Mastercam®

ADVANCED MULTIAXIS

TRAINING TUTORIAL SERIES

X6

Mastercam X6

Advanced Multiaxis Training Tutorial

Mastercam X^6 Training Tutorials - Advanced Multiaxis

Software: Mastercam X^6

Author: Mariana Lendel

ISBN: 978-1-926955-36-0

Date: February 14, 2012

Notice

In-House Solutions Inc. reserves the right to make improvements to this manual at any time and without notice.

Disclaimer Of All Warranties And Liability

In-House Solutions Inc. makes no warranties, either express or implied, with respect to this manual or with respect to the software described in this manual, its quality, performance, merchantability, or fitness for any particular purpose. In-House Solutions Inc. manual is sold or licensed "as is." The entire risk as to its quality and performance is with the buyer. Should the manual prove defective following its purchase, the buyer (and not In-House Solutions Inc., its distributer, or its retailer) assumes the entire cost of all necessary servicing, repair, of correction and any incidental or consequential damages. In no event will In-House Solutions Inc. be liable for direct, indirect, or consequential damages resulting from any defect in the manual, even if In-House Solutions Inc. has been advised of the possibility of such damages. Some jurisdictions do not allow the exclusion or limitation of implied warranties or liability for incidental or consequential damages, so the above limitation or exclusion may not apply to you.

Copyrights

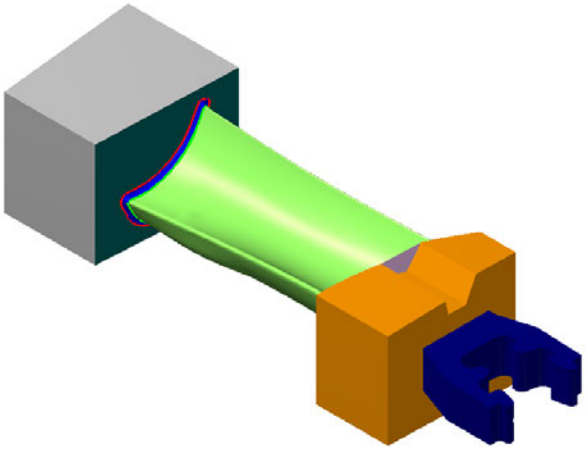
Trademarks

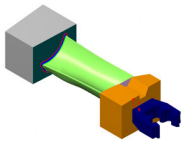
Mastercam is a registered trademark of CNC Software, Inc.

Microsoft, the Microsoft logo, MS, and MS-DOS are registered trademarks of Microsoft Corporation; N-See is a registered trademark of Microcompatibles, Inc.; Windows, Windows XP, Windows Vista, Windows NT and Windows 7 are registered trademarks of Microsoft Corporation.

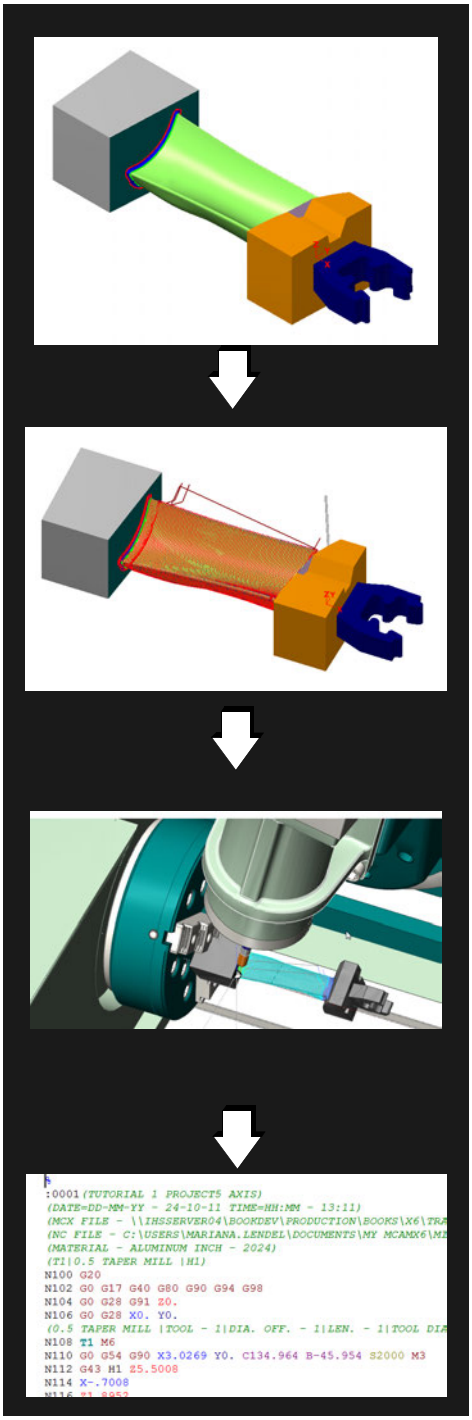
Advanced Multiaxis Training Tutorial
Contents

TUTORIAL #3





OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:



Open the CAD Model:

- The student will download the file that contains the geometry from emastercam.com.

Create the necessary Toolpath to machine the part:

- The student will set up the stock size to be used and the appropriate tool settings.
- A Parallel cuts toolpath will be created to finish the blade surfaces.
- A Morph between 2 curves toolpath will be created for a better finish of the blade surfaces.
- Two Parallel to multiple curves toolpaths will be created to machine the ends of the blade.

Backplot and Machine simulation:

- The Backplot will be used to simulate a step by step process of the tool's movements.
- Machine simulation will be used to check for any collisions between the tool and the part or any other machine components.

Post Process the file to generate the G-code:

- The Student will then post process the file to obtain an NC file containing the necessary code for the machine.

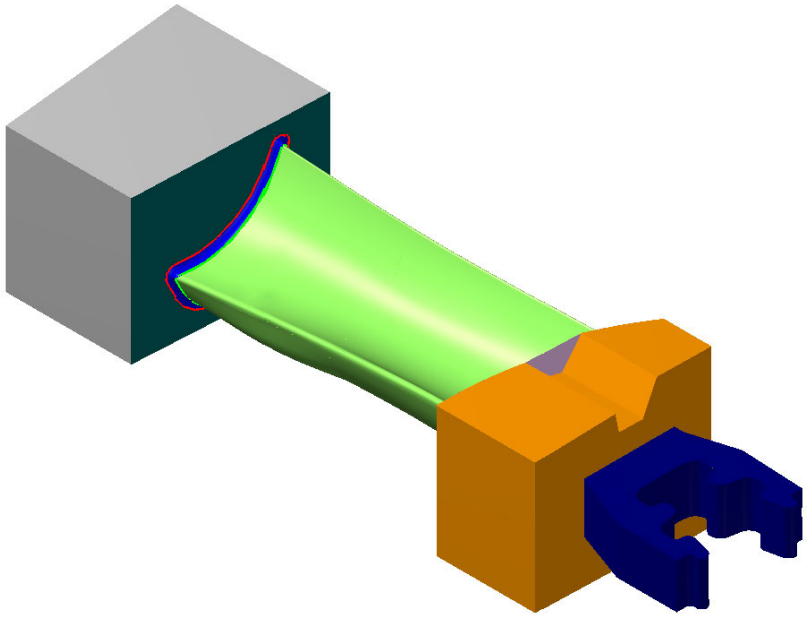
This tutorial takes approximately one hour and half to complete.

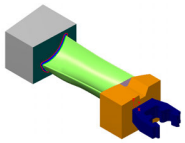
This tutorial covers Mastercam's 5-Axis toolpath functionality. To apply these toolpaths to a 5-Axis machine tool, a customized post processor for your machine is required.
There are no default 5-Axis post processors included with Mastercam due to the variation in multi-axis machine configurations. Contact your Mastercam reseller to request post processor services such as multi-axis post development.
Your post processor may require certain additional programming information not covered in this tutorial. For instance:
You may be required to position your stock in machine space rather than at Mastercam's origin.
You may be required to use the Misc Values.
Your post may also prompt you for tool gage lengths.
The nature of the additional information required depends largely on your machine's configuration. Contact the developer of your post processor for details.

STEP 1: SELECT THE GEOMETRY FILE

Resources - Download the file from www.emastercam.com/files/.
File

- **Open.**
- Select "TUTORIAL 3 FINISH BLADE SURFACES.MCX-6" from the directory you saved it in.
- Use the **Fit** icon to fit the drawing to the screen.





STEP 2: SELECT THE MACHINE AND SET UP THE STOCK

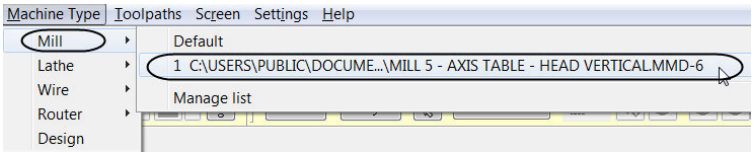
In Mastercam, you select a **Machine Definition** before creating any toolpaths. The **Machine Definition** is a model of your machine tool's capabilities and features and acts like a template for setting up machining jobs. The machine definition ties together three main components: the schematic model of your machine tool's components, the control definition that models your control unit's capabilities and the post processor that will generate the required machine code (G-code). For the purpose of this tutorial, we will be using the 5 - AXIS TABLE - HEAD VERTICAL machine.

If you already have the default machine in the Toolpaths Operations Manager, do not select another machine. Expand **Properties** and select **Files**, and then select **Replace** button and select the Mill 5 - AXIS TABLE - HEAD VERTICAL.MMD-6.

2.1 Select the Machine type

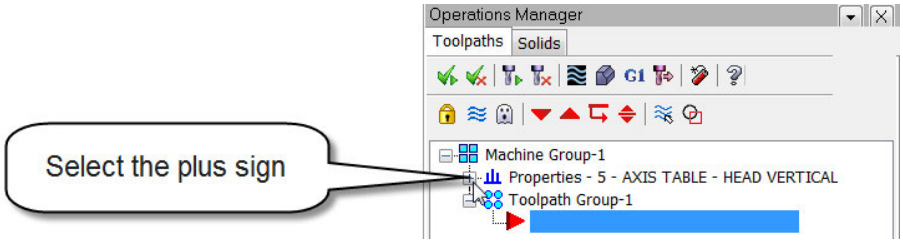
Machine type

- **Mill.**
- Select the **MILL 5 - AXIS TABLE - HEAD VERTICAL.MMD-6**

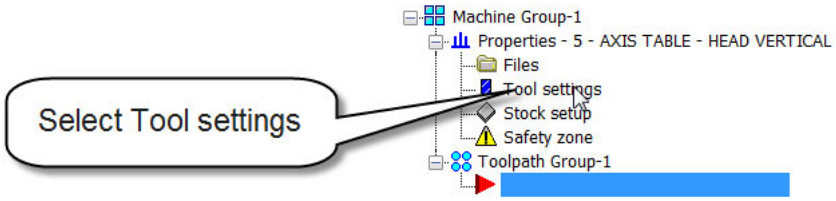


2.2 Set the tool parameters in the Tool settings

- Select the plus sign in front of **Properties** in the **Toolpaths Manager** to expand the **Toolpaths Group Properties.**

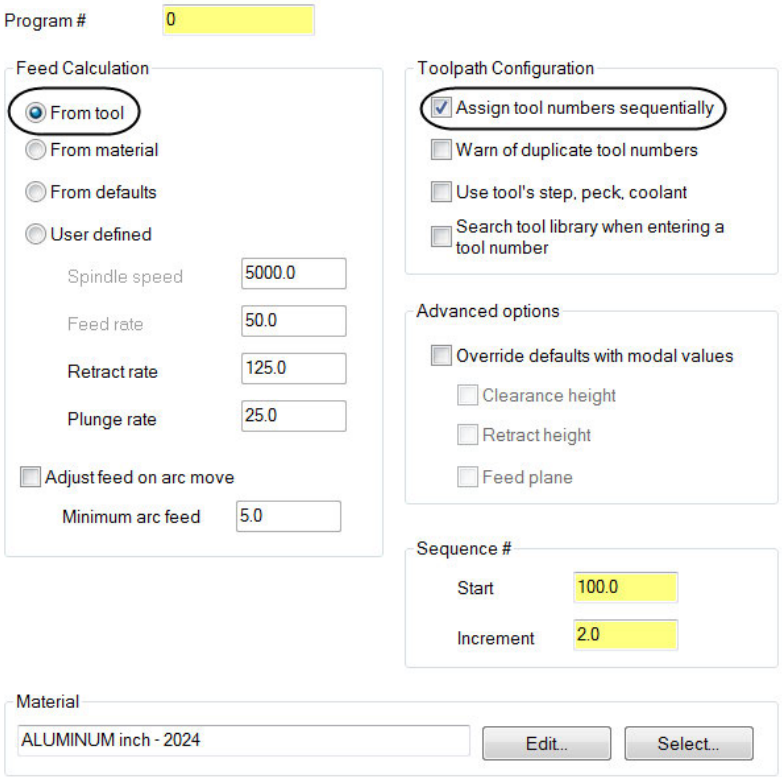


- Select the **Tool settings** to set the tool parameters.



- Change the parameters to match the Figure: 2.2.1.

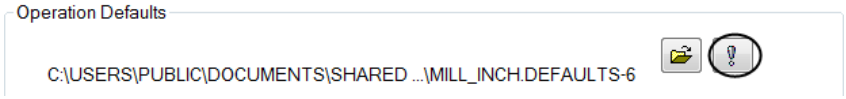
Figure: 2.2.1



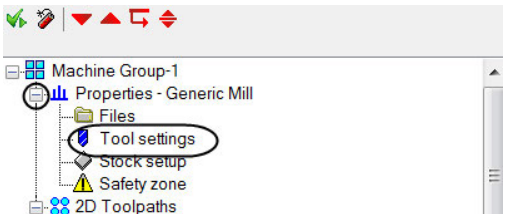
- Select the **OK** button to exit.

2.3 Make the new Tool settings parameters your defaults

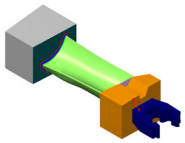
- To make these parameters in the **Tool Settings** your default parameters, from the **Machine Group Properties** select **Files** tab.
- Click the **Options** button inside of the **Operation Defaults**.



- From the **Edit Operations Defaults**, scroll above the operations list and expand **Properties**, and then select **Tool settings.**



- Enable **Assign tool numbers sequentially** as shown in Figure: 2.2.1.
- Select the **OK** button to exit **Machine Group Properties**.



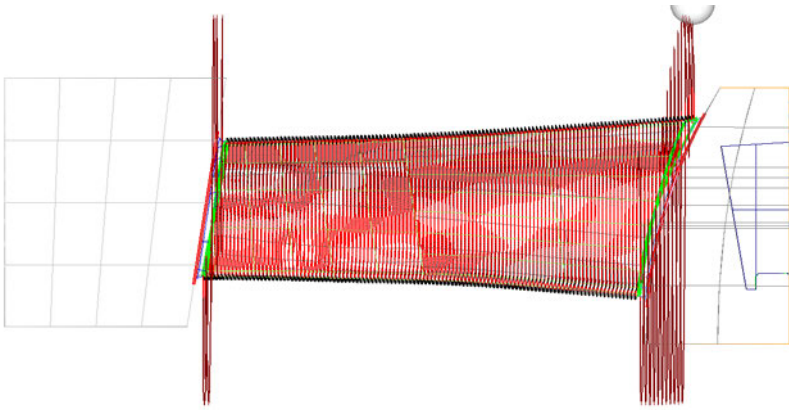
- Select the **OK** button to exit **Edit Operations Defaults**. 
- Select the **OK** button to exit **Machine Group Properties**. 

NOTE: When starting a new file or starting a new session of Mastercam, the parameters that you set in the **Edit Operations Defaults** will be automatically provided. You can also use this dialog box to set other machine group defaults, such as a stock material or to set the defaults for all the toolpaths provided in the operations list.

STEP 3: FINISH THE BLADE USING PARALLEL CUTS TOOLPATH

The **Parallel cuts toolpath** will create tool motion with cuts that are parallel to each other. The direction of the cuts is defined by the two angles: the Machining angle in X, Y and the Machining angle in Z. The toolpath is located in the **Surface/Solid toolpath family.**
Use **Parallel cuts** to finish the blade surfaces.

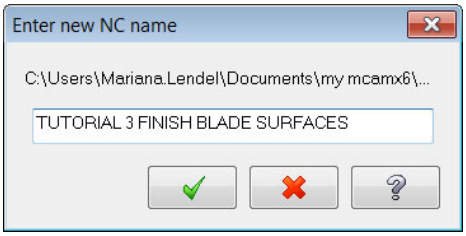
Toolpath Preview:



Create the Parallel cuts toolpath from the Surface/Solid toolpath family using the following parameters:
1/2" Ball Endmill.
Machining angle in X, Y = **0.0.**
Machining angle in Z = **90.0.**
Area type = Full avoid cuts at exact edges.
Cutting method set to One way; Counterclockwise.
Cut Tolerance = 0.0005".
Maximum stepover = **0.05".**
Enable Enforce cutting direction (assume closed contours).
Output format set to 4 Axis; Maximum angle step = 3 degrees.
Lead angle to cutting direction = 0.0.
Tool axis will = Surface with lead/lag.
Tilt angle at side of cutting direction = **0.0.**
Side tilt definition set to **Ortho to cut dir at each pos.**

Toolpaths

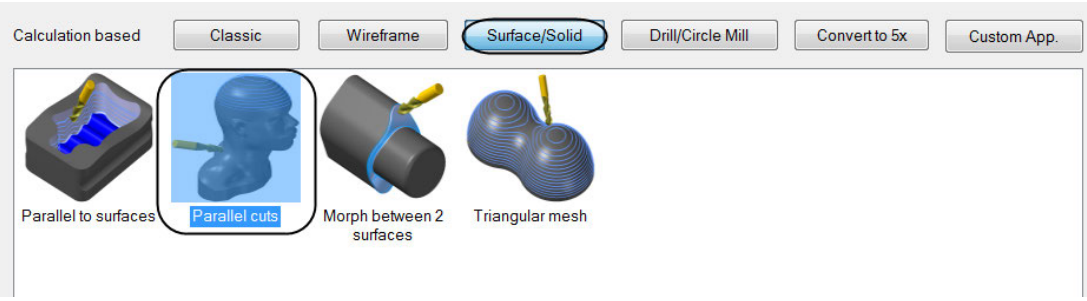
- **Multiaxis.**
- Select the **OK** button to accept the **NC name**.

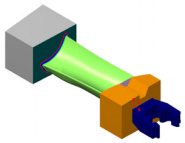


Toolpath Type

Toolpath Type page allows you to establish the type of multiaxis toolpath to create. Toolpath type controls the options available as you proceed down the tree structure. The options are based on the toolpath family selected using the Calculation based on buttons, as well as by selecting the picture of the toolpath you wish to create. The **Toolpath Type** dialog box allows you to select between the different toolpath families Classic, Wireframe, Surface/Solids, Drill/Circle Mill, Convert to 5x or Custom applications. You can then choose one of the toolpaths inside of that family.

- Select **Surface/Solids** button and click on **Parallel cuts.**

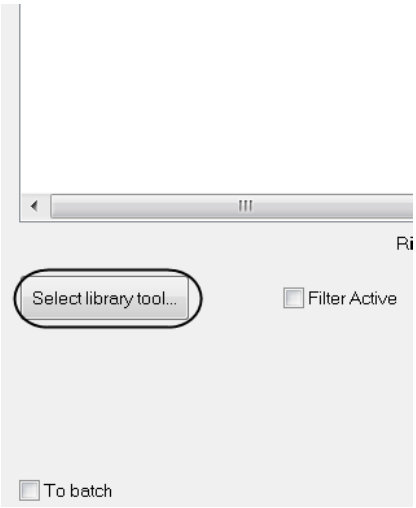




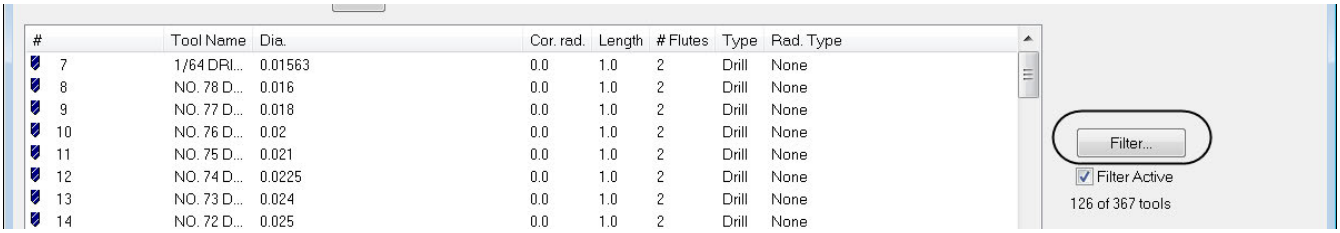
Tool

The **Tool** page allows you to select a tool, set the feeds and speeds, enter a comment about the operation, and set other general toolpath parameters.

- From the **Tree View** list, select **Tool**.
- To select the tool, click on the **Select library tool** button.



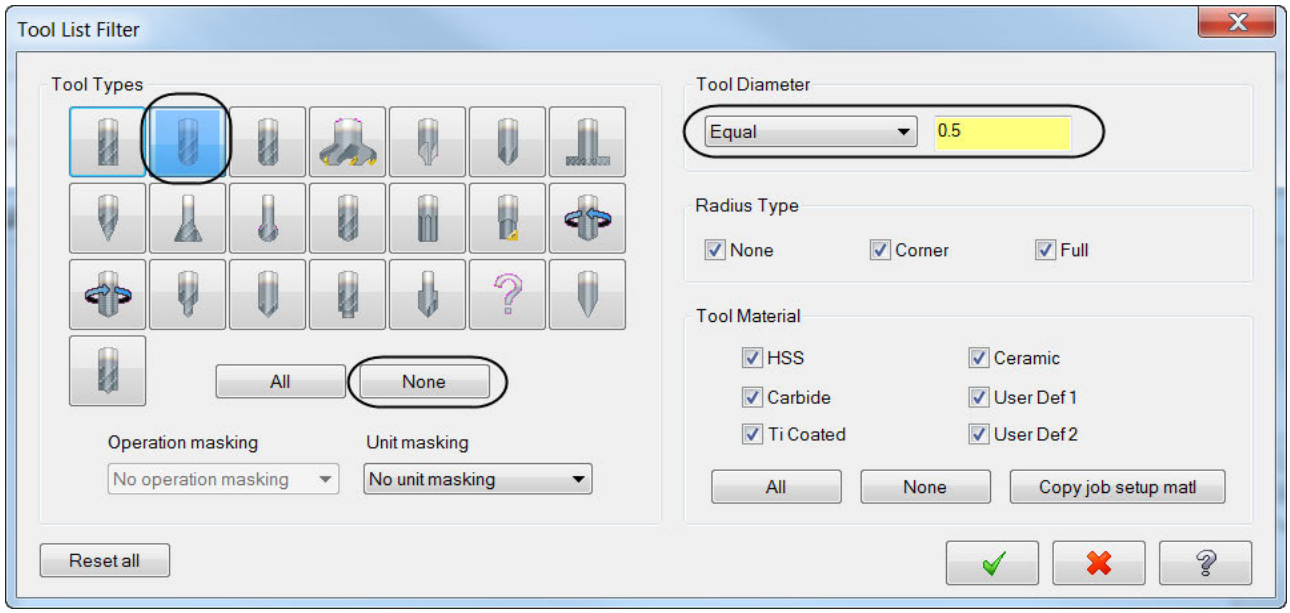
- Select the **Filter** button.



