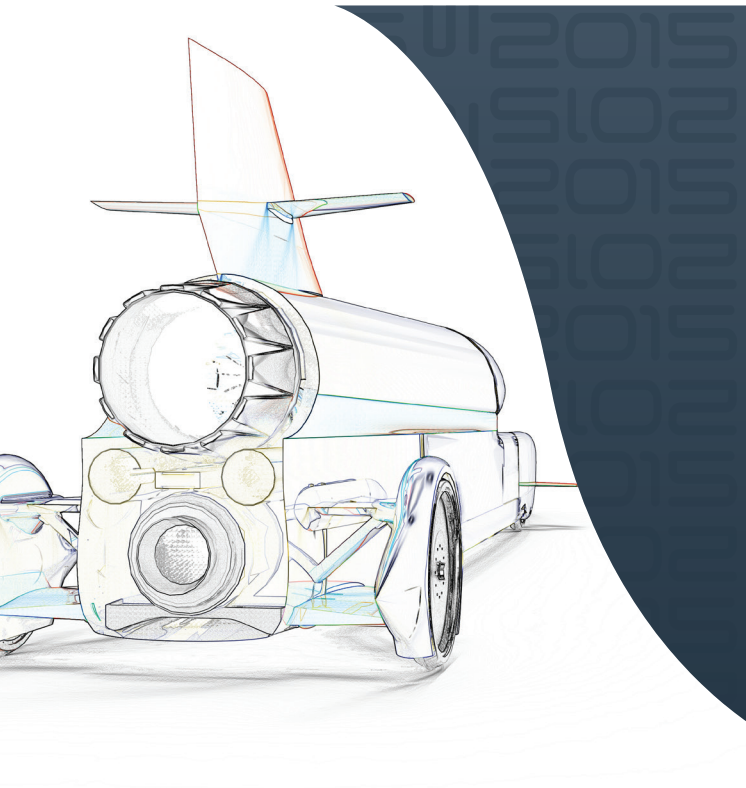


Delcam Advanced Manufacturing Solutions

PartMaker 2015

Advanced Surface Machining – Multi-Axis

Version 2015 or Higher

User Guide

PartMaker 2015

User Manual

User Guide/PartMaker

Advanced Surface Machining - Multi-Axis

Important User Notices

Delcam Ltd has no control over the use made of the software described in this manual and cannot accept responsibility for any loss or damage howsoever caused as a result of using the software. Users are advised that all the results from the software should be checked by a competent person, in accordance with good quality control procedures.

The functionality and user interface in this manual is subject to change without notice in future revisions of software.

Acknowledgements

Patents

PartMaker software is subject to the following patents:
Patent granted: US 6, 112, 133 Visual system and method for generating a CNC program for machining parts with planar and curvilinear surfaces
Patent granted: US 6, 741, 905 Visual system for programming of simultaneous and synchronous machining operations on lathes

PartMaker 2015. Published on 08 December 2014

Contents

Exercise 1: 5 Axis Surface Swarf and Wireframe Swarf Finishing Using Advanced Surface Machining Multi-Axis (ASM-MX-T) for SwissCAM and Turn-Mill

Introduction

This tutorial is designed to help you learn the steps to go through in using PartMaker SwissCAM or PartMaker Turn-Mill to program parts on a lathe utilizing PartMaker's Advanced Surface Machining Multi-Axis module. This tutorial assumes that you have a good working knowledge of either PartMaker SwissCAM or Turn-Mill. Though the PartMaker SwissCAM module is used for this tutorial, the steps can be identically replicated if working in PartMaker Turn-Mill.

This tutorial has been developed for use with PartMaker Versions 2015 and higher and uses the files in **C:\PartMaker\pm-swiss\ASM-MXT_Surface_Machining\5_Axis_Swarfing_Demo_ASM_Sample.**

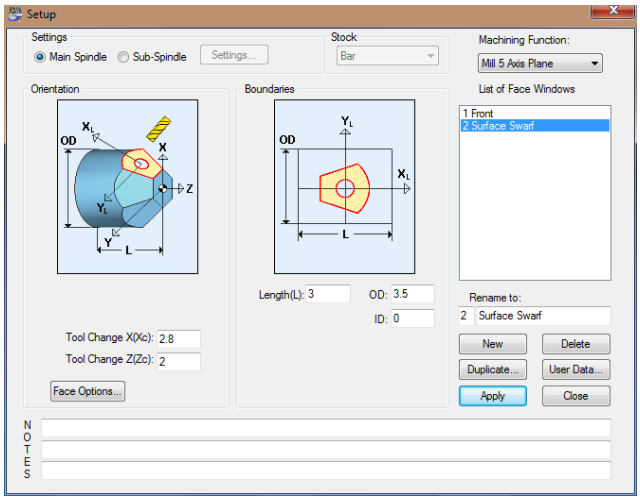
Creating a Surface Swarf Tool Path on the Solid model Part

In this section of the tutorial you will create and verify surfacing tool paths for Surface Swarf Finish machining on the imported solid model using ASM-MX-T.

Create a New Face Window for Surfacing on the part

The first step in creating a surfaced feature in PartMaker Turn-Mill or SwissCAM is to create a new Face Window. To create the Face Window:

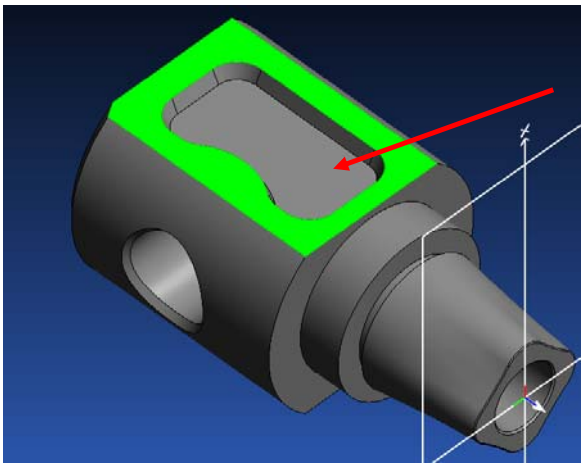
1. Choose **Setup** from the **View** menu
2. In the **Setup** dialog, click the <**New**> button to create a new Face Window.
3. From the **Machining Function** drop down menu, choose **Mill 5 Axis Plane**.
4. In the **Rename To:** field, type "**Surface Swarf**" and click **<Apply>**. Your completed **Setup** dialog should appear as shown below:



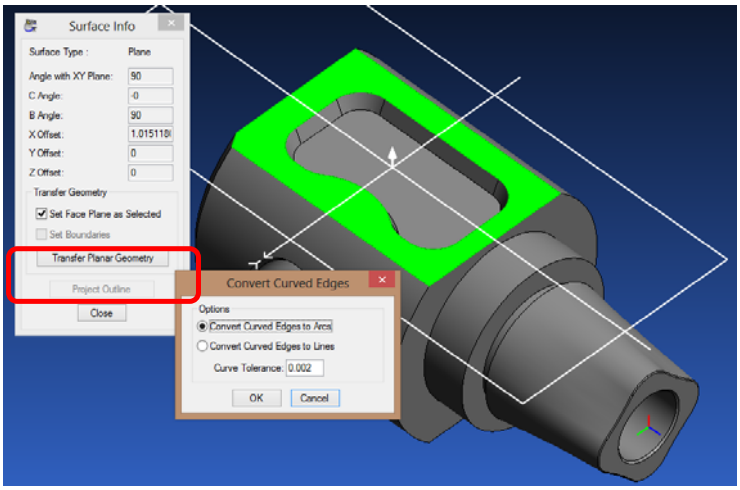
5. Click the **<Close>** button to close the **Setup** dialog.

Create 5 Axis Face Plane for Surface Swarf Machining

1 Double click on the surface as shown below:

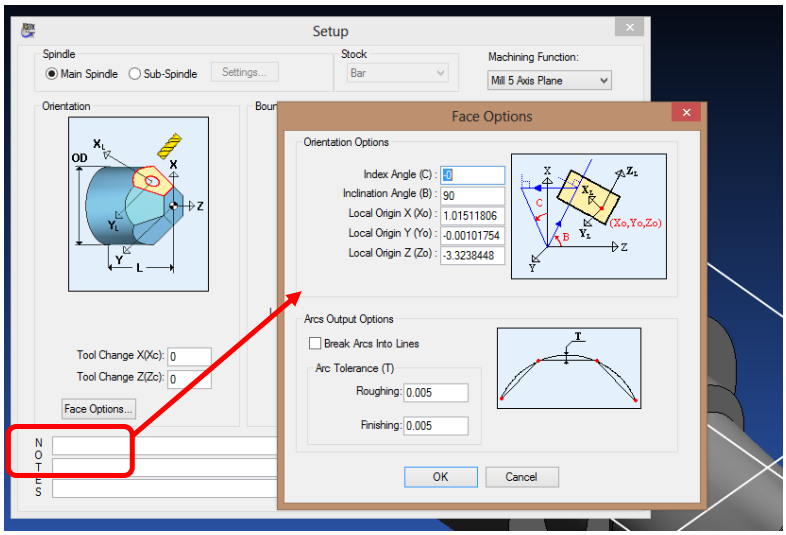


2 Select Transfer Planar Geometry:



3 Click the <OK> button to transfer the geometry and set the face plane for this Surface group.

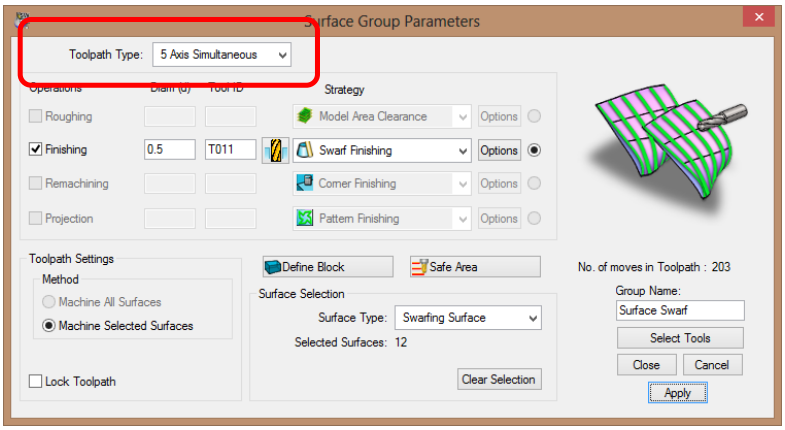
4 In the Setup Dialog the <**Face Options>** have been updated to the Orientation for the new face.



Create a New Surface Group for Surface Swarf Machining

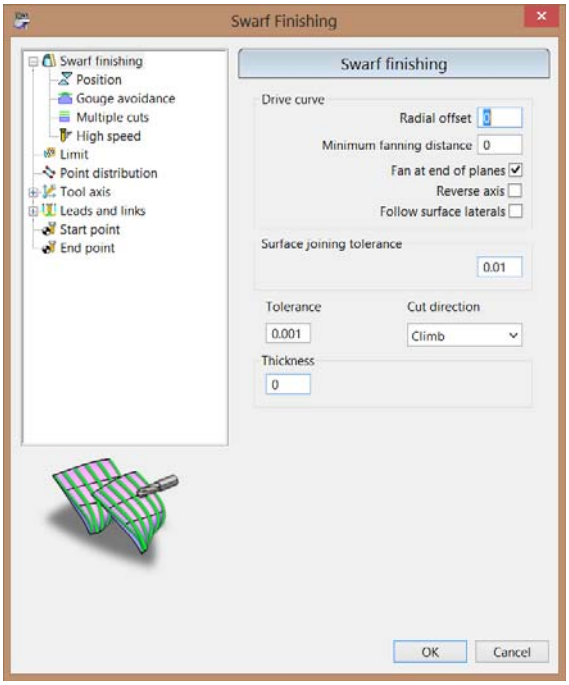
In this section of the tutorial you will create and verify a tool path for Surface Swarf machining on the imported solid model.

1. Choose New Surface Group from the Part Features menu. The Surface Group Parameters dialog will display.
2. Select the **5 Axis Simultaneous** option from the **Toolpath Type** drop down menu.
3. Select the **Swarf Finishing** strategy.
4. Enter a tool diameter of 0.5 in (12.7 mm) in the **Diam (d)** field and click the **<Select Tools>** button.
5. In the **Group Name** field enter "**Surface Swarf**" and click **<Apply>**. The dialog should now appear as shown below:



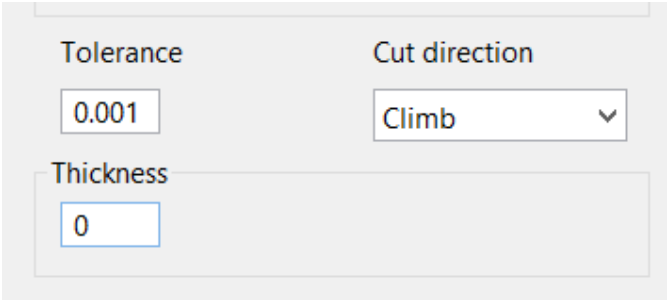
Setting Strategy Options for Surface Swarf Machining

Click the <**Options>** button in the **Surface Group Parameters** dialog which will open the dialog for the Swarf Finishing strategy as shown below in the picture.



General Swarf Finishing Settings

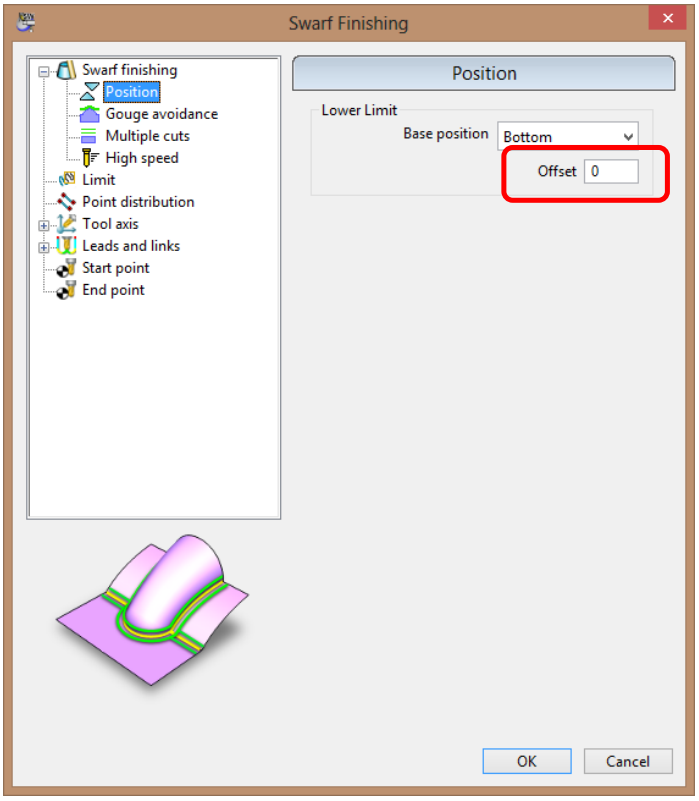
1 In the Swarf finishing dialog, on the main page enter the Tolerance value of 0.001 (0.0254 for metric) as shown below:



2 Select Position.

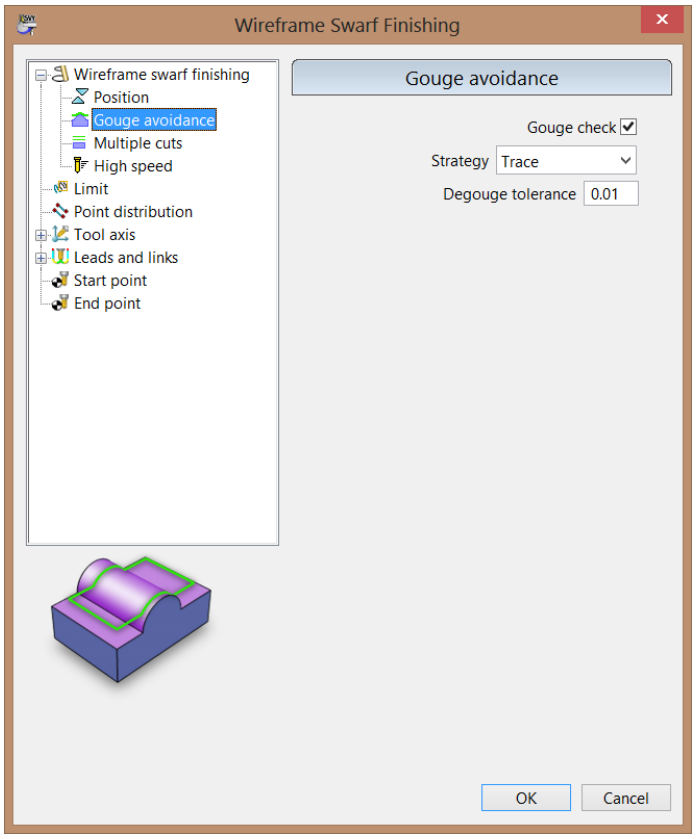
3 Set the Base Position to Bottom.

*If you set **Offset** to a negative value the end mill will track that distance below the bottom curve.*



4 Select **Gouge avoidance**.

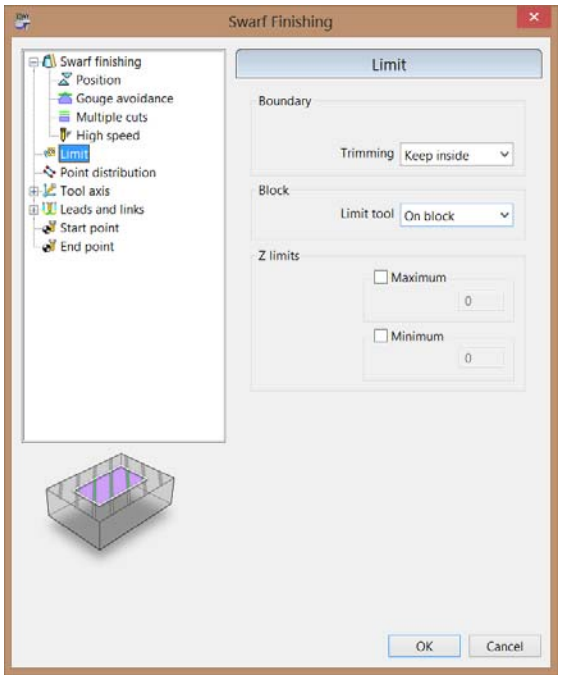
5 Check the box for **Gouge Check.**



6 Select **Limit**.

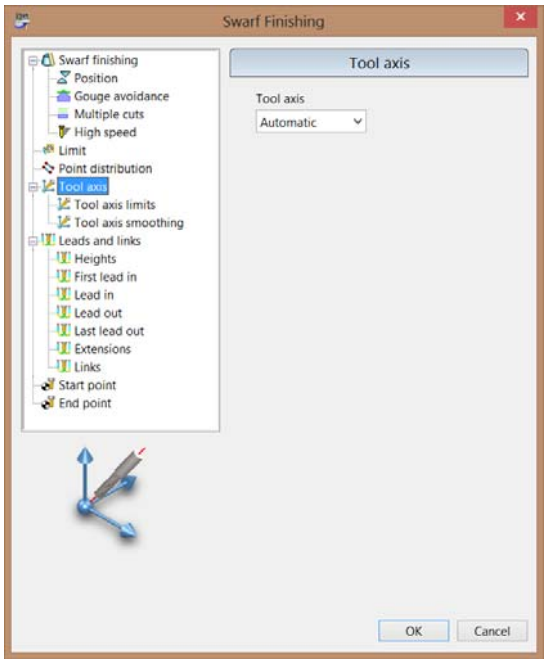
7 Set **Boundary** to **Keep inside**.

8 Set **Block: Limit Tool** to **On Block**.

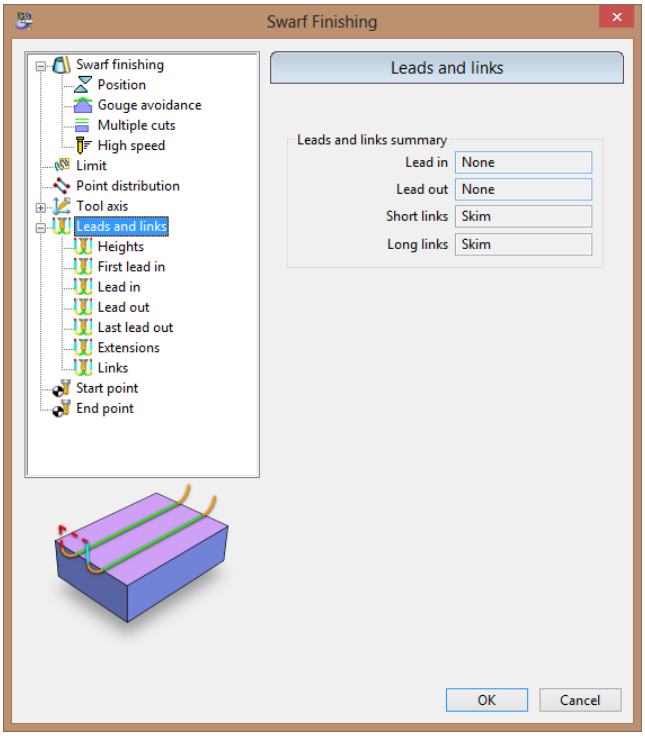


9 Select **Tool Axis**.

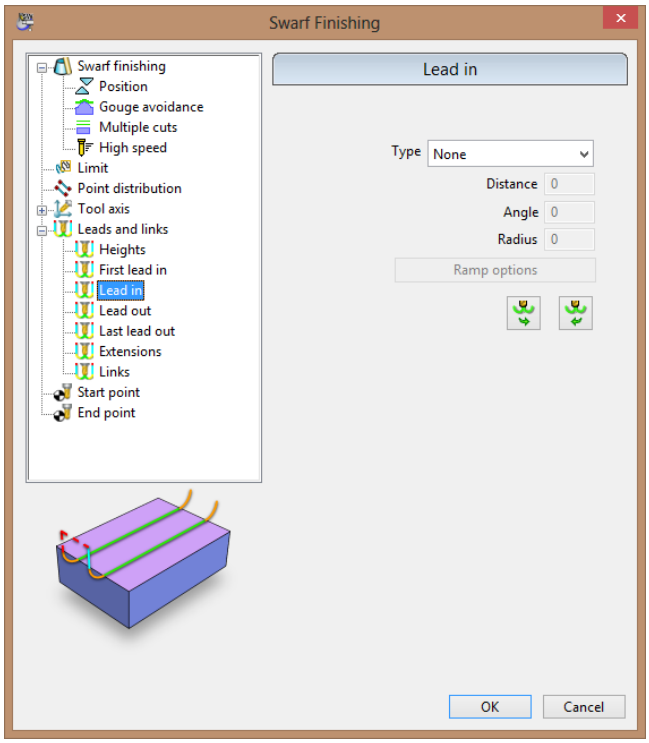
10 Tool Axis will be set to **Automatic**.



