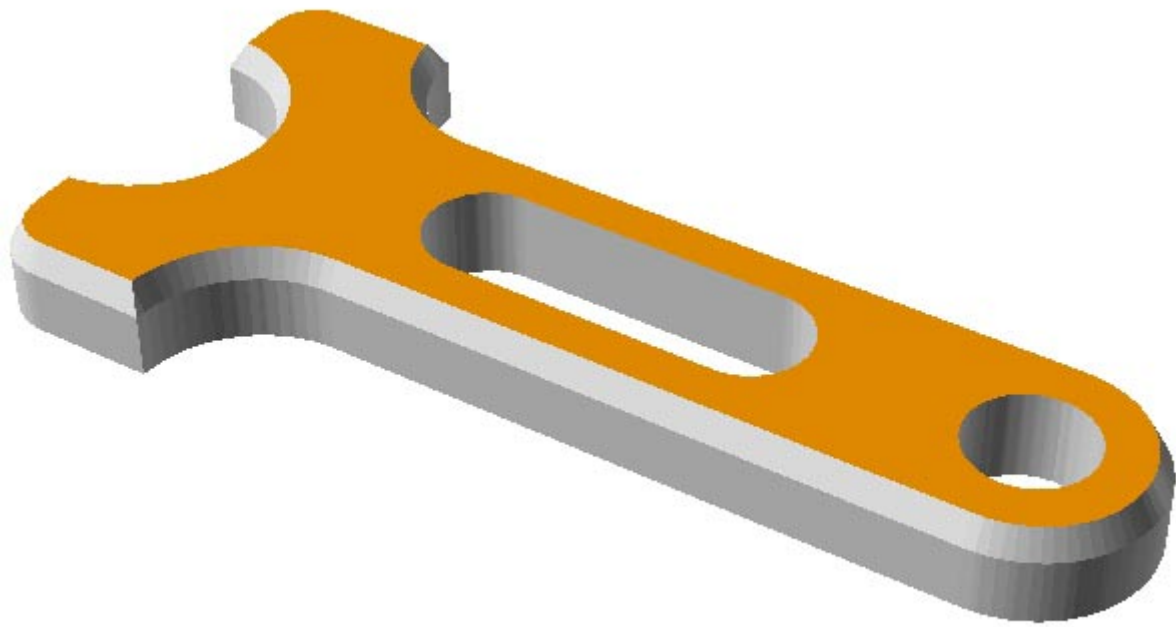## Tutorial 6: Chamfering

Introduction

This tutorial is intended to show an easy way to chamfer and smooth sharp corners by using 2 ½ axis Chamfering operation. A tapered tool is suitable for this purpose.

We plan to chamfer the edges of the part using a 30 degree taper tool that has 0 radius at the tip.

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

- Chamfering is performed using the 2 ½ axis Chamfer Machining Operation.
- 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.

