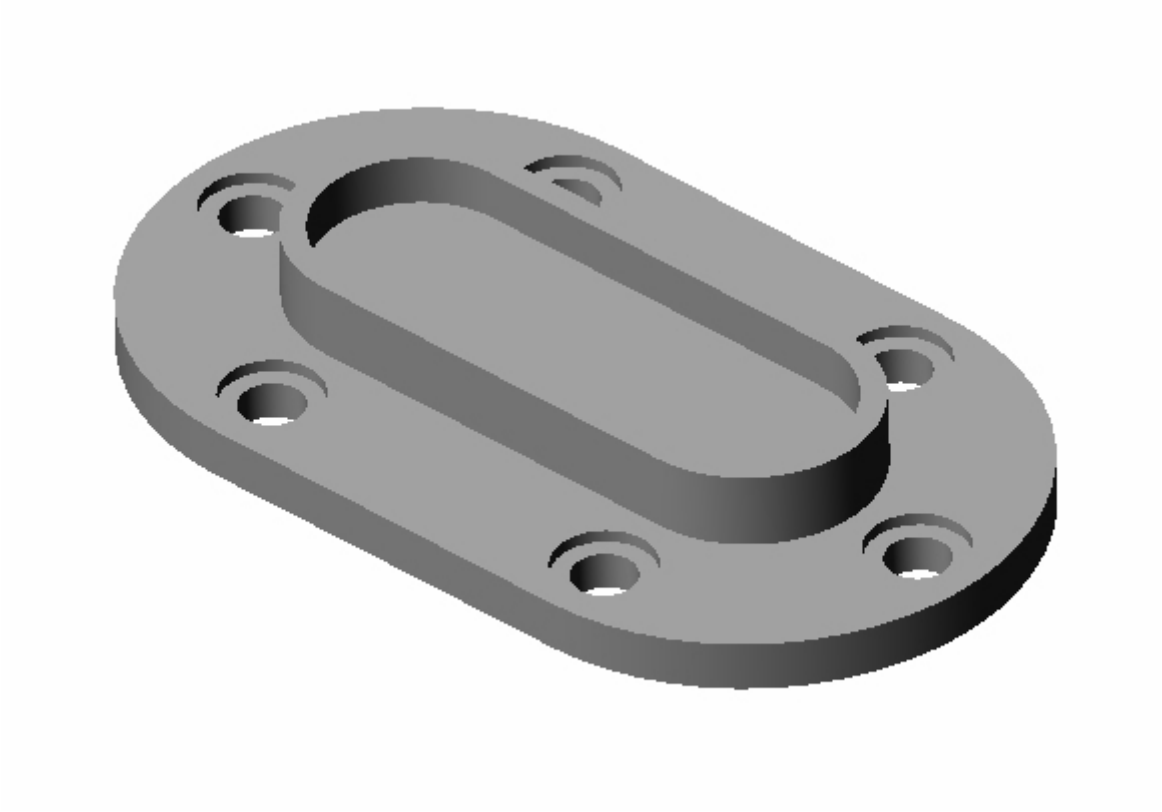
## Tutorial 3: Machining a Shaft Base



Using
2½ Axis Facing,
Pocketing,
Hole Pocketing &
Profiling Toolpath methods

## Introduction

This tutorial will illustrate machining of a prismatic part such as this Shaft Base using 2-1/2 milling operations. Even though we have created a 3-D representation of the part, it will be seen later on that we can machine this using just 2D Sketches. The reason we are able to do this is because of the prismatic nature of this model. This tutorial will introduce the usage of 2 ½ axis machining for a simple one-sided part. We will use profiling, pocketing and hole pocketing operations.

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 Shaft Base

- We will machine the shaft base completely using 2 ½ axis-machining operations.
- The starting material for the Shaft Base is soft wood and the size is 5.5 x 3.25 x 0.625 inches.
- The wooden sheet will be held to the machine table or the spoil sheet on the table using double-sided tape.
- The part will be machined using a single ¼ inch flat end mill.
- Determining the sequence of machining operations
    - The first operation would involve machining around the boss using a 2 ½ Axis Facing Operation.
    - The next step would involve machining the area inside the boss. We will use 2 ½ Axis Pocketing Operation, which is ideal removing material inside a specified region.
    - We are now down to the level where the step holes need to be machined. As the holes are circular we will use 2 ½ Axis Hole Pocketing operation to machine the holes to its depth in 2 separate operations.
    - Finally we will cut out the shape of the part from the rectangular using a contour toolpath. This is accomplished using a 2 ½ Axis Profiling Operation that separates the finished part from the stock material.

## Main Programming Steps

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

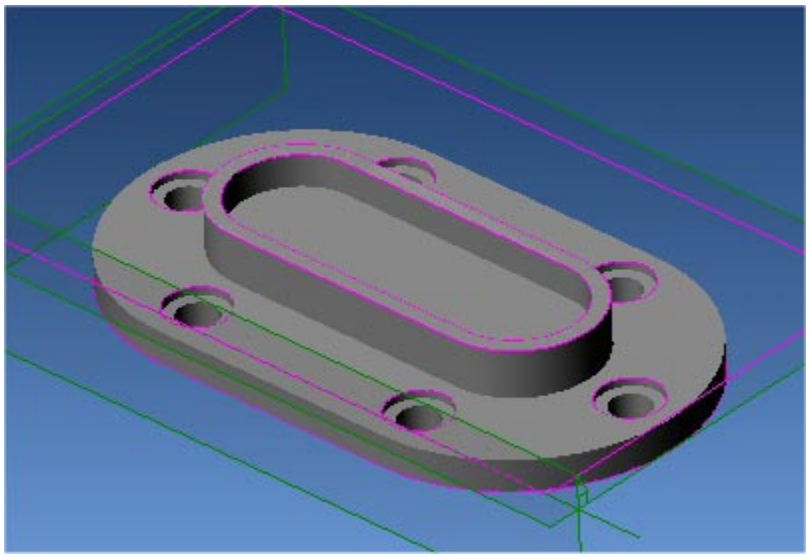
- Create the Stock geometry
- Set the Machine zero point or Locate geometry with respect to the machine coordinates
- Create / Select the tool used for machining
- Set the feeds and speeds
- Set the clearance plane for the non-cutting transfer moves of the cutter
- Select the machining regions for containing 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

Use the Alibre Design menu bar or the Standard toolbar buttons to create, load and save part geometry.

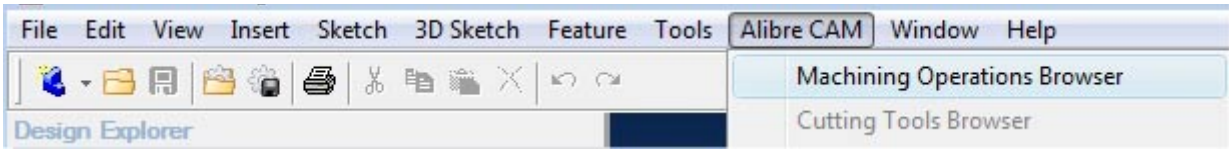
1. Select File / Open Part, or click the Open Part icon from the Alibre Design standard toolbar.
2. From the **Open** dialog box, select the **ShaftBase.AD_PRT** file from the **Tutorials** folder in the Alibre CAM installation folder. (Default location C:\Program Files\MecSoft Corporation\Alibre CAM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

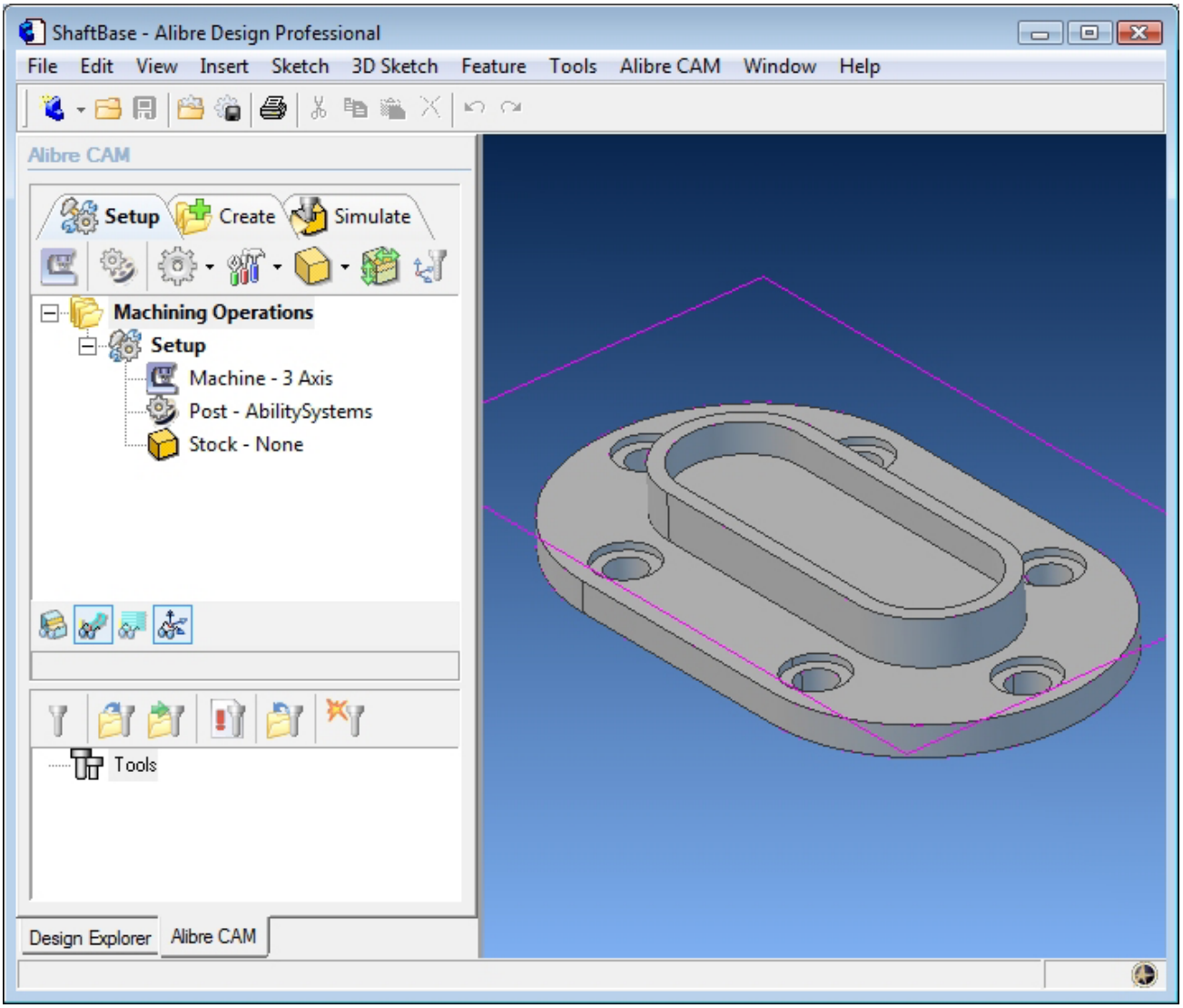


**Note:** You must work in shaded mode in order to be able to visualize toolpaths created in Alibre CAM. It is suggested for best visual performance with Alibre CAM to work with only one view port open and the view port operating in shaded mode.

## Loading the Alibre CAM Browser

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





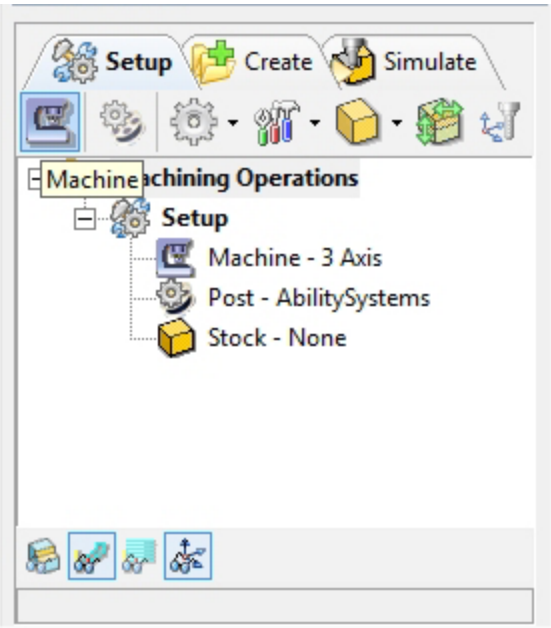

The Alibre CAM browser (MOPs and Cutting Tools) is now loaded and is docked over the Design Explorer. You can toggle between the MOPs browser and the Design Explorer from the tabs at the bottom of the window. It can be hidden by un-checking Browser on the Alibre CAM menu bar. To re-display the hidden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

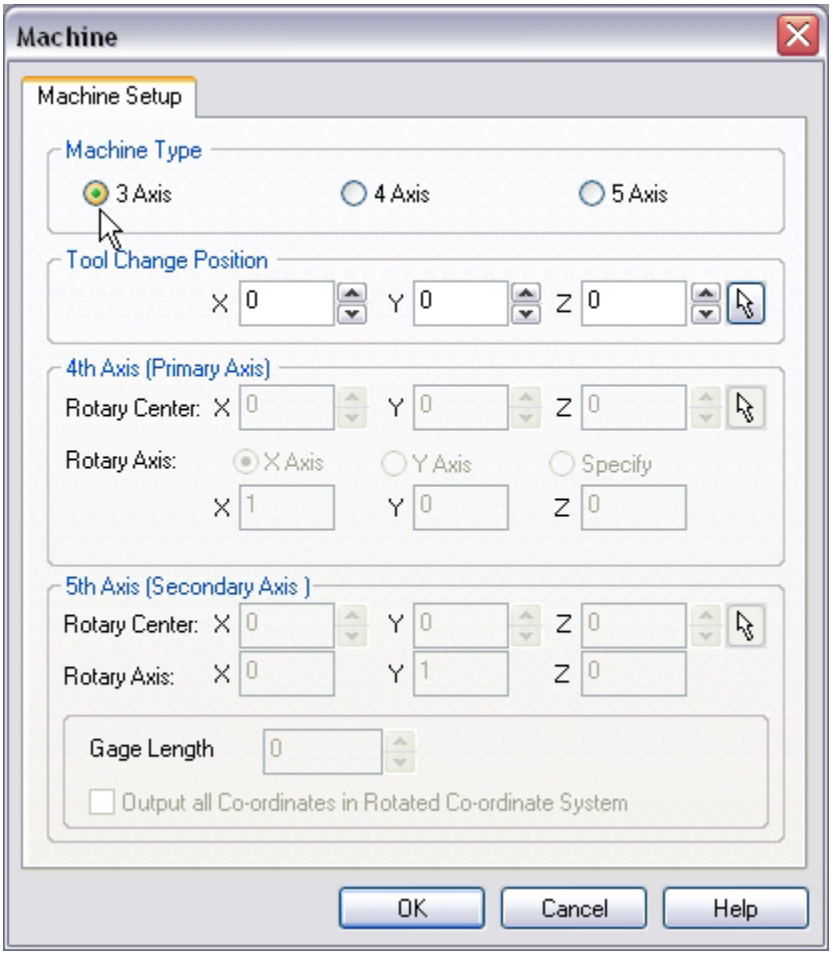
The Setup tab allows the user to specify Machine Setup, Select Post Processor, Stock Geometry, Machine Coordinate System (Machine Zero) & Preferences.

