Tutorial 4: Simple V-Carving

Introduction

This tutorial will illustrate machining a Sign using 2-1/2 axis-engraving operations. We can engrave the sign using 2-D Sketches. This tutorial will introduce the usage of 2-½ axis simple V Carving using V bit.

V carving refers to a cutting strategy employed by sign makers to create sharp corners. V carving is performed using a tapered bit or conical tool (as shown below) usually known in the industry as a V Bit.

The V-bit is made to rise from the cutting depth to the top of the surface at the corners in such a way that the tapered sides of the cutter are always in contact with the corners. When the cutter finally reaches the top surface, only the bottom tip of the tool will be in contact with the corners, thereby creating clean and crisp cuts at the corners.

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 part

- V carving is performed using the 2 ½ axis Machining Operation.
- The part itself will be machined out of a 10.75 inch x 4 inch x ½ inch poplar wood sheet
- The part would be machined using a single V-Groove bit.
- The wooden sheet will be held to the machine table or the spoil sheet on the table using double-sided tape.

Main Programming Steps

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

- Create the Stock geometry
- Set the Machine zero point 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

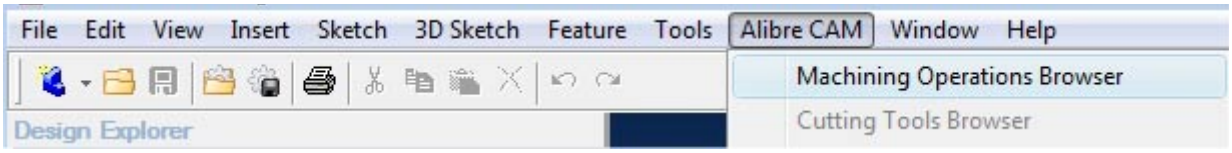
"Part" refers to the geometry that represents the final manufactured product. Typical you would create this in Alibre Design. Use the Alibre Design menu bar or the Standard toolbar buttons to create, load and save part geometry.

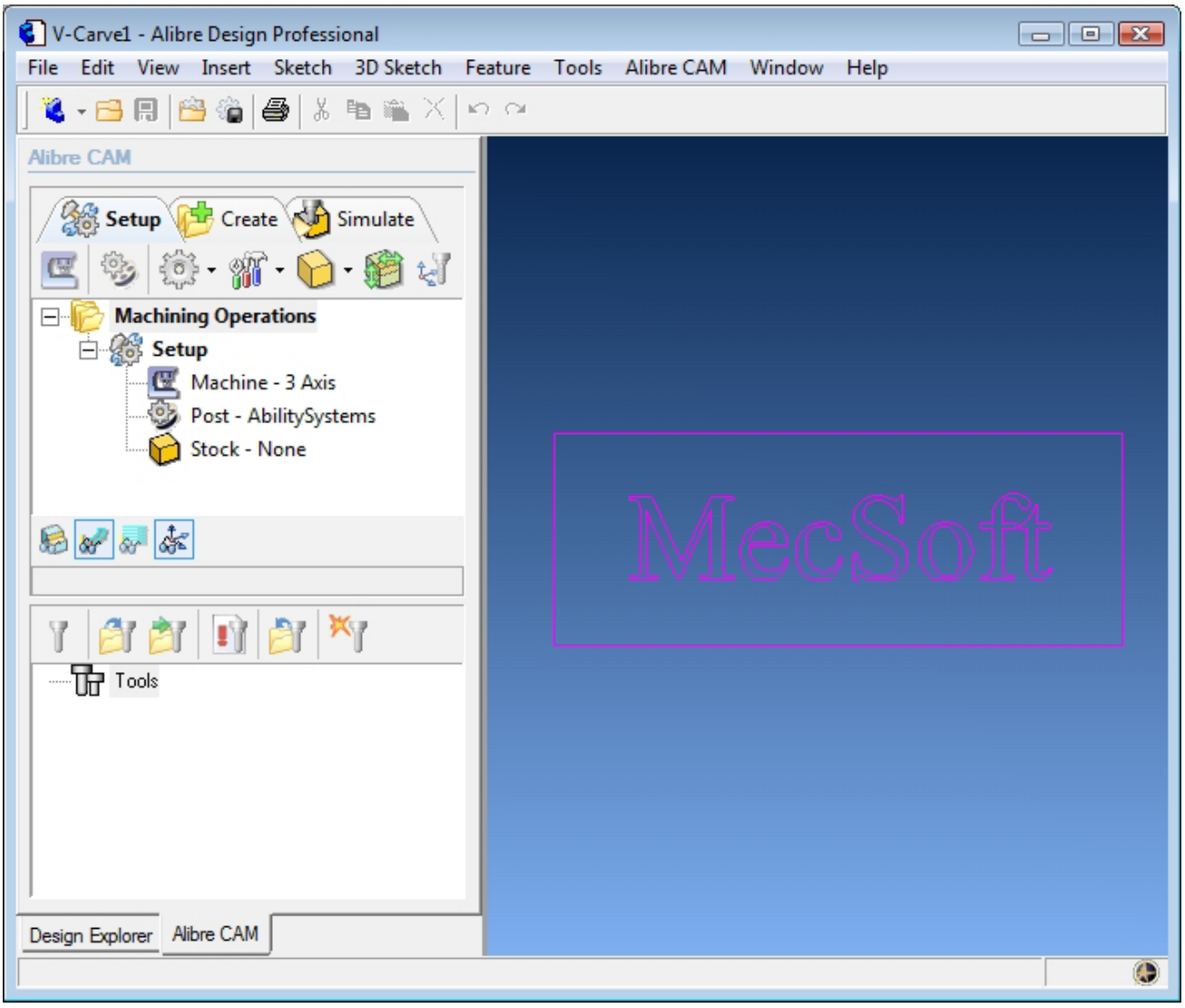
1. Select File / Open Part, or click the Open Part icon from the Alibre Design standard toolbar.
2. From the **Open** dialog box, select the **V-Carve1.AD_PRT** file from the **Tutorials** folder in the Alibre CAM installation folder. (Default location C:\Program Files\MecSoft Corporation\Alibre CAM 2.0\Tutorials)

The imported part appears as shown below

Loading the Alibre CAM Browser

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





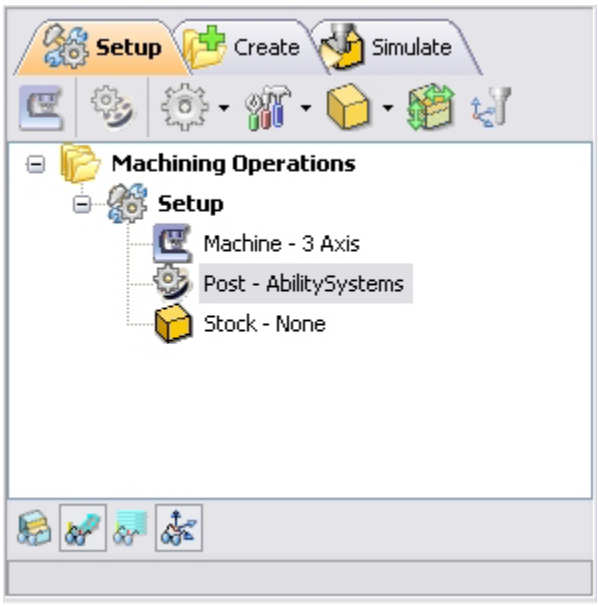
The Alibre CAM browser (MOPs and Cutting Tools) is now loaded and is docked over the Design Explorer. You can toggle between the MOPs browser and the Design Explorer from the tabs at the bottom of the window. It can be hidden by un-checking Browser on the Alibre CAM menu bar. To re-display the hidden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

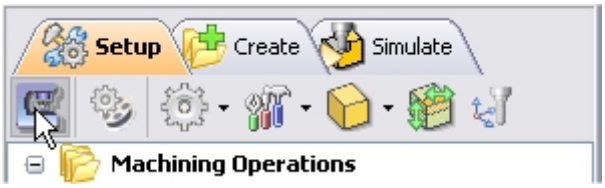
The Setup tab allows the user to specify Machine Setup, Select Post Processor, Stock Geometry, Machine Coordinate System (Machine Zero) & Preferences.

