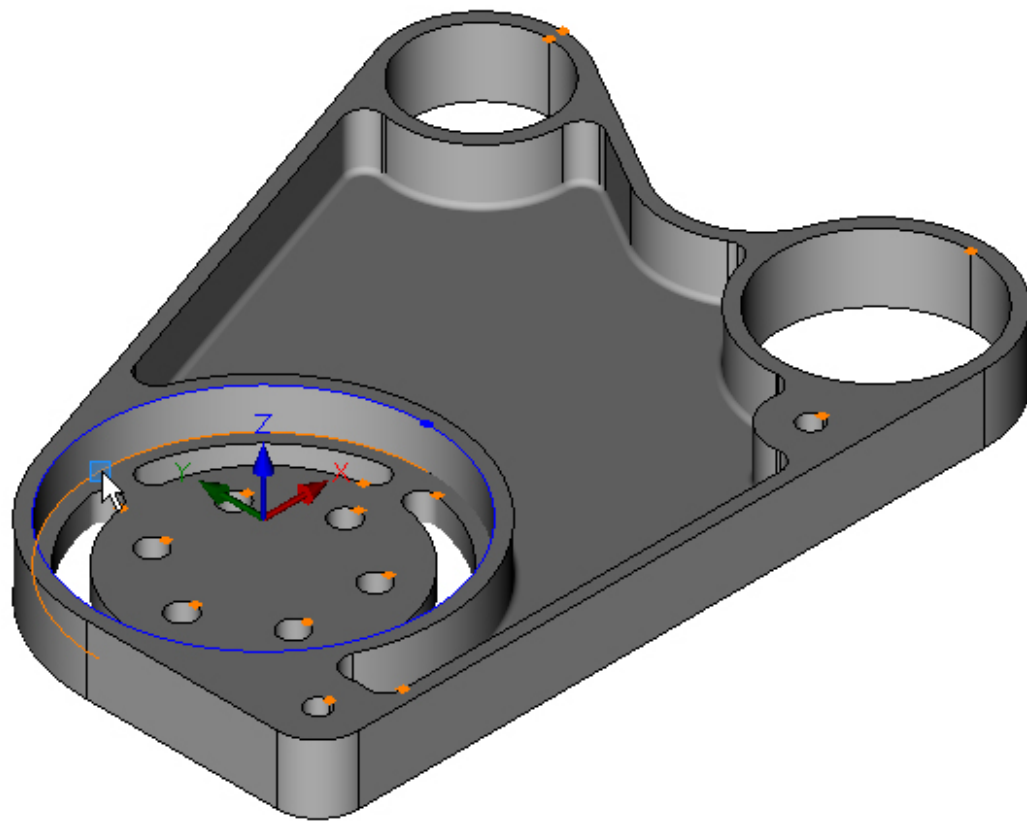
- Highlight bore Contour on top face, left click to select:



Choose the *Set the Contour Heights* command
Click OK to accept the default Top value:



Pick the Bottom value from the model. Highlight a point on the bottom edge of the bore:



Left click to enter value:

Contour Height

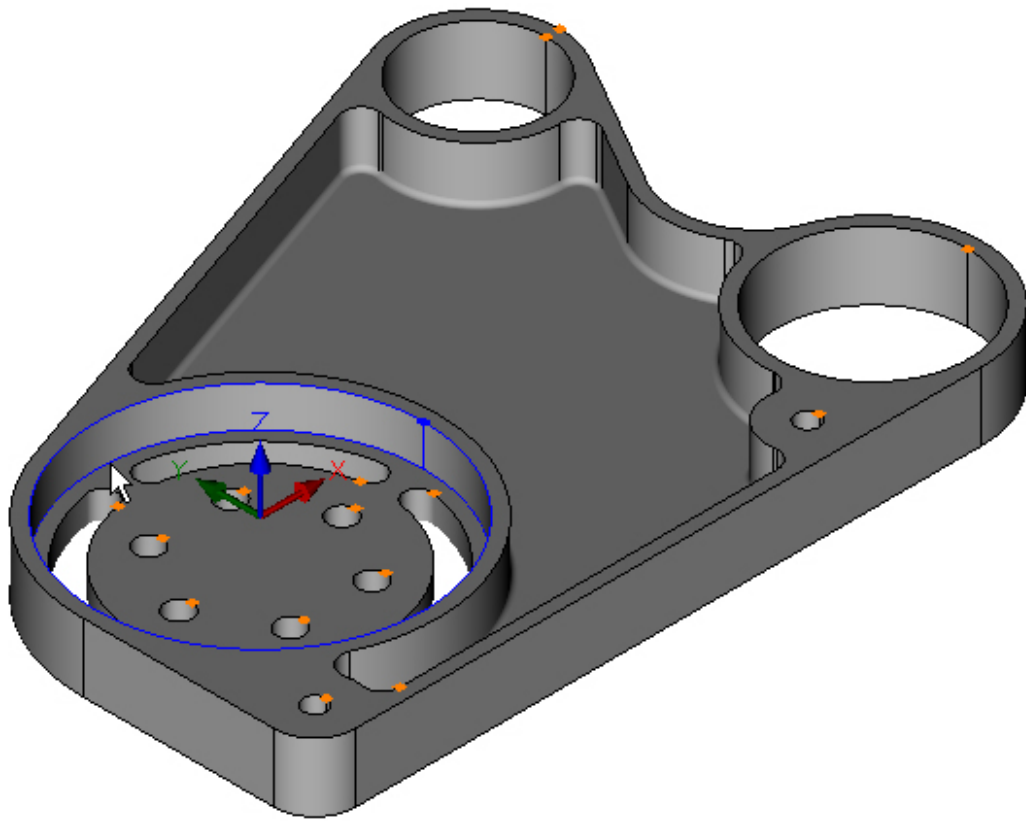
Top


OK

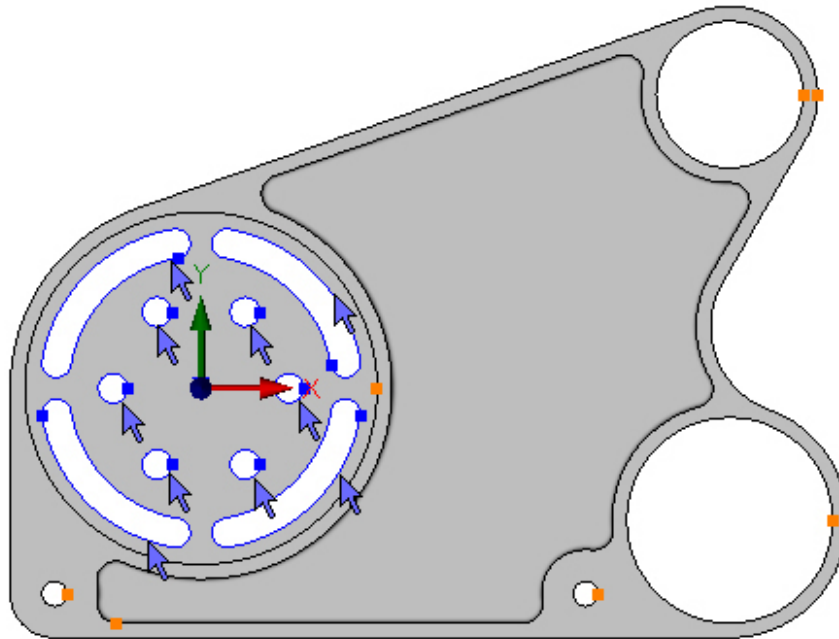
Bottom

OK

The Contour heights will be set at the same time:




- Press the escape key to deselect the bore Contour.
Highlight and select the slots and holes in the bore:
One of the slots is obscured, choose the Top View command to assist in selection.
Menu: View -> Top View
Toolbar button: 



Switch back to Isometric View


Menu: View -> Standard View -> Isometric View

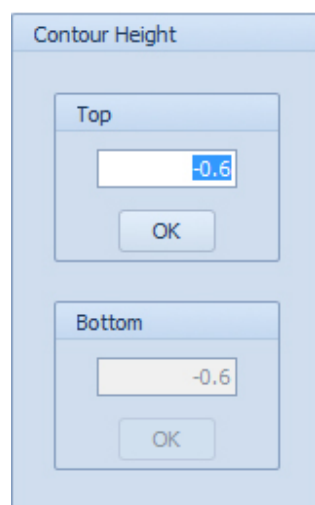
Toolbar button: 

Choose the *Set the Contour Heights* command.

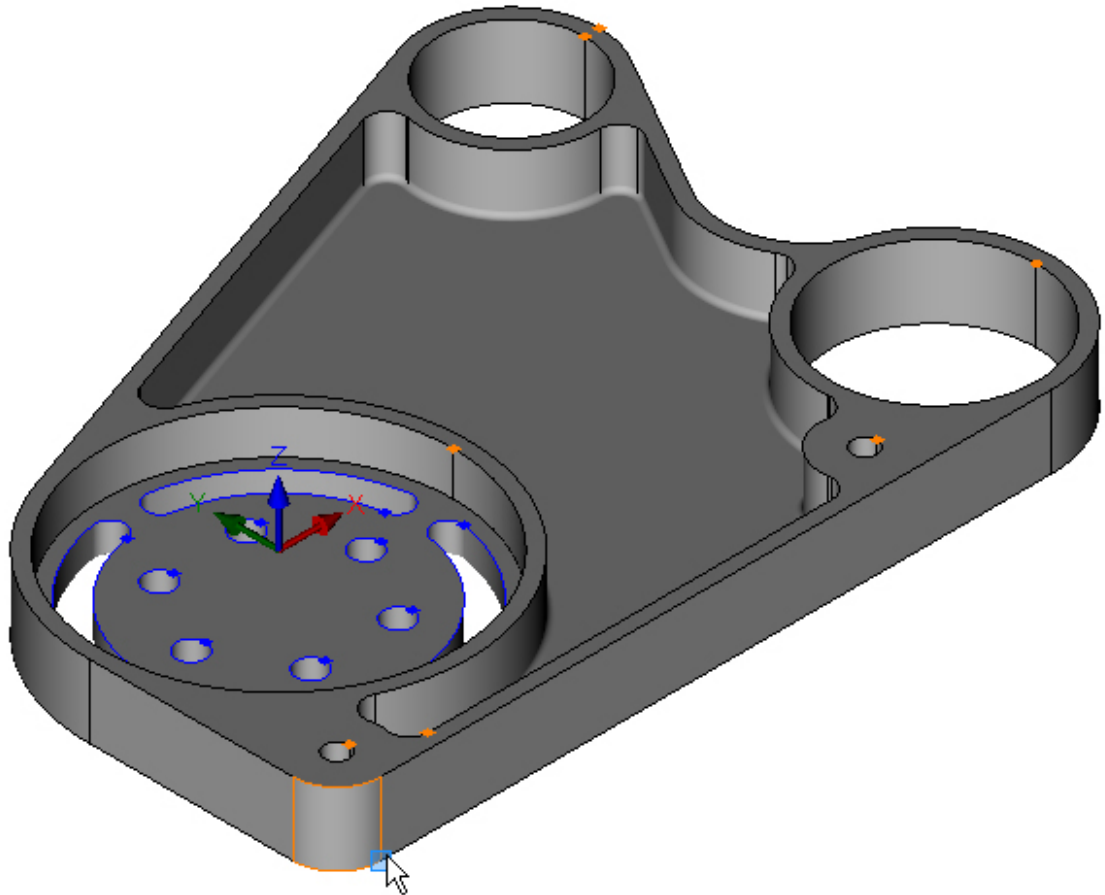
Click OK to accept the default Top value:

Menu: Modify -> Set Contour Height

Toolbar button: 



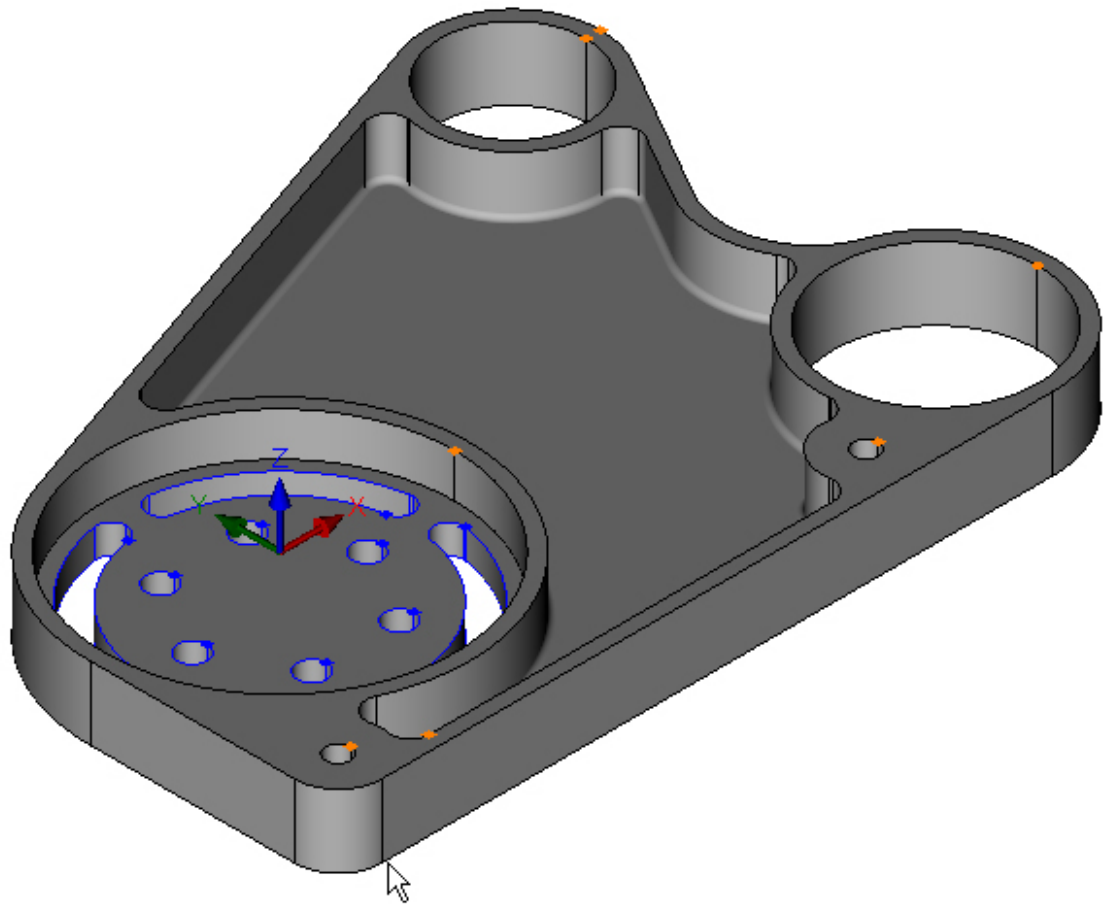
Pick the Bottom value from the model. Highlight a point on the bottom edge of the outside:



Left click to enter value:

Contour Height	
Top	
<input type="text" value="-0.6"/>	OK
Bottom	
<input type="text" value="-1"/>	OK

The Contour heights will be set at the same time:



Gear Housing 3D 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 completely, 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	


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.


Choose the Top View command.

Menu: View -> Top View

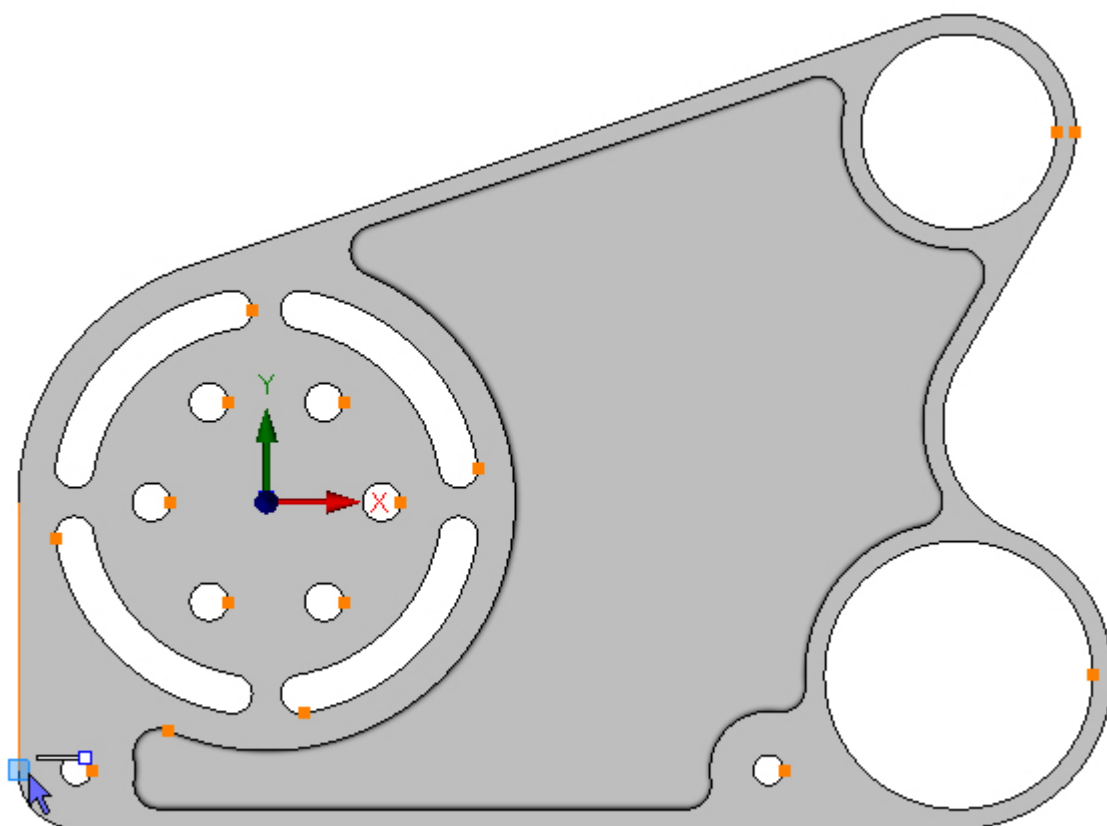
Toolbar button: 

Set the start point:

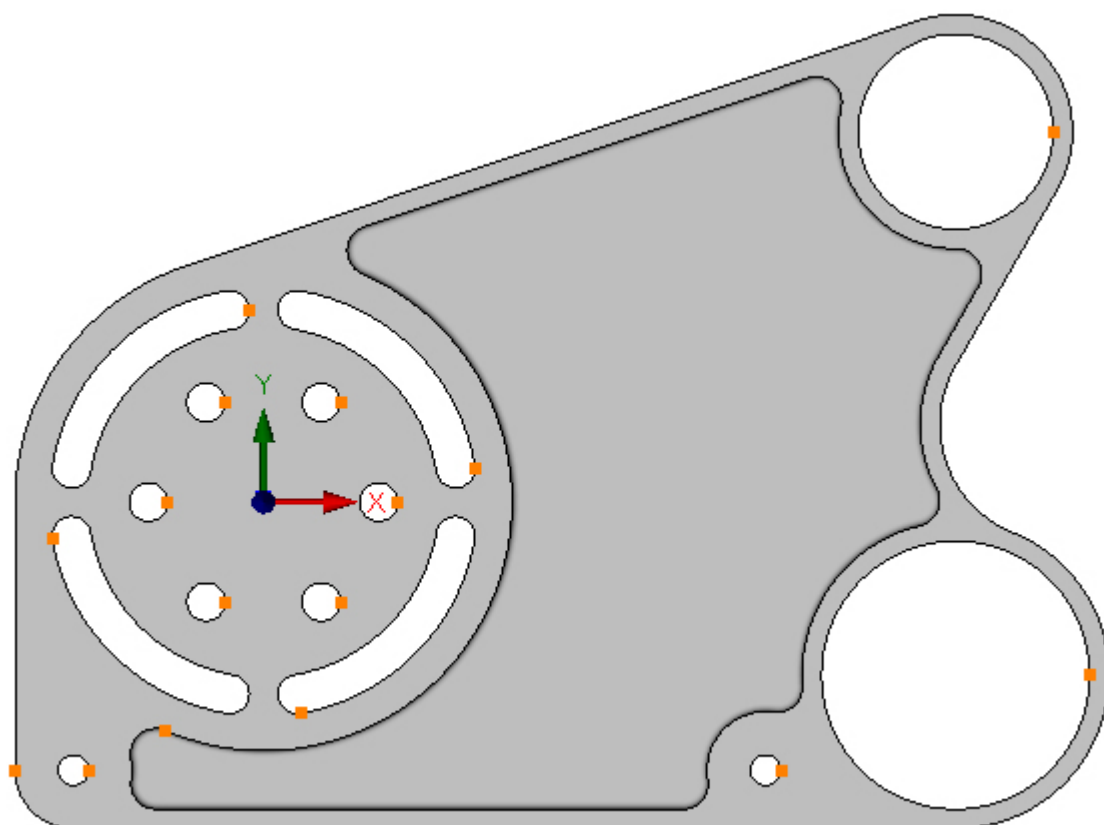
Menu: Modify -> Set Start Point

Toolbar button: 

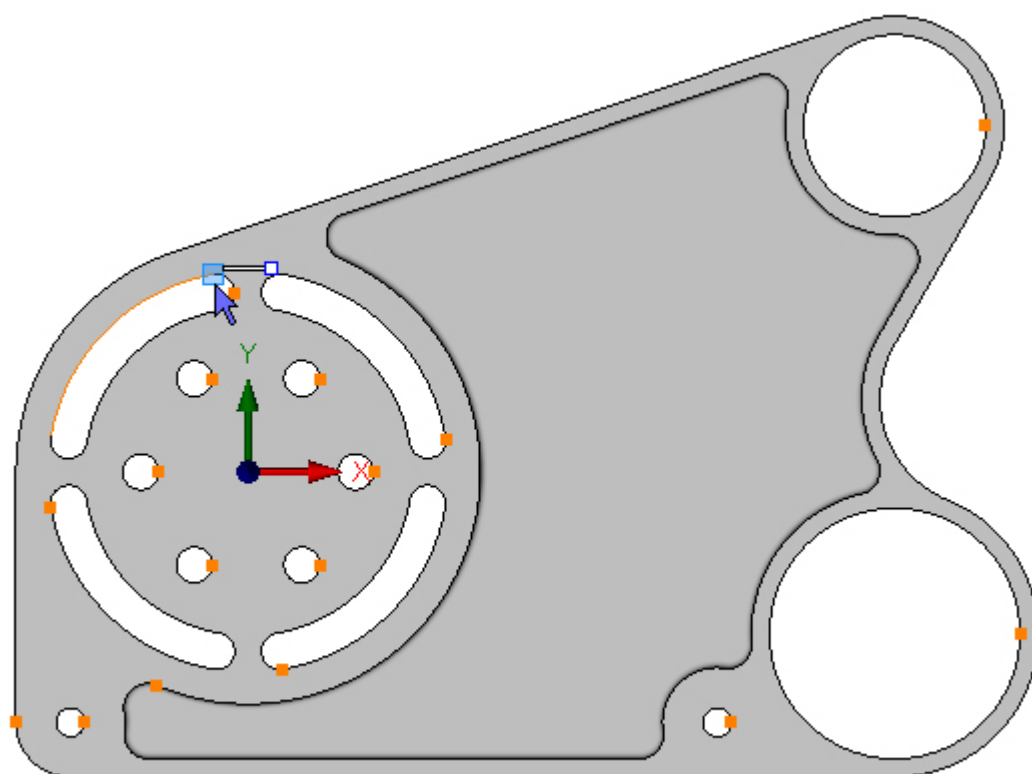
Choose the command and move the cursor near to the end of the line on the left:

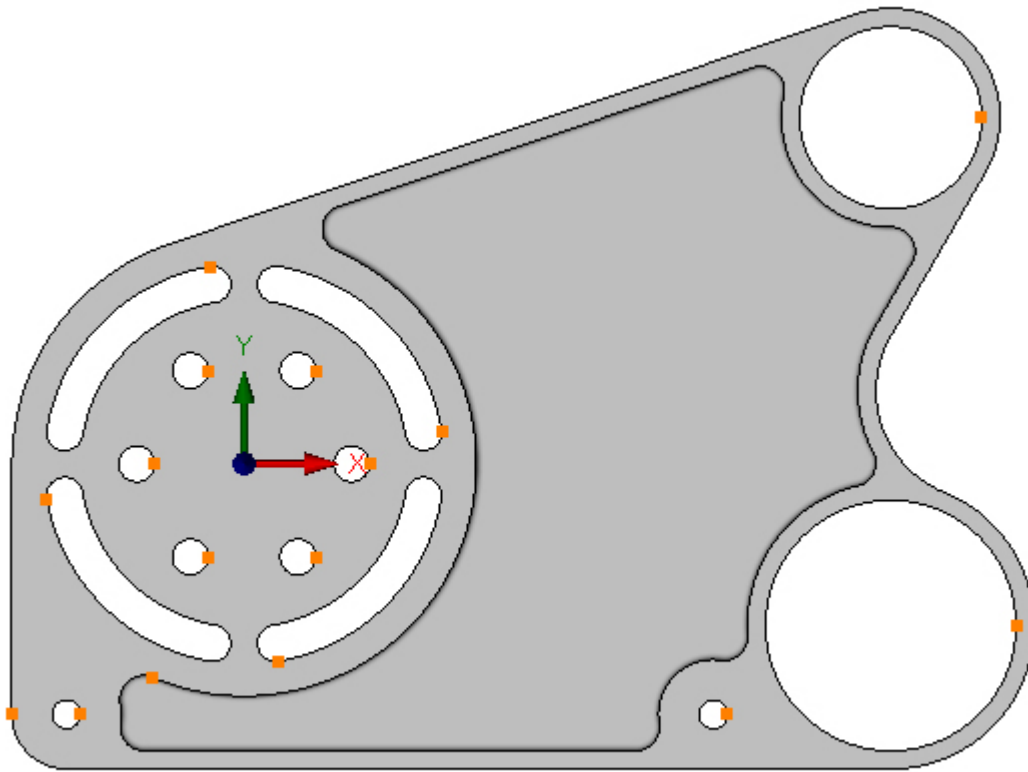


Left mouse click to set the start point:



Also set the start point on the upper left slot:






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:

Gear Housing 3D Tutorial

Rectangle

Start Corner Point

X

Y

OK

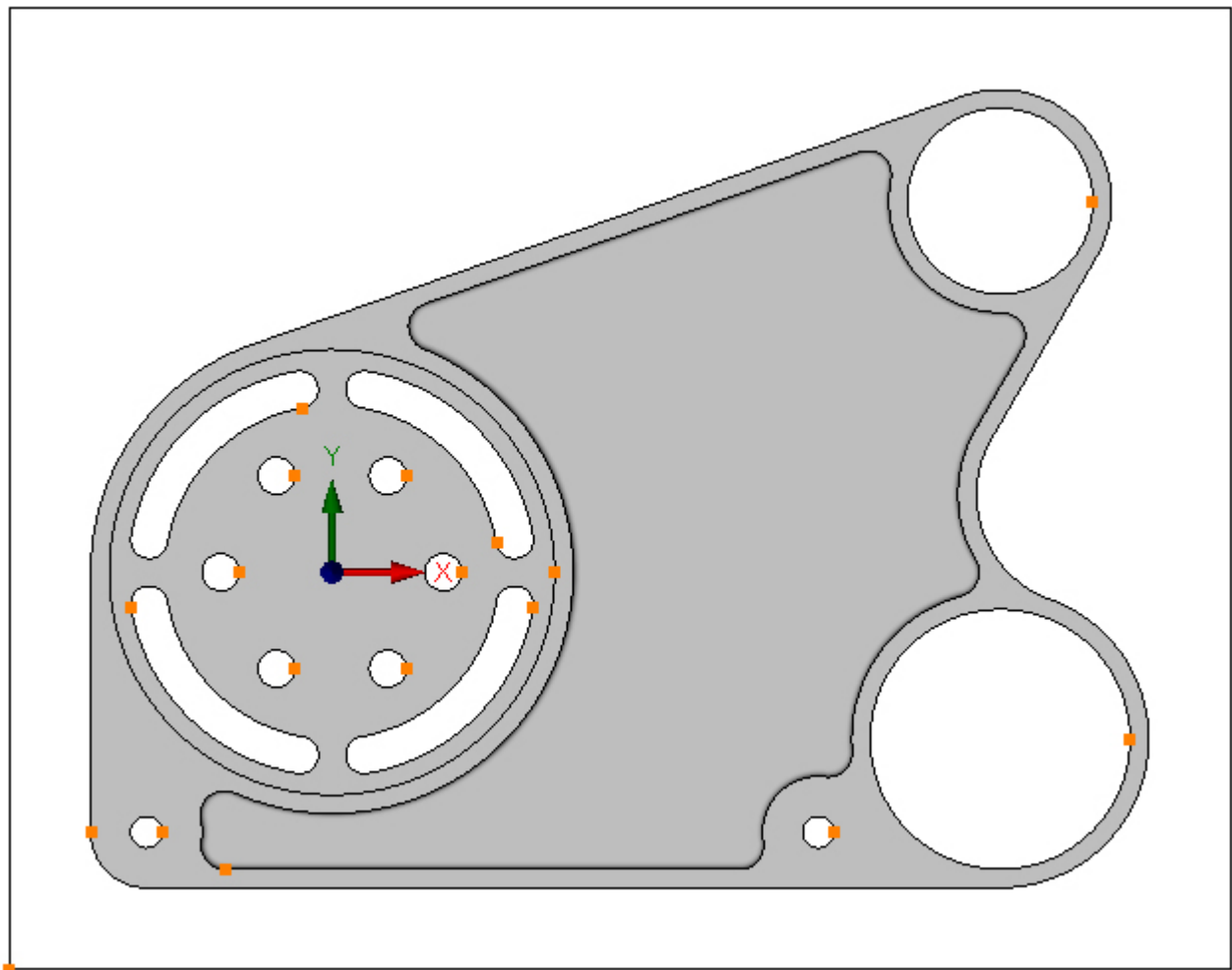
End Corner Point

X

Y

OK

The Part should now look like this:




By default the Contour that forms the rectangle has a Z Top and Z bottom of zero. The outside profile and the rectangle will be used for the pocketing operation, they both must have the same Z Top and Z bottom.

