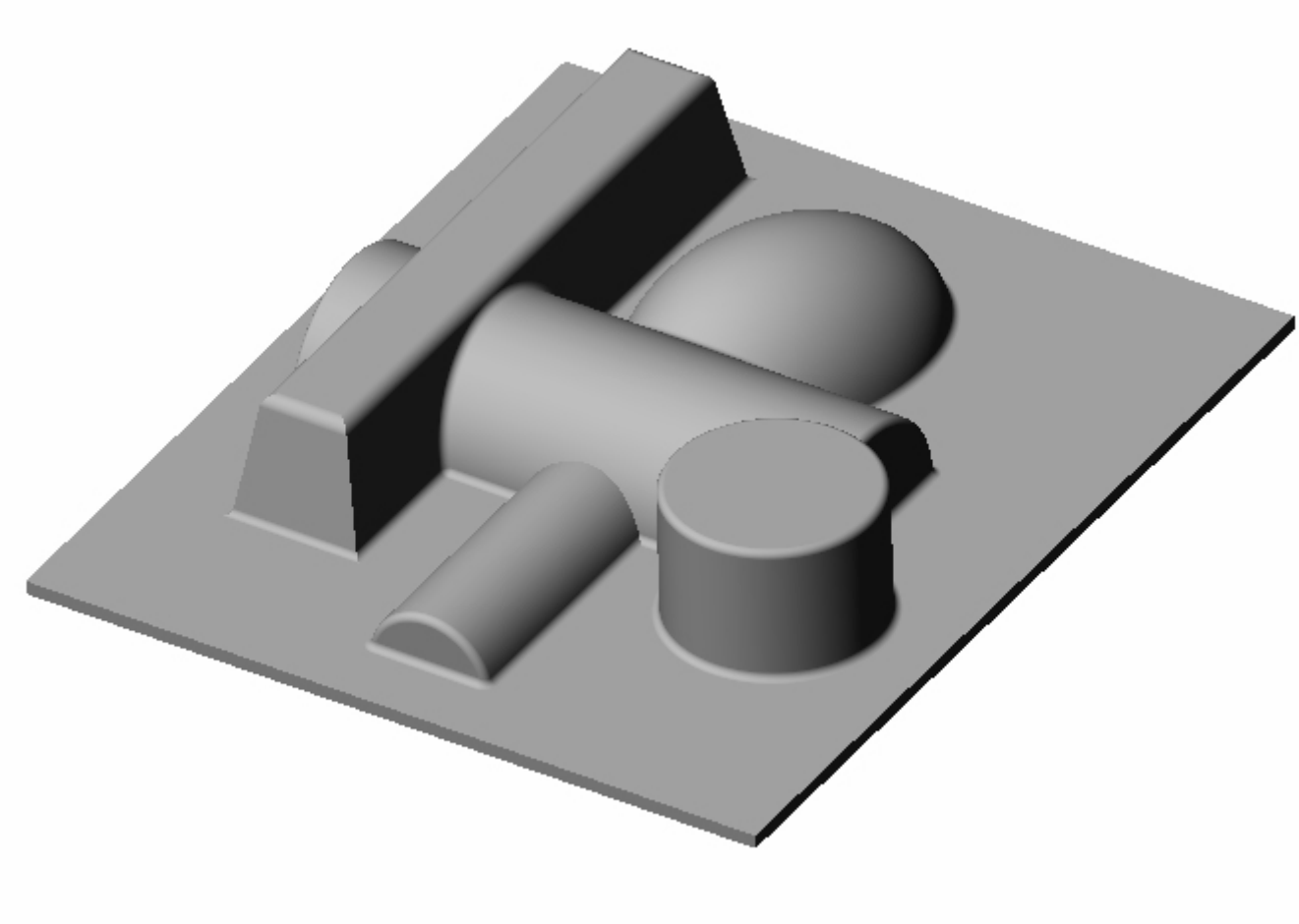
Tutorial 7: Machining a 3D Mold



Using 3 axis Roughing and Finishing Toolpath methods

Introduction

This tutorial will illustrate machining this Mold using 3 axis-milling operations.
This tutorial will introduce the usage of several 3-axis operations such as horizontal roughing, parallel finishing, and horizontal 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 mold

- We will machine the mold completely using 3 axis-machining operations.
- The part itself will be machined out of a 5.5 x 6.5 inch x 1.25-inch wood block.
- The stock may be held to the machine table or the spoil sheet on the table using double-sided tape or by clamps.
- The part will be machined using 0.5" flat end mill, 0.25" & 0.125" ball end mills.

Main Programming Steps

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

- Create the Stock geometry
- Set the Machine zero point or Locate geometry with respect to the machine coordinates
- Create / Select the tool used for machining
- Set the feeds and speeds
- Set the clearance plane for the non-cutting transfer moves of the cutter
- Select the machining regions for containing the cutter to specific areas to cut
- Select the machining operations and set the parameters
- Generate the toolpath
- Simulate the toolpath.

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

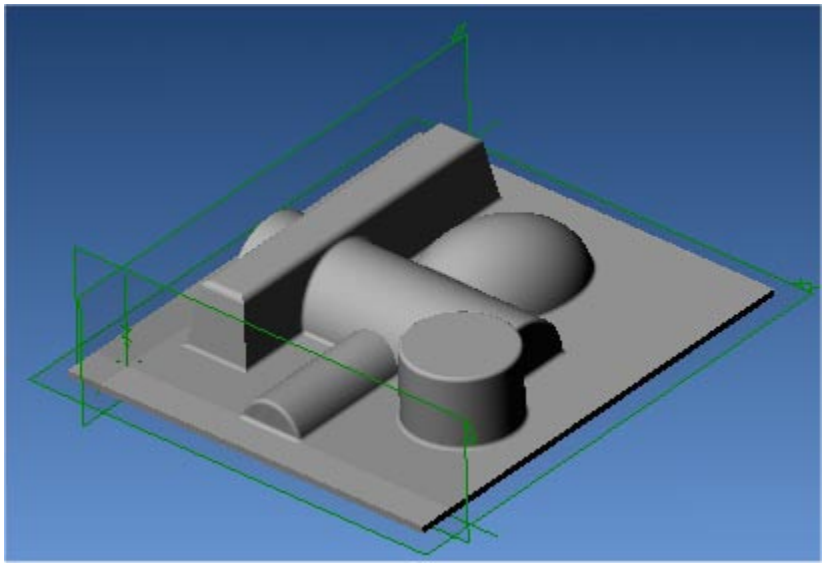
Loading the Part Model

"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 **3dMold.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)

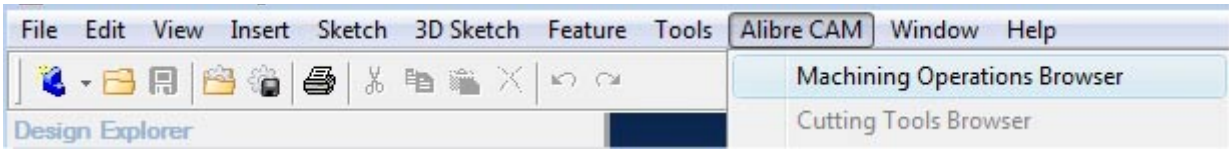
The loaded part appears as shown below.

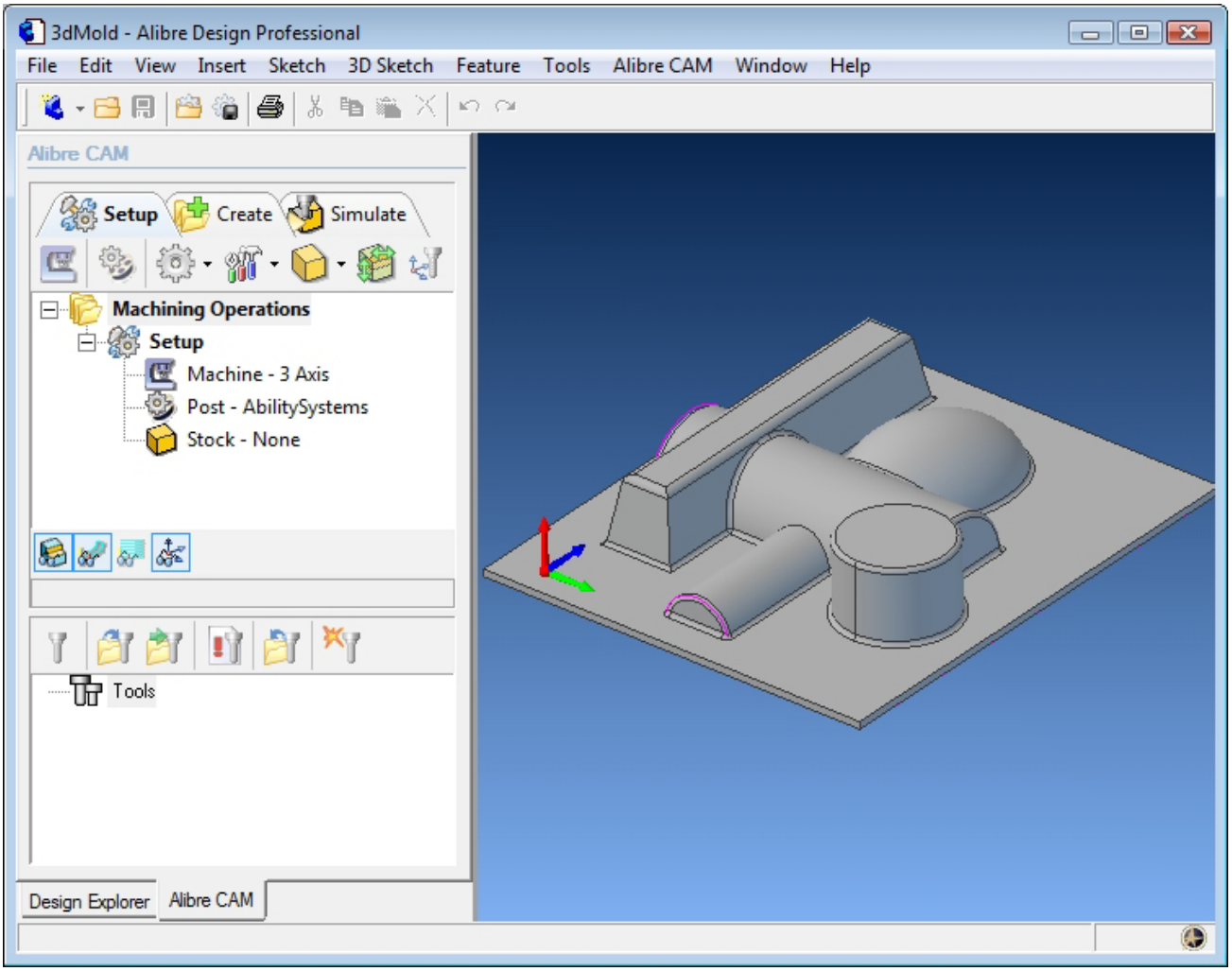


Note: You must work in shaded mode in order to be able to visualize toolpaths created in Alibre CAM. It is suggested for best visual performance with Alibre CAM to work with only one view port open and the view port operating in shaded mode.

