SolidCAM

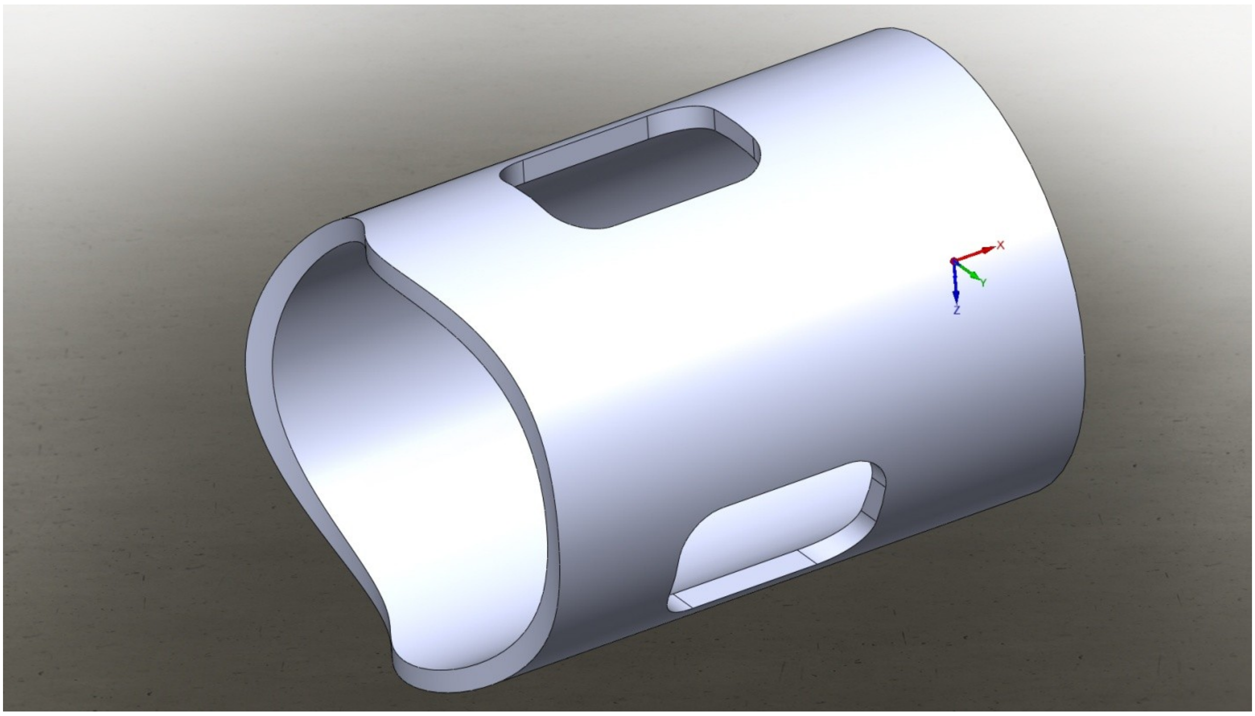
5 Axis Tutorial

Volume 1 – Beginner

Goals of this Tutorial

- **To understand post settings and machine simulation settings**
- **To understand the advanced concepts of 5 Axis**
- **To understand the basic parameters used in defining the toolpath**
- **To understand the parallel strategy of 5 axis machining**
- **To create machine simulation**

Part file used in this exercise:



Basic Training.Sldprt

Post processor used:

DMU100_Monoblock_iTNC530_5x

Machine simulation

DMU_100

Chapter 1

Opening the Part and Starting SolidCAM

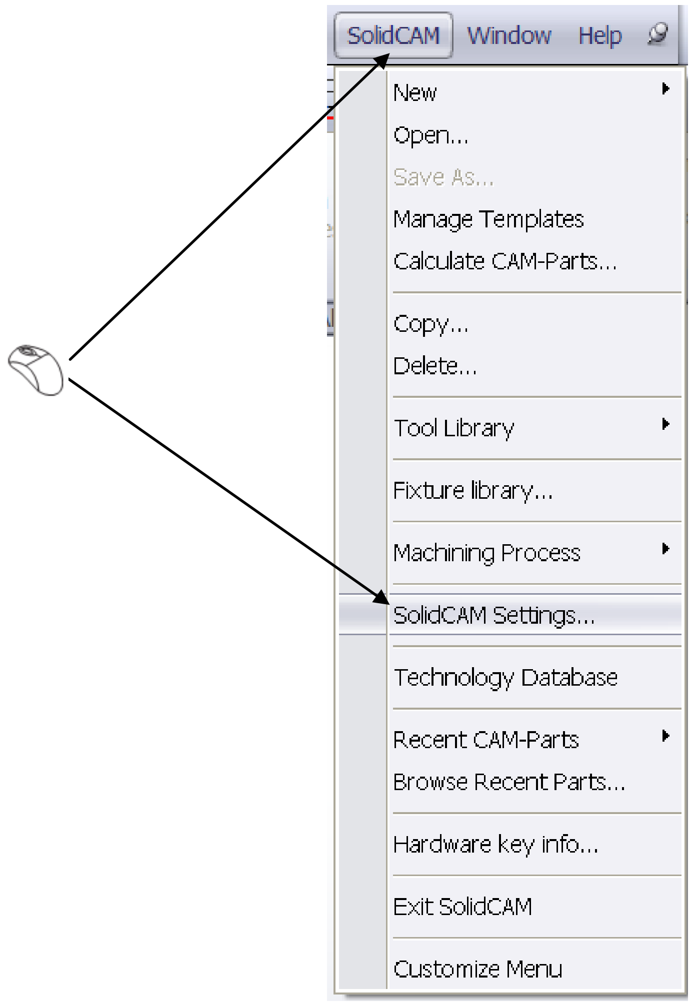
Do the following to open a part:

Double click the **SolidWorks** icon and open **Basic Training.SLDPRT** part.

ⓘ Before starting SolidCAM, you need to do the following settings that will enable you to select the right post processor and the machine associated with the post processor. To do the settings:

Click **SolidCAM** tab in **SolidWorks**.

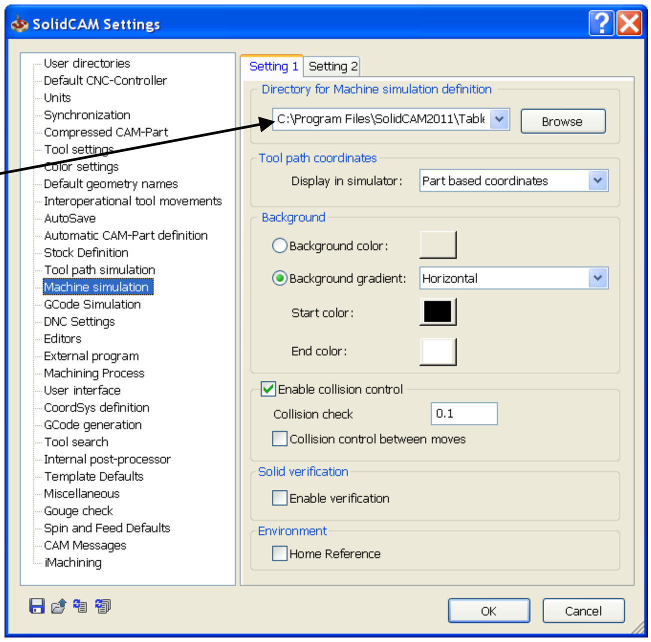
Click **SolidCAM Settings**.



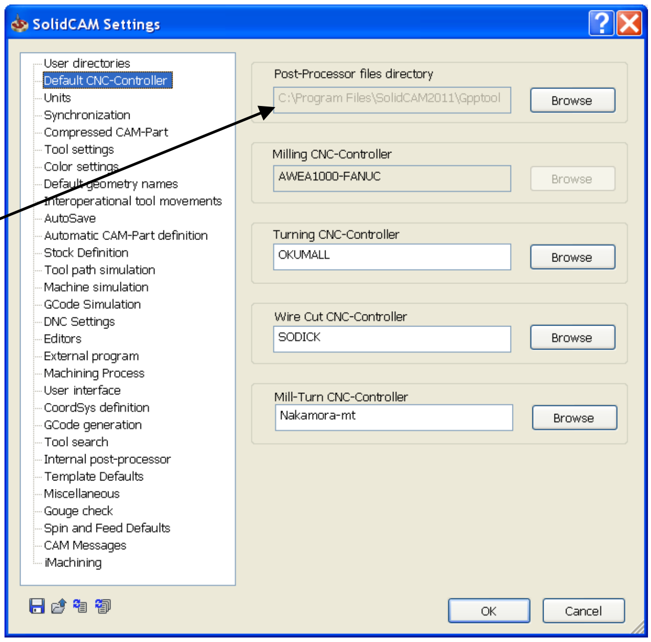
In **SolidCAM Settings**, you must concentrate on the following 2 tabs in the settings area:

- The path for machine simulation files
- The location of the post processor

Path for Machine Simulation Files

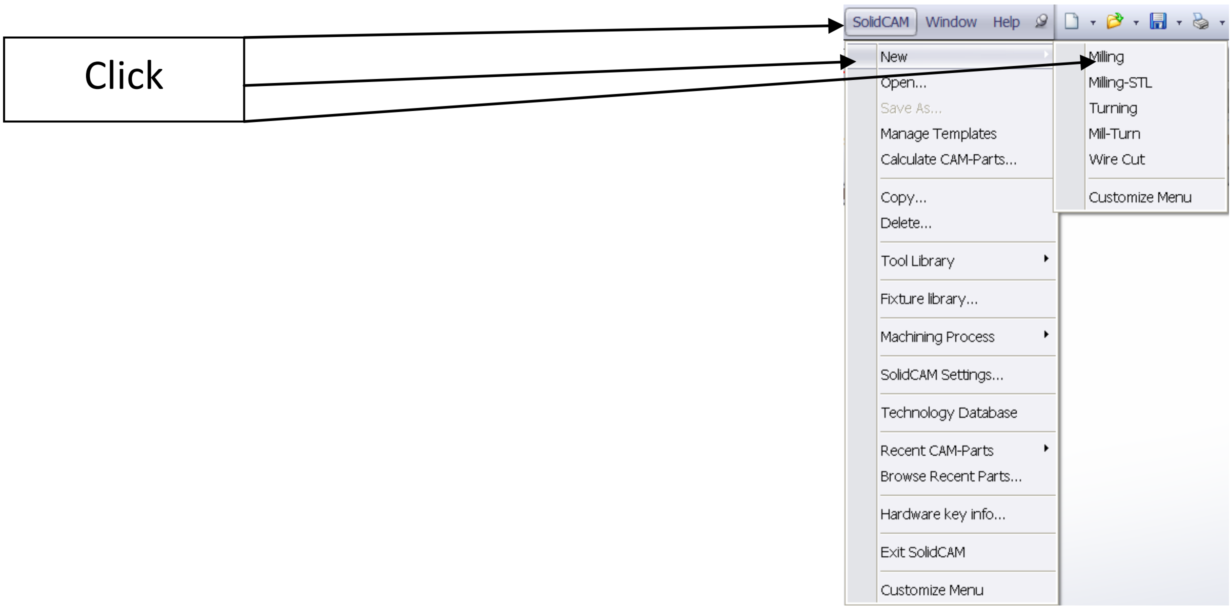


Post Processor Location

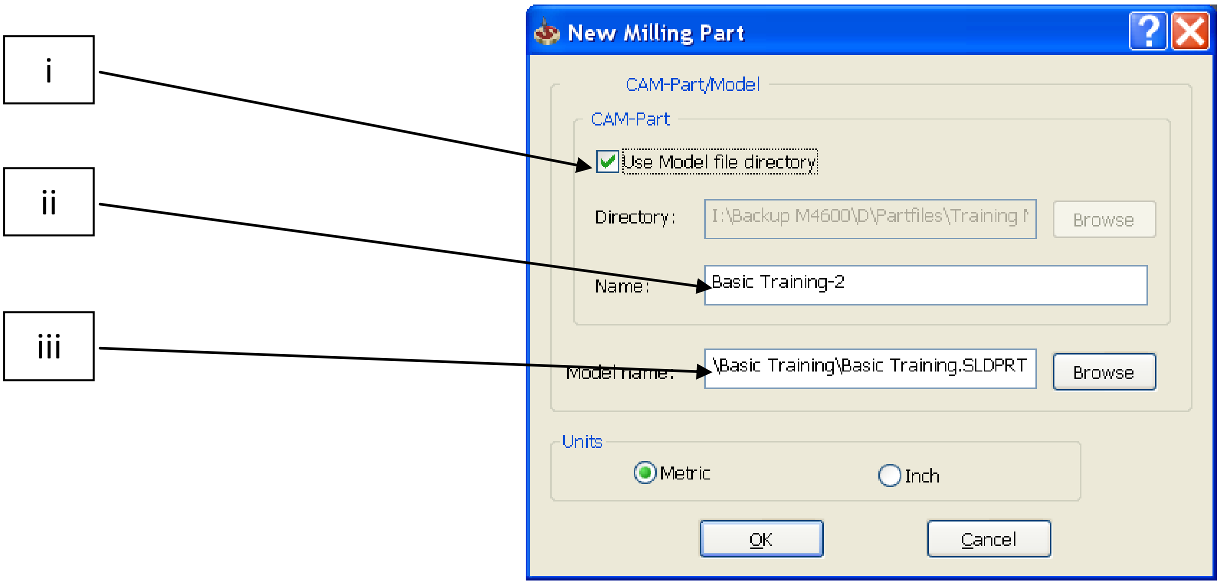


ⓘ By default, the post processor files and the machine simulation files are located on this path: /root/Program Files/SolidCAM/Gpptool. But, it is a good practice to create separate folders in the root directory for the post processor and the machine simulation. For example, C:/GPP & C:/Machine_Sim. The advantage of creating separate folders is that the post processor and machine simulation files remain intact even when the SolidCAM version is uninstalled or upgraded.

- Click **OK** to exit the **SolidCAM Settings** window.
- Click the **SolidCAM** tab.
- Click **New** > **Milling**.



The following dialog box appears:



i. If the **Use Model file directory** check box is checked, SolidCAM saves the entire project in the location of the SolidWorks part file. If the box is not checked, it stores the project in the "User" directory located in /root/Program Files/SolidCAM/. It is recommended that you select **Use Model file directory** check box.

ii. Name of the CAM part. The name of the part is always derived from the SolidWorks part file name. If a CAM part already exists, then SolidCAM uses the suffixes -1, -2 to create alternate names.

iii. Path and Name of the SolidWorks part file.

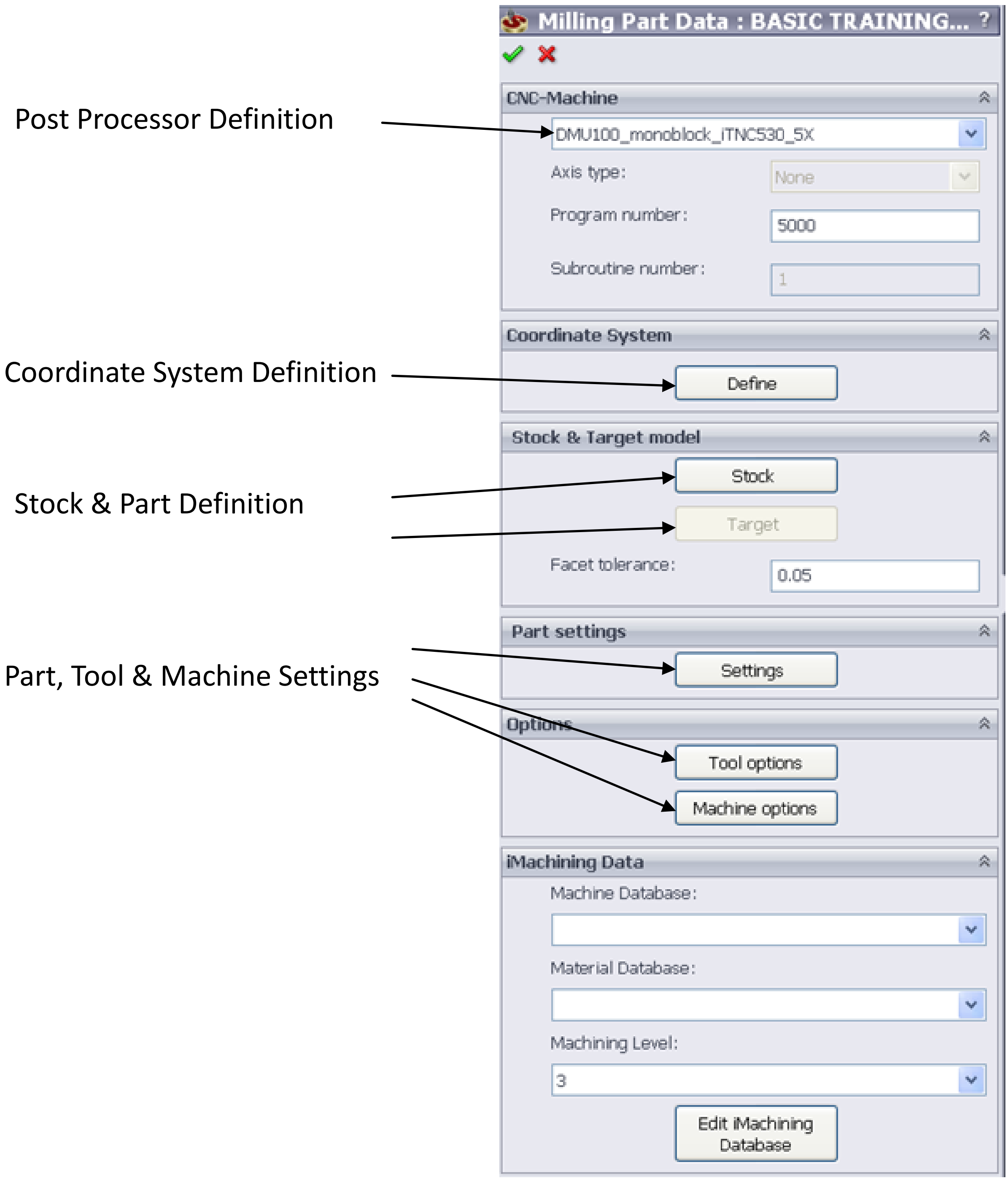
Click **OK** to continue.