- Select the **None** button to unselect any previous tool selection as shown.

- Select the **Endmill Sphere** in the **Tool Types** list.
- Select the drop down arrow in the **Tool Diameter** field and select **Equal**.
- Enter **1/2** in the **Tool Diameter** value box as shown in Figure: 3.0.1.

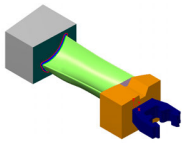
Figure: 3.0.1



- Select **OK** button to exit **Tool List Filter**.
- Make sure that the tool in the **Tool Selection** window is highlighted, otherwise select it.
- Select the **OK** button to exit from the **Tool Selection** dialog box.

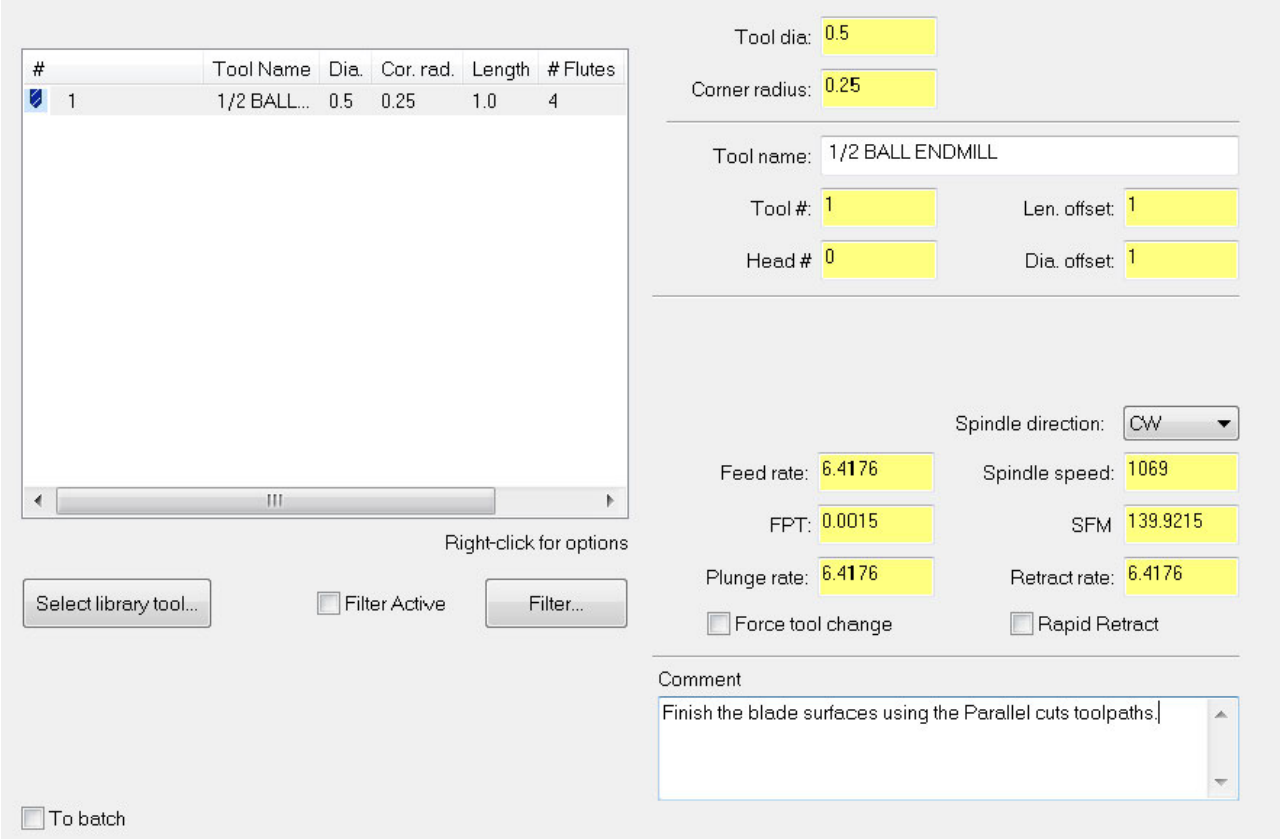
NOTE: The feeds and spindle speed are based on the tool definition. You can overwrite them with the feeds and speed that you want to use. You can also let Mastercam calculate the feeds and speed using the Feed speed calculator.

- Add a comment in the **Comment** area to identify the toopath in the **Toolpath Manager** and also in the NC file.



- The **Tool** page should look as shown in Figure: 3.0.2.

Figure: 3.0.2



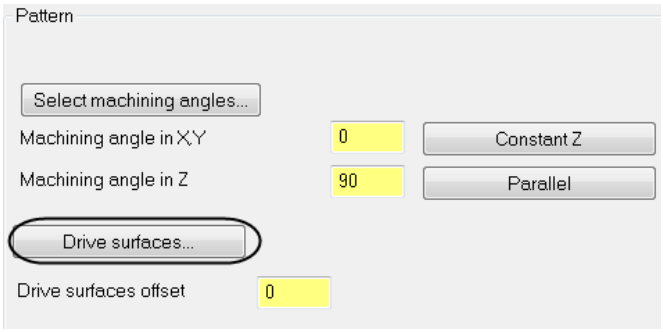
Cut Pattern

Cut pattern settings determine the geometry the tool follows and how it travels along that geometry.
The direction of the cuts is defined by the two angles: the **Machining angle in X, Y** and the **Machining angle in Z**. You can also choose the **Area type** which determines where the tool starts, ends, and area to be cut.
Cut Pattern page also allows you to set the **Cutting method** and the **direction** for one way machining. In the **Cut Pattern** you can set the **Cut tolerance** and the **Maximum stepover**.

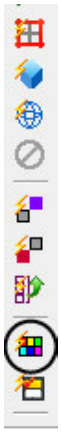
- From the **Tree View** list, select **Cut Pattern.**
- Set the **Machining angle in X, Y** to **0.0** and the **Machining angle in Z** to **90.0.**

NOTE: These settings will create parallel cuts to the Y-Axis of the current plane (top in our case). If the XY angle was set to 90.0 degrees, the resulting cuts are parallel to X-Axis.

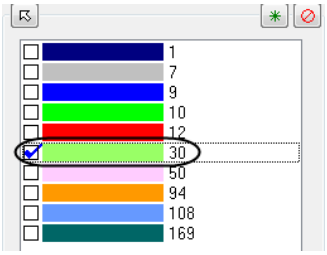
- Click on the **Drive surfaces** button to select the surfaces you want to machine as shown.



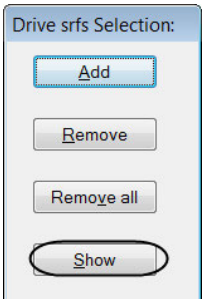
- [Select surfaces for machining]: From the **Quick Mask** toolbar, located to the right side of the graphics window, select the **QM color** icon.

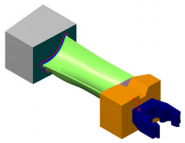


- Select color green no. **30** as shown.



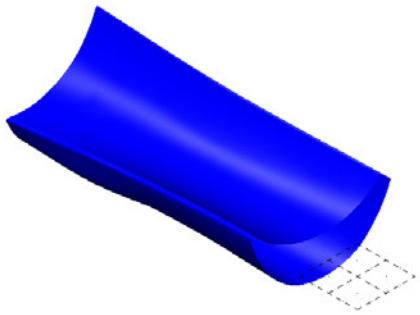
- Select the **OK** button to exit the **Select All** dialog box. 
- Press **Enter** to finish the selection.
- Click on the **Show** button to see the selected surfaces.



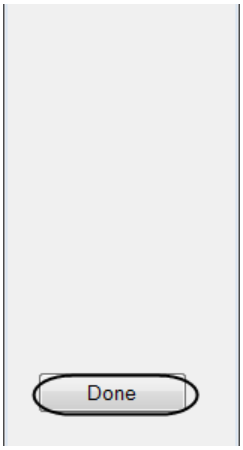


- The surfaces should look as shown in Figure: 3.0.3.

Figure: 3.0.3



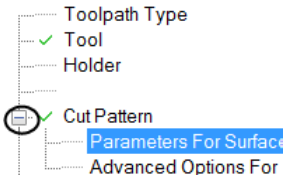
- Press **Enter** to continue.
- Select **Done** to exit **Drive srfs Selection** dialog box as shown.

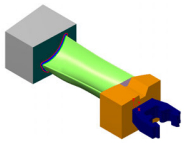


- The rest of the parameters are set as shown in Figure: 3.0.4.

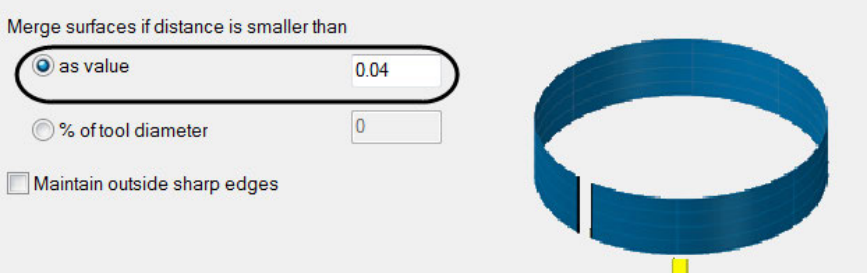
Figure: 3.0.4

- From the **Tree view** area, click on the plus sign in front of the **Cut Pattern** to expand it.





- Select **Parameters For Surface Edge Handling** and ensure that the **Merge surfaces if distance is smaller than** is set as a value to **0.04** as shown in the following picture.



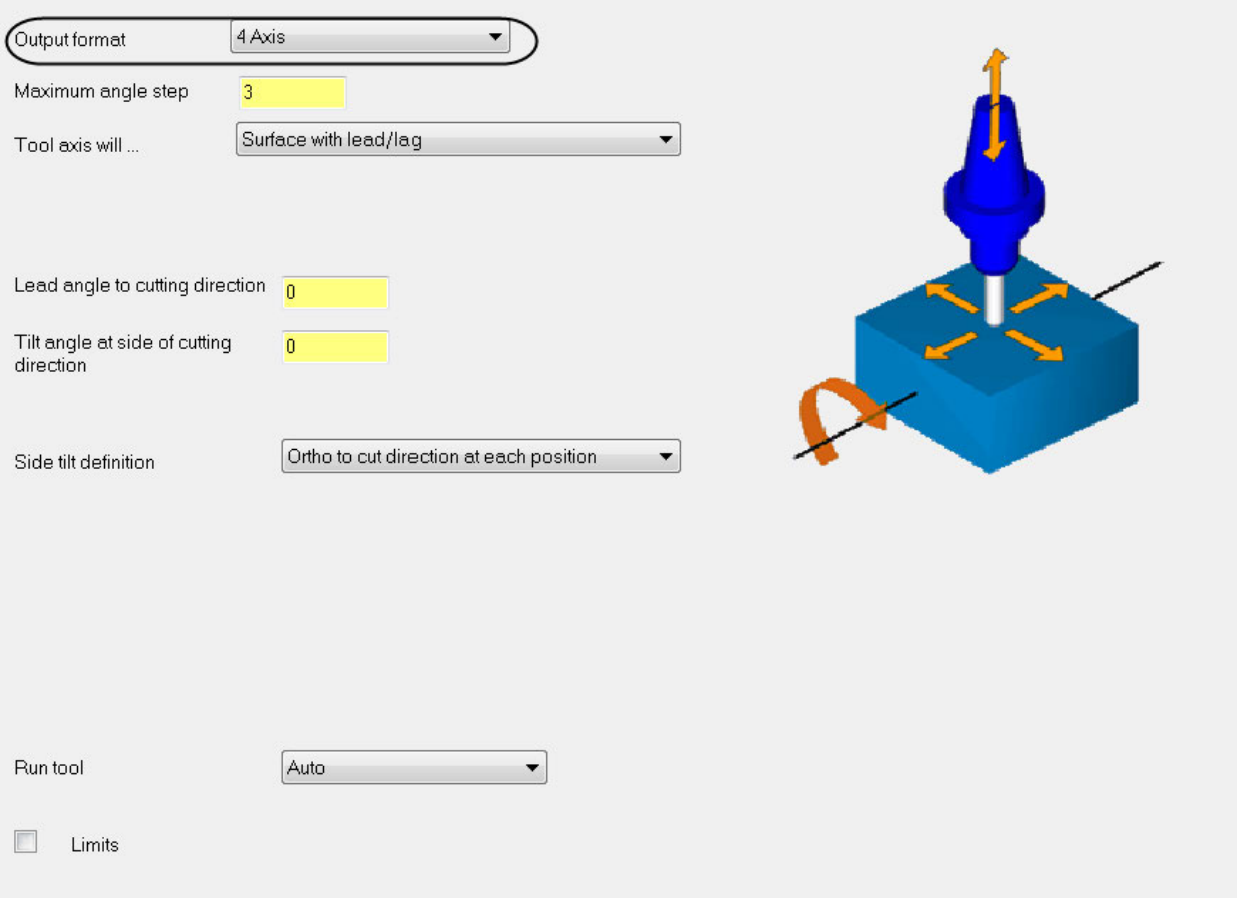
Tool Axis Control

Tool axis control settings determine the tool's orientation in relation to the geometry being cut. You can set the maximum angle the tool will be allowed to move through between adjacent moves.
You can also apply a lead/lag angle to the tool axis in the direction of the cut, measured relative to the surface normal.
The side tilt angle is applied to the tool axis in the direction of the cut, relative to the surface normal. A positive angle tilts the tool to the left; a negative angle tilts the tool to the right.

- From the **Tree View** area, select the **Tool Axis Control**.

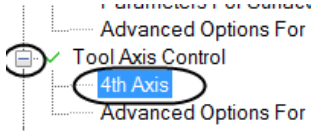
- Set the **Output format** to **4-Axis** and leave the rest of the parameters as shown Figure: 3.0.5.

Figure: 3.0.5

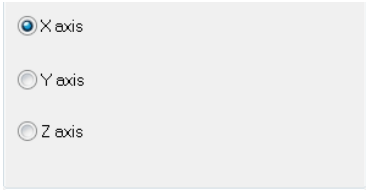


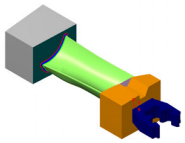
4th Axis

- From the **Tree View** area, click on the plus sign in front of the **Tool Axis Control** to expand it.
- Select **4th Axis**.



- Make sure that the axis is set to **X-Axis** as shown.

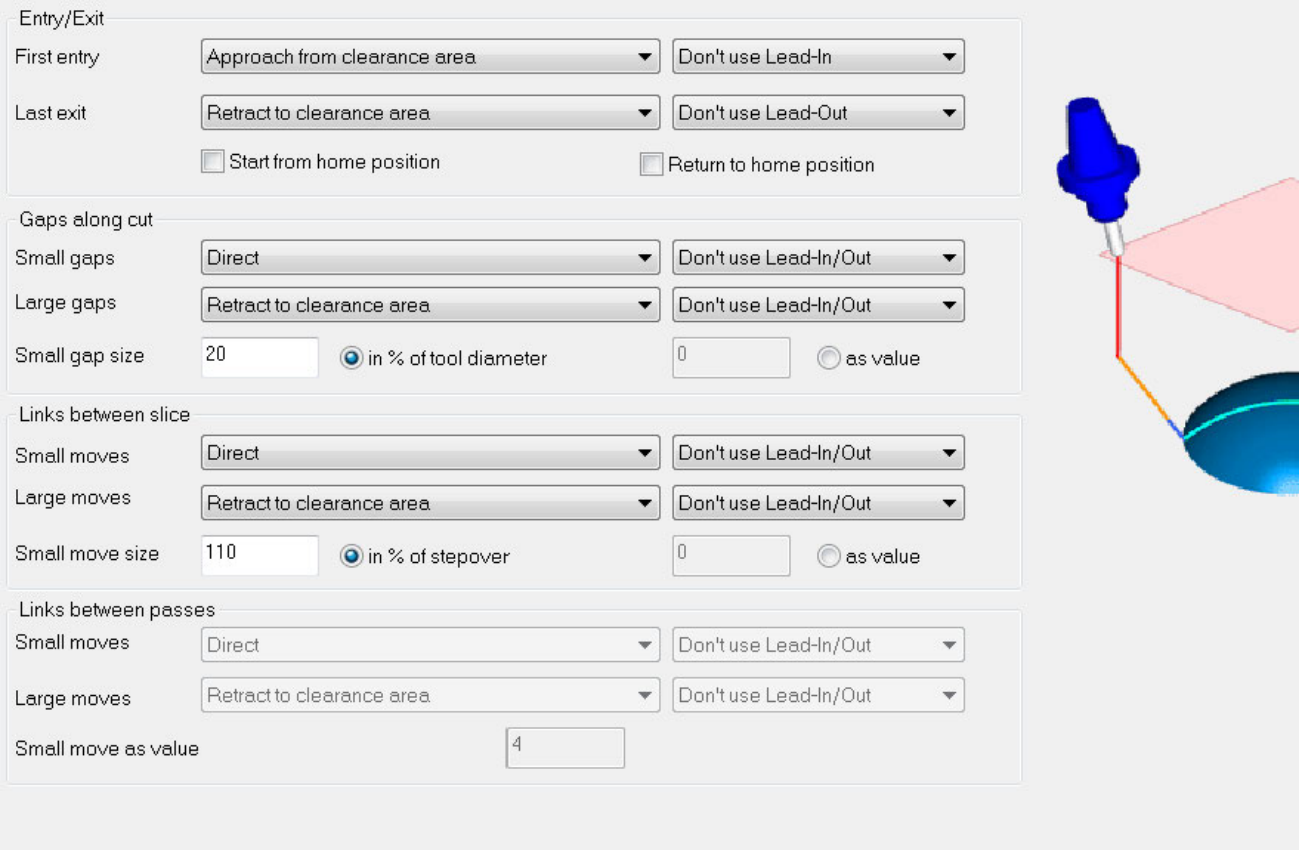




Linking

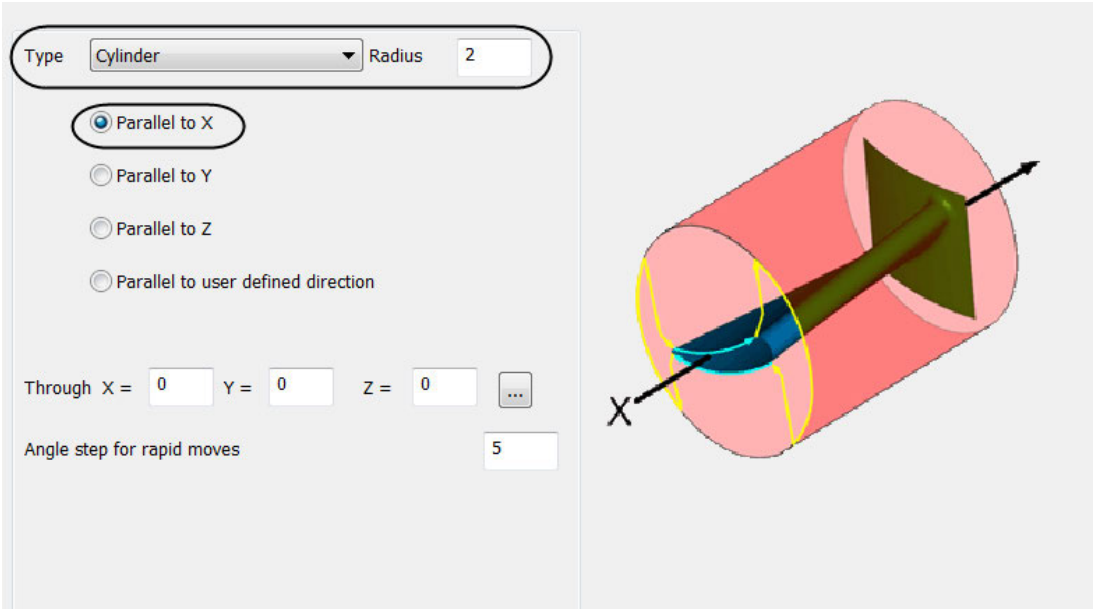
- From the **Tree View** area, select **Linking**.
- Leave the defaults as shown in Figure: 3.0.6.

