SolidCAM

The Leaders in Integrated CAM

SolidCAM
5 Axis Tutorial

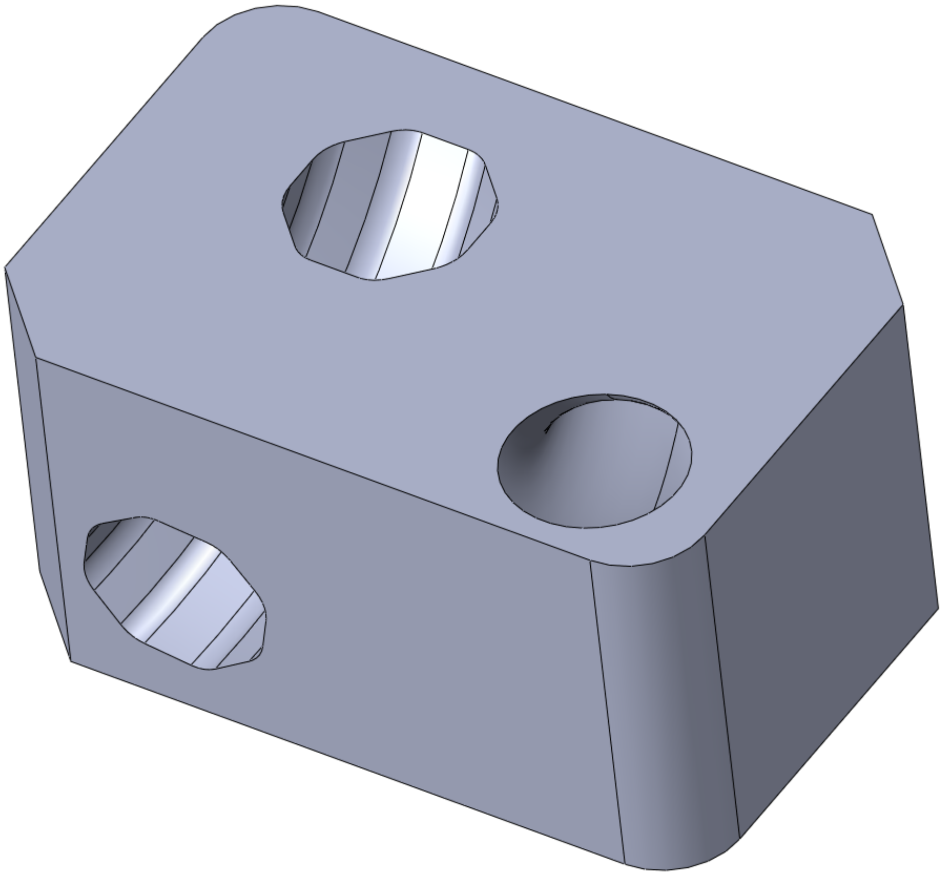
Volume 3 – Beginner

Goals of the Tutorial

- To understand the side tilt definitions
- To understand the strategy of tilted through curve definition
- To understand different sub-tilt definitions
- To understand the gouge check definitions
- To understand the new strategy in perpendicular to curve

Files Used

Part File



Port Inlet

Post Processor

NTU_MAZAK_MPOS

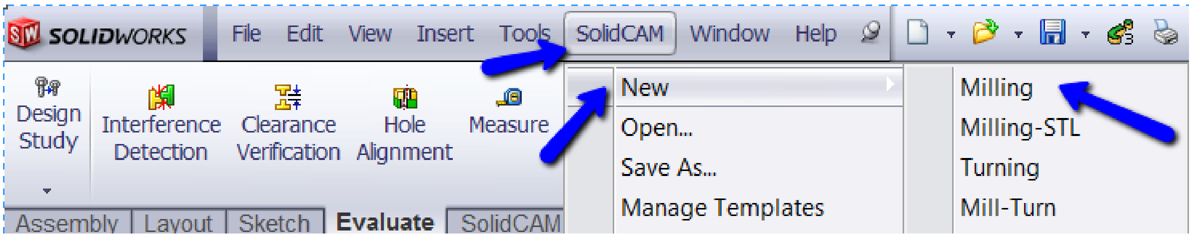
Machine Simulation

5AxMazakVariaxis630

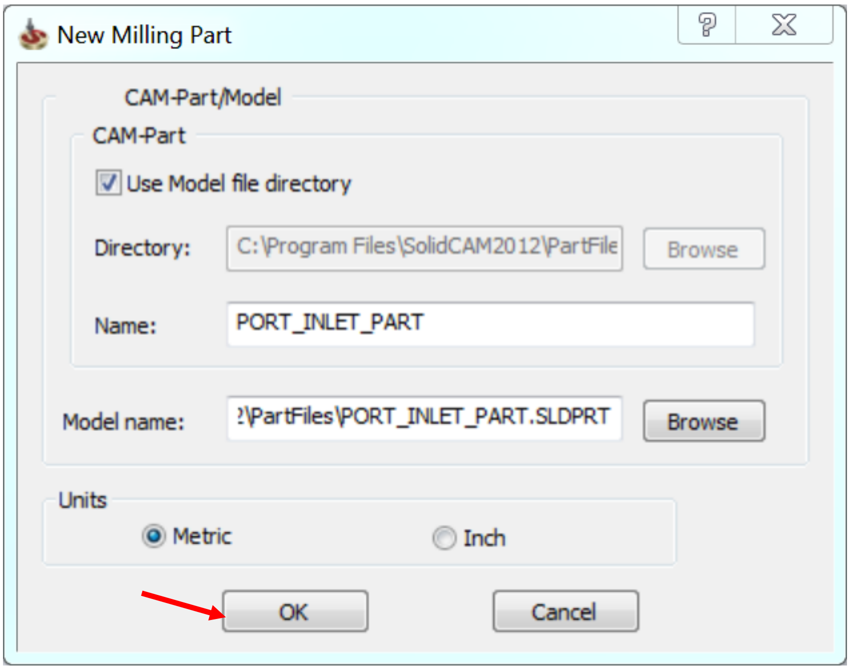
Double click the **SolidWorks** Icon.

Click **File** > **Open** to open the SolidWorks part **PORT_INLET_PART.SLDPRT**.

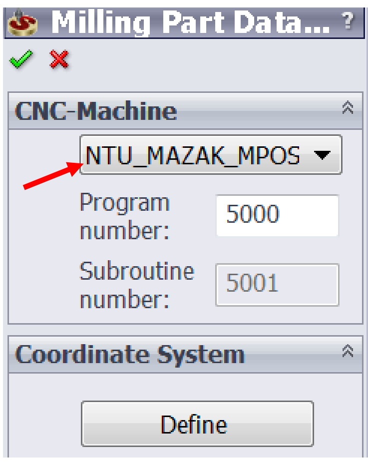
Click **SolidCAM** > **New** > **Milling**.



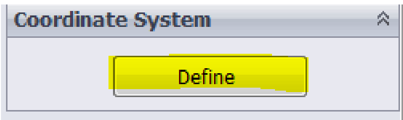
Accept the default part file name and path and Click **OK**.



Select NTU_MAZAK_MPOS as our post processor. Leave the fields of **Program Number** and **Subroutine number** with the default settings.

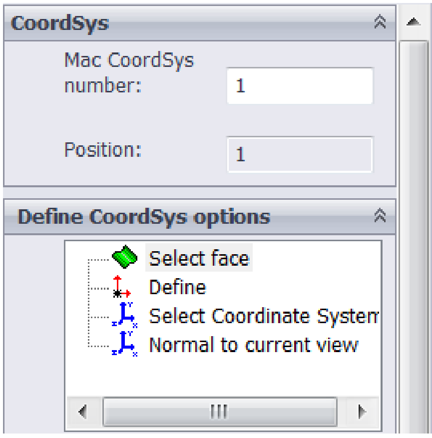


Click **Define** button in **Coordinate System** tab to set the coordinate system for this part.



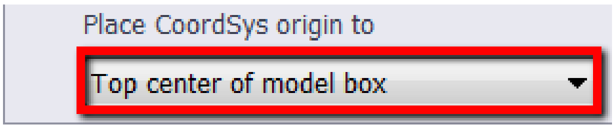
A dialog box with the following options appears:

- Mac CoordSys number
- Position
- Coordinate system definition options

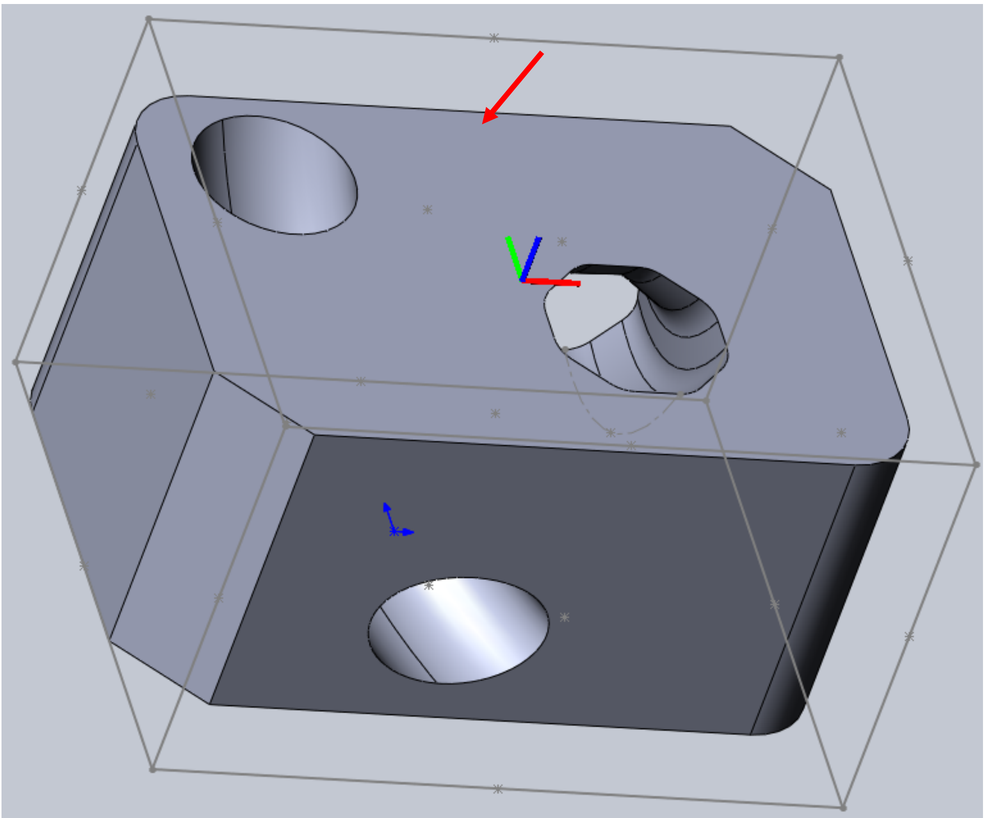


Click **Select Face** option in the **Define CoordSys options** section to define the main coordinate system.

In the **Place CoordSys origin to** section, select **Top center of model box** from the drop down.

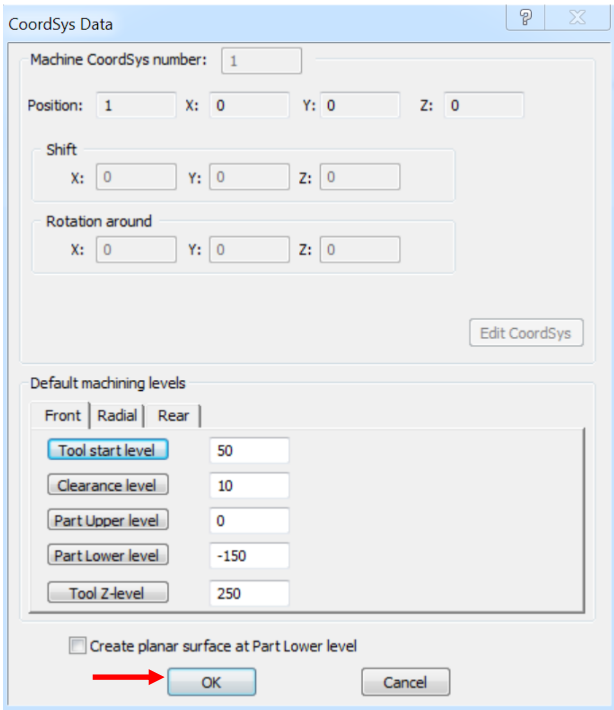


Select the face as shown in the below image.



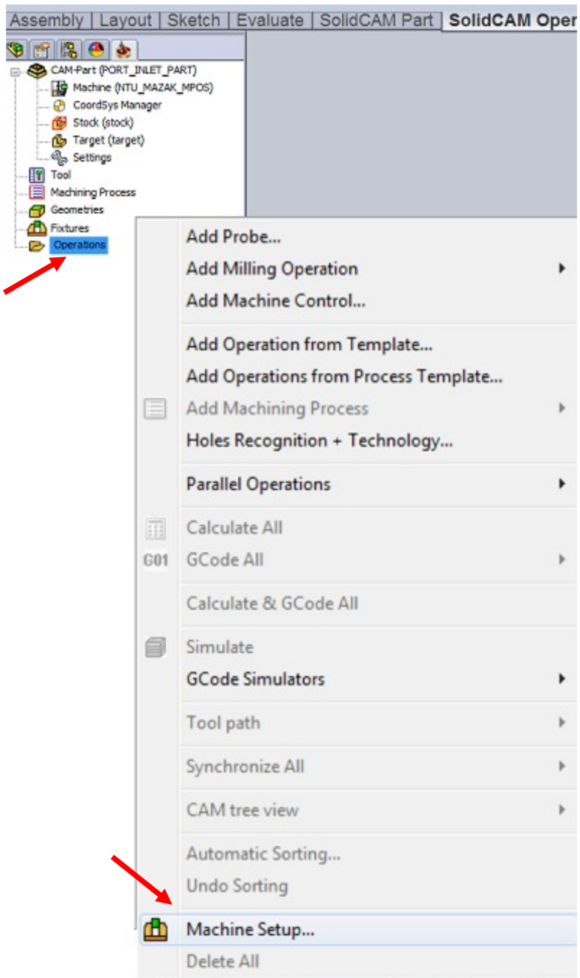
Click ✔ to accept.

Click **OK** in the **CoordSys Data** window.

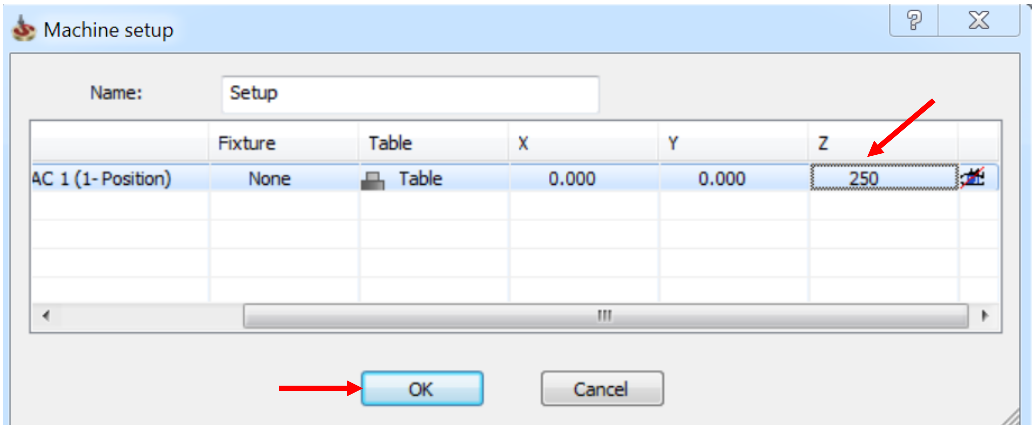


Click ✔

Right click **Operations** > **Machine Setup**.

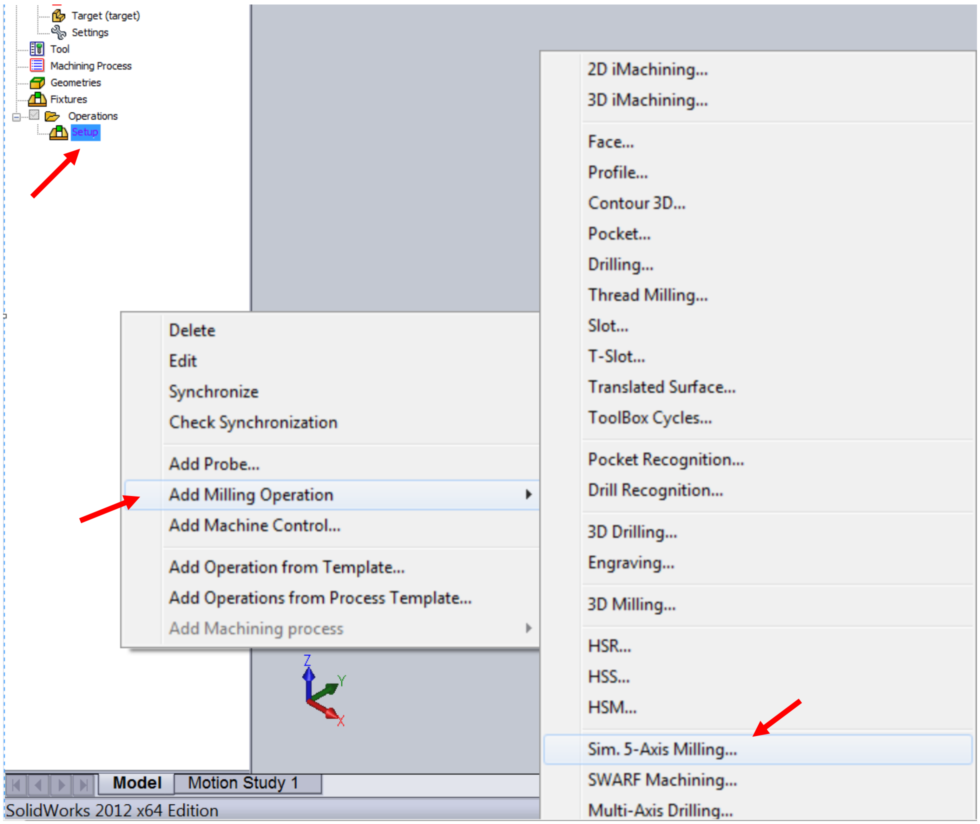


Enter a value of 250 in the Z column.



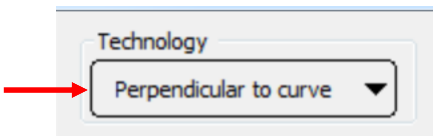
Press the Enter key and then click **OK**.

Right click **Setup** > **Add Milling Operation** > **Sim. 5-Axis Milling**.



The Operations Manager window opens.

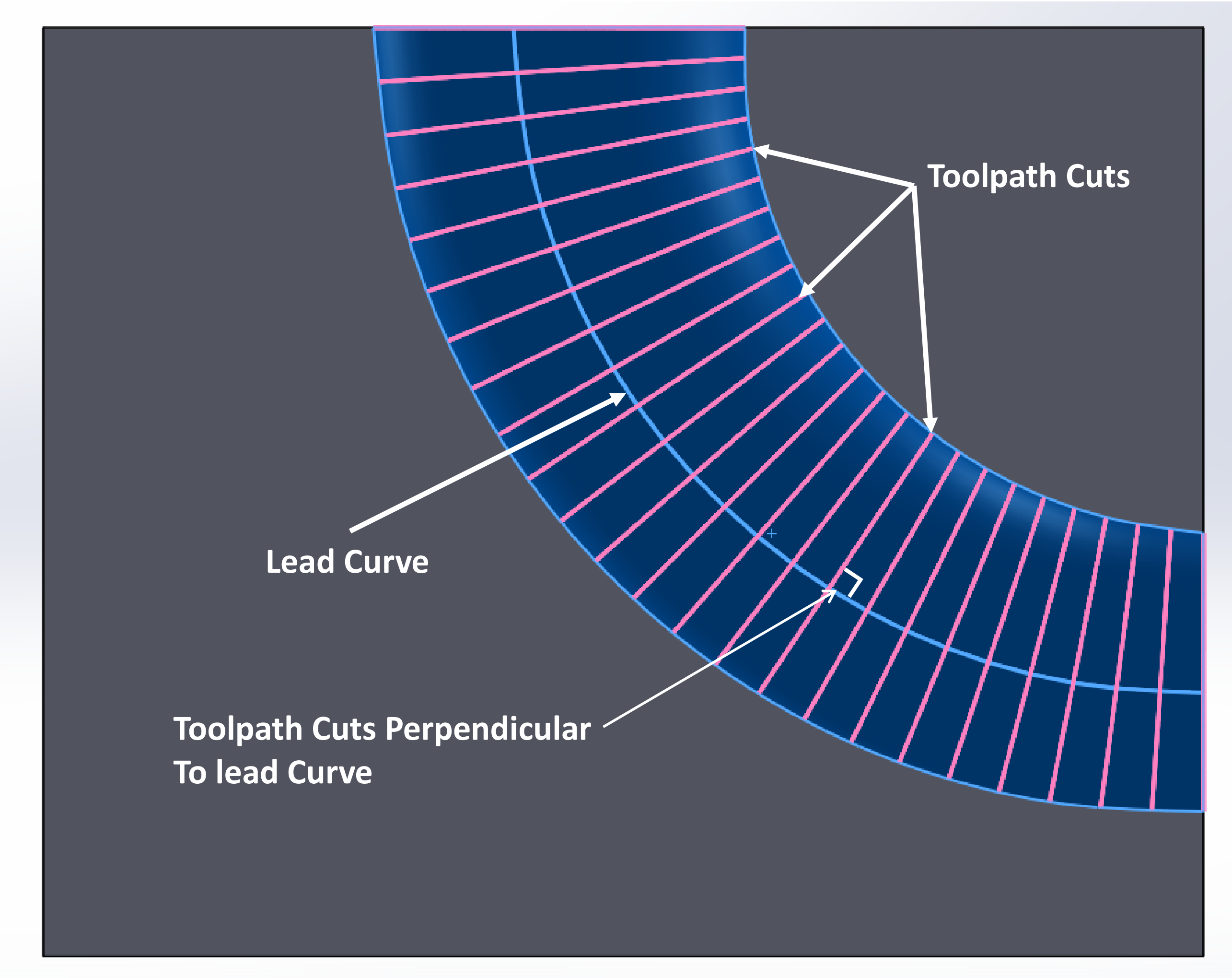
Select **Perpendicular to curve** in the **Technology** section.



Perpendicular to Curve Strategy

The Perpendicular to Curve feature allows you to create a toolpath perpendicular to a Lead curve (drive curve) on a selected Drive Surface. If the curve is a straight line, then the cuts are created parallel to each other. You can use a surface edge as the Lead Curve, or a curve that is on the surface. If the curve is not on the surface then it is projected onto the Drive surface to create the toolpath.

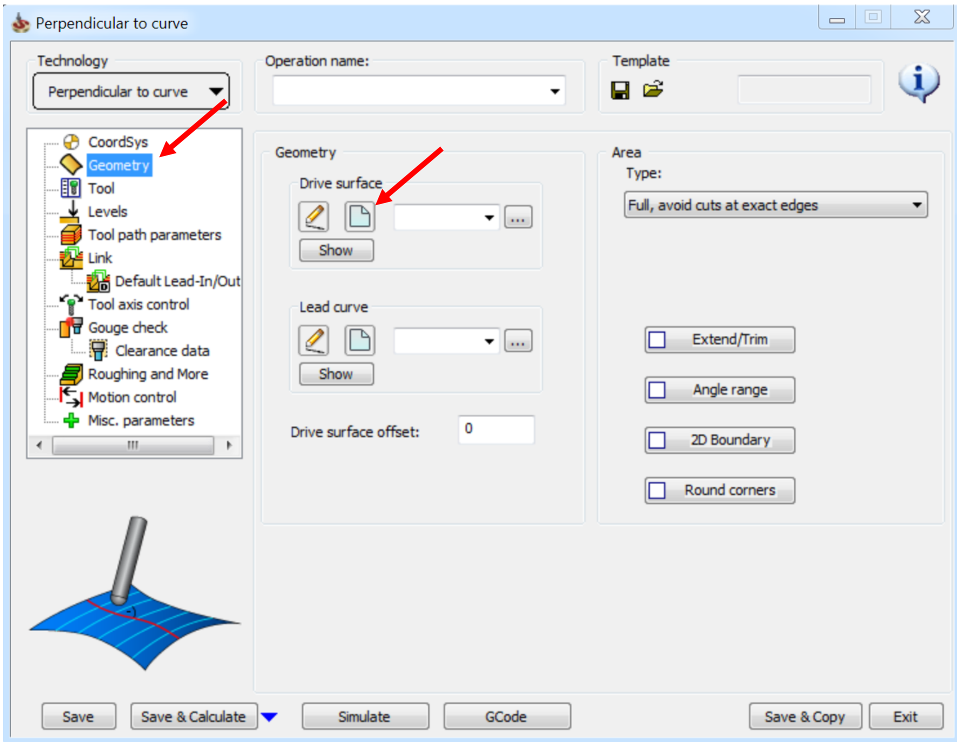
How does this strategy work



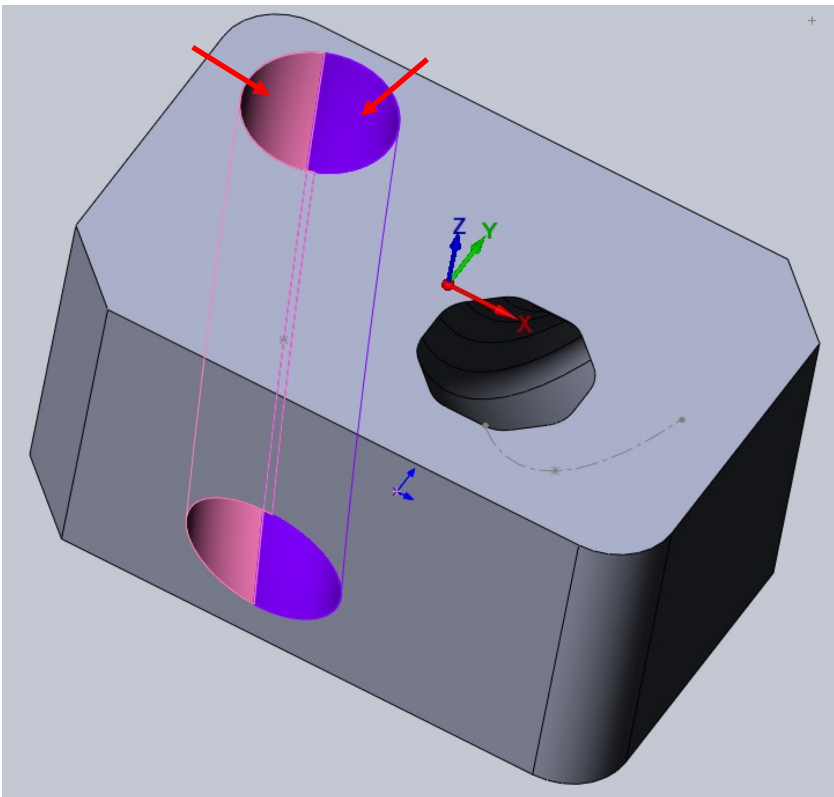
In the above picture you can see that the toolpath cuts are taken perpendicular to the lead curve. This is done by distributing points on the lead curve (Equal to the Step Over) and then taking the slices on the drive surface on each of these points by creating planes normal to the lead cuve at that point. The lead curve need not be the edge of the drive surface , it could be any curve and the cuts will be perpendicular to that curve.

Click **Geometry**.

Click New icon under **Drive surface**.

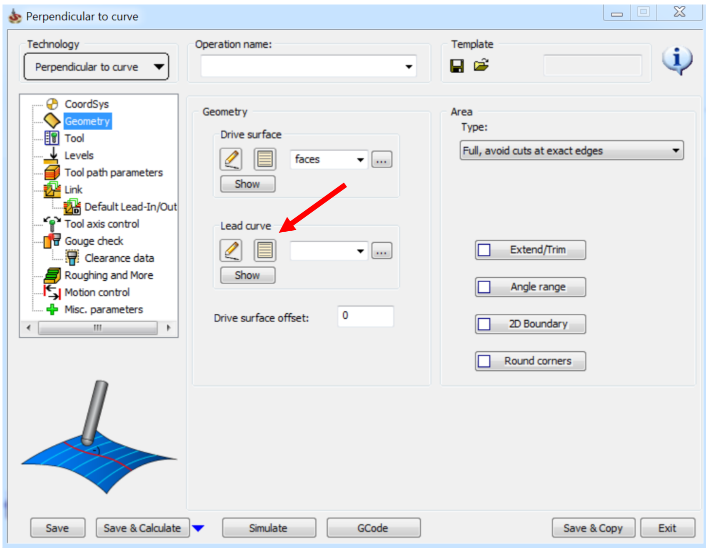


Select the two faces as shown in the image.

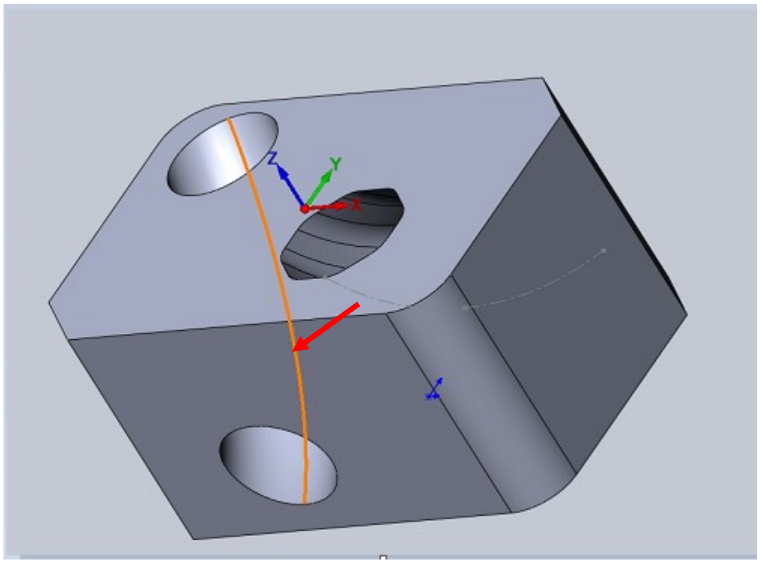


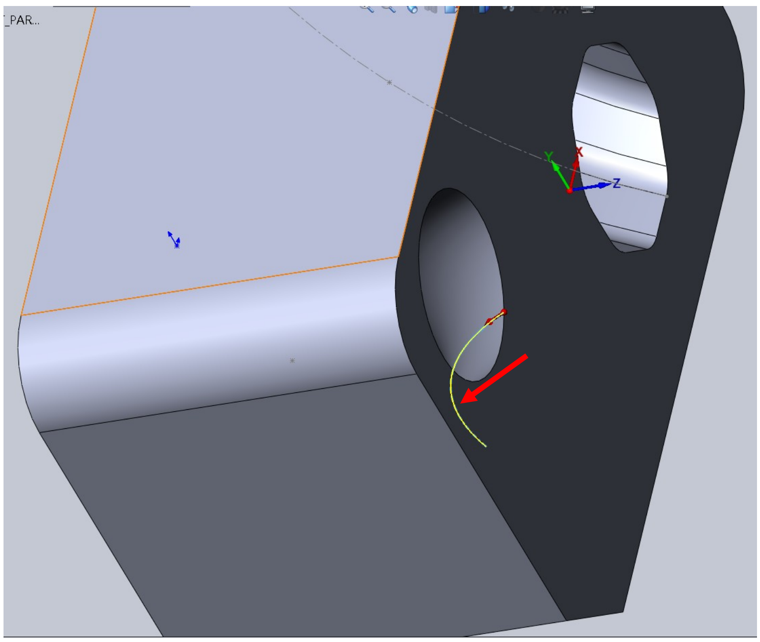
Click ✓ to return to the Operations Manager.