## Setup Tab

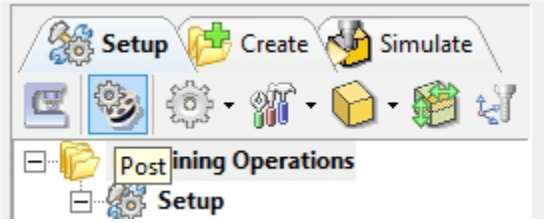
1. Click on the Setup tab located under the MOPs Browser and select Setup Machine



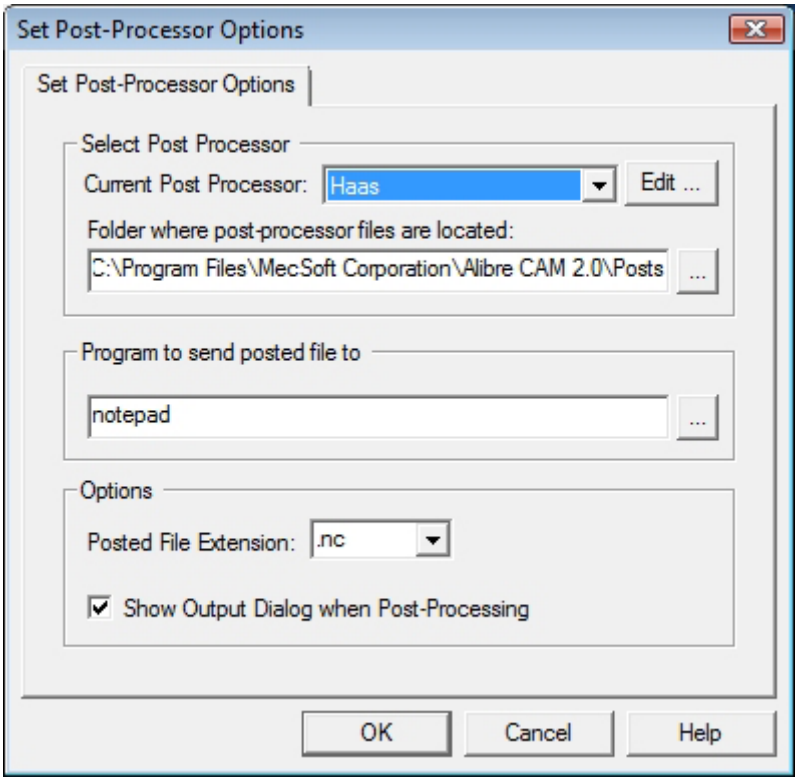

2. Set the Machine type to 3 axis




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



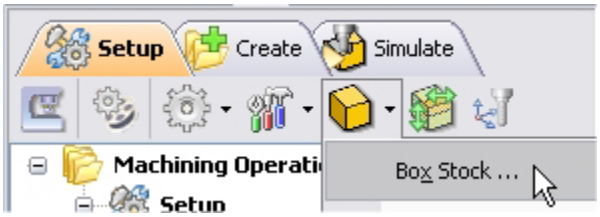

4. Set the current post processor that is on your controller. We will select Haas as the post processor for this exercise. Set the posted file extension type to .nc



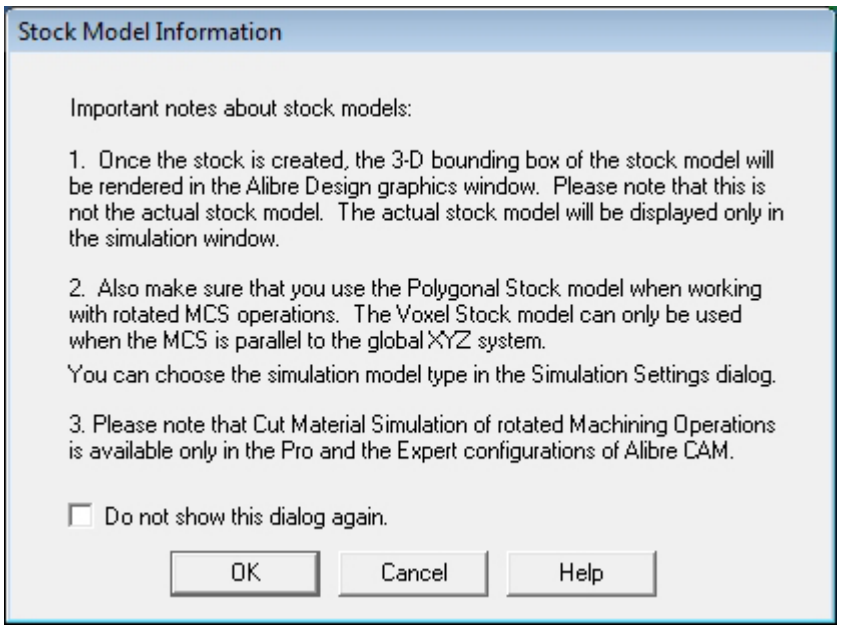

*Note: By default post processor files are located under*
*C:\Program Files\MecSoft Corporation\Alibre CAM 2.0\Posts*
*The program to send the posted output is set to notepad. This would output the G code to a notepad.*

## Create/Load Stock

5. Select Create / Load Stock from the Setup tab and choose Box Stock




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

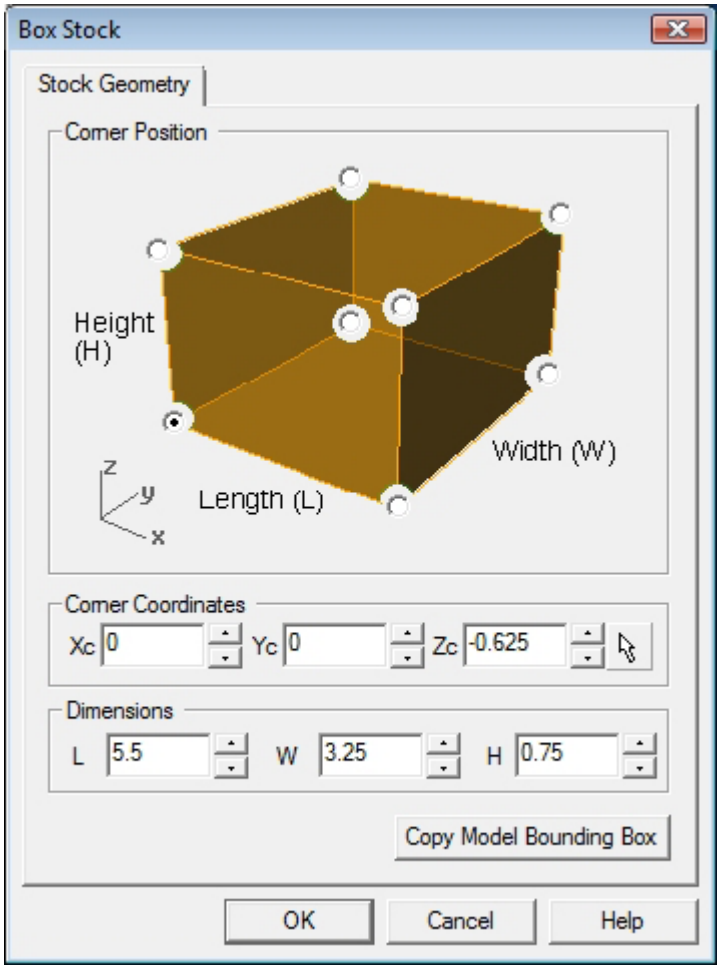



Click OK

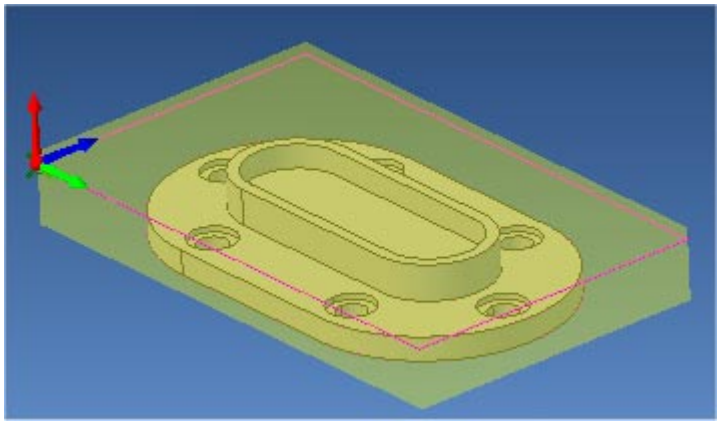
User can turn off this dialog by selecting Do not show this dialog again located on the bottom of the message window.

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

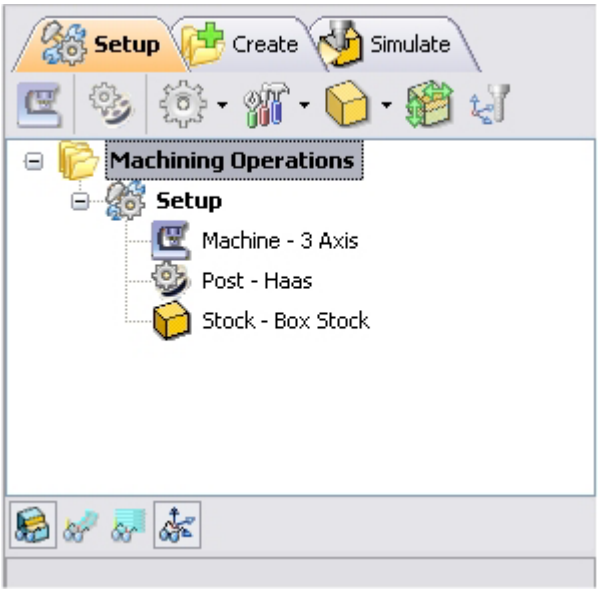
6. This brings up the Box Stock parameters. Set the Length (L) = 5.50, Width W = 3.25, and Height (H) = 0.75. Leave the other parameters as default, and click OK.




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



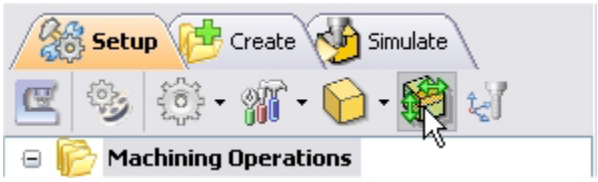
The setup tab now displays the following information: Machine Type, Post Processor, and Stock type as show below.



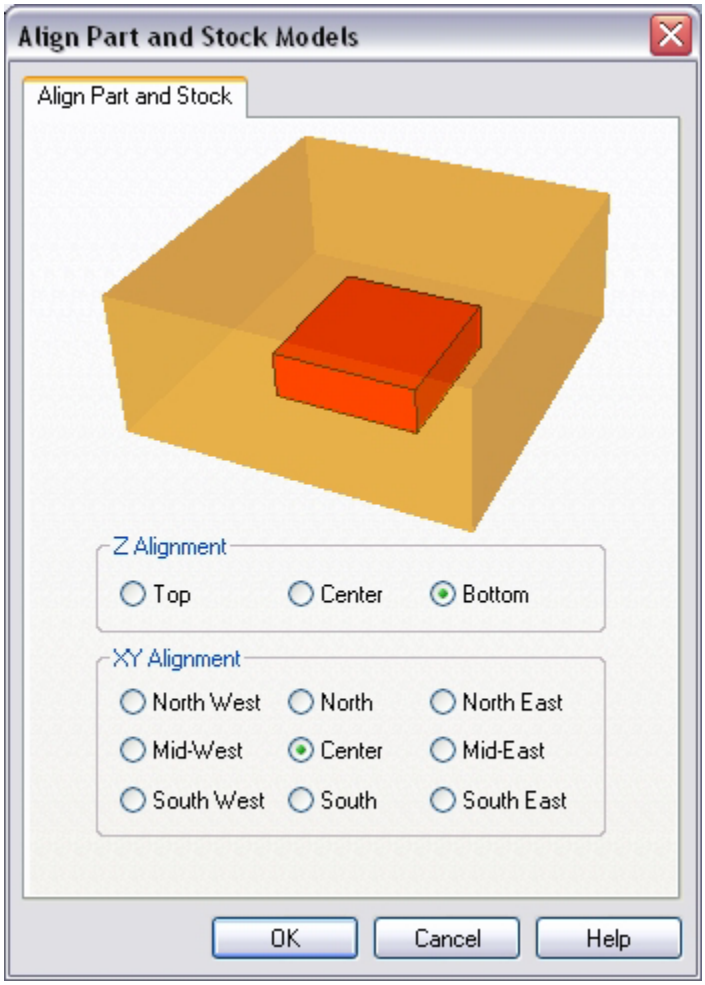

## Align Part and Stock

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



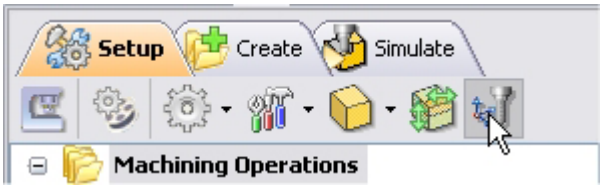

2. Set Z alignment to **Bottom** and XY alignment to **Center.** (This would align the stock to the bottom of the part in Z and center in XY)



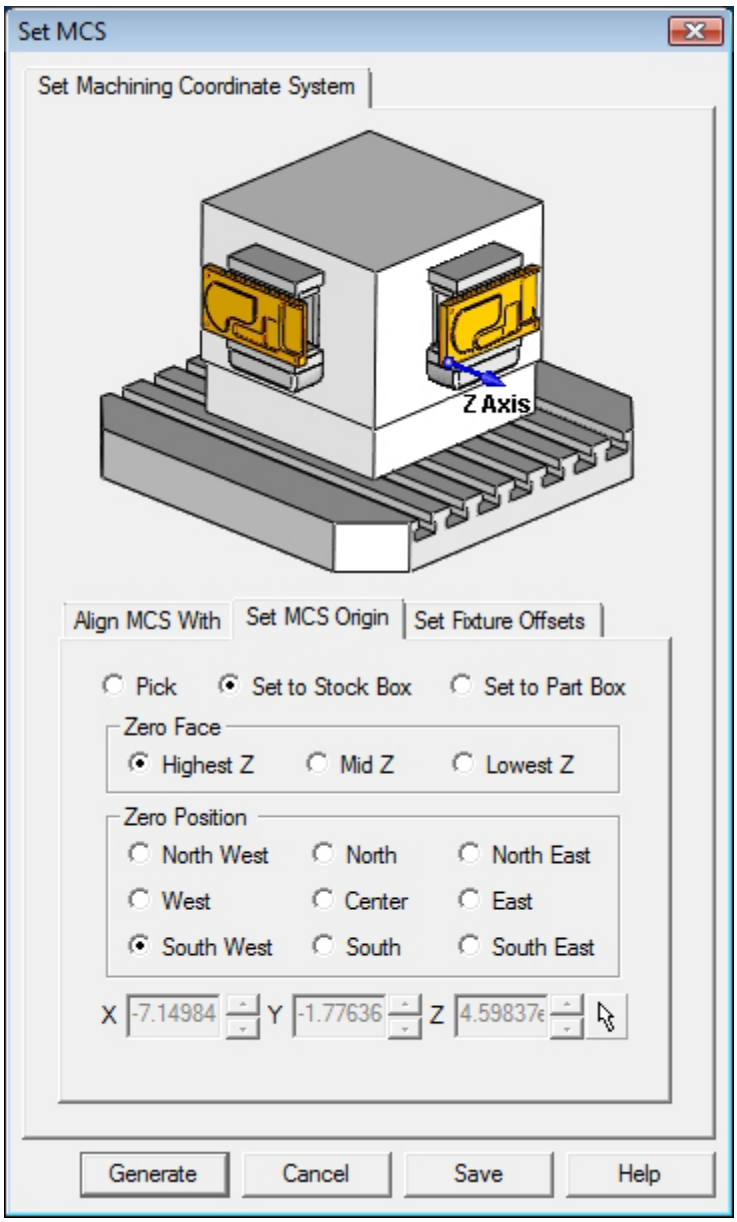

## Set Machine Coordinate System (MCS)

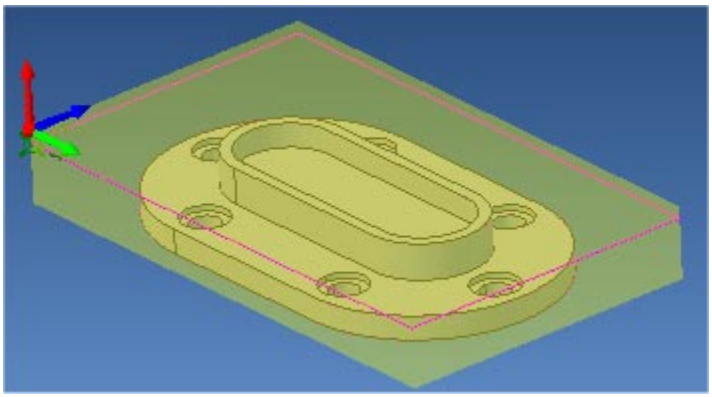
The steps below help you determine the machine home (also known as machine zero or tool touch off point) for the part/stock geometry.

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.