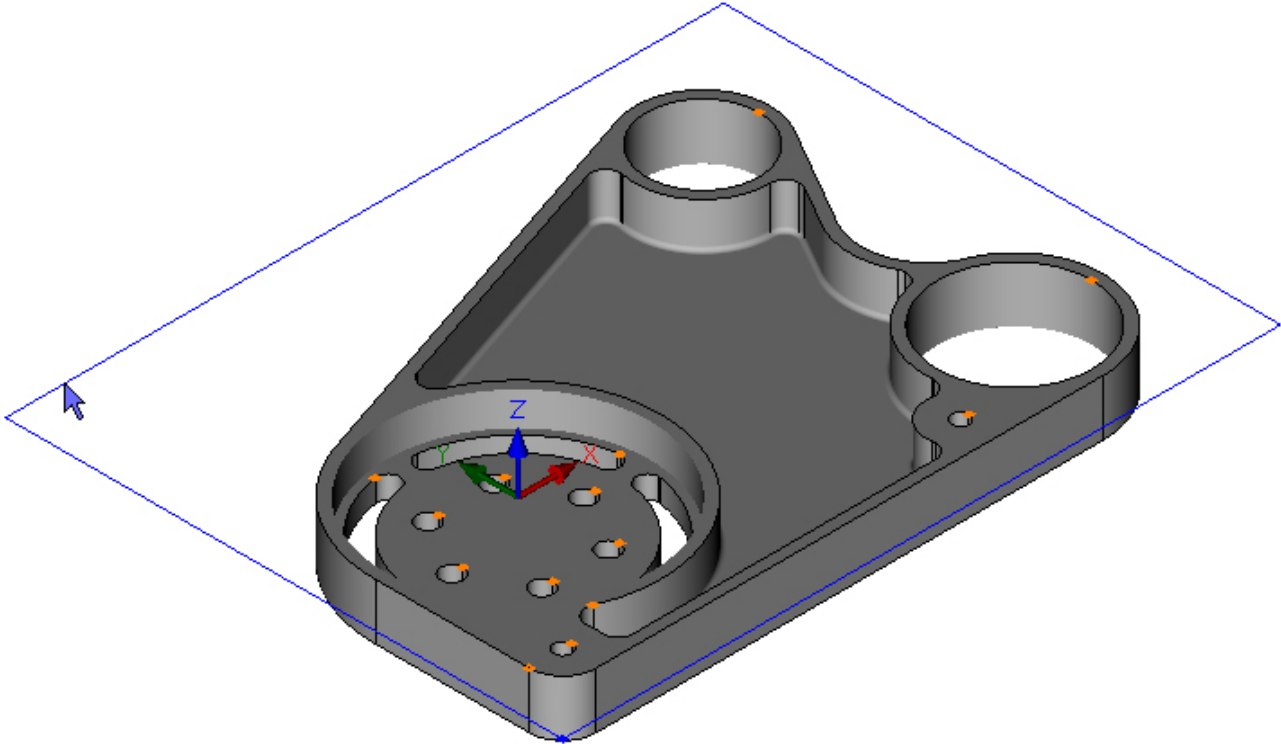
Press the escape key to exit Rectangle mode.

Set the model view to isometric:

Menu: View -> Standard View -> Isometric View


Toolbar button: 

Highlight and left click the rectangle to select:

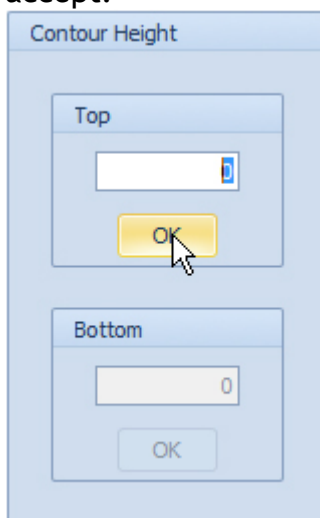


Set the Contour Heights:

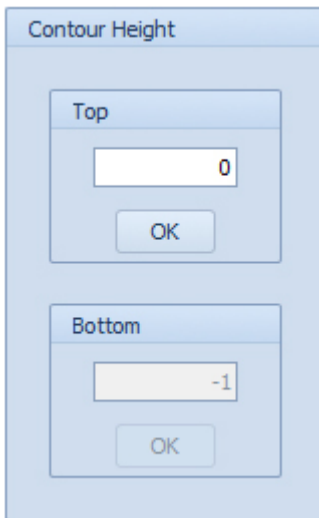
Menu: Modify -> Set Contour Height

Toolbar button: 

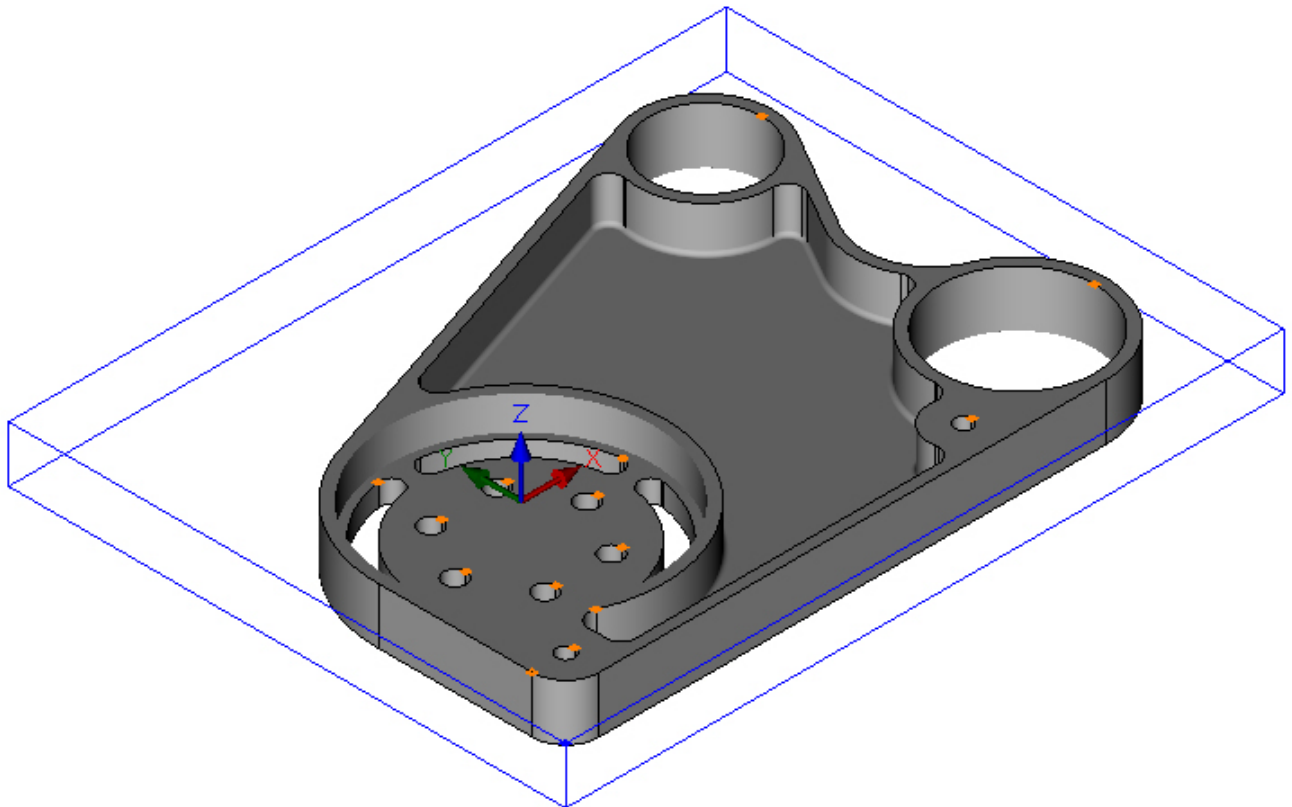
The default value for the Top of the selected rectangle is zero, this is correct. Click OK to accept:



Enter the value Z-1 in the Z Bottom and click OK



The contour height has been set:




Step 4 - Rough outside profile

Rough Outside Profile using Pocket

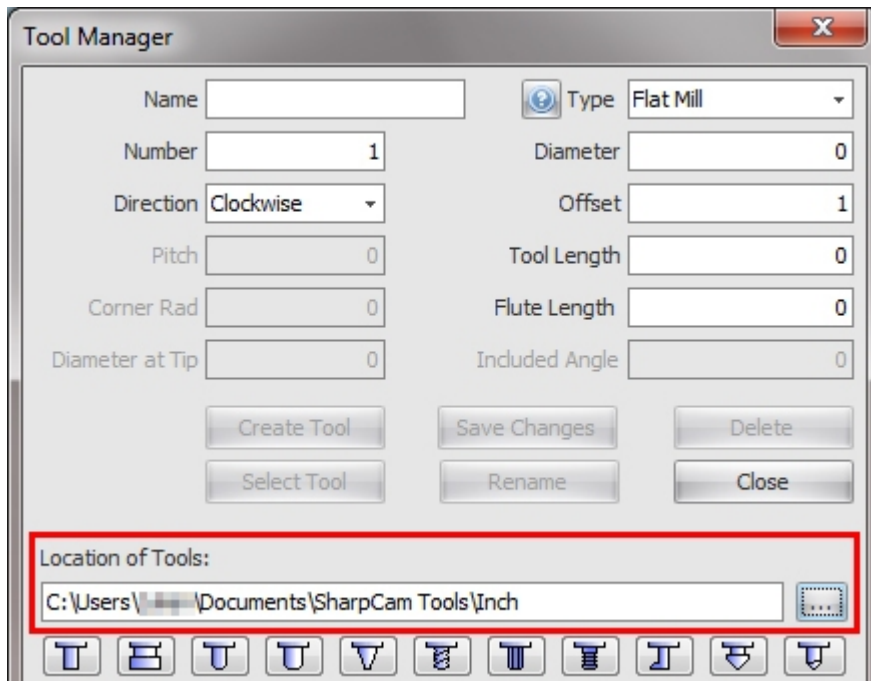
Before choosing the pocket command, first select the cutting tool to be used for the operation. The Tool Manager is used to select a tool:


Menu: Machine-> Tool Manger

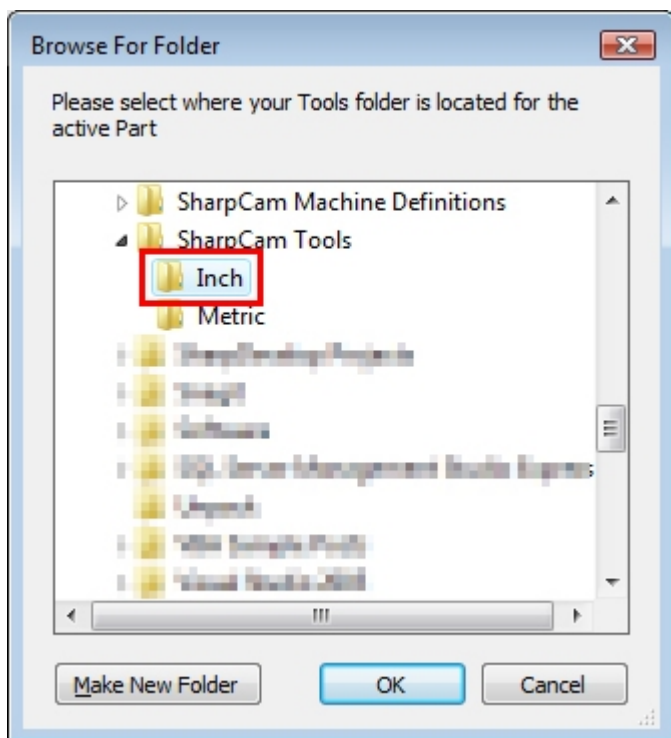
Toolbar button: 

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:



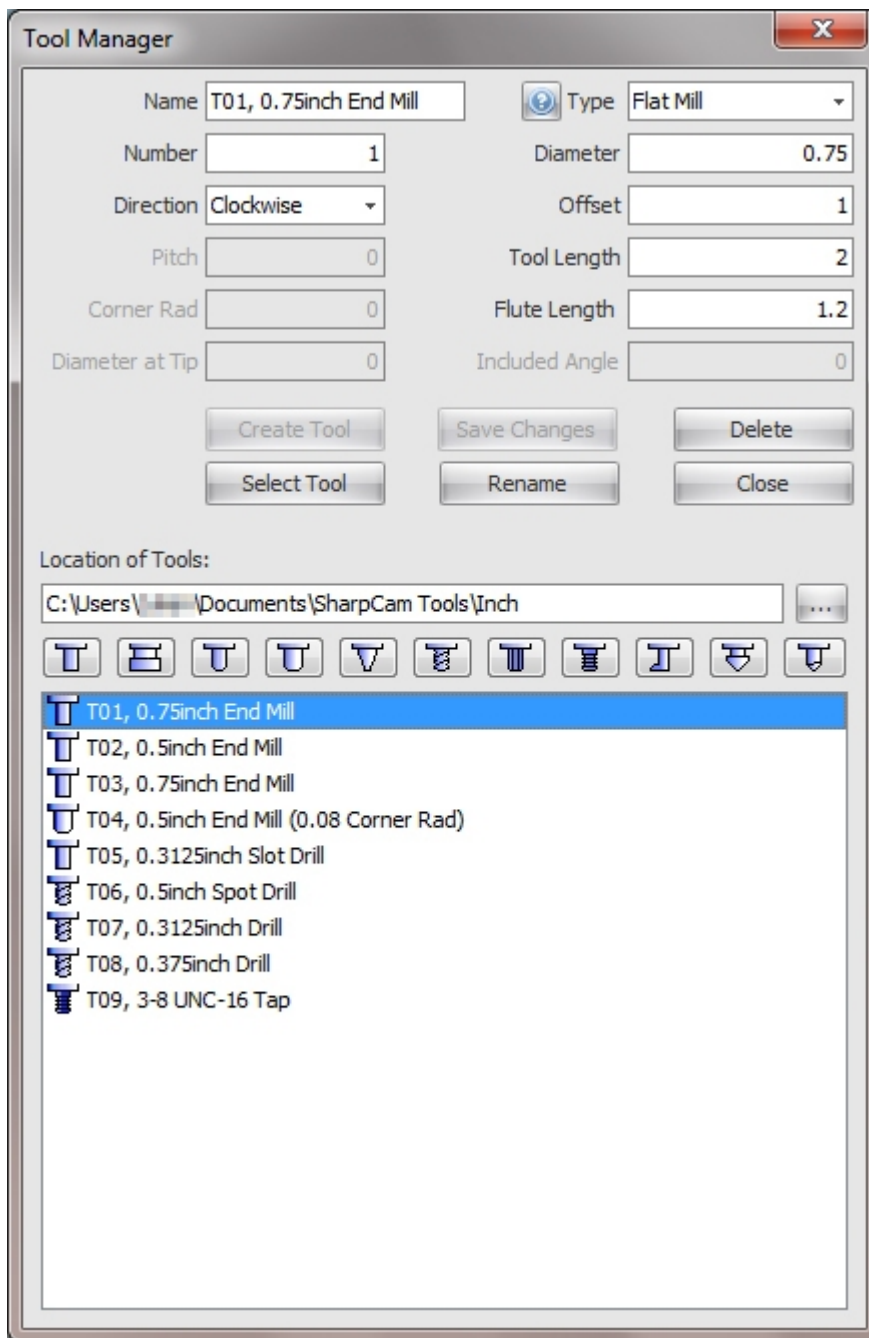
As this is an imperial Part the *Inch* tools folder, that was installed at the same time as SharpCam, should be used. This folder is located in Documents (Vista)/Documents Library (Windows 7, 8, 8.1), inside a folder called *SharpCam Tools*. If the location is wrong, click the browse button  and navigate 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.

Gear Housing 3D Tutorial

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



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

Selected Tool: T01, 0.75inch End Mill


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


Menu: Machine-> Pocket

Toolbar button: 

Enter the following values for the Z Positions:

Z Positions	
Abs <input checked="" type="checkbox"/>	Initial Rapid <input type="text" value="2"/>
	Feed From <input type="text" value="0.1"/>
	Material Surface <input type="text" value="0"/>
	Finish Depth <input type="text" value="-0.04"/>
Abs <input checked="" type="checkbox"/>	Retract <input type="text" value="0.1"/>
	Number of Passes <input type="text" value="4"/>
	Depth of Cut <input type="text" value="0.04"/>

 The Finish Depth is taken from the Bottom of the contour, which is -1, so the actual depth is -1.04

 The Depth of cut will not be correct until the contours have been added to the operation.

Enter the following values for the Cutting Data:

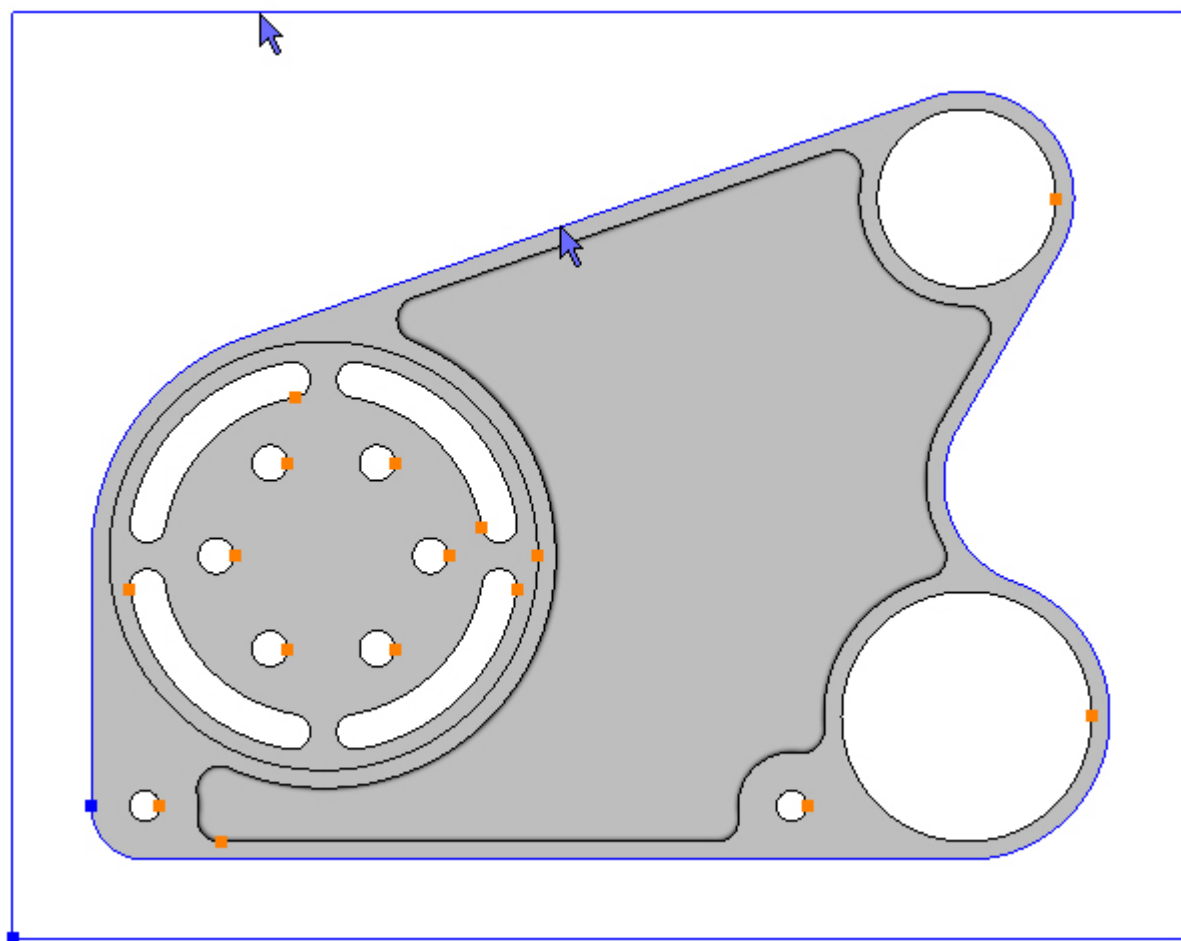
Cutting Data	
Entry	Advanced
Spindle Speed	<input type="text" value="8000"/>
Plunge Feed Rate	<input type="text" value="80"/>
Cut Feed Rate	<input type="text" value="40"/>
Finish Allowance	<input type="text" value="0.02"/>
Step Over	<input type="text" value="0.375"/>
Coolant	<input type="text" value="Flood"/>
Sub Routines	<input type="checkbox"/>


Choose the Top View command.

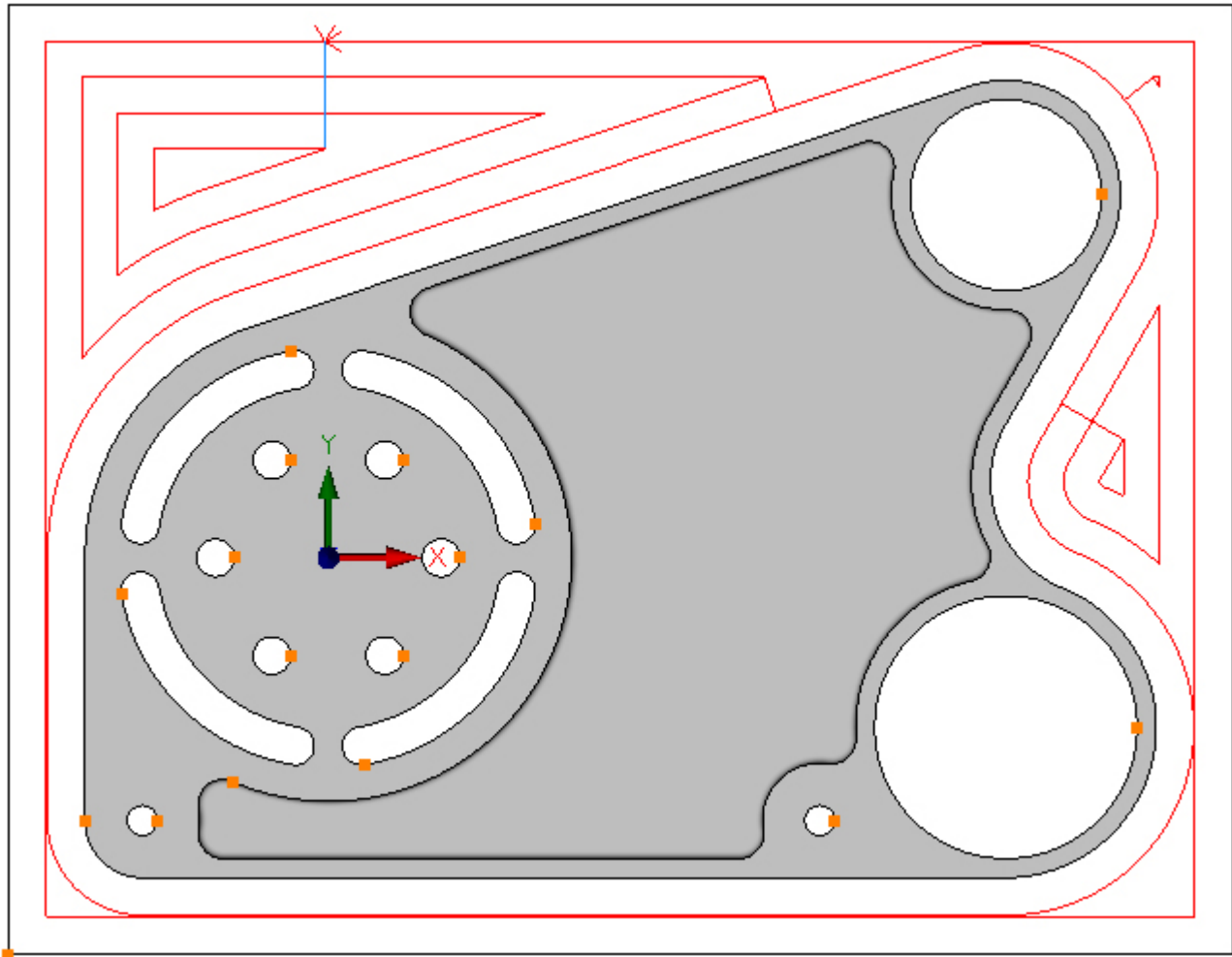
Menu: View -> Top View

Toolbar button: 

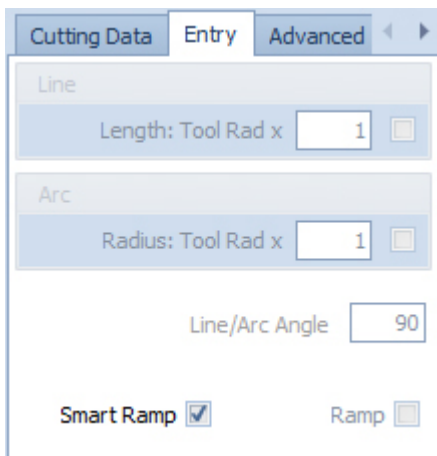
Select the outside profile and the outer rectangle by directly clicking them, the outer rectangle may already be selected:



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:

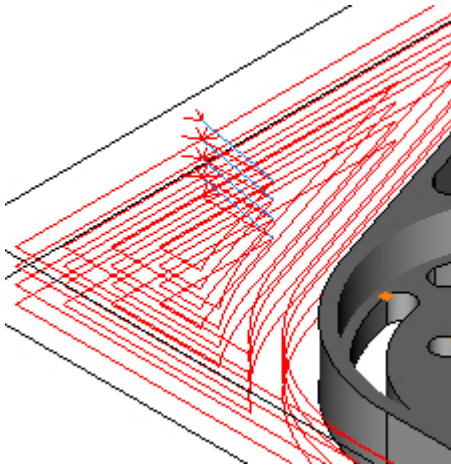
Menu: View -> Standard Views -> Isometric View

Toolbar button: 

Choose the command and zoom in on the Smart Ramp by pointing the cursor at the Smart

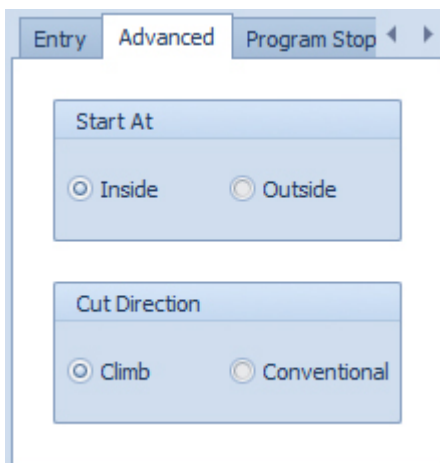
Gear Housing 3D Tutorial

Ramp and rotating the middle mouse button:

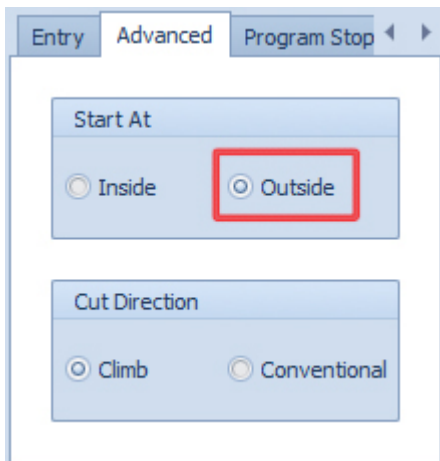


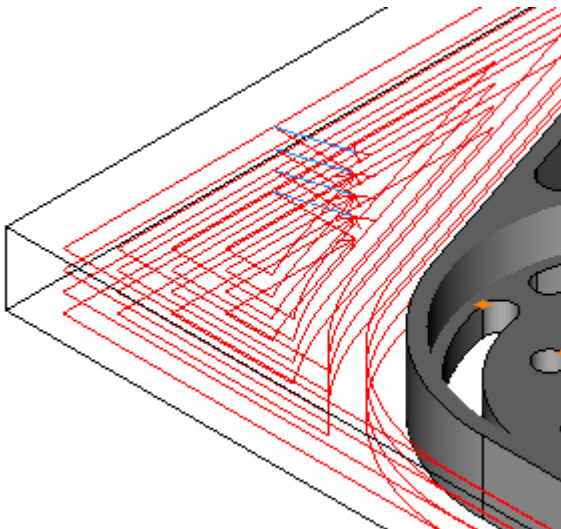
As can be seen the Smart Ramp, as the name implies, automatically ramps to depth at the shallowest angle possible. As it is the outside profile that is being machined Smart Ramp is not needed, because it is possible to plunge to depth outside of the billet.

