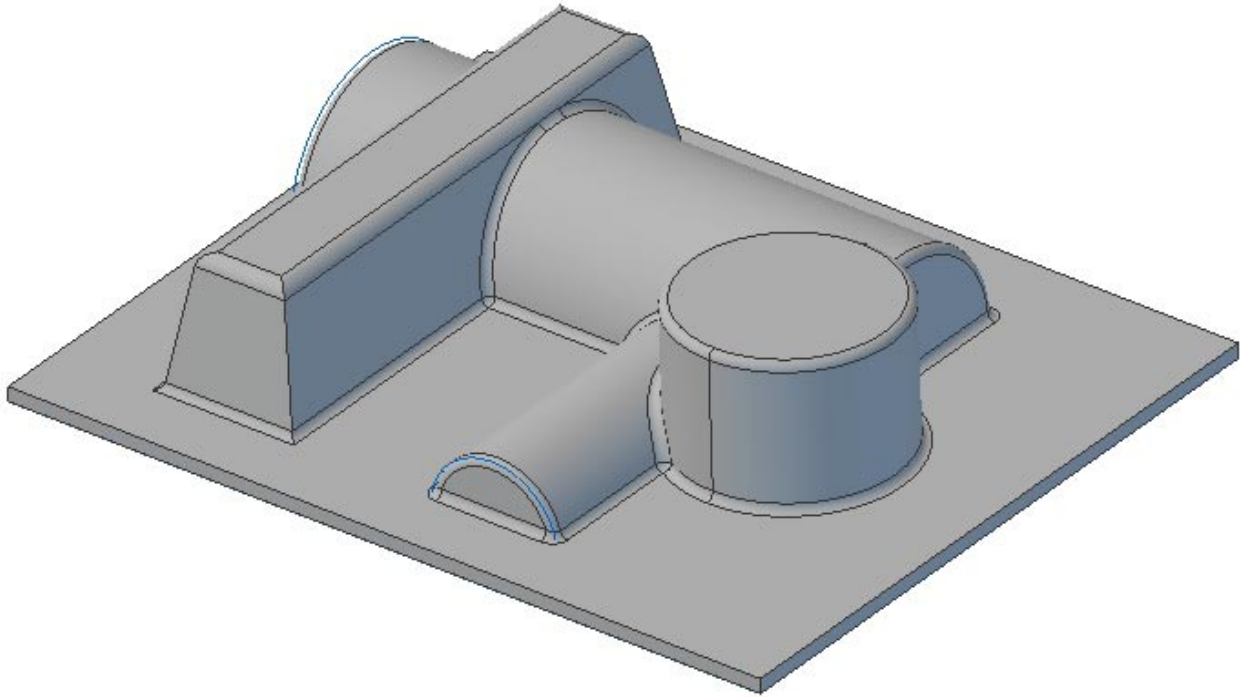


Tutorial 10: Re-Machining a 3D Mold



Using 3 axis Pencil Tracing and Valley Re-Machining Toolpath methods

Introduction

This tutorial will illustrate machining this Mold using advanced 3 axis-milling operations.

This tutorial will introduce the usage of advanced 3-axis operations such as Pencil Tracing & Valley Remachining.

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 4.5 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 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.

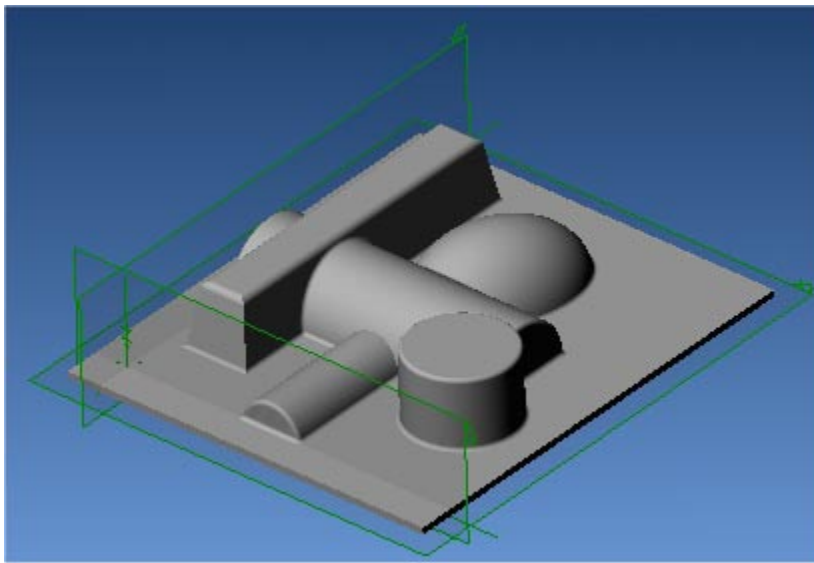
Loading the Part Model

Getting Started with Alibre CAM

"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 **3dMold2.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)

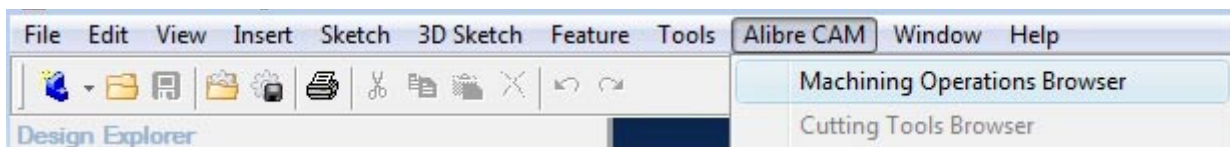
The loaded part appears as shown below.