Click New icon under **Lead curve**.

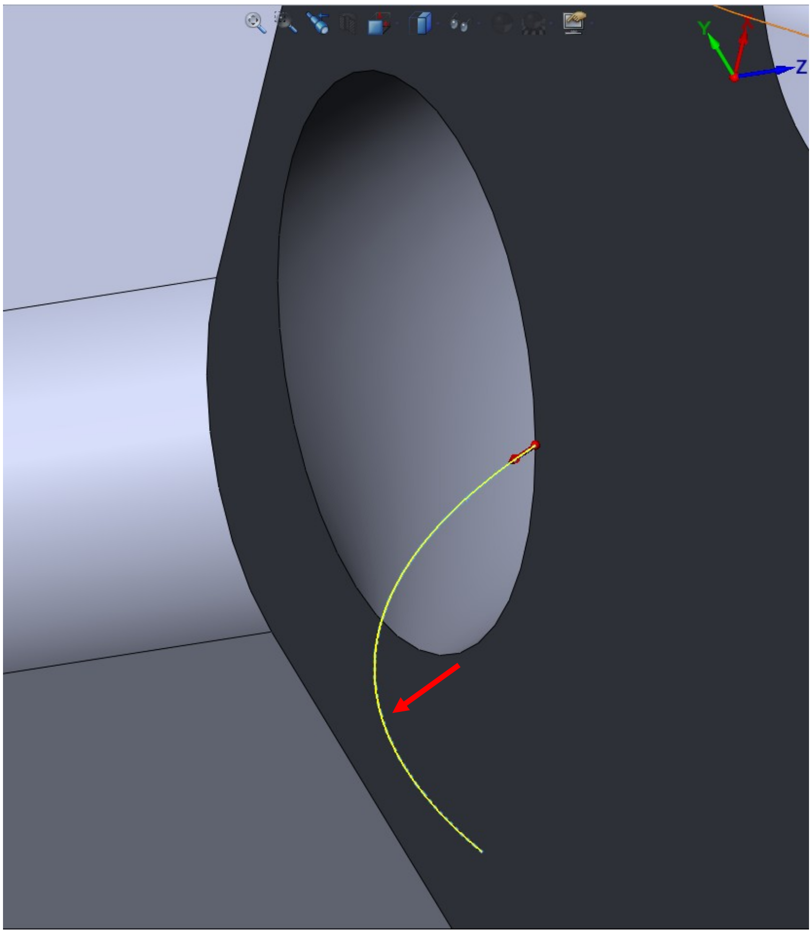


Two curve options are available as shown in the image. One is the main one, other the smaller one is exactly opposite to it.





Select the smaller curve as shown in the image. Selecting the smaller curve does not produce any air cuts.

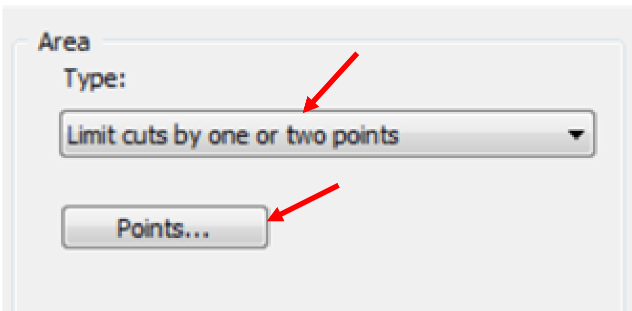


Click in the **Chain List** section.

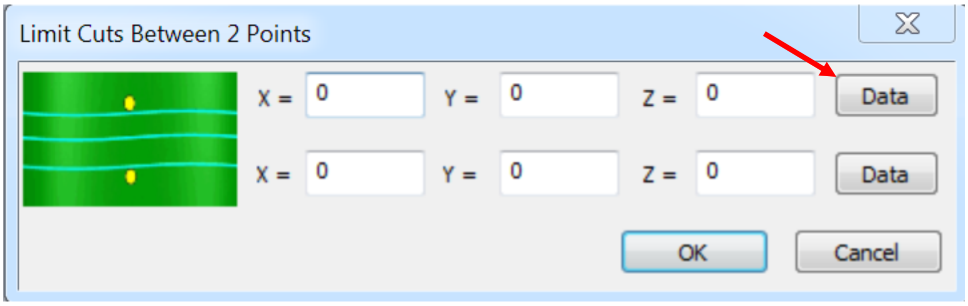
Click to return to the Operations Manager.

Select the **Type** as **Limit cuts by one or two points** from the drop down list in the **Area** section.

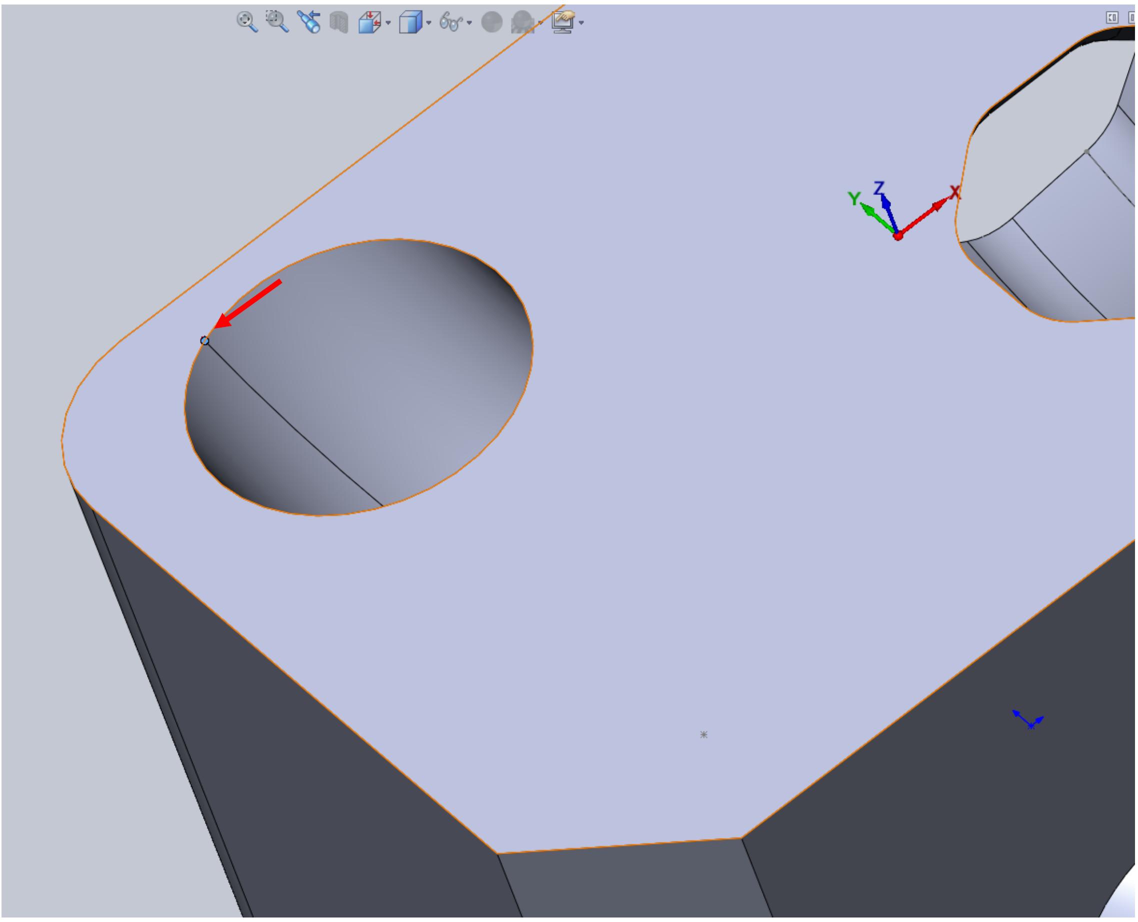
Click **Points**.



Click **Data**.



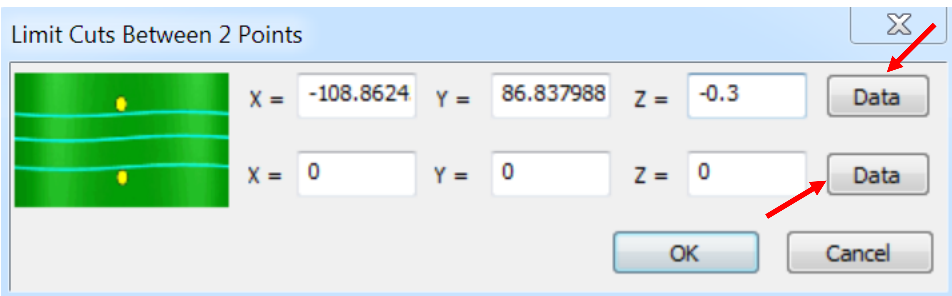
Select the point as shown in the image.



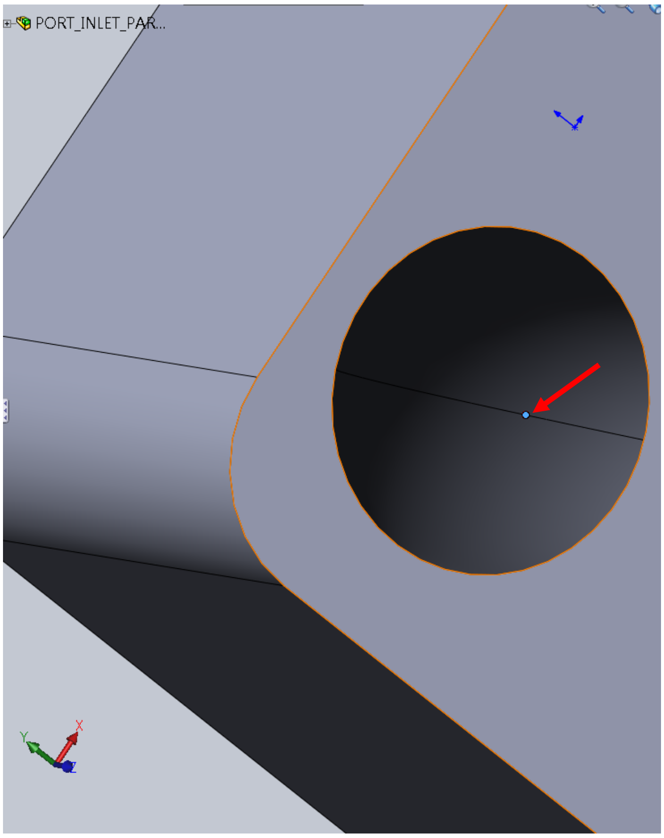
Click 

Define the first margin as -0.3 in the **Z** field. This is needed as there is no way to define margins while selecting the two points option.

Click **Data**.

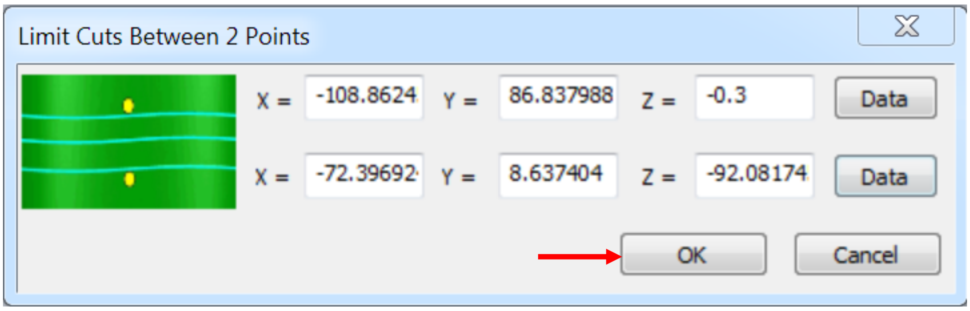


Select the point as shown in the image. The point is already predefined in the part.

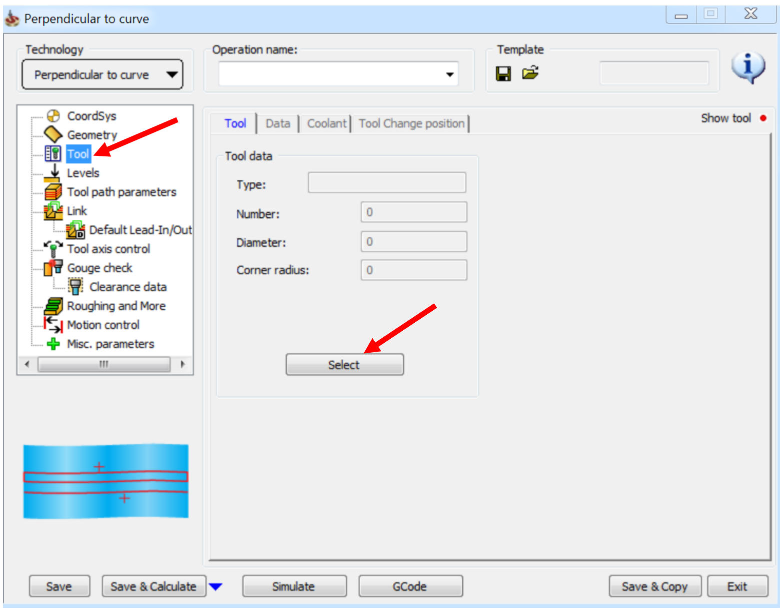


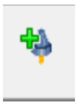
Click ✔

Click **OK** in the Limit Cuts Between 2 Points window.



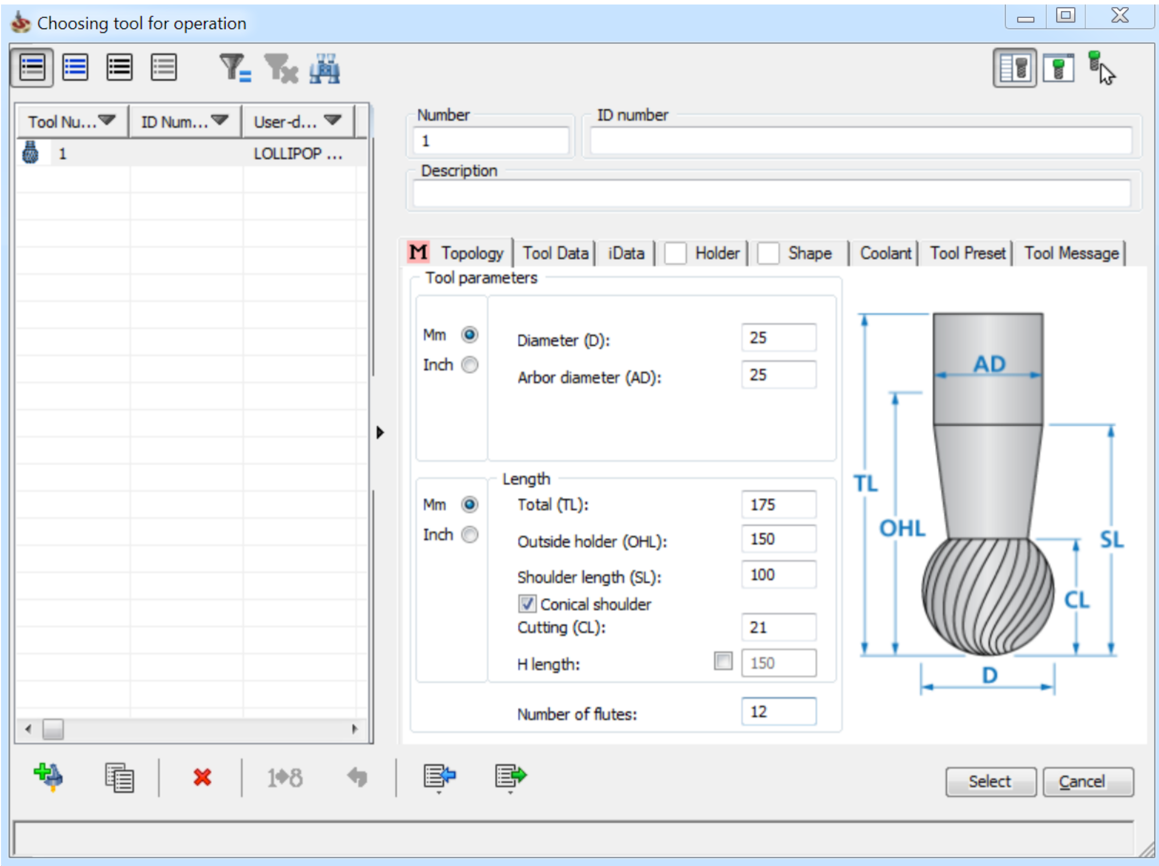
Click **Tool** > **Select**.



Click  icon to add a new tool.

Select Lollipop Mill as our tool. This tool is the tool of choice for machining such geometries.

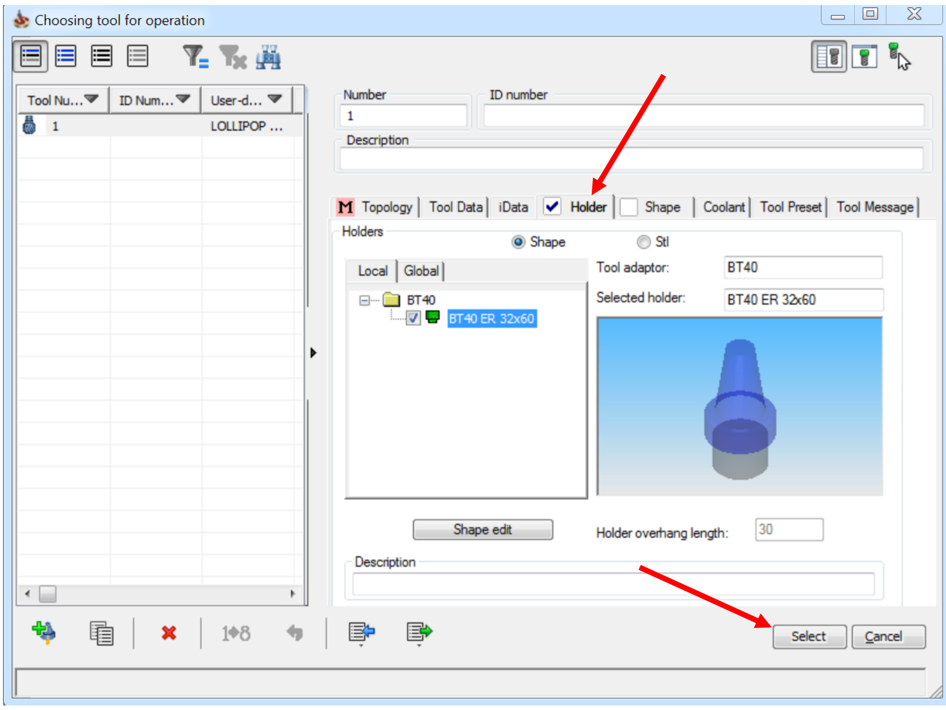
Enter the tool parameters as shown in the image.



Select **Holder**.

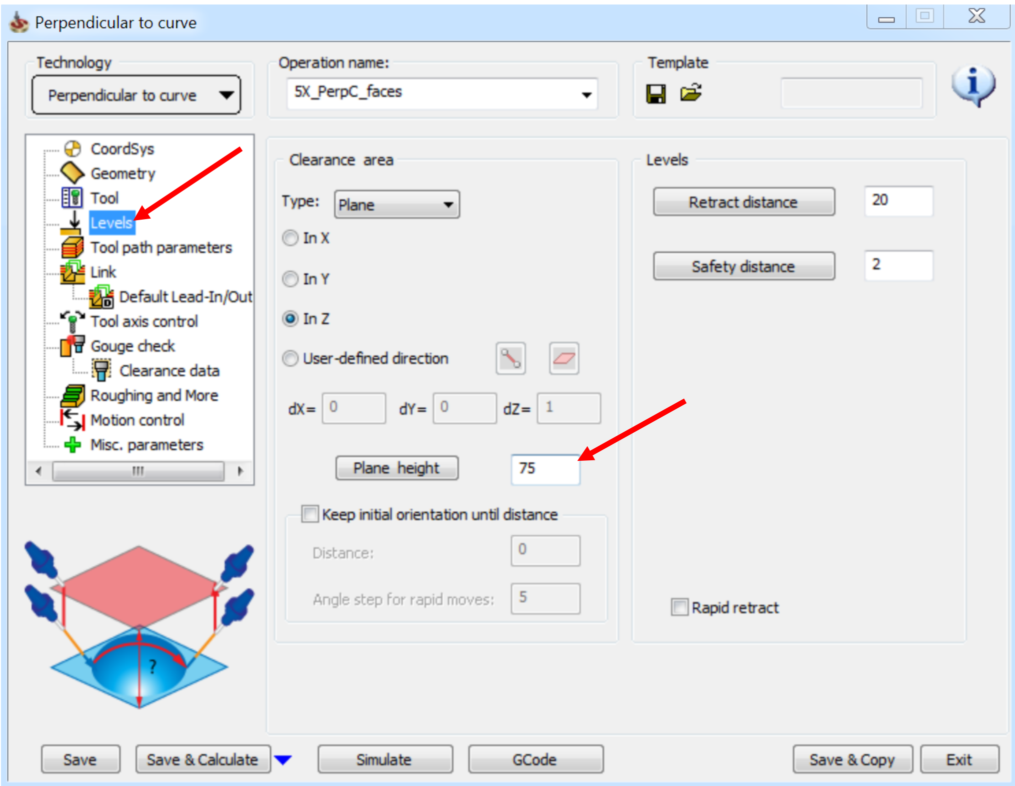
Select BT 40 > BT 40 ER 32X60.

Click **Select**.

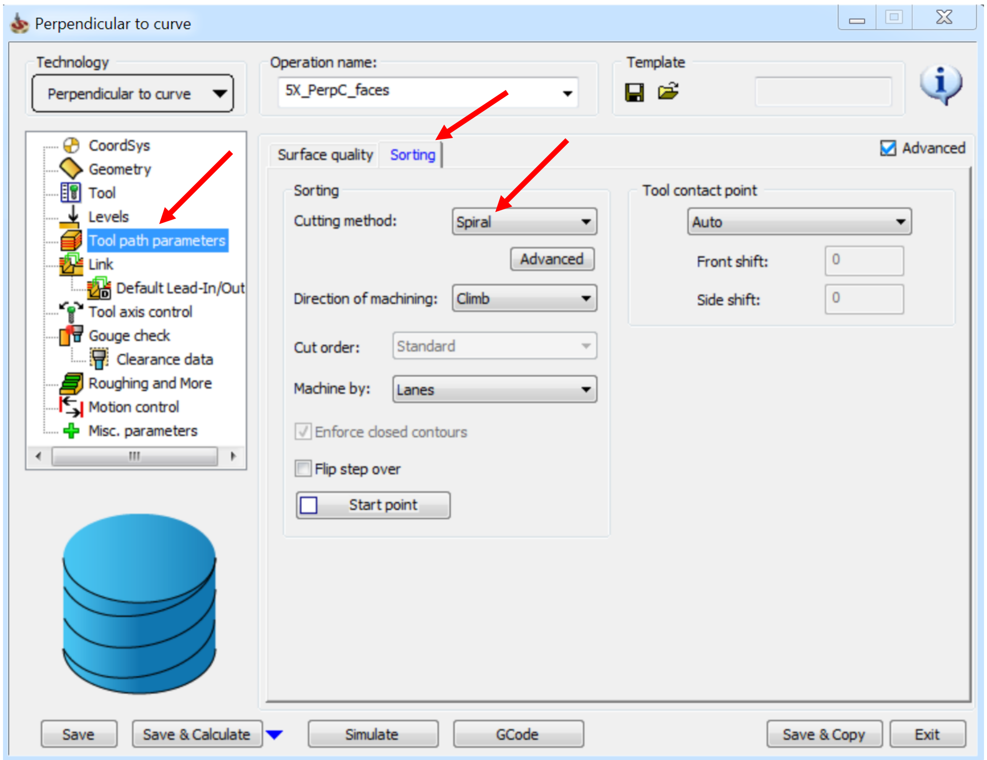


Click **Levels**.

Enter a value of 75 mm in the **Plane height** field.



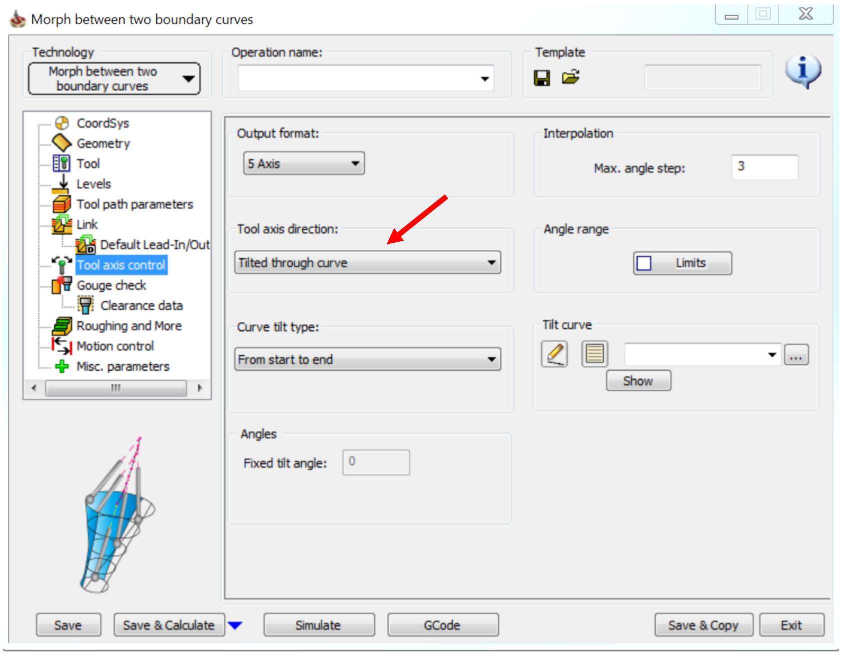
Click **Tool path parameters** > **Sorting**.



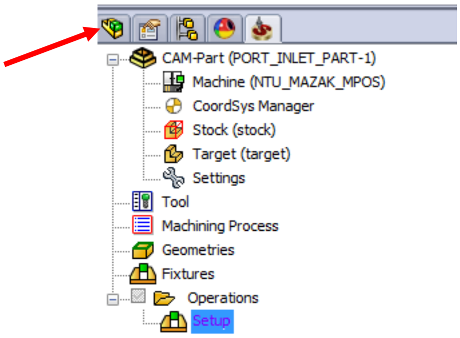
Select **Spiral** as the **Cutting method**.

Click **Tool axis control**.

Select **Tilted through curve** as the **Tool axis direction**.



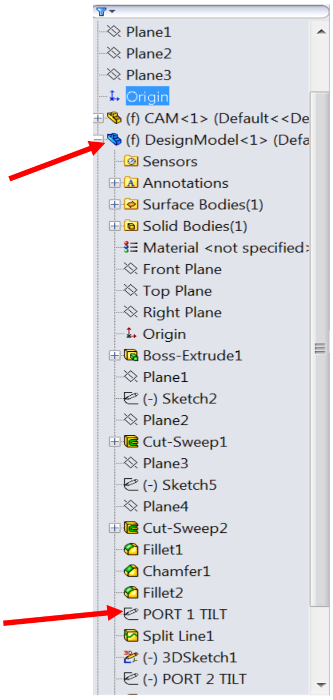
Select the Feature Manager Design Tree icon to switch on the curve.



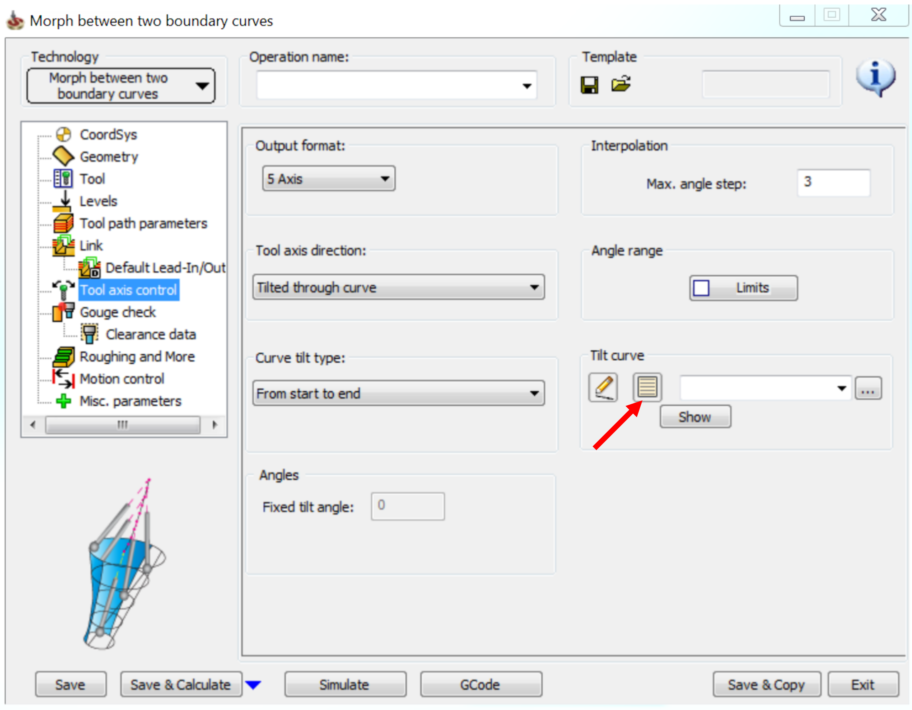
Expand the **Design Model**.