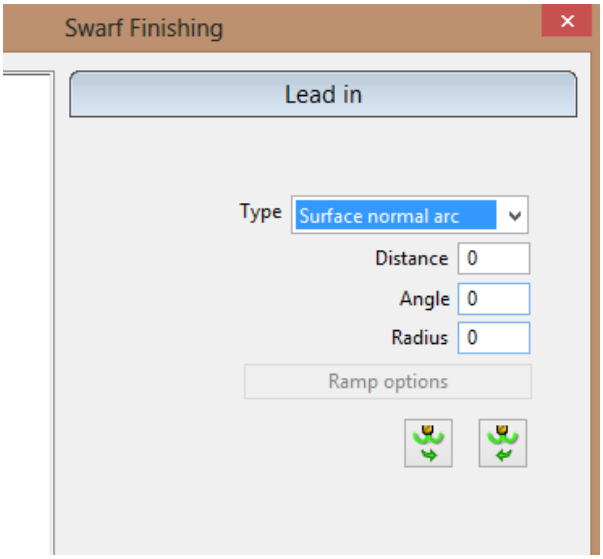
11 Click the **Leads and Links** tab in the tree on the left in the dialog to expand the **Lead and Links** features.



12 Click on **Lead in** on the tree to see the lead-in options.



13 Set **Type** to **Surface Normal Arc.**

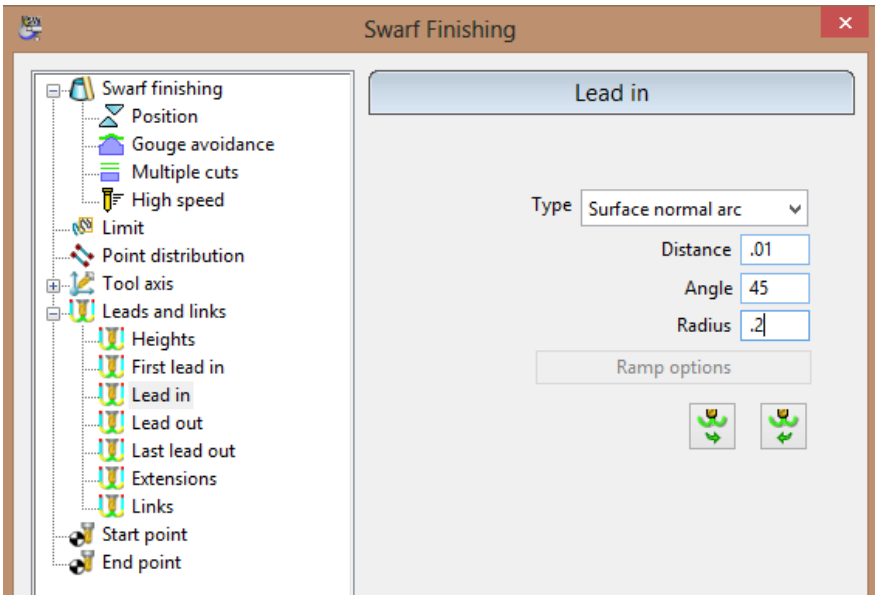


14 The **Distance** field would now be enabled. Enter a value of .01 (.254 mm) in the distance field.

15 Set the **Angle** to 45.

16 Set the **Radius** to .2 (5.08).

The dialog box appears as below:



17 Click on  to copy the same parameters from **Lead In** to **Lead Out.**

18 Click on **Links** on the tree to see the Links options.

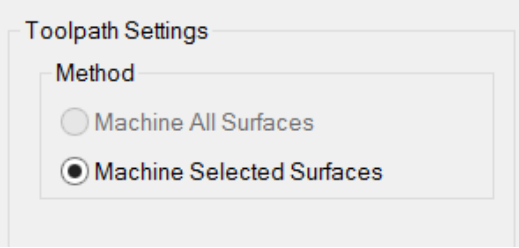
19 Set the **Short** and **Long** link to **Skim**. This will move the tool clear of the part by the specified incremental distance, descend at rapid feed to the specified incremental distance above the contact point, and then plunge the remaining distance.

20 Set the **Retract and Approach** moves to **Tool Axis.** This will make the retract and approach moves have the same orientation as the tool axis.

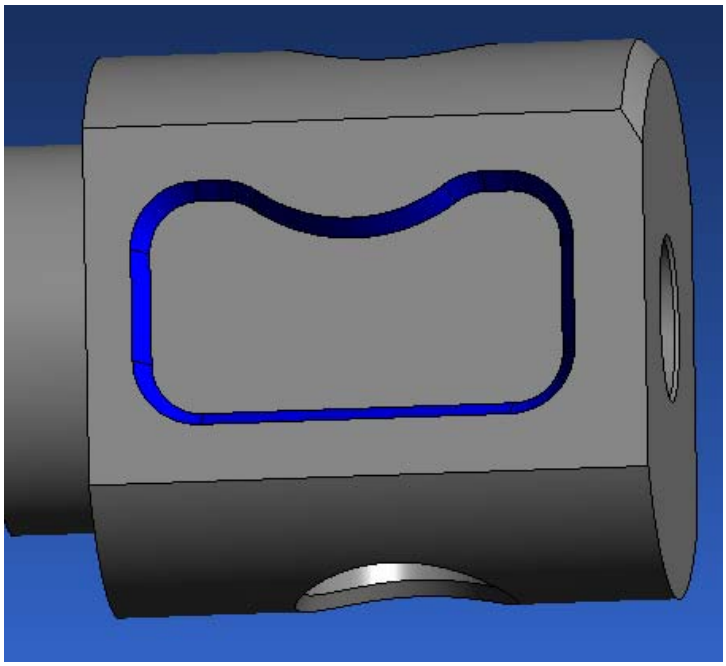
21 Select **<OK>**. You should still have the **Surface Group Parameters** dialog open.

Select the Surfaces for Machining

At this point you should still have the **Surface Group Parameters** dialog open. In the lower left hand corner of this dialog there is a framed section called **Toolpath Settings**. In this section you will control how you select surfaces to be machined by the current operation. There are two choices: **Machine All Surfaces** and **Machine Selected Surfaces**. By default **Machine All Surfaces** will be selected which will select all surfaces of the imported solid model as Machining Surfaces. In this exercise you will select the surfaces to be machined.



1 In the **Solid Model File** window, click on the surfaces on the solid model as indicated in the picture below: This will create the Top and Bottom curve automatically for the surface group.



2 Click the **<Close>** button to return to the CAM Face Window.

Verify the Tool Path

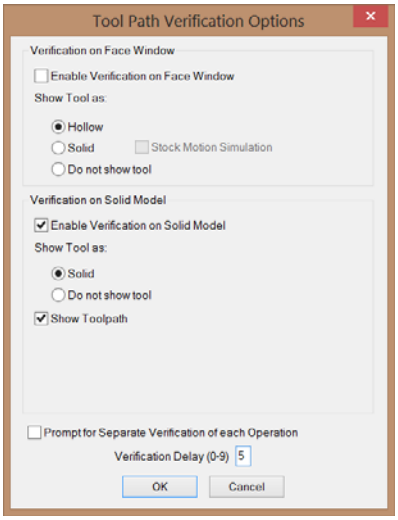
Once you have created the **surface group** you can visually verify the tool path PartMaker has calculated. To do so:

1 From the Part Features menu choose Verify Work Group Tool Path.

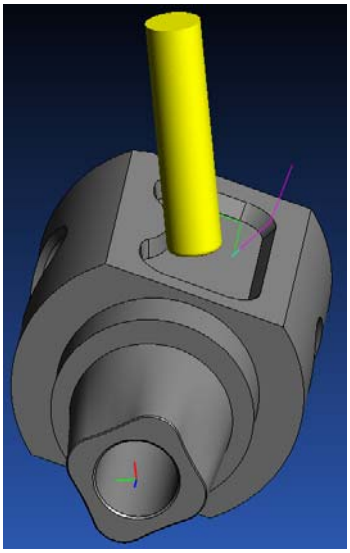
2 In the **Tool Path Verification Options** click the **Hollow** radio button as shown below.

3 Enter a Verification Delay of 5.

Note: *When entering a* ***Verification Delay*** *of greater than 0, you will see the 3D representation of the cutting tool moving along the Solid Model. When* ***Verification Delay*** *is set to 0, you will only see the path of the tool on the Solid Model.*



4 Click **<OK>** to show the calculated tool path. Your screen should appear as shown below:



Note: *Notice above, the line that extends into the programmed tool path. This line represents the* ***Lead In & Lead Out*** *set in the* ***Leads and Links*** *property page. Creating this line assures that the tool will safely approach the stock without risking a tool collision.*

5 From the Part Features menu choose Hide Work Group Tool Path.

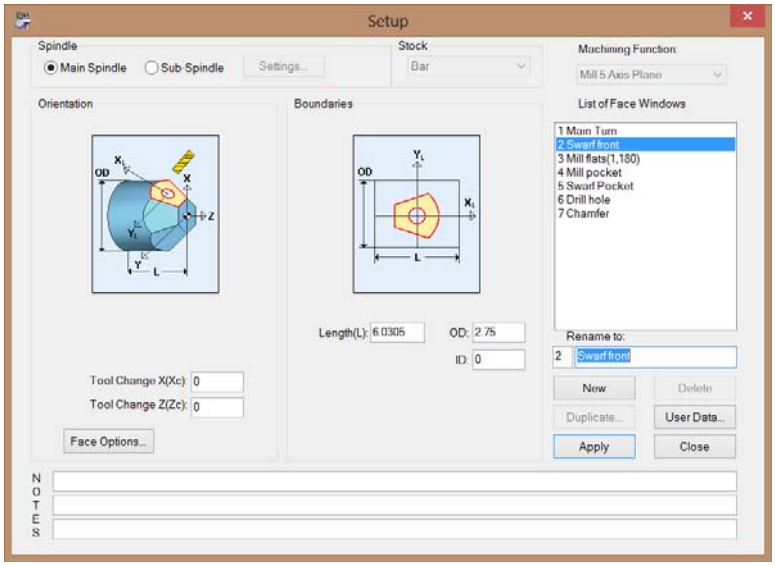
Creating a Wireframe Swarf Tool Path on the Face of the Part

In this section of the tutorial you will create and verify surfacing tool paths for Wireframe Swarf Finish machining on the front of the imported solid model using ASM-MX-T.

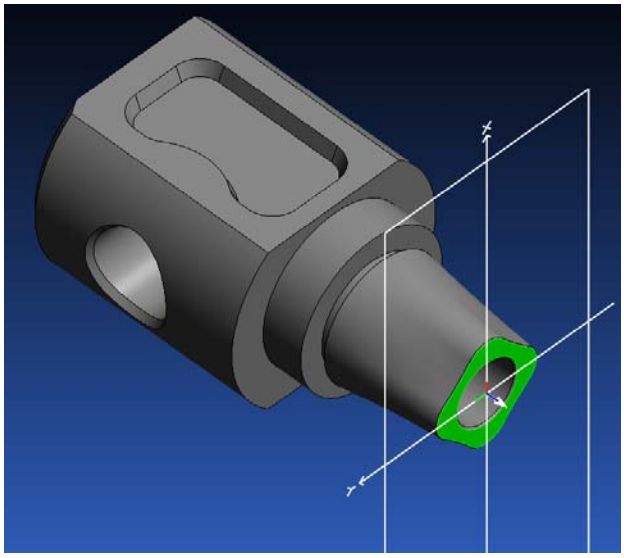
Create a New Face Window for Surfacing on the Face

The first step in creating a surfaced feature in Turn-Mill or SwissCAM is to create a new Face Window. To create the Face Window:

1. Choose **Setup** from the **View** menu
2. In the **Setup** dialog, click the <**New**> button to create a new Face Window.
3. From the **Machining Function** drop down menu, choose **Mill 5Axis Plane**.
4. In the **Rename To:** field, type "**Swarf Front**" and click **<Apply>**. Your completed **Setup** dialog should appear as shown below:



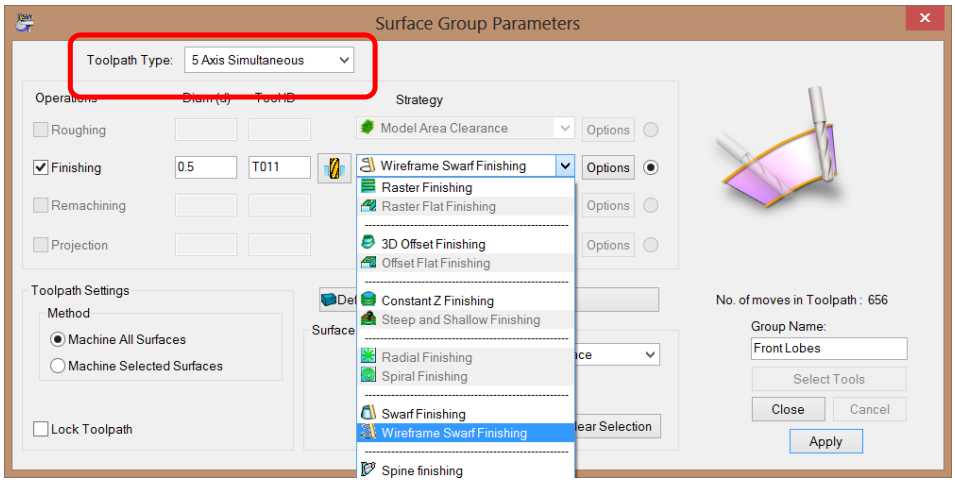
5. Click the **<Close>** button to close the **Setup** dialog.
6. Since we are working on the face of the part the Face Plane is already correctly set.



Create a New Surface Group for Wireframe Swarf Machining

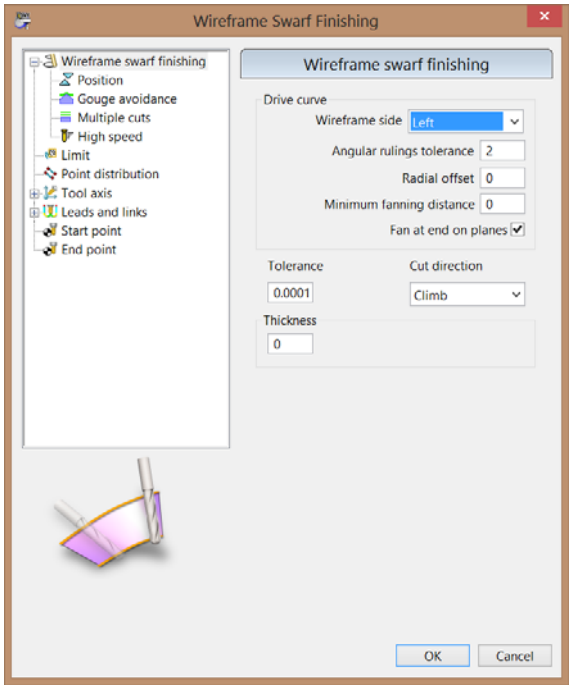
In this section of the tutorial you will create and verify a tool path for Wire frame Swarf machining on the face of the imported solid model.

1. Choose New Surface Group from the Part Features menu. The Surface Group Parameters dialog will display.
2. Select the **5 Axis Simultaneous** option from the **Toolpath Type** drop down menu.
3. Select the Wireframe Swarf Finishing strategy.
4. Enter a tool diameter of 0.5 in (12.7 mm) in the **Diam (d)** field and click the **<Select Tools>** button.
5. In the **Group Name** field enter "**Front Lobes**" and click **<Apply>**. The dialog should now appear as shown below:



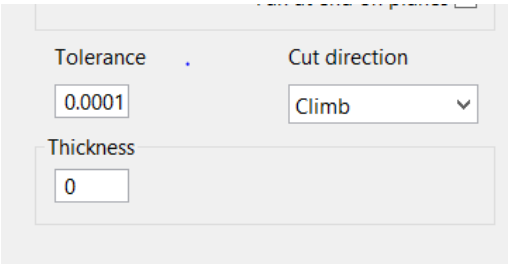
Setting Strategy Options for Wireframe Swarf Machining

Click the <**Options**> button in the **Surface Group Parameters** dialog which will open the dialog for the Wireframe Swarf Finishing strategy as shown below in the picture.



General Wireframe Swarf Finishing Settings

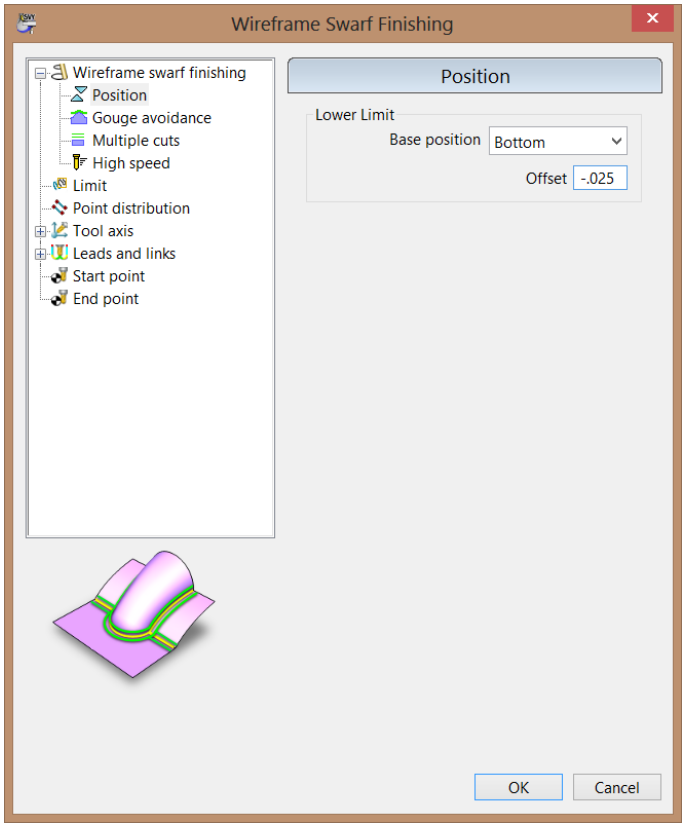
1 In the Wireframe Swarf finishing dialog, on the main page enter the Tolerance value of 0.0001 (0.0025 for metric) as shown below



2 Select Position.

3 Set the **Base Position** to Bottom.

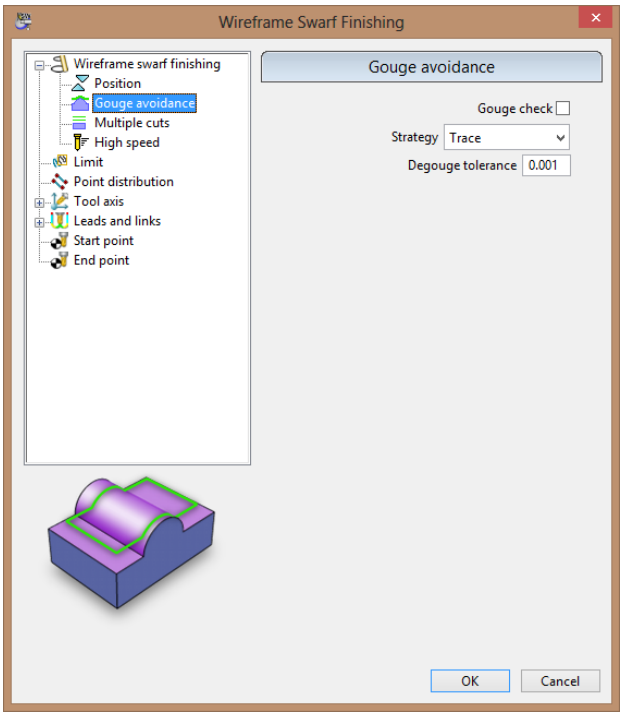
*If you set **Offset** to a negative value the end mill will track that distance below the bottom curve.*



4 Select Gouge avoidance.

5 **Uncheck** the **Gouge Check** box.

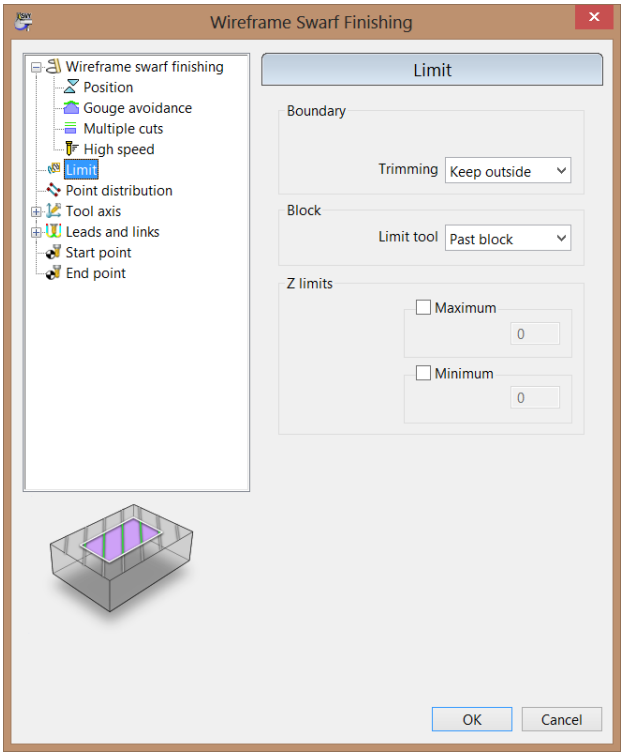
6 Set the **Degouge tolerance** to 0.001



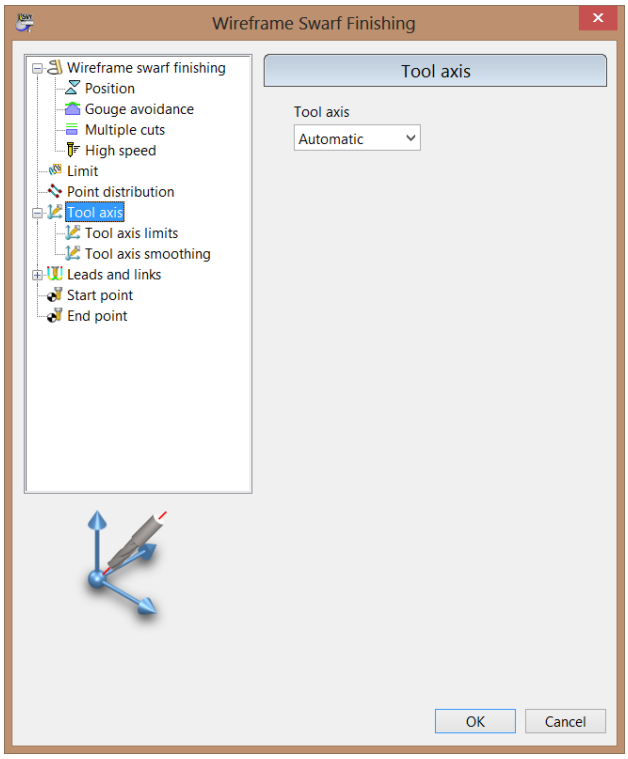
7 Select **Limit**.