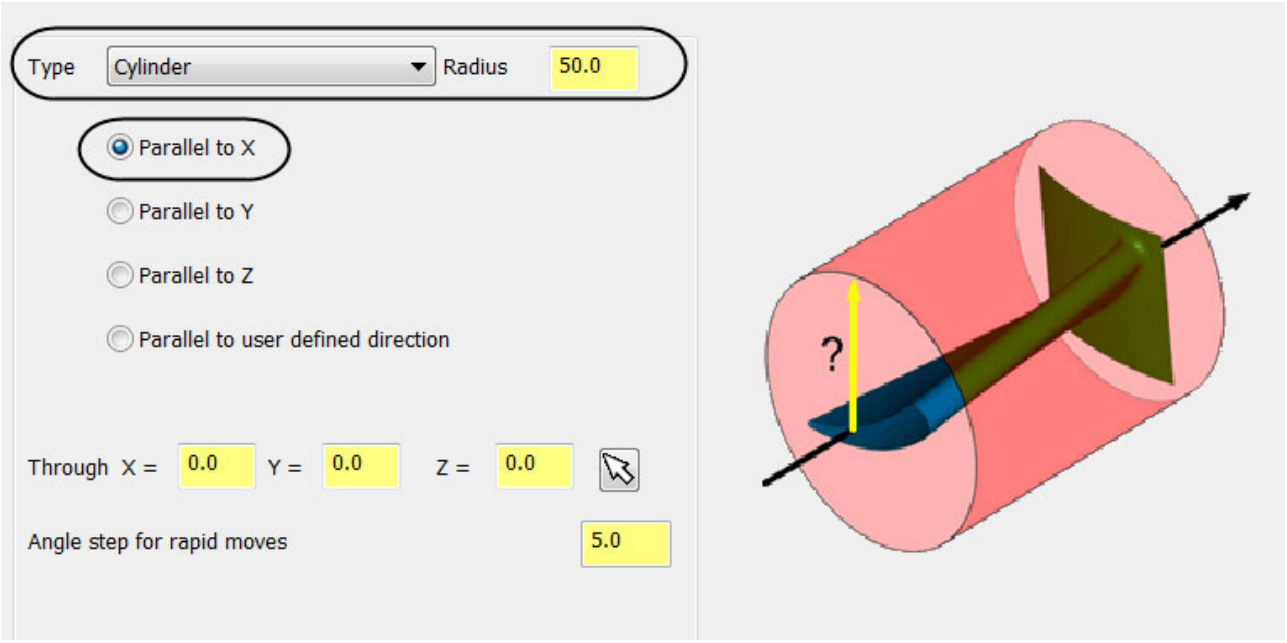
Figure: 3.0.6



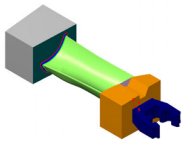
Clearance Area

- Click on the plus sign in front of **Linking** to expand the tree and select **Clearance Area**.
- Set the **Type** to **Cylinder** and **Radius** to **2.**
- Enable **Parallel to X.**





- Select the **OK** button to exit **Multiaxis Toolpath** - **Parallel cuts**.



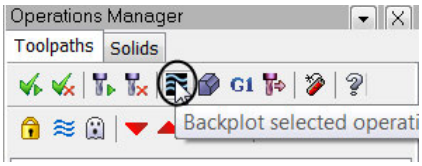
STEP 4: BACKPLOT THE TOOLPATH

Backplotting shows the path the tools take to cut the part. This display lets you spot errors in the program before you machine the part. As you backplot toolpaths, Mastercam displays the current X, Y, and Z coordinates in the lower left corner of the screen.

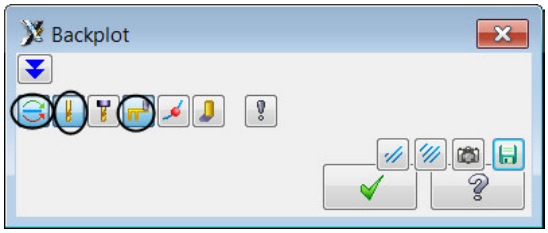
- Make sure that the toolpath is selected (signified by the green check mark on the folder icon).

Backplot the toolpath

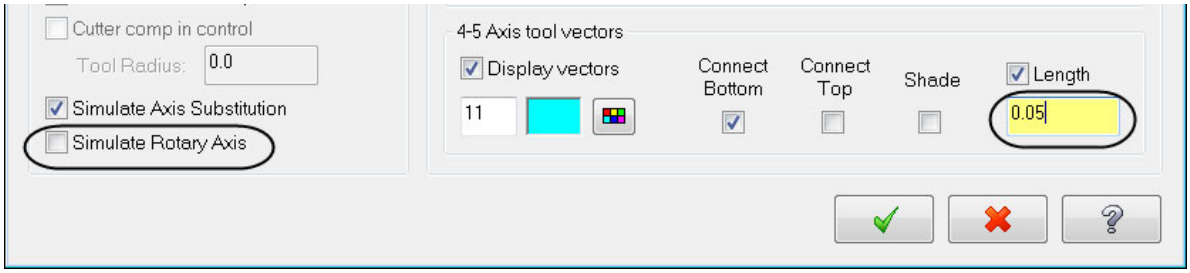
- Select the **Backplot selected operations** button.



- Make sure that you have the following buttons turned on to see the tool and the rapid moves.

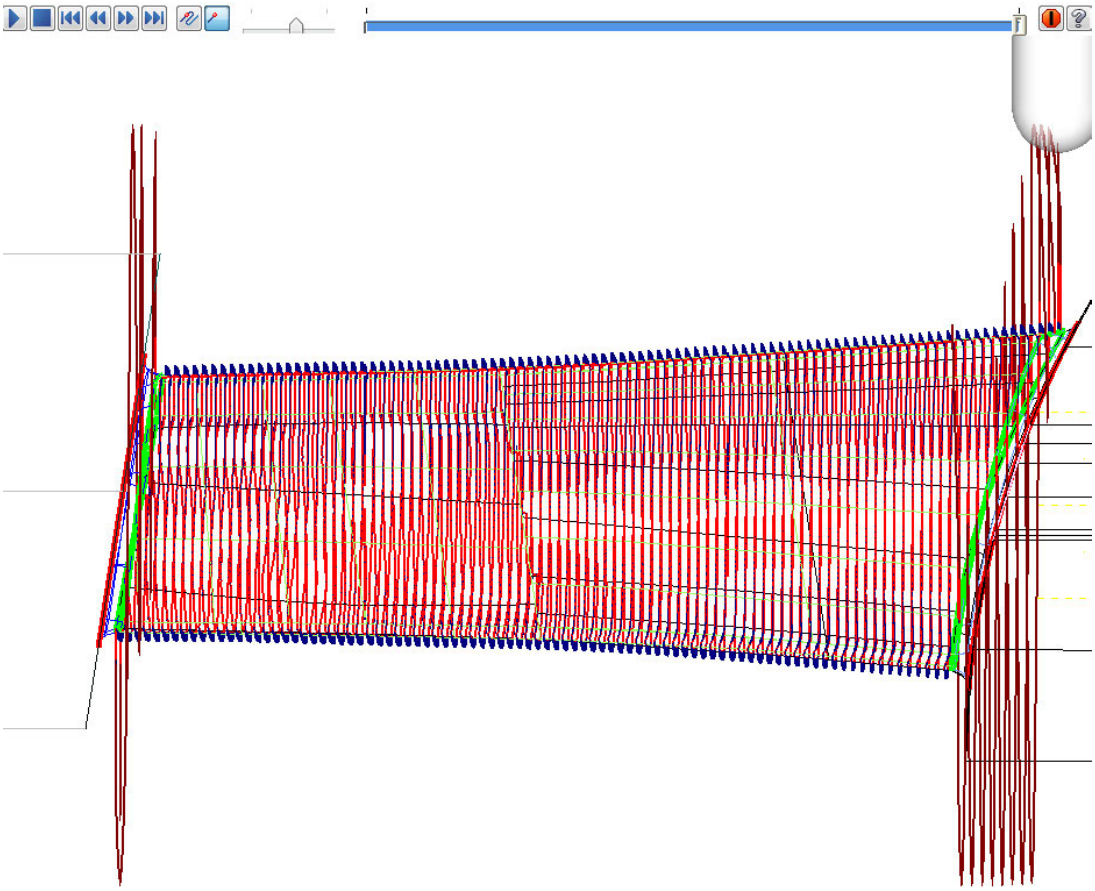


- To be able to easily check the toolpath, click on the **Options** button. 
- To maintain the part in a stationary position, disable **Simulate Rotary Axis**.
- Change the **4-5 Axis tool vector Length** to **0.05**.



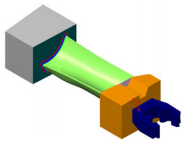
- Select the **OK** button to exit **Backplot Options**. 
- Change the **Graphic view** to **Top**. 
- You can step through the **Backplot** by using the **Step forward**  or **Step back**  buttons.
- You can adjust the speed of the backplot. 
- Select the **Play** button in the **VCR** bar to run **Backplot**. 

- The toolpath should look as shown.



NOTE: The extra rapid movements at the beginning and end of the toolpath. This is the result of the parallel pattern and of the size of the small move in the Links between slices.
In the next step we are comparing this toolpath with the Morph between 2 curves. This toolpath has a better pattern that uses two curves created at both ends of the surface and then approximates the passes between them creating a smooth transition.

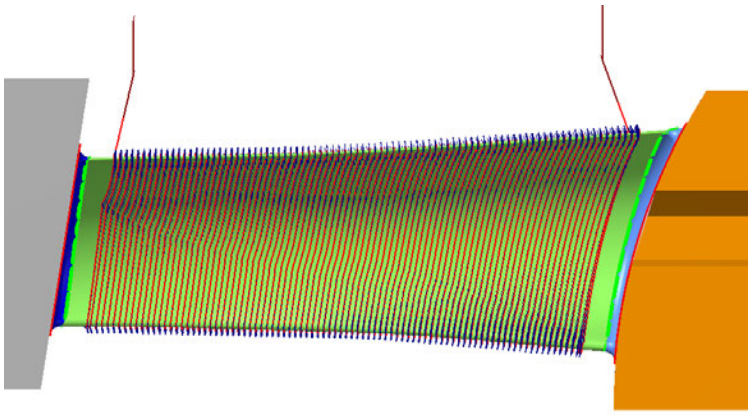
- Select the **OK** button to exit **Backplot**.
- Press **Alt** + **T** to remove the toolpath display.



STEP 5: FINISH THE BLADE USING MORPH BETWEEN CURVES TOOLPATH

The **Morph between 2 curves** will create a morphed toolpath between two leading curves. A "morphed" toolpath is one that is approximated between the tilt curves and evenly spread over the surface. This toolpath will ensure a better finish with less retraction moves. The toolpath is located in the **Wireframe toolpath family.**

Toolpath Preview:



5.1 Finish the blade surfaces using Morph between 2 curves

Create the Morph Between 2 Curves toolpath from the Wireframe toolpath family using the following parameters:

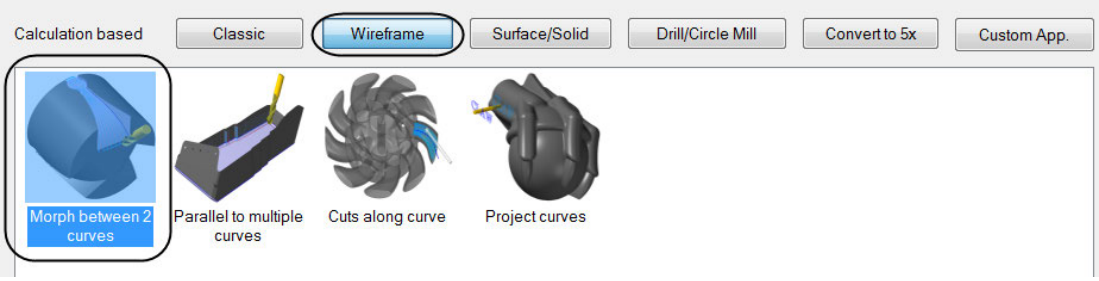
1/2"Ball Endmill.
Area type = Full avoid cuts at exact edges.
Cutting method set to One way; Counterclockwise.
Cut Tolerance = 0.0005".
Maximum stepover = **0.05".**
Enable Enforce cutting direction (assume closed contours).
Output format set to 4 Axis; Maximum angle step = 3 degrees.
Lead angle to cutting direction = 0.0.
Tool axis will = Surface with lead/lag.
Tilt angle at side of cutting direction = **0.**
Side tilt definition set to **Ortho to cut dir at each pos.**
Links between slice set the **Small move size** to **400%.**

Toolpaths

- **Multiaxis.**

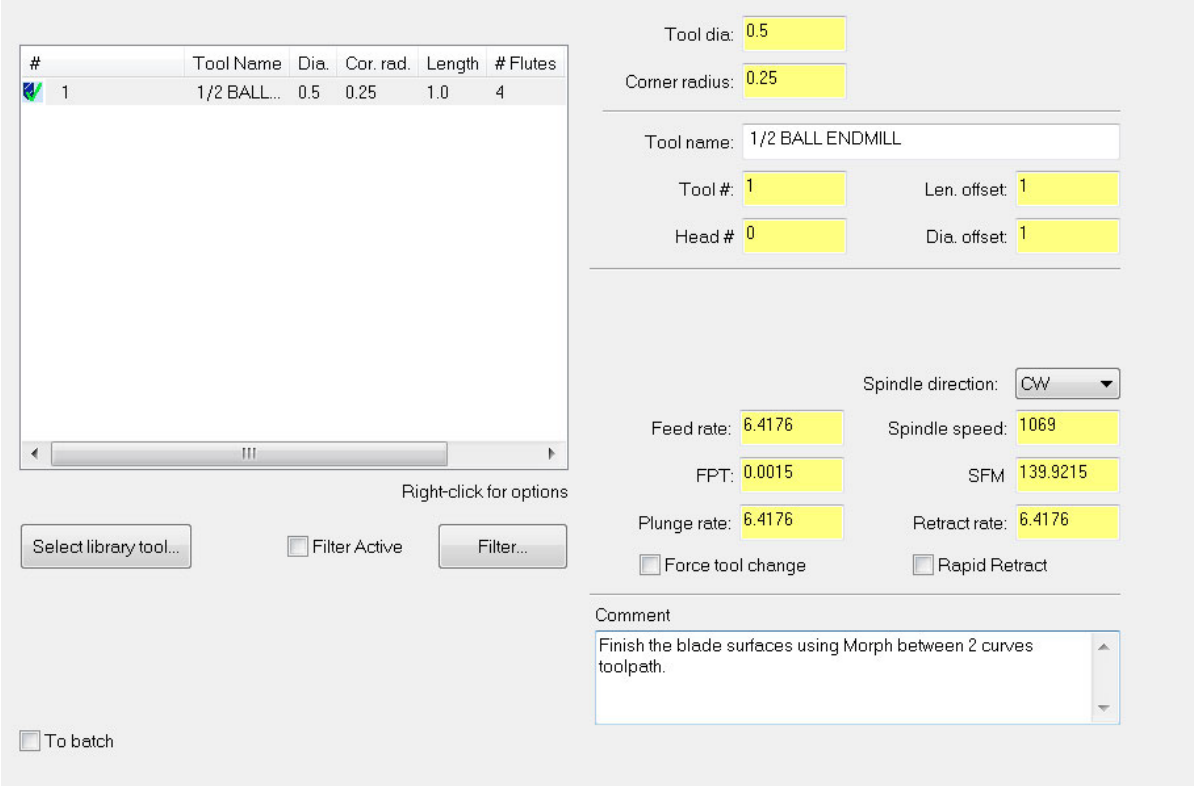
Toolpath type

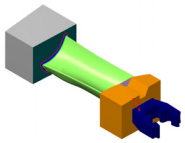
- Select **Wireframe** button and click on the **Morph between 2 curves.**



- From the **Tree View** area, select **Tool** and change the parameters as shown in Figure: 5.1.1.

Figure: 5.1.1



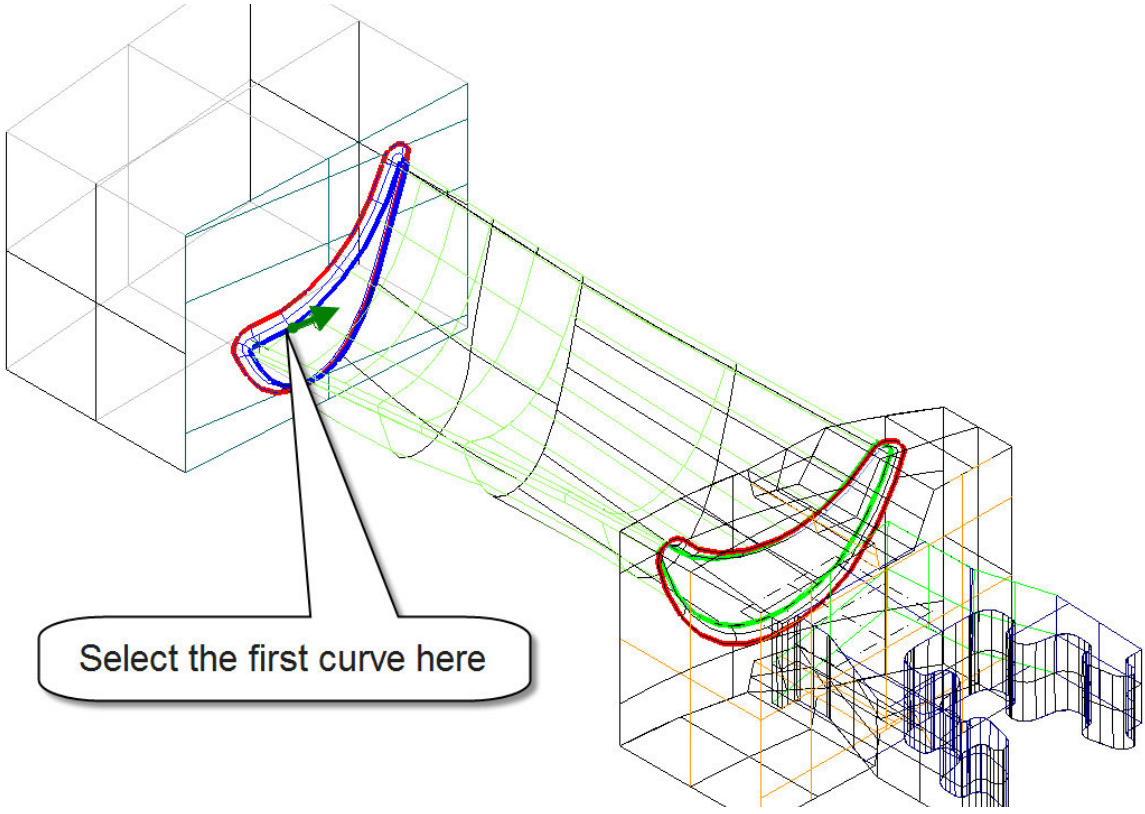


Cut Pattern

Cut pattern settings determine the geometry the tool follows and how it travels along that geometry. The direction of the cuts is approximated between the two curves and evenly spread over the surface.
You can also choose the **Area type** which determines where the tool starts, ends, and area to be cut. **Cut Pattern** page also allows you to set the **Cutting method** and the **direction** for one way machining. In the **Cut Pattern** you can set the **Cut tolerance** and the **Maximum stepover**.

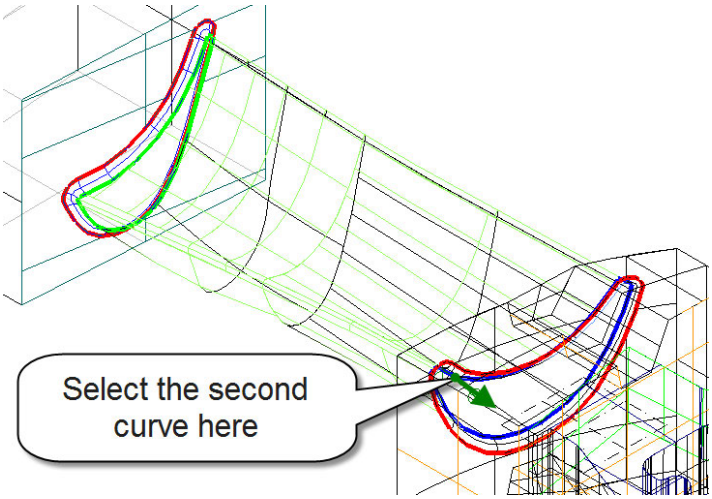
- From the **Tree View** area, select **Cut Pattern.**
- Select the **First** button.

- Change the **Graphics View** to **Isometric**. 
- Select the green color curve at the left end of the turbine blade in **CCW** direction as shown.



- Select the **OK** button to exit the **Chaining** dialog box. 
- Select the **Second** button**.**

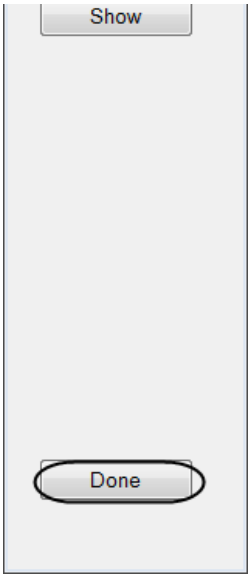
- Select the green color curve at the right end of the turbine blade in the same **CCW** direction.

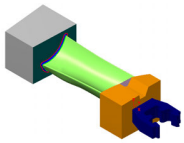


NOTE: Select the chains in the same direction and align their starting locations.

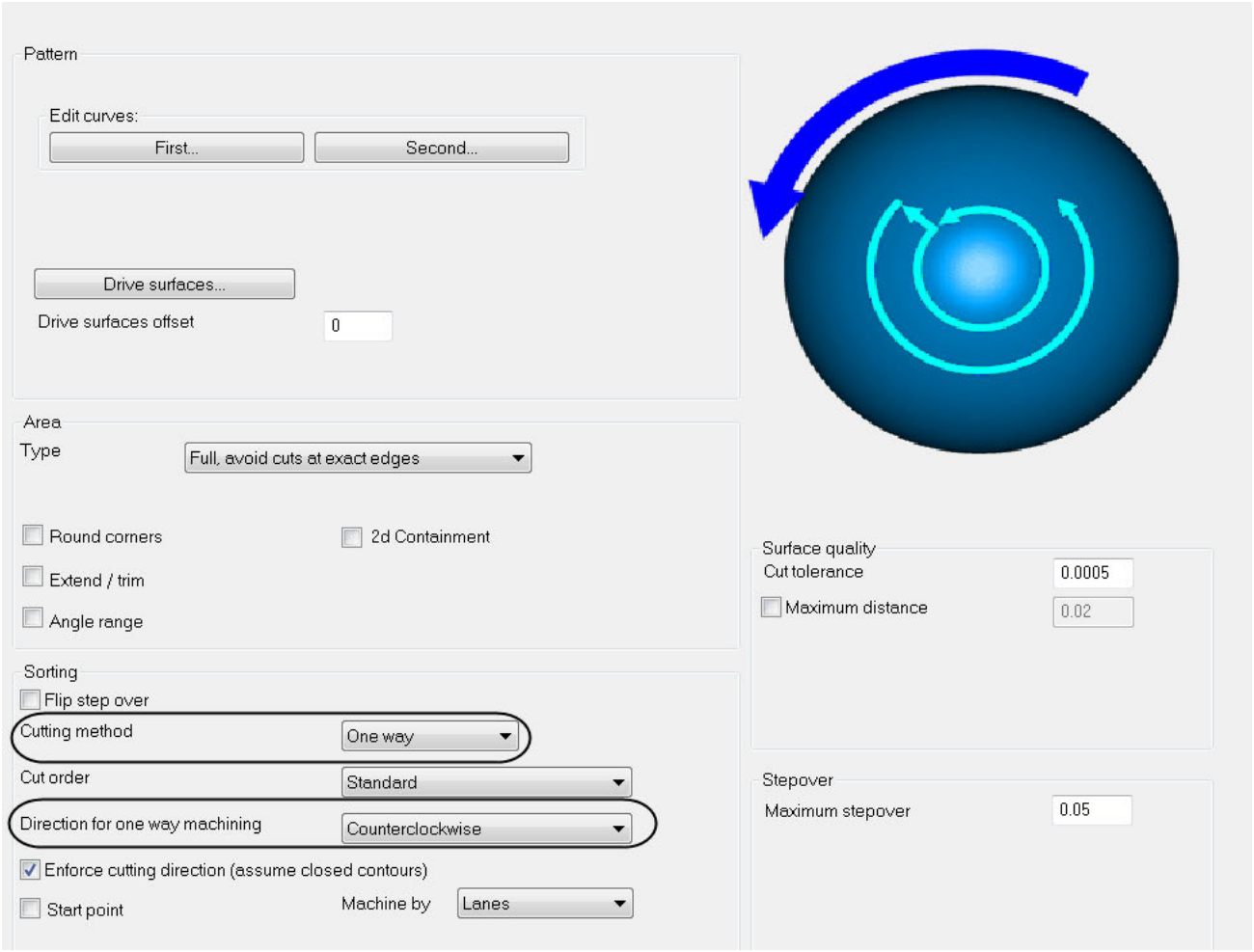
- Select the **OK** button to exit the **Chaining** dialog box.
- Click on the **Drive surfaces** button.