Select **PORT 1 TILT**.

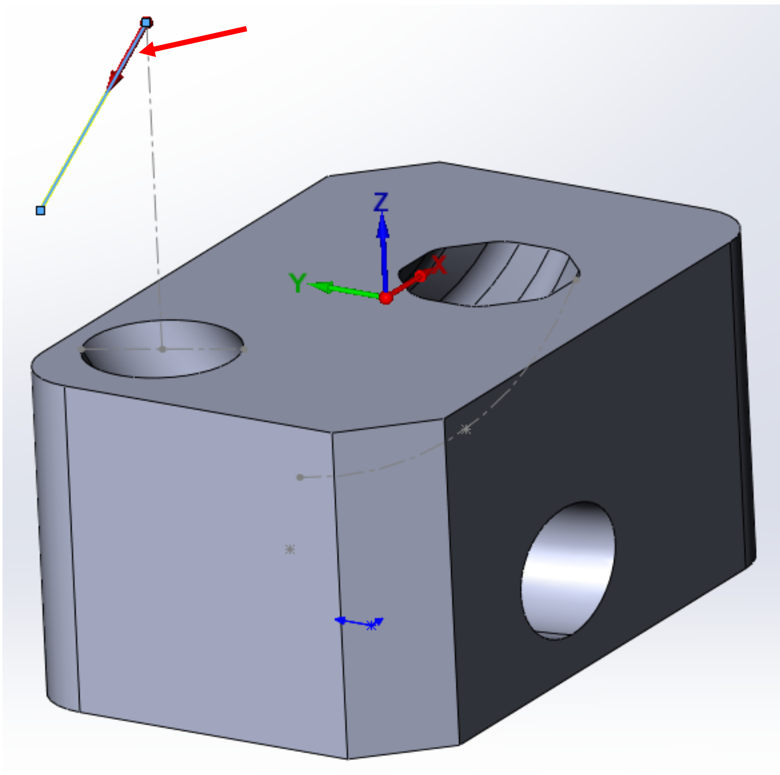
Select the Show icon.



Click the New icon under **Tilt curve** to define the new curve.



Select as shown in the image. Ensure that the selection is from top to bottom, as is the direction of the arrow.

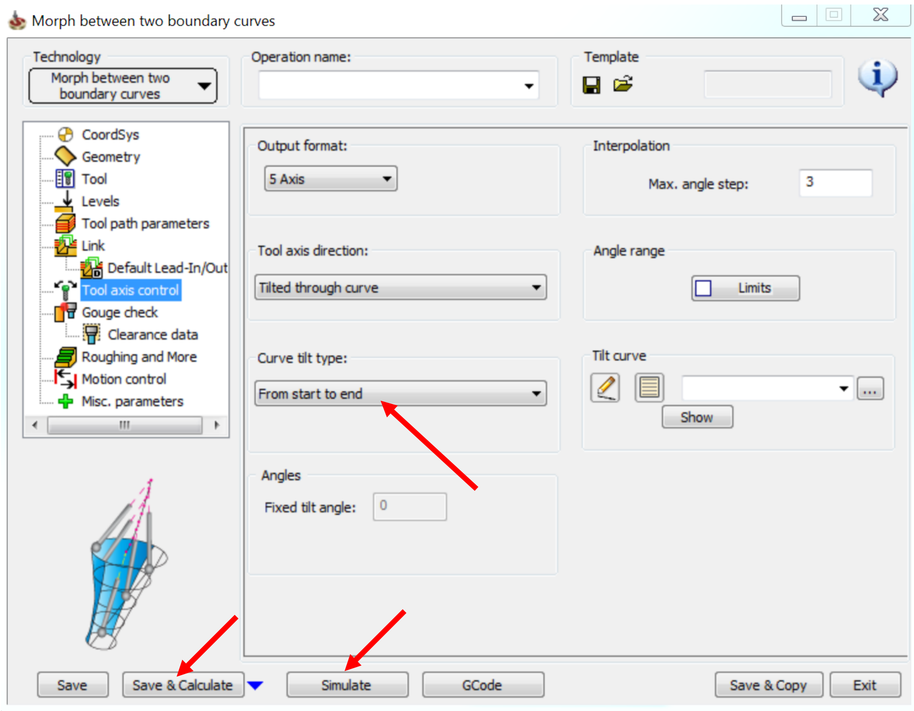


Click 

Click 

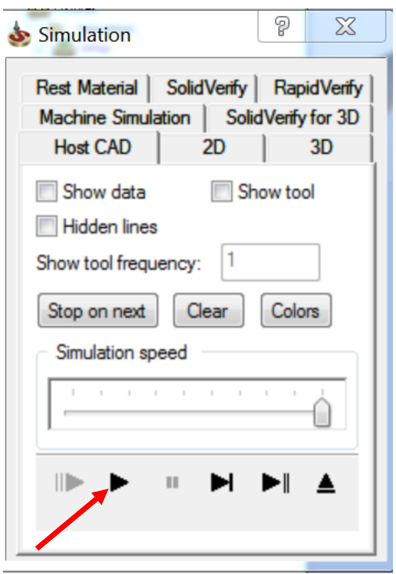
In the **Curve tilt type** select **From start to end**. This tilt type is used to force the tool axis to always follow the tilt curve from start of the curve to the end of the curve. This is the most used tilt type for machining ports.

Click **Save & Calculate**.

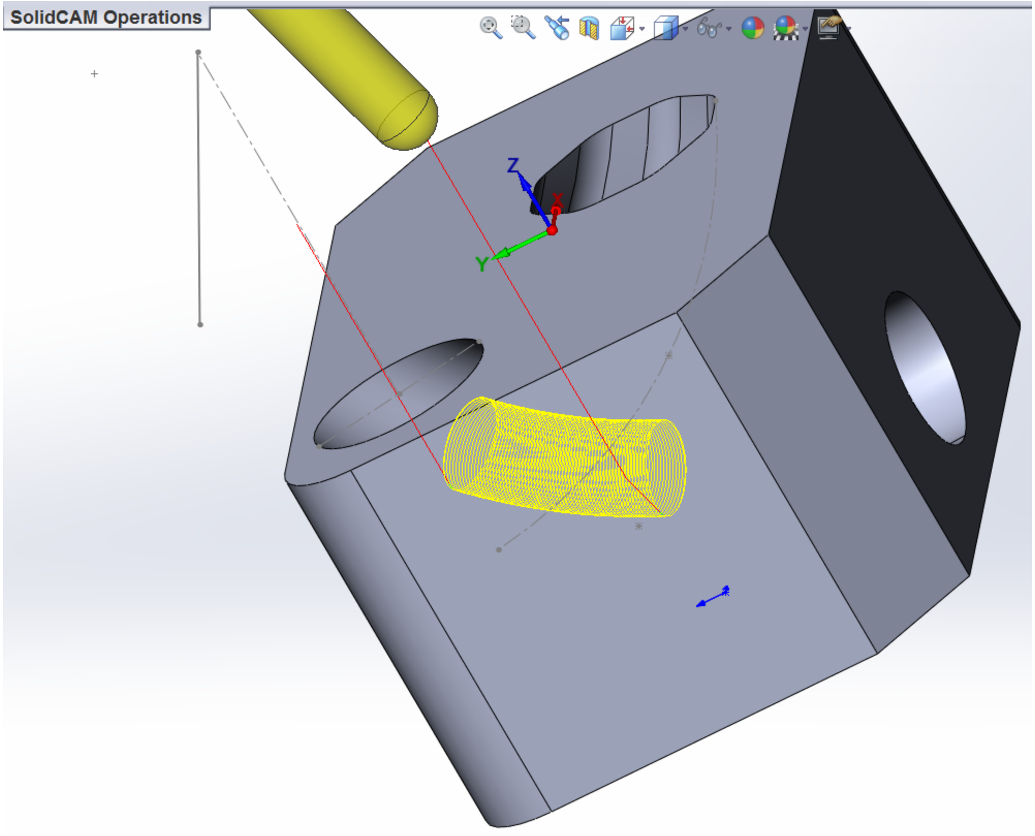


Click **Simulate** once the tool path is calculated.

Click the Play icon.

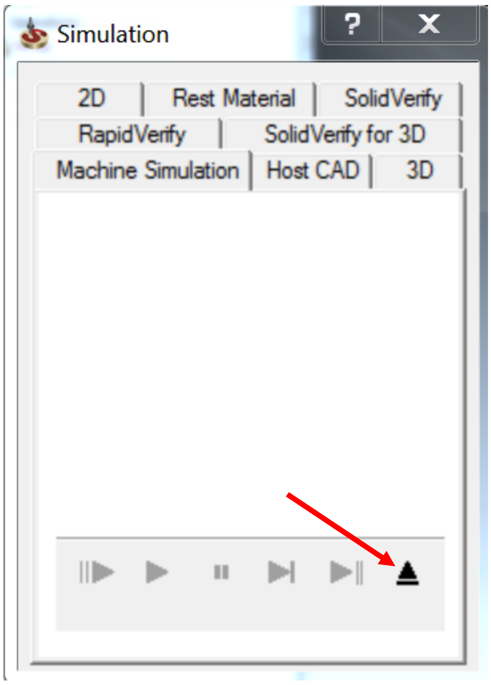


The simulated toolpath looks like this:

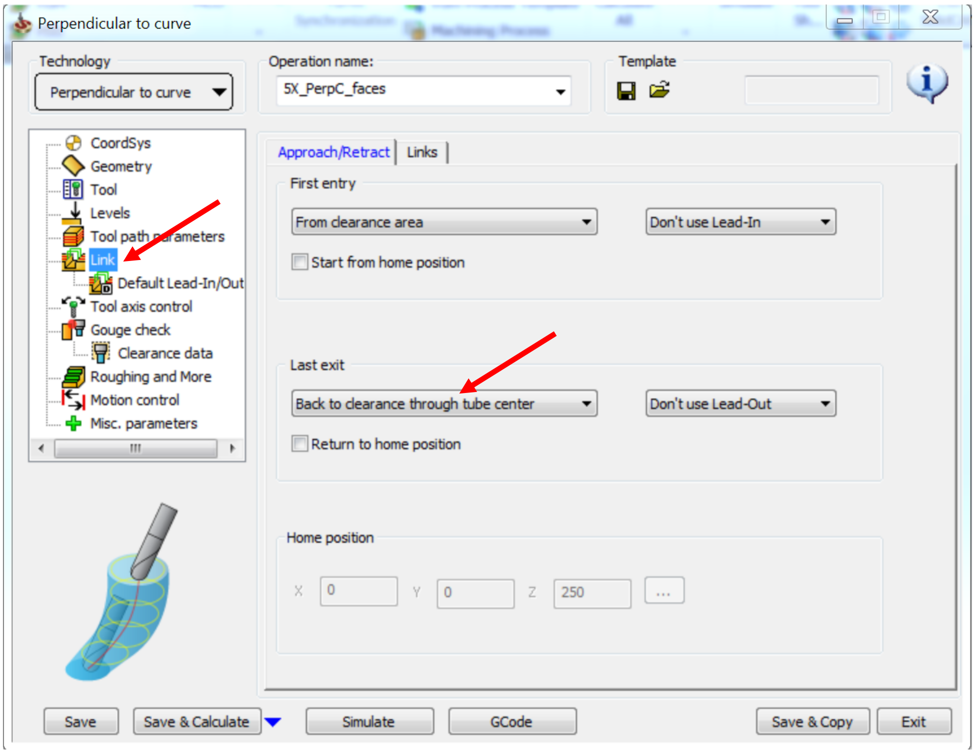


You can see that a retract is coming through the part & would cause a major crash on the machine. We need to fix this.

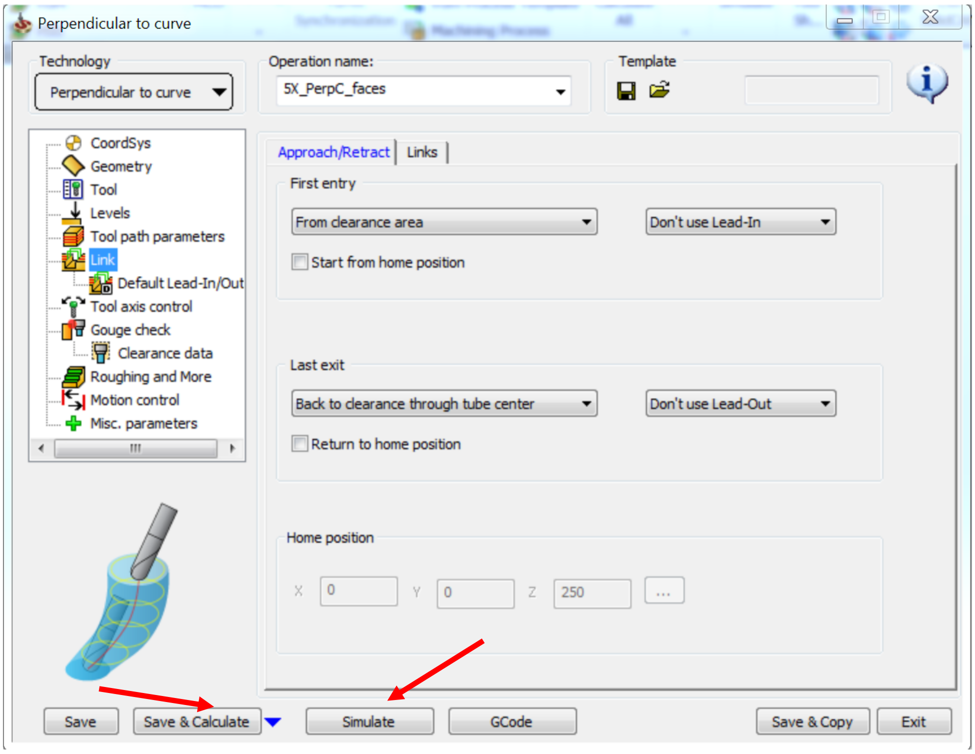
Select the **Exit** icon to exit machine simulation.



Click **Link**.



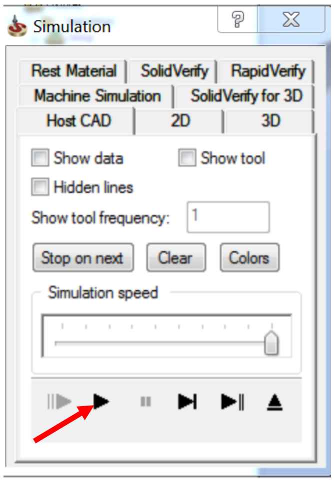
Select **Back to clearance through tube center** as the **Last exit**.



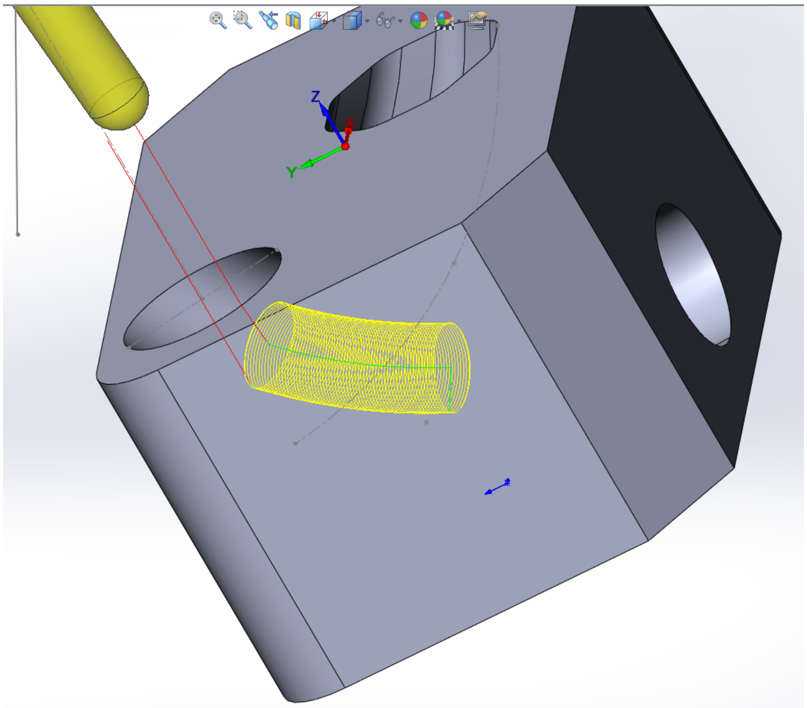
Click **Save & Calculate**.

Click **Simulate**.

Click the Play icon.

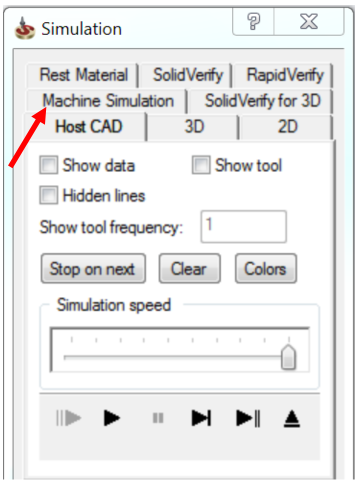


The simulated toolpath looks like this:



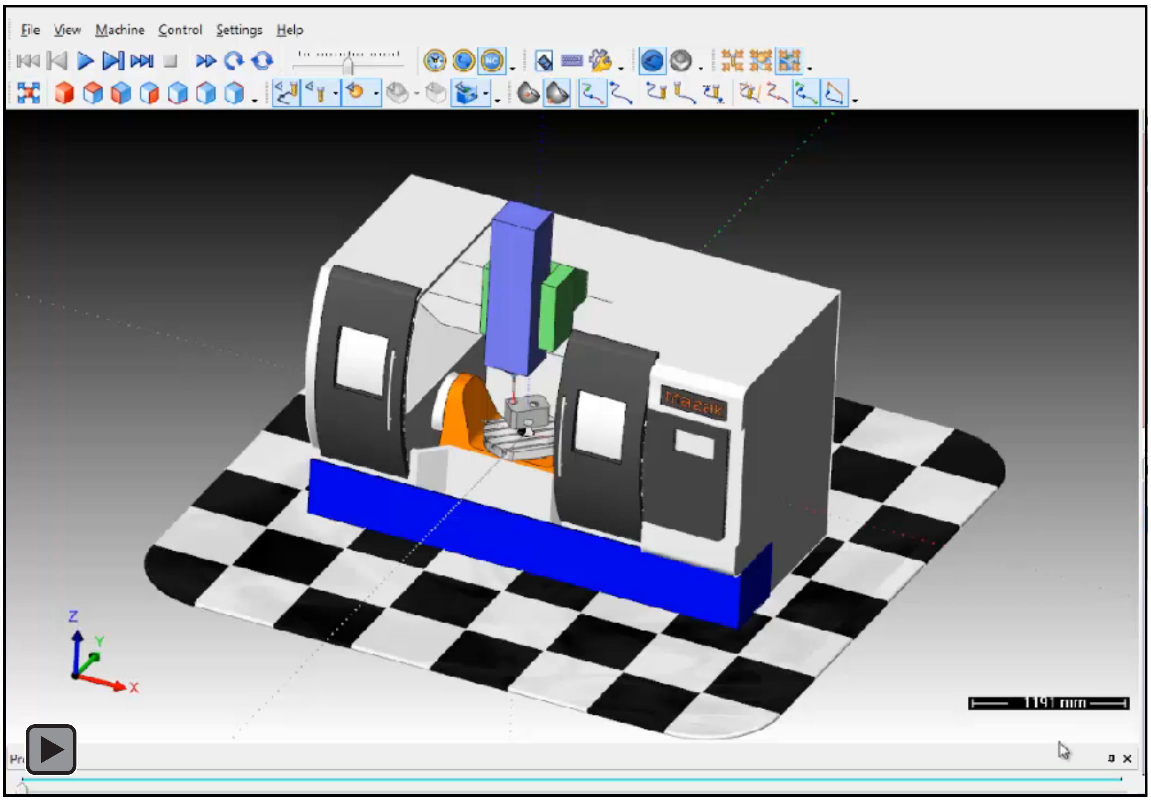
The tool is now retracting out through the center without causing any gouges to the part. This is a new retract option that was added in SolidCAM and is exclusively used for machining port geometries.

Click **Machine Simulation**.

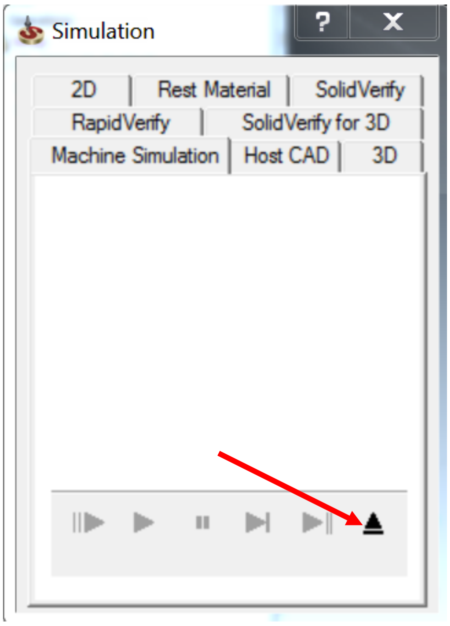


Click the hide  icon.

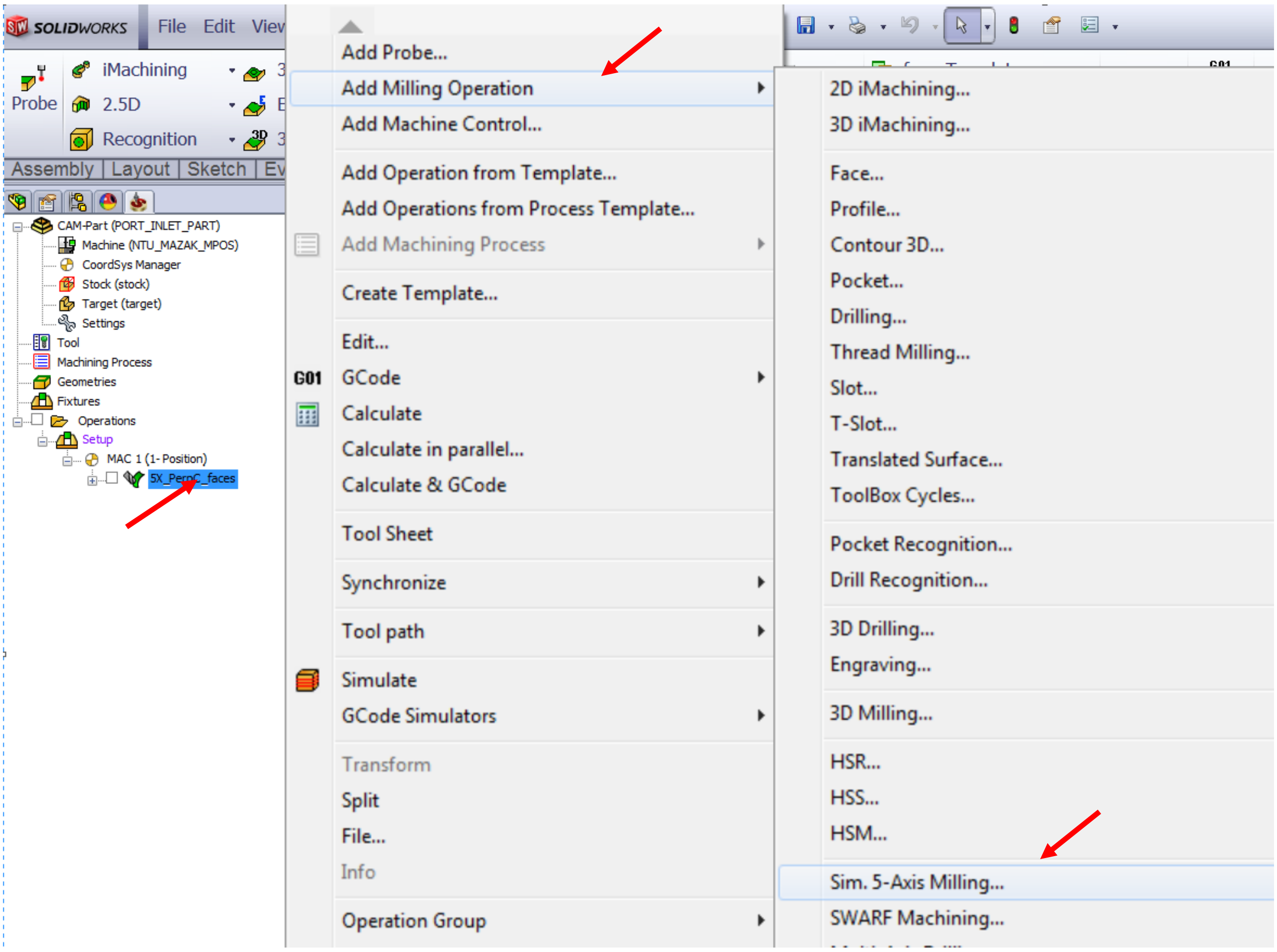
Run machine simulation by clicking the play icon.



Select the **Exit** icon to exit machine simulation.

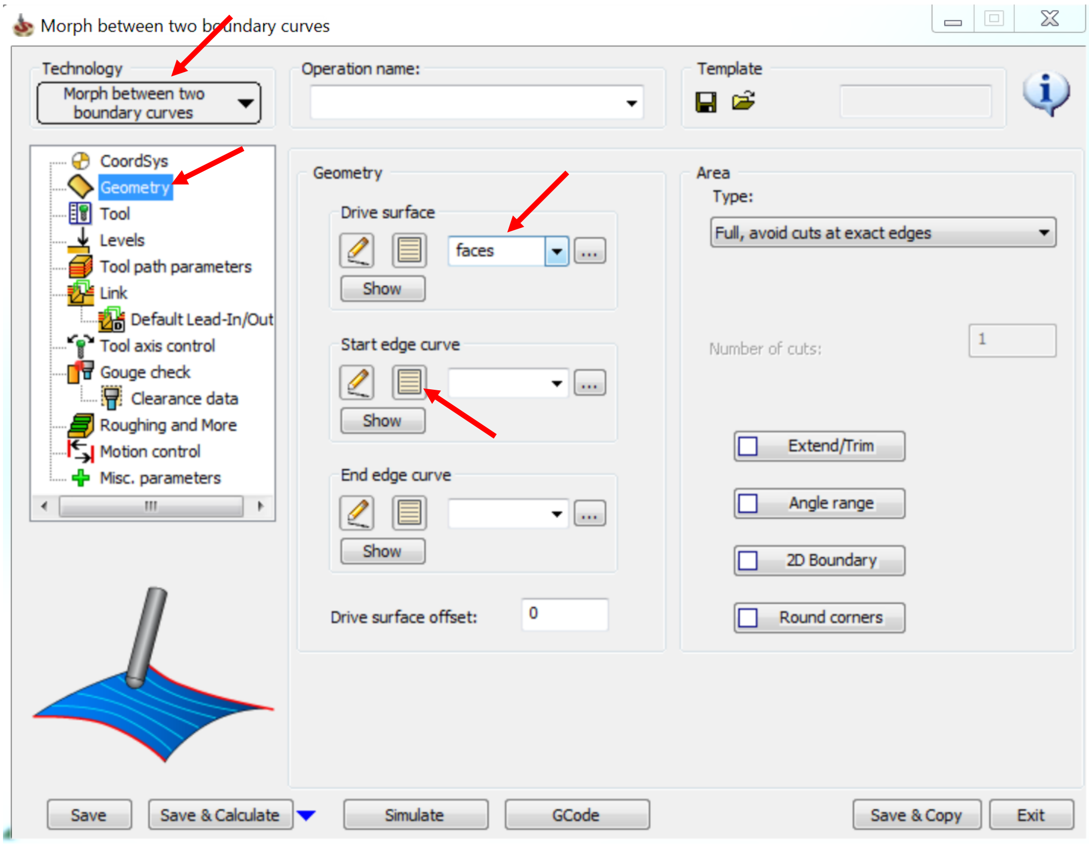


Right click the tool path > **Add Milling Operation** > **Sim.5- Axis Milling**.



The Operations Manager window opens.

Select **Morph between two boundary curves** in the **Technology** section.



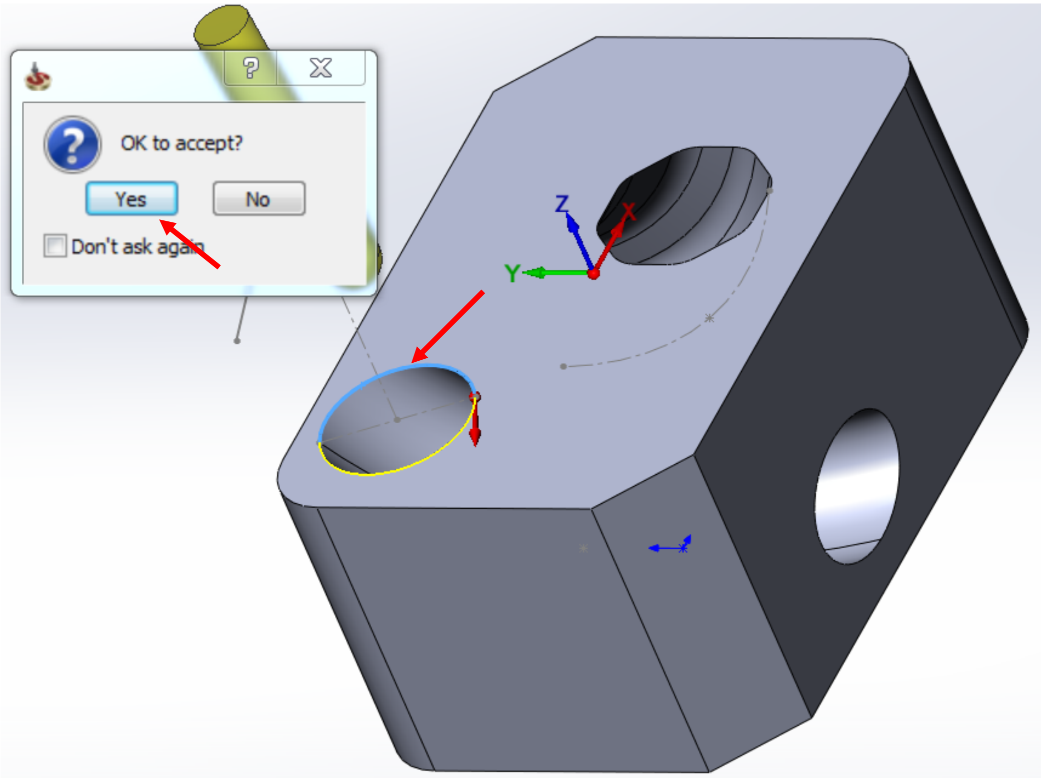
The Morph Between Two Curves feature creates a morph toolpath between two leading curves. Morph means that the generated toolpath gradually interpolates between the two curves and it is evenly spread over the surface. When selecting the two curves, the geometry should be selected directly from the drive surfaces.