Setup Tab

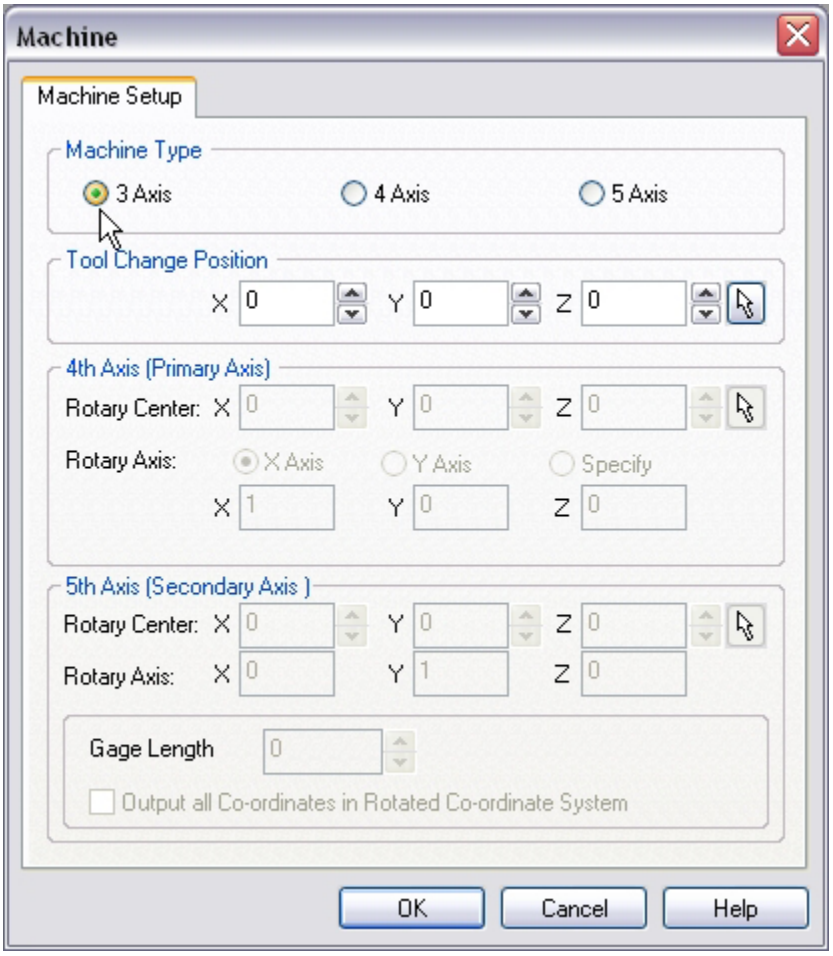
1. Go to the Alibre CAM MOps browser and click on the Setup tab



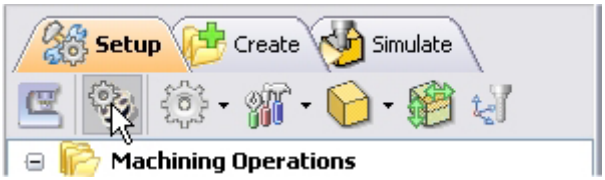
2. Select Machine Setup from the setup tab.



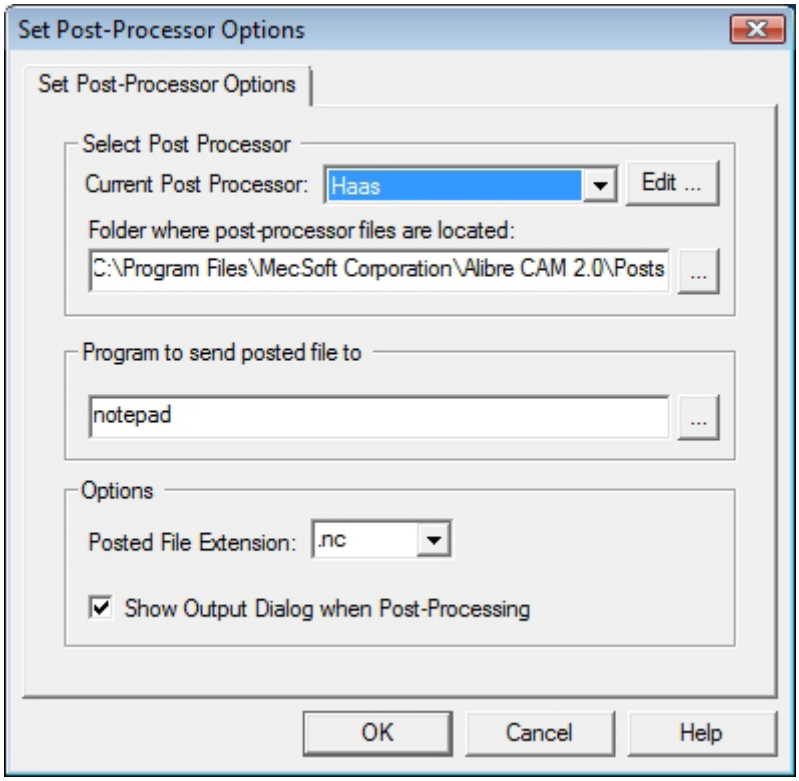
3. Set the Machine type to 3 axis



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



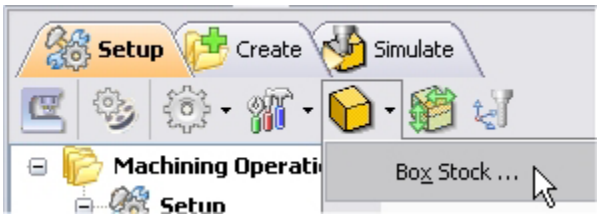
5. Set the current post processor that is on your controller. We will select Haas as the post processor for this exercise. Set the posted file extension type to .nc



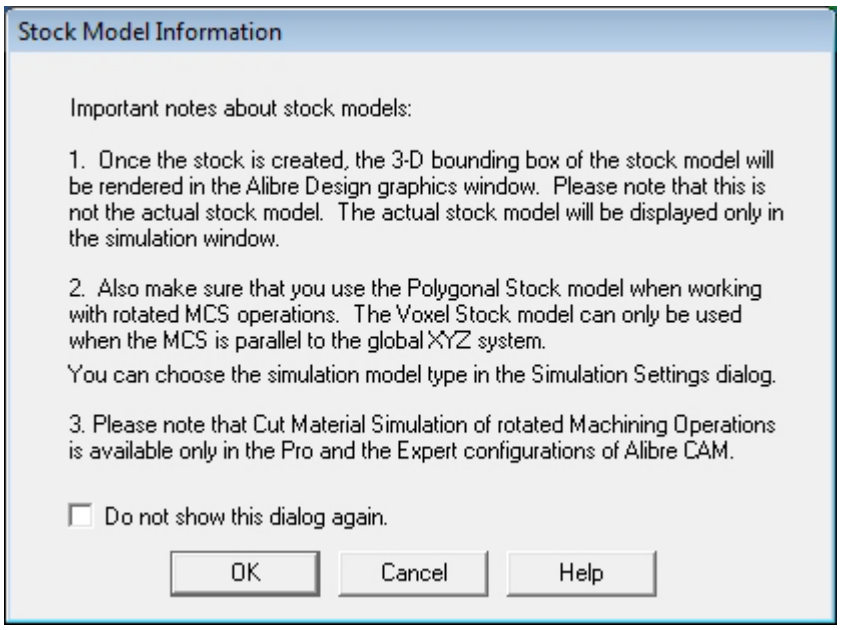
Note: By default post processor files are located under
C:\Program Files\MecSoft Corporation\Alibre CAM 2.0\Posts
The program to send the posted output is set to notepad. This would output the G code to a notepad.