First start from the outside, click the Advanced tab:

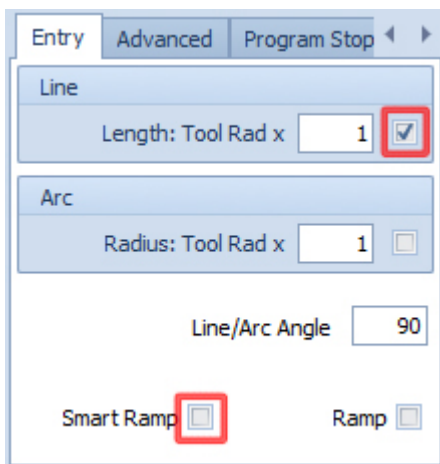


At the moment the pocketing starts from the inside, click the outside radio button and the toolpaths will be recalculated to start from the outside:

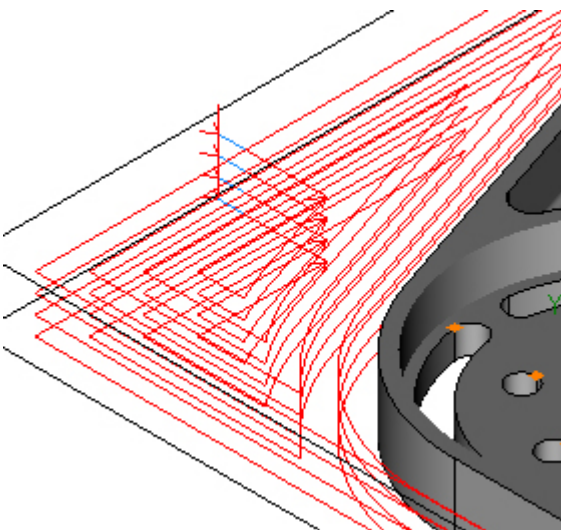




Choose the Entry tab and uncheck the Smart Ramp check box and check the Line check box:



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):

Gear Housing 3D Tutorial

Entry Advanced Program Stop

Line

Length: Tool Rad x ☒

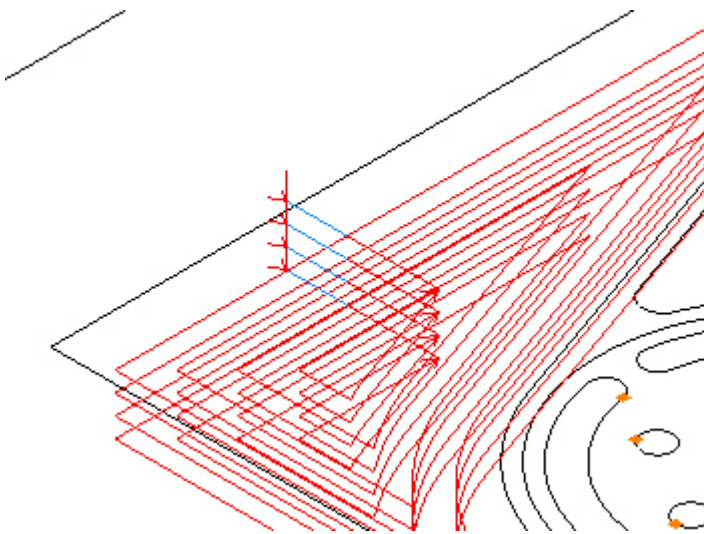
Arc

Radius: Tool Rad x ☐

Line/Arc Angle

Smart Ramp ☐ Ramp ☐

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:

Z Positions

Abs ☒ Initial Rapid


Feed From

Material Surface

Finish Depth

Abs ☒ Retract

Number of Passes


Depth of Cut 

Enter the following values for the Cutting Data:

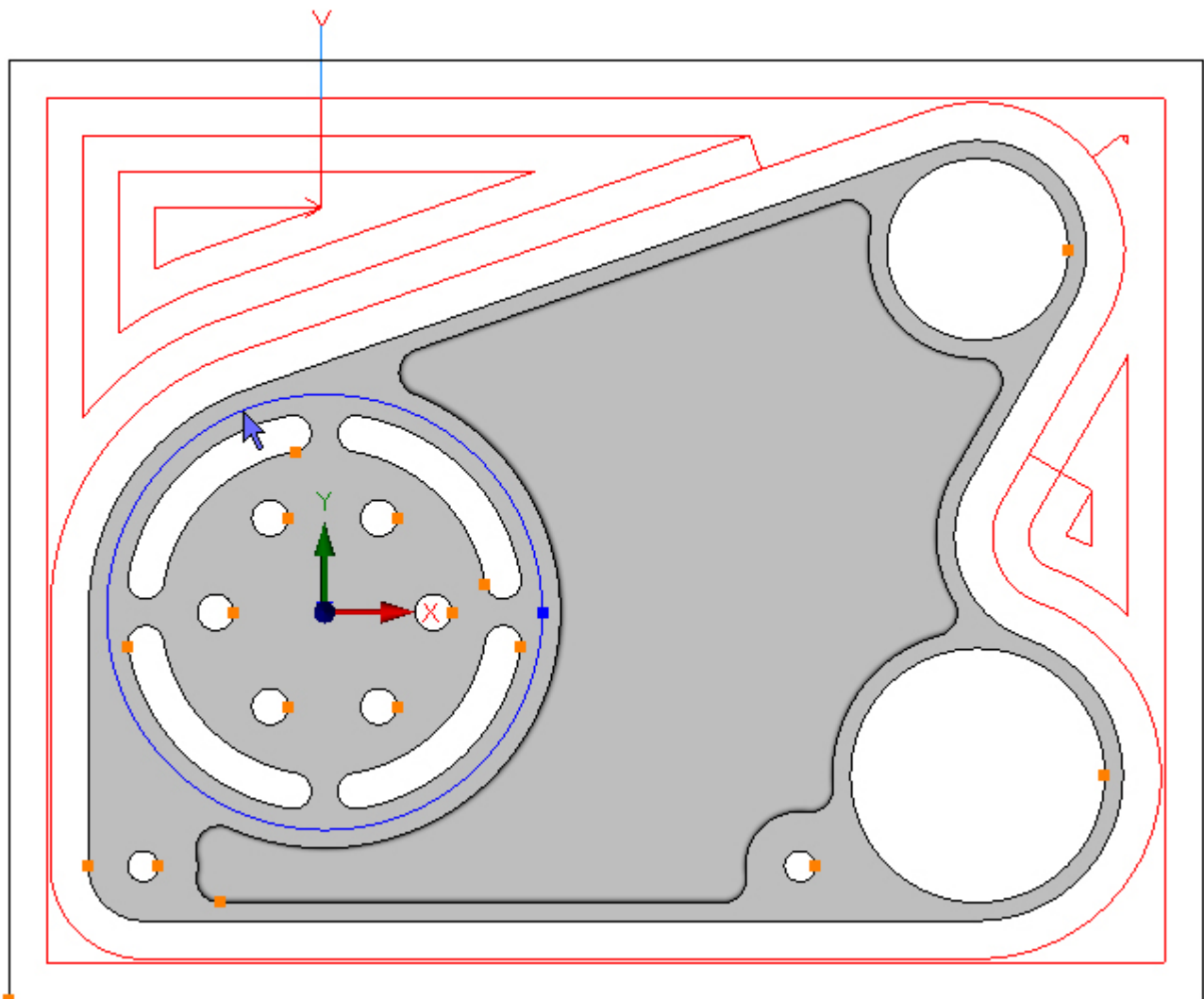
Cutting Data	
Entry	Advanced
Spindle Speed	8000
Plunge Feed Rate	20
Cut Feed Rate	30
Finish Allowance	0.02
Step Over	0.375
Coolant	Flood
Sub Routines	<input type="checkbox"/>

Choose the Top View command.

Menu: View -> Top View

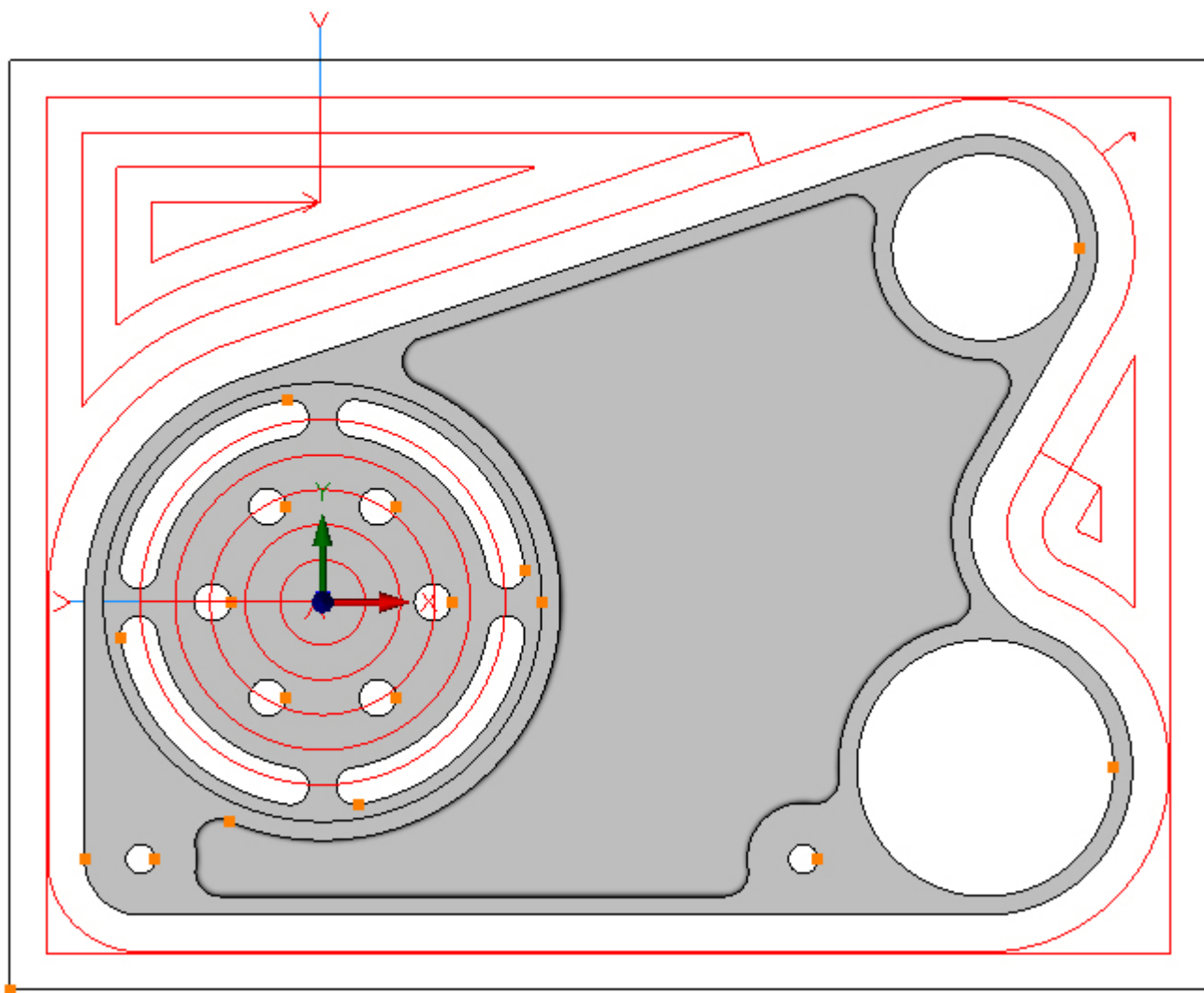
Toolbar button: 

Select the $\varnothing 4.7$ circle by directly clicking it:

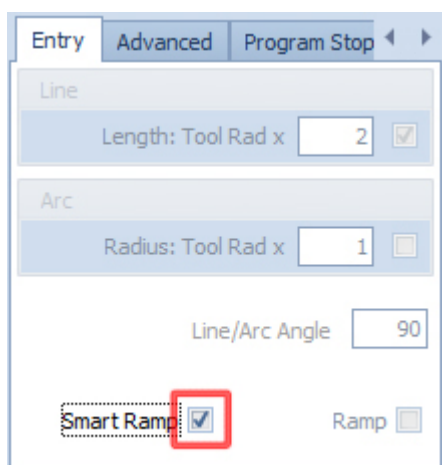


Gear Housing 3D Tutorial

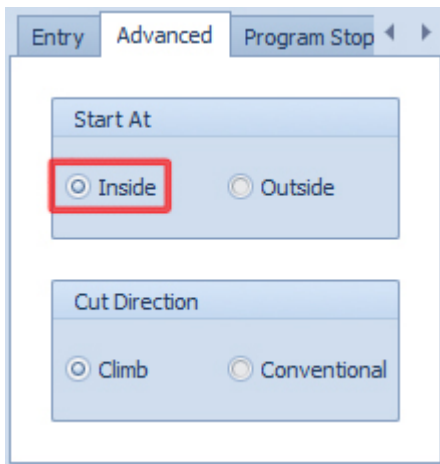
To pocket the selected Contours click the Add button . The toolpath is created:



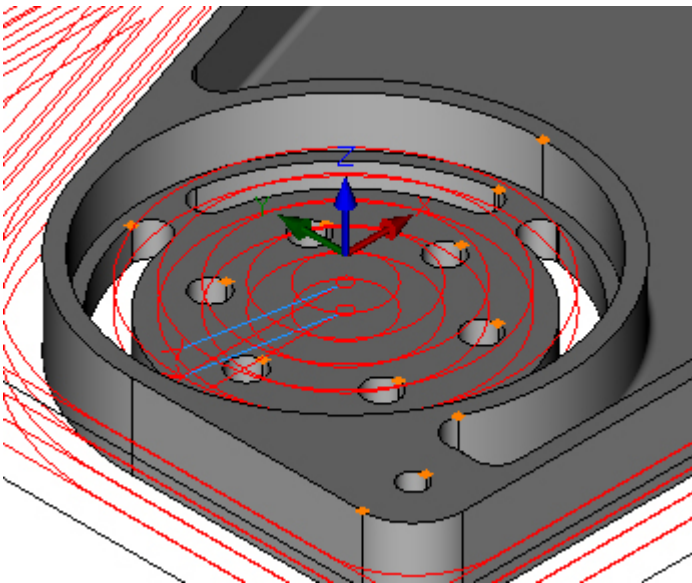
This time Smart Ramp is required, this is still unchecked from the previous pocketing operation. Choose the Entry tab and check Smart Ramp:



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:

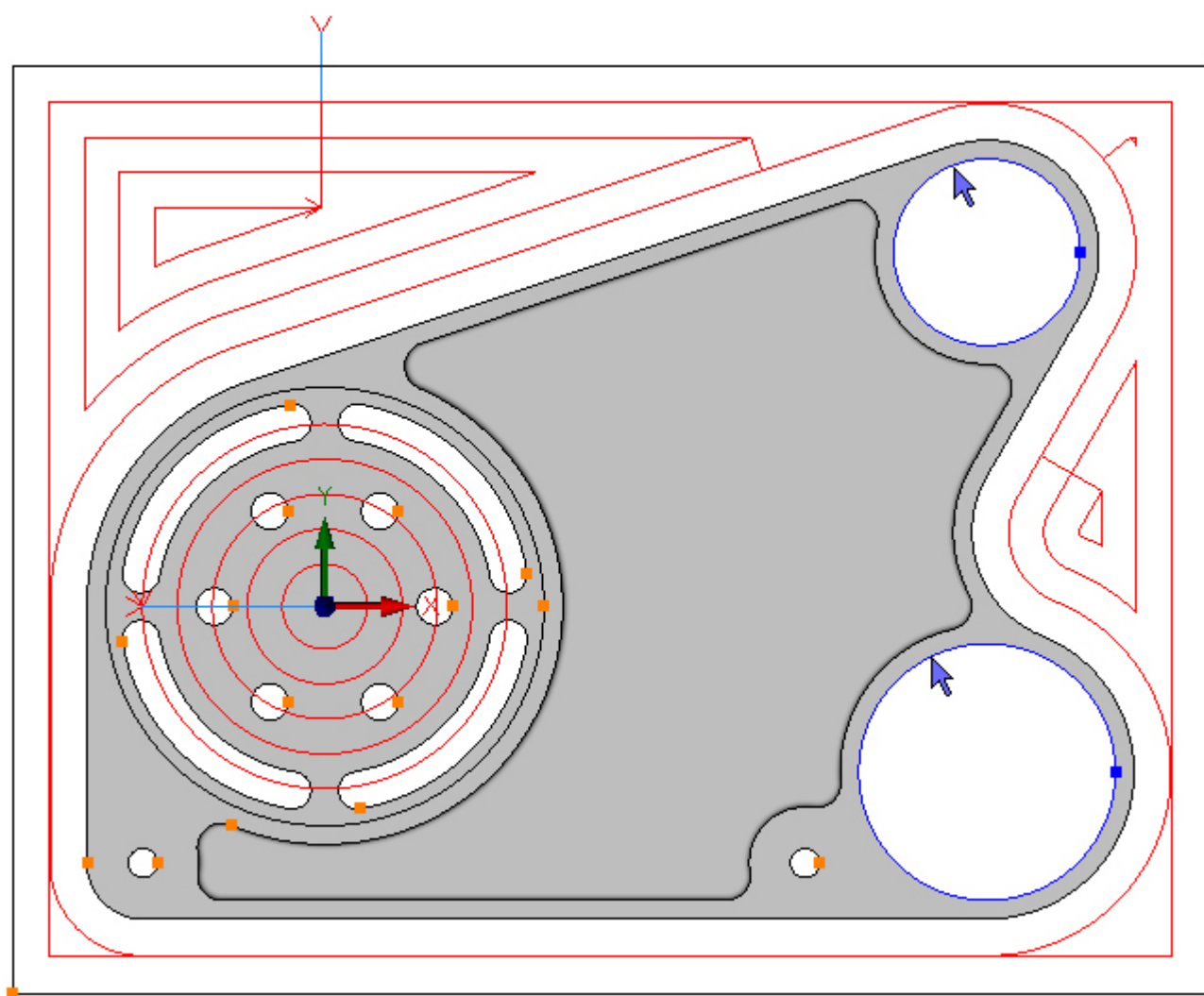
Gear Housing 3D Tutorial

Z Positions		
Abs <input checked="" type="checkbox"/>	Initial Rapid	0.1
	Feed From	0.1
	Material Surface	0
	Finish Depth	-0.1
Abs <input checked="" type="checkbox"/>	Retract	0.1
	Number of Passes	5
	Depth of Cut	 0.22

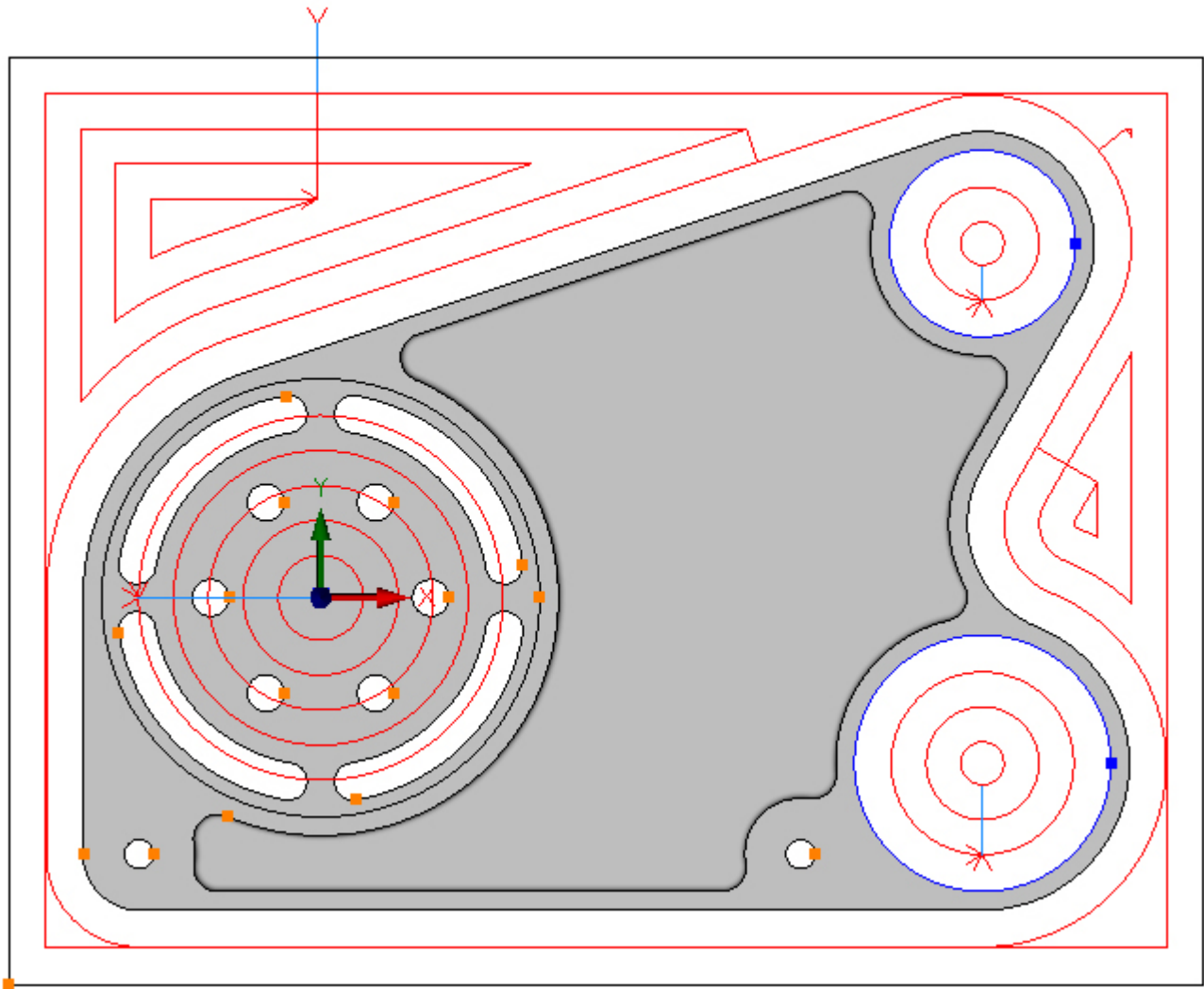
Enter the following values for the Cutting Data:

Cutting Data		Entry	Advanced	◀	▶
Spindle Speed		8000			
Plunge Feed Rate		8			
Cut Feed Rate		20			
Finish Allowance		0.02			
Step Over		0.375			
Coolant	Flood				
Sub Routines	<input type="checkbox"/>				

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:




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.

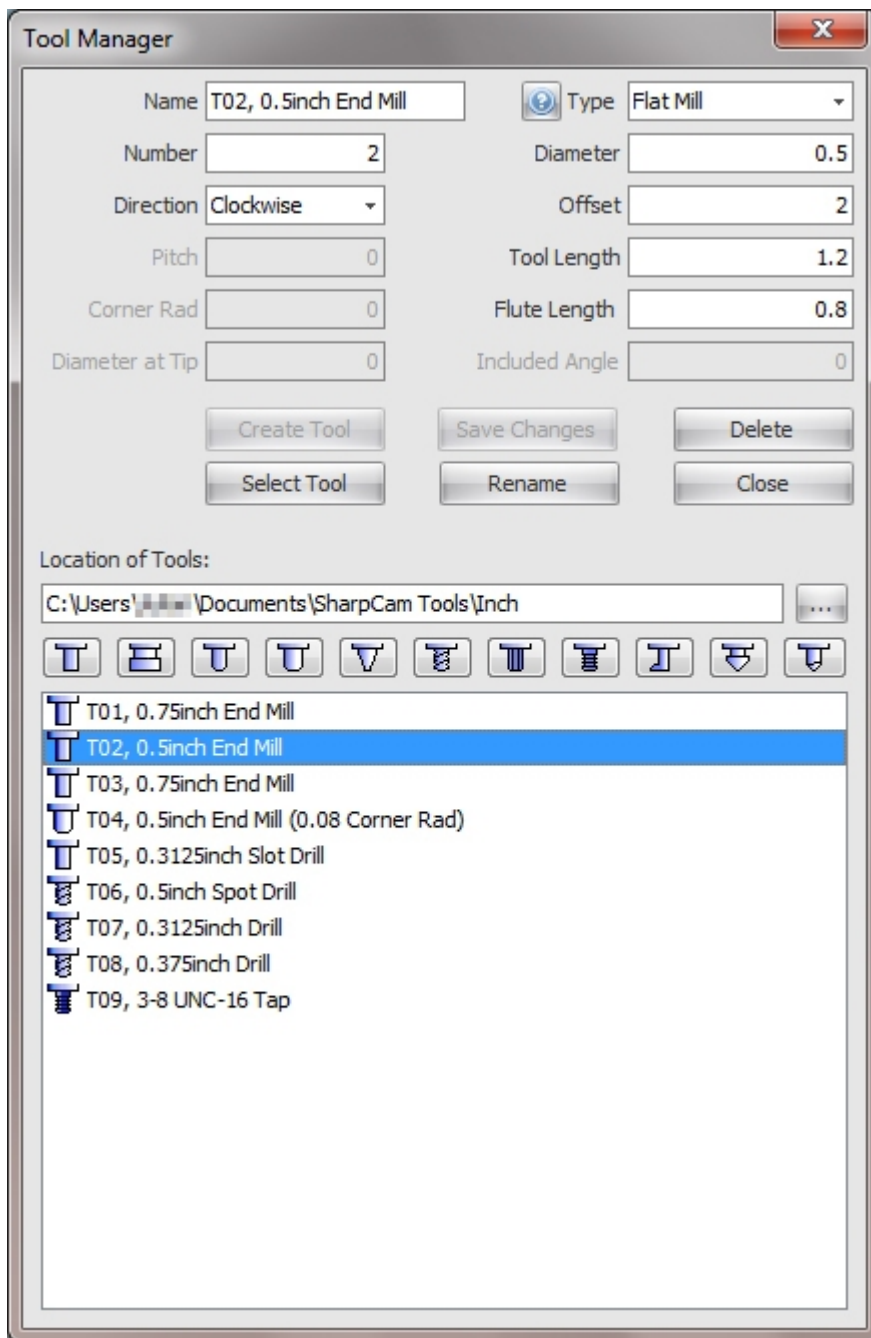
 If a tool is selected, before choosing the Pocket command, SharpCam will assume that the current operation is to be created with a different tool and will regenerate the toolpath accordingly (do not forget that the Ø2.0 and Ø2.75 bores have just been roughed and are still editing the operation).

Choose the Tool Manager:

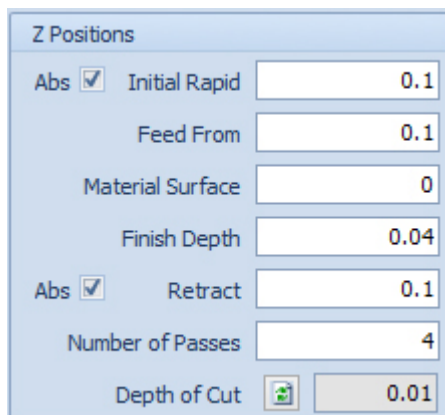
Menu: Machine-> Tool Manger

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:

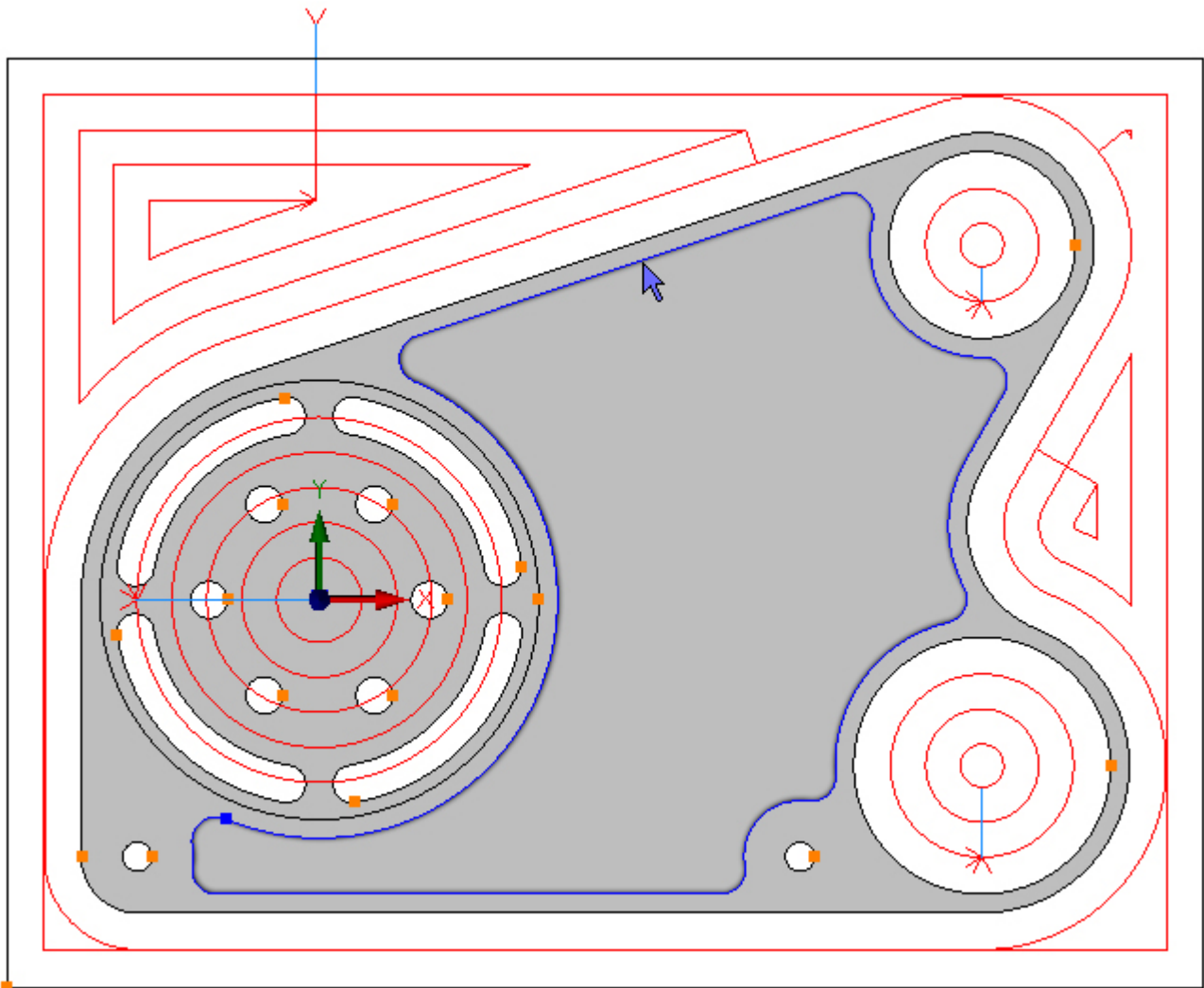


Gear Housing 3D Tutorial

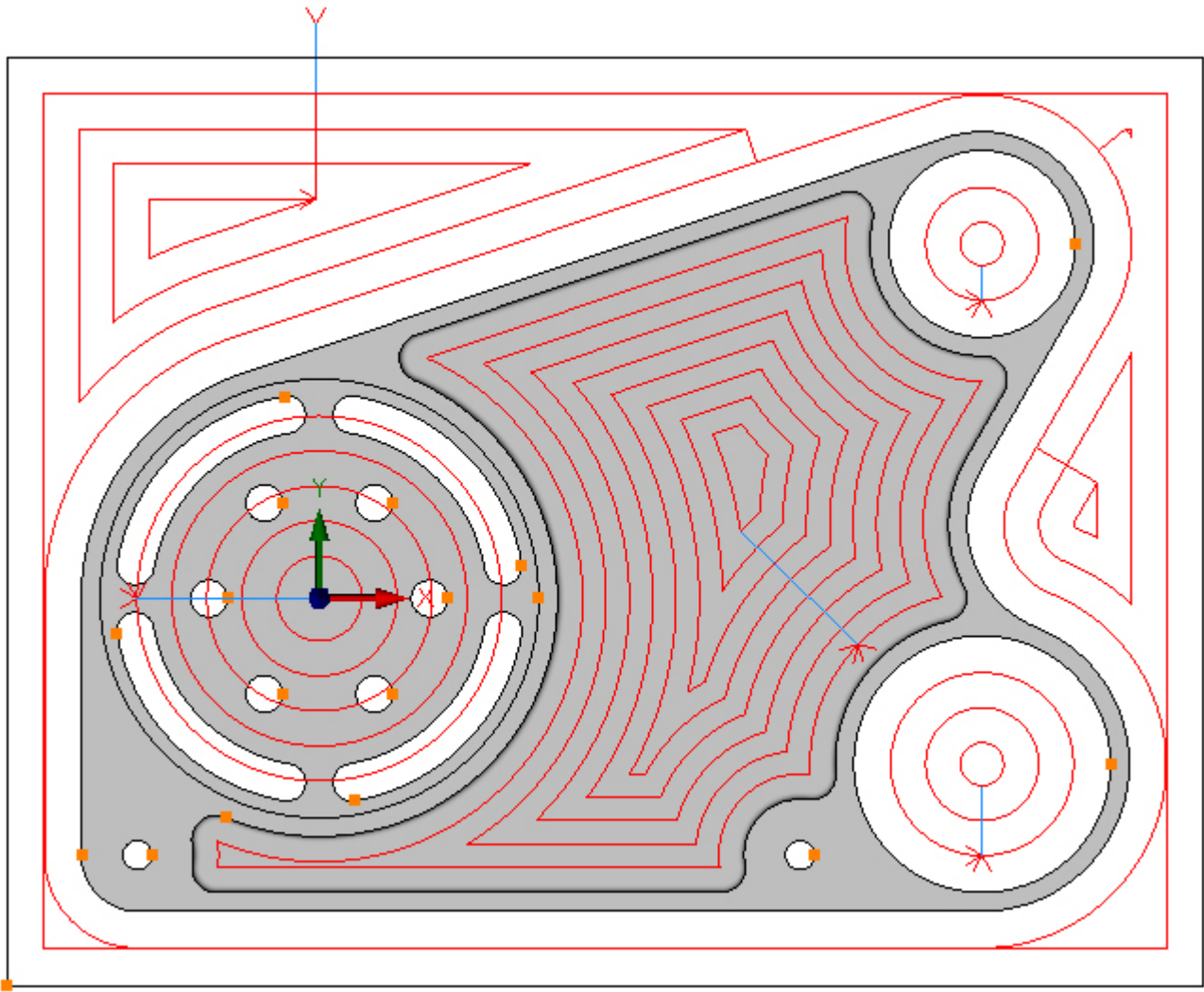
Enter the following values for the Cutting Data:

Cutting Data	
Entry	Advanced
Spindle Speed	10000
Plunge Feed Rate	40
Cut Feed Rate	100
Finish Allowance	0.02
Step Over	0.25
Coolant	Flood
Sub Routines	<input type="checkbox"/>

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: 

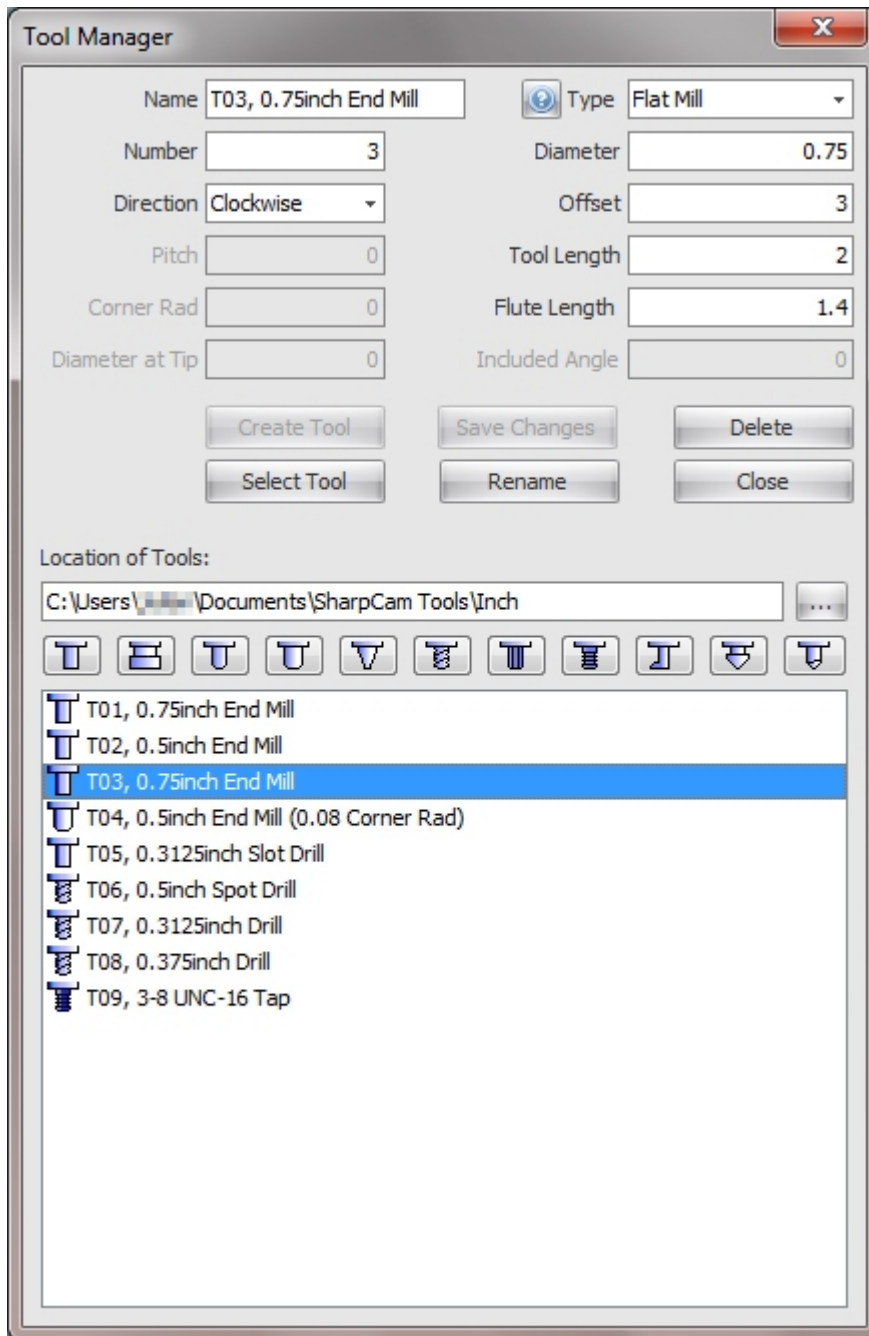
Choose the Profile command then select the tool for this operation:

Menu: Machine-> Tool Manger

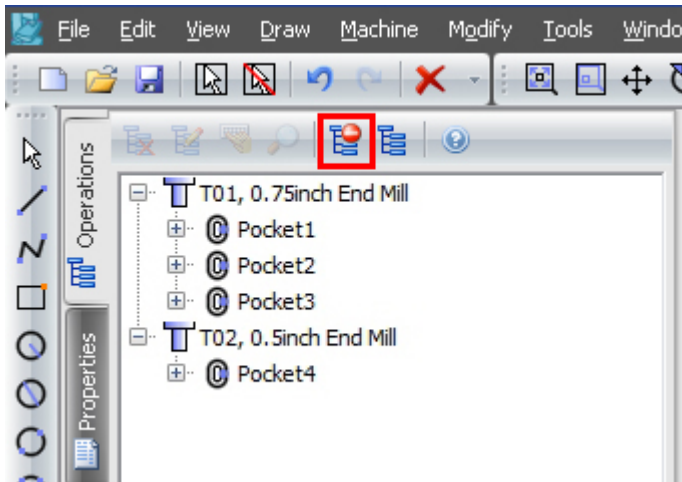
Toolbar button: 

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:

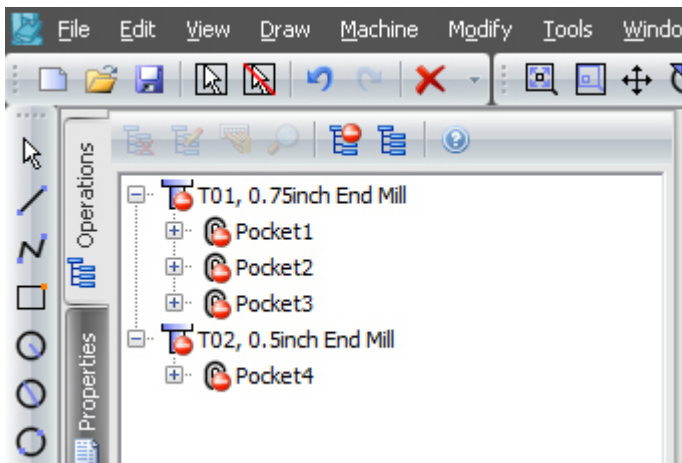
Gear Housing 3D Tutorial



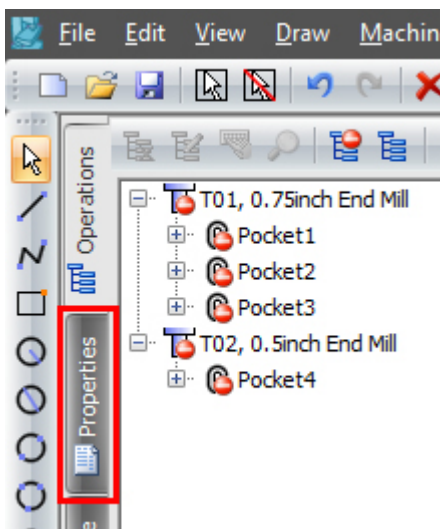
The Part now has a number of toolpaths. In order to prevent the view from being obscured, suppress them so they are hidden. Choose the Operations tab on the Part Manager and click the Suppress All button:



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:

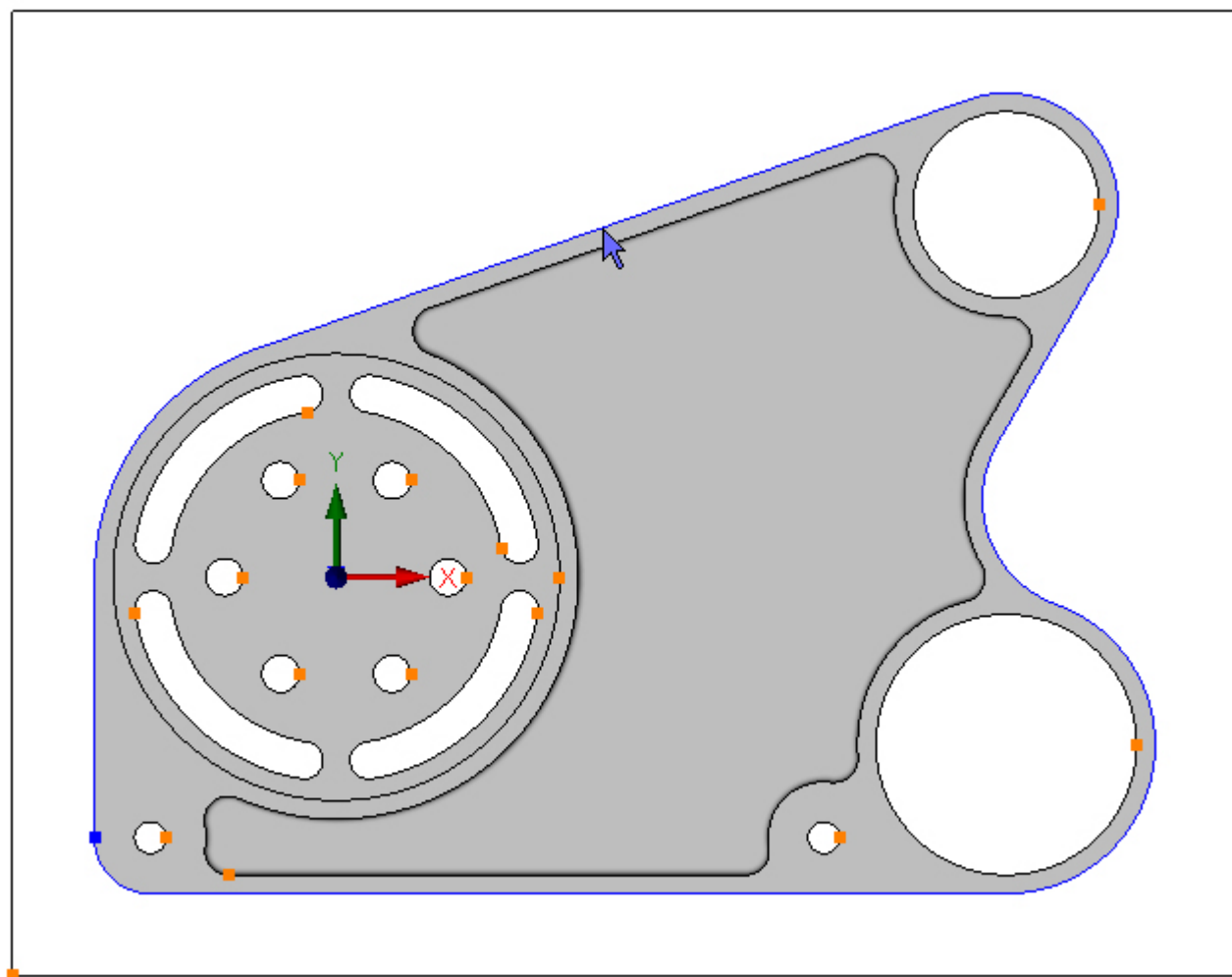
Gear Housing 3D Tutorial


Z Positions	
Abs <input checked="" type="checkbox"/>	Initial Rapid <input type="text" value="2"/>
	Feed From <input type="text" value="0.1"/>
	Material Surface <input type="text" value="0"/>
	Finish Depth <input type="text" value="-0.04"/>
Abs <input checked="" type="checkbox"/>	Retract <input type="text" value="0.1"/>
	Number of Passes <input type="text" value="1"/>
	Depth of Cut  <input type="text" value="0.04"/>

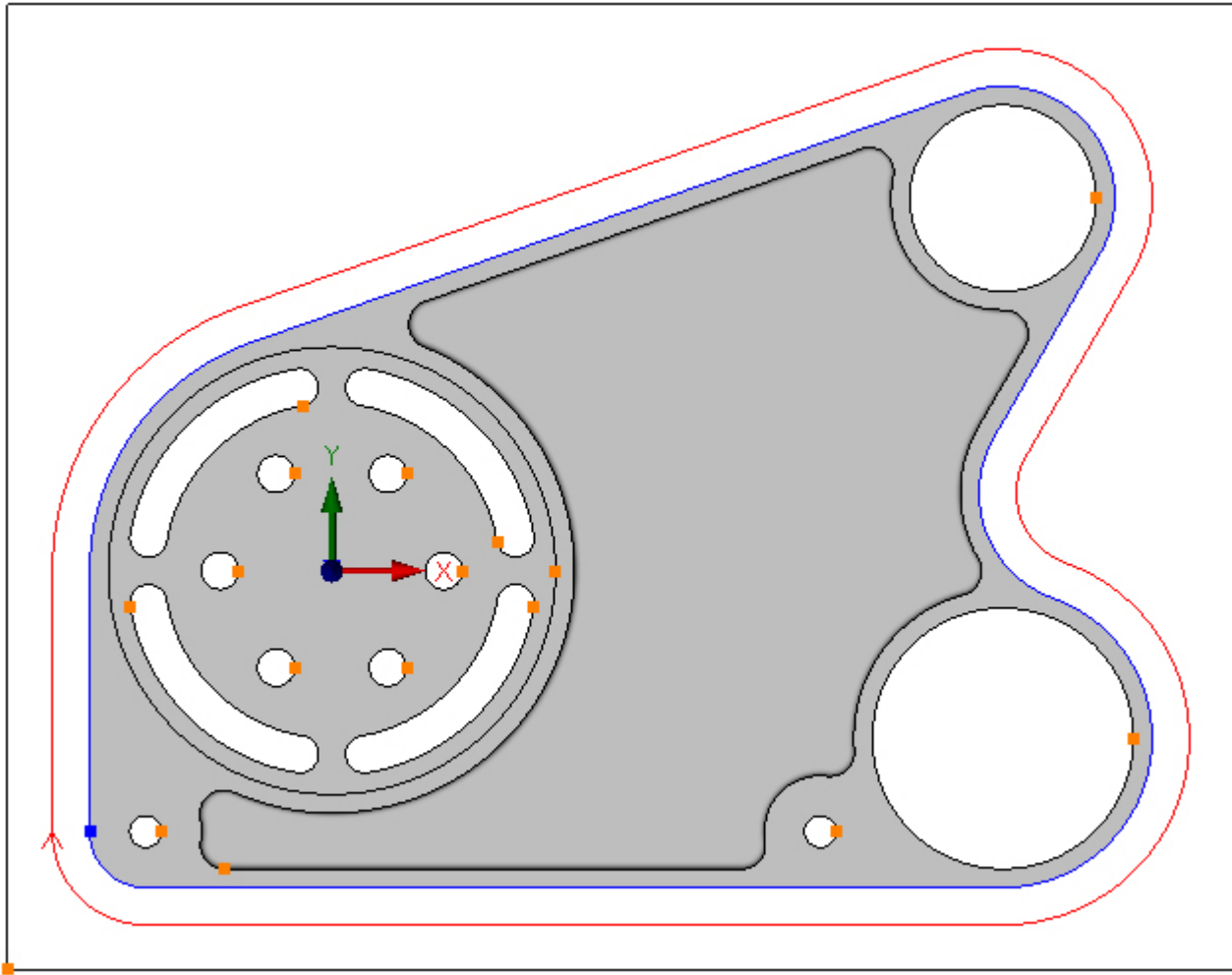
Enter the following values for the Cutting Data:

Cutting Data	Entry	Exit	Proj	◀	▶
Spindle Speed	<input type="text" value="8000"/>				
Plunge Feed Rate	<input type="text" value="20"/>				
Cut Feed Rate	<input type="text" value="50"/>				
Finish Allowance	<input type="text" value="0"/>				
Coolant	<input type="text" value="Flood"/>				
Cutter Radius Comp	<input type="text" value="None"/>				
Sub Routines	<input type="checkbox"/>				

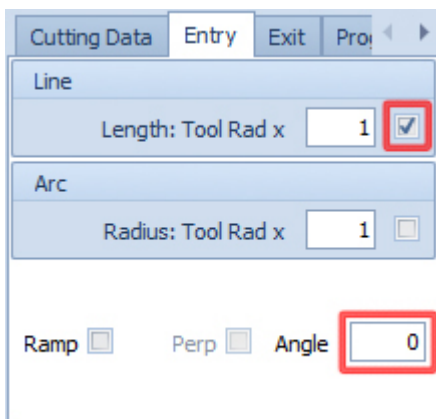
Select the outside Contour by directly clicking it:



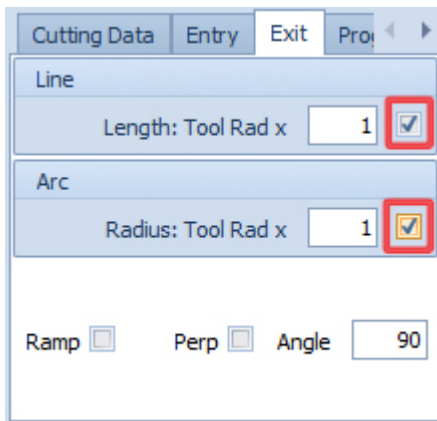
To profile the selected Contour click the Add button . The toolpath is created:



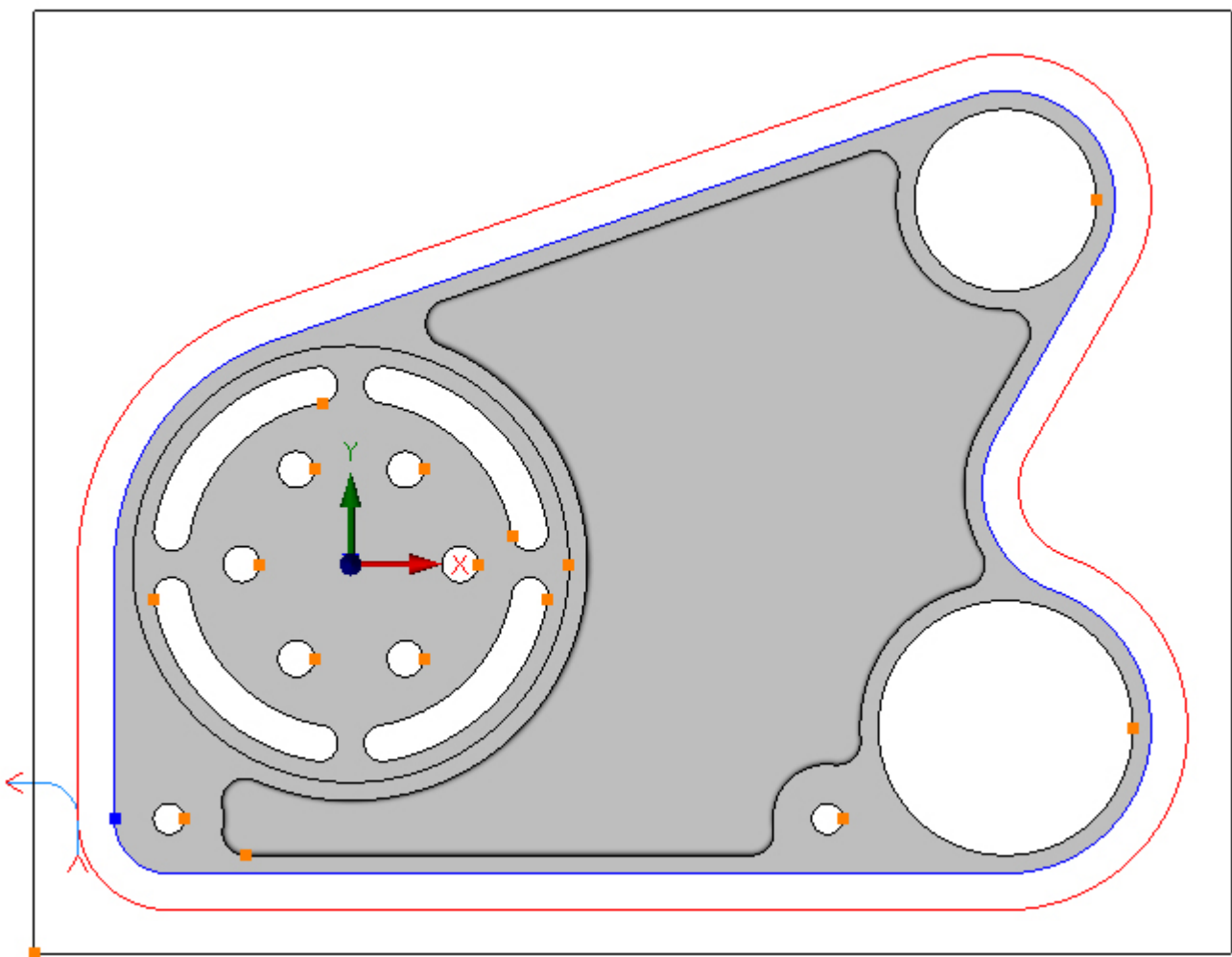
Add an entry and exit. Choose the Entry tab and change the Line/Arc Angle to 0 and check the Line check box:



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

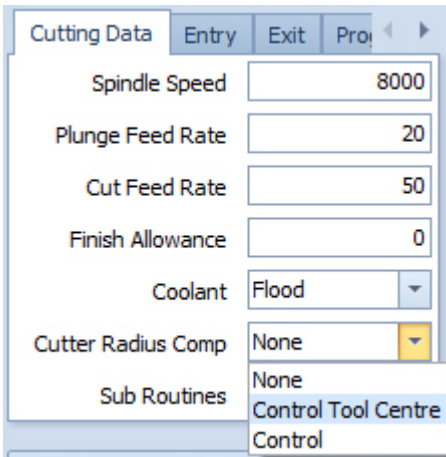


The entry and exit have been added:



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

Gear Housing 3D Tutorial




Parameter	Value
Spindle Speed	8000
Plunge Feed Rate	20
Cut Feed Rate	50
Finish Allowance	0
Coolant	Flood
Cutter Radius Comp	None
Sub Routines	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*.

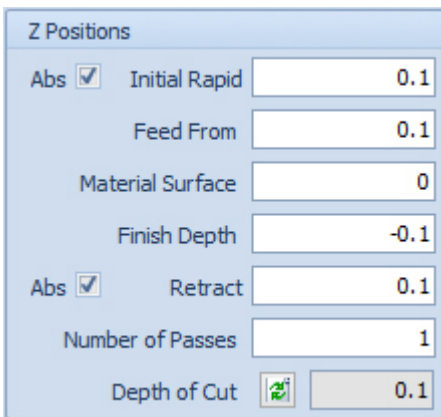
Step 9 - Finish bores


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:

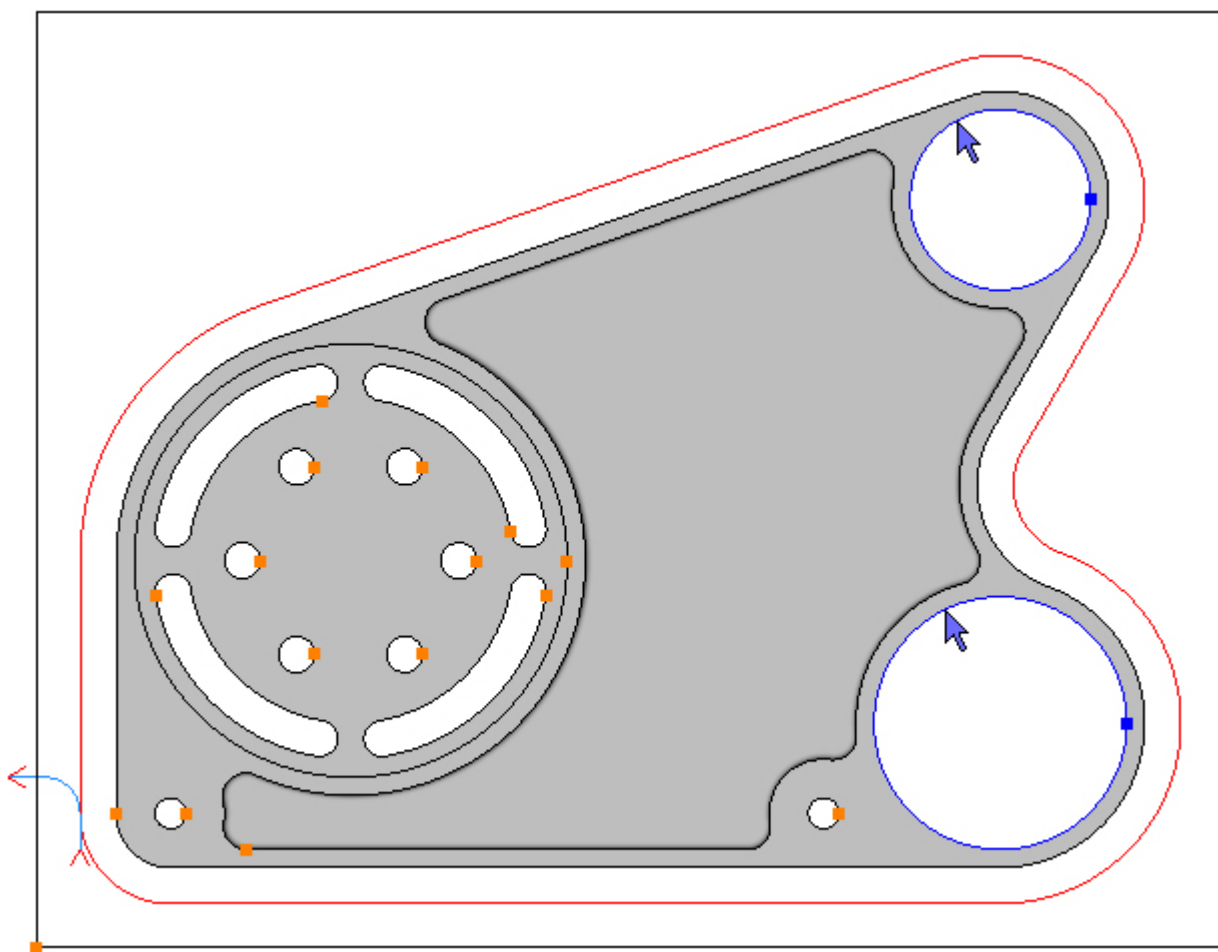


Z Positions	
Abs <input checked="" type="checkbox"/>	Initial Rapid 0.1
	Feed From 0.1
	Material Surface 0
	Finish Depth -0.1
Abs <input checked="" type="checkbox"/>	Retract 0.1
	Number of Passes 1
	Depth of Cut  0.1