- Use the **QM Color** icon to select the green color no. **30** as previously shown.
- Select the **OK** button to exit the **Select All** dialog box.
- Press **Enter** to finish the selection.
- Select **Done** to exit **Drive srfs Selection** dialog box.

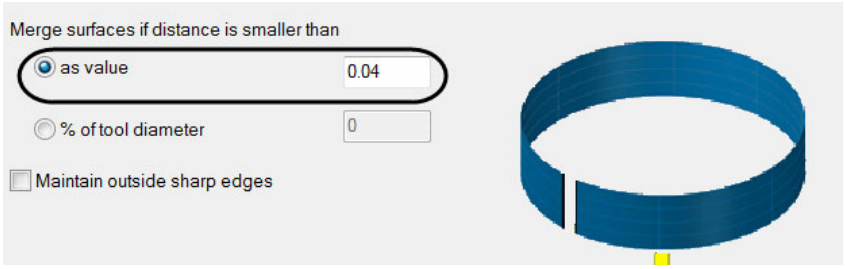




- Change the **Cutting method** to **One way** and set the **Direction for one way machining** to **Counterclockwise.**
- Leave the rest of the parameters as shown.

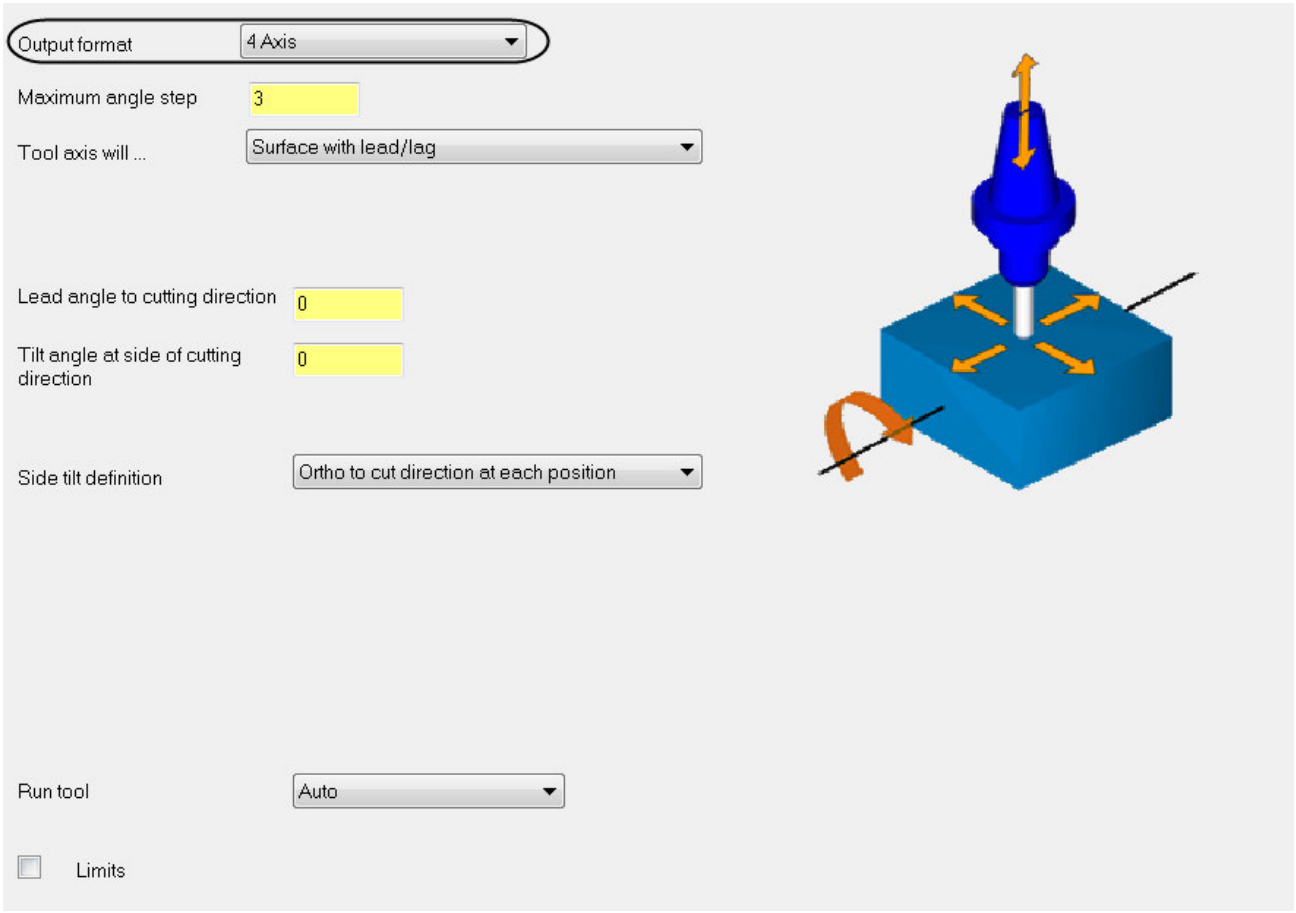


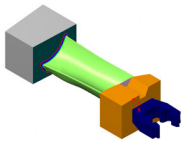
- From the **Tree view** area, select **Parameters For Surface Edge Handling** and ensure that the **Merge surfaces if distance is smaller than** is set as a value to **0.04** as shown in the following picture.



Tool Axis Control

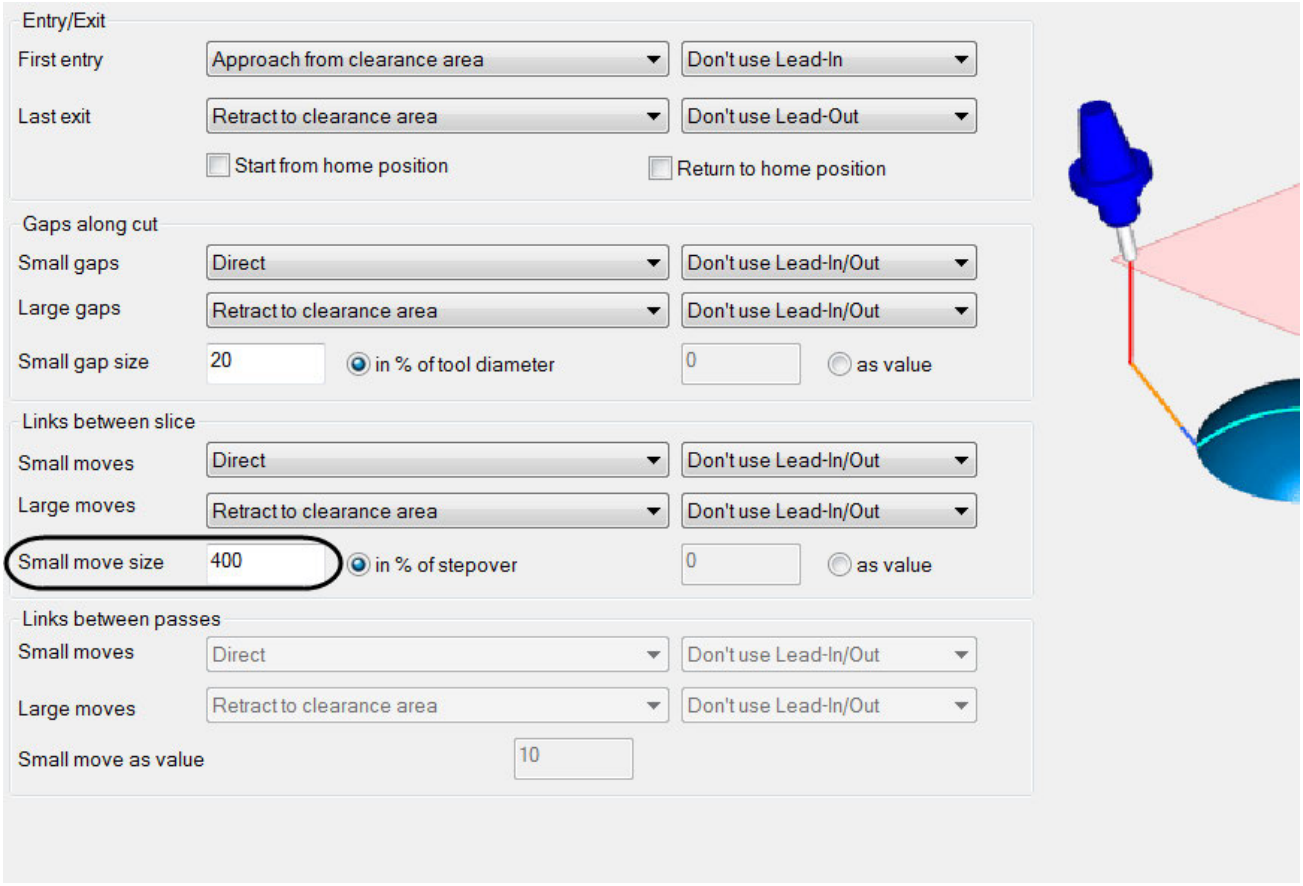
- From the **Tree View** area, select **Tool Axis Control.**
- Make sure that the **Output format** is set to **4-Axis** and leave the rest of the parameters as shown.





Linking

- From the **Tree View** area, select **Linking** and change the**Small move size** to **400**.
- Leave the rest of the parameters as shown.



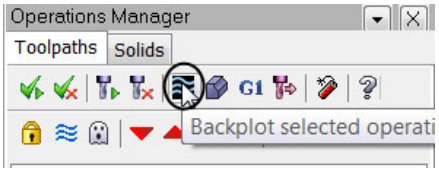
NOTE: Making the Small move size bigger, will keep the tool down avoiding unwanted retract movements between steps.

NOTE: The Clearance Area parameters should be the same as set in the previous operation.

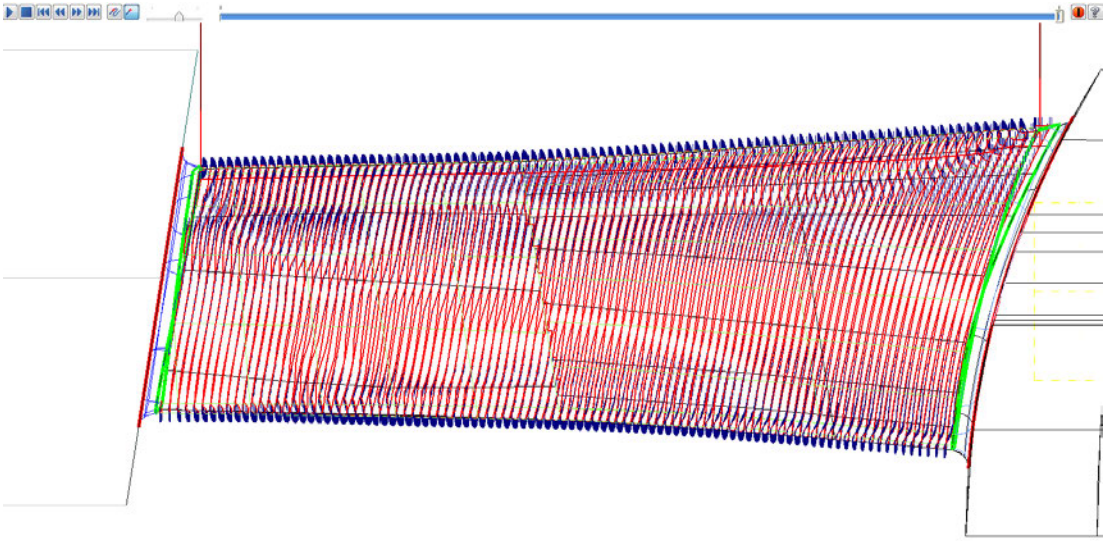
- Select the **OK** button to exit **Multiaxis Toolpath** - **Morph between 2 curves.**

5.2 Backplot the toolpath

- Select the **Backplot selected operations** button.

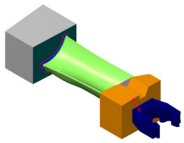


- Select the **Play** button in the **VCR** bar to run **Backplot**.
- Press **Alt + S** to un shade the surfaces if needed.
- Change the **Graphic view** to **Top**.
- The toolpath should look as shown.

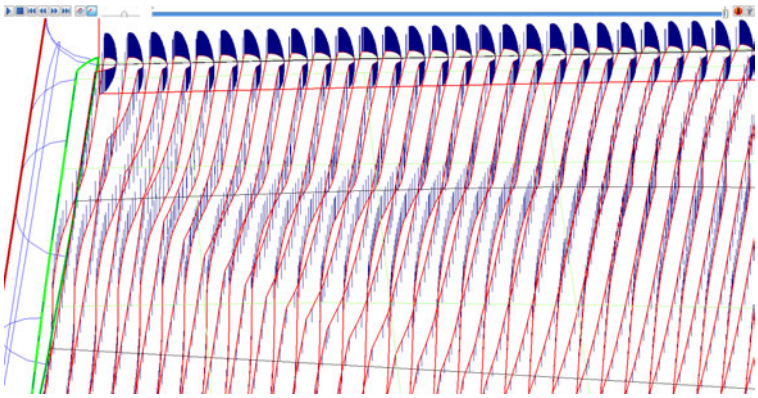


NOTE: The slices are morphing between the two curves and the rapid movements are gone.

- Zoom in the area close to the left fixture and observe the stepover moves.

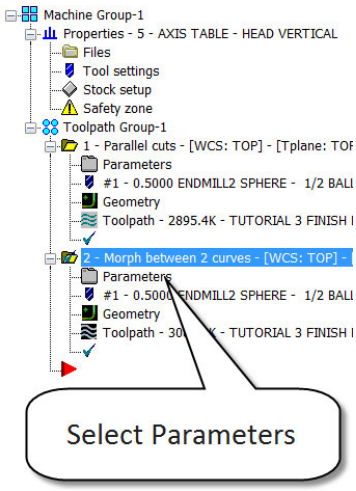


NOTE: To zoom in, move the cursor close to the focal point and scroll up with the mouse wheel.



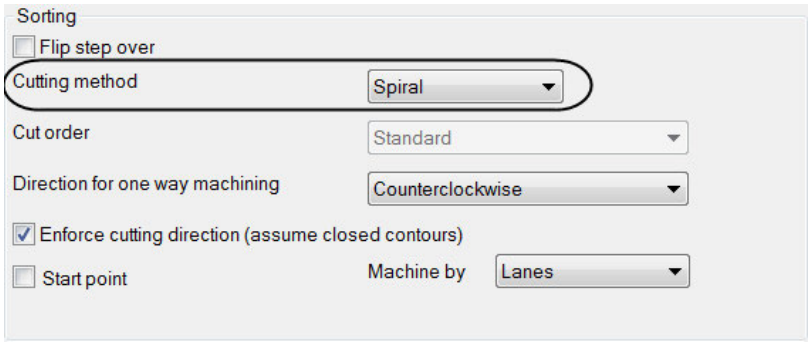
NOTE: The stepover moves are set to direct lines which will leave scallops on the part. Changing the cutting method from one way to spiral will make the transition smoother and leave no scallops on the part.

- Select the **OK** button to exit **Backplot**. 
- Press **Alt** + **T** to remove the toolpath display.
- Modify the **Cutting method** to **Spiral**.
- From the **Toolpaths Operations Manager**, select the **Parameters** in the second operations.



Cut Pattern

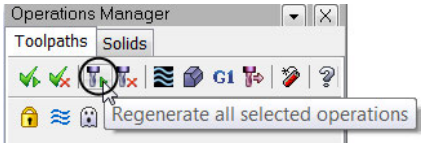
- From the **Tree View** list, select **Cut Pattern** and change the **Cutting method** to **Spiral.**



- Select the **OK** button to exit **Multiaxis Toolpath** - **Morph between 2 curves.**

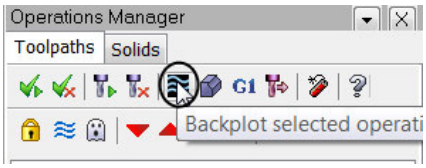
5.3 Regenerate the toolpath

- From the **Toolpaths Operations Manager**, select the **Regenerate all selected operations**.

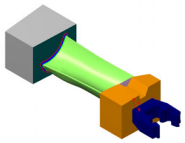


5.4 Backplot the toolpath again

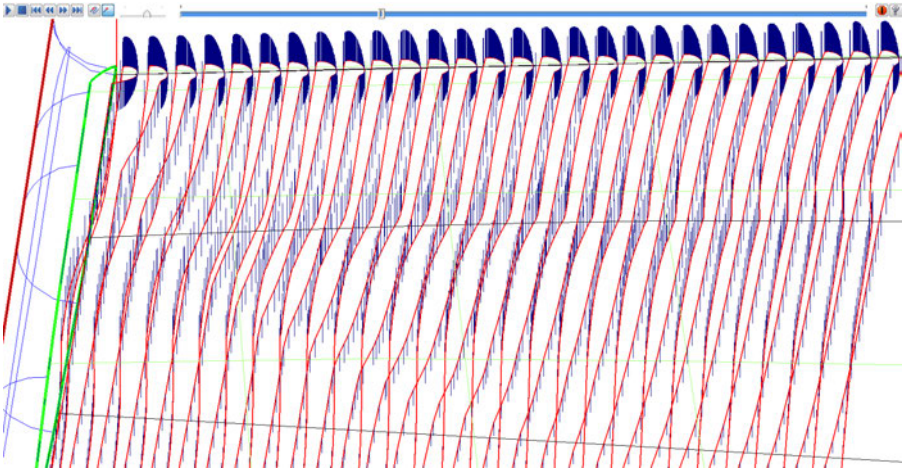
- Select the **Backplot selected operations** button.



- Select the **Play** button in the **VCR** bar to run **Backplot**.



- The toolpath should look as shown.

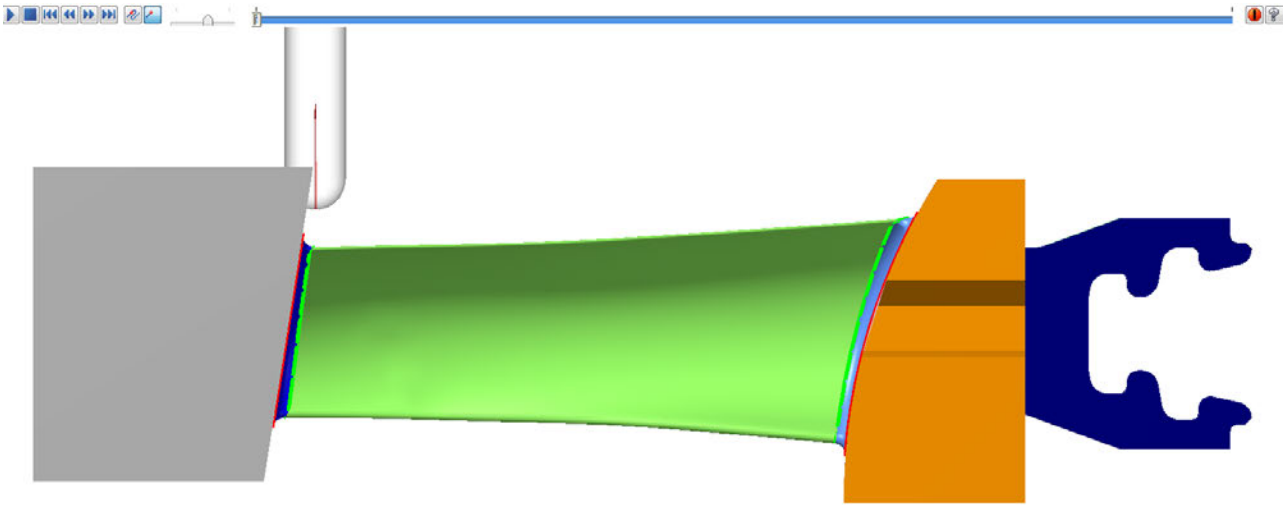


NOTE: The direct lines between the steps are gone.

- Select the **Fit** icon to fit the geometry to the screen. 
- From the **Backplot VCR** bar, select the **Previous Stop** button to restart the toolpath backplotting.

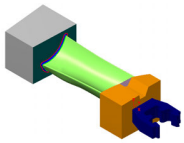
- Press **Alt** + **S** to shade the geometry.

- From the **Backplot VCR** bar, click on the **Step forward** button several times, until the tool position is as shown.