8 Set **Boundary** to **Keep outside**.

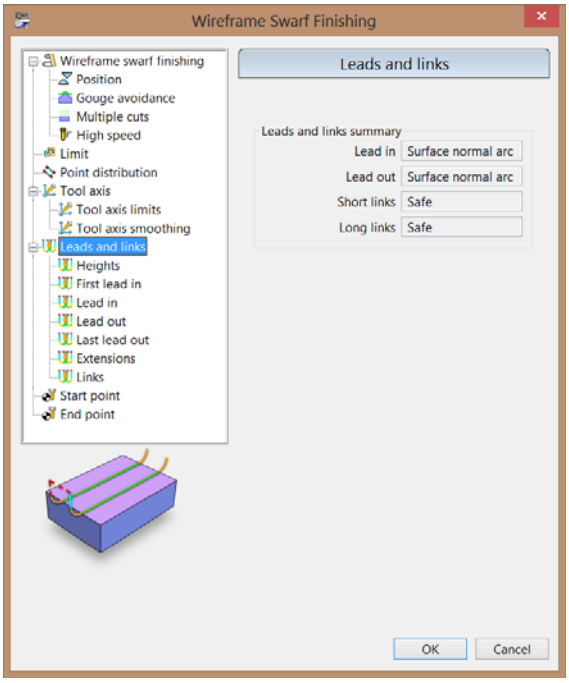
9 Make sure the **Limit Tool** option is set to **Past block** to allow the tool to move outside the block in order to follow the pattern.



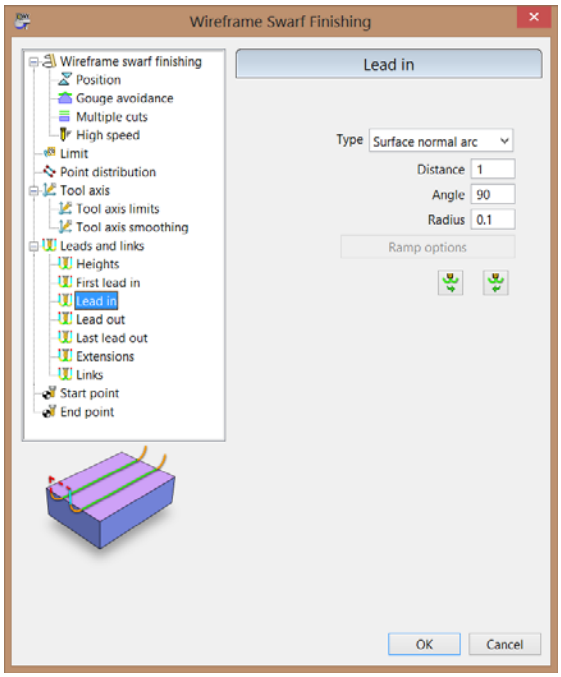
10 Select **Tool Axis** on the tree. Tool Axis will be set to **Automatic**.



11 Click the **Leads and Links** tab in the tree on the left in the dialog to expand the **Lead and Links** features.



12 Click on **Lead in** on the tree to see the lead-in options.



13 Set **Type** to **Surface Normal Arc**.

14 The **Distance** field would now be enabled. Enter a value of 1 (25 mm) in the distance field.

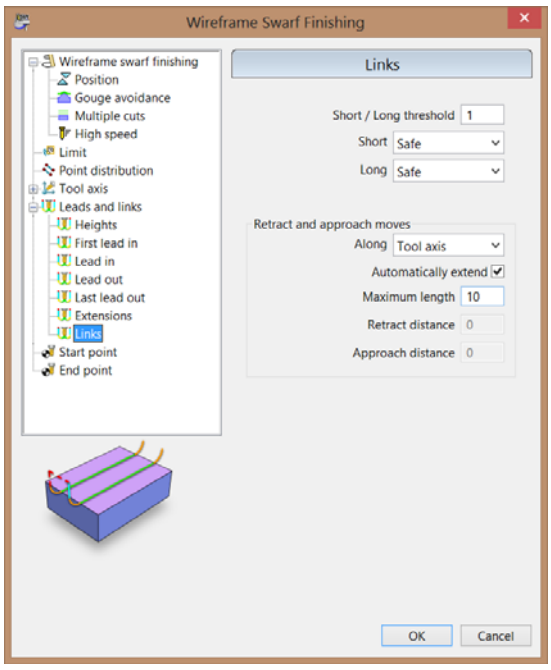
15 Set the **Angle** to 90 and the **Radius** to .1 (2.54).

16 Click the **Lead out same as lead in** button to copy the same parameters from **Lead In** to **Lead Out.**

17 Click on **Links** on the tree to see the Links options.

18 Set the **Short** and **Long** link to **Safe**. This will move the tool to the safe area before making the link move.

19 Set the **Retract and Approach** moves to **Tool Axis.** This will make the retract and approach moves have the same orientation as the tool axis.



20 Select <**OK**>. You should still have the **Surface Group Parameters** dialog open.

At this point you should still have the **Surface Group Parameters** dialog open. In the lower left hand corner of this dialog there is a framed section called **Toolpath Settings**. In this section you will control how you select surfaces to be machined by the current operation. There are two choices: **Machine All Surfaces** and **Machine Selected Surfaces**. By default **Machine All Surfaces** will be selected which will select all surfaces of the imported solid model as Machining Surfaces.

21 Click the **<Close>** button to return to the CAM Face Window

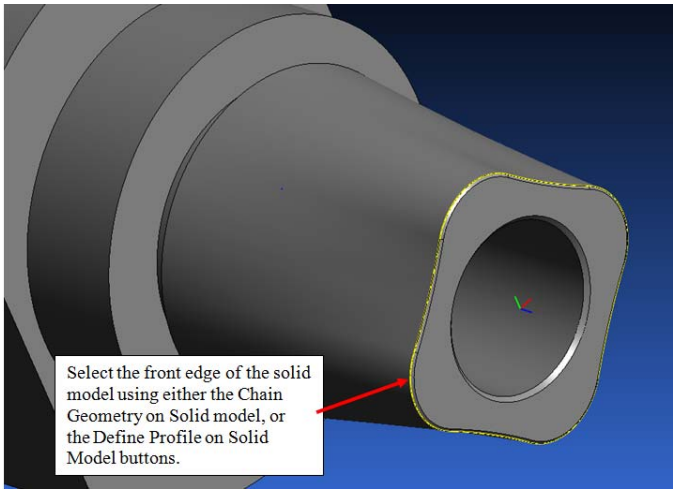
Select the Wire frames for Machining

1 In the **Solid Model File** window, click on the curve at the front of the solid model as indicated in the picture below:

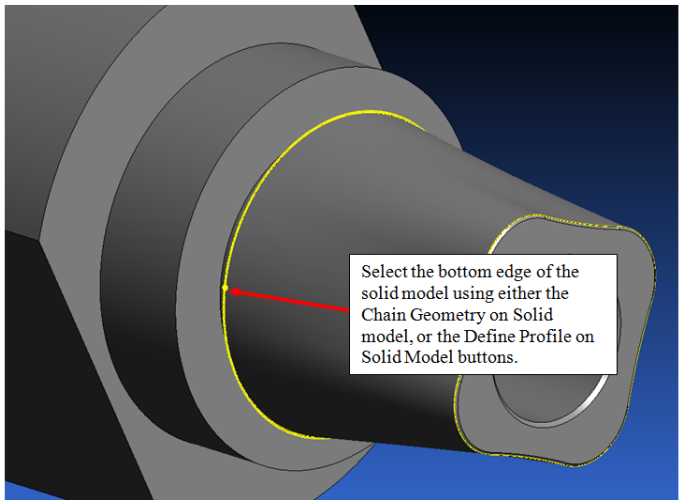
2 Choose the **Define Profile on Solid Model** icon

3 Click your cursor on the front edge of the solid model and follow the curve around the face of the part. This will set the top Swarfing Curve.

4 Your Face Window should now appear as shown below:

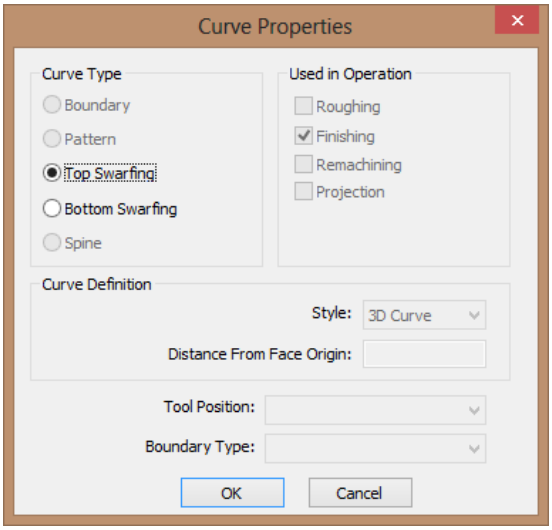


5 Choose the **Chain Geometry on Solid Model** icon and select the bottom curve on the solid model.



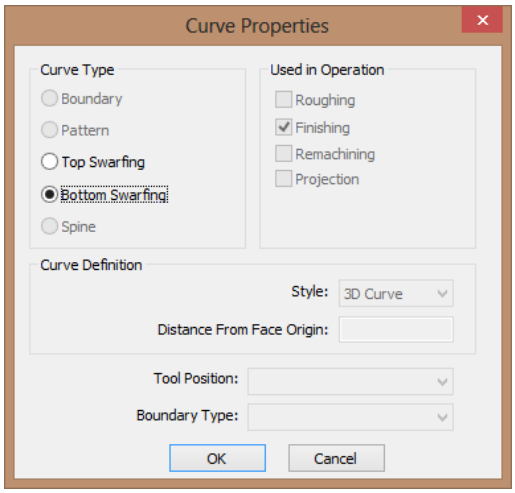
6 Zoom in on the two curves to verify that both curves are going in a clockwise direction.

7 Double click on the pattern curve at the face of the part using the **Selection** icon to display the **Curve Properties** dialog below.



8 Set the curve at the face of the part to **Top Swarfing.**

9 Double click on the pattern curve at the Bottom of the part using the **Selection** icon to display the **Curve Properties** dialog below.



10 Set the curve at the face of the part to **Bottom Swarfing.**

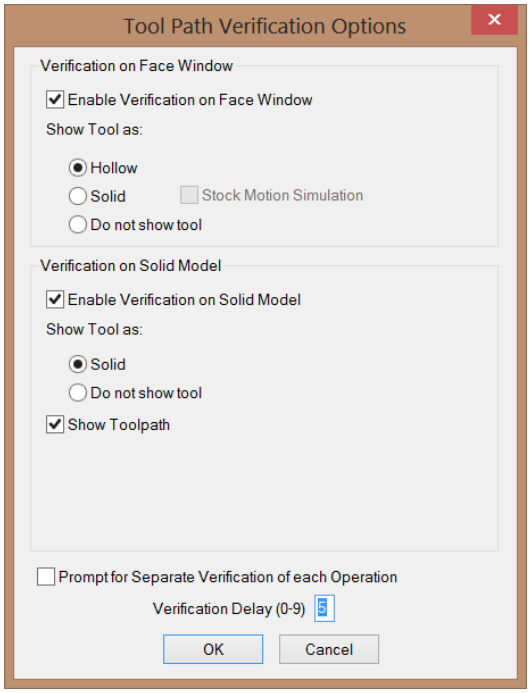
11 Click the **<OK>** button to return to the CAM Face Window.

Verify the Tool Path

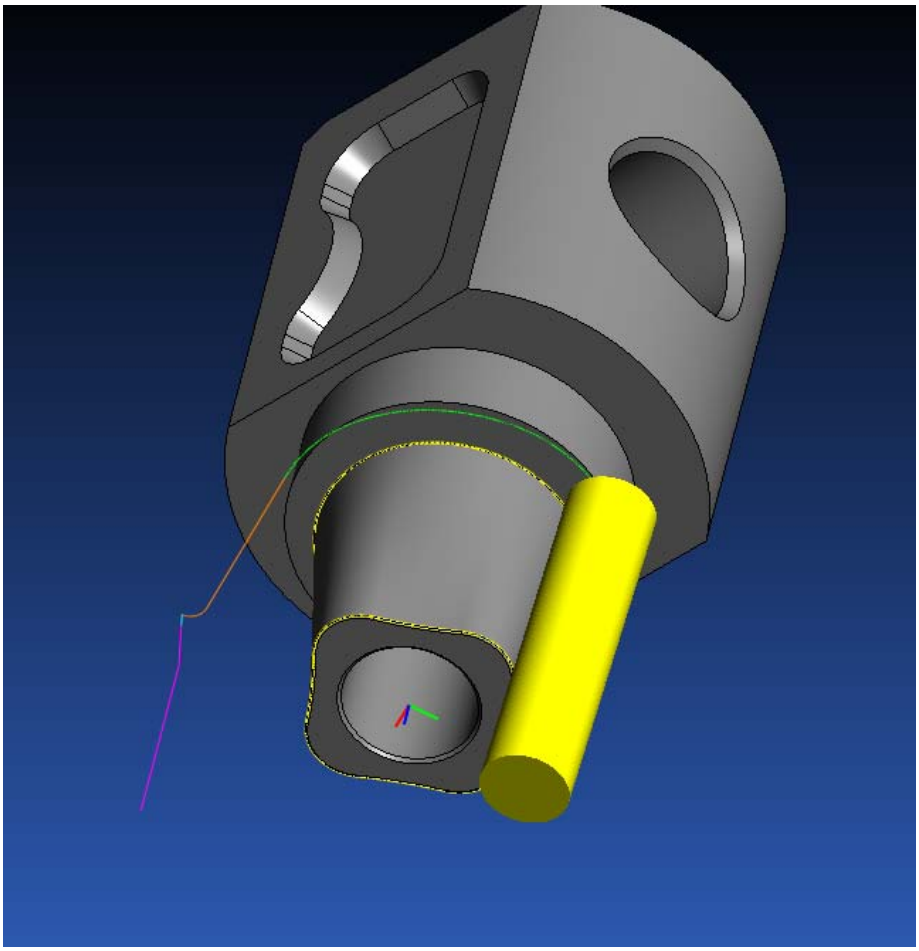
Once you have created the **surface group** and established the **Boundary Curves** you can visually verify the tool path PartMaker has calculated. To do so:

1. From the Part Features menu choose Verify Work Group Tool Path.
2. In the **Tool Path Verification Options** click the **Hollow** radio button as shown below.
3. Enter a Verification Delay of 5.

Note: *When entering a* ***Verification Delay*** *of greater than 0, you will see the 3D representation of the cutting tool moving along the Solid Model. When* ***Verification Delay*** *is set to 0, you will only see the path of the tool on the Solid Model.*



4 Click **<OK>** to show the calculated tool path. Your screen should appear as shown below:



Note: *Notice above, the line that extends into the programmed tool path. This line represents the 1-inch* ***Lead In & Lead Out*** *set in the* ***Leads and Links*** *property page. Creating this line assures that the tool will safely approach the stock without risking a tool collision.*

5 From the Part Features menu choose Hide Work Group Tool Path.

Exercise 2: 4-Axis Spine Finishing Using PartMaker Advanced Surface Machining Multi-Axis (ASM-MX-T) for SwissCAM and Turn-Mill

Introduction

This tutorial is designed to help you learn the steps for using PartMaker SwissCAM or PartMaker Turn-Mill to program parts on a lathe utilizing PartMaker's Advanced Surface Machining Multi-Axis module. This tutorial assumes that you have a good working knowledge of either PartMaker SwissCAM or Turn-Mill. Though the PartMaker SwissCAM module is used for this tutorial, the steps can be identically replicated if working in PartMaker Turn-Mill.

This tutorial has been developed for use with PartMaker Versions 2015 and higher and uses the files in the **C:\PartMaker\pm-swiss\ASM-MXT_Surface_Machining\4_Axis_Abutment_ASM_MXT_Sample** folder**.**

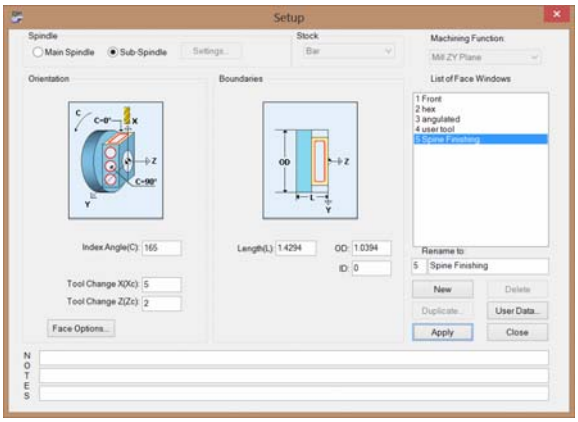
Creating a Spine Finishing Tool Path on the Solid model Part

In this section of the tutorial you will create and verify surfacing tool paths for Spine Finish machining on the imported solid model using ASM-MX-T.

Create a New Face Window for Surfacing on the part

The first step in creating a surfaced feature in PartMaker Turn-Mill or SwissCAM is to create a new Face Window.

1. Choose **Setup** from the **View** menu
2. In the **Setup** dialog, click the <**New**> button to create a new Face Window.
3. From the **Machining Function** drop down menu, choose **Mill ZY Plane**.
4. In the **Rename To:** field, type "**Spine Finishing**" and click **<Apply>**. Your completed **Setup** dialog should appear as shown below:

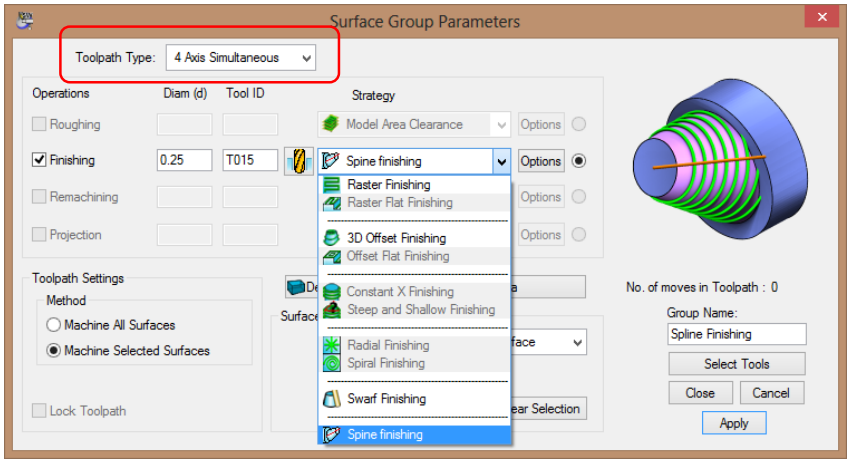


5. Click the **<Close>** button to close the **Setup** dialog.

Create a New Surface Group for Spine Finishing Machining

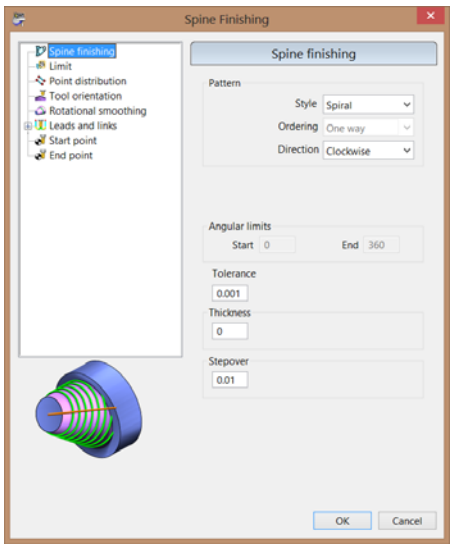
In this section of the tutorial you will create and verify a tool path for Spine Finishing on the imported solid model.

1. Choose New Surface Group from the Part Features menu. The Surface Group Parameters dialog will display.
2. Select the **4-Axis Simultaneous** option from the **Toolpath Type** drop down menu.
3. Select the **Spine Finishing** strategy.
4. Enter a tool diameter of 0.05 in (1.25 mm) in the **Diam (d)** field and click the **<Select Tools>** button.
5. In the **Group Name** field enter "**Spine Finishing**" and click **<Apply>**. The dialog should now appear as shown below:



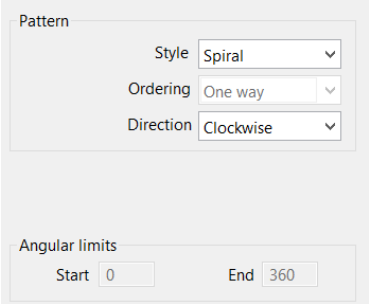
Setting Strategy Options for Spine Finishing

Click the <**Options**> button in the **Surface Group Parameters** dialog which will open the dialog for the Spine Finishing strategy as shown below in the picture.