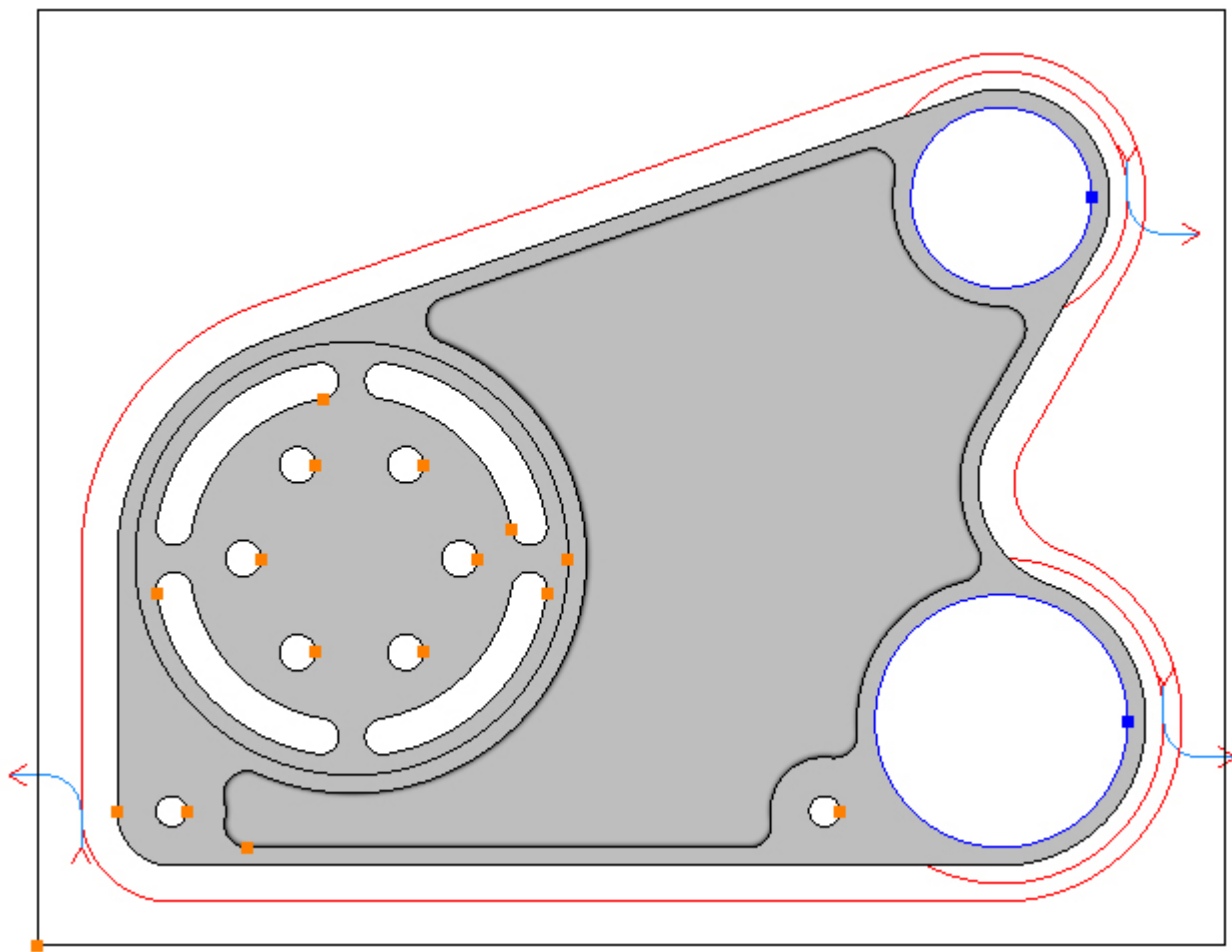
Enter the following values for the Cutting Data:

Cutting Data	Entry	Exit	Proj	
Spindle Speed	8000			
Plunge Feed Rate	80			
Cut Feed Rate	40			
Finish Allowance	0			
Coolant	Flood			
Cutter Radius Comp	None			
Sub Routines	<input type="checkbox"/>			

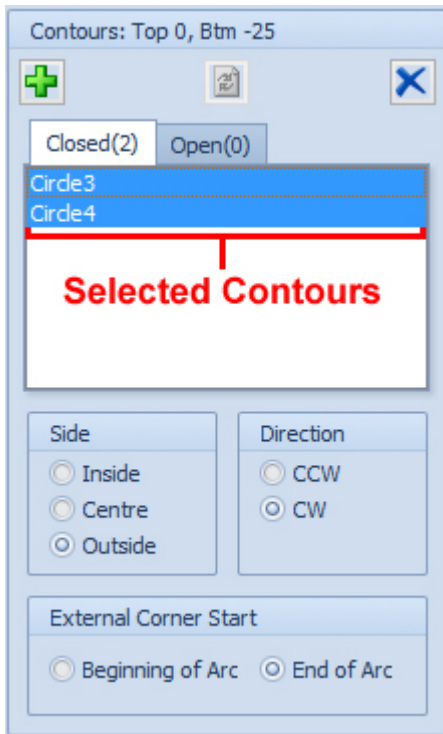
Select the Ø2.0 and Ø2.75 circles by directly clicking them:



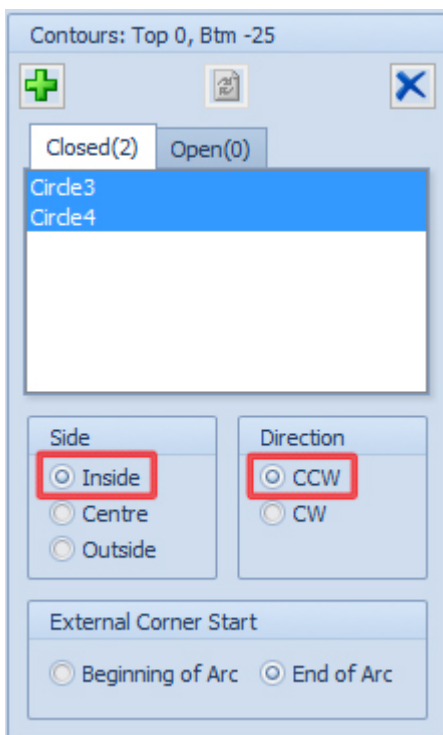
To profile the selected Contours click the Add button . The toolpaths are created:



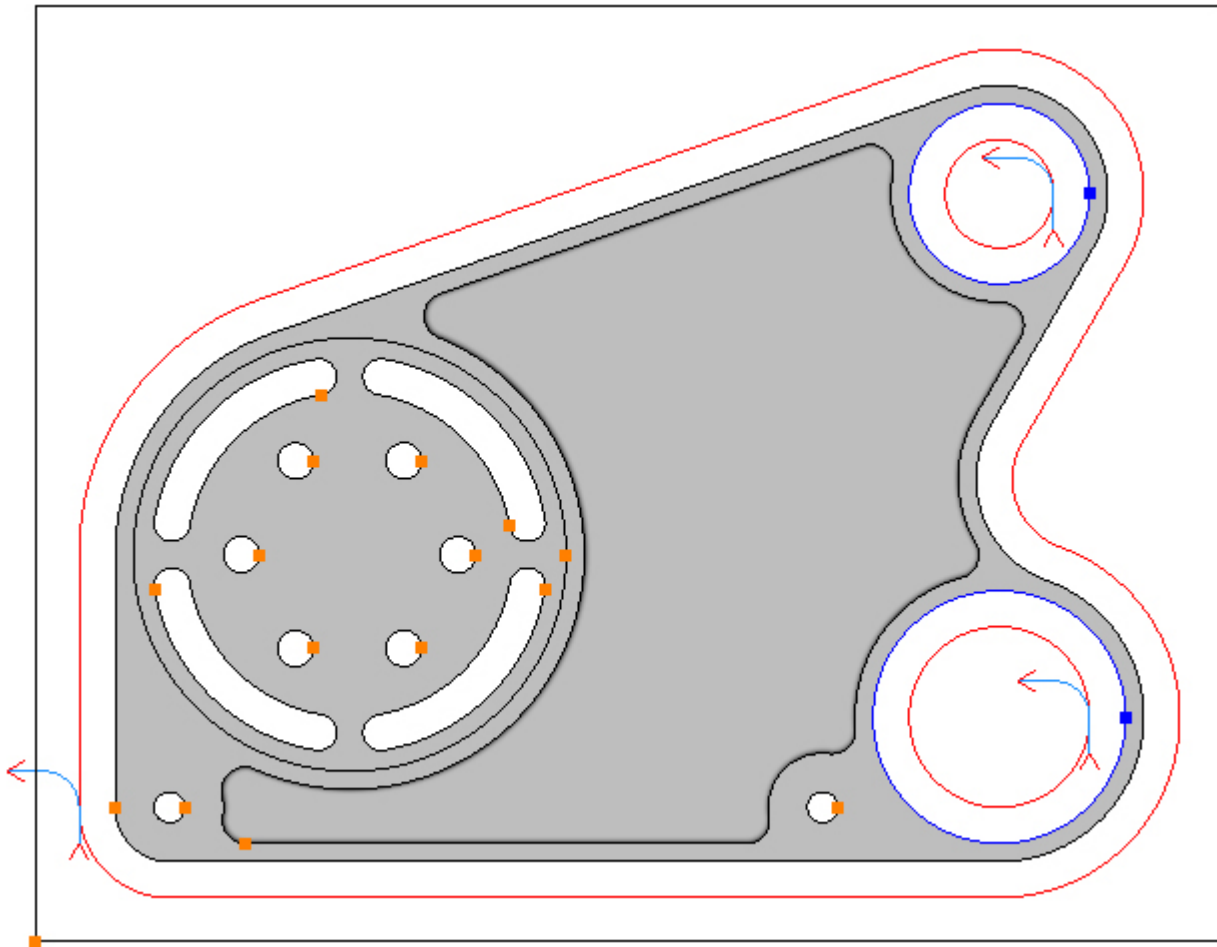
As can be seen the toolpaths are on the outside and in a clockwise direction, this is the default. This needs to be changed to the inside in a counter-clockwise direction. The Contours must be selected in order to change the side and direction of the resulting toolpaths. When Contours are added to an 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* group box:



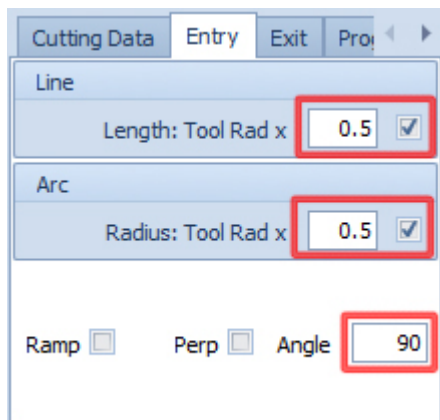
To change the side and direction click the appropriate radio button, in this case *Inside* and *CCW*:




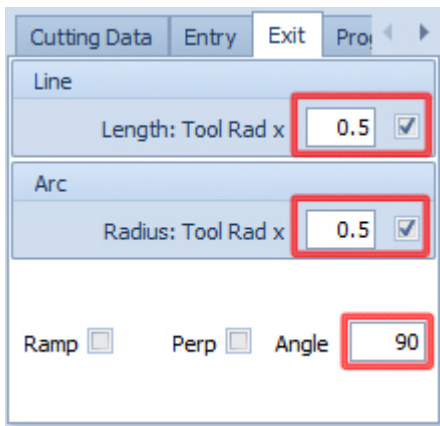
The toolpath is now on the inside and in a counter-clockwise direction:



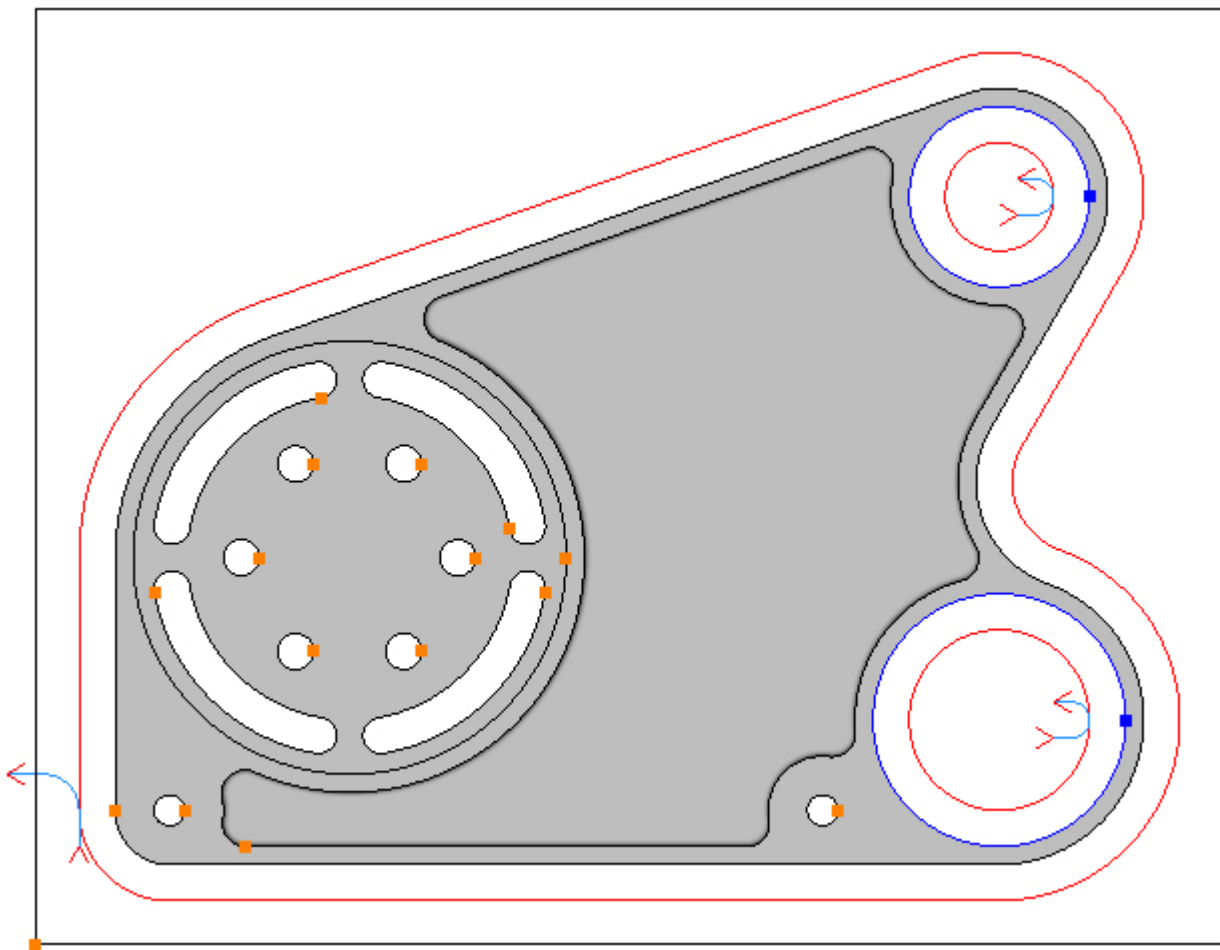
Change the entry and exit because the settings from the previous operation are still in effect. Choose the Entry tab and set the values (before checking the boxes) as below and check the line and arc check box:



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*, if not already selected:

Gear Housing 3D Tutorial

Cutting Data		Entry	Exit
Spindle Speed		8000	
Plunge Feed Rate		80	
Cut Feed Rate		40	
Finish Allowance		0	
Coolant		Flood	
Cutter Radius Comp		Control Tool Ce	
Sub Routines		<input type="checkbox"/>	

Deselect the Contours in readiness for the next operation:


Menu: Edit -> Deselect All

Toolbar button: 


Step 10 - Finish counterbore

Finish Ø4.7 counterbore using Pocket

Menu: Machine-> Pocket

Toolbar button: 

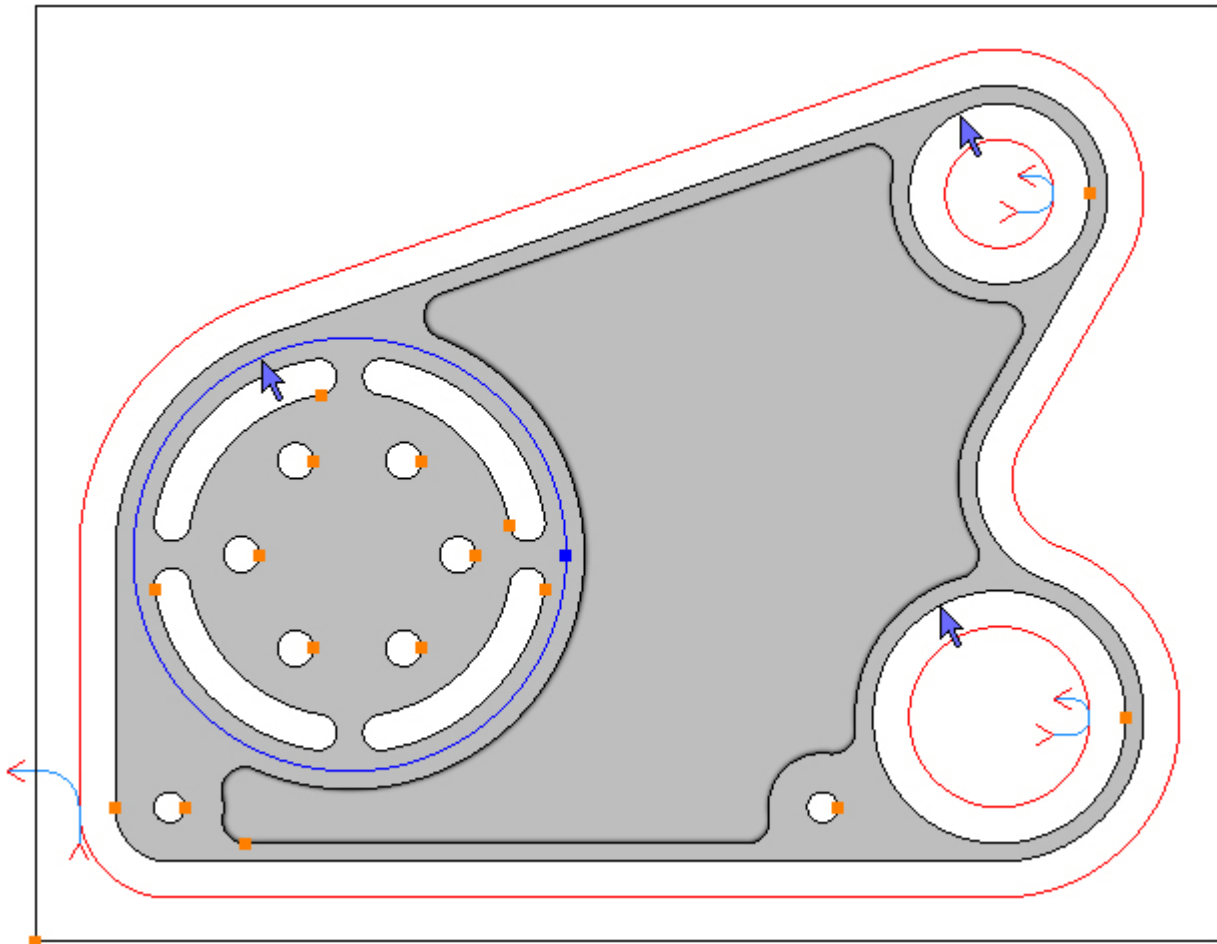
Choose the command and enter the following values for the Z Positions:

Z Positions	
Abs <input checked="" type="checkbox"/>	Initial Rapid <input type="text" value="0.1"/>
	Feed From <input type="text" value="-0.5"/>
	Material Surface <input type="text" value="-0.58"/>
	Finish Depth <input type="text" value="0"/>
Abs <input checked="" type="checkbox"/>	Retract <input type="text" value="0.1"/>
	Number of Passes <input type="text" value="1"/>
	Depth of Cut  <input type="text" value="0.02"/>

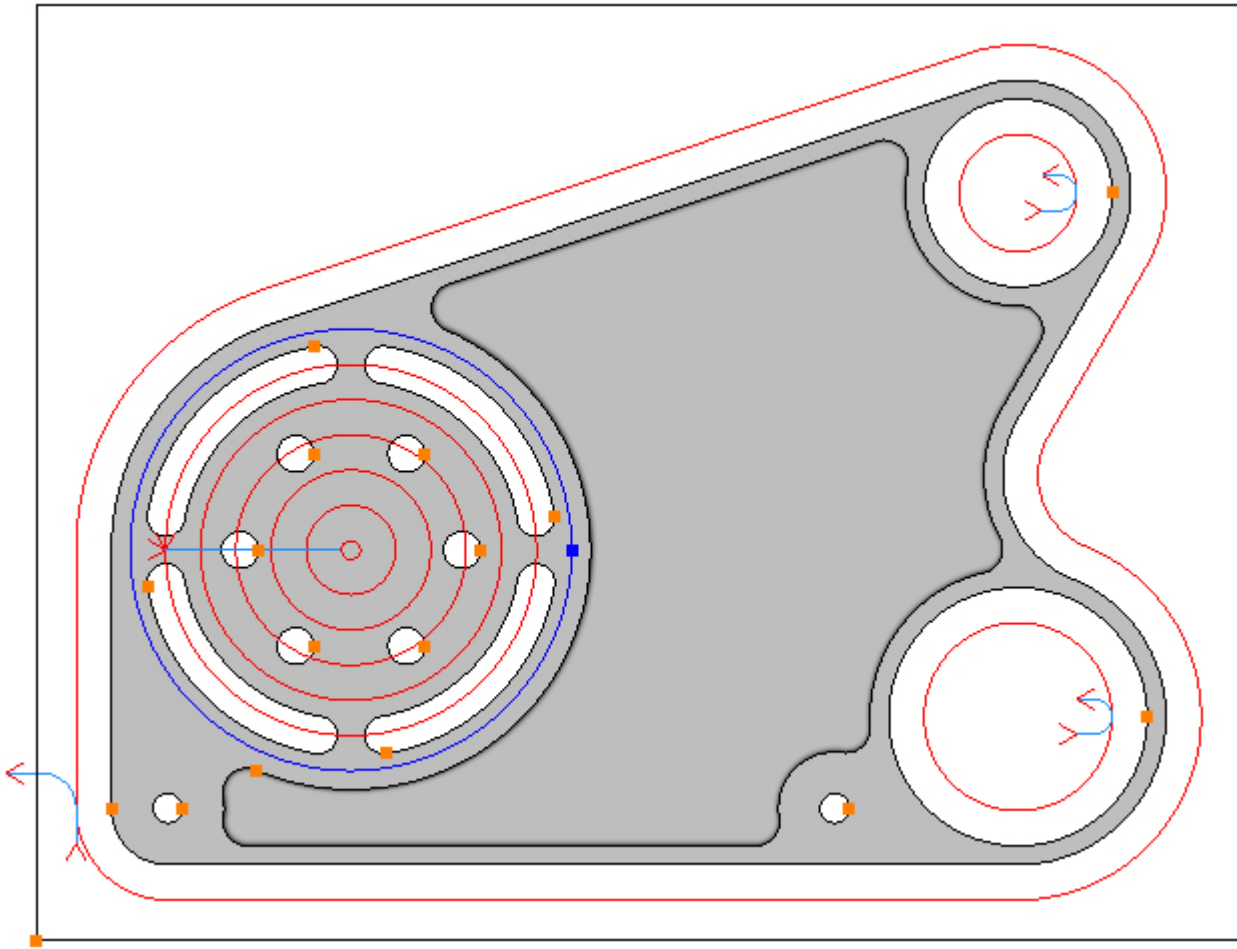
Enter the following values for the Cutting Data:

Cutting Data	Entry	Advanced
Spindle Speed	<input type="text" value="8000"/>	
Plunge Feed Rate	<input type="text" value="40"/>	
Cut Feed Rate	<input type="text" value="80"/>	
Finish Allowance	<input type="text" value="0"/>	
Step Over	<input type="text" value="0.375"/>	
Coolant	<input type="text" value="Flood"/>	
Sub Routines	<input type="checkbox"/>	

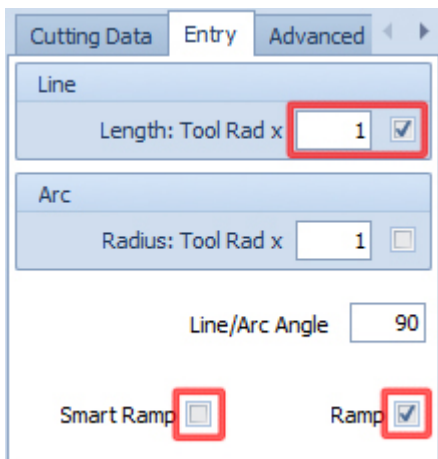
First deselect the bores from the previous operation, if required, by directly clicking them, then select the counterbore Contour by directly clicking it:



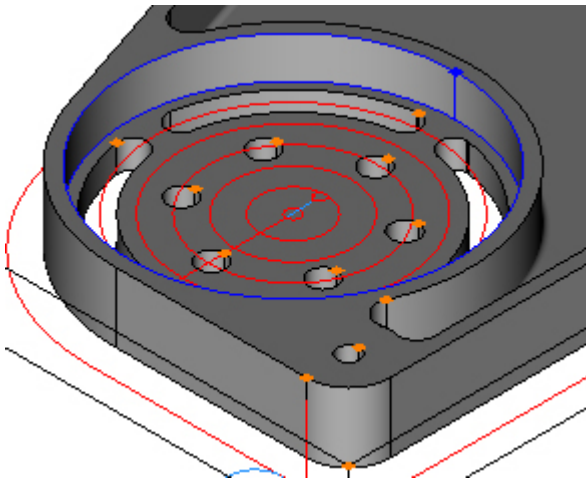
To profile the selected Contour click the Add button . The toolpath is created:



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 (if not already selected) and then the Ramp check box:




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



Step 11 - Finish inside


Finish inside pocket using Pocket

Menu: Machine-> Pocket

Toolbar button: 

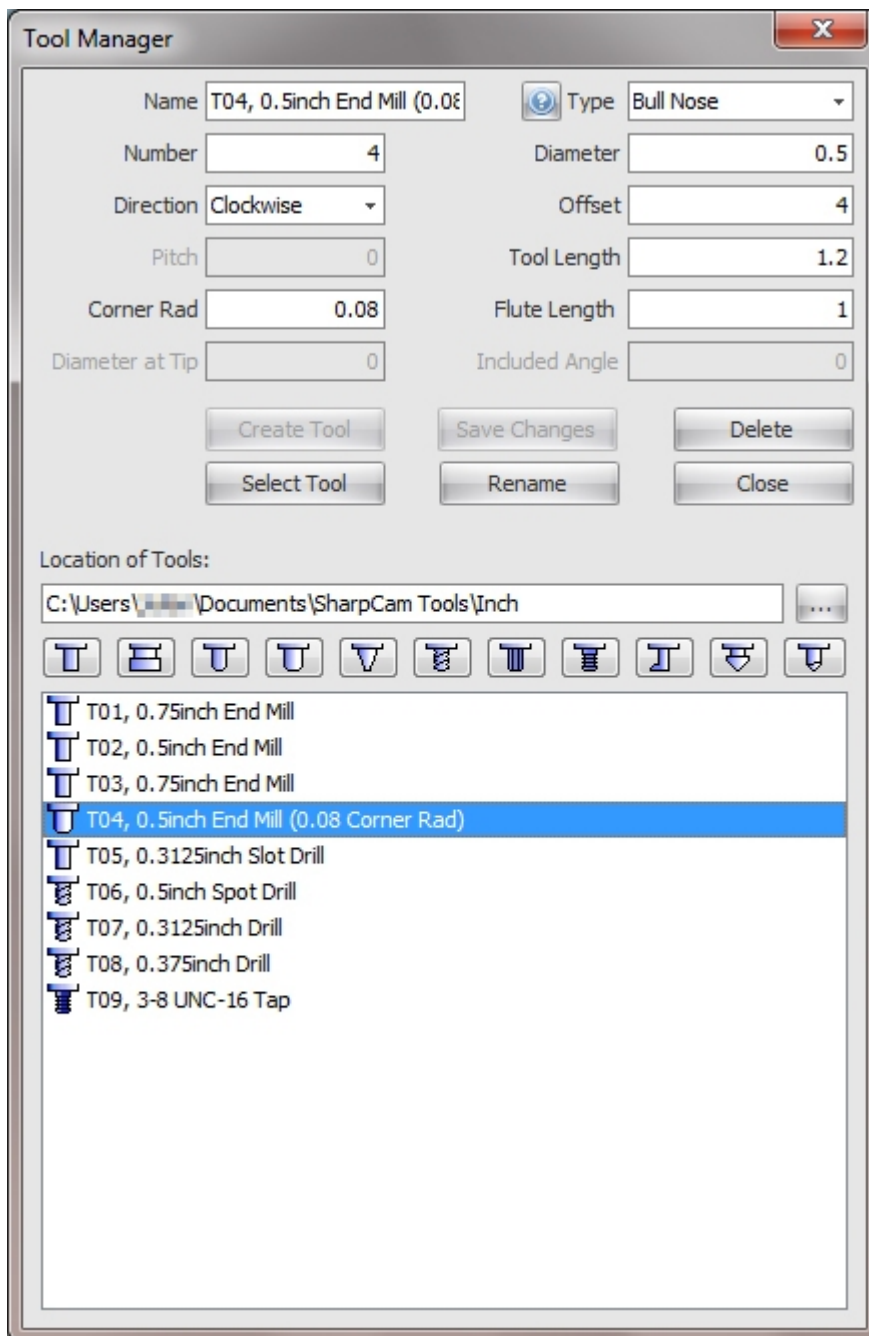
Choose the command and then select the tool for this operation:

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:

Gear Housing 3D Tutorial



The Tool Manager dialog box is used to configure and manage tools. It includes fields for Name, Number, Direction, Pitch, Corner Rad, Diameter at Tip, Type, Diameter, Offset, Tool Length, Flute Length, and Included Angle. It also has buttons for Create Tool, Save Changes, Delete, Select Tool, Rename, and Close. A section for Location of Tools shows the path C:\Users\...\Documents\SharpCam Tools\Inch and a list of tools, with T04, 0.5inch End Mill (0.08 Corner Rad) selected.

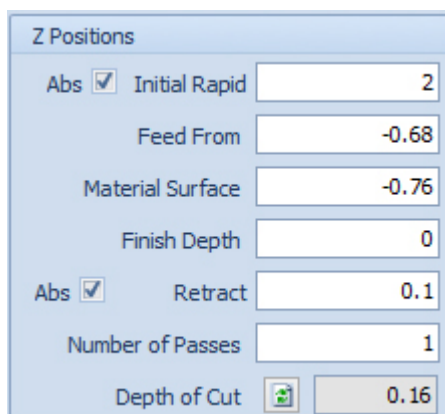
Field	Value
Name	T04, 0.5inch End Mill (0.08
Number	4
Direction	Clockwise
Pitch	0
Corner Rad	0.08
Diameter at Tip	0
Type	Bull Nose
Diameter	0.5
Offset	4
Tool Length	1.2
Flute Length	1
Included Angle	0

Location of Tools:
C:\Users\...\Documents\SharpCam Tools\Inch

Tools List:

- T01, 0.75inch End Mill
- T02, 0.5inch End Mill
- T03, 0.75inch End Mill
- T04, 0.5inch End Mill (0.08 Corner Rad)**
- T05, 0.3125inch Slot Drill
- T06, 0.5inch Spot Drill
- T07, 0.3125inch Drill
- T08, 0.375inch Drill
- T09, 3-8 UNC-16 Tap

Enter the following values for the Z Positions:



The Z Positions dialog box is used to set Z-axis parameters. It includes checkboxes for Abs and Initial Rapid, and input fields for Feed From, Material Surface, Finish Depth, Retract, Number of Passes, and Depth of Cut.