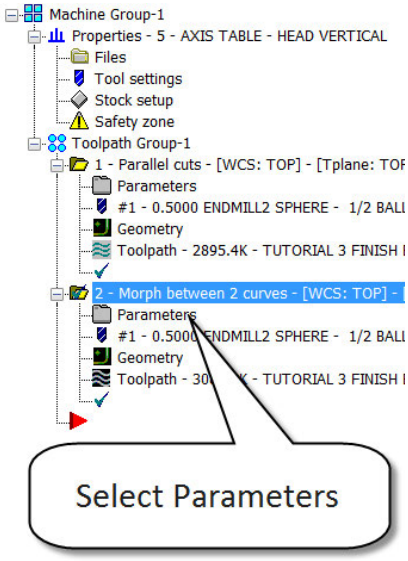
NOTE: The tool collision with the fixture. To fix this problem you will change the **Area Type** to **Full, start and end at exact surface edges**. This option allows you to add **Margins** at both the start and the end of the toolpath to compensate for the tool radius.

- Select the **OK** button to exit **Backplot**.



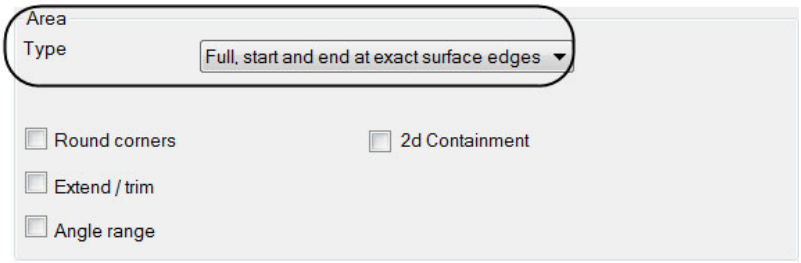
5.5 Add Margins in the toolpath to avoid tool collisions with the fixtures

- From the **Toolpaths Operations Manager**, select the **Parameters** in the second operations.

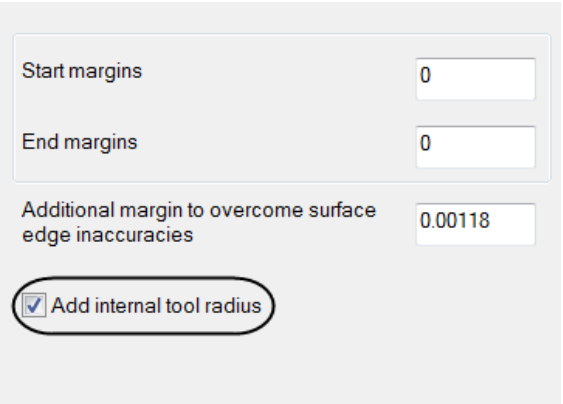


Cut Pattern

- From the **Tree View** list, select **Cut Pattern** and change the **Area Type** to **Full, start and end at exact surface edges.**



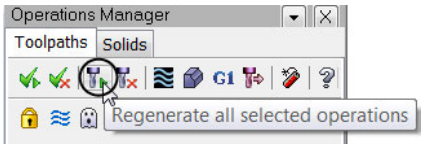
- From the **Tree View** area, click on the plus sign in front of **Cut Pattern** to expand it.
- Select **Margins** and enable **Add internal tool radius**.
- Leave the **Additional margin to overcome surface edge inaccuracies** to **0.001180.03**.



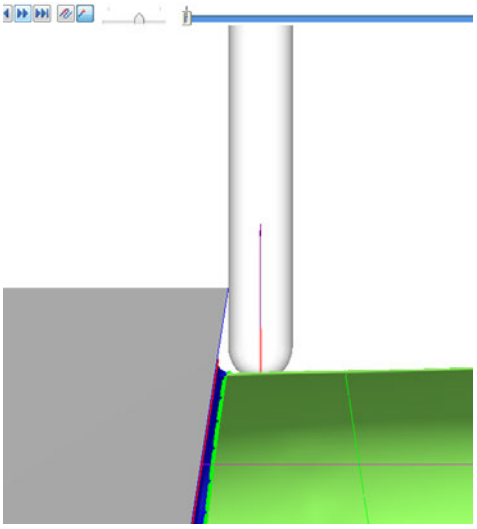
- Select the **OK** button to exit **Multiaxis Toolpath** - **Morph between 2 curves.**

5.6 Regenerate and Backplot the toolpath

- From the **Toolpaths Operations Manager**, select the **Regenerate all selected operations**.



- The toolpath should look as shown.

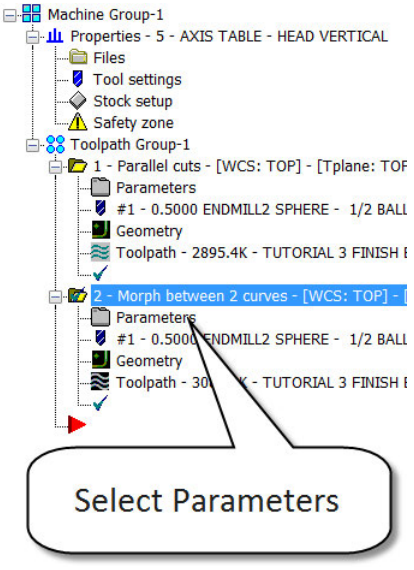


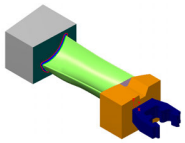
NOTE: Although the tool is moved from the surface edges it might still gouge into the fixture. A solution to this problem is to approach the part with the tool axis tilted away from the fixture. In the next step, you are going to gradualy tilt the tool.

- Select the **OK** button to exit **Backplot**.

5.7 Change the output format to 5-Axis; use lead and tilt angle with gradual side tilt angle

- From the **Toolpaths Operations Manager**, select the **Parameters** in the second operations.



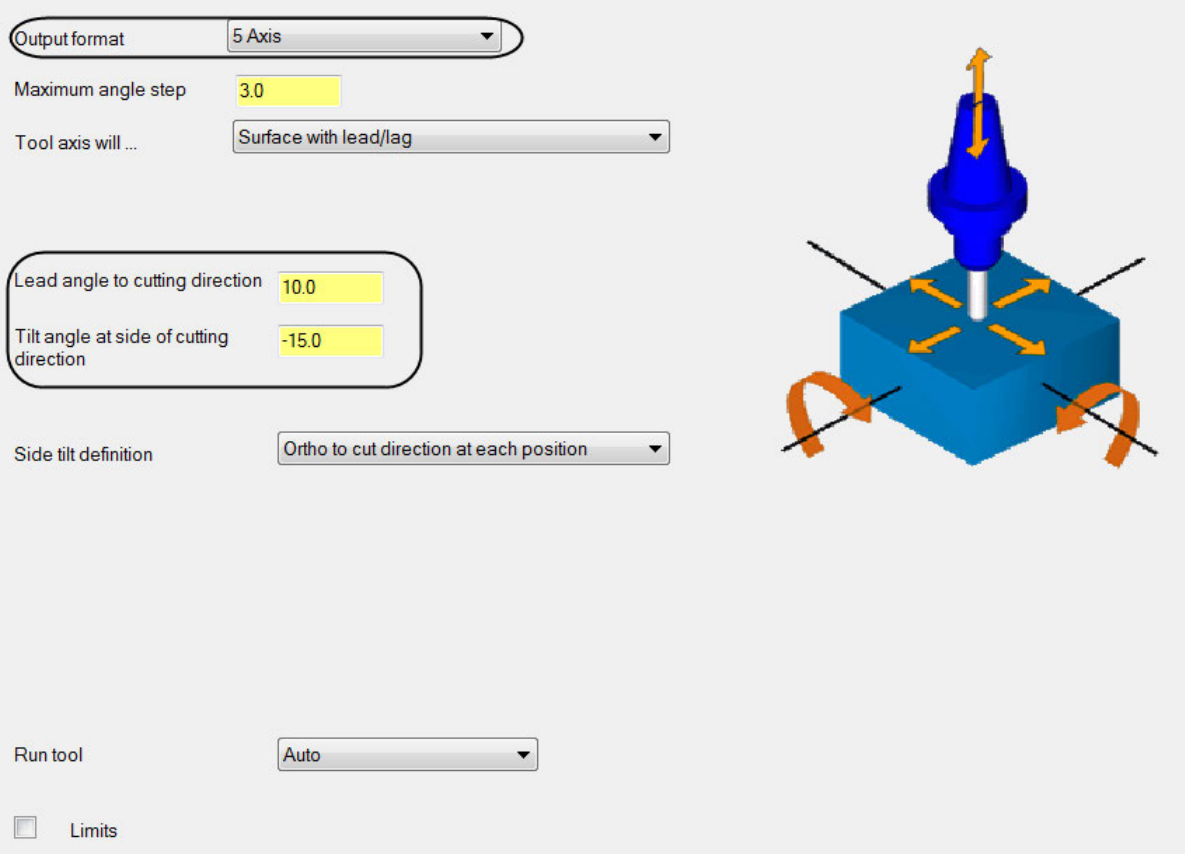


Tool Axis Control

Lead angle sets the angle applied to the tool axis in the direction of the cut, relative to the surface normal.
Tilt angle at side of cutting direction sets the angle applied to the tool axis in the direction of the cut, relative to the surface normal. A positive angle tilts the tool to the left; a negative angle tilts the tool to the right.

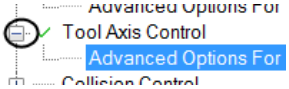
- From the **Tree View** list, select **Tool Axis Control** and change the **Output format** to **5-Axis**.
- Set the **Lead angle** to **10.0** and the **Tilt angle** to **-15.0** as shown in Figure: 5.7.1.

Figure: 5.7.1

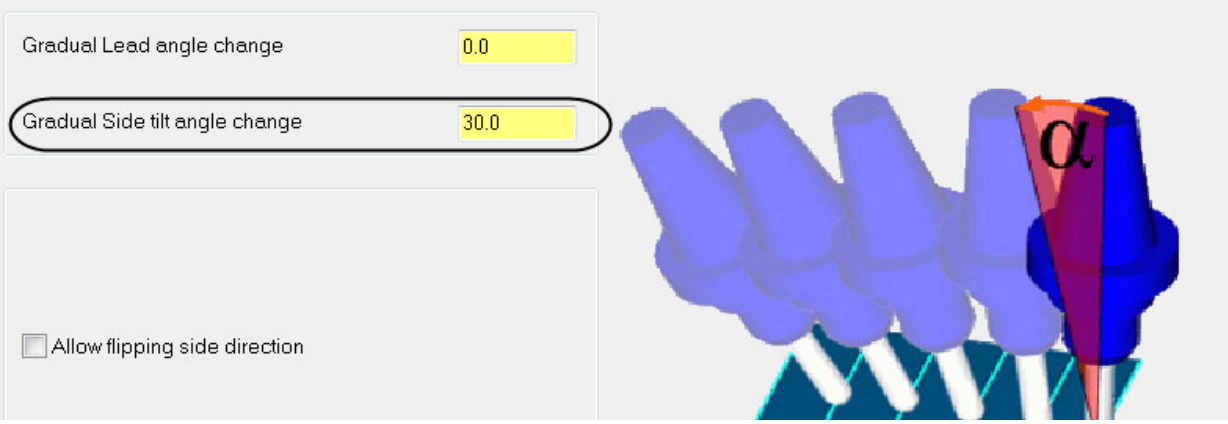


Advanced Options for Tilting Relative to Cutting Direction

- From the **Tree View** area, expand the **Tool Axis Control** if needed and select the **Advanced Options for Tilting Relative to Cutting Direction**.



- Set the **Gradual Side tilt angle** to **30** degrees.

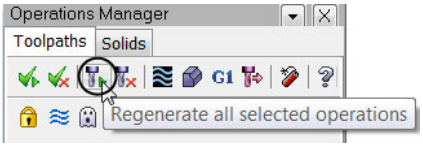


NOTE: The angle change is applied gradually over the series of cuts created for the operation. The first cut will use only the side tilt angle. The last cut will use the side tilt angle plus the change value. Cuts in between will use side tilt angle plus a percentage of the change value.

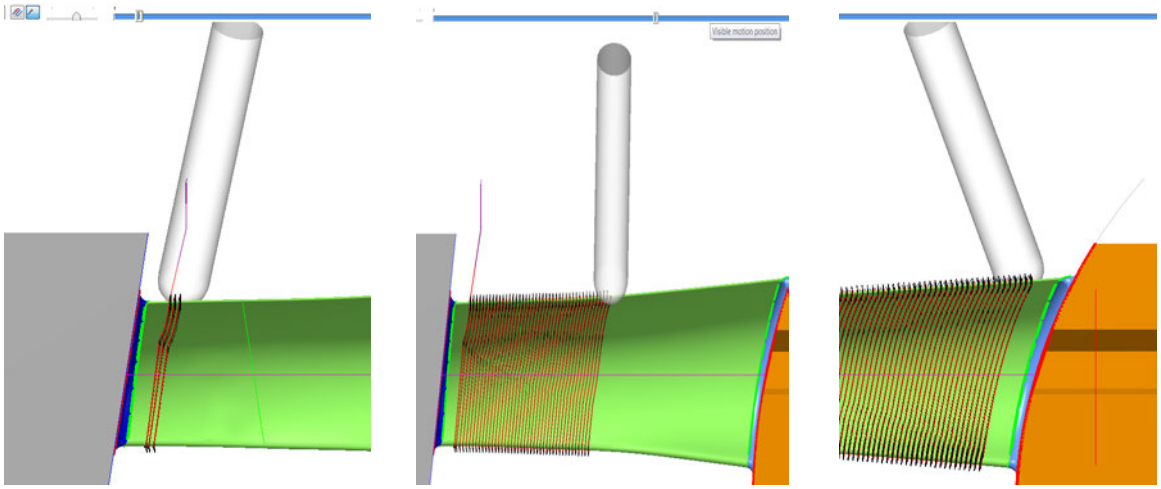
- Select the **OK** button to exit **Multiaxis Toolpath** - **Morph between 2 curves.**

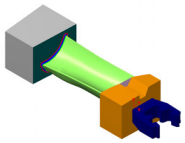
Regenerate and Backplot the toolpath

- From the **Toolpaths Operations Manager**, select the **Regenerate all selected operations**.



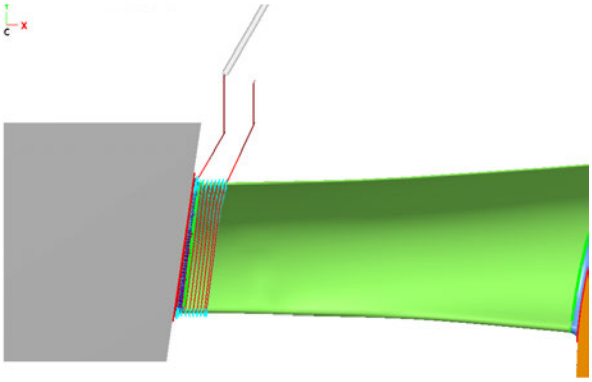
NOTE: The gradual change in the side tilt angle.





STEP 6: FINISH THE FILLETS - PARALLEL TO MULTIPLE CURVES WITH LIMITS

Toolpath Preview:



6.1 Finish the left side fillet surfaces using Parallel to multiple curves toolpath

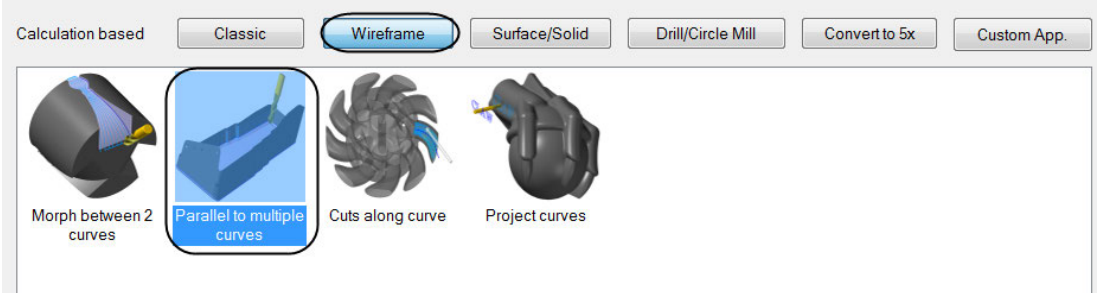
Create the Parallel to multiple curves toolpath from the Wireframe toolpath family using the following parameters:
1/16" Ball Endmill.
Area type = Determine by number of cuts; Number of cuts = 10.
Cutting method = Spiral.
Cut Tolerance = 0.0005".
Maximum stepover = **0.0325".**
Enable Enforce cutting direction (assume closed contours).
Output format set to 5 Axis; Maximum angle step = 3 degrees.
Tool axis will = Surface with lead/lag.
Lead angle to cutting direction = 10 degrees.
Tilt angle at side of cutting direction = **0.**
Side tilt definition set to **Ortho to cut dir at each pos.**
Limits; Contain tool within conical angles; W1 =80; W2 =80; Cone axis set to **X-axis.**

Toolpaths

-  **Multiaxis.**

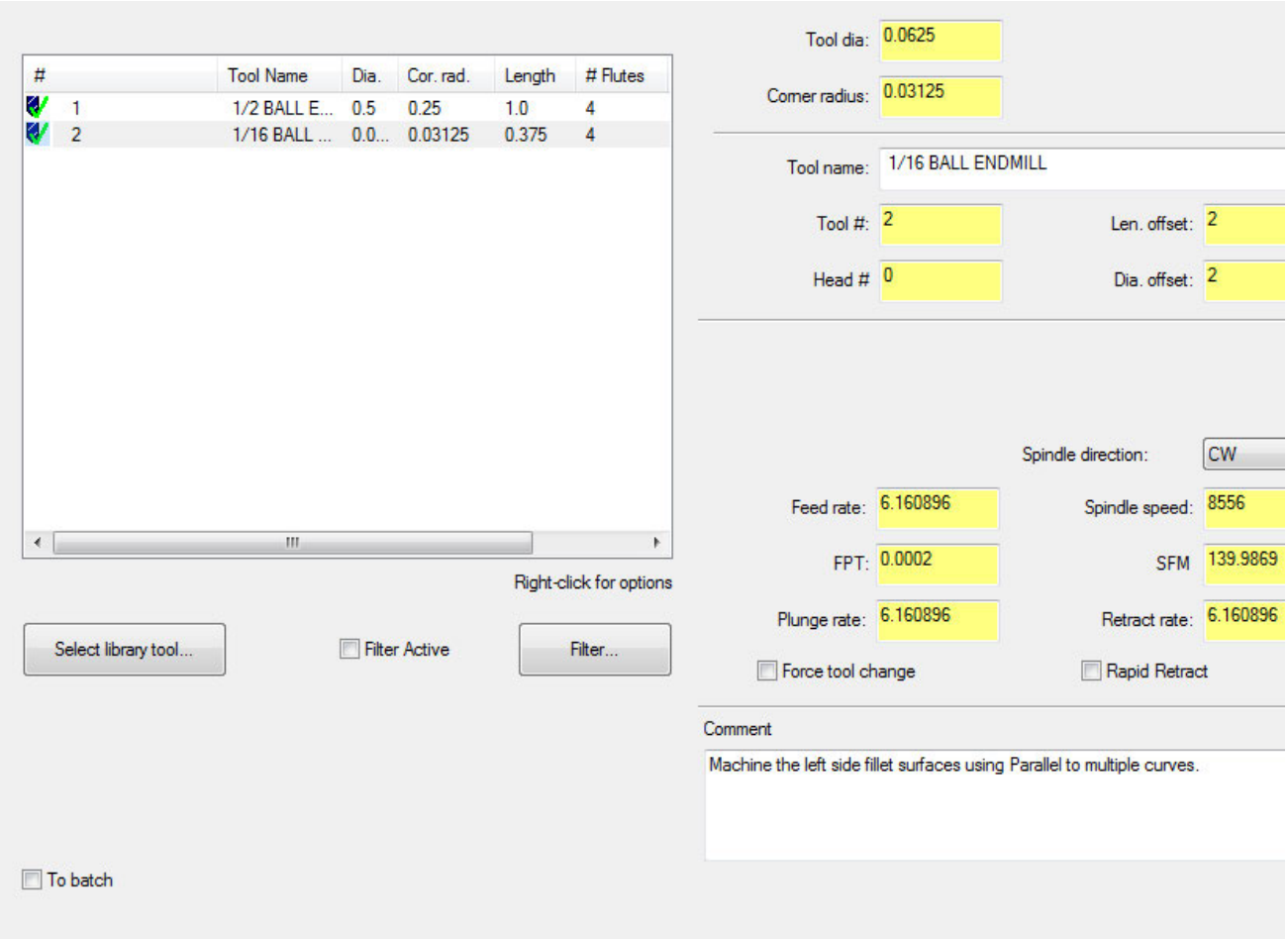
Toolpath type

- Select **Wireframe** button.
- Select **Parallel to multiple curves.**



Tool

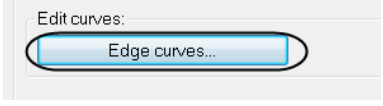
- From the **Tree View** area, select **Tool**.
- Click on the **Select tool library** and using the **Filter** options select the **1/16" Ball Endmill** as previously shown.
- Add a comment in the **Comment** area.



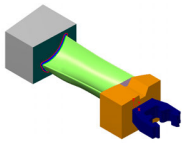
Cut Pattern

Cut pattern settings determine the geometry the tool follows and how it travels along that geometry. You can select the **Edge curves** and the **surfaces** to be machined. You can also choose the area type which determines where the tool starts, ends, and area to be cut. **Cut Pattern** page also allows you to set the **Cutting method** and the direction for one way machining. In the **Cut Pattern** you can set the **Cut tolerance** and the **Maximum stepover**.

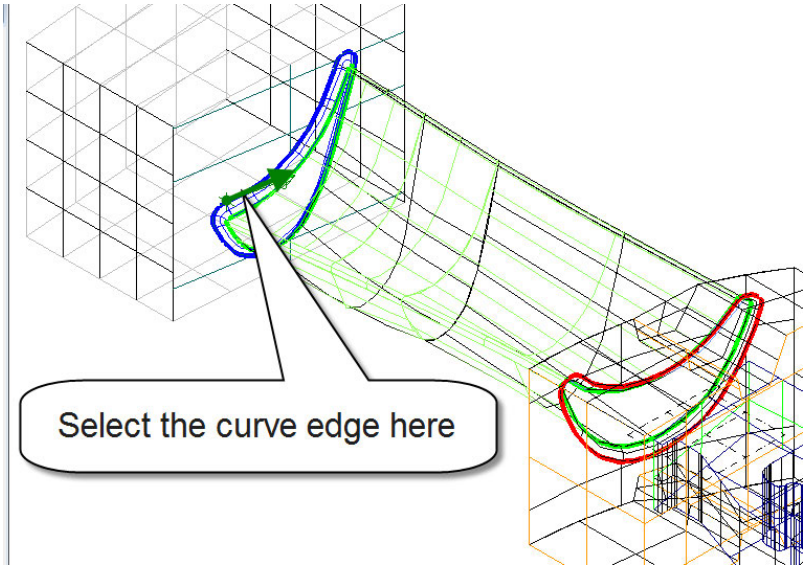
- From the **Tree View** area, select **Cut Pattern**.
- Click on the **Edge curves** button.



