SolidCAM
5 Axis Tutorial

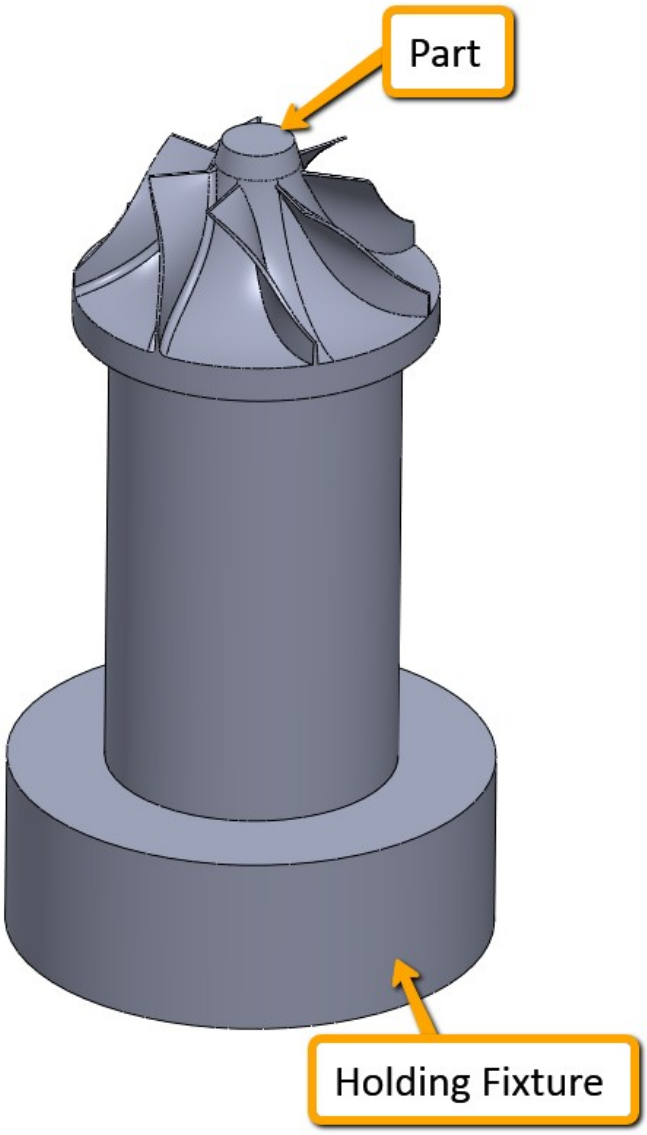
Volume 2 – Beginner

Goals of the Tutorial

- To understand the strategy of morph between two curves
- To understand the strategy of morph between two surfaces
- To understand the strategy of parallel to curve
- To understand different side tilt definitions

Files Used

Part File



IMPELLER

Post Processor

DMU-60T-MonoBlock

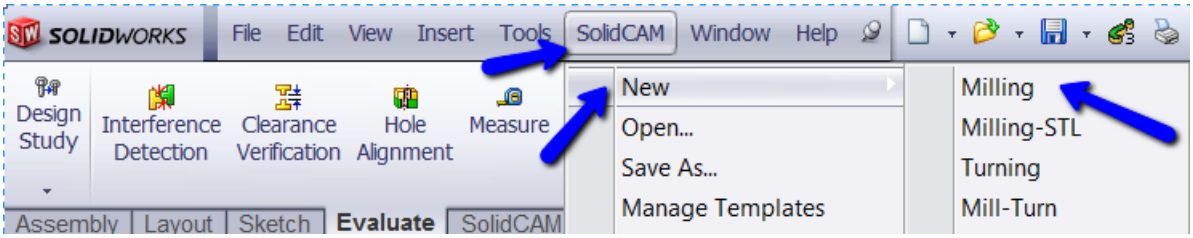
Machine Simulation

DMU60_NEW

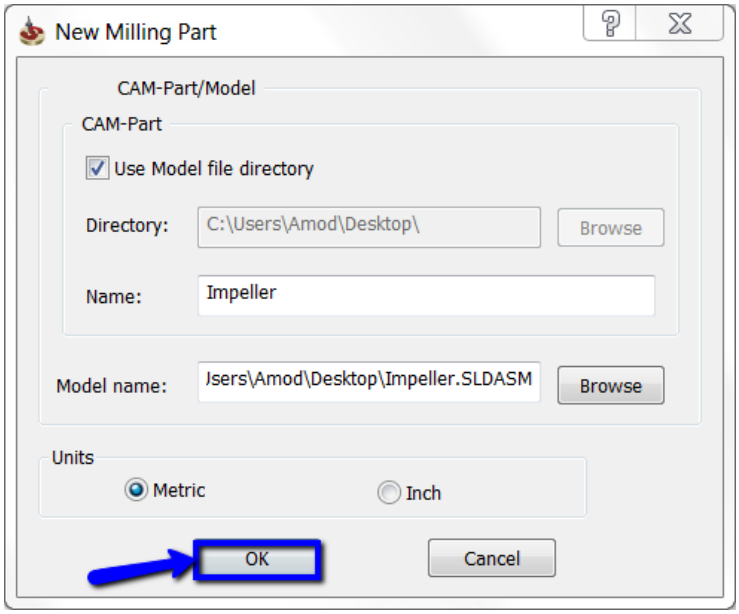
Double click the **SolidWorks** Icon.

Click **File** > **Open** to open the SolidWorks part Impeller.SLDASM.

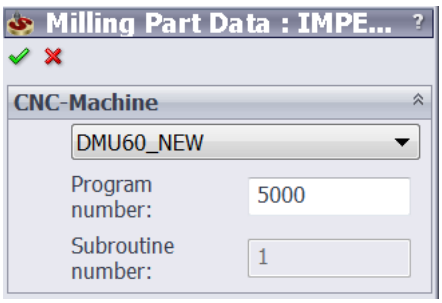
Click **SolidCAM** > **New** > **Milling**.



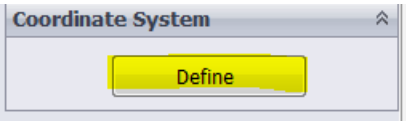
Accept the default part file name and path and Click **OK**.



Select DMU60_NEW as our post processor. Leave the fields of **Program Number** and **Subroutine number** with the default settings.

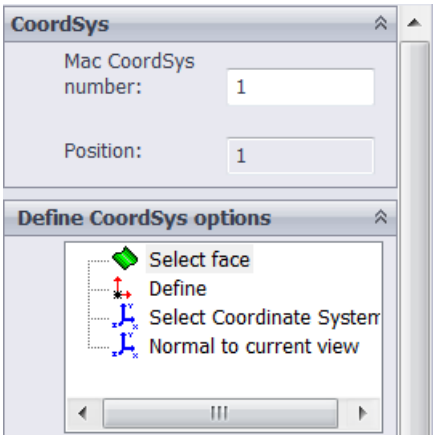


Click **Define** button in **Coordinate System** tab to set the coordinate system for this part.



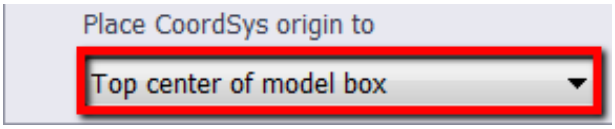
A dialog box with the following options appears:

- Mac CoordSys number
- Position
- Coordinate system definition options

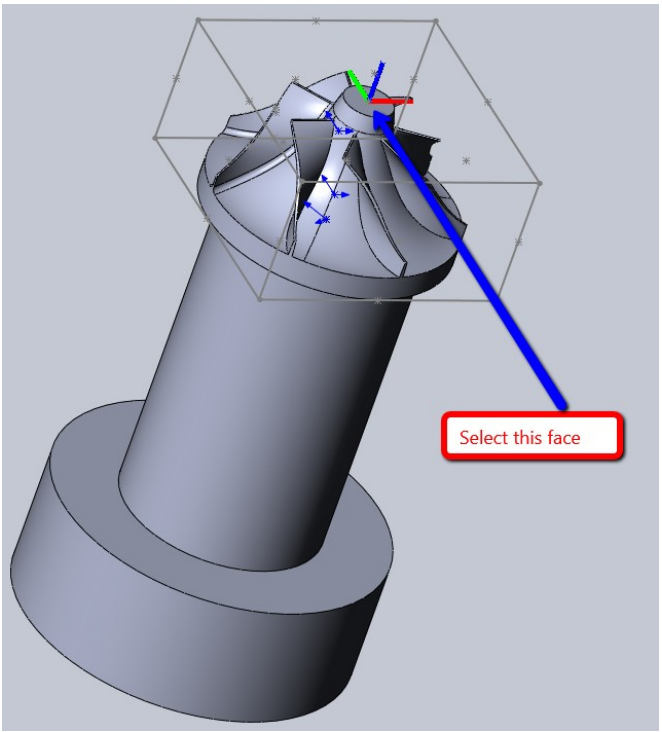


Click **Select Face** option in the **Define CoordSys options** section to define the main coordinate system.

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

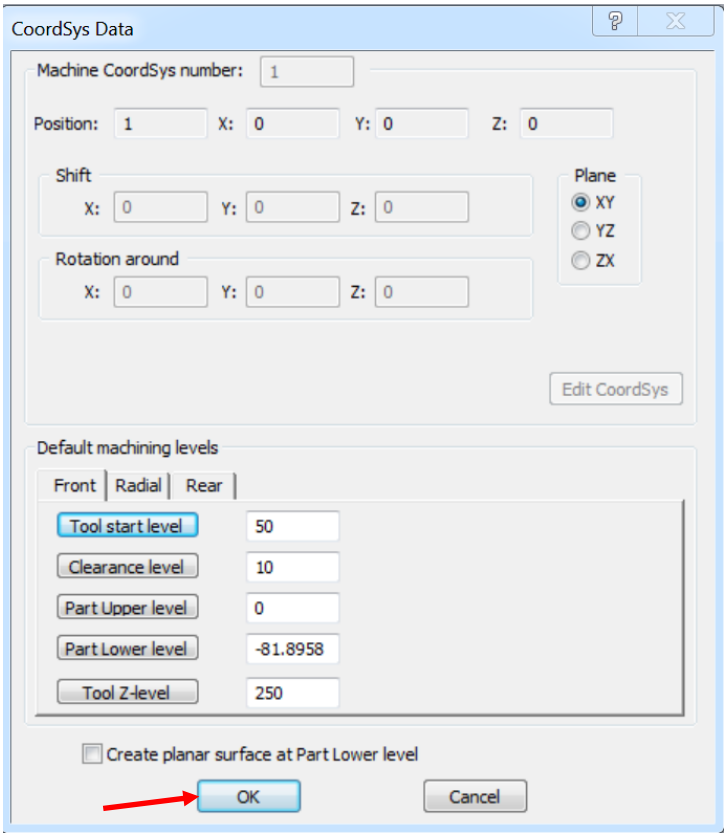


Select the face as shown in the below image.



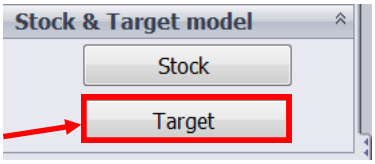
Click ✔ to accept.

Click **OK** to accept the default values in the **CoordSys Data** pop-up.

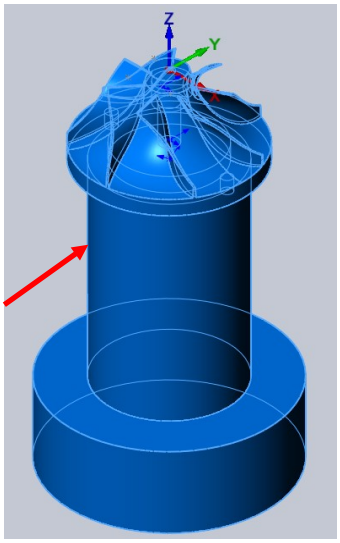


Click ✔ to return to the Milling Part Data.

Click **Target** in the Stock & Target model section.

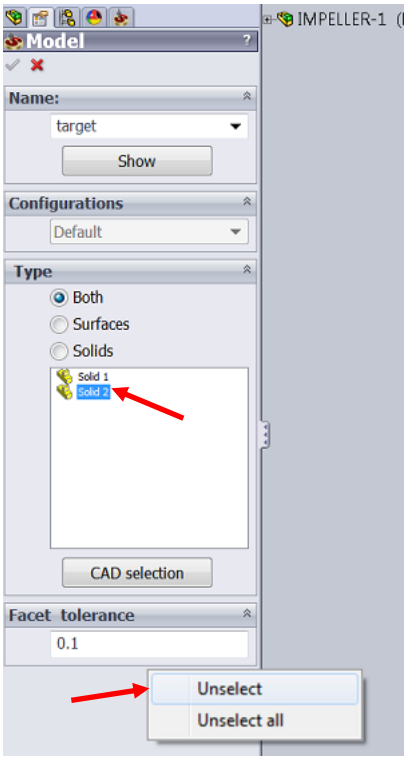


This image appears:

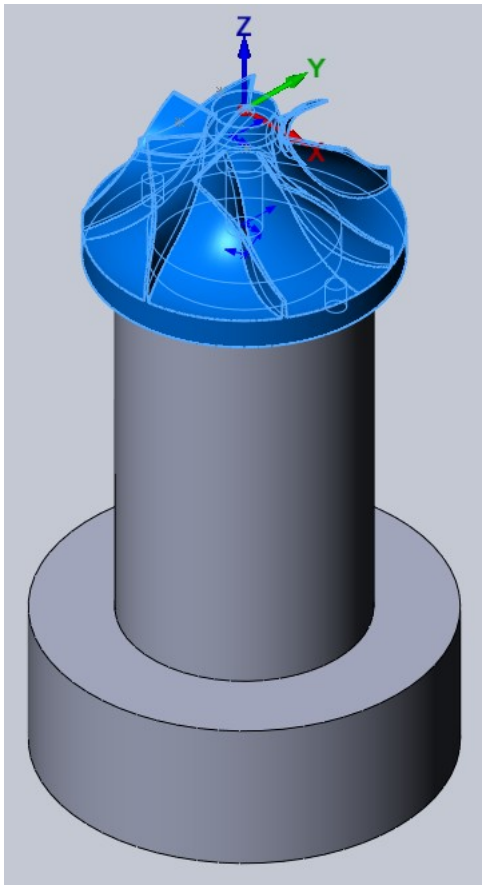


Presently, the target and the fixture both are selected as the target. We will Edit this to separate the definitions of Part & Fixture.

Right click Solid 2 and click **Unselect**, in the **Type** section.



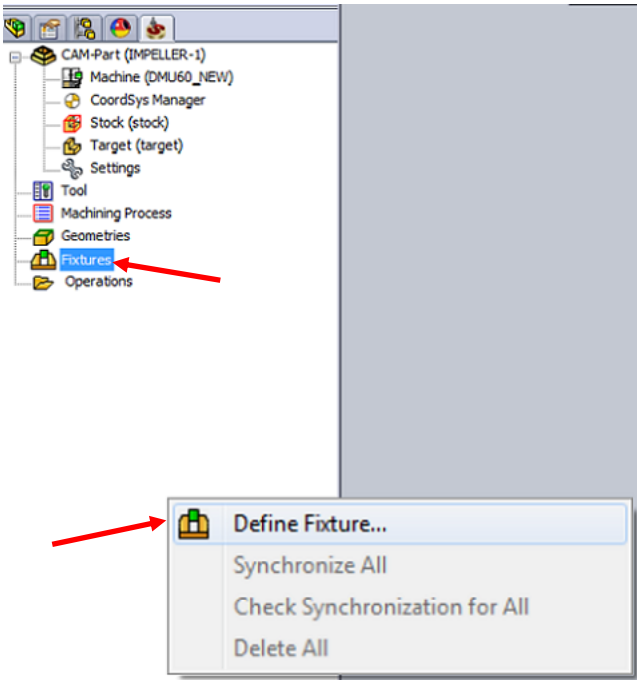
The image now looks like this as we have separated the fixture from the part.



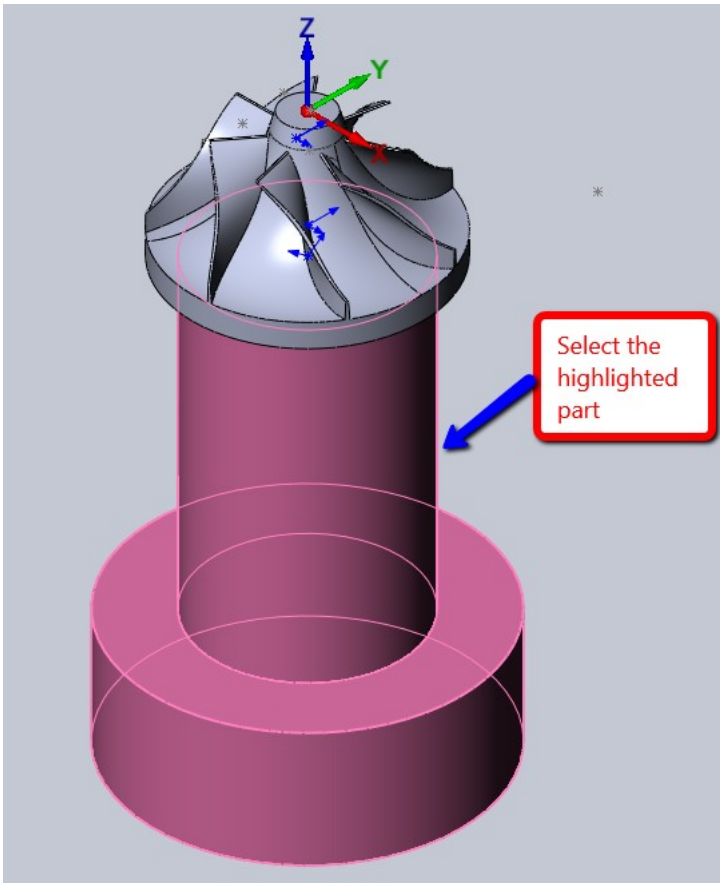
Click ✔ to return to Milling Part Data.

Click  to come out of the Milling Part Data.

Right click **Fixtures** and select **Define Fixture**.

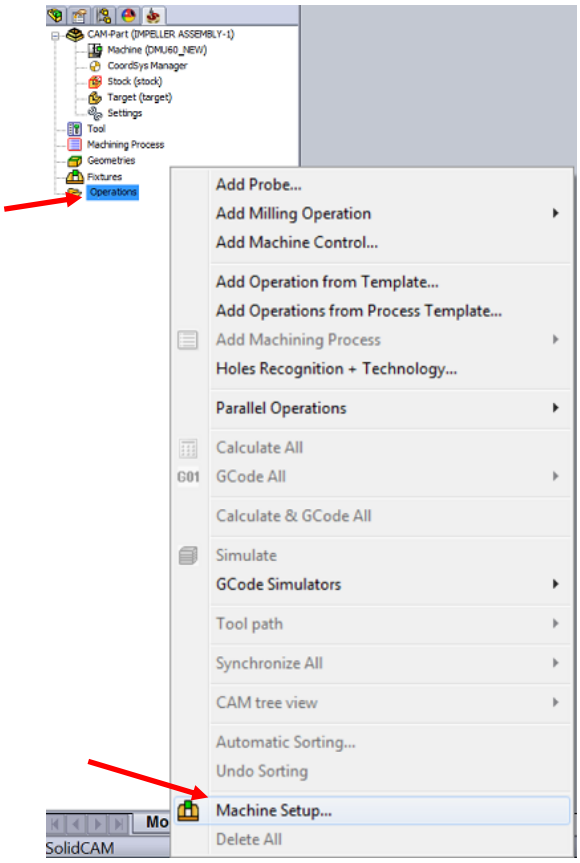


Select the highlighted part as the fixture.

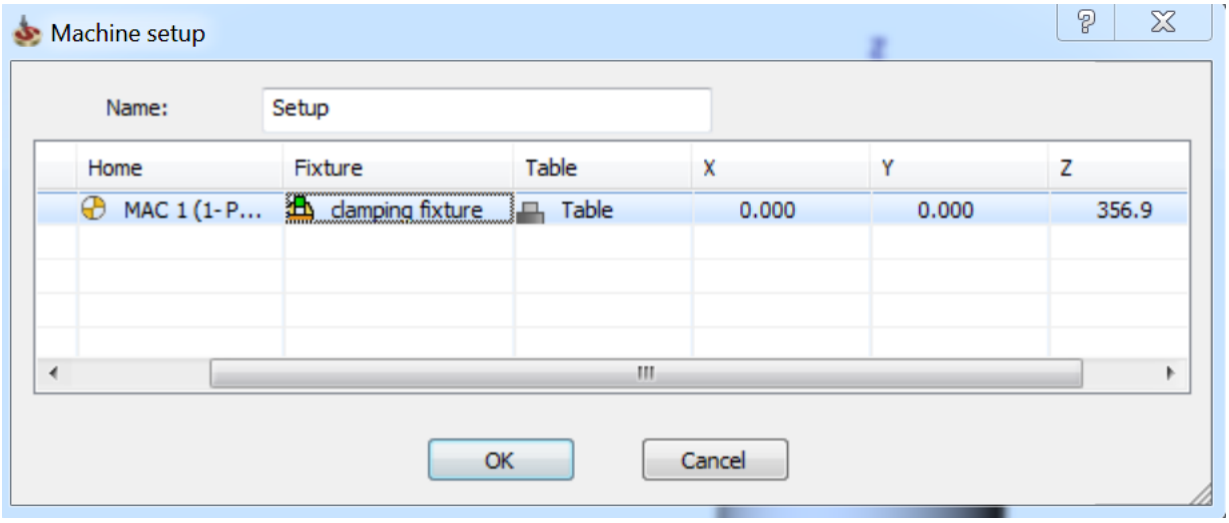


Click 

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



Enter a value of **356.9** in the **Z** column and select clamping fixture in the **Fixture** column. This is the distance from the top face of the part to the bottom of the holding fixture. This is needed to "push" the part above the table of the machine. This will essentially be used in the machine simulation.



Click **OK**.

Morph Between 2 Curves

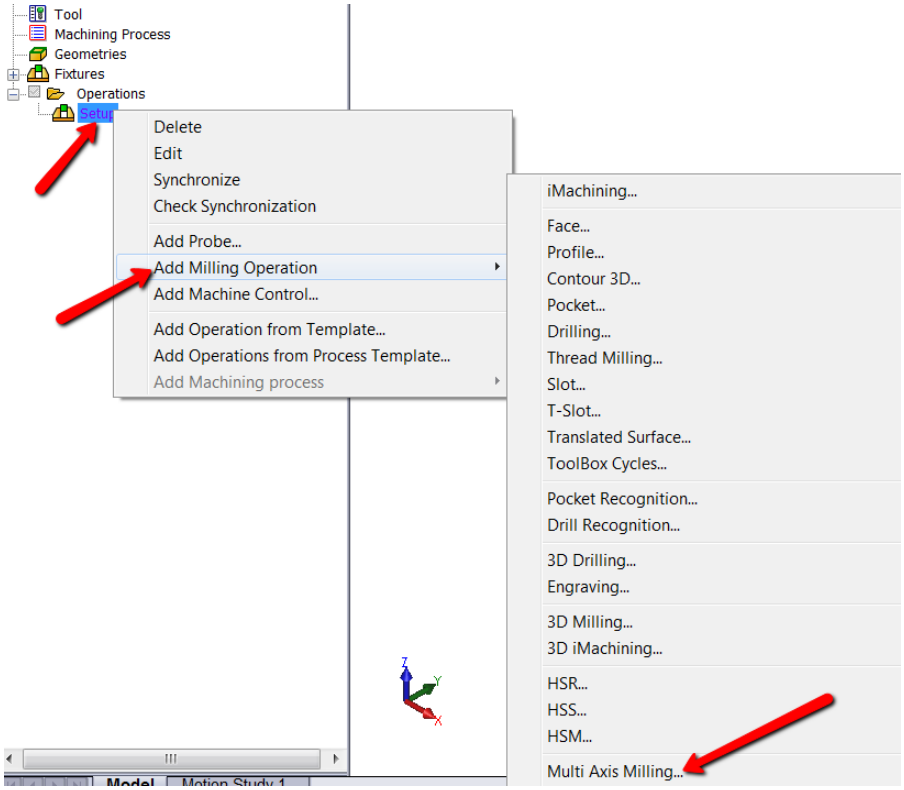
The Morph Between Two Curves feature creates a morph toolpath between two leading curves. Morph means that the generated toolpath gradually interpolates between the two curves and it is evenly spread over the surface. This option is very suitable to machine steep areas for mould making , blade surfaces of impeller and Turbine blades. When selecting the set of two curves, the geometry should be selected directly from the drive surfaces.

The main advantage of this strategy is that the toolpath is created on several surfaces by following a pattern created by two curves and there are no lifts. The toolpath is homogenous from start to end.

Care must be taken while using this strategy while machining Impeller surfaces as varying surface widths can cause the toolpath to “squeeze” at certain areas thereby creating a rubbing effect.

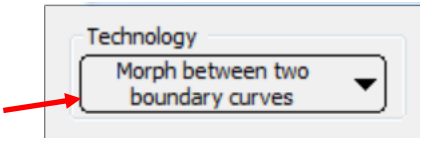
We would be discussing various tilt mechanisms available inside SolidCAM 5 Axis. An interesting tilt mechanism “Tilt through Curve” helps us to create toolpath on surfaces with blending fillets.

Right click **Setup** > **Add Milling Operation** > **Multi Axis Milling**.



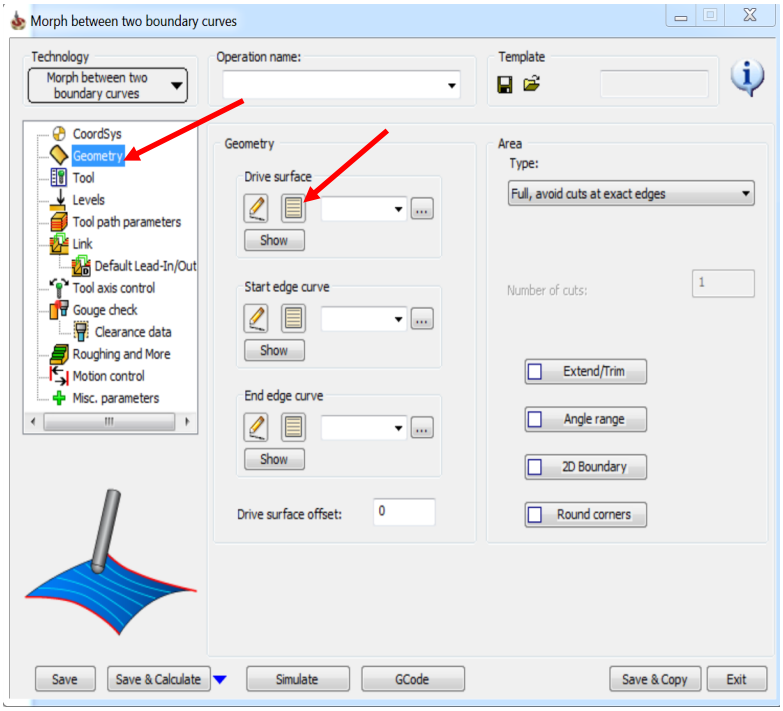
The Operations Manager window opens.

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

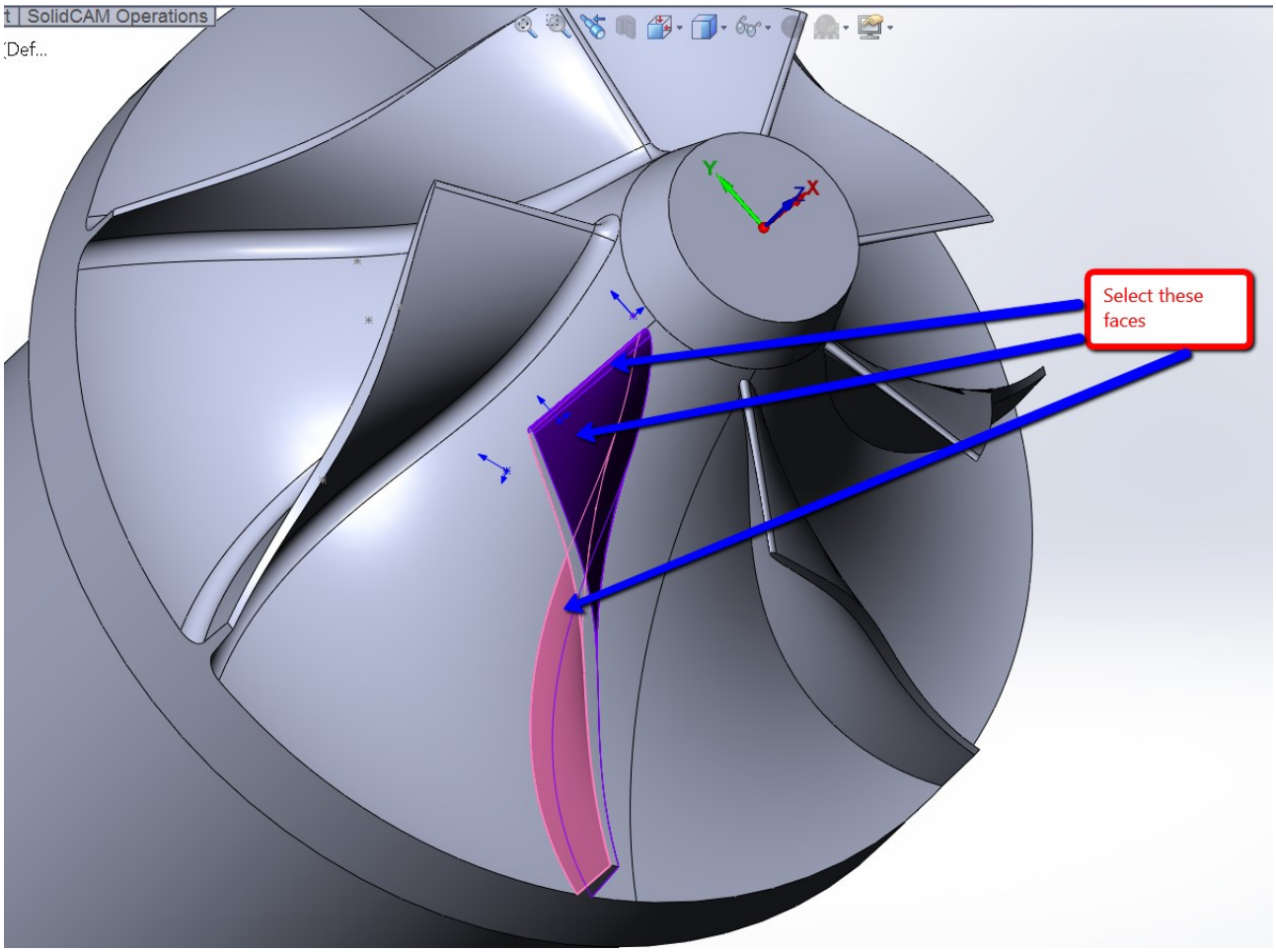


Click **Geometry**.

Click New icon under **Drive surface**.

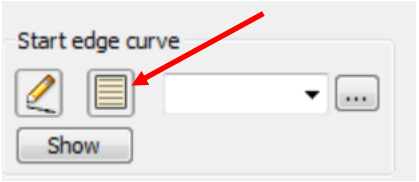


Select the three faces as shown in the image.

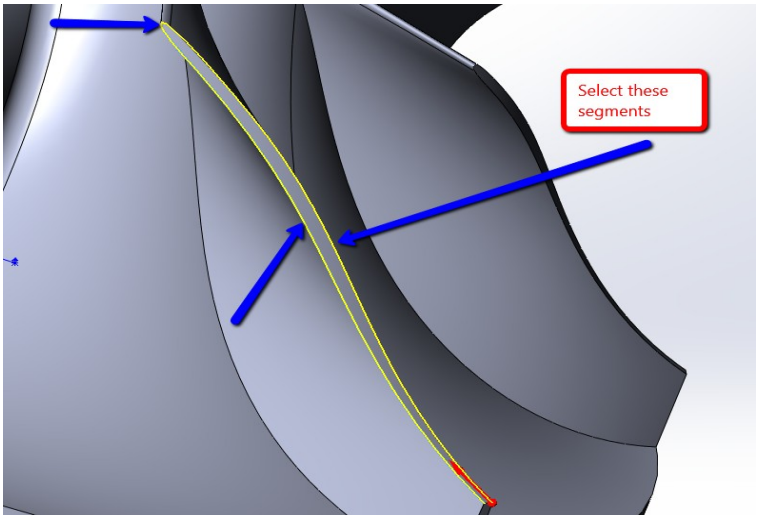


Click ✔ to return to the Operations Manager.

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



Select the three segments as shown in the image.



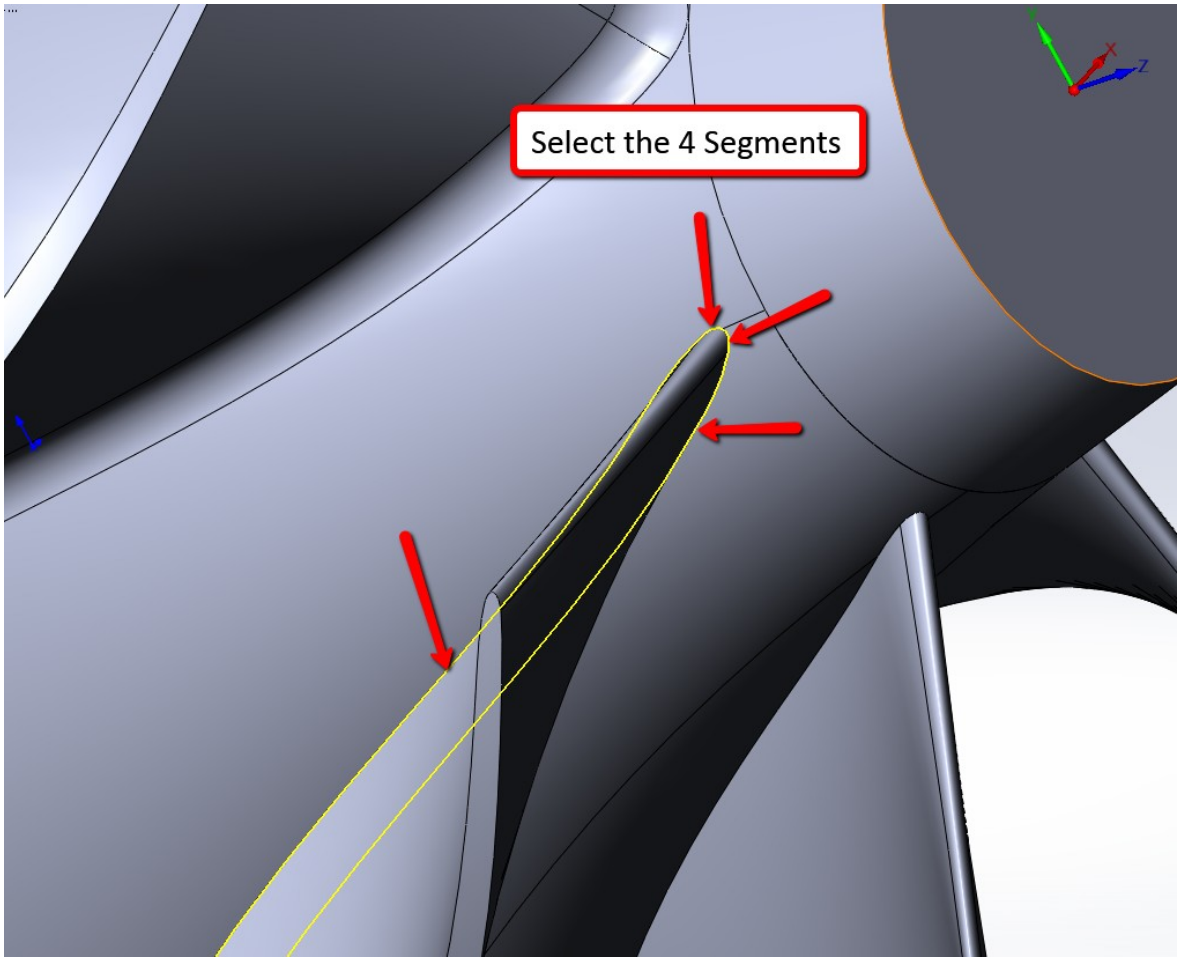
Click  in the **Chain List** section.

Click  to return to the Operations Manager.

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

Select the four segments as shown

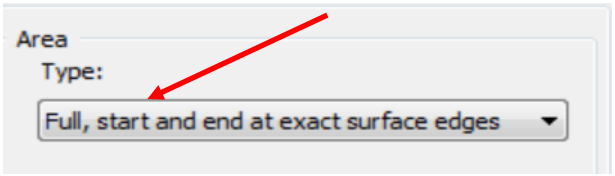
in the image.



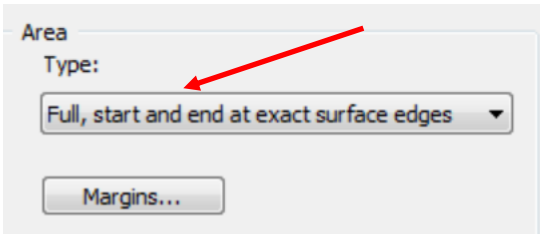
Click  in the **Chain List** section.

Click  to return to the Operations Manager.

Select the **Type** as **Full, start and end at exact surface edges** from the drop down list in the **Area** section.

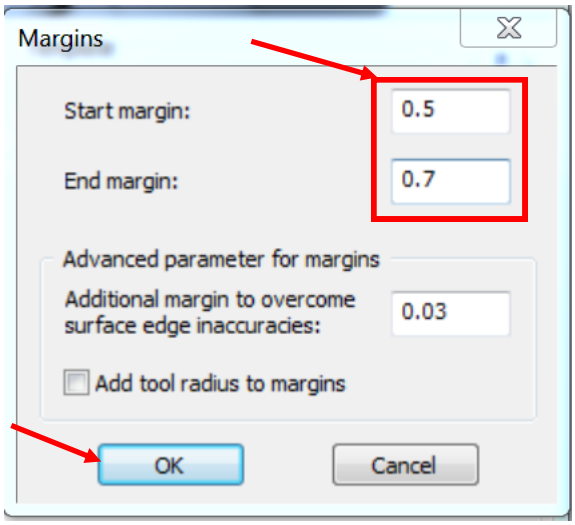


Click **Margins**.

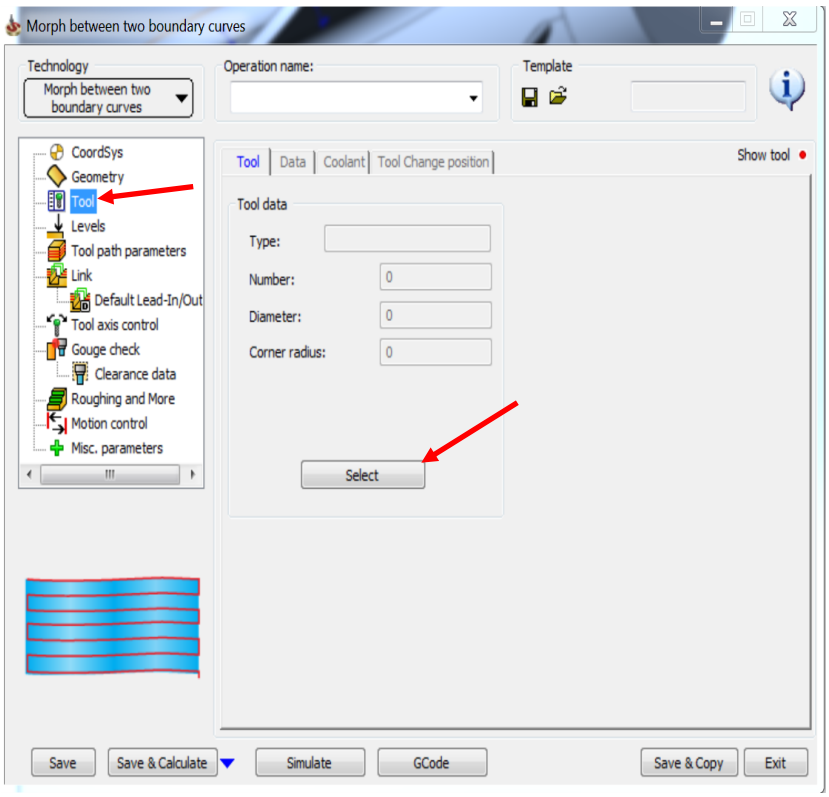


Enter the Start and End margins as shown in the image.

Click **OK**.