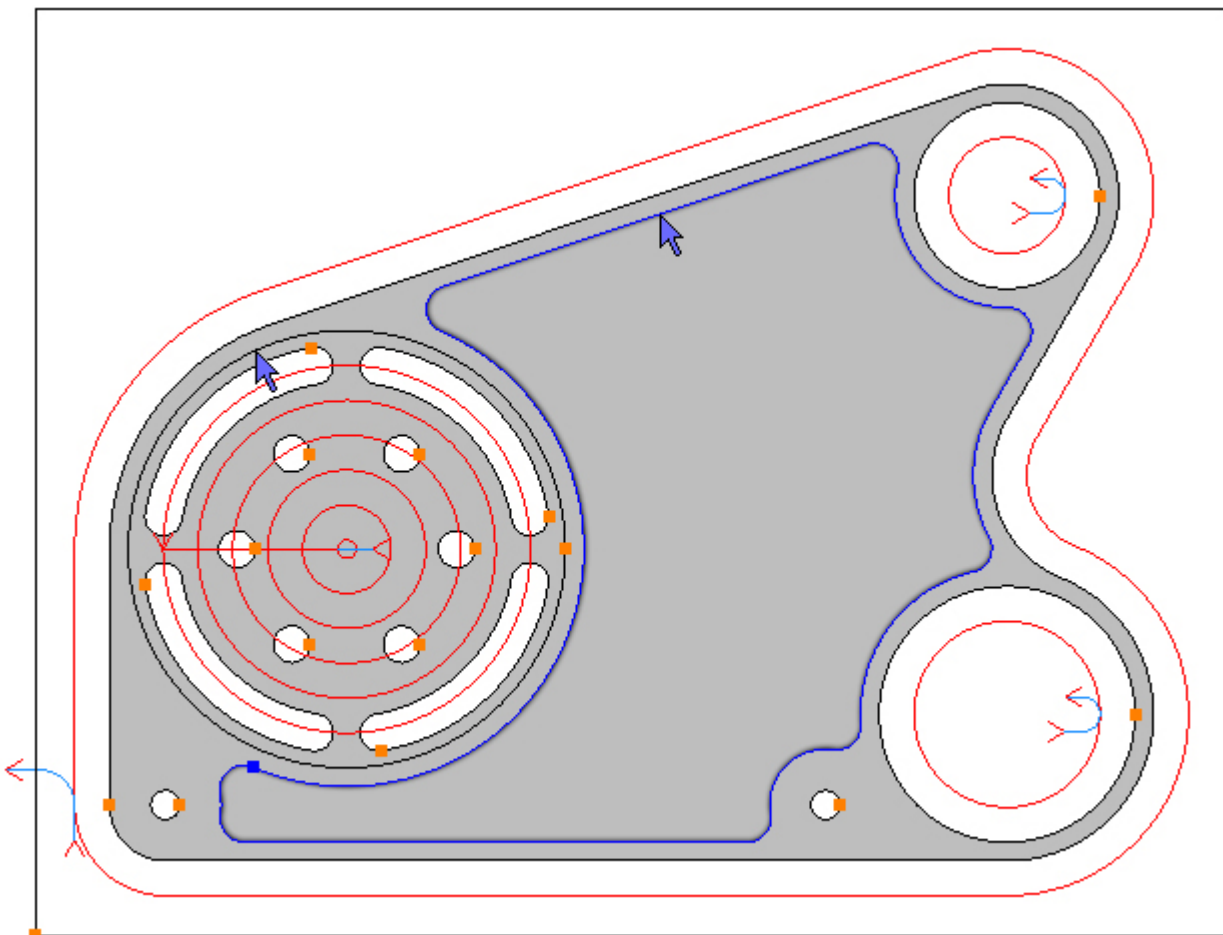
Field	Value
Abs	<input checked="" type="checkbox"/>
Initial Rapid	<input checked="" type="checkbox"/>
Feed From	-0.68
Material Surface	-0.76
Finish Depth	0
Retract	0.1
Number of Passes	1
Depth of Cut	0.16

Enter the following values for the Cutting Data.

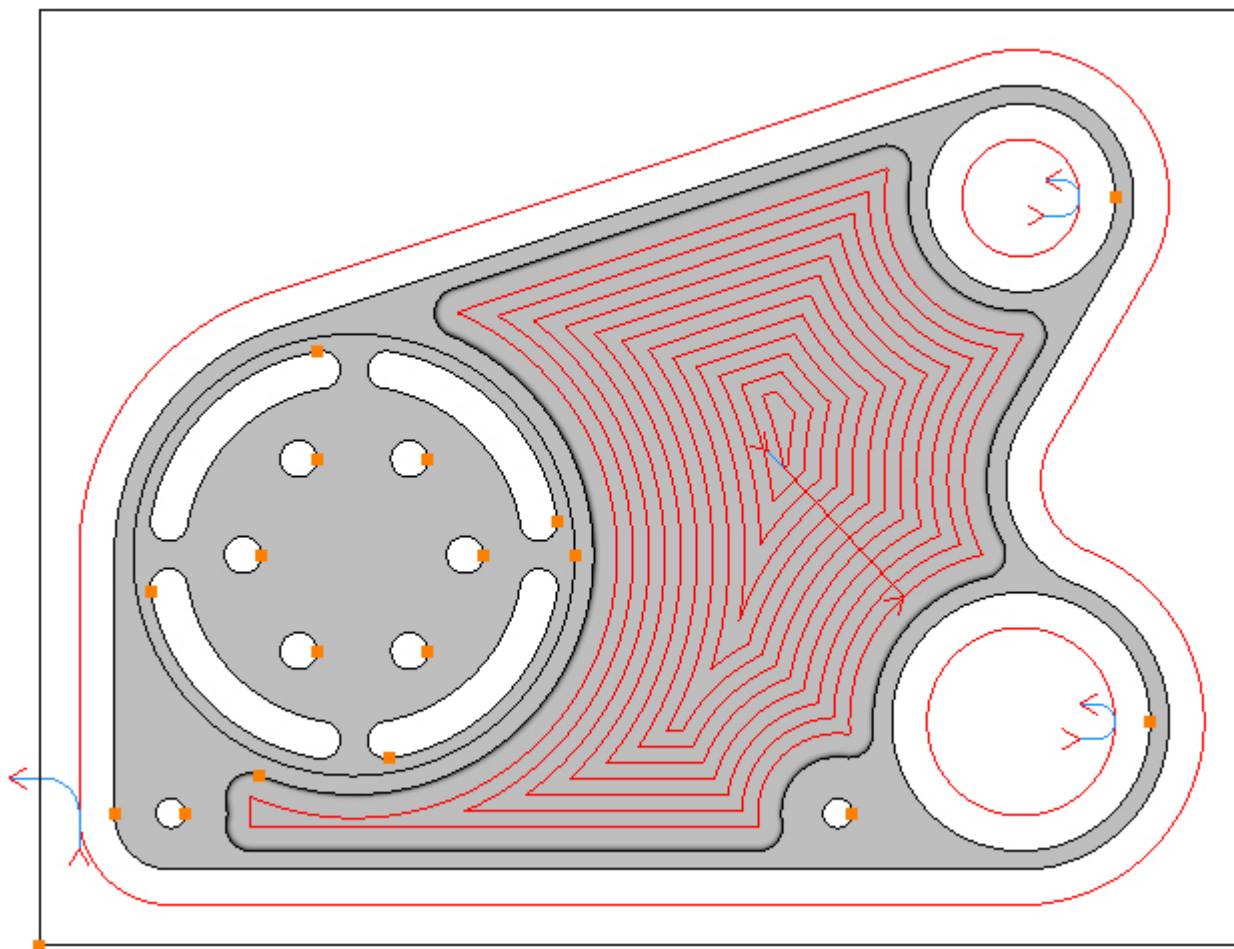
Note that the step over value has been changed from the default value of 0.25 to 0.17 to ensure that pocket fully cleans up on the bottom, don't forget that the cutter has a 0.08 corner radius:

Cutting Data	
Entry	Advanced
Spindle Speed	10000
Plunge Feed Rate	40
Cut Feed Rate	80
Finish Allowance	0
Step Over	0.17
Coolant	Flood
Sub Routines	<input type="checkbox"/>

First deselect the counterbore from the previous operation, if required, by directly clicking it, then 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.

Step 12 - Machine radial slots


Machine radial slots using Profile

Menu: Machine-> Profile

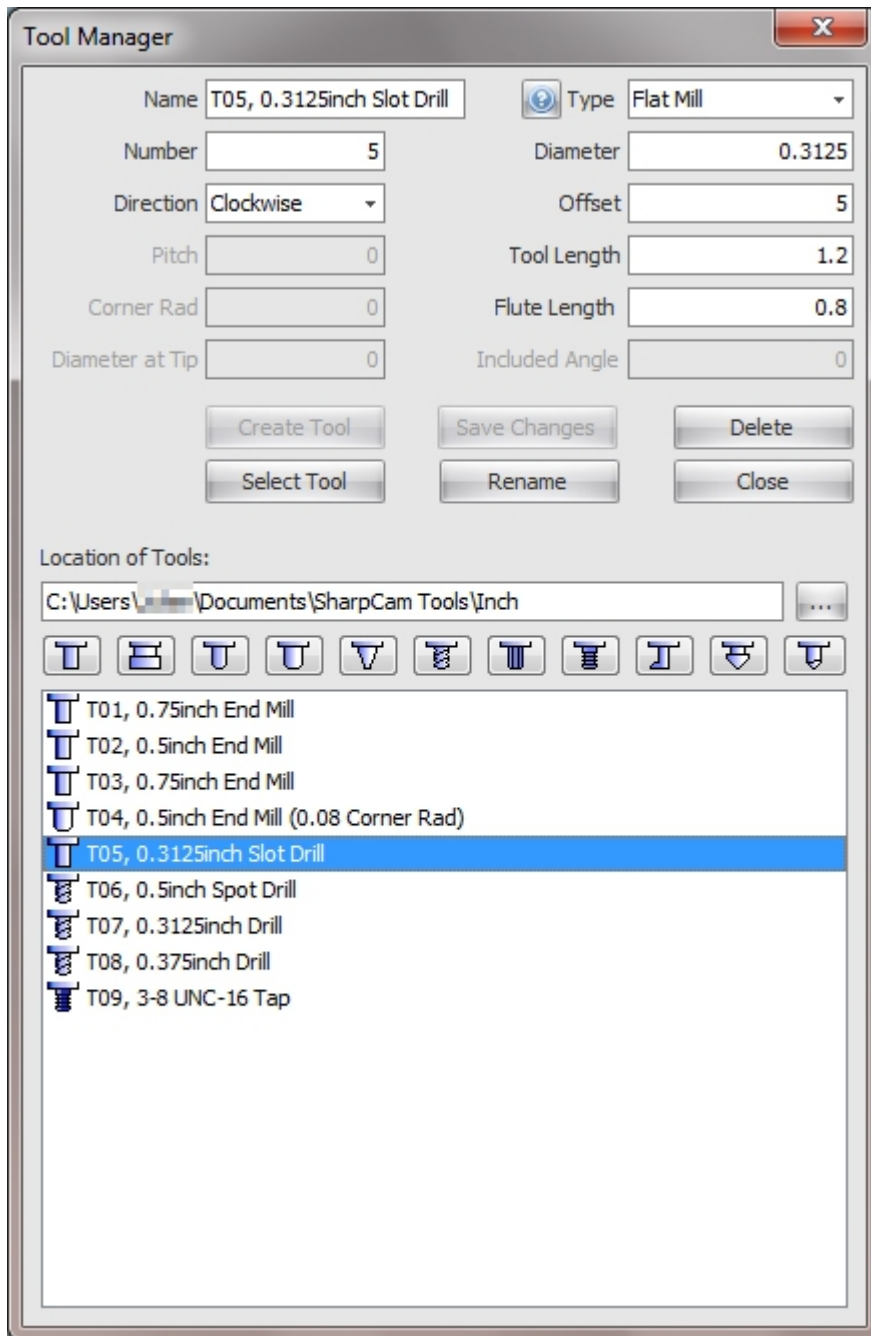
Toolbar button: 

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

Menu: Machine-> Tool Manger

Toolbar button: 

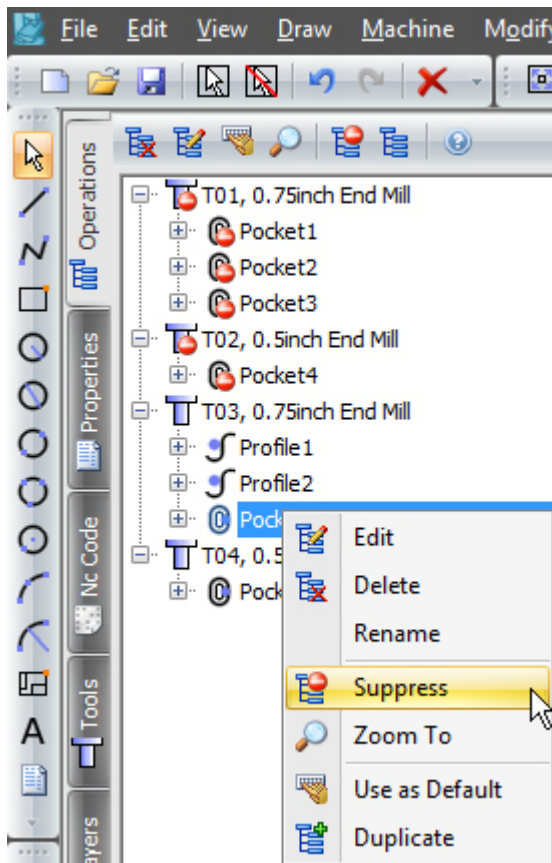
Choose the command to display the Tool Manager and click on 'T05, 0.3125inch Slot Drill', then click the 'Select Tool' button. Alternatively double click a tool to select it:



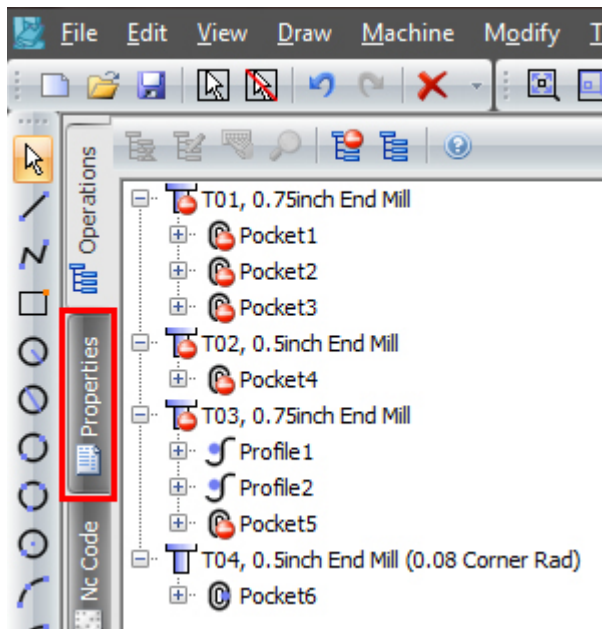
Before machining the slots suppress the operation that machined the $\varnothing 4.7$ counterbore, so it does not obscure the view.

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:

Gear Housing 3D Tutorial



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



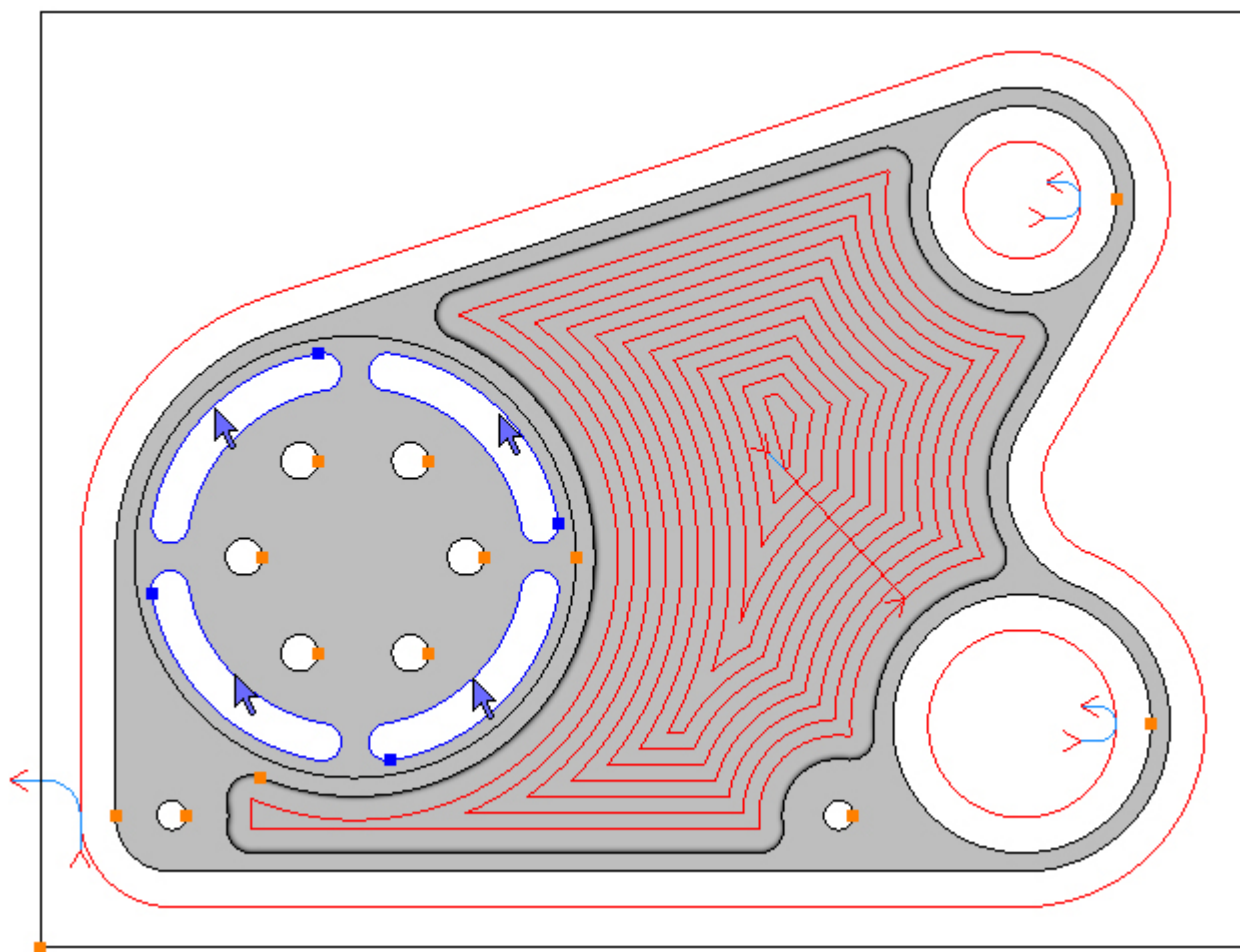
Enter the following values for the Z Positions:


Z Positions	
Abs <input checked="" type="checkbox"/>	Initial Rapid <input type="text" value="2"/>
	Feed From <input type="text" value="0.1"/>
	Material Surface <input type="text" value="0"/>
	Finish Depth <input type="text" value="-0.04"/>
Abs <input checked="" type="checkbox"/>	Retract <input type="text" value="0.1"/>
	Number of Passes <input type="text" value="11"/>
	Depth of Cut <input type="text" value="0.0036"/>

Enter the following values for the Cutting Data:

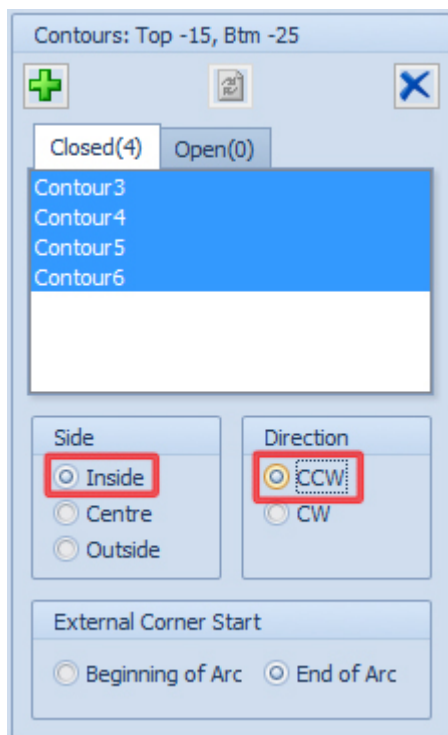
Cutting Data	
Spindle Speed	<input type="text" value="12000"/>
Plunge Feed Rate	<input type="text" value="4"/>
Cut Feed Rate	<input type="text" value="20"/>
Finish Allowance	<input type="text" value="0"/>
Coolant	<input type="text" value="Flood"/>
Cutter Radius Comp	<input type="text" value="None"/>
Sub Routines	<input type="checkbox"/>

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

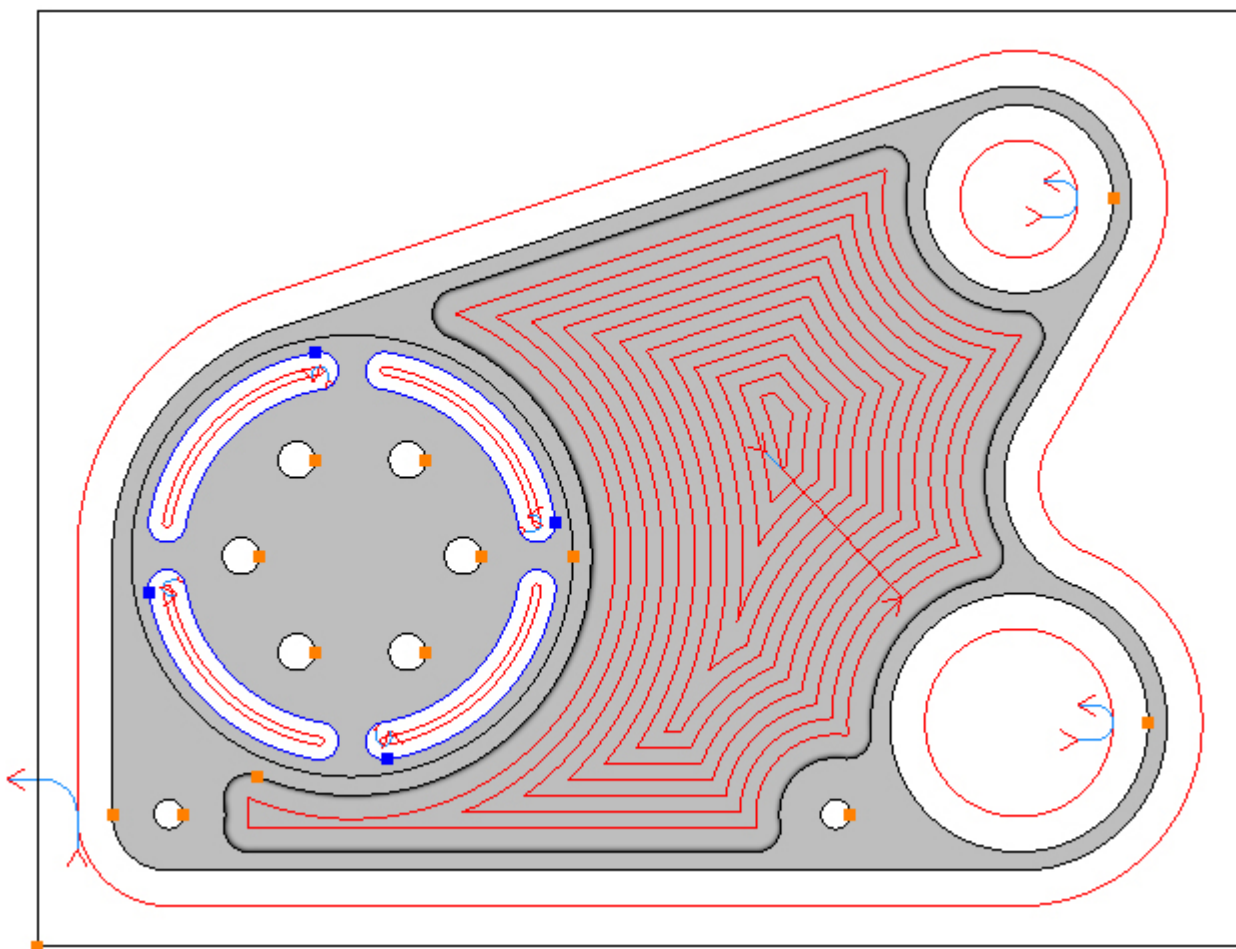


To profile the selected Contours click the Add button . The toolpaths are created on the outside by default, so you will not be able to see them as they are obscured by the model.

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

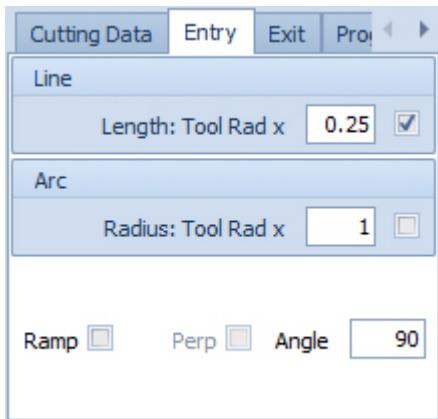


The toolpath is now on the inside and in counter-clockwise direction:



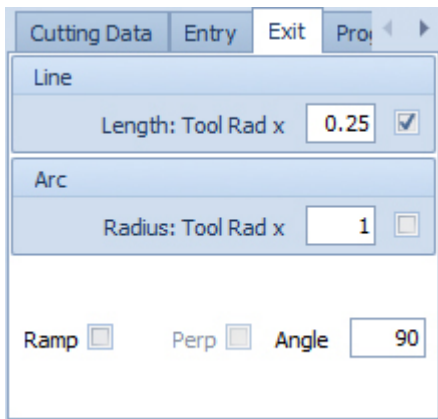
Gear Housing 3D Tutorial

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



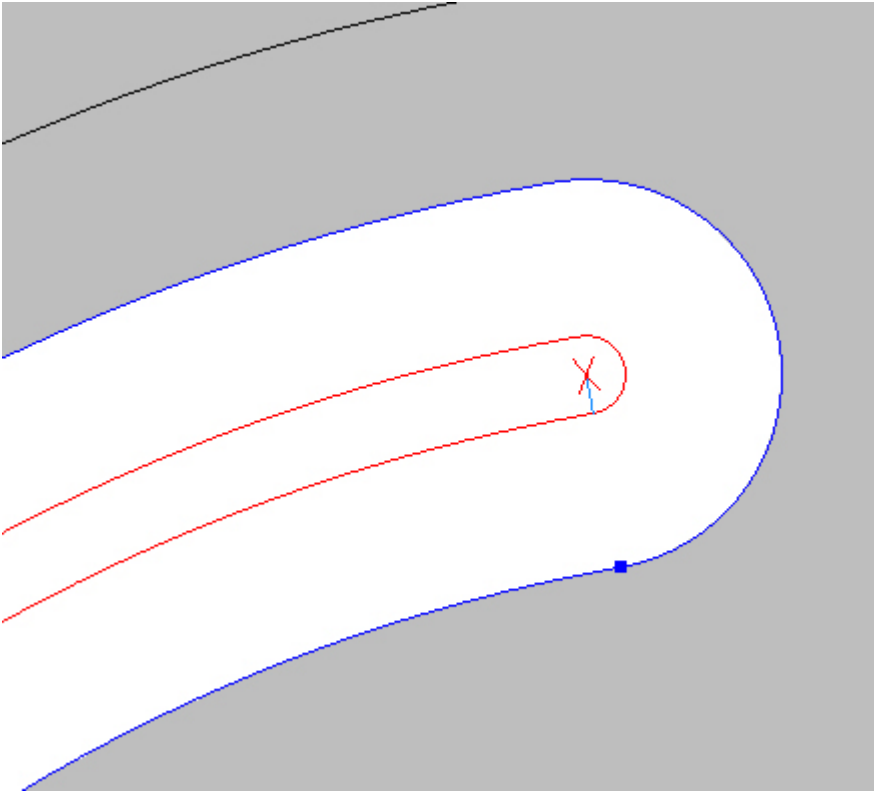
The screenshot shows a software interface with four tabs: 'Cutting Data', 'Entry', 'Exit', and 'Proj'. The 'Entry' tab is selected. It contains two main sections: 'Line' and 'Arc'. The 'Line' section has a label 'Length: Tool Rad x' followed by a text input field containing '0.25' and a checked checkbox. The 'Arc' section has a label 'Radius: Tool Rad x' followed by a text input field containing '1' and an unchecked checkbox. At the bottom, there are three options: 'Ramp' with an unchecked checkbox, 'Perp' with an unchecked checkbox, and 'Angle' with a text input field containing '90'.

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



The screenshot shows the same software interface as before, but the 'Exit' tab is now selected. The settings are identical to the 'Entry' tab: 'Line' section with 'Length: Tool Rad x' set to '0.25' and checked; 'Arc' section with 'Radius: Tool Rad x' set to '1' and unchecked; and 'Ramp' and 'Perp' options unchecked with 'Angle' set to '90'.

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




Step 13 - Spot drill holes

Spot Drill holes using Drilling

Deselect radial slots by pressing the Escape key twice.