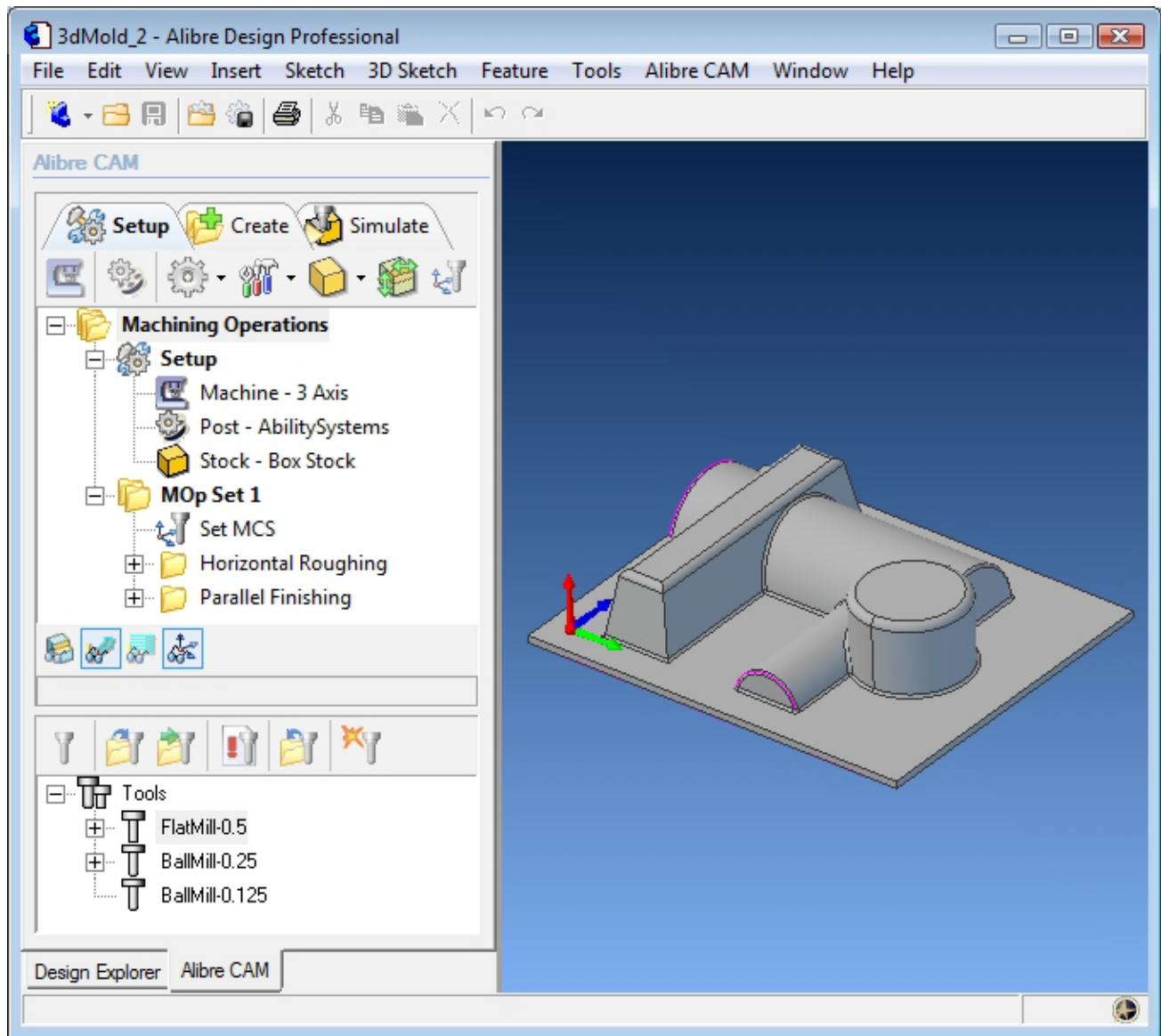


Note: You must work in shaded mode in order to be able to visualize toolpaths created in Alibre CAM. It is suggested for best visual performance with Alibre CAM to work with only one view port open and the view port operating in shaded mode.

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.

Create Machining Operations

Note: The part is pre-programmed with 2 machining operations. Horizontal Roughing and Parallel Finishing.

We will now use Remachining to finish the mold using 2 different machining operations – Pencil Tracing and Valley Re-Machining.