Click **Geometry**. The geometry remains the same.

Select faces from the drop down list.

Click New icon under **Start edge curve**.

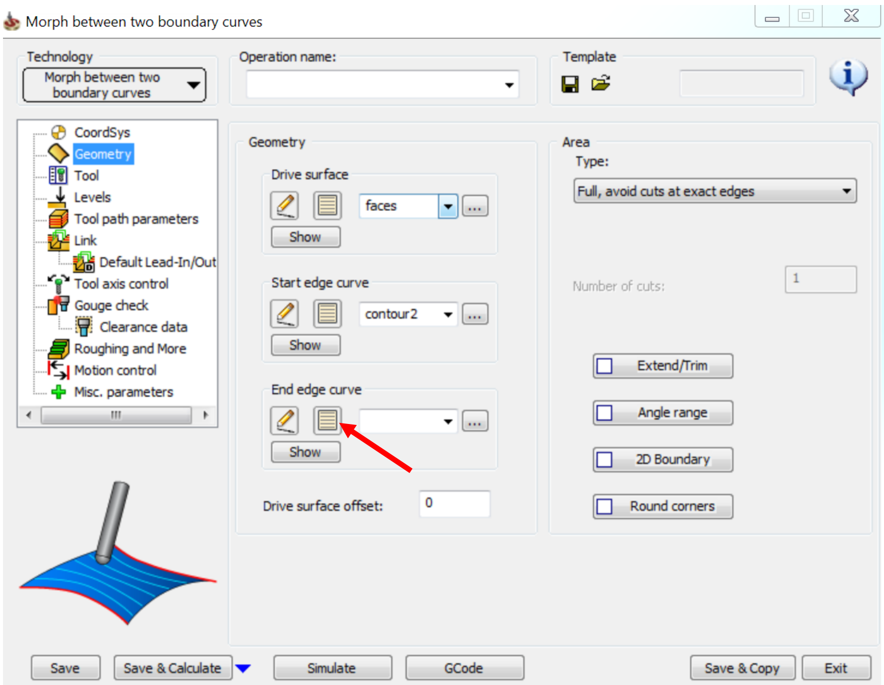
Select the curve as shown in the image.



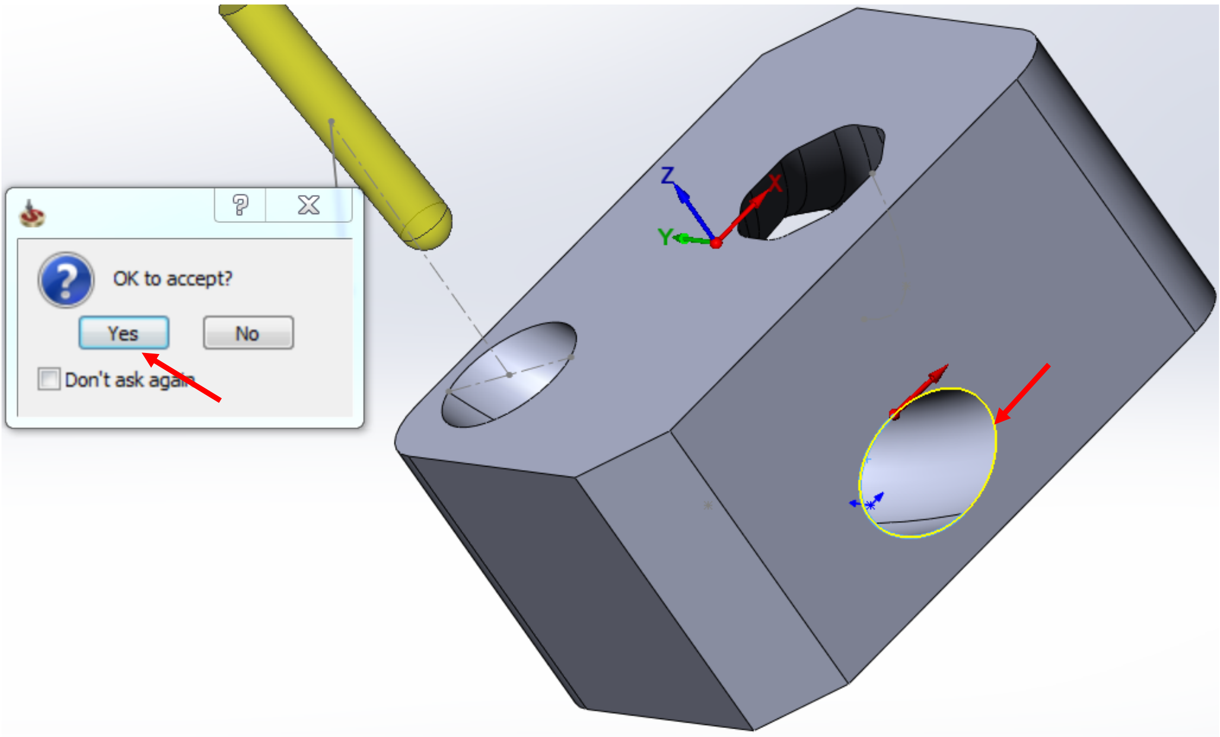
Click **Yes** in **OK to accept** pop up window. The OK to accept message will ap pear only if this option is selected under “SolidCAM Settings ==> CAM Messages”.

Click

Click the New icon under **End edge curve**.



Select the curve as shown in the image.

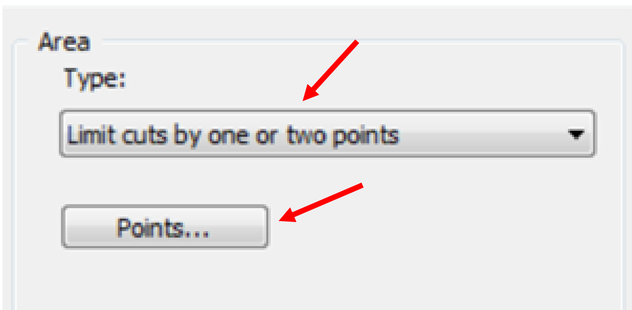


Click **Yes** in **OK to accept** pop up window.

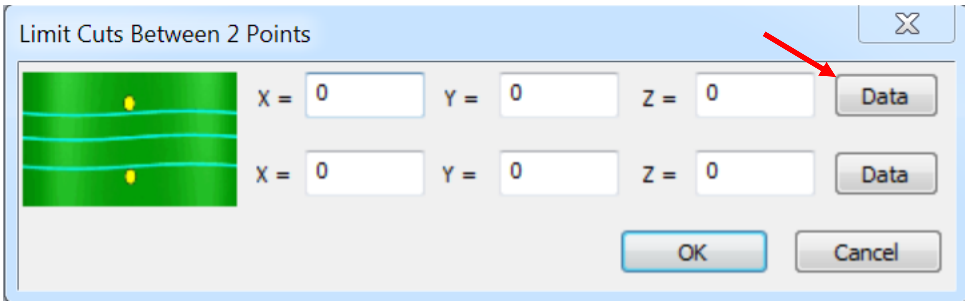
Click 

Select the **Type** as **Limit cuts by one or two points** from the drop down list in the **Area** section.

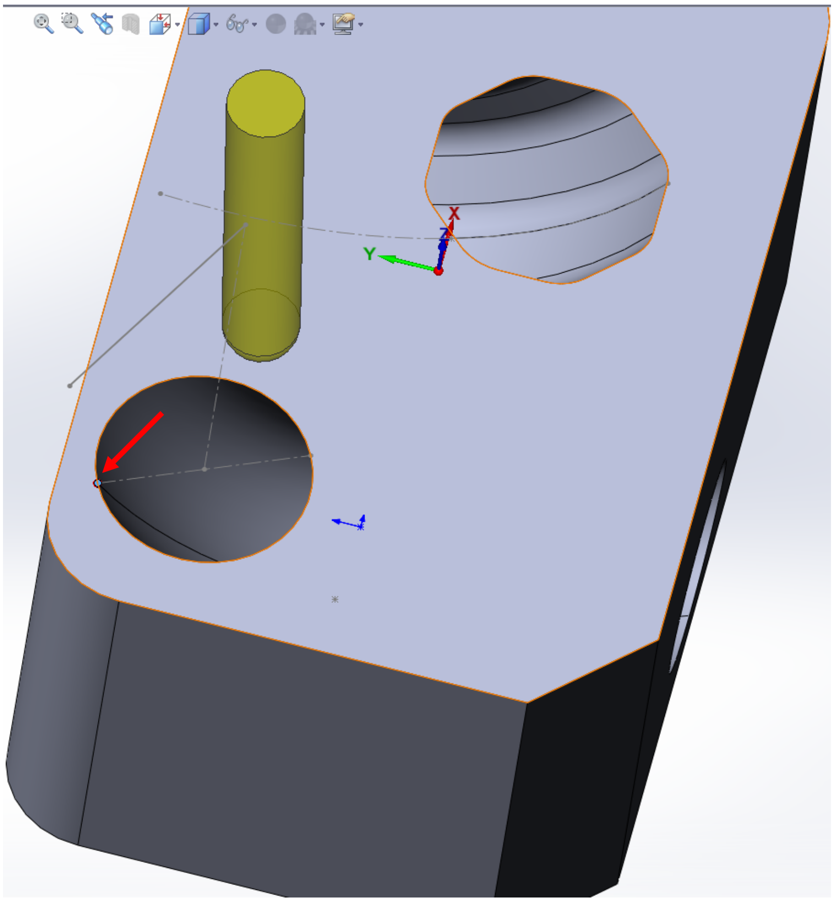
Click **Points**.



Click **Data**.

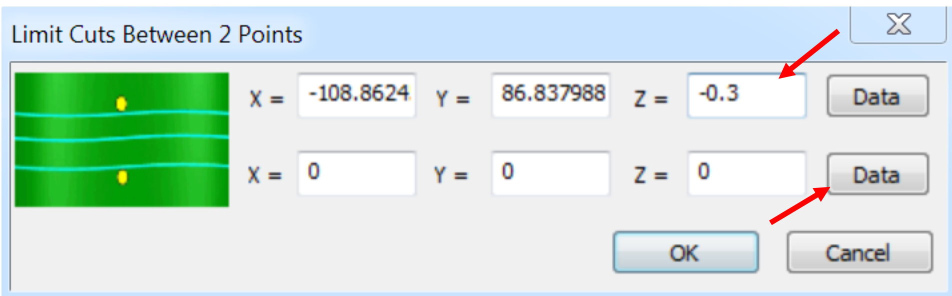


Select the point as shown in the image.



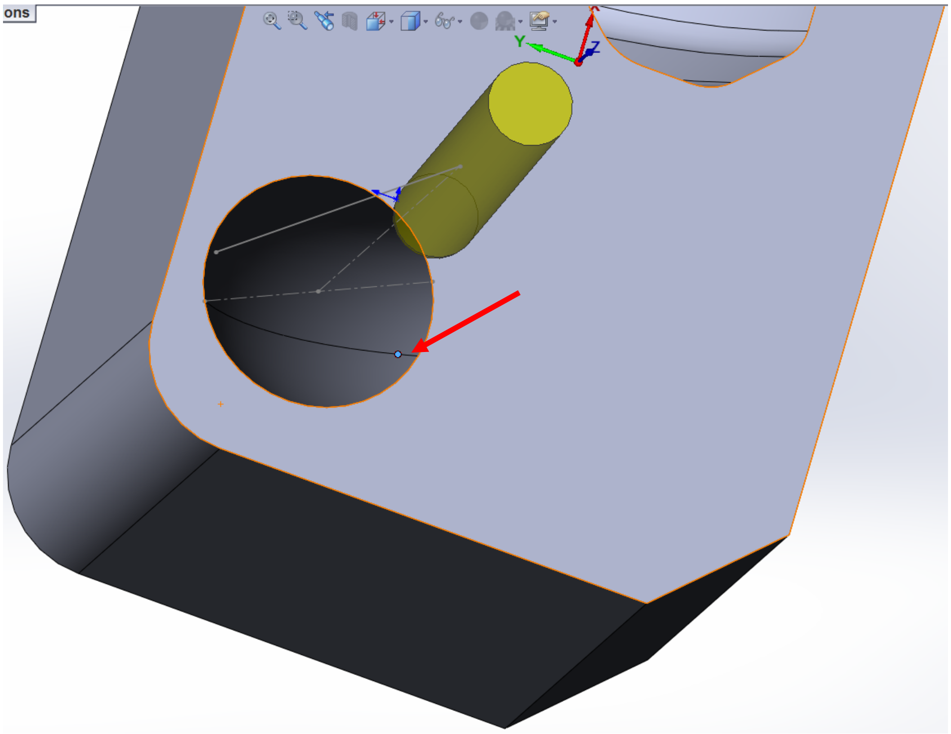
Click ✔

Define the first margin again as -0.3 in the **Z** field.



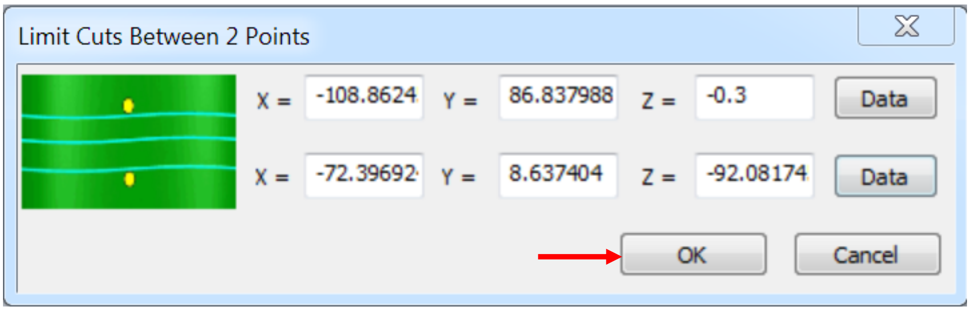
Click **Data**.

Select the point as shown in the image.



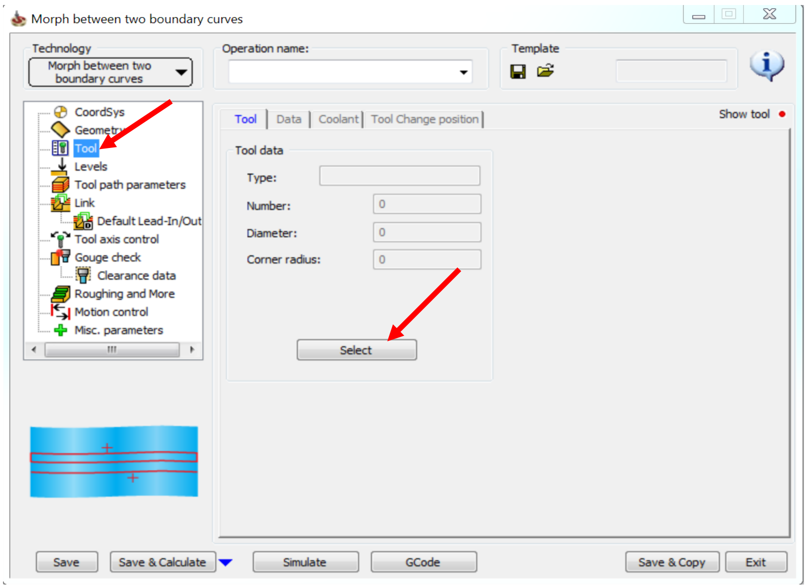
Click ✓

Click **OK** in the Limit Cuts Between 2 Points window.

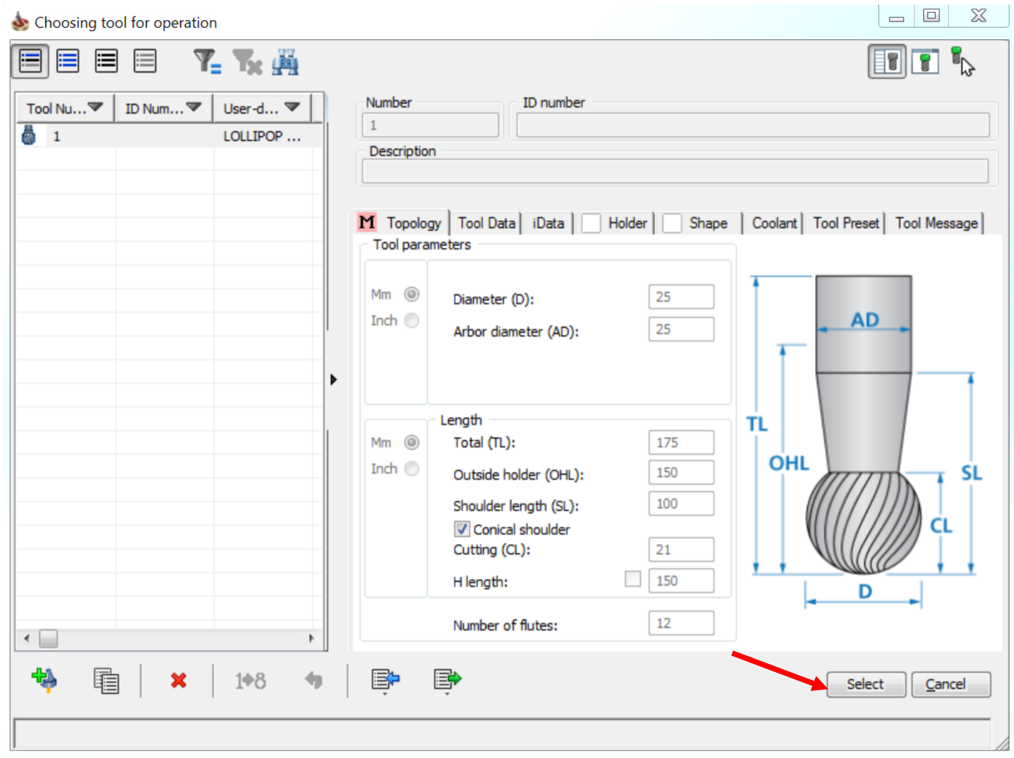


Click **Tool**.

Click **Select**.

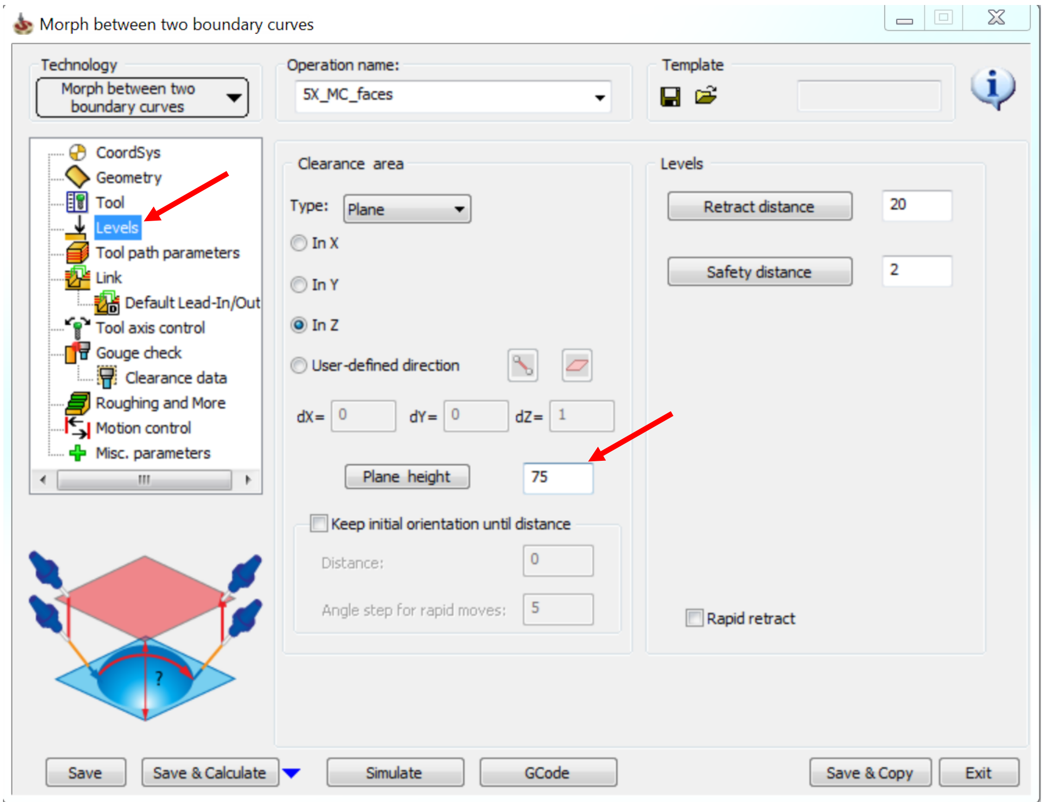


Click **Select** as we will use the same tool.

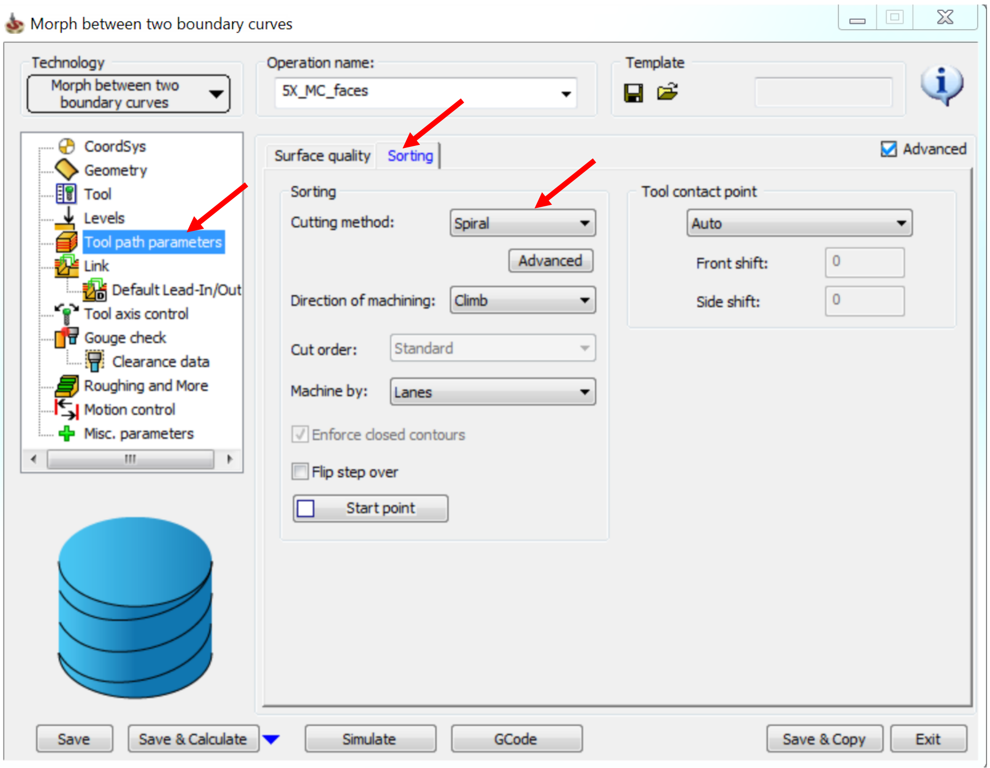


Click **Levels**.

Enter a value of 75 mm in the **Plane height** field.



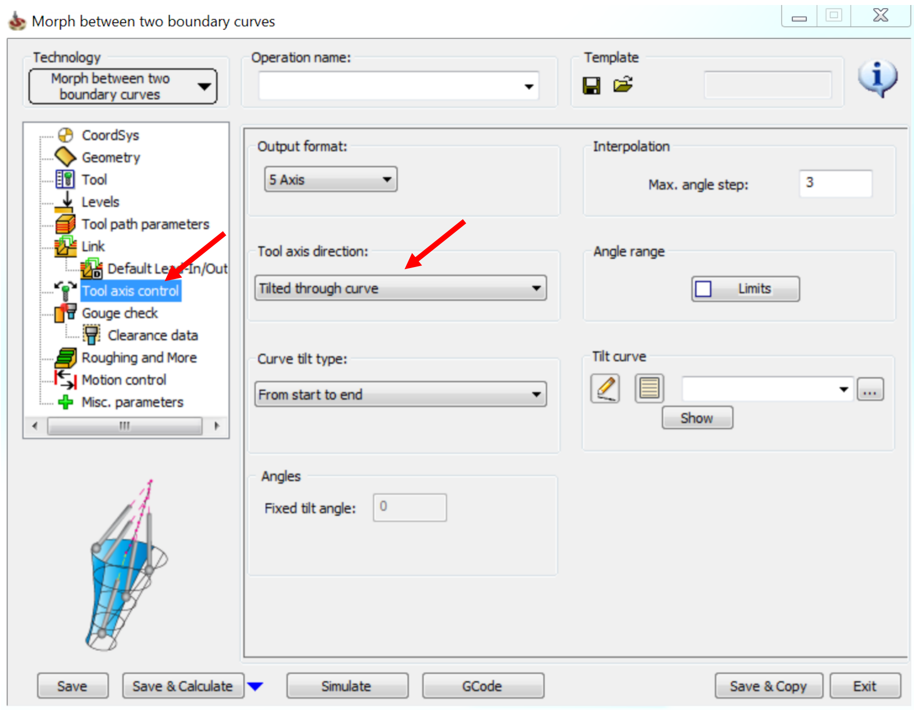
Click **Tool path parameters** > **Sorting**.



Select **Spiral** as the **Cutting method**.

Click **Tool axis control**.

Select **Tilted through curve** as the **Tool axis direction**.

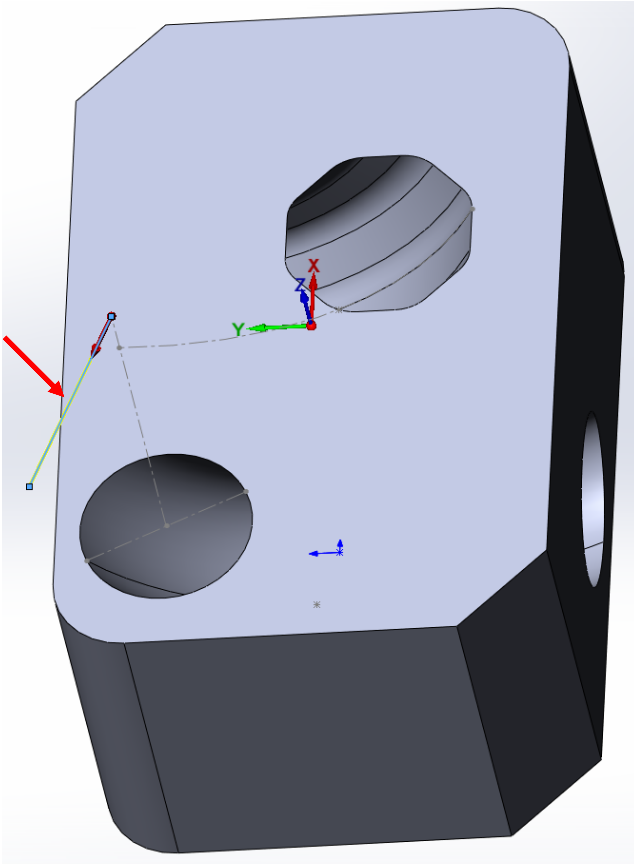


Click New icon under **Tilt curve**.

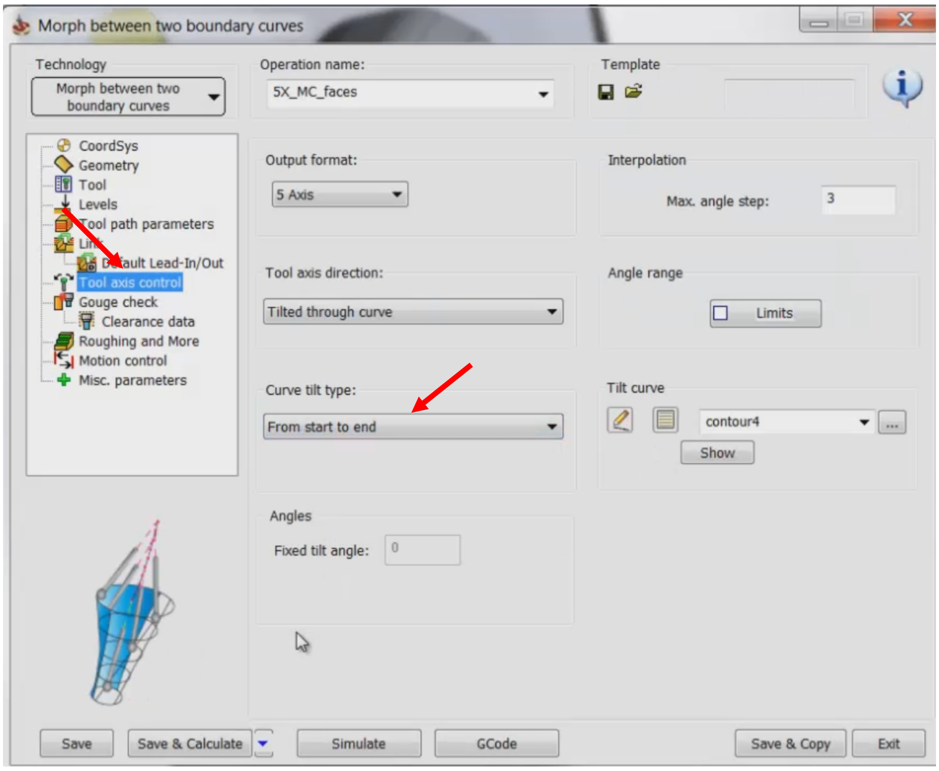
Select the curve as shown in the image. Remember the direction of the curve.