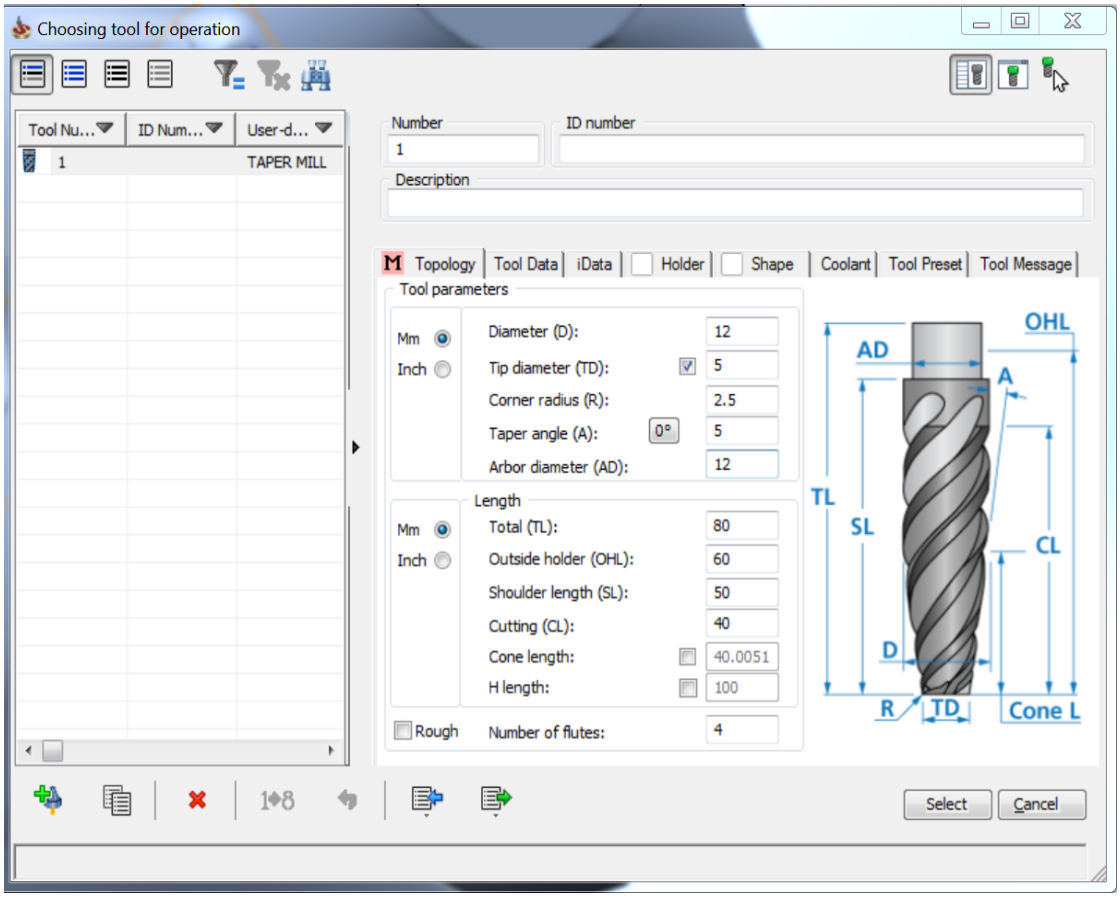
Click **Tool** > **Select**.



Click  icon to add a new tool.

Select Taper Mill as our tool.

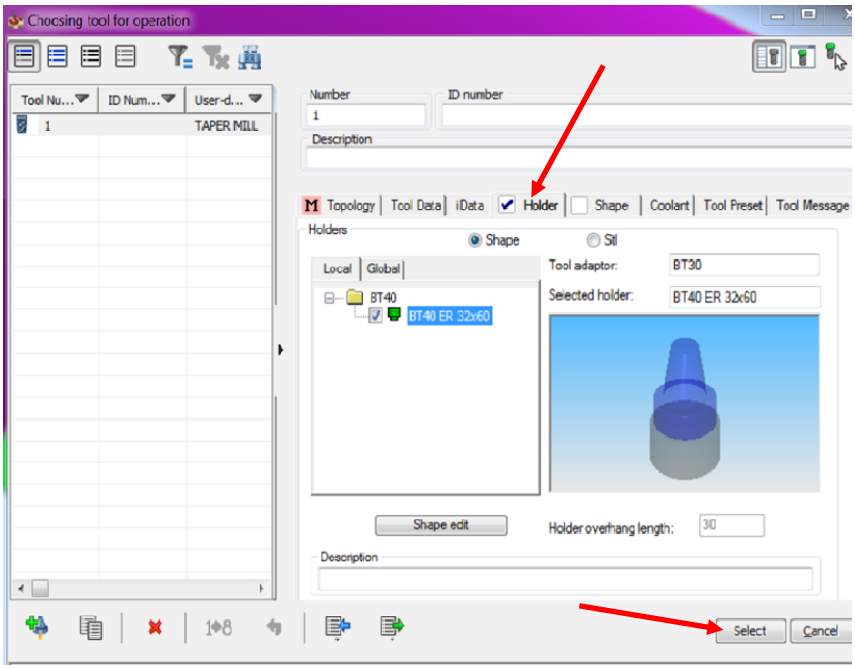
Enter the tool parameters as shown in the image.



Select **Holder**.

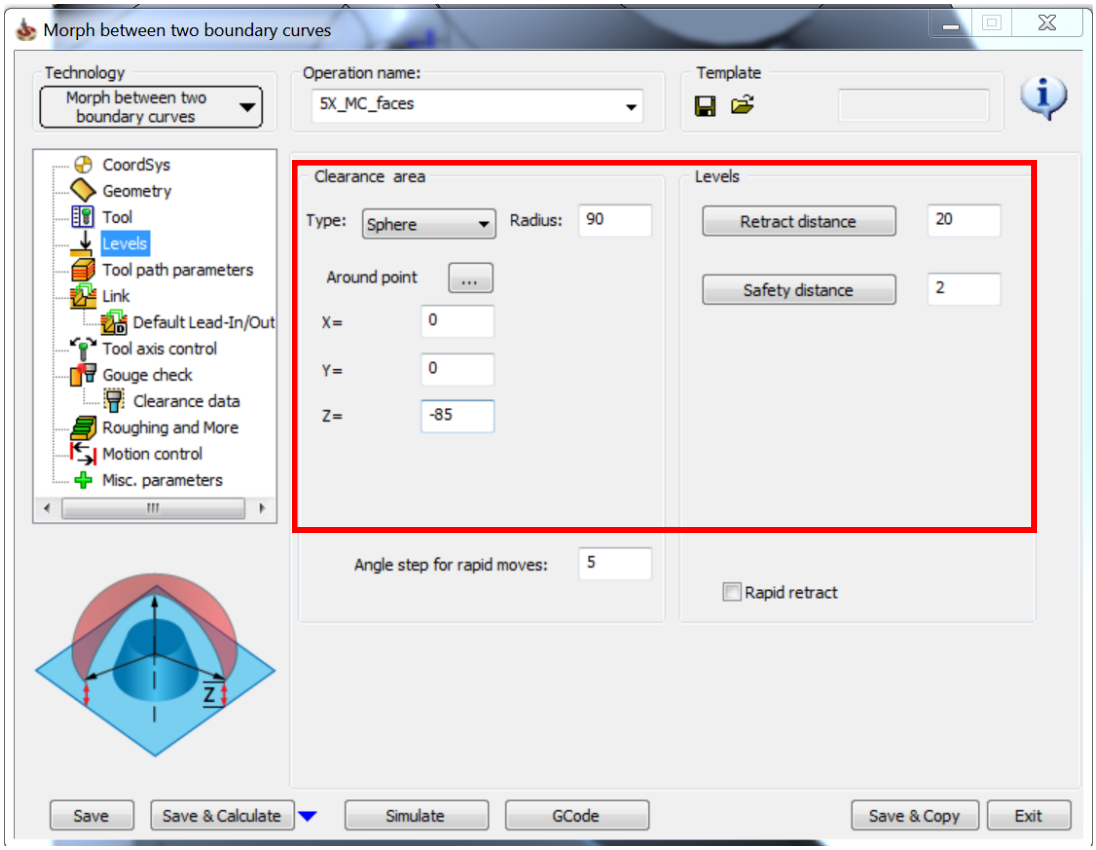
Select BT 40 > BT 40 ER 32X60.

Click **Select**.



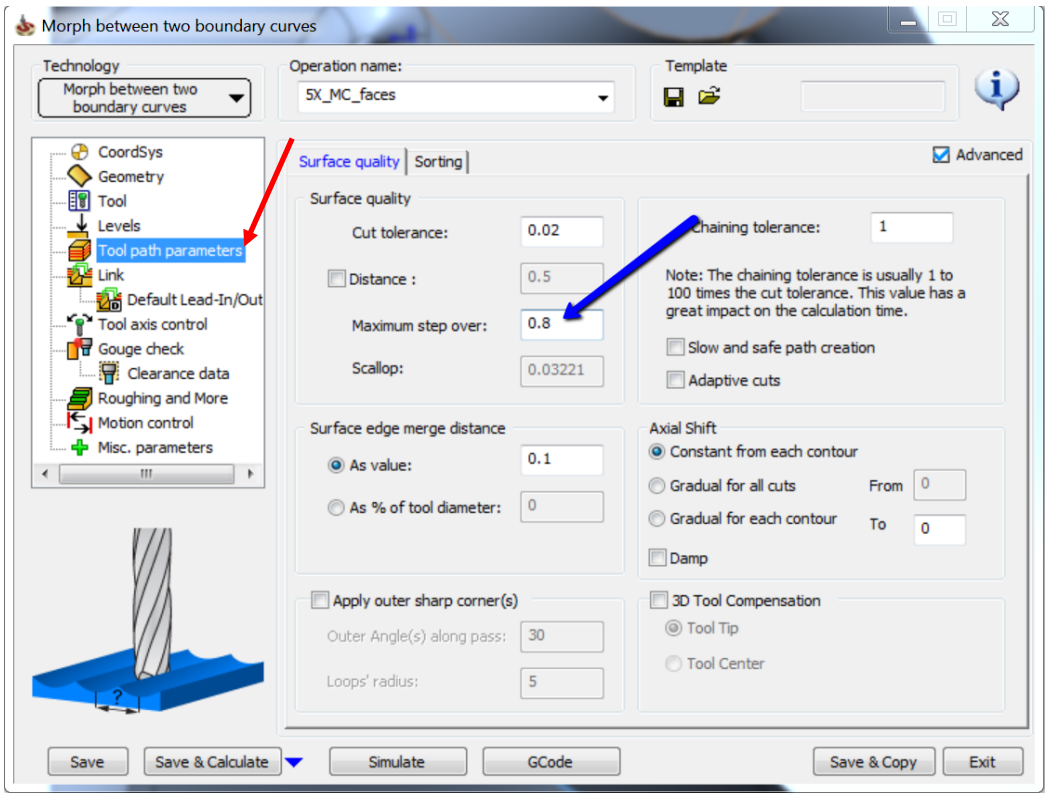
Click **Levels**.

Select the **Levels** parameters as shown in the image.

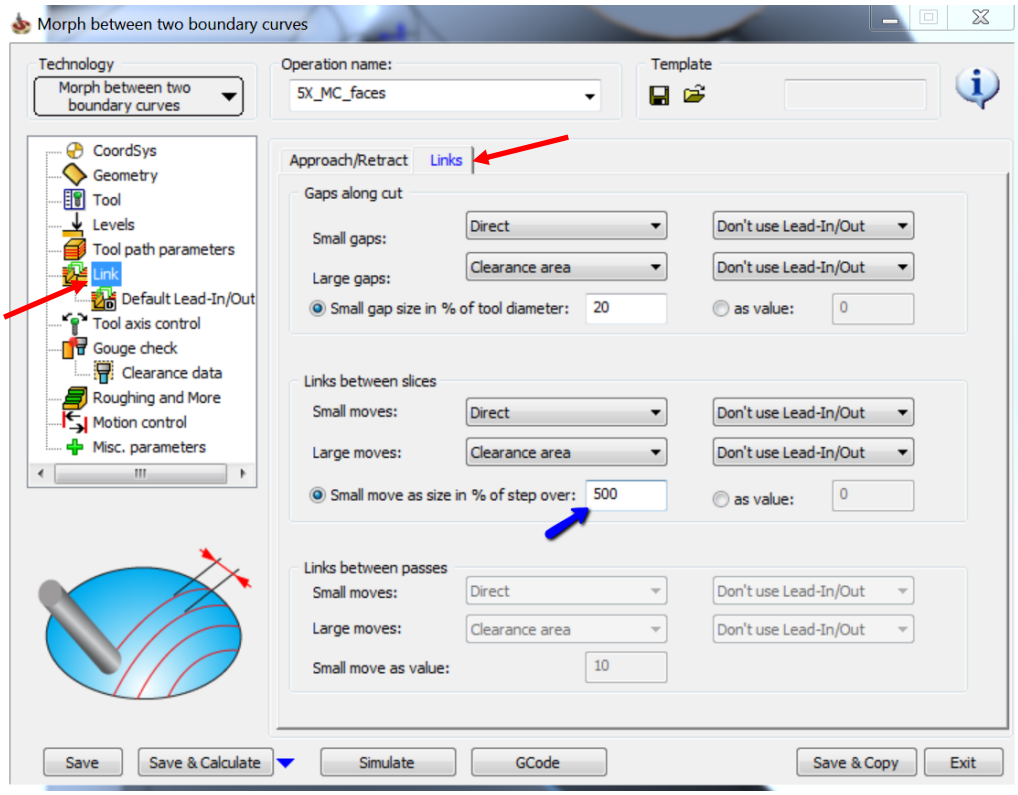


Click **Tool path parameters**.

Enter a value of 0.8 in **Maximum step over**.



Click **Link** > **Links** sub-tab.

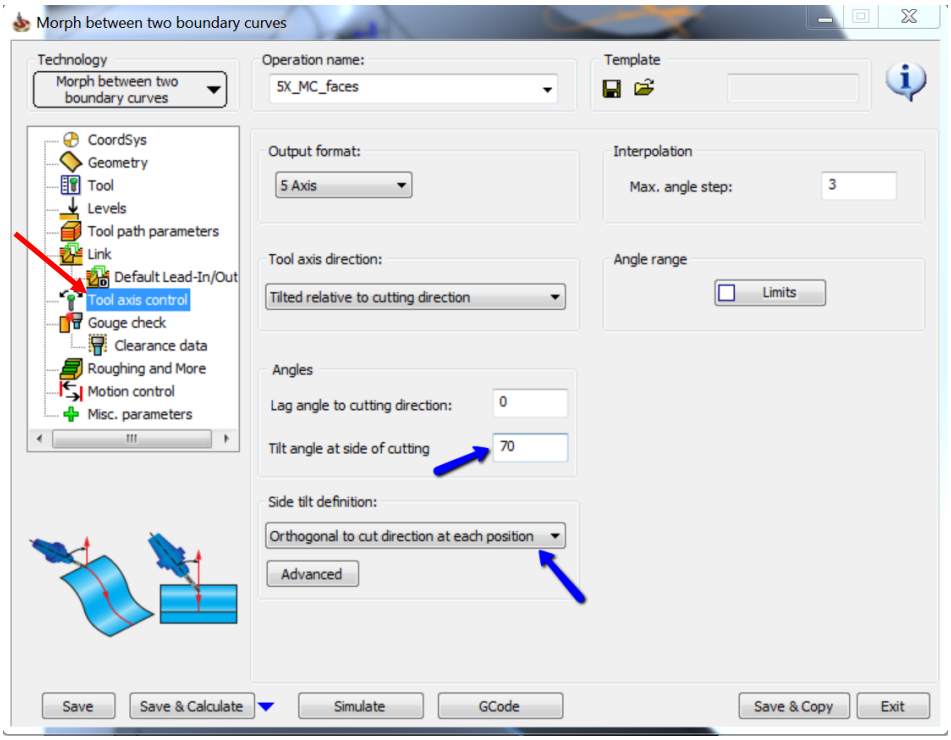


Enter a value of 500 in **Small move as size in % of step over**.

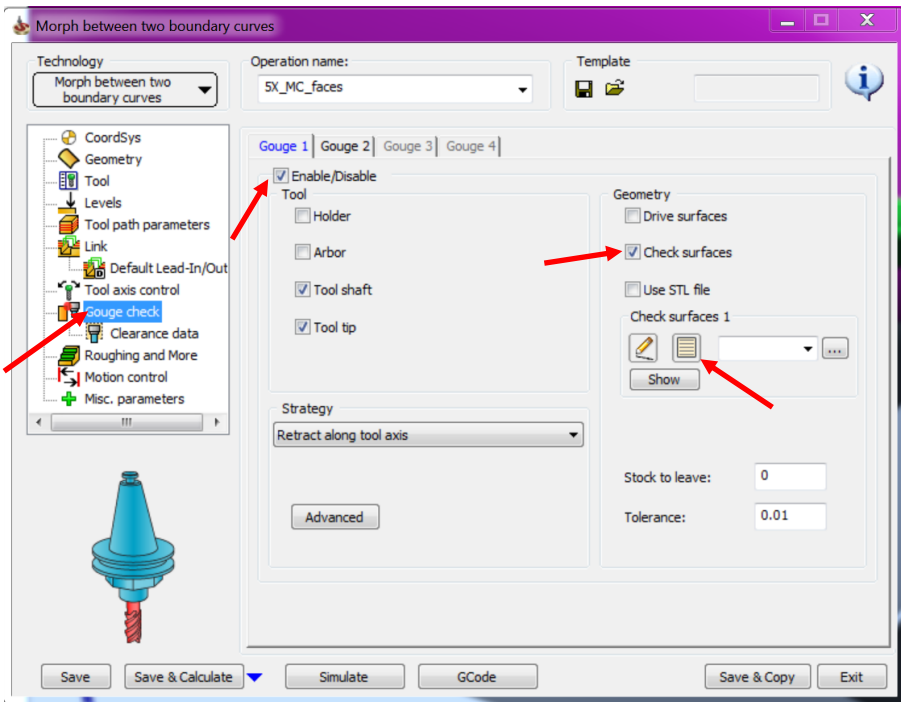
Click **Tool axis control**.

Enter a value of 70 in **Tilt angle at side of cutting** in the **Angles** section.

Select **Orthogonal to cut direction at each position** from the drop down, in the **Side tilt definition** section.



Click **Gouge check**.

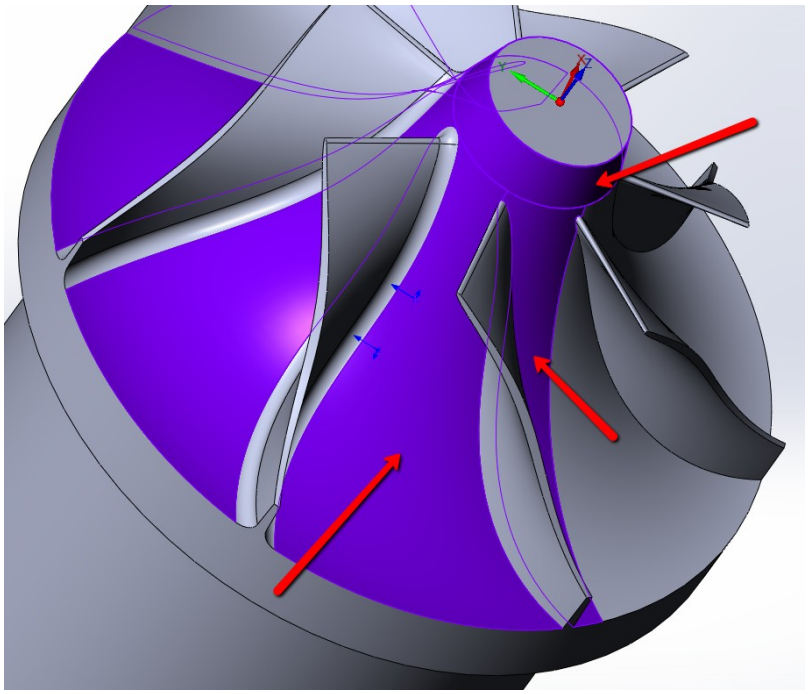


Select the **Enable/Disable** check box.

Select the **Check surfaces** check box.

Click New icon in **Check surfaces 1** section.

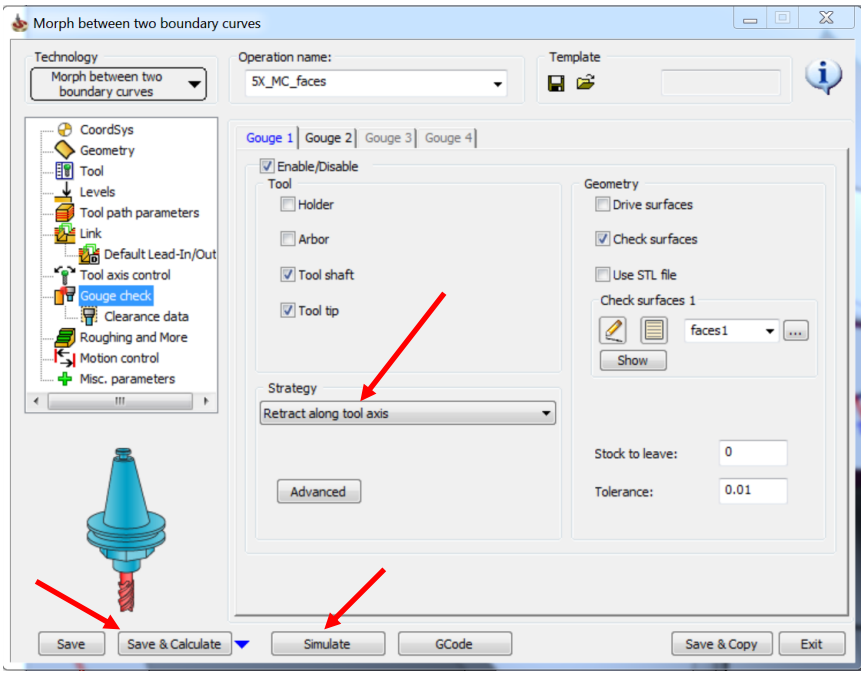
Select the three surfaces as shown in the image.



Click 

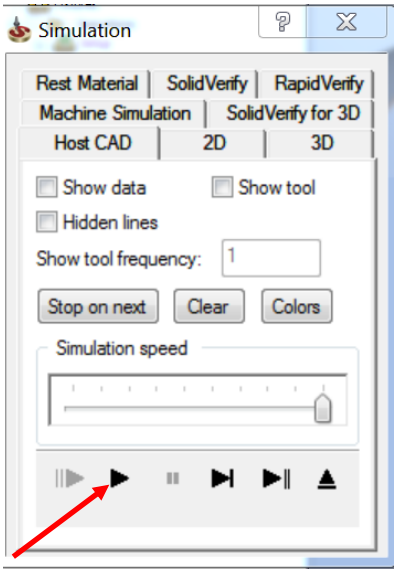
Ensure that **Retract along tool axis** is chosen in the **Strategy** section.

Click **Save & Calculate**.

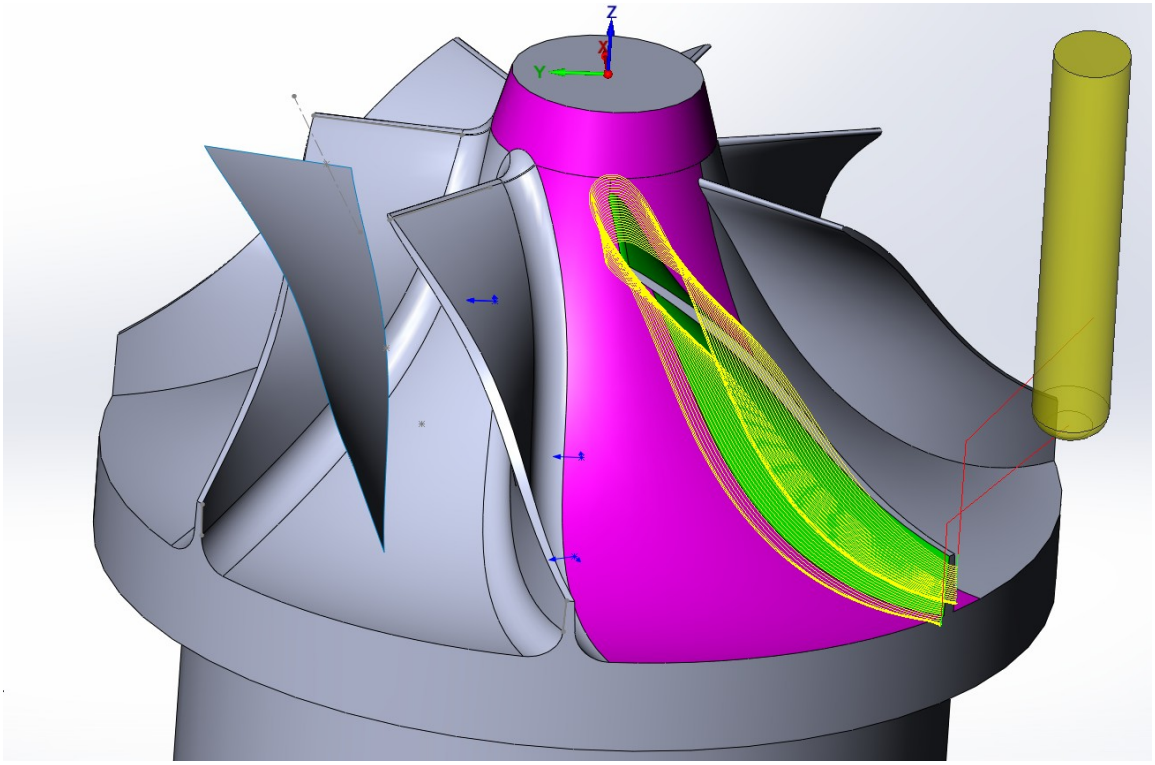


Click **Simulate** once the tool path is calculated.

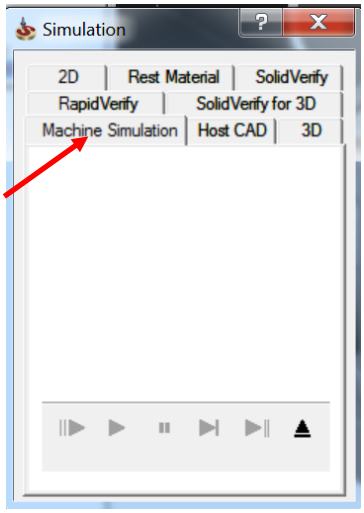
Click the Play icon.



The simulated toolpath looks like this:

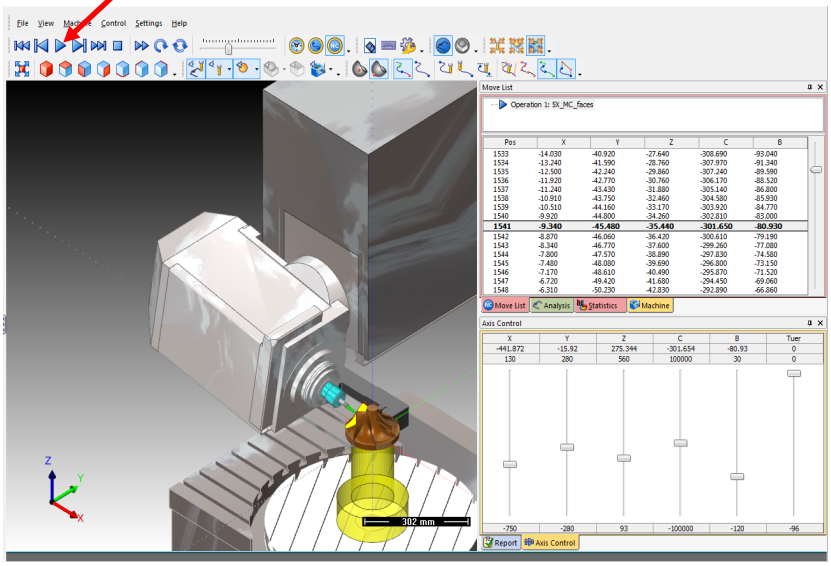


Click **Machine Simulation**.



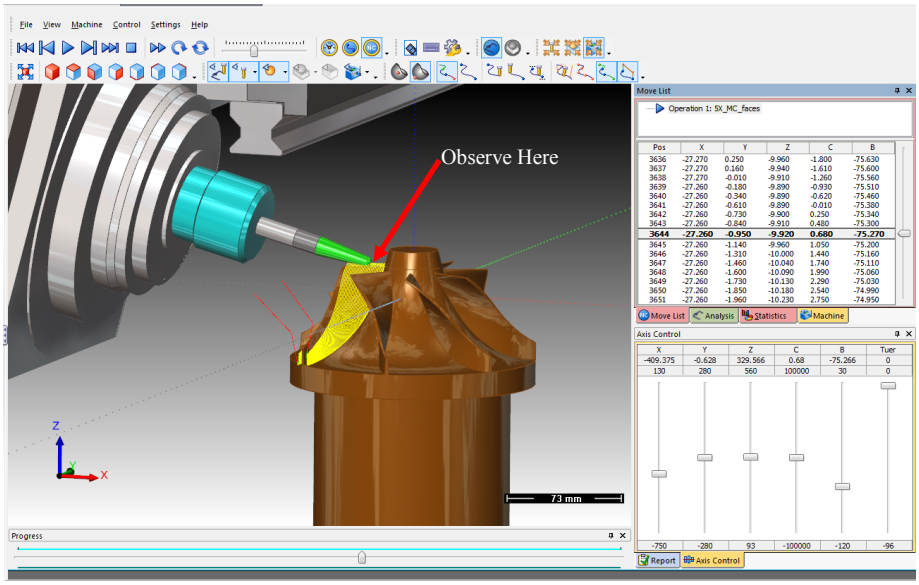
Click the hide icon.

Run machine simulation by clicking the play icon.



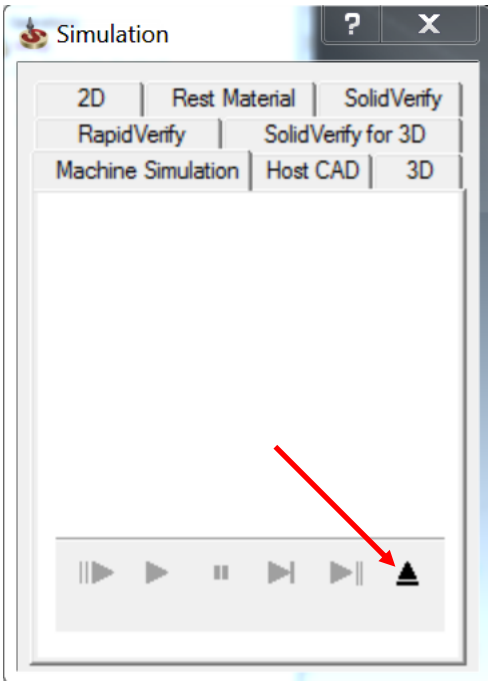
Pull the slider on **Move List** to 50% of the tool path.

Use the down key on your keyboard and move step by step to observe the movement on the fillet.

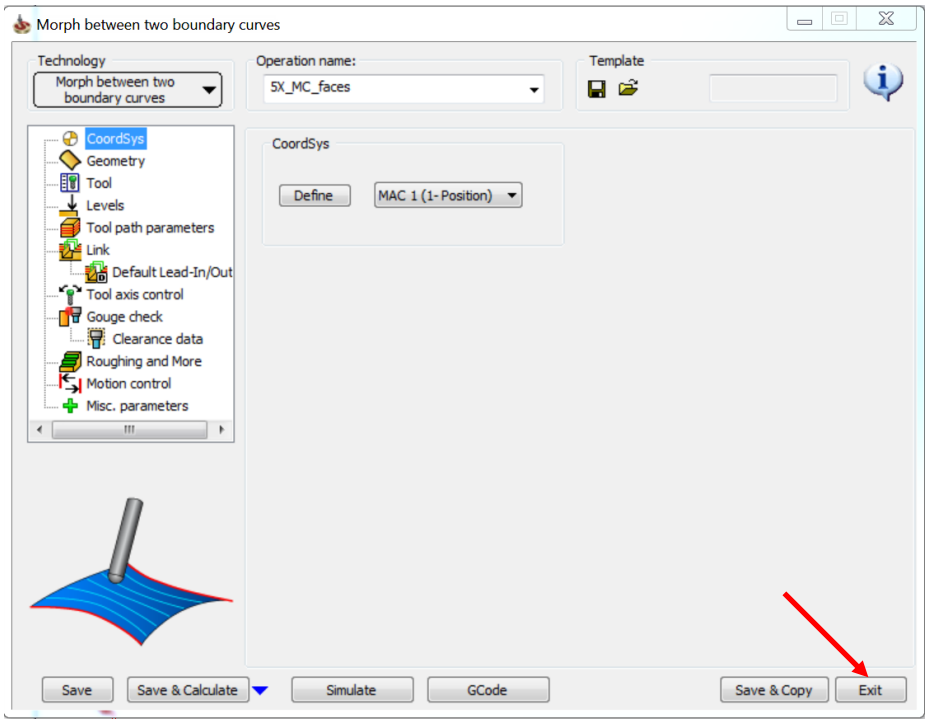


Observe that the B angle changes drastically due to the chosen side tilt definition i.e. Orthogonal to cut direction at each position. Also observe that the tool side sticks to the fillet while rolling creating a funny situation where the head and table are trying to roll over.

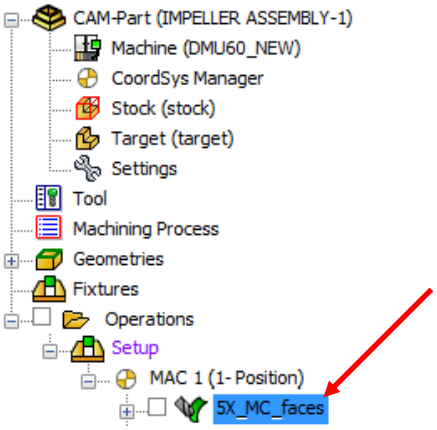
Select the **Exit** icon to exit machine simulation.



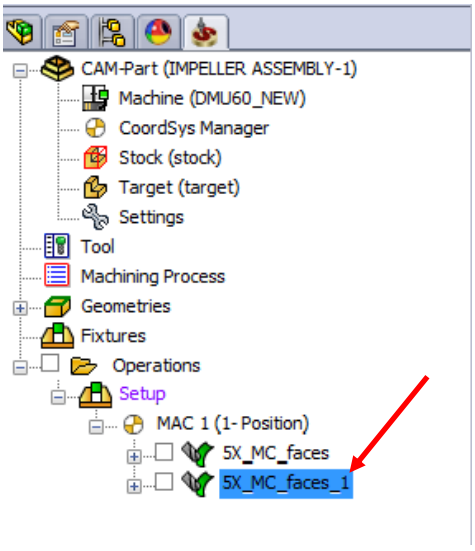
Click **Exit** to exit Operations Manager.



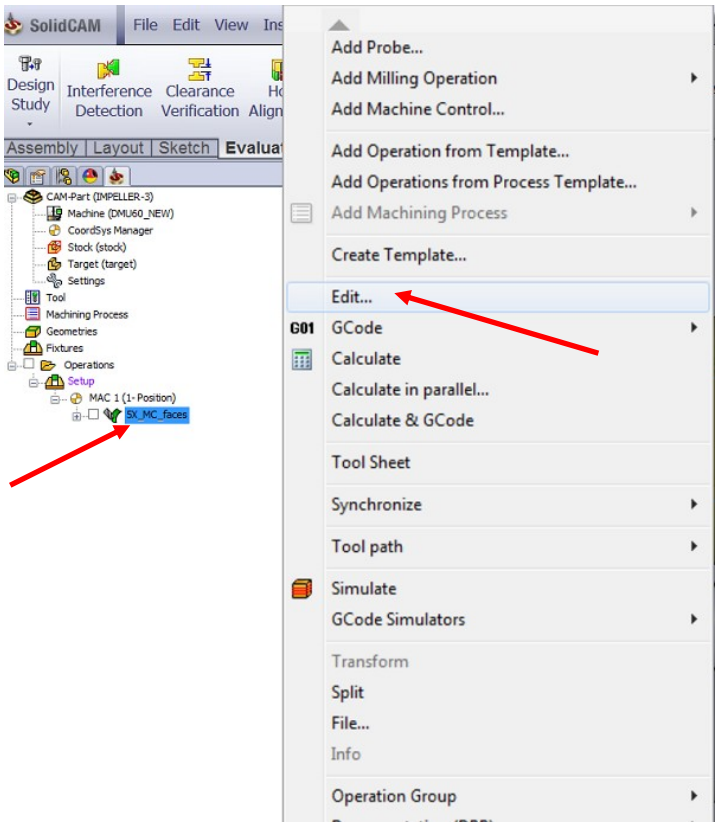
Right click > **Copy** the tool path.



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

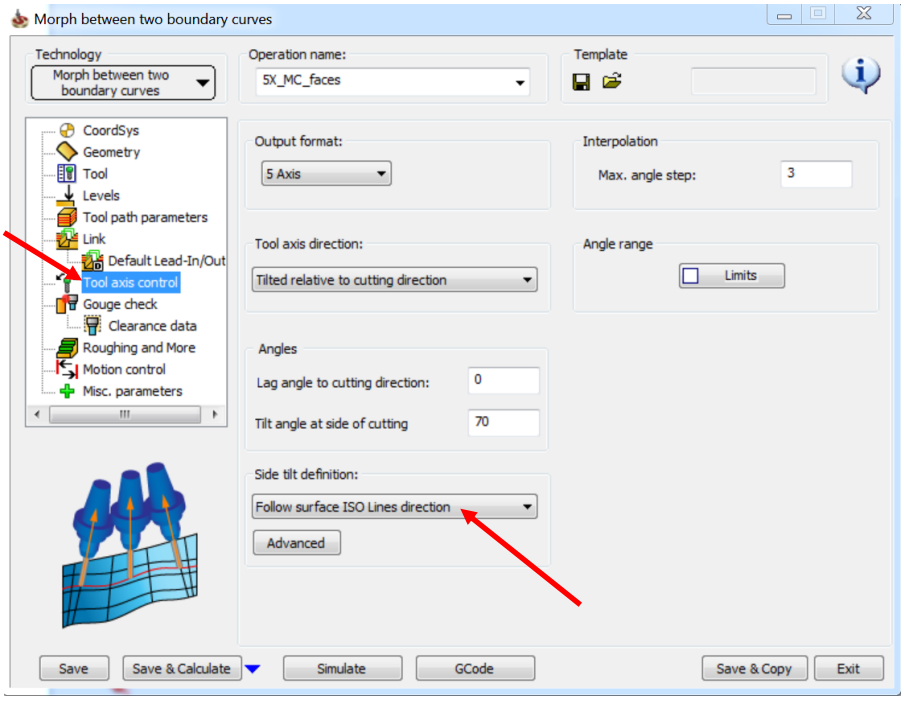


Right click > **Edit** the pasted tool path.



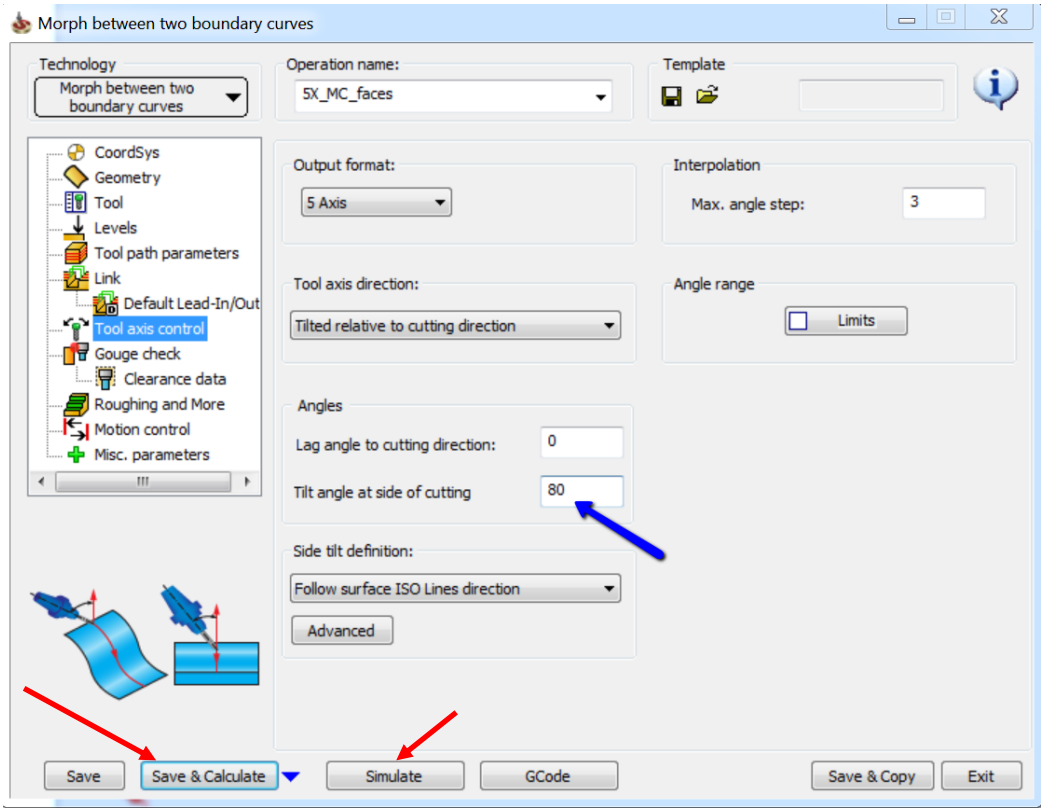
The Operations Manager now opens.

