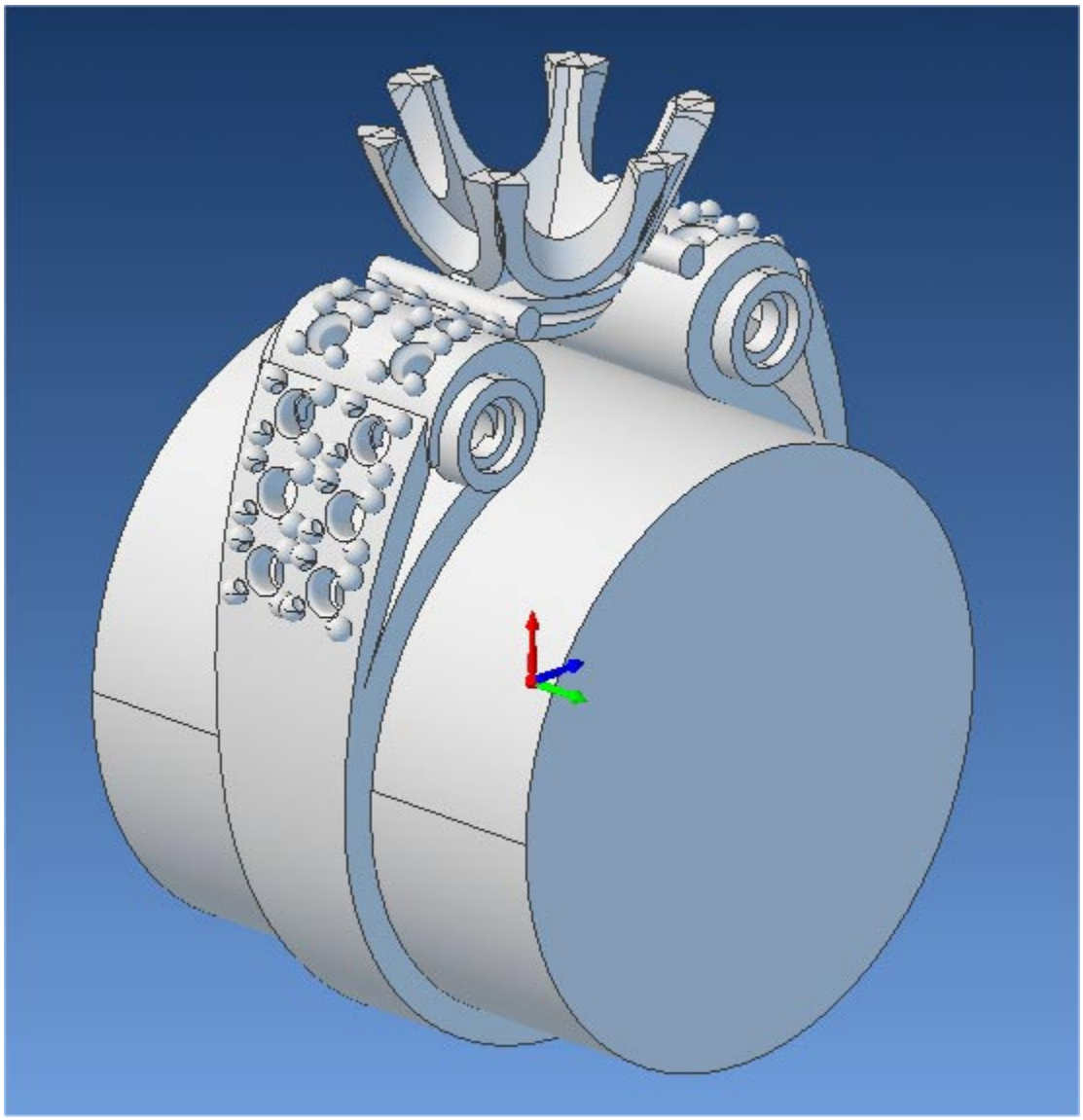
Tutorial 11: Machining a Ring



Introduction

This tutorial will illustrate machining this Ring using 4 axis-milling operations.

This tutorial will introduce the usage of several 4-axis operations such as 4 axis roughing and finishing.

The stepped instructions are accompanied by explanatory and introductory text. Reading this text will help you understand the tutorial methodology and provide information about additional options available.

Don’t forget to save your work periodically! You may want to save the file under a different name so that the original file will be preserved.

Strategy to Machine the ring

- We will machine the ring completely using 4 axis-machining operations.
- The part itself will be machined out of a cylindrical blank.
- The stock will be held to the machine table using a rotary chuck.
- The part will be machined using 0.125”, 0.0625”, and 0.03125” ball end mills.

Main Programming Steps

In creating programs for each setup, the following steps will be followed:

1. Create the Stock geometry
2. Set the Machine zero point respect to the machine coordinates.
3. Set the rotary axis and rotary center.
4. Create / Select the tool used for machining
5. Set the feeds and speeds
6. Set the clearance plane for the non-cutting transfer moves of the cutter
7. Select the machining operations and set the parameters
8. Generate the toolpath
9. Simulate the toolpath.

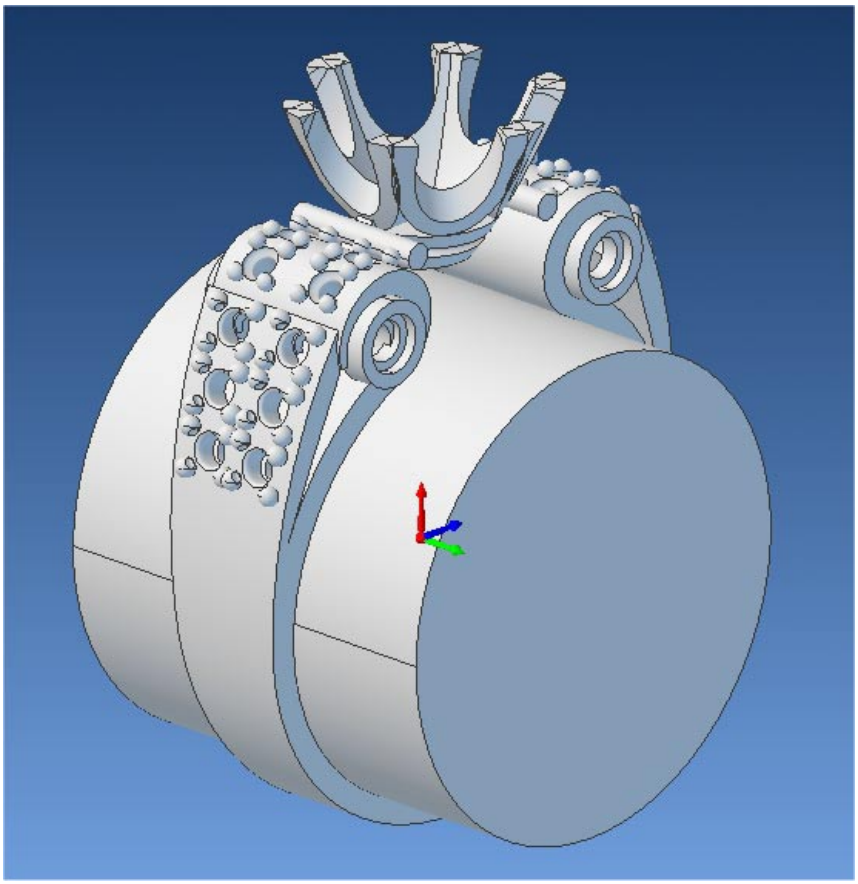
You may have to repeat either all or part of these steps for subsequent operations.

Loading the Part Model

“Part” refers to the geometry that represents the final manufactured product. Typical you would create this in Alibre Design. Use the Alibre Design menu bar or the Standard toolbar buttons to create, load and save part geometry.

1. Select File / Open Part, or click the Open Part icon from the Alibre Design standard toolbar.
2. From the **Open** dialog box, select the **RingExample_1.AD_PRT** file from the **Tutorials** folder in the Alibre CAM installation folder. (Default location C:\Program Files\MecSoft Corporation\Alibre CAM 2.0\Tutorials)

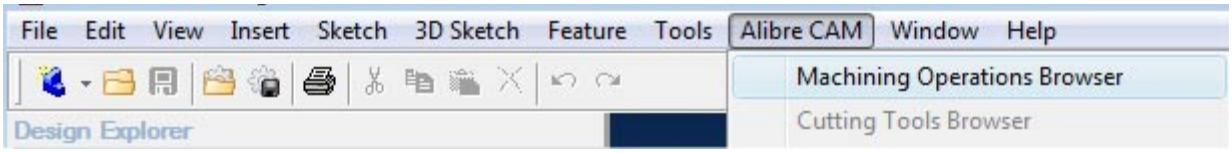
The loaded part appears as shown below.

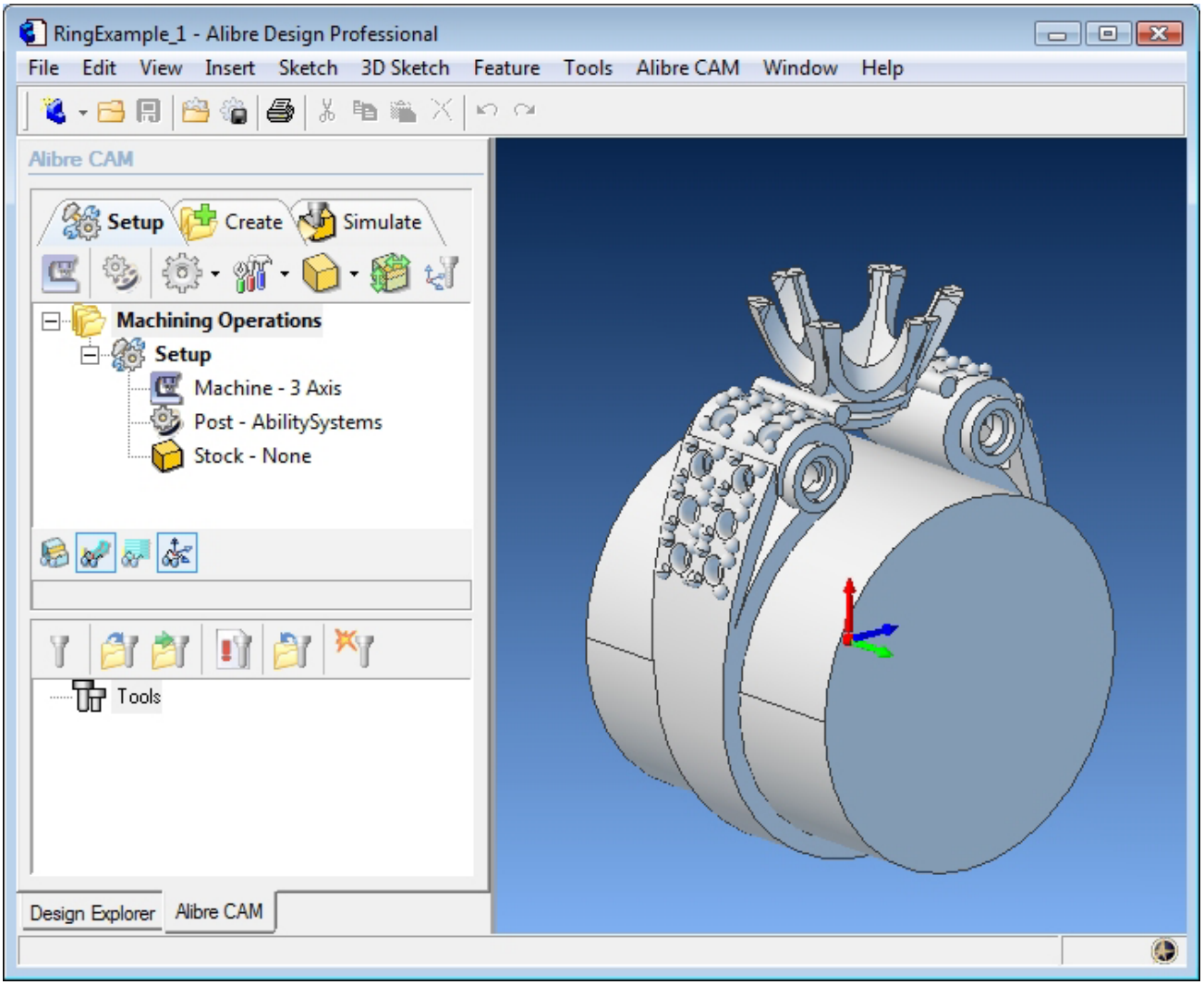


Note: You must work in shaded mode in order to be able to visualize toolpaths created in Alibre CAM. It is suggested for best visual performance with Alibre CAM to work with only one view port open and the view port operating in shaded mode.