Create Stock Geometry

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



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

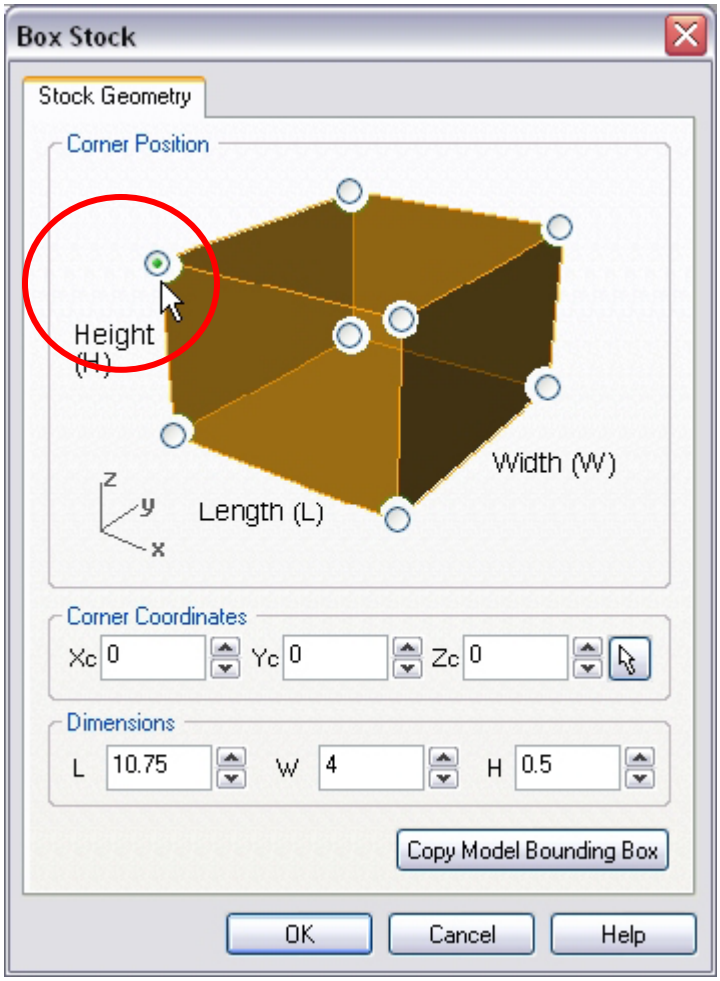


Click OK

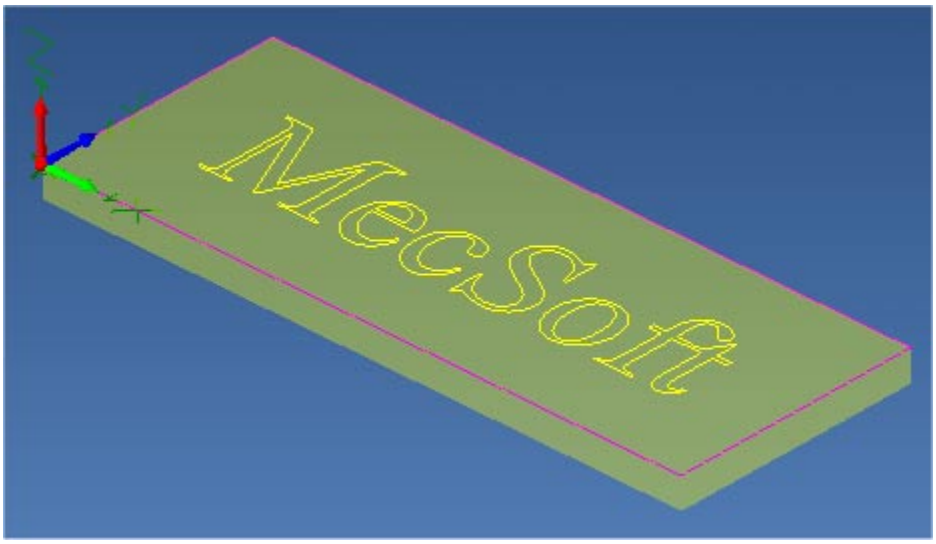
User can turn off this dialog by selecting Do not show this dialog again located on the bottom of the message window.

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

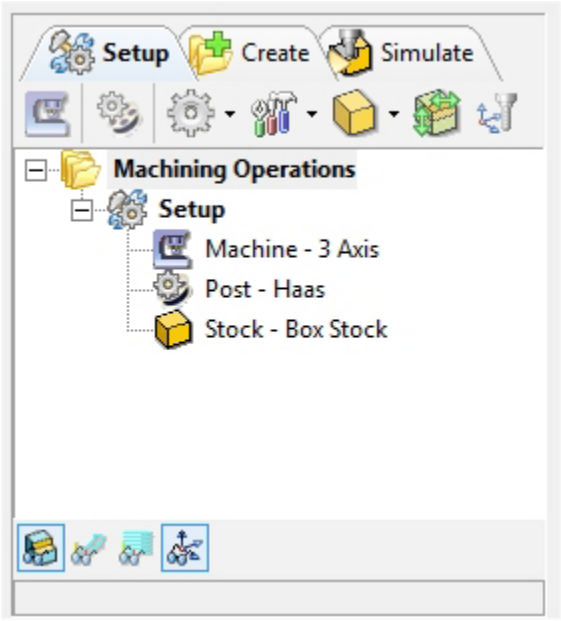
2. This brings up the Box Stock parameters. Set the Length (L) = 10.75, Width W = 4.00 and Height (H) = 0.5. Make sure to set the corner position to Southwest corner Top of Stock as shown below.



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



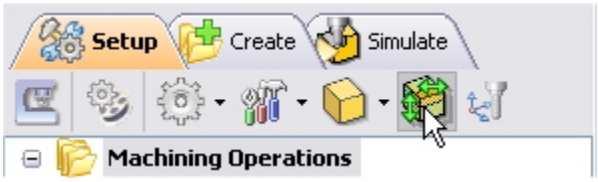
4. The setup tab now displays the following information: Machine Type, Post Processor, and Stock type as show below.



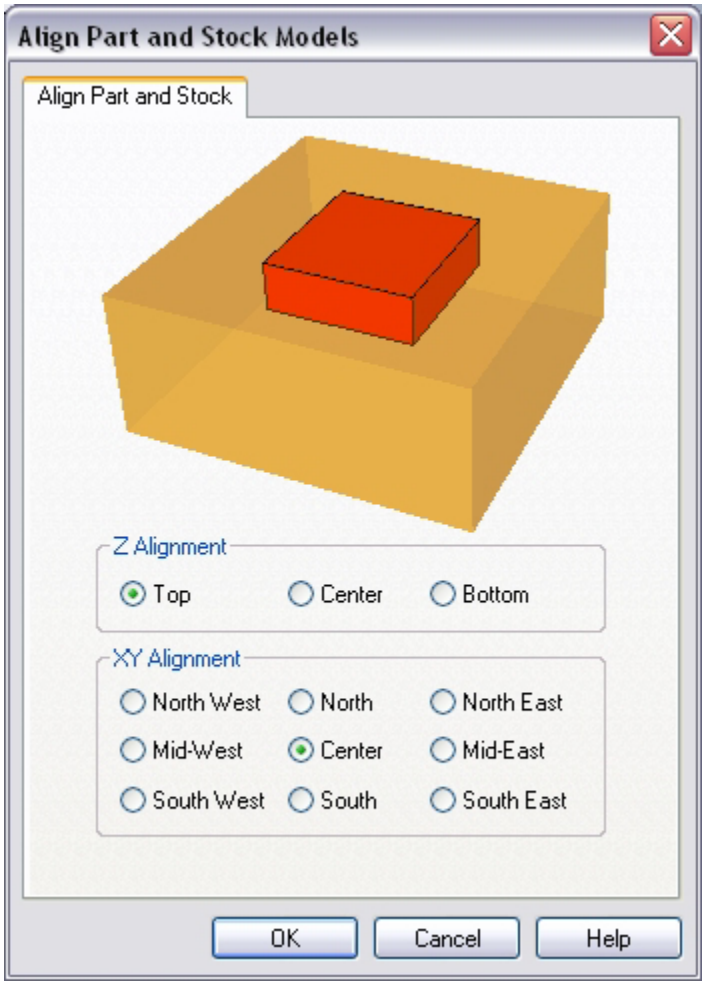
Align Part and Stock

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



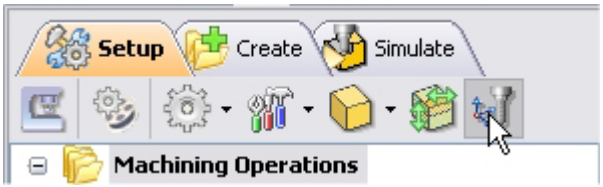
2. Set Z alignment to **Top** and XY alignment to **Center.** (This would align the stock to the top of the part in Z and center in XY)