Click **Tool axis control**.



Select **Follow surface ISO Lines direction** as the **Side tilt definition**.

Enter a value of 80 in the **Tilt angle at side of cutting**.

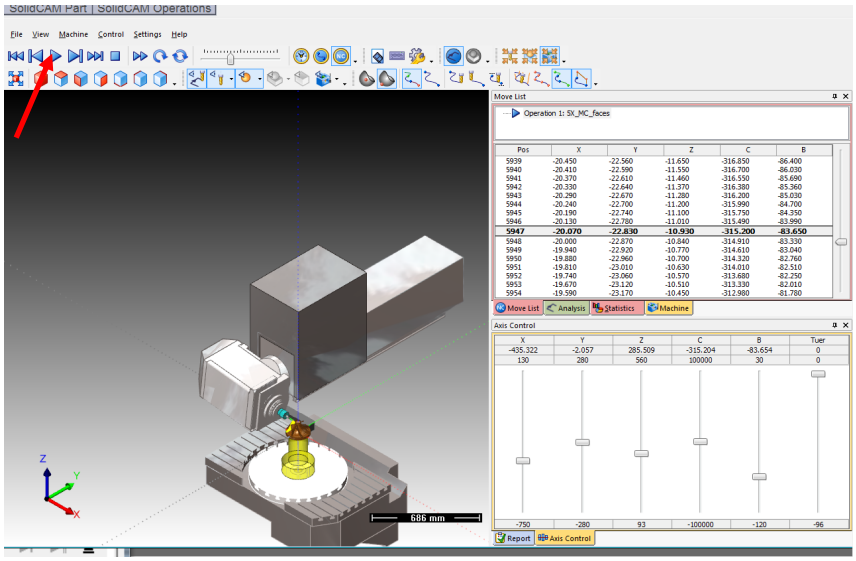


Click **Save & Calculate**.

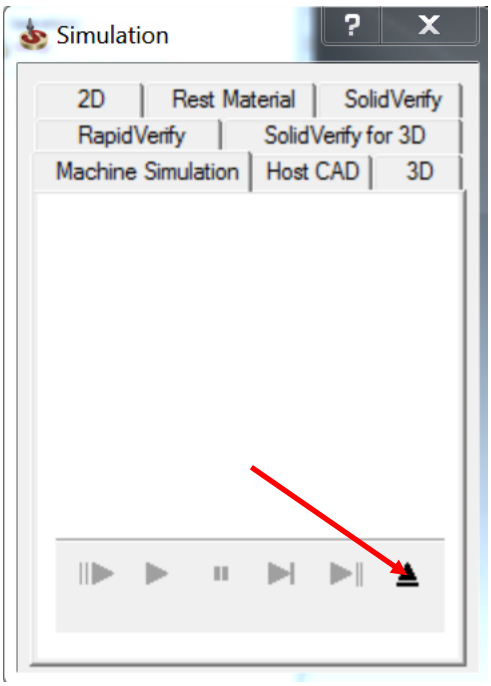
Click **Simulate** once the tool path is calculated.

Click the hide  icon.

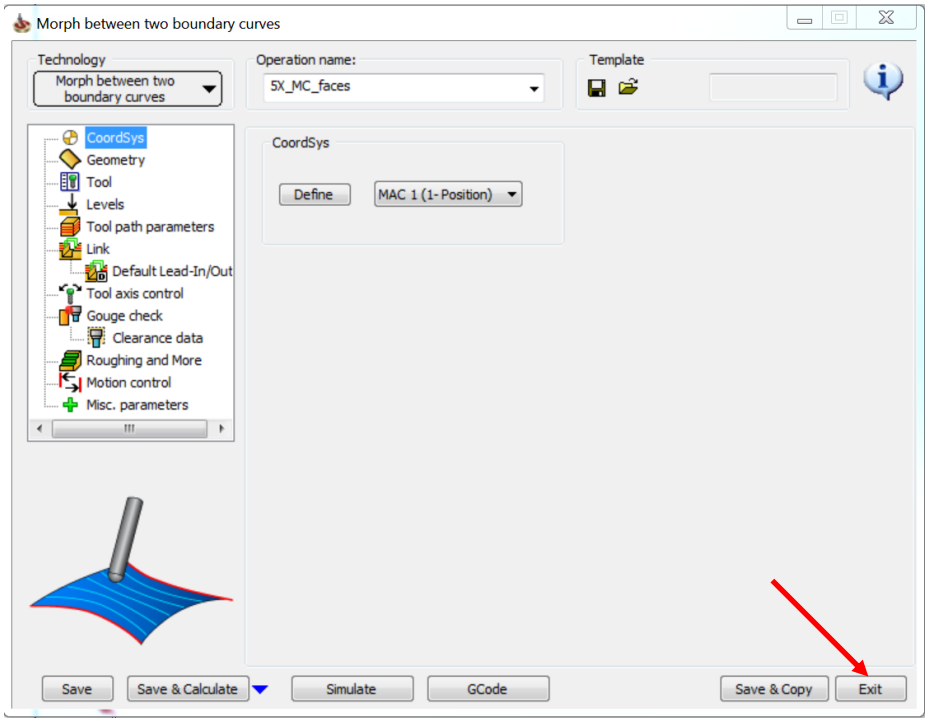
Run machine simulation by clicking the play icon.



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

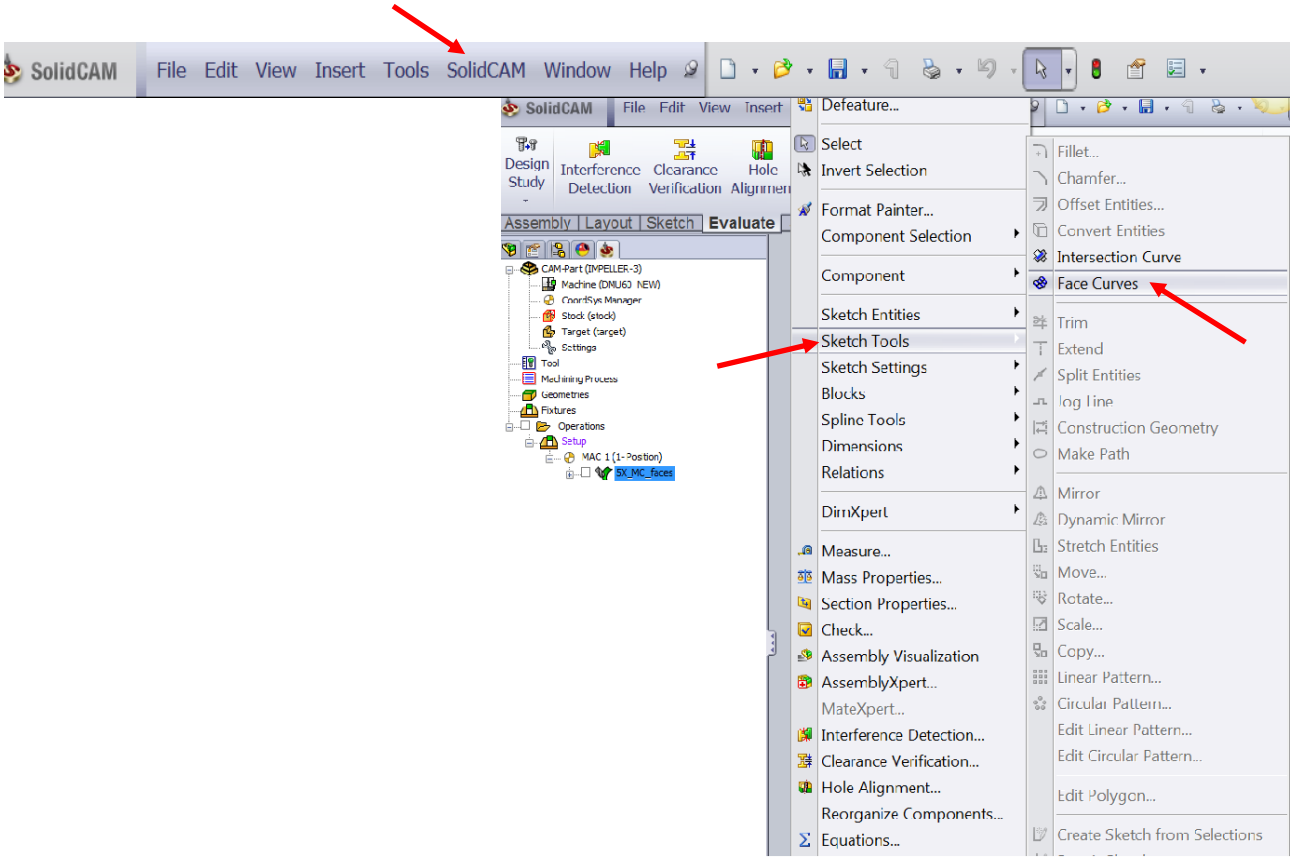


Click **Exit** to exit Operations Manager.

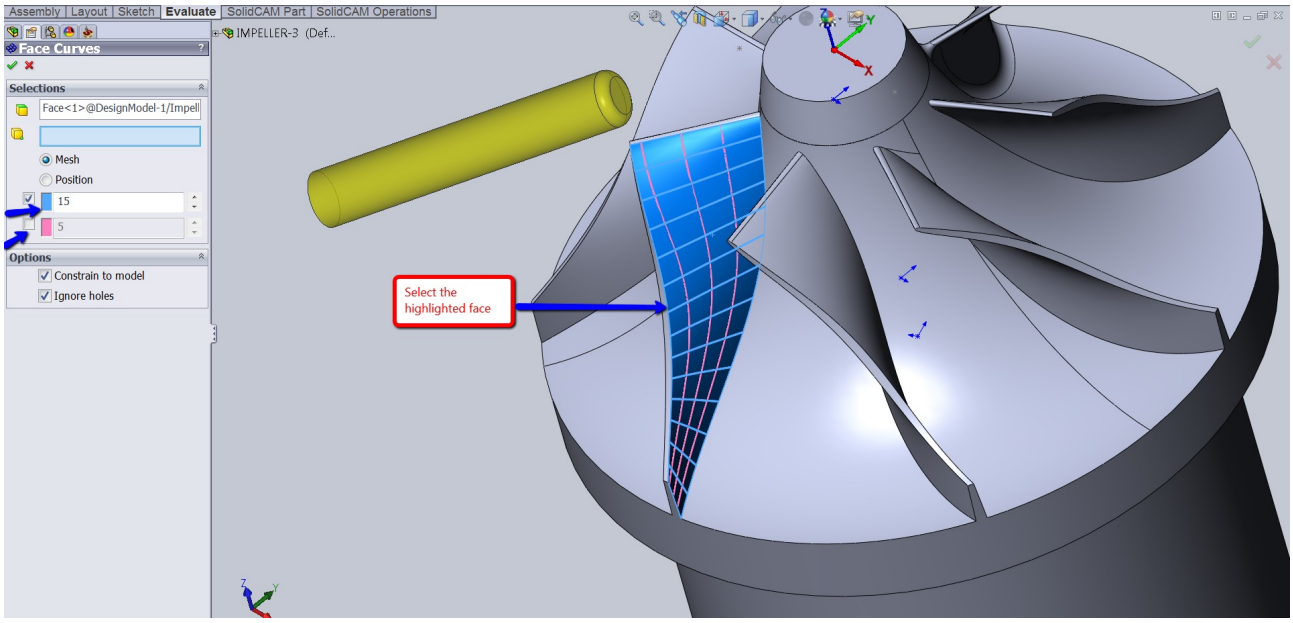


To find out the ISO directions of this tool path, do the following:

Right click **Tools** > **Sketch Tools** > **Face Curves**.



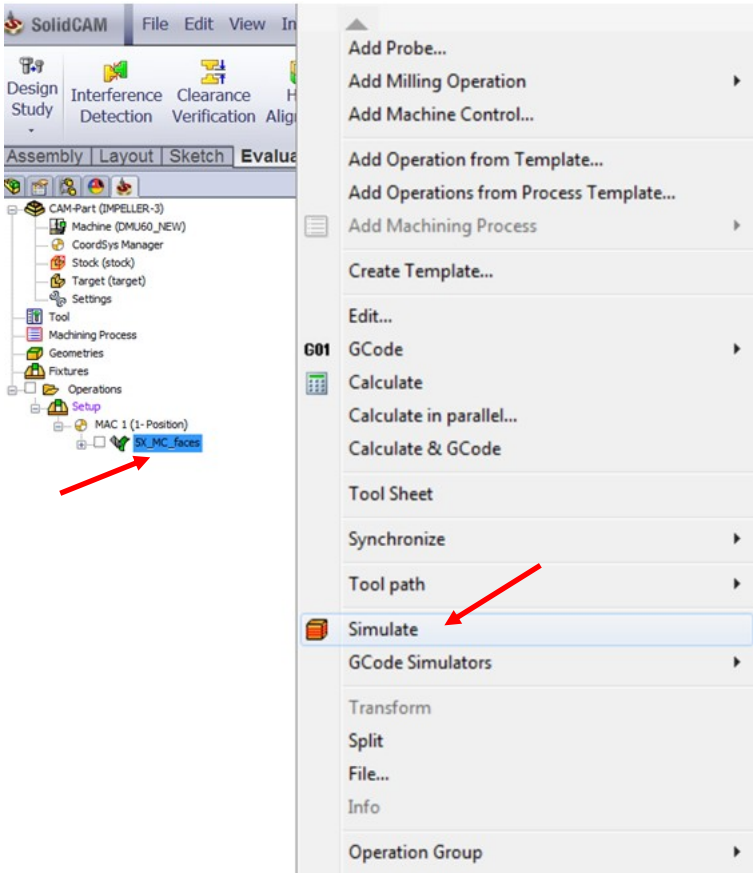
Select the highlighted face as shown in the image.



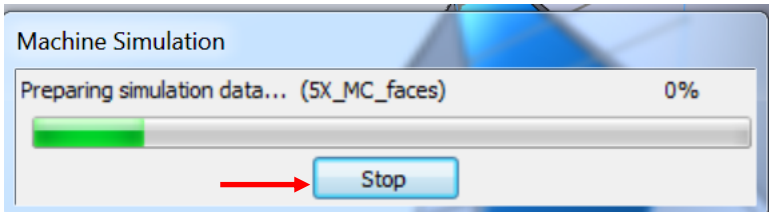
Deselect the pink lines checkbox and enter a value of 15 in the blue lines check box as shown in the above image.

Click 

Right click the tool path and select **Simulate**.



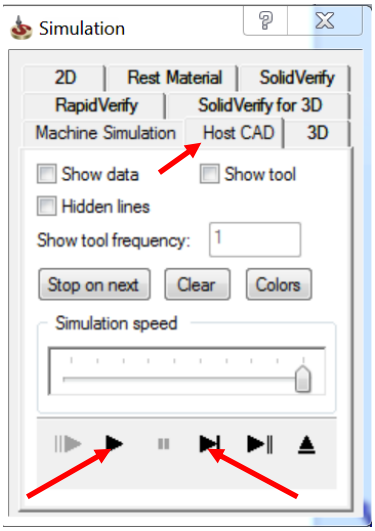
Stop the **Machine Simulation**.



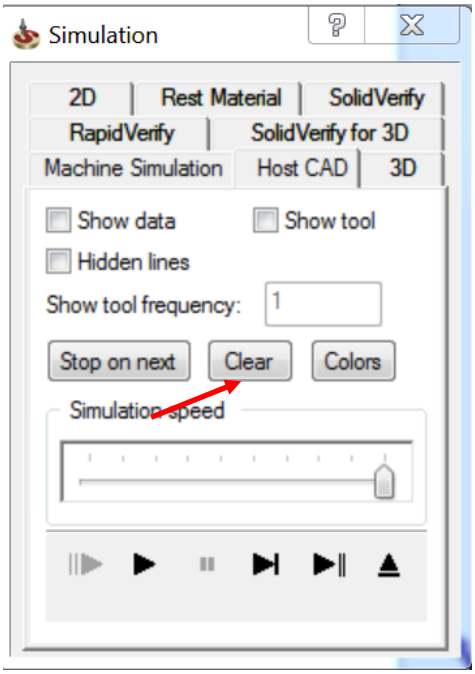
Click **Host CAD**.

Select the single step mode icon.

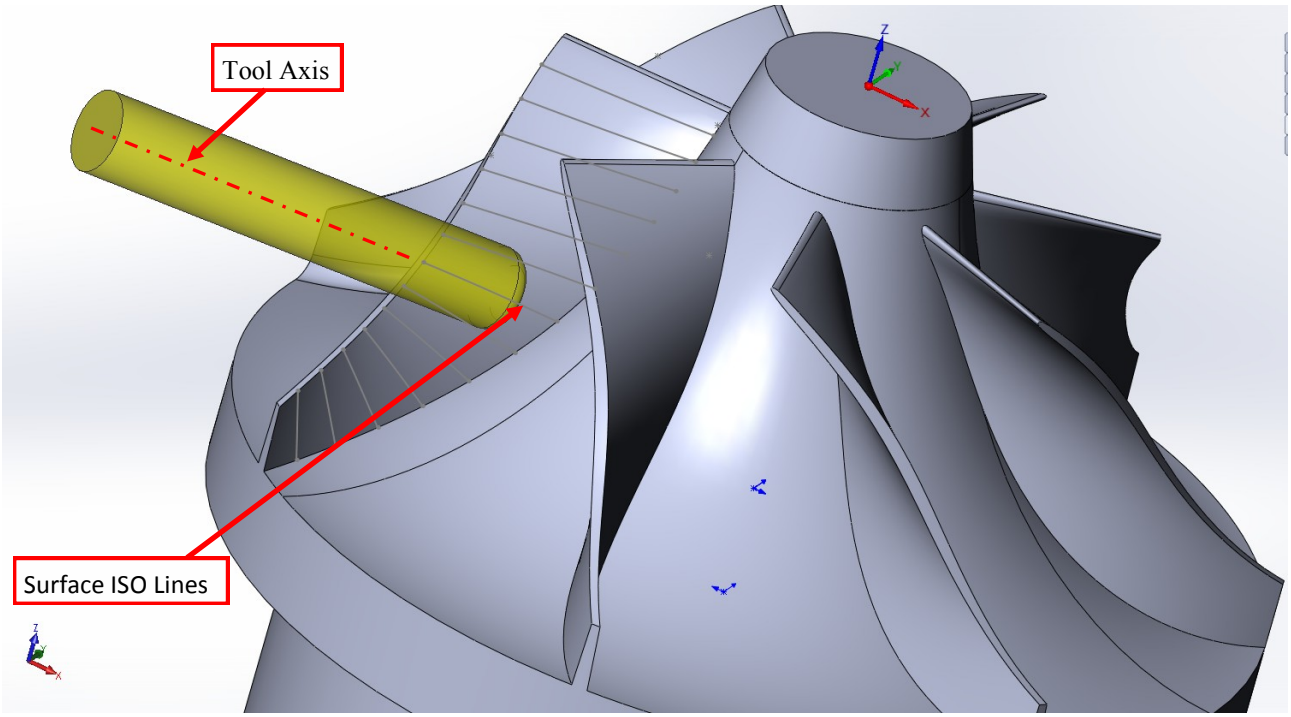
Click the play icon.



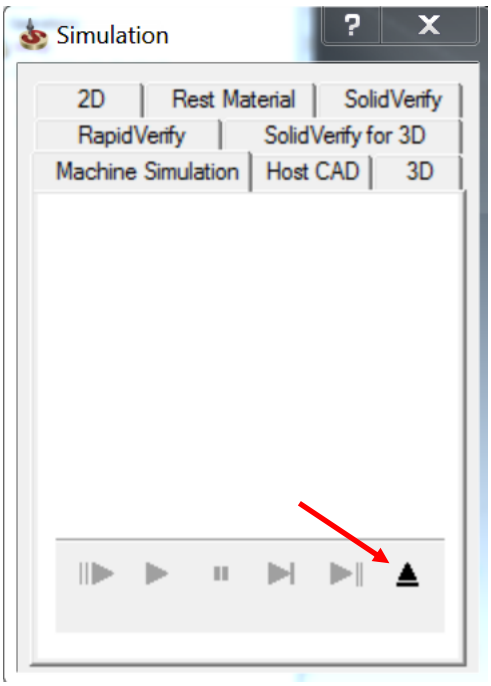
Click **Clear** to clear the simulated tool path.



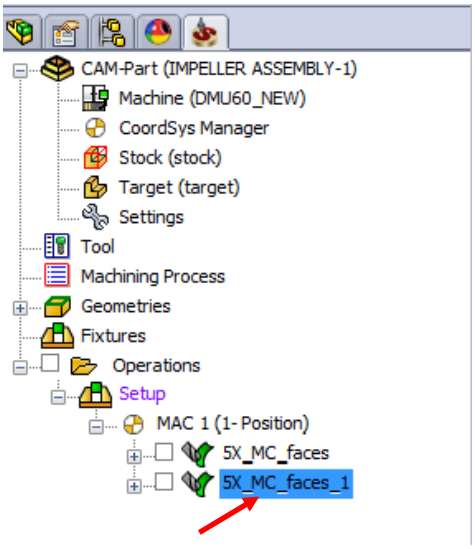
Observe that the tool follows the axis of the ISO parameters. This is because of the side tilt definition we chose i.e. follow surface ISO lines direction.



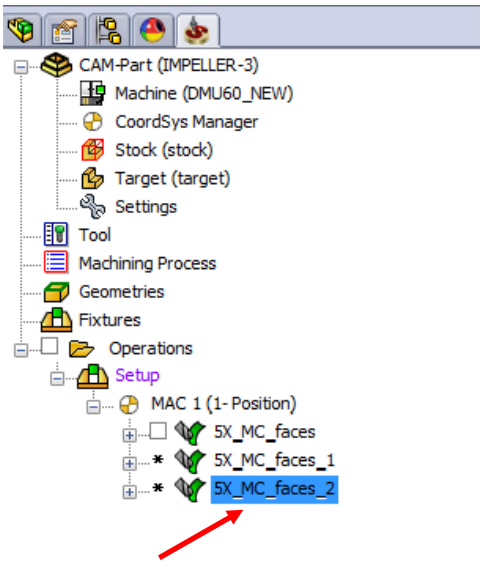
Select the **Exit** icon to exit machine simulation.



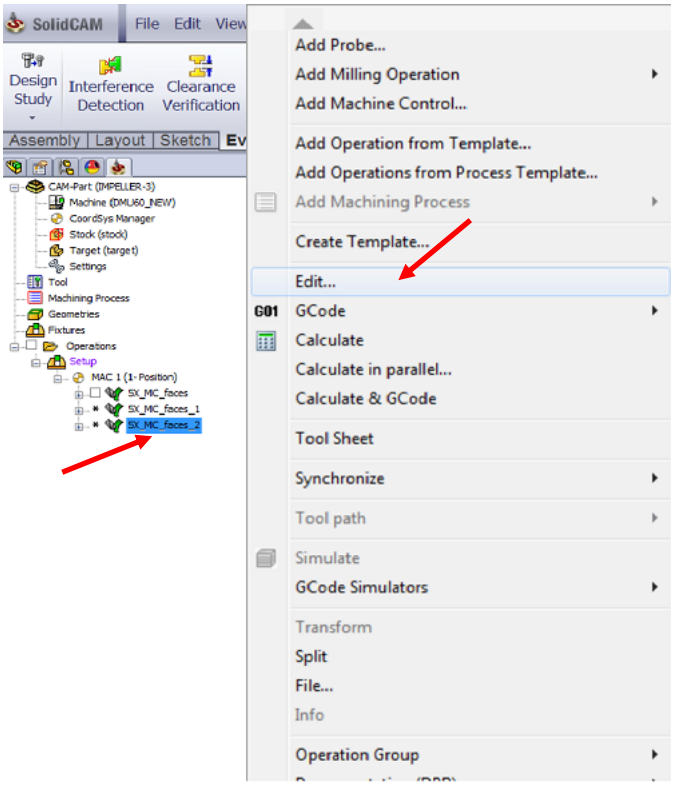
Right click > **Copy** the tool path.



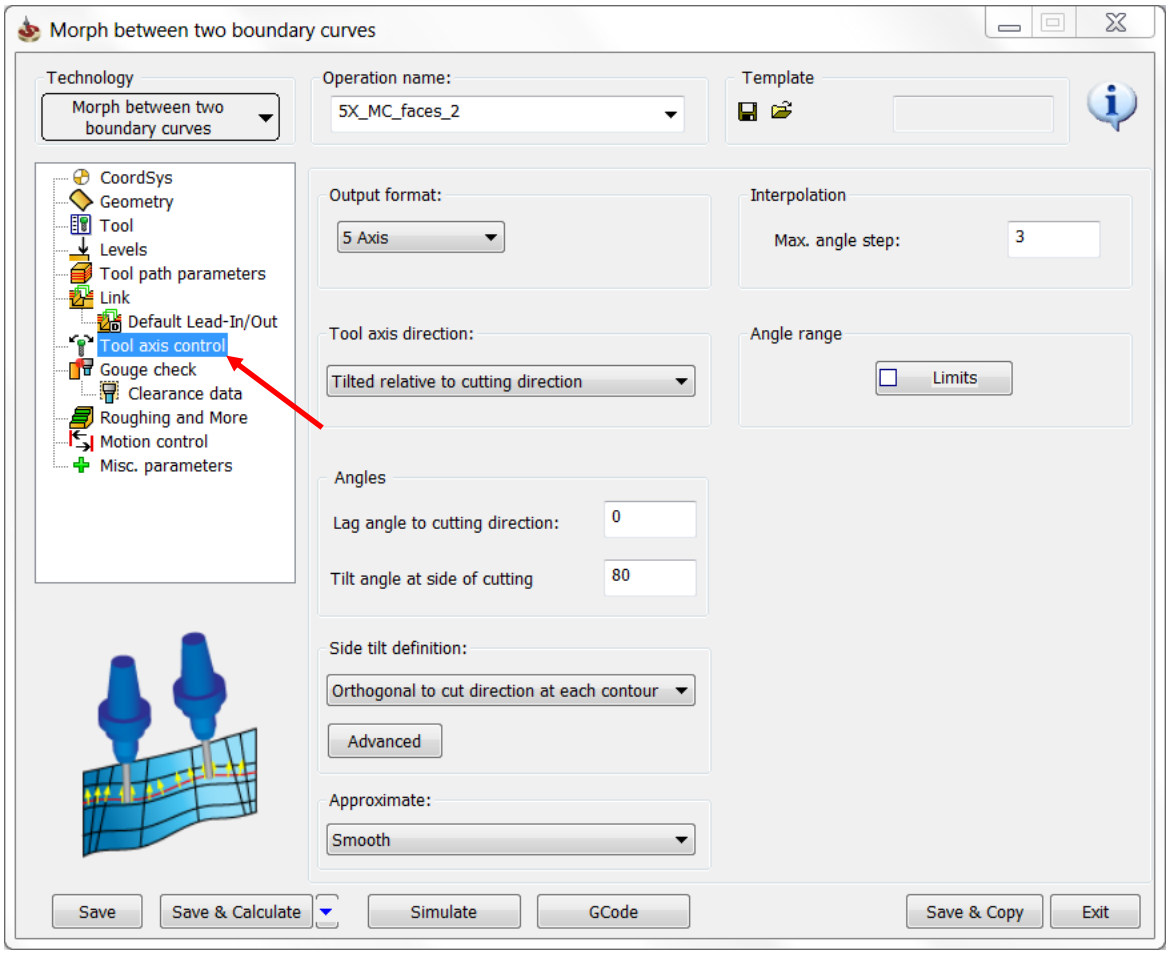
Right click > **Paste** the tool path.



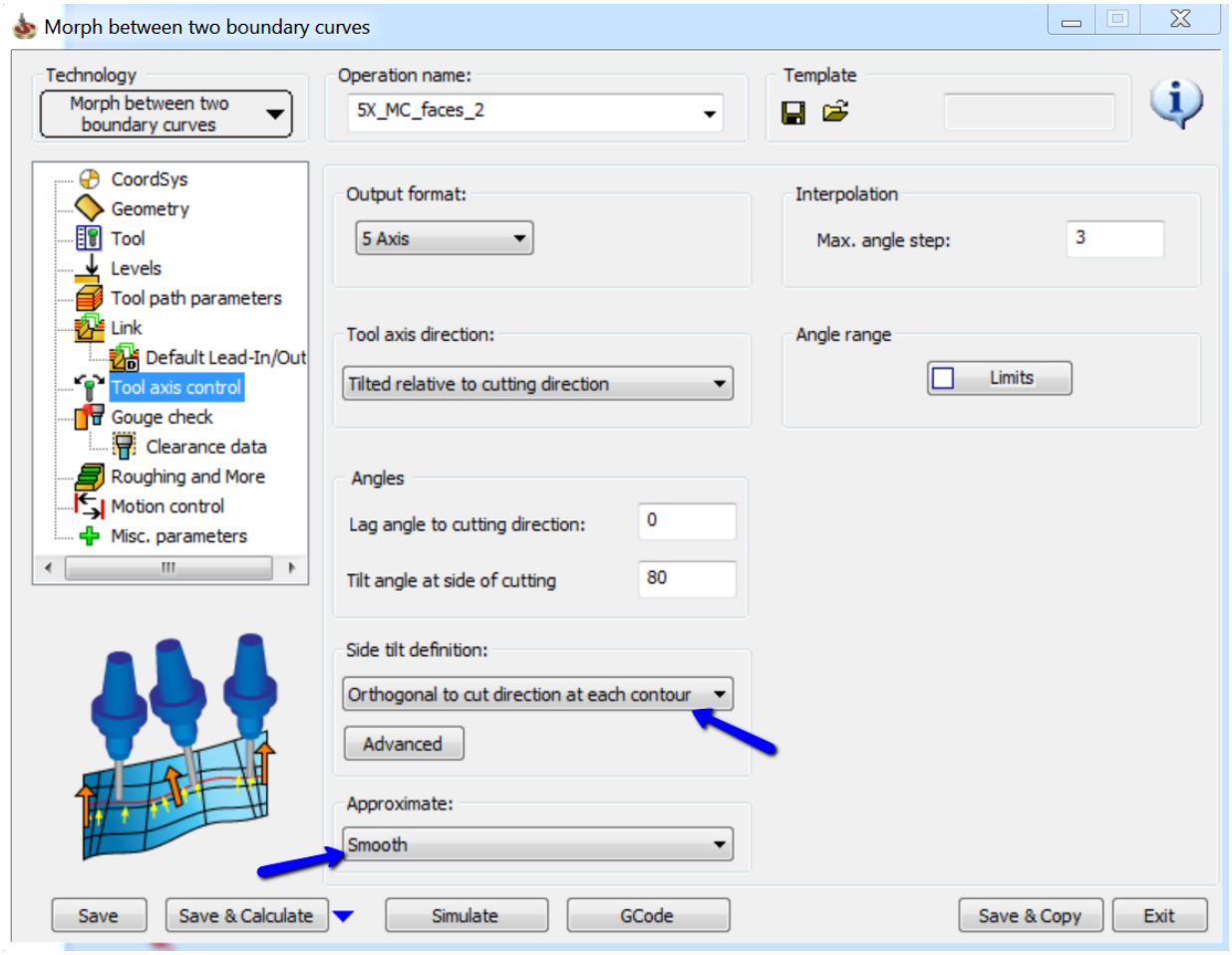
Right click > **Edit** the pasted tool path.



Click **Tool axis control**.



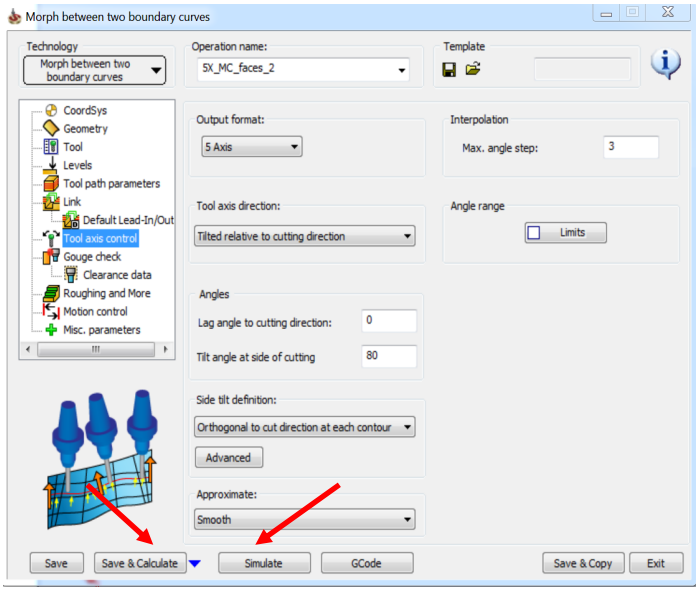
Select **Orthogonal to cut direction at each contour** as the **Side tilt definition**.



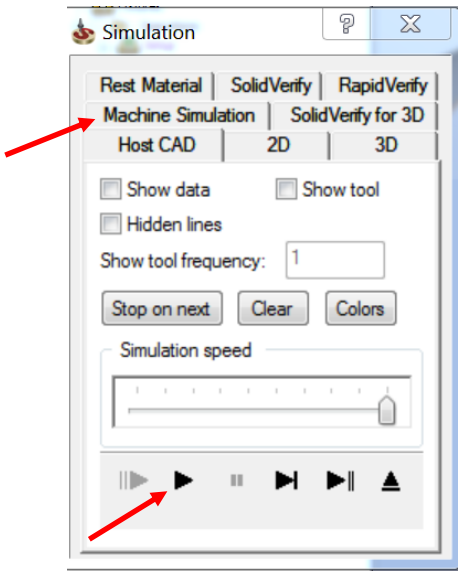
Select **Smooth** in the **Approximate** section.

Click **Save & Calculate**.

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

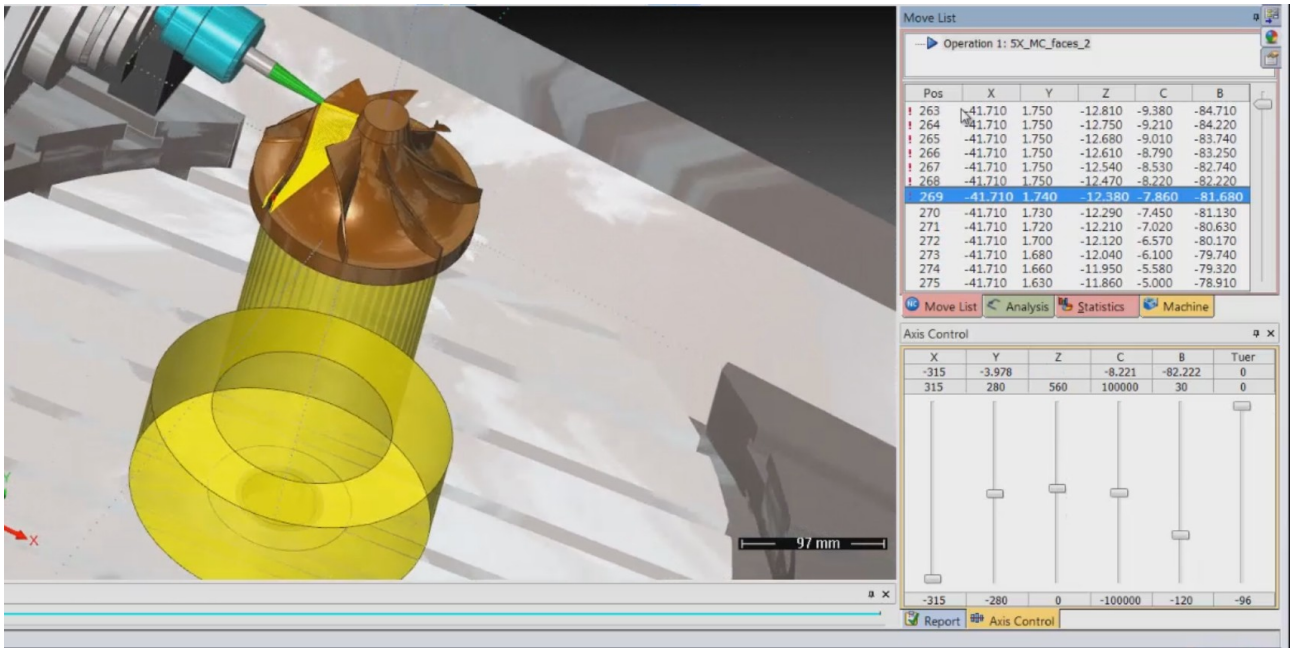


Click the play icon.



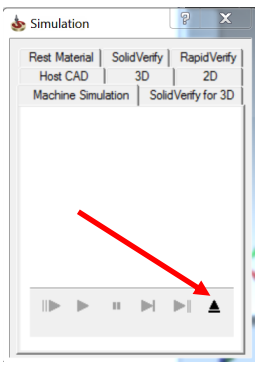
Click **Machine Simulation**.

The simulated tool path looks like this:



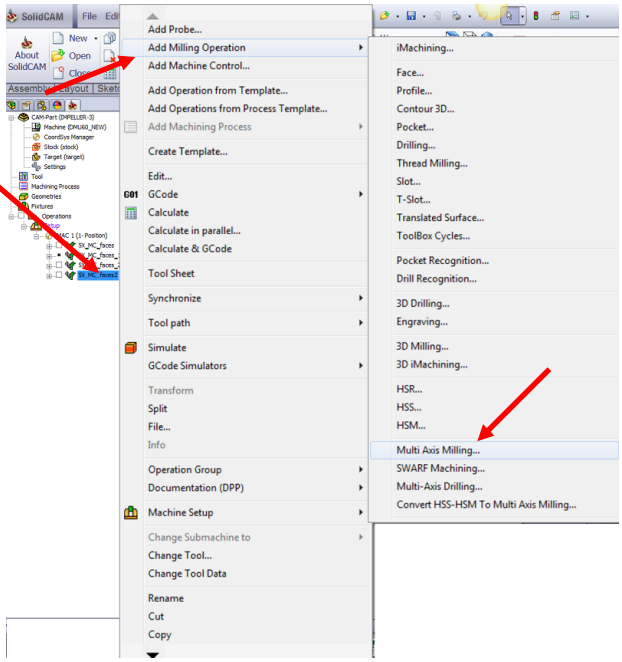
The tool maintains orthogonal position to each contour. This is because of the chosen side tilt definition i.e. orthogonal to cut direction at each contour. Observe the tool **"Dance"** on the front fillet as the tool is maintaining it's position orthogonal to the contour.

Select the exit icon to come out of machine simulation.

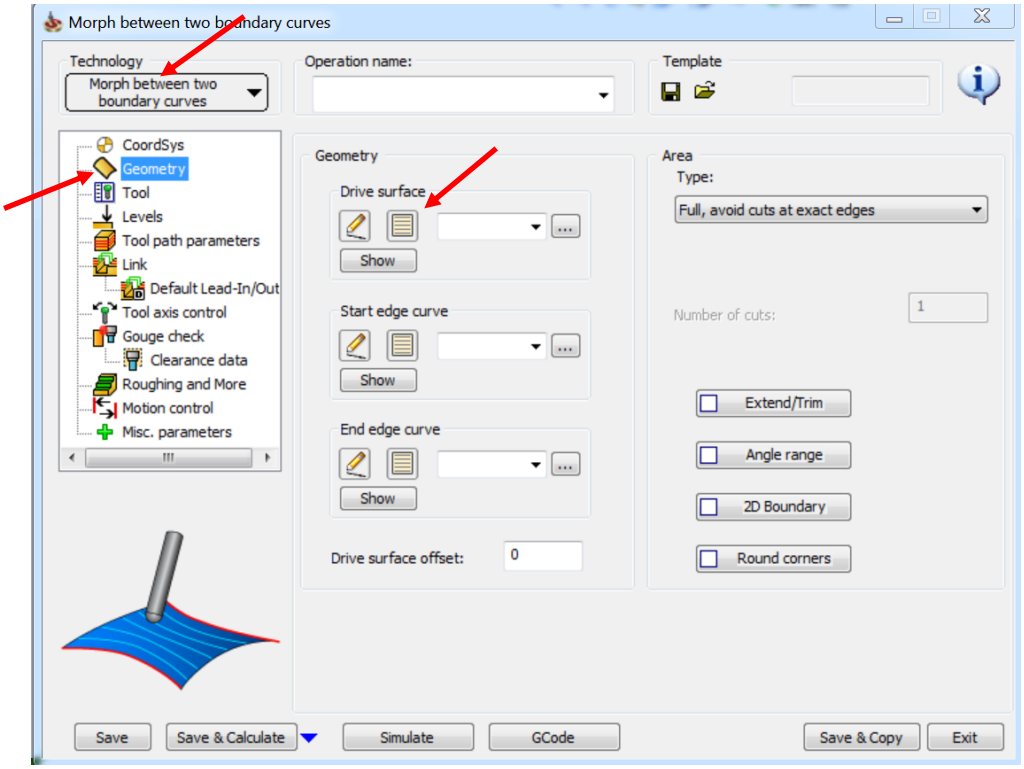


Right click the toolpath.