Loading the Alibre CAM Browser

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





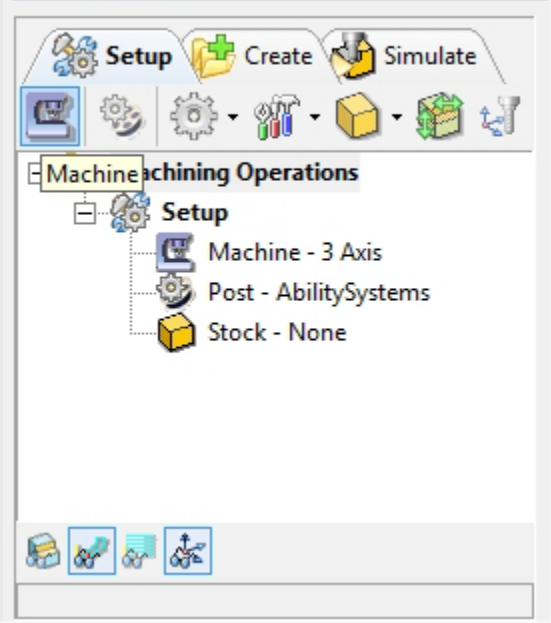
The Alibre CAM browser (MOPs and Cutting Tools) is now loaded and is docked over the Design Explorer. You can toggle between the MOPs browser and the Design Explorer from the tabs at the bottom of the window. It can be hidden by un-checking Browser on the Alibre CAM menu bar. To re-display the hidden 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

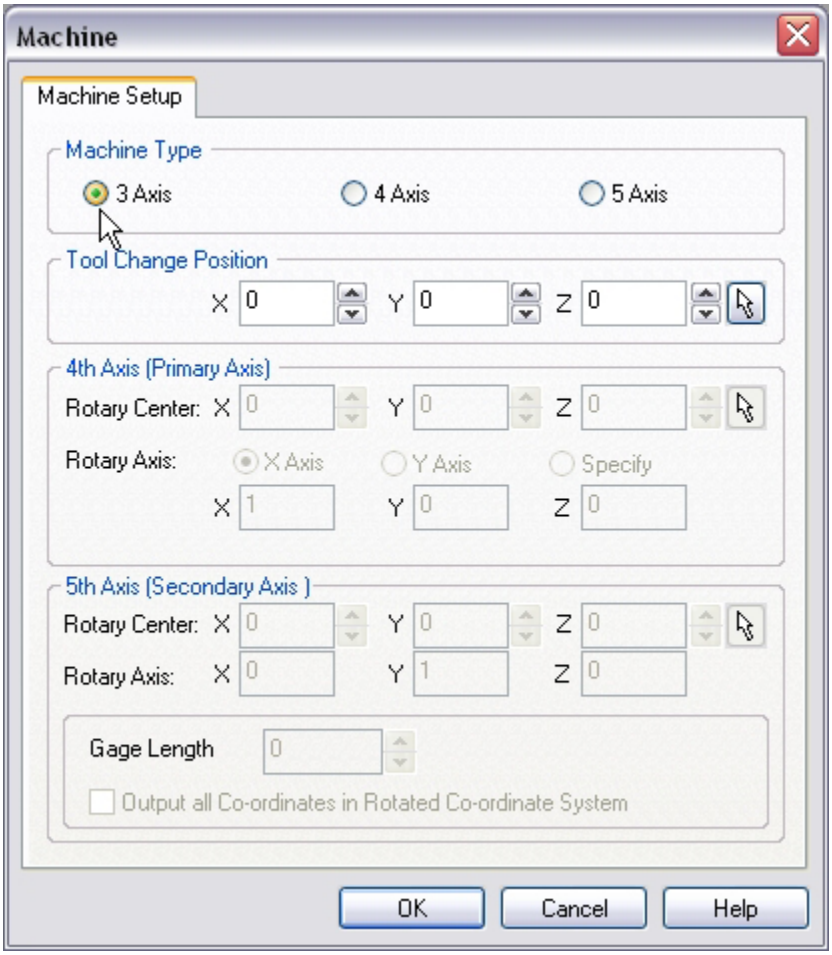
The Setup tab allows the user to specify Machine Setup, Select Post Processor, Stock Geometry, Machine Coordinate System (Machine Zero) & Preferences.

Setup Tab

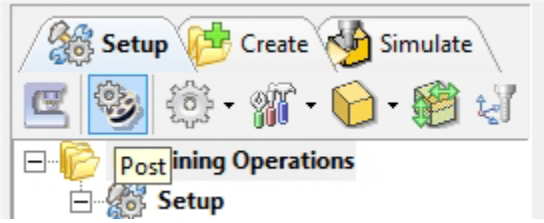
1. Click on the Setup tab located under the MOPs Browser and select Setup Machine



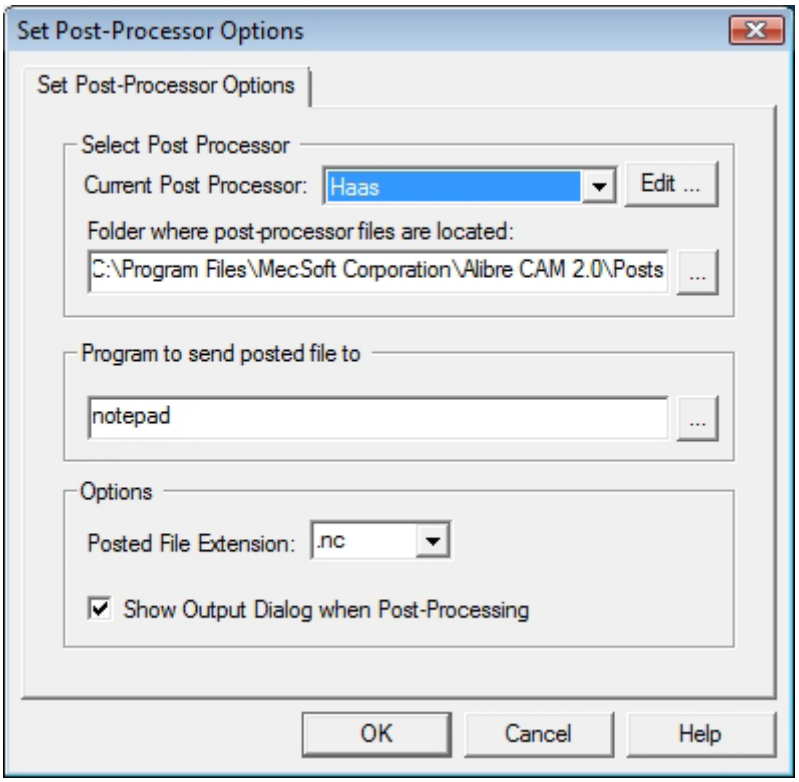
2. Set the Machine type to 3 axis



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



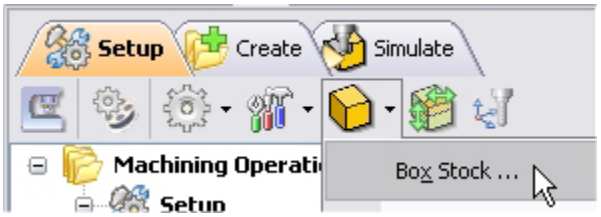
4. Set the current post processor that is on your controller. We will select Haas as the post processor for this exercise. Set the posted file extension type to .nc



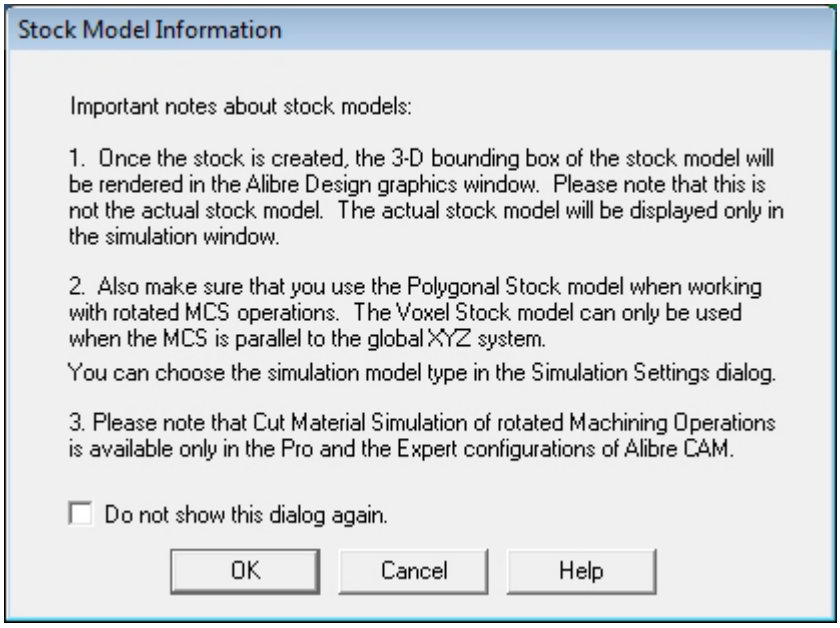
Note: By default post processor files are located under
C:\Program Files\MecSoft Corporation\Alibre CAM 2.0\Posts
The program to send the posted output is set to notepad. This would output the G code to a notepad.

Create Stock Geometry

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



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

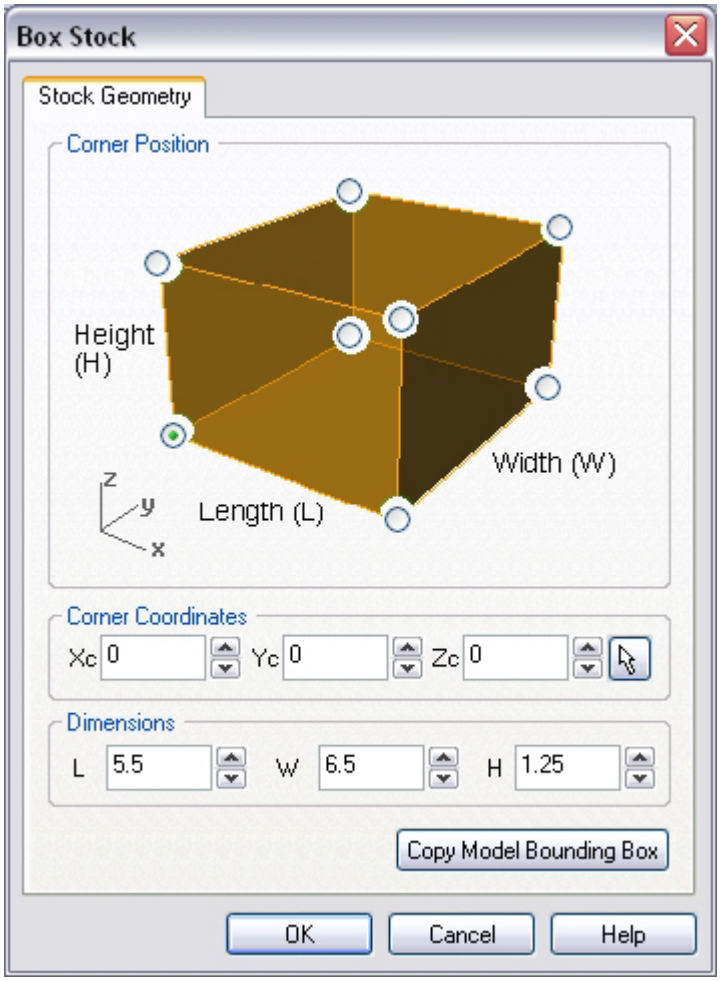


Click OK

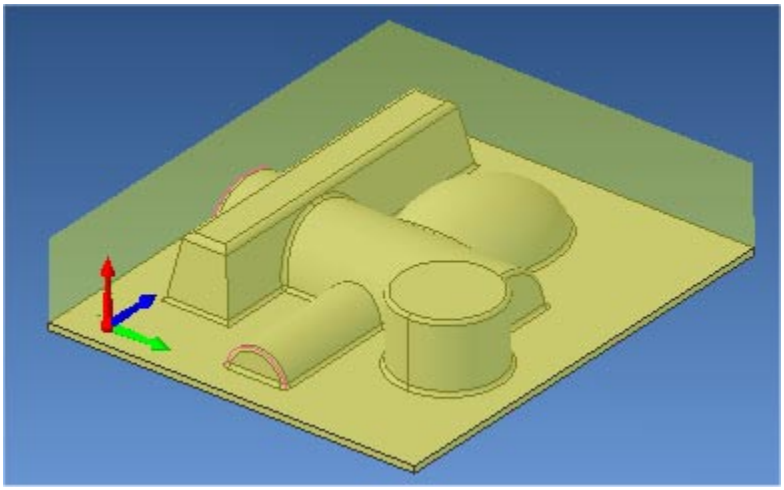
User can turn off this dialog by selecting Do not show this dialog again located on the bottom of the message window.

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

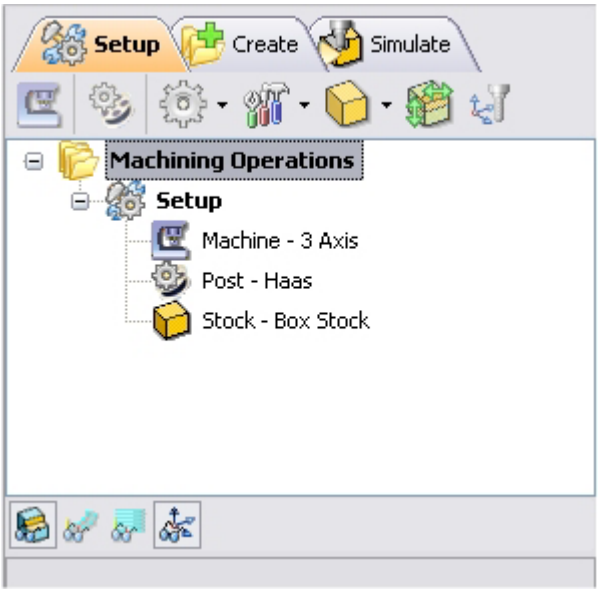
2. This brings up the Box Stock parameters. Set the Length (L) = 5.50, Width W = 6.5 and Height (H) = 1.25. Leave the other parameters as default and Click OK. Set Corner coordinates for Xc, Yc and Zc = 0.



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



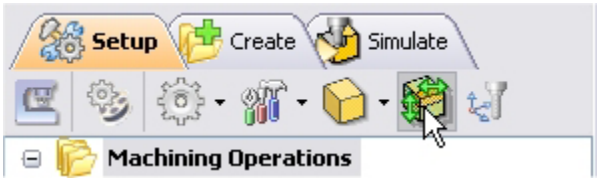
3. The setup tab now displays the following information: Machine Type, Post Processor, and Stock type as show below.