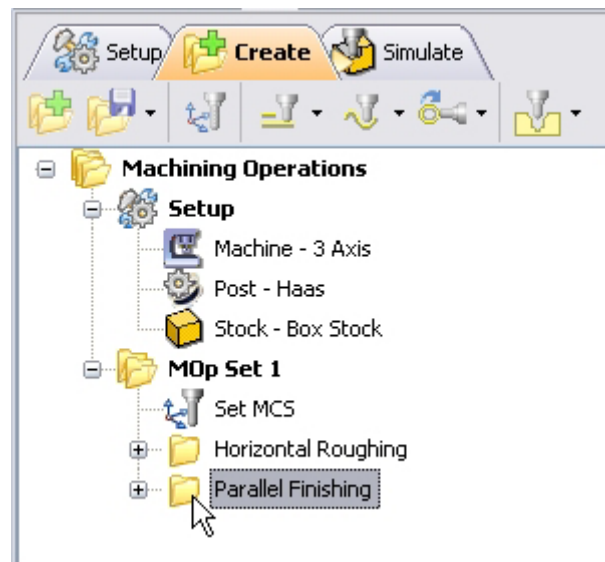
Getting Started with Alibre CAM

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

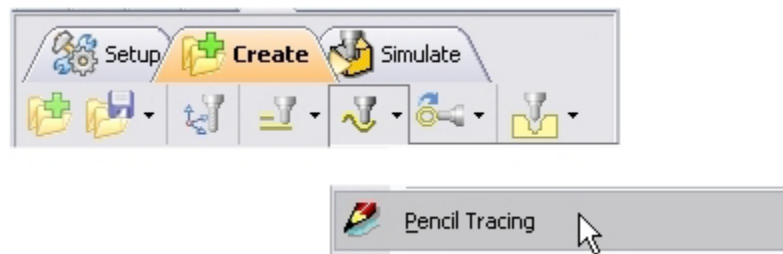


3 axis Pencil Tracing

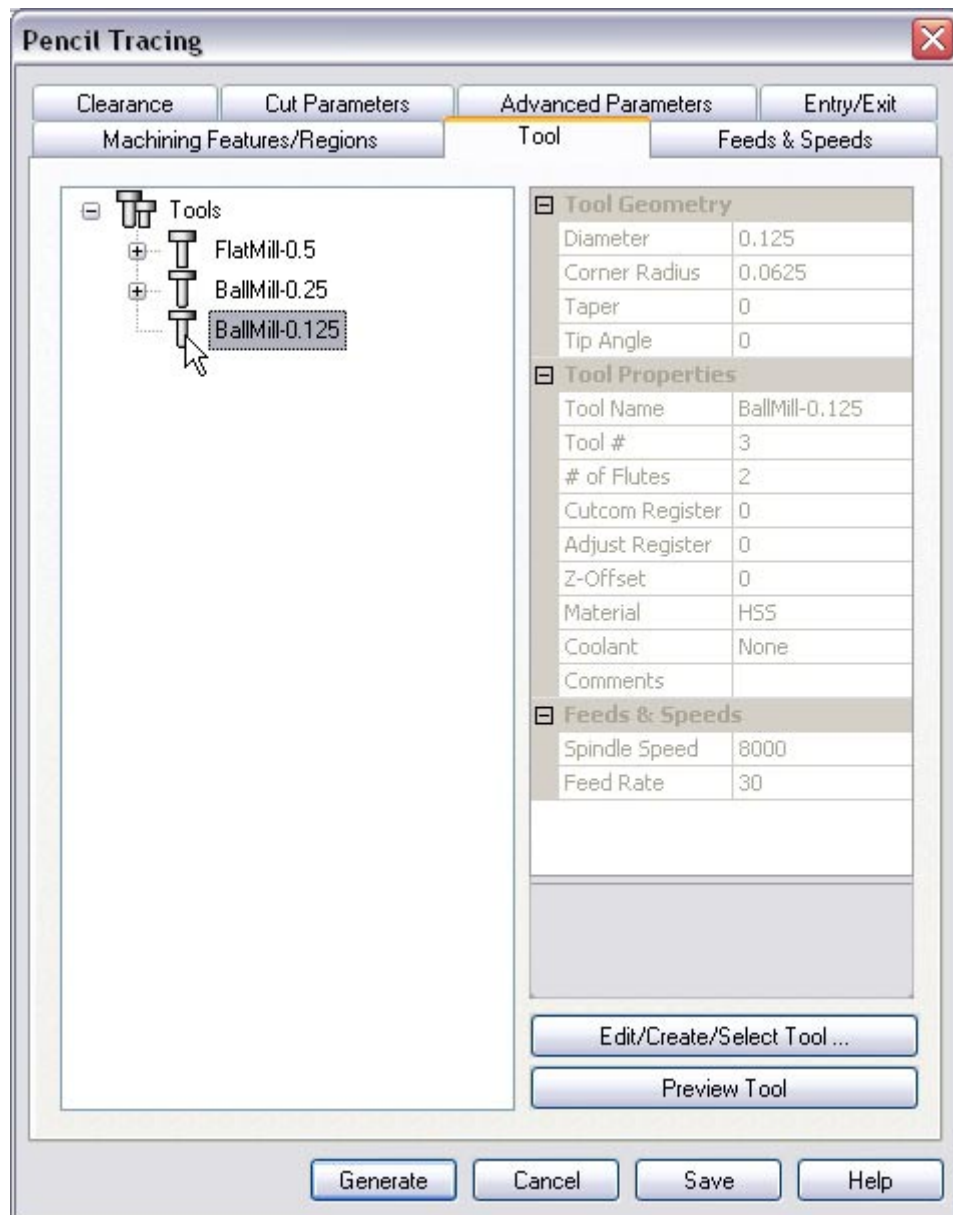
1. Highlight the Parallel Finishing operation in the Create operations tab. This would ensure the next machining operation created would be below the Parallel Finishing toolpath.