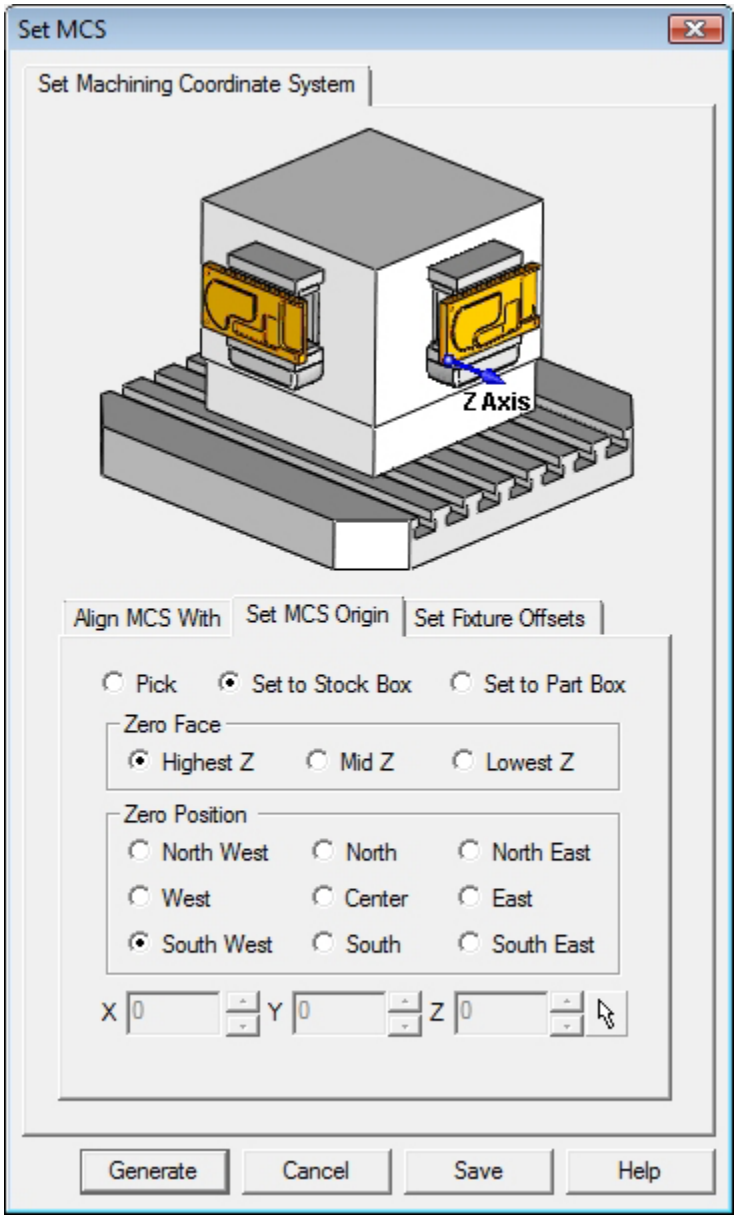
Set Machine Coordinate System (MCS)

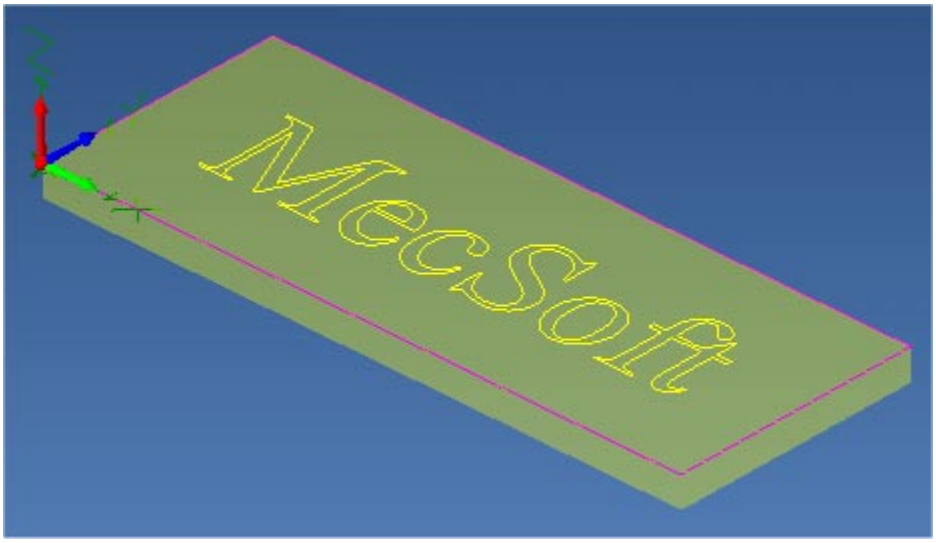
The steps below help you determine the machine home (also known as machine zero or tool touch off point) for the part/stock geometry.

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
Y
Z
Generate
Cancel
Save
Help



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

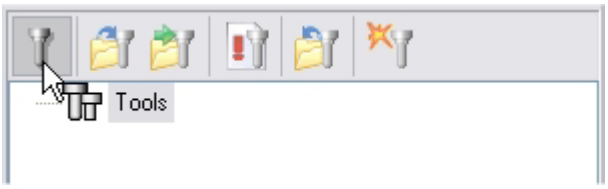
Note: You can change the stock model transparency under standard mode by selecting Simulation Preferences that is located at the bottom of the MOps browser.

Click Save As to save the work and specify a file name as V-Carve1-Rev1.

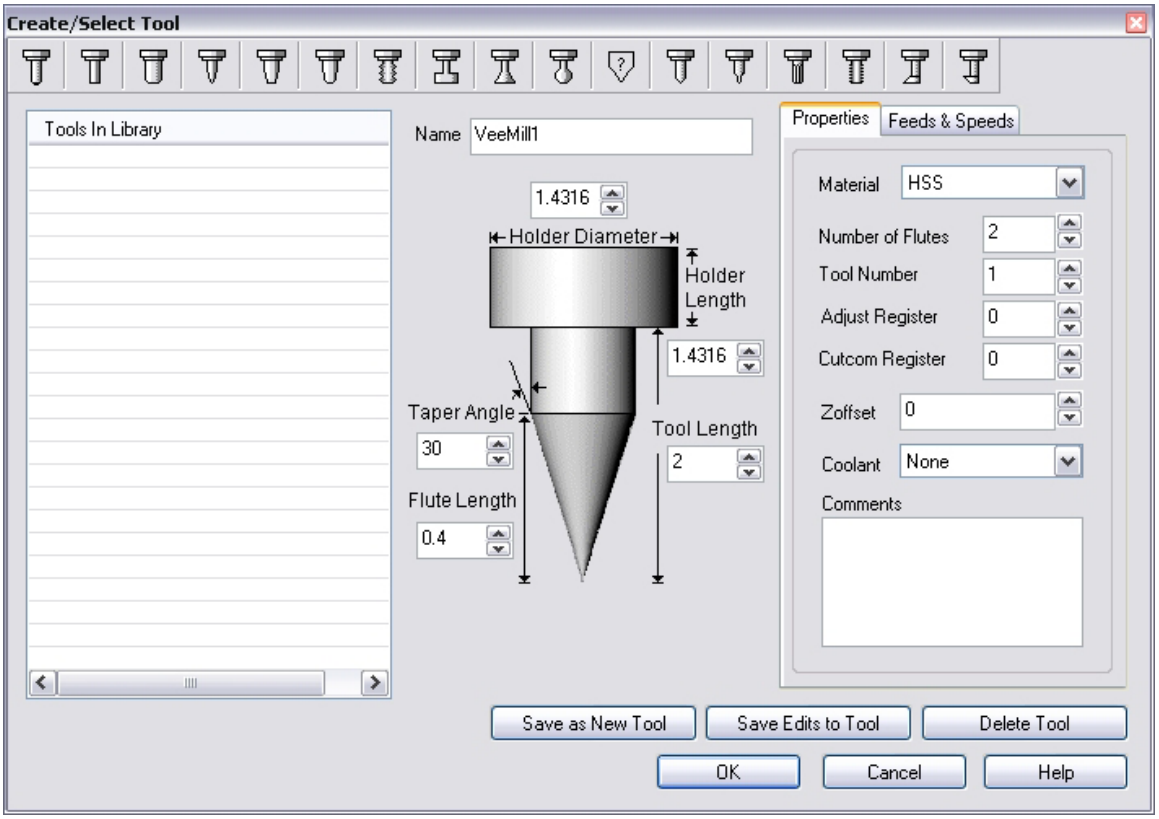
Create Tools

To machine the above part, we will now create a 60-degree Taper Tool.