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

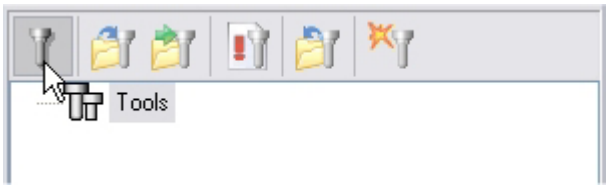
*Note: You can change the stock model transparency under standard mode by selecting Simulation Preferences that is located at the bottom of the MOps browser.*

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

## Create Tools

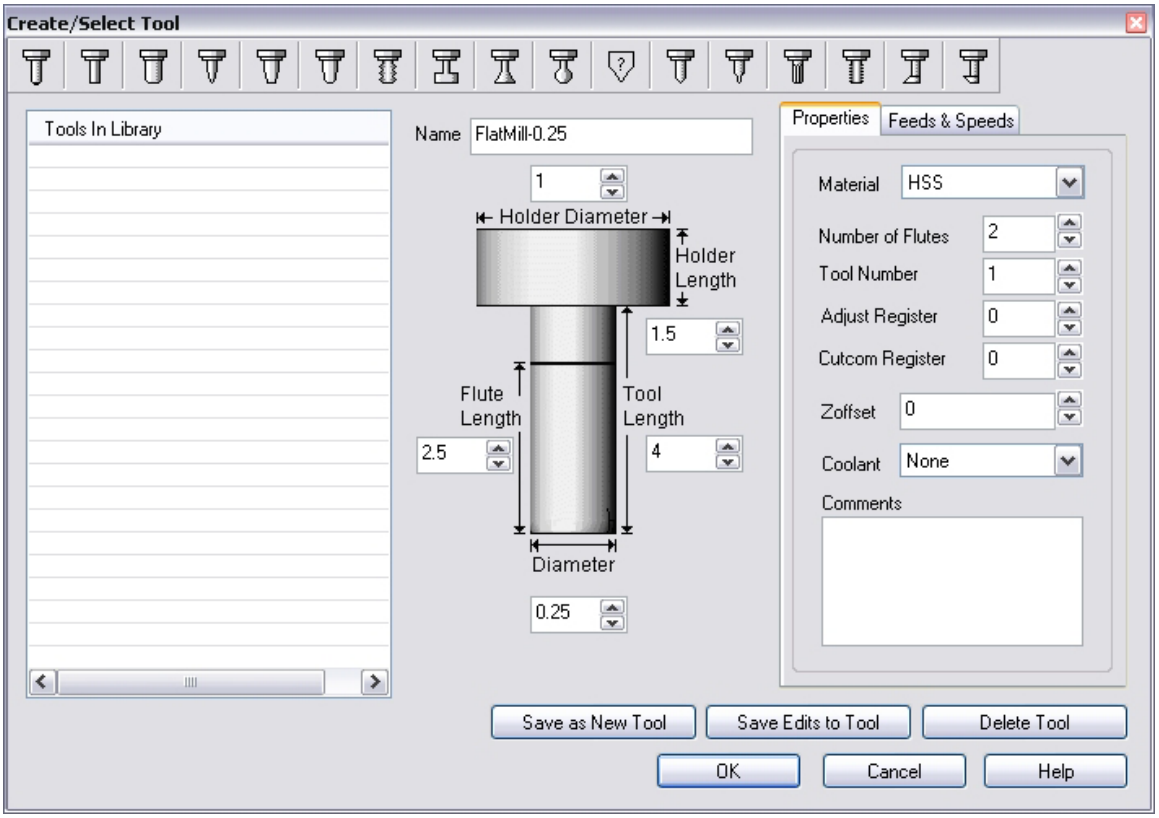
To machine the above part we will now create a ¼ inch (0.25”) Flat End Mill.

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





2. Set the tool name as **FlatMill-0.25**, Tool Diameter = **0.25.** Under the Properties tab, set Tool Number = **1**.

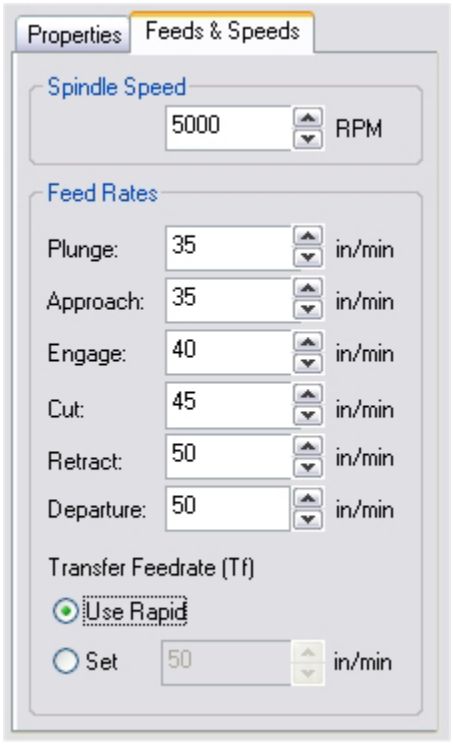


## Setting Feeds and Speeds

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.

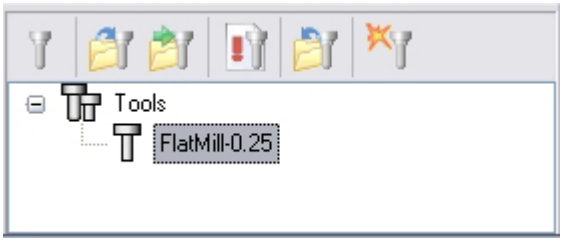
3. Switch to the Feeds & Speeds tab inside the create/select tool dialog.

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.*




The created tool is now listed under the Alibre CAM Tools browser.

## Create Machining Operations

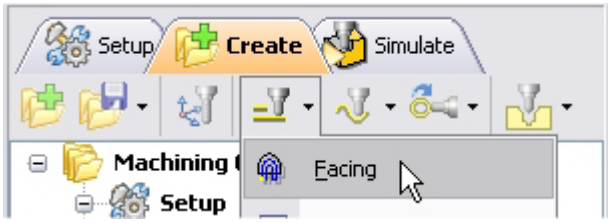
We will machine the Shaft Base using 4 different machining operations – Facing, Pocketing, Hole Pocketing and Profiling.

The stock geometry has a thickness of 0.75” and the finished part is 0.625”. We will create a 2.5 axis facing operation to mill the 0.125” thickness of material from the stock geometry.

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


## 2 ½ Axis Facing

1. Select 2.5 Axis Milling and choose Facing.




2. This brings up the 2 ½ Axis Facing Operation Dialog. We will now go over the steps for creating the toolpath.

## Select Machining Features/Regions

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

Regions are sketches that already exist in your model, or separate new sketches you create within Alibre Design that coexist with your part, but do not contribute to its geometry (the sketch is not used for a feature). Regions serve different purposes in 2½ and 3 axis milling. In 2½ axis milling, however, regions determine the entire area to be milled. Part and stock geometry are not considered, and the tool always moves at a fixed Z level. Therefore, regions act as the drive curves, and their geometry is projected onto the stock.

Regions must be selected before they can be used in an operation. Creating a region does not make it active; you must use one of the **Select Regions** tools before creating the toolpath.

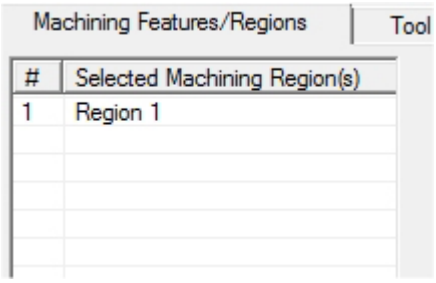
4. Switch to Design Explorer and select **Sketch7**. This selects the outside rectangle.

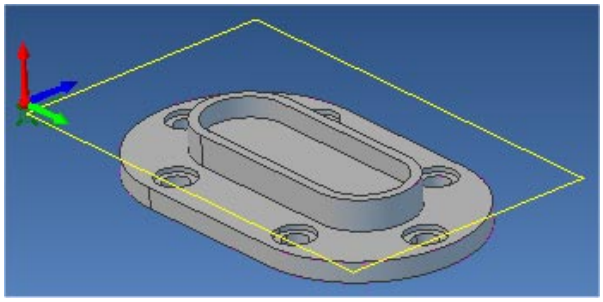
   Note: You can also select the sketch from Alibre Design View. When select containment region is selected, Alibre CAM turns on Sketch selection filter to allow the user to select sketches. Hold the Shift key down to select multiple sketches.

5. Click 

to complete the selection.

The 2 ½ Axis Facing operation dialog comes back up displaying the selected region. The selected region is also highlighted on the part.



6. Switch to the Tools tab inside the 2 ½ Axis Facing operation and select FlatMill-0.25.

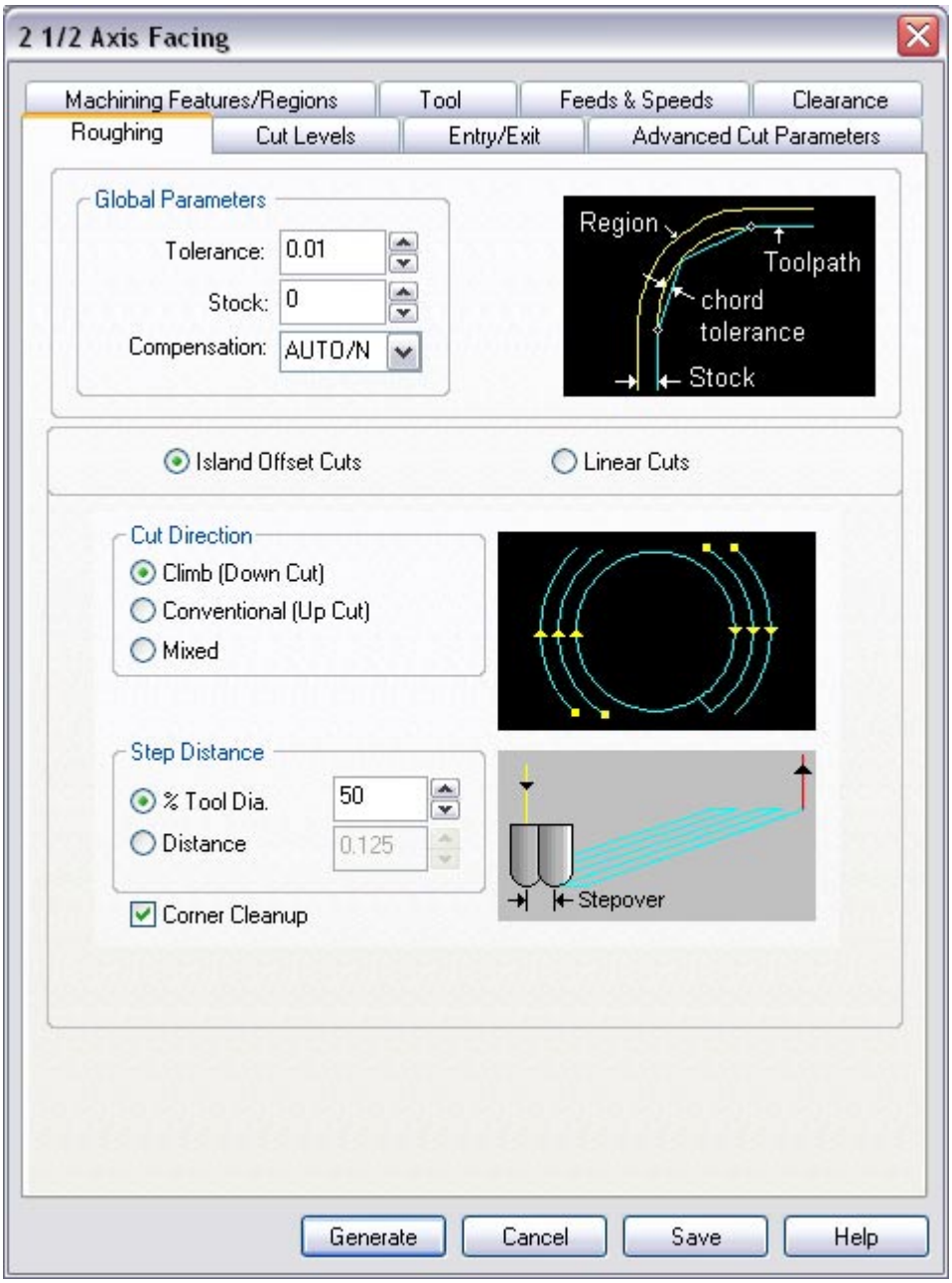
7. Click on the Feeds and Speeds tab and select Load From Tool. Alibre CAM will now get the feeds and speeds information that was set when the tool was defined.

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

## Specify Cut Parameters

9. Click on the Roughing tab.

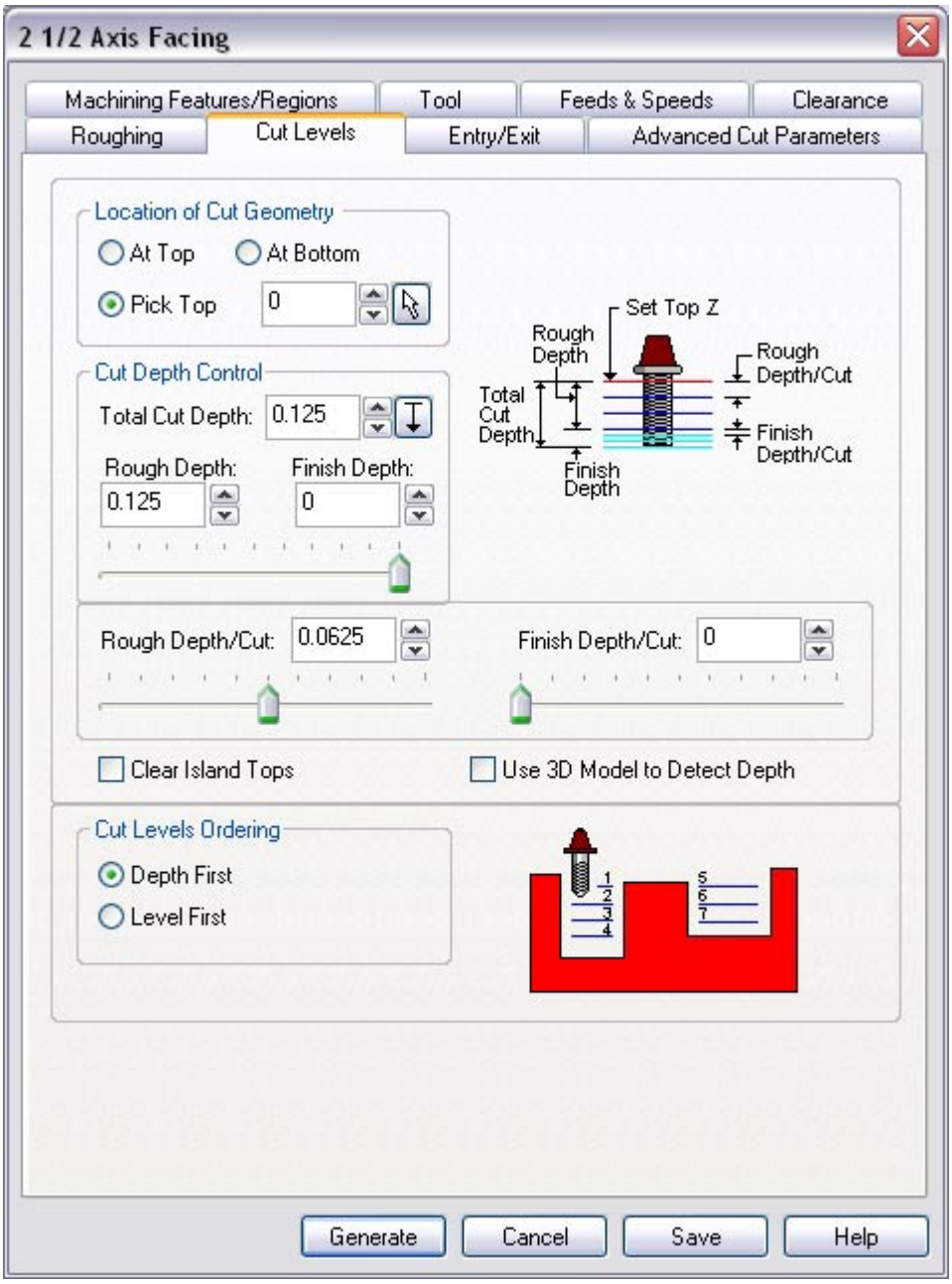
10. Set the Tolerance to **0.01**, Stock to leave to **0**, Cut Pattern to **Island Offset Cuts**, and Step Distance to **50** (% Tool Diameter).




11. Switch to the Cut Levels Tab.

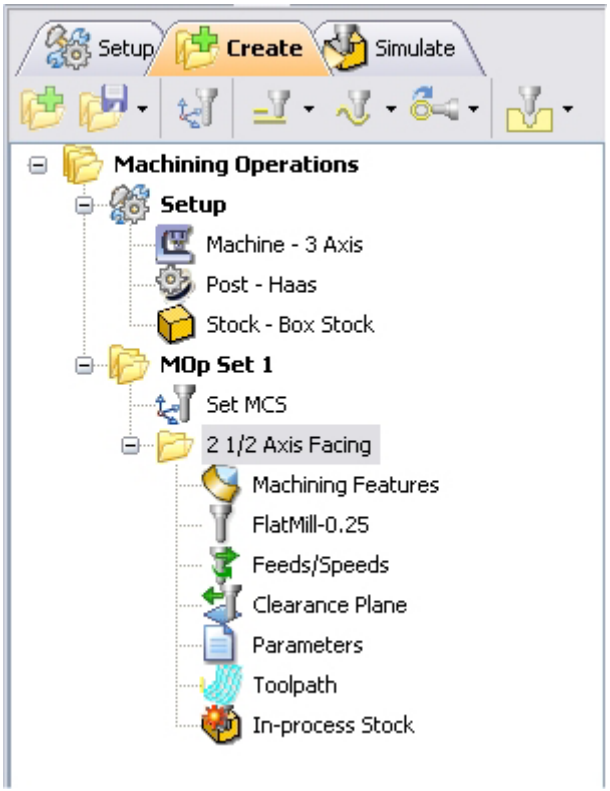
12. Use the Following Settings.