2. Select 3 Axis Milling and choose Pencil Tracing.

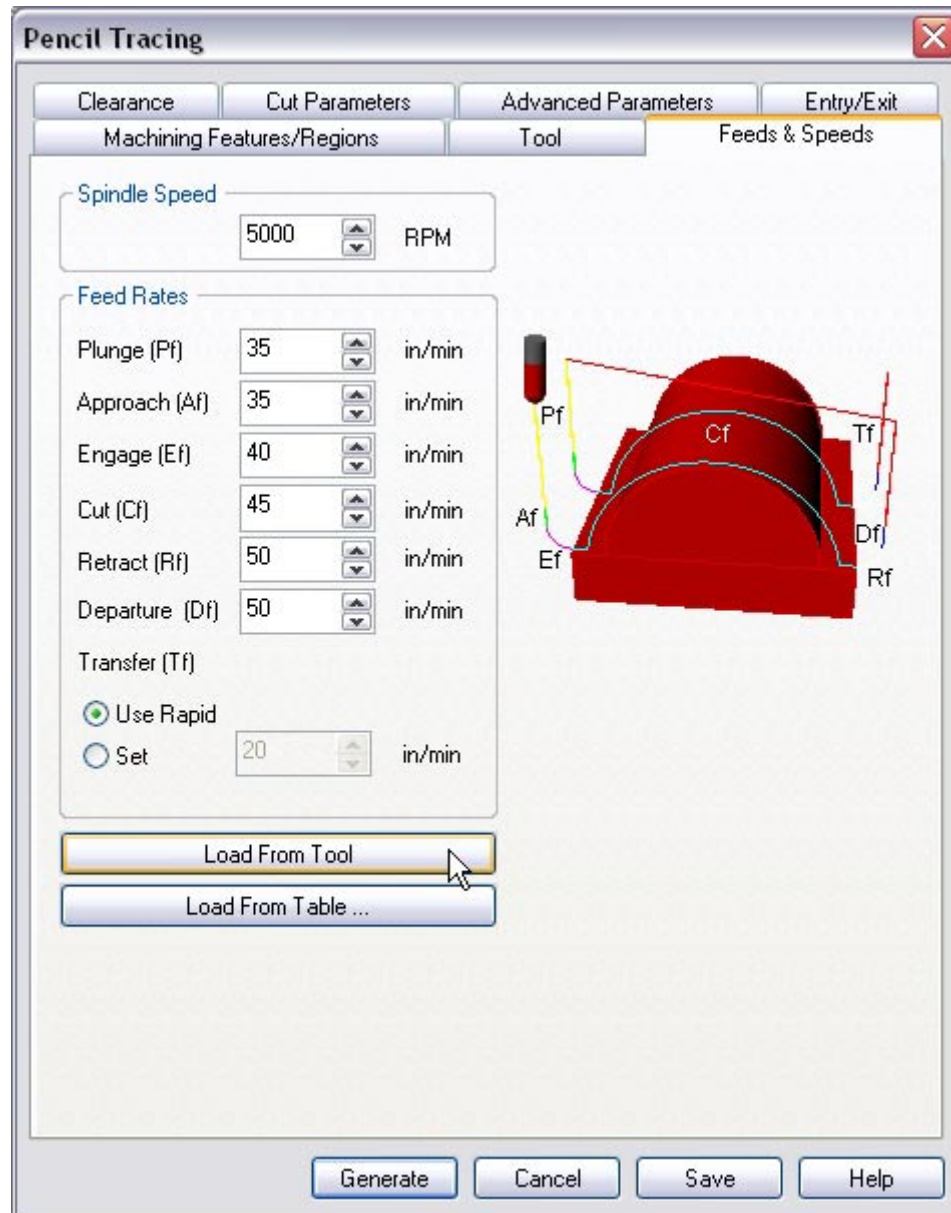


3. This brings up the 3 Axis Pencil Tracing Operation Dialog. We will now go over the steps for creating the toolpath.
4. Switch to the Tools tab inside the Pencil Tracing operation and select BallMill-0.125.

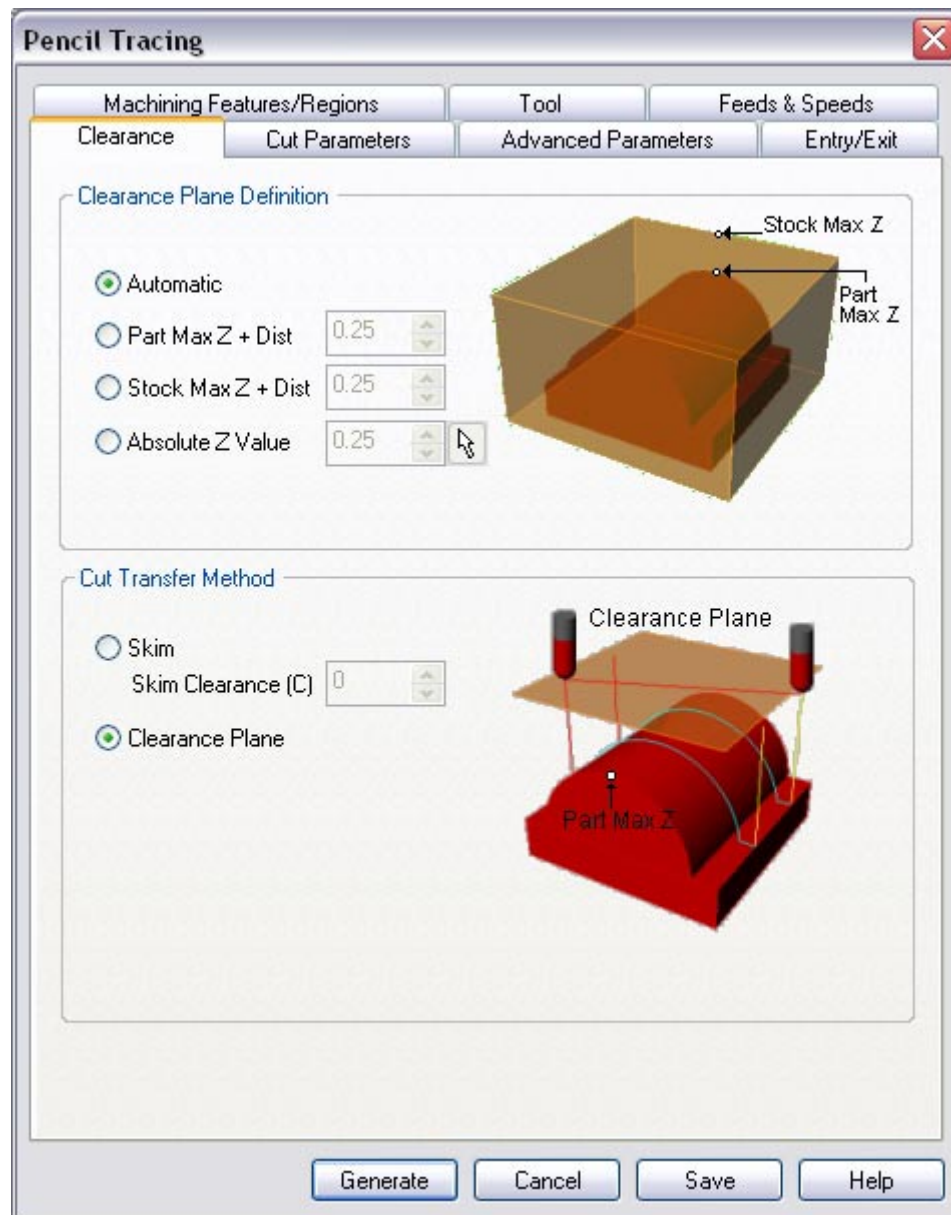


Getting Started with Alibre CAM

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.