End of Chapter 1

Chapter 2

Milling Part Data

In the SolidCAM environment, do the following steps before defining the first 5 Axis toolpath:

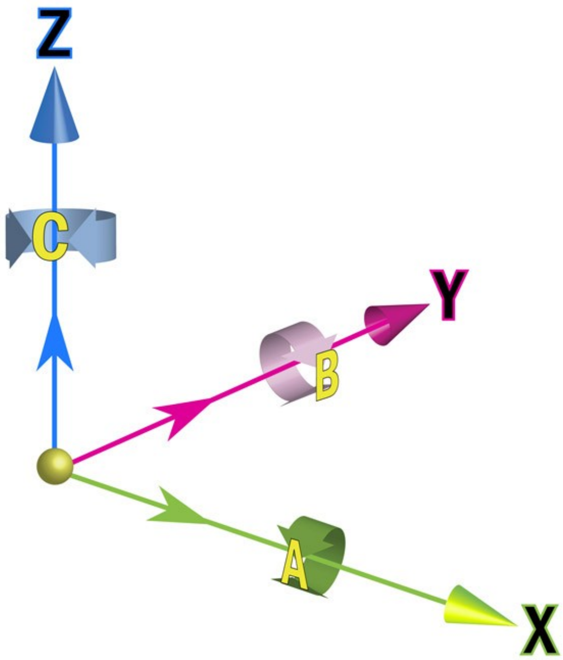


Post Processor Definition

To define a post processor:

Select the post processor/machine that will be used to machine the part. The selection of the post/machine depends on the part to be machined and the availability of the machine with the customer.

ⓘ In this example, if you closely observe the part, you will notice that we need to machine the periphery of the part and also cut some pockets. Although a standard Table-Table Machine would serve the purpose, it is better to machine this on a Head-Table configuration of the machine. As we have a cylindrical part, it is advisable to use the DMU 100 Monoblock kind of a machine which has a tilting head and a rotary table. In our case, the rotary axis of the table is around X axis. This axis is also known as the A axis. The following figure explains the industry standard nomenclature that is used to represent axis:



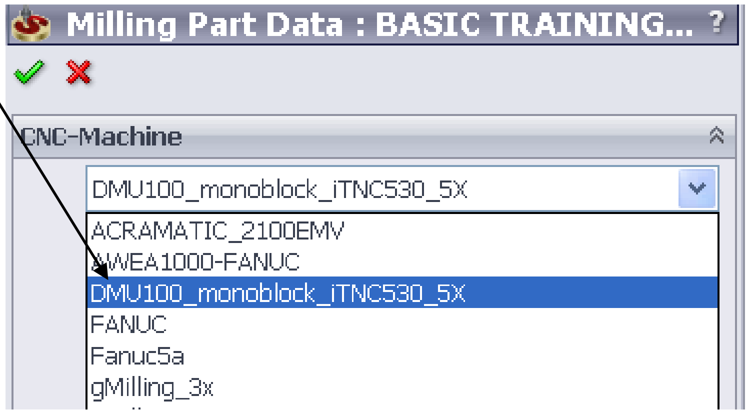
Rotation Around X Axis = A Axis

Rotation Around Y Axis = B Axis

Rotation Around Z Axis = C Axis

ⓘ Our part will rotate around X axis and the head will tilt around Y axis. So, this machine will have A and B axis.

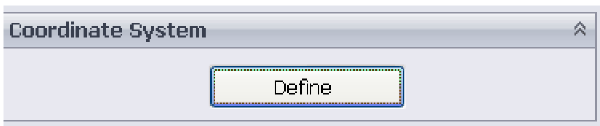
Select DMU100_monoblock_iTNC530_5X.



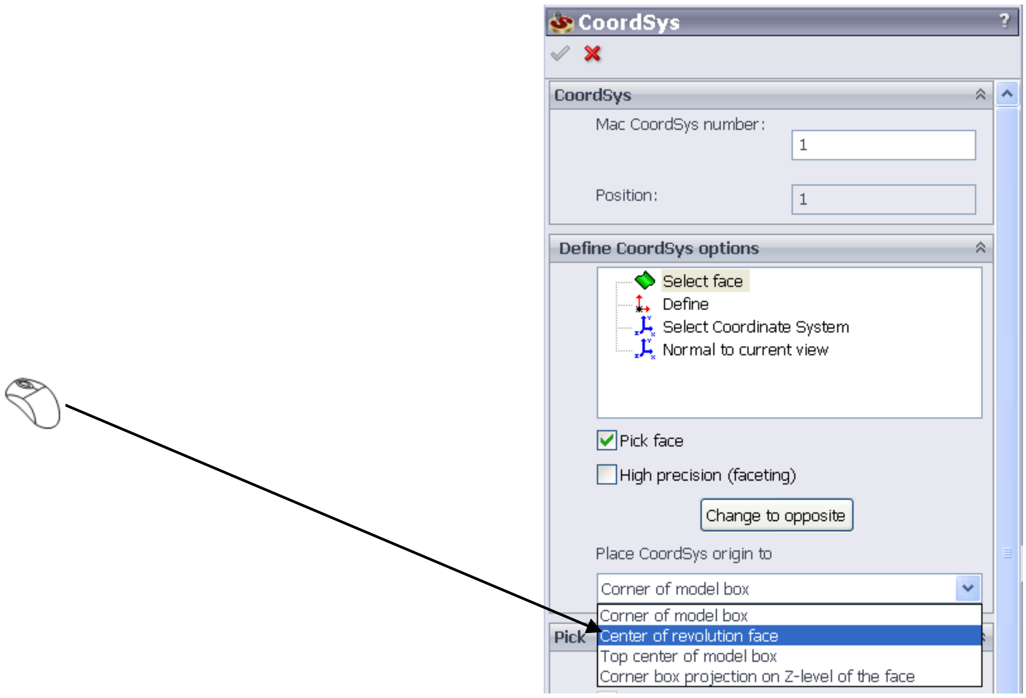
Coordinate System Definition

ⓘ The coordinate system on the part must reflect the machine axis. In our case, we have a cylindrical part, and one of the axis of the machine is rotating around X axis. Thus, the X axis of our coordinate system must be the rotating axis. Do the following:

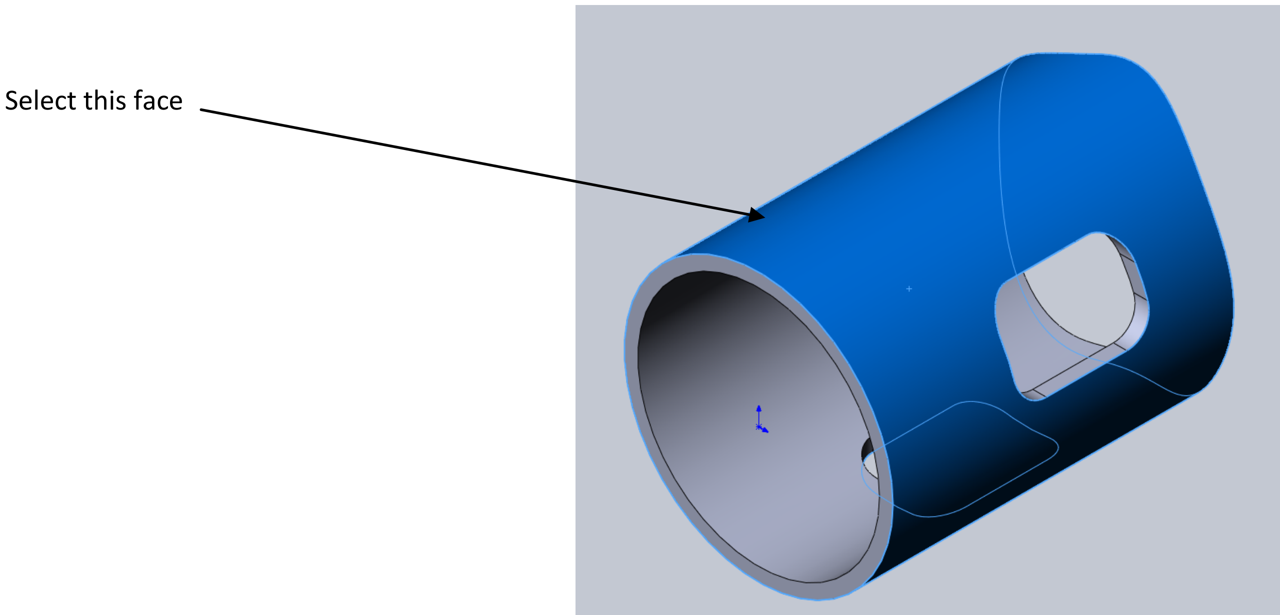
Click the **Define** button on the **Coordinate System** tab.



A window opens which has the options to define a coordinate system. Use the most relevant option for cylindrical parts. The most relevant option is to align one axis along the axis of revolution of the cylinder. Select **Center of revolution face** from the drop-down for defining **Place CoordSys origin to**.

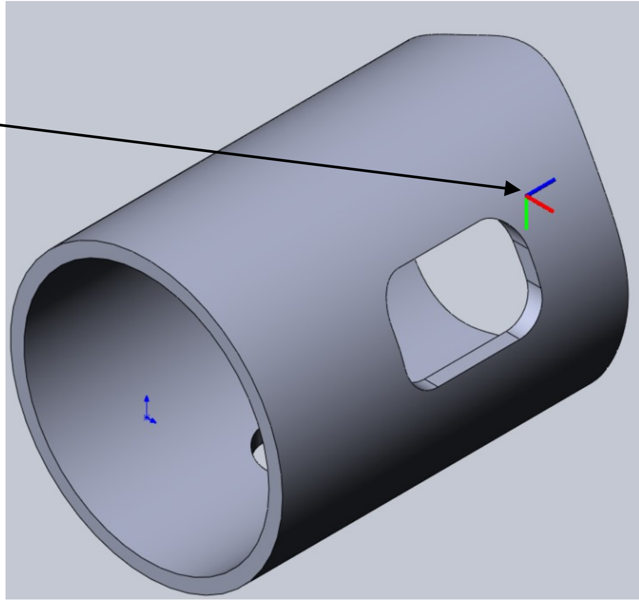


Select the outer or inner cylindrical face of the part.



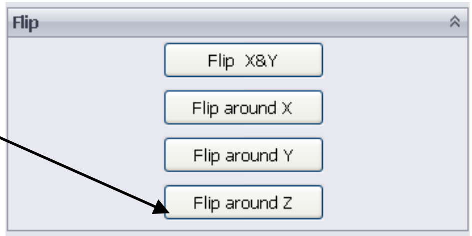
The coordinate system is now set with one of the axis. In our case, the Z axis is aligned to the axis of revolution of the cylinder.

"Z" Axis Aligned to axis of revolution



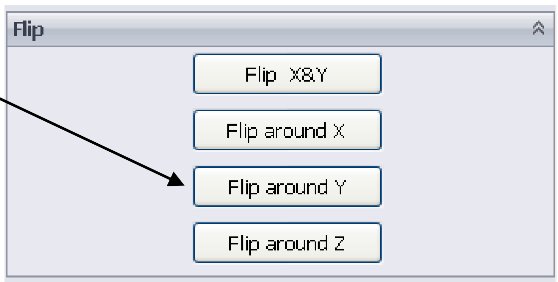
As our axis of rotation on the machine table is around X axis, we must align our X axis along the axis of revolution of the cylinder. And, the Z axis must denote the depth axis.

In short, we need to swap the place of Z axis with X axis and the place of Y axis with the Z axis. You can do this using the **Flip** functionality available in SolidCAM.



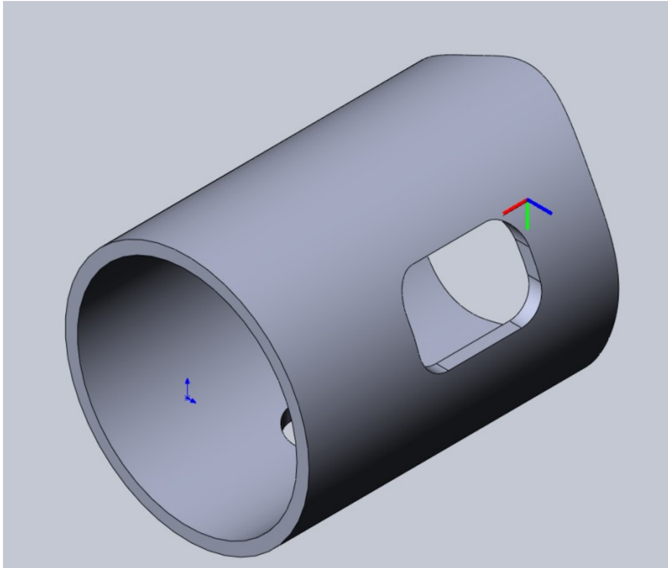
To get the X axis in place of the current Z axis, you must flip the axis around Y.

Click the **Flip Around Y** button.

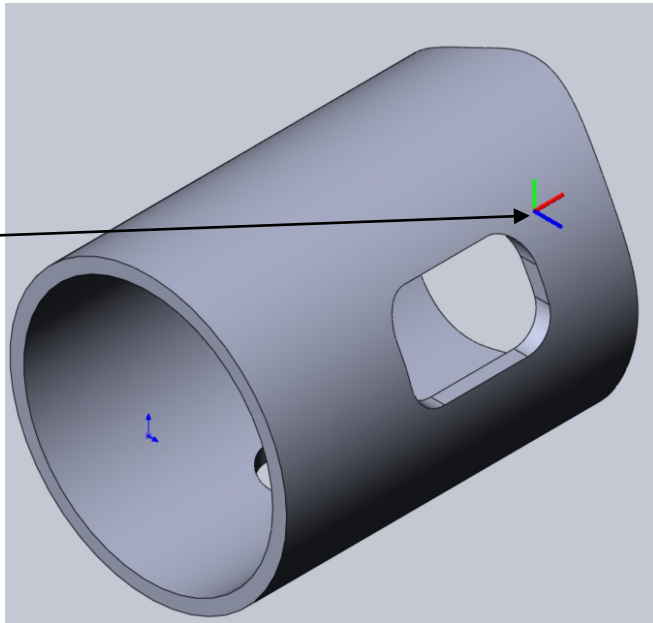


The coordinate system will look like this:

Now, the X axis is aligned to the axis of revolution of the cylinder, the Z axis is pointing to the depth, and the Y axis is denoting the tilting head. Depending on which side of the part is on the + X, click **Flip around Z** button twice. The resultant coordinate system will look like the below image:



Final coordinate system

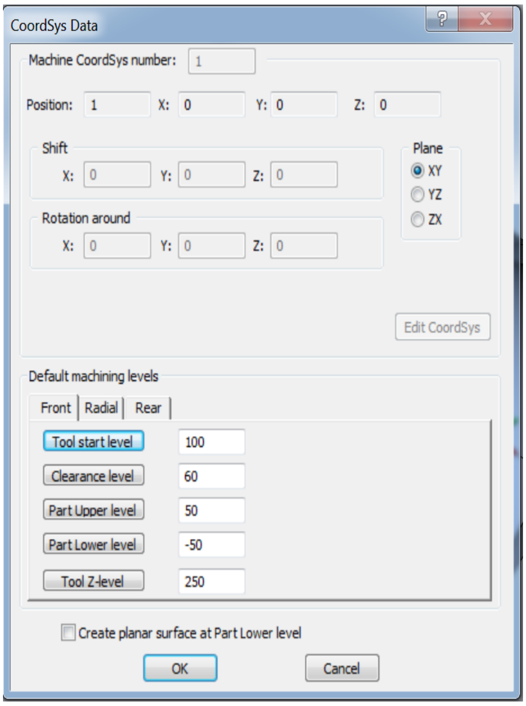


Click icon. The following dialog box appears:

Click icon to return to the **Milling Part Data**.