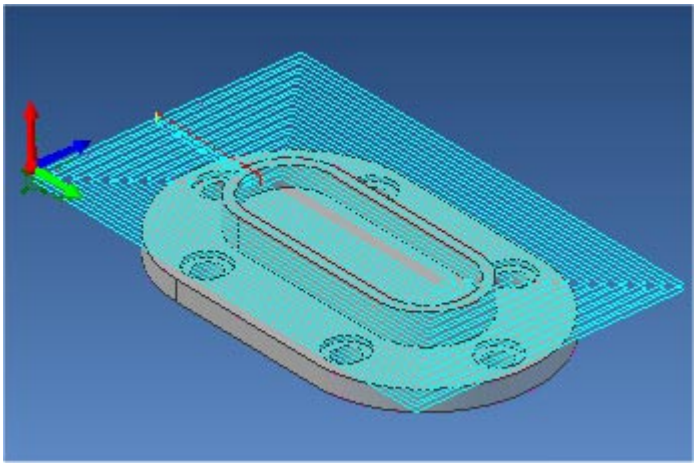
    a. Pick Top = **0** (As the selected region at Z = -0.125”, we would need to start the first cut from Z =0).
    b. Total Cut Depth = **0.125**, Rough Depth = **0.125**, and Rough Depth/Cut = **0.0625**.




13. Switch to the Entry/Exit tab and set the Entry and Exit parameters to none.

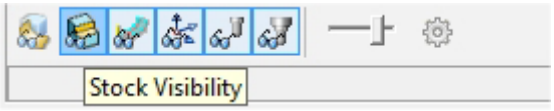
14. Click **Generate**. The 2½ Axis Facing toolpath is now generated, and the Operation is listed under the Alibre CAM MOps browser.



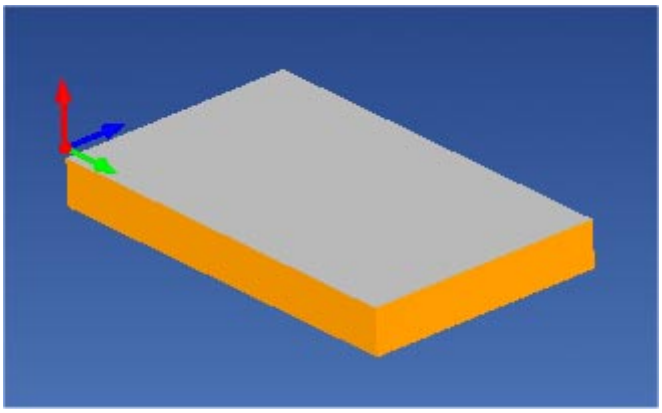
## Simulate Toolpath

Make sure to turn on Stock Visibility under the Simulate tab.




1. Switch to Simulate tab, Select the 2 ½ Axis Facing Operation and click  to launch the Alibre CAM Stock Simulation window.

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

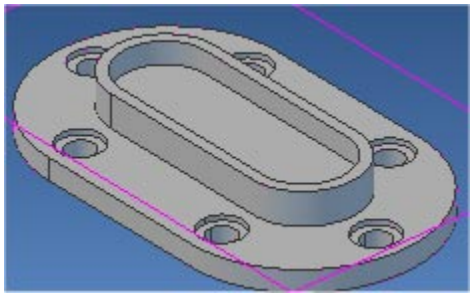


Note: You can pause/stop, step, and skip the simulation using the controls available on the Simulation window. To control the simulation speed, select Simulation Settings from the Stock Simulation window.

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

## 2 ½ Axis Pocketing

We will now use 2½ axis Pocketing operation to machine the area inside the boss.



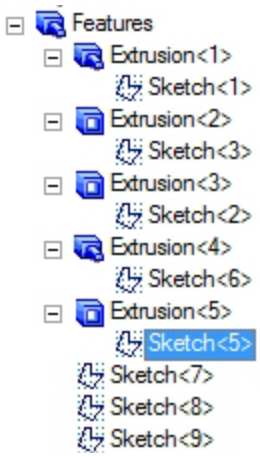
## Creating the Pocketing Operation #1

1. From the Create Operations tab, select 2½ axis Milling and Pocketing.


This brings up the 2 ½ Axis Pocketing Operations dialog. We will go over the steps for creating the pocketing operation.

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

3. Switch to Design Explorer and select **Sketch5**.

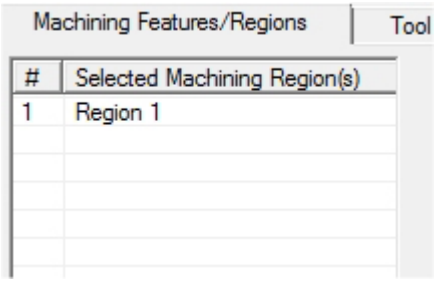


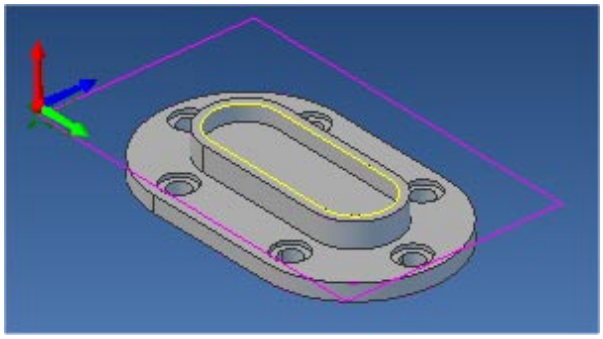

Note: You can also select the sketch from Alibre Design View. When select containment region is selected, Alibre CAM turns on Sketch selection filter to allow the user to select sketches. Hold the Shift key down to select multiple sketches.

4. Click 

to complete the selection.

The 2 ½ Axis Pocketing operation dialog comes back up displaying the selected. The selected regions are also highlighted on the part.



5. Switch to the Tools tab inside the 2½ Axis Pocketing operation and select FlatMill-0.25.

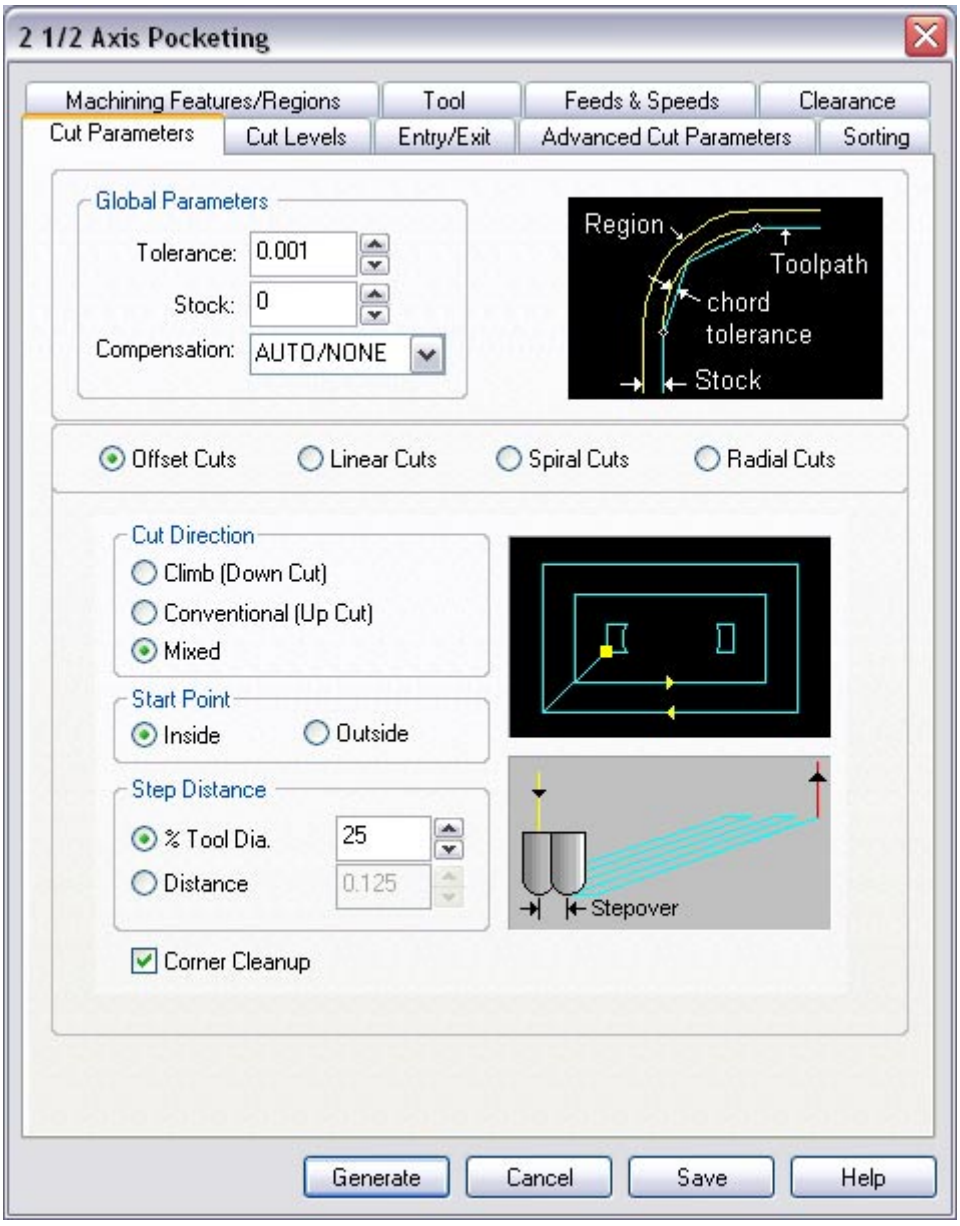
6. Click on the Feeds and Speeds tab and select Load From Tool. Alibre CAM will now get the feeds and speeds information that was set when the tool was defined.

7. 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 to **0**, Cut Pattern to **Offset Cuts**, and Step over distance to **25** (% Tool Diameter).

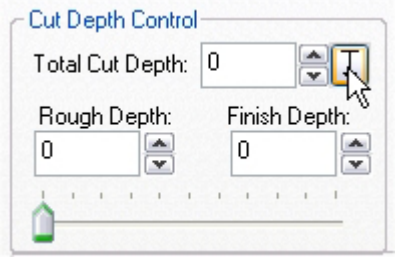



3. Switch to the Cut Levels Tab.

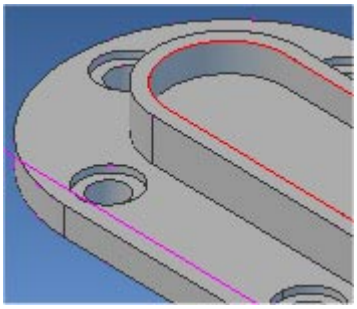
4. Use the Following Settings.
    a. Location of Cut Geometry at **Top**.

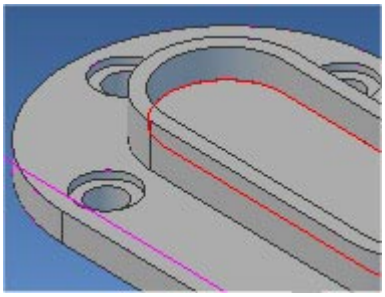
    We will determine the Total Cut Depth from the 3D model by snapping at 2 points.

    b. Select the Depth measuring tool located to the right of Total Cut Depth. This will minimize the Pocketing Operation parameters dialog.



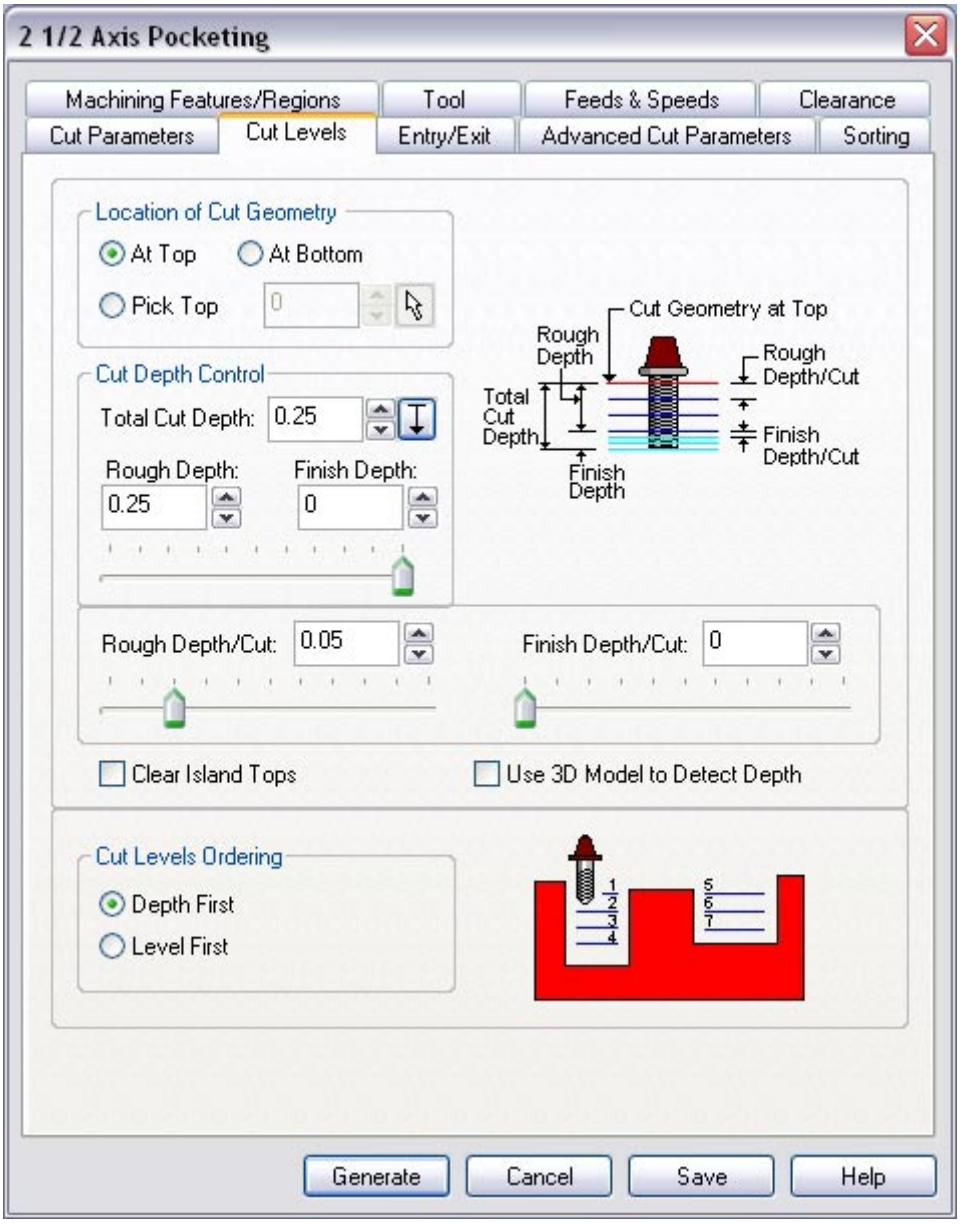

    c. Pick the top edge of the boss as the start point of axis vector and the bottom edge of the boss as the end point as shown below.







    d. The pocketing operation dialog shows up and determines Total Cut Depth = **0.25**.
    e. Set the Rough Depth = **0.25** and Rough Depth/Cut = **0.05**.

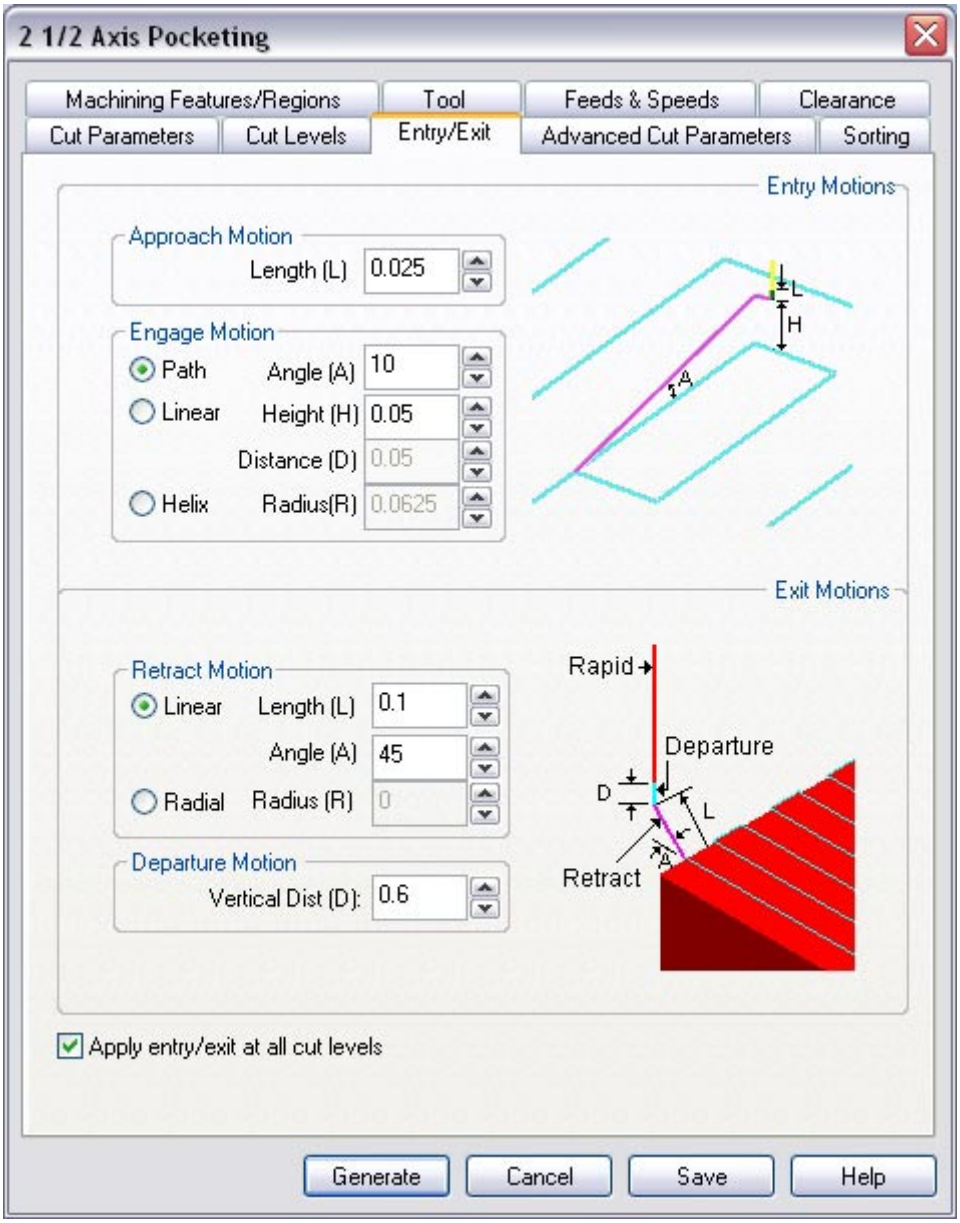



This would machine the pocket in steps of 0.05 resulting in 5 cut levels.