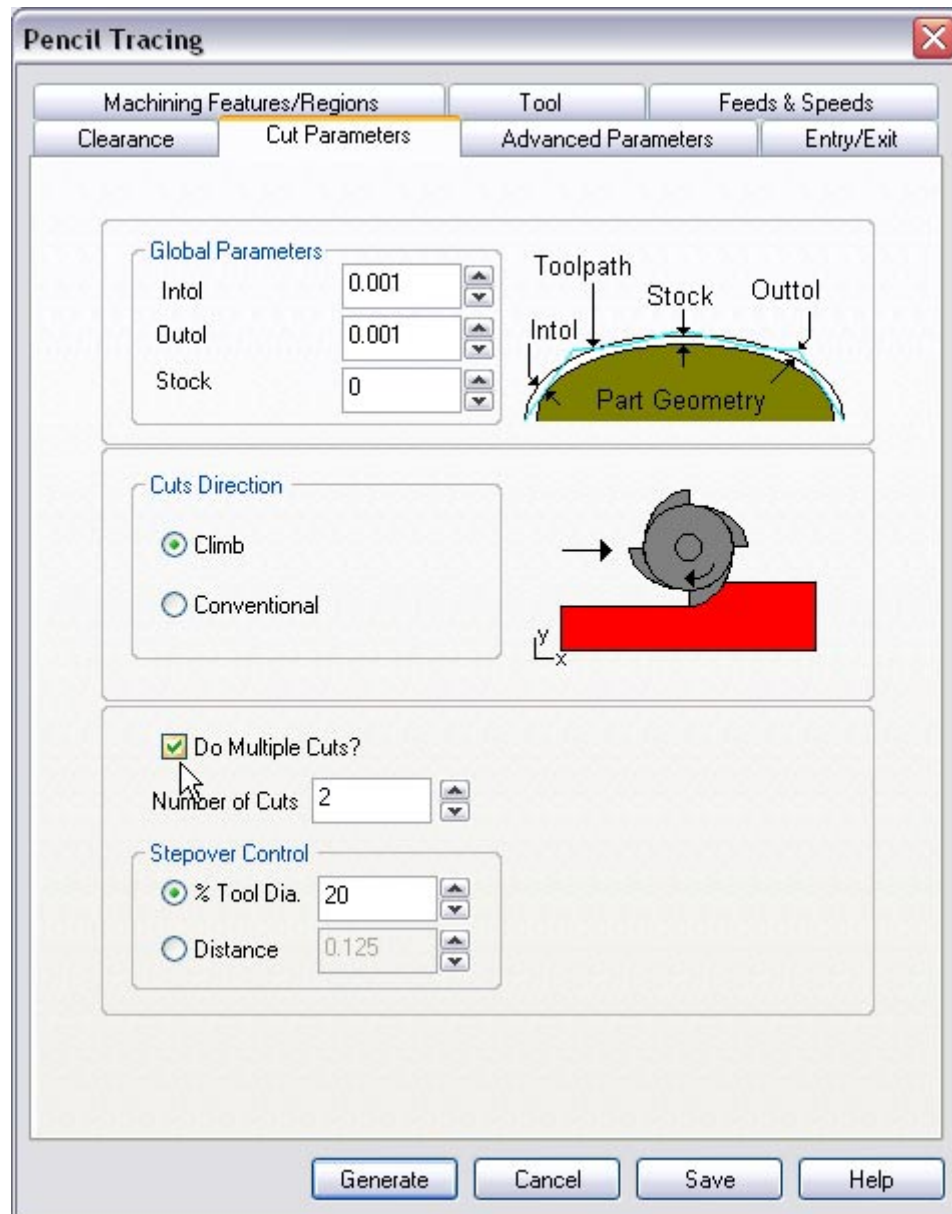


6. Switch to the Clearance Tab and set the Clearance Plane Definition to Automatic and Cut Transfer Method to Clearance Plane.



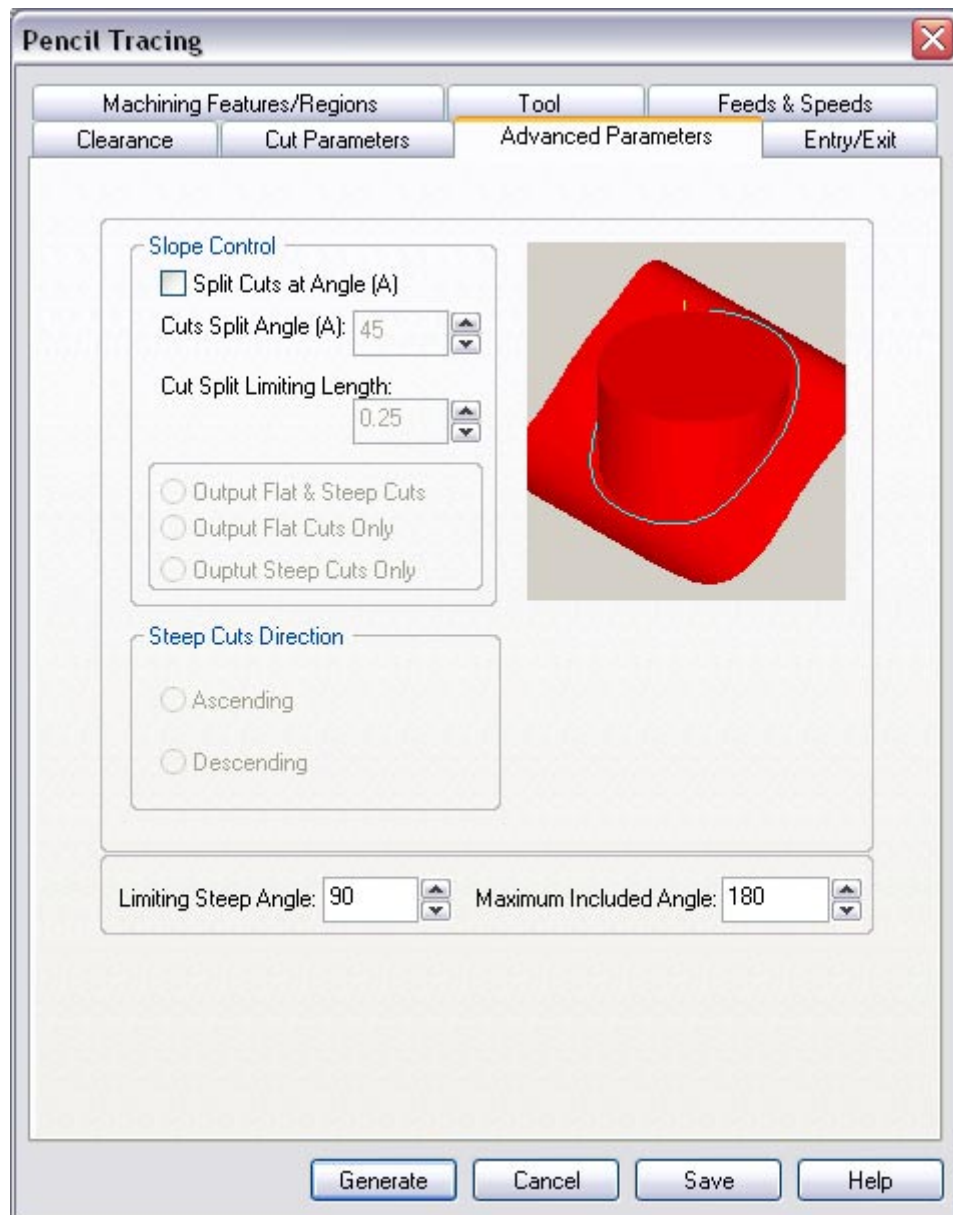
Specify Cut Parameters

- Click on the Cut Parameters tab.
- Set the Intol and Outol = **0.001**, Stock to leave = **0.0**, Cut Direction to **Climb**, Select **Do multiple cuts** and set Number of Cuts = **2** and Step over control = **20** (% Tool Diameter).