Accept the default values as shown in the table and click **OK**.

Revisit **Machine Setup** , when you are about to simulate using the machine.



Click icon to return to the **Milling Part Data**.

Stock & Part Definition

We will skip the stock definition now as this will be done later during the exercise.

To define a part, do the following:

In **Milling Part Data**, click the **Target** button in the **Stock & Target model** option.

SolidCAM has already defined the target by selecting the available geometry.

Click icon to return to the **Milling Part Data**.

Click icon and we are now ready to define our first 5 Axis Toolpath.

End of Chapter 2

Chapter 3

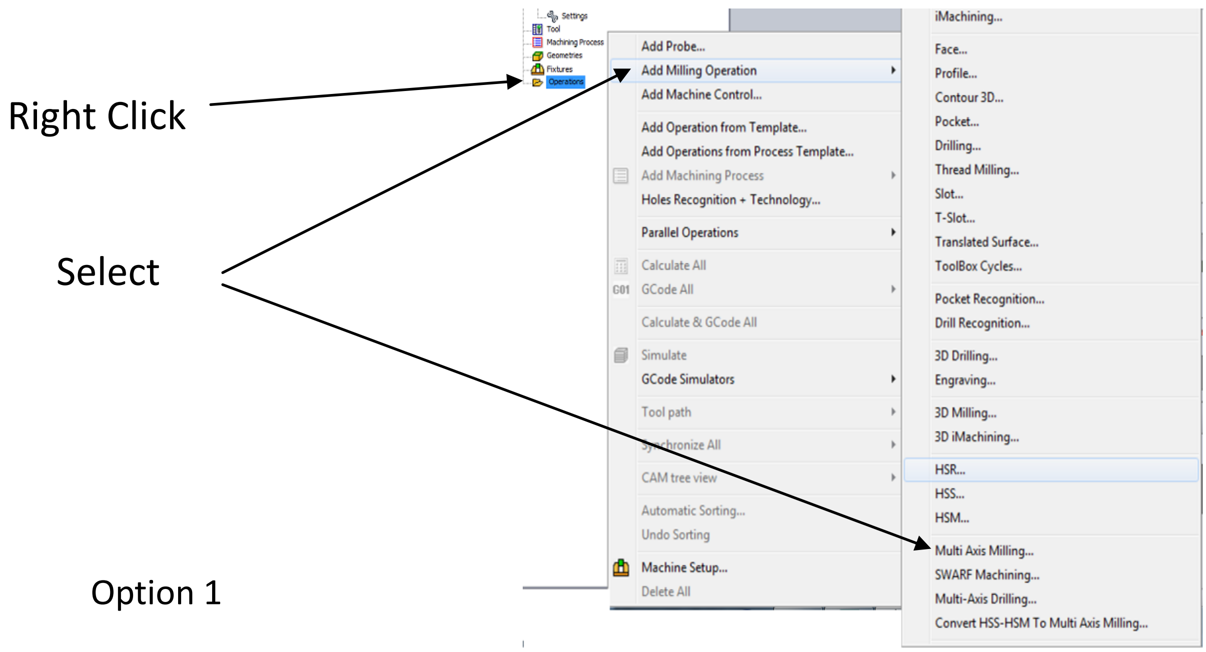
Creating 5 Axis Toolpath

We will now create our first 5 Axis Toolpath and learn some basic functionality in SolidCAM 5 Axis Simultaneous Milling.

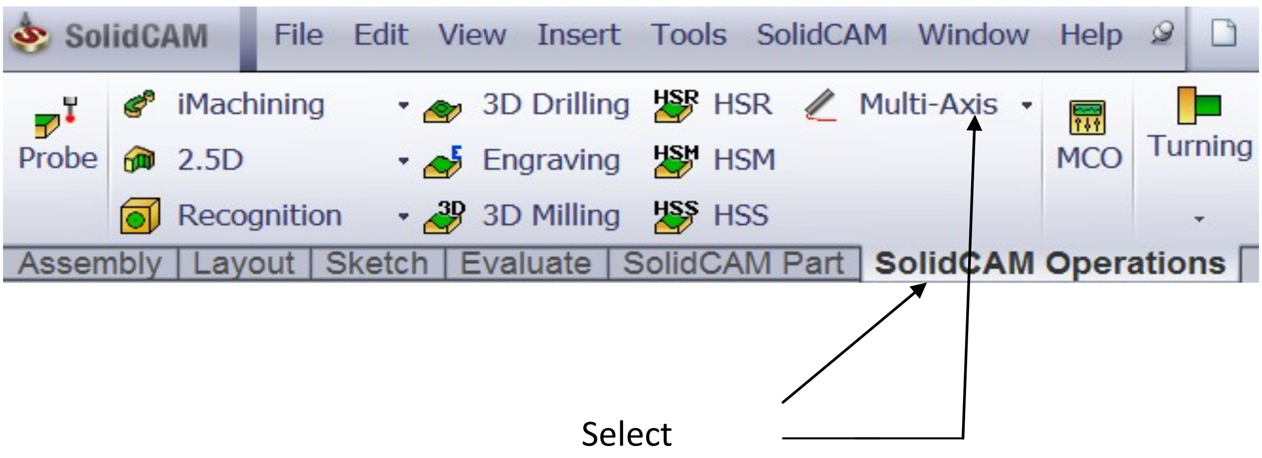
Do the following:

- In the toolpath manager, right click **Operations**.
- Click **Add Milling Operation**.
- Select **Multi Axis Milling**.

You can also select the 5 Axis function by selecting the **SolidCAM Operations** tab and then selecting the Multi-Axis icon.

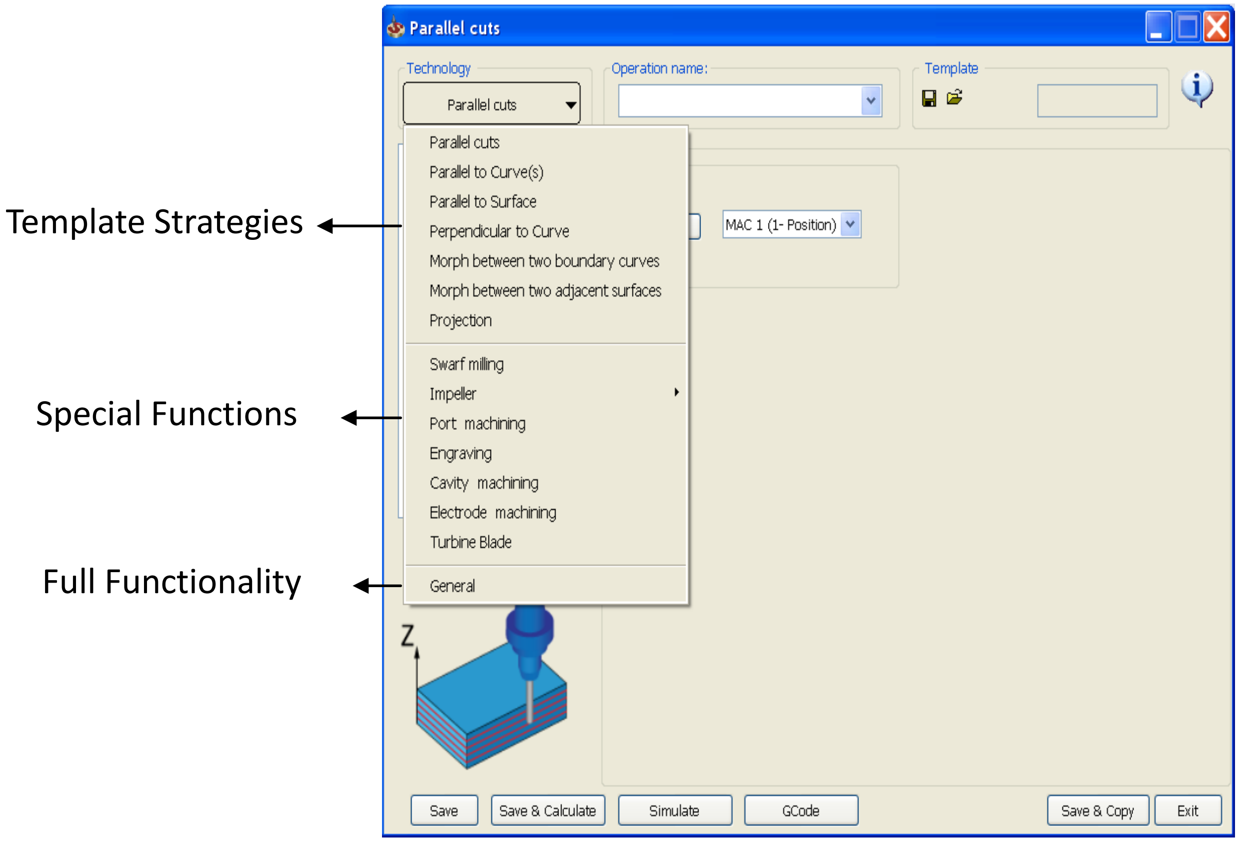


Option 1



Option 2

The following dialog box appears:

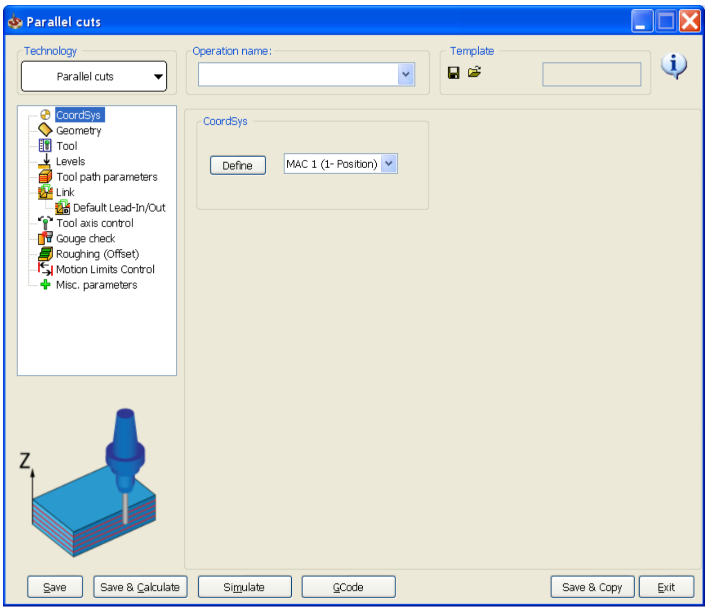


For this exercise, we will use the template strategies and learn about the special functions in our next training volumes. The template strategy used in this exercise is **Parallel cuts**.

Do the following:

Select **Parallel cuts** in the **Technology** section.

For this exercise, we will follow a chronological order of parameter definition. This will help you understand the functions in a better way.



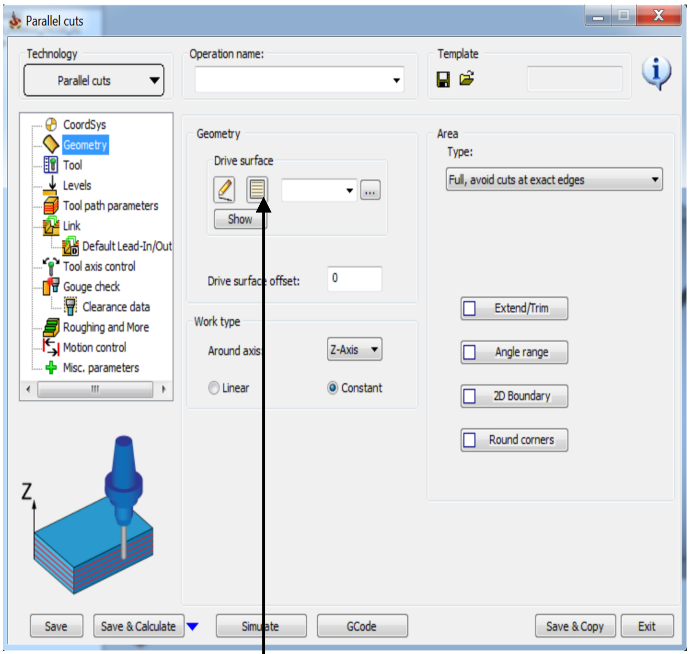
1. Geometry Selection
2. Tool Selection
3. Clearance Area Setting (levels)
4. Toolpath Parameters
5. Leads & Links
6. Tool Axis Control
7. Gouge Check Parameters
8. Roughing
9. Motion Limits Control
10. Misc Parameters

ⓘ SolidCAM 5 Axis always works with the 1st position of any coordinate system. For example, MAC1 POS1 , MAC2 POS1 , MAC3 POS1 etc. It never allows you to select any other positions of the coordinate system.

Click **Geometry**. The following dialog box appears:

The Geometry area is subdivided into 3 groups.

1. Geometry selection area
2. Work type
3. The area to be cut and other parameters

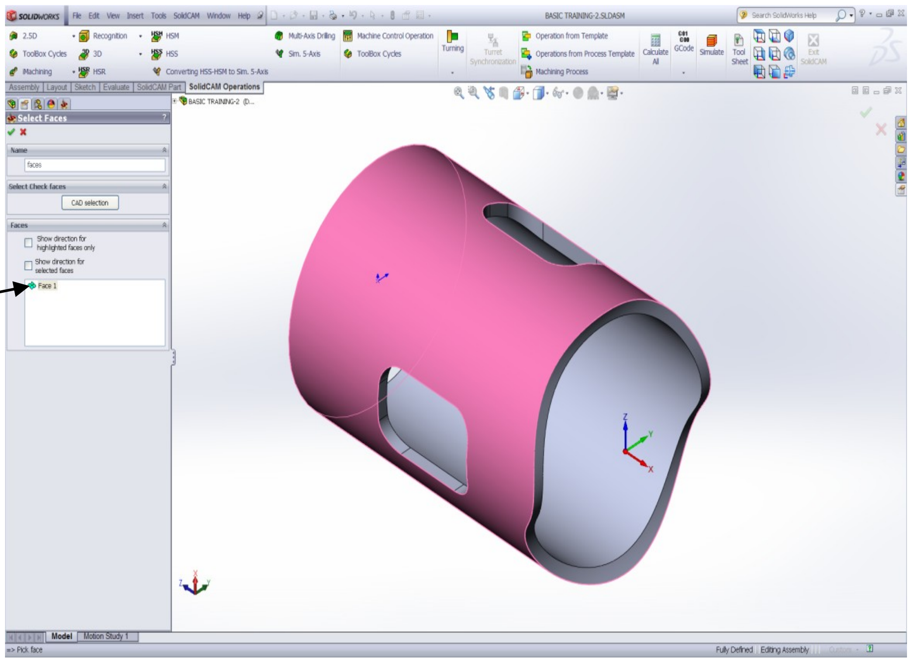


Select the geometry to be cut.

From now on in SolidCAM 5 Axis Milling, all the geometry to be cut will be referred as **Drive surface**.

Click New icon under **Drive surface**.

Geometry selected area

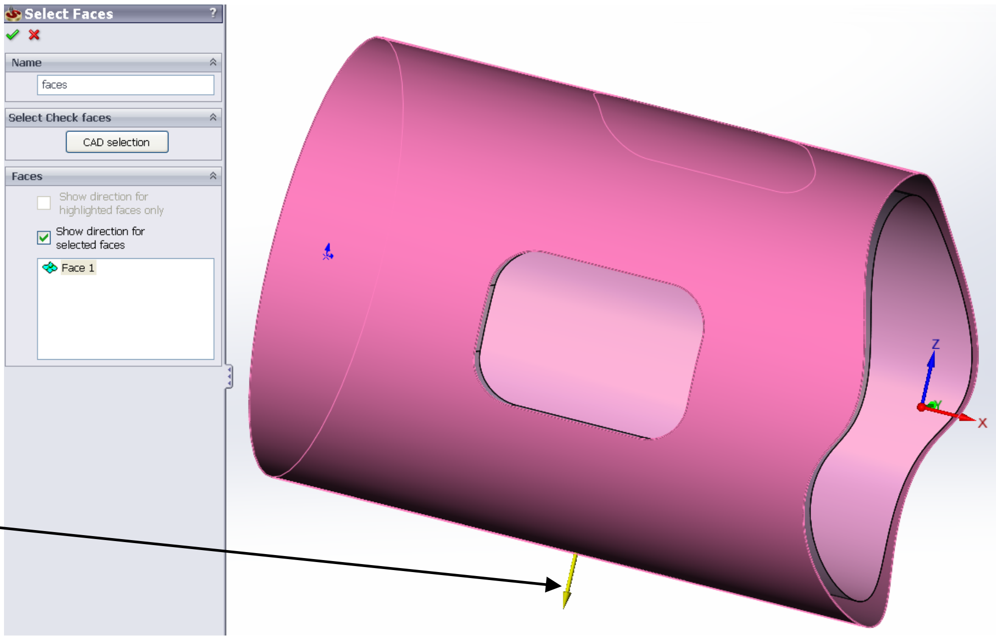


Select the highlighted face. The face is displayed in the geometry selected area on the left side.