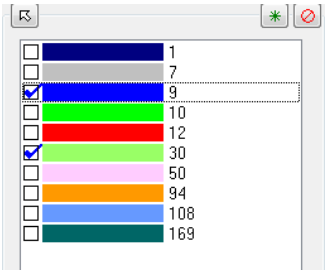
- Change the **Graphics view** to **Isometric**. 



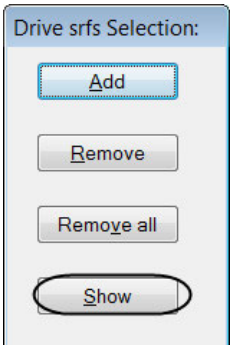
- Select the **Fit** icon.
- Press **Alt** + **S** to un shade the surfaces if needed.
- Select the red color contour in the **CW** direction as shown.



- Select the **OK** button to exit **Chaining** dialog box.
- Click on the **Drive surfaces** button and using **QM Color** icon, hold down **Ctrl** key and select the green color no. **30** and color blue no. **9**.

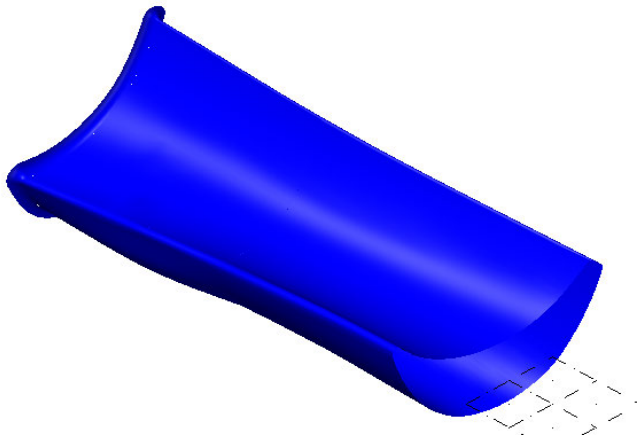


- Select the **OK** button to exit the **Select All** dialog box.
- Press **Enter** to finish the selection.
- Click on the **Show** button to see the selected surfaces.

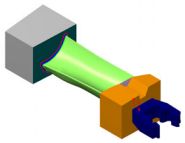


- The surfaces should look as shown in Figure: 6.1.1.

Figure: 6.1.1

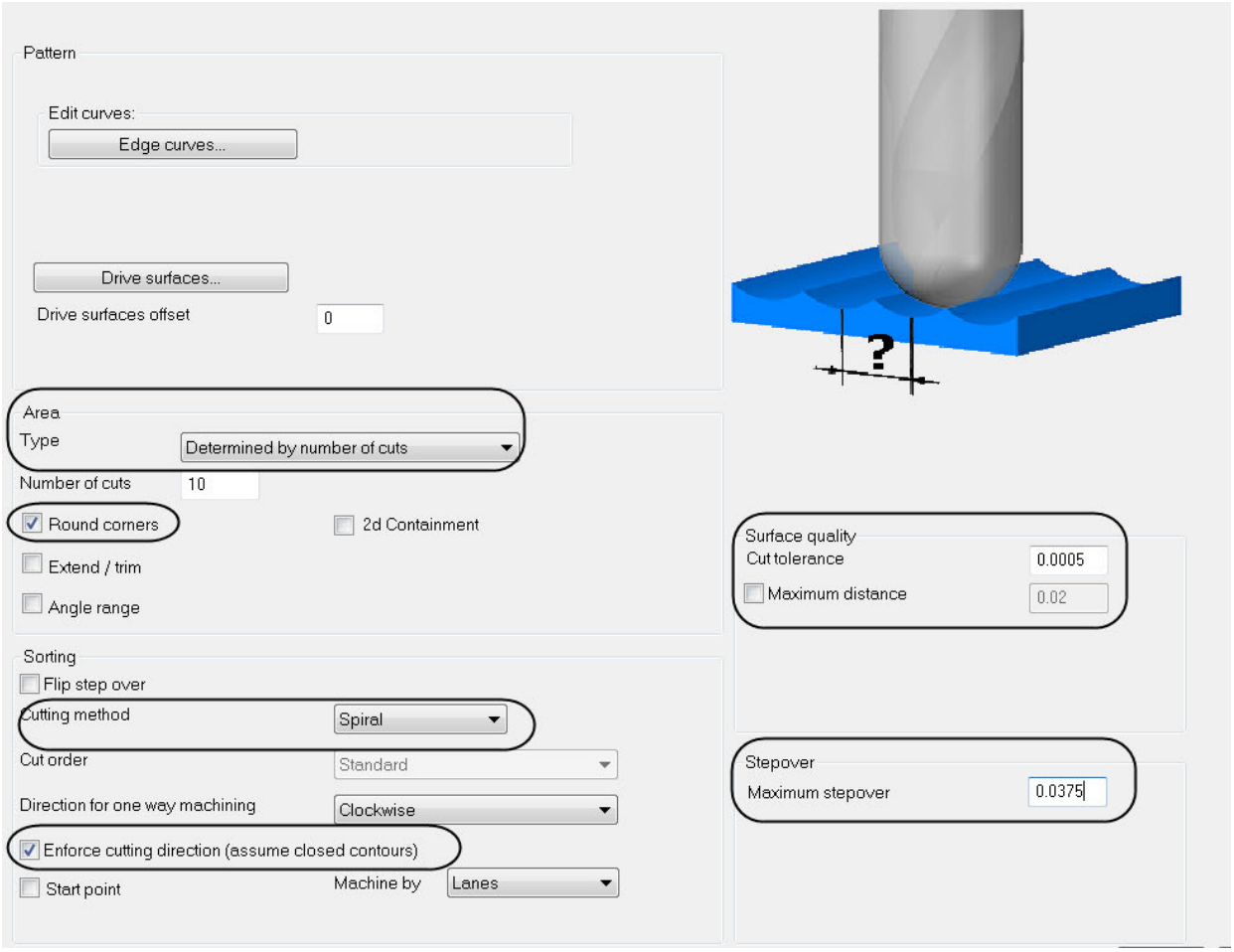


- Press **Enter** to continue.
- Select **Done** to exit **Drive srfs Selection** dialog box.

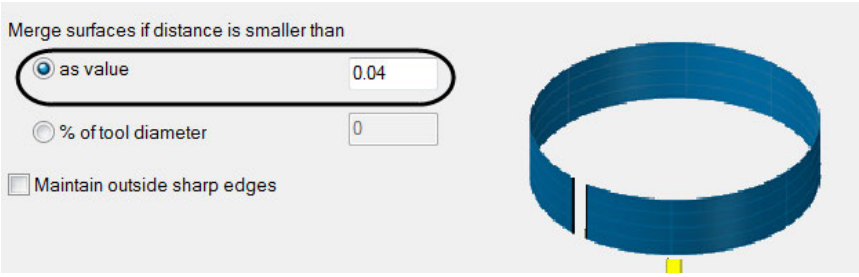


- Set the **Area type** to **Determine by number of cuts** and set the **Number of cuts** = **10** as shown in Figure: 6.1.2.
- Enable **Round corners** to round the surface model to avoid the small radii and internal sharp corners in the direction of cutting as shown.
- Set the **Cutting method** set to **Spiral** and **Direction for one way machining** set to **Clockwise.**
- Enable **Enforce cutting direction (assume closed contours).**
- Set the **Cut Tolerance** to **0.0005** and **Maximum stepover** = **0.0375.**

Figure: 6.1.2



- From the **Tree view** area, select **Parameters For Surface Edge Handling** and ensure that the **Merge surfaces if distance is smaller than** is set as a value to **0.04** as shown in the following picture.

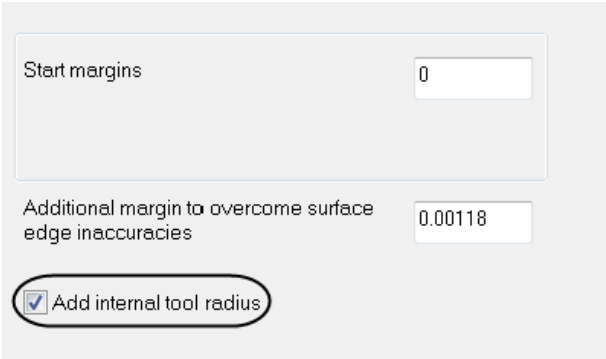


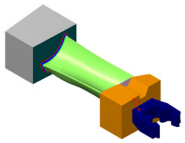
Margins

Margins create additional space so the toolpath is not starting directly on the surface edge. You are machining the fillet surfaces with a tool that has the radius the same radius as the fillet surfaces. To move the toolpath to the middle of the fillet enable add internal tool radius for margins.

- Click on the plus sign in front of **Cut pattern** to expand it and select **Margins**.
- Enable **Add internal tool radius** and leave the rest of the parameters as shown in Figure: 6.1.3.

Figure: 6.1.3



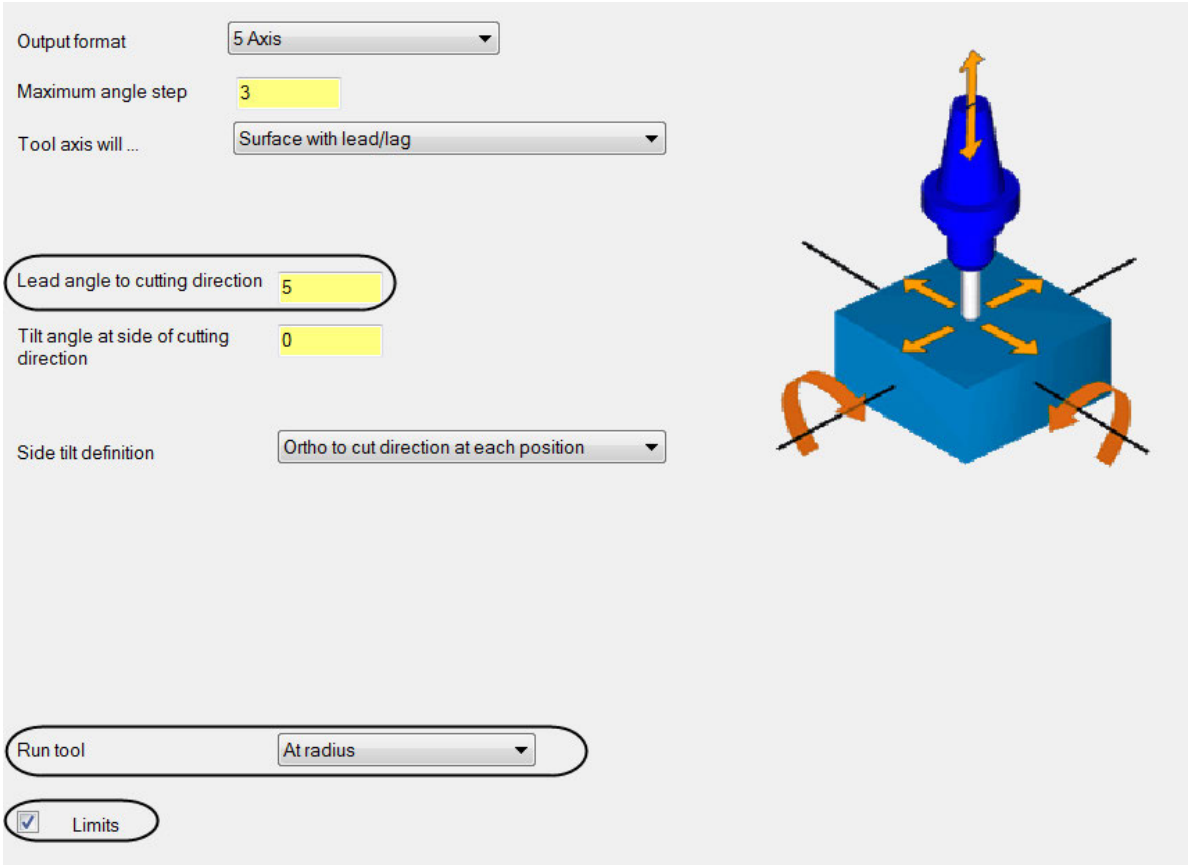


Tool Axis Control

Tool axis control settings determine the tool's orientation in relation to the geometry being cut. You can set the maximum angle the tool will be allowed to move through between adjacent moves.
The **Limit** parameters allow you to limit your tool axis orientation along the toolpath.

- From the **Tree view** area, select the **Tool Axis Control**.
- Set the **Run tool** to **At radius** to move the contact point from the tip of the tool to the radius of the tool and enable **Limits** as shown in Figure: 6.1.4.

Figure: 6.1.4

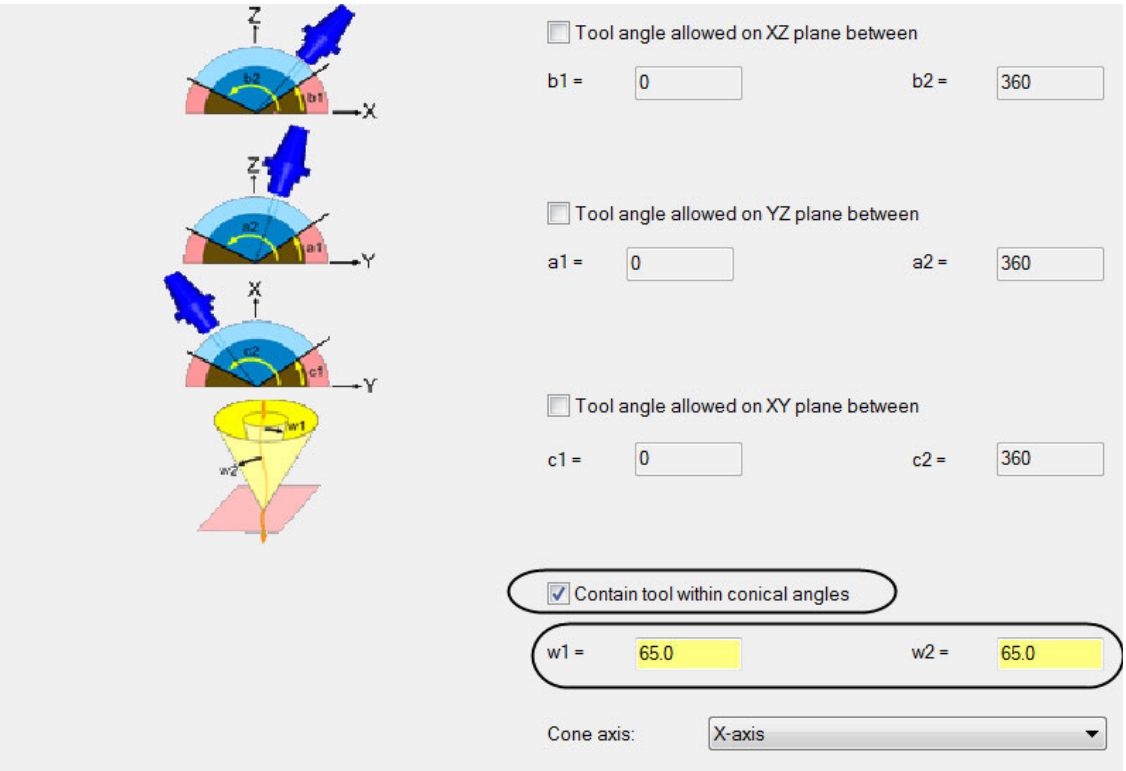


Limits

Contain tool within conical angles limits the tool axis orientation by two angles starting from toolpath slice normal vector. As an example, imagine two cones with different opening angles w1 and w2. The tool axis direction must be between these two cones.

- From the expanded **Tool Axis Control**, select **Limits** and enable **Contain tool within conical angles** as shown in Figure: 6.1.5.
- Set both angles **W1** and **W2** to **65** and set the **Cone axis** to **X-Axis** as shown in Figure: 6.1.5.

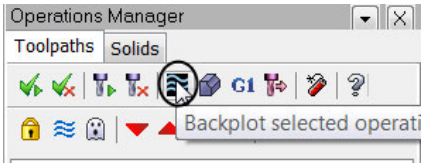
Figure: 6.1.5

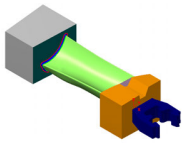


- Leave the rest of the parameters as previously set.
- Select the **OK** button to exit **Multiaxis Toolpath** - **Parallel to multiple curves.**

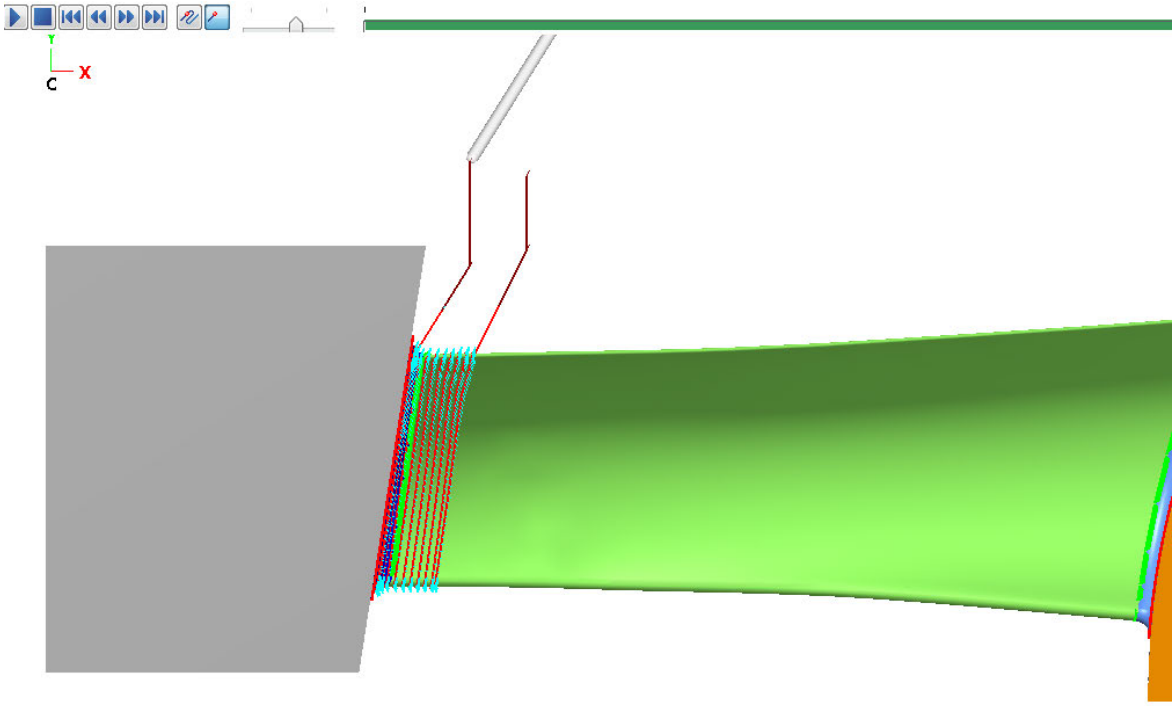
6.2 Backplot the toolpath

- Select the **Backplot selected operations** button.





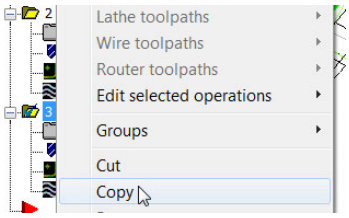
- Select the **Play**  button in the **VCR** bar to run **Backplot**.
- Press **Alt + S** to shade the surfaces if needed.
- The toolpath should look as shown.



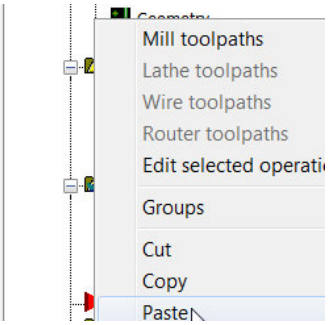
- Select the **OK** button to exit **Backplot.** 

6.3 Finish the right side end of the blade

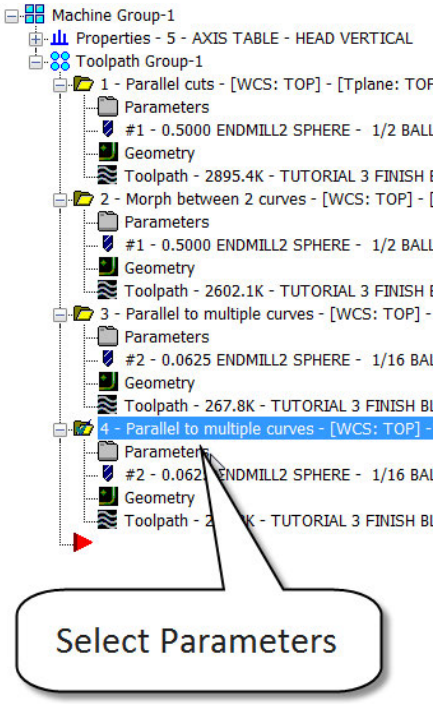
- From the **Toolpaths Operation Manager** select the last operation only.
- Right mouse click on the last toolpath and select **Copy**.

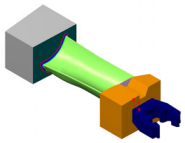


- Right mouse click on the last toolpath and select **Paste.**



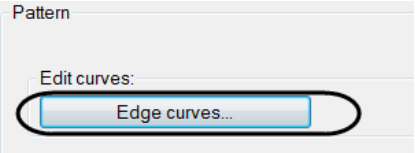
- Click on the **Parameters** in the last toolpath (fourth operation).