*Note: You can also specify the Total Cut Depth by entering the depth values under Total Cut Depth.*

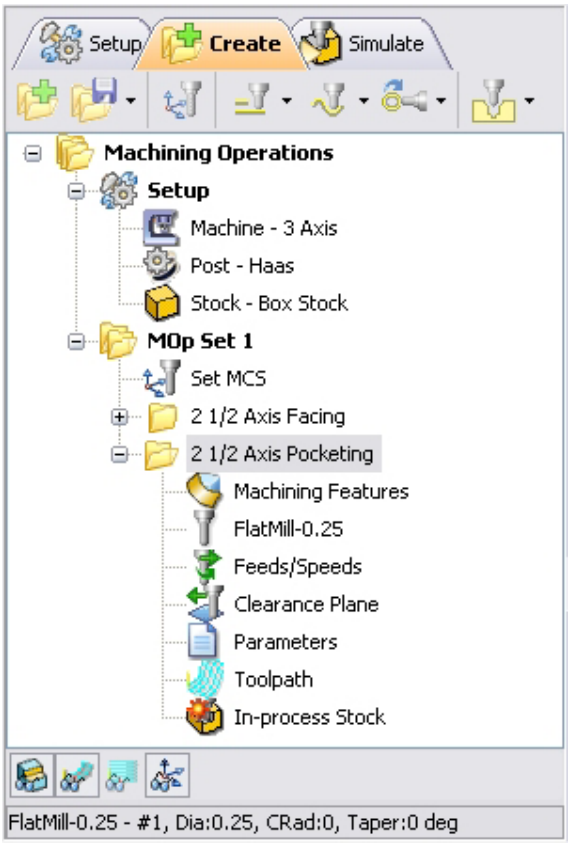
5. Switch to the **Entry/Exit** tab.

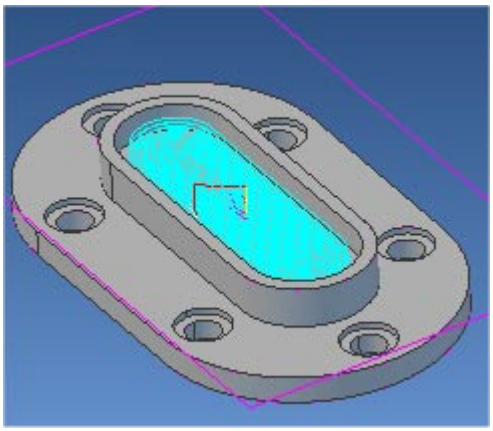
6. Use the following settings for Entry/Exit. Make sure to check Apply Entry/Exit at all cut levels.




7. Click **Generate**. The 2½ Axis Pocketing toolpath is now generated and the Operation is listed under the 2 ½ Axis Facing Operation in the Alibre CAM MOps browser.

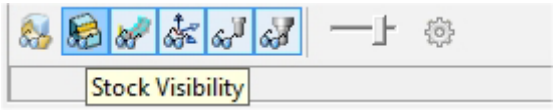
*Note: You can rearrange the operations in the MOps browser by selecting the operation and dragging and dropping.*



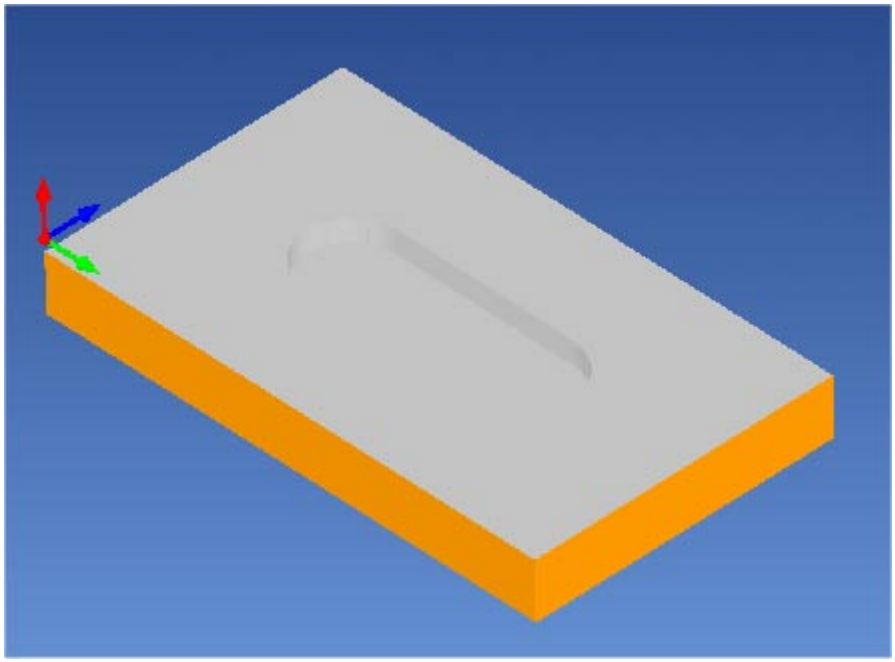
## Simulate Toolpath

Make sure to turn on Stock Visibility under the Simulate tab.




1. Switch to Simulate tab, Select the 2 ½ Axis Pocketing Operation and click  to launch the Alibre CAM Stock Simulation window.

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



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

## Creating the Pocketing Operation #2

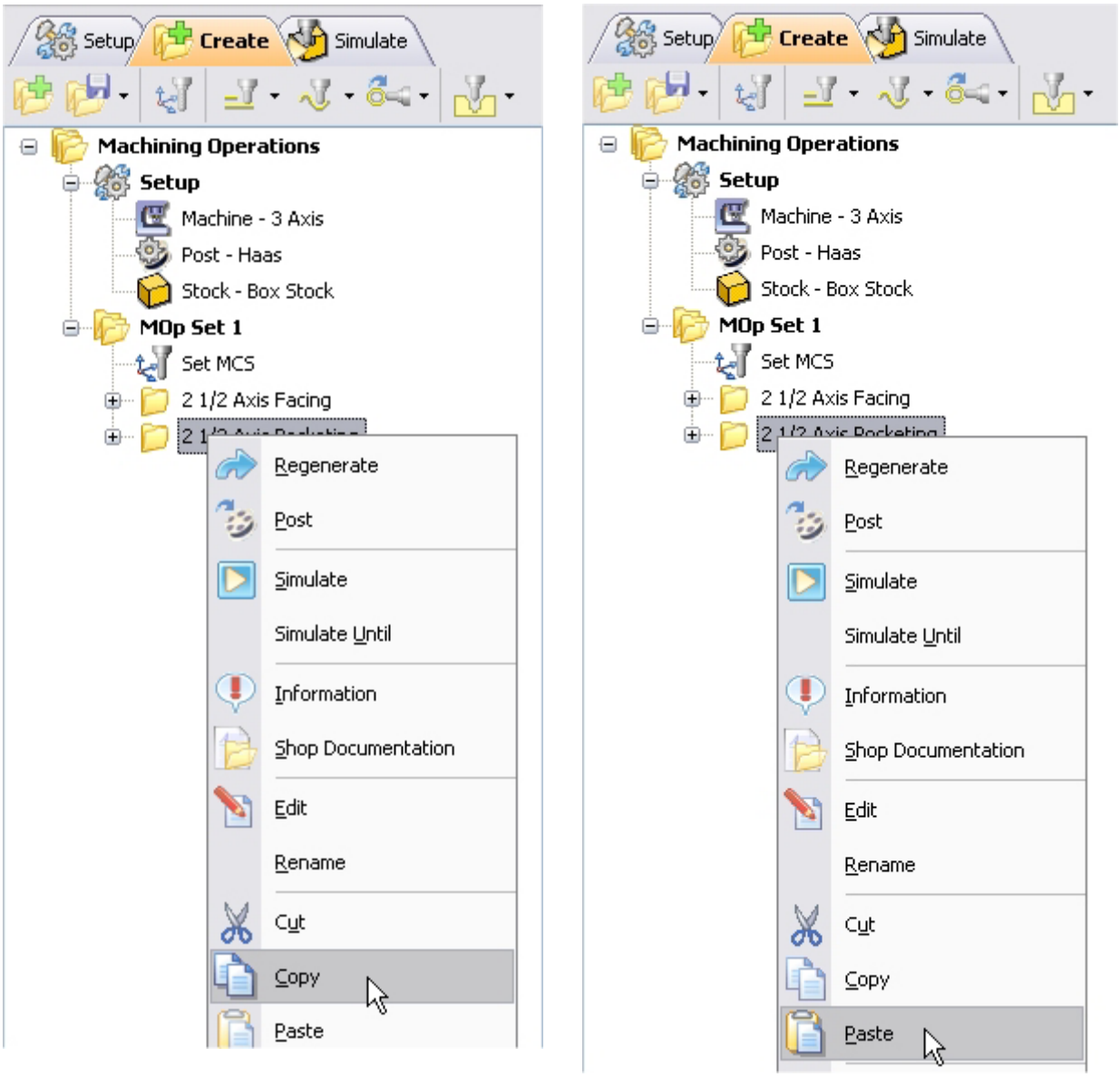
We will now create a 2nd pocketing operation for machining the region around the boss.

1. Switch to the Create Operations tab.

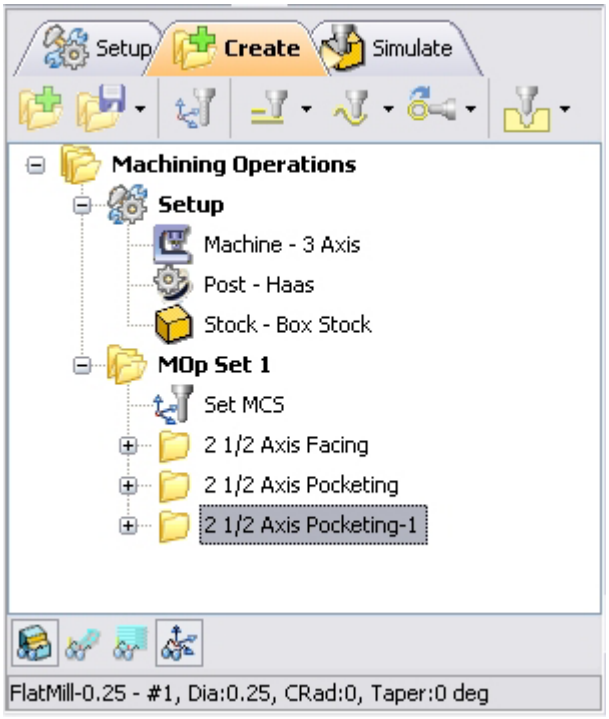
## Copying a MOp

2. Select the 2 ½ axis Pocketing Operation created from the previous step, right mouse click, and select **Copy**.

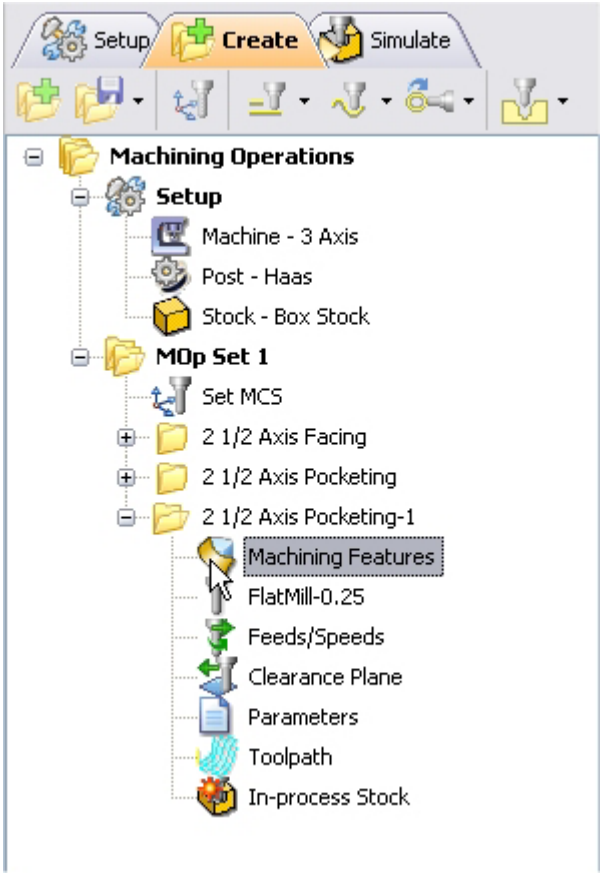
3. Right click and select **Paste**.




4. This would create a copy of the 2 ½ axis Pocketing Operation listed below the first pocketing operation as show below.

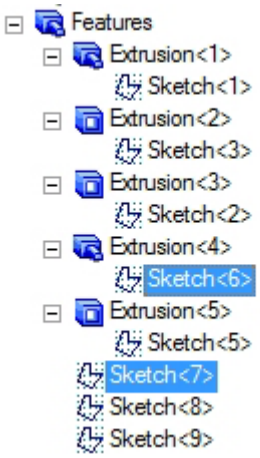


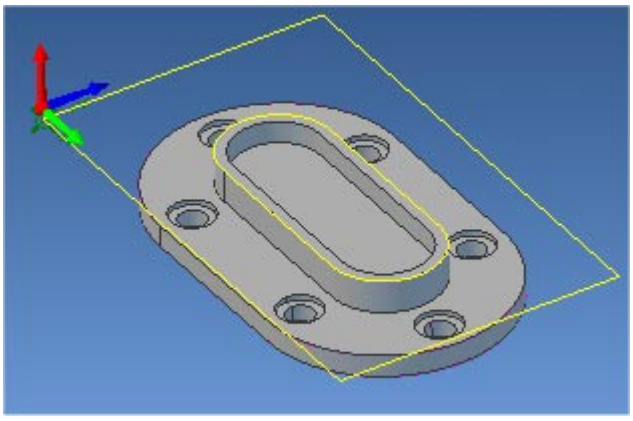

5. Expand the 2 ½ Axis Pocketing-1 folder and double click on Machining Features.



6. Click Remove All under Machining Features and click **Select Containment Regions**.

7. Select **Sketch7** and **Sketch6** from the Design Explorer by holding the Ctrl key down.



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.