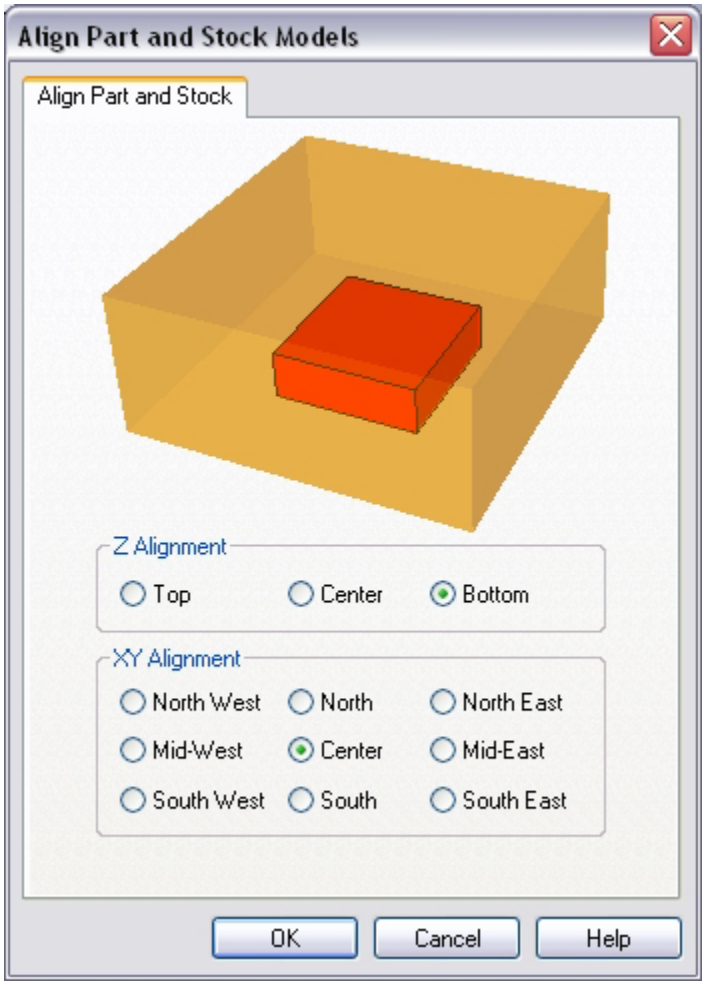
Align Part and Stock

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



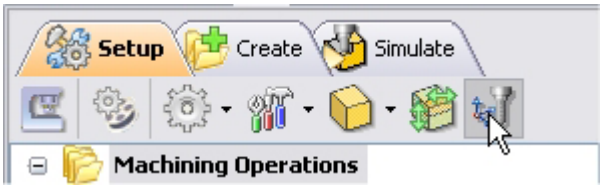
2. Set Z alignment to **Bottom** and XY alignment to **Center.** (This would align the stock to the bottom of the part in Z and center in XY)



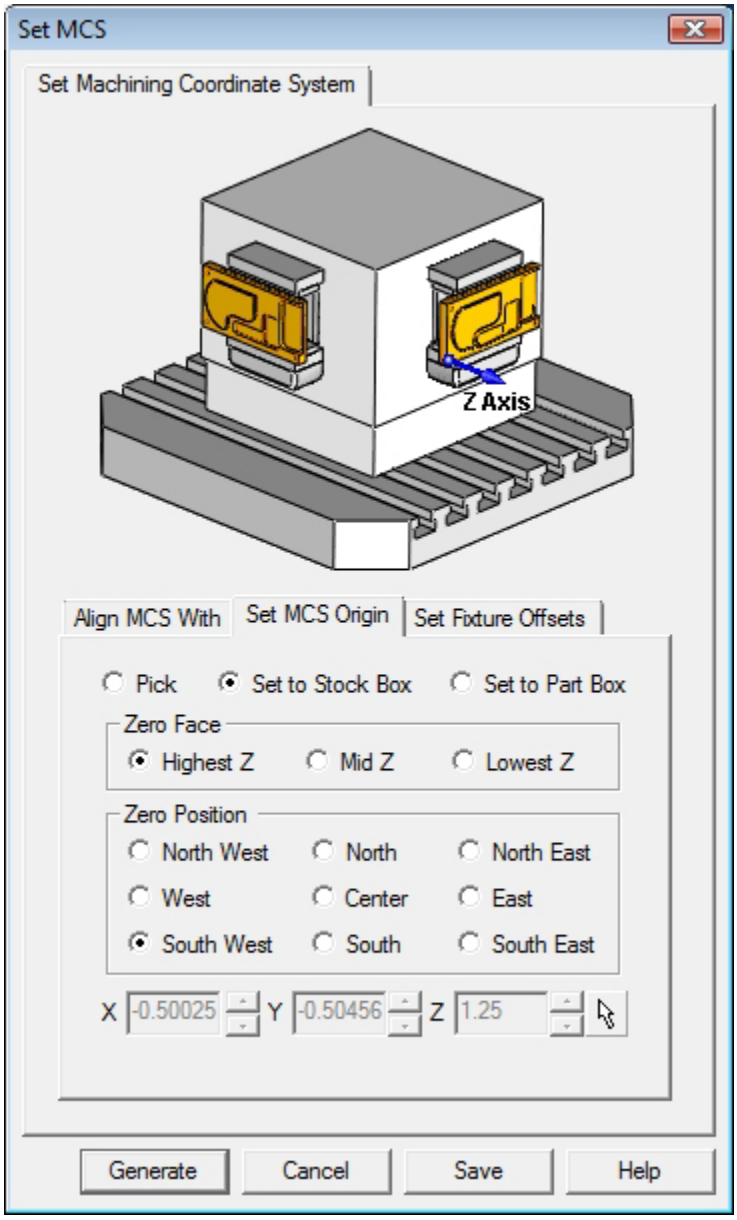
Set Machine Coordinate System (MCS)

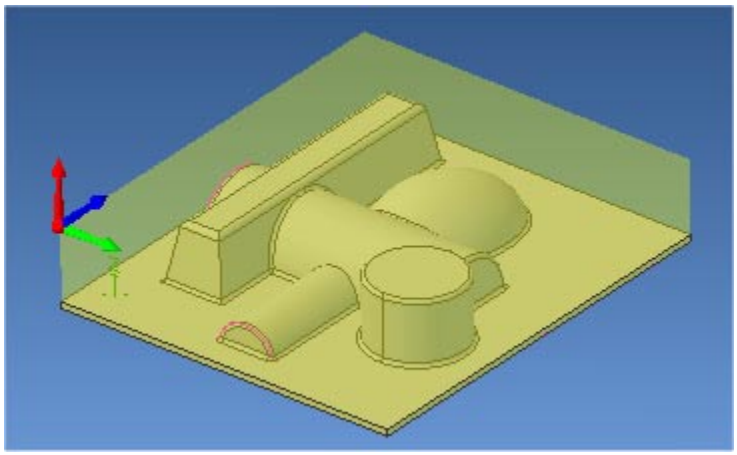
The steps below help you determine the machine home (also known as machine zero or tool touch off point) for the part/stock geometry.

1. Select Set MCS from the Setup tab



2. Switch to SetMCS Origin tab and choose **Set to Stock Box**, the Zero Face to **Highest Z,** and Zero Position to **South West** corner. This sets the machine home to the top of the stock material and the southwest corner of the part geometry.


Set MCS
Set Machining Coordinate System
Z Axis
Align MCS With
Set MCS Origin
Set Fixture Offsets
Pick
Set to Stock Box
Set to Part Box
Zero Face
Highest Z
Mid Z
Lowest Z
Zero Position
North West
North
North East
West
Center
East
South West
South
South East
X -0.50025
Y -0.50456
Z 1.25
Generate
Cancel
Save
Help



(This sets the machine home to Southwest corner and top of the stock material).

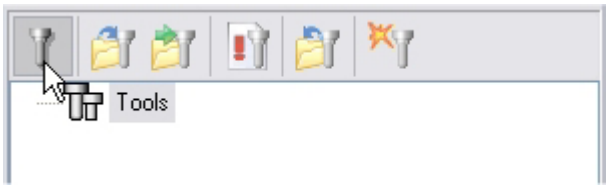
Click Save As to save the work and specify a file name as 3dMold-Rev1.

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.

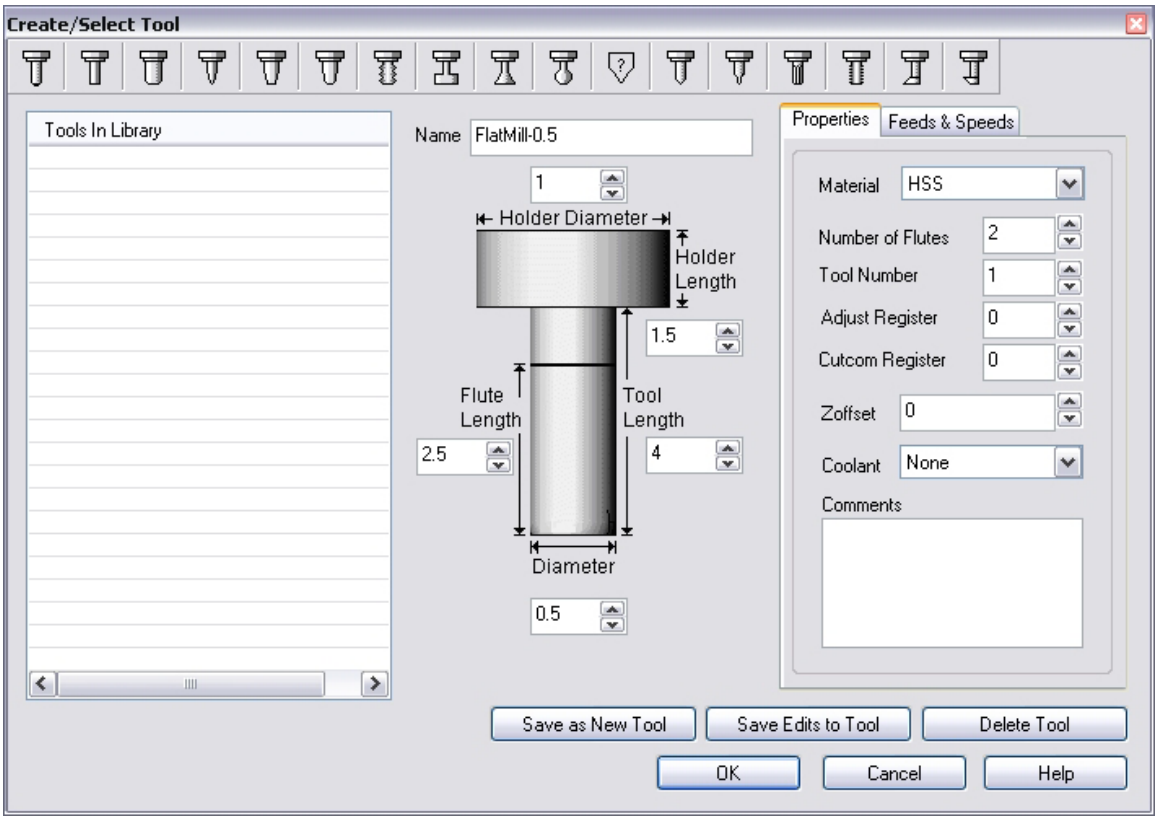
Create Tools

To machine the above part we will now create a ½ inch (0.5”) Flat End Mill.

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 Flat End Mill.



2. Set the tool name as **FlatMill-0.5**, Tool Diameter = **0.5,** Under the Properties tab set Tool Number = **1**.

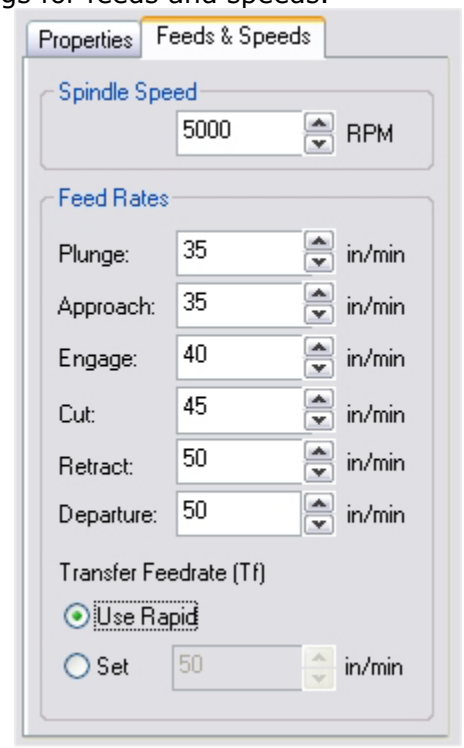


Setting Feeds and Speeds

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.