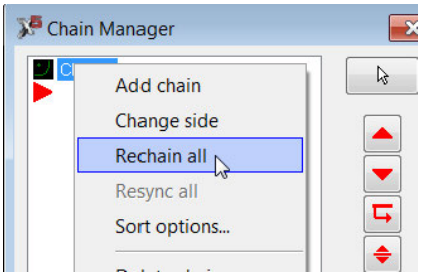


Cut Pattern:

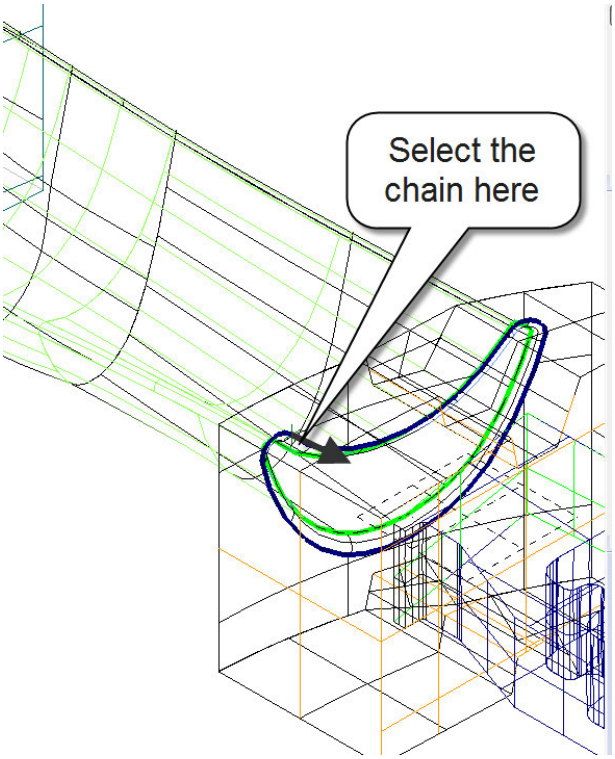
- From the **Tree View** area, select **Cut Pattern.**
- Click on the **Edge curve** button.



- In the **Chain Manager** dialog box, right mouse click and select **Rechain all** to reselect the chain.

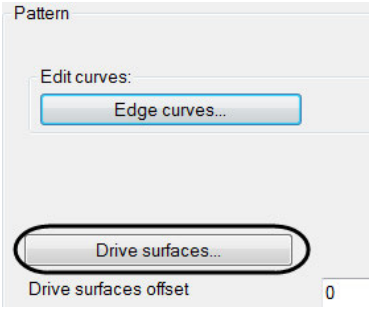


- Select the red curve at the right side end in the **CW** direction as shown.



- Select the **OK** button to exit **Chaining** dialog box. 
- Select the **OK** button to exit **Chaining Manager** dialog box. 

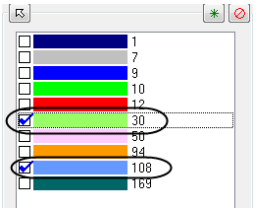
- Click on the **Drive surfaces button.**



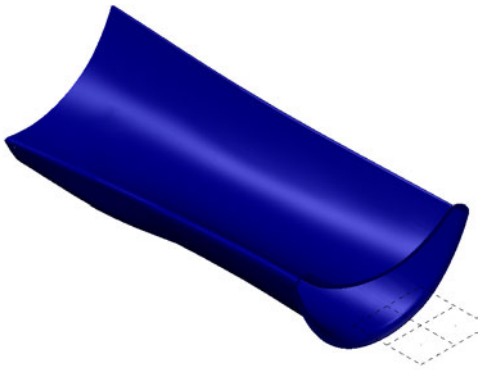
- In the **Drive srfs Selection** dialog box, click on the **Remove all** button to remove the existing surfaces as shown in Figure: 6.3.1.

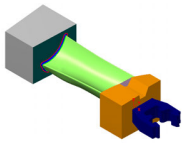
Figure: 6.3.1

- Select the **Add** button as shown in Figure: 6.3.1.
- Using the **QM color** icon as shown before, then hold down the **Ctrl** key and select color **108** and color **30.**



- Select the **OK** button to exit **Select All** dialog box. 
- Press **Enter** to end the selection.
- From the **Drive srfs Selection** dialog box click on the **Show** button to see the selected surfaces.

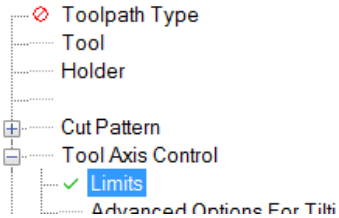




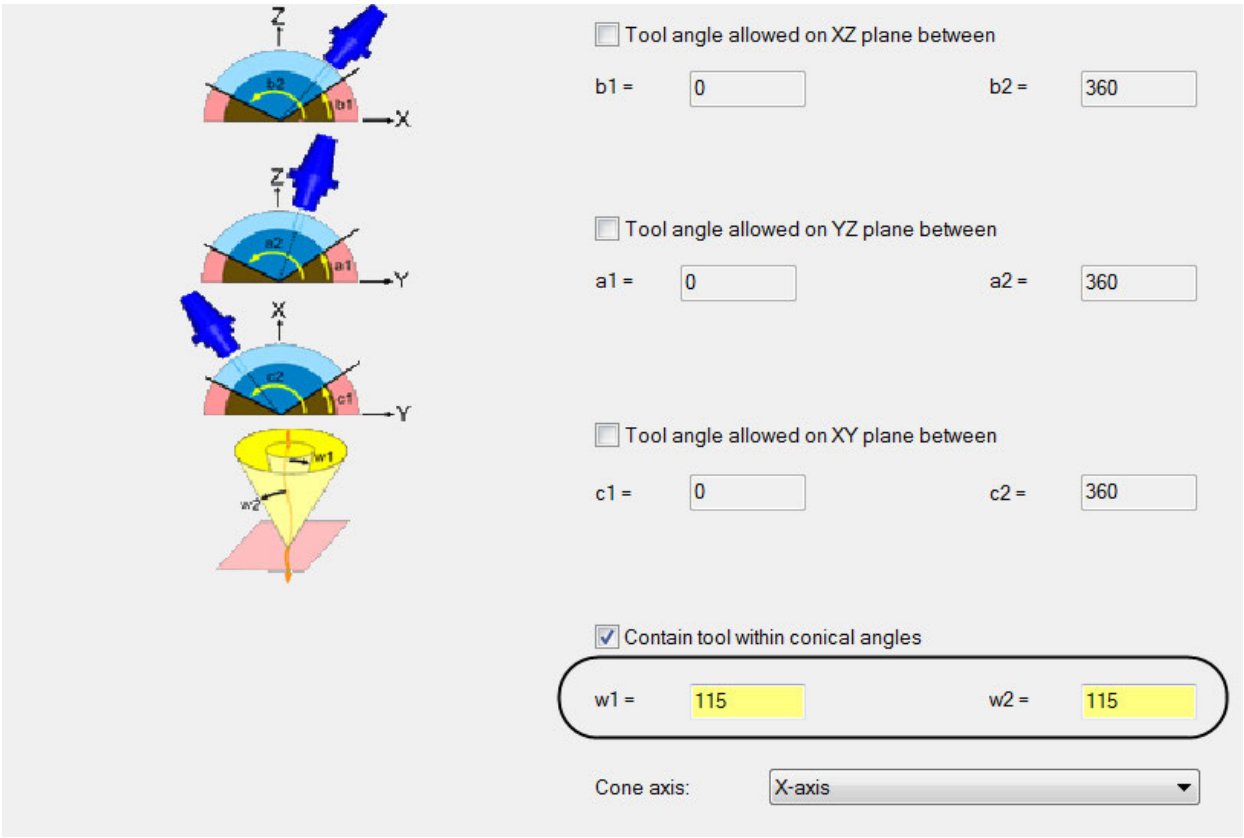
- Press **Enter** to continue.
- Select **Done** to exit the **Drive srfs Selection** dialog box.

Limits

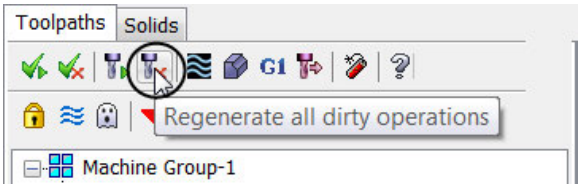
- From the **Tree view** area, expand **Tool Axis Control** and select **Limits.**



- From **Limits** change the **w1 and w2** to **100**.

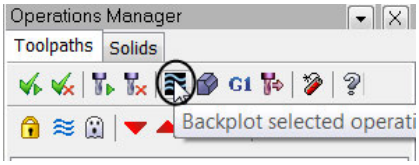


- Select the **OK** button to exit **Multiaxis Toolpath - Parallel to multiple curves**.
- Select **Regenerate all dirty operations** button to regenerate the 5-Axis toolpaths.

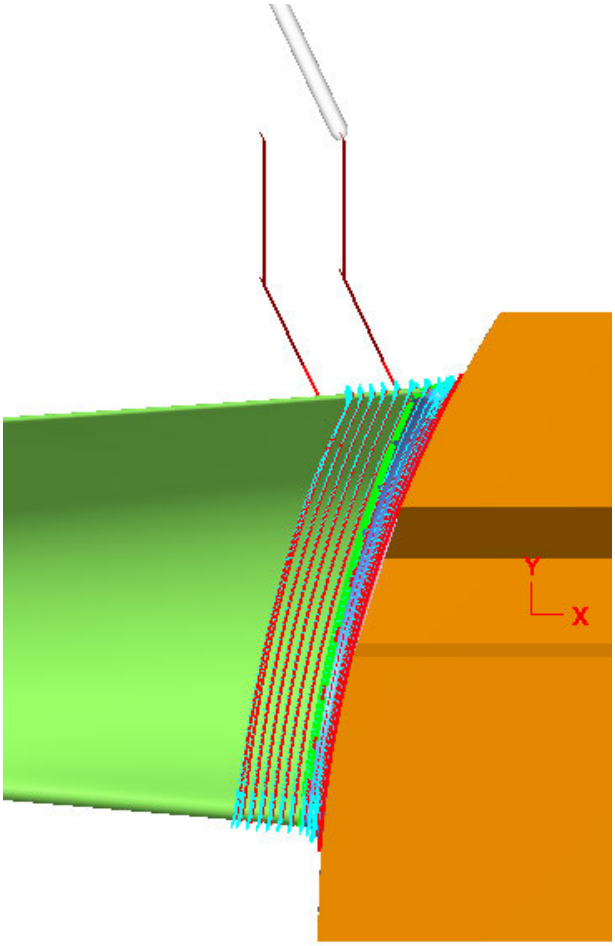


Backplot the toolpath

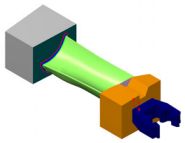
- Select the **Backplot selected operations** button.



- Select the **Play** button in the **VCR** bar to run **Backplot**.
- Press **Alt + S** to shade the surfaces if needed.
- The toolpath should look as shown.



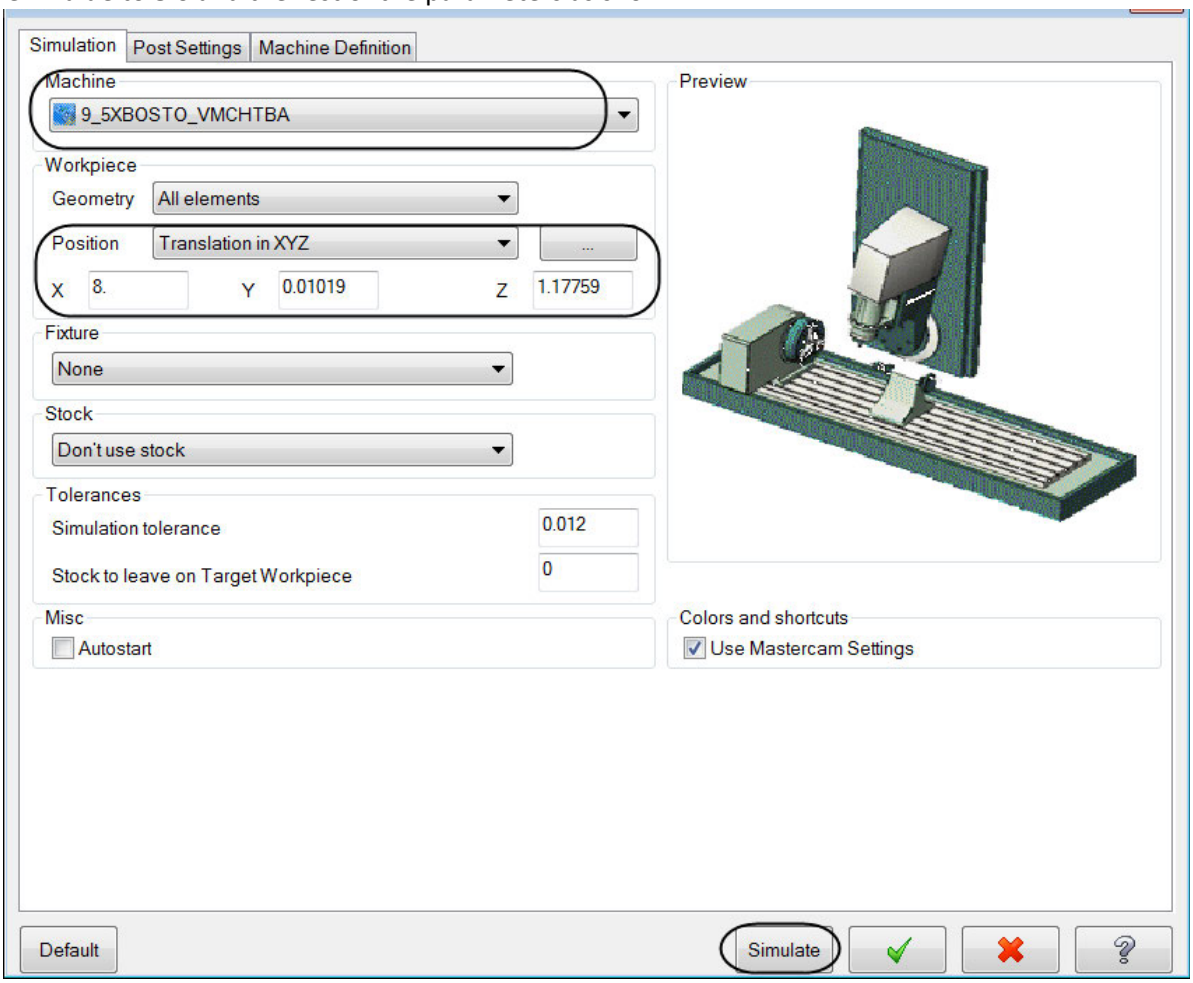
- Select the **OK** button to exit **Backplot.**



STEP 7: MACHINE SIMULATION

Machine simulation is an advanced verification module capable of backplotting, verifying stock removal, and full machine tool simulation with collisions check.
The machine simulation dialog box allows you to select the machine to be simulated, the work piece that will be shown in the machine simulation, the stock, the fixtures and the tolerances.

- From the **Toolpaths Operations Manager**, hold down the **Ctrl** key and select only the last three 5-Axis toolpaths.
- From the **Machine Simulation** toolbar, select the **Simulation startup settings** icon. 
- Select as **Machine** the **9_5XBOSTO_VMCHTAB.**
- As **Workpiece Geometry** select **All elements.**
- Click on the drop down arrow next to **Position** and select **Translation in XYZ** to set the location of the part.
- Change the **X** value to **8.0** and the rest of the parameters as shown.

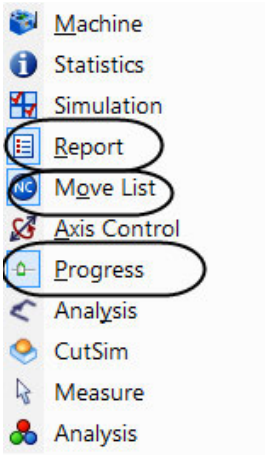


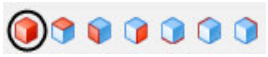
- Click on the **Simulate** button as shown above.

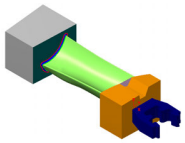
7.1 Setting the windows in the machine simulation

Settings

- **Windows.**
- Make sure that the following **Windows** are selected.

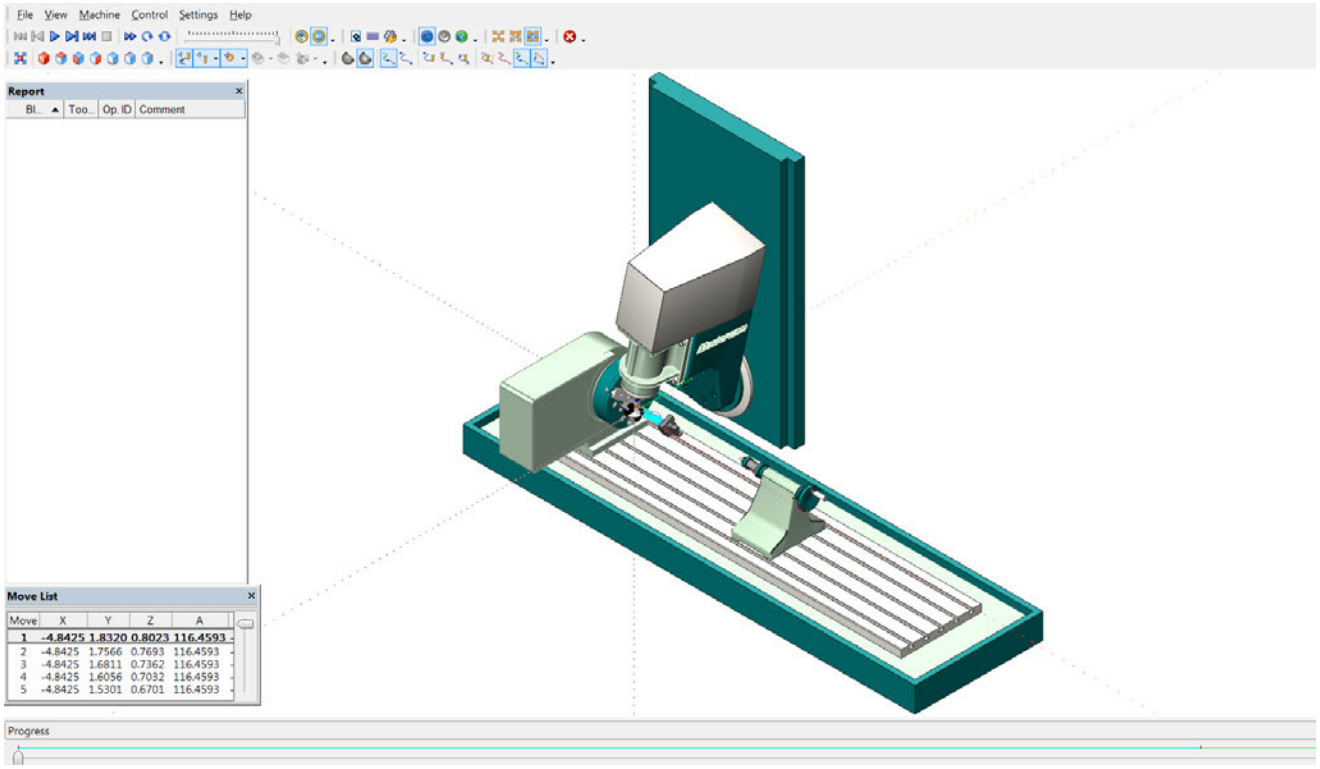


- Select the **Isometric** icon. 
- Select the **Fit** icon. 



NOTE: The windows can be resized and can be horizontally or vertically aligned and then docked at the desire location.
Figure: 7.1.1 shows you an example of how to set the windows.

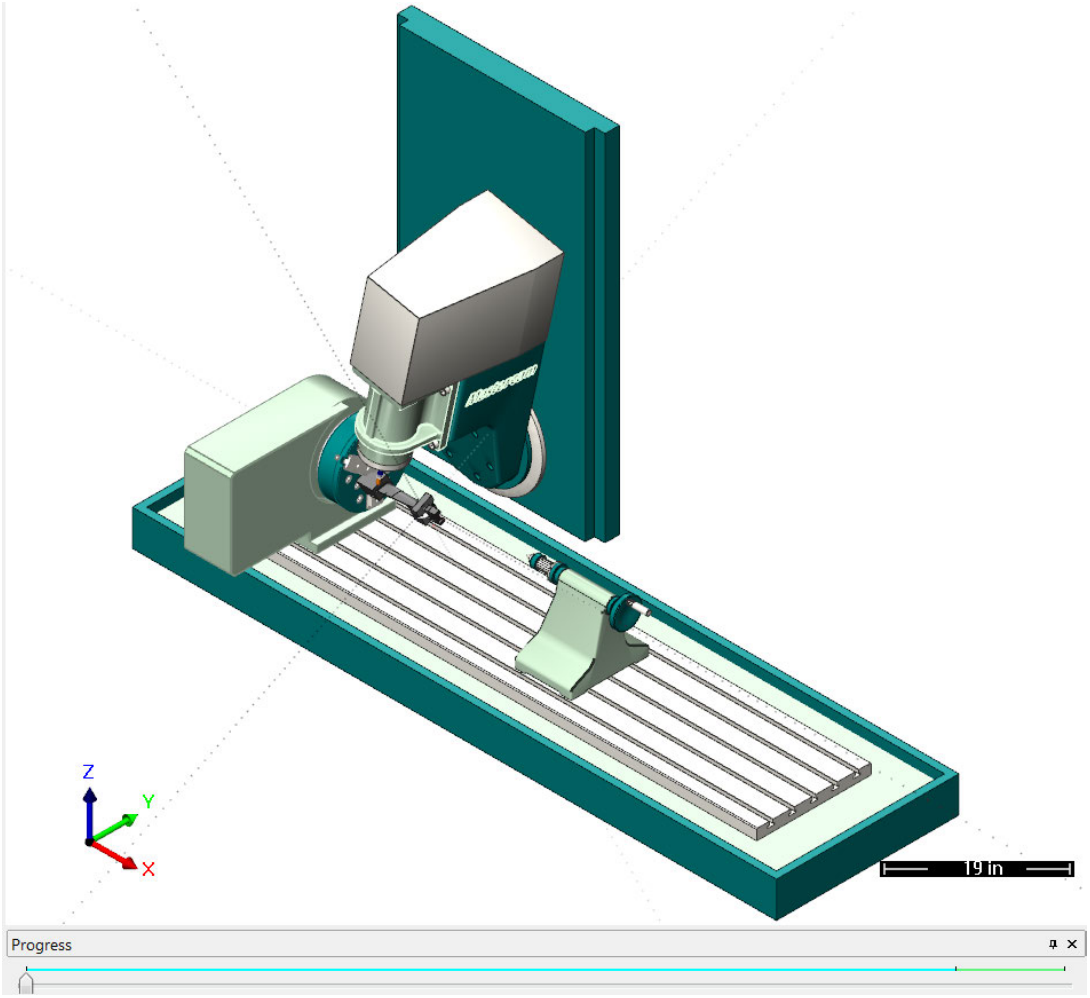
Figure: 7.1.1

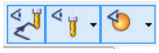


7.2 Simulate the toolpath in Toolpath Mode (backplotting)

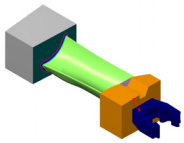
- Select as simulation mode, the **Toolpath Mode** icon. 
- Enable **Machine Focus**. 