Loading the Alibre CAM Browser

1. Select Alibre CAM from the menu bar and click Machining Operations Browser.





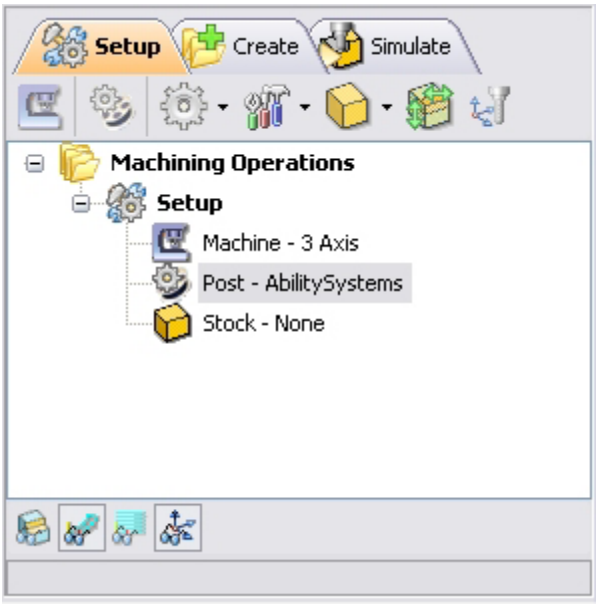
The Alibre CAM browser (MOPs and Cutting Tools) is now loaded and is docked over the Design Explorer. You can toggle between the MOPs browser and the Design Explorer from the tabs at the bottom of the window. It can be hidden by un-checking Browser on the Alibre CAM menu bar. To re-display the hidden Browser window, you can re-check **Browser** from the Alibre CAM menu entry. You can also resize it by dragging one of its sides.

Preparing the part for Machining

The Setup tab allows the user to specify Machine Setup, Select Post Processor, Stock Geometry, Machine Coordinate System (Machine Zero) & Preferences.

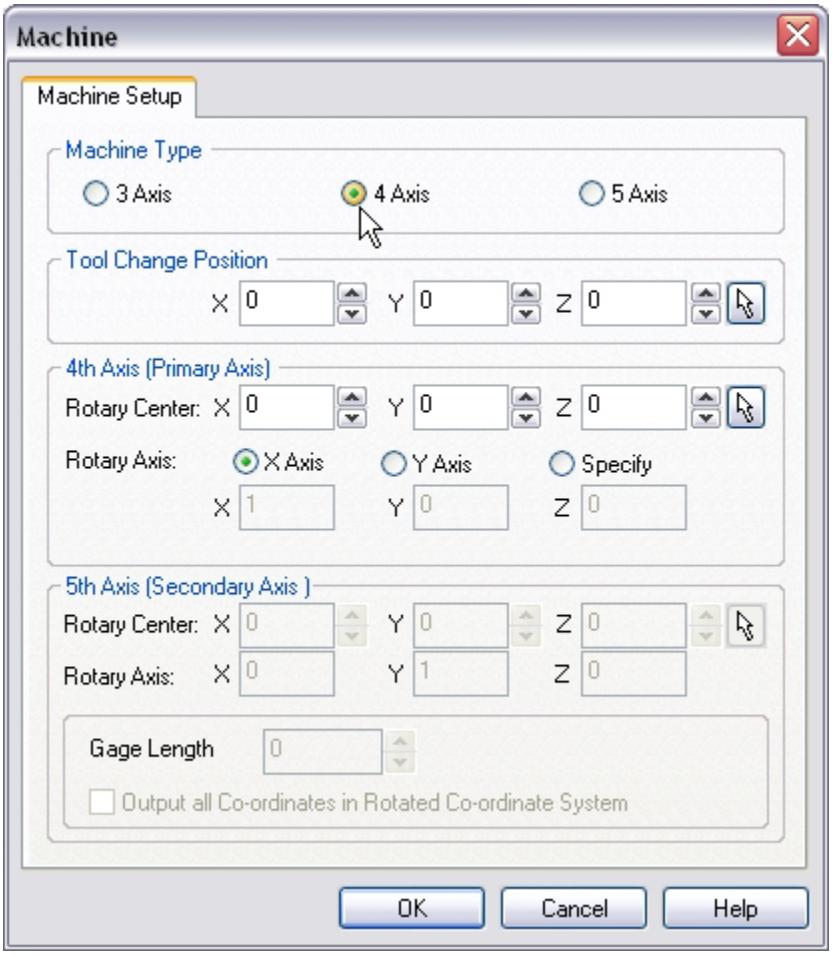
Setup Tab

1. Go to the Alibre CAM MOps browser and click on the Setup tab



2. Select Machine Setup from the setup tab.

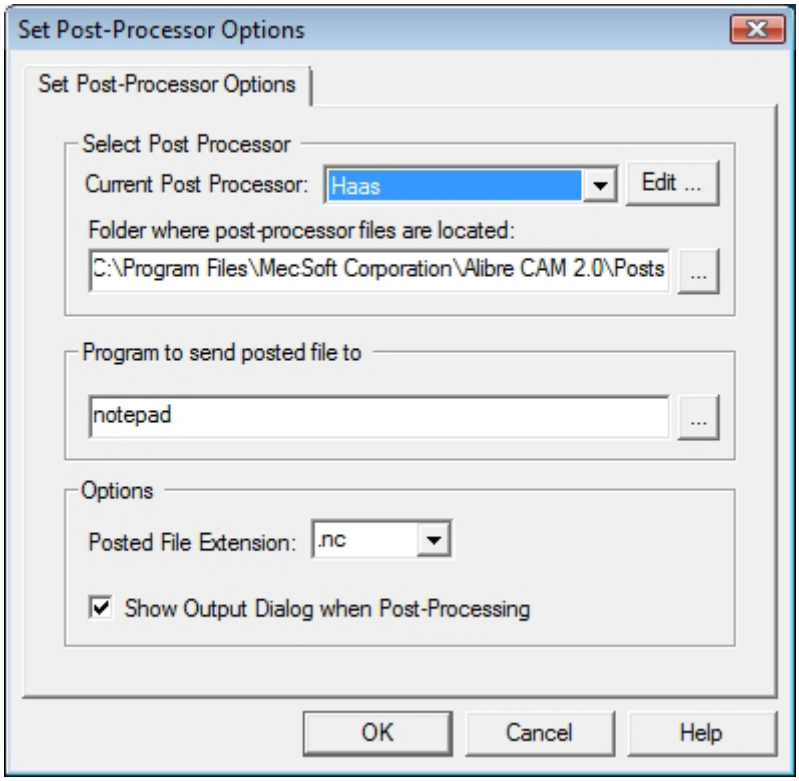
3. Set the Machine type to 4 Axis and Rotary Axis to X Axis. For most controllers rotation along X represents A axis and rotation along Y represents B axis. We will set the Rotary Center once we determine the Machine Zero.



4. Select **Post** from the setup tab to specify the post processor options

5. Set the current post processor that is on your controller. We will select Haas as the post processor for this exercise. Set the posted file extension type to .nc

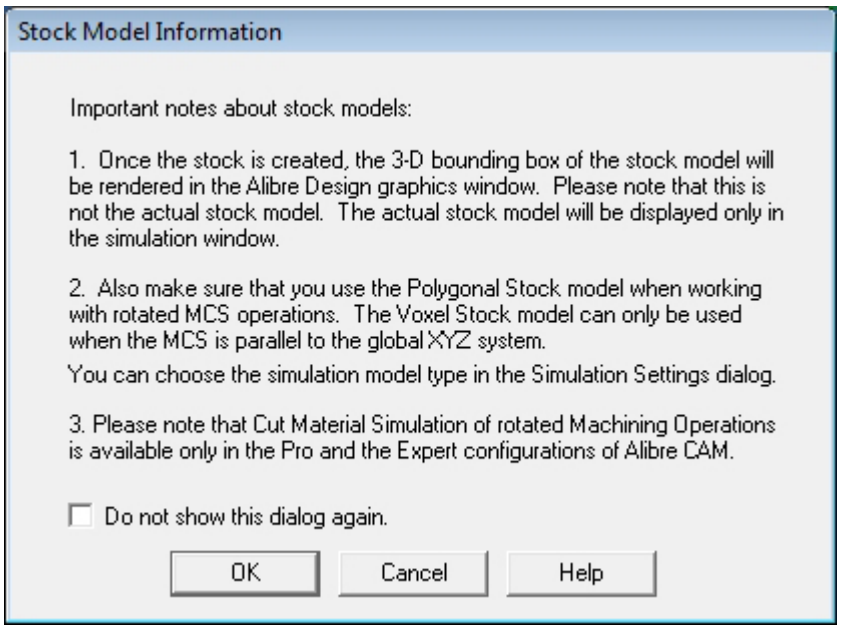
Note: By default post processor files are located under
C:\Program Files\MecSoft Corporation\Alibre CAM 2.0\Posts
The program to send the posted output is set to notepad. This would output the G code to a notepad.



Create Stock Geometry

1. Select Create/Load stock from the setup tab and create a Cylinder Stock.

The stock model information dialog may be displayed when a stock geometry is created.

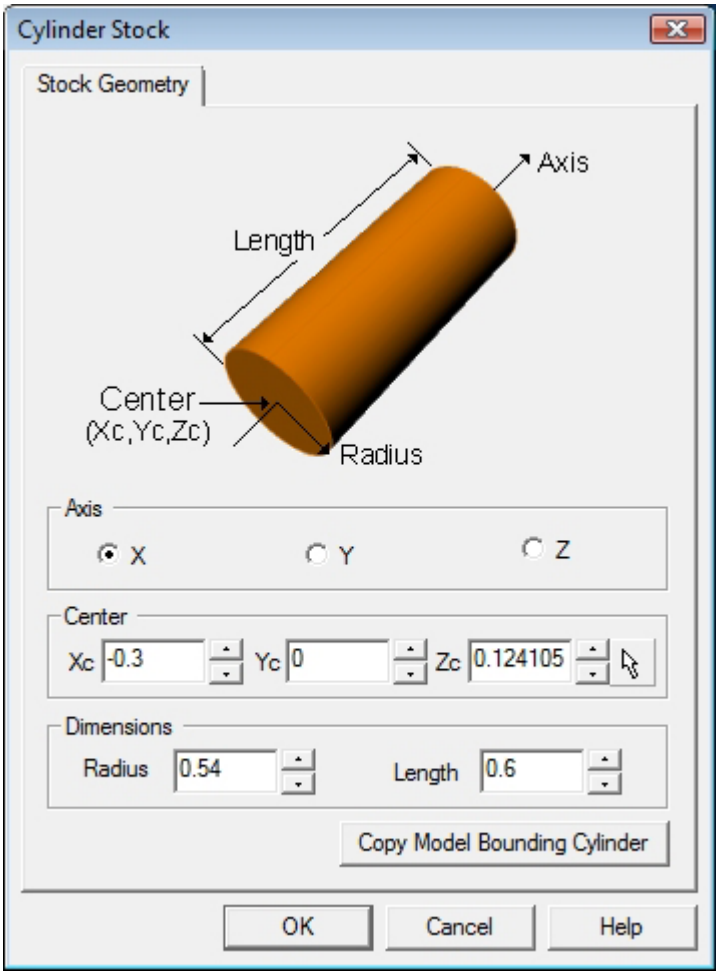


Click OK

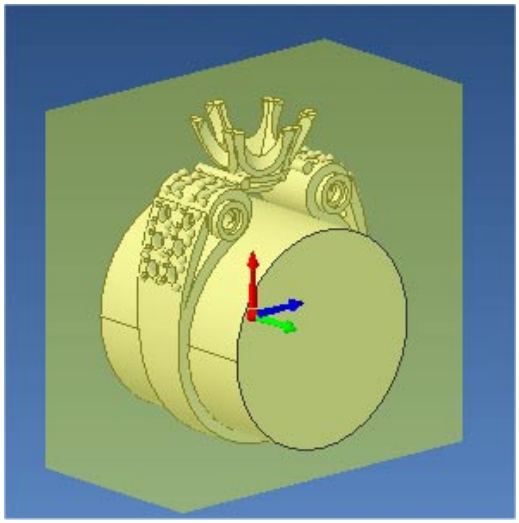
User can turn off this dialog by selecting Do not show this dialog again located on the bottom of the message window.

To display this dialog during stock creation, select Alibre CAM Preferences->Simulation Preferences and select Invoke 'Stock Model Information' dialog.

2. This brings up the Cylinder Stock parameters. Set the Axis (rotary) = X, Radius = 0.52" and Length (L) = 0.6. Leave the other parameters as default and click OK.