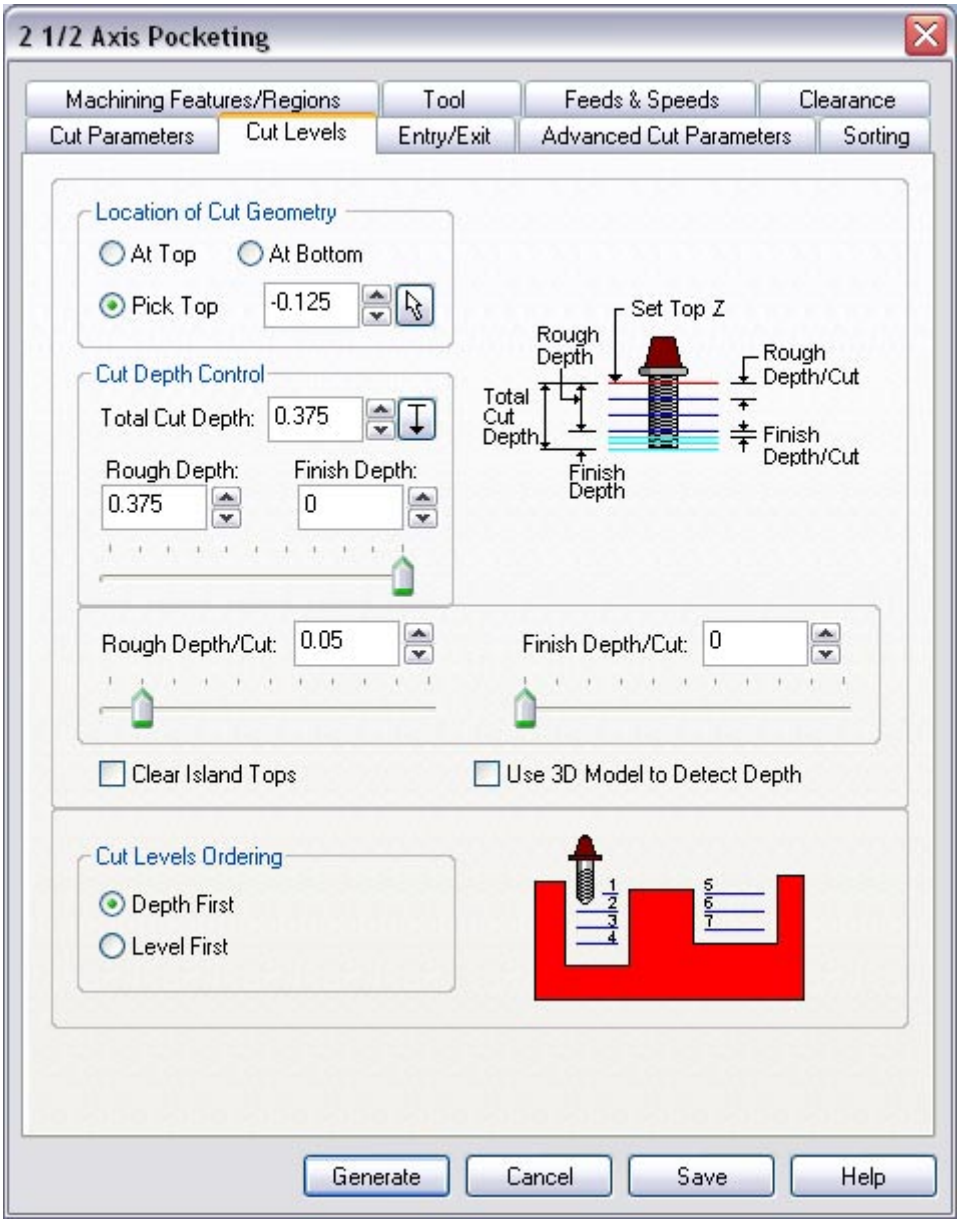
8. Click 

to complete the selection.

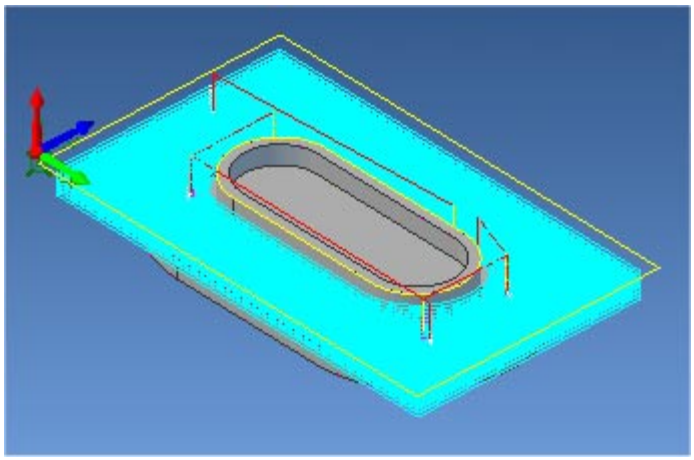
9. Regions are now listed under Machining Features/Regions.

10. Switch to the **Cut Levels Tab**.

11. Use the Following Settings.
    a. Location of Cut Geometry – Select Pick at Top = **– 0.125**
    b. Total Cut Depth – Set this to **0.375**
    c. Set the Rough Depth = **0.375** and Rough Depth /Cut = **0.05**
    d. Switch to the Entry/Exit tab and set the Retract Motion to Linear, Length = **0.1** and Angle = **0**
    e. Click **Generate**.

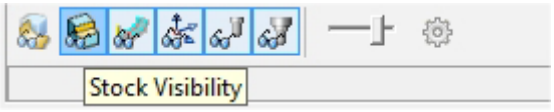



12. The pocketing toolpath is now created and displayed in the MOps browser.



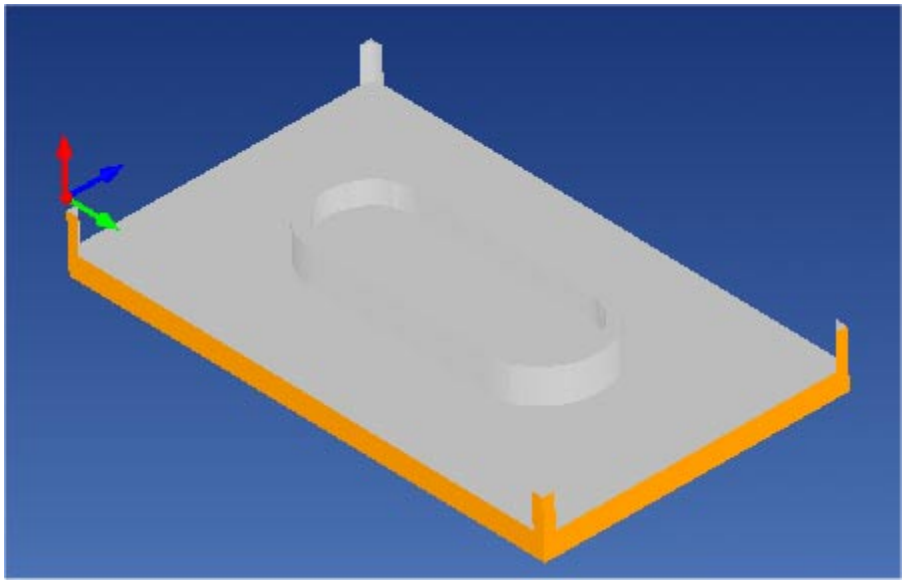
## Simulate Toolpath

Make sure to turn on Stock Visibility under the Simulate tab.




1. Switch to Simulate tab, Select the 2 ½ Axis Pocketing-1 Operation and click  to launch the Alibre CAM Stock Simulation window.

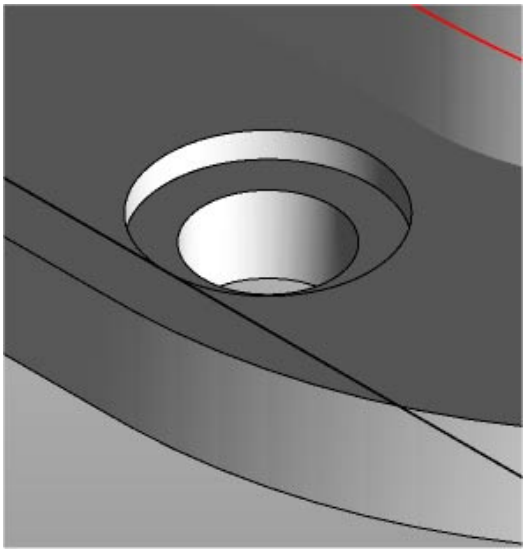
2. Click Simulate from the Stock Simulation window to run simulation.



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

## Hole Pocketing Operation

In order to machine the 6 holes, we will now use 2 ½ axis hole pocketing operation.

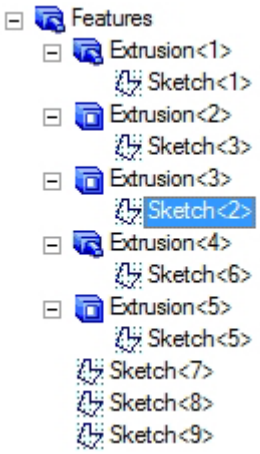


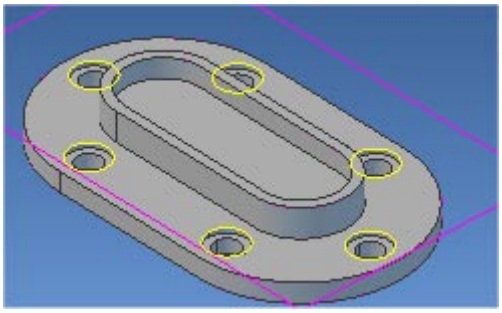
## Creating the Hole Pocketing Operation #1

1. Select 2 ½ Axis Milling and Hole Pocketing.

2. Go to the Machining Features/ Regions tab and click Select Containment Regions.

3. Switch to Design Explorer and select **Sketch2** located under Extrude4. This selects the 6 holes.



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.

4. Click 

to complete the selection.

5. Switch to the Tools tab inside the Hole Pocketing operation and select FlatMill-0.25.

6. 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.

7. 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. Use the following Settings

    a. Tolerance to **0.001,**

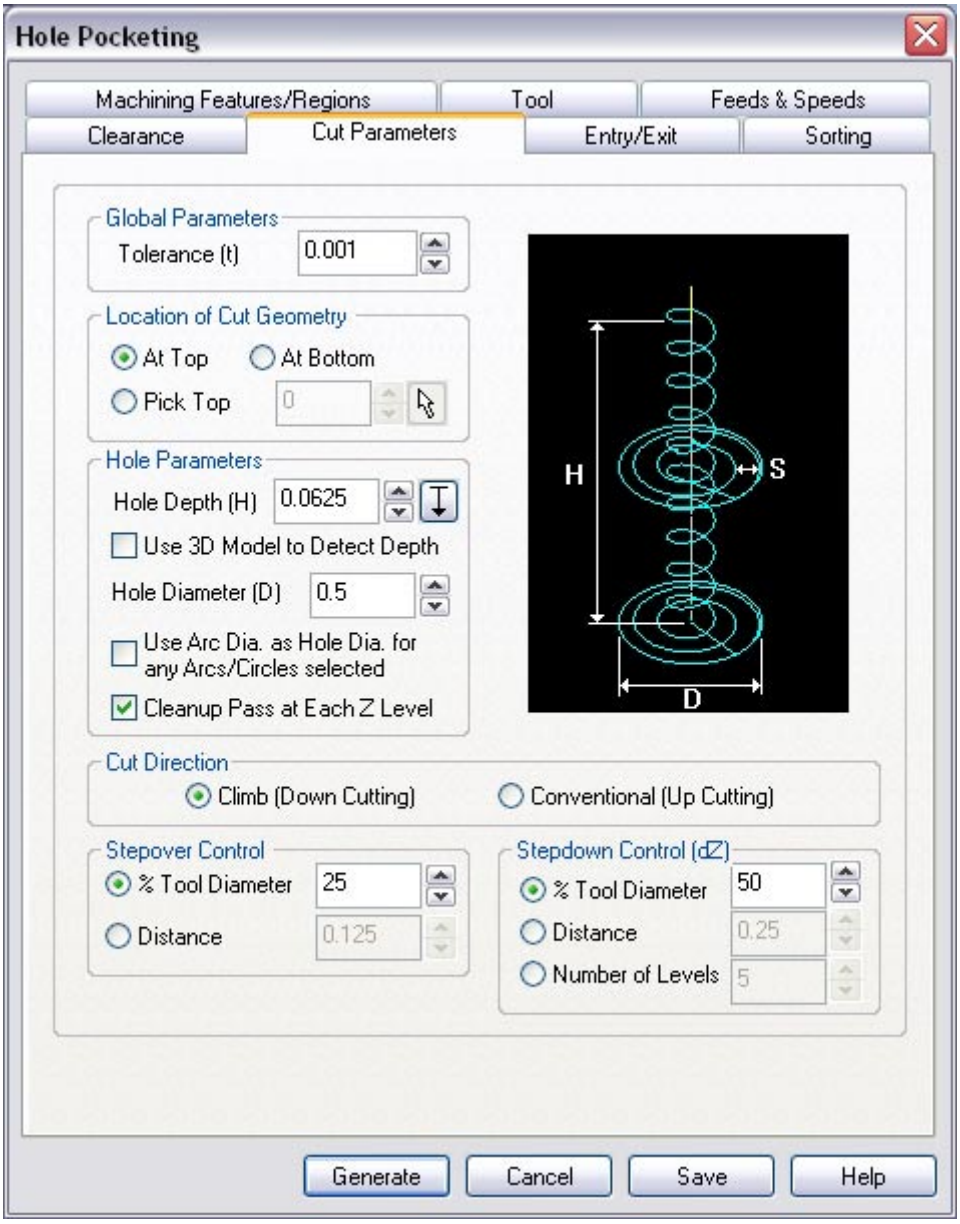
    b. Hole Depth (H) =**0.0625**, Uncheck Use 3D model to Detect Depth,

    c. Hole Diameter (D) = **0.5**, Check Cleanup pass at each Z level

    d. Step over distance = **25** (% Tool Diameter),

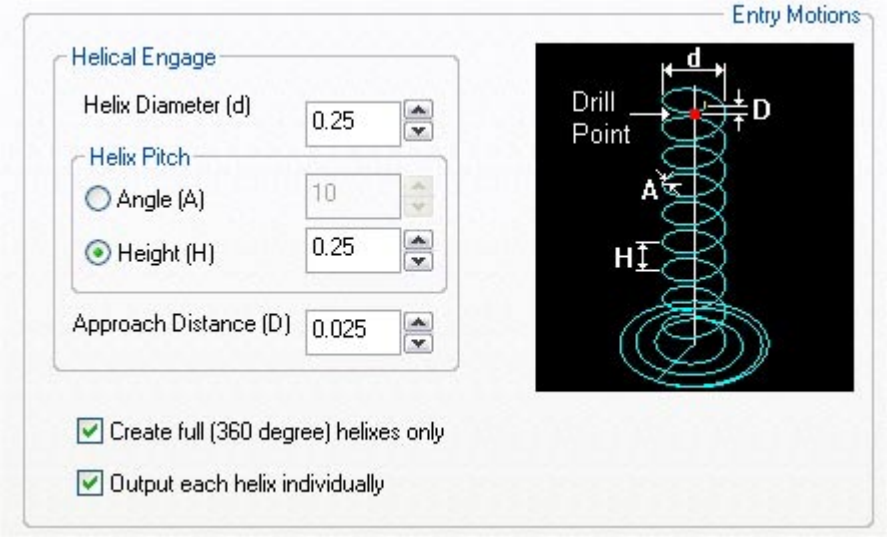
    e. Step Down Control (dZ) = **50** (% Tool Diameter),

    f. Cut Direction = Climb (Down Cutting).