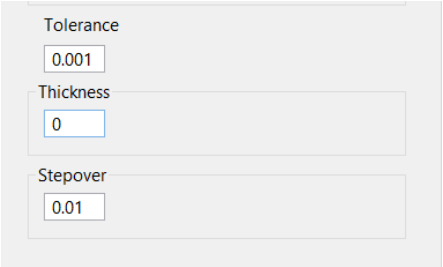


General Spine Finishing Settings

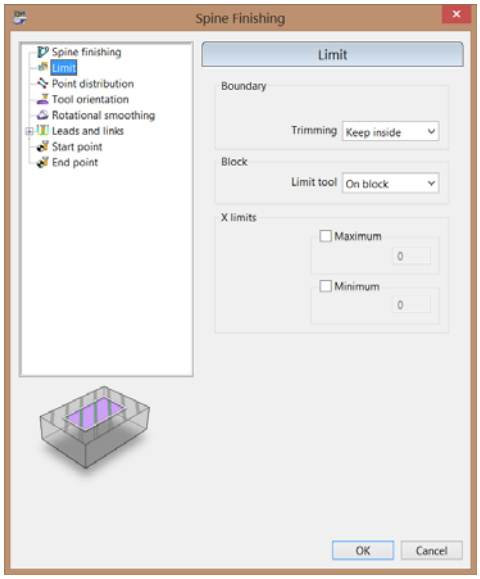
1. In the Spine finishing dialog, on the main page set the **Pattern Style** to Spiral and the **Direction** to Clockwise as shown below:



2. In the Spine finishing dialog, on the main page enter the **Tolerance** value of 0.001 (0.0254 for metric) as shown below:
3. Set **Thickness** to 0. This is the amount of material to be left behind.
4. Set the Step over amount to 0.01. This is the distance between successive machining passes.

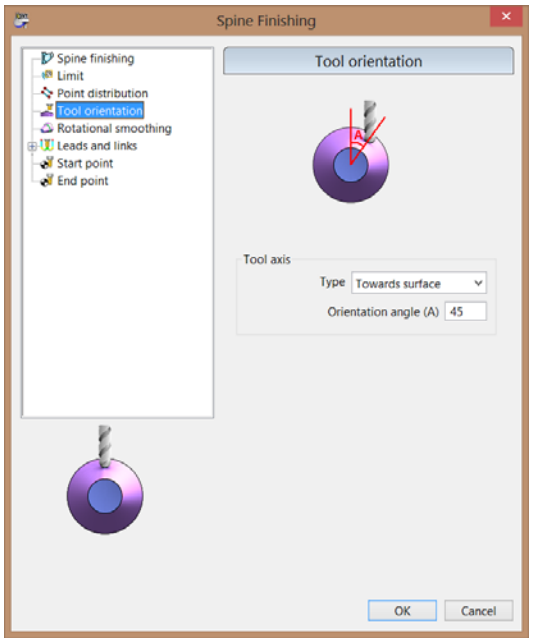


5. Select the **Limit** tab in the tree on the left in the dialog.
6. Set Boundary to Keep inside.
7. Set Block Limit Tool to On Block.

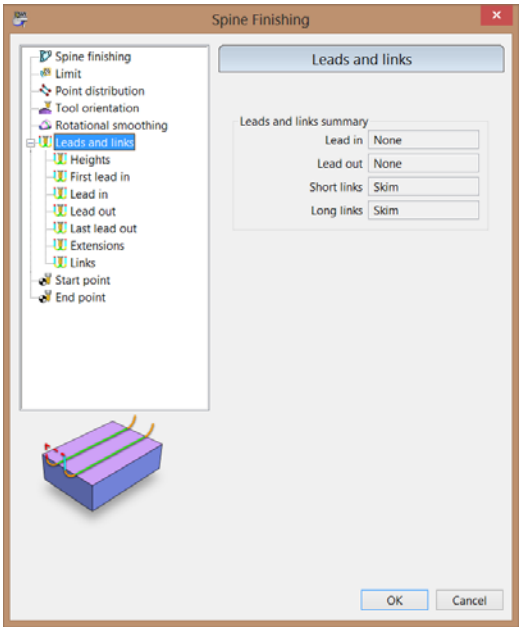


8. Select **Tool orientation** tab in the tree on the left in the dialog.
9. Set the Tool Axis Type to Towards surface.

10 Set the **Orientation angle (A)**. This is an optional stock rotation angle, which PartMaker applies after calculating the toolpath that achieves the desired tool orientation. You can use this angle to control the tool contact point and so achieve a better surface finish.

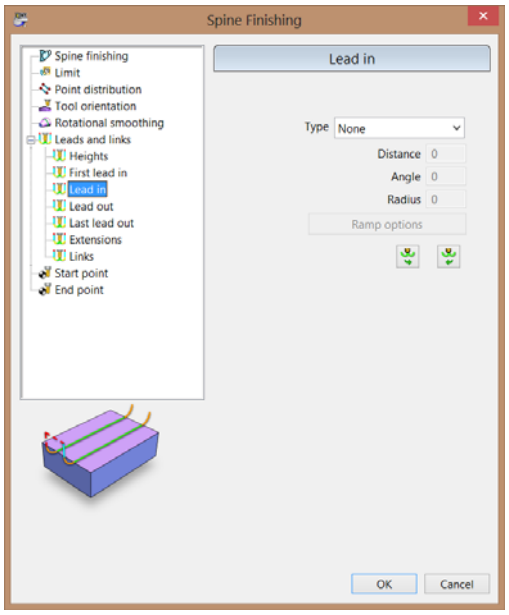


11 Click the **Leads and Links** tab in the tree on the left in the dialog to expand the **Lead and Links** features.



12 Click on **Lead in** on the tree to see the lead-in options.

13 Set the **Lead in** Type to None.

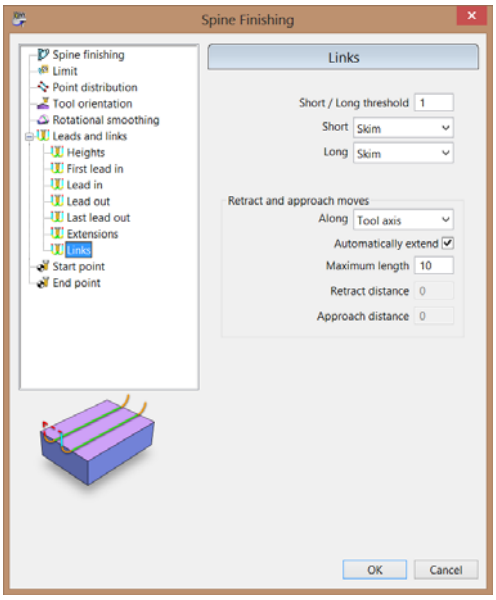


14 Click on  to copy the same parameters from **Lead In** to **Lead Out.**

15 Click on **Links** on the tree to see the Links options.

16 Set the **Short** and **Long** link to **Skim**. This will move the tool clear of the part by the specified incremental distance, descend at rapid feed to the specified incremental distance above the contact point, and then plunge the remaining distance.

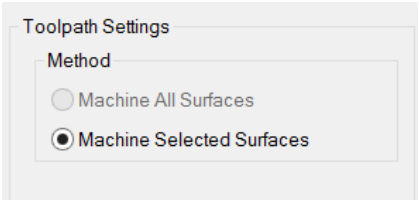
17 Set the **Retract and Approach** moves to **Tool Axis.** This will make the retract and approach moves have the same orientation as the tool axis.



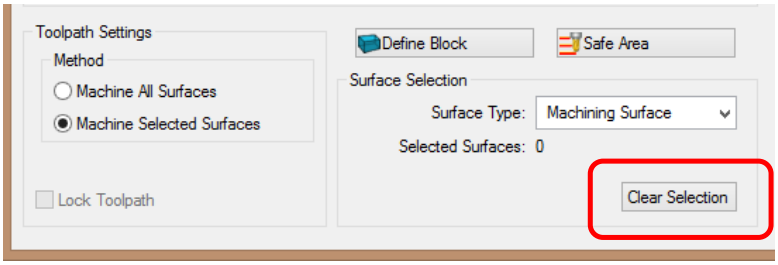
18 Select <**OK**>I to return to the **Surface Group Parameters.**

Select the Surfaces for Machining

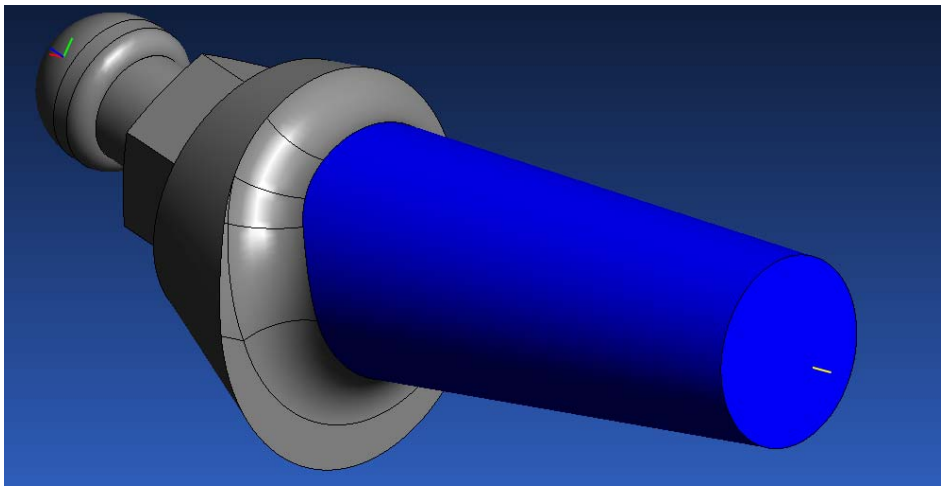
At this point you should still have the **Surface Group Parameters** dialog open. In the lower left hand corner of this dialog there is a framed section called **Toolpath Settings**. In this section you will control how you select surfaces to be machined by the current operation. There are two choices: **Machine All Surfaces** and **Machine Selected Surfaces**. By default **Machine All Surfaces** will be selected which will select all surfaces of the imported solid model as Machining Surfaces. In this exercise you will select the surfaces to be machined.



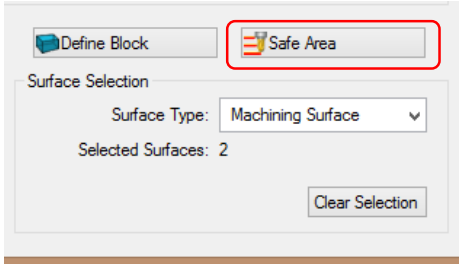
1 Click on the <**Clear Selection**> button.



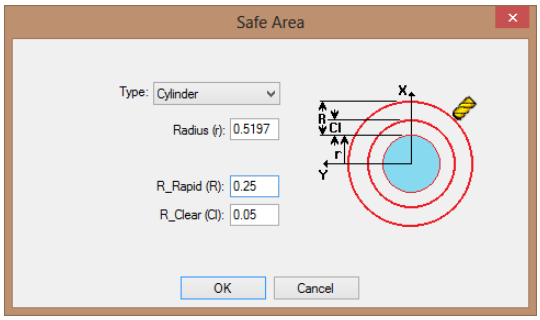
2 In the **Solid Model File** window, click on the surfaces on the solid model as indicated in the picture below:



3 Select the <**Safe Area**> button.



4 Set the Safe Area Type to Cylinder:



5 Select <**OK**> to close the Safe Area dialog

6 Click the **<Close>** button to return to the CAM Face Window.

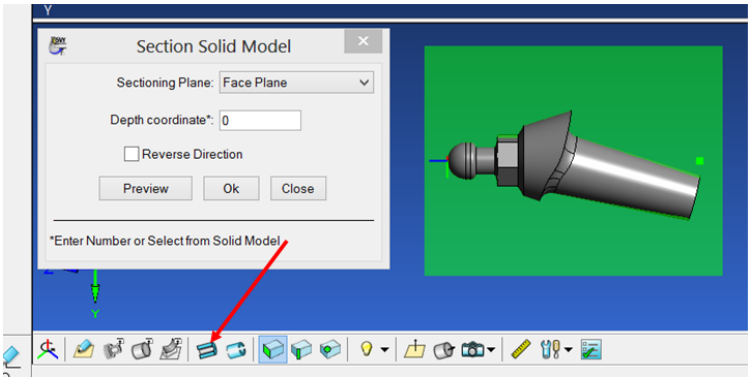
Create CAD to apply the Spine Finishing Machining Curve

Boundary curves are created the same way as 2D profile curves. They are used as tool path boundaries to restrict the machining area. In this example, you will section the model and Transfer the Planar Geometry to the CAD window.

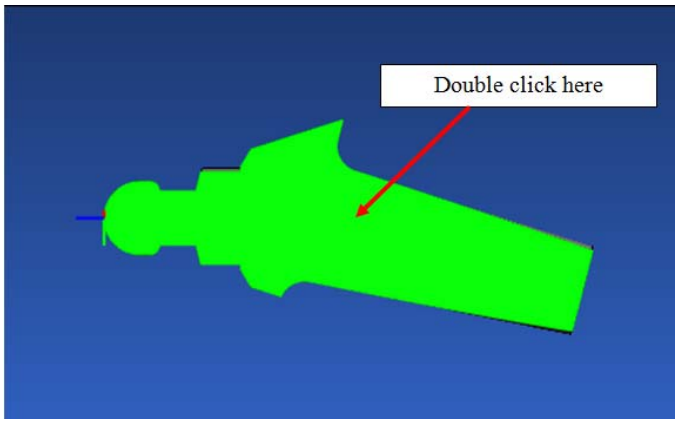
To create the boundary curves for this feature:

1 Click the CAD/CAM switch to enter the CAD mode.

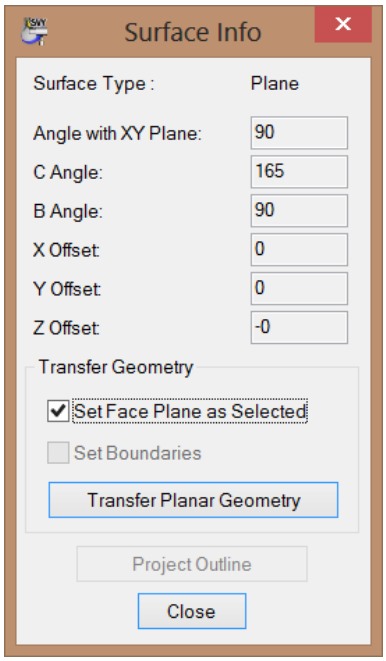
2 Section the model along the Face Plane by selecting the Sectional Part View button on the Solid Model tool bar and click <**OK**>.



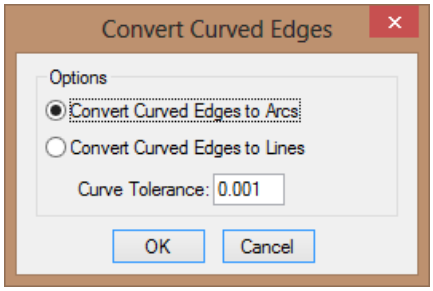
3 Double click on the face of the solid model as indicated below to open the **Surface Info** dialog.



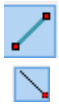
4 Click the <**Transfer Planar Geometry** > button in the **Surface Info** dialog shown below to transfer the profile into the CAD Face Window.



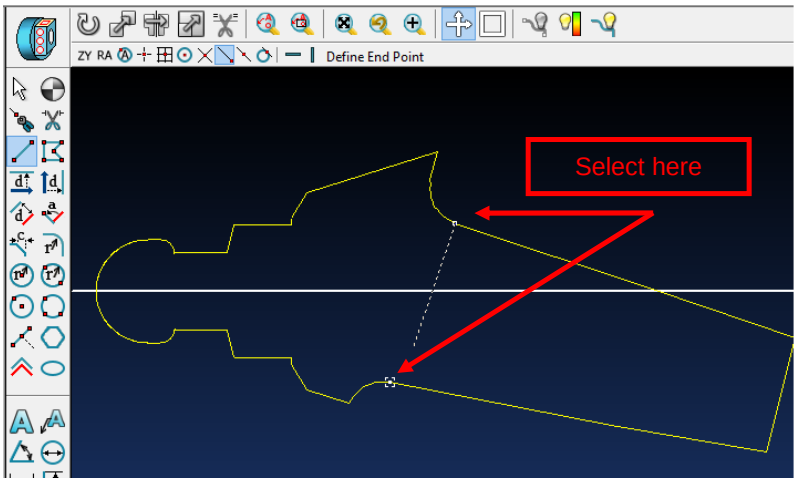
5 Click the Convert Curved Edges to Arcs radio button.



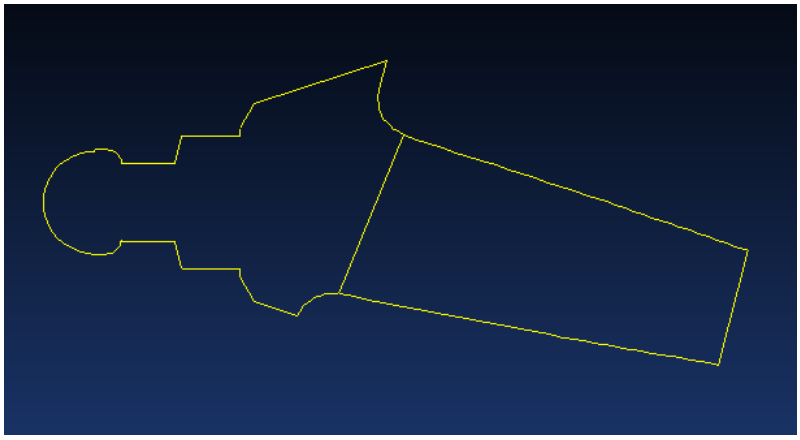
The CAD window as shown below:



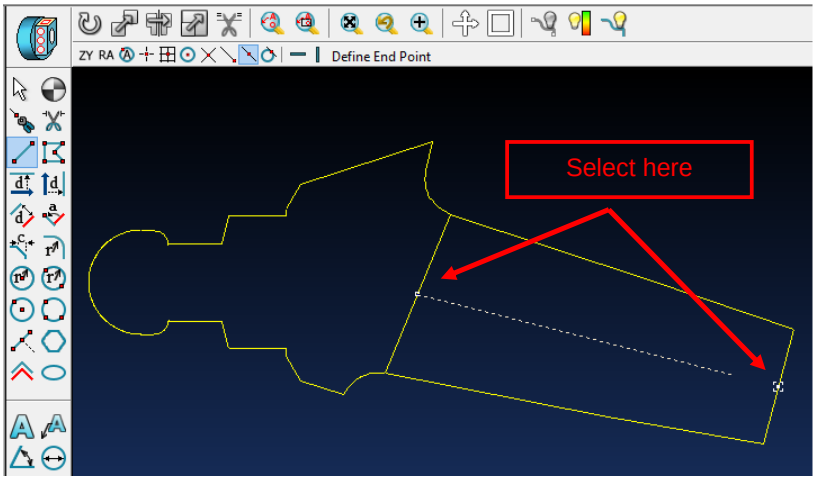
6 Create a line through two points at the base of the abutment by selecting the **Line Through two points Icon** and the **End of an Element snap mode** icon and then select the base of the two radiuses.



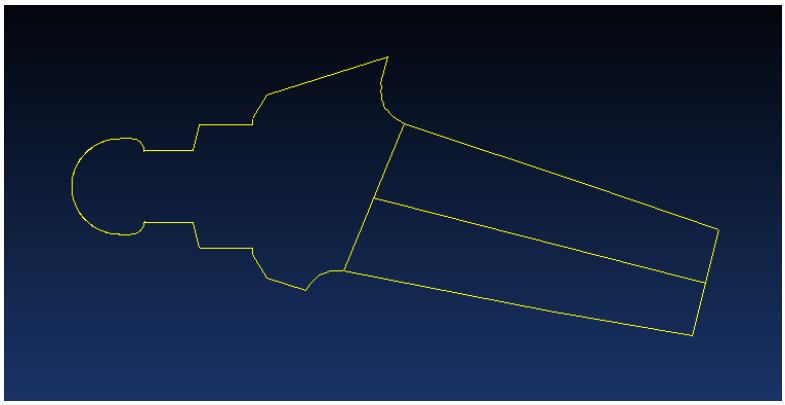
Finished line shown below:



7 Make a line through the middle of this line and the end of the part by selecting the **Line Through two points Icon** and the **Middle of an Element snap mode** icon and then select the middle of the two lines as shown.



Finished line shown below:

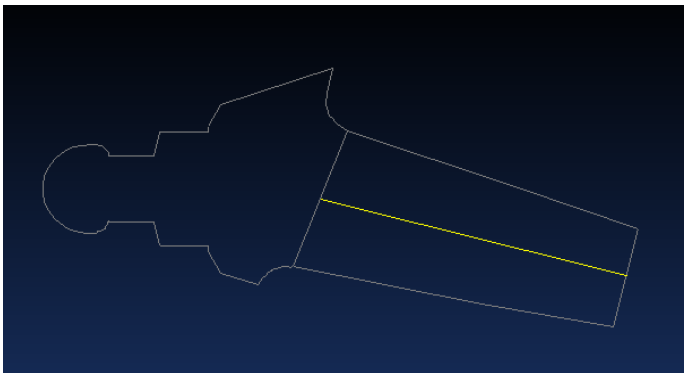


8 Click anywhere in the Face Window to deselect the geometry.

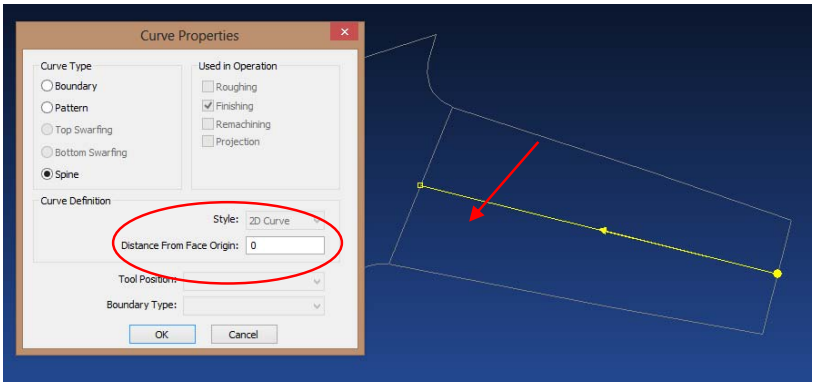
9 Switch to the **CAM** mode.

10 Choose the **Chain Geometry** icon

11 With the **Chain Geometry** icon click on the line just created as shown in the picture below to create a boundary curve for the toolpath.