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 VeeMill.



2. Set the tool name as **VeeMill1**, Taper Angle = **30,** Flute Length = **0.4**, Tool Length = **2**. Under the Properties tab set Tool Number = **1**.



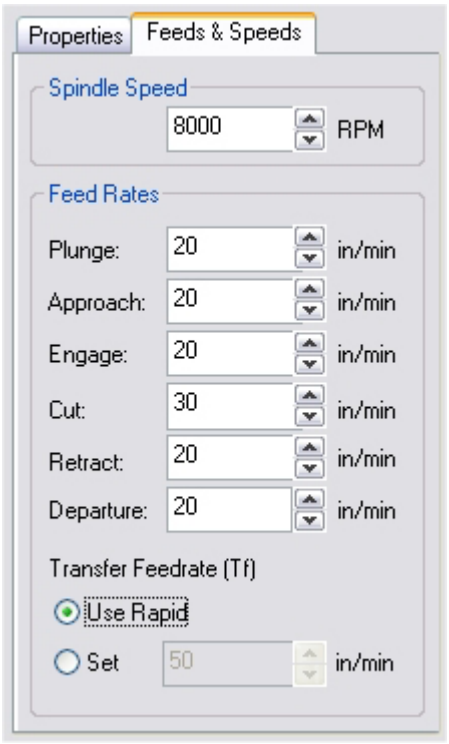
Note: Taper Angle represents the included angle for a taper tool. For example a 60-degree taper tool would have a included angle of 30 degrees. If you have a taper tool with a diameter select Chamfer Mill or Taper Mill under Create/Select Tool.

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.

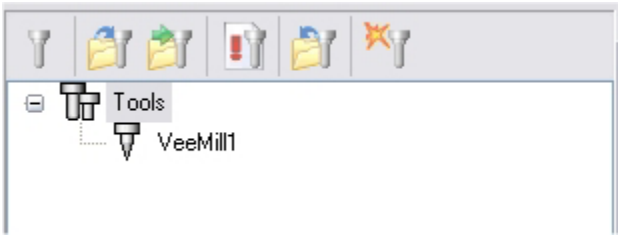
3. Switch to the Feeds & Speeds tab inside the create/select tool dialog.

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 tools are now listed under the Cutting Tools browser.

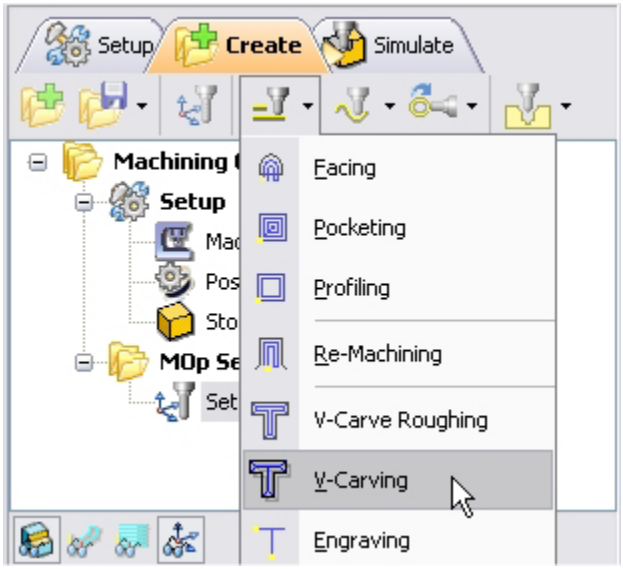
Create Machining Operations

In this process we will create a 2.5 axis V-Carve operation.

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

V-Carving

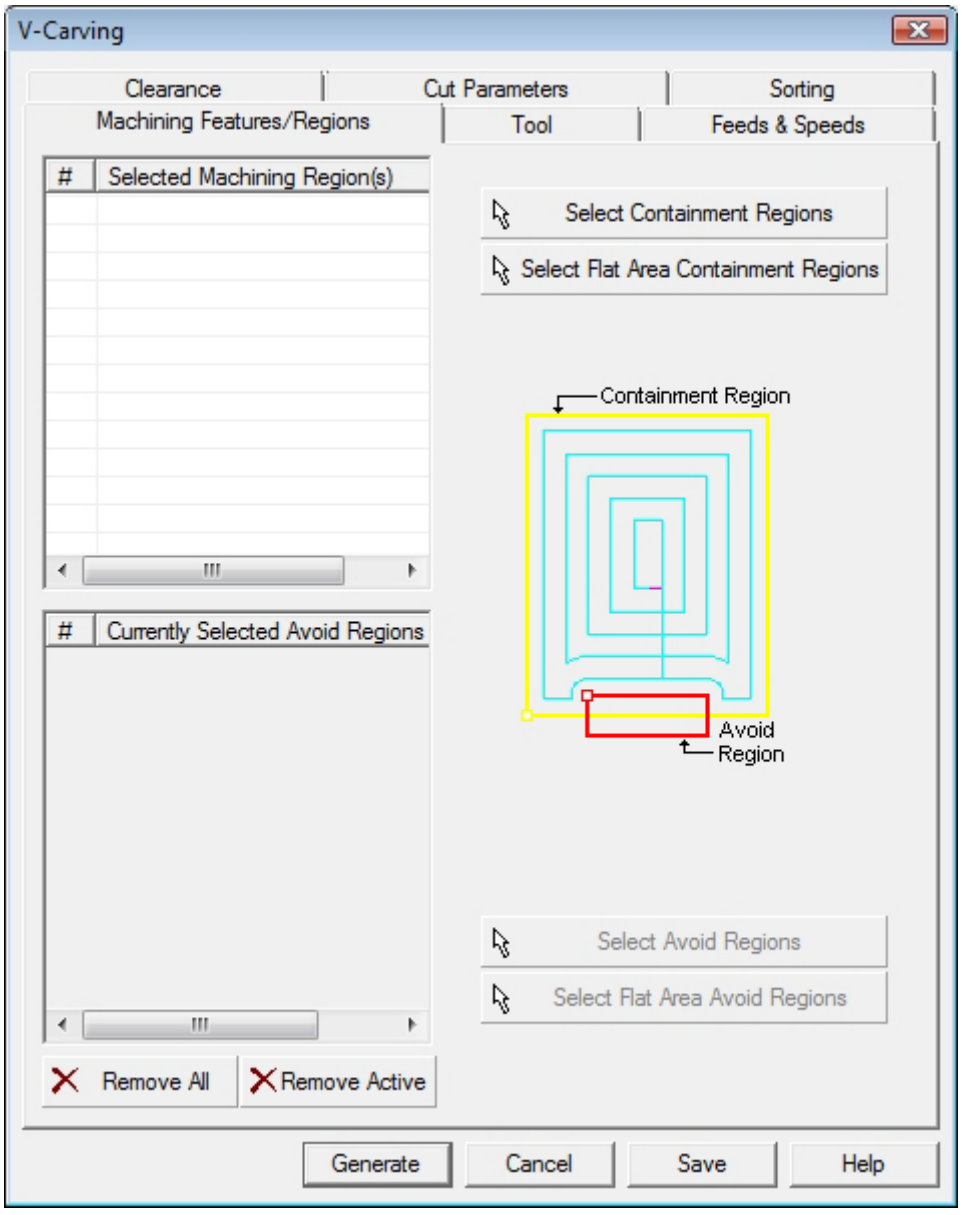
2. Select 2.5 Axis Mill and choose V-Carving.



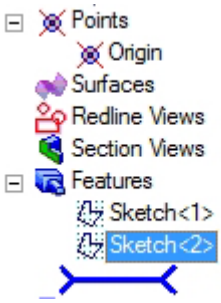
This brings up the V-Carving Operations dialog. We will go over the steps for creating the toolpath.

Select Machining Features/Regions

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



4. Switch to Design Explorer and select **Sketch2**. This will select the text.

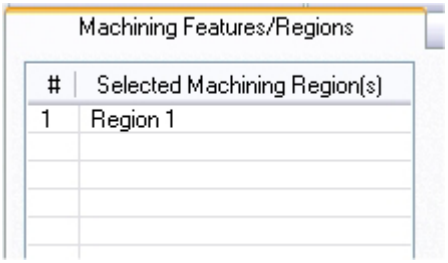


Note: You can also select the sketch from Alibre Design View. When select containment region is selected, Alibre CAM turns on Sketch selection filter to allow the user to select sketches. Hold the Shift key down to select multiple sketches.