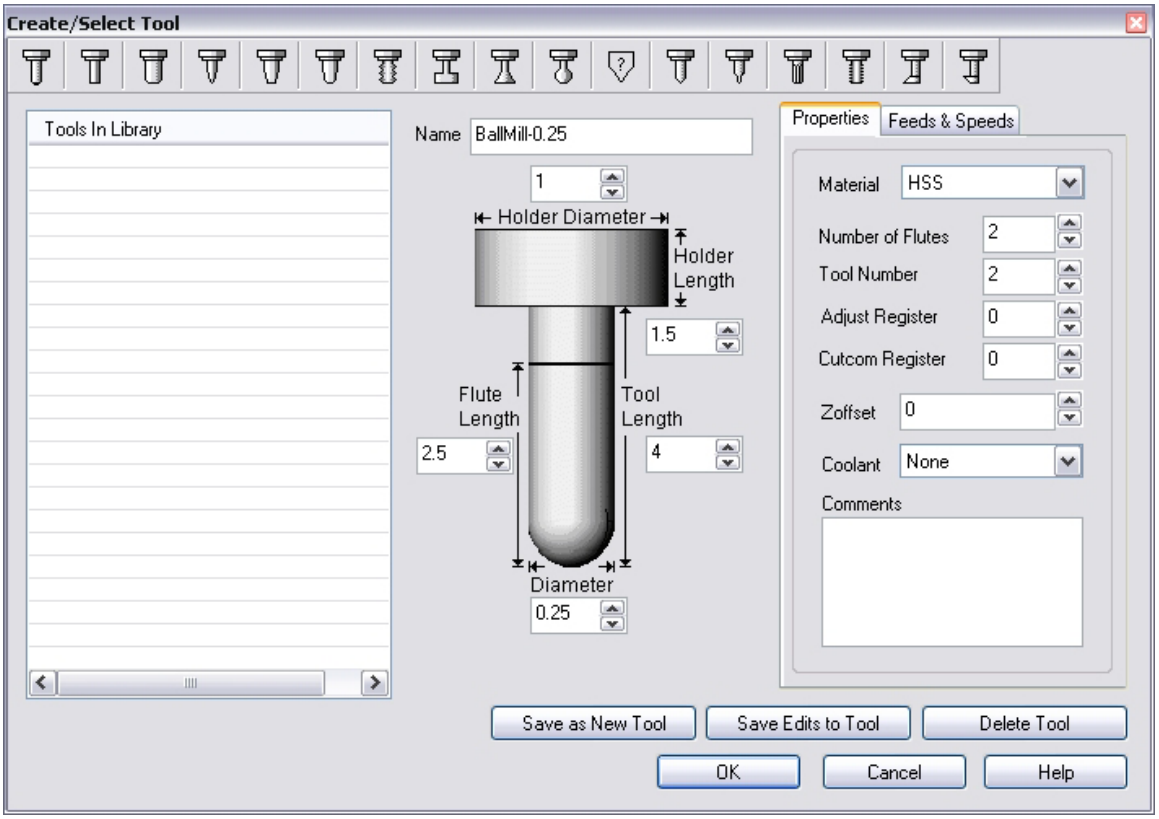
4. Use the following settings for feeds and speeds.



5. Click **Save as New Tool**. The tool is now created and listed under Tools in Library. Click OK to close the dialog.

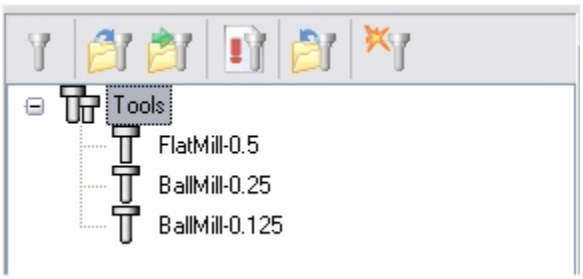
Note: You can edit the tool properties and click Save Edits to Tool to save the changes. You can create additional tools by assigning a different name and specify the tool parameters.

6. Create a Ball End Mill with the following parameters.
    a. Tool Name: **BallMill-0.25**, Tool Number = **2**.
    b. Switch to Feeds & Speeds tab set Spindle Speed = 5000 rpm, plunge & approach feed = 35 ipm, approach feed = 40 ipm, cut feed = 45 ipm, retract and departure feeds = 50 ipm. Set the Transfer Feedrate to Use Rapid.
    c. Click Save as New Tool.



7. Create another Ball End Mill with the following parameters.
    a. Tool Name: **BallMill-0.125**, Tool Number = **3**.
    b. Switch to Feeds & Speeds tab set Spindle Speed = 5000 rpm, plunge & approach feed = 35 ipm, approach feed = 40 ipm, cut feed = 45 ipm, retract and departure feeds = 50 ipm. Set the Transfer Feedrate to Use Rapid.
    c. Click Save as New Tool.

The created tools are now listed under the Alibre CAM Tools browser.



Create Machining Operations

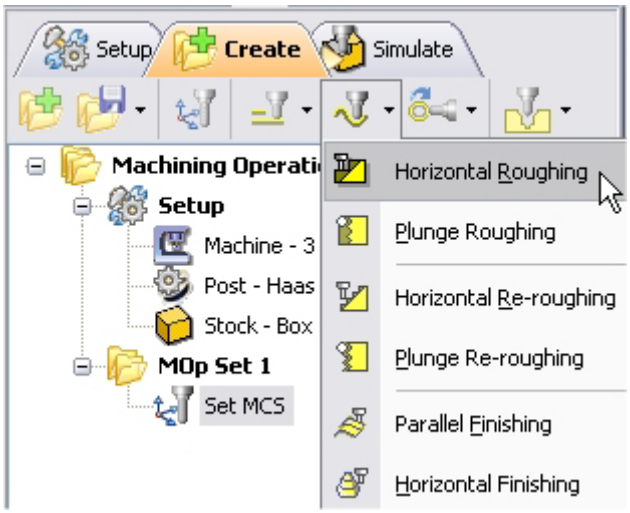
We will machine the mold using 3 different machining operations – Horizontal Roughing, Parallel Finishing, and Horizontal Finishing.

The first step in machining the mold will be a roughing operation. 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.

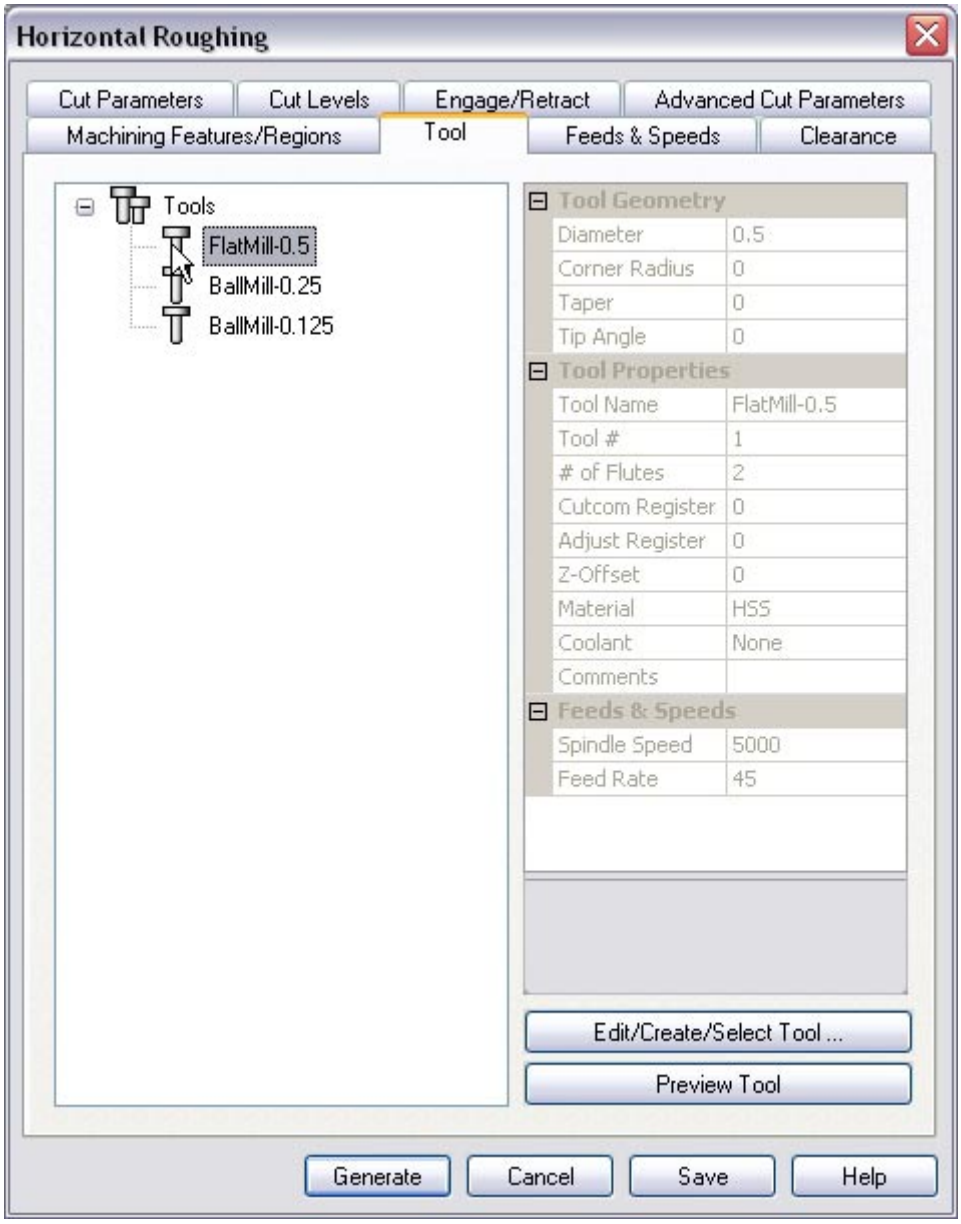
3 axis Horizontal Roughing

1. Select 3 Axis Milling and choose Horizontal Roughing.

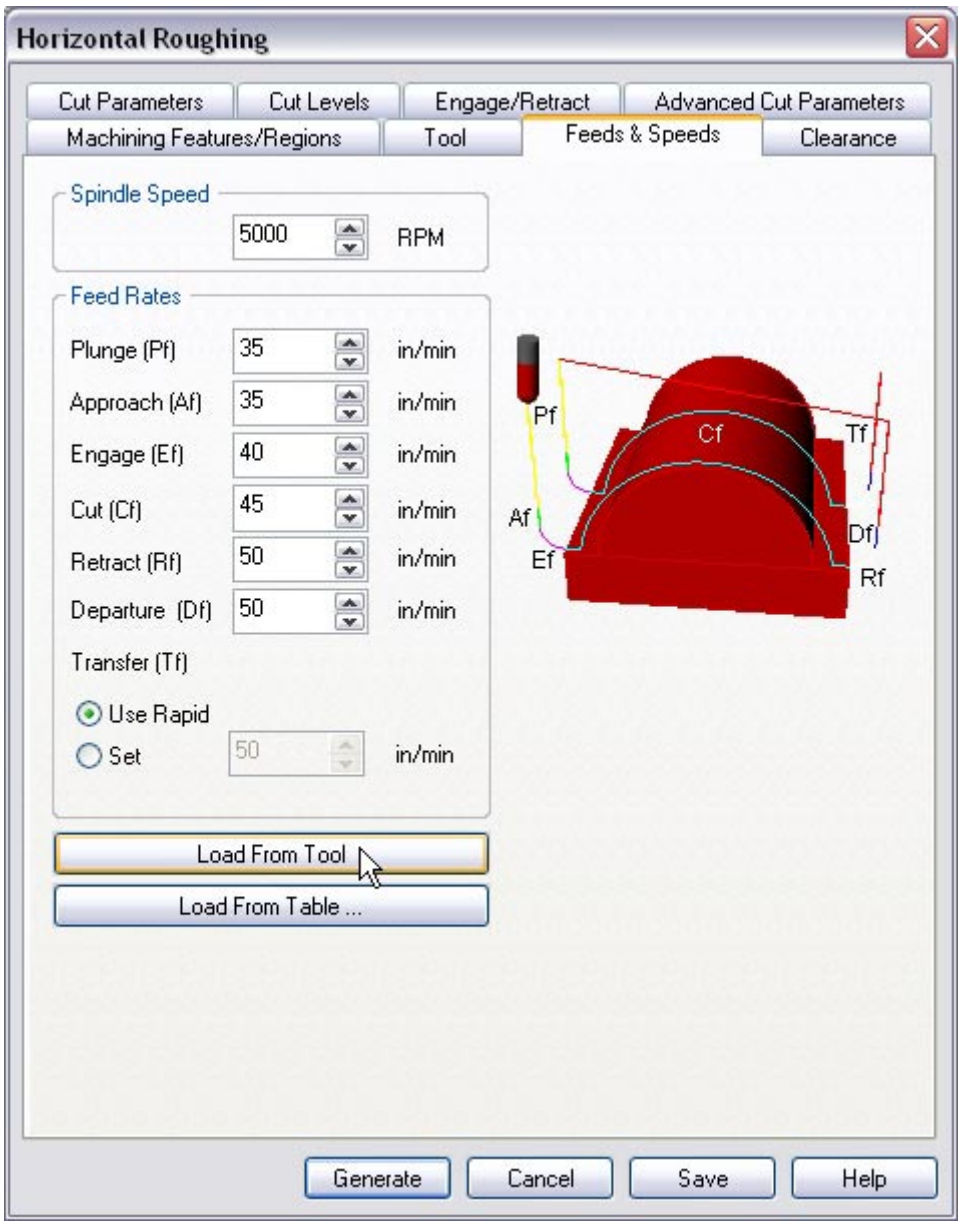


2. This brings up the 3 Axis Horizontal Roughing Operation Dialog. We will now go over the steps for creating the toolpath.