Click

Click

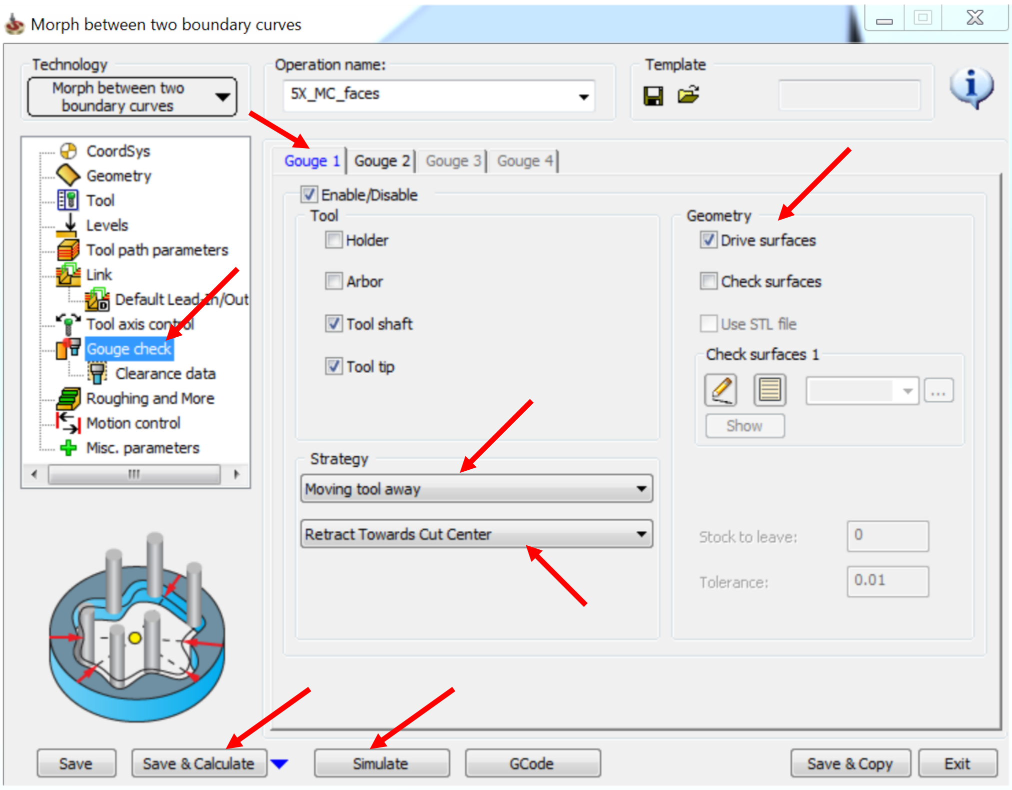


Click **Tool axis control**.



Select **From start to end** as the **Curve tilt type**.

Click **Gouge check**.

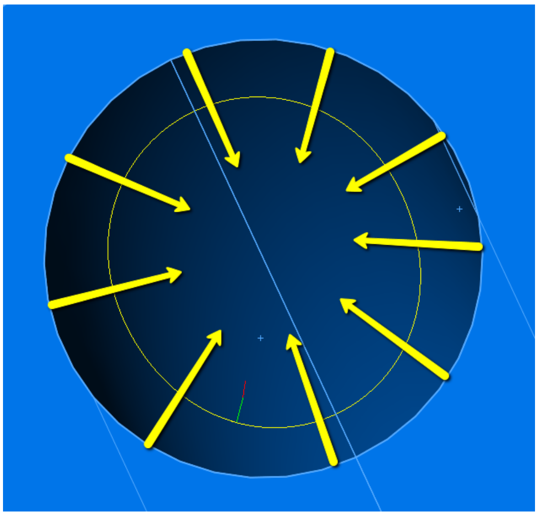


Select the **Enable/Disable** check box.

Ensure that only the **Drive surfaces** check box is selected in the **Geometry** section.

Select **Moving tool away** and **Retract Towards Cut Center**.

What this gouge check does it to push the tool-path to the centre of the geometry so that it does not gouge the geometry anymore. This would ensure that the tool sticks to the geometry without gouging the part surfaces.



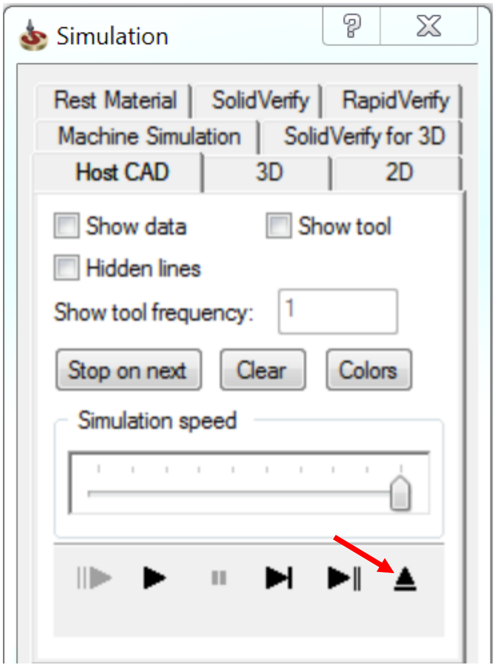
Click **Save & Calculate**.

Click **Simulate** once the tool path is calculated.

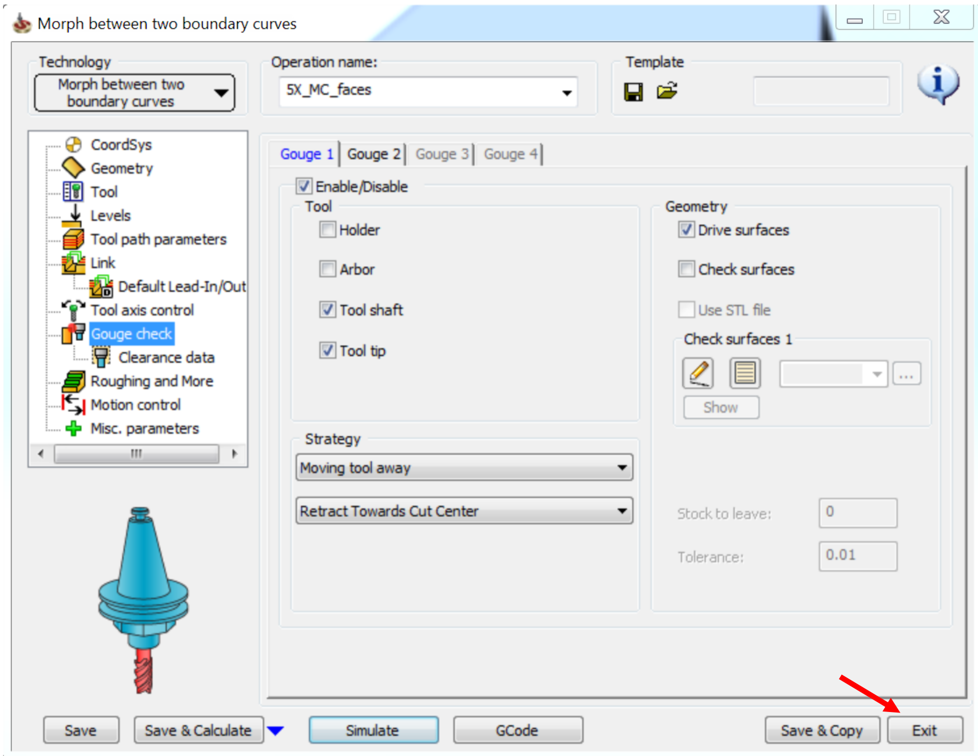
The Machine simulation now looks something like this as shown in the video.

The machine simulation still has gouges which need to be fixed.

Click the Exit icon to exit the machine simulation.

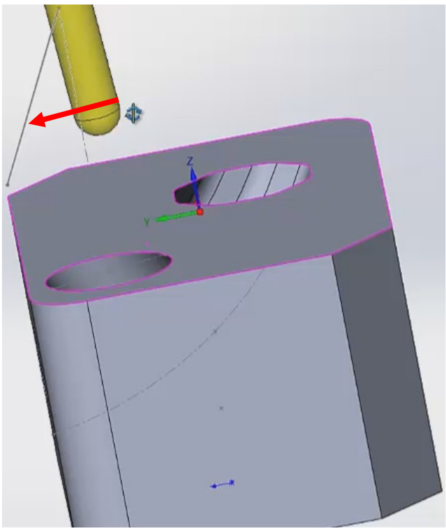


Click **Exit** from the Operations Manager.

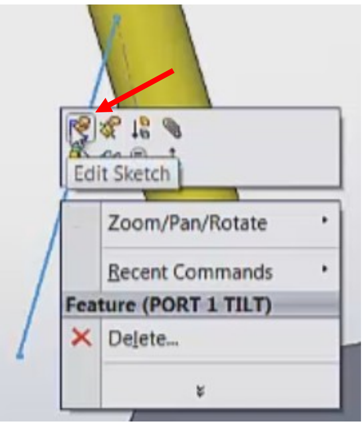


We will now fix the gouges by changing the angle of the tilt curve. To do this we need to edit the primary sketch of the curve and change the angle.

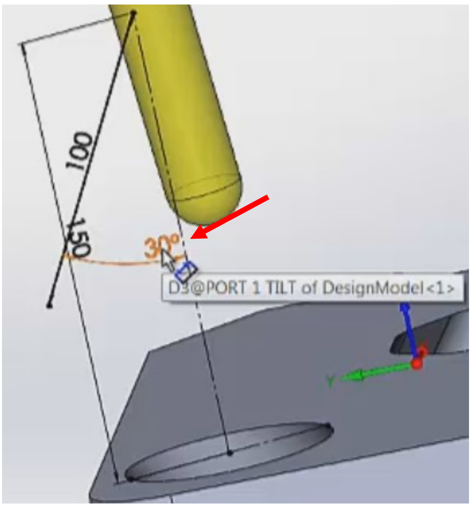
Select the curve.



Right click > Edit sketch.

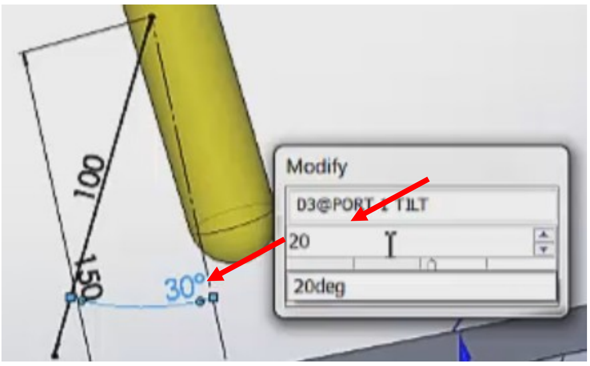


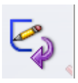
You can see that this curve has an angle of 30 degrees. This angle controls the tilt of the tool, how deep it can go and to which areas it can go. Let us reduce this angle.



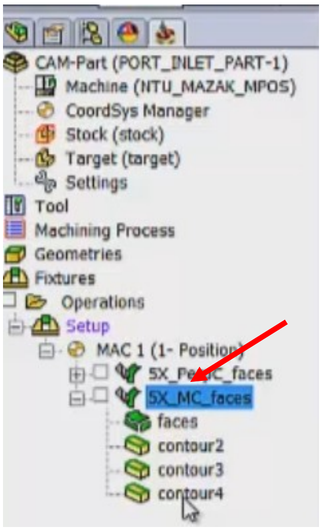
Double click the angle and enter a value of 20 in the Modify window.

Press the Enter key on your keyboard to accept the new value.

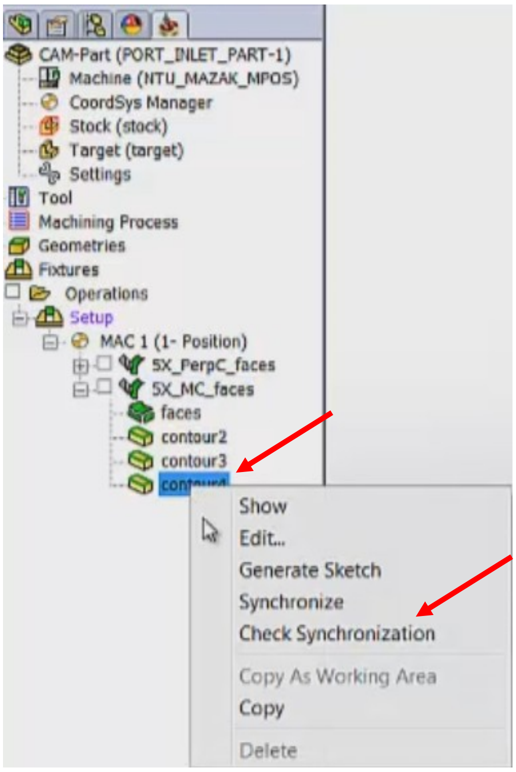


Click the Exit sketch  icon.

Expand the tool path.

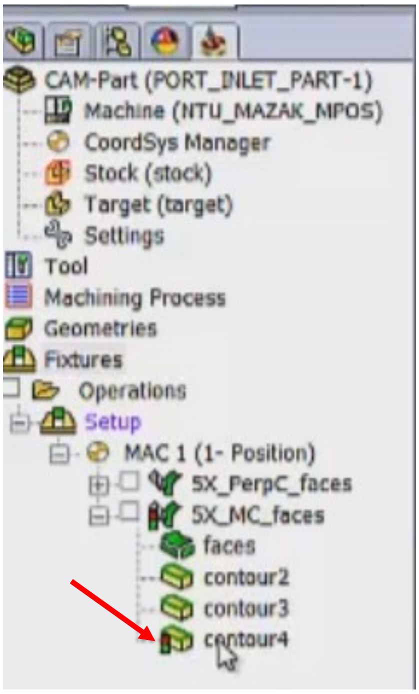


Right click **contour 4** > **Check Synchronization**.

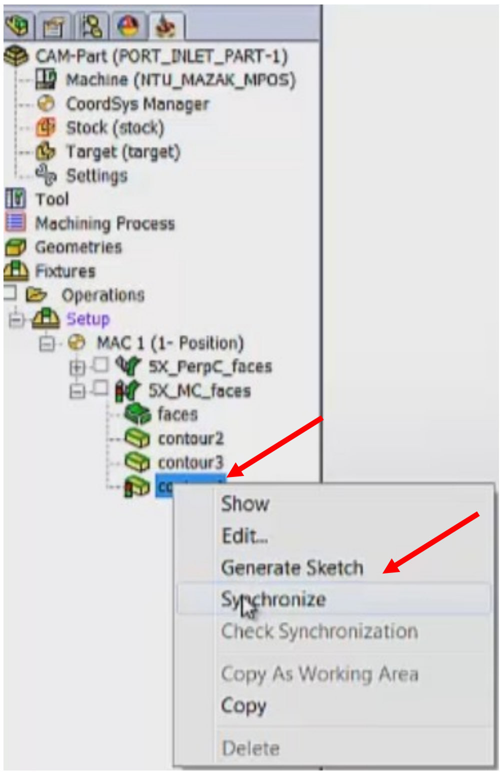


The synchronization symbol appears on contour 4 and we will have to rebuild this contour.

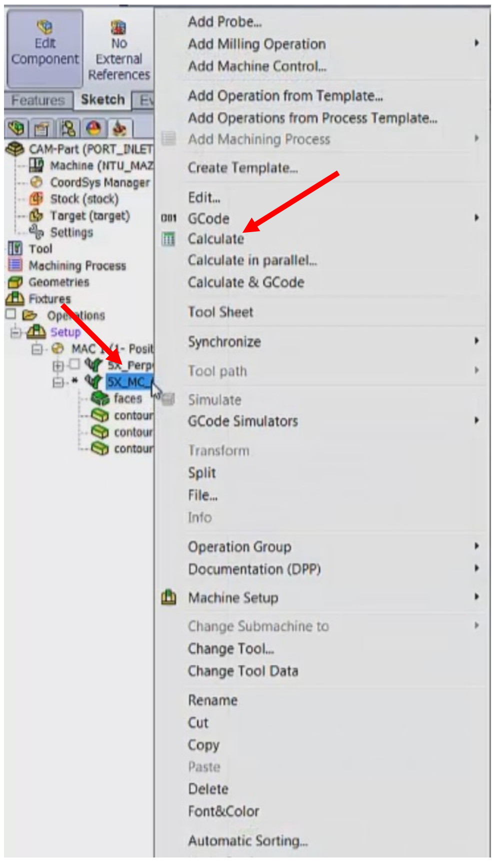
SolidCAM is smart enough to understand which geometry has changed and puts the rebuild symbol only on the geometry which needs to be rebuilt.



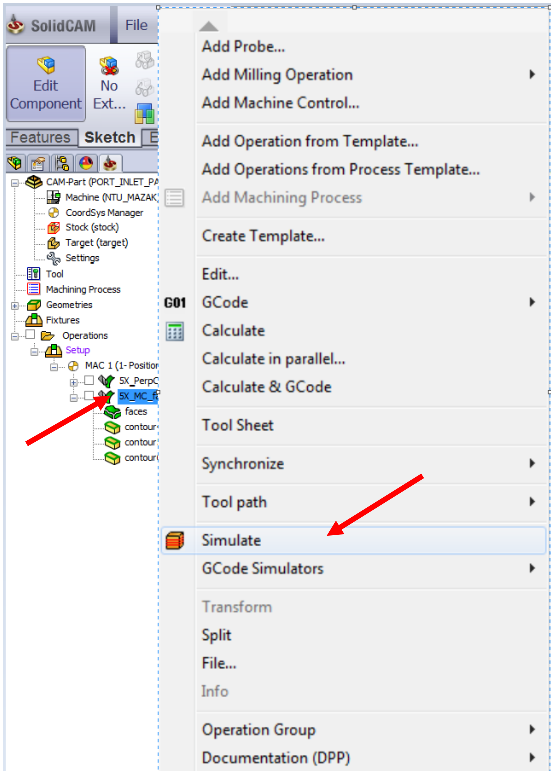
Right click **contour 4** > **Synchronize**.



Right click > **Calculate** the tool path.



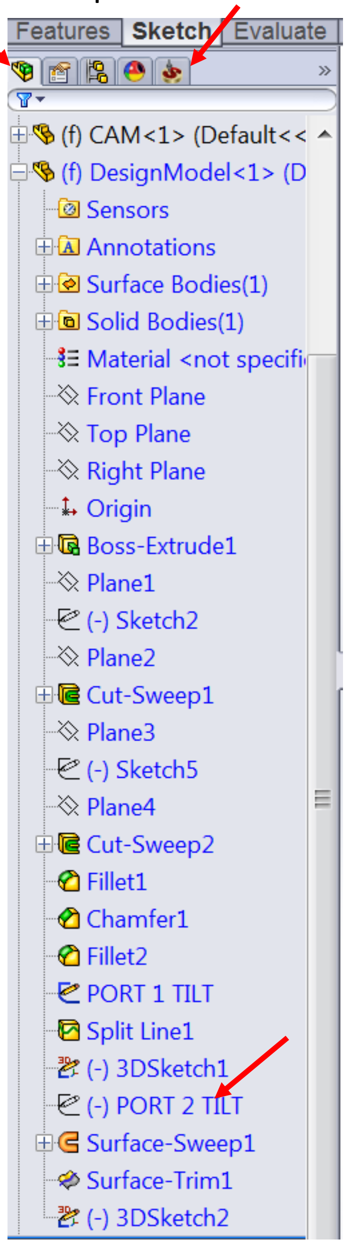
Right click > **Simulate** the tool path.



The simulated tool path looks like this:

You can see that there are no collisions now and the tool path looks clean. The angle of curve plays a major role in determining how deep the tool can get into the port without causing any kind of collisions.

Let us now look at machining another port which is octagonal in shape. Lets switch on the sketch for our next tool path.



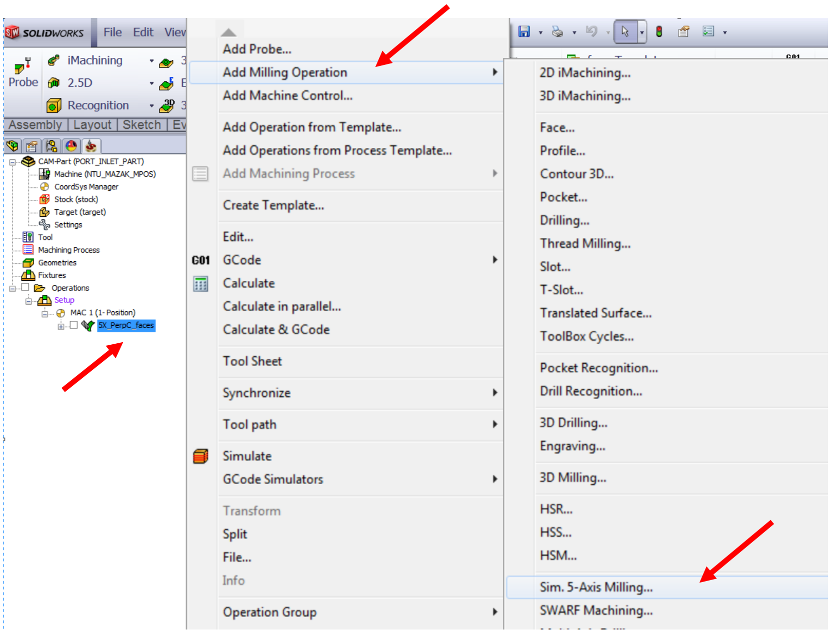
Click the Feature Manager Design Tree icon.

Select **PORT 2 TILT** under **Design Model**.

Click the show  icon.

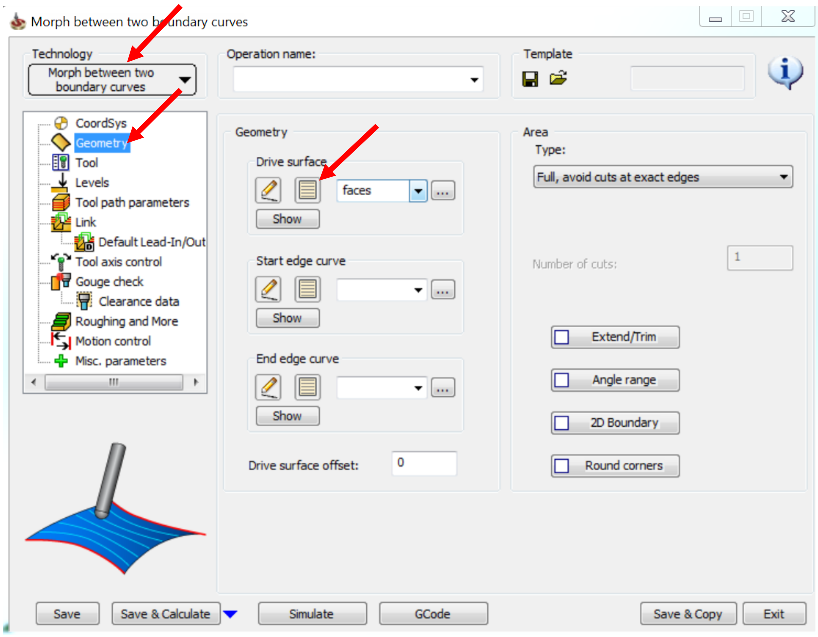
Click the SolidCAM Manager icon.

Right click the tool path > **Add Milling Operation** > **Sim.5 - Axis Milling**.



The Operations Manager window opens.

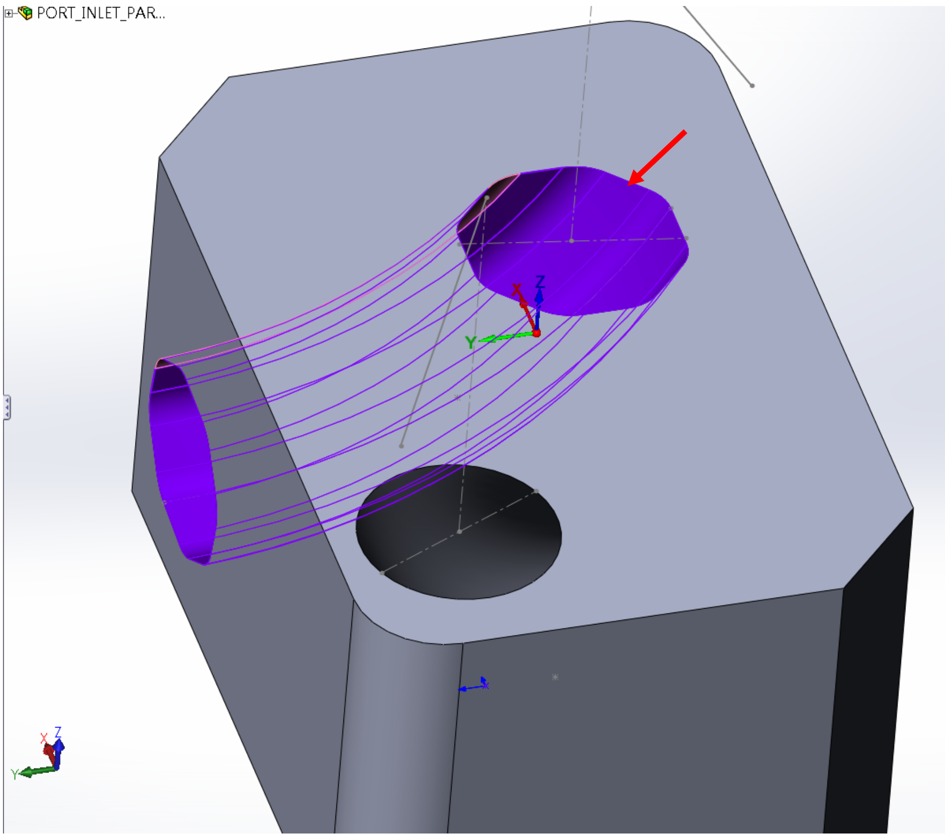
Select **Morph between two boundary curves** in the **Technology** section.



Click **Geometry**.

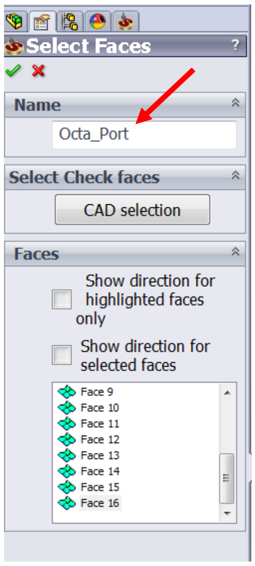
Click the new icon under **Drive surface**.

Select all the surfaces of the octagonal curve as shown in the image.

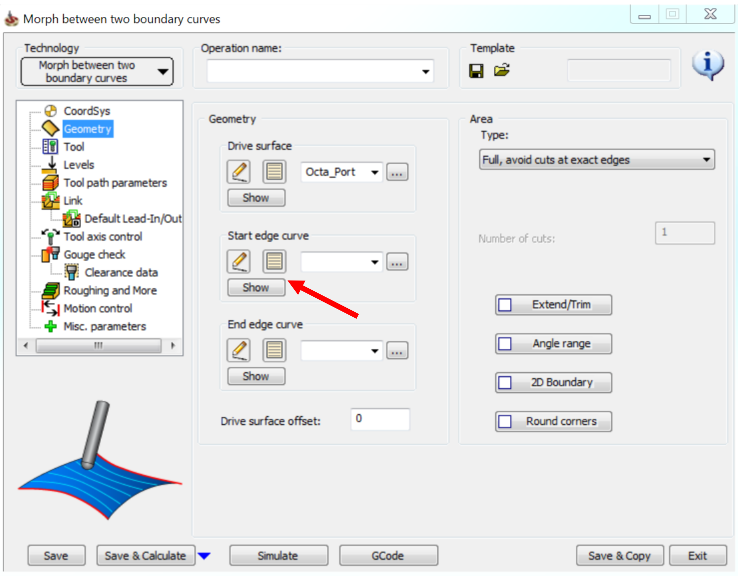


Change the Name as Octa_Port.

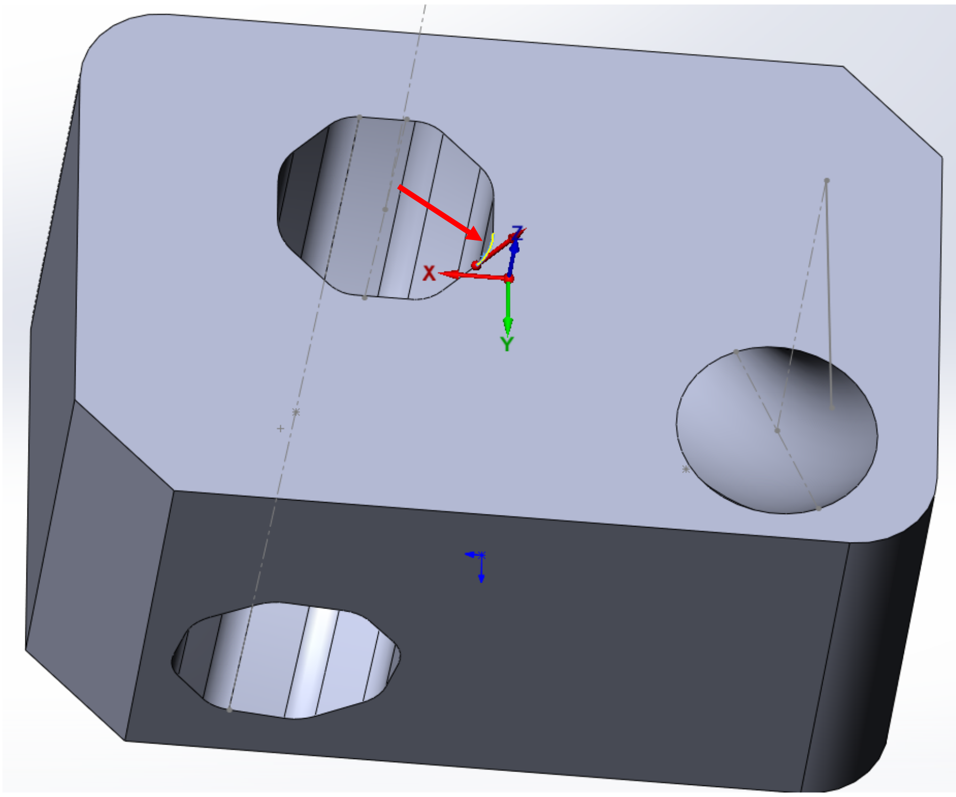
Click



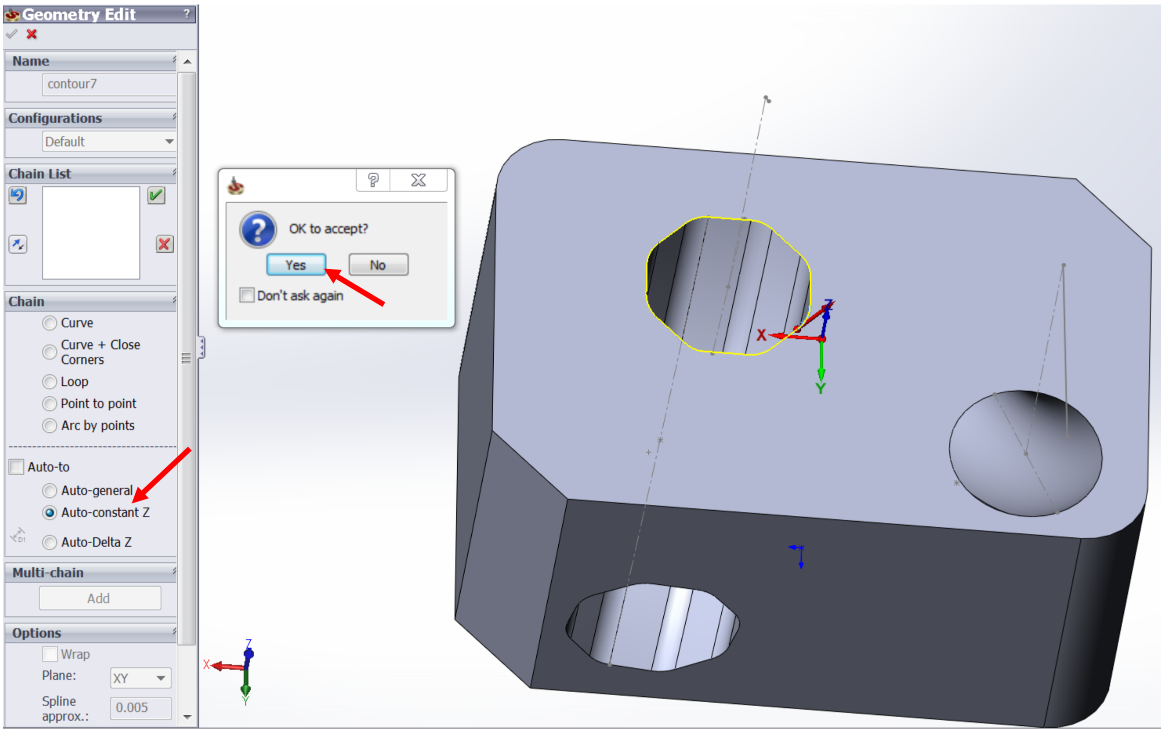
Click the new icon under **Start edge curve**.