5. Click 

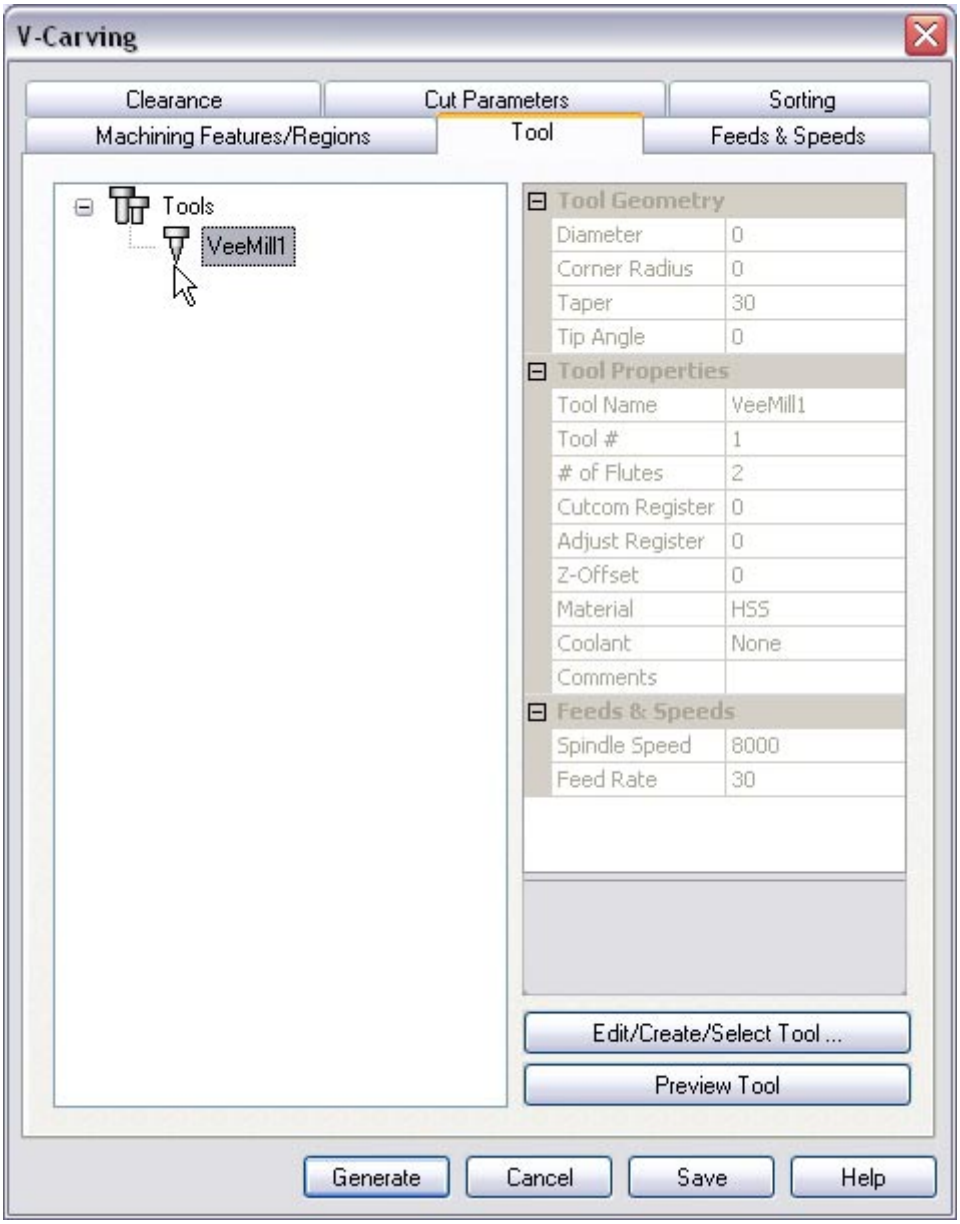
to complete the selection.

6. V-Carve operation dialog comes back up displaying the selected regions. The selected regions are also highlighted on the part



Selecting the Tool

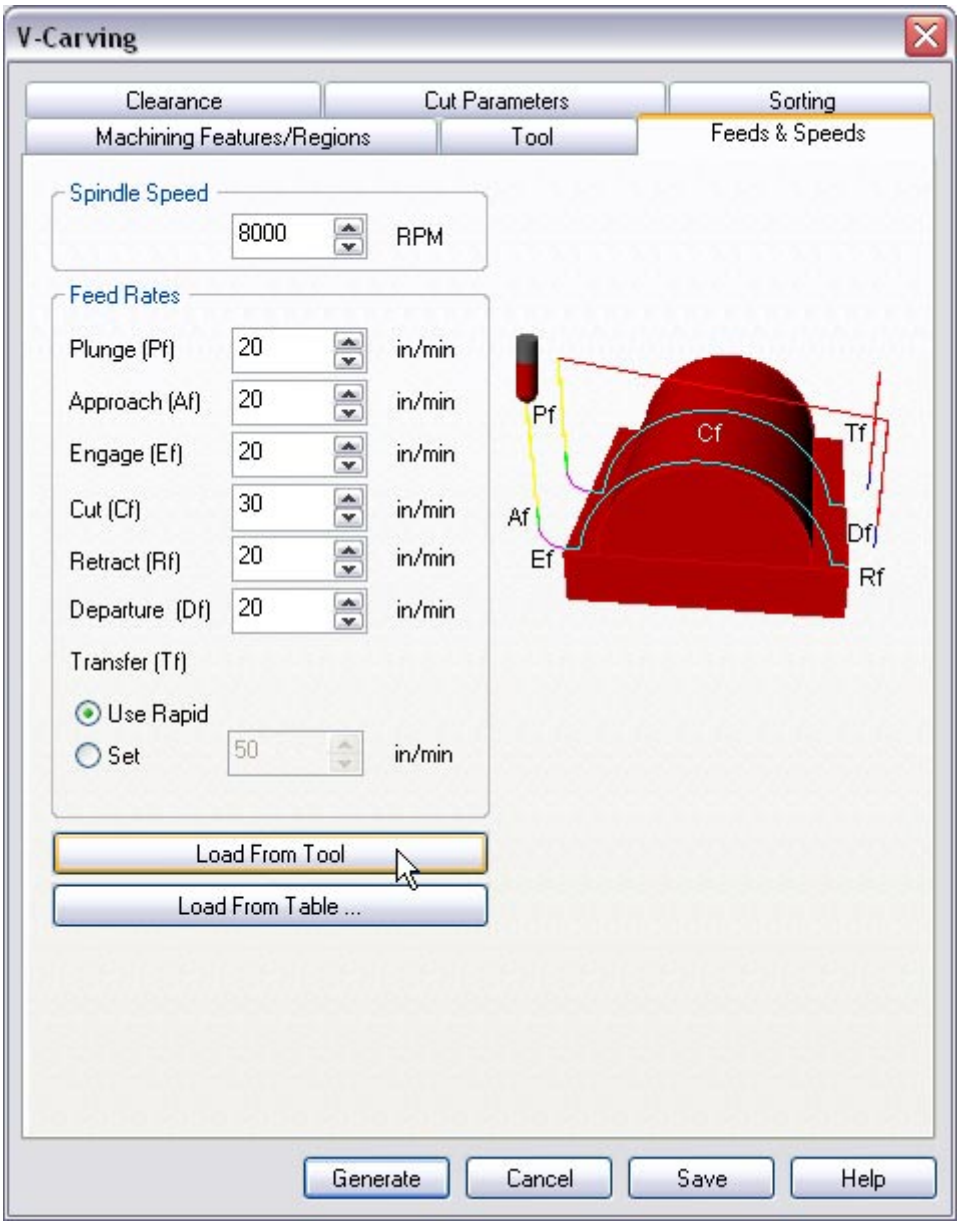
7. Switch to the Tools tab inside the V-Carving operation.



8. Select VeeMill1. VeeMill1 is now selected as the active tool and the Tool parameters are displayed to the right of the Tools window.