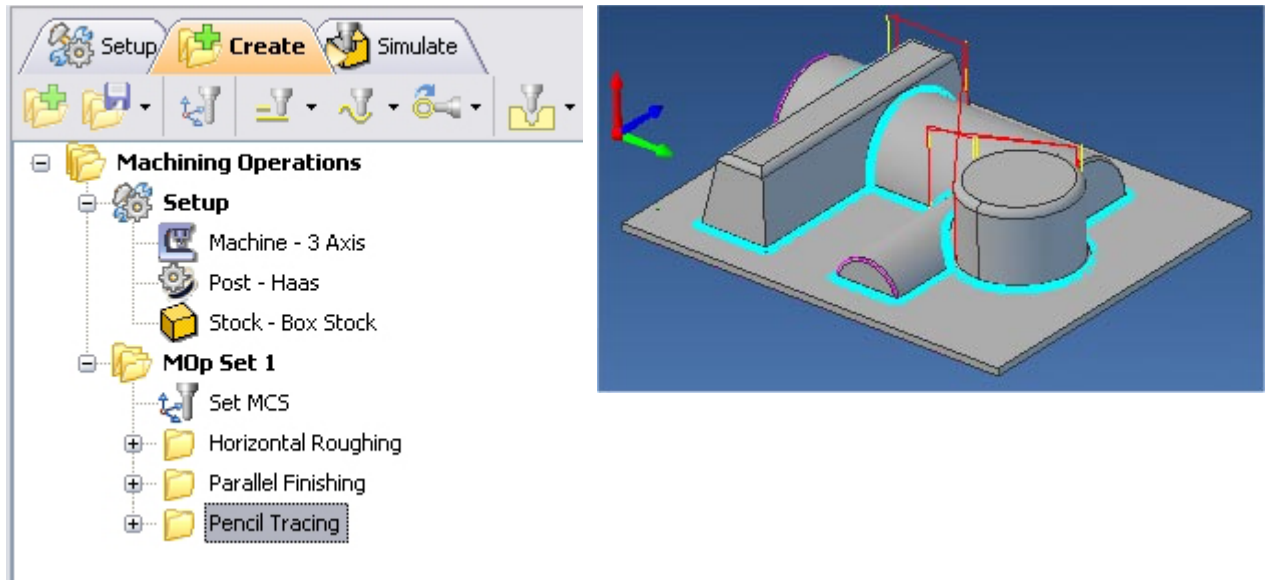
Advanced Cut parameters

9. Switch to the advanced cut parameters Tab.
10. Use the Following Settings.
 - a. Limiting Steep Angle = 90 and Maximum included angle =180.



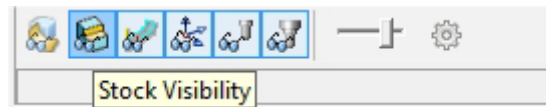
11. Click **Generate**. The 3-axis Pencil Tracing toolpath is now generated, and the Operation is listed under the Alibre CAM MOPs browser.




Getting Started with Alibre CAM

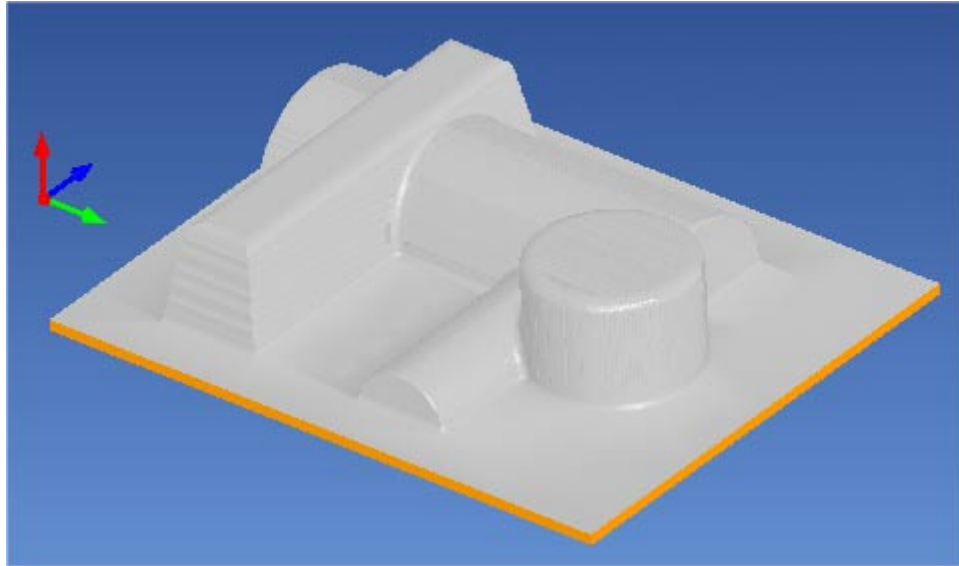


Simulate Toolpath

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



1. Select the 3 axis Roughing Operation and click  to launch the Alibre CAM Stock Simulation window.
2. After simulating the roughing operation, close the Stock Simulation window, select the Parallel Finishing Operation and click  to launch the Alibre CAM Stock Simulation window.
3. Select the Pencil Tracing operation and click  to launch the Alibre CAM Stock Simulation window.
4. 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.