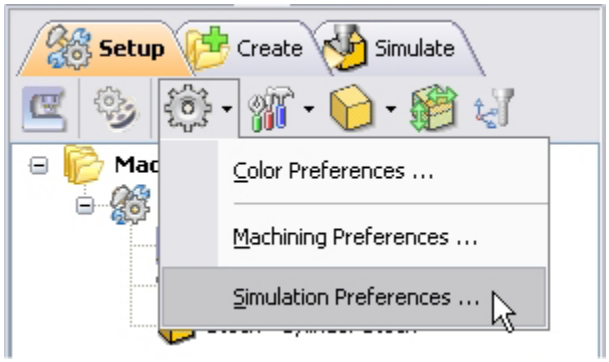


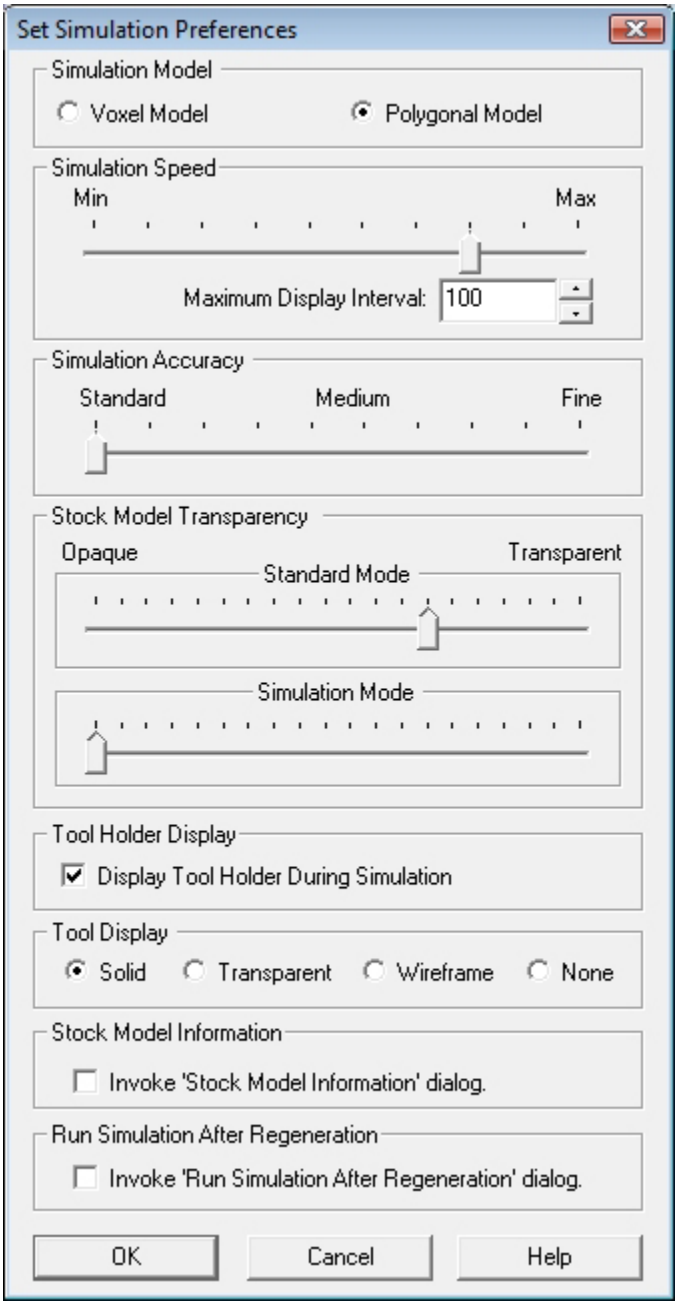
3. The stock geometry is now created, and a semi-transparent stock is displayed on top of the part geometry.



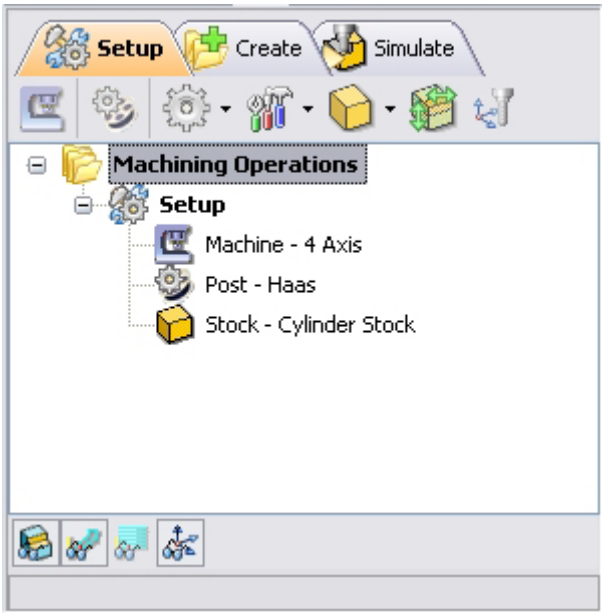
Note: The stock model created in Alibre Design Graphics window is a 3d bounding box of the stock dimension. The actual stock model will be displayed only in the stock simulation window.

4. You must switch the simulation model to Polygonal model to run 4 axis simulations. Select Preferences->Simulation Preferences from the Setup Tab and switch the simulation model to Polygonal if set to Voxel.





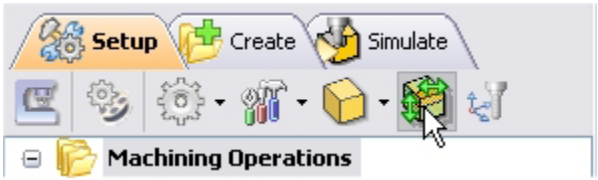
The setup tab now displays the following information: Machine Type, Post Processor, and Stock type as show below.



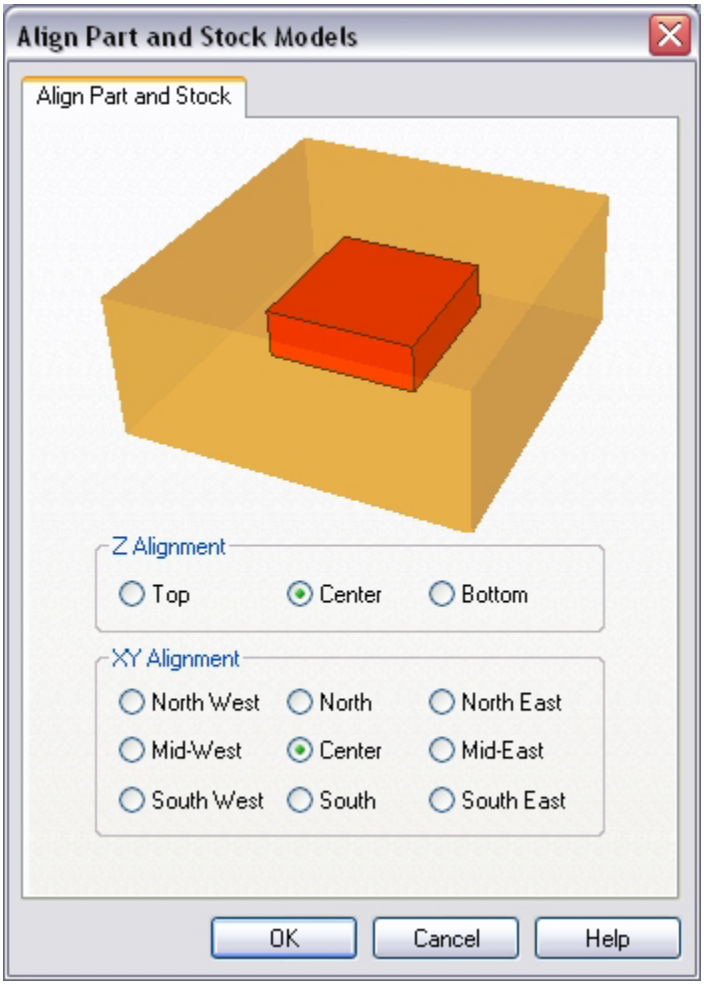
Align Part and Stock

Once the stock model is created, user can move the stock geometry relative to the part geometry and use the stock box to specify the machine zero (home position).

1. Select Align Part and Stock from the Setup tab



2. Set Z alignment to **Center** and XY alignment to **Center.** (This would align the stock to the top of the part in Z and center in XY)

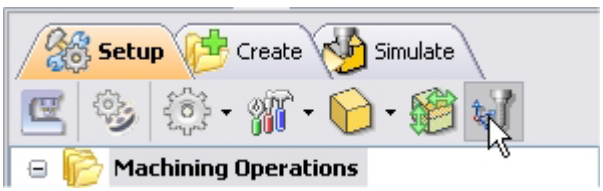


Set Machine Coordinate System (MCS)

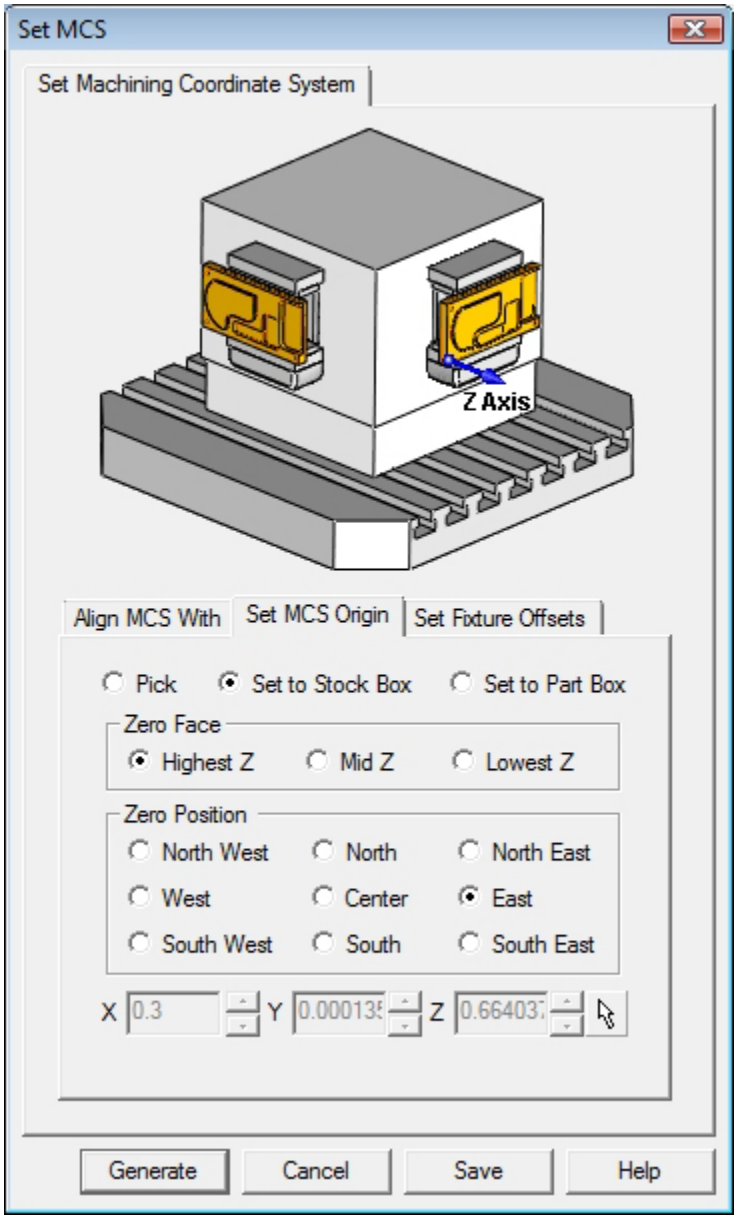
The steps below help you determine the machine home (also known as machine zero or tool touch off point) for the part/stock geometry.

The MCS is represented as a triad with 3 arrows, the Green arrow indicating X axis, Blue the Y axis and Red the Z axis. It is always a good practice to set the machine zero before generating a toolpath.

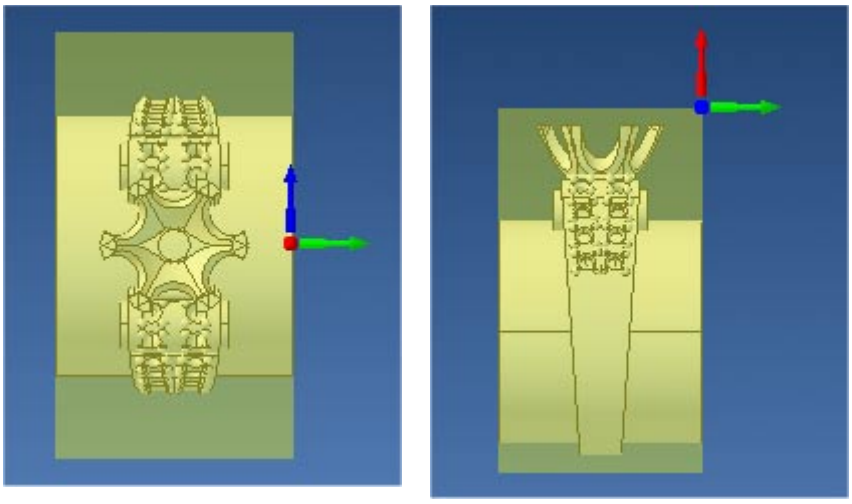
3. Select Set MCS from the Setup tab



4. Switch to SetMCS Origin tab and choose **Set to Stock Box**, the Zero Face to **Highest Z,** and Zero Position to **East** corner. This sets the machine home to the top of the stock material and the right most edge of the part geometry.



Click Save As to save the work and specify a file name as RingExample_1-Rev1.