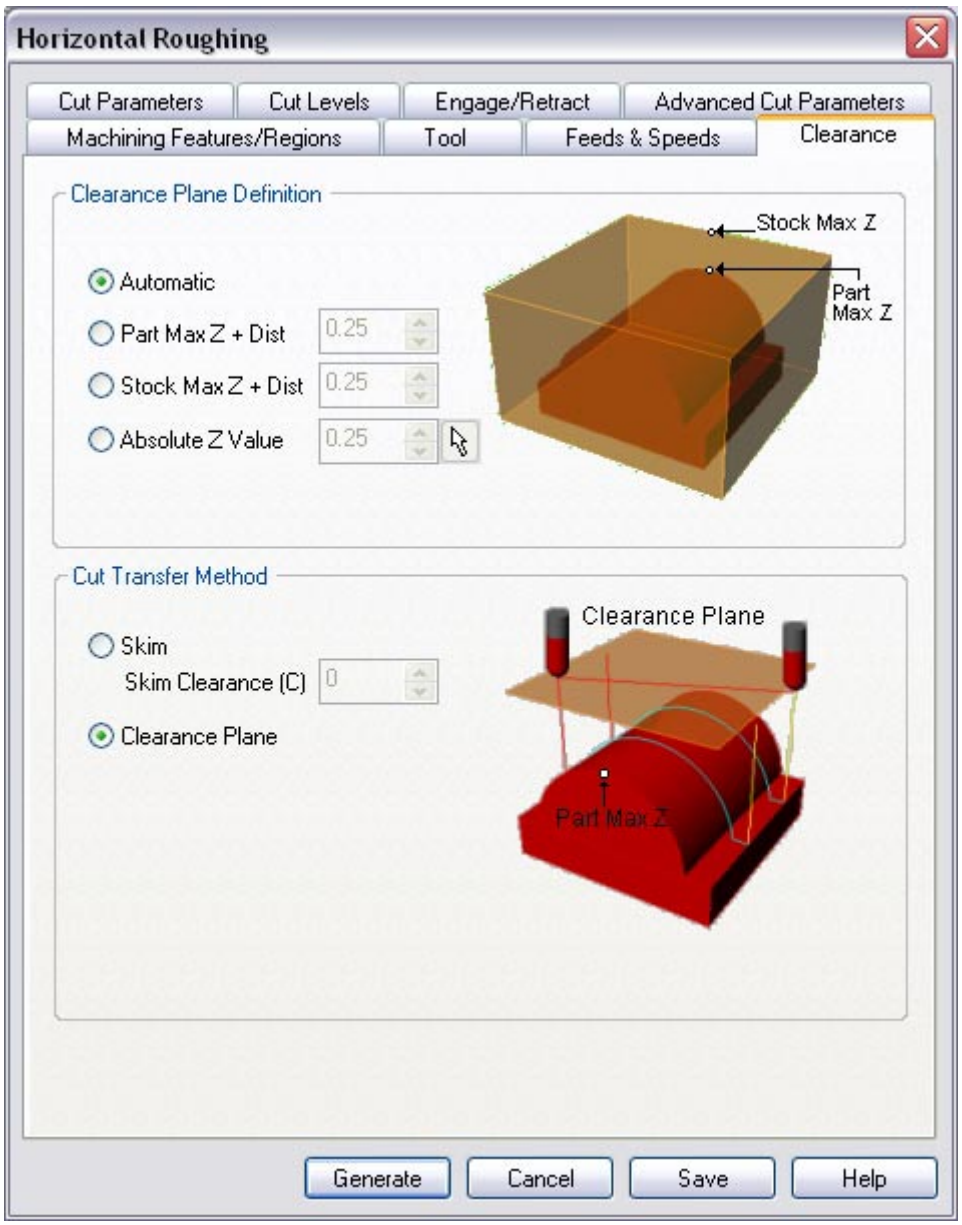
3. Switch to the Tools tab inside the 3 axis Horizontal Roughing operation and select FlatMill-0.5.


Horizontal Roughing
Cut Parameters
Cut Levels
Engage/Retract
Advanced Cut Parameters
Machining Features/Regions
Tool
Feeds & Speeds
Clearance
Tools
FlatMill-0.5
BallMill-0.25
BallMill-0.125
Tool Geometry
Diameter 0.5
Corner Radius 0
Taper 0
Tip Angle 0
Tool Properties
Tool Name FlatMill-0.5
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 45
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.



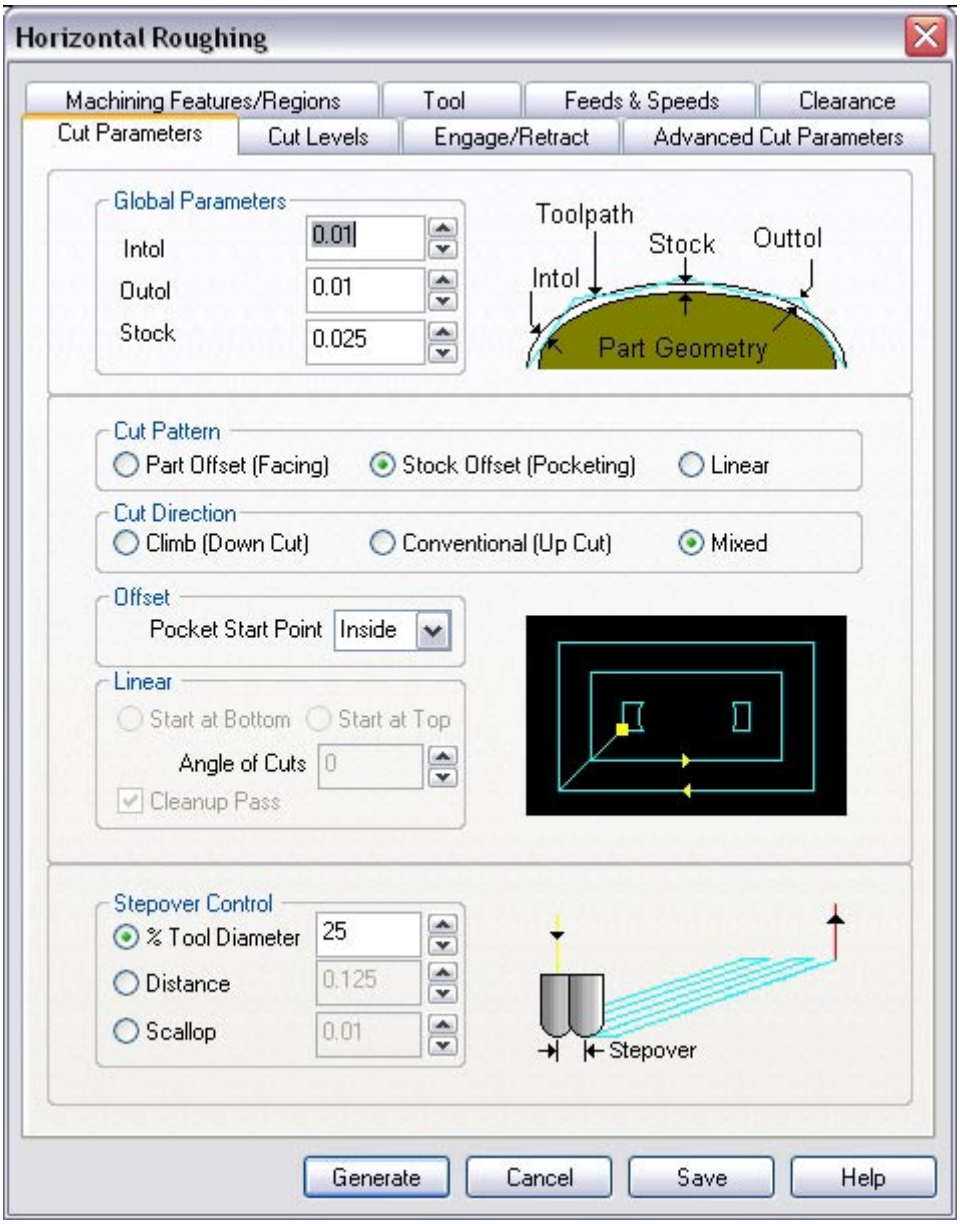
5. Switch to the Clearance Tab and set the Clearance Plane Definition to Automatic and Cut Transfer Method to Clearance Plane.



Specify Cut Parameters

6. Click on the Cut Parameters tab.

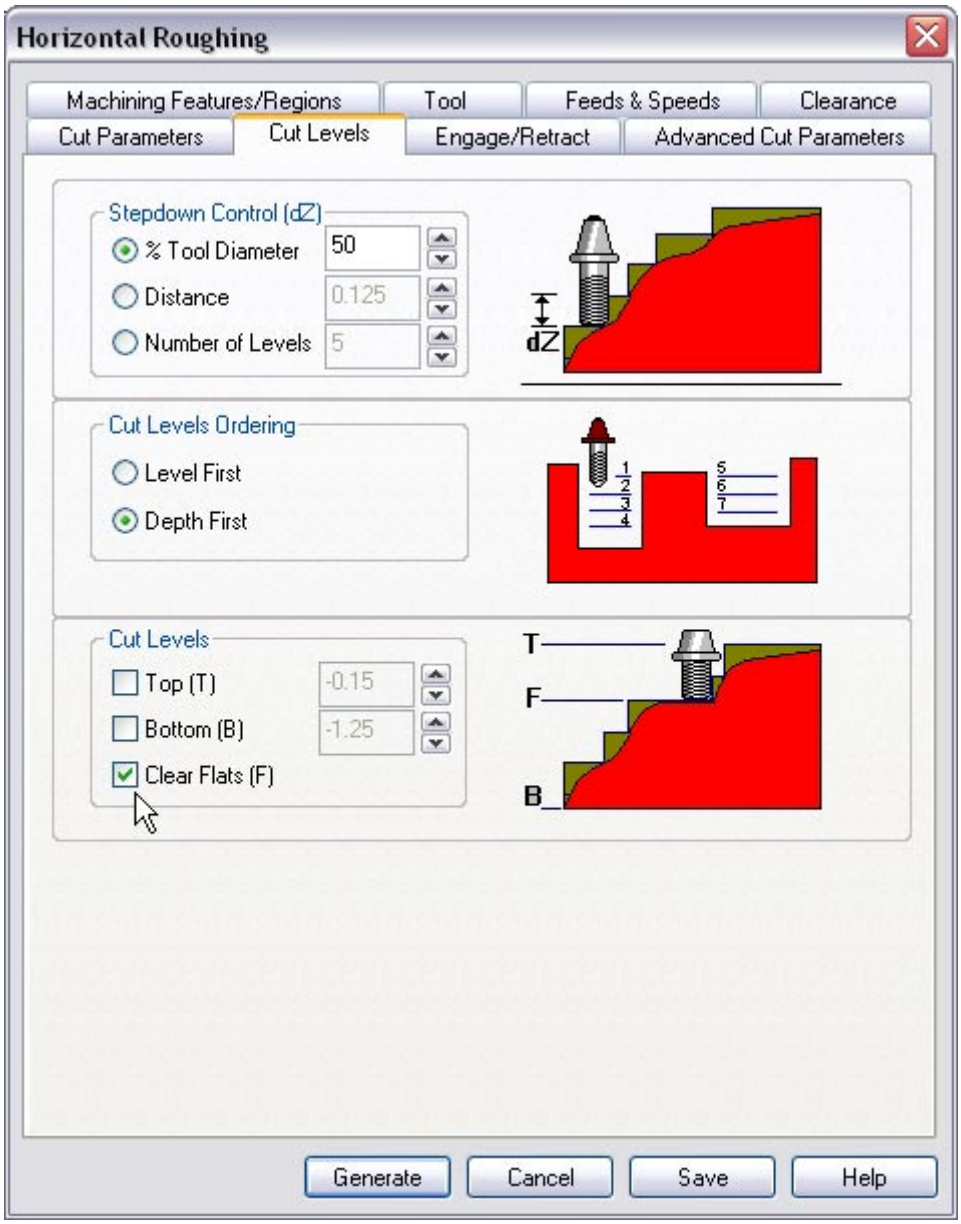
7. Set the Intol and Outol = **0.01**, Stock to leave =**0.025**, Cut Pattern to **Stock Offset**, Cut Direction = **Mixed**, Step over distance = **40** (% Tool Diameter).