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

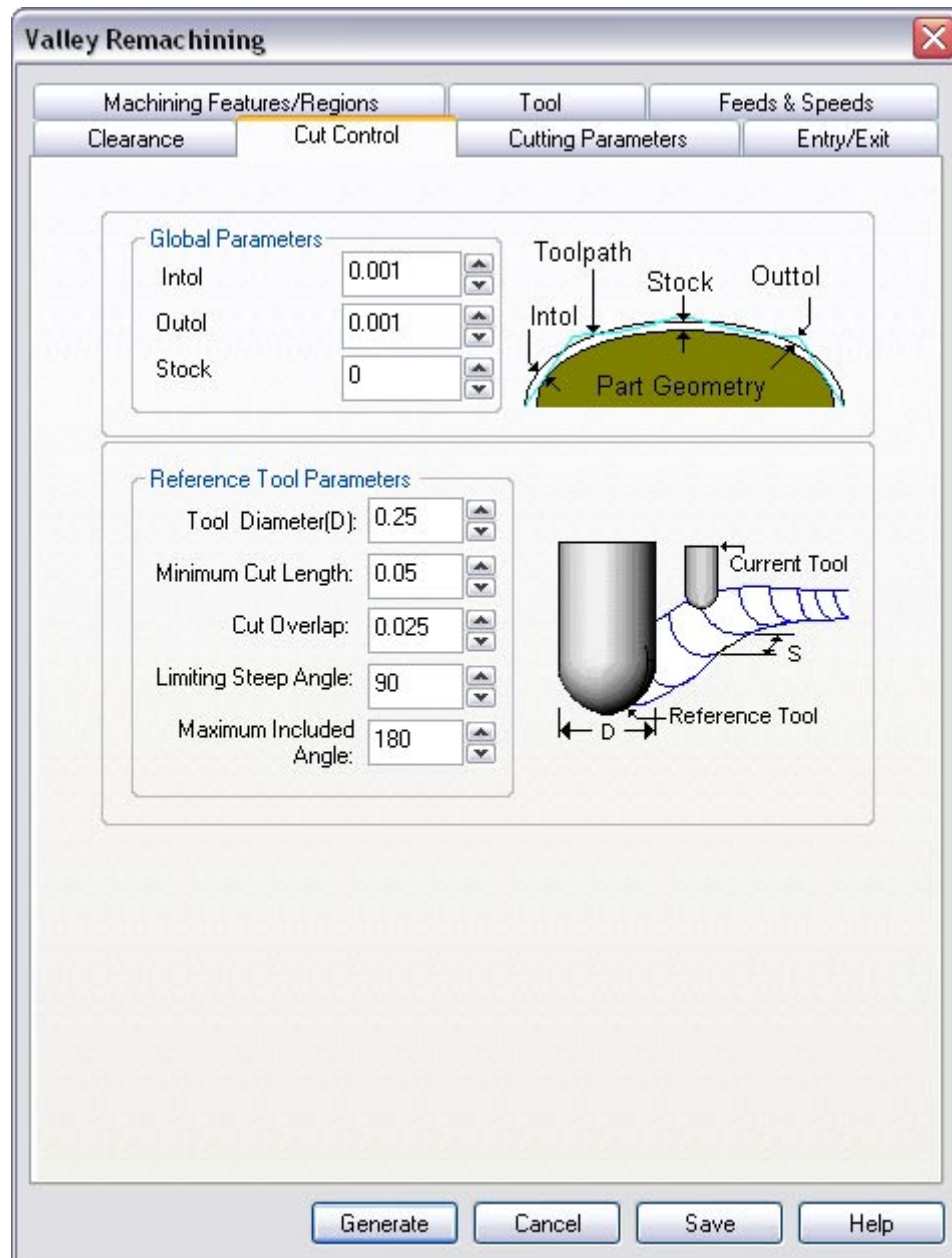
3 axis Valley Re-Machining

We will now use 3 axis Valley Re-Machining operation to finish the part using a 0.125" Ball End Mill.

1. From the Create Operations tab, select 3 axis Milling and Valley Re-Machining.
2. This brings up the Valley Re-Machining Operations dialog. We will go over the steps for creating the pocketing operation.
3. Switch to the Tools tab inside the Valley Re-Machining operation and select BallMill-0.125.
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.
6. Click on the Cut control tab.

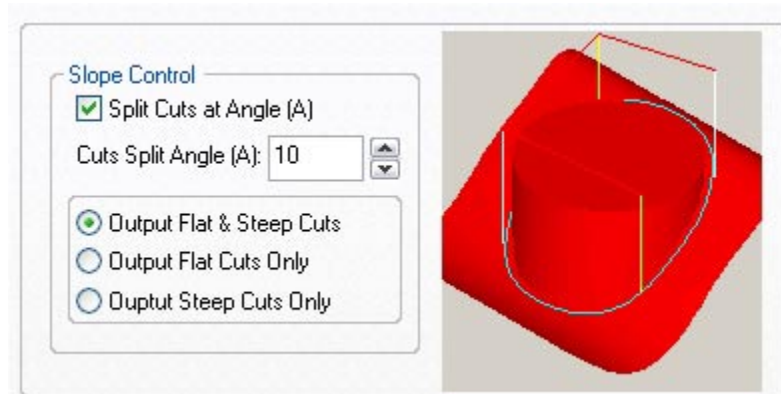
Specify Cut Control

- Click on the cut control tab.
- Set the Tolerance to **0.001**, Stock to leave = **0**, Under Reference Tool Parameters set Tool Diameter (D) = 0.25 (This is the diameter of the tool that was used for parallel finishing).

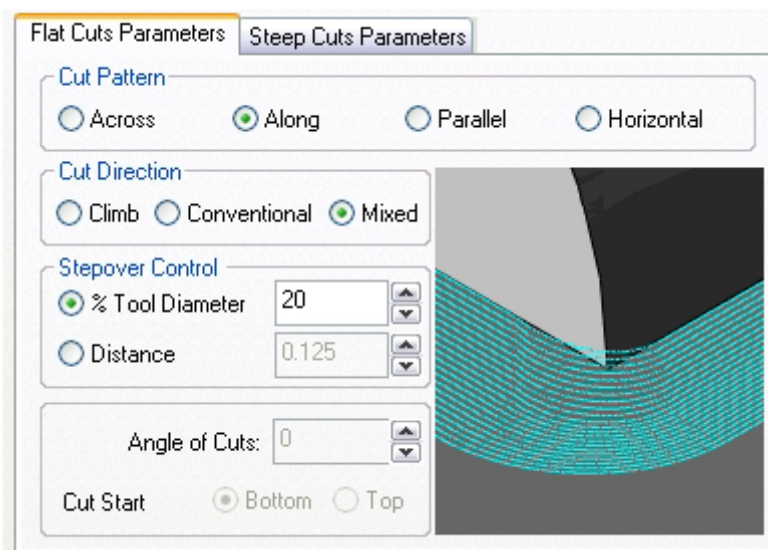


Specify Cutting Parameters

9. Click on the Cutting Parameters tab.
10. Use Split Cuts under slope Control. Specify the Cuts Split Angle (A) = **10**
11. Select Output Flat and Steep Cuts.