Set Feeds and Speeds

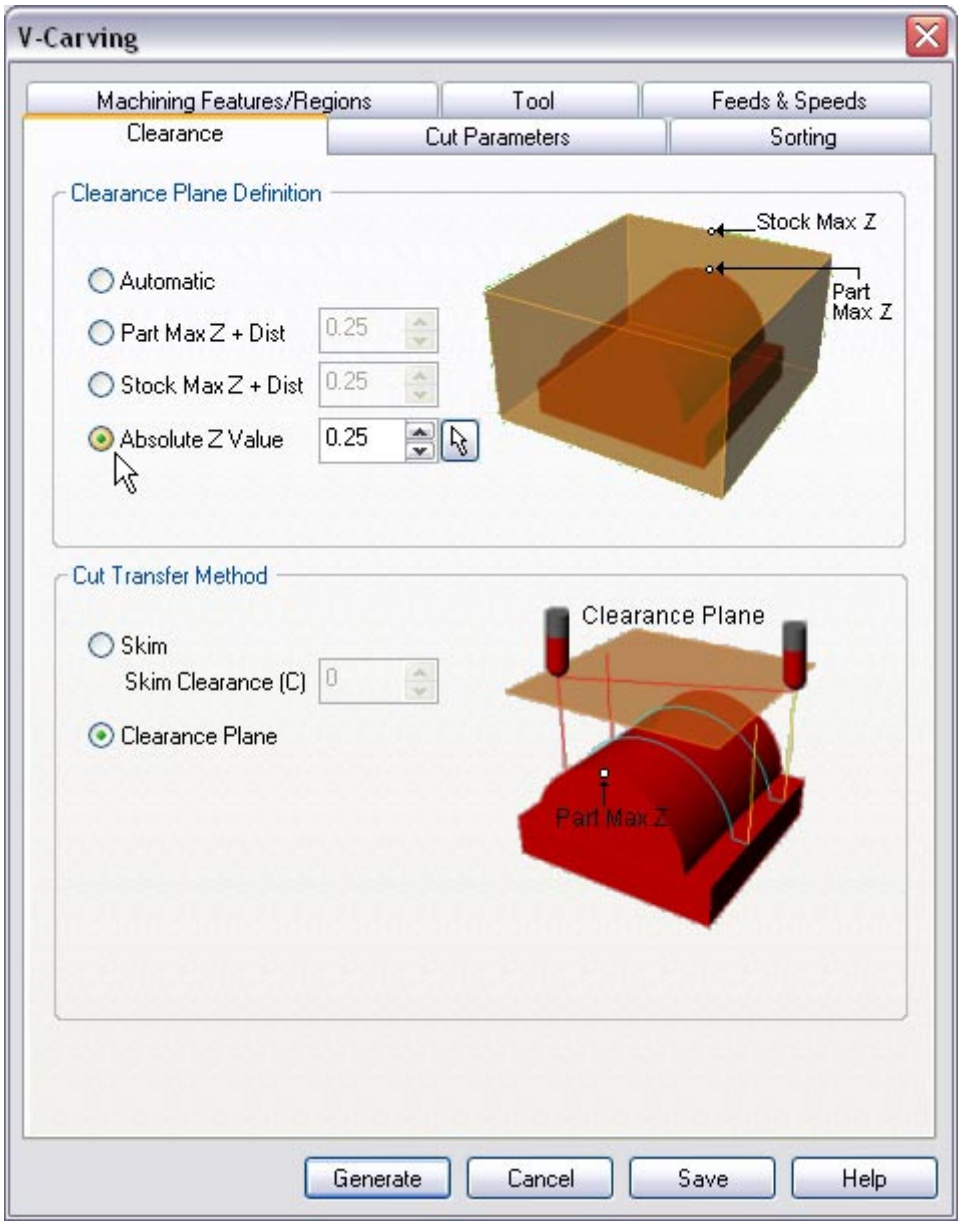
9. Click on the Feeds and Speeds tab.



10. Select Load From Tool. Alibre CAM will now get the feeds and speeds information that was set when the tool was defined.

Clearance Control

11. Switch to Clearance Tab.

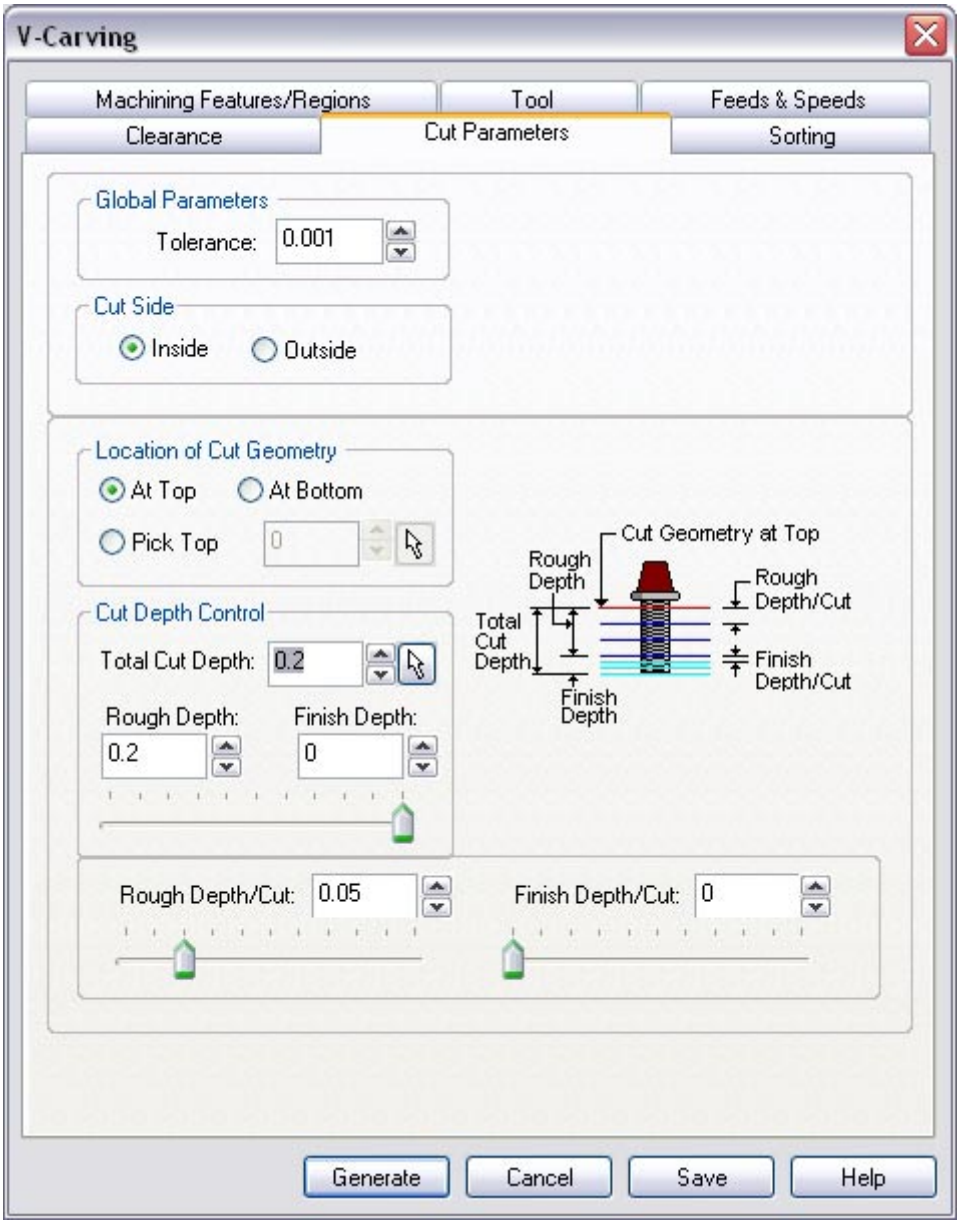


12. Set the Clearance Plane Definition to Absolute Z Value = **0.25** and Cut Transfer Method to Clearance Plane.

Setting Cut Transfer to Clearance Plane would apply the Absolute Z value clearance between transfers when the tool moves from a machining region to another.