12 Double click on the profile line with the Selection Arrow and set the Distance from Face Origin to 0.

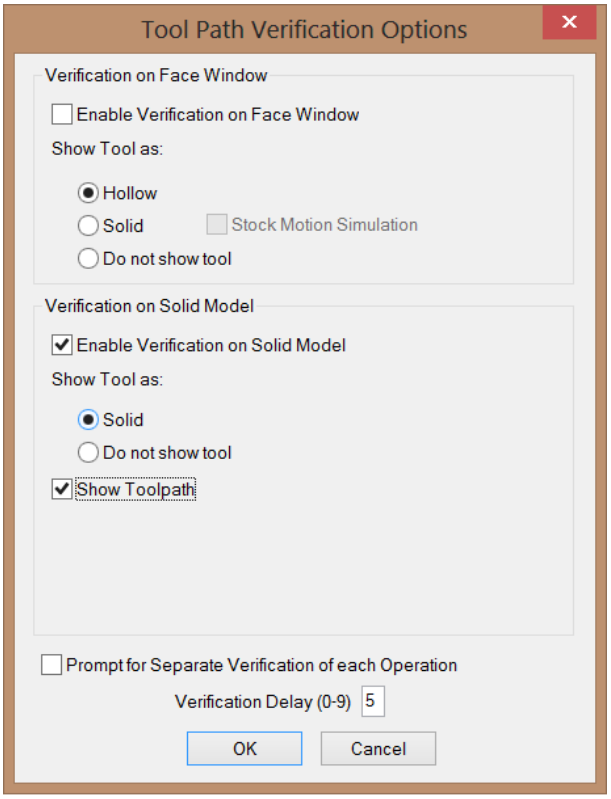


Verify the Tool Path

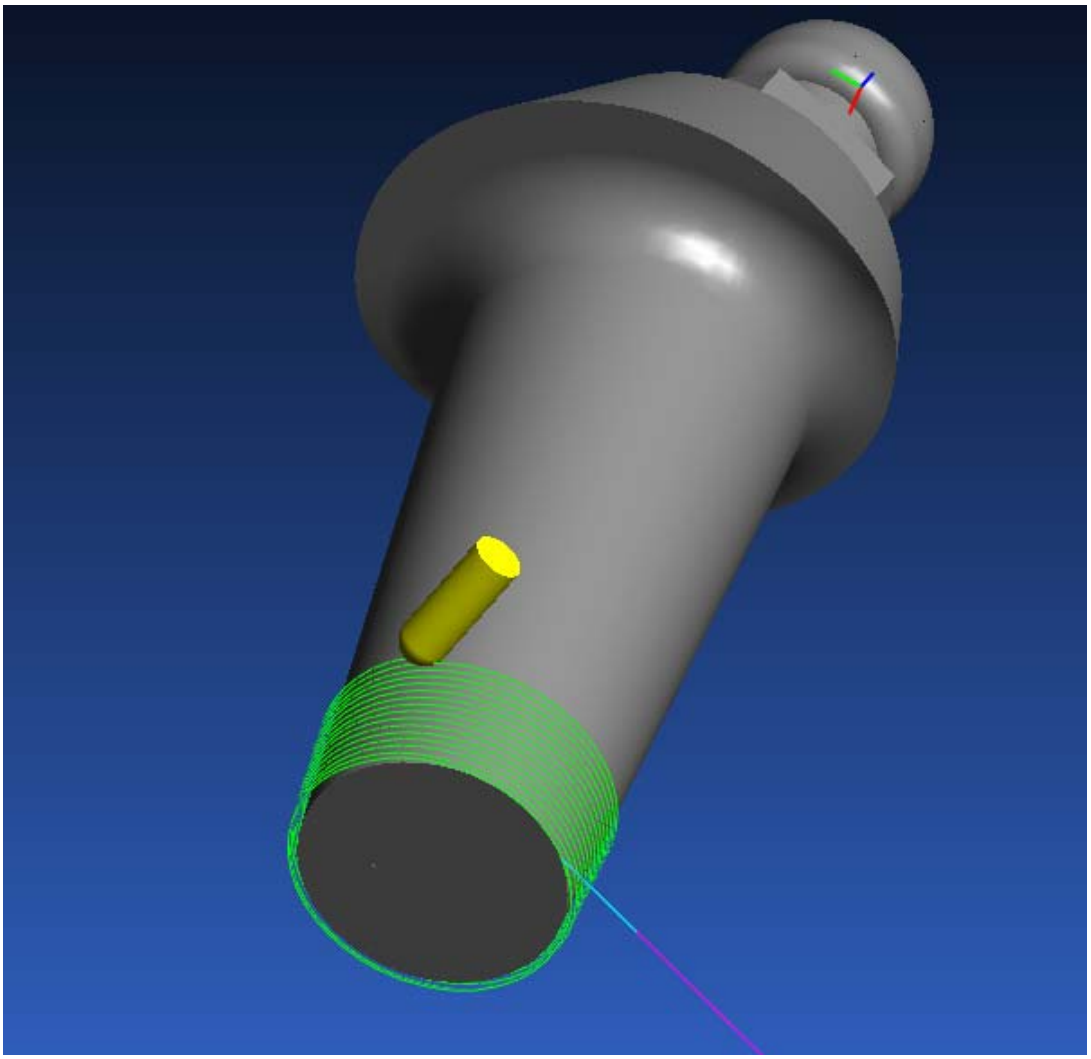
Once you have created the **surface group** you can visually verify the tool path PartMaker has calculated. To do so:

1. From the Part Features menu choose Verify Work Group Tool Path.
2. In the **Tool Path Verification Options** click the **Hollow** radio button as shown below.
3. Enter a Verification Delay of 5.

Note: *When entering a* ***Verification Delay*** *of greater than 0, you will see the 3D representation of the cutting tool moving along the Solid Model. When* ***Verification Delay*** *is set to 0, you will only see the path of the tool on the Solid Model.*



4 Click **<OK>** to show the calculated tool path. Your screen should appear as shown below:



Note: *Notice above, the line that extends into the programmed tool path. This line represents the* ***Lead In & Lead Out*** *set in the* ***Leads and Links*** *property page. Creating this line assures that the tool will safely approach the stock without risking a tool collision.*

5 From the Part Features menu choose Hide Work Group Tool Path.

Exercise 3: Lead/Lean Finishing Using PartMaker Advanced Surface Machining Multi-Axis (ASM-MX-T) for SwissCAM and Turn-Mill

Introduction

This tutorial is designed to help you learn the steps to go through in using PartMaker SwissCAM or PartMaker Turn-Mill to program parts on a lathe utilizing PartMaker's Advanced Surface Machining Multi-Axis Module. This tutorial assumes that you have a good working knowledge of either PartMaker SwissCAM or Turn-Mill. Though the PartMaker SwissCAM module is used for this tutorial, the steps can be identically replicated if working in PartMaker Turn-Mill.

This tutorial has been developed for use with PartMaker Version 2015 and higher and uses files in the **C:\PartMaker \pm-swiss\ASM-MXT_Surface_Machining \Tool_Axis_ASM-MXT_Sample** folder.

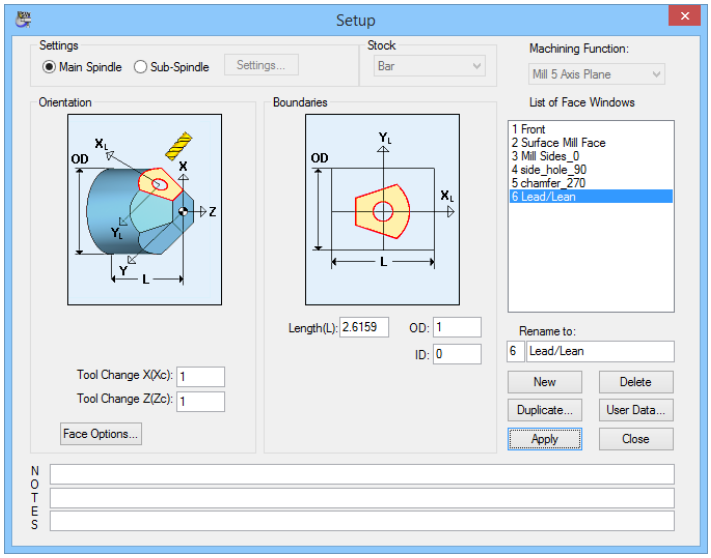
Creating a Lead/Lean Finishing Tool Path on a Solid model Part

In this section of the tutorial you will create and verify surfacing tool paths for Lead/Lean Finish machining on the imported solid model using ASM-MX-T.

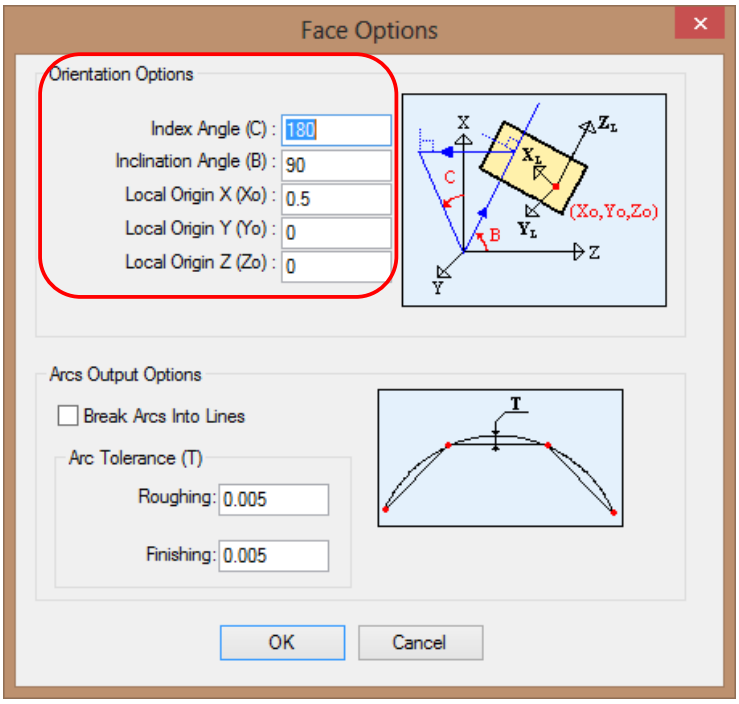
Create a New Face Window for Surfacing on the part

The first step in creating a surfaced feature in PartMaker Turn-Mill or SwissCAM is to create a new Face Window. In this case, you will use the Mill 5-Axis Face Window:

1. Choose **Setup** from the **View** menu
2. In the **Setup** dialog, click the **<New>** button to create a new Face Window.
3. From the **Machining Function** drop down menu, choose **Mill 5-Axis Plane**.
4. In the **Rename To:** field, type "**Lead/Lean**" and click **<Apply>**. Your completed **Setup** dialog should appear as shown below:



5 Select the <**Face Options**> button: Set the Orientation Options as shown below:



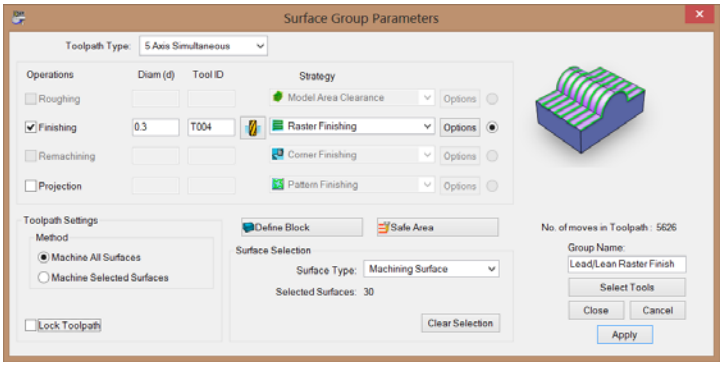
6 Click the **<OK>** button to close the **Face Options** dialog.

7 Click the **<Close>** button to close the **Setup** dialog.

Create a New Surface Group for Lead/Lean Finishing Machining

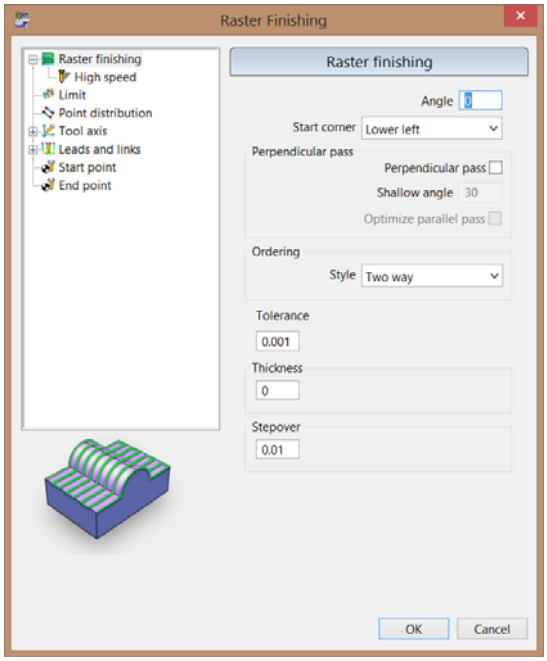
In this section of the tutorial you will create and verify a tool path using Lead/Lean Finishing on the imported solid model.

1. Choose New Surface Group from the Part Features menu. The Surface Group Parameters dialog will display.
2. Select the **5 Axis Simultaneous** option from the **Toolpath Type** drop down menu.
3. Select the **Raster Finishing** strategy. (Lead/Lean is also available in Spine Finishing and 3D Offset Finishing)
4. Enter a tool diameter of 0.300 in (7.62 mm) in the **Diam (d)** field and click the **<Select Tools>** button.
5. In the **Group Name** field enter "**Lead/Lean Raster Finish**" and click **<Apply>**. The dialog should now appear as shown below:



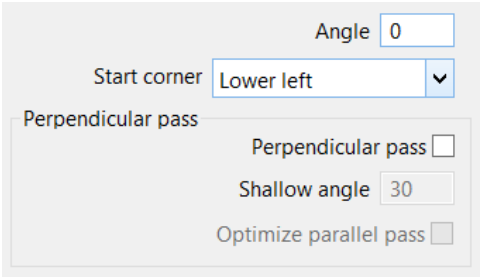
Setting Strategy Options for Raster Finishing

Click the <**Options**> button in the **Surface Group Parameters** dialog which will open the dialog for the Raster Finishing strategy as shown below in the picture.

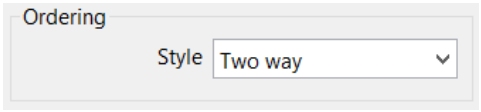


Raster Finishing settings

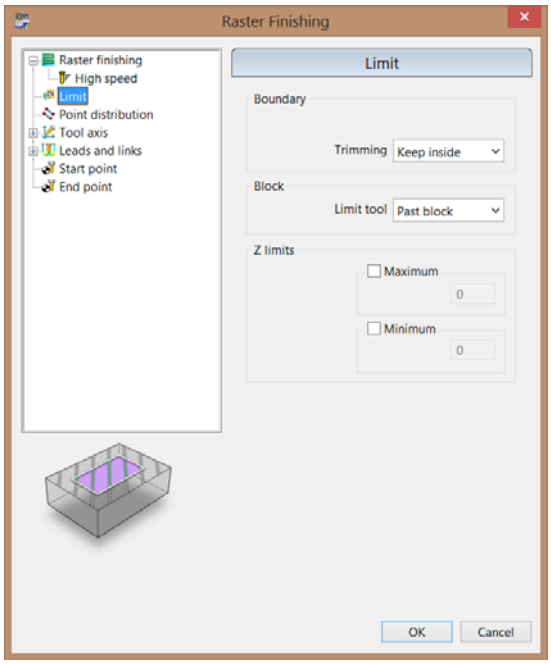
1. In the Raster finishing dialog, on the main page set the **Start corner** and the Angle as shown below:



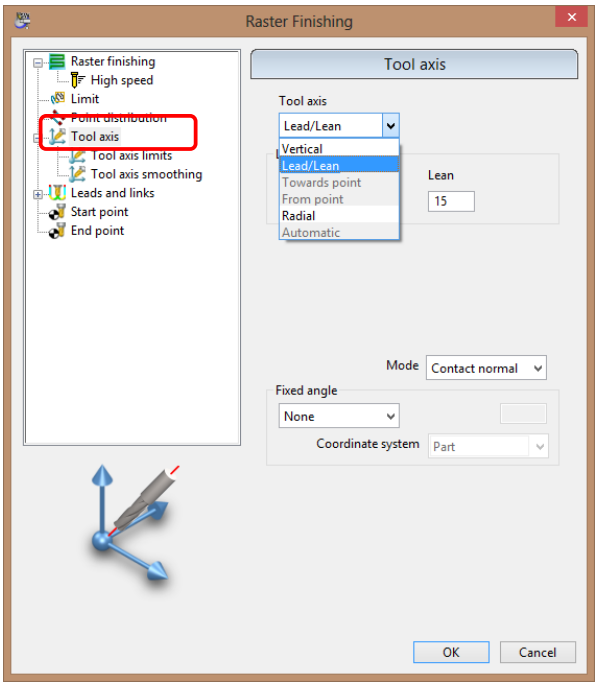
2. Set the **Ordering Style** to Two way as shown below:



3. In the Raster finishing dialog, on the main page enter the **Tolerance** value of 0.001 (0.0254 for metric).
4. Set **Thickness** to 0. This is the amount of material to be left behind.
5. Set the **Stepover** amount to 0.01. This is the distance between successive machining passes.
6. Select the **Limit** tab in the tree on the left in the dialog.
7. Set **Boundary** to Keep inside.
8. Set **Block Limit Tool** to Past Block.

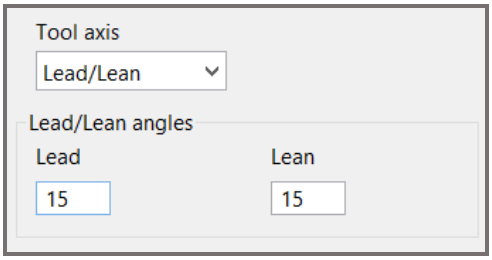


9 Select **Tool axis** tab in the tree on the left in the dialog.



10 Set the **Tool axis** to Lead/Lean.

11 Set the Lead/Lean angles:



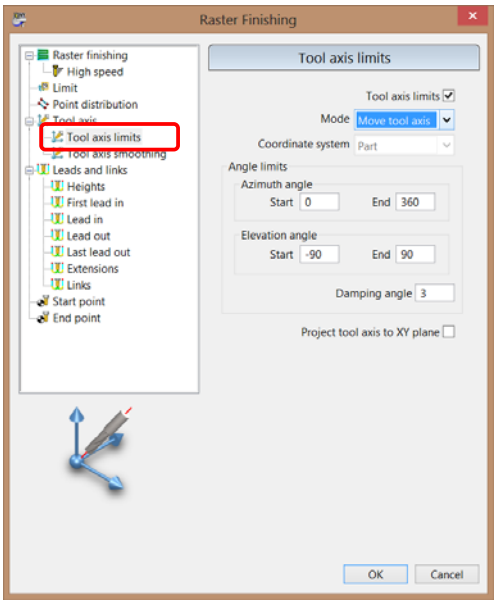
Lead: Enter the angle of the tool axis in the direction of travel. It is measured from the perpendicular to the direction of travel. 0 is vertical. Typically, this is used to avoid cutting at the center of a ball nosed tool on flattish areas. A typical **Lead Angle** is 15.

Lean: Enter the angle of the tool axis at right angles to the direction of travel. 0 is vertical. Typically, a **Lean Angle** is used to avoid the tool holder colliding with the part, to avoid collisions caused by a step, or to allow you to use a smaller tool when machining up to a step.

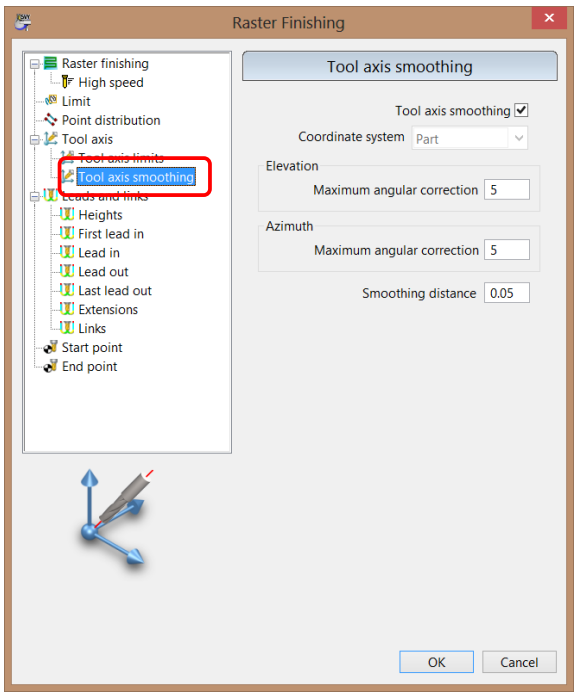
12 Select **Tool axis limits** from the tree and set the Azimuth and Elevation angles. Elevation is the B axis and Azimuth is the C axis.

Azimuth angle: Enter **Start** and **End** angles to define the angular limits of the machine tool in the XY plane. 0 is along the X axis; 90 is along the Y axis.

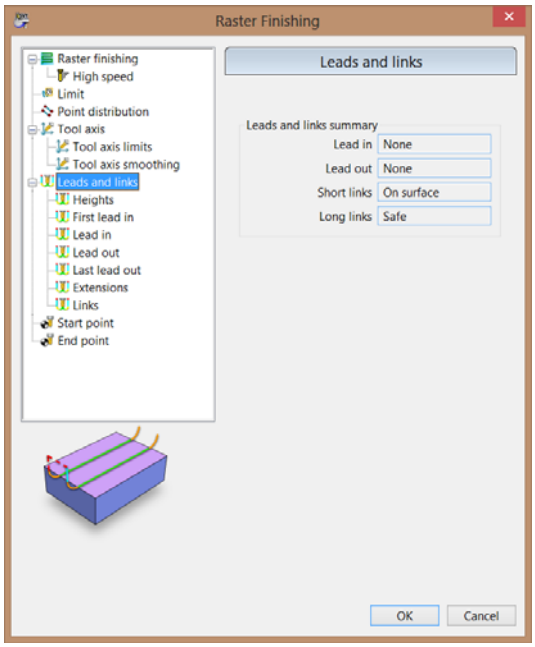
Elevation angle: Enter **Start** and **End** angles to define the angular limits of the machine tool above the XY plane. 0 is in the XY Plane; 90 is along the Z axis.