Select the curve as shown in the image.



Select the **Auto-constant Z** radio button in the **Auto-to** section, so that the entire chain on the Z level is picked up.

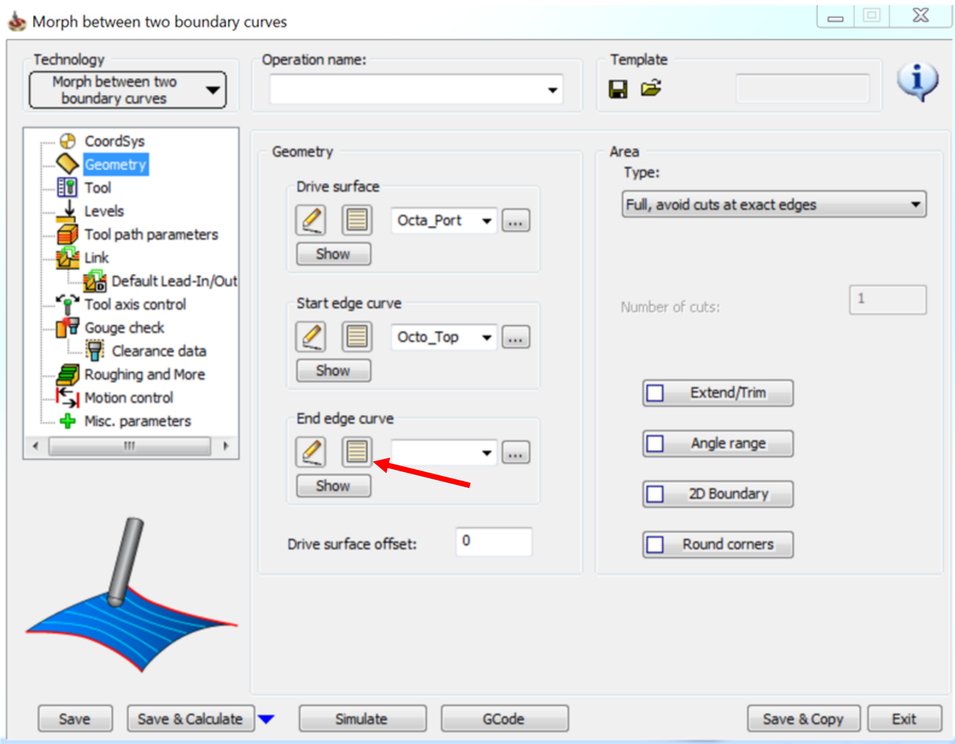


Click **Yes** in the **OK to accept** pop-up window.

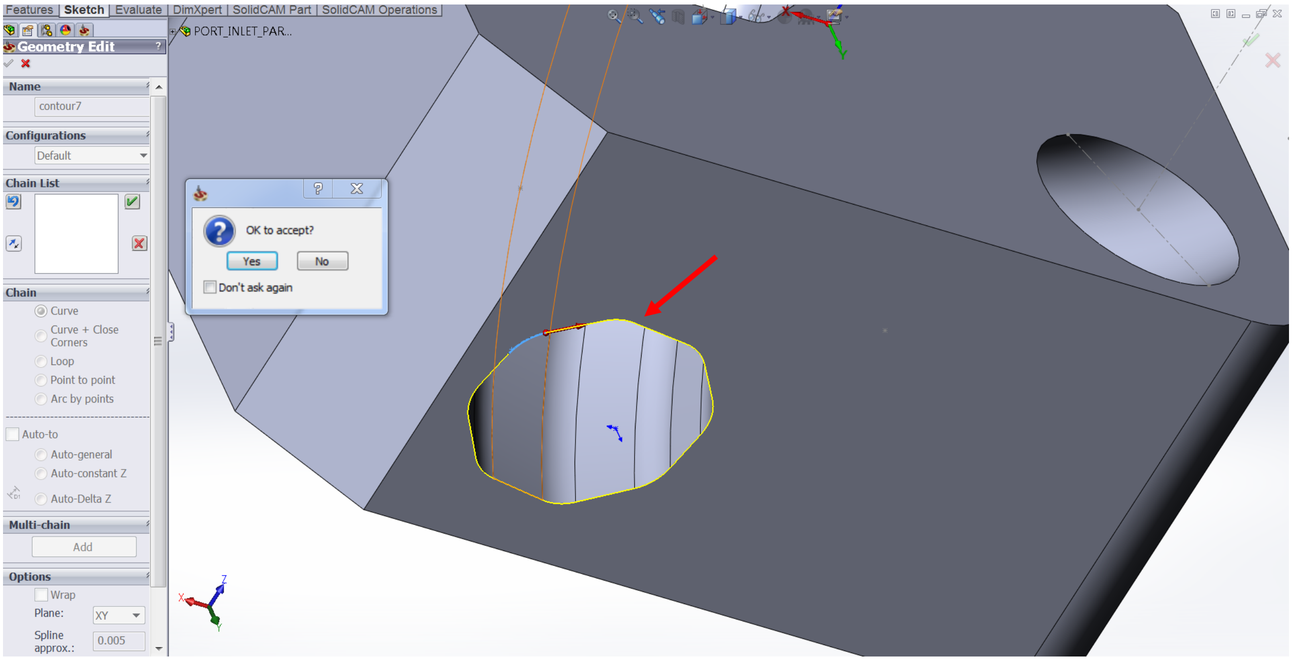
Change the Name as Octo_Top.

Click

Click the new icon under End edge curve.



Select all the edges one by one as shown in the image.

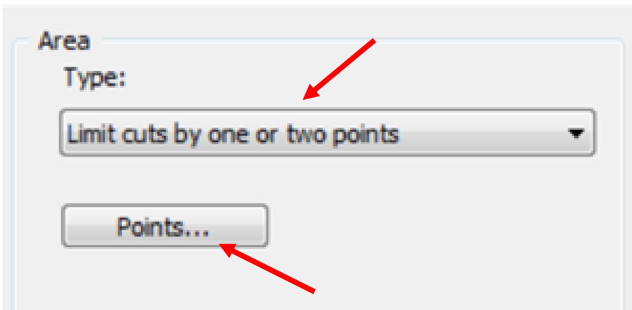


Click **Yes** in the **OK to accept** pop-up window.

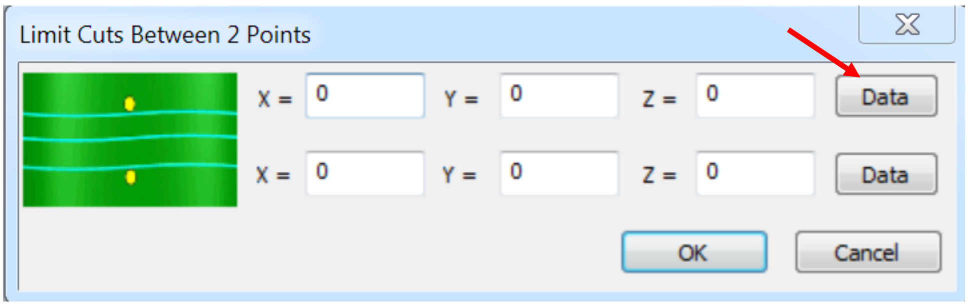
Click ✔

Select the **Type** as **Limit cuts by one or two points** from the drop down list in the **Area** section.

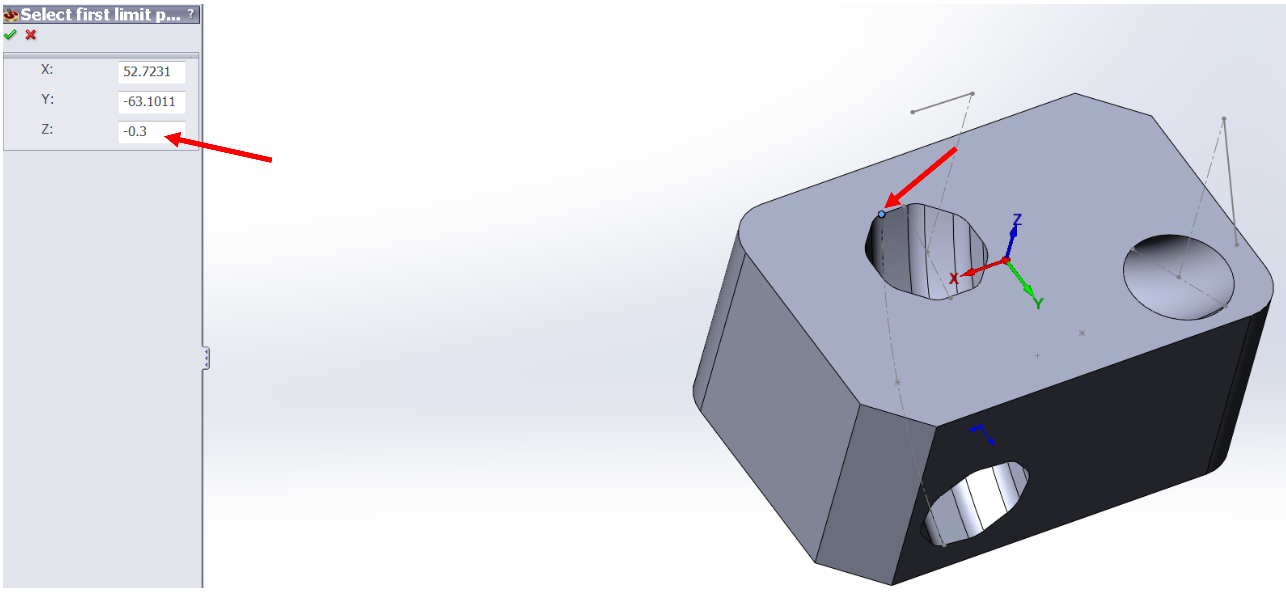
Click **Points**.



Click **Data**.



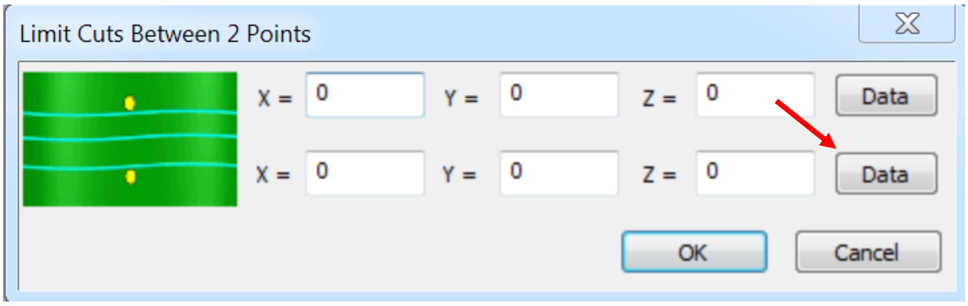
Select the point as shown in the image.



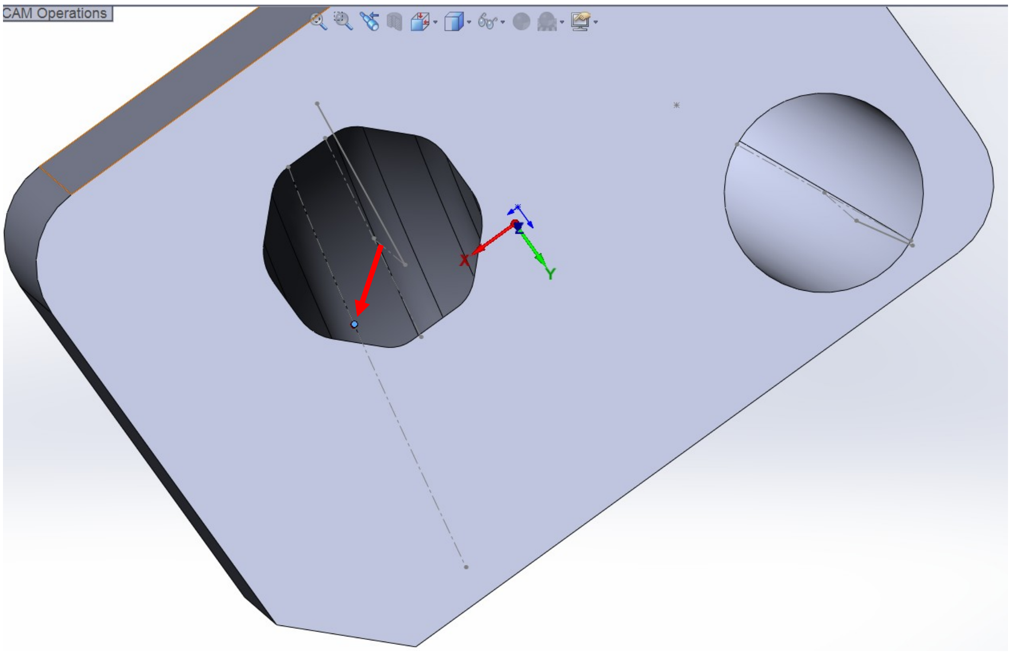
Enter a value of -0.3 in the **Z** field.

Click

Click **Data**.

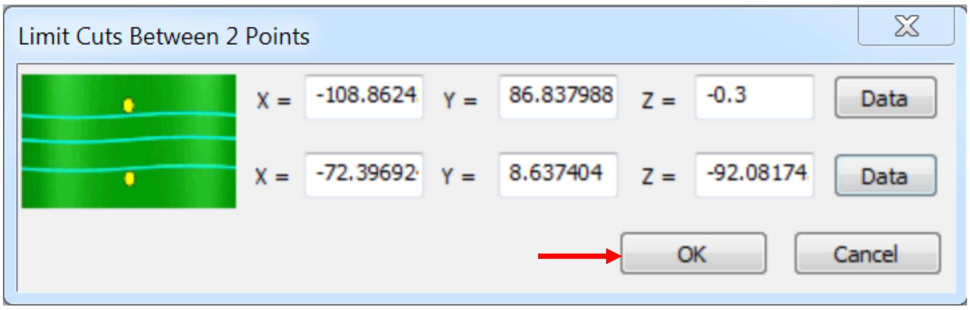


Select the point as shown in the image.



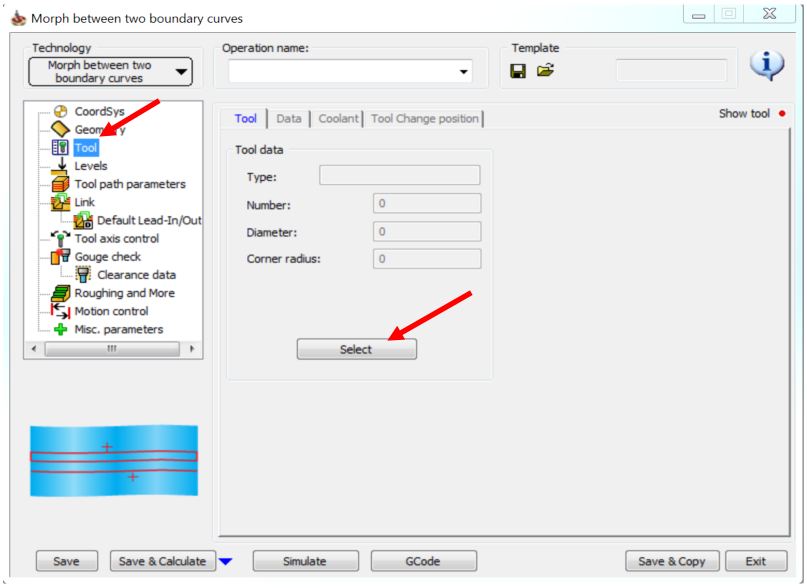
Click 

Click **OK** in the Limit Cuts Between 2 Points window.

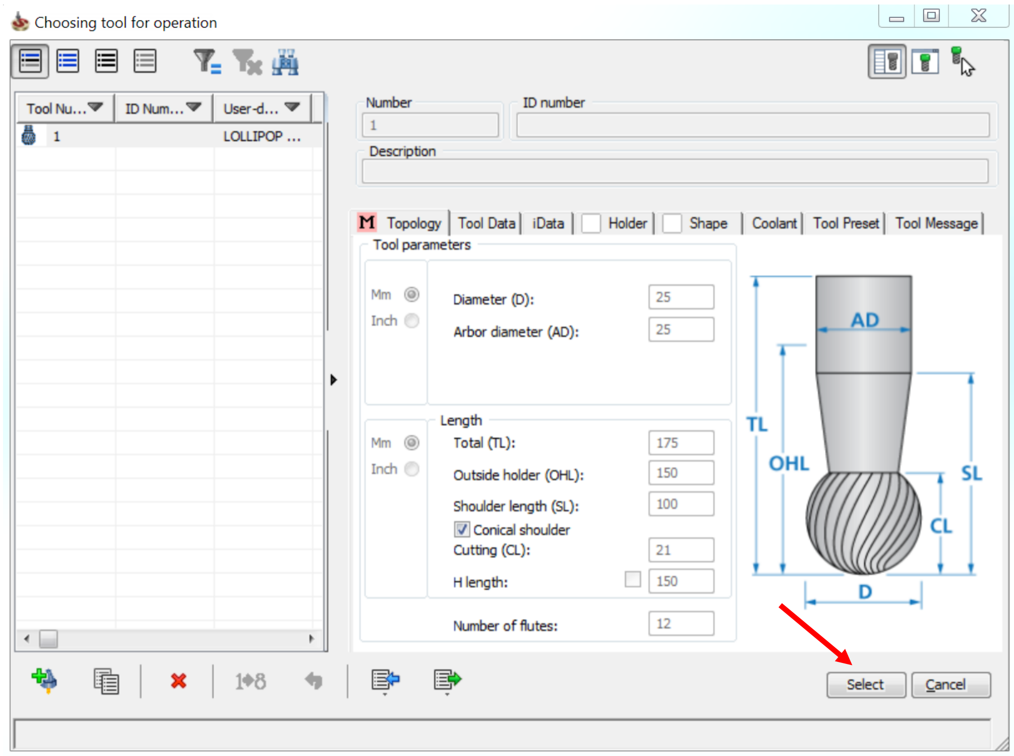


Click **Tool**.

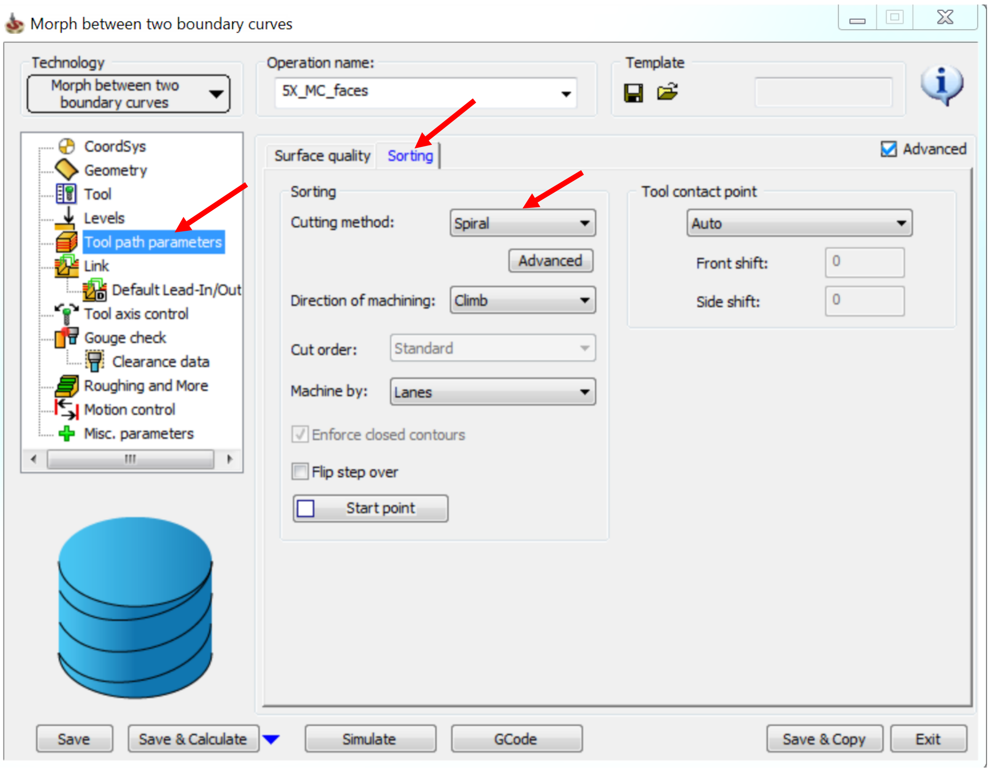
Click **Select**.



Click **Select** as we will use the same tool.



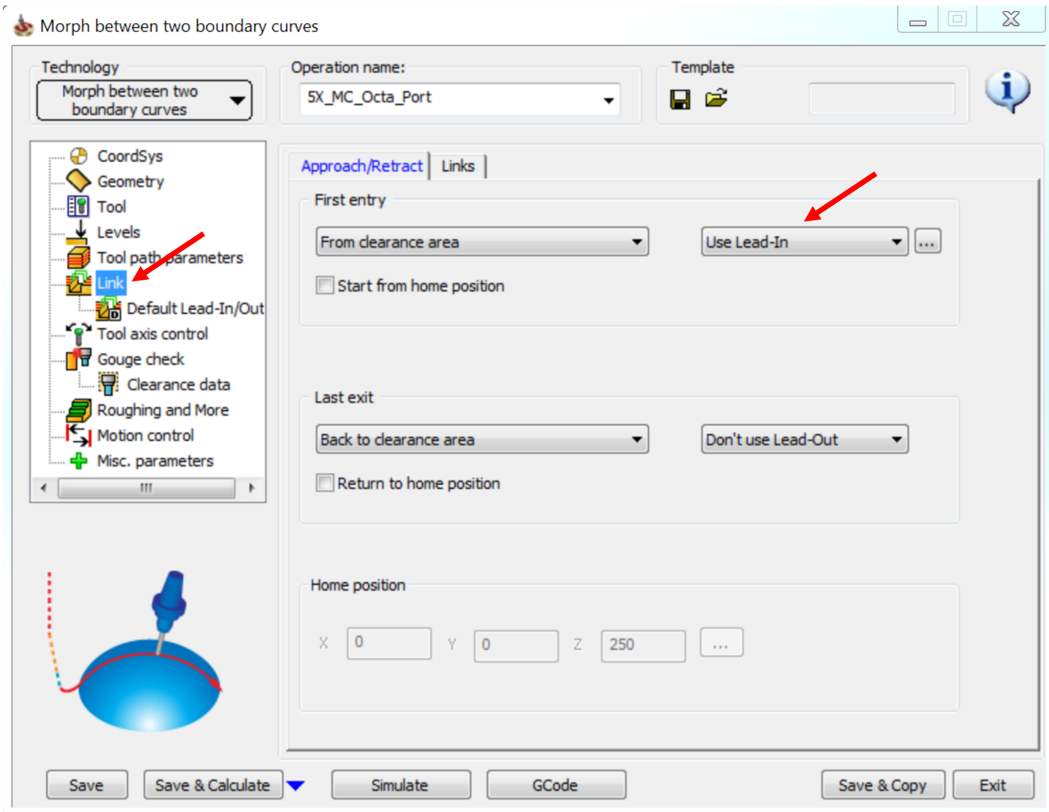
Click **Tool path parameters** > **Sorting**.



Select **Spiral** as the **Cutting method**.

Click **Link**.

Select **Use Lead-In** in the **First entry** section.

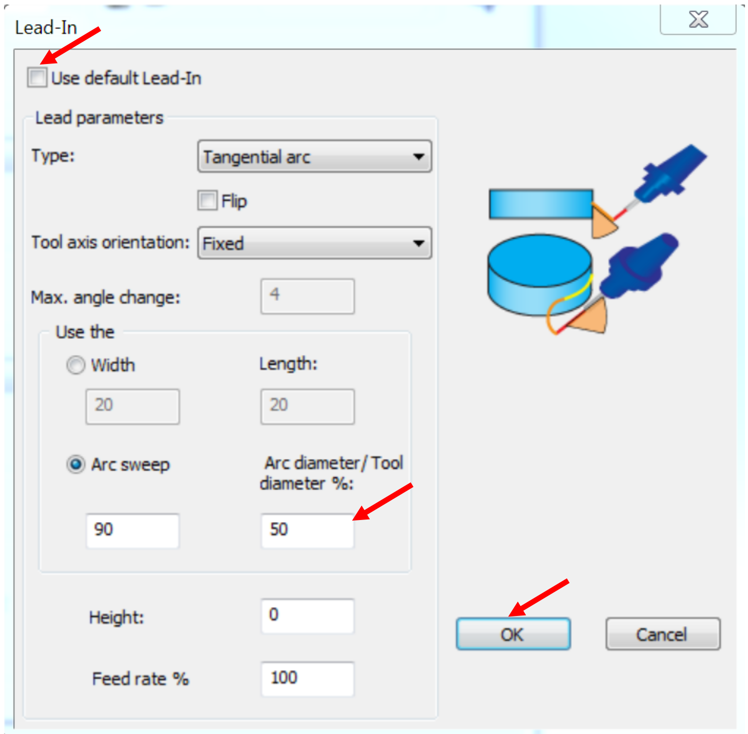


Click ...

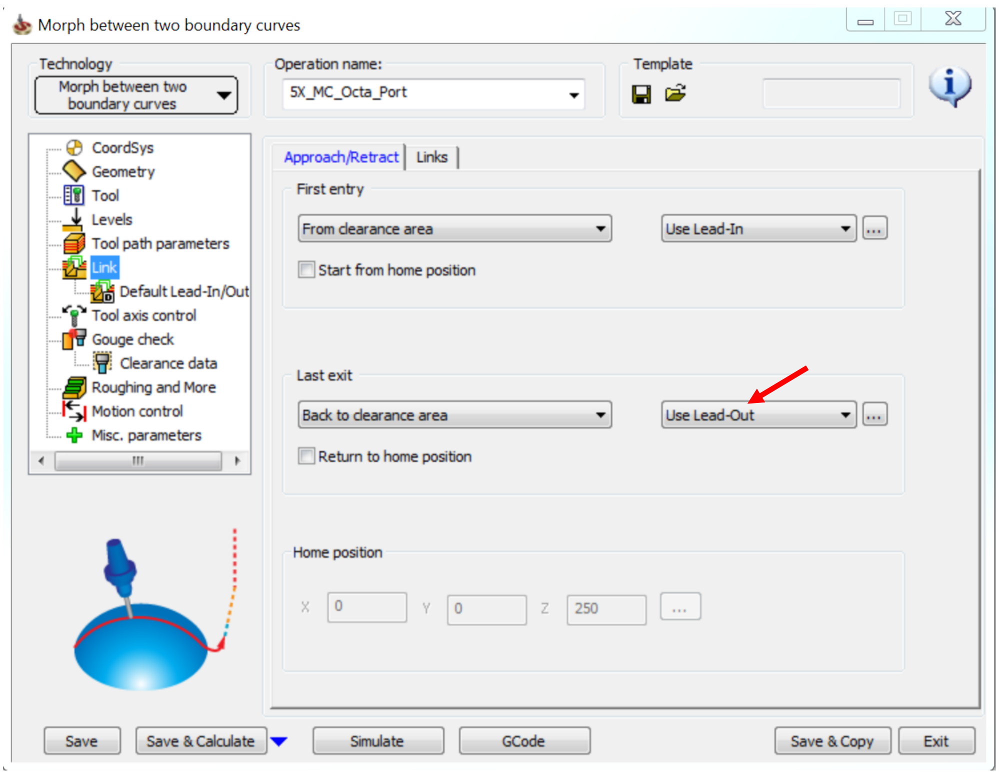
Clear the **Use default Lead-In** check box.

Enter a value of 50 in the **Arc diameter/Tool diameter %** field.

Click **OK**.



Select **Use Lead-Out** in the **Last exit** section.