- The machine with the part should automatically be loaded.

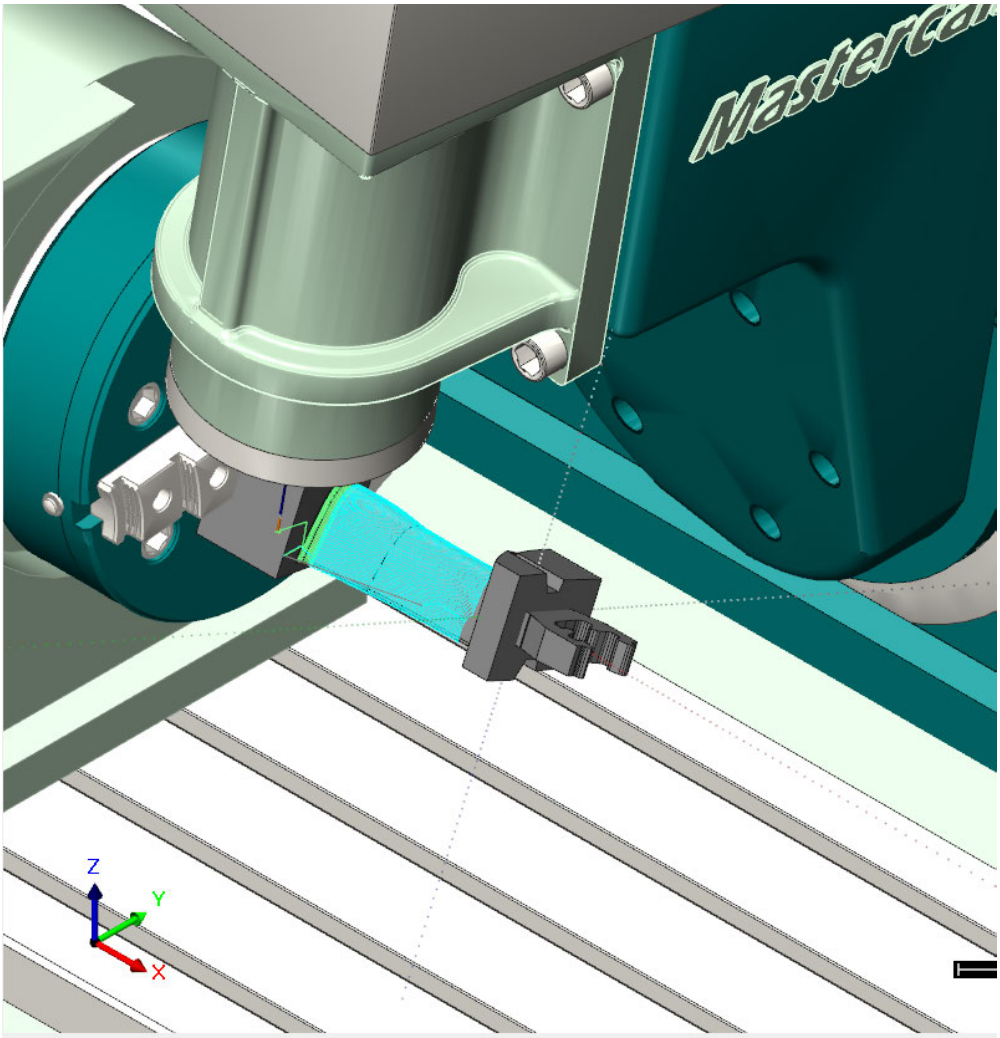


- From the **Visibility** toolbar, enable the **Toolpath**, **Tool** and **Workpiece** icons to see them displayed while running the simulation. 
- From the **Toolpath Rendering** toolbar make sure that the **Tool Tip**, **Display Current Operation**, **Follow**, **Leads** and **Links** icons are selected as shown.

- Move the cursor close to the center of the part and scroll down the mouse wheel to zoom in.
- Select the **Run** button in the **Control** bar to simulate the toolpath.



- The simulation should look as shown.



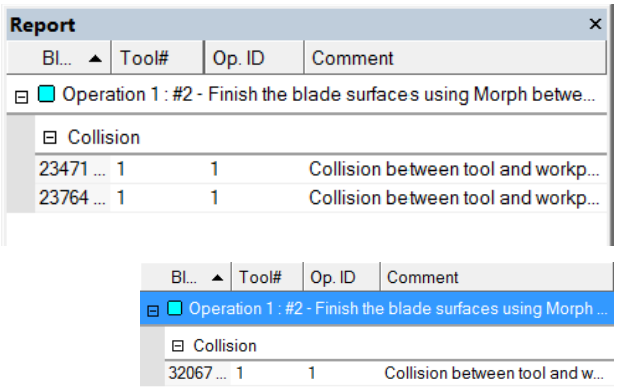
NOTE: To run the simulation and check the current position you can also move the slider to the left of the **Move list**.

Move List

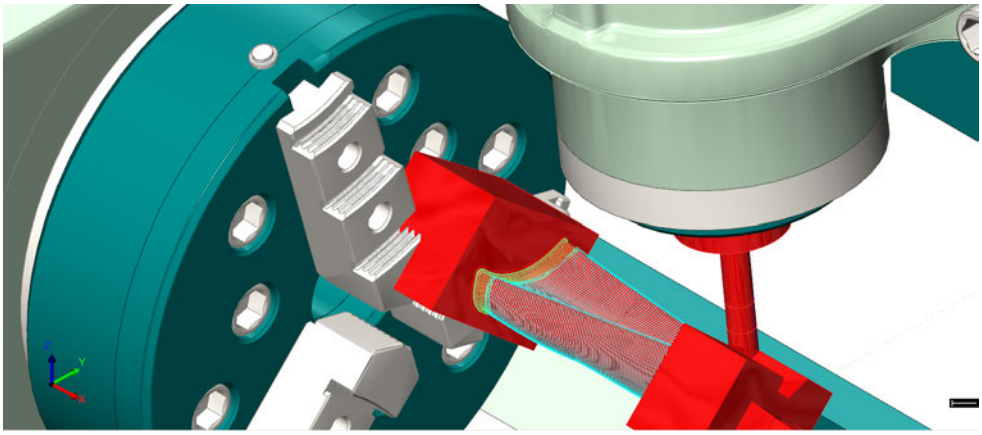
Blk	X	Y	Z	A	B
2319	-5.49	-0.69	-0.09	3503.20	-10.00
2320	-5.50	-0.70	-0.07	3504.41	-10.00
2321	-5.50	-0.70	-0.06	3506.67	-10.00
2322	-5.50	-0.70	-0.06	3508.36	-10.00
2323	-5.50	-0.70	-0.06	3510.05	-10.00
2324	-5.50	-0.71	-0.05	3512.29	-10.00
2325	-5.50	-0.71	-0.05	3514.78	-10.00
2326	**-5.50**	**-0.71**	**-0.04**	**3516.60**	**-**
2327	-5.50	-0.71	-0.04	3518.41	-10.00
2328	-5.50	-0.71	-0.04	3520.15	-10.00
2329	-5.50	-0.71	-0.03	3522.95	-10.00
2330	-5.50	-0.71	-0.03	3525.91	-10.00
2331	-5.50	-0.71	-0.02	3528.20	-10.00
2332	-5.50	-0.71	-0.02	3529.71	-10.00

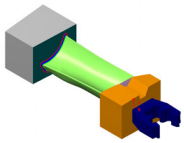
NOTE: The second operation has collisions between the tool and the workpiece reported in the **Report list** as shown in Figure: 7.2.1.

Figure: 7.2.1

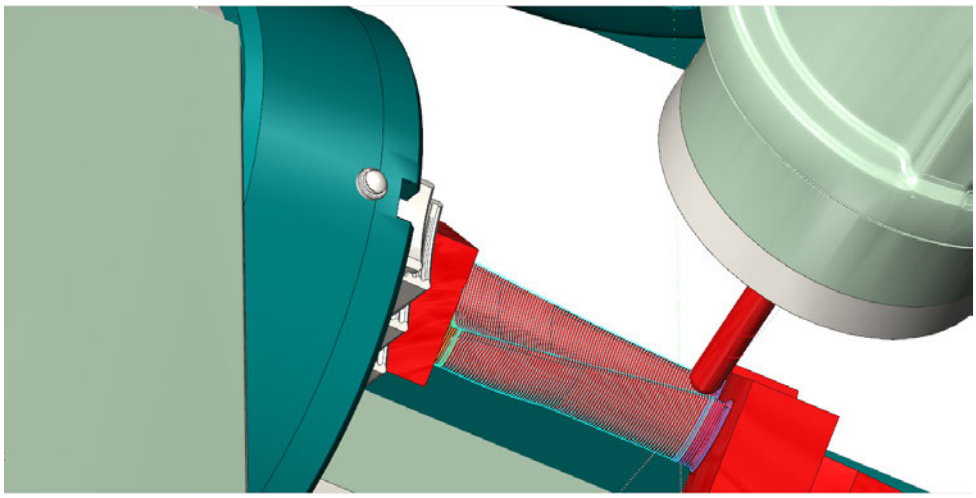


- Click on the first collision in the **Report** to check it.





- To rotate the part, left mouse click, hold down and drag it.

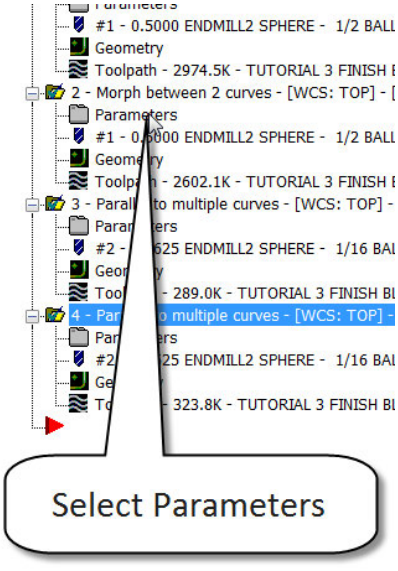


NOTE: To fix the problem you will need to increase the gradual tilt angle.

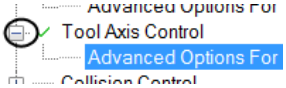
- To exit **Machine Simulation** select the **Exit** icon. 

STEP 8: MODIFY THE MORPH BETWEEN 2 CURVES TOOLPATH TO REMOVE COLLISIONS

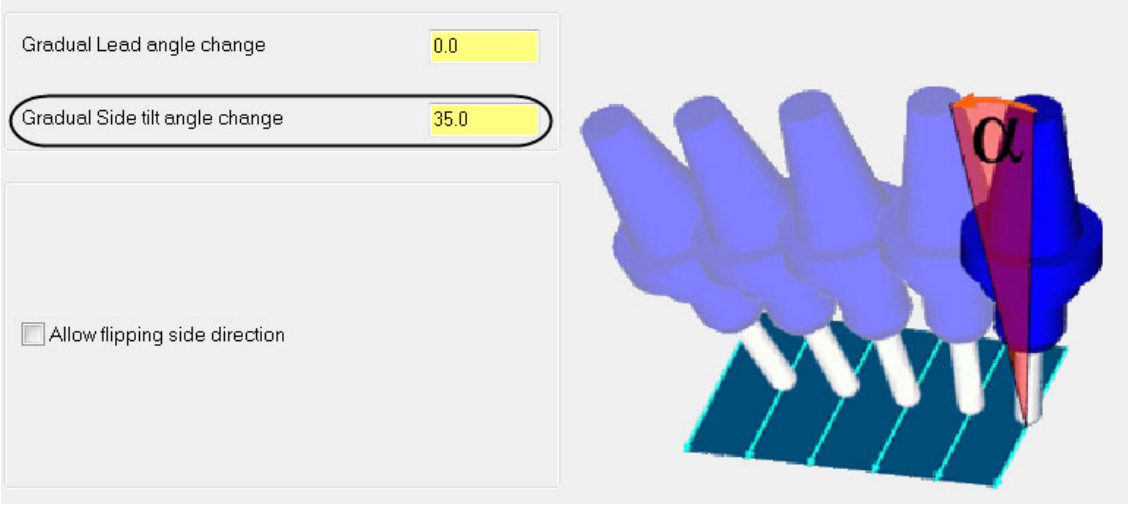
- From the **Toolpaths Operations Manager**, select **Parameters** in the **Morph between 2 curves** toolpath.



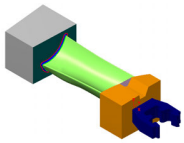
- From the **Tree view** area, expand **Tool Axis Control** and select **Advanced Options For Tilting.**



- Change the **Gradual Lead** angle change to **35.0** as shown.

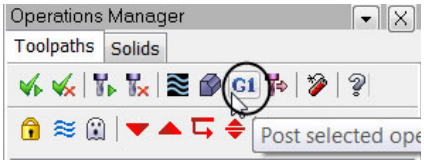


- Restart **Machine simulation** and run again the three toolpaths. No collisions should appear in the **Report** window.



STEP 9: POST THE FILE TO GENERATE THE NC CODE

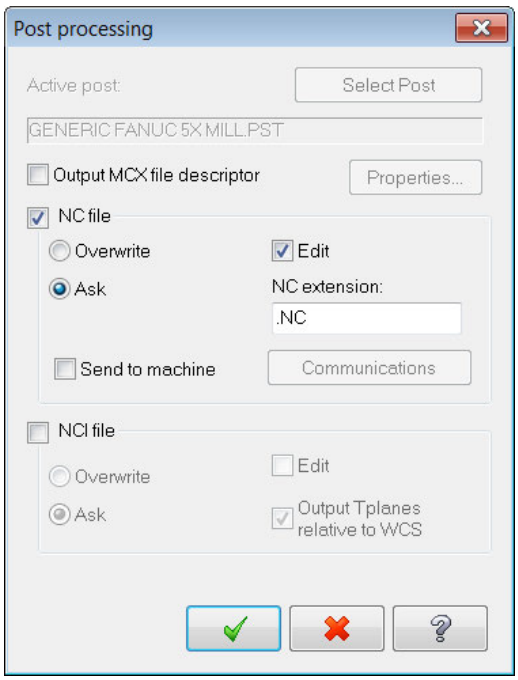
- Hold down the **Ctrl** key and select the last three toolpaths.
- Select **Post selected operations** button from **Toolpath Operations Manager**.



NOTE: The active **Post Processor** is a generic fanuc post processor. Please check the information about the post processor at page 1-3.

- In the **Post processing** window, if needed, make any necessary changes as shown in Figure: 9.0.1.

Figure: 9.0.1



- Select the **OK** button to continue.
- We are post process just the last 3 toolpath, please select **No** to post all operations warning.
- Enter the same name as the file name in the **NC File name** field.
- Save the NC file to continue.

- The NC file will appear on the screen as shown in Figure: 9.0.2.

Figure: 9.0.2

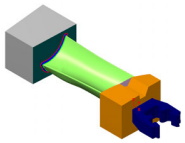
```
:0001 ( PROGRAM -  TUTORIAL 4 FINISH BLADE SURFACES )
N100 ( DATE -  04-10-10  TIME -  15:02 )
N102 G20
N104 G0 G17 G40 G80 G90 G94 G98
N106 G0 G28 G91 Z0.
N108 G0 G28 X0. Y0.
N110 (  1/2 BALL ENDMILL   TOOL - 1  DIA. OFF. - 1  LEN. - 1  DIA. - .5 )
( FINISH THE BLADE SURFACES USING MORPH BETWEEN 2 CURVES TOOLPATH. )
N112 T1 M6
N114 G0 G54 G90 X1.0154 Y-5.1335 C13.255 B-115.847 S1069 M3
N116 G43 H1 Z.2558
N118 X.6448 Y-5.0297
N120 Z-.0544
N122 Z-.4544
N124 G1 Z-.8544 F6.42
N126 X.5091 Y-5.0265 Z-.8402 C13.339 B-115.999
N128 X.326 Y-5.0218 Z-.8227 C13.467 B-116.323
N130 X.1566 Y-5.0176 Z-.8012 C13.535 B-116.62
N132 X.0326 Y-5.0183 Z-.752 C13.203 B-116.485
N134 X.0153 Y-5.0205 Z-.7248 C12.937 B-115.604
N136 X-.0244 Y-5.0297 Z-.6354 C11.97 B-114.694
N138 X-.0543 Y-5.0447 Z-.4924 C10.35 B-115.081
N140 X-.0702 Y-5.0503 Z-.4329 C9.685 B-115.881
N142 X-.1011 Y-5.0588 Z-.3294 C8.539 B-116.957
N144 X-.1176 Y-5.0632 Z-.2703 C7.905 B-118.557
N146 X-.1333 Y-5.0671 Z-.2146 C7.315 B-120.233
N148 X-.1513 Y-5.0703 Z-.1662 C6.825 B-122.063
N150 X-.1615 Y-5.0727 Z-.1271 C6.461 B-124.602
N152 X-.1891 Y-5.0744 Z-.0977 C6.252 B-126.883
N154 X-.223 Y-5.0758 Z-.078 C6.188 B-129.039
N156 X-.2631 Y-5.0767 Z-.0658 C6.247 B-131.12
N158 X-.311 Y-5.0769 Z-.0612 C6.445 B-133.138
N160 X-.3666 Y-5.0763 Z-.0622 C6.766 B-135.136
N162 X-.4444 Y-5.0738 Z-.0725 C7.394 B-137.89
N164 X-.5573 Y-5.0653 Z-.1224 C8.555 B-139.1
N166 X-.6852 Y-5.0516 Z-.1725 C9.888 B-140.746
N168 X-.8325 Y-5.0312 Z-.2207 C11.432 B-142.824
N170 X-1.0021 Y-5.0019 Z-.2624 C13.208 B-145.379
N172 X-1.0998 Y-4.9823 Z-.2818 C14.238 B-146.811
N174 X-1.2636 Y-4.945 Z-.3052 C15.969 B-149.197
```

NOTE: The G code that you have created will appear on the screen, if the code looks okay you can shut the window down without saving it. If you need to change the code, make sure you save it before you close the window down.
How the program is sent to the machine depends on the shop setup.

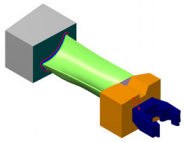
- Select the **red "X"** box at the upper right corner to exit the **Editor.**

STEP 10: SAVE THE UPDATED MCX-6 FILE

- Select the **Save** icon.



NOTES:



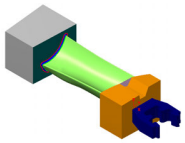
TUTORIAL #3 QUIZ

- How is the Parallel cuts toolpath generated?

- How is the Morph between curves toolpath generated?

- When would you use margins in a Multaxis toolpath?

- How does the Gradual side tilt angle work?



Thank you for previewing the

Mastercam X6 Advanced Multiaxis Training Tutorial.

We hope you like what you see. If you haven't already, be sure to check out the Table of Contents included at the beginning of this sample for a full listing of topics covered. If you found the first tutorial useful, we're certain the rest of the book will help you along your way to mastering Mastercam.

To thank you for considering our Mastercam Training Solutions,

>> We would like to extend a special offer to you.

Take 10% off your next purchase of any Mastercam X6 Training Tutorial (Imperial or Metric versions, Print or eBook formats) at our eMastercam Store using coupon code:

X6TRAIN

If you need some help deciding which title is right for you, email us at books@inhousesolutions.com. Likewise, feel free to drop us any suggestions of what you might like to see in future products.

>> Be in the loop!

Also consider signing up for the eMastercam.com Community Updates email list. You'll receive more offers like this and be the first to know about new product releases and free downloads.

Sign me up!

Other Available Mastercam Training Solutions:

Training Tutorial Series
Mill Level 1 (2D) | Mill Level 3 (3D) | Lathe | Solids | Beginner | Design | Router | Art | Intro to Multiaxis
Advanced Multiaxis | Wire | Indexing | Project Workbook

Instructor Kits for the Training Tutorial Series
Instructor Kit (for Mill Level 1 & Mill Level 3) | Instructor Kit for Mill & Lathe (Mill Level 1, Mill Level 3 & Lathe)

Metric Training Tutorial Series
Mill Level 1 | Mill Level 3 | Lathe | Solids | Beginner | Design | Router | Art | Intro to Multiaxis
Advanced Multiaxis | Wire | Indexing

Professional Courseware Series
Mill Level 1 | Mill Level 3 | Lathe | Multiaxis | Router

Handbook Series
Volume 1 | Volume 2 | Volume 3 | Volume 1 Learning Kit | Volume 1 Curriculum | Volume 2 Curriculum

CNC Courseware
Single Set | Site License | Premium Site License

Multimedia
Mill Level 1 | Mill Level 3 | Lathe | Intro to Multiaxis | Handbook Video Series

To order: contact your local reseller or visit www.emastercam.com

Mastercam X6 TRAINING TUTORIAL SERIES

About the Author:

Mariana Lendel has a Bachelor Degree in Mechanical Engineering and over ten years of experience as a CNC programmer. For the past decade, Mariana has been a Mastercam instructor at In-House Solutions where she teaches Mill Level 1, Level 3 and Multiaxis; Lathe, Wire EDM, Router, Solids and Art. Her trainees include members of the industrial workforce as well as instructors from High Schools and Colleges across Canada. Mariana and her team continue to develop the most extensive collection of Mastercam Training Solutions available.

In-House Solutions
240 Holiday Inn Drive, Unit A
Cambridge, ON, Canada N3C 3X4
T 800-529-5517 | F 519-658-1335
www.inhousesolutions.com

Advanced Multiaxis Tutorial

ISBN 926955-36-0-4
9 789269 553601