13 Turn on **Tool axis smoothing**: Enter the distance over which to smooth the tool axis movement. With sudden changes in direction in a toolpath (such as a right-angled corner), rapid changes of orientation of the tool axis leaves dwell marks. To prevent this, the **Smoothing distance** blends the change in orientation to produce a much improved surface finish.

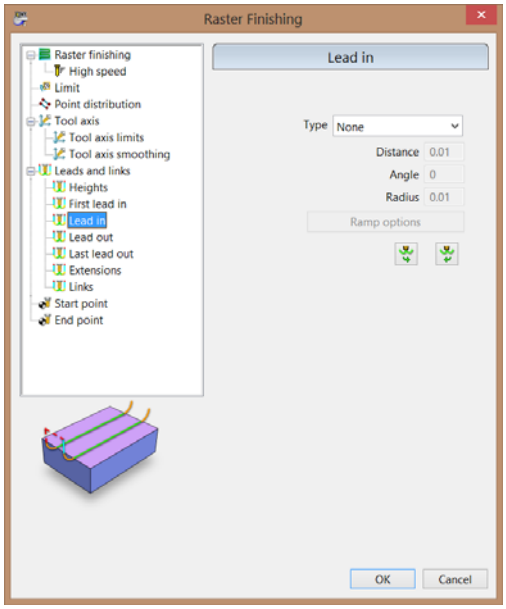


14 Click the **Leads and Links** tab in the tree on the left in the dialog to expand the **Lead and Links** features.



15 Click on **Lead in** on the tree to see the lead-in options.

16 Set the **Lead in** Type to None.



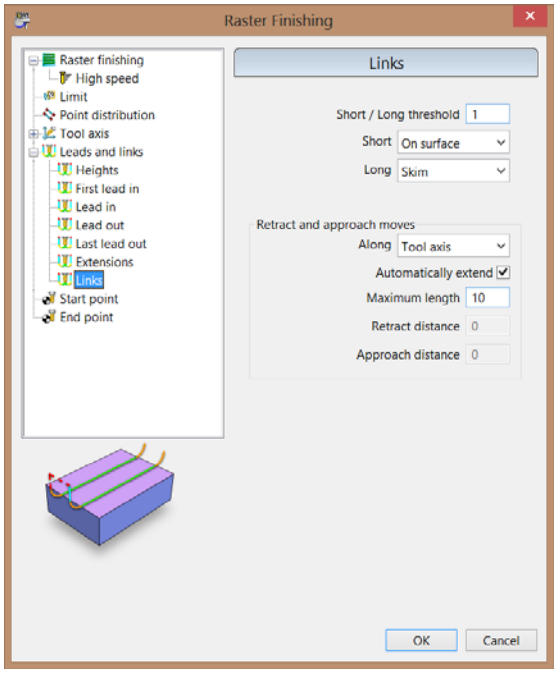
17 Click on  to copy the same parameters from **Lead In** to **Lead Out.**

18 Click on **Links** on the tree to see the Links options.

19 Set the **Short** Link to **On surface:** This will make links follow the surface of the triangulated model, and prevent gouging.

20 Set the **Long** link to **Skim**. This will move the tool clear of the part by the specified incremental distance, descend at rapid feed to the specified incremental distance above the contact point, and then plunge the remaining distance.

21 Set the **Retract and Approach** moves to **Tool Axis.** This will make the retract and approach moves have the same orientation as the tool axis.

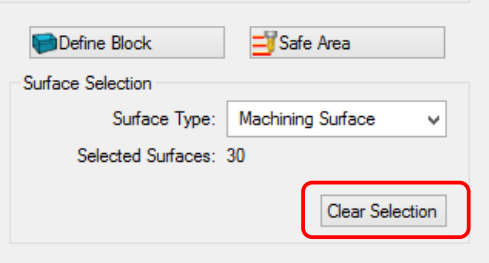


22 Select <OK> to return to the Surface Group Parameters.

Select the Surfaces for Machining

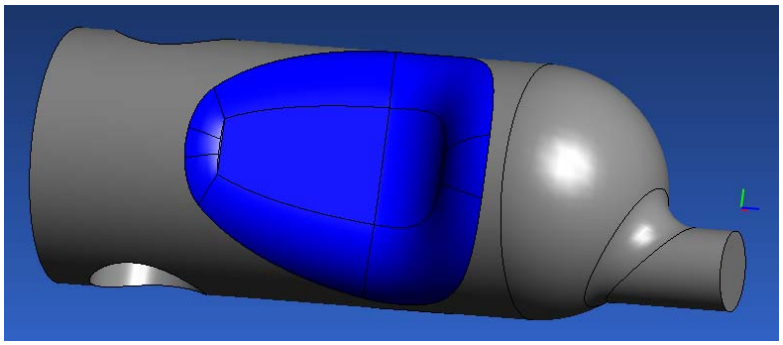
At this point you should still have the **Surface Group Parameters** dialog open. In the lower left hand corner of this dialog there is a framed section called **Toolpath Settings**. In this section you will control how you select surfaces to be machined by the current operation. There are two choices: **Machine All Surfaces** and **Machine Selected Surfaces**. By default **Machine All Surfaces** will be selected which will select all surfaces of the imported solid model as Machining Surfaces. In this exercise you will select the surfaces to be machined.

1 Select the **<Clear Selection>** button.



2 Now select the surfaces to be machined.

3 In the **Solid Model File** window, click on the surfaces on the solid model as indicated in the picture below: You should have 11 surfaces selected.

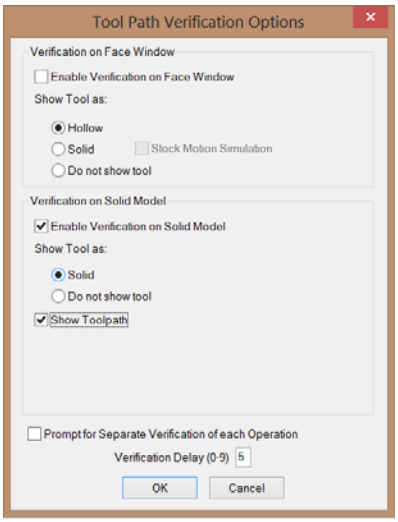


4 Click the **<Close>** button to return to the CAM Face Window.

Verify the Tool Path

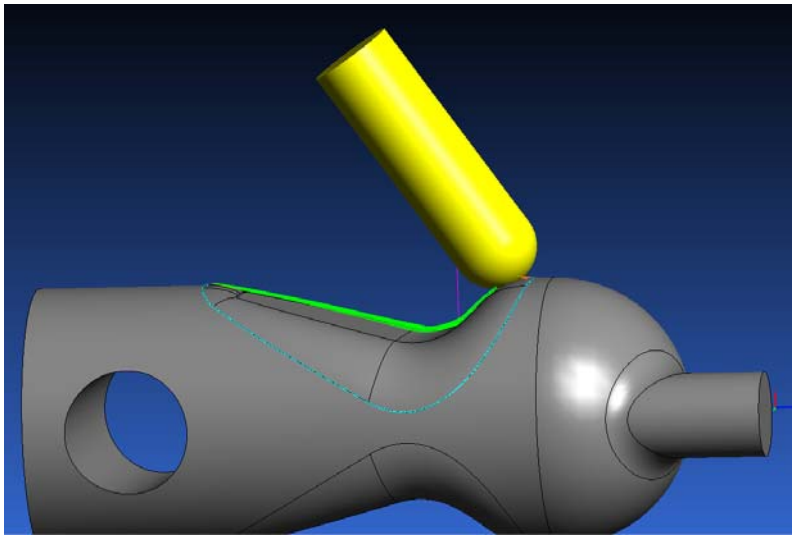
Once you have created the **surface group** you can visually verify the tool path PartMaker has calculated. To do so:

1. From the Part Features menu choose Verify Work Group Tool Path.
2. In the **Tool Path Verification Options** click the **Hollow** radio button as shown below.
3. Enter a Verification Delay of 5.



Note: *When entering a* ***Verification Delay*** *of greater than 0, you will see the 3D representation of the cutting tool moving along the Solid Model. When* ***Verification Delay*** *is set to 0, you will only see the path of the tool on the Solid Model.*

4. Click **<OK>** to show the calculated tool path. Your screen should appear as shown below:



Note: *Notice above, the line that extends into the programmed tool path. This line represents the* ***Lead In & Lead Out*** *set in the* ***Leads and Links*** *property page. Creating this line assures that the tool will safely approach the stock without risking a tool collision.*

5. From the Part Features menu choose Hide Work Group Tool Path.

Exercise 4: 5 Axis Surface Swarf Finishing Using PartMaker Advanced Surface Machining (ASM-MX-M) for PartMaker Mill

Introduction

This tutorial is designed to help you learn the steps to go through in using PartMaker Mill's Advanced Surface Machining Milling (ASM-MX-M). This tutorial assumes that you have a good working knowledge of PartMaker Mill.

This tutorial has been developed for use with PartMaker Version 2015 and higher and uses files in the **C:\PartMaker\pm-mill\ASM-MXM_Surface_Machining\ASM-MXM_Swarf_Tutorial** folder.

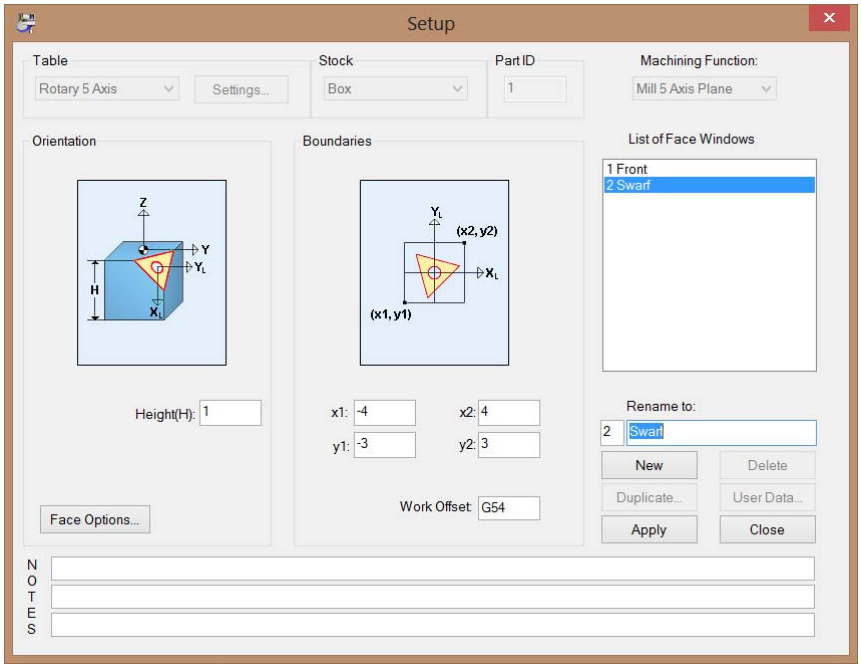
Creating a Surface Swarf Tool Path on the Solid model Part

In this section of the tutorial you will create and verify surfacing tool paths for Surface Swarf Finish machining on the imported solid model using ASM-MX-M.

Create a New Face Window for Surfacing on the part

The first step in creating a surfaced feature in PartMaker Mill is to create a new Face Window.

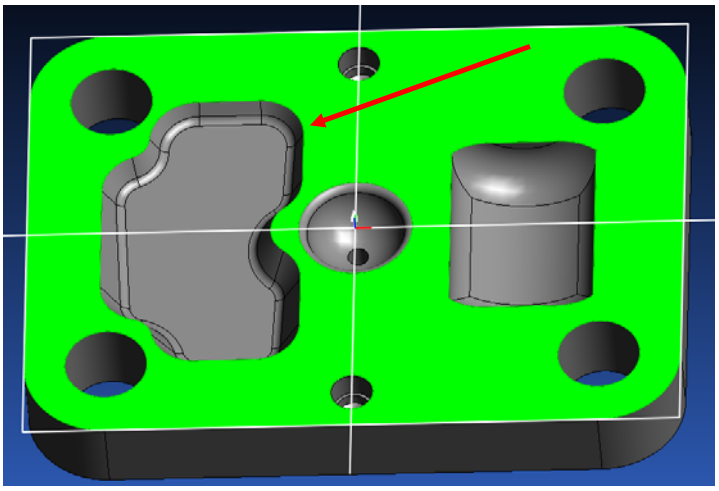
1. Choose **Setup** from the **View** menu
2. In the **Setup** dialog, click the <**New**> button to create a new Face Window.
3. From the **Machining Function** drop down menu, choose **Mill 5Axis Plane**.
4. In the **Rename To:** field, type "**Swarf**" and click **<Apply>**. Your completed **Setup** dialog should appear as shown below:



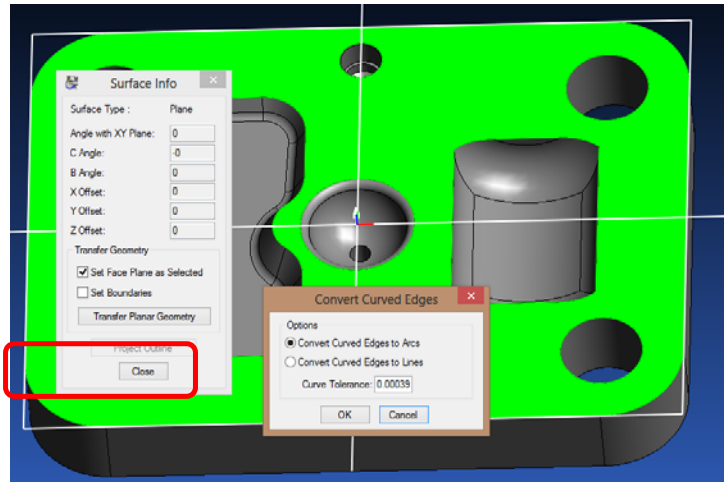
5. Click the **<Close>** button to close the **Setup** dialog.

Create 5 Axis Face Plane for Surface Swarf Machining

1 Double click on the surface as shown below:

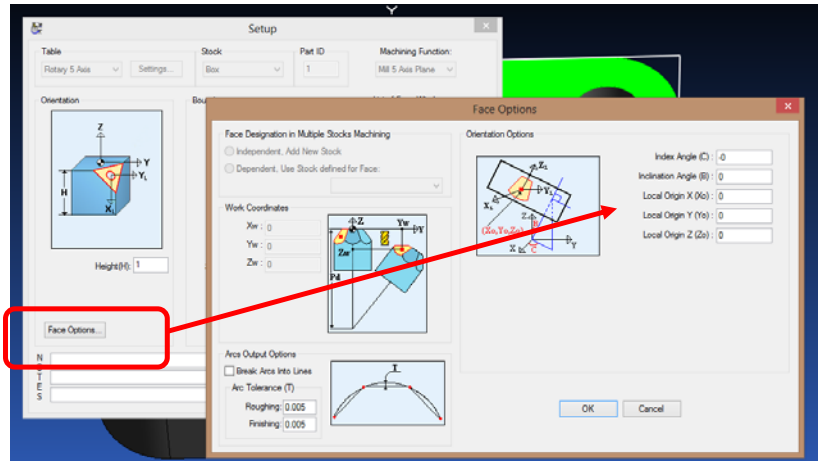


2 Select Transfer Planar Geometry:



3 Click the <OK> button to transfer the geometry and set the face plane for this Surface group.

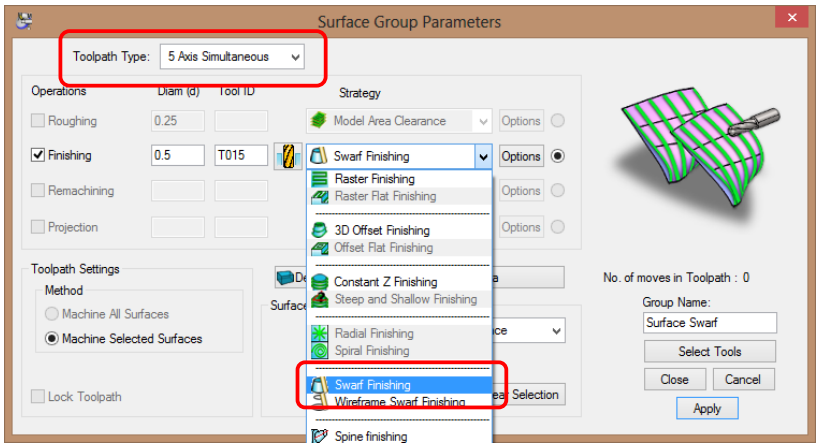
4 In the Setup Dialog the <**Face Options>** have been updated to the Orientation for this face.



Create a New Surface Group for Surface Swarf Machining

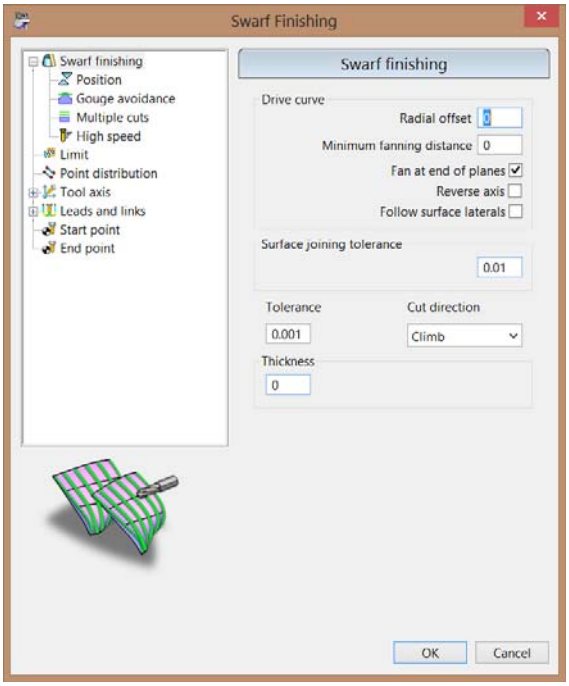
In this section of the tutorial you will create and verify a tool path for Surface Swarf machining on the imported solid model.

1. Choose New Surface Group from the Part Features menu. The Surface Group Parameters dialog will display.
2. Select the **5 Axis Simultaneous** option from the **Toolpath Type** drop down menu.
3. Select the **Swarf Finishing** strategy.
4. Enter a tool diameter of 0.5 in (12.7 mm) in the **Diam (d)** field and click the **<Select Tools>** button.
5. In the **Group Name** field enter "**Surface Swarf**" and click **<Apply>**. The dialog should now appear as shown below:



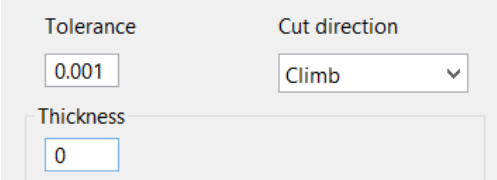
Setting Strategy Options for Surface Swarf Machining

Click the **<Options>** button in the **Surface Group Parameters** dialog which will open the dialog for the Swarf Finishing strategy as shown below in the picture.

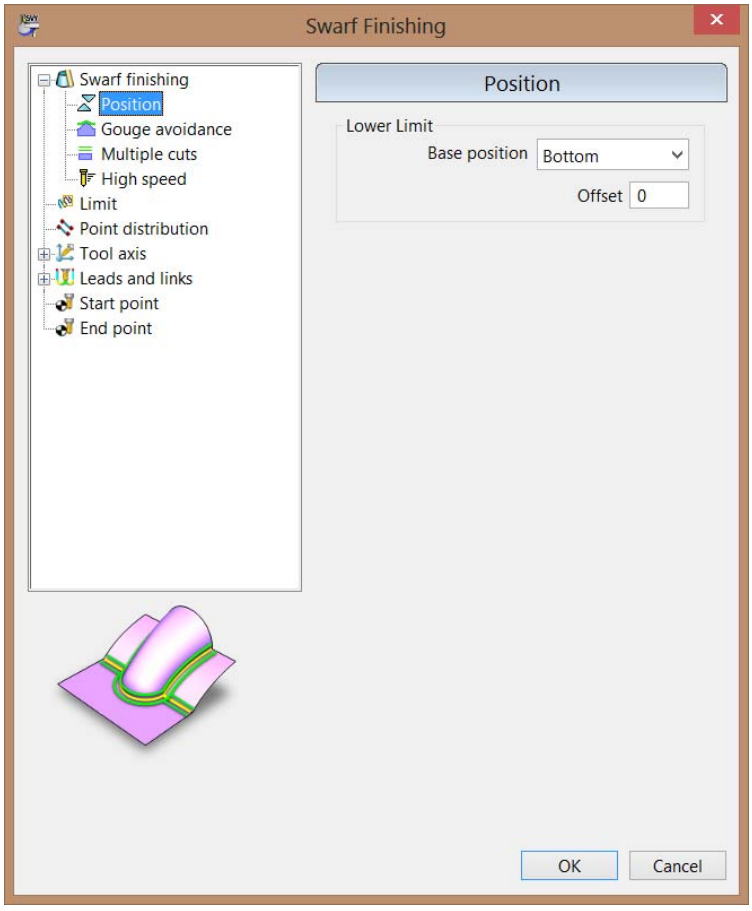


General Swarf Finishing Settings

1. In the **Swarf finishing** dialog, on the main page enter the **Tolerance** value of 0.001 (0.0254 for metric) as shown below:

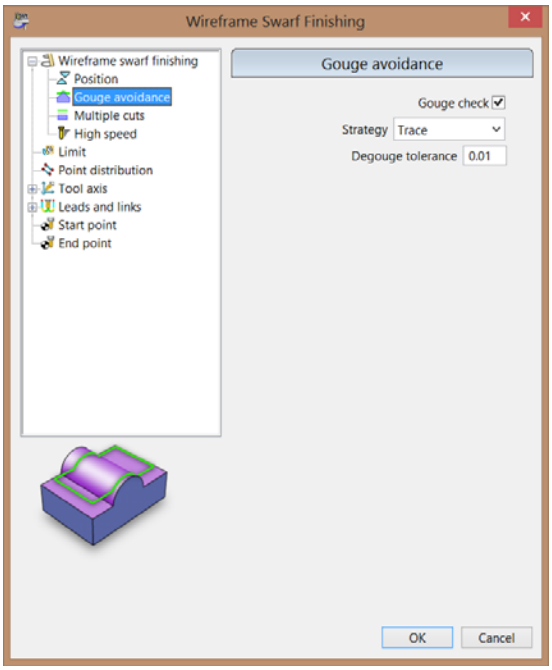


2. Select **Position**.
3. Set the **Base Position** to Bottom.
4. If you set **Offset** to a negative value the end mill will track that distance below the bottom curve.