Click **Add Milling Operation** > **Multi Axis Milling**.



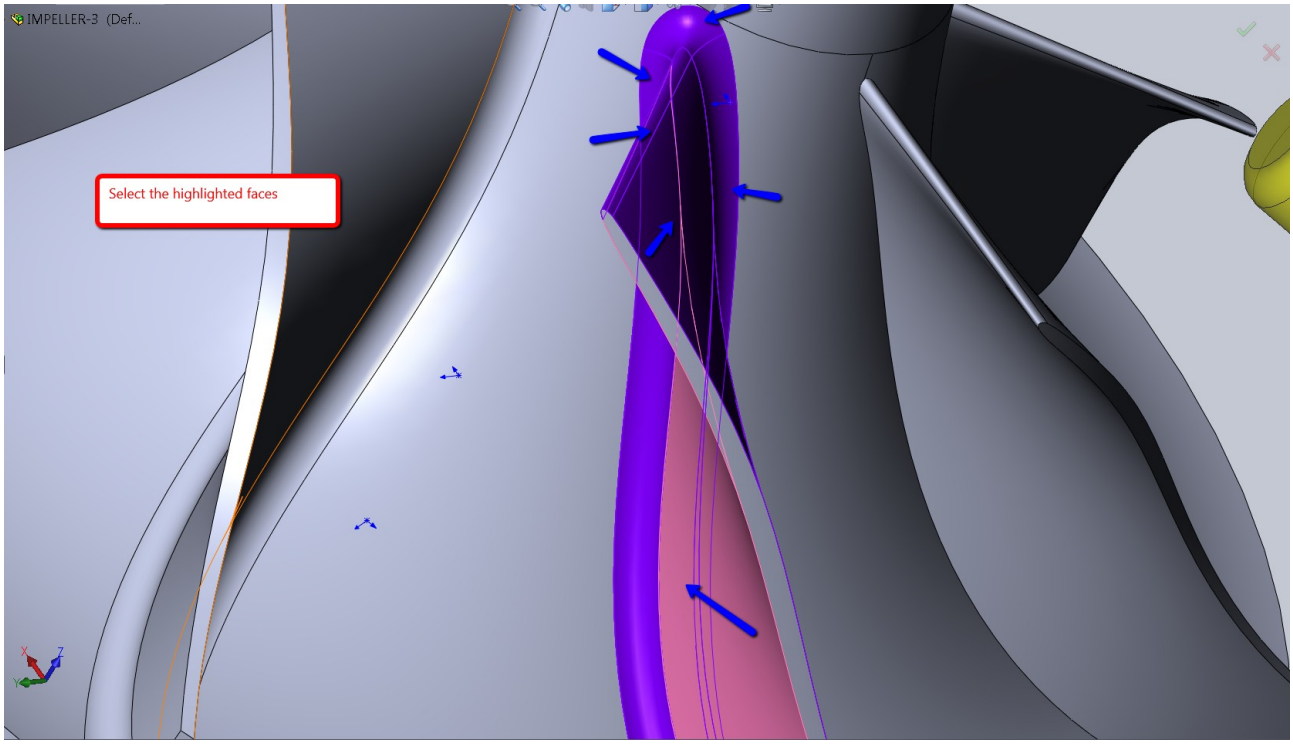
Select **Morph between two boundary curves** in the **Technology** section.



Click **Geometry**.

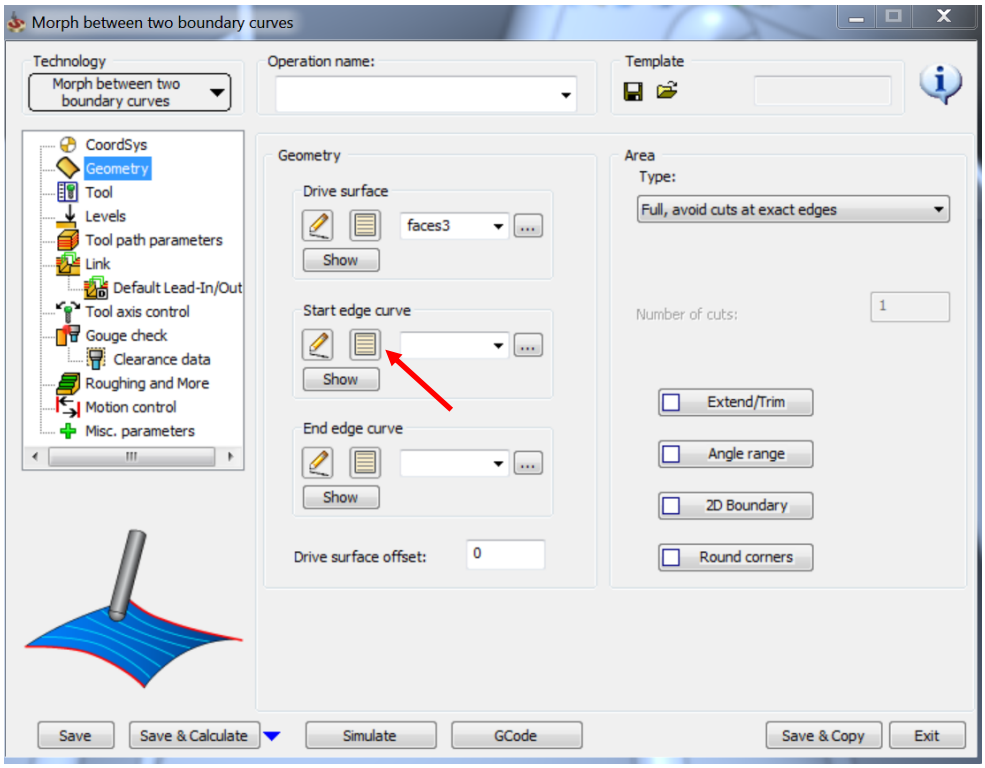
Click the new icon in the **Drive Surface**.

Select the faces as shown in the image.

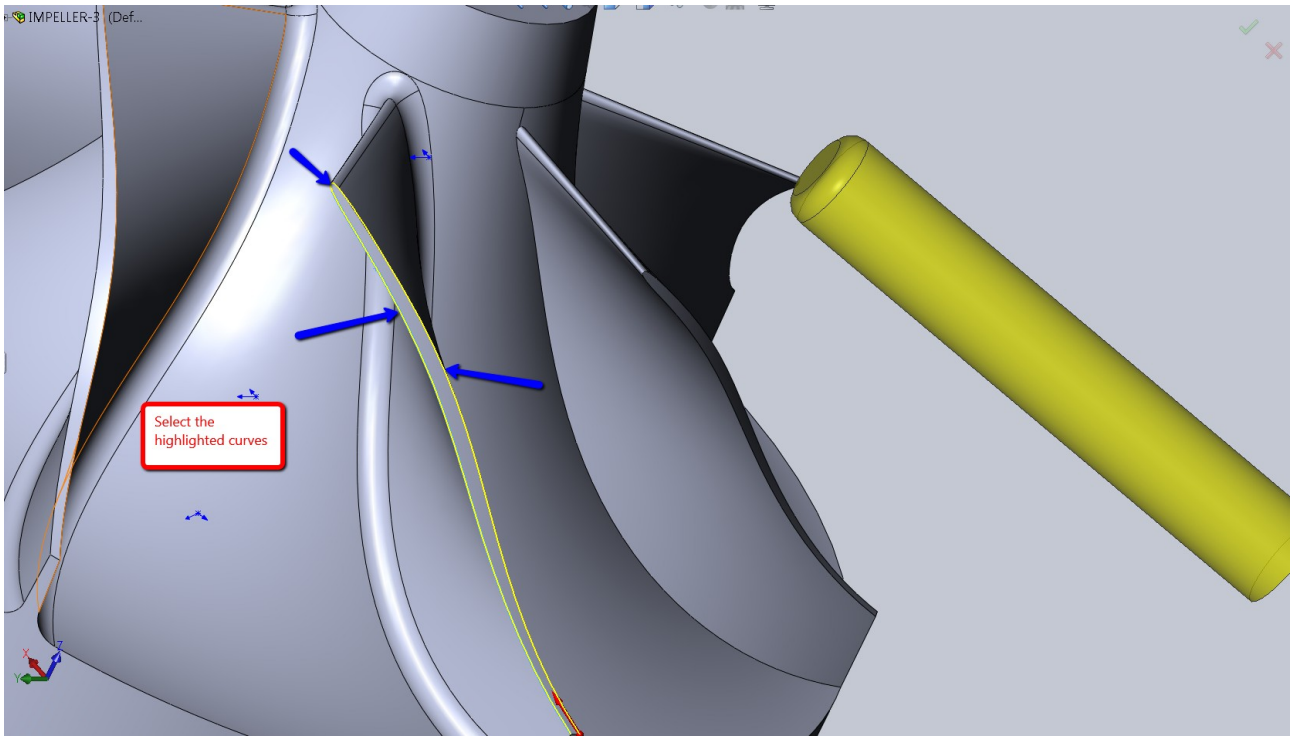


Click 

Click the new icon in **Start edge curve** section.



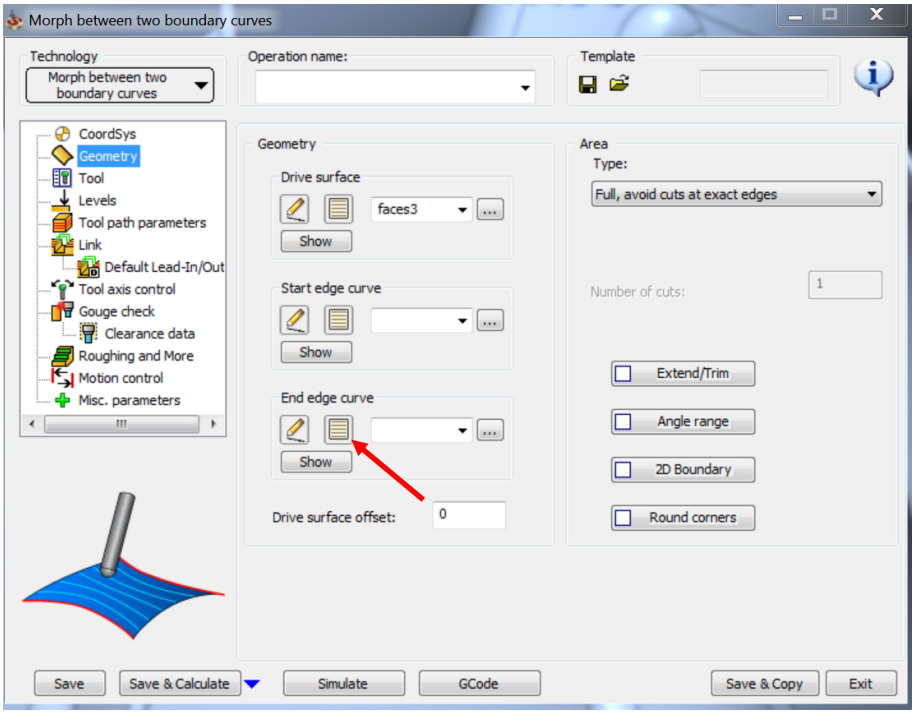
Select the curves as shown in the image:



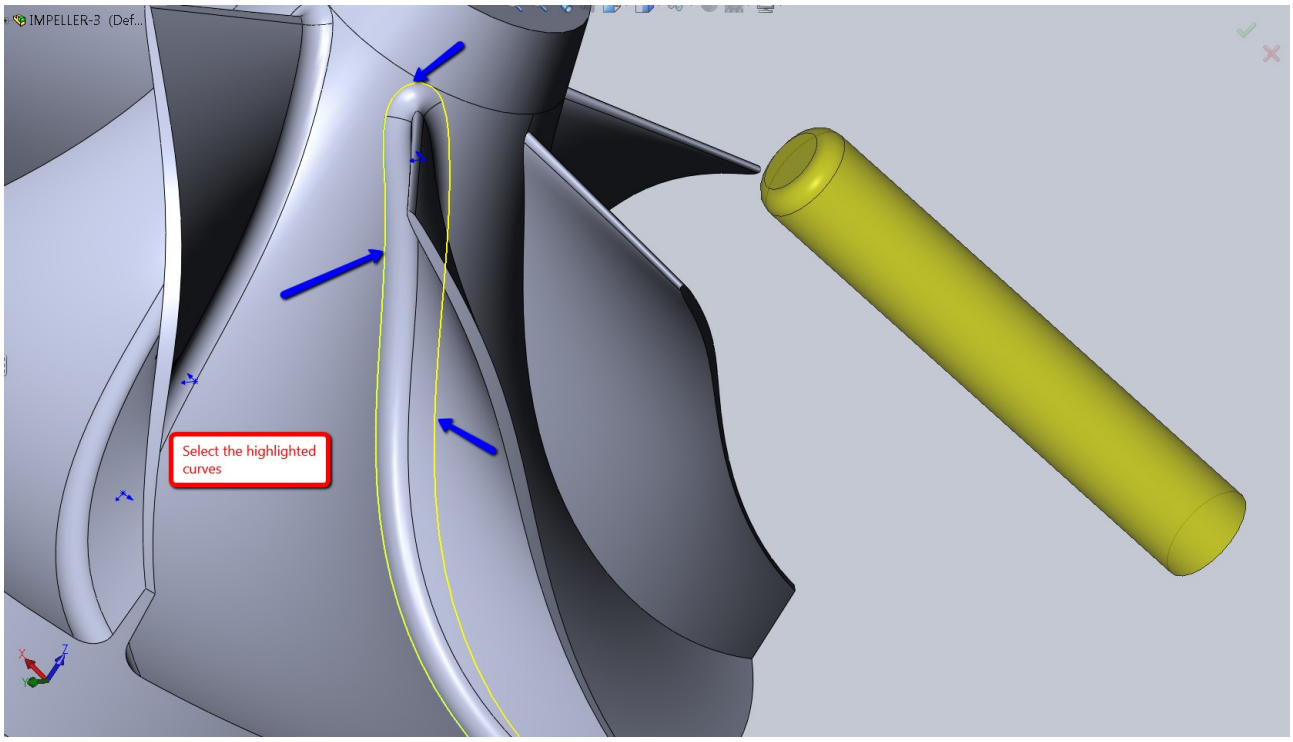
Click 

Click 

Click the new icon in the **End edge curve** section.



Select the curves (Bottom Fillet Edges) as shown in the image.

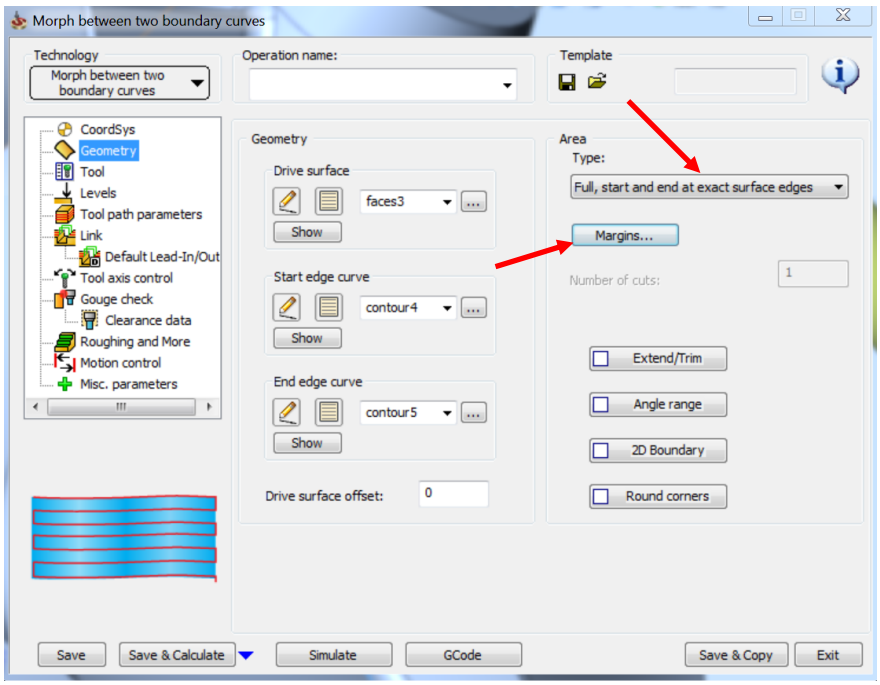


Click 

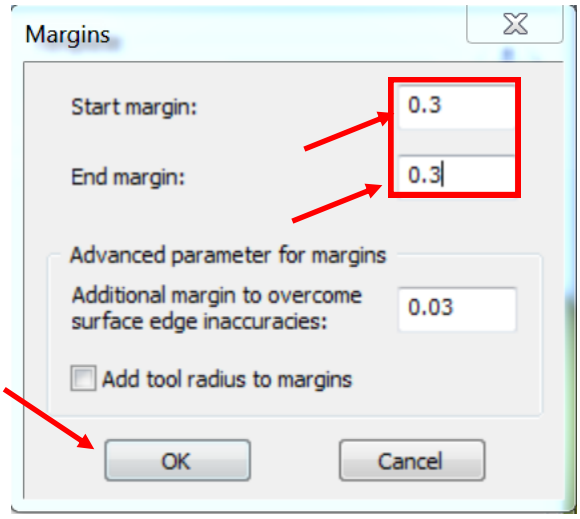
Click 

Select the **Type** as **Full, start and end at exact surface edges** in the **Area** section.

Select **Margins**.

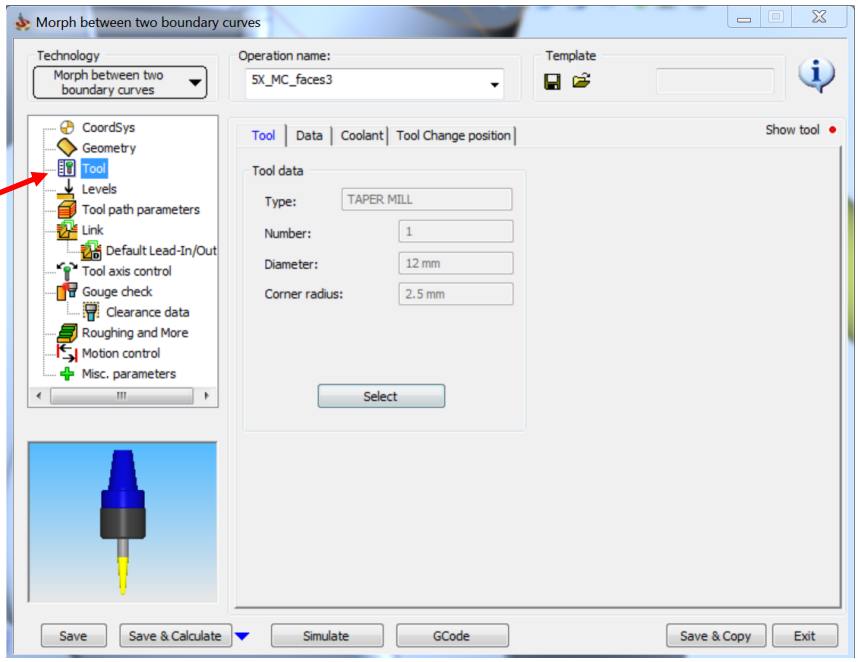


Enter the start and end margin values as shown in the image:



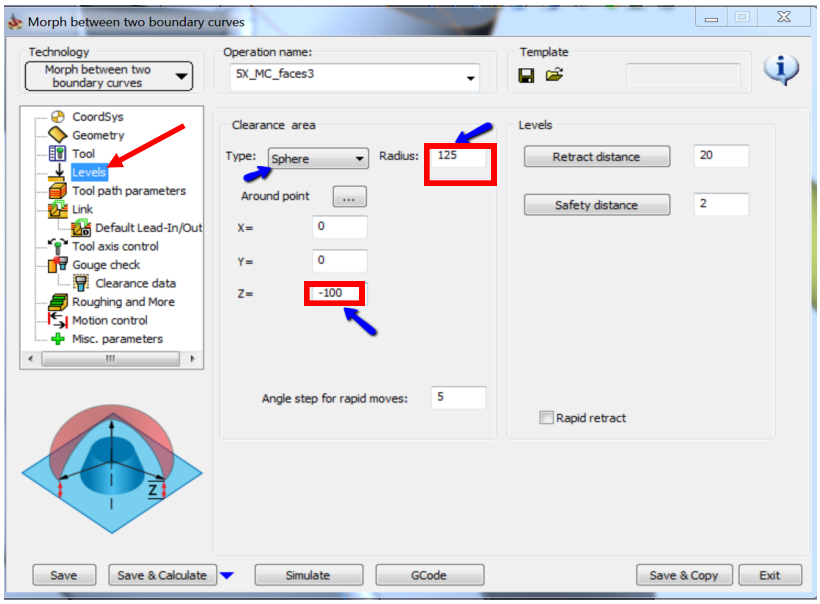
Click **OK**.

Click **Tool** and select the same tool (Taper Mill) that was used earlier.

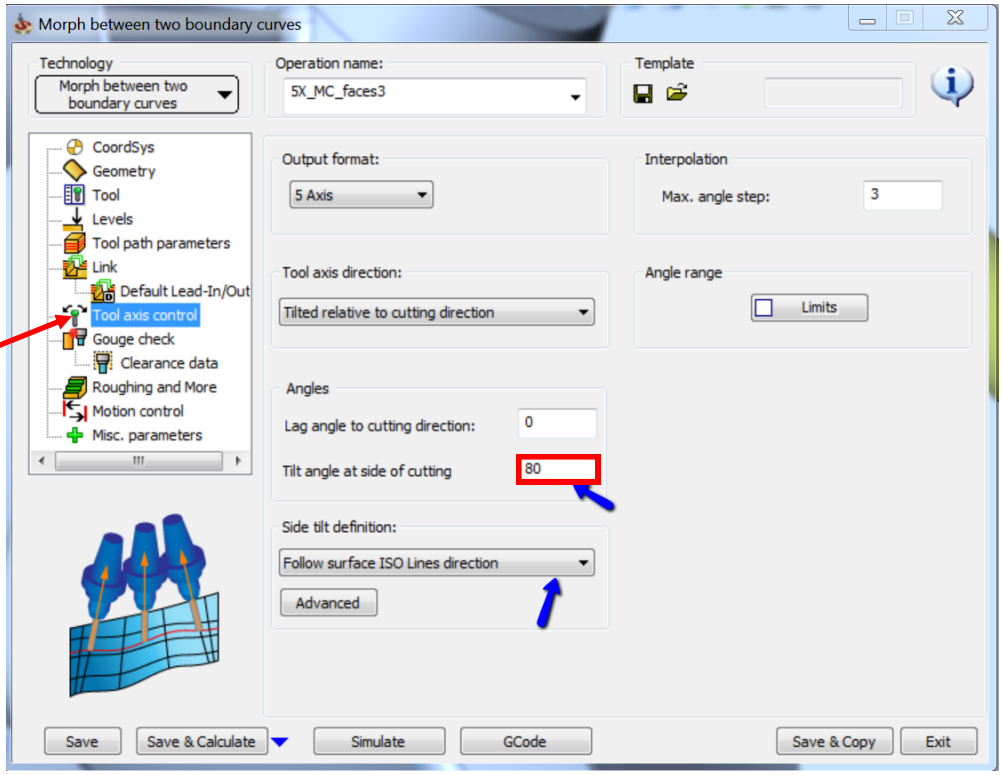


Click **Levels**.

Select “Sphere” as the clearance type and enter the values as shown in the image.



Click **Tool axis control**.



Enter a value of 80 as the **Tilt angle at side of cutting**.

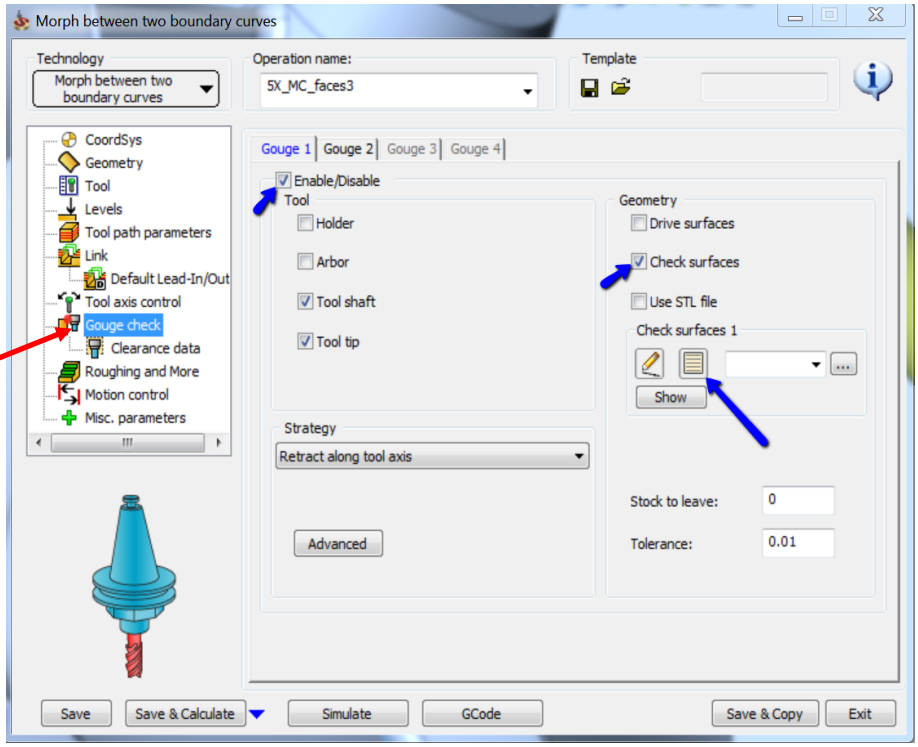
Select **Follow surface ISO Lines direction** as the **Side tilt definition**.

Click **Gouge check**.

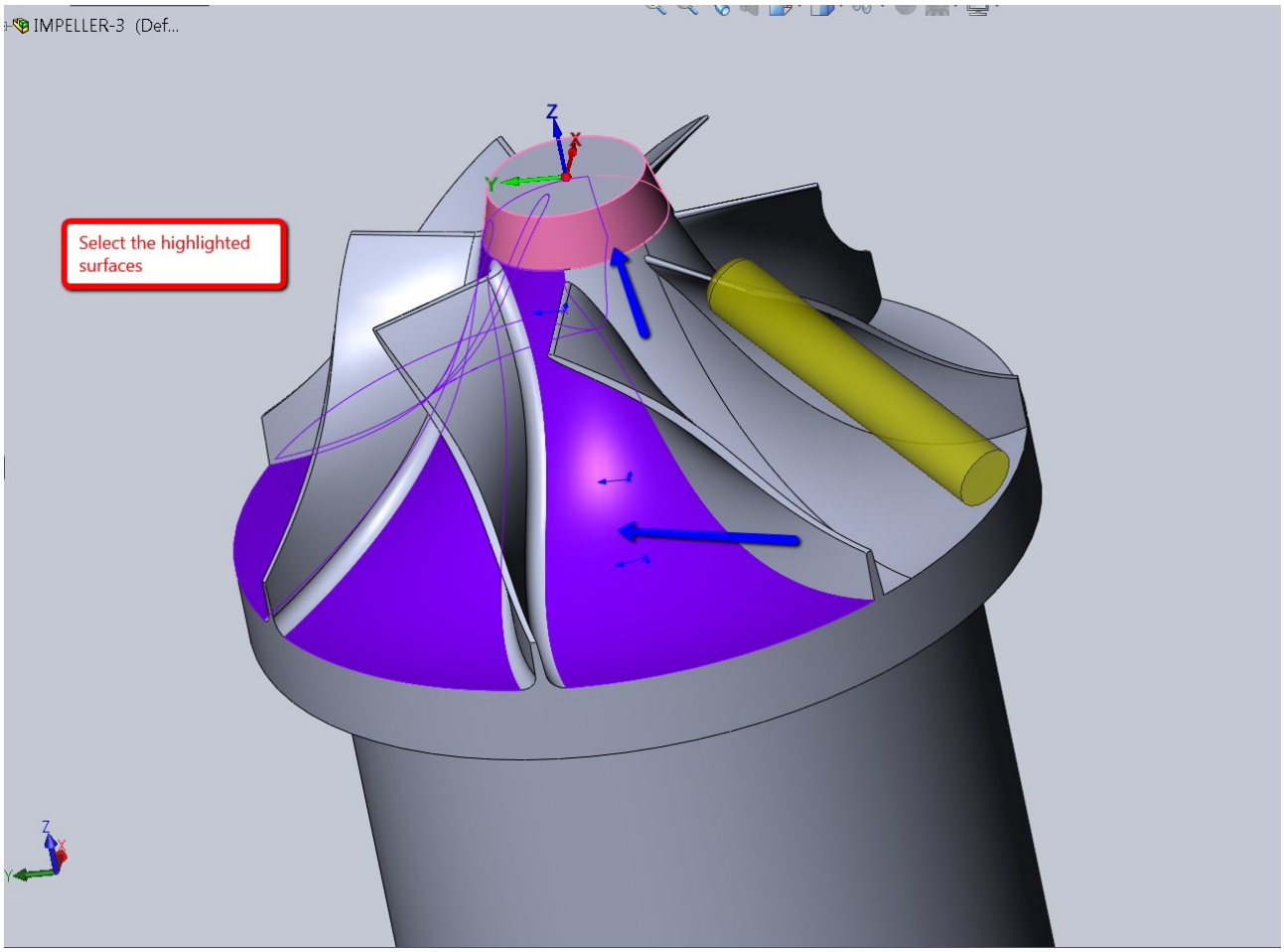
Select the **Enable/ Disable** check box.

Select the Check surfaces checkbox.

Click the new icon under **Check surfaces 1** section.



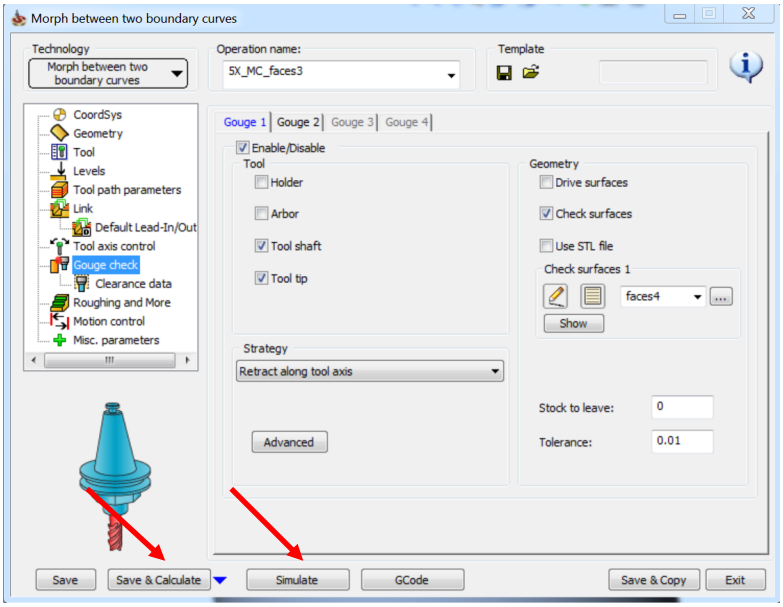
Select the surfaces as shown in the image:



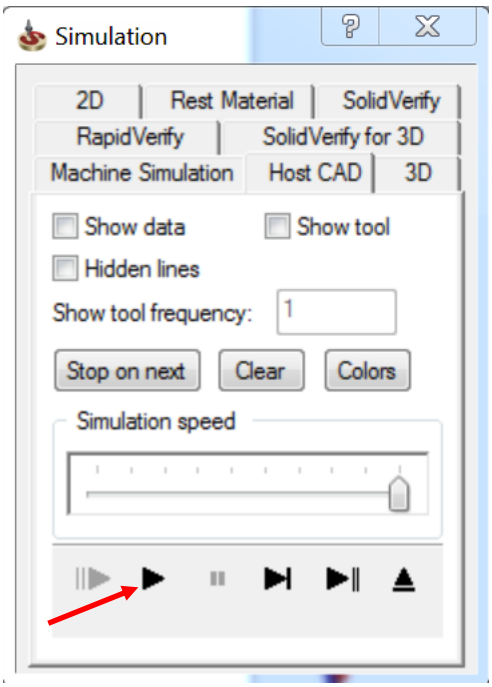
Click 

Click **Save & Calculate**.

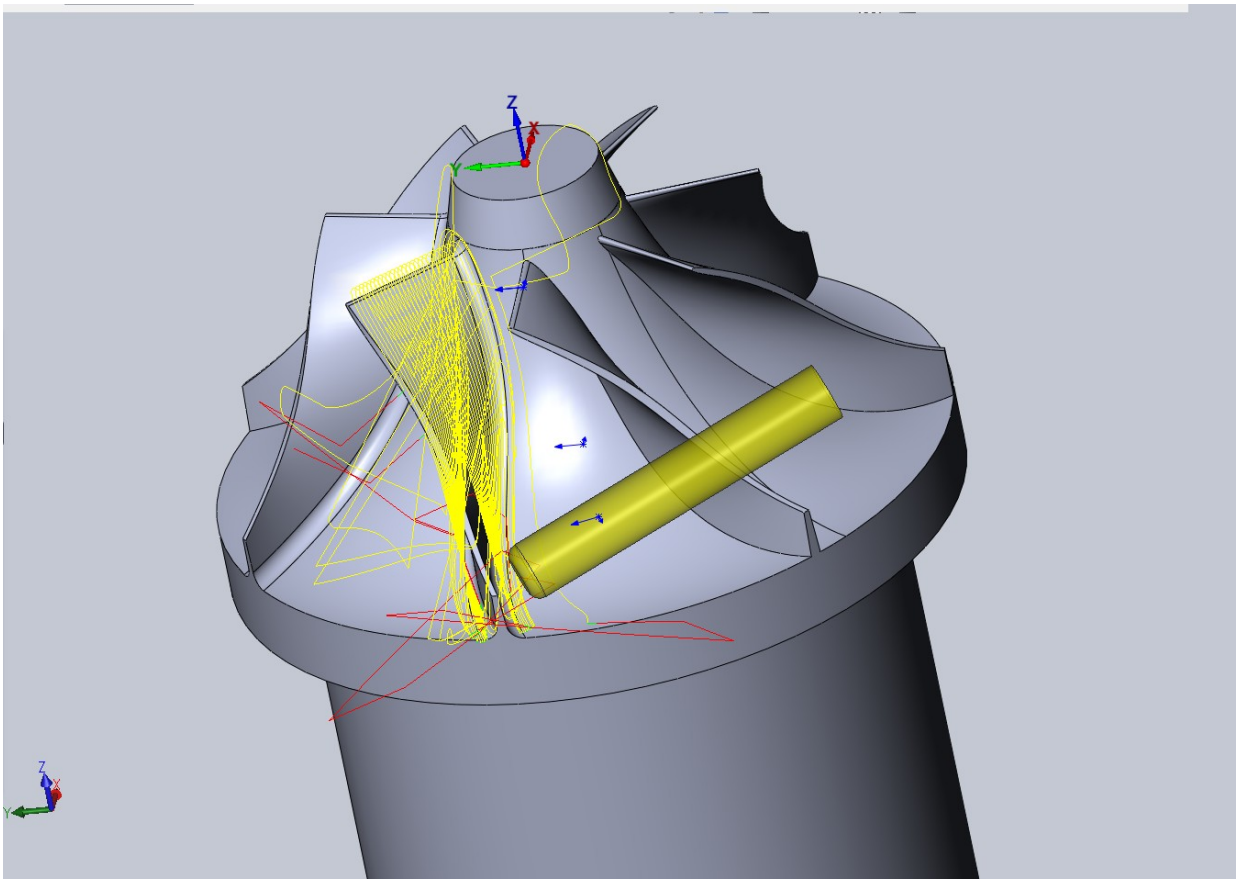
Click **Simulate** once the tool path is calculated.



Click the play icon.



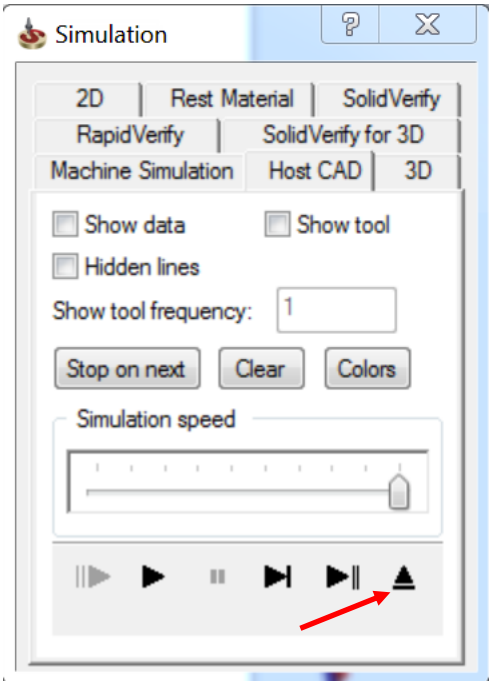
The tool path looks like this:



We can see that the toolpath really looks very bad as there are gouges , wrong retracts etc. We need to fix this issue.

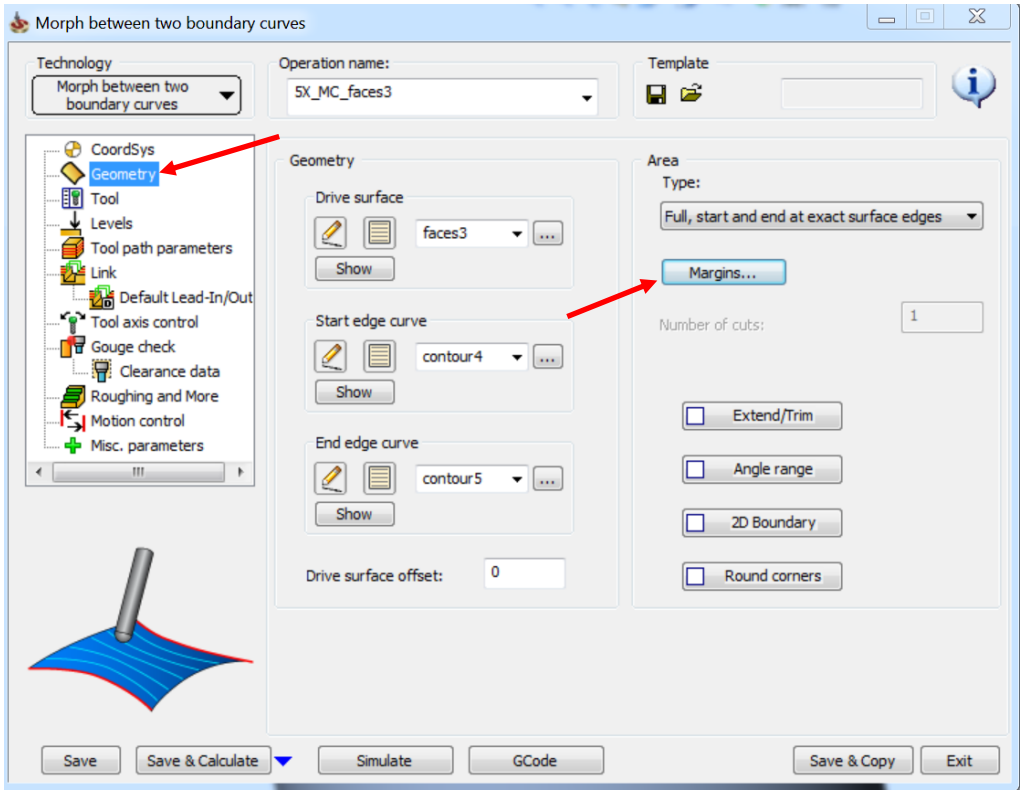
Let us go back to the Operations Manager and fix the issues in the tool path.