8. Switch to the Cut Levels Tab.

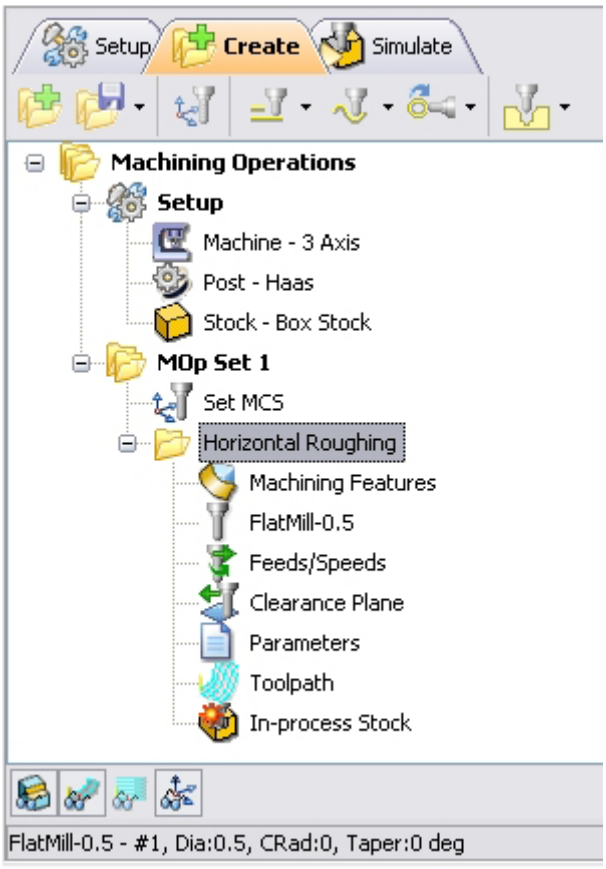
9. Use the Following Settings.

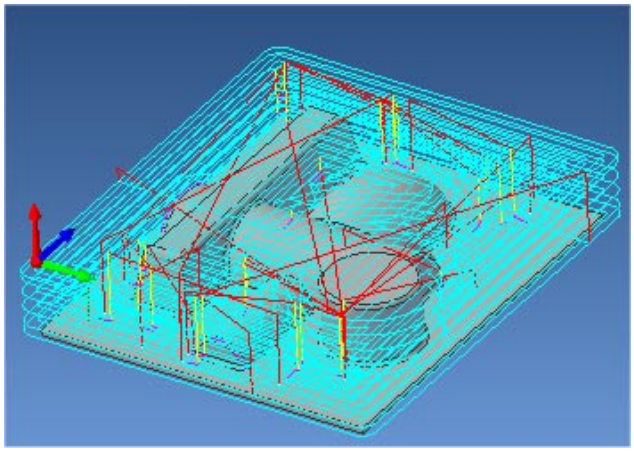
    a. Step Down Control (dZ) = **50** (% Tool Diameter).
    b. Check **Clear Flats** under Cut Levels.



10. Switch to the Engage/Retract tab and leave the entry/exit parameters as default.

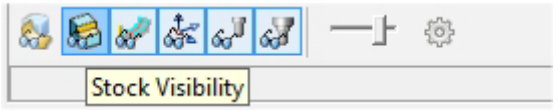
11. Click **Generate**. The 3 axis Roughing toolpath is now generated, and the Operation is listed under the Alibre CAM MOps browser.



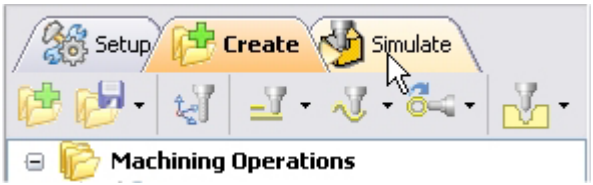


Simulate Toolpath

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.

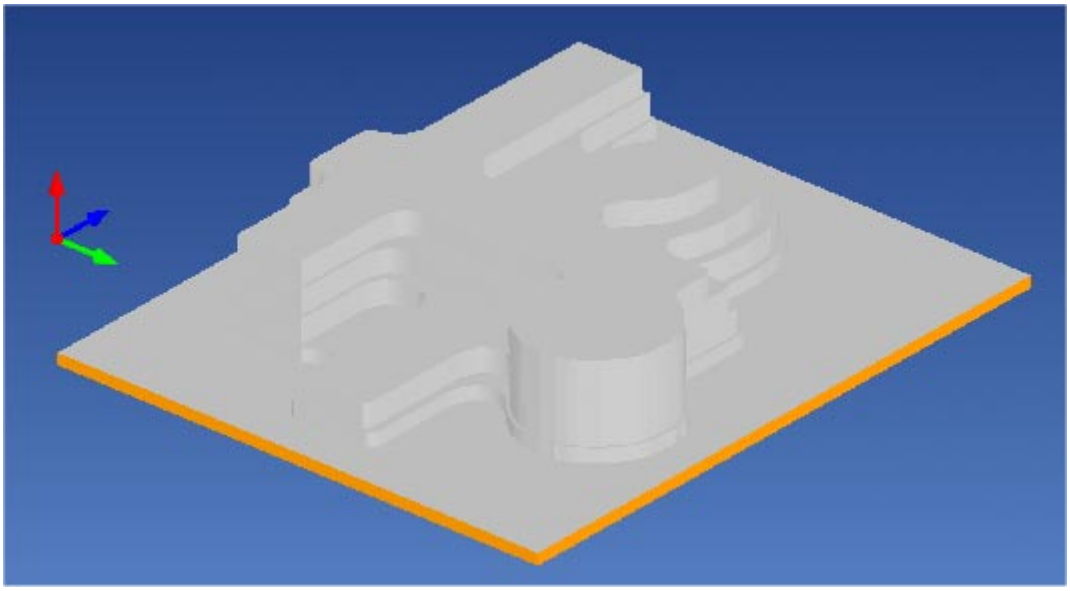


2. Select the 3 Axis Horizontal Roughing Operation and click  to launch the Alibre CAM Stock Simulation window.

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

The simulated part is as shown below.

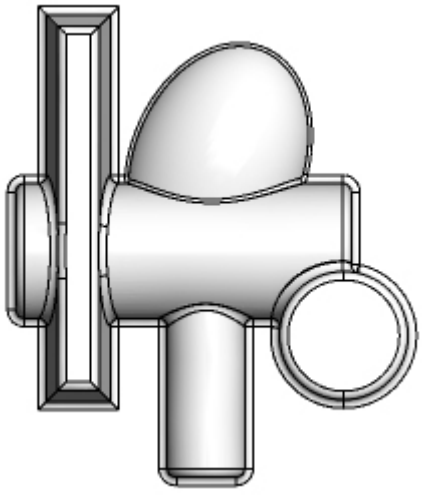
Note: You can adjust the simulation speed by selecting Simulation Preferences that is located to the bottom right corner of the Simulate tab Alibre CAM-MOps browser or from the Stock simulation window.



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

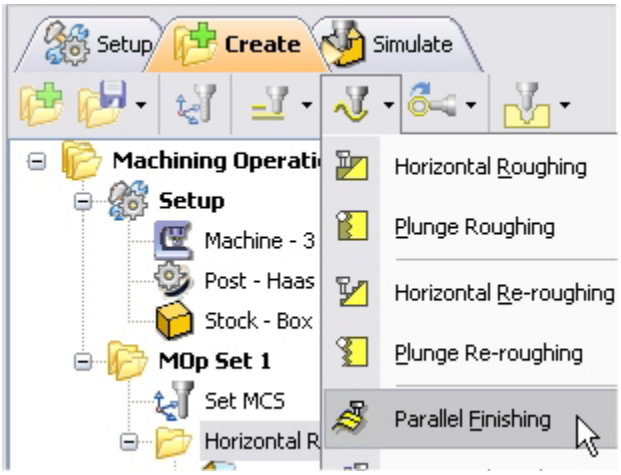
3 axis Parallel Finishing

We will now use 3 axis Parallel Finishing operation to pre-finish the part using a 0.25” Ball End Mill.



This is an efficient method of finishing or pre-finishing, typically used when part surfaces are relatively flat. A 2D linear zigzag pattern is generated on the XY plane above the part geometry. The tool moves along this cut pattern, following the contours of the part geometry below.

1. From the Create Operations tab, select 3 axis Milling and Parallel Finishing.



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

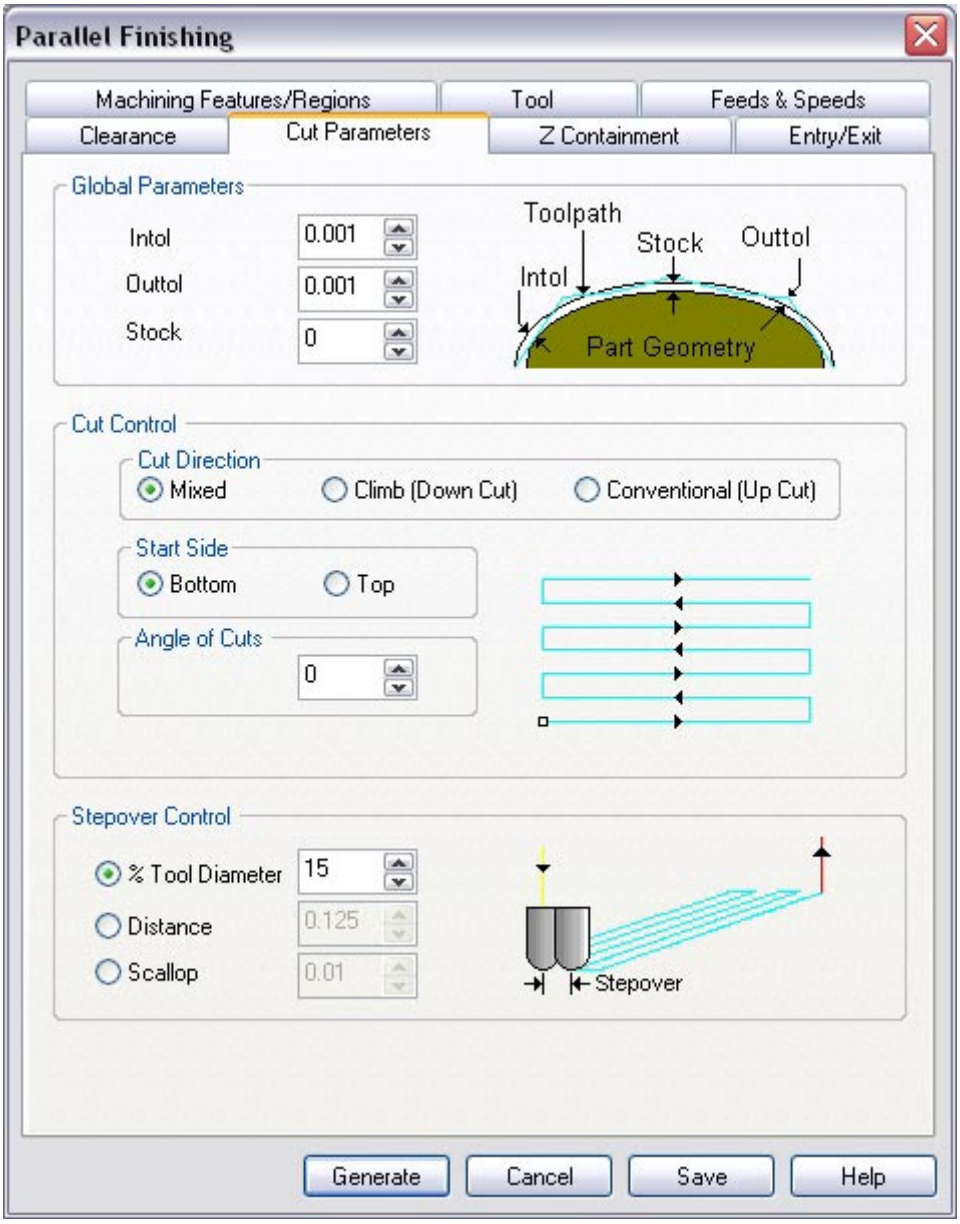
2. Switch to the Tools tab inside the Parallel Finishing operation and select BallMill-0.25.

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.