5 Select Gouge avoidance.

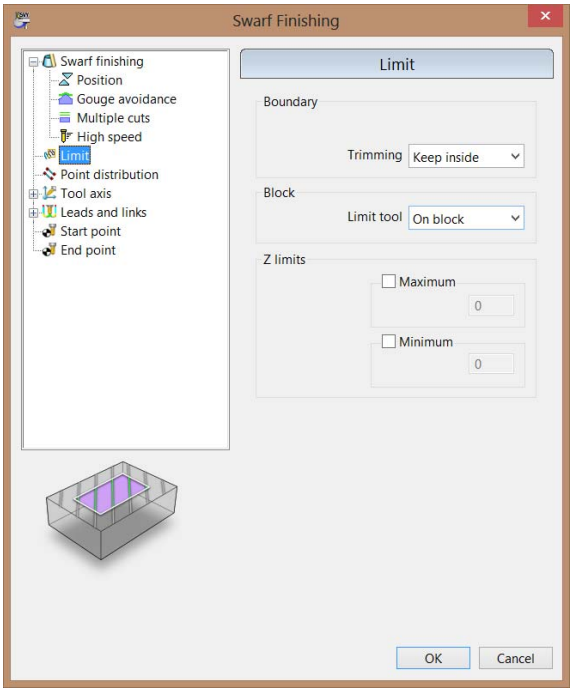
6 Check the box for Gouge Checking.



7 Select **Limit**.

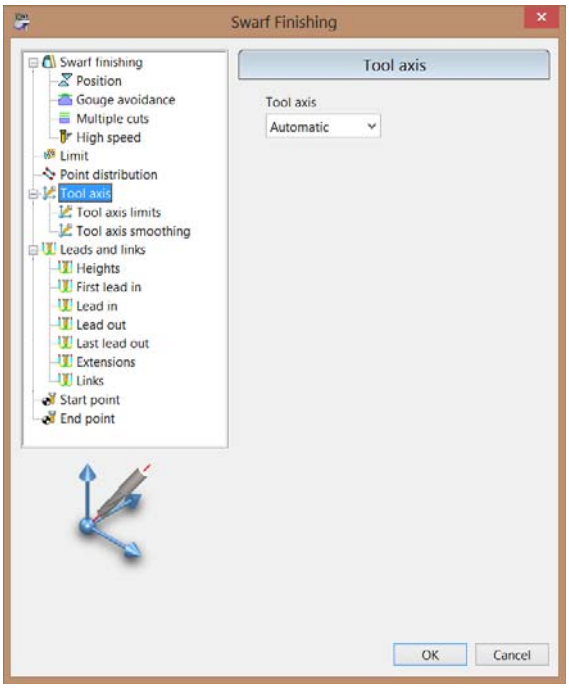
8 Set Boundary to Keep inside.

9 Set Block Limit Tool to On Block.



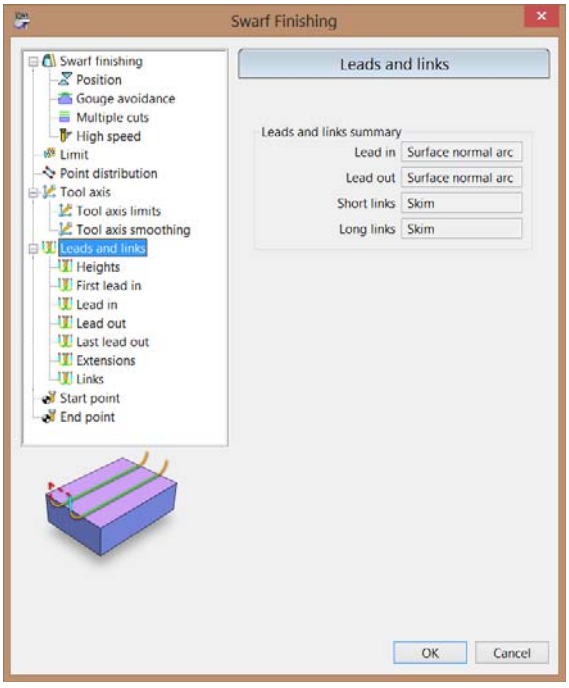
10 Select Tool Axis.

11 Tool Axis will be set to **Automatic**.

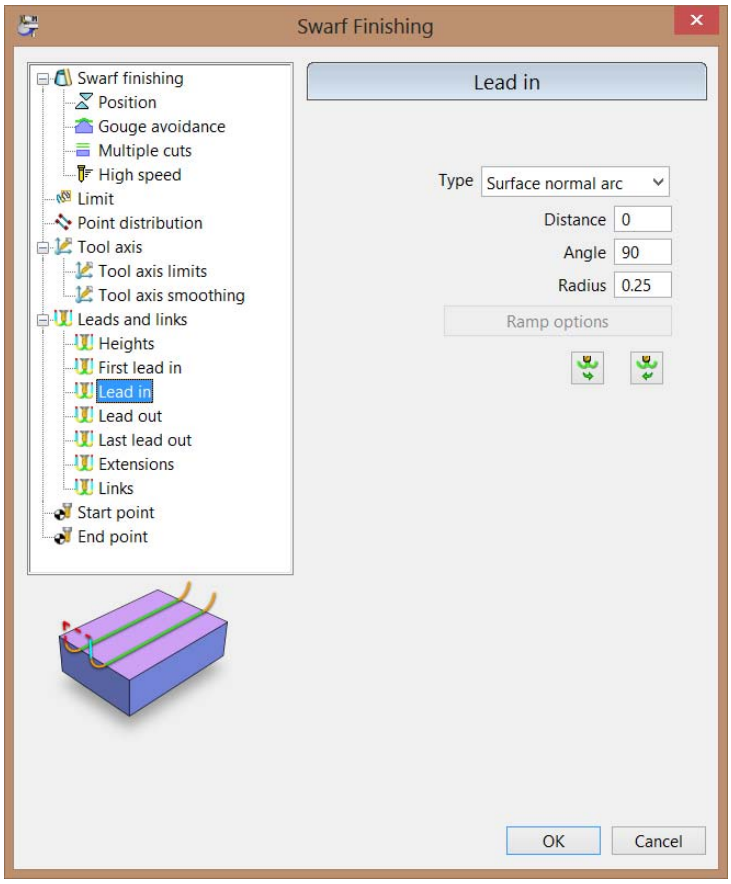


12 Click the **Leads and Links** tab in the tree on the left in the dialog to expand the **Lead and Links** features.

13 Click on **Lead in** on the tree to see the lead-in options.



14 Set Type to Surface Normal Arc.



15 The **Distance** field would now be enabled. Enter a value of 0 in the distance field.

16 Set the **Angle** to 90.

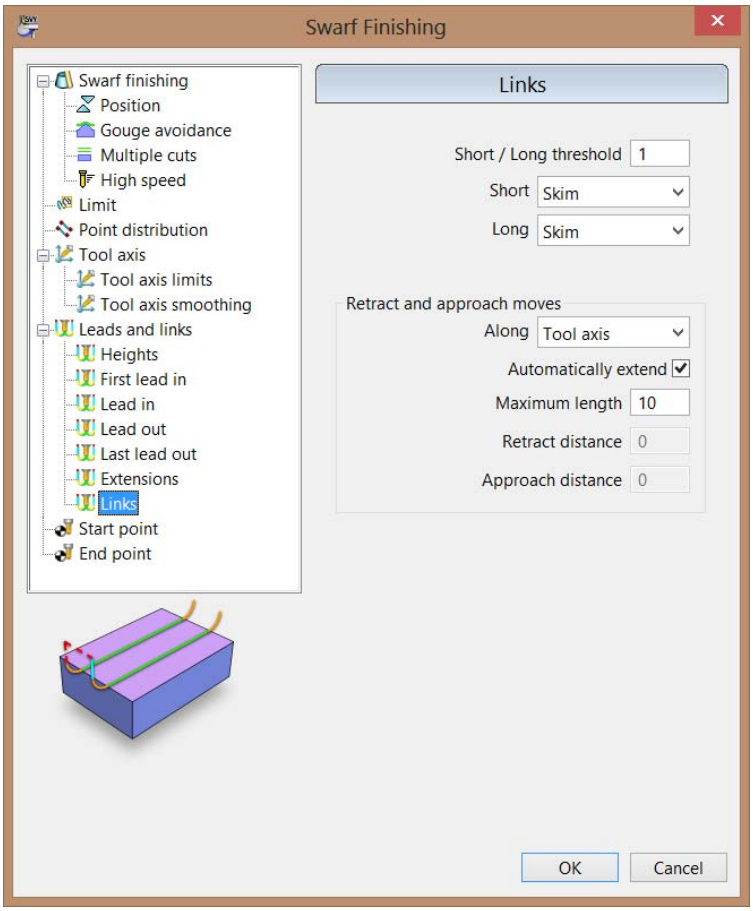
17 Set the **Radius** to .250 (6.350).

18 Click on  to copy the same parameters from **Lead In** to **Lead Out.**

19 Click on **Links** on the tree to see the Links options.

20 Set the **Short** and **Long** link to **Skim**. This will move the tool clear of the part by the specified incremental distance, descend at rapid feed to the specified incremental distance above the contact point, and then plunge the remaining distance.

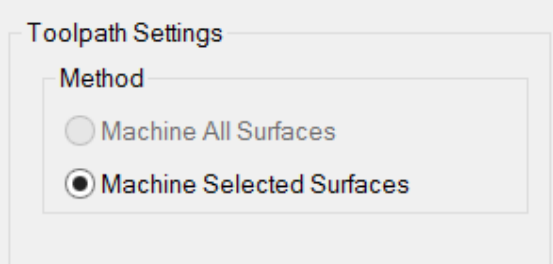
21 Set the **Retract and Approach** moves to **Tool Axis.** This will make the retract and approach moves have the same orientation as the tool axis.



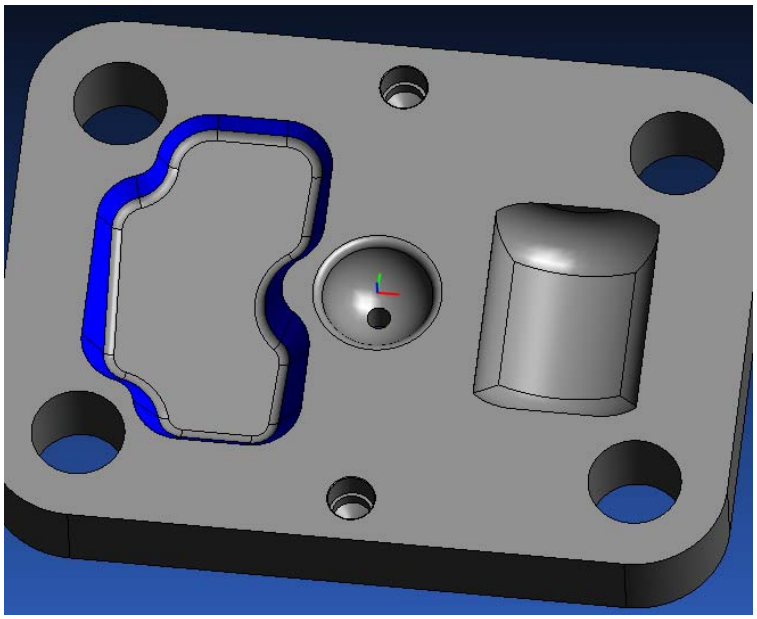
22 Select **<OK>** to return to the **Surface Group Parameters** dialog.

Select the Surfaces for Machining

At this point you should still have the **Surface Group Parameters** dialog open. In the lower left hand corner of this dialog there is a framed section called **Toolpath Settings**. In this section you will control how you select surfaces to be machined by the current operation. There are two choices: **Machine All Surfaces** and **Machine Selected Surfaces**. By default **Machine All Surfaces** will be selected which will select all surfaces of the imported solid model as Machining Surfaces. In this exercise you will select the surfaces to be machined.



1. In the **Solid Model File** window, click on the surfaces on the solid model as indicated in the picture below: This will create the Top and Bottom curve automatically for the surface group.



2. Click the **<Close>** button to return to the CAM Face Window.

Verify the Tool Path

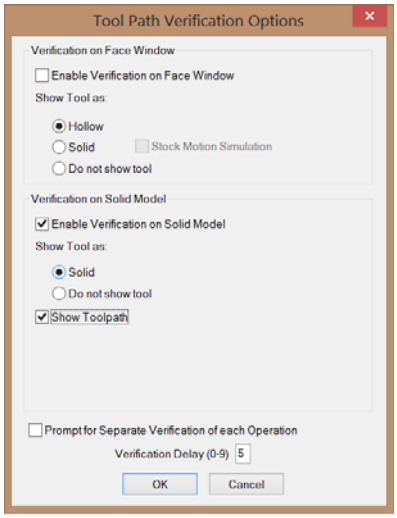
Once you have created the **surface group** you can visually verify the tool path PartMaker has calculated. To do so:

1 From the Part Features menu choose Verify Work Group Tool Path.

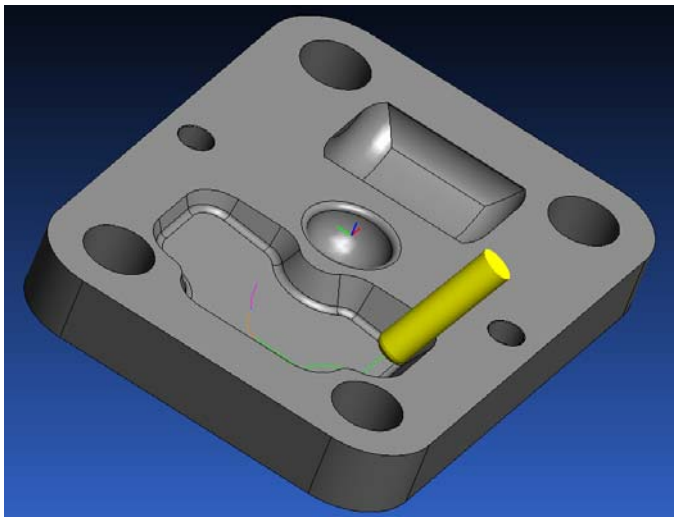
2 In the **Tool Path Verification Options** click the **Hollow** radio button as shown below.

3 Enter a Verification Delay of 5.

Note: *When entering a* ***Verification Delay*** *of greater than 0, you will see the 3D representation of the cutting tool moving along the Solid Model. When* ***Verification Delay*** *is set to 0, you will only see the path of the tool on the Solid Model.*



4 Click **<OK>** to show the calculated tool path. Your screen should appear as shown below:



Note: *Notice above, the line that extends into the programmed tool path. This line represents the* ***Lead In & Lead Out*** *set in the* ***Leads and Links*** *property page. Creating this line assures that the tool will safely approach the stock without risking a tool collision.*

5 From the Part Features menu choose Hide Work Group Tool Path.

Exercise 5: Tool axis control Finishing Using PartMaker Advanced Surface Machining (ASM-MX-M) for PartMaker Mill

Introduction

This tutorial is designed to help you learn the steps to go through in using PartMaker Mill to program parts on a 5-Axis mill utilizing PartMaker Advanced Surface Machining Multi-Axis module. This tutorial assumes that you have a good working knowledge of PartMaker Mill.

This tutorial has been developed for use with PartMaker Versions 2015 and higher and uses files in the **C:\PartMaker\pm-mill\ASM-MXM_Surface_Machining \ASM-MXM_Tool_Axis_Tutorial** folder.

Creating a Tool Axis control Finishing Tool Path on the solid model

In this section of the tutorial you will create and verify surfacing tool paths for Tool Axis control Finish machining on the imported solid model using ASM-MX-M.

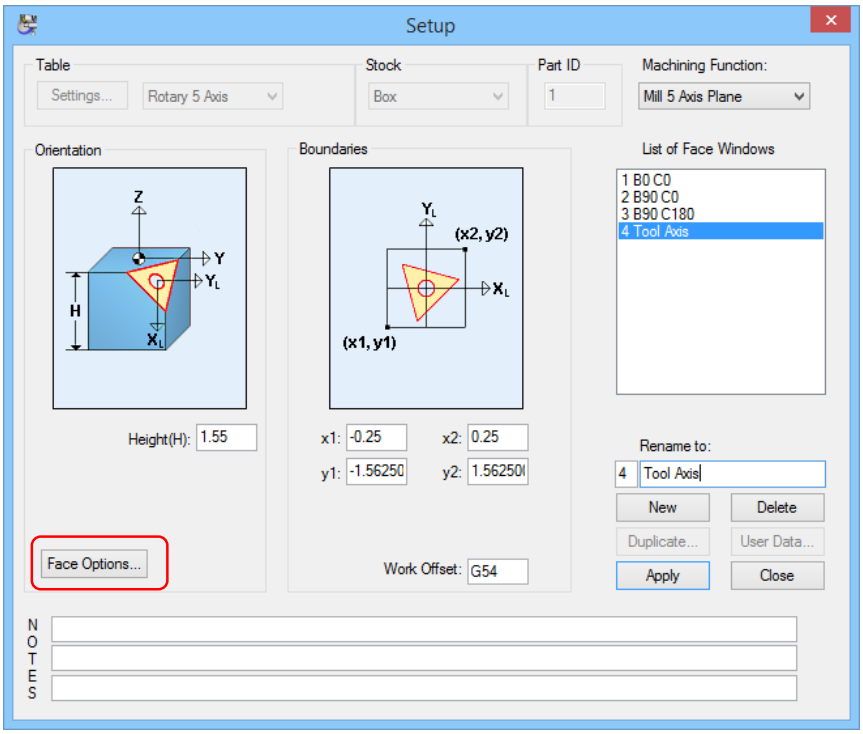
Fixed Angle: This fixes the tool axis; using **Azimuth** or **Elevation**, to a specified value after the orientation has been calculated using the orientation selected as the **Tool axis**. This fixes one rotational axis of a machine tool wherever possible, giving an improved surface finish and increases the overall feed rate by reducing acceleration and deceleration. The locked axis is overridden only to avoid a collision and to ensure the tool stays on the part.

Create a New Face Window for Surfacing on the part

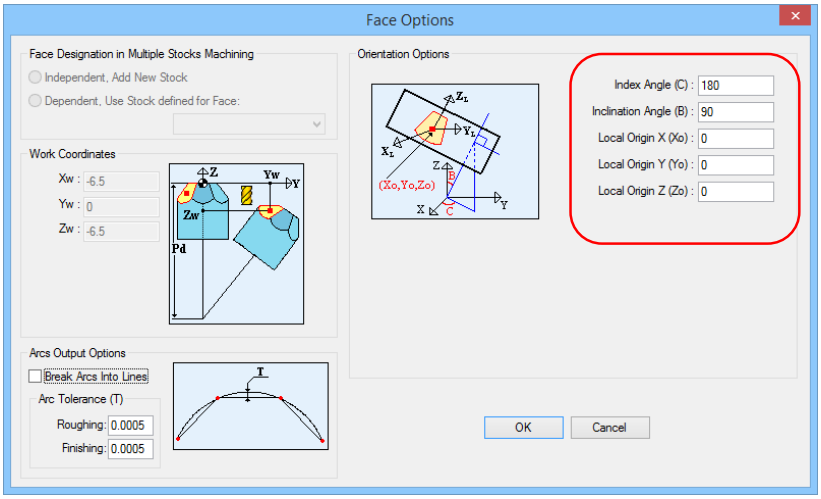
The first step in creating a surfaced feature in PartMaker Mill is to create a new Face Window. In this case, you will use the Mill 5-Axis Face Window:

1. Choose **Setup** from the **View** menu
2. In the **Setup** dialog, click the **<New>** button to create a new Face Window.
3. From the **Machining Function** drop down menu, choose **Mill 5-Axis Plane**.

4 In the **Rename To:** field, type "**Tool Axis**" and click **<Apply>**. Your completed **Setup** dialog should appear as shown below:



5 Select the <**Face Options>** button. Set the orientation as shown below:



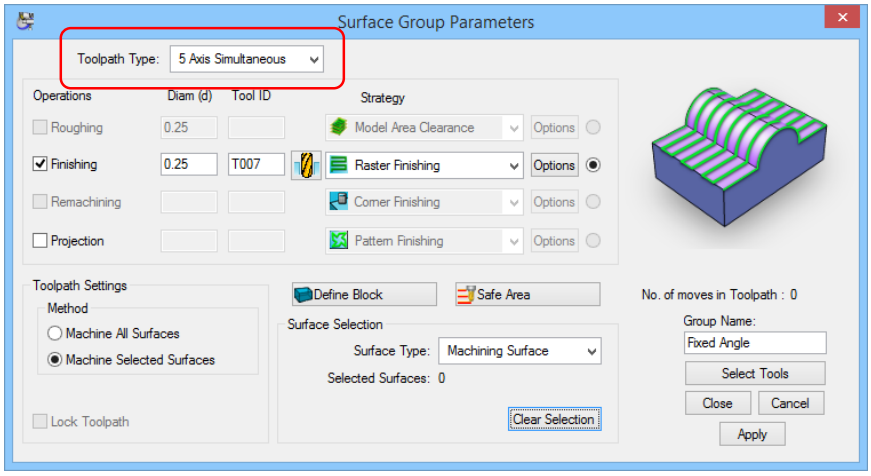
6 Select <**OK**> to close the Face Options.