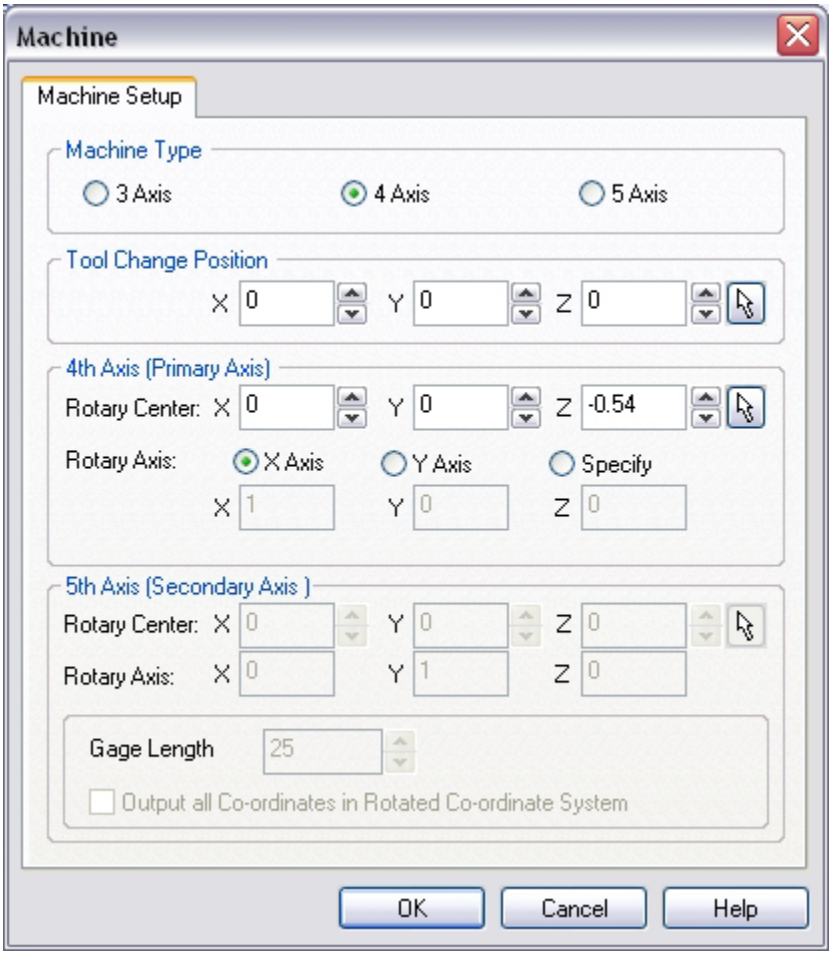
Note: You can change the stock model transparency under standard mode by selecting Simulation Preferences that is located at the bottom of the MOps browser.

Specify Rotary Center

In this step we will determine the rotary center for the part geometry. The rotary center must pass thro' the entire part geometry. Alibre CAM will not compute a toolpath if the part/feature is below the rotary center as this is considered as an undercuts in the part.

1. Select Machine Setup from the setup tab.

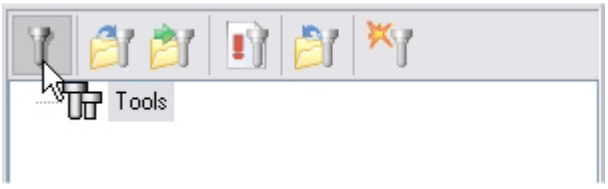
2. Set the rotary center in X =0, Y = 0 and Z = -0.54 which is the center of the stock geometry.



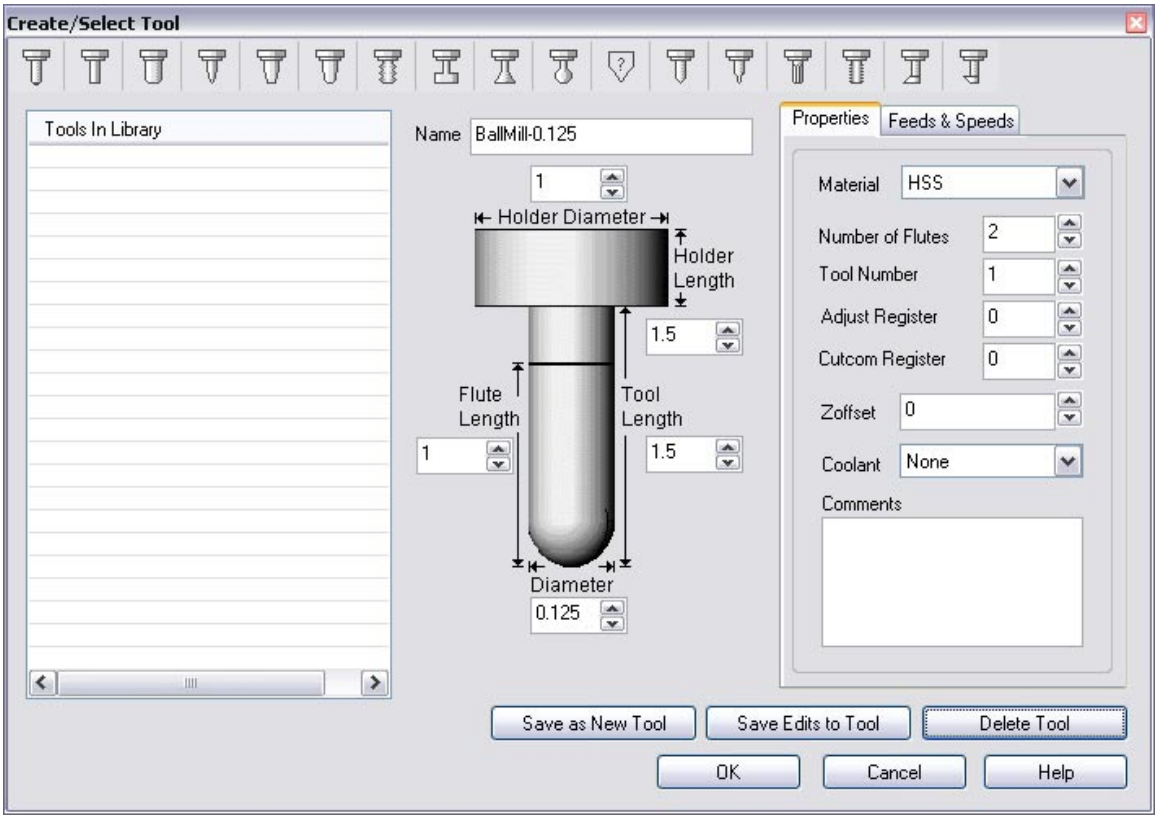
Create Tools

To machine the above part, we will now create a 0.125”, 0.0625” and 0.03125” Ball End Mills.

1. Go to the Cutting Tools browser that is located below the Alibre CAM MOps browser and select Create/Edit Tools. Select the Tool Type to Ball End Mill.



2. Set the tool name as BallMill-0.125 and Tool Diameter = 0.125. Under the Properties tab, set Tool Number = 1.

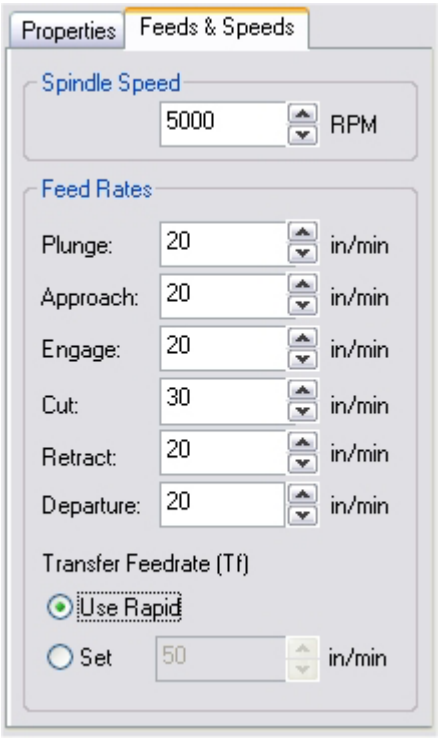

Create/Select Tool
Tools In Library
Name
BallMill-0.125
1
Holder Diameter
Holder Length
1.5
Flute Length
Tool Length
1
1.5
Diameter
0.125
Properties
Feeds & Speeds
Material
HSS
Number of Flutes
2
Tool Number
1
Adjust Register
0
Cutcom Register
0
Zoffset
0
Coolant
None
Comments
Save as New Tool
Save Edits to Tool
Delete Tool
OK
Cancel
Help

Setting Feeds and Speeds

You can assign Feeds & Speeds to a tool or you can load from a table. In this exercise, we will assign feeds and speeds to the tool.

3. Switch to the Feeds & Speeds tab inside the create/select tool dialog.

4. Use the following settings for feeds and speeds.



5. Click **Save as New Tool**. The tool is now created and listed under Tools in Library.

Note: You can edit the tool properties and click Save Edits to Tool to save the changes. You can create additional tools by assigning a different name and specify the tool parameters.

6. Create a 2nd Ball End Mill with the following parameters.

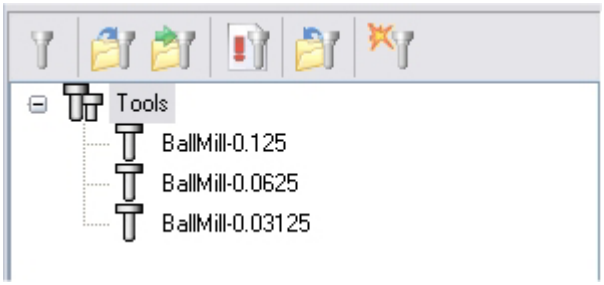
    a. Tool Name: **BallMill-0.0625**, Tool Number = **2**.

    b. Switch to Feeds & Speeds tab set Spindle Speed = 5000 rpm, plunge, approach & engage feed = 20 ipm, cut feed = 30 ipm, retract and departure feeds = 20 ipm. Set the Transfer Feedrate to Use Rapid.

    c. Click Save as New Tool.

7. Create a 3rd Ball End Mill with the following parameters
    a. Tool Name: **BallMill-0.03125**, Tool Number = 3.
    b. Switch to Feeds & Speeds tab set Spindle Speed = 5000 rpm, plunge, approach & engage feed = 20 ipm, cut feed = 30 ipm, retract and departure feeds = 20 ipm. Set the Transfer Feedrate to Use Rapid.

The created tools are now listed under the Cutting Tools browser.



Create Machining Operations

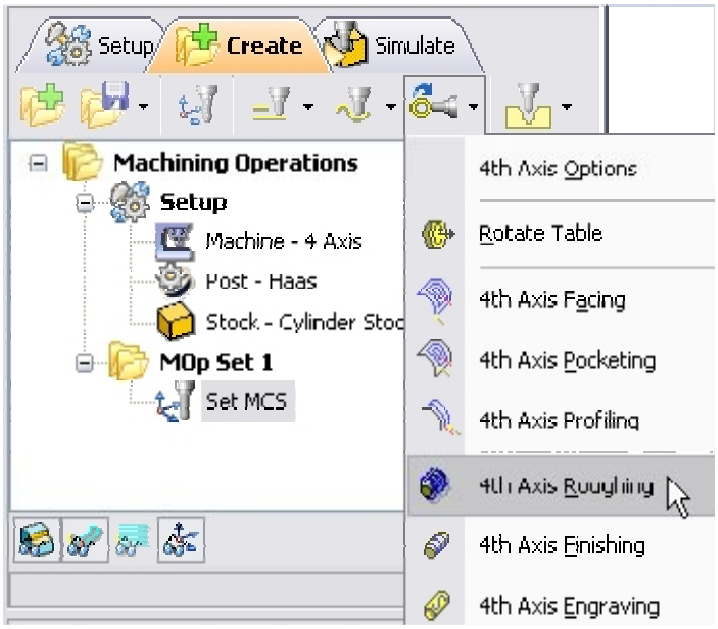
We will machine the ring using 2 different machining operations – 4 axis Roughing and Finishing.

The first step in machining the ring will be a roughing operation. In this cut method, the tool cuts the stock in successive levels. The spacing between these levels are specified by the user. This type of machining is very efficient for removing large volumes of material and is typically performed with a large tool. Roughing is typically followed by semi-finishing or finishing toolpaths.

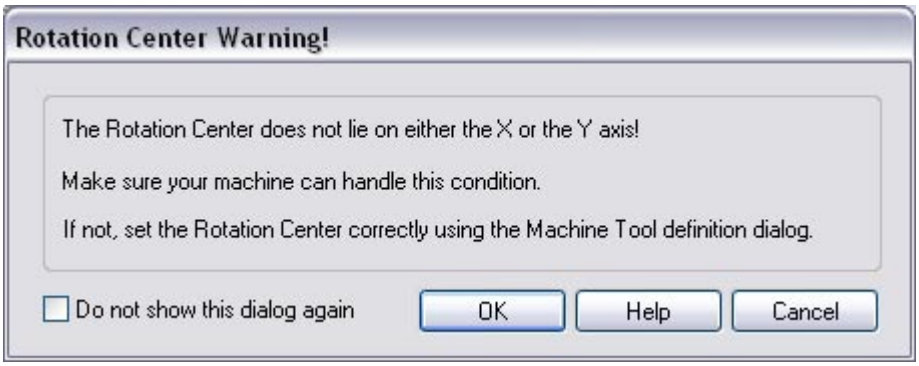
Switch to the Create Operations tab in Alibre CAM Mops browser.