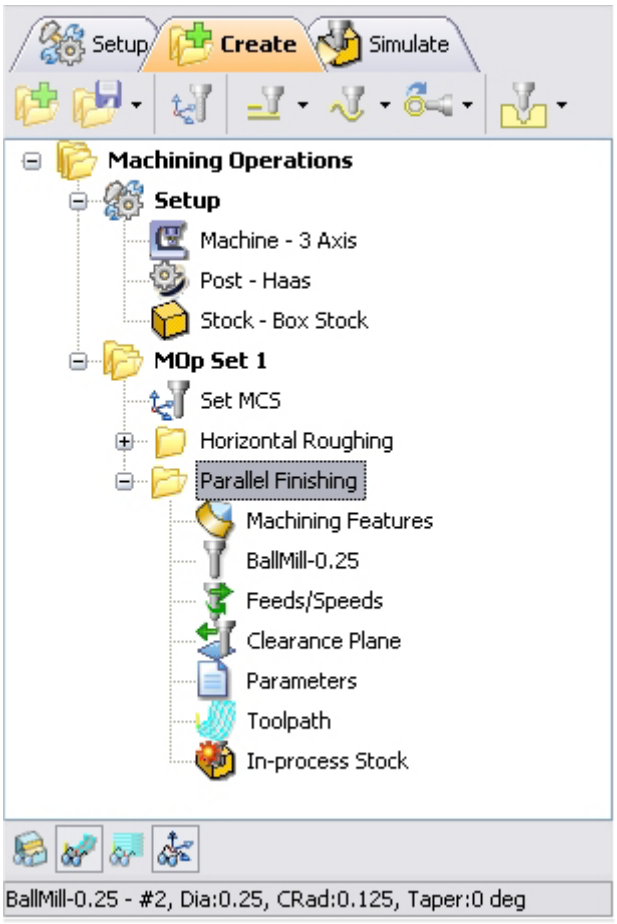
4. Switch to the Clearance Tab and set the Clearance Plane Definition to Automatic and Cut Transfer Method to Clearance Plane.

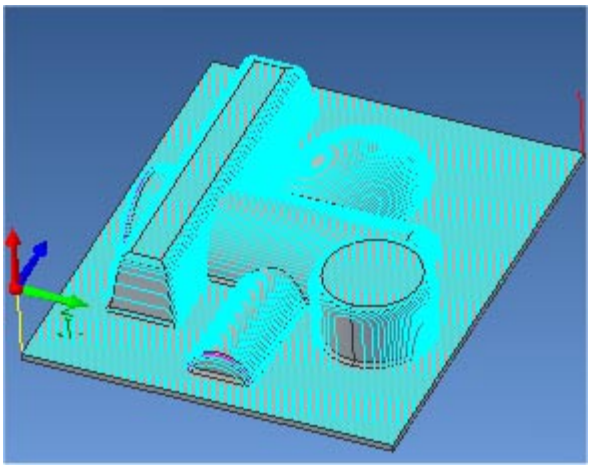
Specify Cut Parameters

5. Click on the Cut Parameters tab.
6. Set the Tolerance to **0.001**, Stock to leave =**0**, Cut Direction = Mixed, Step over distance = **15** (% Tool Diameter).



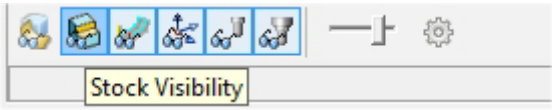
7. Click **Generate**. The Parallel Finishing toolpath is now generated, and the Operation is listed under the 3 Axis Horizontal Roughing Operation in the Alibre CAM MOps browser.





Simulate Toolpath

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

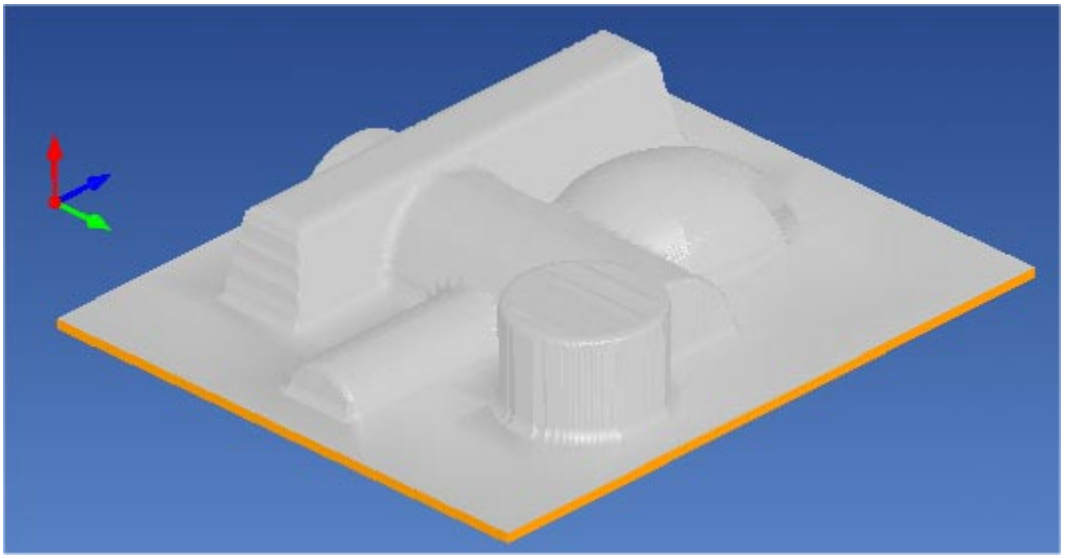


1. Switch to Simulate Tab, Select the 3 Axis Parallel Finishing and click  to launch the Alibre CAM Stock Simulation window.

2. 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.



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

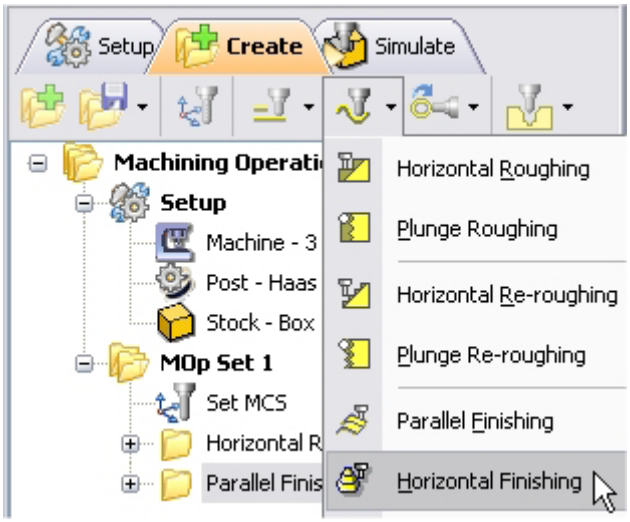
3 axis Horizontal Finishing

We will now create a Horizontal Finishing operation for machining steep area regions.

This method is used for pre-finishing or finishing in constant Z levels, typically used when the part has large vertical surfaces and when Parallel Finishing will not yield satisfactory results.

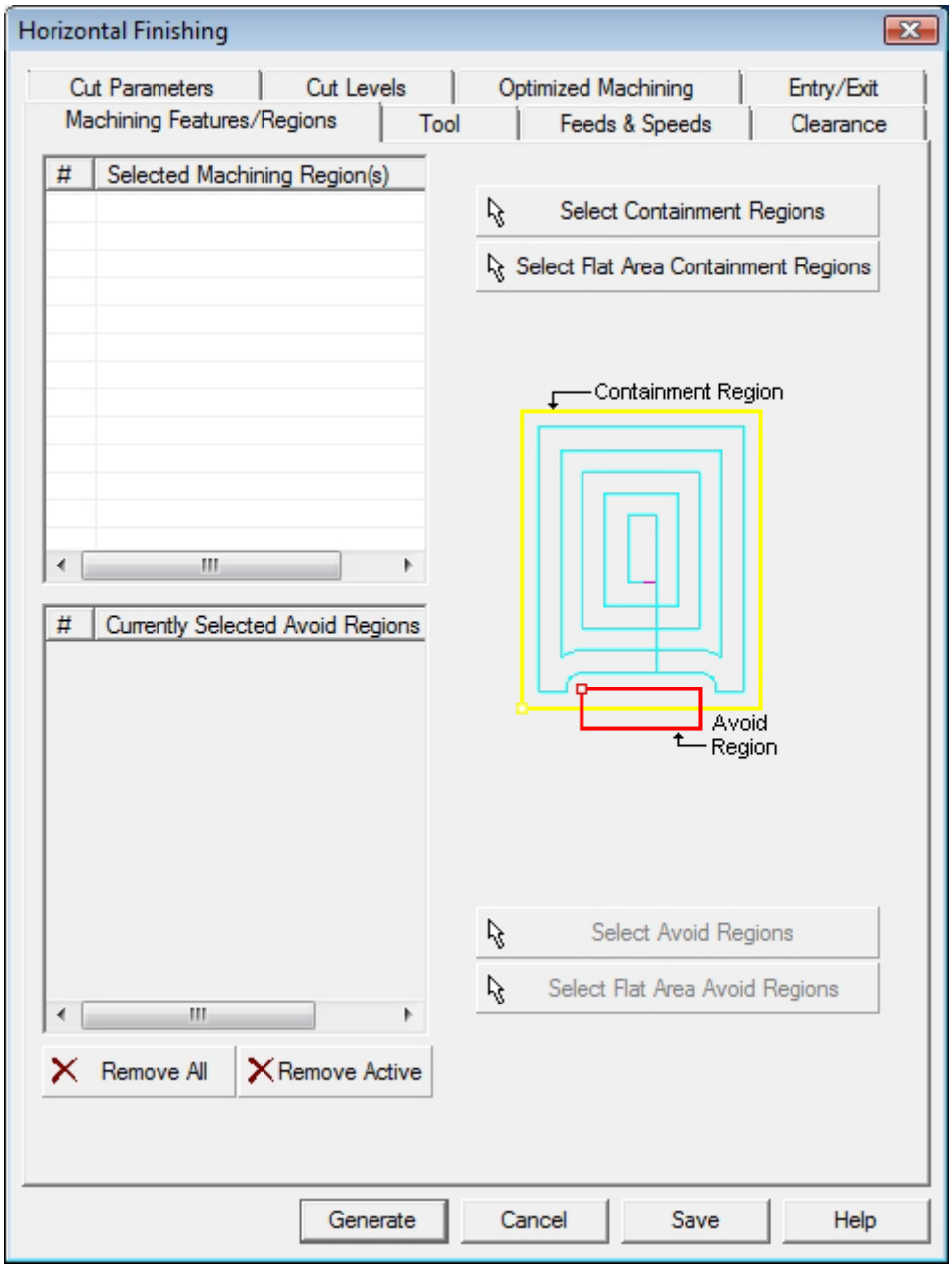
1. Switch to the Create Operations tab.

2. Select 3 axis Milling and **Horizontal Finishing**.

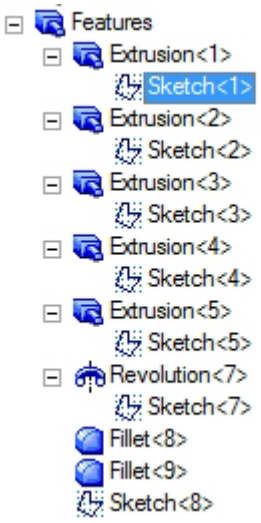


Select Machining Features/Regions

1. Go to the **Machining Features/ Regions** tab and click **Select Containment Regions.** The Horizontal Finishing operation dialog is now minimized and allows selection of the sketch geometry.



2. Switch to Design Explorer and select **Sketch1** located under Extrusion1.

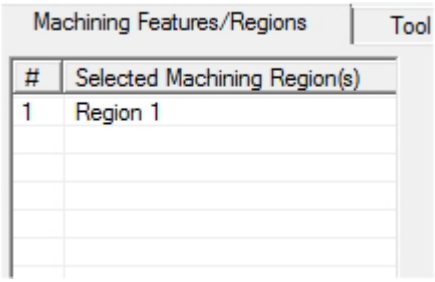


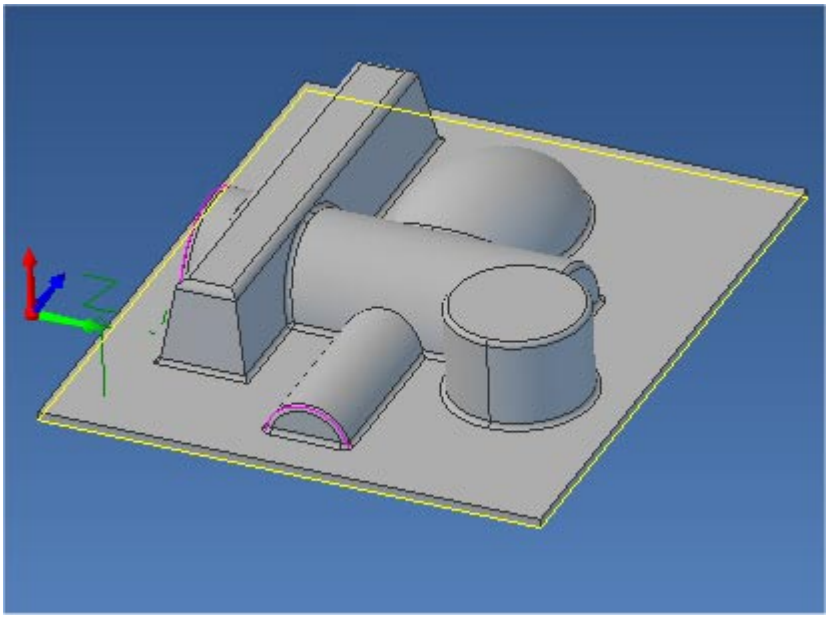
Note: You can also select the sketch from Alibre Design View. When select containment region is selected, Alibre CAM turns on Sketch selection filter to allow the user to select sketches. Hold the Shift key down to select multiple sketches.