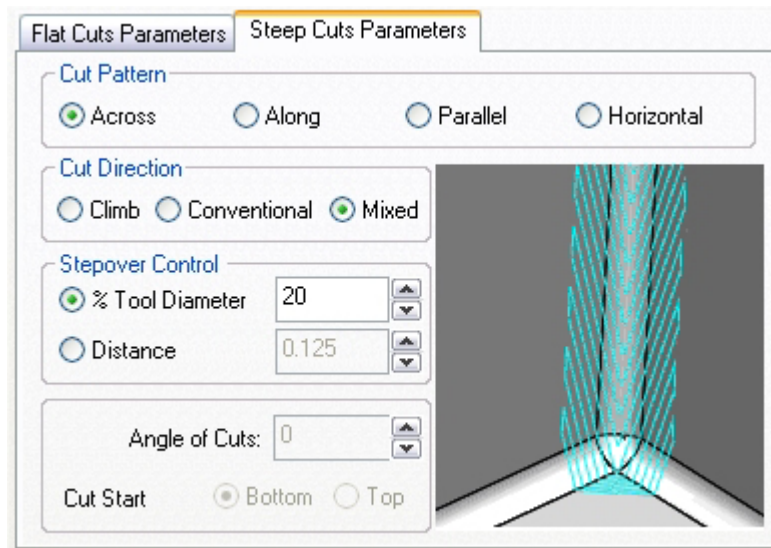


12. Select **Flat Cut Parameters**.
13. Select Cut Pattern as Along, Cut Direction as Along, Cut Direction as Mixed and specify Step Over control = 20 (% tool diameter).

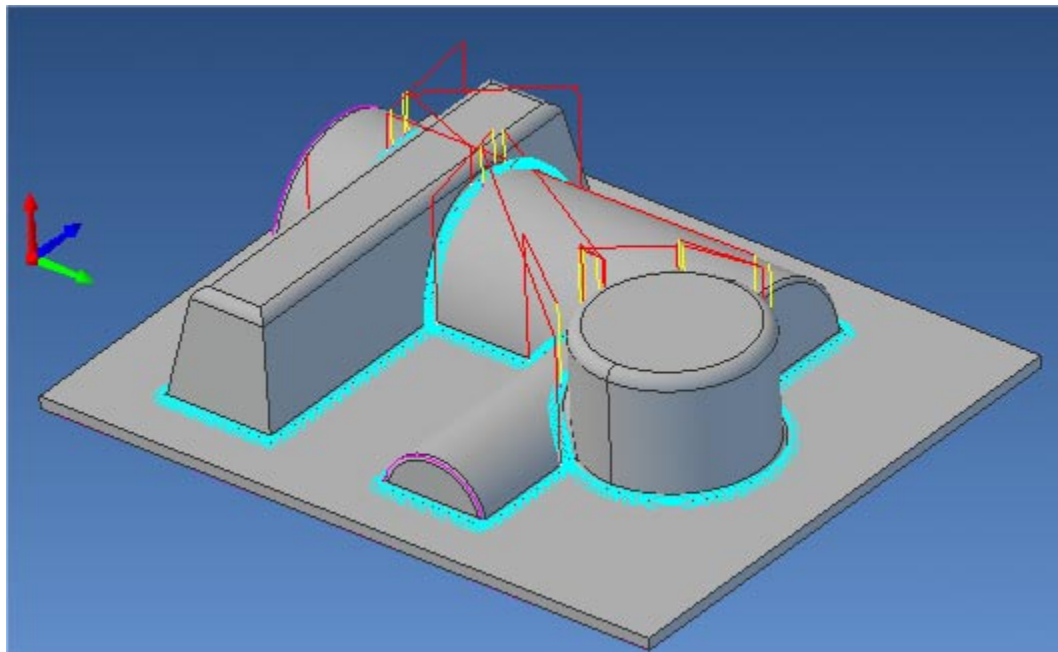


14. Select **Steep Cut Parameters**.
15. Select Cut Pattern as Across, Cut Direction as Along, Cut Direction as Mixed and specify Step Over control = 20 (% tool diameter).

Getting Started with Alibre CAM



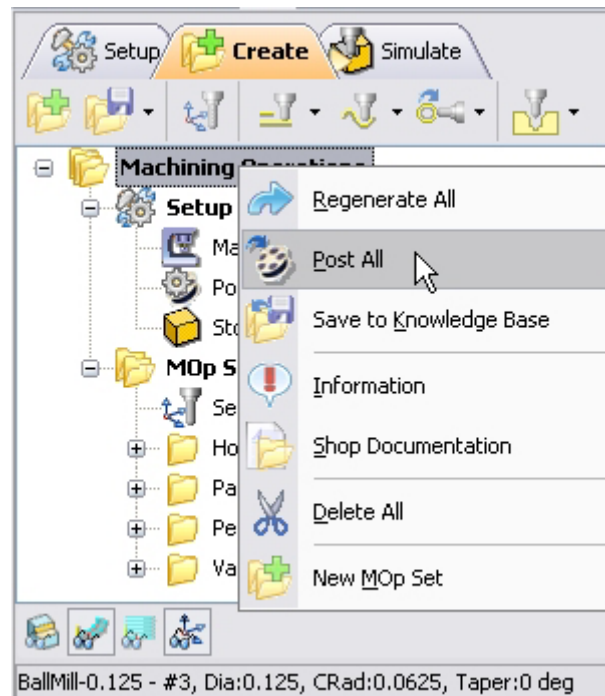
16. Click **Generate**. The Valley Remachining toolpath is now generated, and the Operation is listed in the Alibre CAM Mops browser.



17. Switch to Simulate tab, select Valley Remachining, and click Simulate to run the simulation.

Post Processing

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



2. Specify the File Name as **3axisMold2.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.

Lista Studio srl www.lista.it info@lista.it