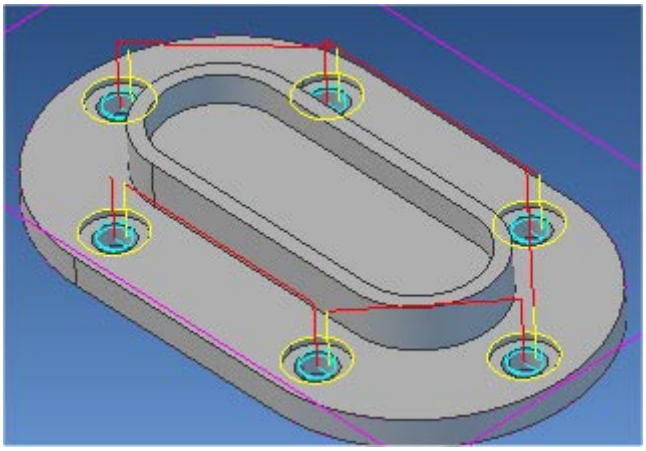


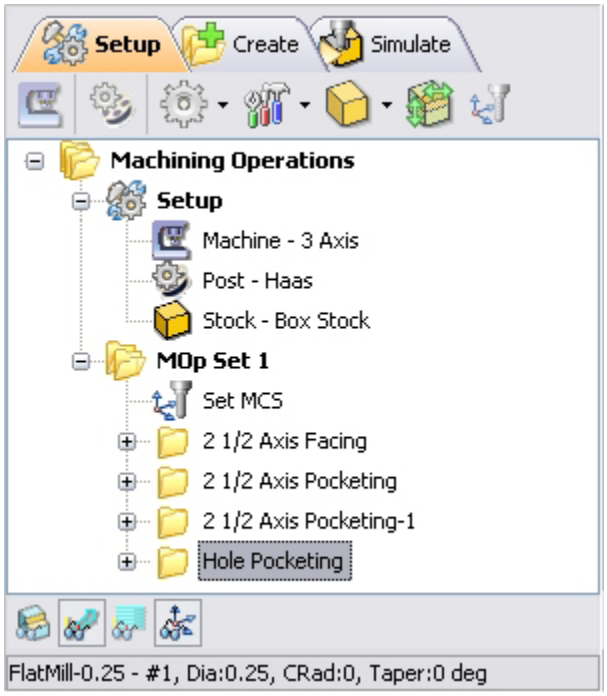

3. Switch to the Entry Exit Tab and set the Helix Diameter = **0.25**.

4. Check Create full (360 degree) helixes only and output each helix individually.



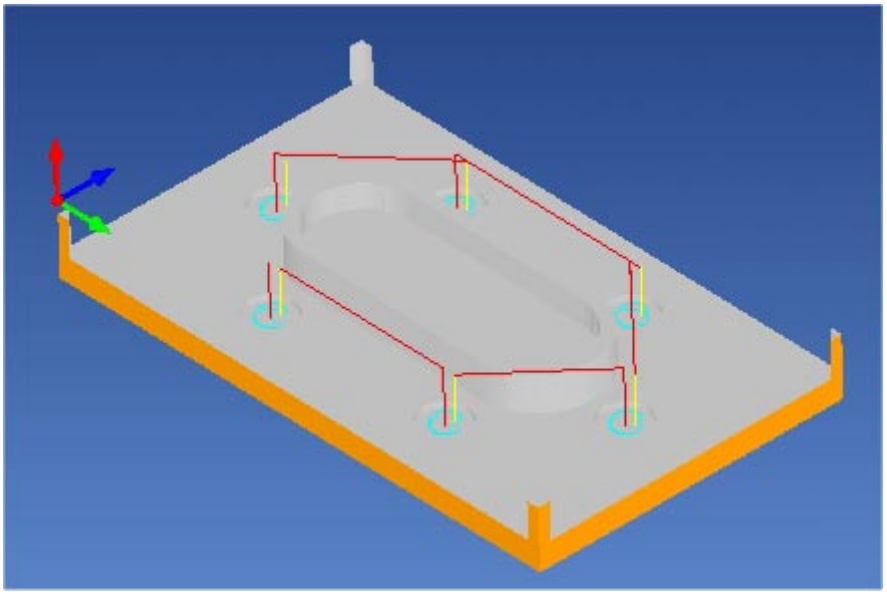

5. Click **Generate**. The Hole Pocketing operation is now created and is listed under the MOps browser.






4. Switch to the Simulate tab, select Hole Pocketing, and click  to launch the Alibre CAM Stock Simulation window.

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



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

## Creating the Hole Pocketing Operation #2

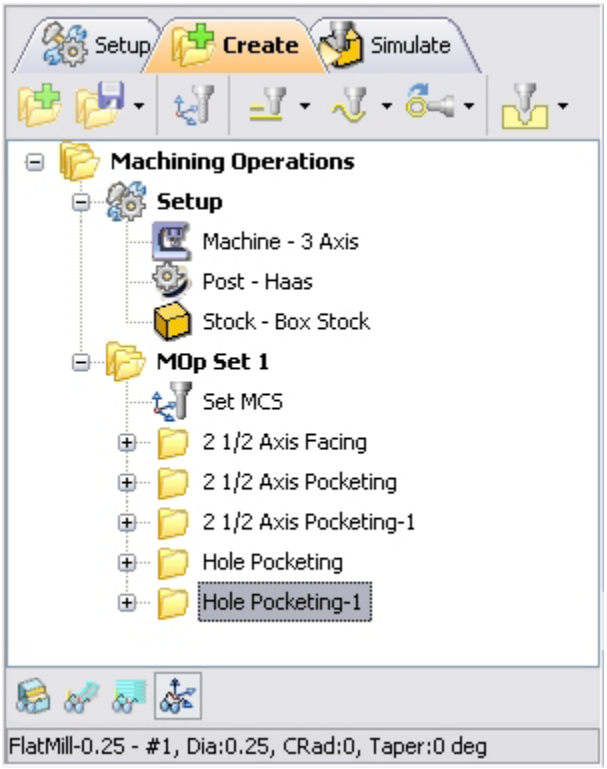
1. Switch to the Create Operations tab.

## Creating the Hole Pocketing Operation

2. Select the Hole Pocketing Operation created from the previous step, right mouse click, and select **Copy**.

3. Right click and select **Paste**.

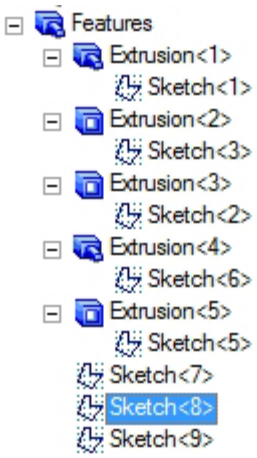
4. This would create a copy of the Hole Pocketing Operation listed below the first Hole Pocketing operation as show below.

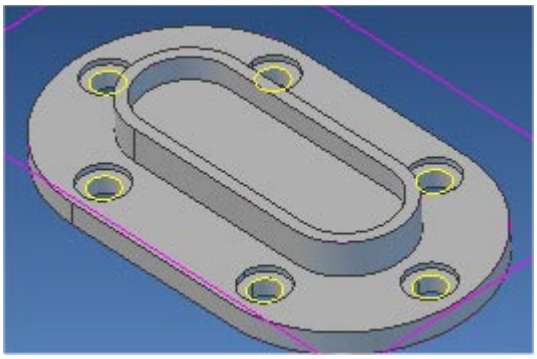



5. Expand the Hole Pocketing-1 folder and double click on Machining Features.

6. Click Remove All under Machining Features and click **Select Containment Regions**.

7. Switch to Design Explorer and select **Sketch8**.



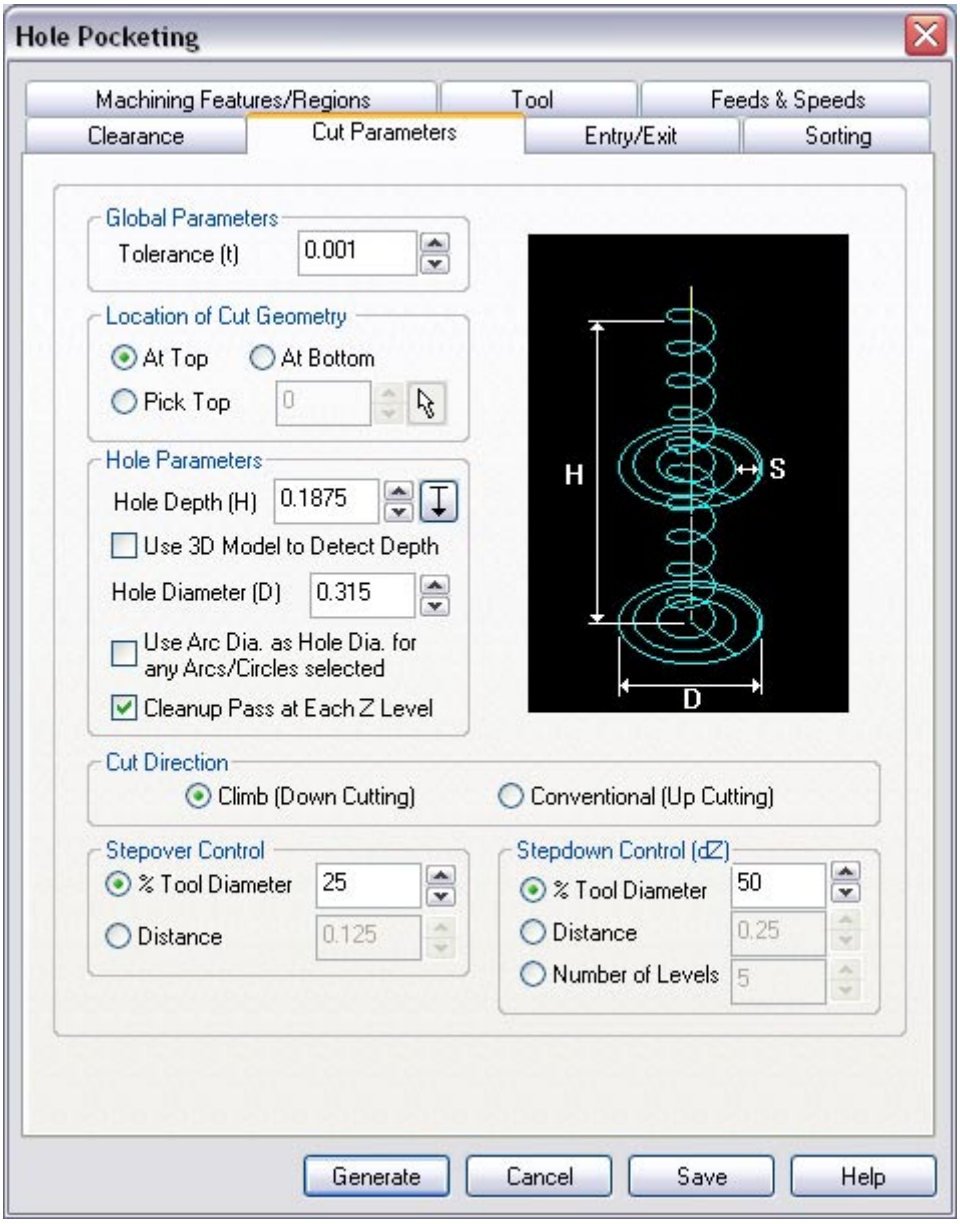



Note: You can also select the sketch from Alibre Design View. When select containment region is selected, Alibre CAM turns on Sketch selection filter to allow the user to select sketches. Hold the Shift key down to select multiple sketches.

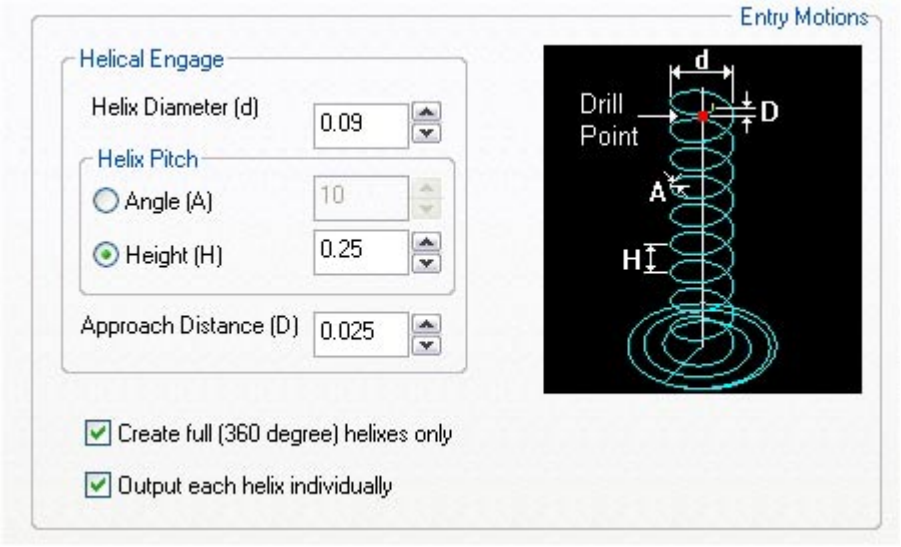
8. Click 

to complete the selection.

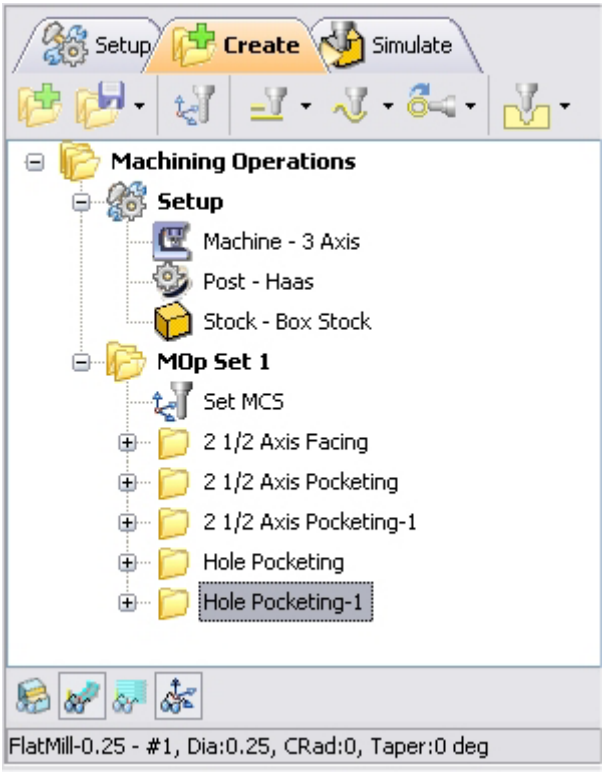
9. Switch to Cut Parameters tab and set the Hole Depth = **0.1875**, Hole Diameter = **0.315.**

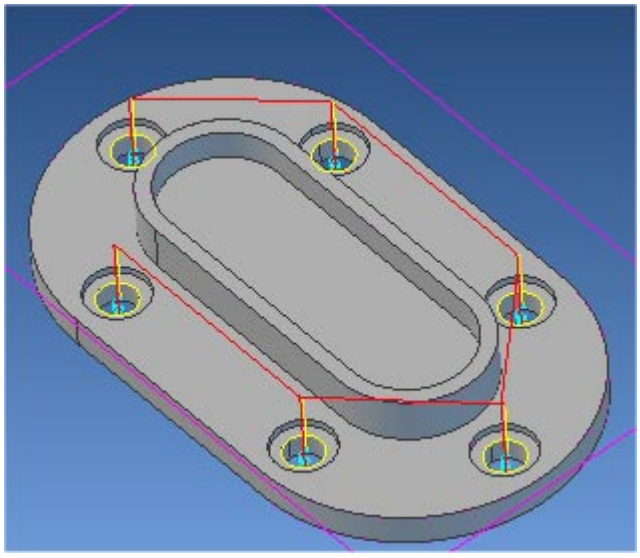



10. Switch to the Entry/Exit tab and set the Helix Diameter to **0.09**.



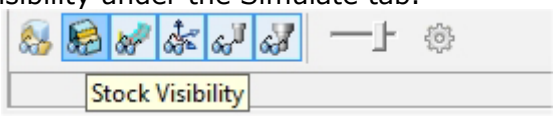

11. Click **Generate**. The Hole Pocketing Operation for the inner holes is now created.



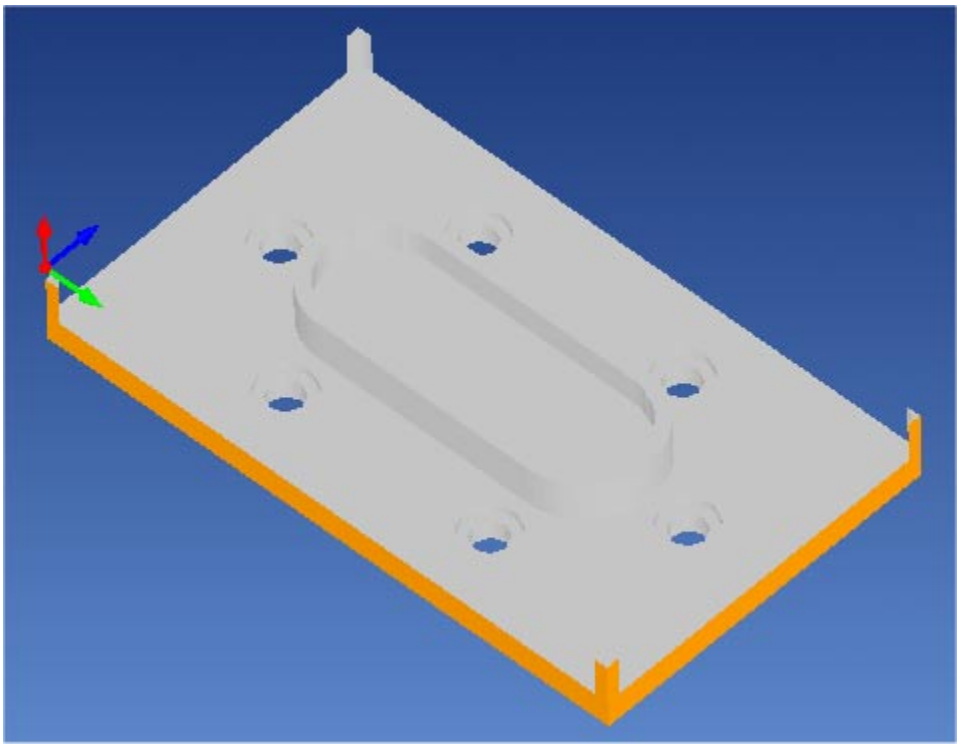



12. Switch to the Simulate tab, select Hole Pocketing-1, and click  to launch the Alibre CAM Stock Simulation window.

Make sure to turn on Stock Visibility under the Simulate tab.



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



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

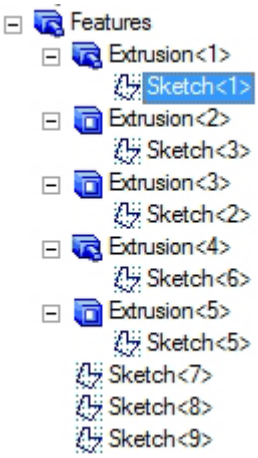
## Creating the Outer Profile Toolpath

We will not create a 2 ½ Axis Profiling toolpath to cut out the shaft base. We will use the same Tool (FlatMill-0.25) and settings for Feeds /Speeds and Clearance plane for the profiling toolpath.

1. Switch to the Create Operations tab and select 2.5 Axis Milling and choose Profiling.

2. Go to the **Machining Features/ Regions** tab, click Remove All under Machining Features, and click **Select Containment Regions**.