Every surface has 2 directions. The outer direction and the inner direction. Depending on which side of the part has to be machined you can select the direction. However, if there is a solid model, then the outer direction is automatically set.

Select the **Show direction for selected faces checkbox** to show the direction.

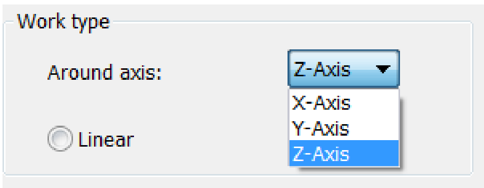
SolidCAM shows the outer direction of the surface. This direction is also the side which will be machined. The direction can be reversed by right clicking on Geometry and selecting “Reverse”.



Click  icon.

Set the **Work type**. The following options are available to us:

Around "X" Axis
Around "Y" Axis
Around "Z" Axis



Around "X" Axis : The cuts must be normal to X Axis

Around "Y" Axis : The cuts must be normal to Y Axis

Around "Z" Axis : The cuts must be normal to Z Axis

To set the Work type, do the following:

In **Around axis,** select **X-Axis** from the drop-down.

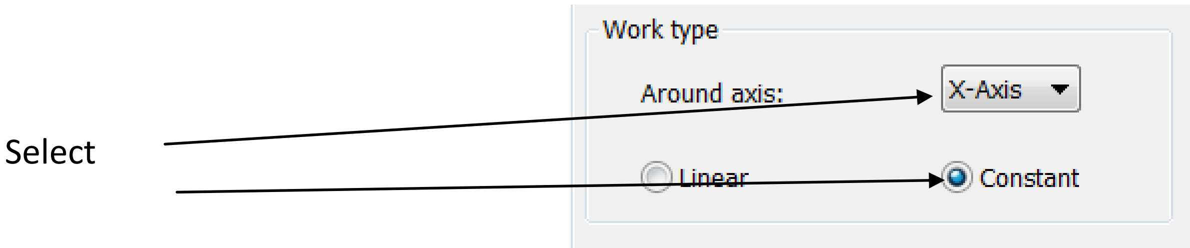
Select the **Linear** radio button to define an angle on the plane that is normal to the selected axis. For example,

Linear Option for Around X Axis allows you to define an angle in the YZ plane.

Linear Option for Around Y Axis allows you to define an angle in the ZX plane.

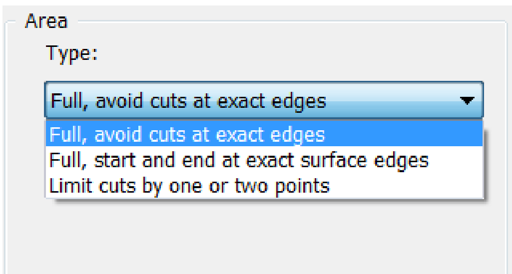
Linear Option for Around Z Axis allows you to define an angle in the XY plane.

For this example, we will use the **Constant** option for Around X Axis. Select **X Axis**. By default, the **Constant** is selected.



Select the **Type** in the **Area** section. This option allows us to set how much area of the drive surface should be machined. Following three options available for the selected strategy (i.e. Parallel cuts):

- Full, avoid cuts at exact edges
- Full, Start and end at exact surface edges
- Limit cuts by one or two points.



Let us examine the three scenarios.

Full, avoid cuts at exact edges:

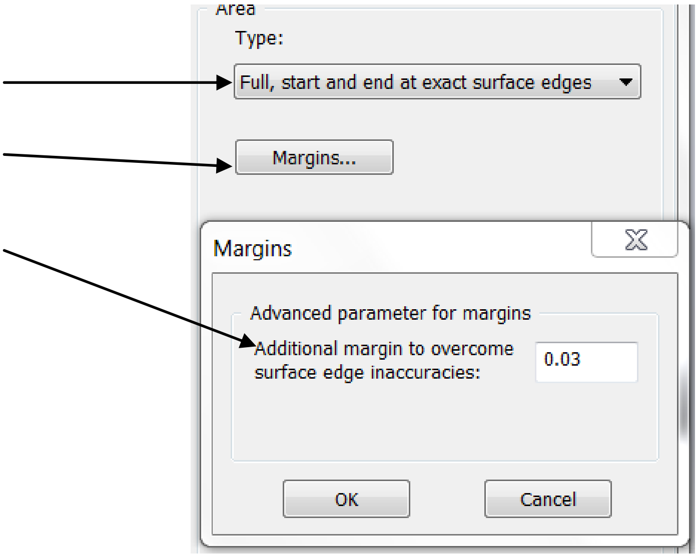
This option creates a toolpath for the entire surface. However, it avoids creating the toolpath at the trimmed edges of the surface. This is useful when we do not want to blunt/spoil an existing edge which is already machined or, when the model is imported from another CAD system as the surfaces which are imported from other CAD system can have imperfect trimmed edges. This can result in the toolpaths having many jumps to the retract plane.

ⓘ The value by which the toolpath starts from inside the surface edge is determined by the **Maximum Step Over** parameter in the **Toolpath Parameters** tab.

Full, Start and end at exact surface edges:

This option creates a toolpath from the exact edge (both start and end edges). In this option we can give a margin by which the toolpath will start from inside the surface edge. The value is to compensate for the inaccuracies in the trimmed edge.

If you select Full, Start and end at exact surface edges, you see these options.

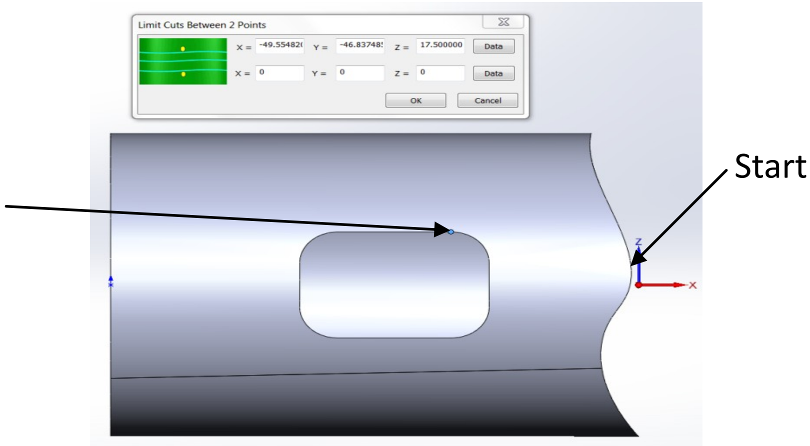


ⓘ It is always a good practice to give an allowance which is either default (0.03mm) or something close to it. Making the additional margins close to 0 creates unnecessary jumps to the clearance plane because there are inaccuracies in the trimmed surface edges.

Limit cuts by one or two points:

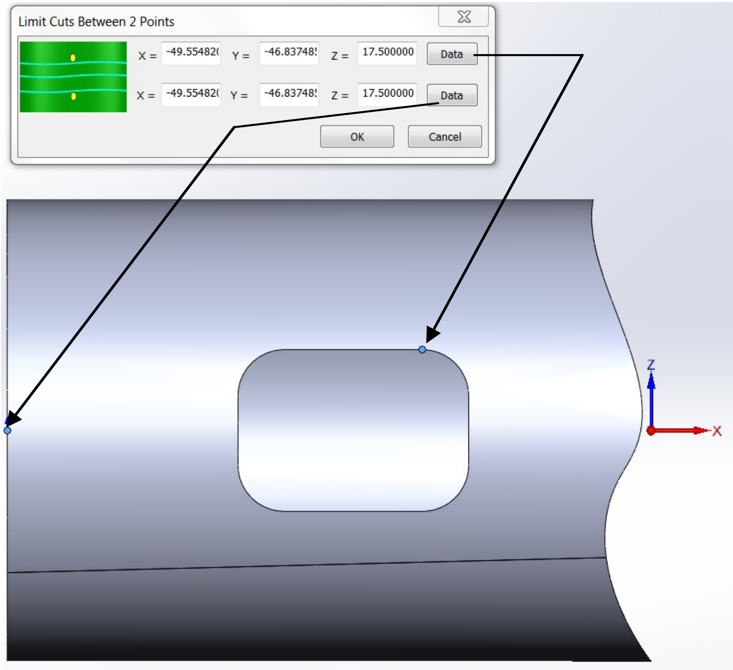
This option allows us to create a toolpath on the surface using the following scenarios:

i. Selecting only the start point ends the toolpath at that point and from the start edge of the surface. The following figure explains this scenario:



ii. Selecting the second point ends the toolpath at that point similar to what is explained in option 1.

iii. Selecting both the points creates the toolpath between the two points. This option is very similar to defining an area for machining. The only difference is that two points are used to define the area.



Select the first option - **Full, avoid cuts at exact edges**.

The next step is to define the tool. We will define a 6 MM Ball Nose End Mill with a suitable holder. To define a 6 MM Ball Nose End Mill, do the following:

Select **Tool** in the toolpath manager.

Click the **Select** button.

Click  (Add Milling Tool) icon.

Click Ball Nose Mill.

Click the **Select** button.

The next step is to define the clearance area type and values for clearance definition. We will look at this step after we have generated the toolpath, as this step will be clearer after the toolpath is created. For now, we will select/accept the default level and value settings.

Let us now define the toolpath parameters. Do the following:

Click the **Tool path parameters** in the toolpath manager. The **Tool path parameters** tab is sub divided into **Surface quality** and **Sorting** sub-tabs. As the name indicates, the **Surface quality** sub-tab greatly influences the quality of the surface that is achieved after machining. The **Sorting** sub-tab is used in sorting cuts as **Climb**/**Conventional** milling etc. Let us see some of the parameters under **Surface quality** tab.

We will look at the following three main parameters which will be defined in this toolpath:

i. Cut tolerance

ii. Distance

iii. Maximum Step Over/Scallop

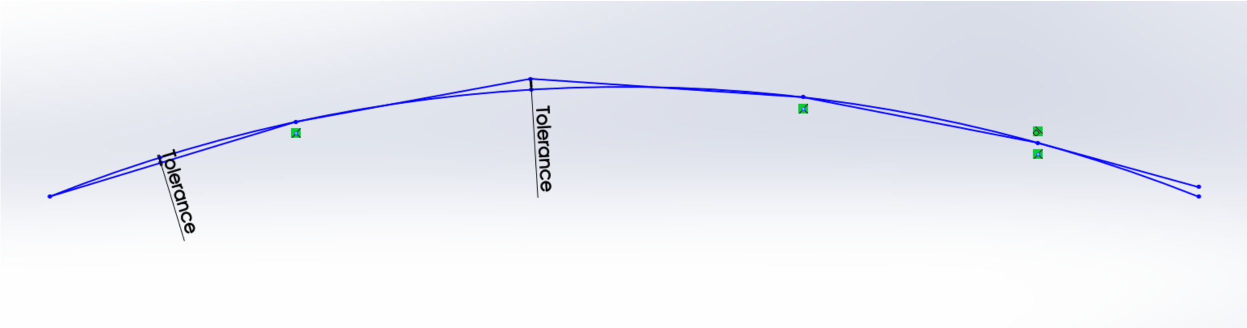
Cut tolerance :

This parameter is the tolerance with which the toolpath is generated. This parameter is "Bi-tol" which means any value defined as the tolerance is taken as +/-

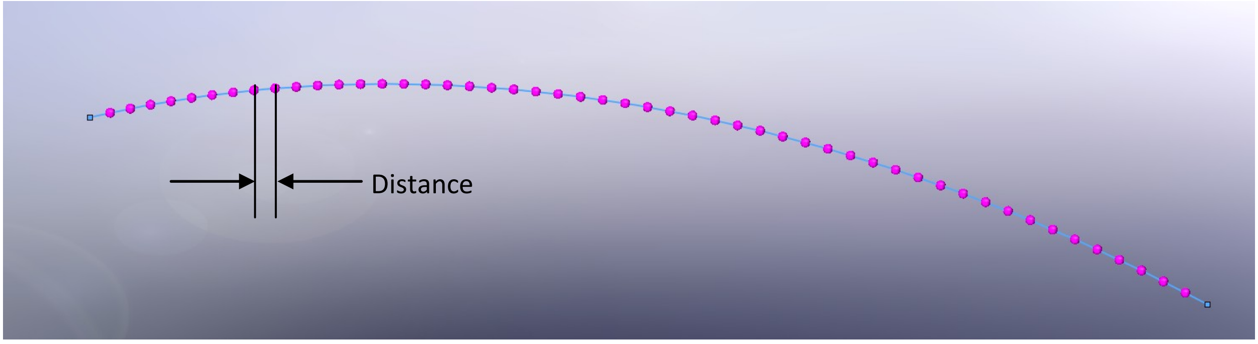
Note that here the 5 Axis Simultaneous directly uses surfaces to perform the calculations. Unlike HSM which uses intermediate mesh to do the calculations. A mesh is generally made up of triangles which is the simplest geometry to calculate on. Therefore, if the surfaces are of good quality, even a slightly loose tolerance in the 5 Axis Simultaneous gives a very high quality surface finish. Tightening the tolerance results in higher calculation times and more number of points in the resultant Gcode file.

Distance :

This parameter is used to generate more number of points with equal spacing between them. Generally, SolidCAM calculates points on a given surface based on tolerance. However, the number of points can sometimes be too less. For example, on a surface which has extremely large curvature radius or on small surfaces which can lose out on detail because of less number of points representing the surface. To overcome this, the distance parameter is used.



Points Generated on surface with a given tolerance



Points Generated on surface with a given tolerance & Distance

ⓘ Note that using the distance parameter increases the number of points appearing in the Gcode file in multiples. This can sometimes slow down old machines as they do not have the processing power to read in the large number of points. Therefore, this parameter must always be used judiciously.

Maximum step over :

This parameter is the maximum distance between two subsequent toolpath passes. Remember that if the distance is "Maximum" it means at some areas the distance can also be lower than what is defined. Changing the maximum step over influences the scallop value and vice versa.

Let us now set the above explained values:

Cut tolerance = 0.02

Distance = We will not use this parameter now

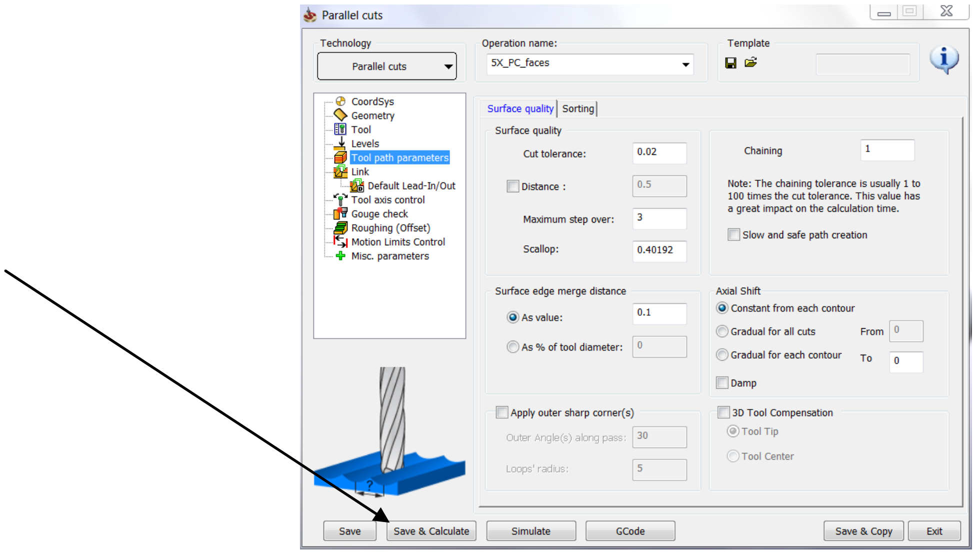
Maximum step over = 3

Right now, we will not touch the other parameters in the **Tool path parameters** tab, as these will be explained and demonstrated in the subsequent volumes.

We will use the default values in the other tabs as we will explain the effects of the same in the pages to follow.

Let us now create and visualize our first 5 axis toolpath.