Specifying Cut Parameters

13. Switch to Cut Parameters tab.

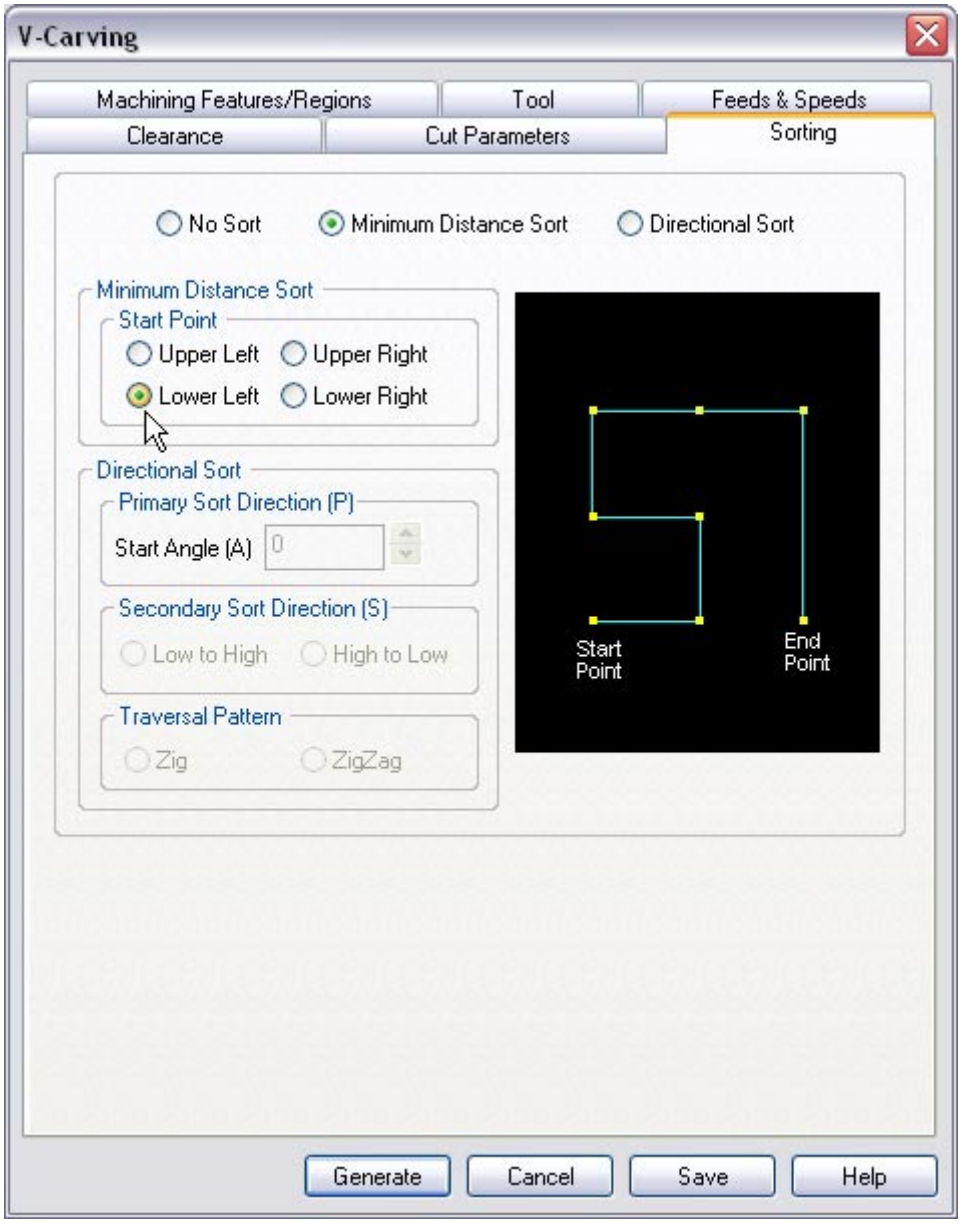


14. Set the Tolerance = **0.001**, Cut Side = **Inside**, Location of Cut Geometry = **At Top**, Total Cut Depth = **0.2**, Rough Depth = **0.2** and Rough Depth/Cut = **0.05**.

Sorting

15. Select the Sorting Tab.

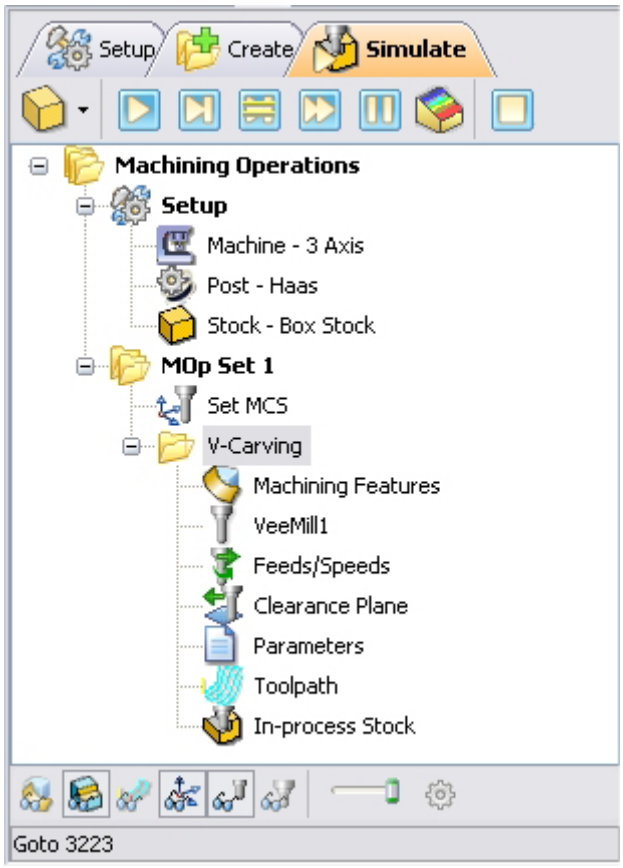
16. Use Minimum Distance Sort & set Start Point as Lower Left.

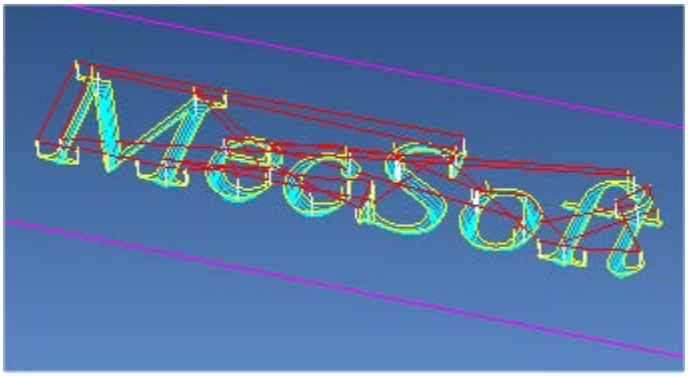


This will sort based on the minimum distance between 2 text based on their start points.

17. Click **Generate**. The V-Carving toolpath is now generated and the Operation is listed under the Alibre CAM MOps browser.

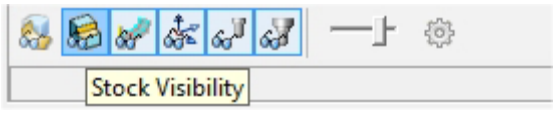
Note: Toolpath display can be turned on/off by selecting Toolpath Visibility under the MOps browser.





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 the V-Carving Operation and click  to launch the Alibre CAM Stock Simulation window.

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

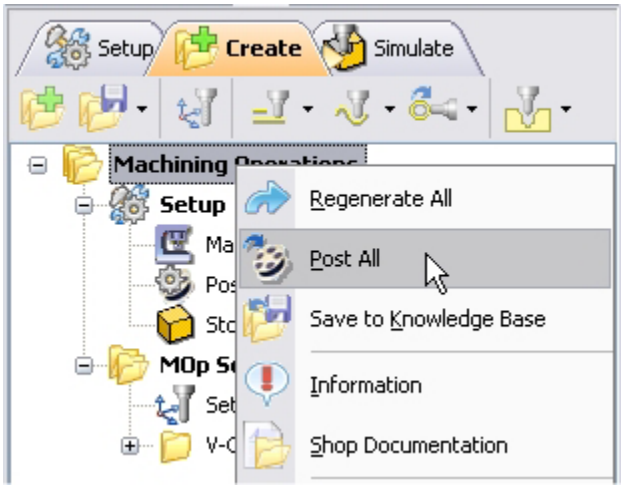
The simulated part is as shown below.

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 **V-Carve.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.