3. Switch to Design Explorer and select **Sketch1**.

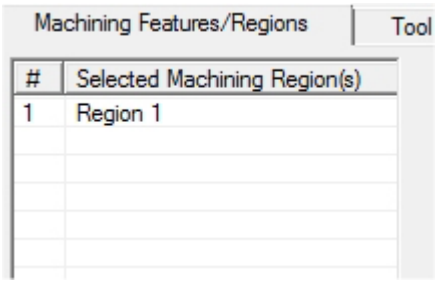


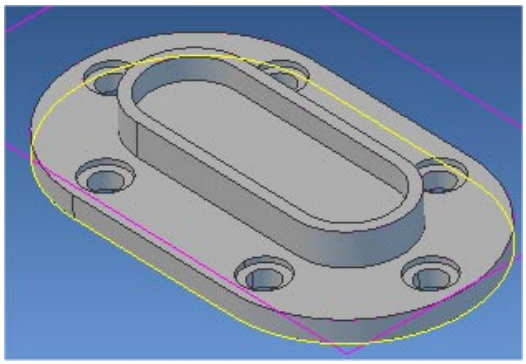

Note: You can also select the sketch from Alibre Design View. When select containment region is selected, Alibre CAM turns on Sketch selection filter to allow the user to select sketches. Hold the Shift key down to select multiple sketches.

4. Click 

to complete the selection.

5. The selected region is now displayed under Machining Features/Regions.



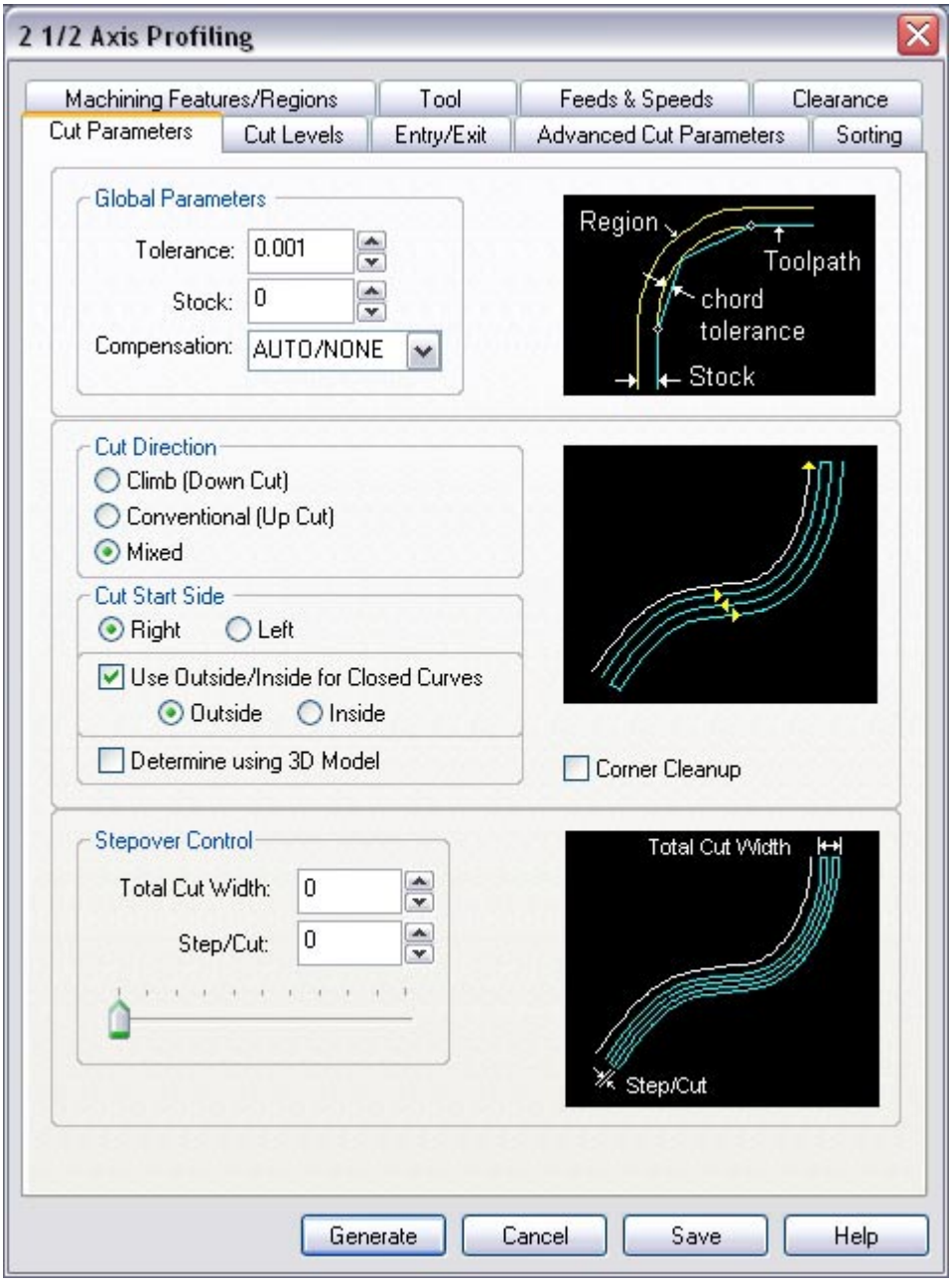
6. Switch to the Tools tab inside 2 ½ Axis Profiling operation and Select the FlatMill-0.25.

7. 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.

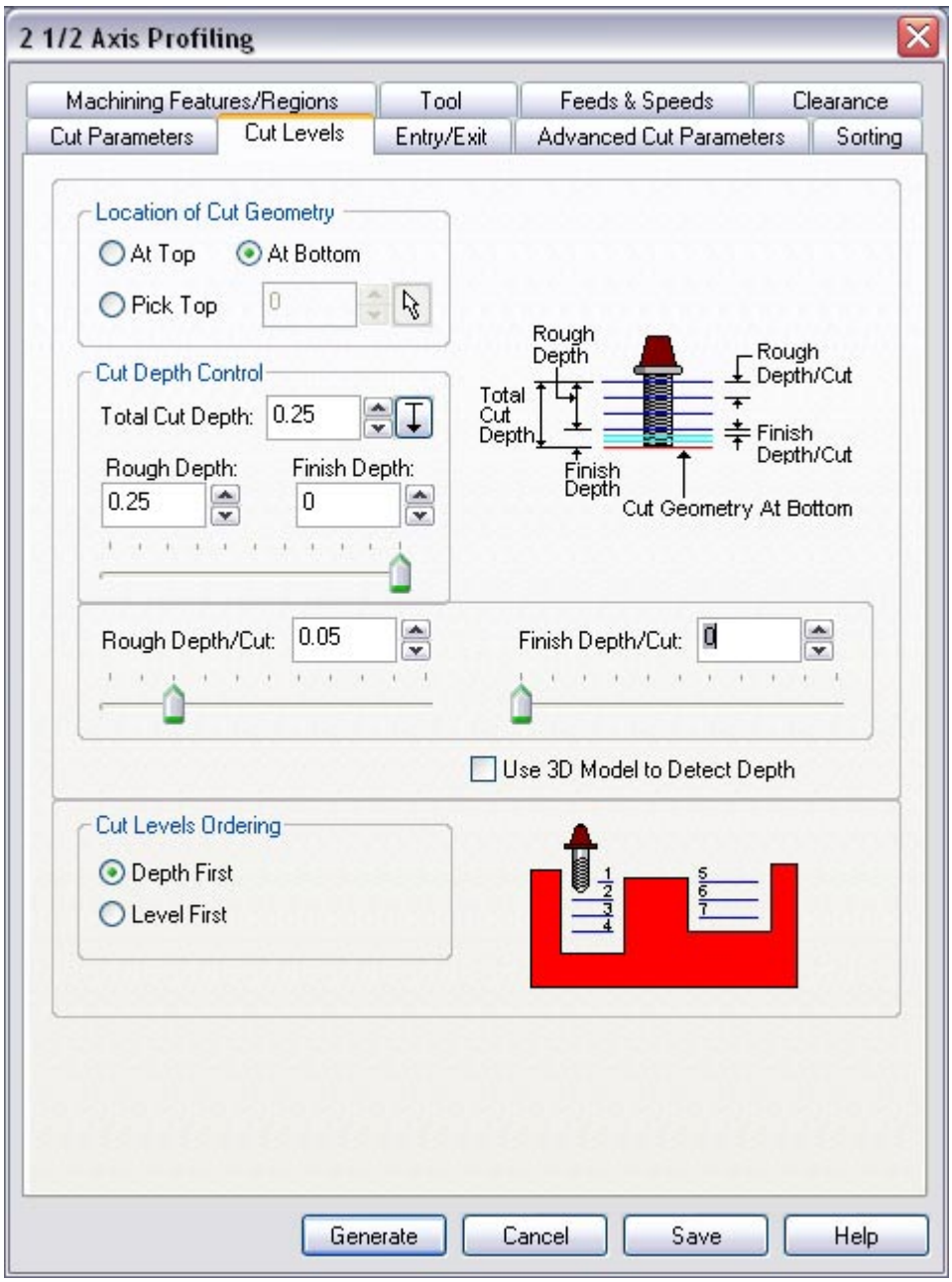
8. Switch to Clearance Tab. Set the Clearance Plane Definition to Automatic and Cut Transfer Method to Clearance Plane.

## Specifying Cut Parameters

1. Switch to Cut Parameters tab and use the following Settings
    a. Tolerance = **0.001**,
    b. Stock = **0**,
    c. Cut Start Side- Check Use Outside/Inside for closed curves and pick **Outside**.



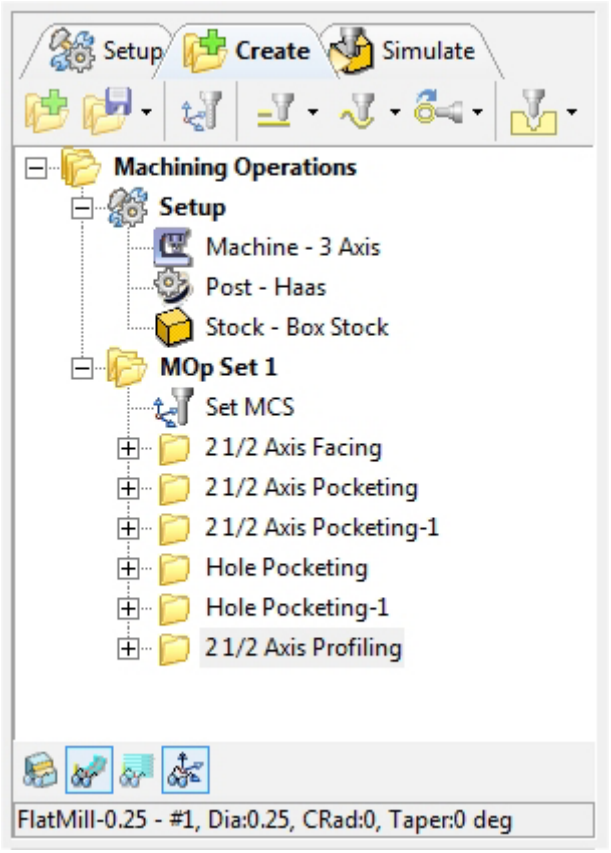
2. Select the Cut Levels Tab and specify Location of Cut Geometry At Bottom, Total Cut Depth = **0.25**, Rough Depth/Cut = **0.05**.

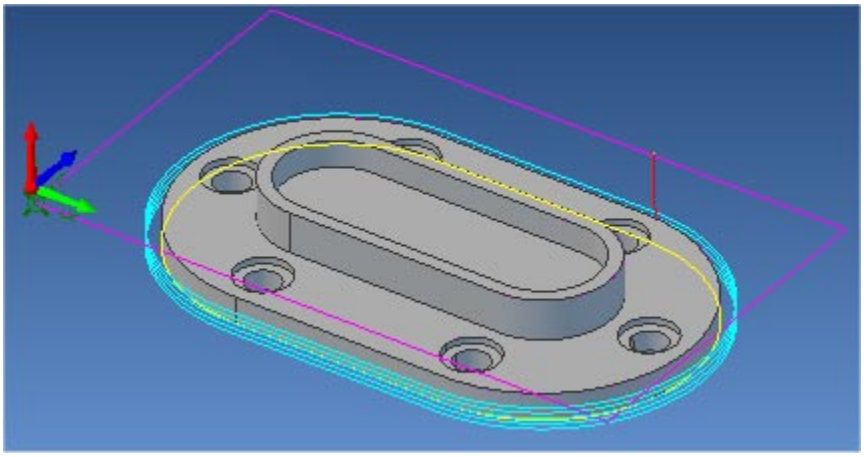



3. Switch to Entry/Exit Tab and Set the Entry and Exit Type to None.

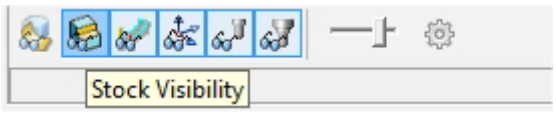
4. Click **Generate**. The 2½ Axis Profile toolpath is now generated and the Operation is listed under the Alibre CAM-MOps browser.

Toolpath display can be turned on/off by selecting Toolpath Visibility under the MOps browser.



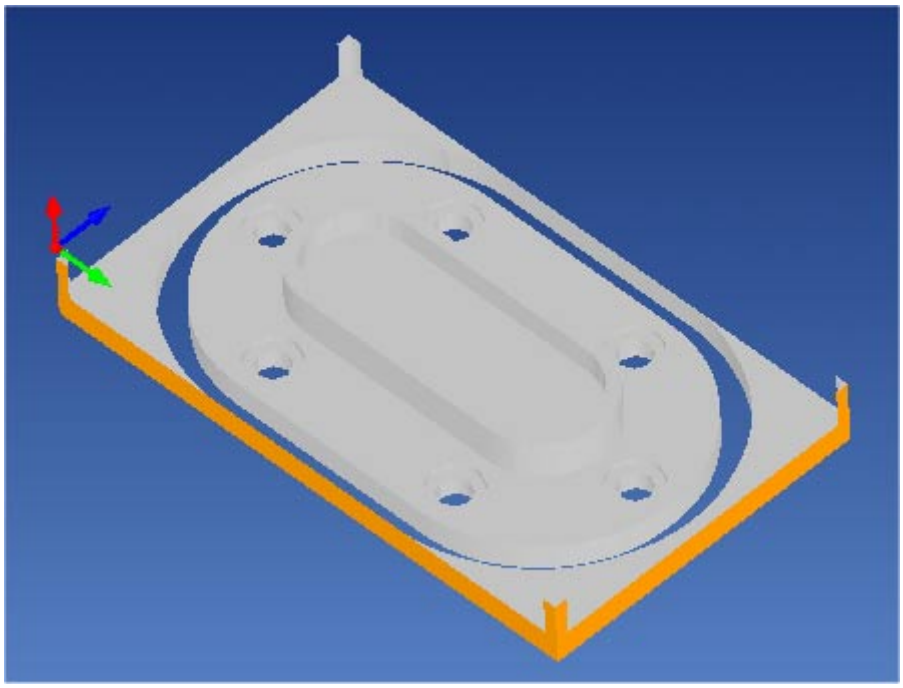
Make sure to turn on Stock Visibility under the Simulate tab.




5. Switch to the Simulate tab, Select the 2 ½ Axis Profiling Operation and click  to launch the Alibre CAM Stock Simulation window.

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

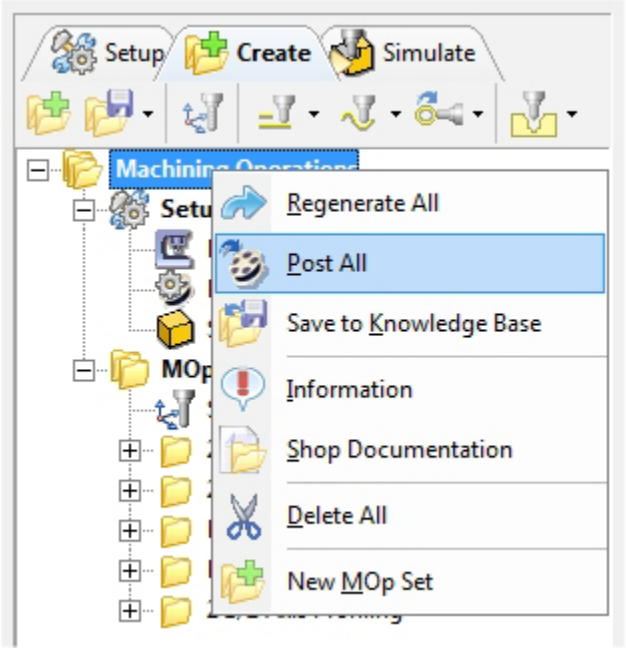
7. Once the simulation is complete, you can close the Stock Simulation window and return to the Alibre CAM browser.



## Post Processing

Once the toolpath has been generated, it can be post-processed to a specific machine controller.

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




2. Specify the File Name as **Shaftbase.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.*