4 axis Roughing

1. Select 4 Axis Milling and choose Roughing.



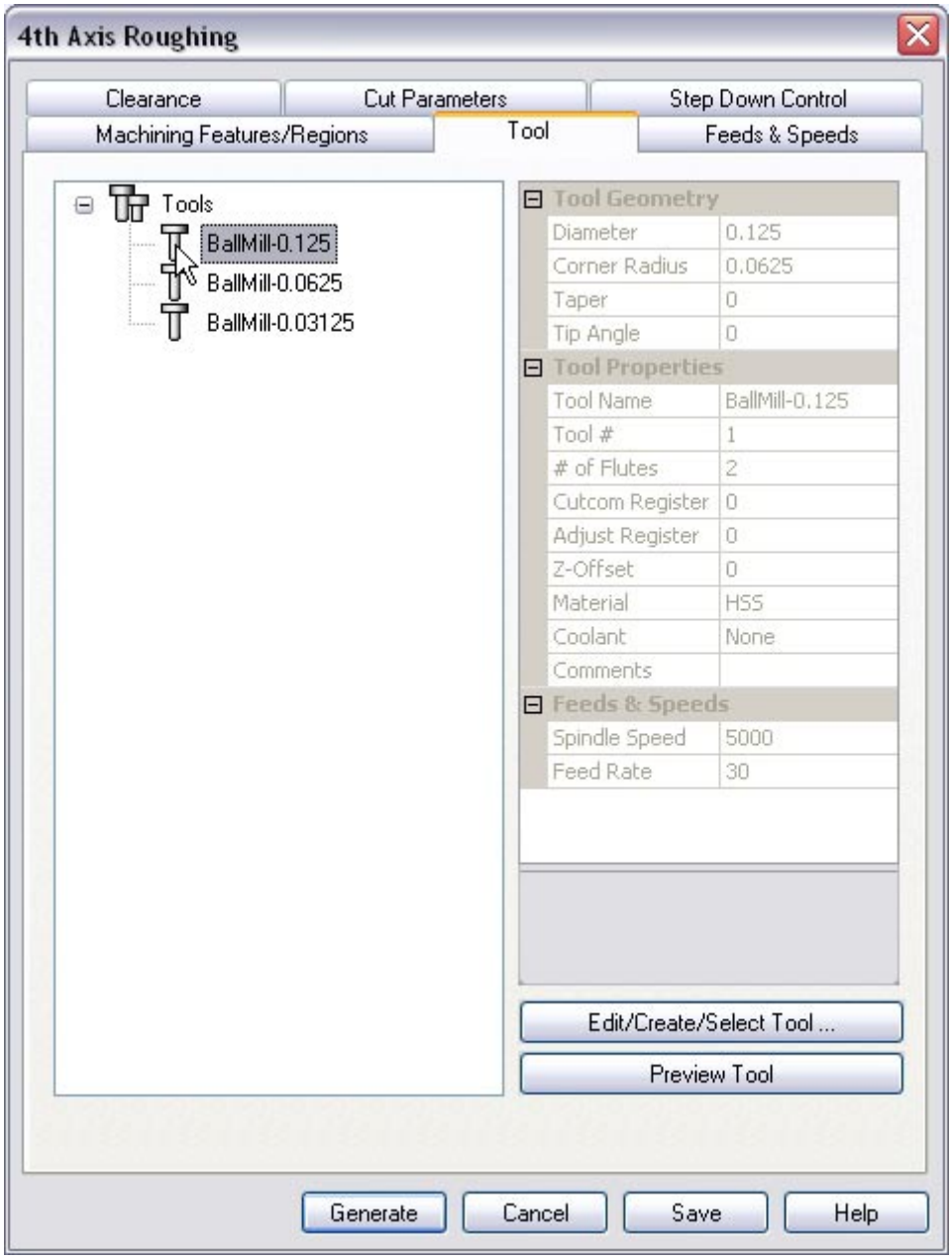
If the rotary center is not set to the same location as the Machine Zero, a warning message dialog would be displayed at all times when a 4 axis machining operation is selected. Users can override this message by clicking OK in the dialog.



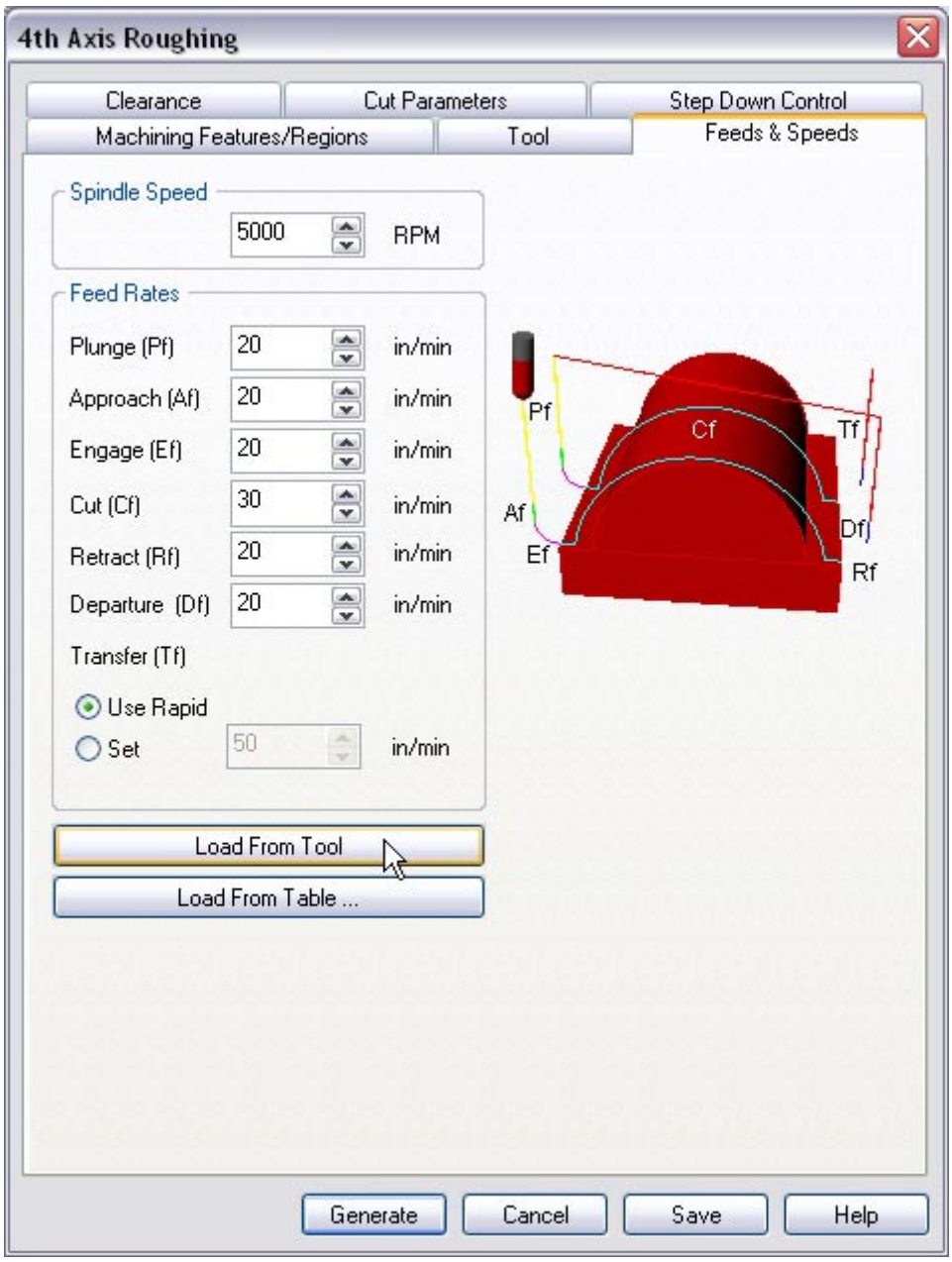
Note: You can check Do no show this dialog again to stop the warning message appearing again when you create/edit a 4 axis machining operation.

2. This brings up the 4 Axis Roughing Operation Dialog. We will now go over the steps for creating the toolpath.

3. Switch to the Tools tab inside the 4th axis Roughing operation and select BallMill-0.125.


4th Axis Roughing
Clearance
Cut Parameters
Step Down Control
Machining Features/Regions
Tool
Feeds & Speeds
Tools
BallMill-0.125
BallMill-0.0625
BallMill-0.03125
Tool Geometry
Diameter 0.125
Corner Radius 0.0625
Taper 0
Tip Angle 0
Tool Properties
Tool Name BallMill-0.125
Tool # 1
# of Flutes 2
Cutcom Register 0
Adjust Register 0
Z-Offset 0
Material HSS
Coolant None
Comments
Feeds & Speeds
Spindle Speed 5000
Feed Rate 30
Edit/Create/Select Tool ...
Preview Tool
Generate
Cancel
Save
Help

4. Click on the Feeds and Speeds tab and select Load From Tool. Alibre CAM will now get the feeds and speeds information that was set when the tool was defined.

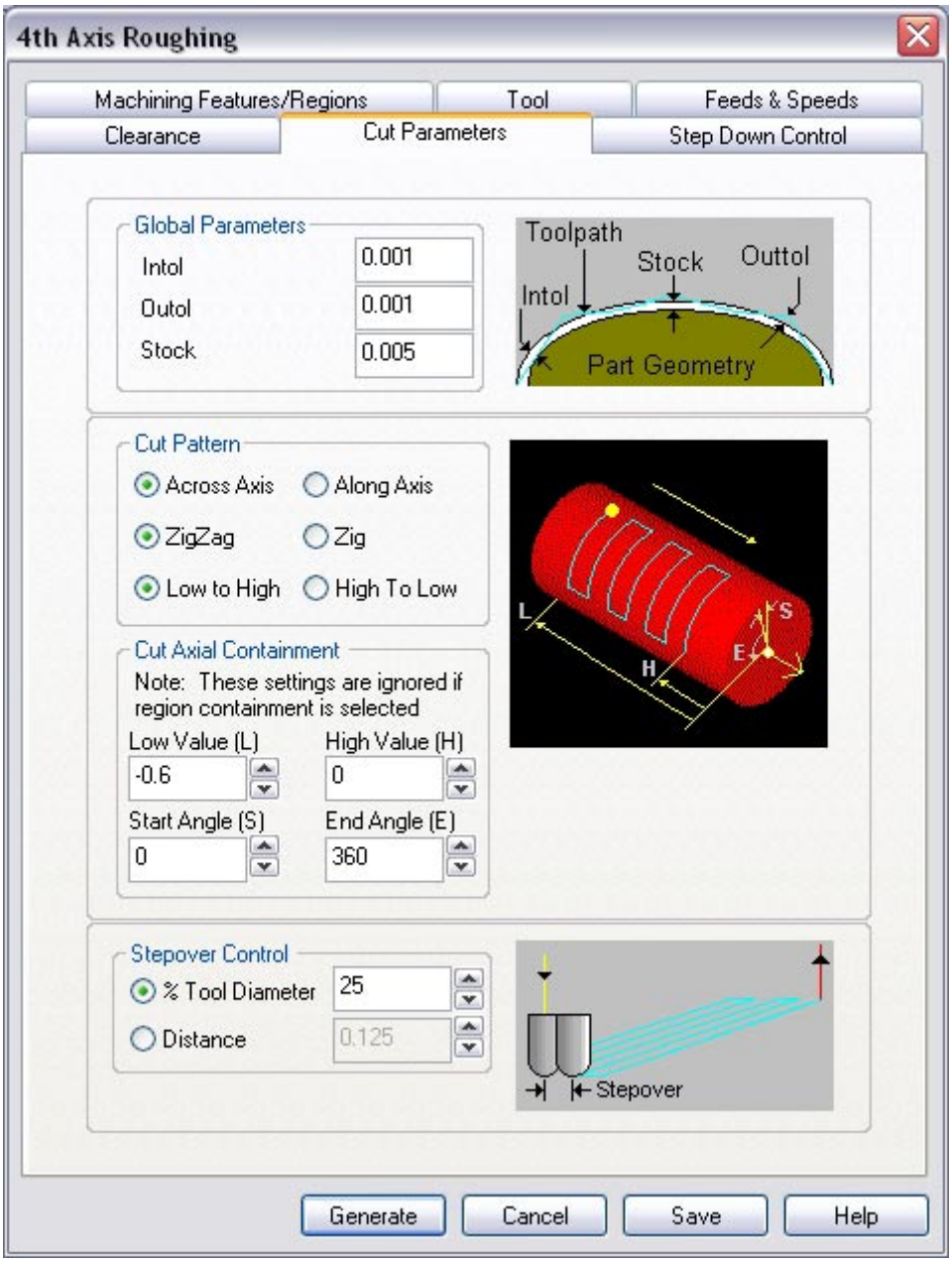


5. Switch to the Clearance Tab and set the Clearance Plane Definition to Automatic and Cut Transfer Method to Clearance Plane.