Select the exit icon to go back to the Operations Manager.

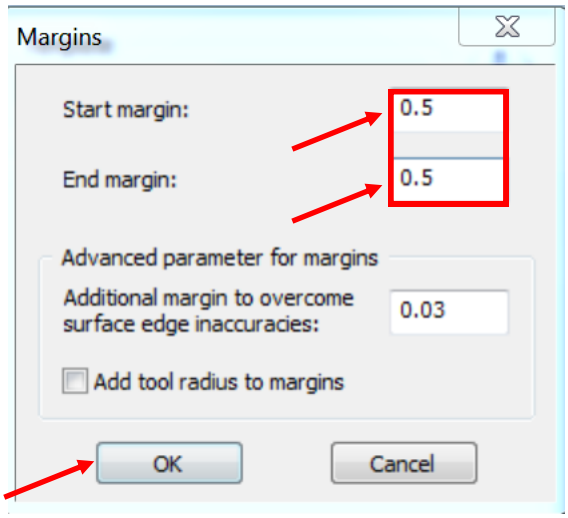


Click **Geometry**.

Click **Margins**.



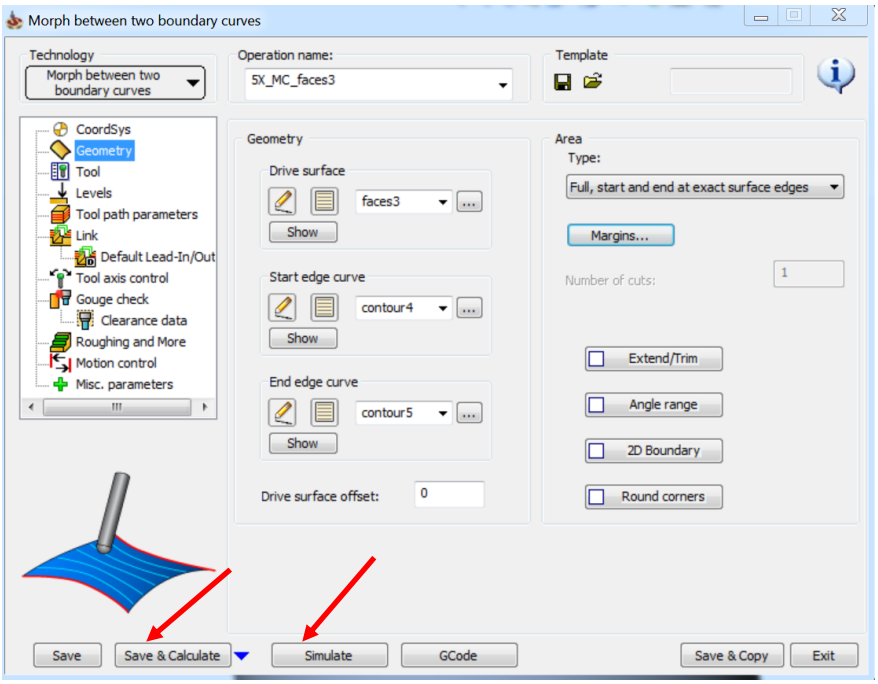
Change the start and end margins as shown in the image:



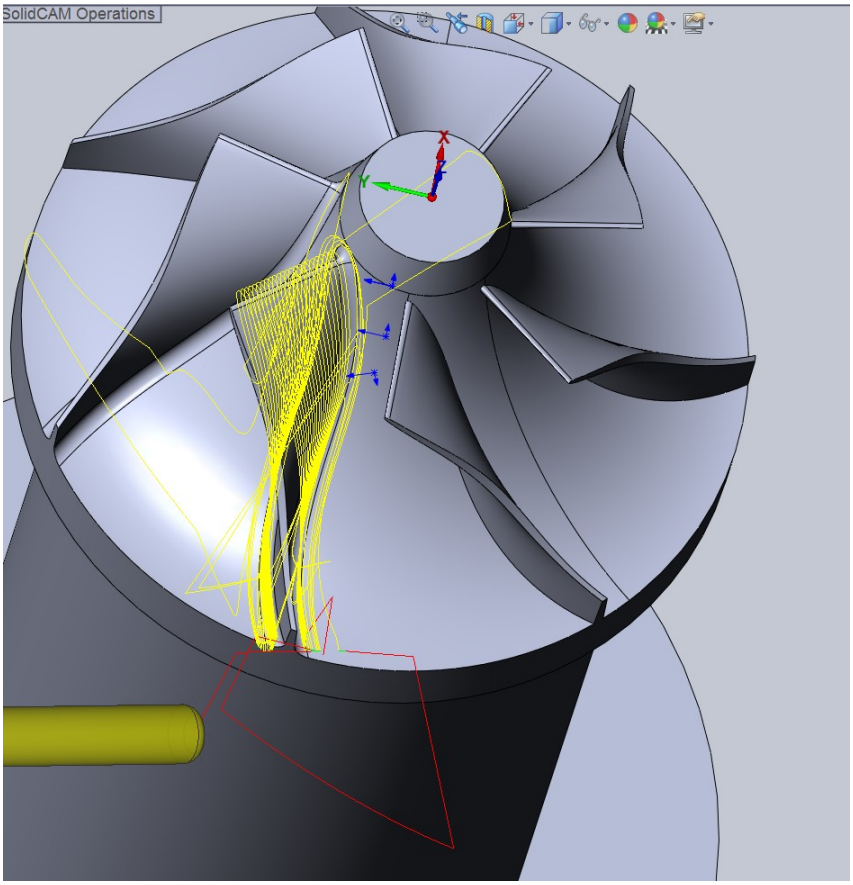
Click **OK**.

Click **Save & Calculate**.

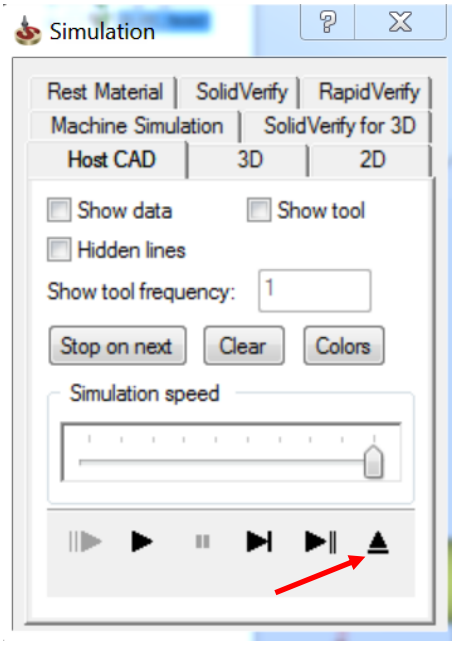
Click **Simulate** once the tool path is calculated.



The simulated tool path looks the same as there is not much change in the quality of the toolpath. We need to understand as to what is causing these issues.



Click the exit icon to return to the Operations Manager.

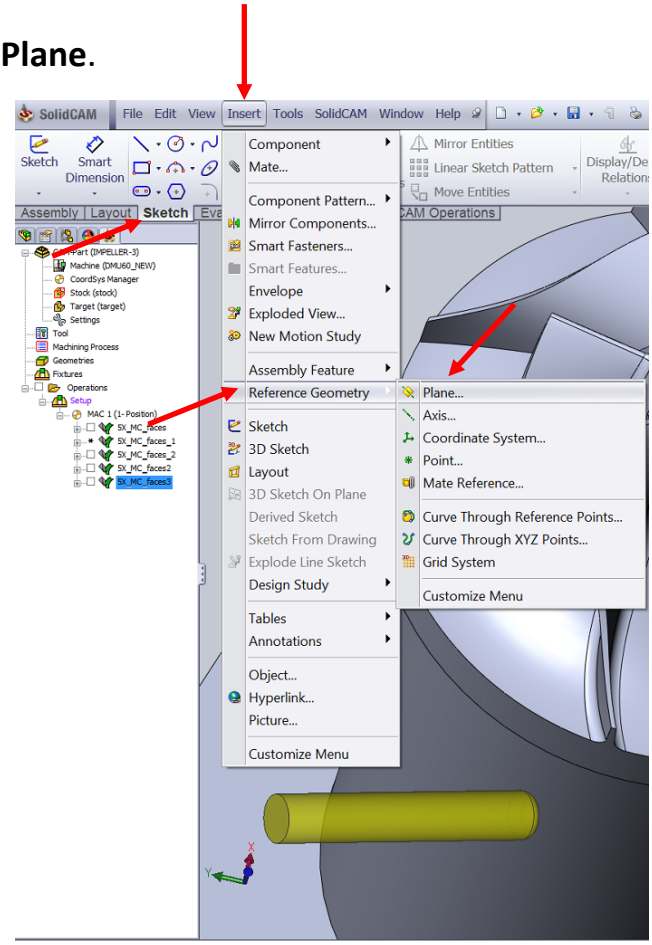


Minimize the Operations Manager.

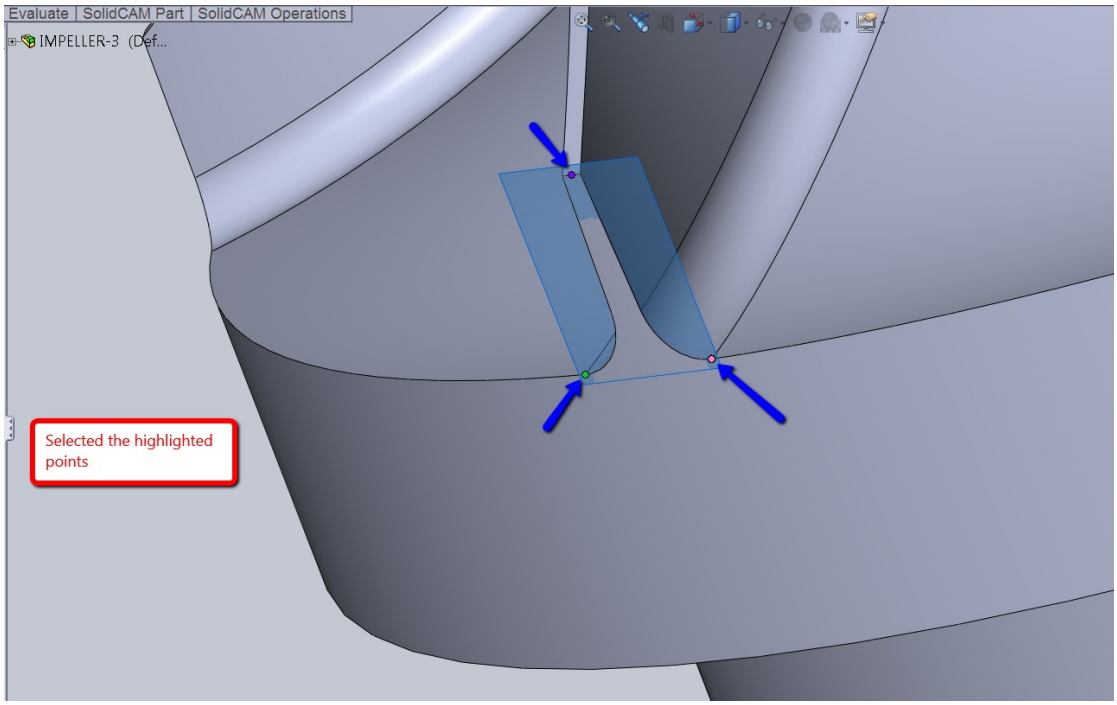
The tool path is fine till the fillet comes up. To understand and evaluate it better, let us create a sketch.

Click **Sketch**.

Click **Insert** > **Reference Geometry** > **Plane**.



Select the highlighted points.

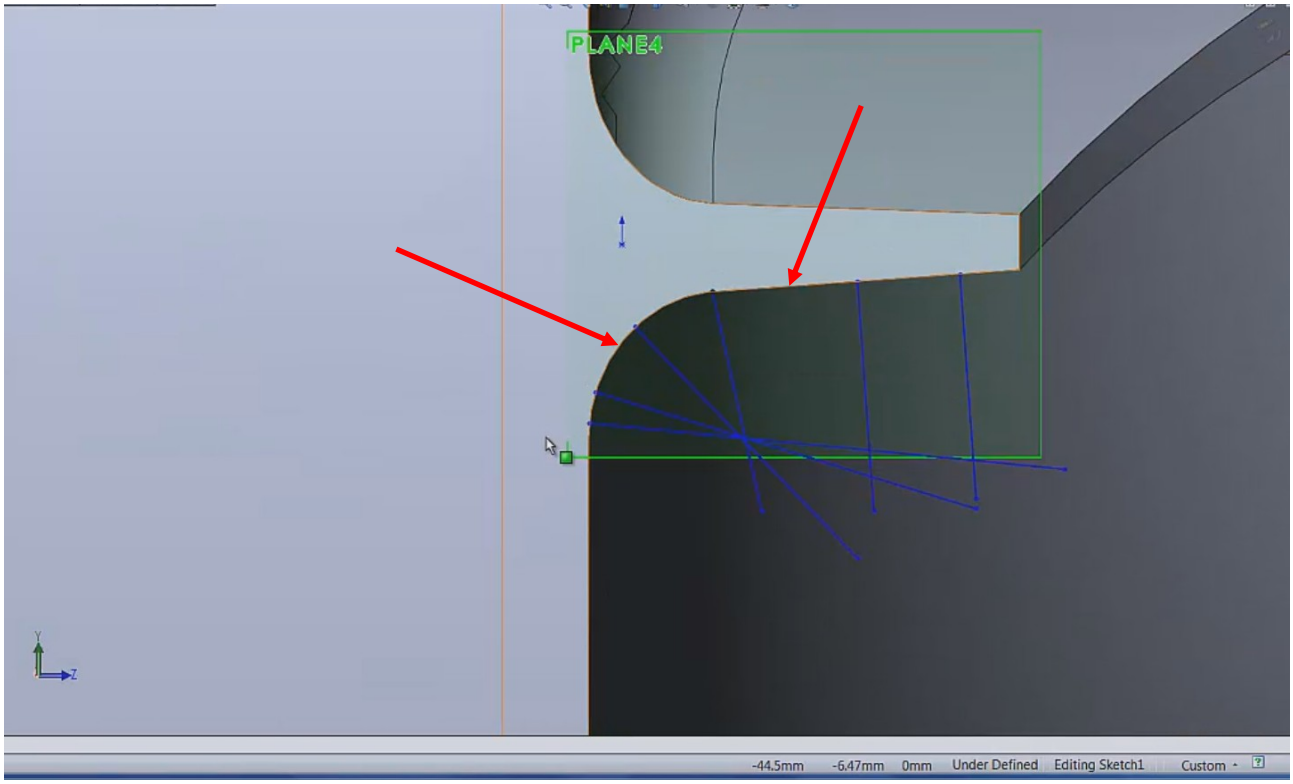


Click 

Click 

to create a sketch.

Let us make a sketch as shown in the image:

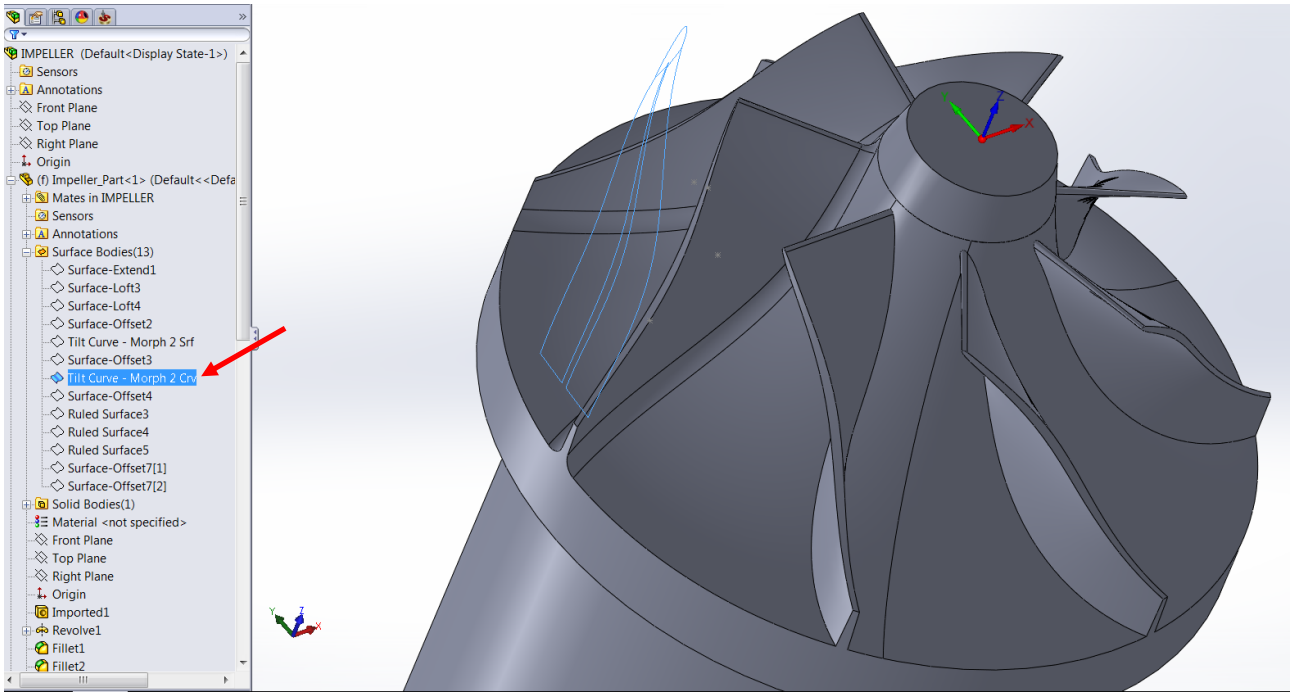


The lines in the sketch must be created nearly normal to the edge and the fillet as marked with the arrows. These lines represent the normal direction of the surfaces. You can see that the normal angle changes according to the curvature of the surface. Since we had kept the tool axis control at 80 degrees, the tool will always maintain 80 degrees to the lines that we just created, which is resulting in a bad toolpath. Thus, we need to use a different tilt axis control. Whenever we get a blade with a bottom fillet, the below mentioned tilt strategy holds good.

Click  to discard the changes to the sketch.

Click the feature manager design tree icon. 

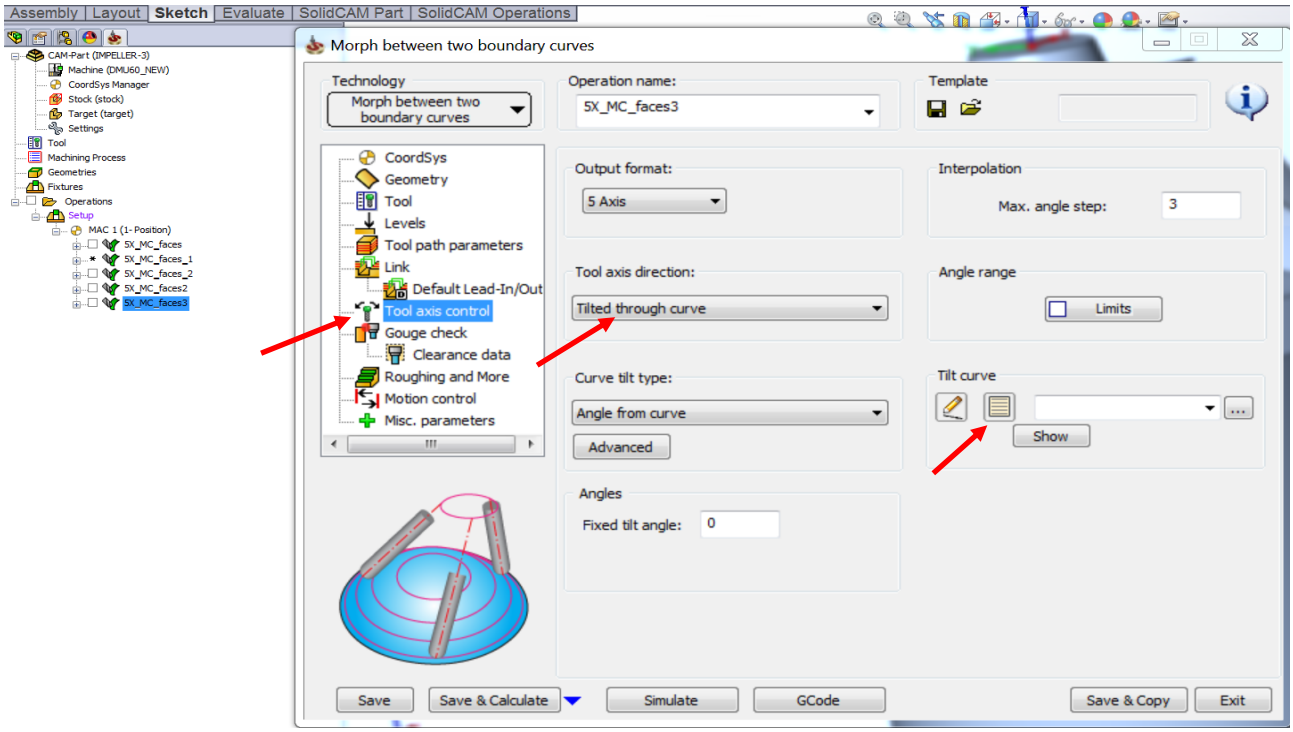
Select **Tilt Curve – Morph 2 Crv** (Under Surface Bodies of Impeller_Part)



Switch on the surface by clicking 

Go back to the SolidCAM Manager to edit the tool path.

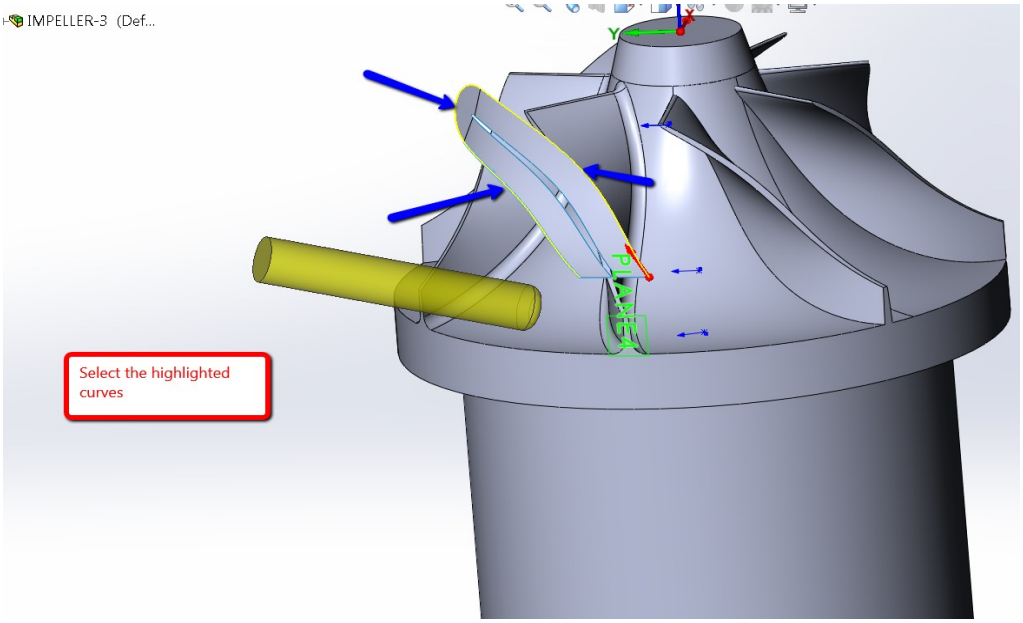
Click **Tool axis control**.



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

Click new icon in the **Tilt curve** section.

Select the curves as shown in the image:

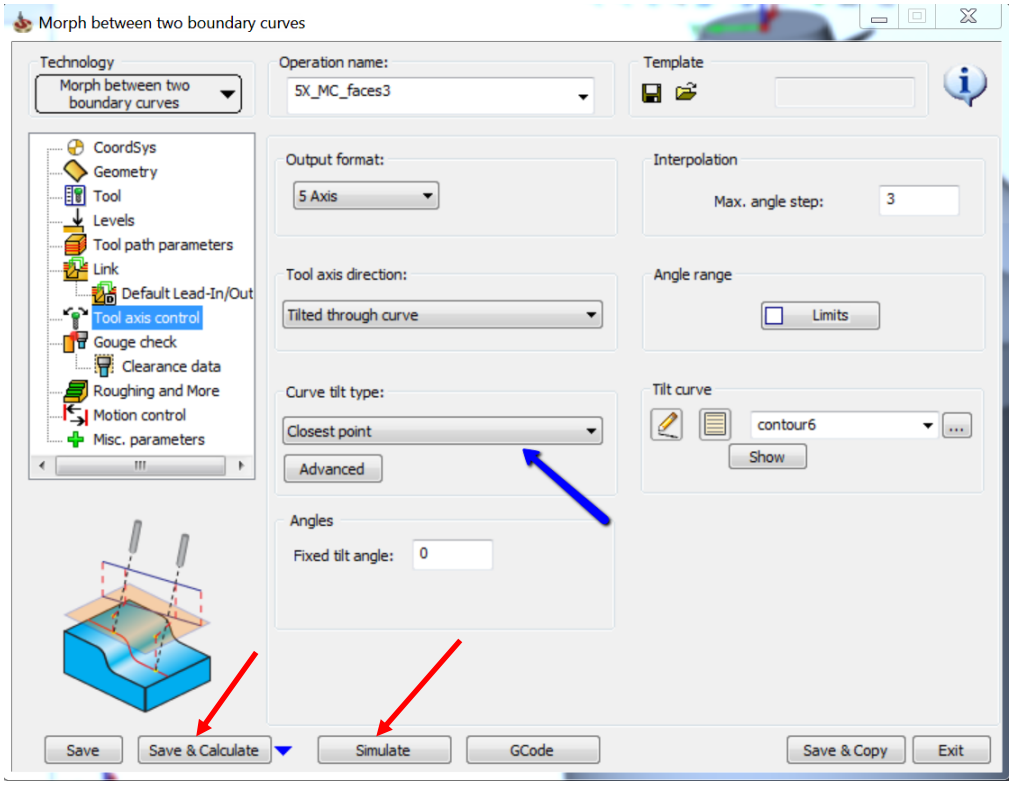


Note: For information on how to create the surface, refer to the movie attached with this tutorial.

Click 

Click 

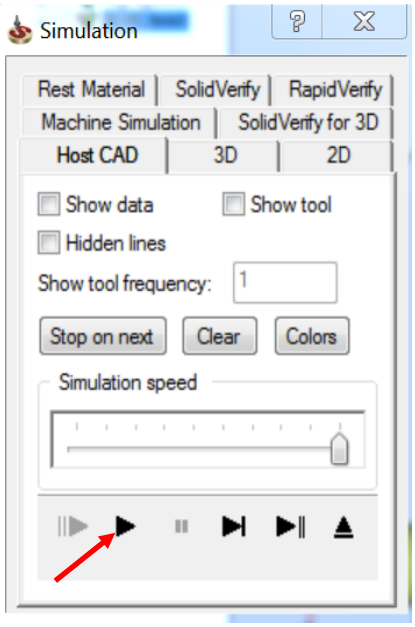
In **Tool axis control**, select **Closest point** as the **Curve tilt type**.



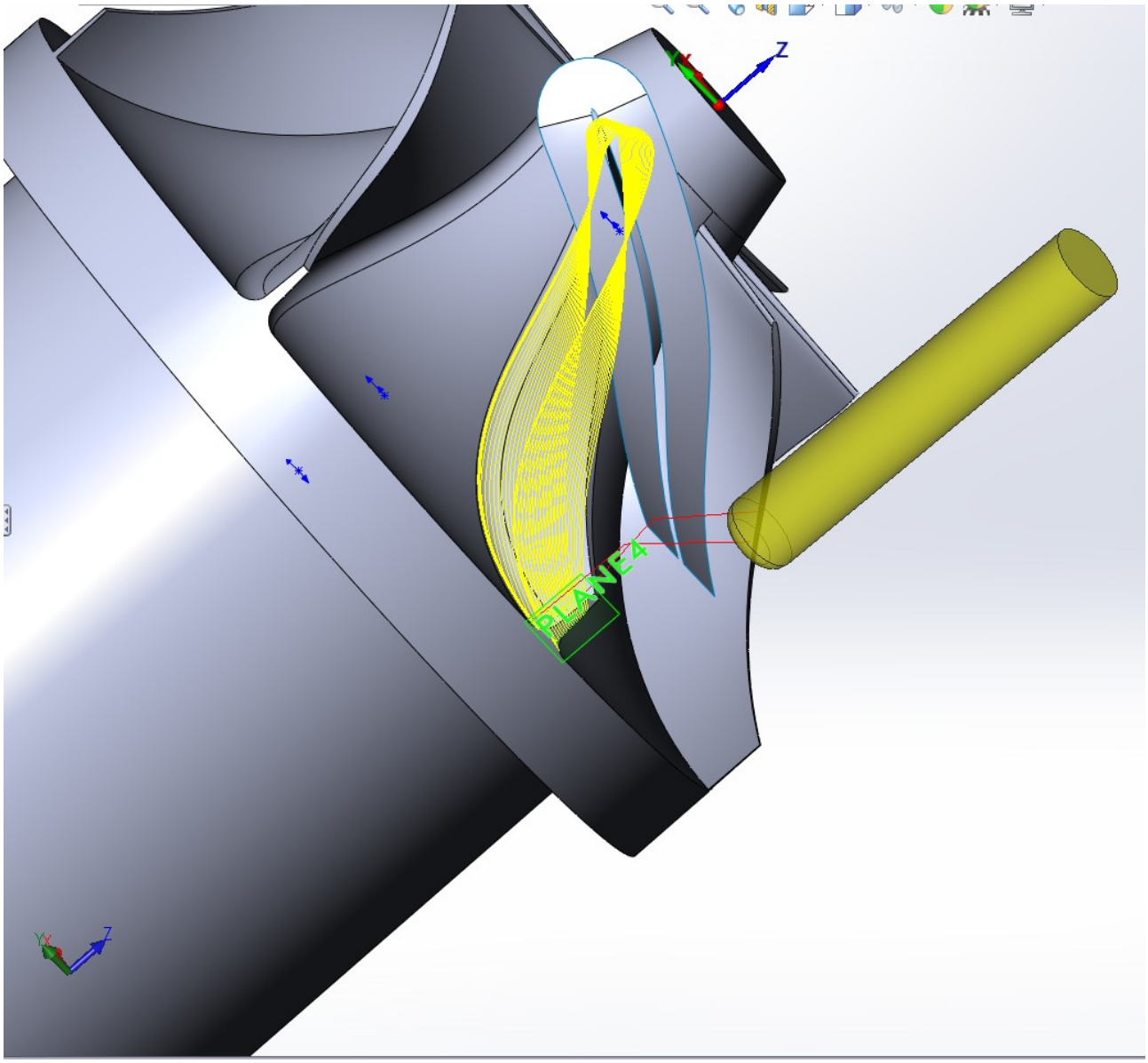
Click **Save & Calculate**.

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

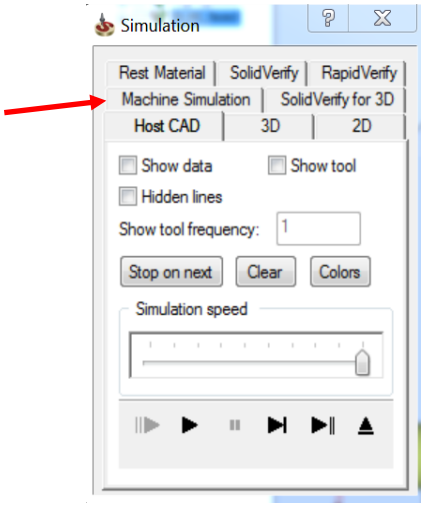
Click the play icon.



The simulated tool path looks much smoother now:



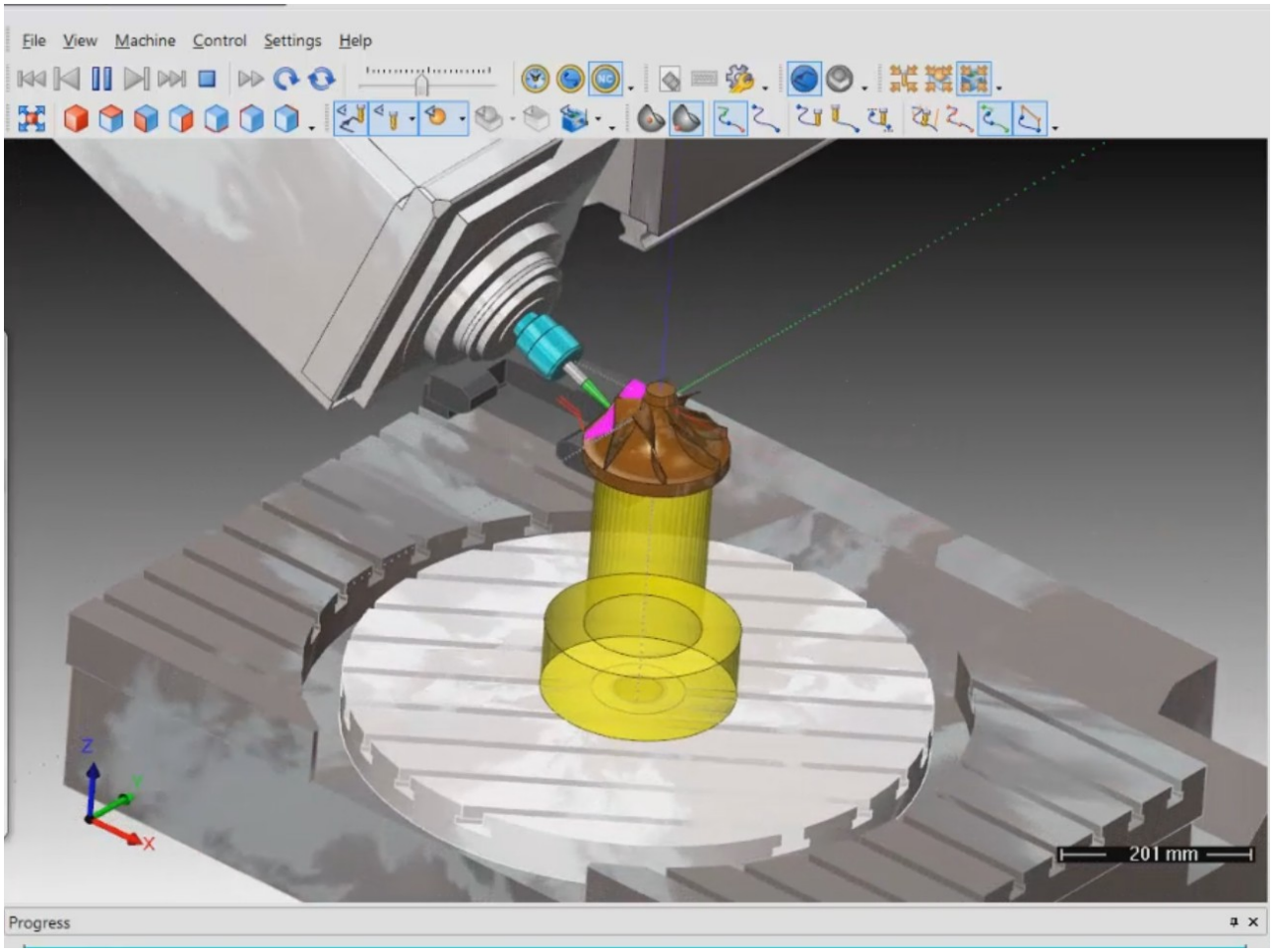
Go to the feature manager design tree and hide the created surface by clicking 



Click **Machine Simulation**.

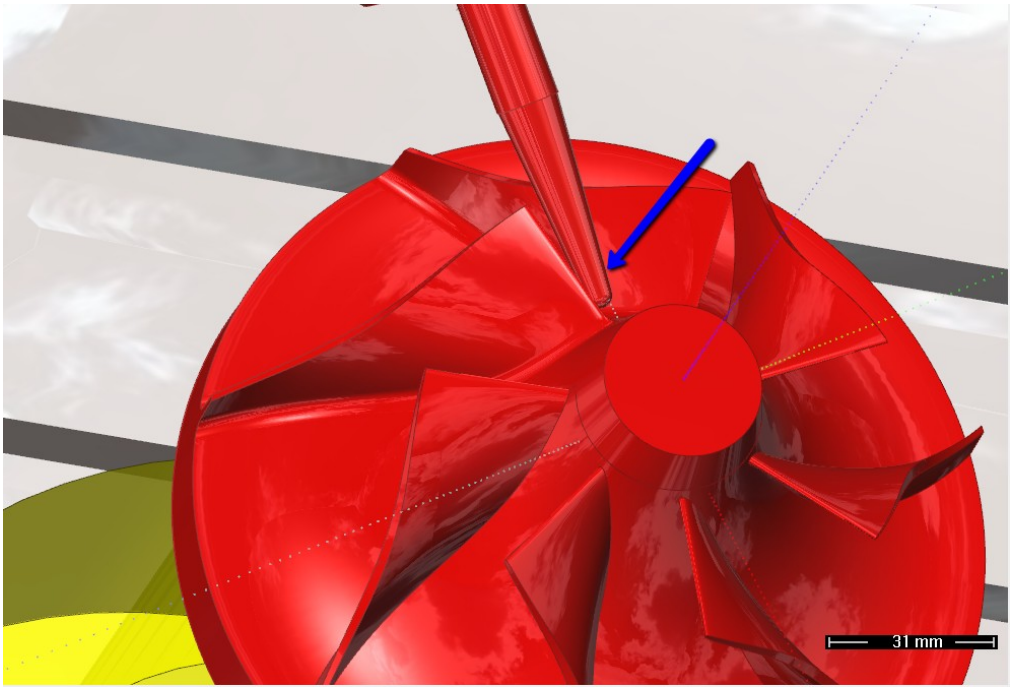
Click the play icon 

The machine simulation is much smoother now as the tool negotiates the blade surfaces & the fillet below smoothly. This type of tool tilting is very useful when negotiating surfaces with fillets or surfaces with varying curvatures which cause the normal's of the surface to change drastically.



Observe that the final 5-6 passes are gouging the part as seen below in the machine simulation. We need to fix this.

The gouge is a result of an angle change within a very short travel of the tool which is causing the tool to "touch"the part thereby causing this gouge.

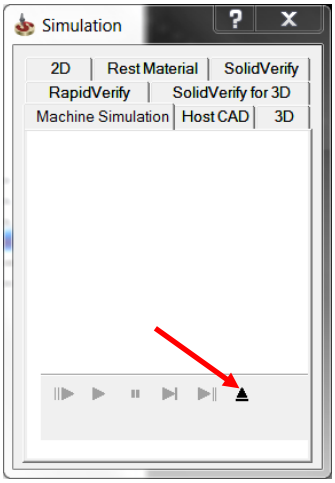


In order to fix this gouge we will now open a new parameter box and study the effect of the same.

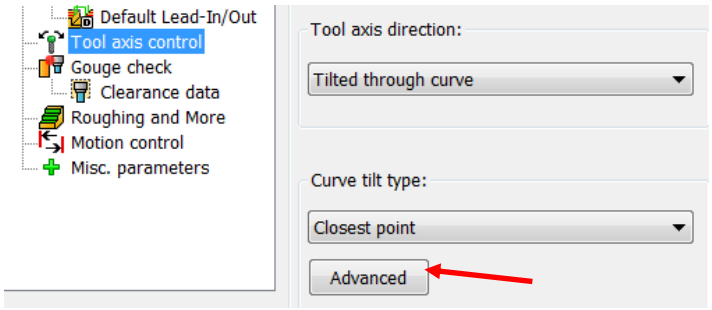
This parameter can be found in the tool axis control tab and more than often kept as default. However this parameter can have a great degree of effect in smoothing tilt motions and thereby avoiding gouges in this case.

This parameter is called “Side Tilt Fanning Distance”.

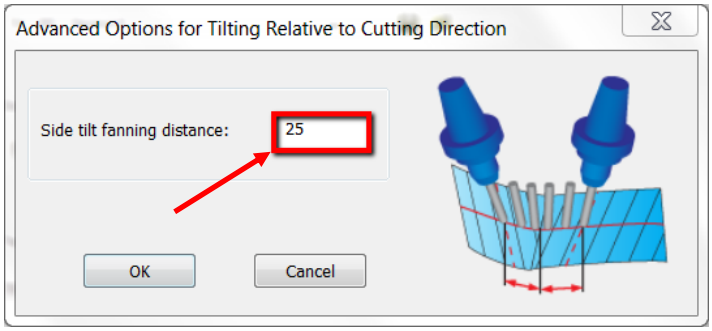
Click the exit button to return to Operation Manager



In the tool axis control tab & curve tilt type sub tab click on **Advanced.**



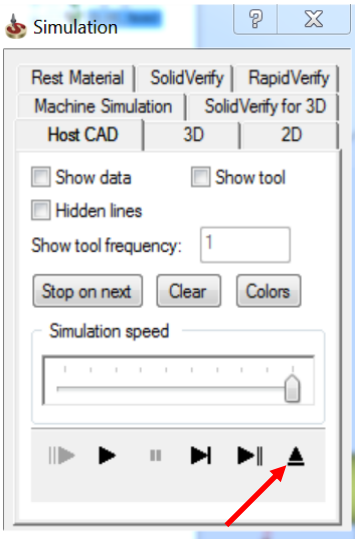
In the Side tilt fanning Distance change the value from **2 to 25** & click OK.