Click



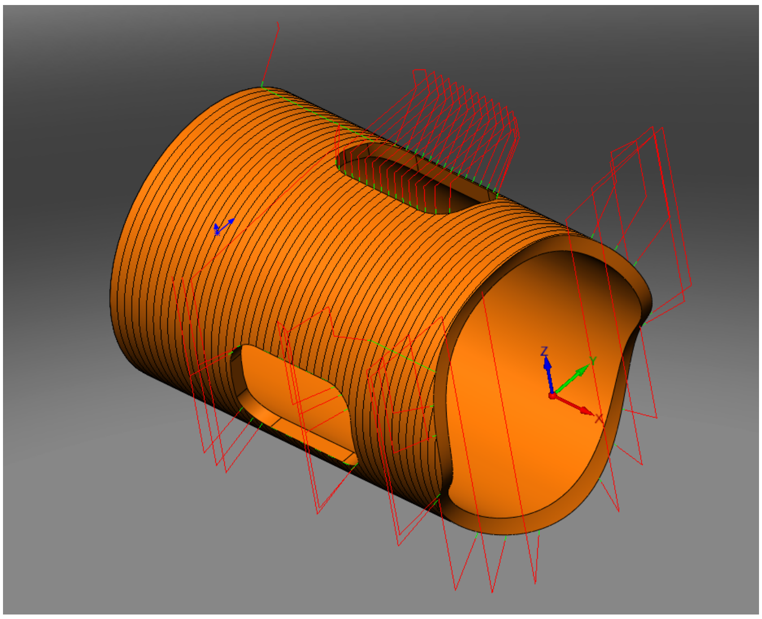
Click **Save & Calculate**.

The toolpath calculation in SolidCAM 5 Axis generally happens in the following three stages:

- Creating cut sections on the drive surfaces
- Linking the cut sections based on sorting and links
- Checking for gouges and applying the gouge check reactions

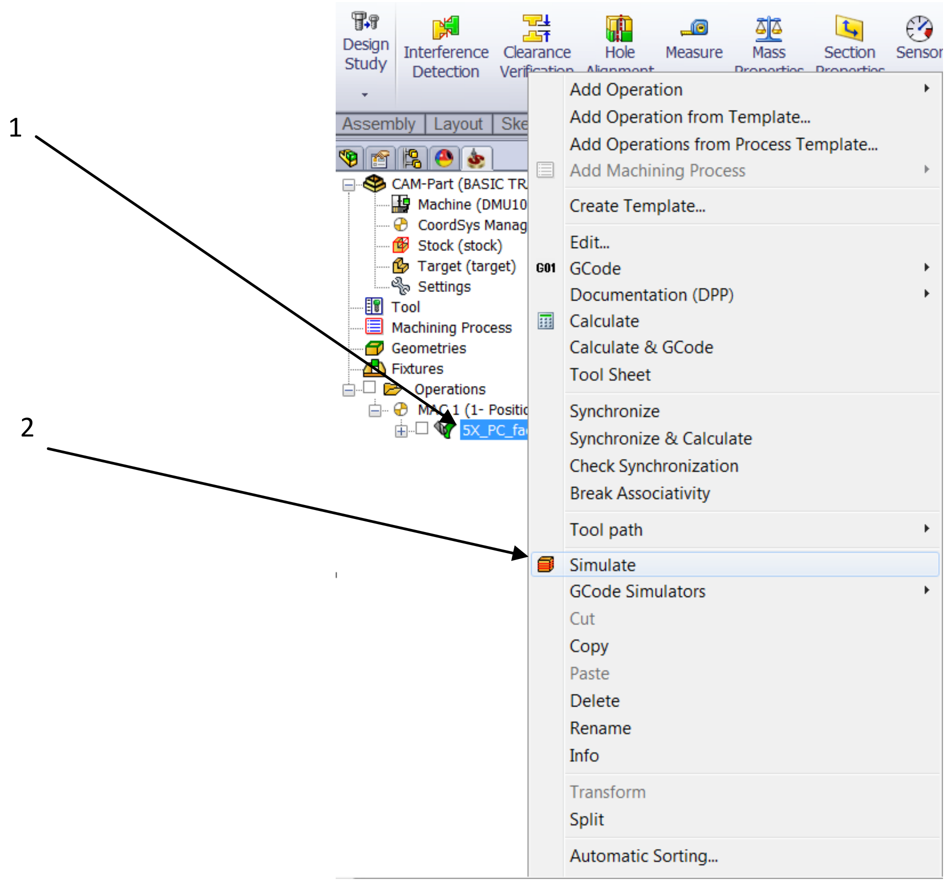
Click **Simulate**.

Select the  (play) icon to visualize our toolpath. Our tool path looks something like this:



Let us run the machine simulation to understand some crucial problems in our current toolpath. We will use DMU 100 Monoblock machine to simulate our tool-path.

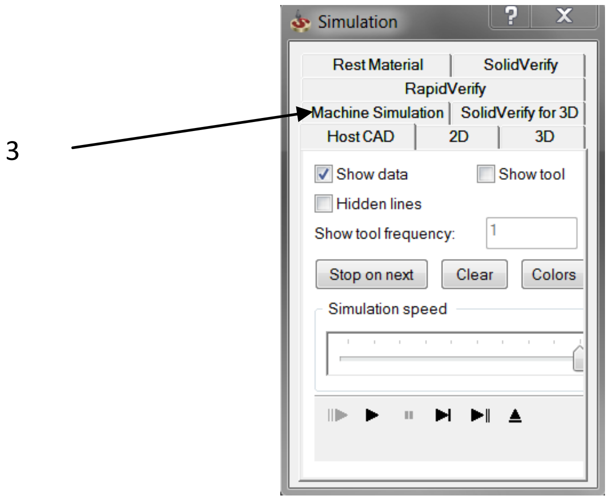
To start the simulation follow these steps:



Right click the toolpath.

Click **Simulate**. The simulation window opens.

Click **Machine Simulation**.

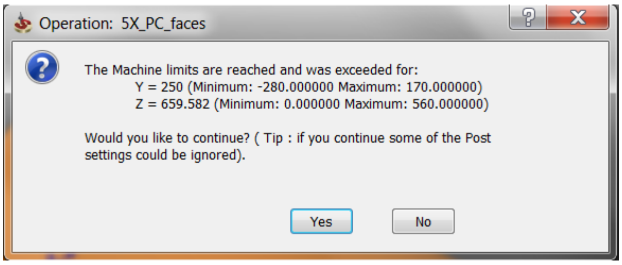


This step does the following :

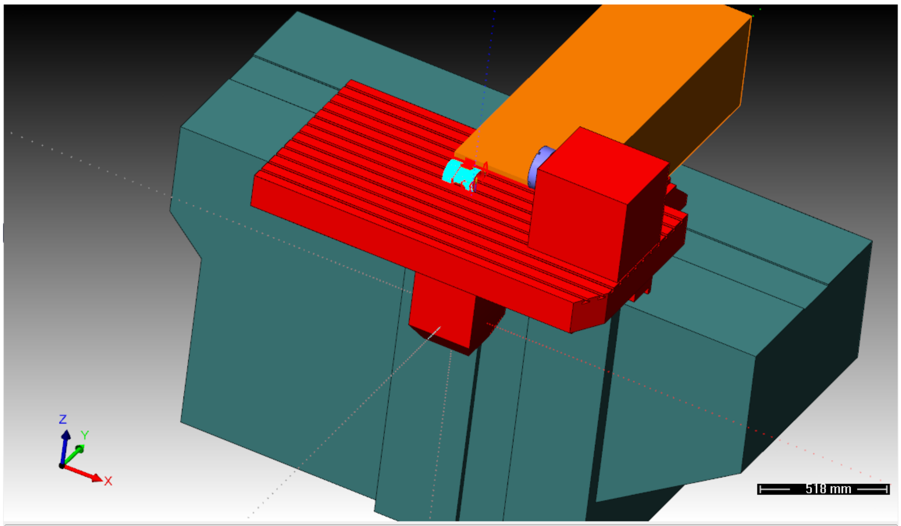
- Creates the part & stock STL and loads into the simulation
- Loads the toolpath with tools & holders
- Loads the machine from the machine sim folder

ⓘ Depending on the number of toolpaths and tolerance of the part, it may take several minutes to start the machine simulation.

The following error message occurs. Click **Yes** to continue and examine why this error occurred:



If you click **Yes** on the error message, you will see the following orientation of the machine inside the simulation:

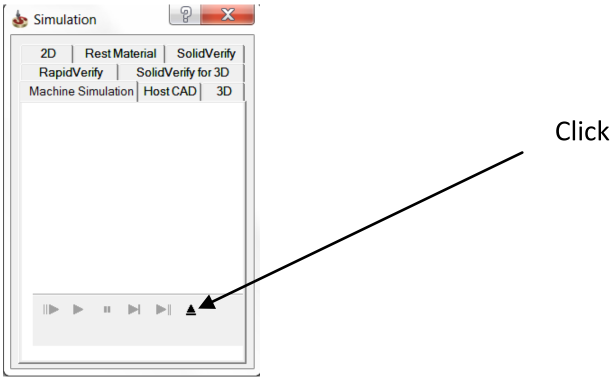


Here, the head has tilted in the opposite side of what was expected. Although this is mathematically correct, it is incorrect as far as the machine's kinematic movements are concerned.

ⓘ For any given part in such machines, the tool can approach the start point and machine the same in two orientations. Although in some cases both the orientations will work but in majority of the cases, only one orientation provides the right solution.

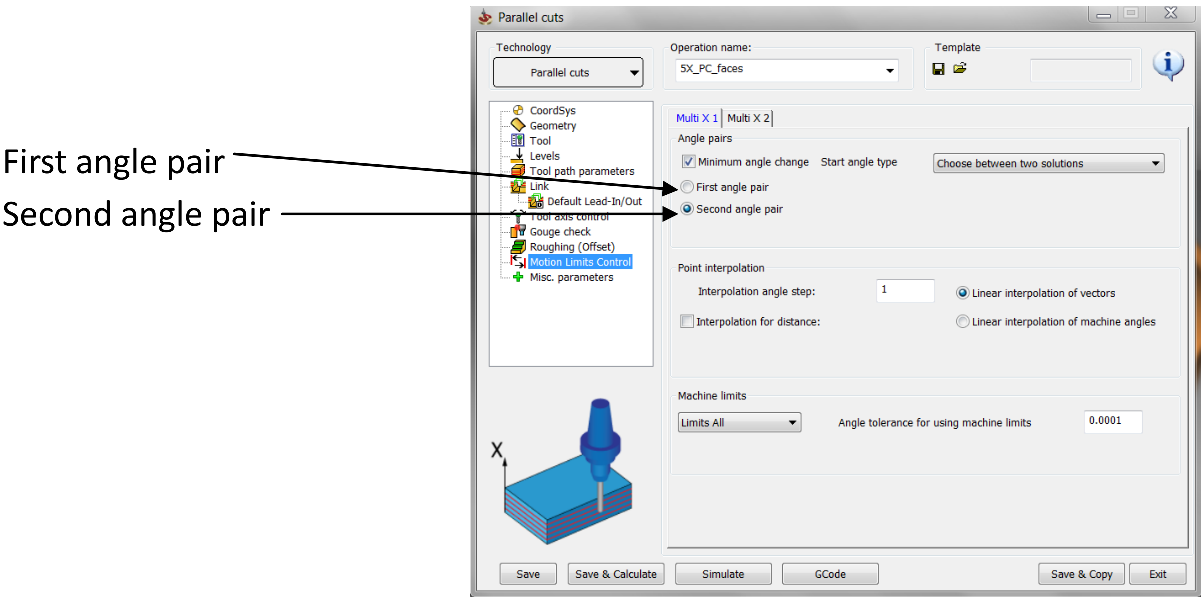
Let us now see how to fix this issue in SolidCAM.

Click the  (exit) icon to exit from the machine simulation.



Right click the toolpath and click **Edit**.

Select **Motion Control**. In the **Angle pairs** section, you can see that there are two orientations possible.



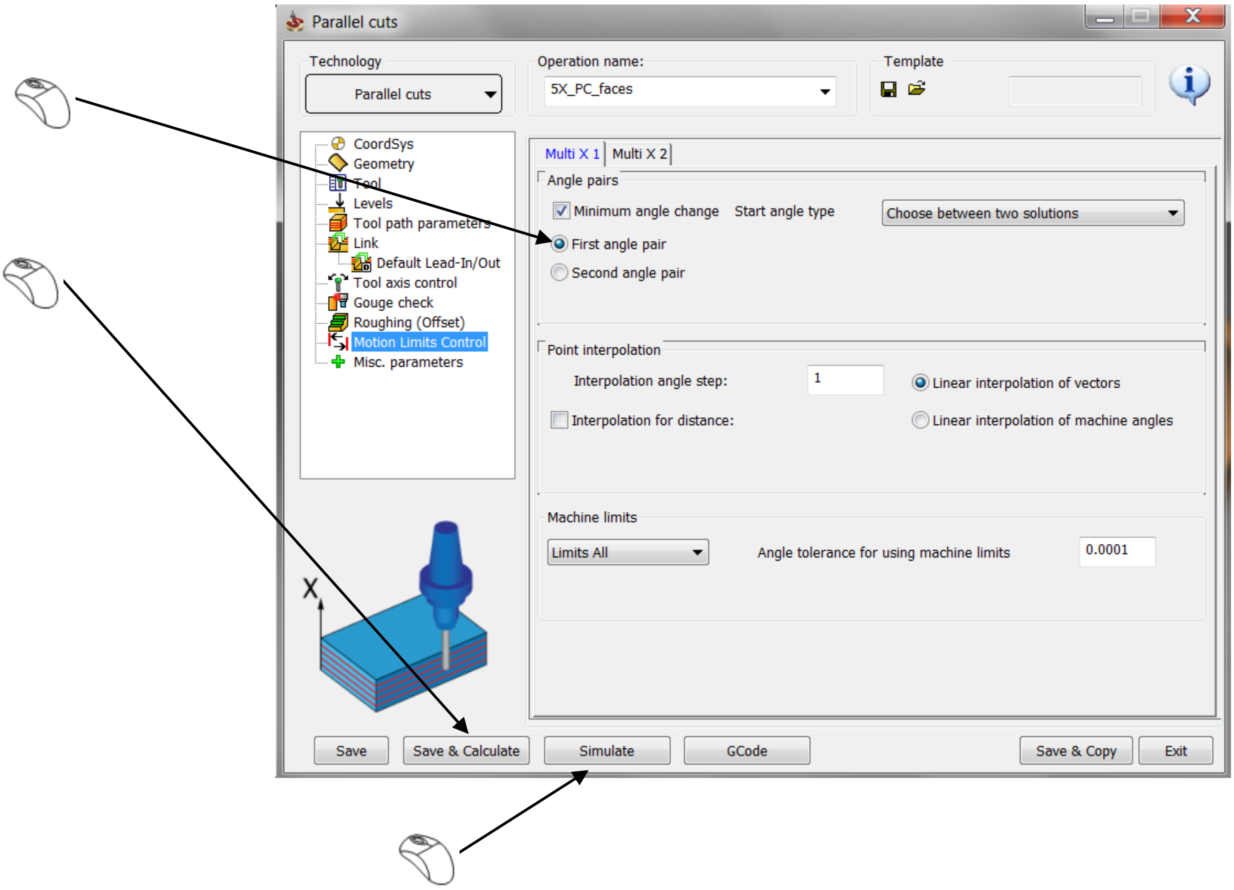
As the **Second angle pair** was selected, it created the second possible orientation. The default settings can be changed in the .PRP file of the post processor under **Multi Axis Milling** tab.

Select **First angle pair** radio button.

Click **Save & Calculate**.

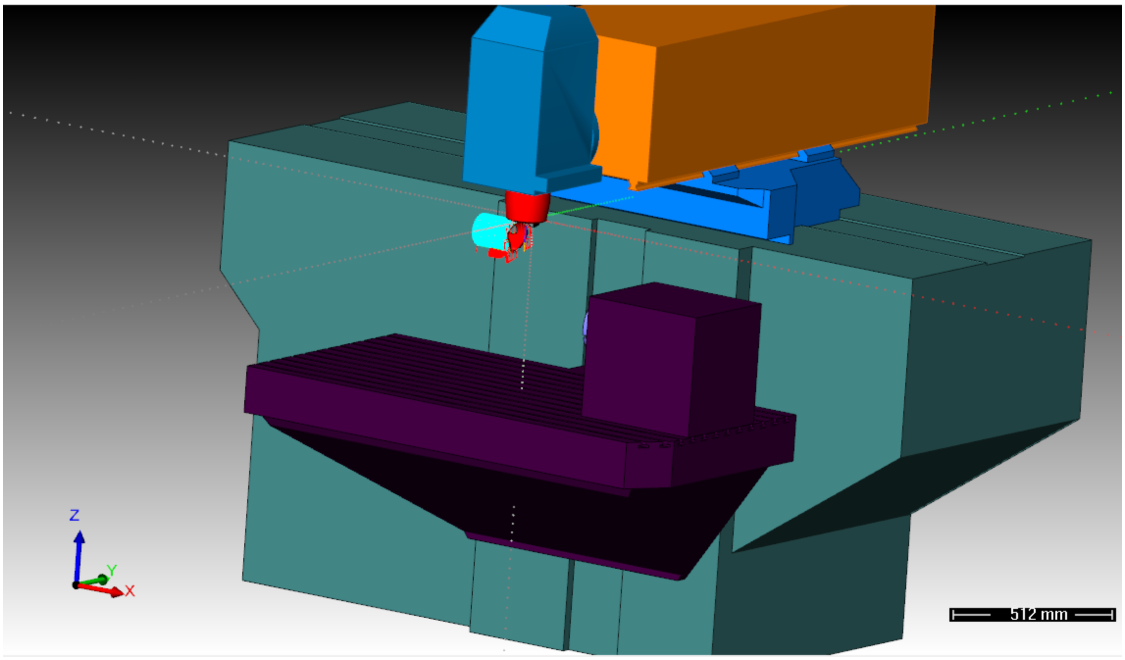
Click **Simulate** once the calculation is complete.