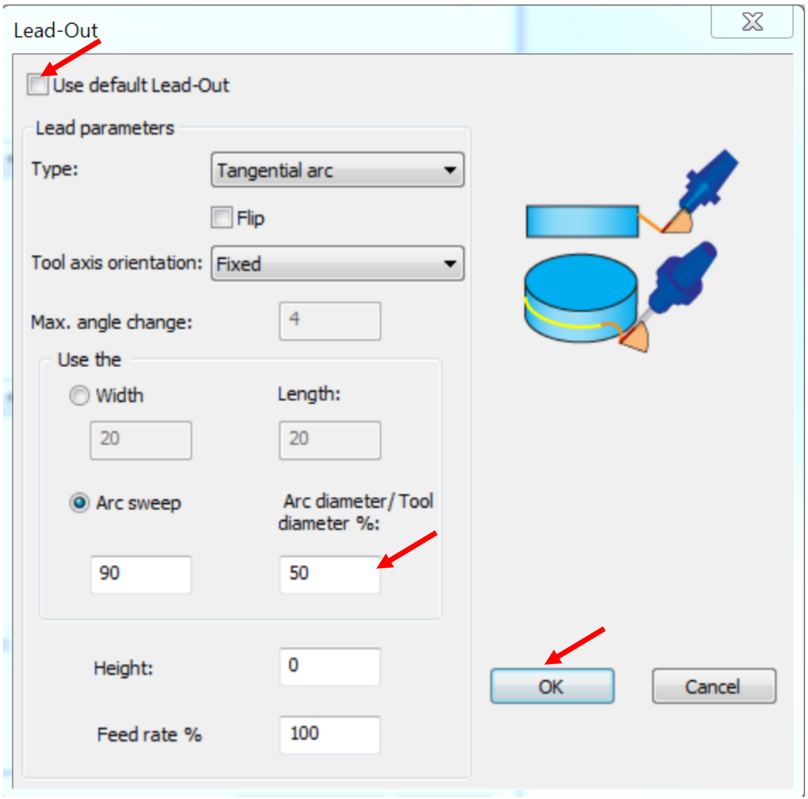


Click ...

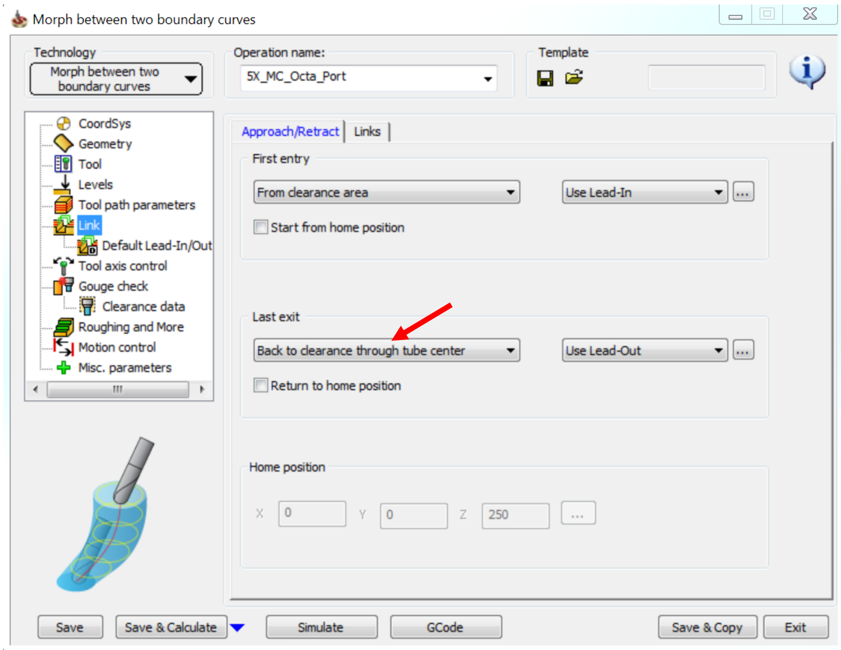
Clear the **Use default Lead-Out** check box.

Enter a value of 50 in the **Arc diameter/Tool diameter %** field.

Click **OK**.



Select **Back to clearance through tube center** in the **Last exit** section.

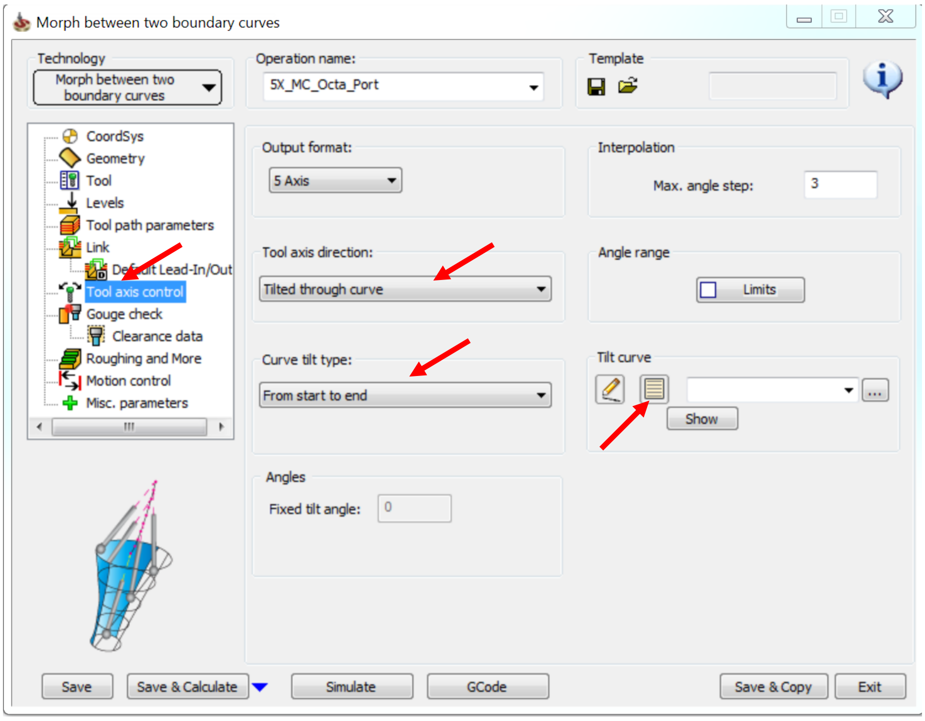


Click **Tool axis control**.

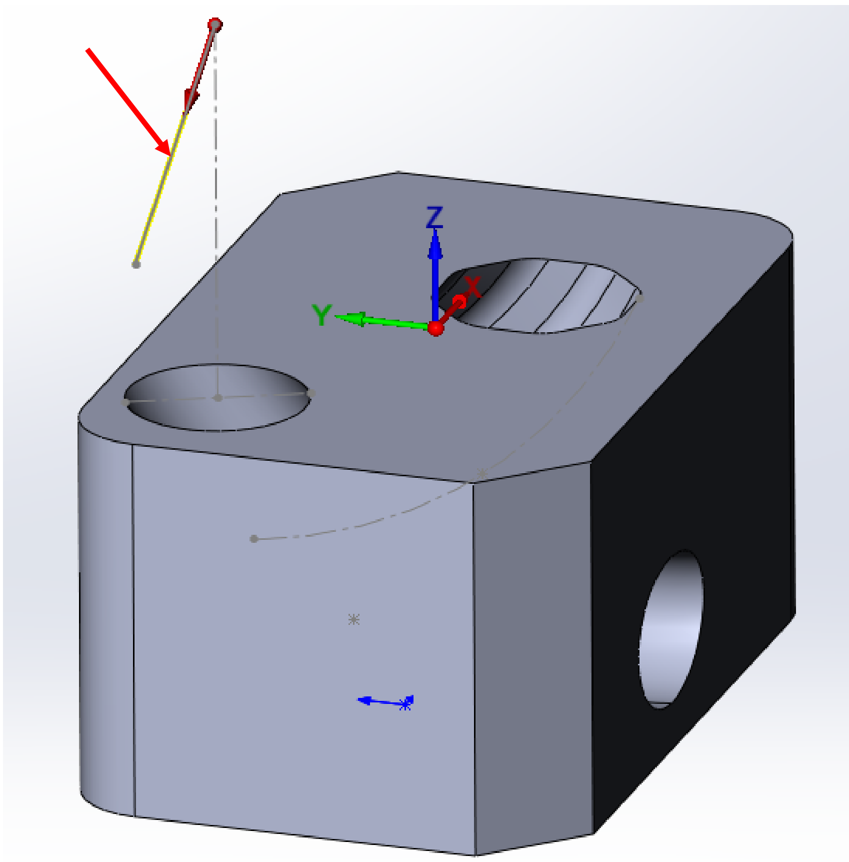
Select **Tilted through curve** as the **Tool axis direction**.

Select **From start to end** as the **Curve tilt type**.

Select the new icon under **Tilt curve**.



Select the curve as shown in the image.



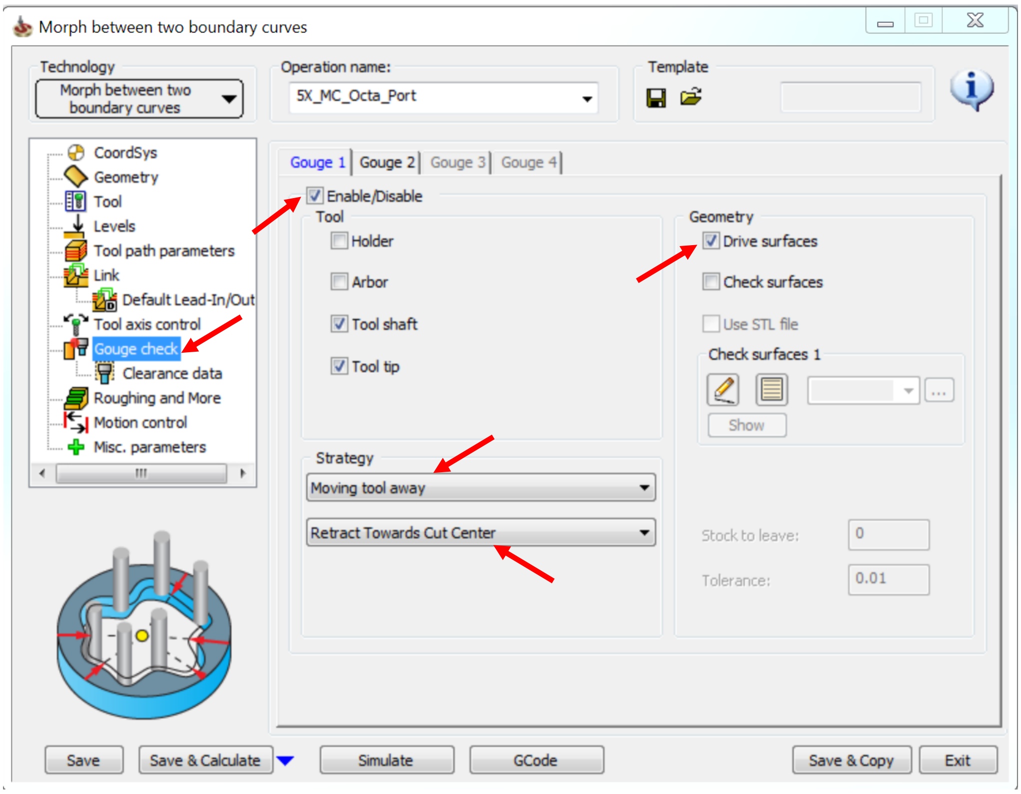
Click

Click

Click **Gouge check**.

Select the **Enable/Disable** check box.

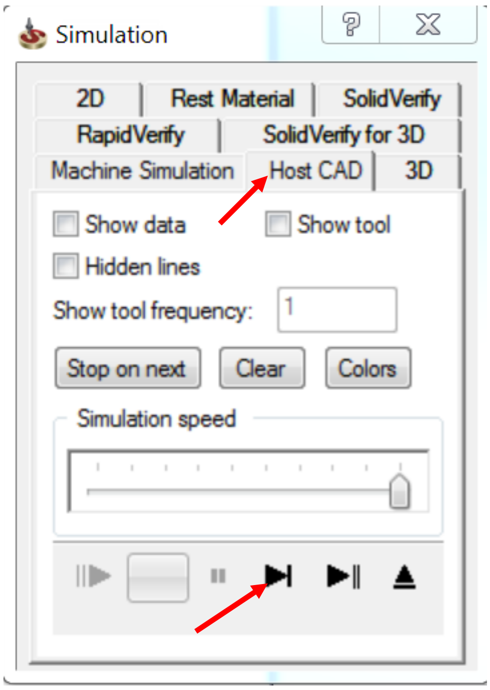
Ensure that only the Drive surfaces check box is selected under the geometry section.



Click **Save & Calculate**.

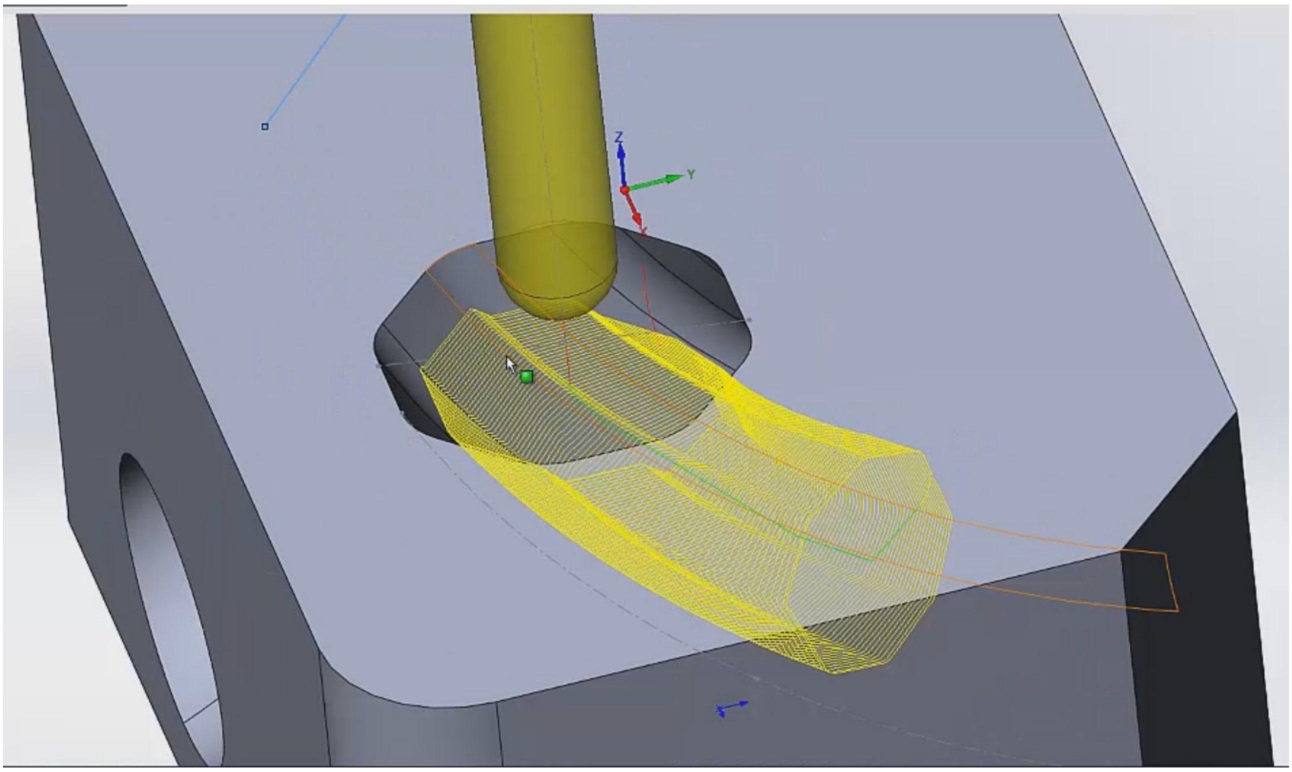
Click **Simulate** once the tool path is calculated.

Click **Host CAD**.



Click the play icon.

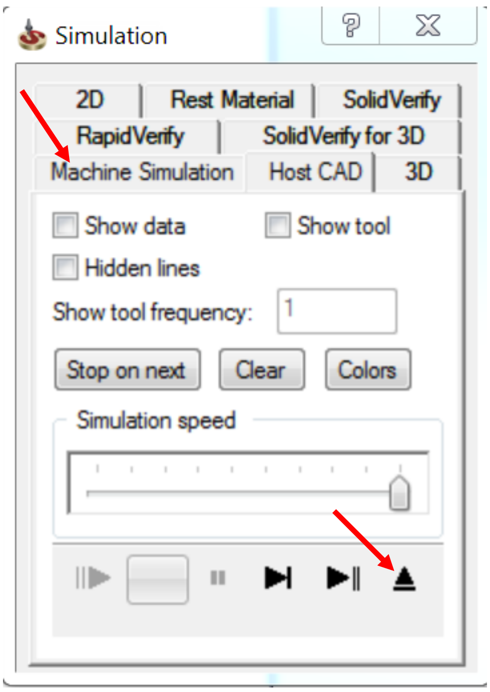
The simulated tool path looks like this:



You can see that the tool is trying to use the geometry of the surfaces and create a tool path. The tool is going up and down at a lot of places resulting in an uneven tool path. We need to fix this issue.

Click the exit icon.

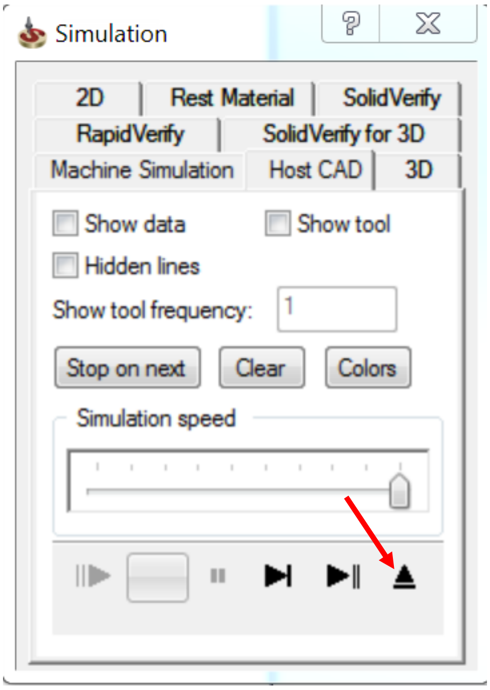
Click Simulate once again in the Operations Manager.



Click **Machine Simulation**.

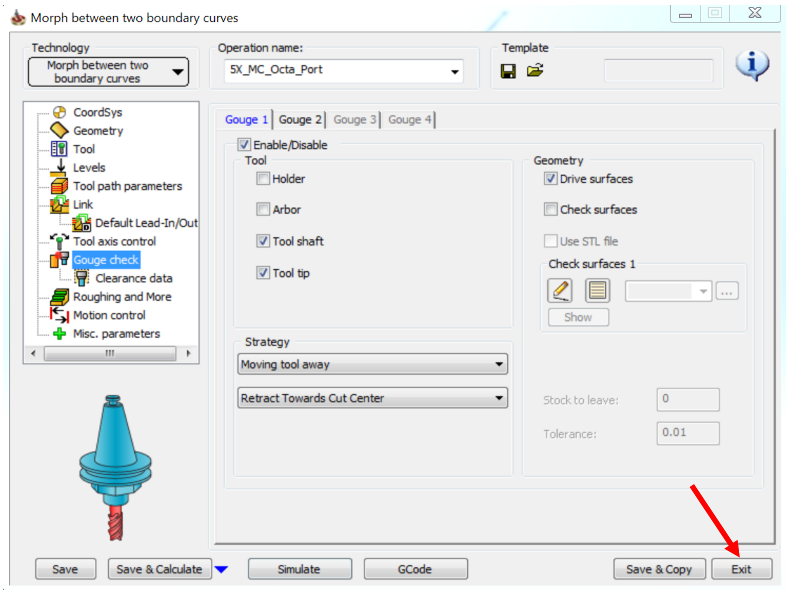
The simulated tool path looks like this:

There are no gouges but the tool path does not look very smooth. Let's Fix this.

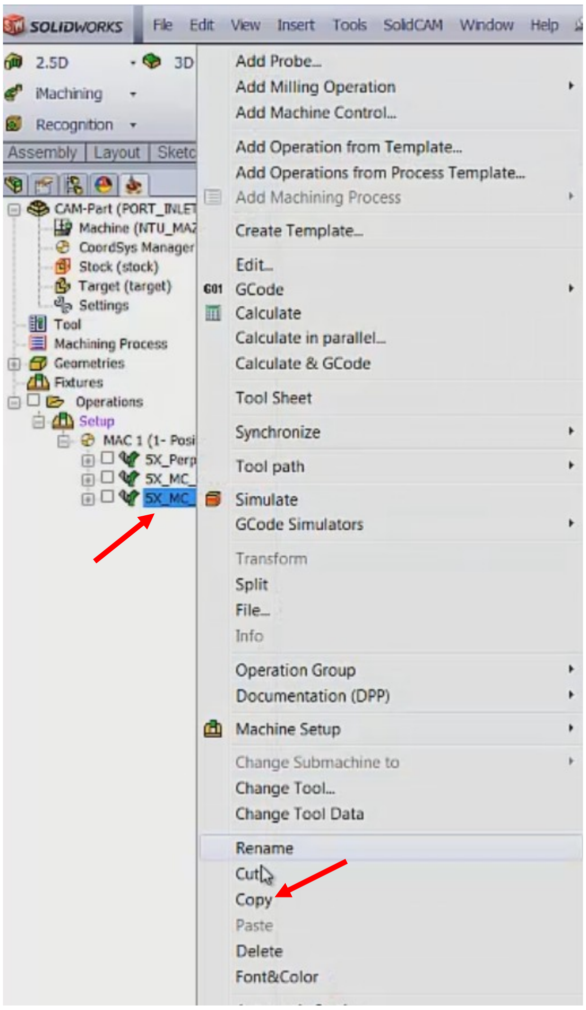


Click the exit icon.

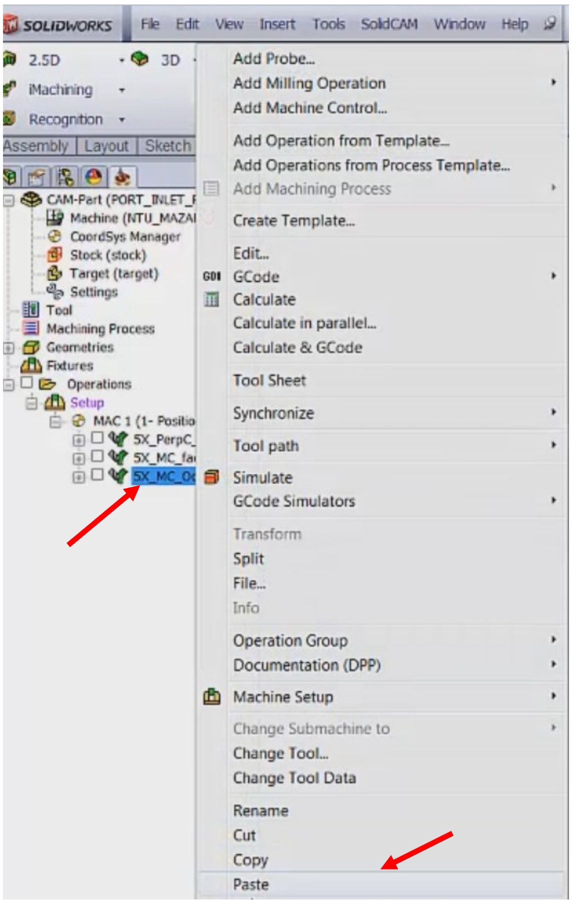
Click **Exit** to exit the Operations Manager.



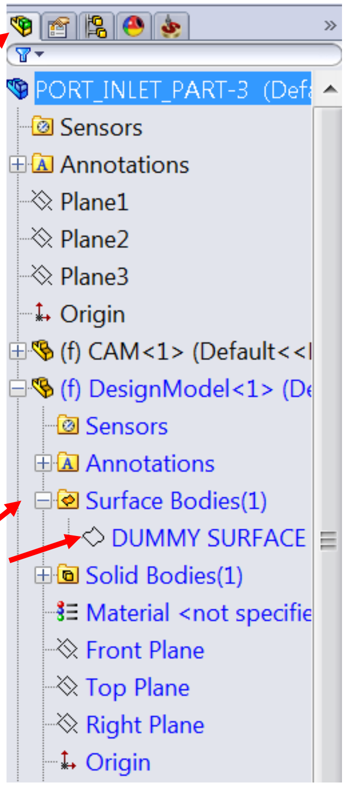
Right click > **Copy** the tool path.



Right click > **Paste** the tool path.



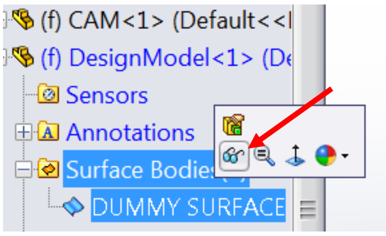
Select the Feature Manager Design Tree icon.



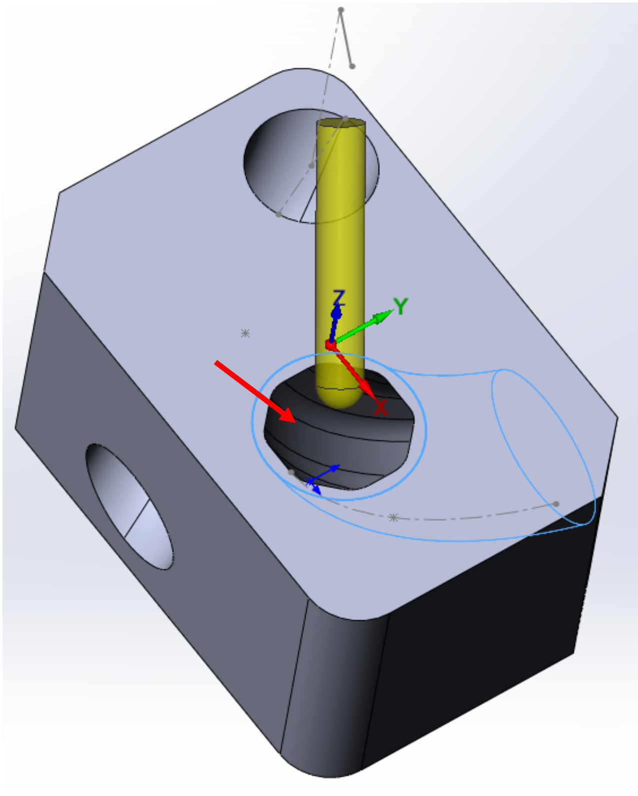
Expand **Surface Bodies**.

Select **DUMMY SURFACE**.

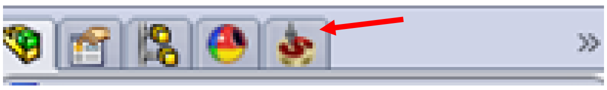
Click the Show icon.



We can see that the dummy surface is a circle driven along the curve. It uses the same guide curve that was used to create the octagonal surface.



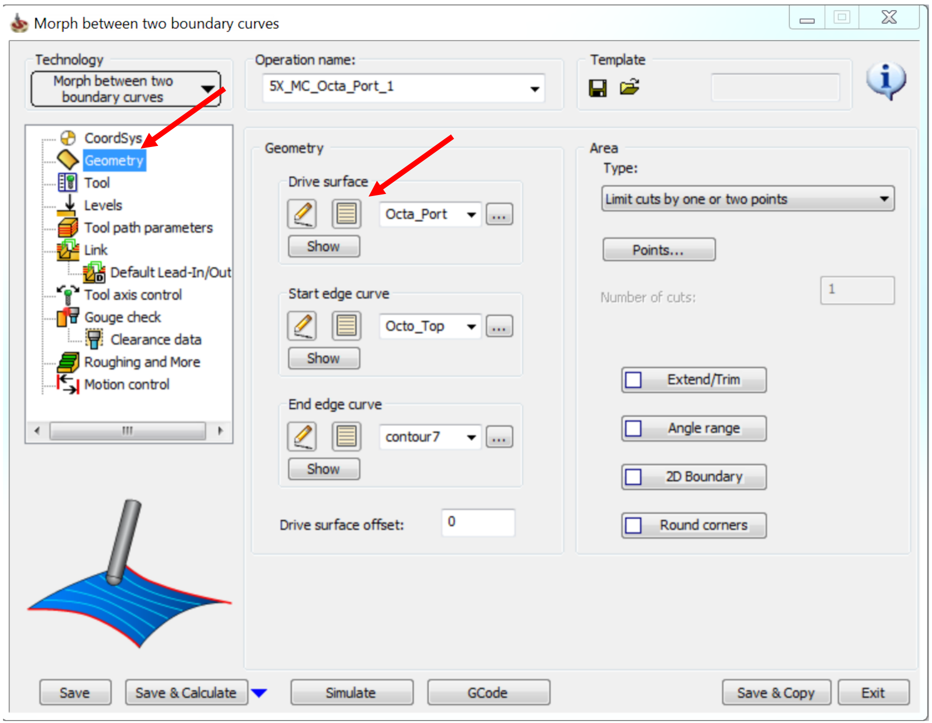
Click the SolidCAM Manager icon.



Right click > **Edit** the tool path.

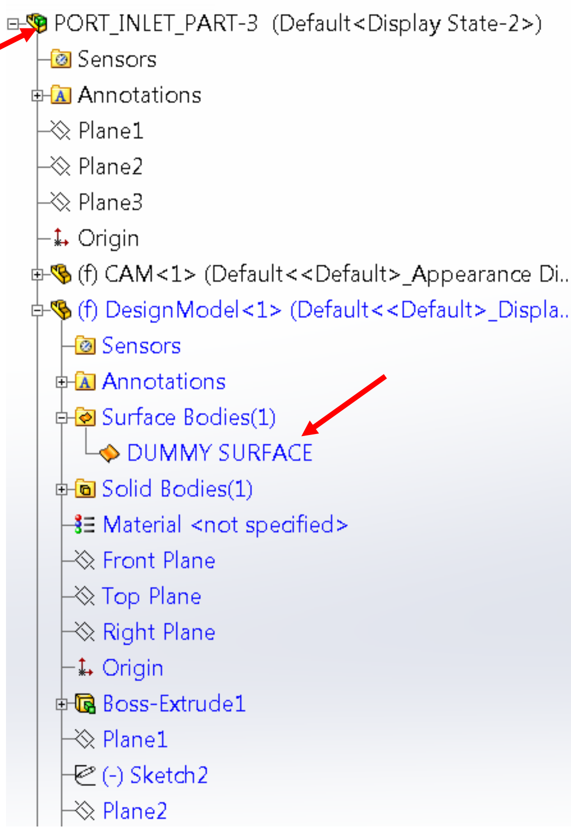
Click **Geometry**.

Click the new icon under **Drive surface**.



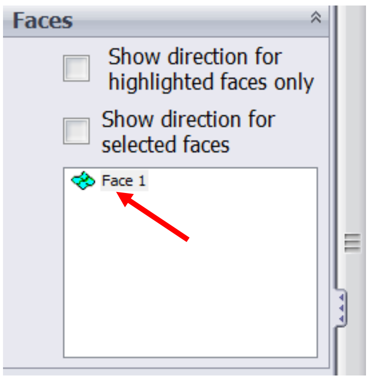
We will use the cylinder. As the cylinder is hidden inside the solid model, it is difficult for us to select the geometry. We will therefore select it from the design model.