Specify Cut Parameters

6. Click on the Cut Parameters tab.

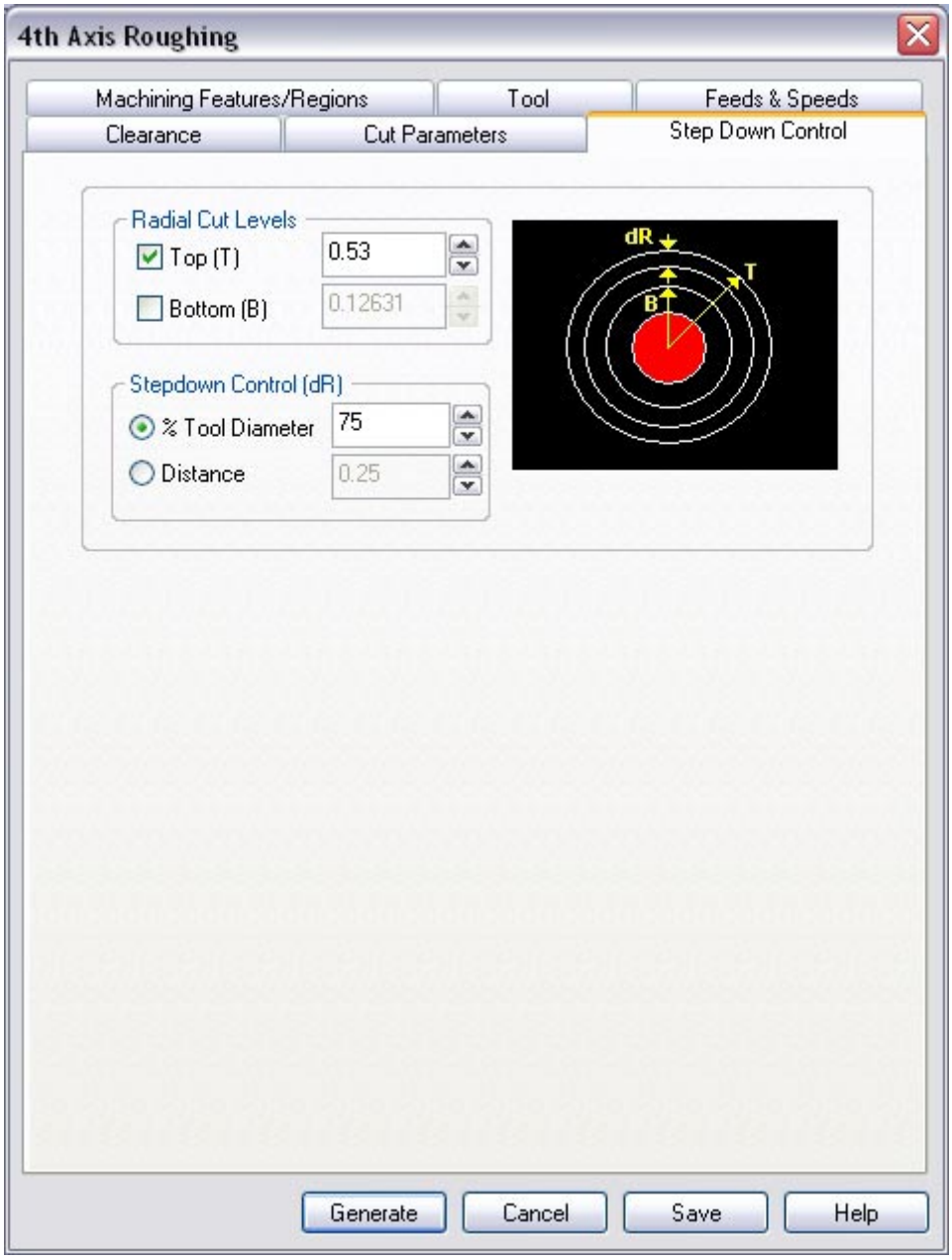
7. Set the Intol and Outol = **0.001**, Stock to leave =**0.005**, Cut Pattern to **Across Axis**, **Zig Zag** and **Low to High**, Cut Containment Low Value = **-0.6**, High Value = **0** (as the machine zero is set to the right edge of the stock/ part), Step over distance = **25** (% Tool Diameter).



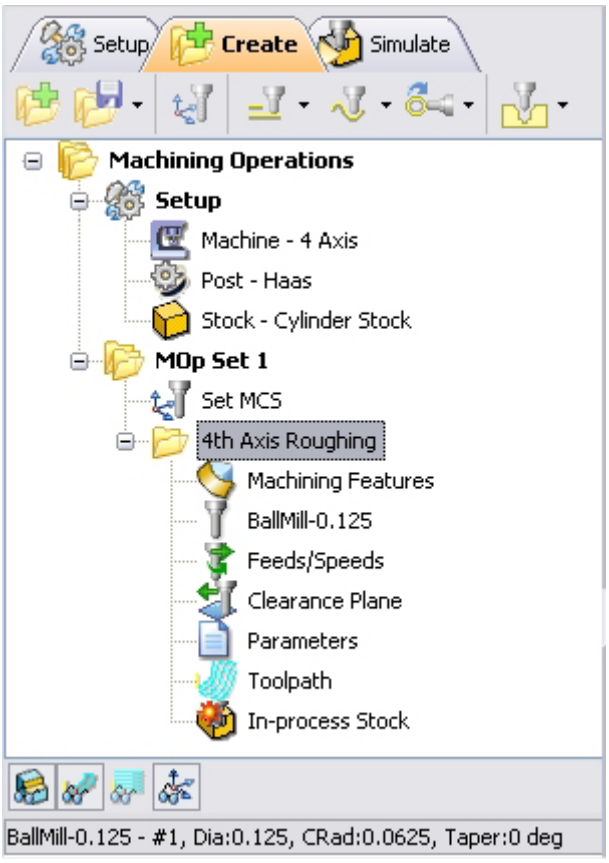
8. Switch to the Step Down Control Tab.

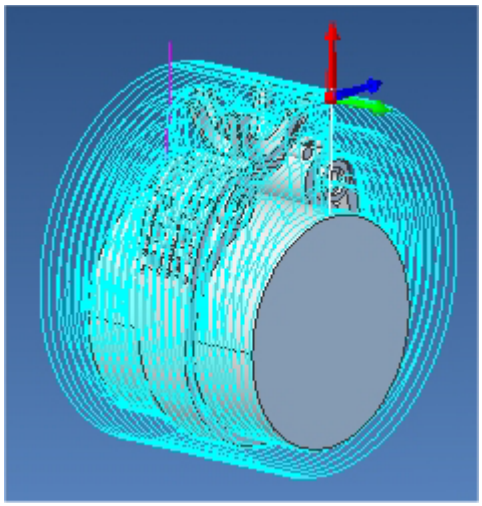
Step Down Control

9. Use the Following Settings.
    a. Cut Levels, Check Top (T) and specify Top (T) = **0.53** (radius of stock material)
    b. Step Down Control (dR) = **75** (% Tool Diameter).



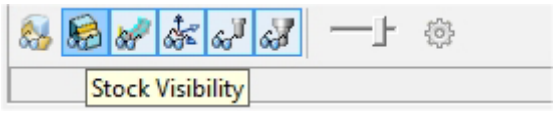
10. Click **Generate**. The 4th axis Roughing toolpath is now generated, and the Operation is listed under the Alibre CAM MOps browser.



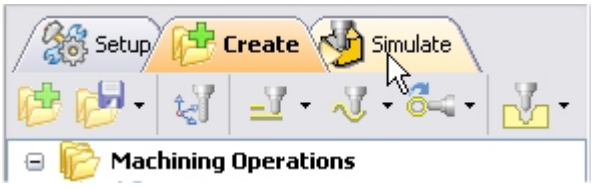


Simulate Toolpath

The generated toolpath can now be simulated. Make sure to turn on Stock Visibility under the Simulate tab.



1. Switch to the Simulate tab in the Alibre CAM -MOps browser.

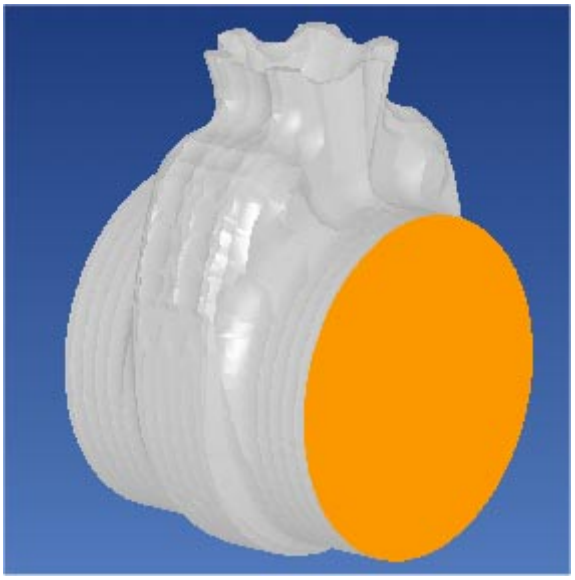


2. Select 4th Axis Roughing Operation and click  to launch the Alibre CAM Stock Simulation window.

3. Click Simulate from the Stock Simulation window to run simulation.

The simulated part is as shown below.

Note: You can adjust the simulation speed by selecting Simulation Preferences that is located to the bottom right corner of the Simulate tab Alibre CAM-MOps browser or from the Stock simulation window.



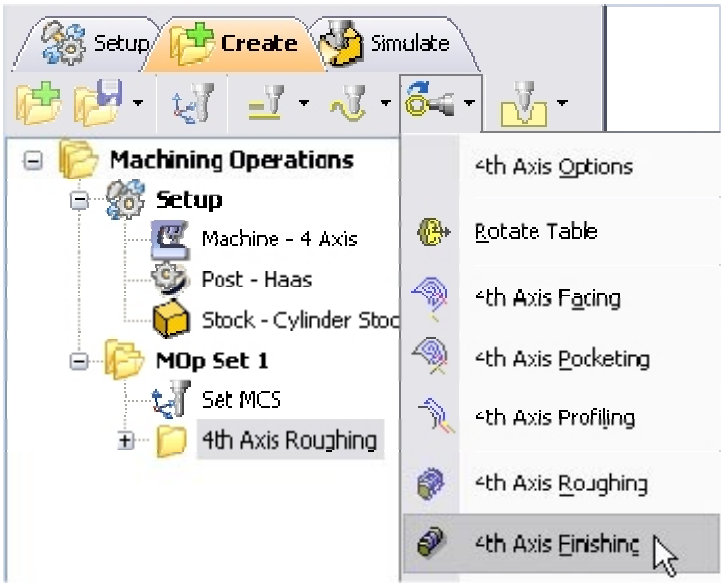
4. Once the simulation is complete, you can close the Stock Simulation window and return to the Alibre CAM browser.

4 axis Finishing

We will now use 4th Axis Finishing operation to finish the part using a 0.0625” Ball End Mill.

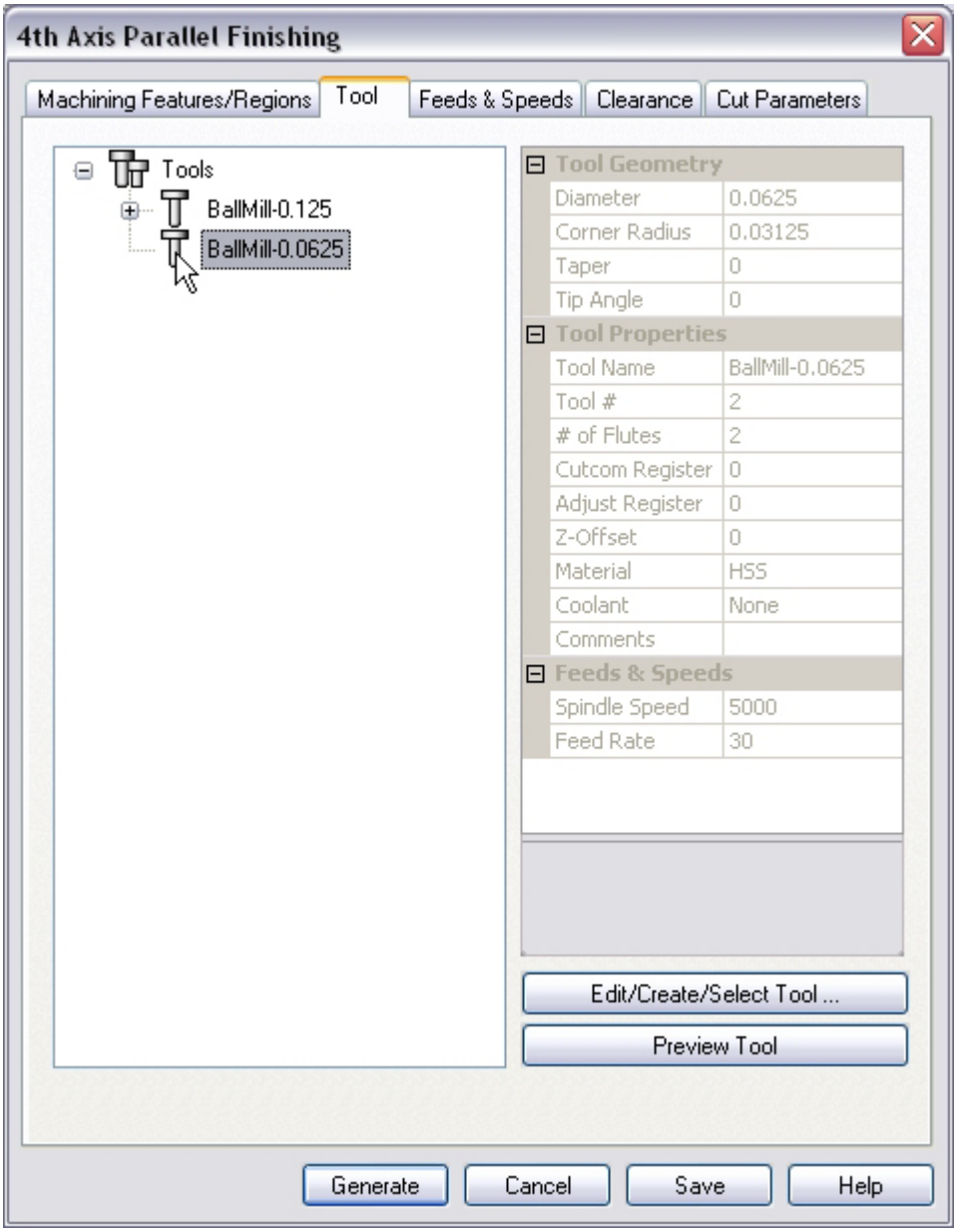
In this method, the tool is always normal to the axis of table rotation (continuous mode). The tool motions can be parallel to or normal to the rotation axis.