Menu: Machine-> Drilling

Toolbar button: 

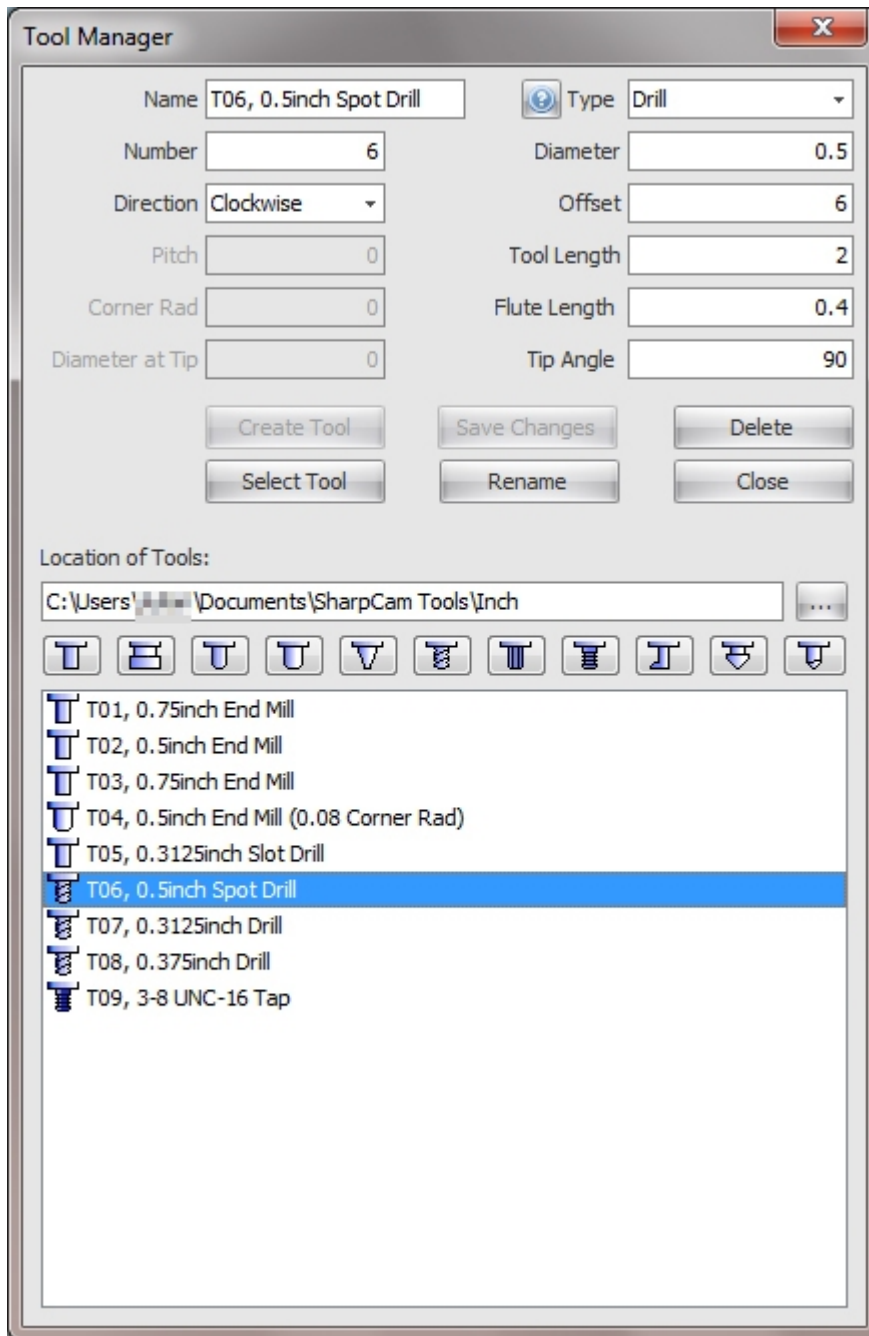
Choose the Drilling command then select the tool for this operation:

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:

Gear Housing 3D Tutorial



Enter the following Z Positions and Cutting Data as below, remember the *Finish Depth* is from the bottom of the Contour which is -1:

Z Positions

Abs
☒
Initial Rapid

Feed From

Material Surface

Finish Depth

Retract To:

☐ Initial Rapid
☒ Feed From

Cutting Data
Program Stop

Spindle Speed

Feed Rate

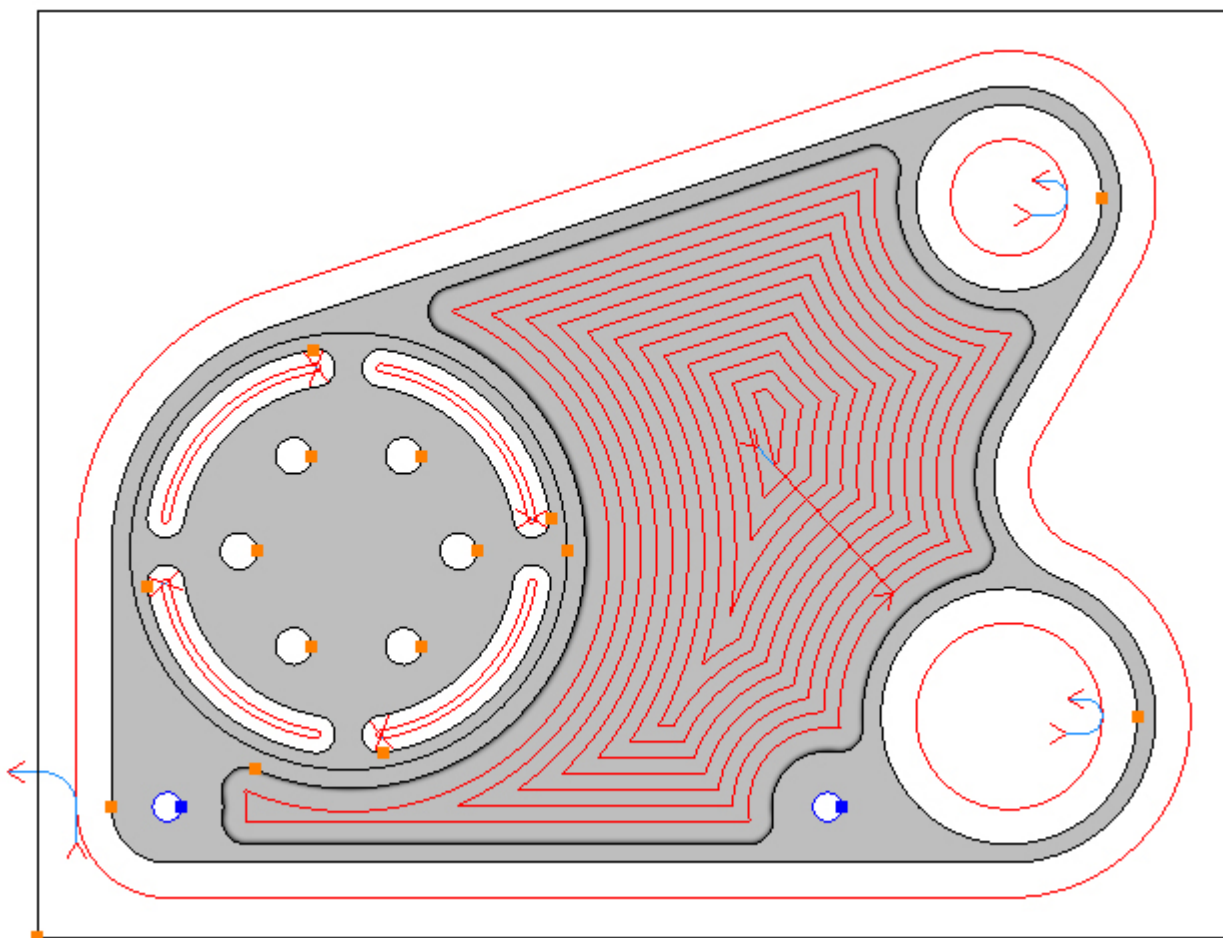
Peck Amount


Dwell

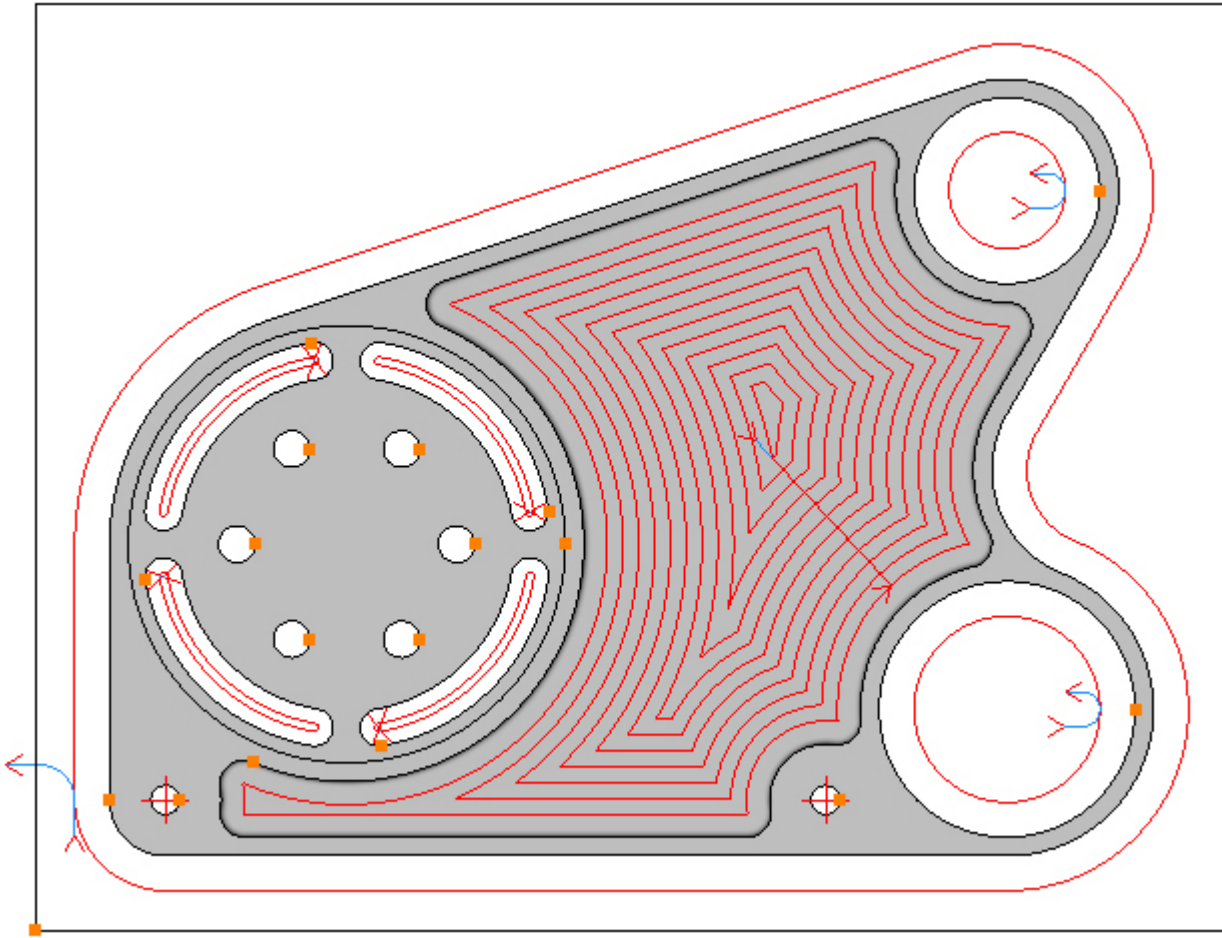
Coolant
Flood

Sub Routines
☐

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 $\varnothing 0.375$ 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:

Z Positions

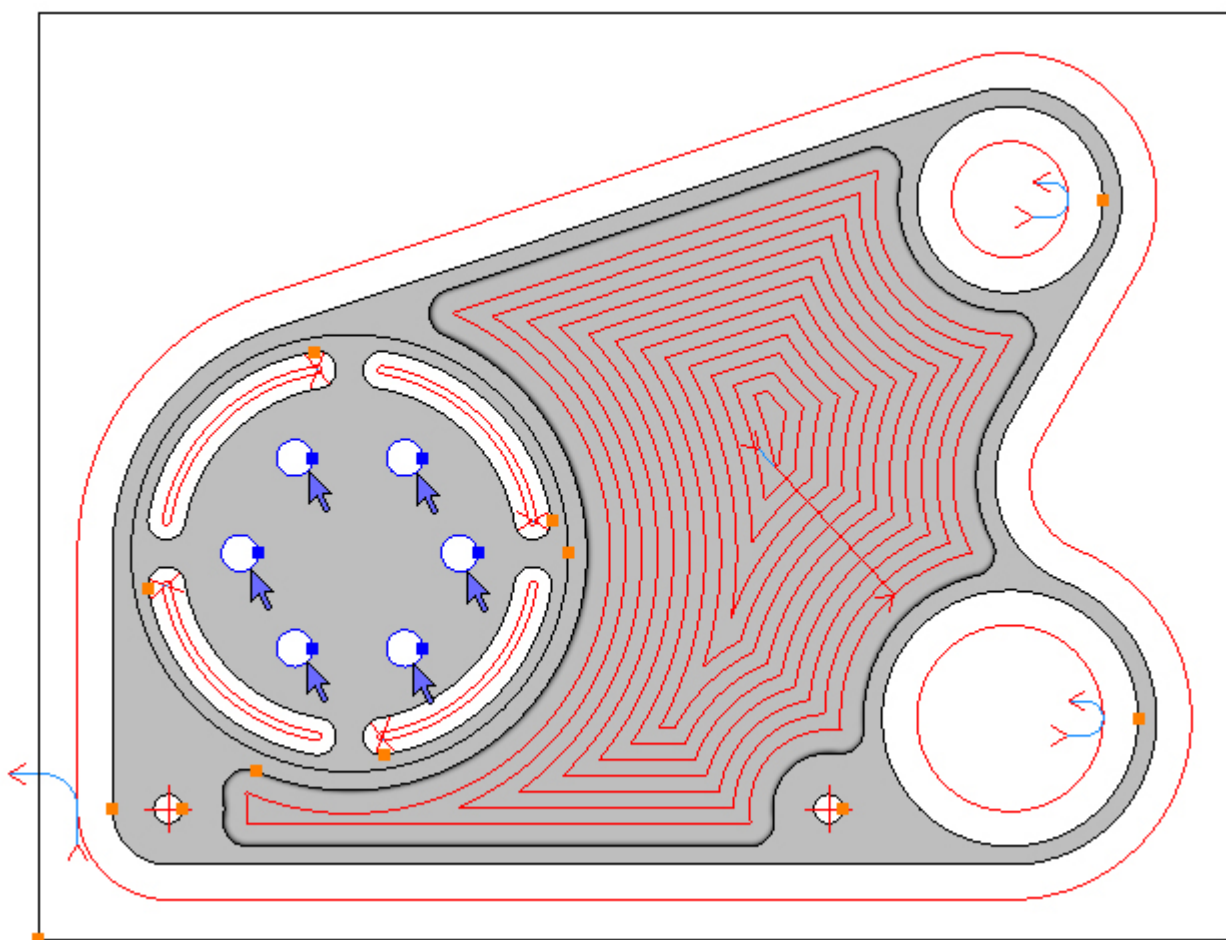
Abs ☒ Initial Rapid
Feed From
Material Surface
Finish Depth
Retract To:
☐ Initial Rapid
☐ Feed From


Cutting Data

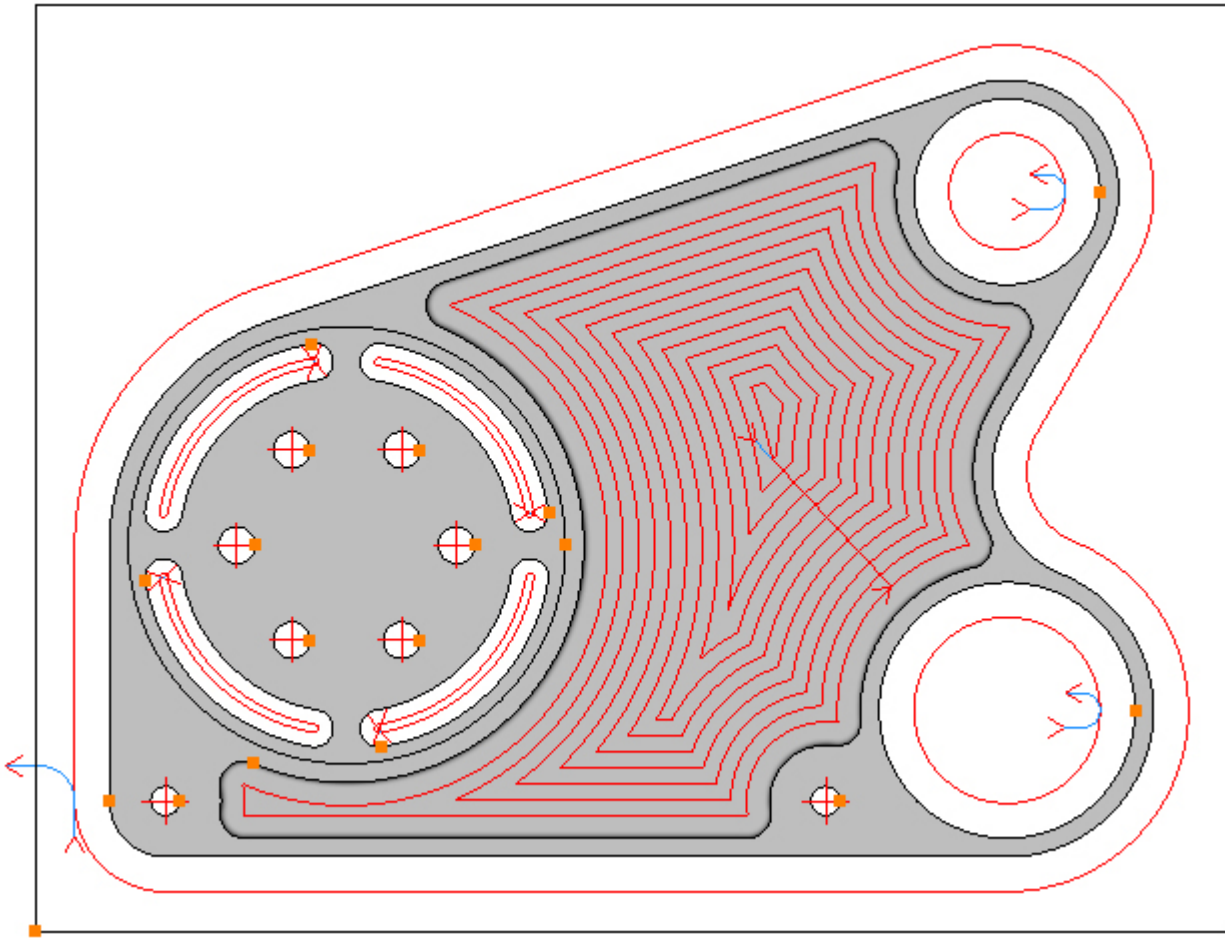
Program Stop

Spindle Speed
Feed Rate
Peck Amount
Dwell
Coolant
Sub Routines ☐

Select the Circles by directly clicking them:




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



Step 14 - Drill holes


Drill Ø0.3125 holes using Drilling

Menu: Machine-> Drilling

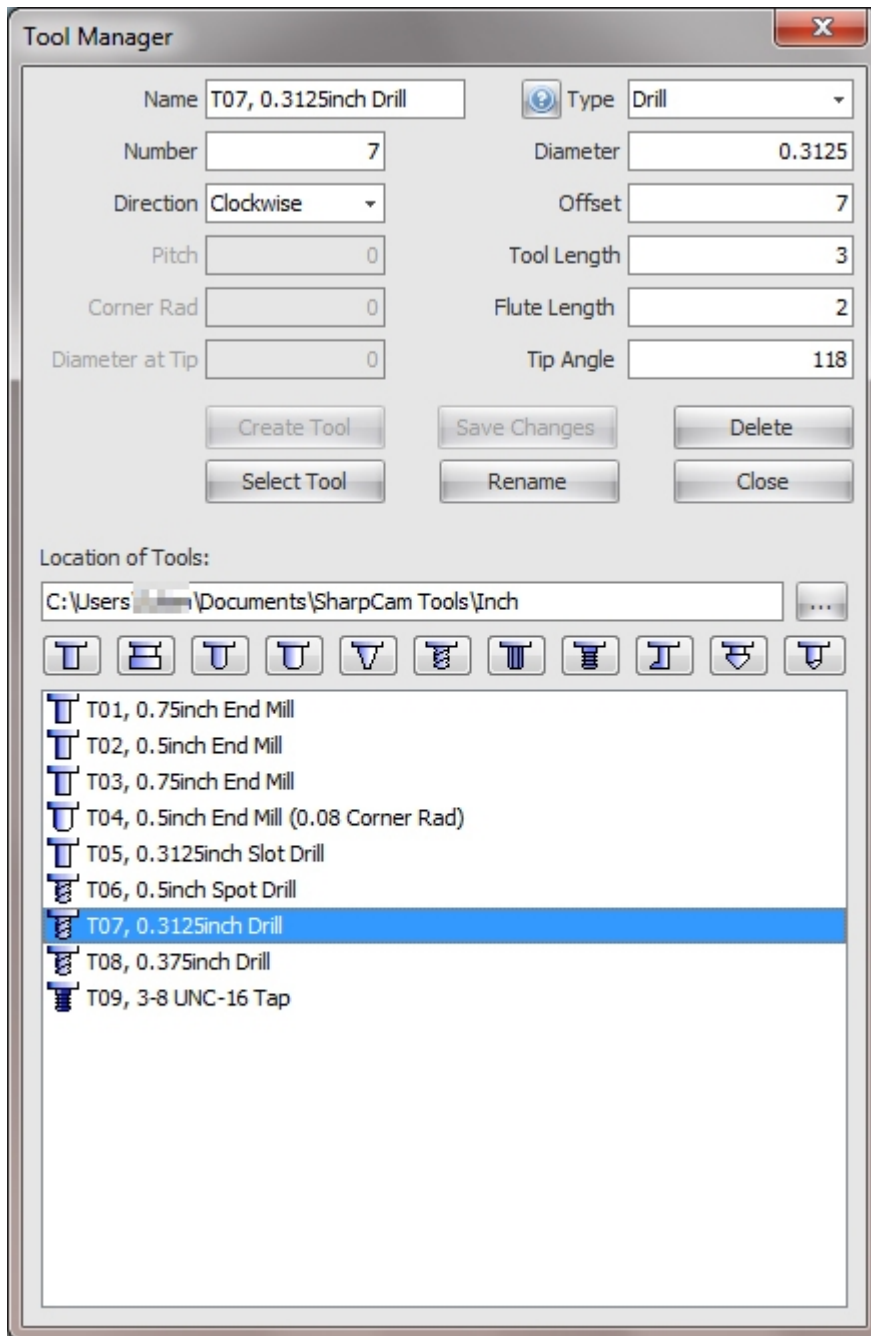
Toolbar button: 

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

Menu: Machine-> Tool Manger

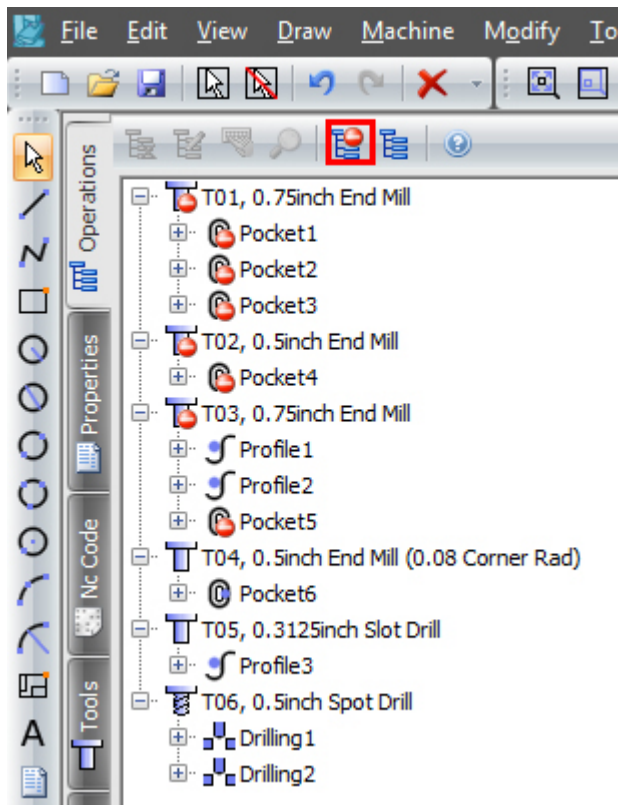
Toolbar button: 

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:

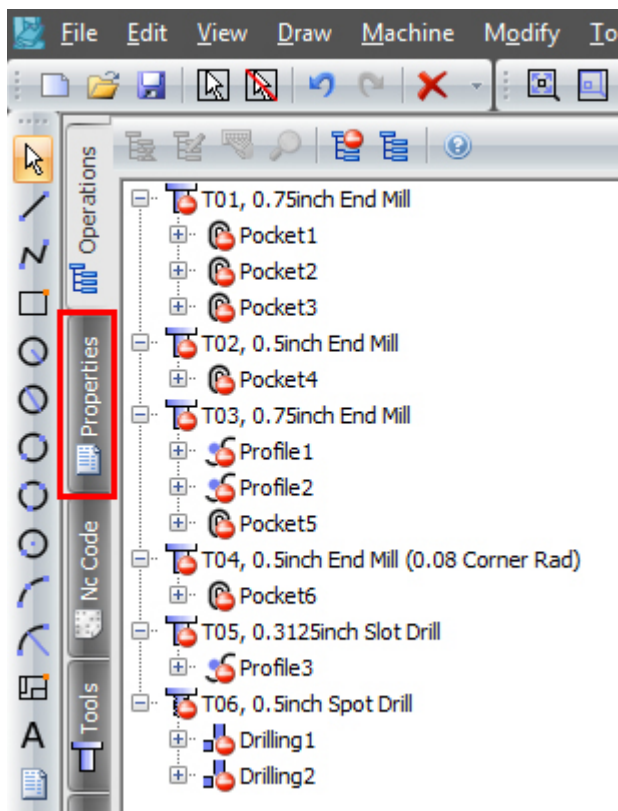


The Part now has a number of toolpaths. In order to prevent the view from being obscured, suppress them so they are hidden. Choose the Operations tab on the Part Manager and click the Suppress All button:

Gear Housing 3D Tutorial



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:

Type
Peck Drill

Z Positions

Abs ☒ Initial Rapid

Feed From

Material Surface

Finish Depth

Retract To:
☐ Initial Rapid
☒ Feed From

Cutting Data
Program Stop

Spindle Speed

Feed Rate

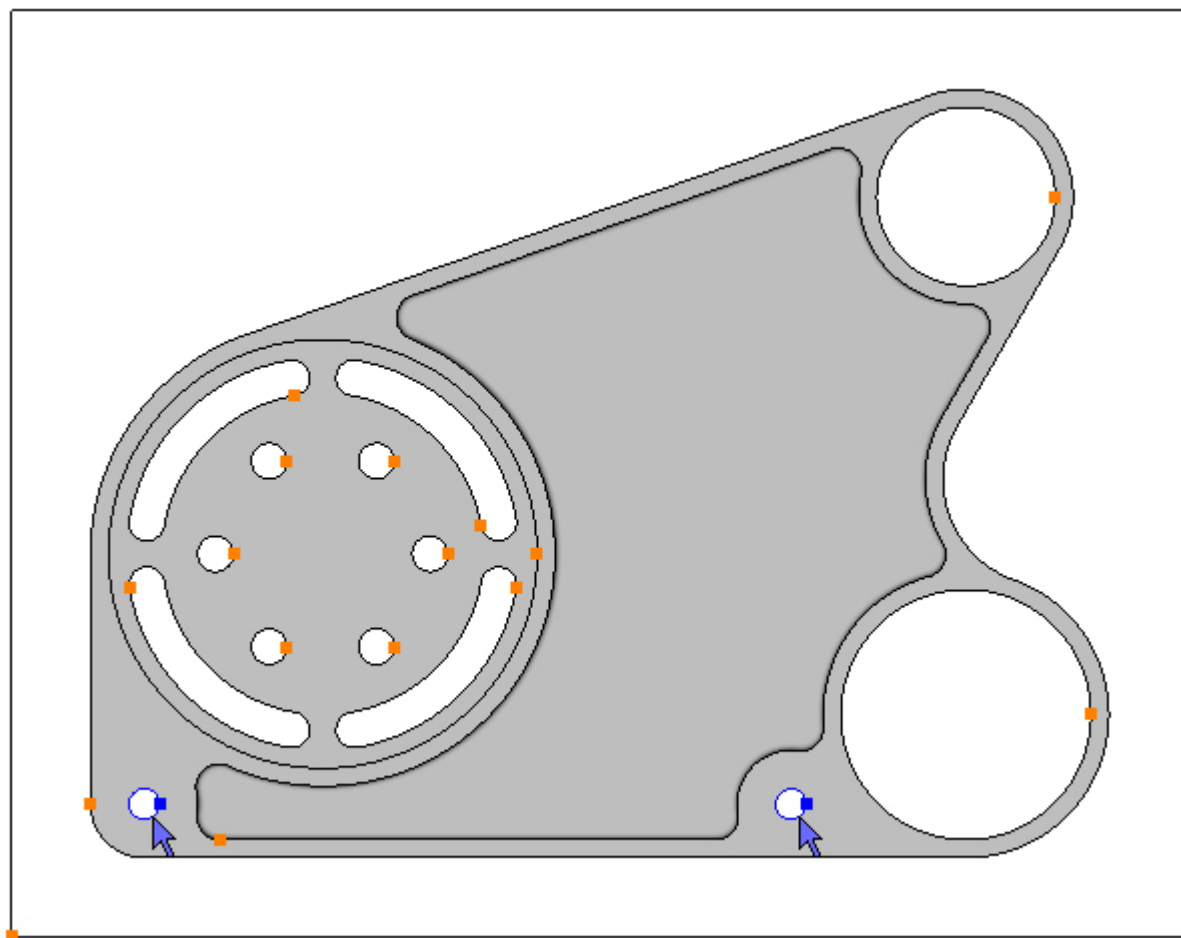
Peck Amount


Dwell

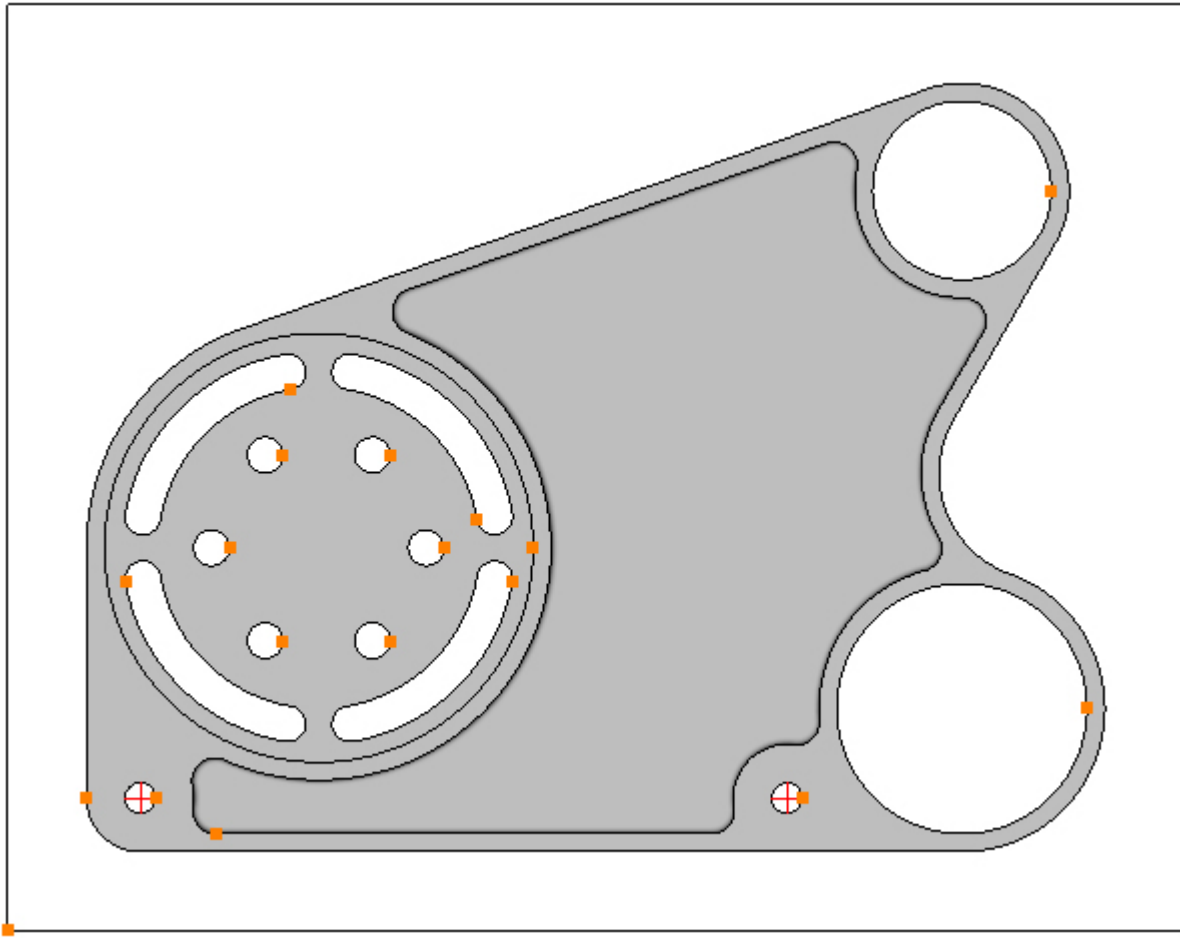
Coolant

Sub Routines ☐

Select the Circles by directly clicking them:




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



Step 15 - Drill holes


Drill Ø0.375 holes using Drilling

Menu: Machine-> Drilling

Toolbar button: 

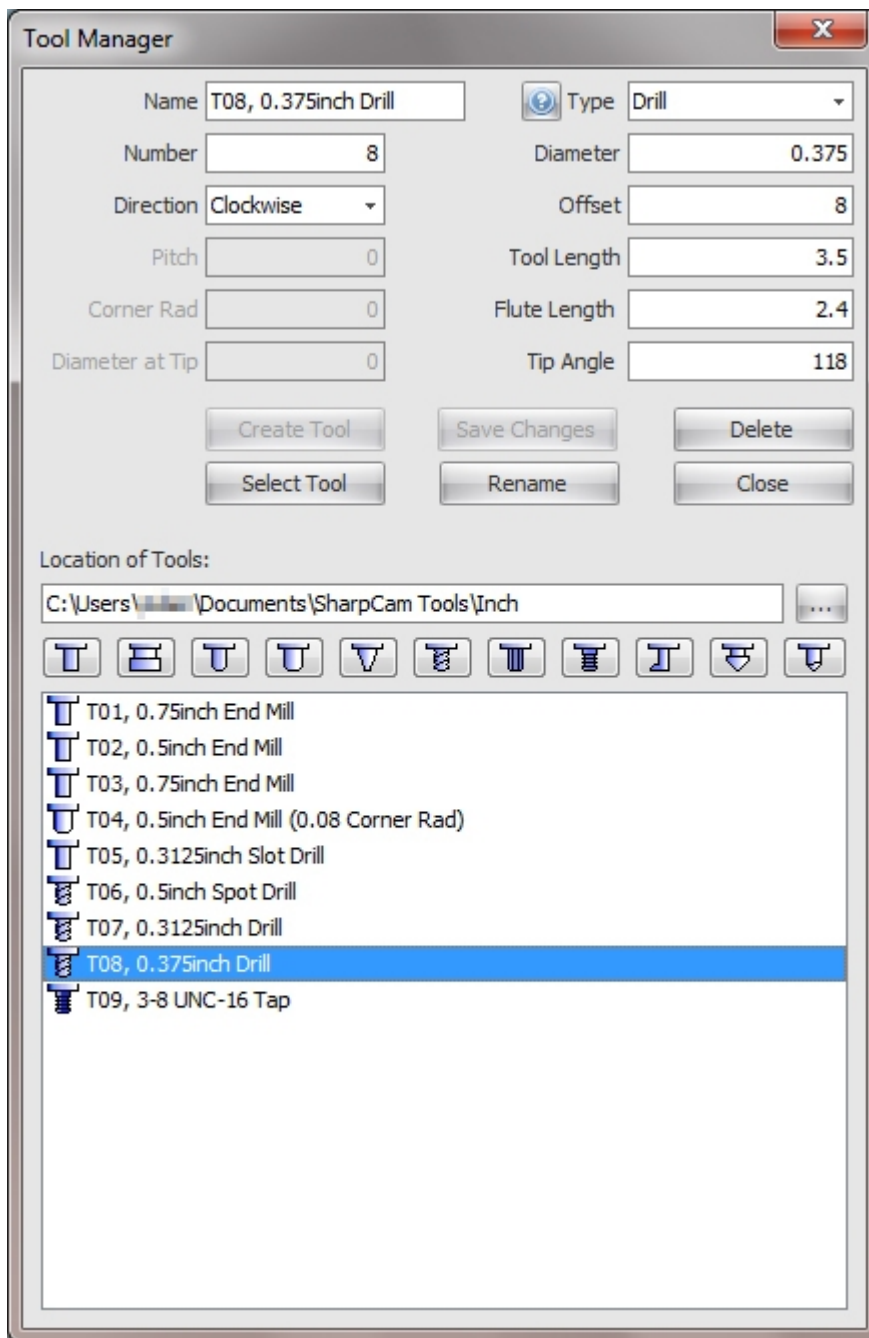
Choose the Drilling command then select the tool for this operation:

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:

Gear Housing 3D Tutorial



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

Type Drilling

Z Positions

Abs ☒ Initial Rapid

Feed From

Material Surface

Finish Depth

Retract To: ☒ Initial Rapid ☐ Feed From

Cutting Data
Program Stop

Spindle Speed

Feed Rate

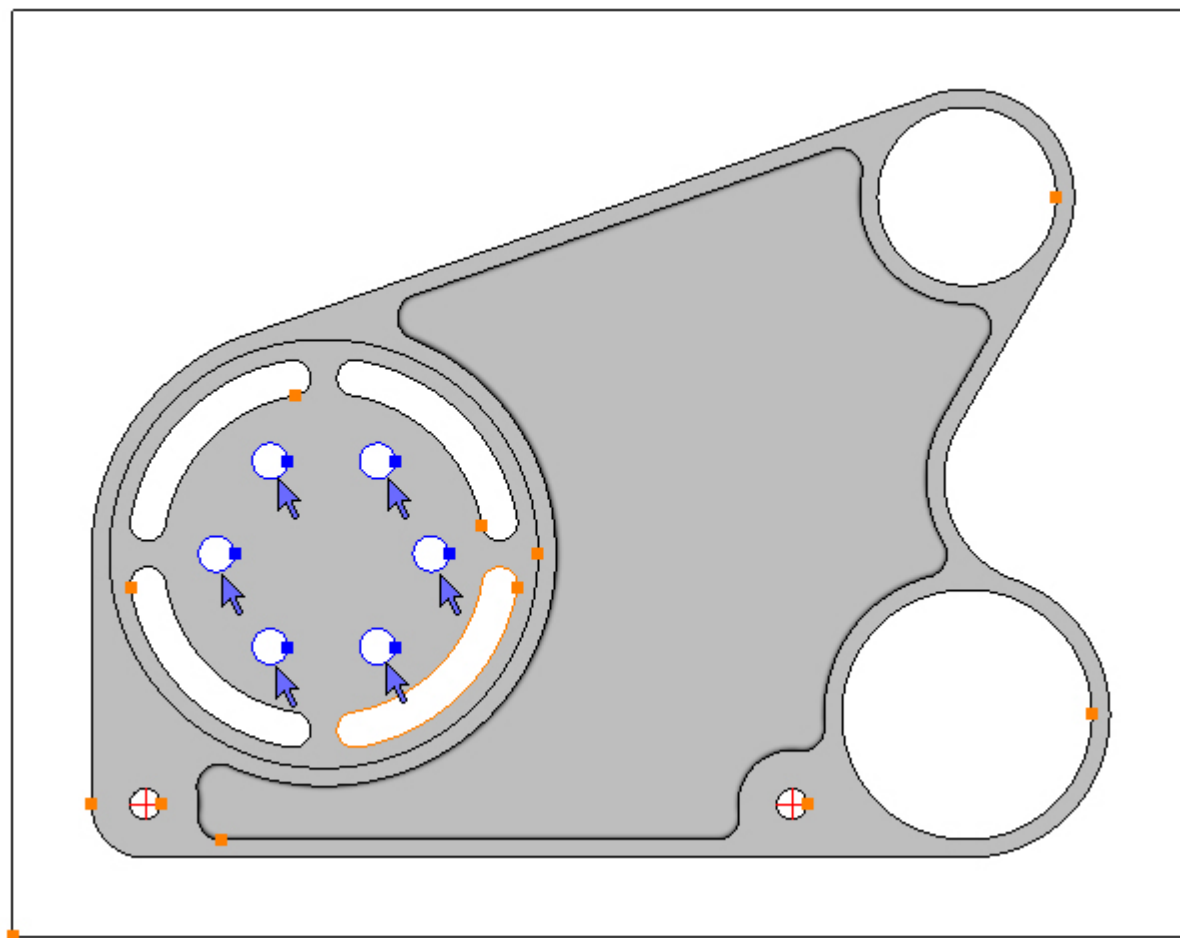
Peck Amount


Dwell

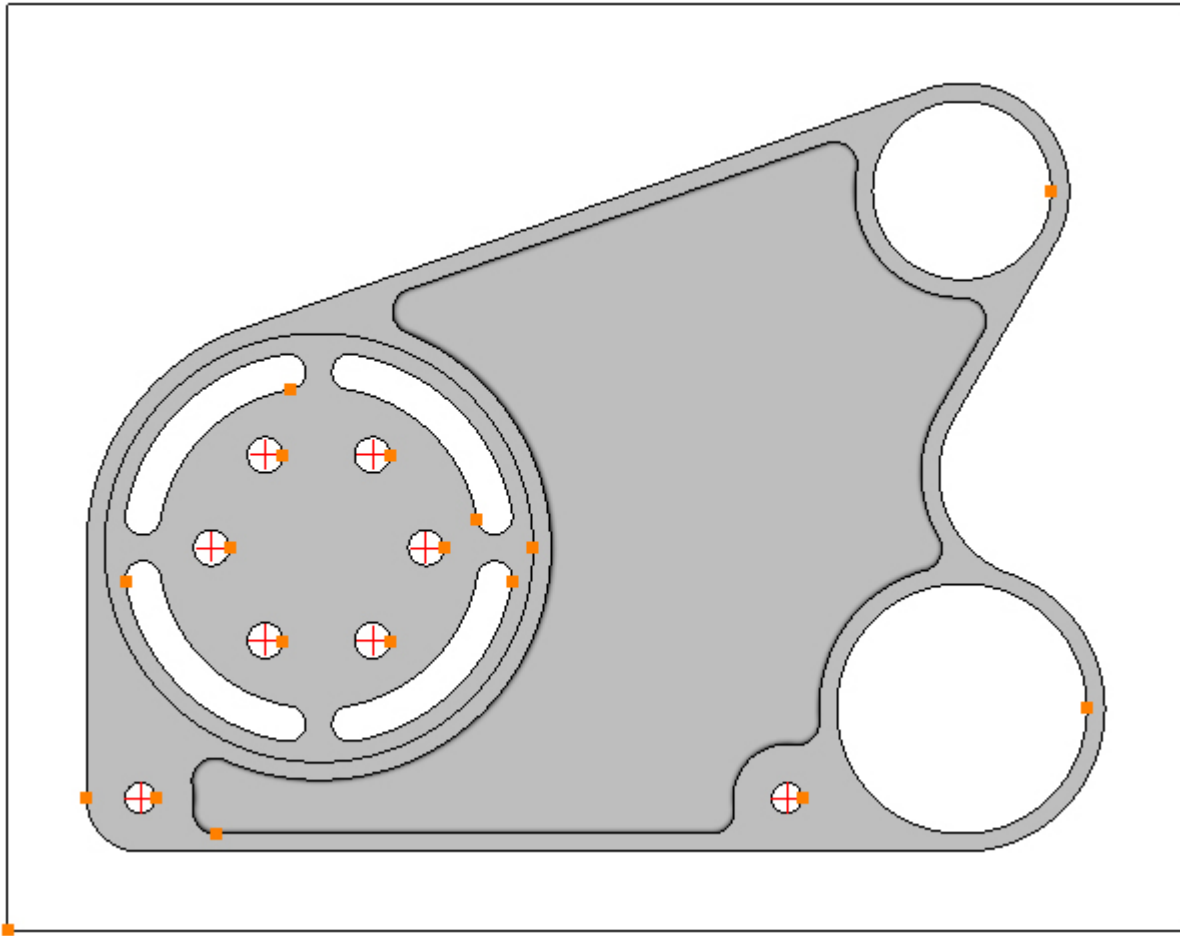
Coolant Flood

Sub Routines ☐

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:

Gear Housing 3D Tutorial

Rectangle

Start Corner Point

X

Y

OK

End Corner Point


X

Y

OK

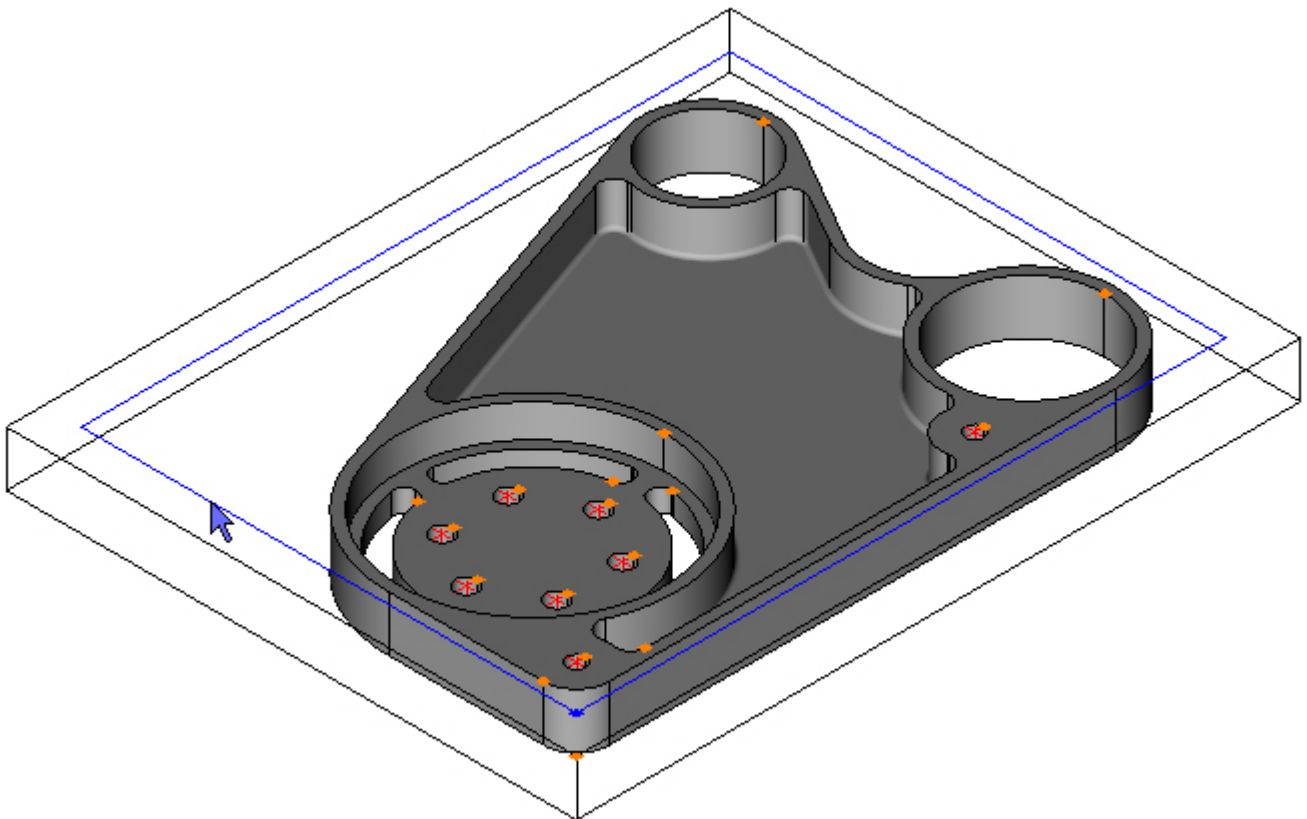
Set the model view to isometric:

Menu: View -> Standard View -> Isometric View

Toolbar button: 


Press the Escape Key to exit Rectangle mode.

Select the rectangle by directly clicking it:

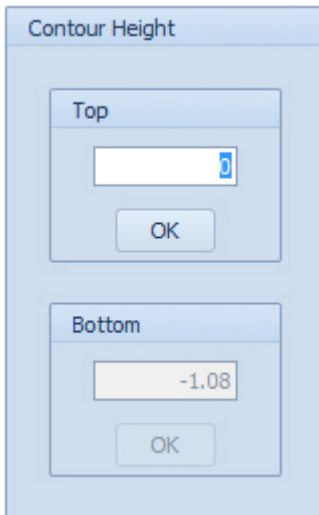


Set the Contour Heights:

Menu: Modify -> Set Contour Height


Toolbar button: 

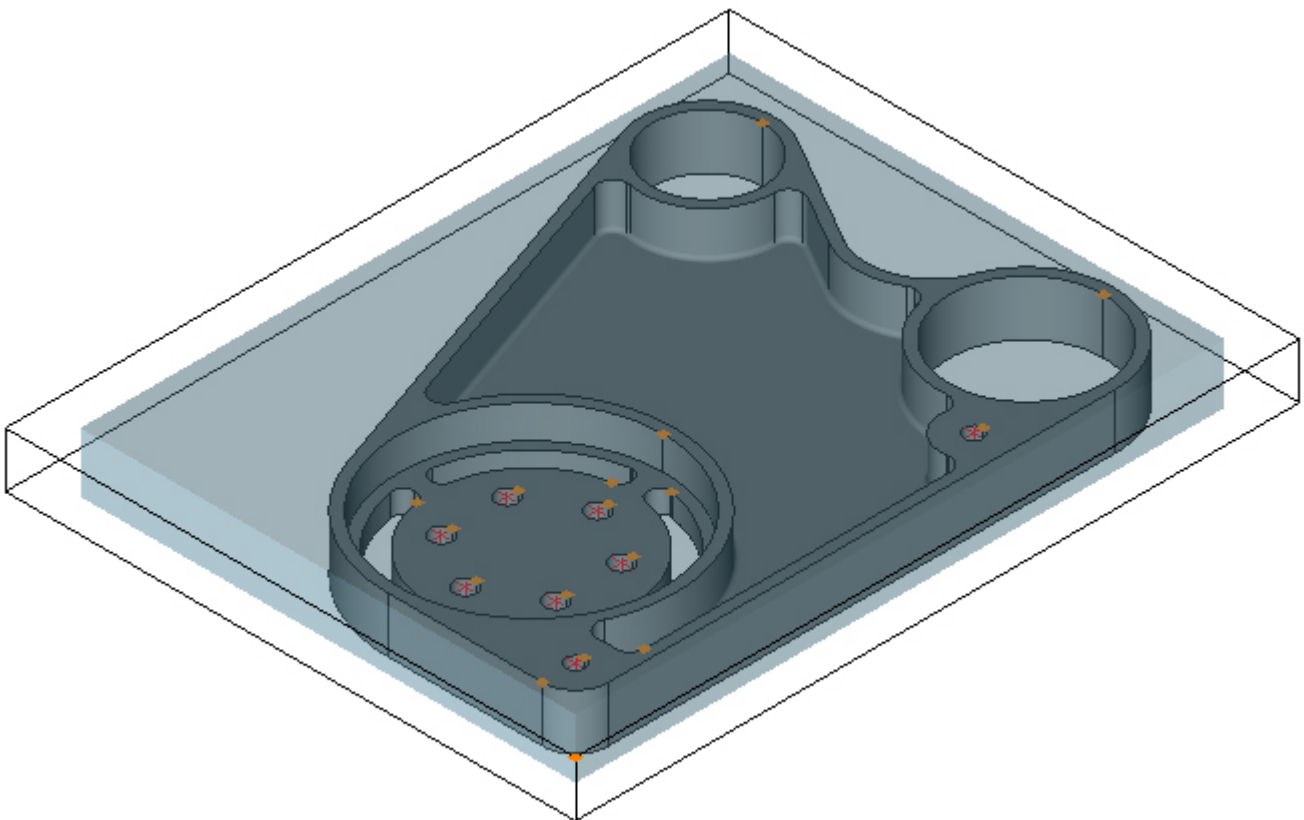
Click OK to accept the default value of zero for the Z Top, then enter -1.08 in Z Bottom and click OK:



Create the Material Stock by perform the Material From Contours command:

Menu: View -> Material Stock -> From Contours

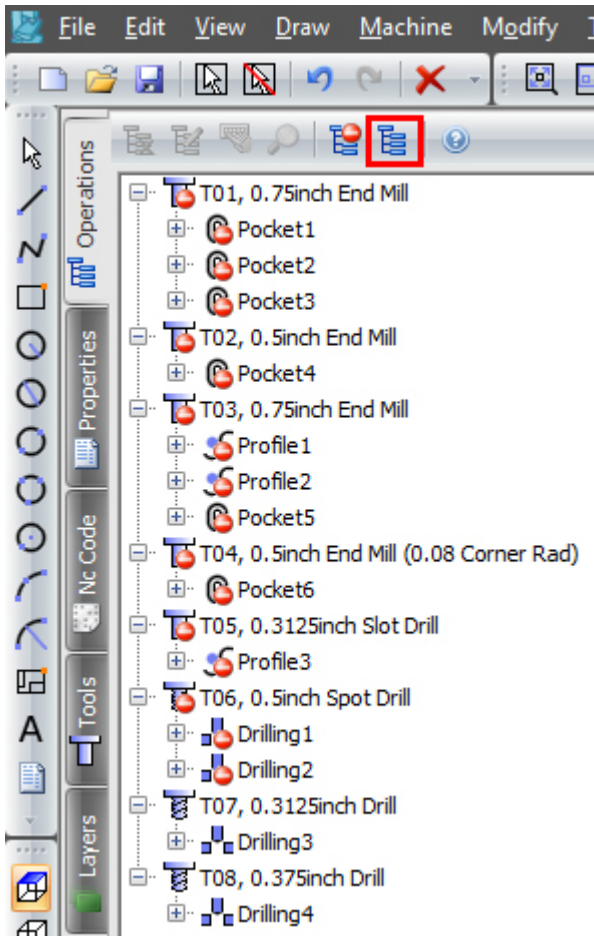
Toolbar button: 



Step 17 - Solid Simulation


Gear Housing 3D Tutorial



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

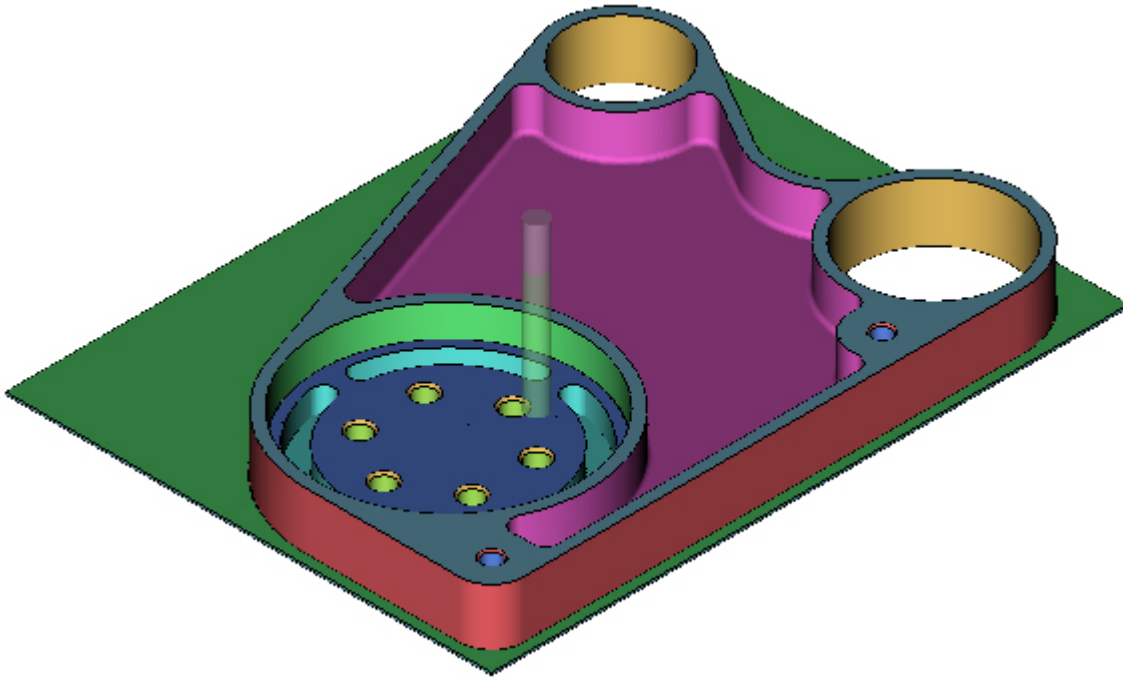



Simulate part being machined

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:



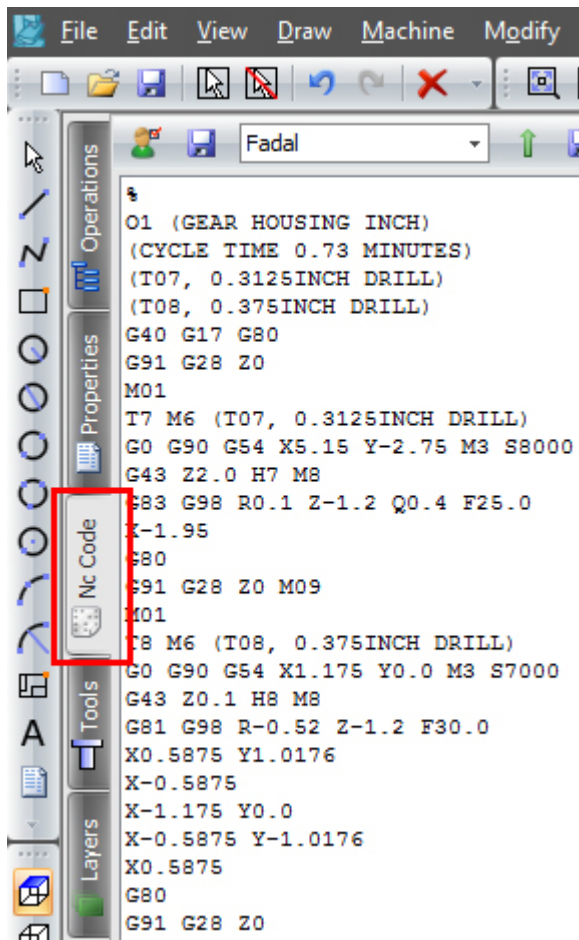
 Simulating the Part is optional and does not need to be performed to create the Nc code.

Step 18 - Output Nc code



Outputting Nc Code

Choose the Nc Code tab on the Part Manager :

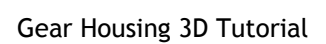
Gear Housing 3D Tutorial



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.

93



1 Index

Create the Contours

Step 1 - Extract Contours from Model, 5-9

Step 2 - Set Contour Heights, 9-23

Creating the Toolpaths

Step 1 - Process plan, 24

Step 10 - Finish counterbore, 60-63

Step 11 - Finish inside, 63-66

Step 12 - Machine radial slots, 66-73

Step 13 - Spot drill holes, 73-78

Step 14 - Drill holes, 78-83

Step 15 - Drill holes, 83-87

Step 16 - Create Material Stock, 87-89

Step 17 - Solid Simulation, 89-91

Step 18 - Output Nc code, 91-92

Step 2 - Set start point, 24-27

Step 3 - Prepare for roughing, 27-30

Step 4 - Rough outside profile, 30-38

Step 5 - Rough counterbore, 38-41

Step 6 - Rough bores, 41-44

Step 7 - Rough inside, 44-47

Step 8 - Finish outside profile, 47-54

Step 9 - Finish bores, 54-60

Gear Housing Drawing, 93

Getting Started

Opening Solid Model File, 2-4

Opening Solid Model File, 2-4

Step 1 - Extract Contours from Model, 5-9

Step 1 - Process plan, 24

Step 10 - Finish counterbore, 60-63

Step 11 - Finish inside, 63-66

Step 12 - Machine radial slots, 66-73

Step 13 - Spot drill holes, 73-78

Step 14 - Drill holes, 78-83

Step 15 - Drill holes, 83-87

Step 16 - Create Material Stock, 87-89

Step 17 - Solid Simulation, 89-91

Step 18 - Output Nc code, 91-92

Step 2 - Set Contour Heights, 9-23

Step 2 - Set start point, 24-27

Step 3 - Prepare for roughing, 27-30

Step 4 - Rough outside profile, 30-38

Step 5 - Rough counterbore, 38-41

Step 6 - Rough bores, 41-44

Step 7 - Rough inside, 44-47

Step 8 - Finish outside profile, 47-54

Step 9 - Finish bores, 54-60