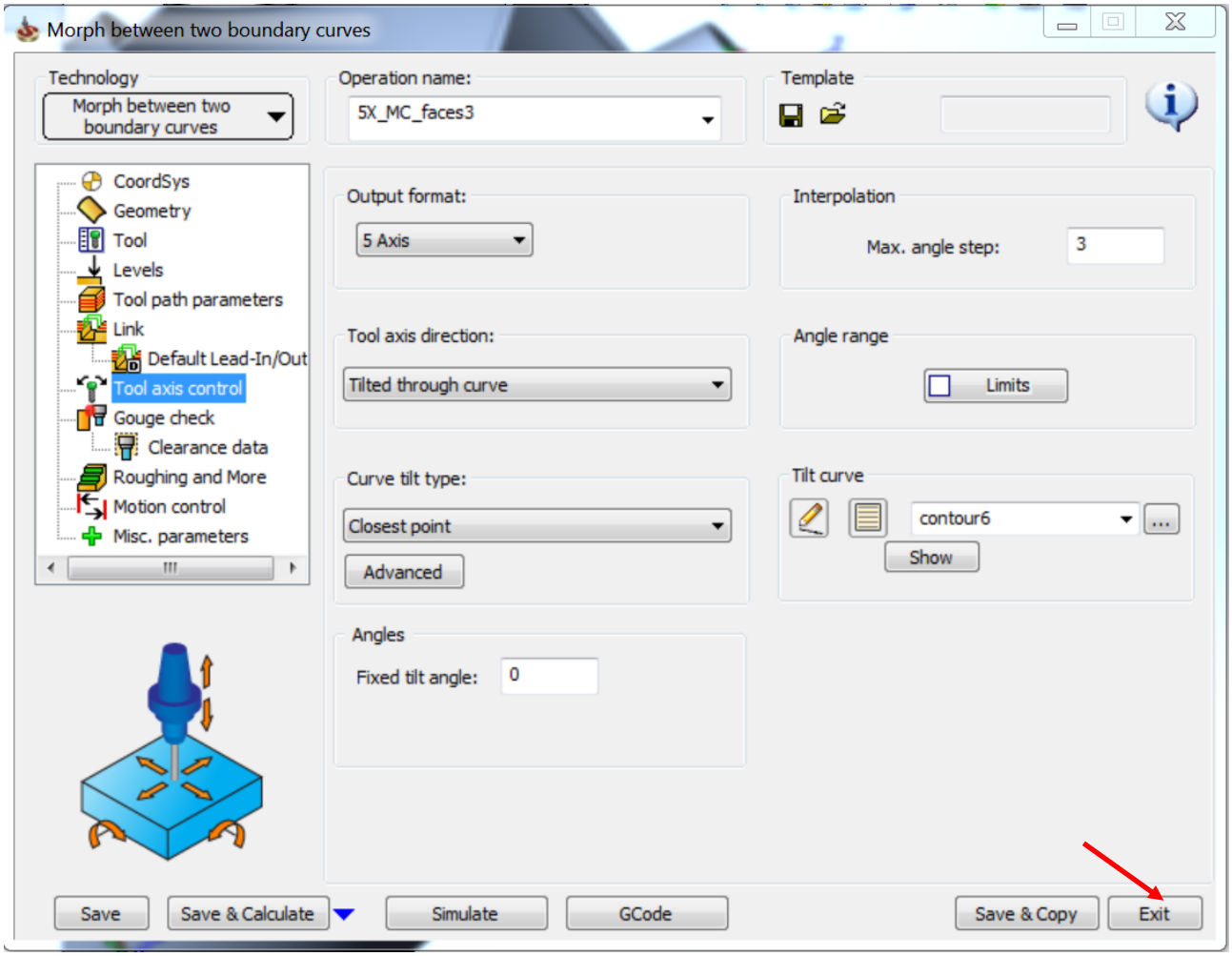
Click Save & Calculate button and after the calculation is completed click on Simulate button & select Machine Simulation (if the previous simulation was set to HostCAD). Click play and observe that there are no gouges now.

! What the “Side Tilt Fanning distance” does is to evenly distribute the changes in the tilt angles over a larger distance (25mm in our case) thereby smoothing the tilt. Thus gouges are now avoided and we get a much better tilting motion.

Click the exit icon to return to the Operations Manager.



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

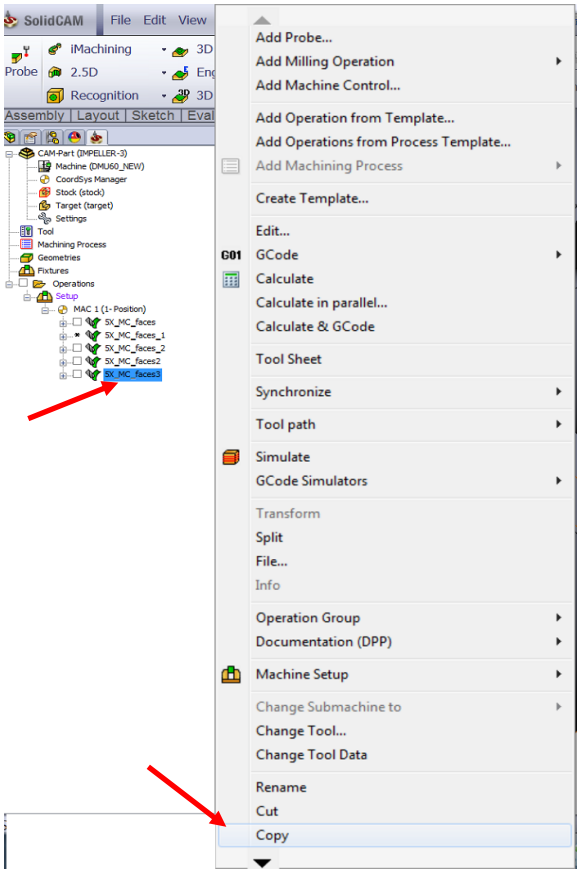


Parallel to Curve

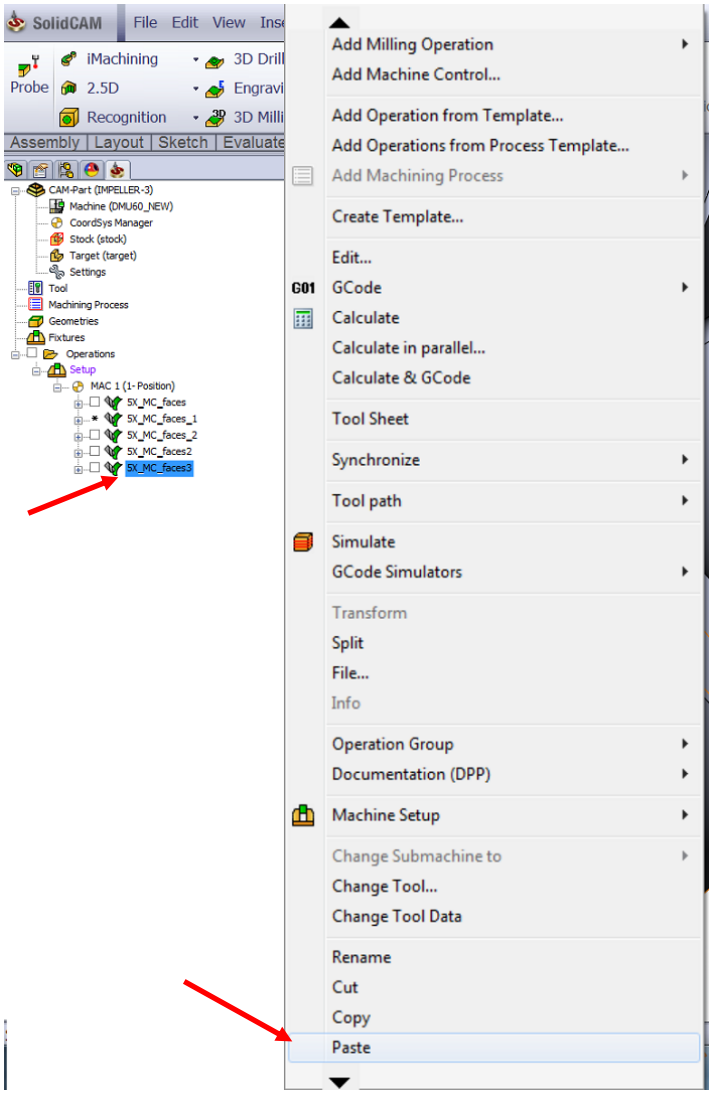
The Parallel to Multiple Curves feature creates toolpath segments parallel to a leading curve. The neighbouring toolpath segments are parallel to each other. An important point here is that the cuts are not simply copied next to each other; every new cut is an offset of the previous cut. When selecting the curve geometry, select the curves directly from the part. When using a single drive surface, you can only select one curve from which to create a parallel toolpath. When using multiple drive surfaces, you can then select multiple curves to use. Each curve is only used for the drive surface nearest to that curve.

This type of strategy is very useful in machining impeller blades as it provides uniform spacing of cuts.

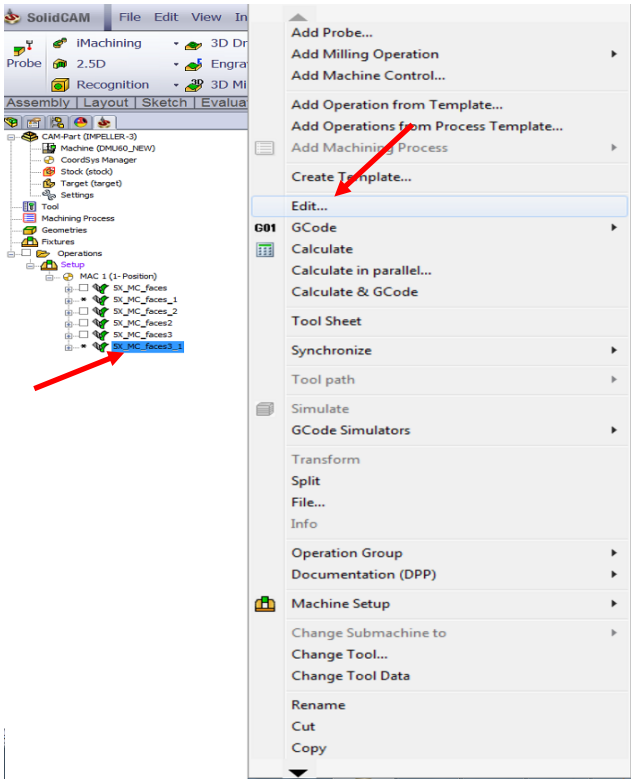
Right click > **Copy** the tool path.



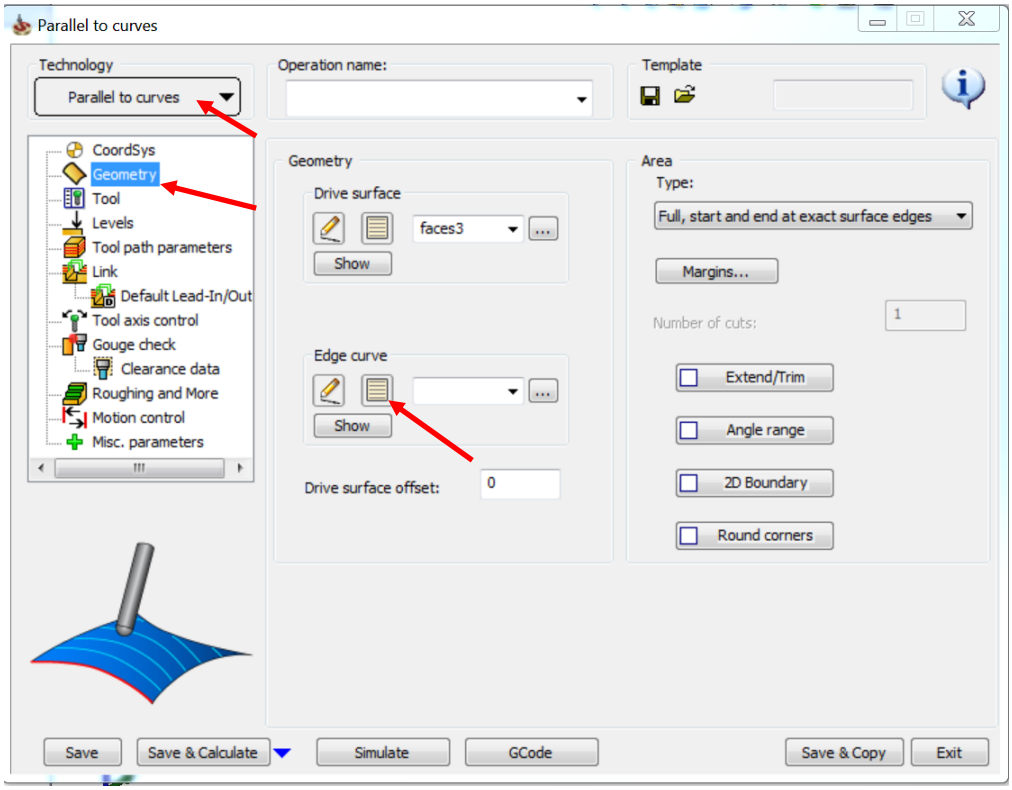
Right click > **Paste** the tool path.



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



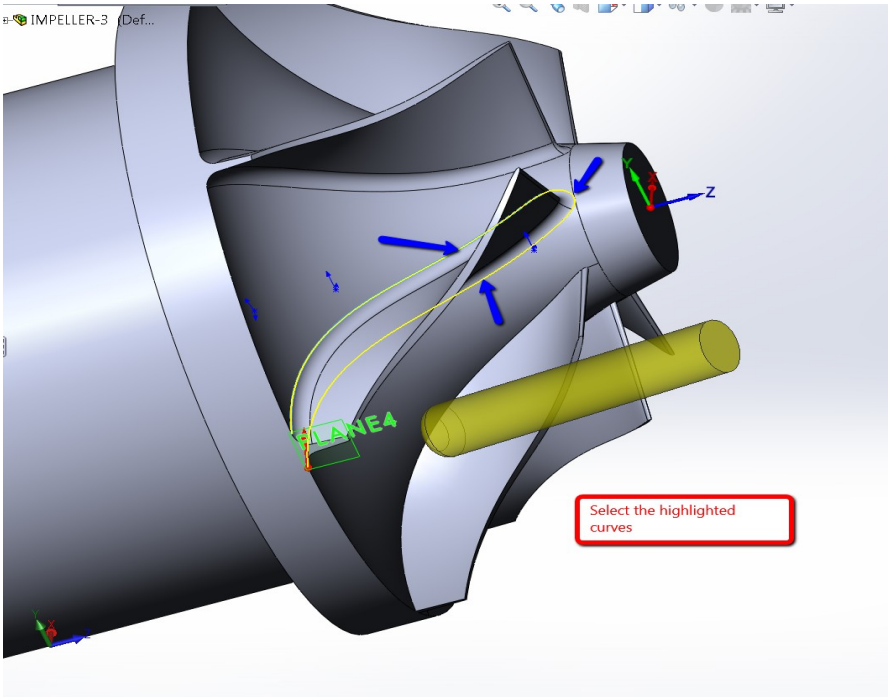
Select **Parallel to curves** in the **Technology** section. We will now study another strategy which is popular when machining these types of parts.



Click **Geometry**.

Click the new icon in the **Edge curve** section.

Select the curves as shown in the image.

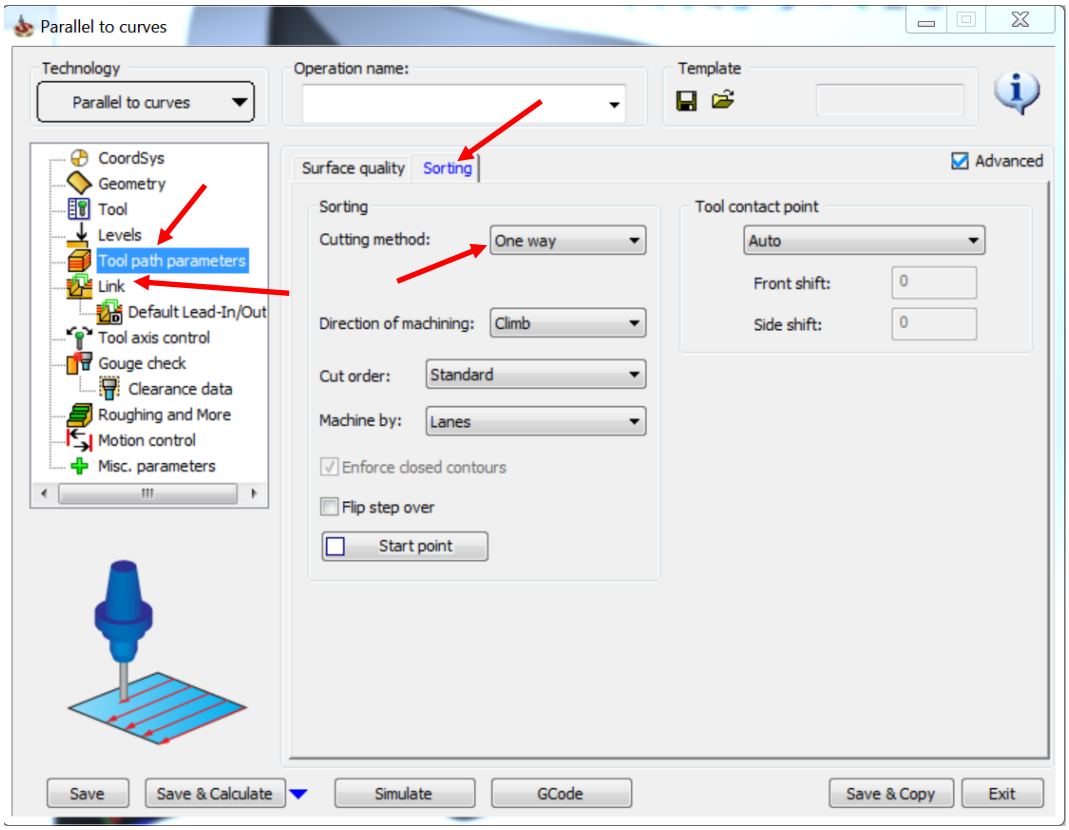


Click 

Click 

Click **Tool** and select the same tool that has been defined already.

Click **Tool path parameters**.

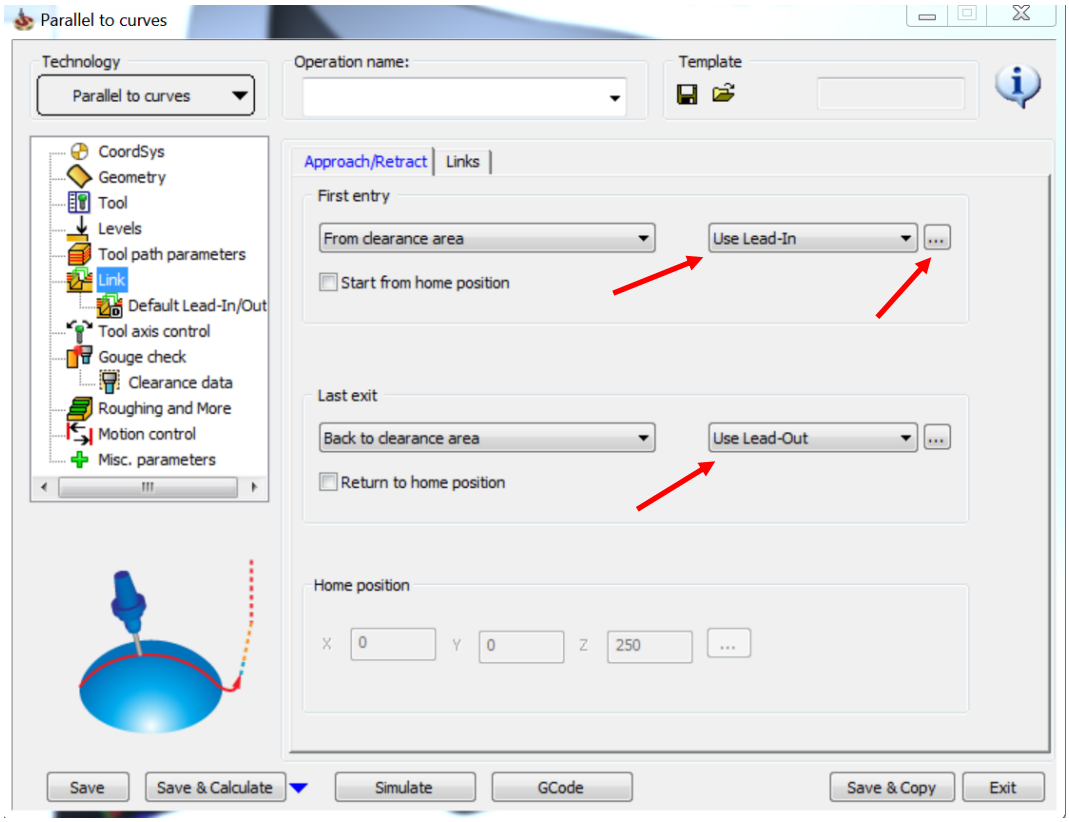


Click the **Sorting** sub-tab.

Select **One way** as the **Cutting method**.

Click **Link**.

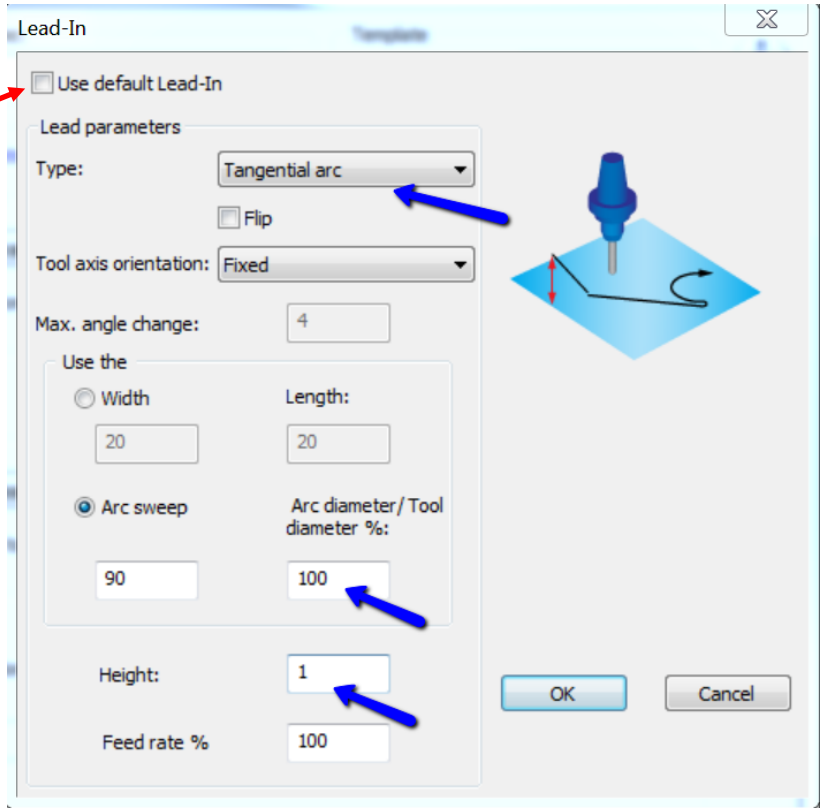
Select **Use Lead-In** in the **First entry** section.



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

Click ...

The Lead-In window opens.

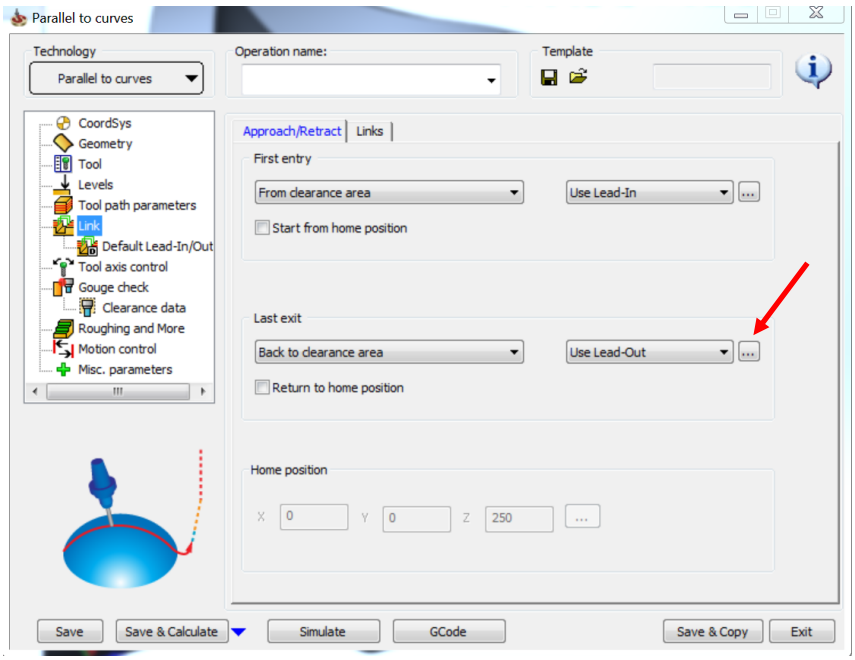


Deselect the **Use default Lead-In** check box.

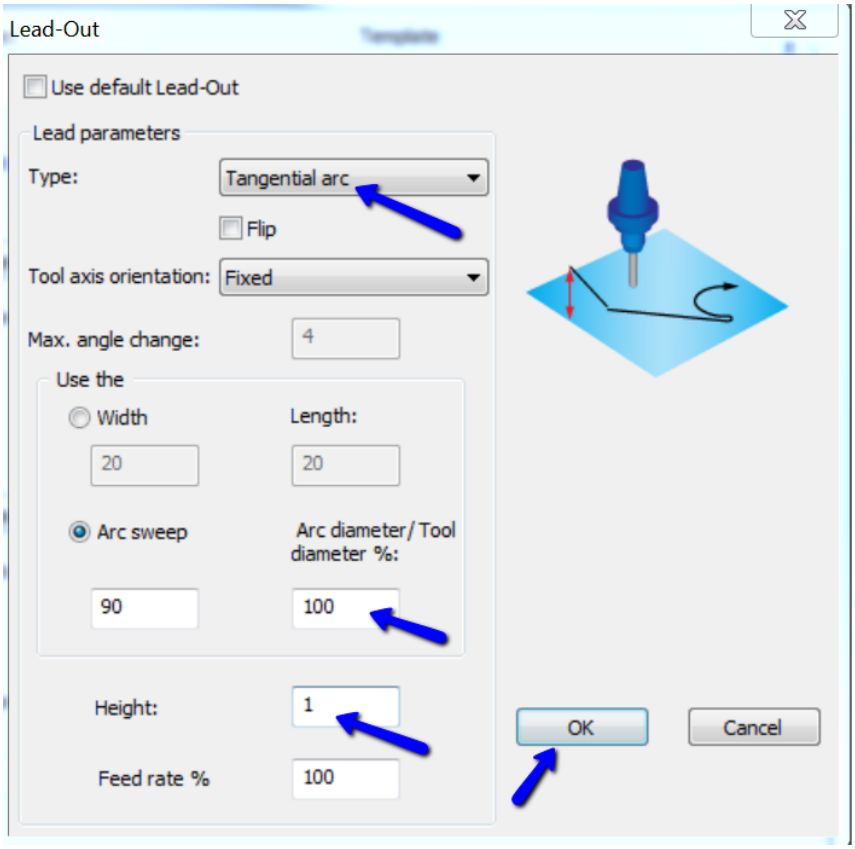
Make the changes as shown in the image.

Click **OK**.

Click ...

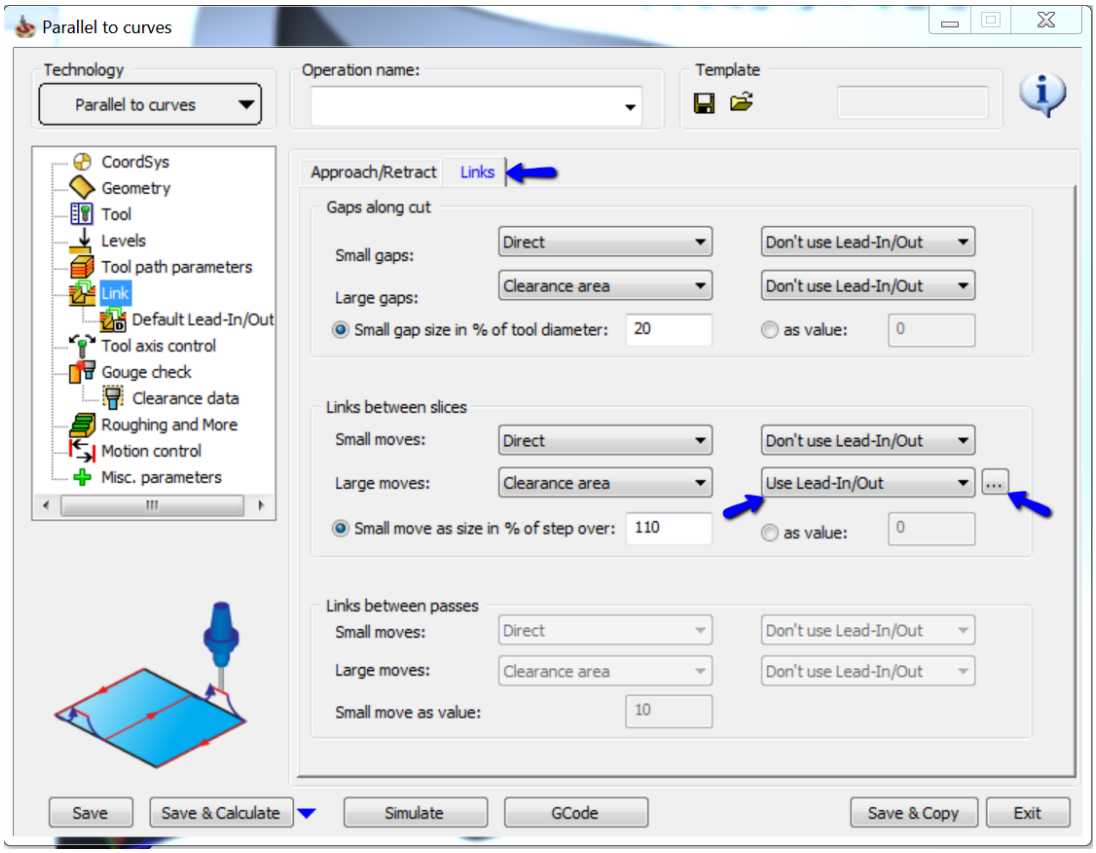


Use the same parameters for Lead– Out also.



Click **OK**.

Click the **Links** sub-tab.

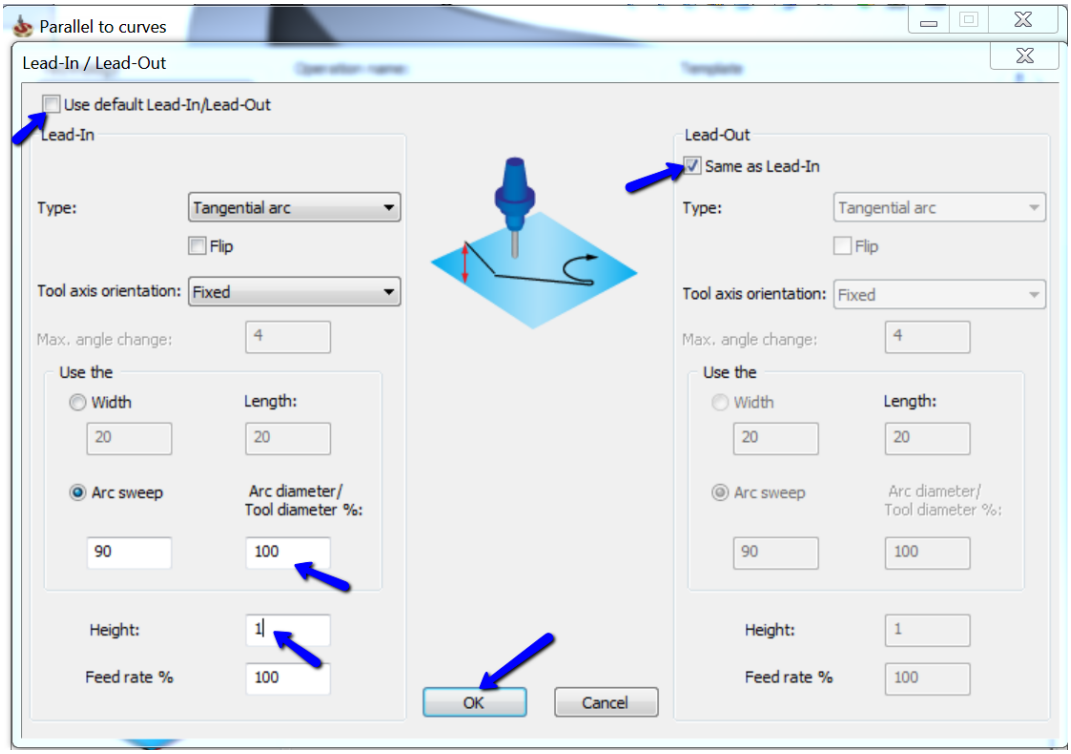


Select **Use Lead-In/Out** in Large moves.

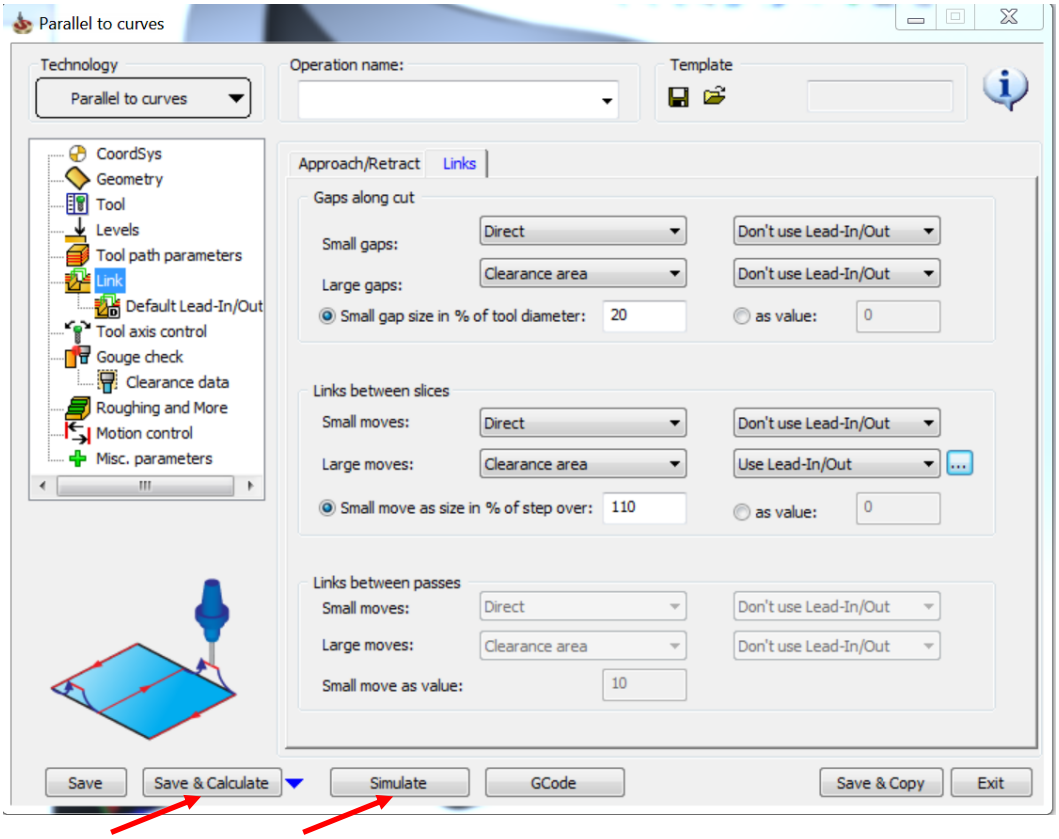
Click ...

Make the changes as shown in the image.

Click **OK**.

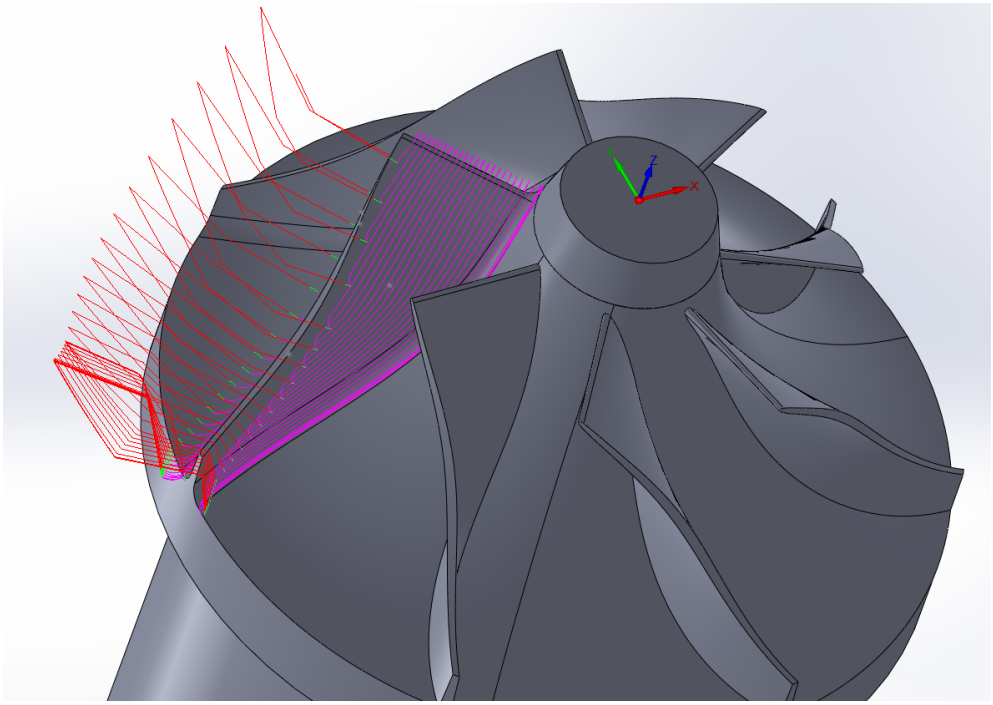


Click **Save & Calculate**.



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

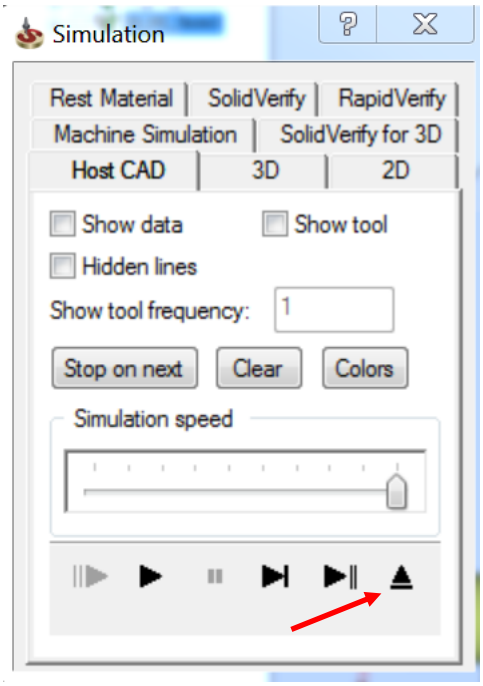
The simulated tool path looks like this. The parallel to curve strategy gives a uniform spacing between passes on the entire set of surfaces unlike the morph between curves which cause passes to converge thereby creating a rubbing action.