3. Click 

to complete the selection.

The 3 Axis Horizontal Finishing operation dialog comes back up displaying the selected regions. The selected regions are also highlighted on the part.





4. Switch to the Tools tab inside Horizontal Finishing operation and select BallMill-0.125.

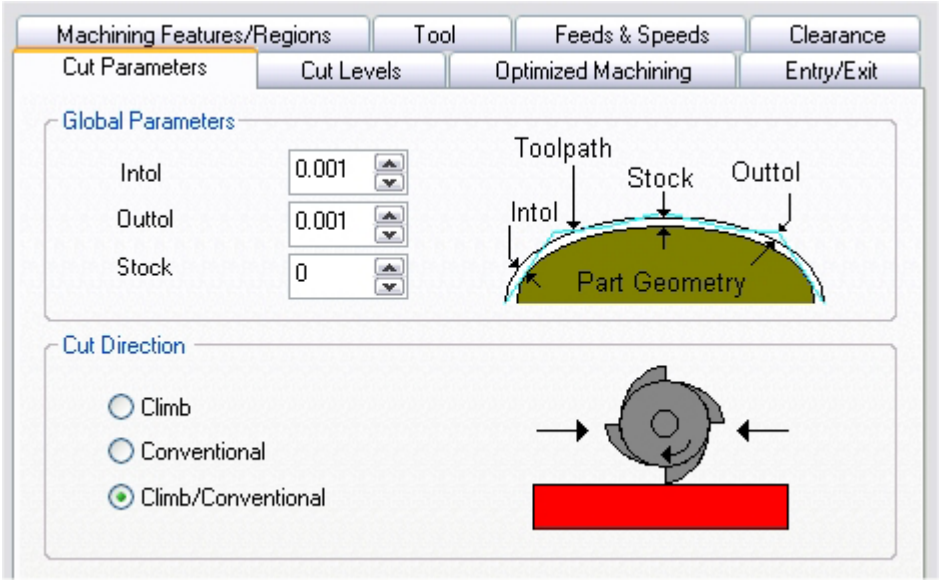
5. Click on the Feeds and Speeds tab and select Load From Tool. Alibre CAM will now get the feeds and speeds information that was set when the tool was defined.

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

Specify Cut Parameters

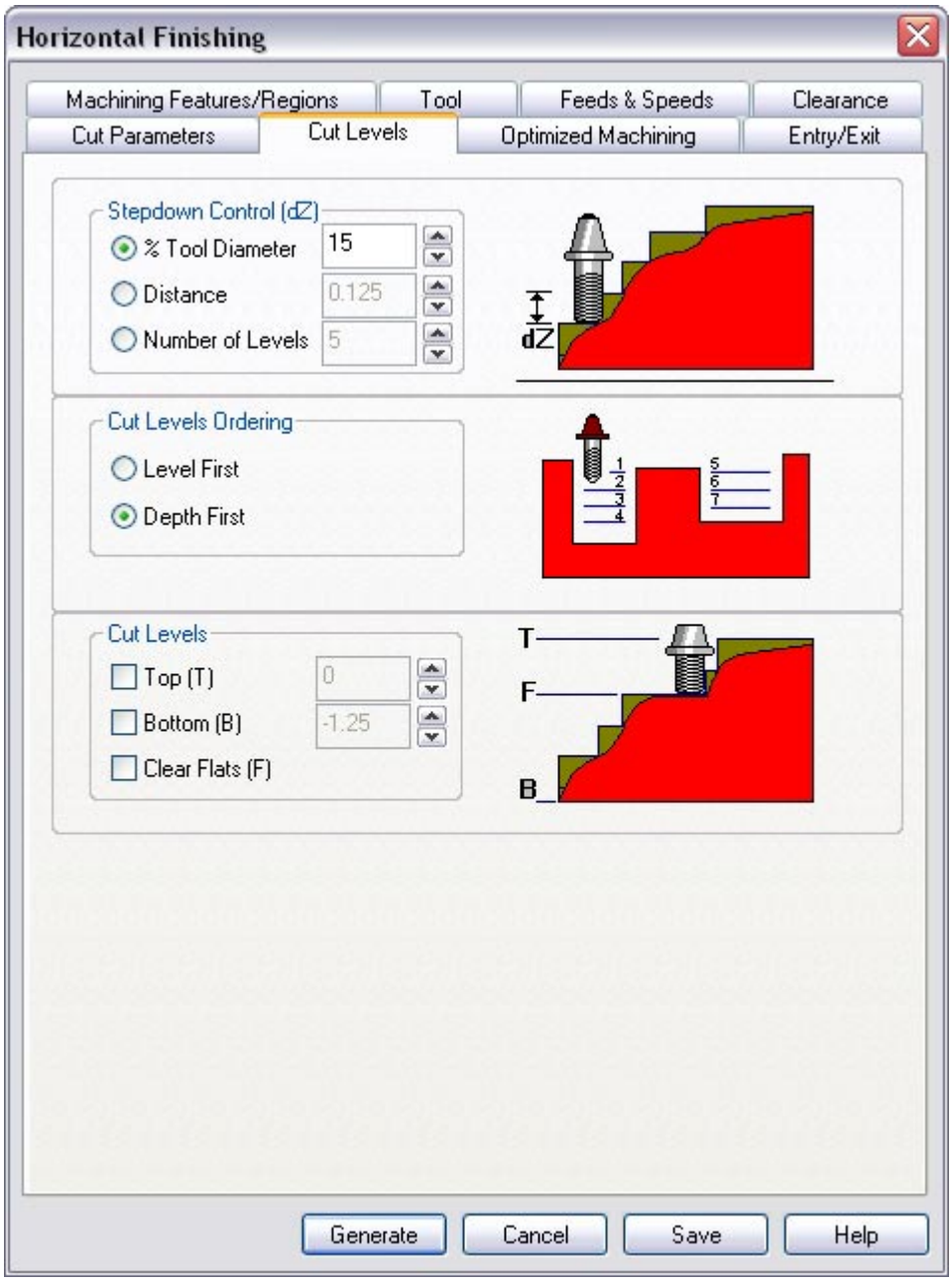
7. Click on the Cut Parameters tab.

8. Set Intol, Outol = **0.001,** Cut Direction = Climb/Conventional.



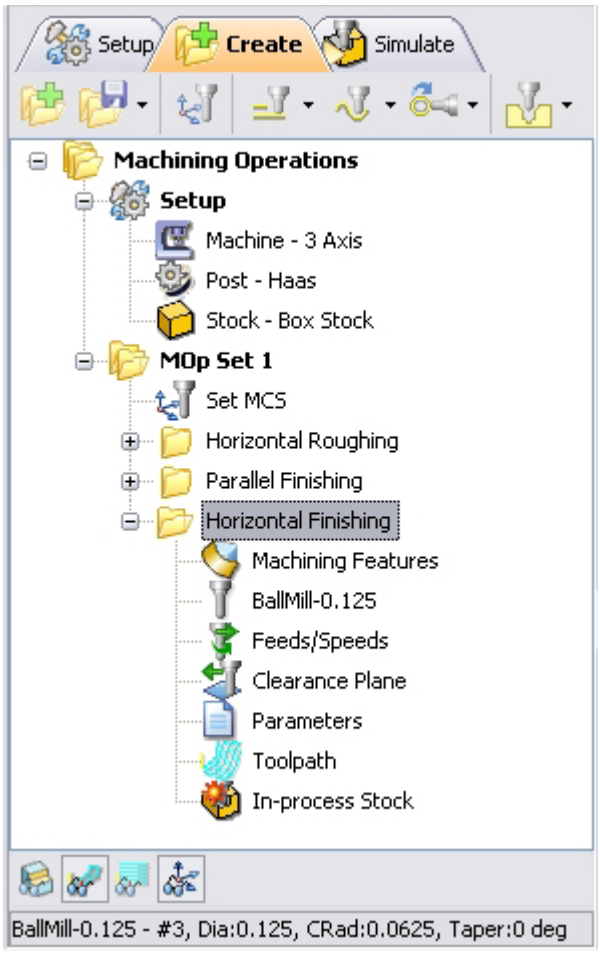
9. Switch to the Cut Levels tab.

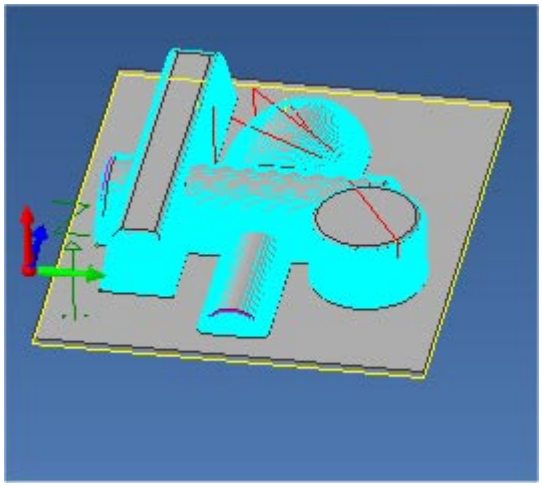
10. Set the Step Down Control (dZ) = **15** (% Tool Diameter).



11. Leave the entry/exit parameters at default.

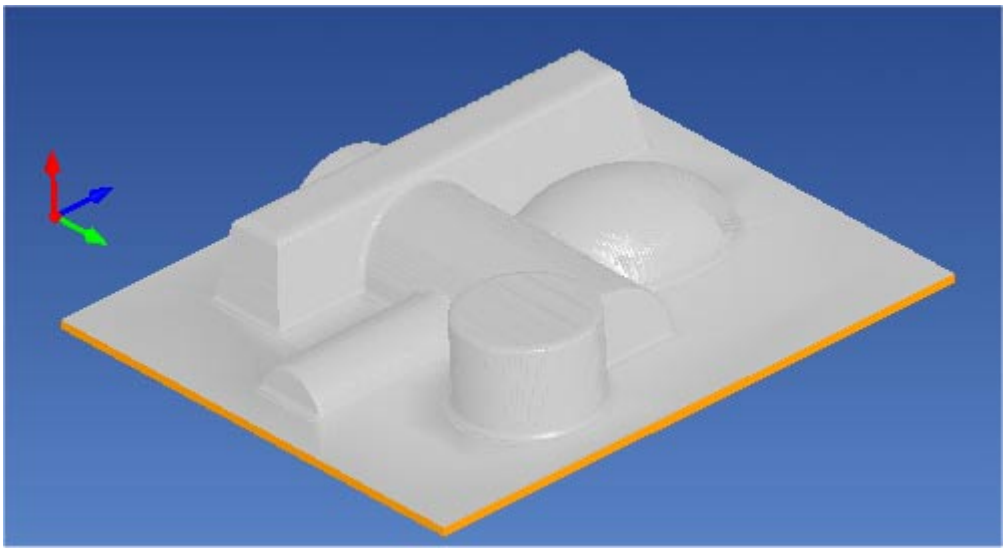
12. Click **Generate**. The Horizontal Finishing operation is now created and is listed under the MOps browser.





13. Switch to the Simulate tab, select Horizontal Finishing, and click  to launch the Alibre CAM Stock Simulation window.

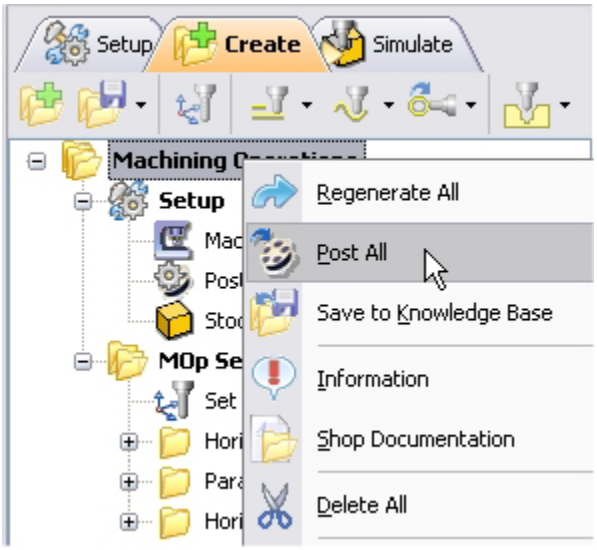
14. Click Simulate from the Stock Simulation window to run simulation.



15. 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 **3dMold.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.