Select **Machine Simulation**.



The error which is seen in step 4, occurs again. Click **Yes** and continue. We will fix this error in the next few steps.

In the machine simulation environment, we can see that the orientation is corrected, and the machine simulation looks something like this:



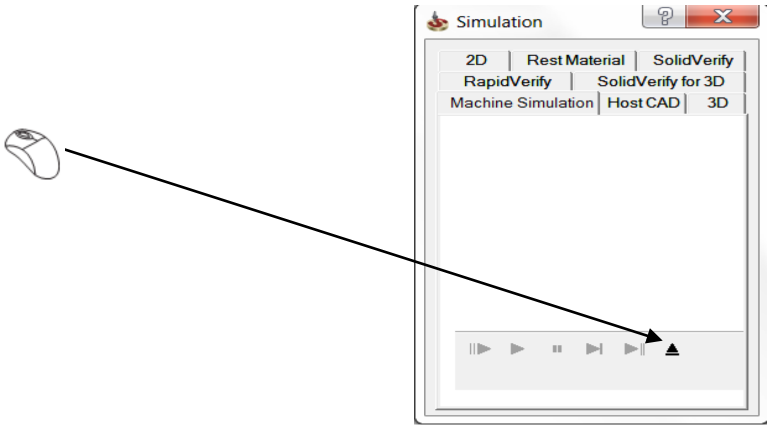
We still have two issues to fix.

Click  (run) icon and observe the following:

- There is a collision on the very first move between the head and part.
- The part is rotating off-centre from the rotating axis (A Axis).

Let us first fix the second problem and then the first one.

Click  (exit) icon to exit the machine simulation environment. The **Parallel cuts** pop-up window appears.

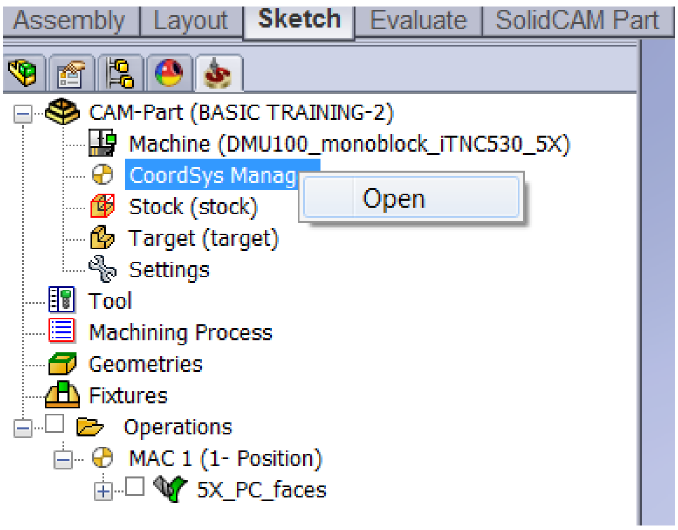


Click **Exit** to return to the toolpath manager.

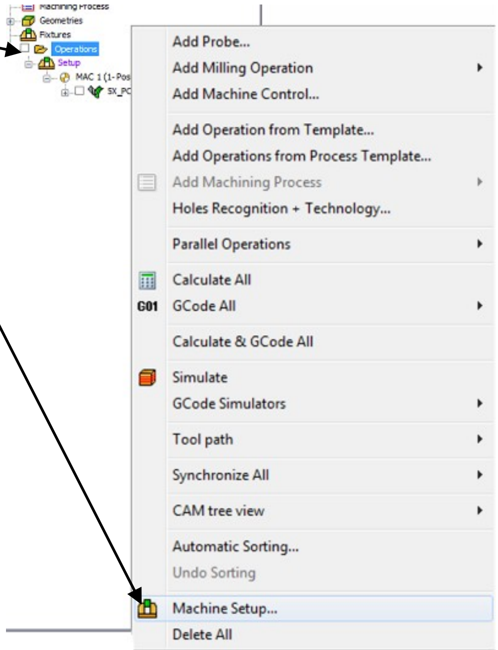
To fix the off centre issue, do the following:

Right click **CoordSys Manager**.

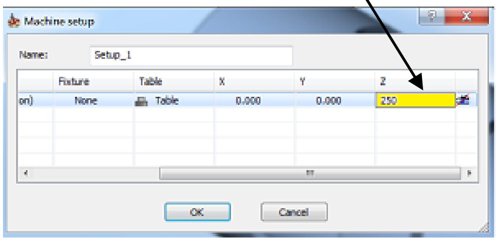
Click **Open**.



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



Enter a value of 250 in the Z column.



In the **Part's Origin Position** section, we need to enter the offset value in Z axis. Enter value 250. This value is the offset of the part's origin from the top face of the bed of the machine to the centre of the rotary axis. This value is needed as the origin of the machine (Z=0) is defined on the top face of the machine and SolidCAM will always align the origin of the part with the origin of the machine (In this case the Z0 of the part is aligned to the Z0 of the machine which is basically the top face of the machine bed). The distance from the top face of the machine bed to the rotary axis centre is 250 mm. Thus, we enter this distance.

Click **OK**.

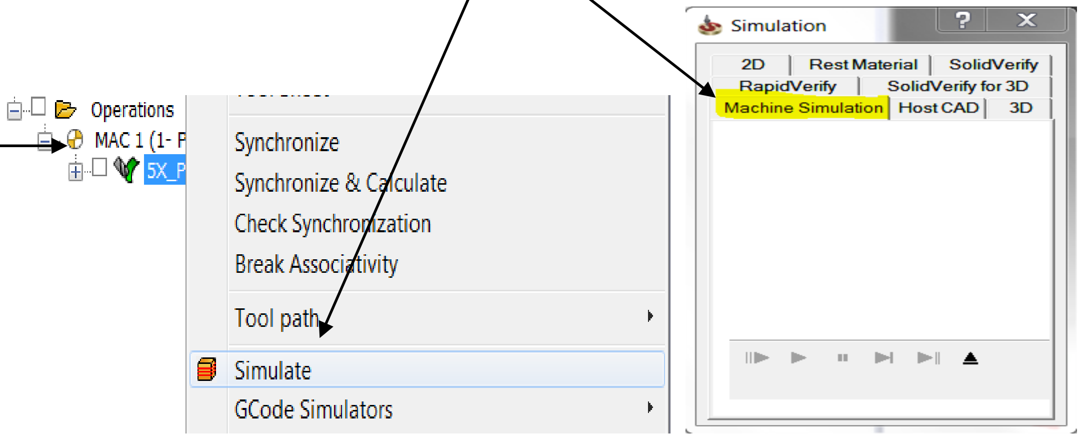
Right click the toolpath.

Click **Calculate**.

Right click the toolpath once the toolpath is calculated.

Click **Simulate**. The simulation window opens.

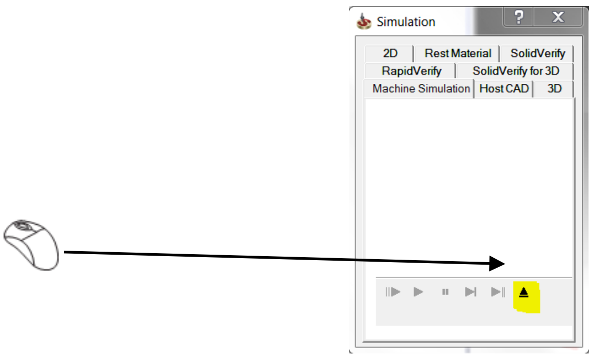
Click **Machine Simulation**.



Click  (run) icon once the machine simulation is loaded. We notice that the part is now rotating perfectly at the centre of the rotary axis.

We have fixed the second problem and will now focus on the first problem which still is persisting.

Click  (exit) icon in the simulation window.



To understand what causes the gouge in the very first move of the part, we need to understand the **Levels** option in the 5 Axis Toolpath Definition Box. To open the **Levels** option, do the following:

Right click the toolpath.

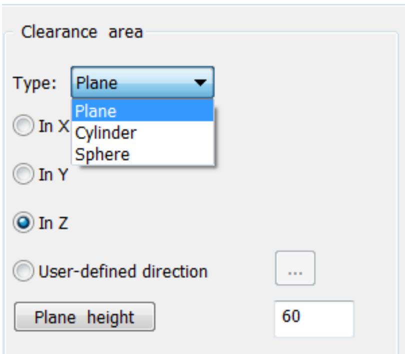
Click **Edit**. The parallel cuts window opens.

Select **Levels**.

Levels in simple term is Clearance Plane. There are three types of levels possible in

SolidCAM 5 Axis.

- Clearance to a Plane
- Clearance to a imaginary Cylinder
- Clearance to a imaginary Sphere



Clearance to a Plane :

Using this option the retracts of the tool always take to the plane specified. The plane could be in either X Direction, Y Direction or, Z Direction. The plane can also be user defined using two points.

Clearance to an Imaginary Cylinder :

Using this option the retracts of the tool are taken to the nearest point on an imaginary cylinder. This cylinder can have its axis running either through X axis, Y axis or, Z axis. The axis can also run through a user defined direction which can be defined using two points in space. The other parameter that the user is allowed to define is the radius of this imaginary cylinder.

ⓘ This option is best suited for this part as this is rotating around X axis and the retracts must take place on the nearest point on an imaginary cylinder.

Clearance to an Imaginary Sphere :

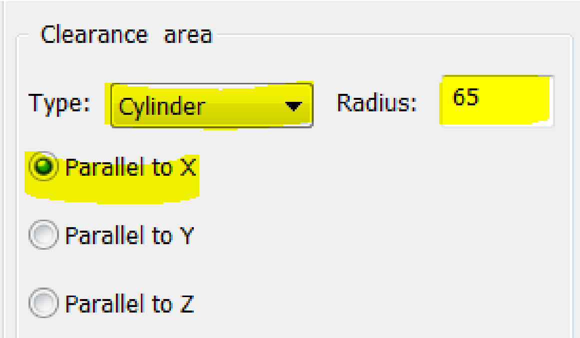
Using this option the retracts of the tool take place on an imaginary sphere. The radius and centre position of this sphere is user defined.

ⓘ This option is best suited for impeller kind of parts.

Select **Cylinder** from the drop-down list in the **Type** field.

Define a **Radius** of 65 mm (The part has a radius of 50mm, so we will have a constant clearance of 15mm throughout). The axis of the Cylinder will pass though X axis.

Select the **Parallel to X** radio button.



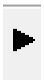
Click **Save & Calculate**. The toolpath is now calculated.

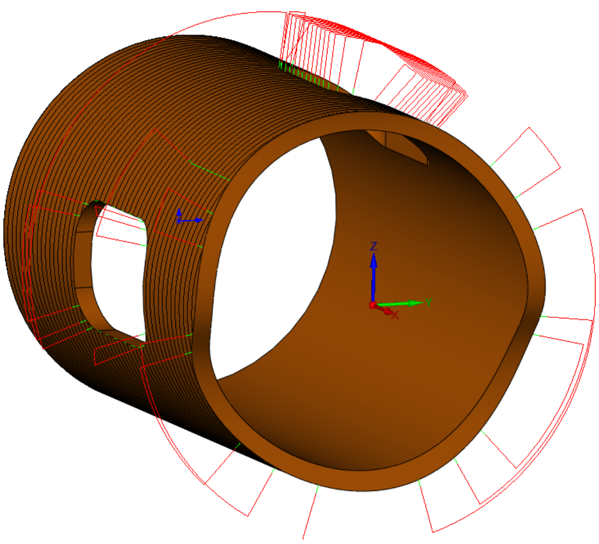
Click **Exit**.

Right click the toolpath.

Click **Simulate**. The simulation window opens.

Click **Host CAD**.

Click  (play) icon. You can see that the retracts look much better than before.



Click **Machine Simulation**.

Click (run) icon. We now see that the initial gouge is no longer there and also the retracts are much better and easy on the machine.

Click (exit) icon.

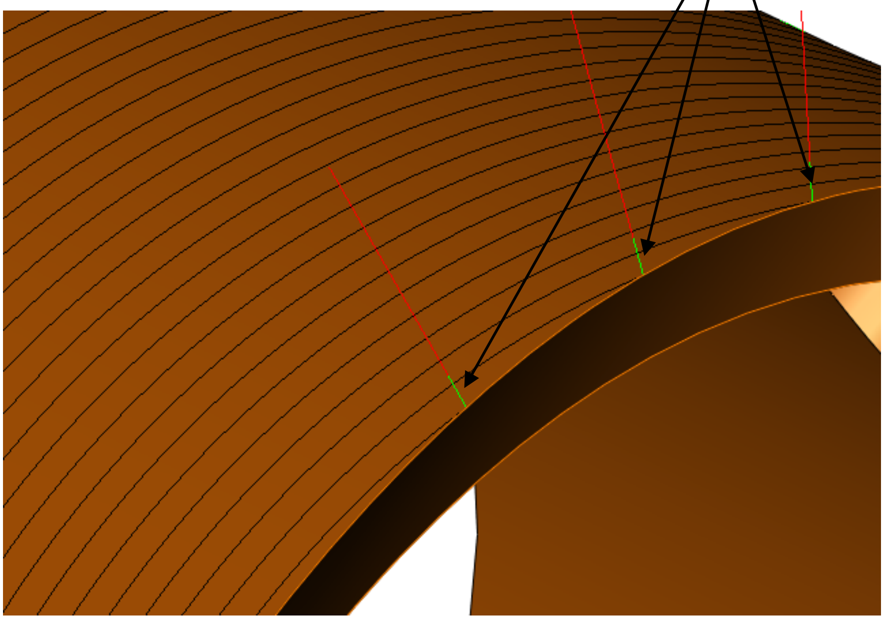
We have fixed the two main issues in the toolpath.

We will now look at two more issues which are not critical, but must be fixed.

1. Leads and links for entry, exit and retracts.
2. Tilt angle to avoid cutting exactly at the tip of the cutter.

Leads & Links

These are very important to allow the tool to enter and exit the material smoothly. As you can see in the below picture, the tool is directly entering into the material, this can cause the tool to chip off or wear early.



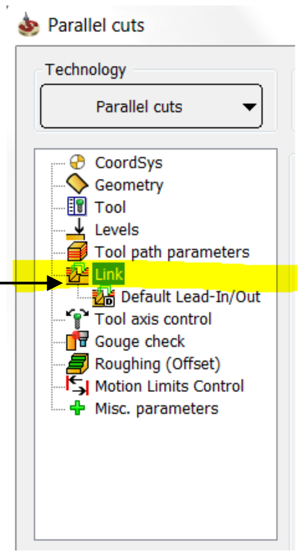
Let us see some of the common leads and links used in this part.

The first leads are applied to the main entry and exit of the tool into and out of the part. Do the following:

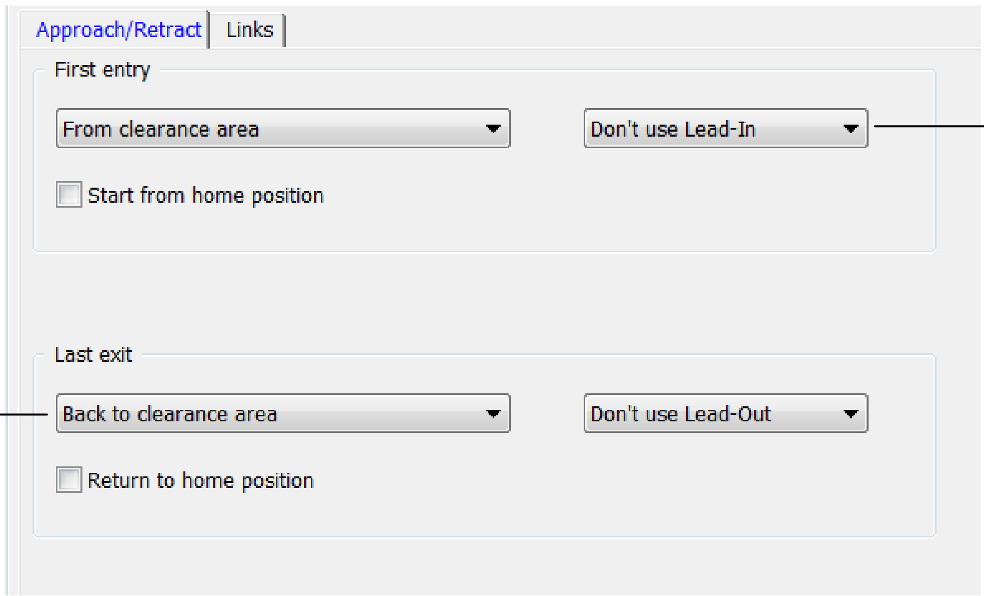
Right click the toolpath.