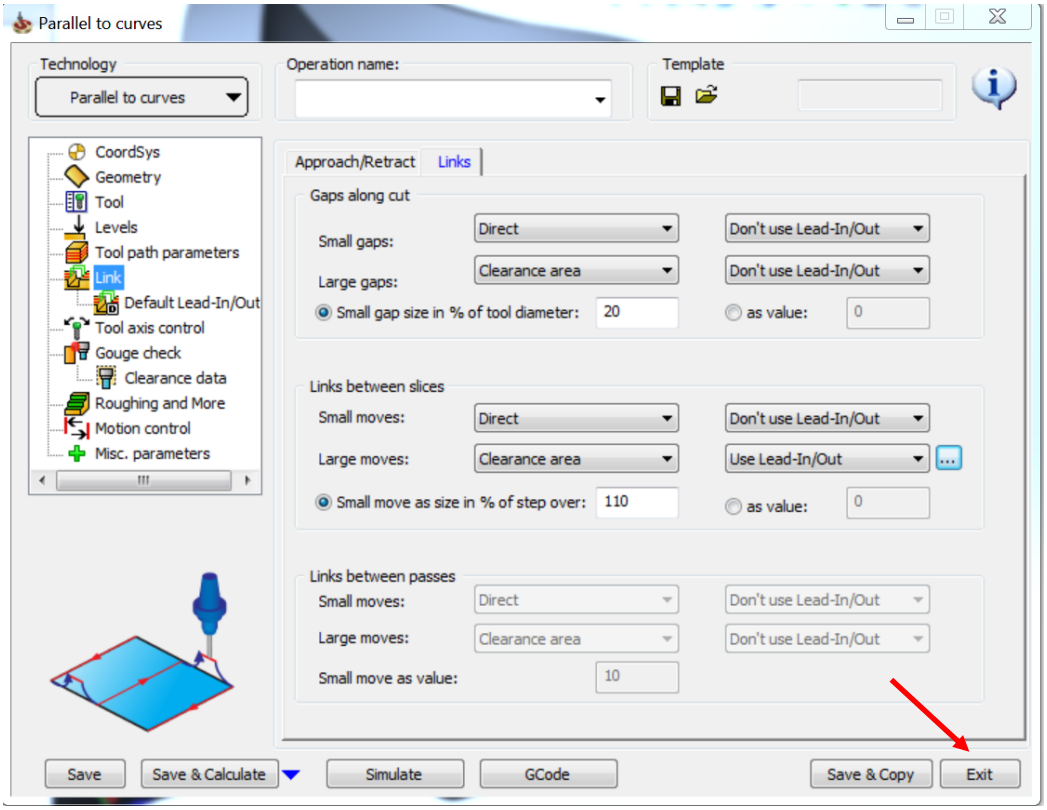
Click the play icon

You can see that the tool is performing one way cutting. This is a very effective strategy for cutting impeller blades as explained in the previous page.

Select the exit icon to return to the Operations Manager.



Click **Exit** to close Operations Manager.

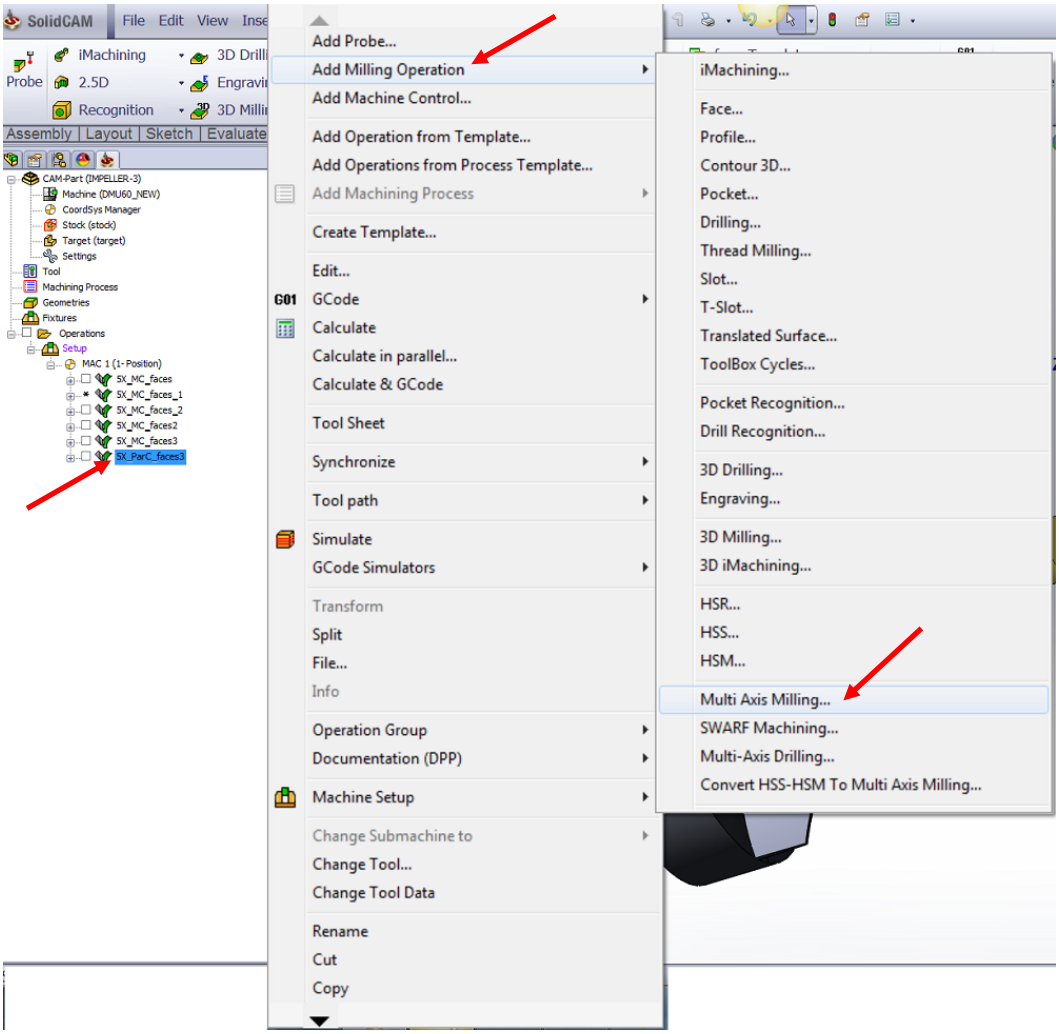


Morph Between 2 Surfaces

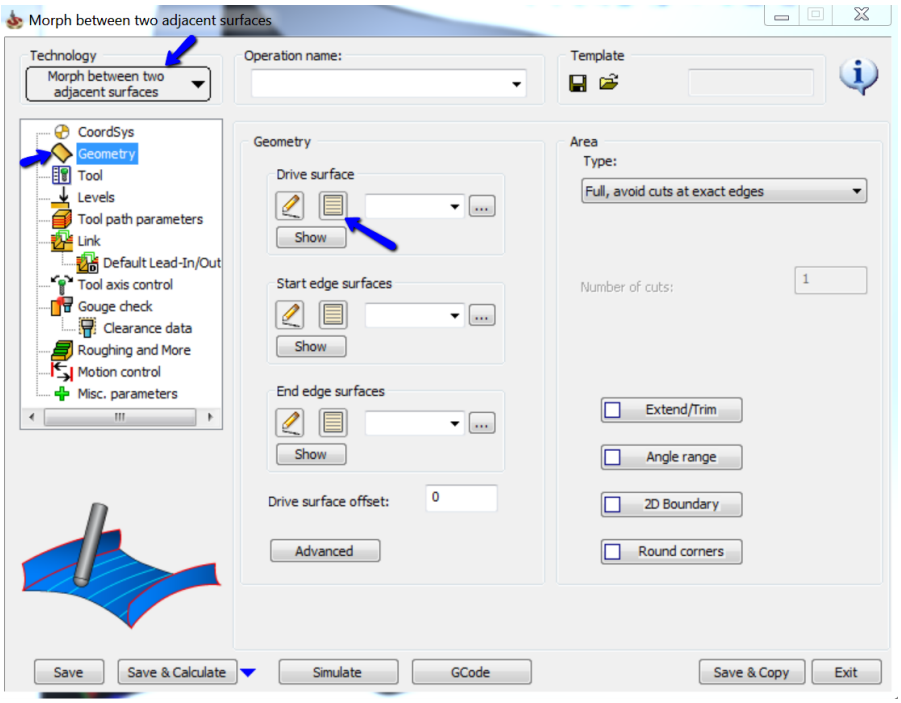
The Morph Between Two Surfaces feature creates a morph toolpath on the drive surface. The drive surface is enclosed by two check surfaces. Morph means that the generated toolpath is approximated between the check surfaces and evenly spread over the drive surface. This is great for impeller machining with twisted turbine blades. The main advantage is the possibility to compensate the tool to the drive surface and check surface in the left and right corner of the work piece. All you need is the tool radius which is the distance (Margin) between the tool center and the surfaces.

We would also be discussing couple of side tilt definations while using the above strategy.

Right click the tool path > **Add Milling Operation** > **Multi Axis Milling**.



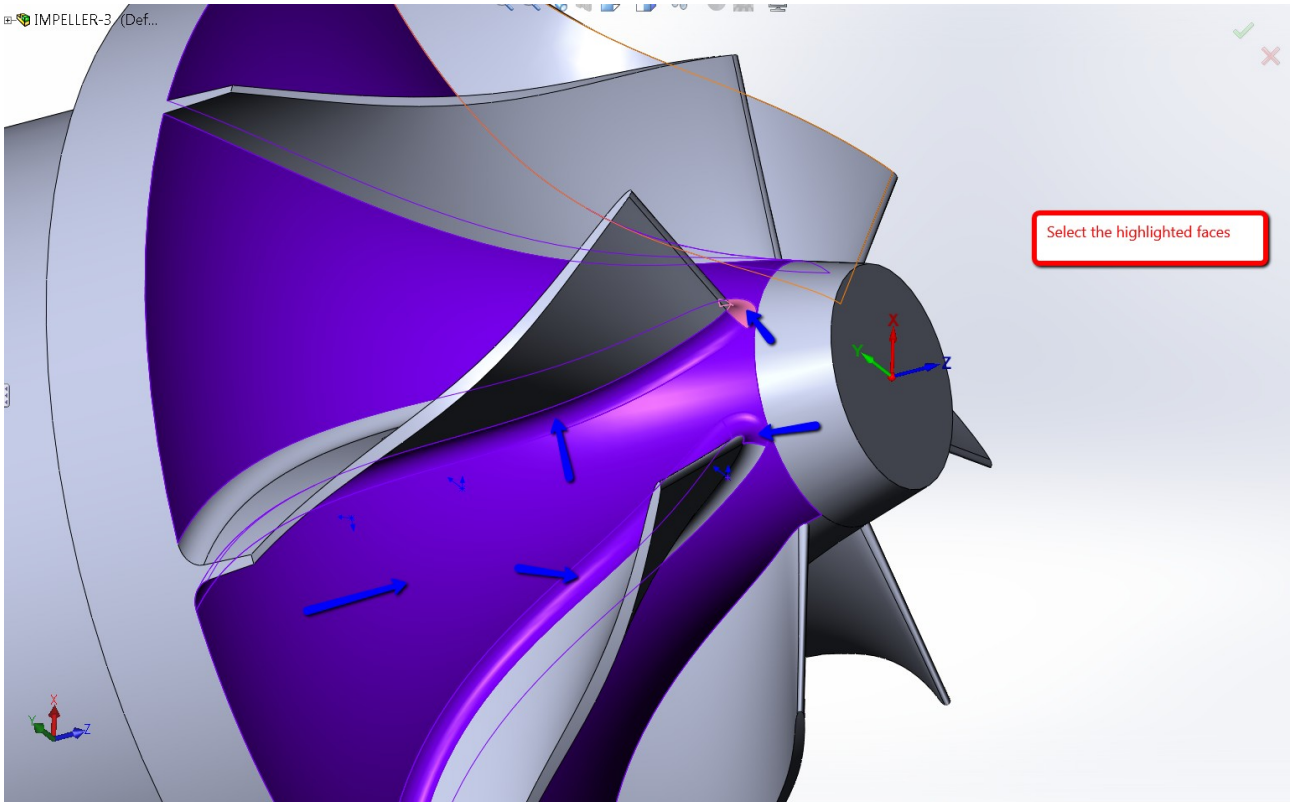
Select **Morph between two adjacent surfaces** in the **Technology** section.



Click **Geometry**.

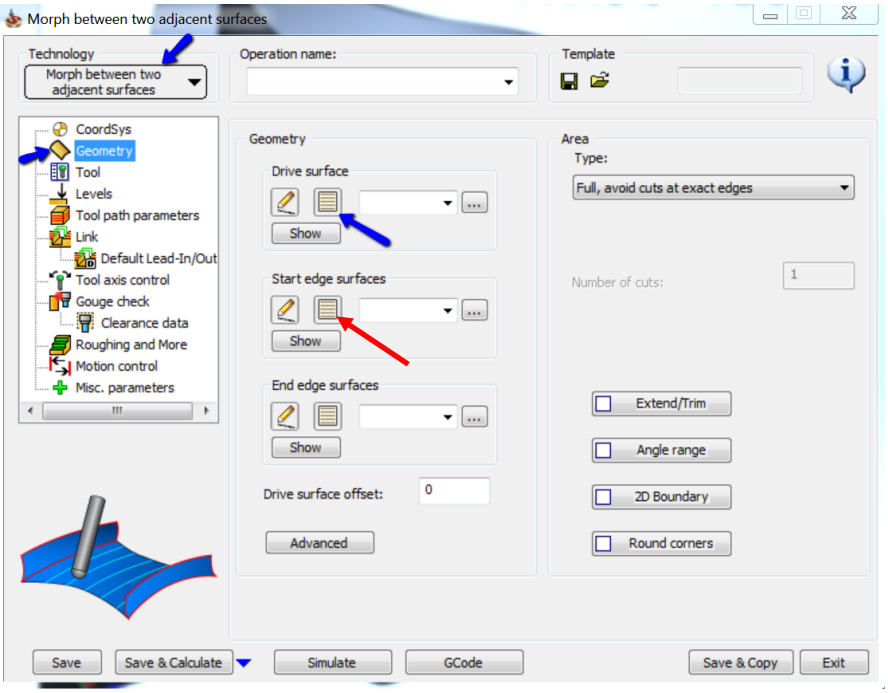
Select the new icon under **Drive surface**.

Select the faces as shown in the image:



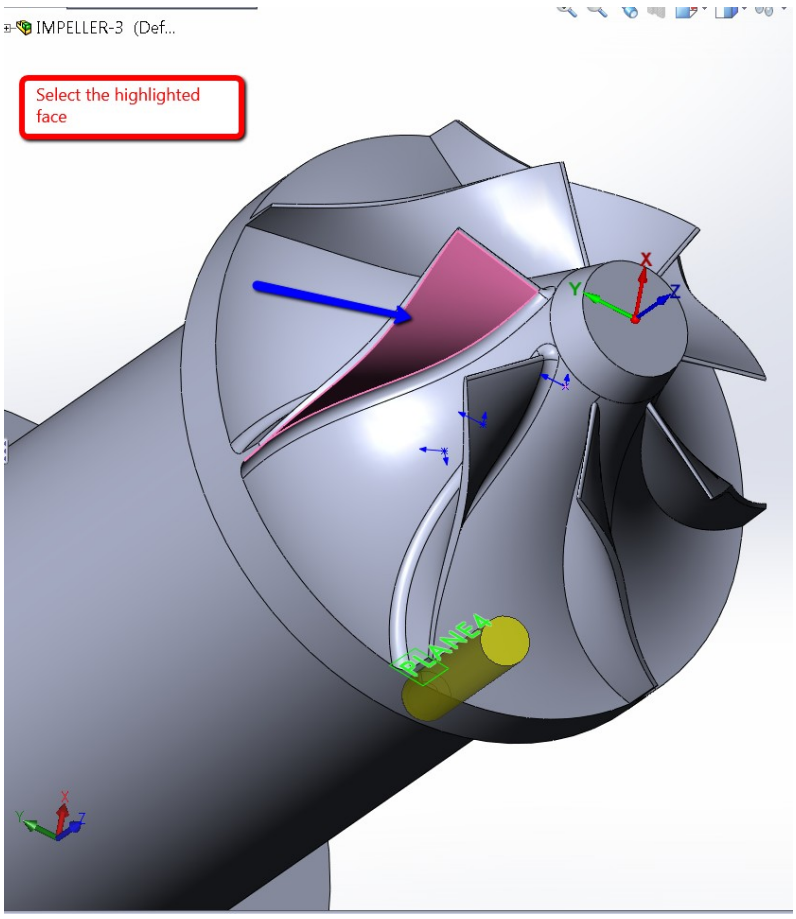
Click 

Select the new icon under **Start edge surfaces** section.

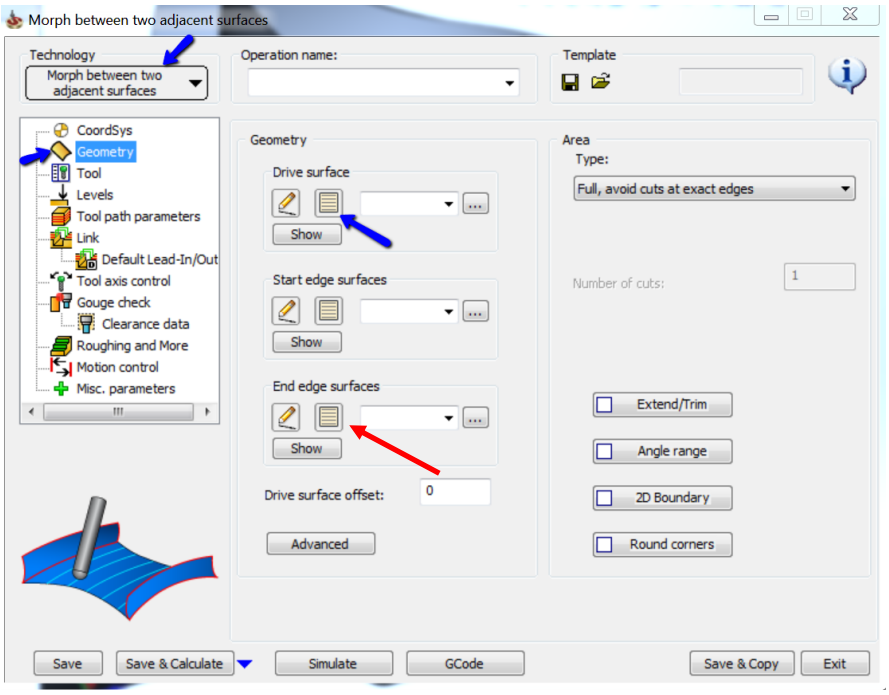


Select the highlighted face.

Click 

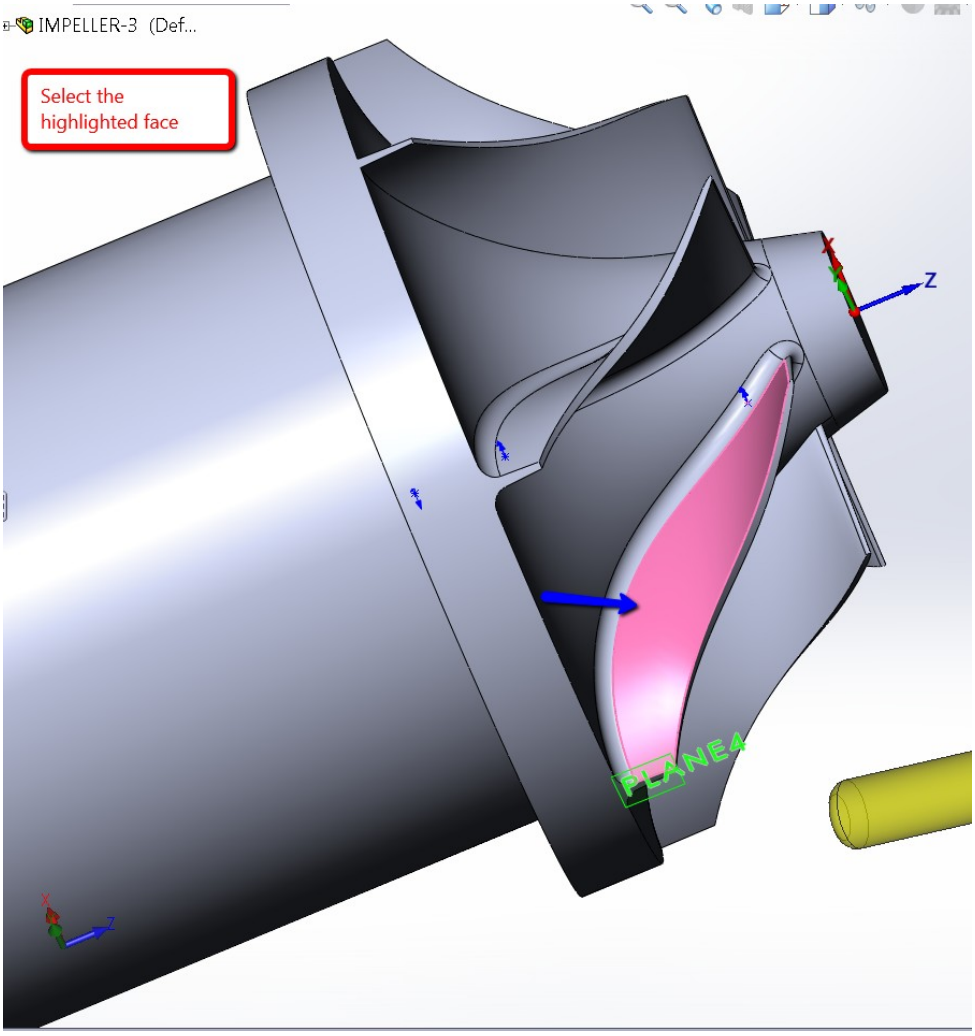


Select the new icon under **End edge surfaces**.

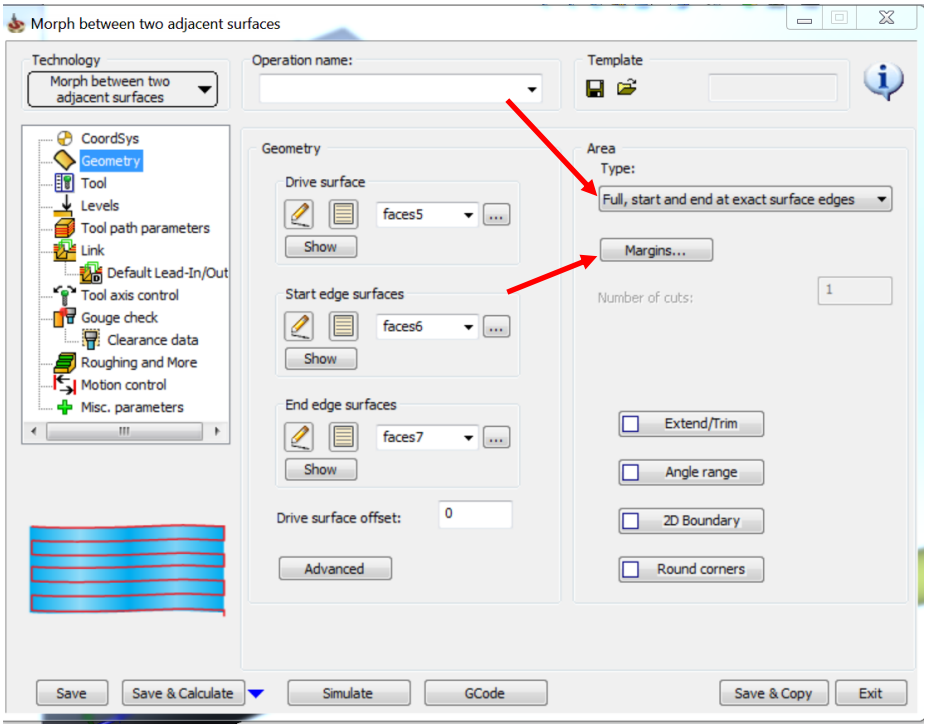


Select the highlighted face.

Click 

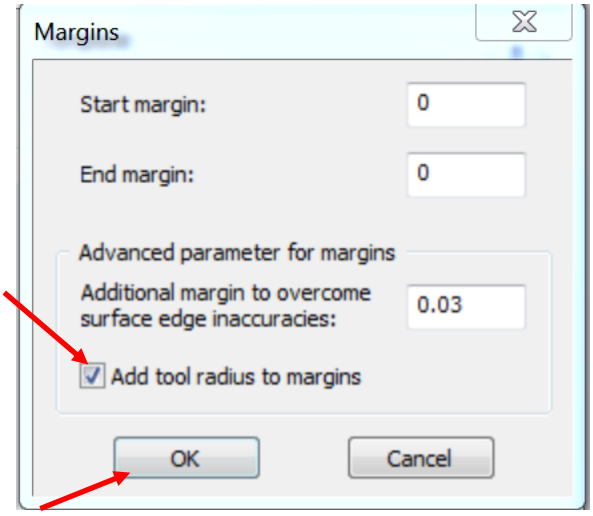


Select the **Type** as **Full, start and end at exact surface edges** in the **Area** section.



Click **Margins.**

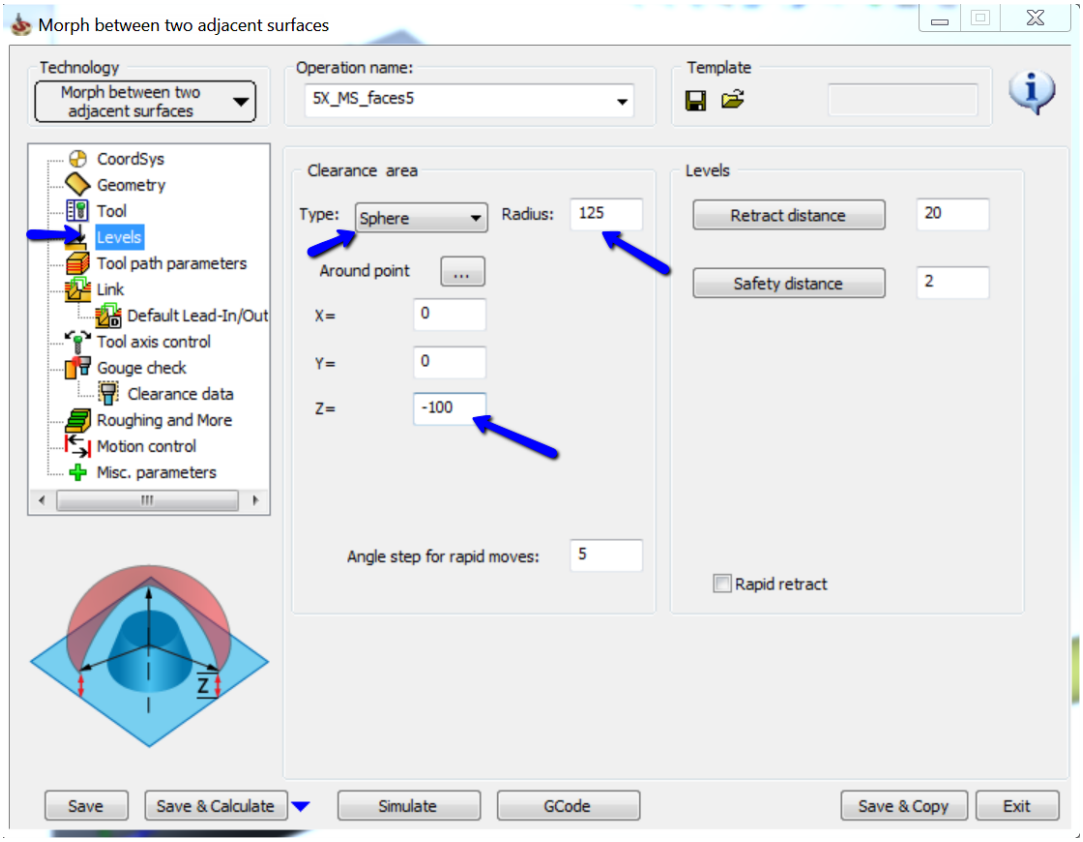
Select **Add tool radius to margins** check box.



Click **OK**.

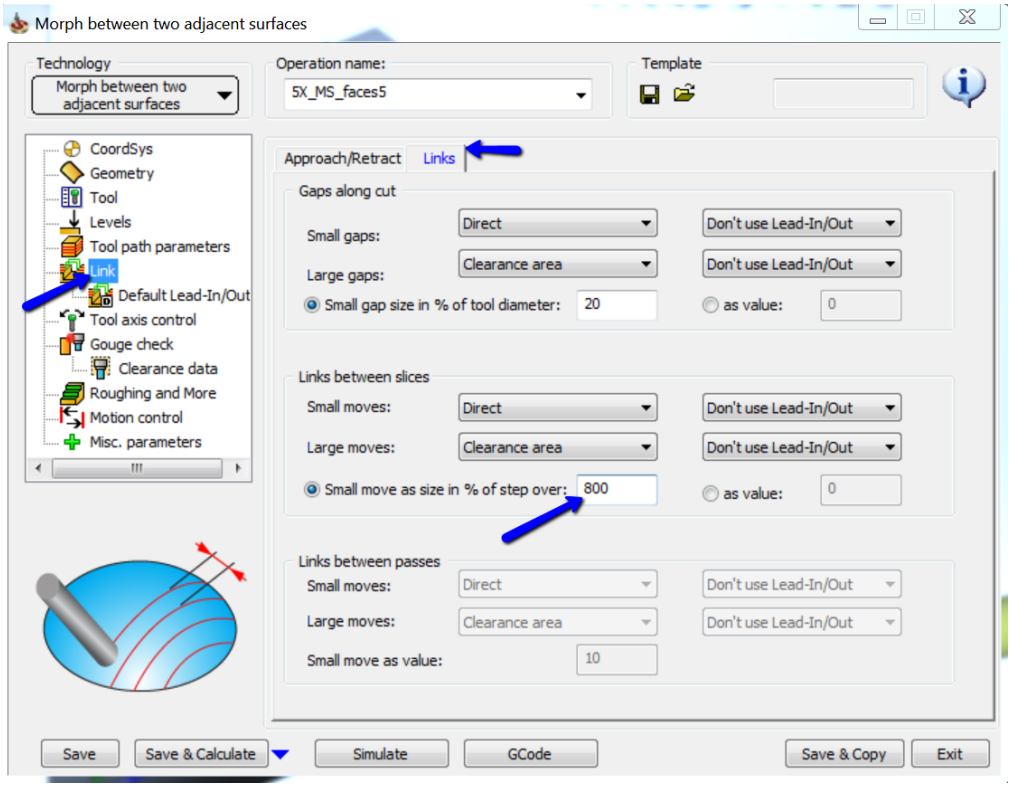
Click **Tool** and select the same tool that has already been defined.

Click **Levels** and enter the parameters as shown in the image.



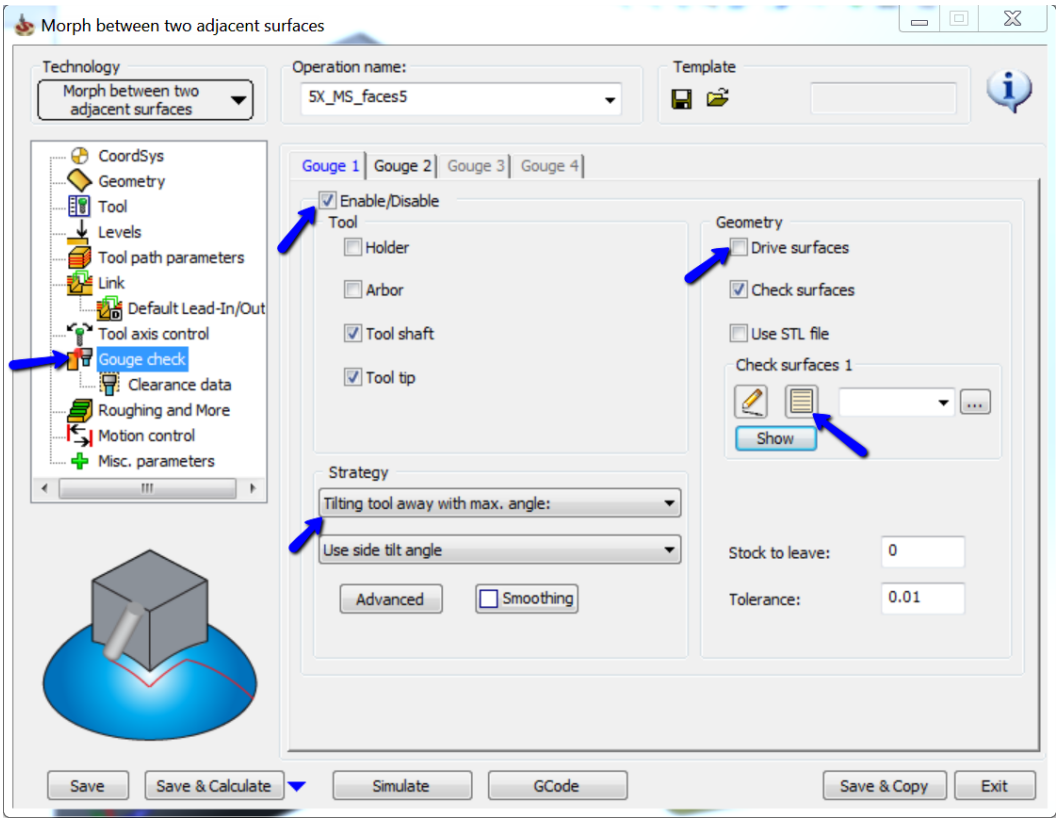
Click **Link**.

Click **Links** sub-tab.



Enter a value of 800 in **Small move as size in % of step over**.

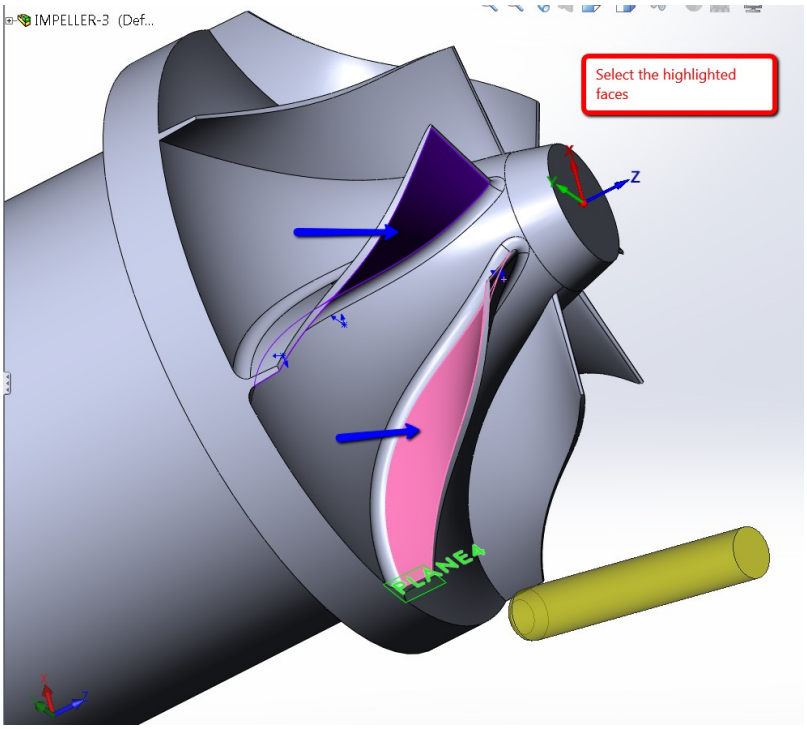
Click **Gouge check**.



Make the changes as shown in the image.

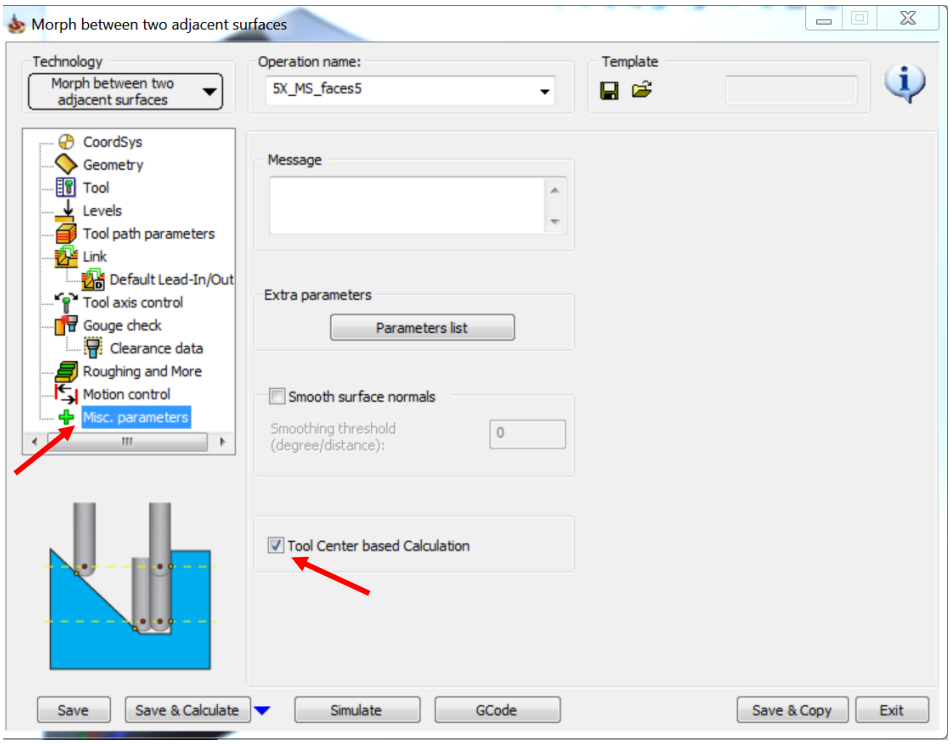
Click new icon under **Check surfaces 1**.

Select the highlighted faces.



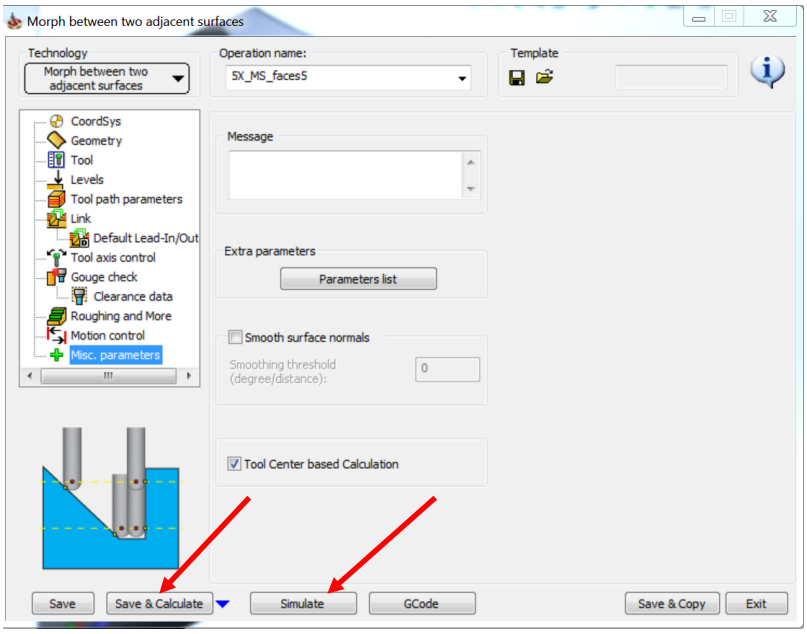
Click 

Click **Misc. parameters**.