1. From the Create Operations tab, select 4 axis Milling and 4 Axis Finishing.



This brings up the Finishing Operations dialog. We will go over the steps for creating the pocketing operation.

2. Switch to the Tools tab inside the 4 Axis Finishing operation dialog and select BallMill-0.0625.



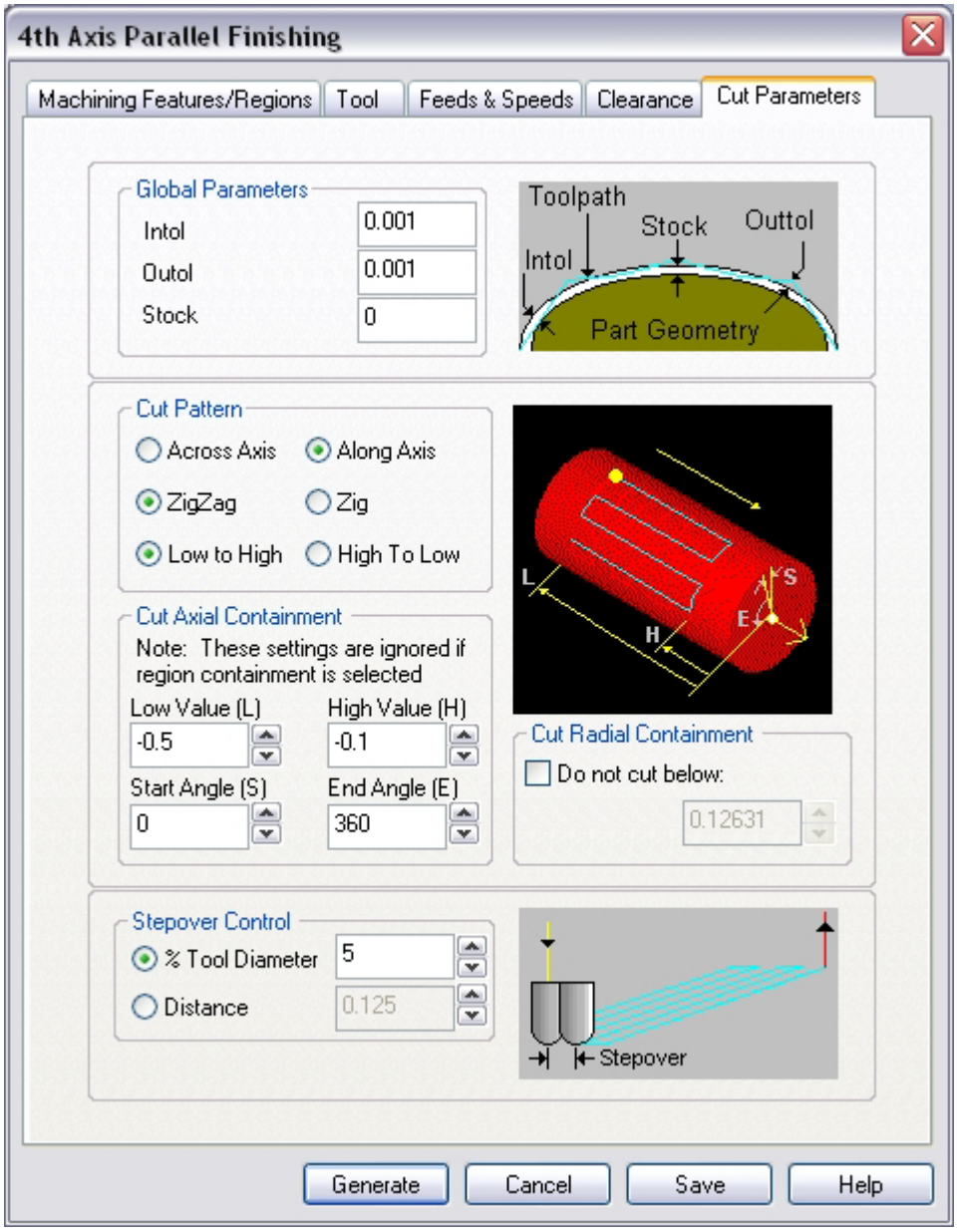
3. Click on the Feeds and Speeds tab and select Load From Tool. Alibre CAM will now get the feeds and speeds information that was set when the tool was defined.

4. Switch to the Clearance Tab and set the Clearance Plane Definition to Automatic and Cut Transfer Method to Clearance Plane.

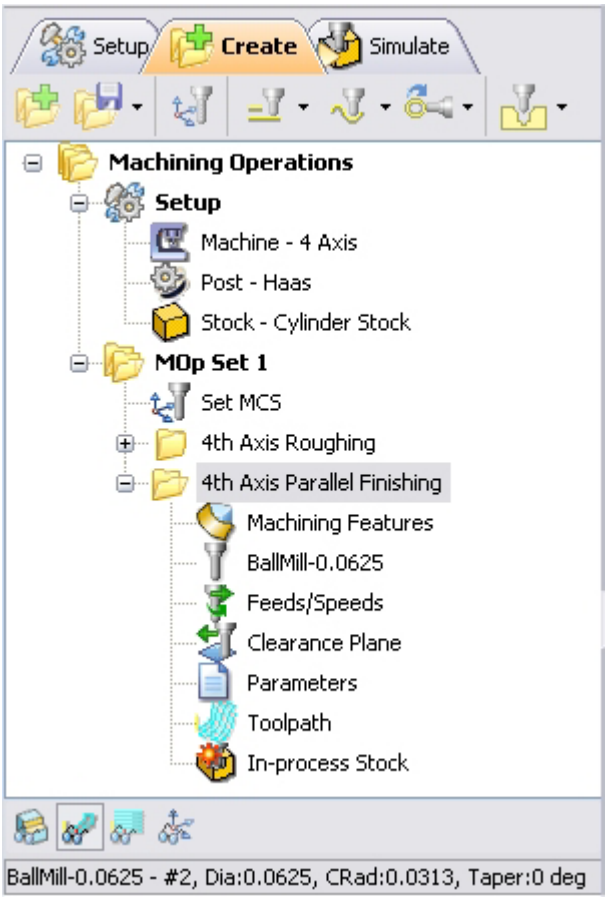
Specify Cut Parameters

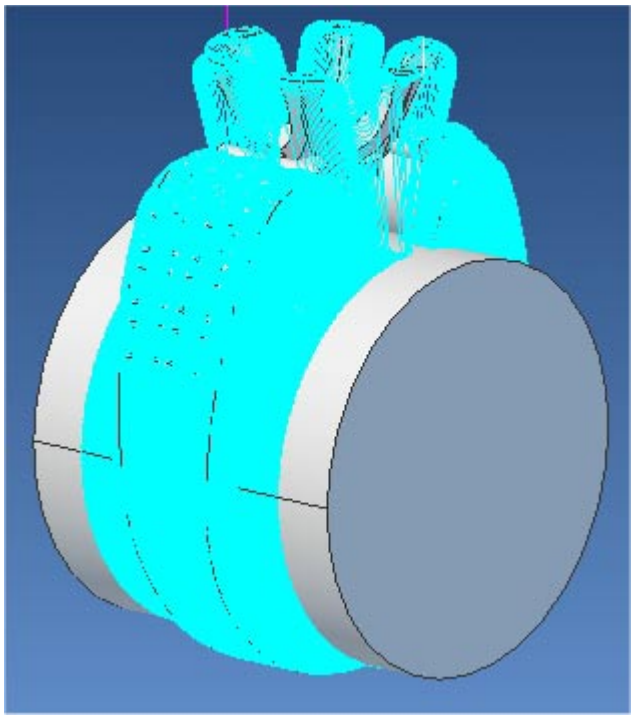
1. Click on the Cut Parameters tab.

2. Set the Tolerance to **0.001**, Stock to leave =**0**, Cut Pattern = **Along Axis**, Zig Zag, Low to High, Set Cut Containment Low Value = **-0.5**, High Value = **-0.1**, Step over distance = **5** (% Tool Diameter).



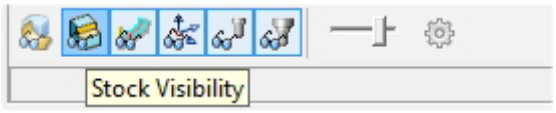
3. Click **Generate**. The Finishing toolpath is now generated, and the Operation is listed under the 4th Axis Roughing Operation in the Alibre CAM MOps browser.





Simulate Toolpath

The generated toolpath can now be simulated. Make sure to turn on Stock Visibility under the Simulate tab.



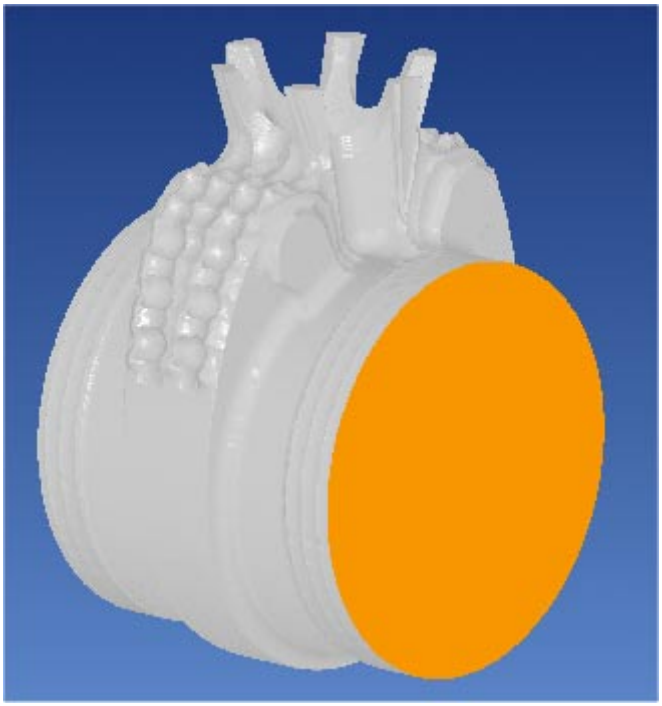
1. Switch to the Simulate tab in the Alibre CAM -MOps browser.

2. Select 4th Axis Parallel Finishing Operation and click  to launch the Alibre CAM Stock Simulation window.

3. Click Simulate from the Stock Simulation window to run simulation.

The simulated part is as shown below.

Note: You can adjust the simulation speed by selecting Simulation Preferences that is located to the bottom right corner of the Simulate tab Alibre CAM-MOps browser or from the Stock simulation window.



4. Once the simulation is complete, you can close the Stock Simulation window and return to the Alibre CAM browser.

4th Axis Finishing Operation #2

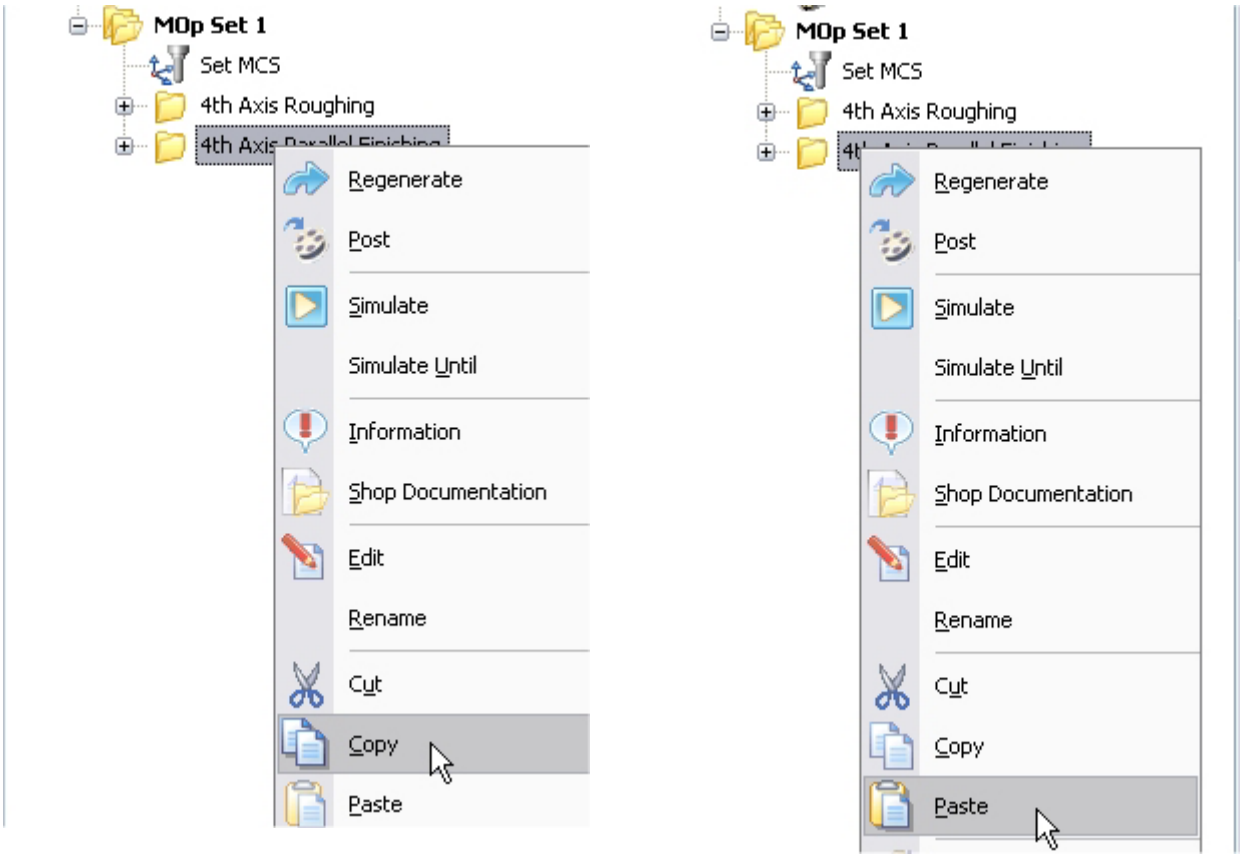
We will now create a second 4 Axis finish operation to machine the prongs and pocket areas around the ring by limiting the toolpath using start, end angle cut containments.