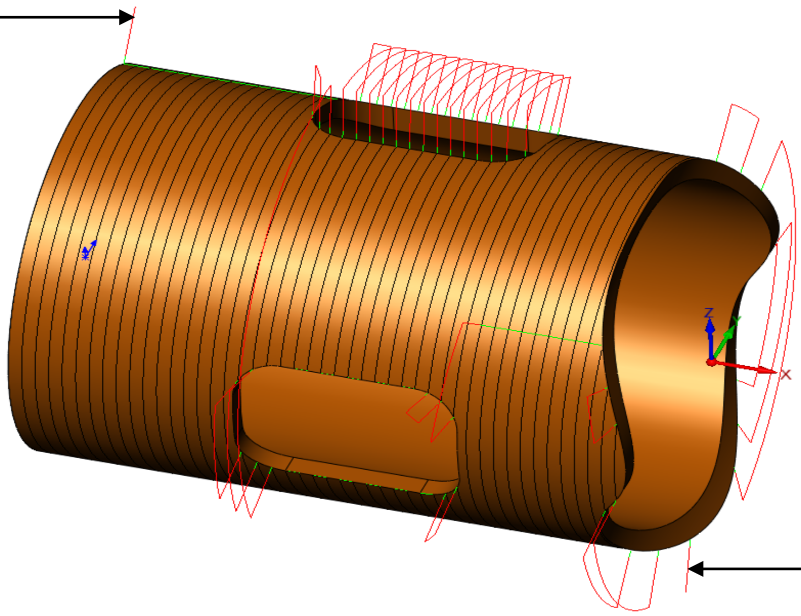
Click **Edit**. The **Parallel cuts** window opens.

Click **Link**. The leads and links are in the **Link** option.

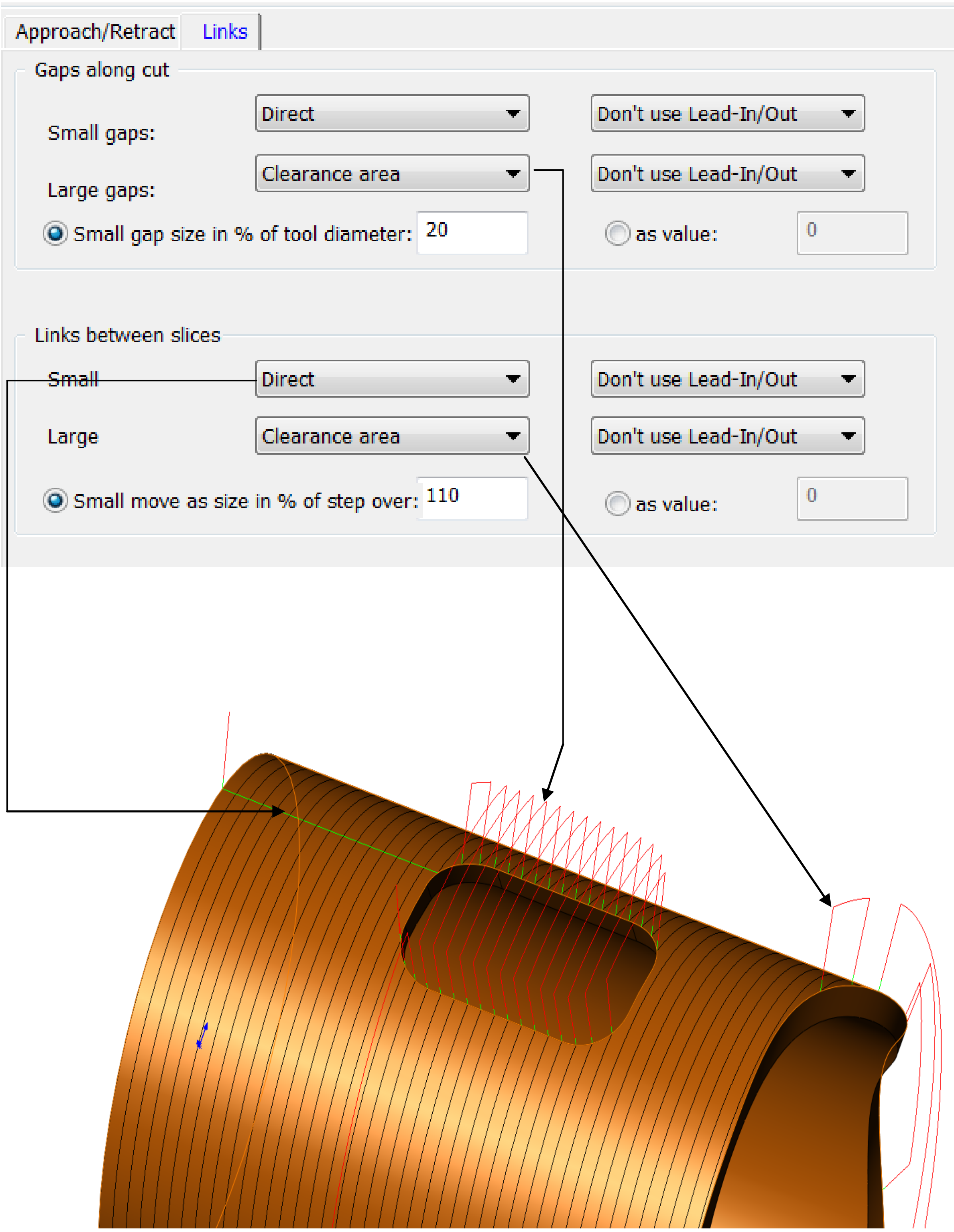


First Entry / Last Exit Leads & Links





Lead-In & Lead-Out



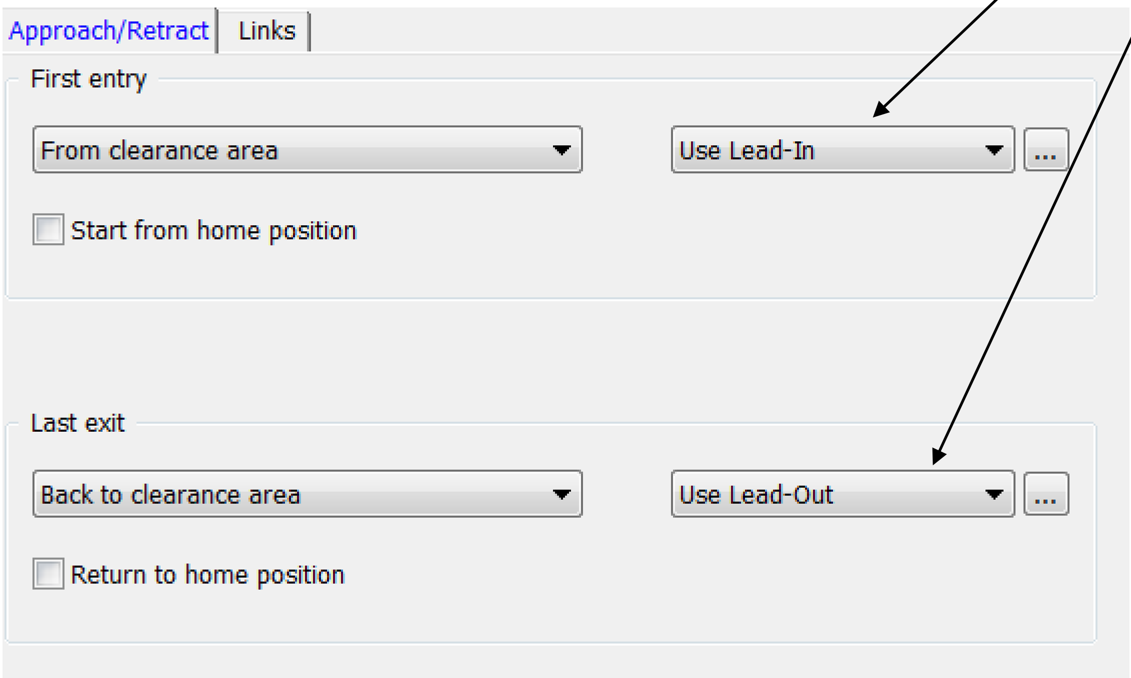
Modifying Lead-Ins & Lead-Outs

Let us set the First lead-in and lead-out parameters. Do the following:

Click **Approach/Retract** sub-tab.

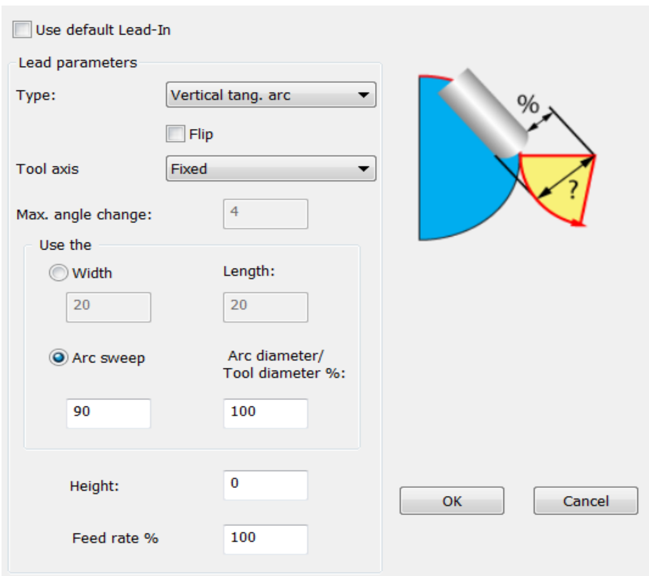
Select **Use Lead-In** from the drop-down list, in the **First entry** section.

Select **Use Lead-Out** from the drop-down list, in the **Last exit** section.



Click the  icon to set the lead-in and lead-out parameters.

Deselect the **Use default Lead-In** check box to set the parameters. Set the parameters as shown in the below image. The parameters are same for Lead-In and Lead-Out.



Set **Vertical tang arc** with a radius value of 100% of the tool diameter.

Set **Arc sweep** value of 90 Deg.

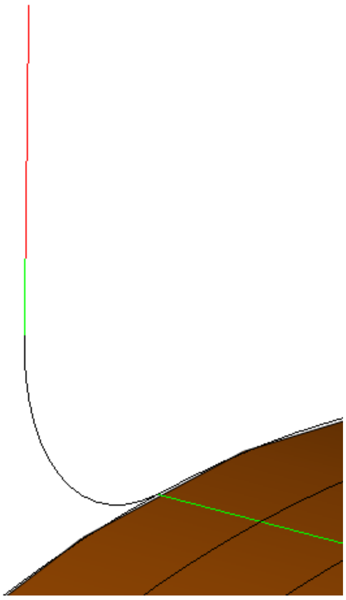
Click **OK**.

Click **Save & Calculate**.

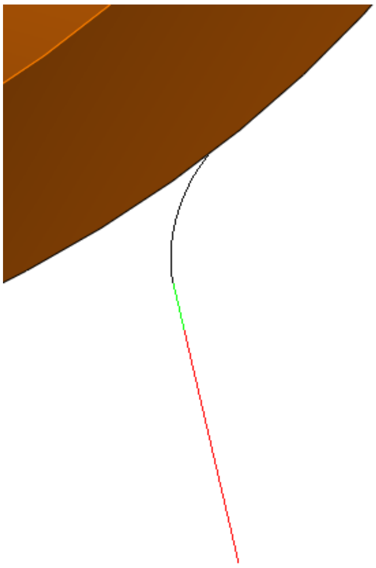
Click **Simulate**.

Click **Host CAD**.

Click ▶ (play) icon.

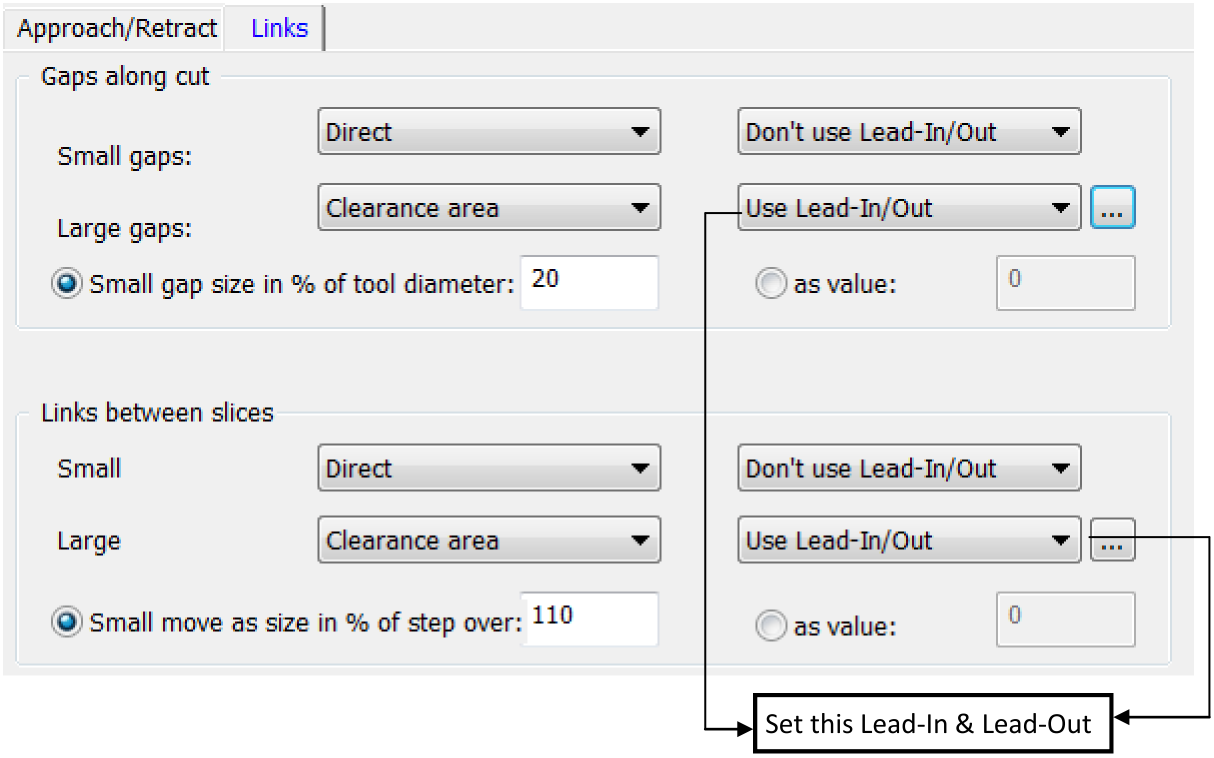


Lead-In

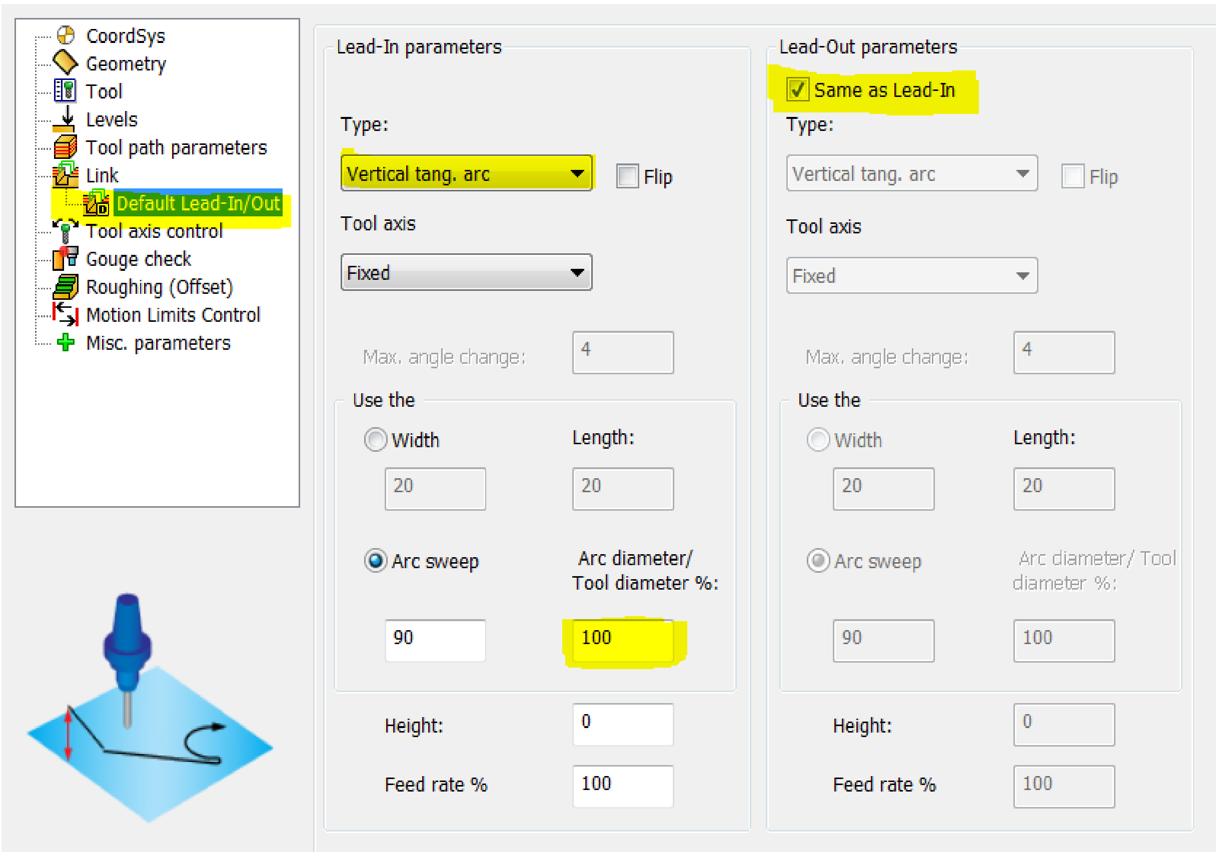


Lead-Out

Let us set some other lead in and lead out parameters



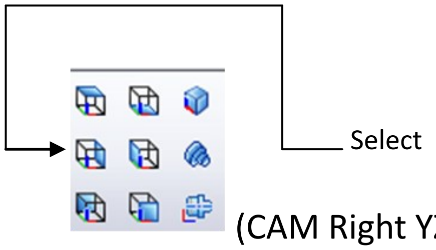
As we will not set any parameters for this Lead-In and Lead-Out, the default values and parameters are used. These values and parameters can be found in the **Default Lead-In/Out** parameter option. Let us set the parameters as shown in the image.