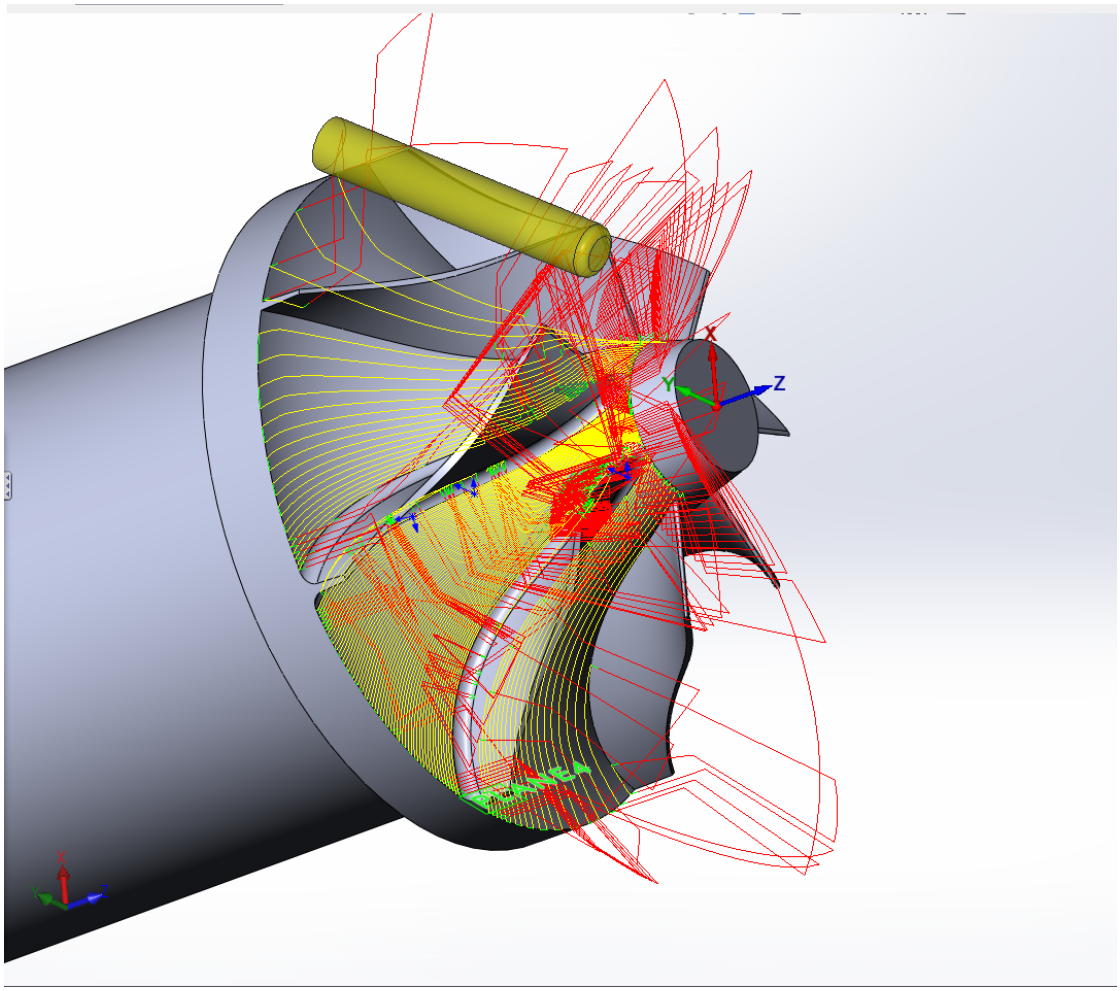
Select the **Tool Center based Calculation** check box.

Click **Save & Calculate**.



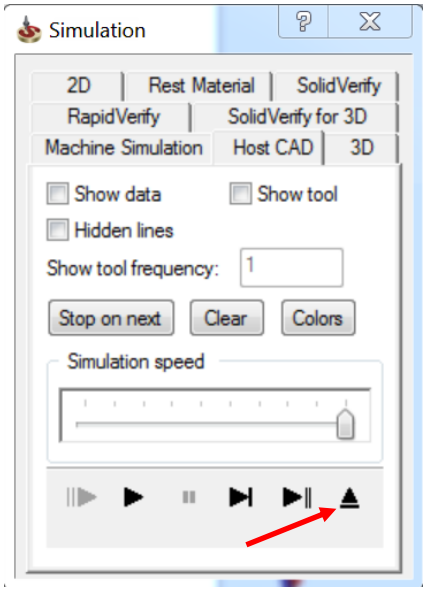
Click **Simulate** once the tool path is calculated.

The simulated tool path looks like this again a bad toolpath , we need to fix this.



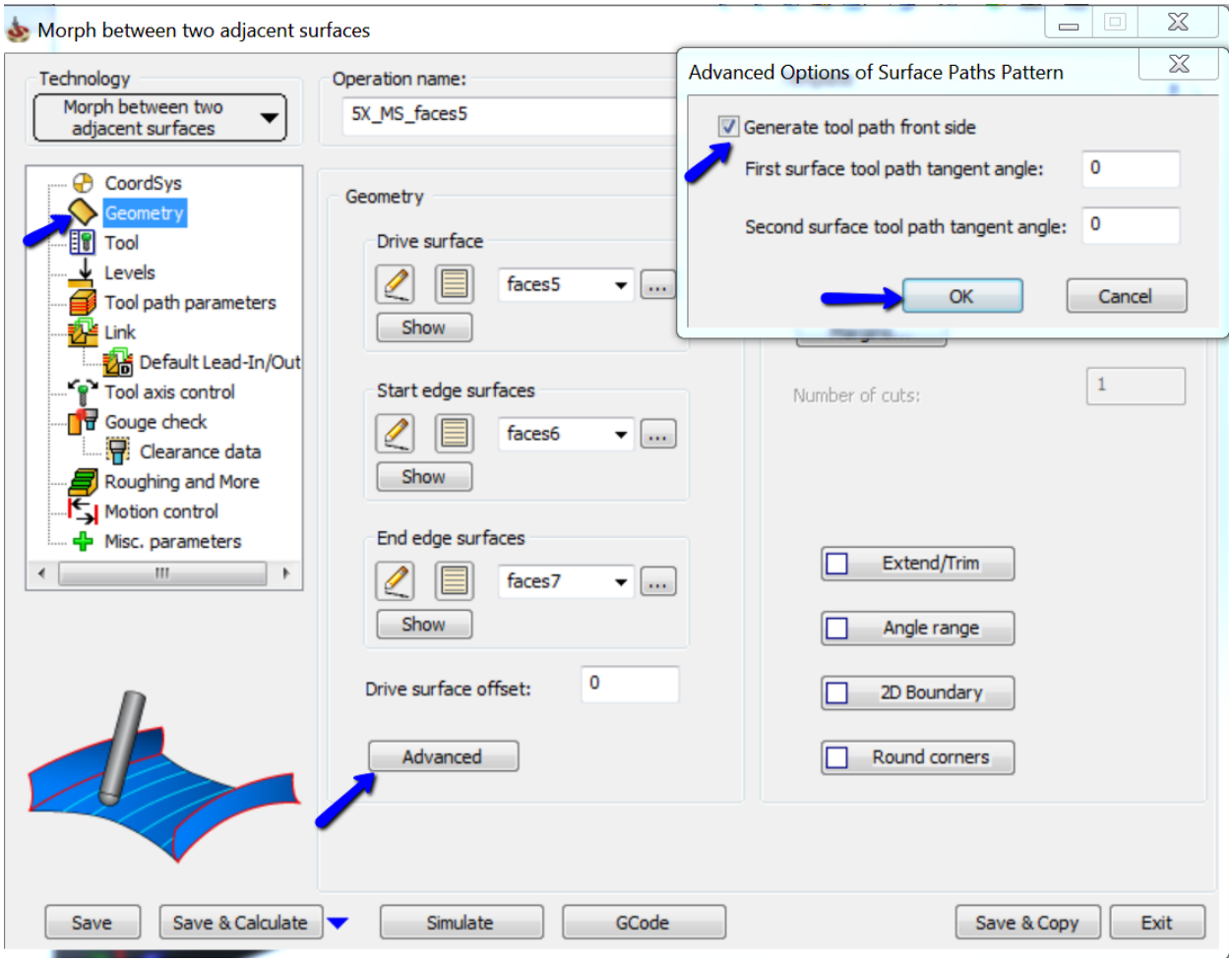
You can see that apart from the area to be machined, some other areas are also getting machined.

Select the exit icon to return to Operations Manager.



We can fix this issue by going back to Geometry.

Click **Geometry**.

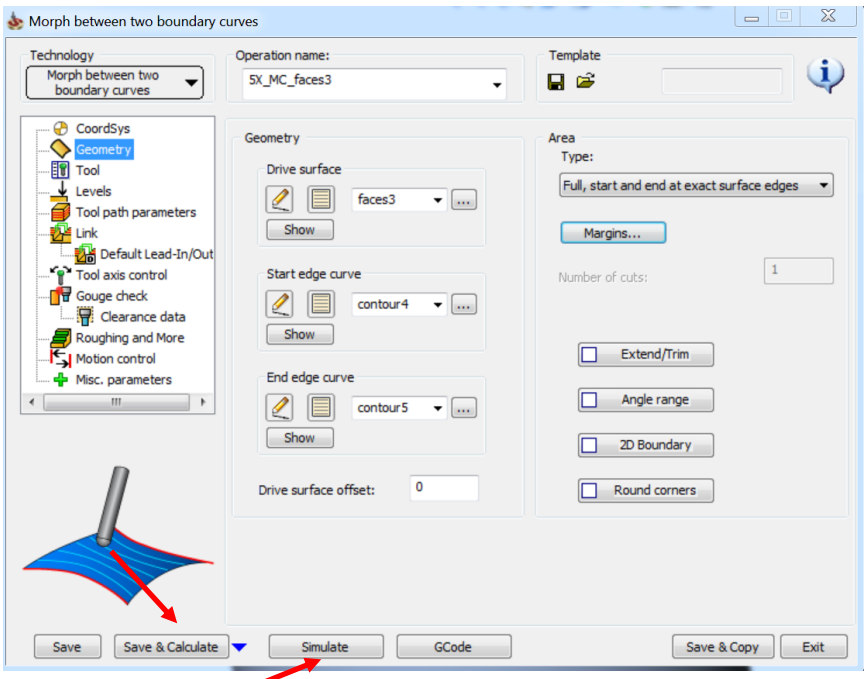


Click **Advanced**.

Select the **Generate tool path front side**.

Click **OK**.

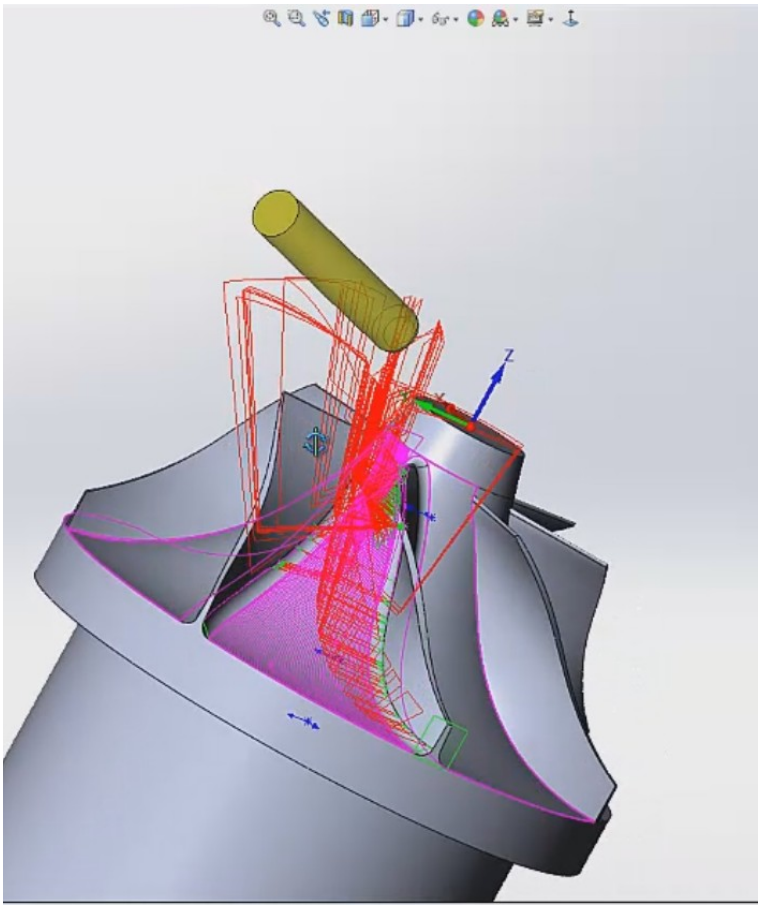
Click **Save & Calculate**.



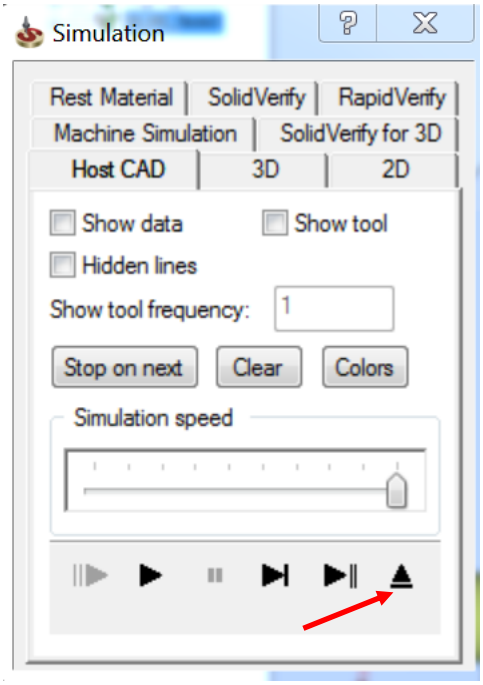
Click **Simulate** once the tool path is calculated.

The simulated tool path looks like this:

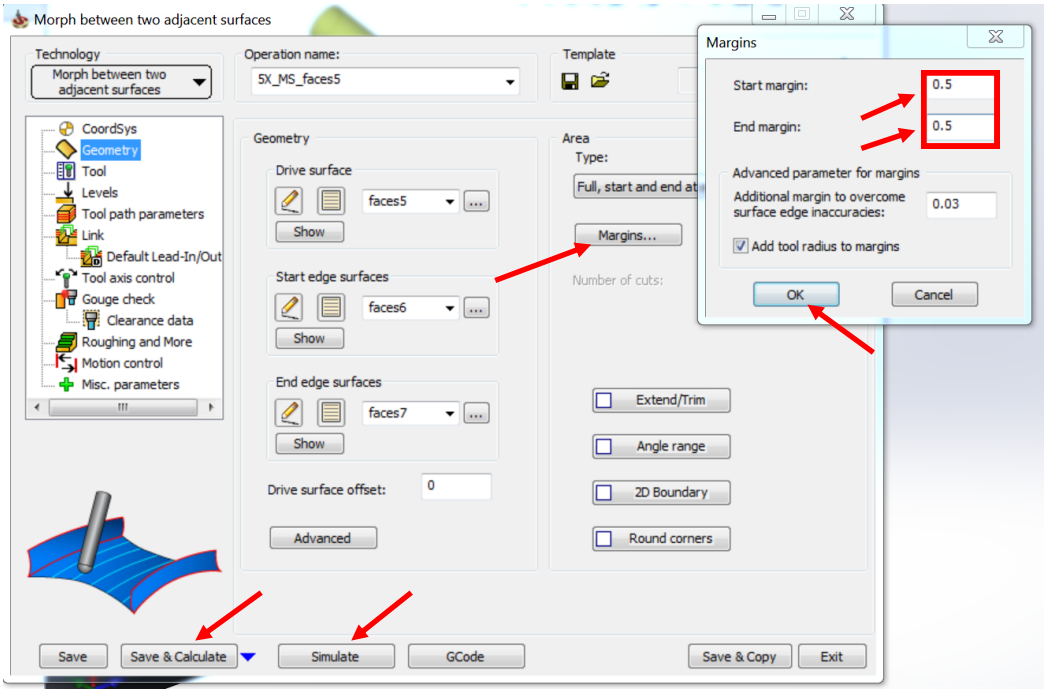
A lot of unwanted areas can still be seen in the tool path.



Click the exit icon to return to the Operations Manager.



Click **Margins**.



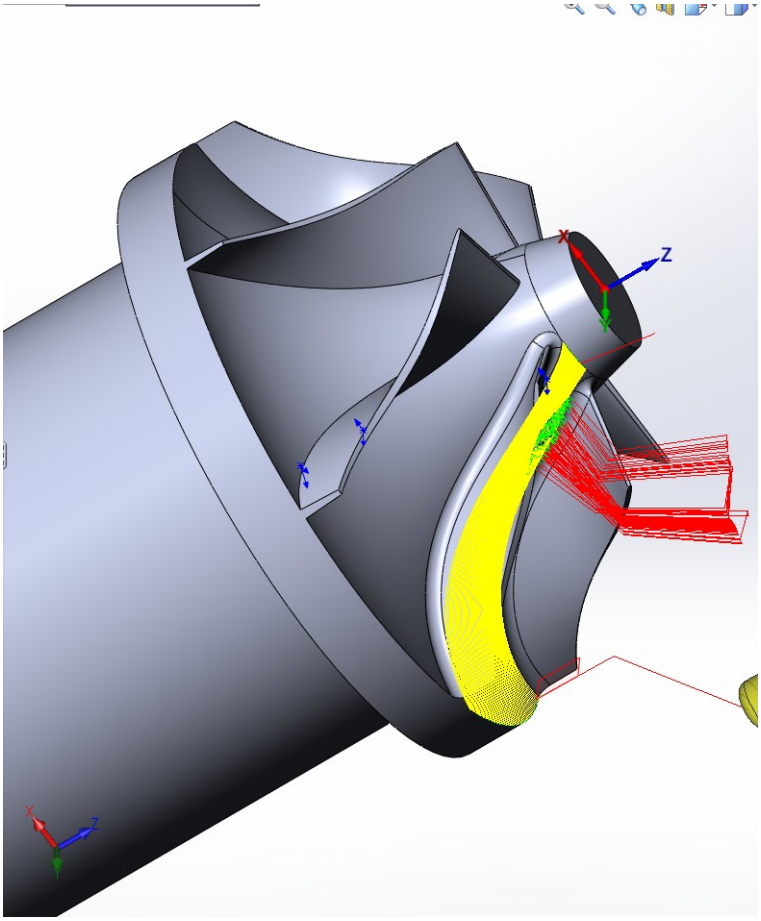
Enter a value of 0.5 in the start and end margin fields.

Click **OK**.

Click **Save & Calculate**.

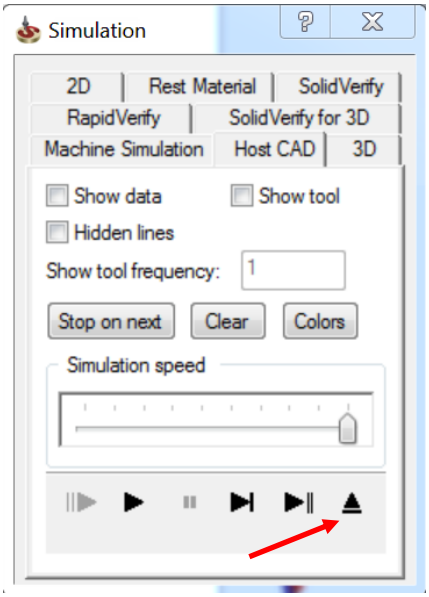
Click **Simulate** once the tool path is calculated.

The simulated tool path looks like this:



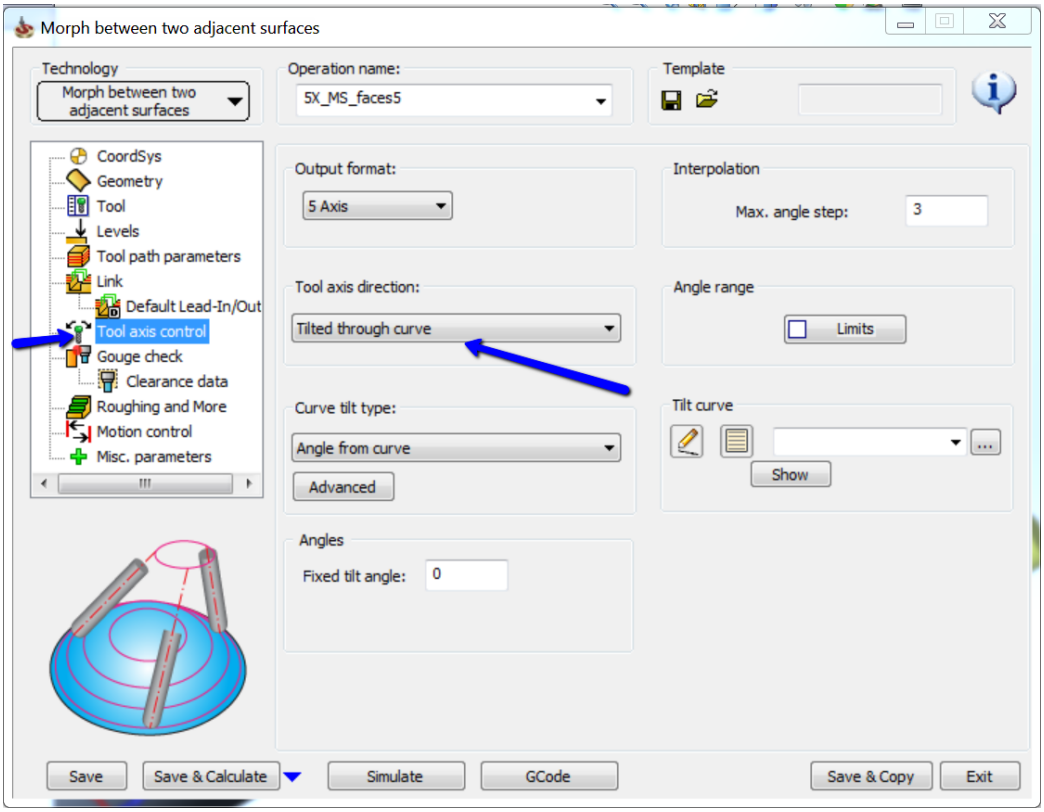
The tool path still has some problems which we must fix.

Click the exit icon to return to the Operations Manager.



This problem is the result of a wrong tool axis control & we need to change the type of tool axis control to fix this issue.

Click **Tool axis control**.

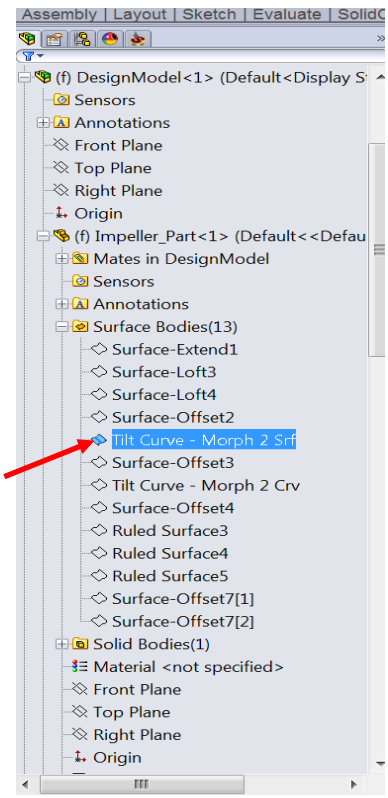


Select **Tilted through curve** as the **Tool axis direction**.

Select the feature manager design tree.

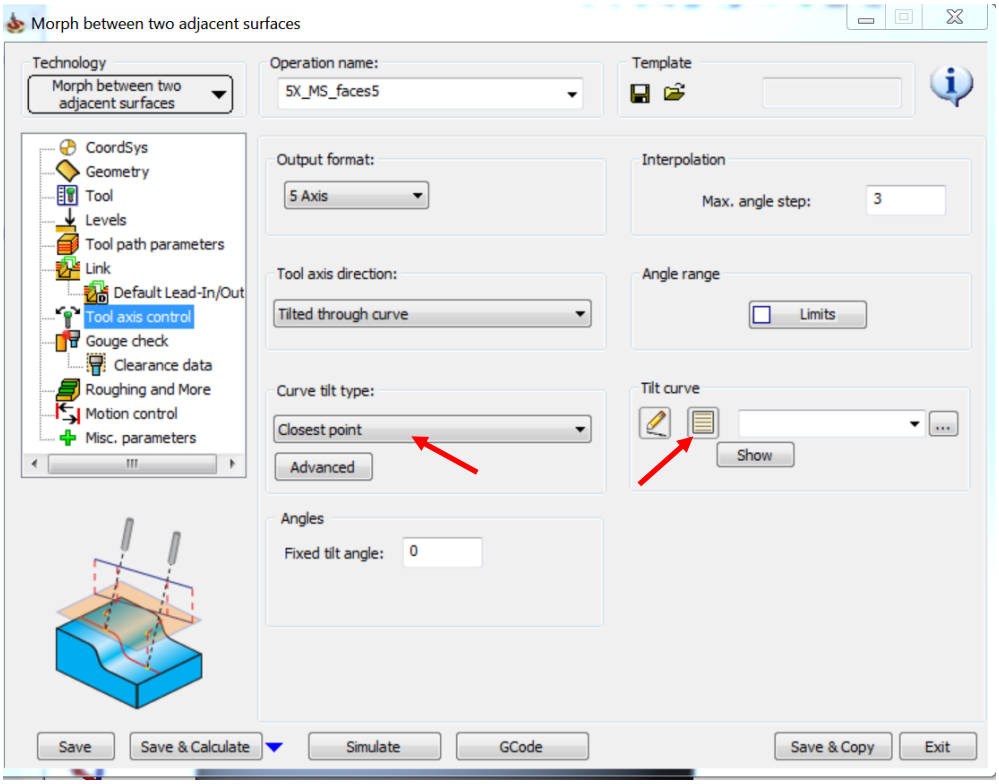
Select **Tilt Curve—Morph 2 Srf**.

Click to unhide the surface.

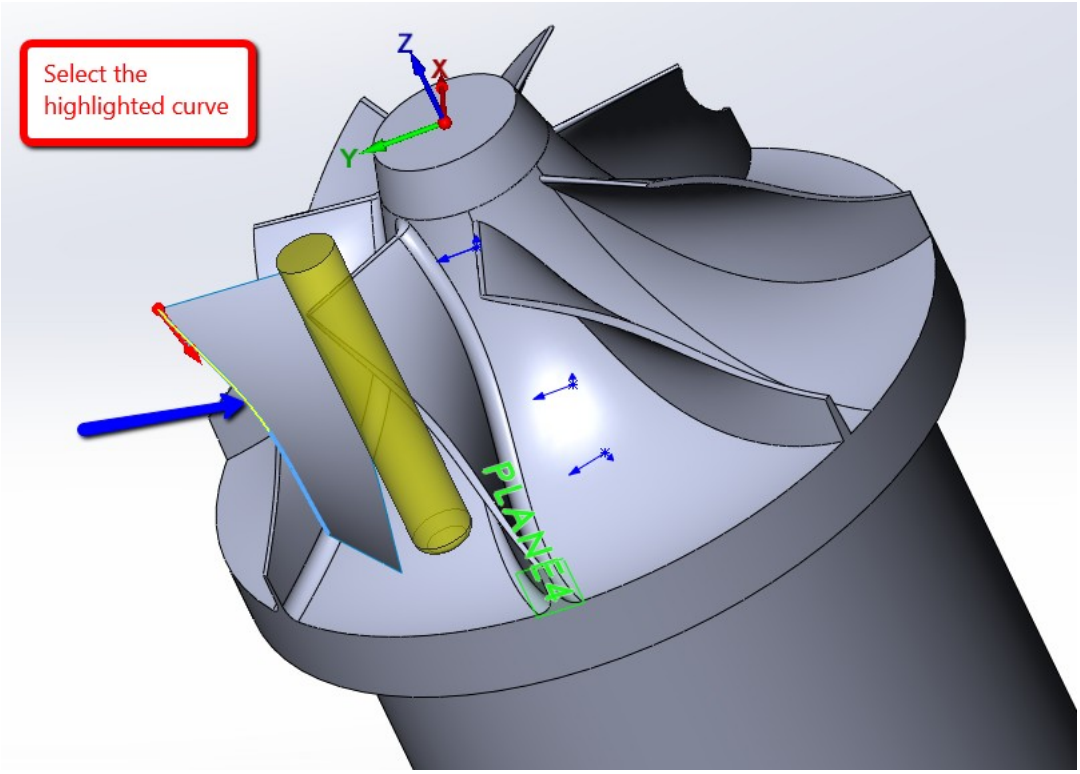


Select **Closest point** as the **Curve tilt type**.

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



Select the highlighted curve.

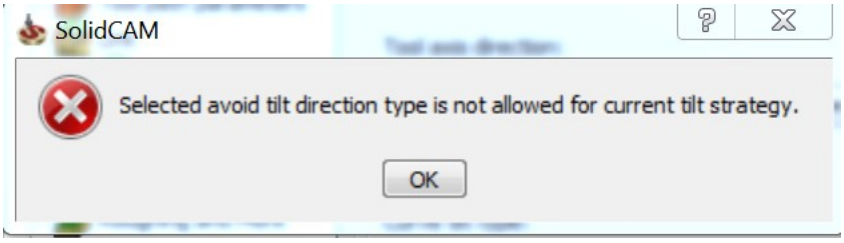


Click 

Click 

Click **Save & Calculate**.

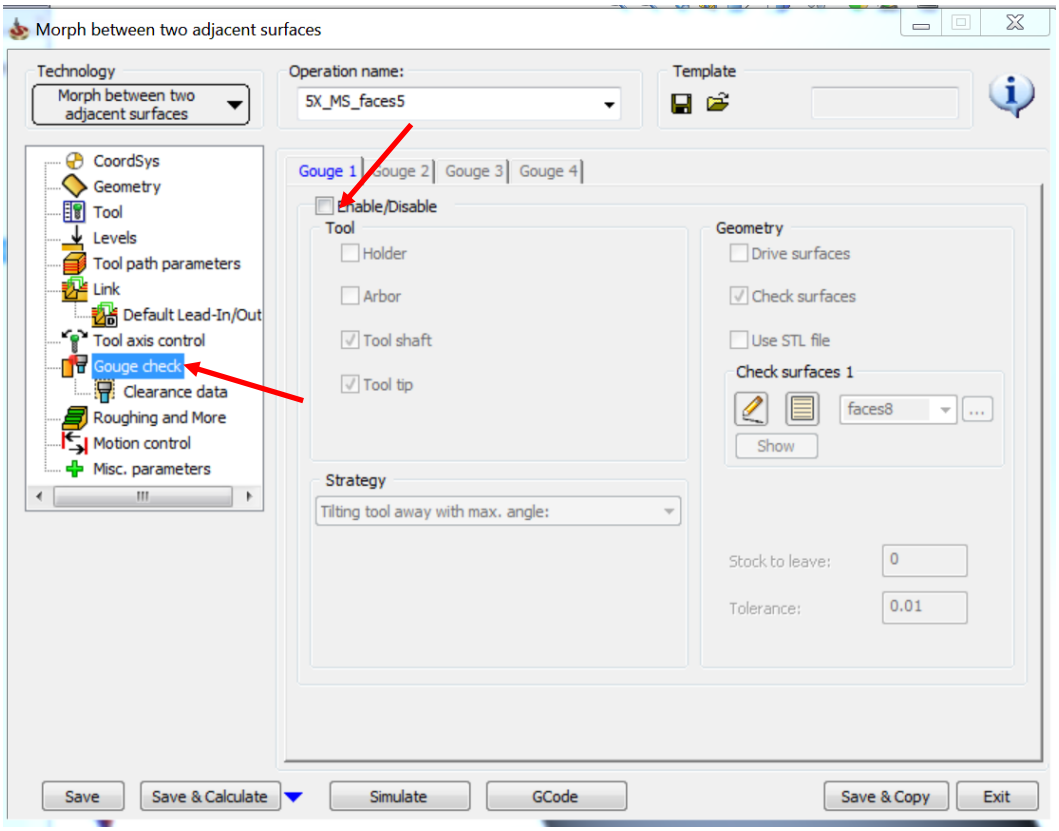
The following error displays:



This is because the tilted through curve strategy does not allow gouge check using the tilt tool as the tilt is being generated by the curve and cannot be changed by the gouge check strategy.

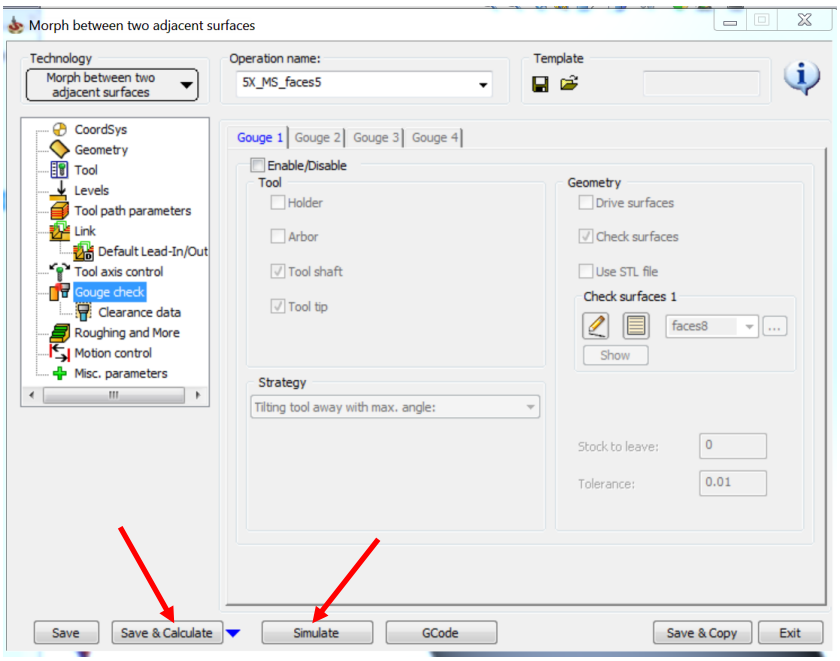
To fix this issue, click **Gouge check**.

Deselect the **Enable/Disable** check box on the Gouge 1 (Which is using the Tilting tool with max angle).

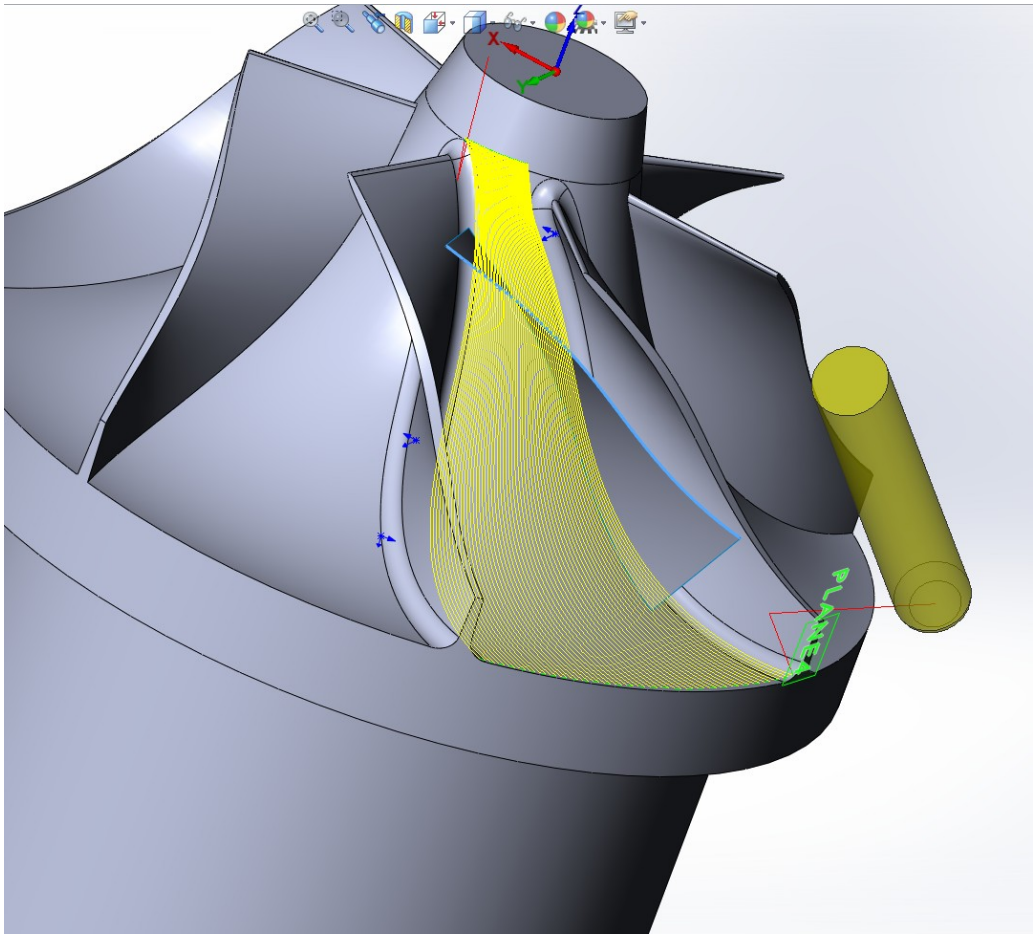


Click **Save & Calculate**.

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

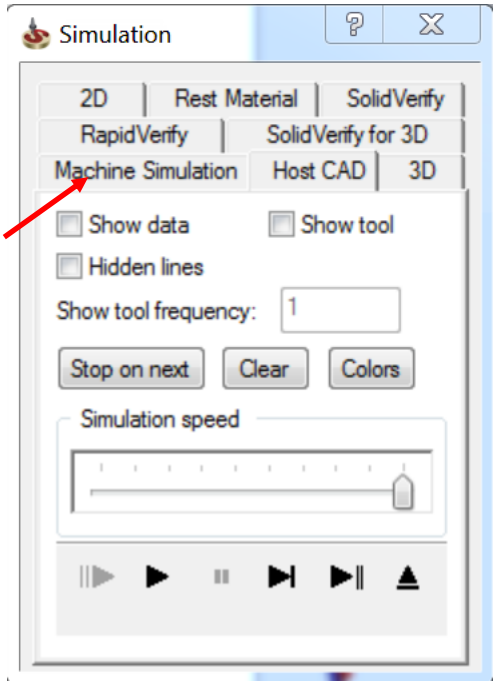


The simulated tool path looks like this:



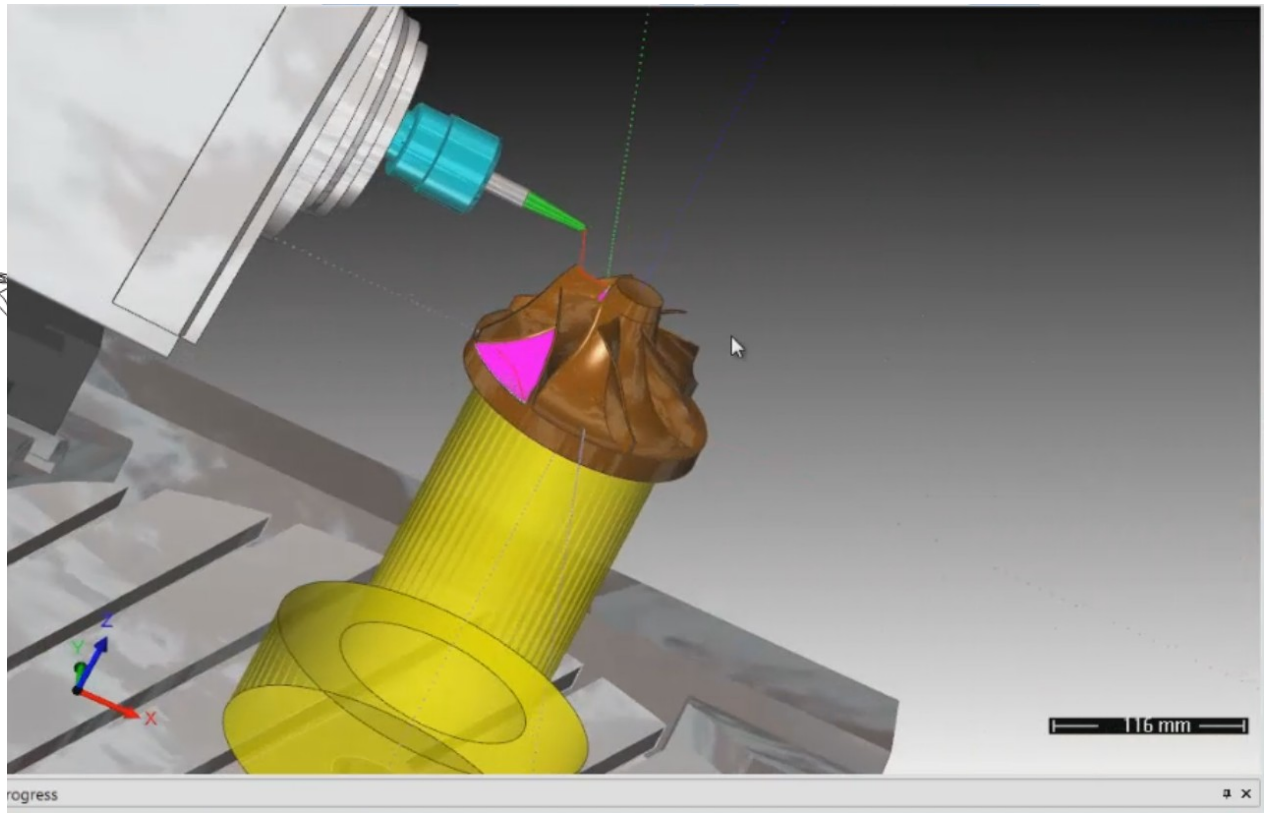
You can see that the tool path is much better with this strategy now. Let us now run the machine simulation.

Click **Machine Simulation**.



Click the play icon to run the simulation.

The machine simulation looks like this:



End of tutorial