7 Click the **<Close>** button to close the **Setup** dialog.

Create a New Surface Group for Tool Axis control Finishing Machining

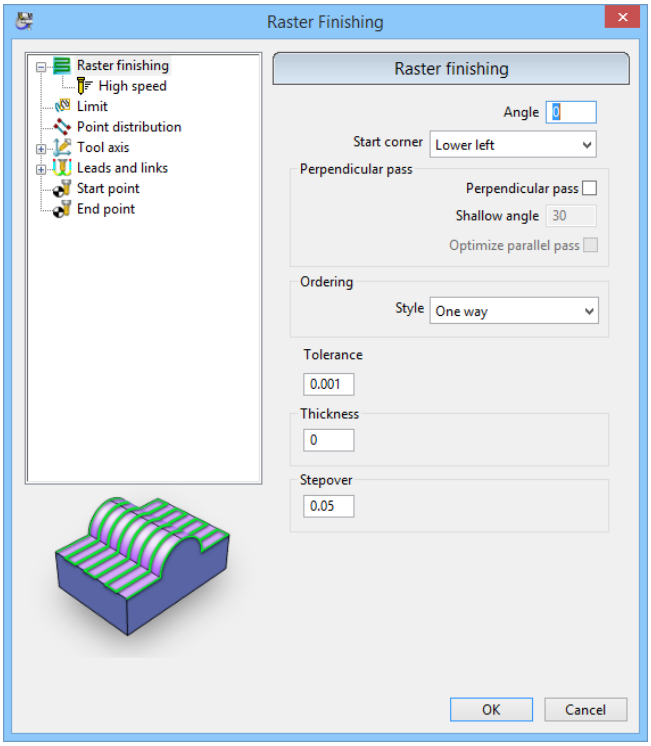
In this section of the tutorial you will create and verify a tool path using Tool axis control Finishing on the imported solid model.

1. Choose New Surface Group from the Part Features menu. The Surface Group Parameters dialog will display.
2. Select the **5 Axis Simultaneous** option from the **Toolpath Type** drop down menu.
3. Select the **Raster Finishing** strategy. (Tool Axis control is also available in 3D Offset, Constant Z, and Spine Finishing)
4. Enter a tool diameter of 0.250 in (6.35 mm) in the **Diam (d)** field and click the **<Select Tools>** button.
5. In the **Group Name** field enter "**Fixed Angle**" and click **<Apply>**. The dialog should now appear as shown below:



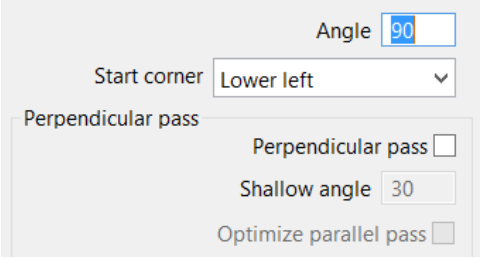
Setting Strategy Options for Raster Finishing

Click the <**Options**> button in the **Surface Group Parameters** dialog which will open the dialog for the Raster Finishing strategy as shown below:

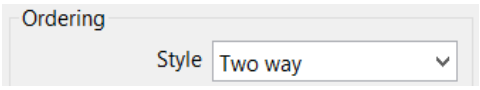


General Raster Finishing settings

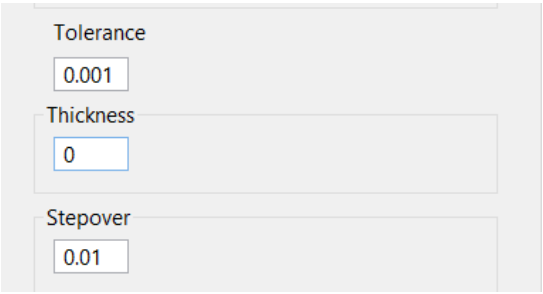
1. In the Raster finishing dialog, on the main page set the Start corner and the Angle as shown below:



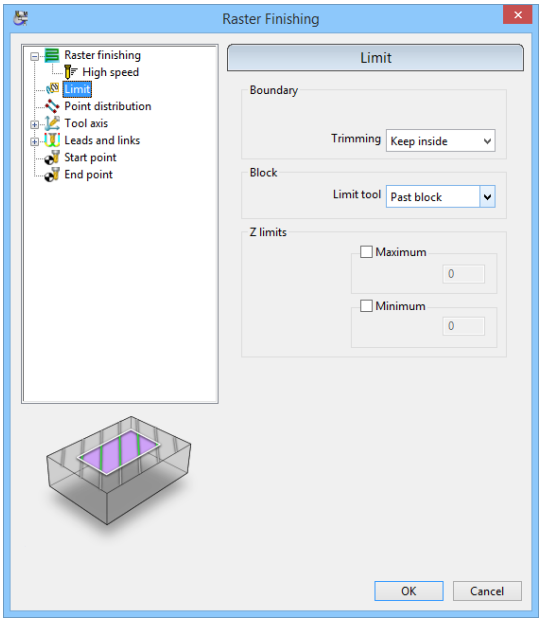
2. Set the Ordering Style to Two way as shown below:



3. In the Raster finishing dialog, on the main page enter the Tolerance value of 0.001 (0.0254 for metric) as shown below:
4. Set Thickness to zero. This is the amount of material to be left behind.
5. Set the Step over amount to .010. This is the distance between successive machining passes.



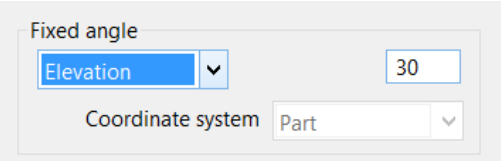
6. Select the **Limit** tab in the tree on the left in the dialog.
7. Set Boundary to Keep inside.
8. Set Block Limit Tool to Past Block.



9 Select **Tool axis** tab in the tree on the left in the dialog.

10 Set the Tool axis to Vertical.

11 Set the Fixed angle to Elevation 30:

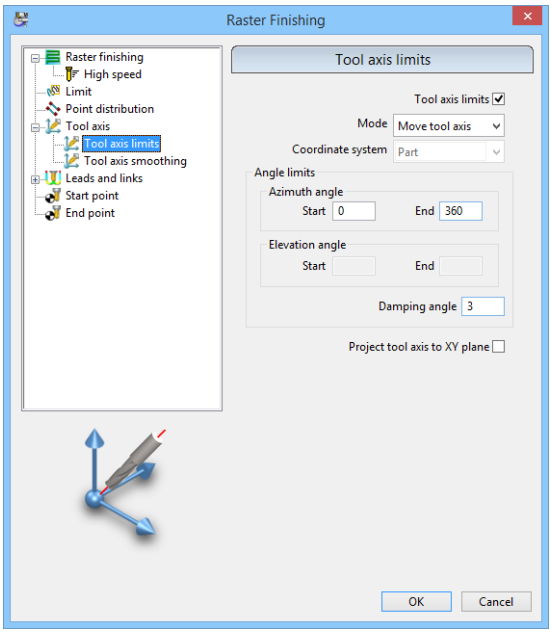


Elevation angle: is the angle from the Tool Axis projection onto the XY plane toward the positive Z-axis. The elevation angle is between -90 and 90 degrees.

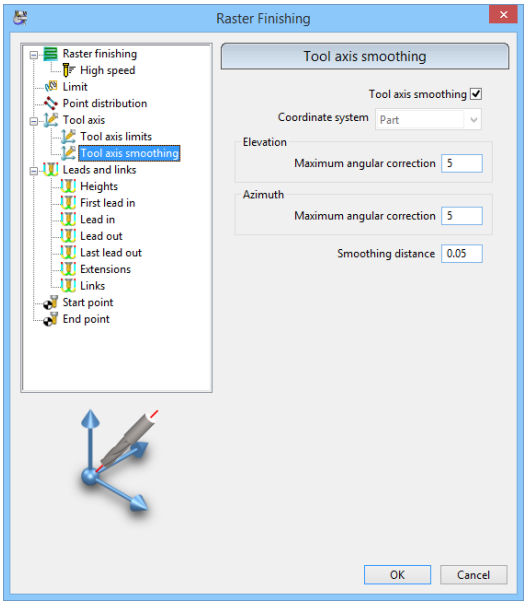
Azimuth angle: is the angle from the positive X axis toward the positive Y axis, to the Tool Axis vector's orthogonal projection onto the XY plane. The azimuth angle is between 0 and 360 degrees.

12 Select **Tool axis limits**: Set the Azimuth and Elevation angles. Elevation is the B axis and Azimuth is the C axis.

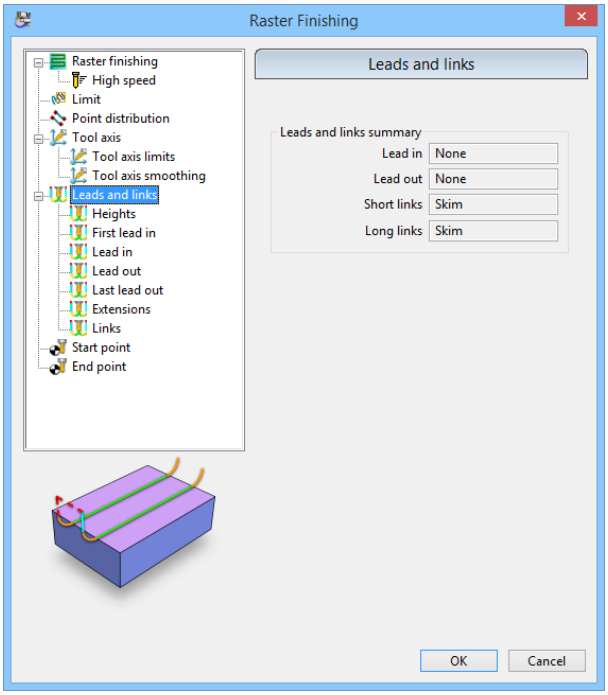
Azimuth angle: Enter **Start** and **End** angles to define the angular limits of the machine tool in the XY plane. 0 is along the X axis; 90 is along the Y axis.



13 Turn on **Tool axis smoothing**: Enter the distance over which to smooth the tool axis movement. With sudden changes in direction in al toolpath (such as a right-angled corner), rapid changes of orientation of the tool axis leaves dwell marks. To prevent this, the **Smoothing distance** blends the change in orientation to produce a much improved surface finish.

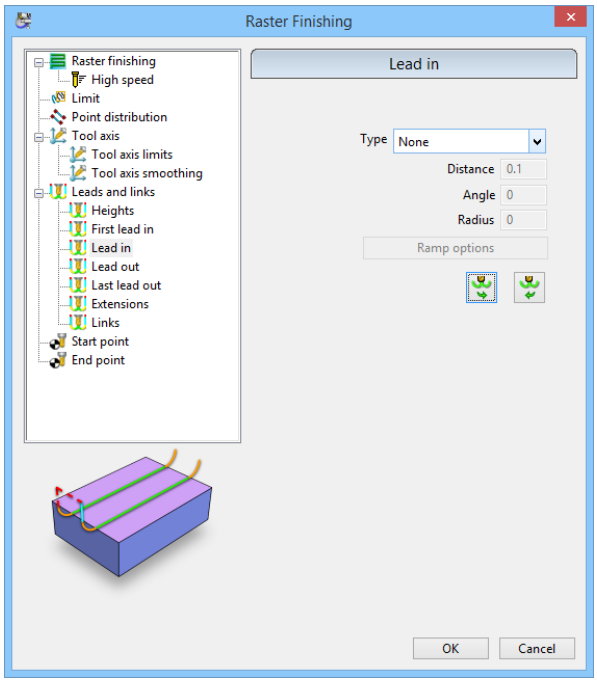


14 Click the **Leads and Links** tab in the tree on the left in the dialog to expand the **Lead and Links** features.



15 Click on **Lead in** on the tree to see the **lead-in** options.

16 Set the **Lead in** Type to None.



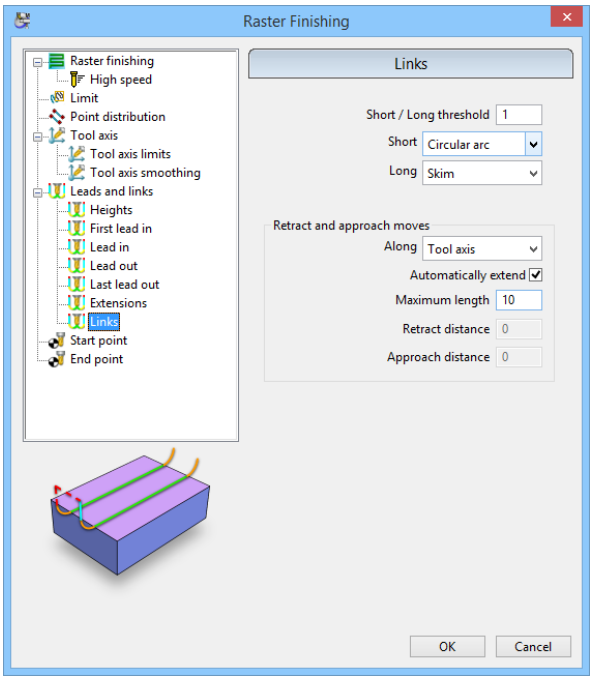
17 Click on  to copy the same parameters from **Lead In** to **Lead Out.**

18 Click on **Links** on the tree to see the Links options.

19 Set the **Short** Link to **Circular arc:** This will make the tool moves circular.

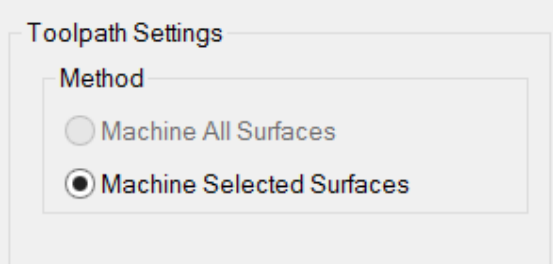
20 Set the **Long** link to **Skim.** This will move the tool clear of the part by the specified incremental distance, descend at rapid feed to the specified incremental distance above the contact point, and then plunge the remaining distance.

21 Set the **Retract and Approach** moves to **Tool Axis.** This will make the retract and approach moves have the same orientation as the tool axis.

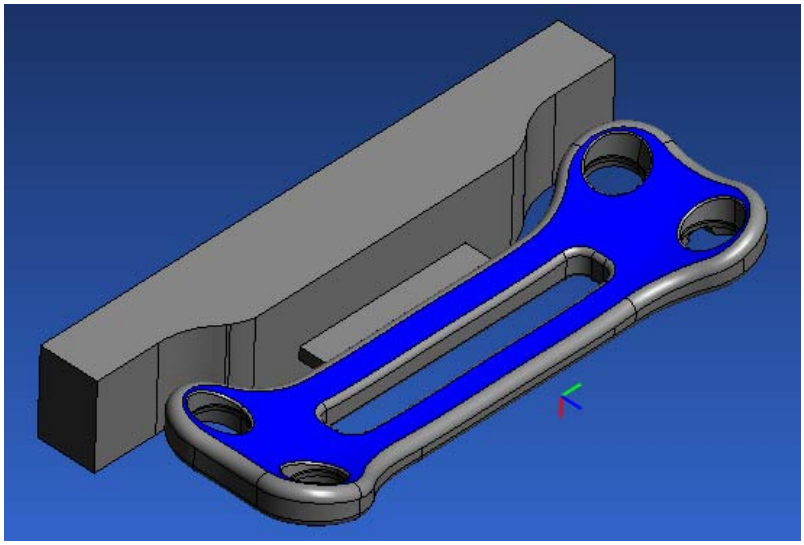


Select the Surfaces for Machining

At this point you should still have the **Surface Group Parameters** dialog open. In the lower left hand corner of this dialog there is a framed section called **Toolpath Settings**. In this section you will control how you select surfaces to be machined by the current operation. There are two choices: **Machine All Surfaces** and **Machine Selected Surfaces**. By default **Machine All Surfaces** will be selected which will select all surfaces of the imported solid model as Machining Surfaces. In this exercise you will select the surfaces to be machined.



1 In the **Solid Model File** window, click on the surfaces on the solid model as indicated in the picture below:



2 Click the **<Close>** button to return to the CAM Face Window.

Verify the Tool Path

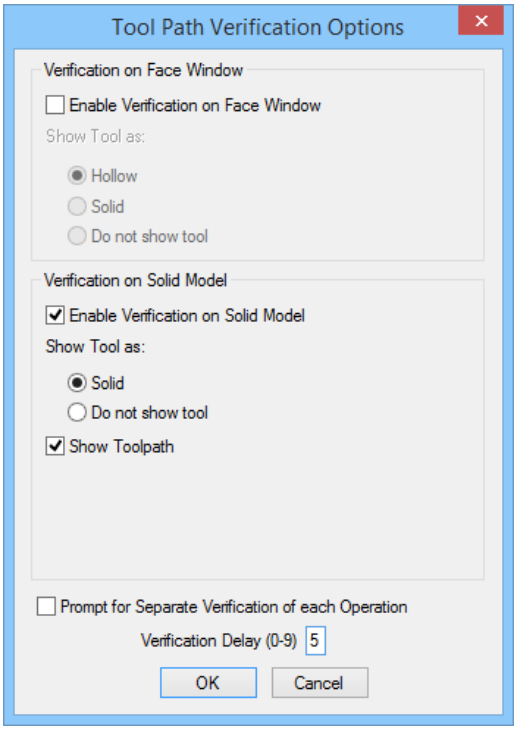
Once you have created the **surface group** you can visually verify the tool path PartMaker has calculated. To do so:

1 From the Part Features menu choose Verify Work Group Tool Path.

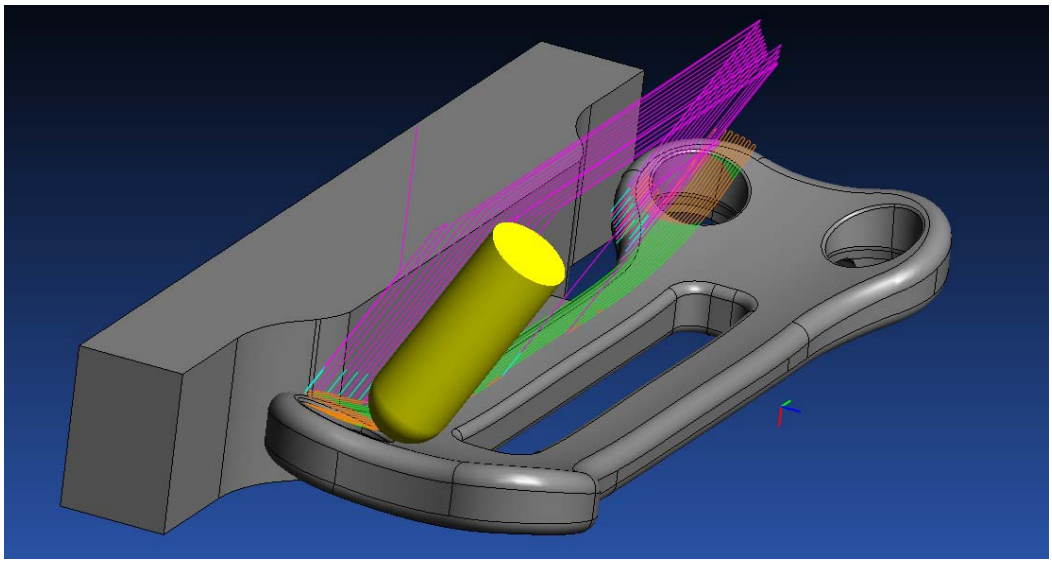
2 In the **Tool Path Verification Options** click the **Hollow** radio button as shown below.

3 Enter a Verification Delay of 5.

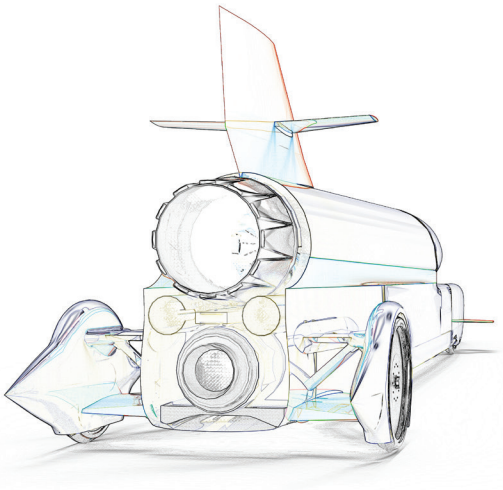
Note: *When entering a* ***Verification Delay*** *of greater than 0, you will see the 3D representation of the cutting tool moving along the Solid Model. When* ***Verification Delay*** *is set to 0, you will only see the path of the tool on the Solid Model.*



4 Click <OK> to show the calculated tool path. Your screen should appear as shown below:



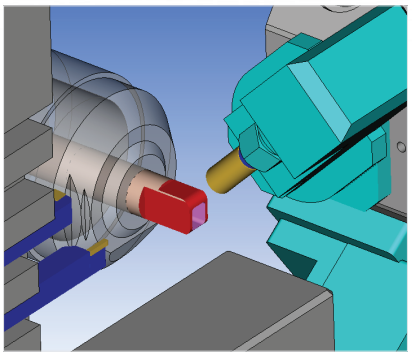
5 From the Part Features menu choose Hide Work Group Tool Path.



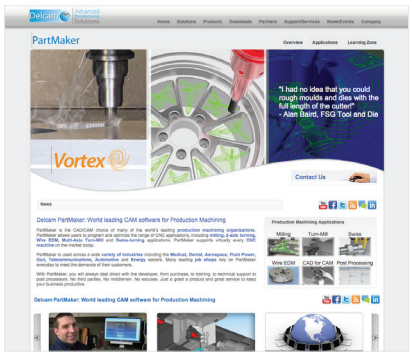
PartMaker 2015

World-leading CAM software for production machining

www.partmaker.com



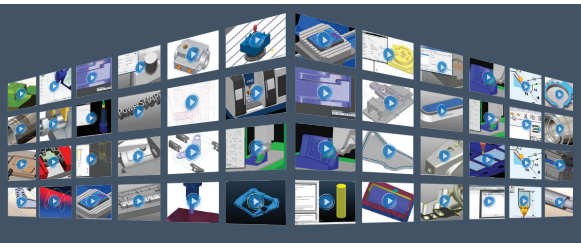
PartMaker 2015



www.partmaker.com

www.delcam.tv

Unlock the power of Delcam software



LEARNING ZONE

Visit **www.delcam.tv/lz** and see how Delcam software could benefit your business!

Delcam Advanced Manufacturing Solutions

Powering your productivity

Delcam
Small Heath Business Park, Birmingham, B10 0HJ, United Kingdom
T: +44 (0) 121 766 5544 | E: marketing@delcam.com
W: www.delcam.com

PartMaker Inc.
550 Pinetown Road, Suite 470, Ft. Washington PA 19034 USA
T: +215 643 5077 | E: info@partmaker.com
W: www.partmaker.com

To contact your local reseller, visit **www.delcam.com/resellers**

AUTODESK.