Expand the feature manager.



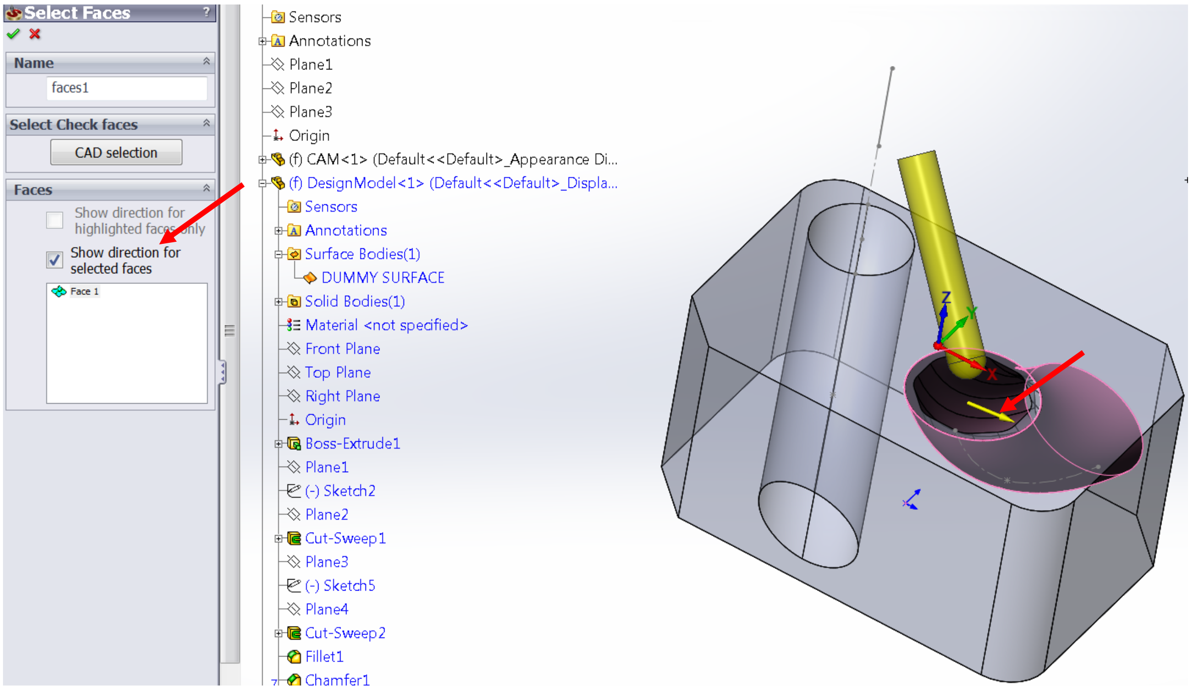
Click **DUMMY SURFACE**.

You can see that it is already selected.



We must ensure that the direction of the surface is proper because we need to machine the inside of the tube.

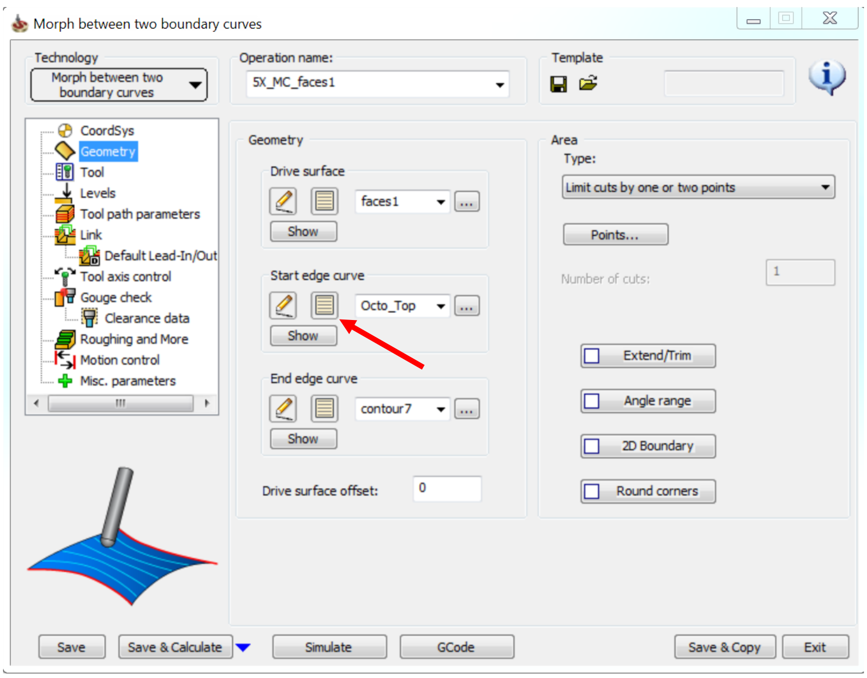
Click **Show direction for selected faces**.



You can see that the arrow is indicating the correct direction.

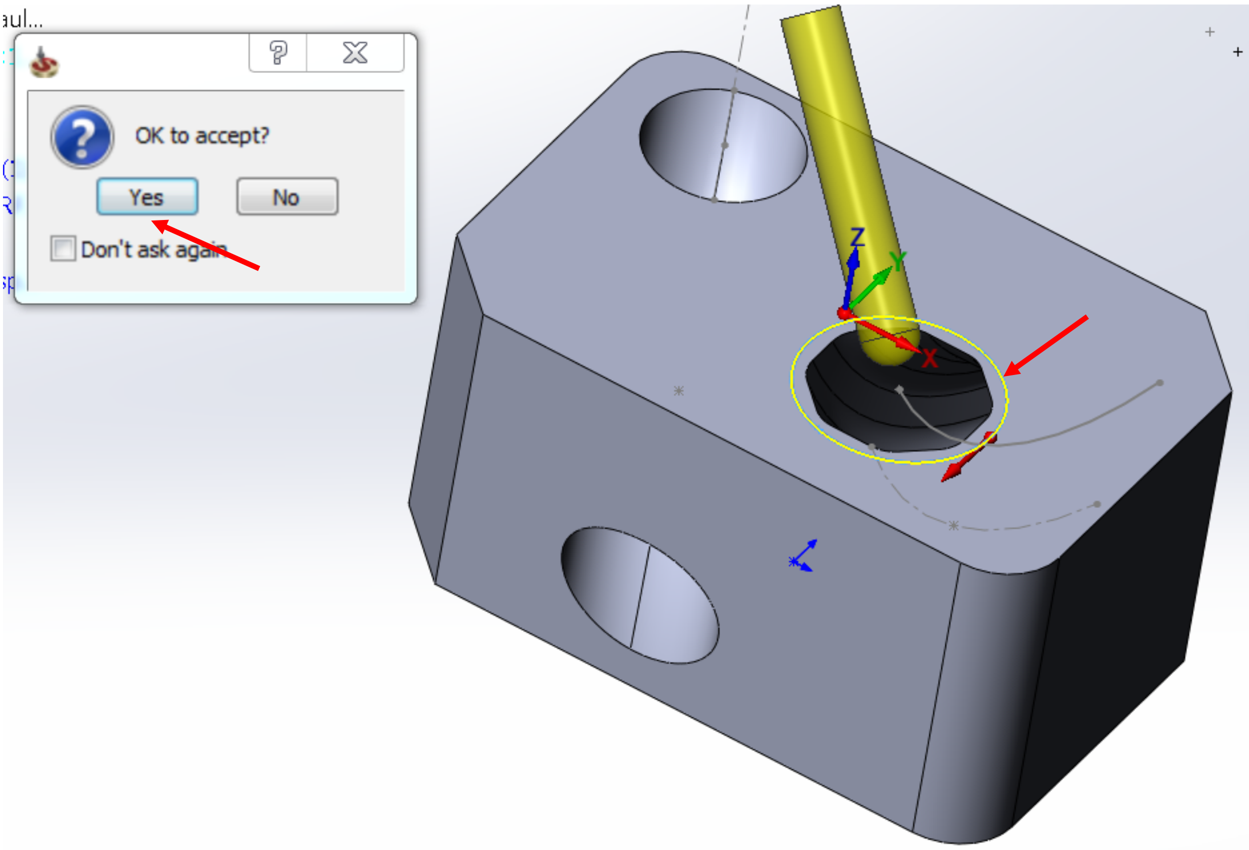
Click 

Click the new icon under the **Start edge curve**.



We will select the edge of the dummy surface as our curve.

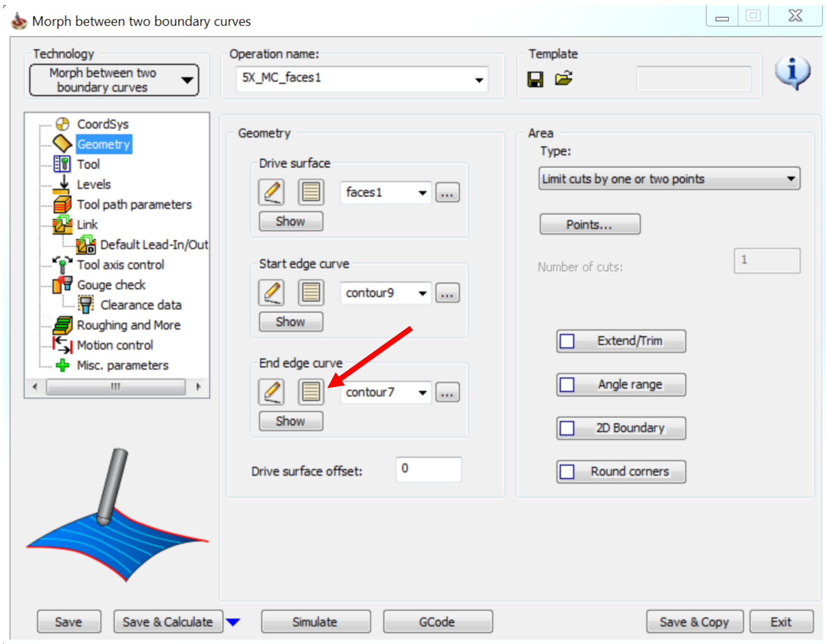
Select the surface as shown in the image.



Click **Yes** in the **OK to accept** pop up window.

Click

Click the new icon under the **End edge curve**.

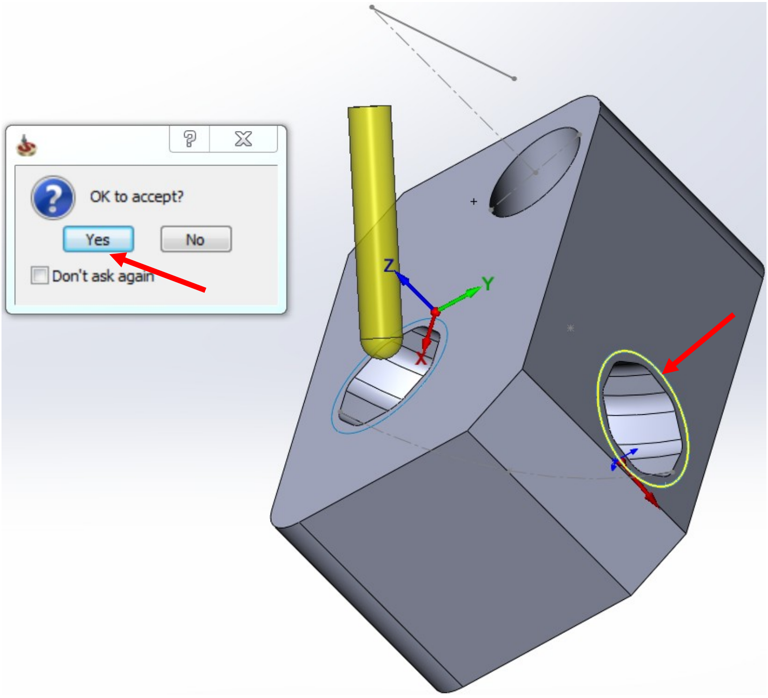


We will select the end edge of the dummy surface as our curve.

Select the surface as shown in the image.

Click **Yes** in the **OK to accept** pop up window.

Click ✓



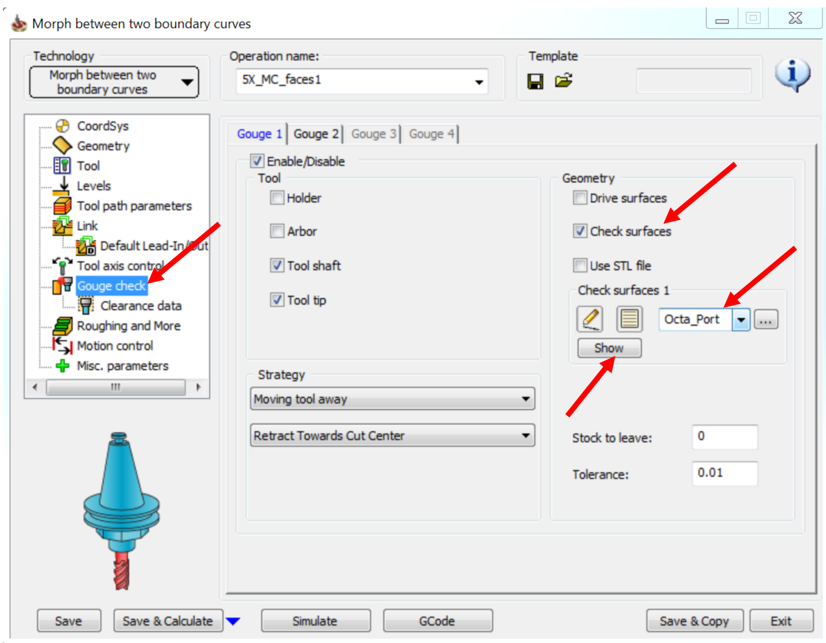
We have now finished selecting the surface to be machined. In this strategy, we have allowed the tool path to be generated in the tube, and now we will run the gouge check.

Click **Gouge check**.

Select the **Check surfaces** checkbox.

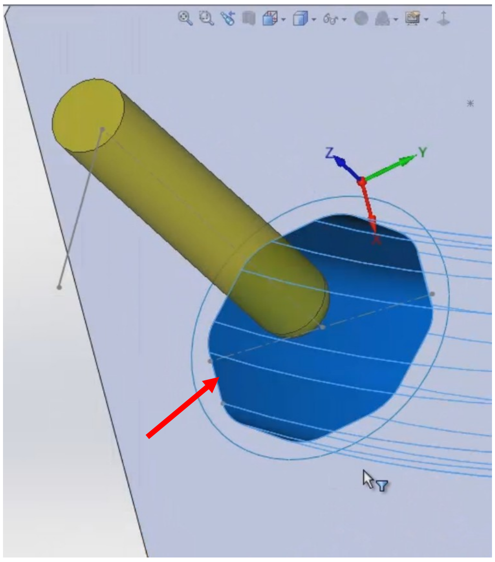
Select **Octa_Port** in the **Check surfaces 1** section.

Click **Show**.



The gouge check is done on the highlighted surfaces that are shown in the image.

Click

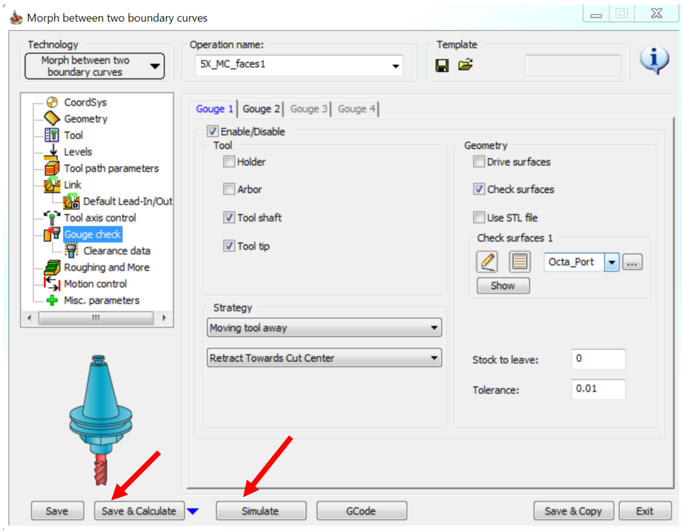


Once the tool finds the gouges, it will move the tool away till it retracts towards the cut center so that all the gouges disappear. This will result in a gouge free tool path.

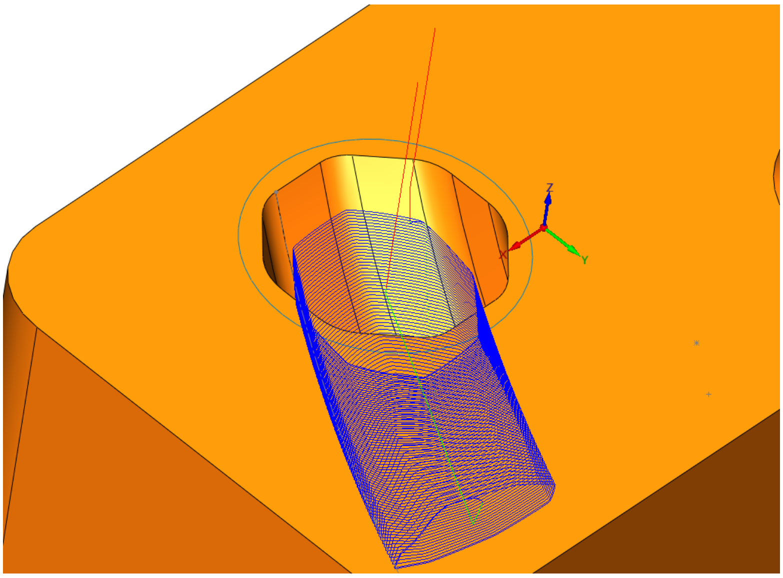
With this strategy, we are effectively projecting a tool path on these octo surfaces in such a way that no gouges remain.

Click **Save & Calculate**.

Click **Simulate** once the tool path is calculated.



The simulated tool path looks like this:

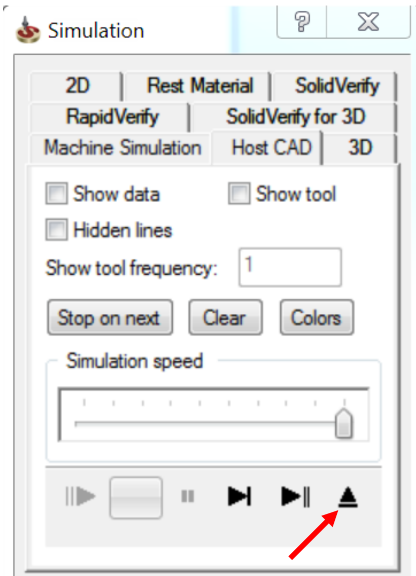


The tool path is looking much better as it has taken the circular tool path and projected it on these octagonal surfaces.

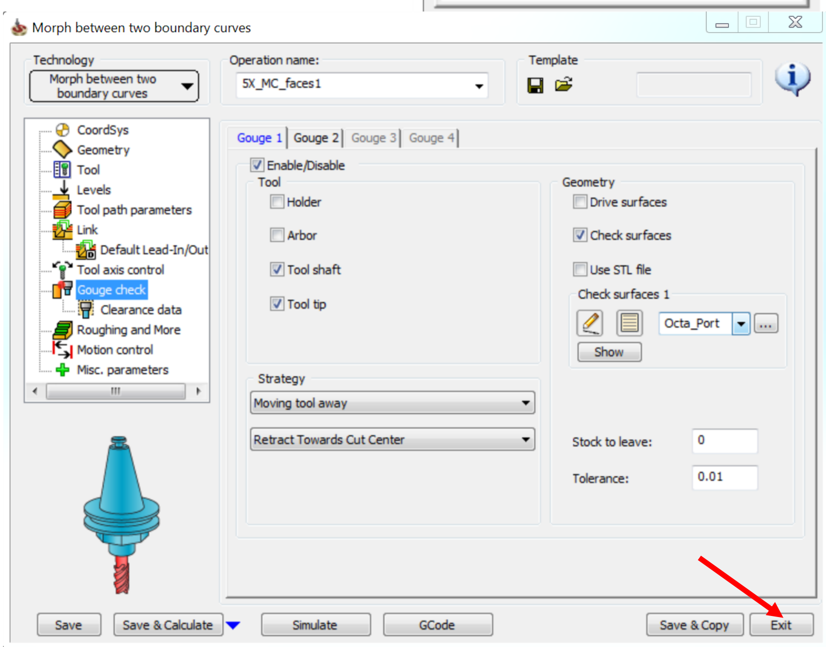
Run the machine simulation to check if there are any gouges.

.

Click the exit icon to exit machine simulation.

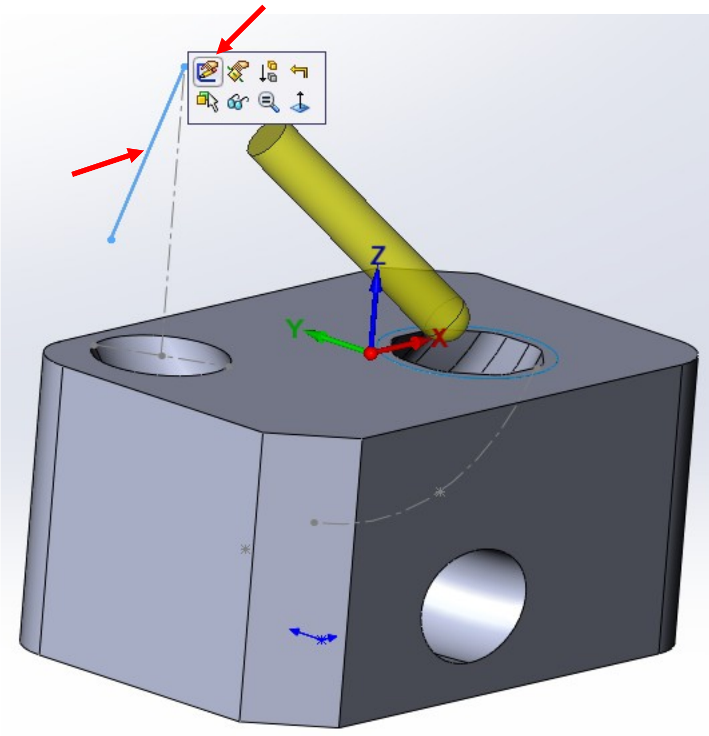


Click **Exit**.



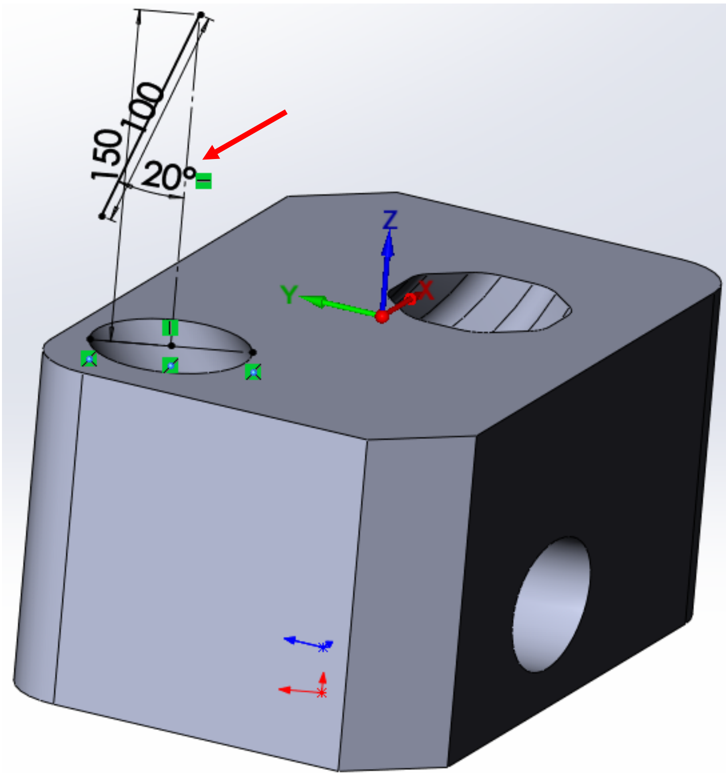
Select the curve.

Click the Edit Sketch icon.



Change the angle from 35 to 20 degrees as shown in the image.

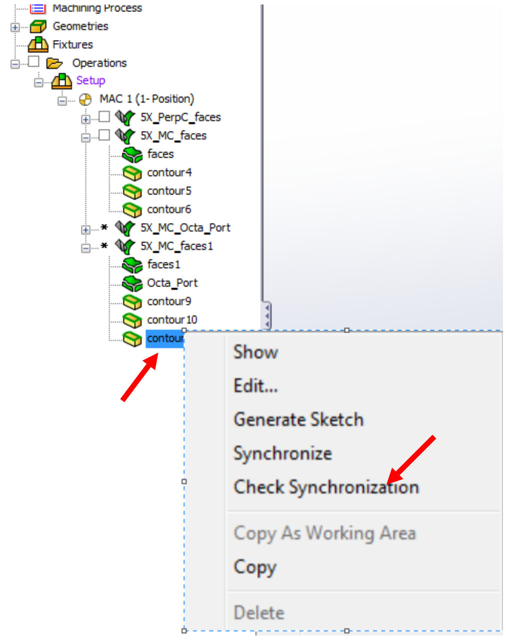
Press enter on your keyboard.



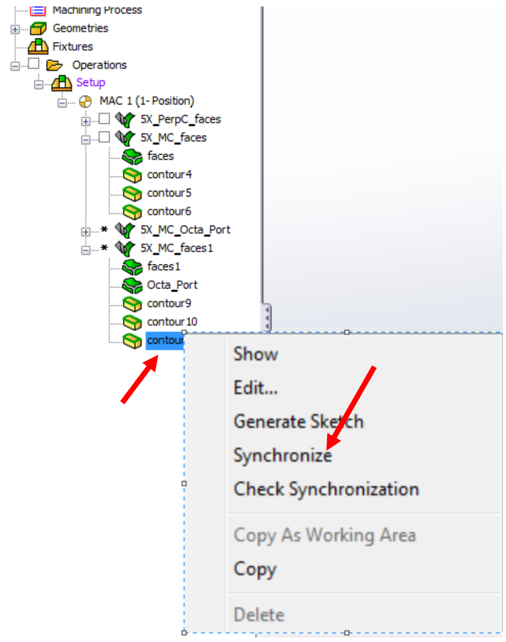
Click the exit sketch icon.

Expand the tool path.

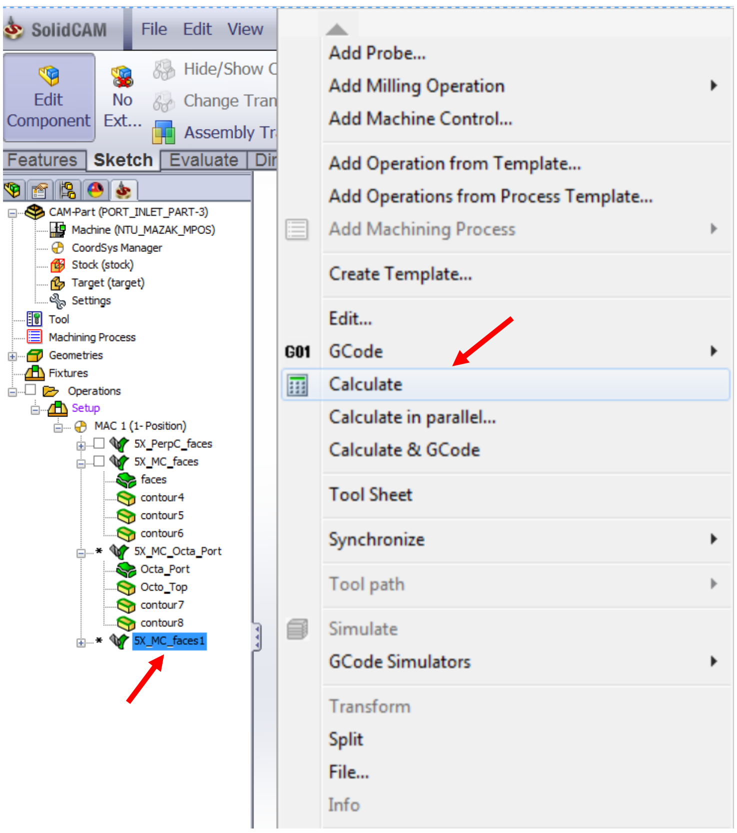
Right click **contour 11** > **Check Synchronization**.



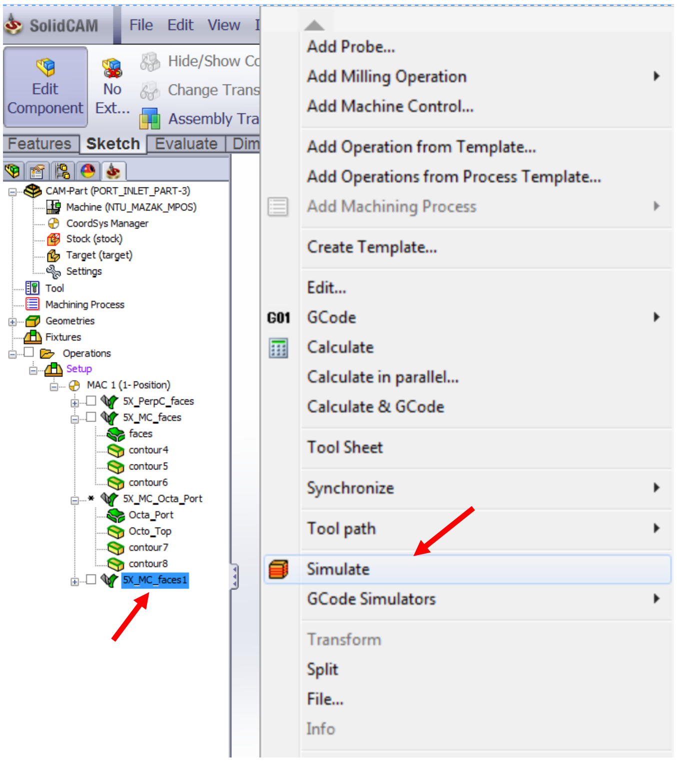
Right click **contour 11** > **Synchronize**.



Right click > **Calculate** the tool path.



Right click > **Simulate** the tool path.



Run the machine simulation.

The simulated tool path looks like this:

You can see that the gouges have disappeared.

The guiding curve plays a very important role in determining the tilt of the tool into the geometry. We could machine a different surface which was much smoother and then project the tool path onto the surfaces inside. This results in a smoother tool path.

End of the volume