Copying a MOp

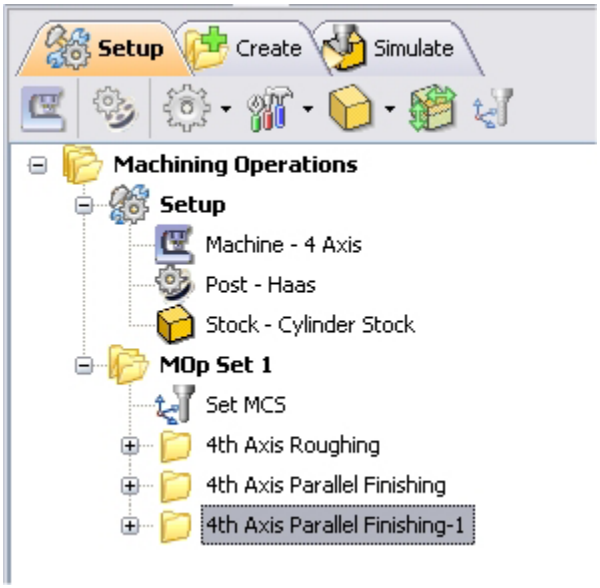
1. Switch to the Create Operations tab in the Mops Browser.

2. Select the 4 Axis Parallel Finishing, right click **copy**.

3. Now right click on 4 Axis Parallel Finishing and select **paste**.



4. A copy of the finish operation is created below the 4th Axis Finishing operation. The operation name is labeled 4th Axis Parallel Finishing-1 and is as shown below.

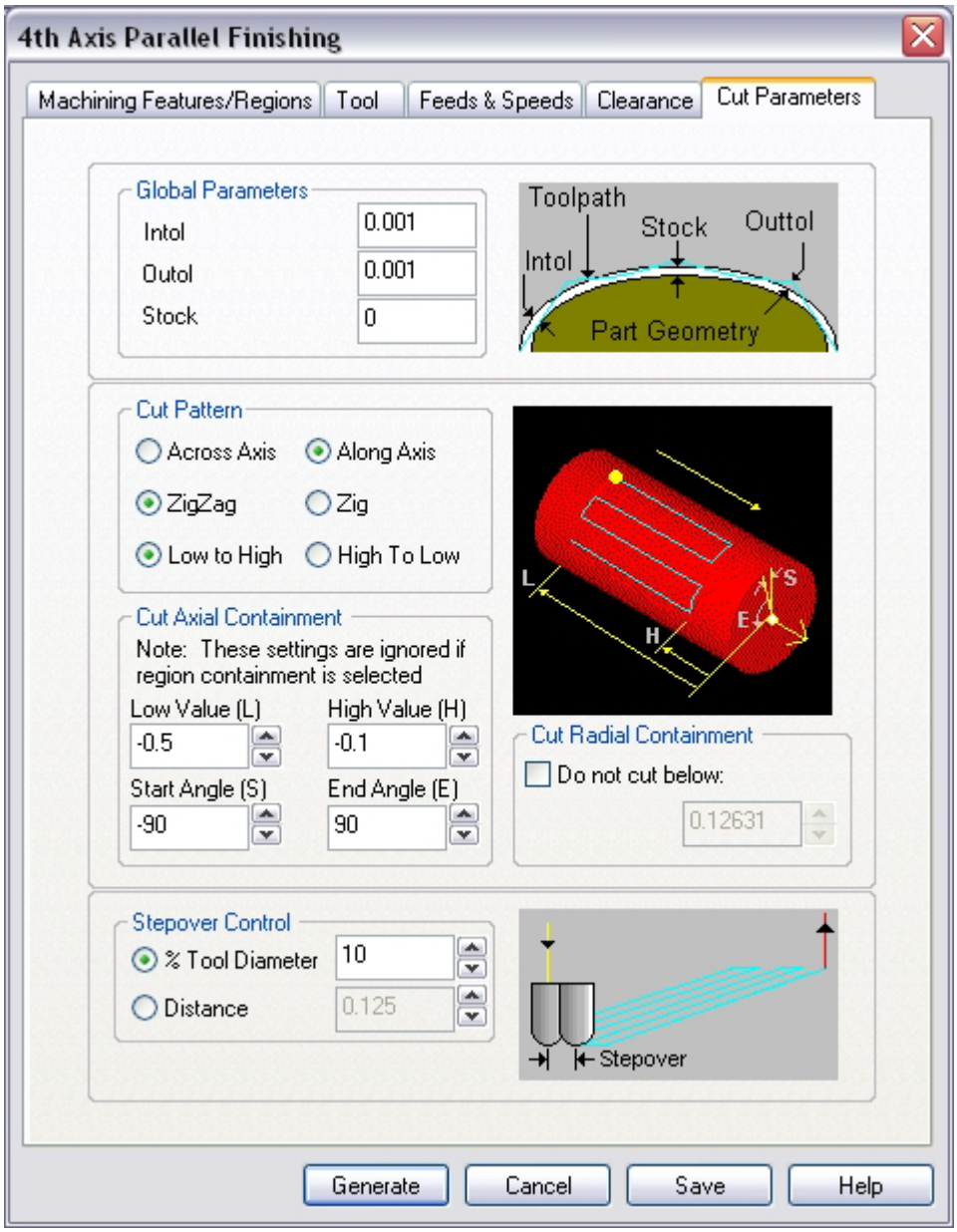

Setup
Create
Simulate
Machining Operations
Setup
Machine - 4 Axis
Post - Haas
Stock - Cylinder Stock
MOp Set 1
Set MCS
4th Axis Roughing
4th Axis Parallel Finishing
4th Axis Parallel Finishing-1

Specify Cut Parameters

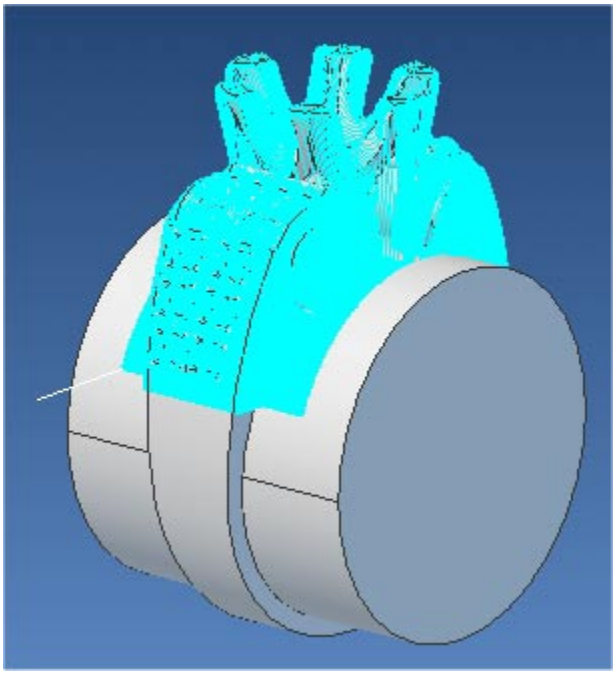
5. Double click on 4th Axis Parallel Finishing-1 to edit the operation.

6. Switch to the tools tab and select **BallMill-0.03125**.

7. Switch to the Cut parameters tab and use the following parameters.

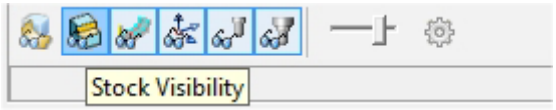


8. Click Generate. The 4th Axis Parallel Finishing-1 toolpath is now generated, and the Operation is listed under the MOps browser.



Simulate Toolpath

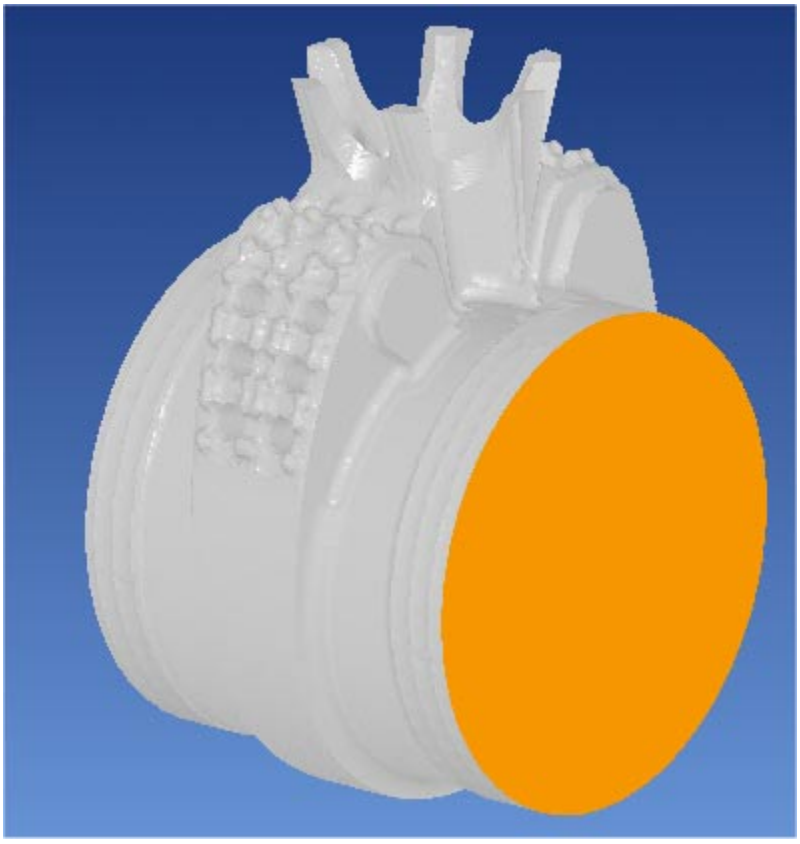
The generated toolpath can now be simulated. Make sure to turn on Stock Visibility under the Simulate tab.



1. Switch to the Simulate tab in the Alibre CAM -MOps browser.
2. Select 4th Axis Parallel Finishing-1 Operation and click  to launch the Alibre CAM Stock Simulation window.
3. Click Simulate from the Stock Simulation window to run simulation.

The simulated part is as shown below.

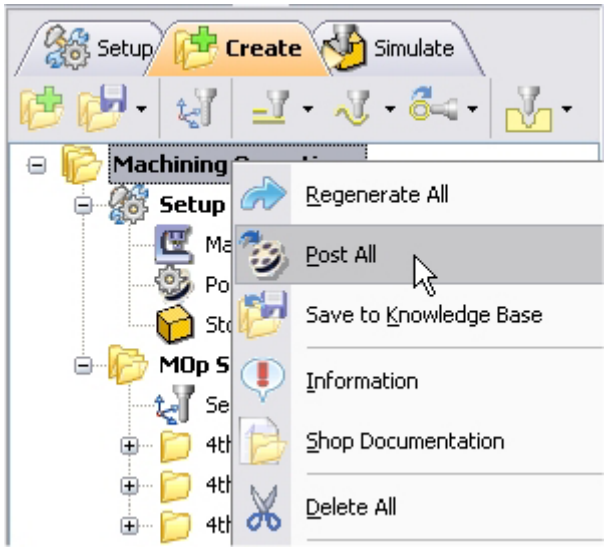
Note: You can adjust the simulation speed by selecting Simulation Preferences that is located to the bottom right corner of the Simulate tab Alibre CAM-MOps browser or from the Stock simulation window.



4. Once the simulation is complete, you can close the Stock Simulation window and return to the Alibre CAM browser.

Post Processing

1. Select Machining Operations from the Create Operations tab and right click and select post process.



2. Specify the File Name as **Ring.nc** and click save.

The post by default is set to Haas as specified under the Post processor setup. You can change the post processor by selecting a different one from the drop down menu in the list. The posted g code by default will be saved to the folder where the part file is located.