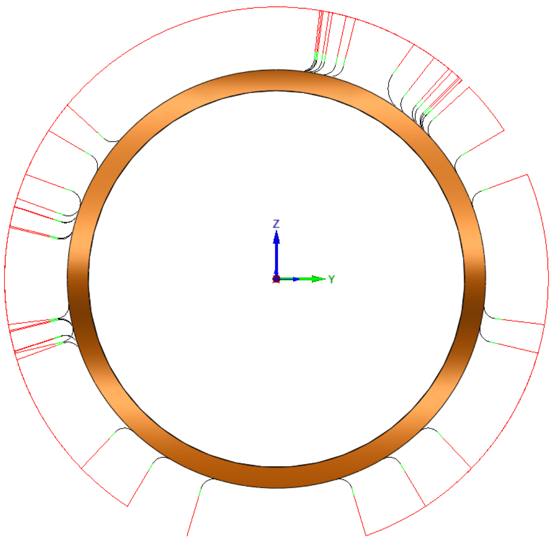
Click **Save & Calculate**.

Click **Simulate**.

Click  (play) icon.



Click the **SolidCAM Operations** tab. Select (CAM Right YZ View) and visualize the toolpath. The Lead-Ins and Lead-Outs look like this:



Let us now move to the last section of this tutorial.

We will now look at tool axis control which is the most important part in 5 Axis Machining. This is important as the **Tool Axis Control** defines the orientation of the tool axis during 5 Axis Machining. An improper tool axis control definition can cause jerks on the machine and can also gouge into the part.

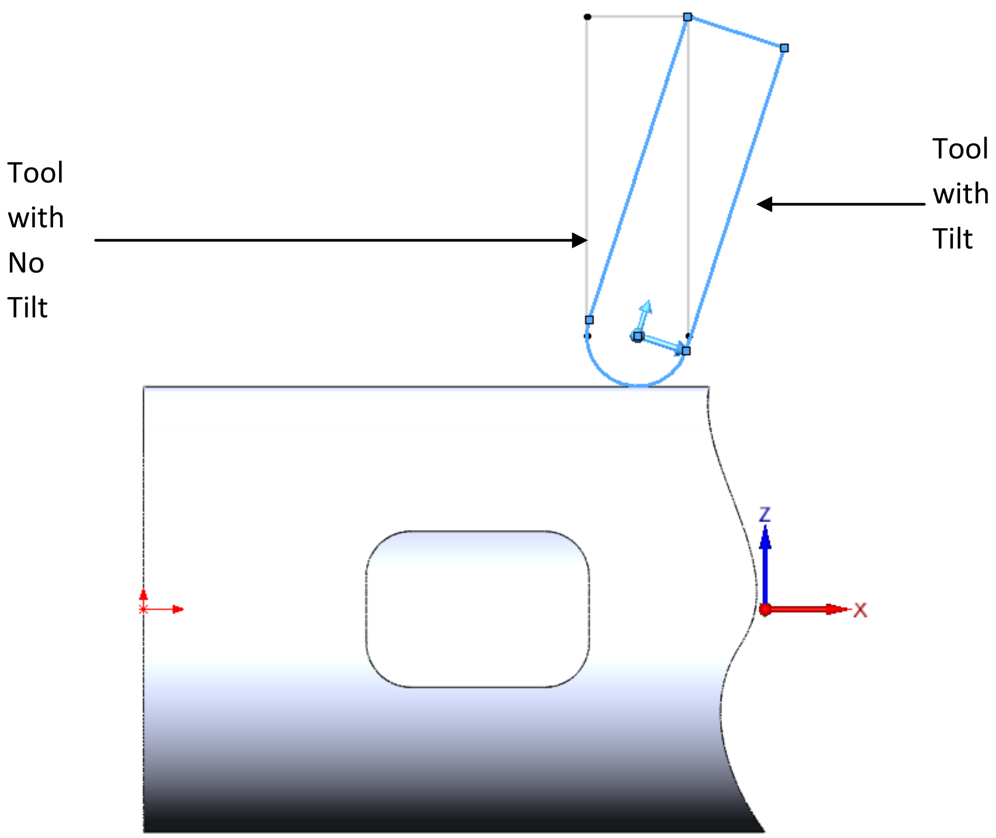
In our exercise we will use the tool axis control to effectively use the tool to cut the material and not Grind/Rub against the material.

Observe the following picture:

In the above scenario, the tip of the tool is doing all the cutting. This is not the best method to cut as the tip of the tool has "0" RPM. This creates a Grinding/ Rubbing effect on the material and wears out the tool. The surface finish appears jaded.

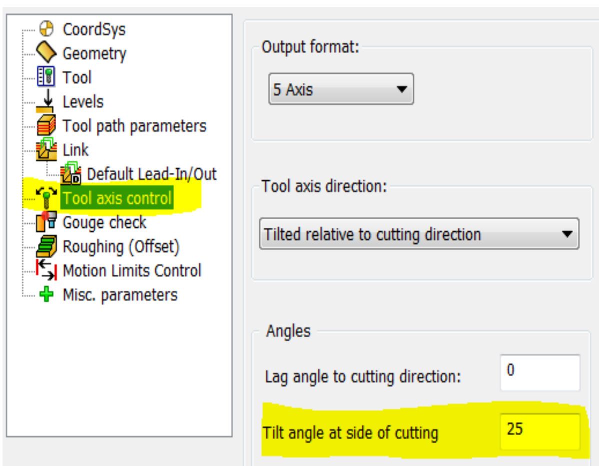
To make the tool cut, and not rub or grind, a small tilt angle must be applied so that we push the tip of the tool away from the cutting area and use the point on the spherical part of the tool which has the "RPM" to cut the material.

Take a look at this picture now:

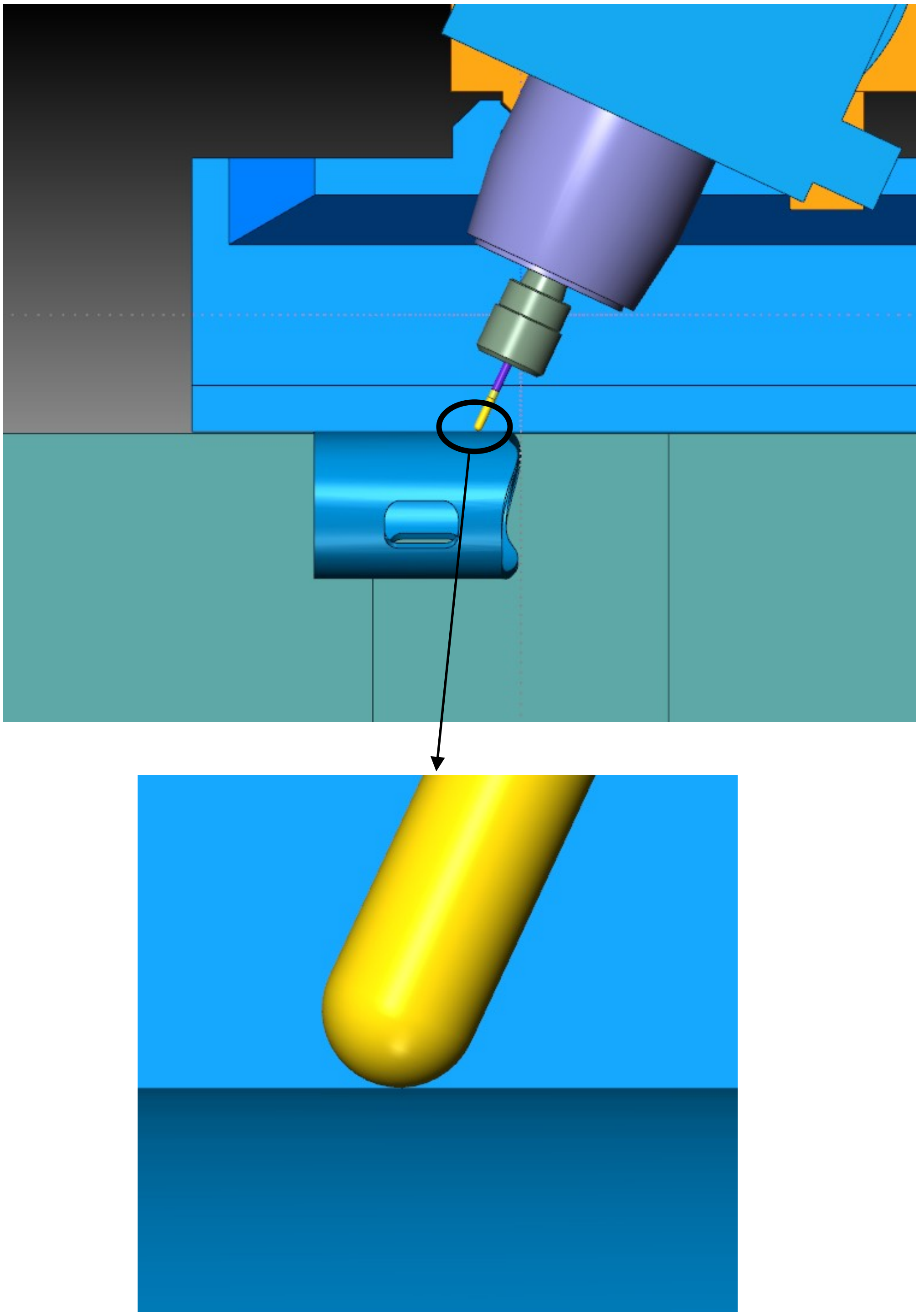


To apply the Tilt Angle, follow these steps:

1. Edit the toolpath
2. Go to Tool Axis Control Tab
3. Enter a value of 25 in Tilt Angle at side cutting box.
4. Click Save & Calculate.
5. Simulate the Toolpath in Machine Simulation to observe the result.

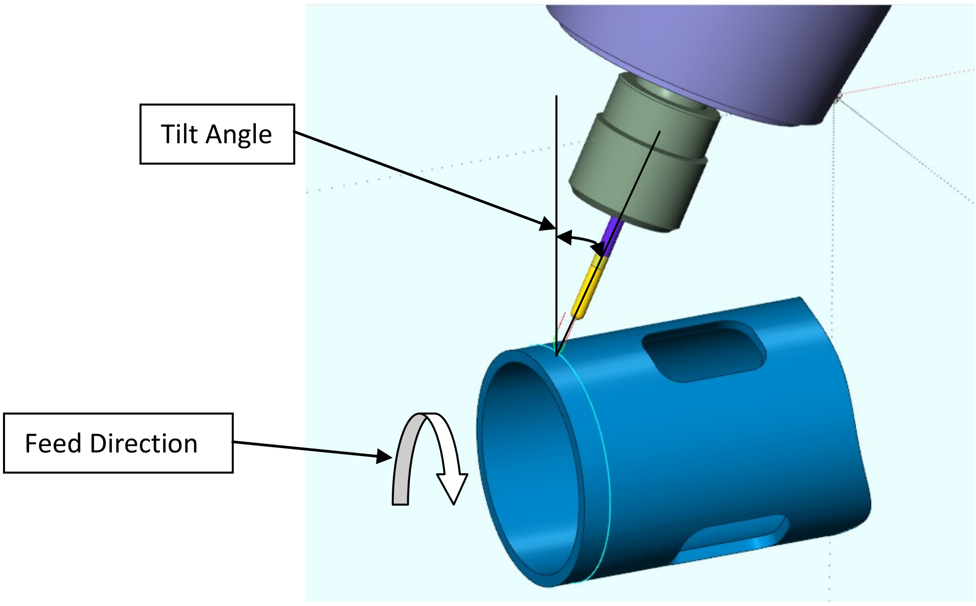


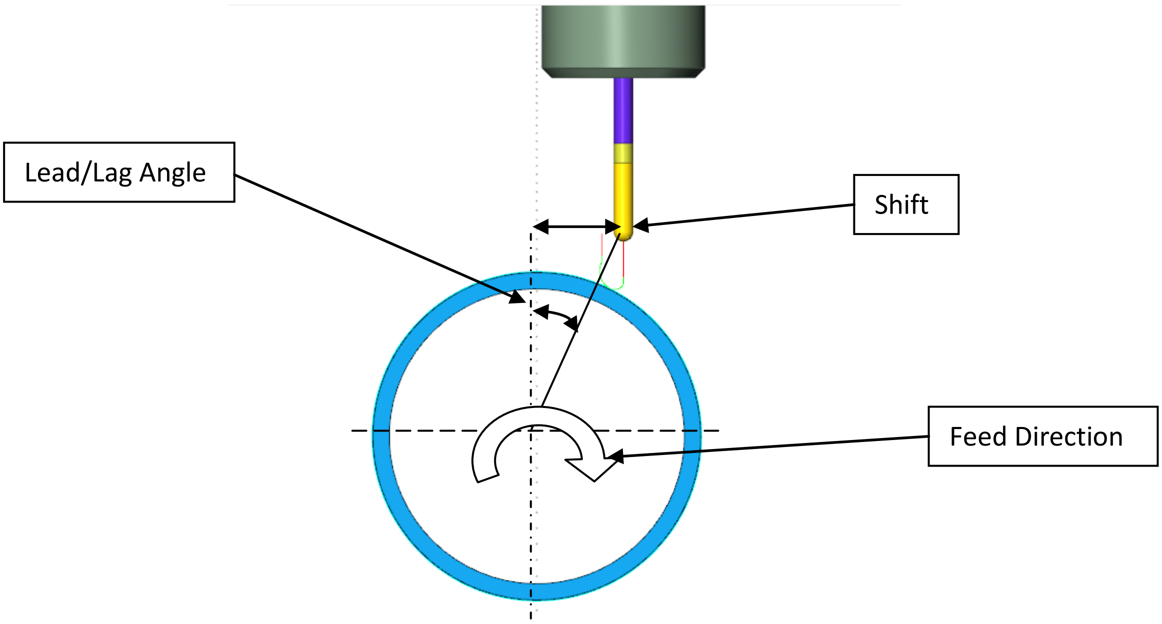
Observe the results in Machine Simulation. The following result is now visible:



You can see that the tip is no longer cutting the part but the side of the tool is cutting the material. Depending on the diameter of the cutter and the machine kinematics, this angle can either be increased or decreased to get the desired results. This angle is more popularly applied when Bull Nose Cutters are used.

ⓘ Tilt Angle is the angle of the tool axis across the toolpath in the feed direction, while Lead/Lag angle is the angle of the tool axis along the toolpath in the feed direction.





Unlike tilt angle which physically tilts the tool, lead/lag angle in this machine produces a shift from the centre thereby producing the same effect of a tilt angle (Pushing the tip of the tool away from the cutting area).

<u>End of Volume - 1</u>