3. From the **Open** dialog box, select the **Chamfer.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)
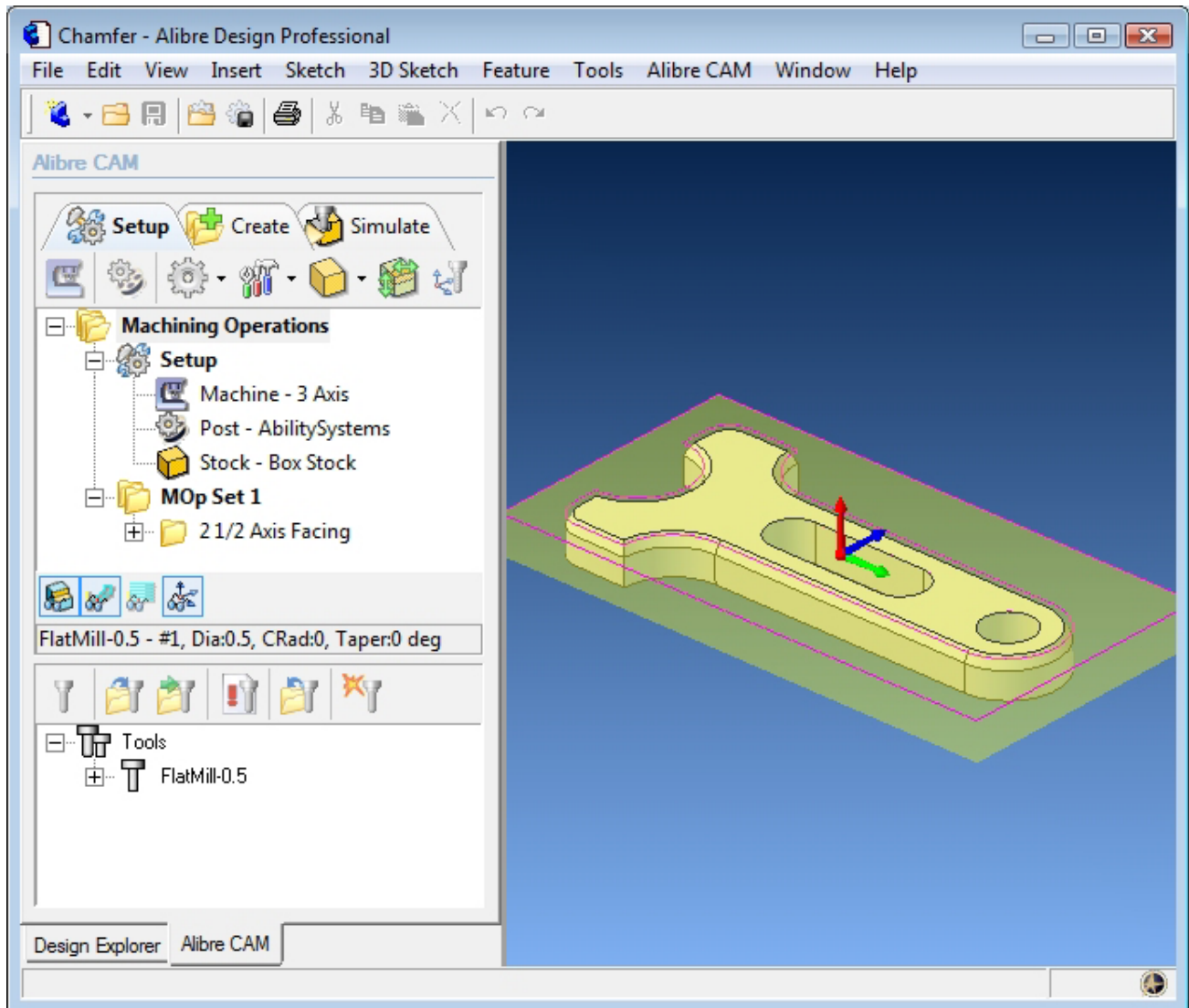
The imported part appears as shown below
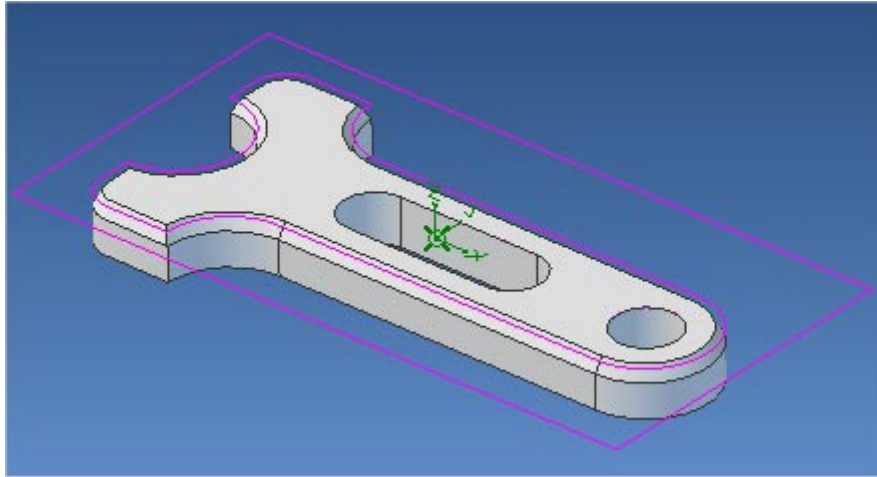
Loading the Alibre CAM Browser

1. Select Alibre CAM from the menu bar and click Machining Operations Browser.
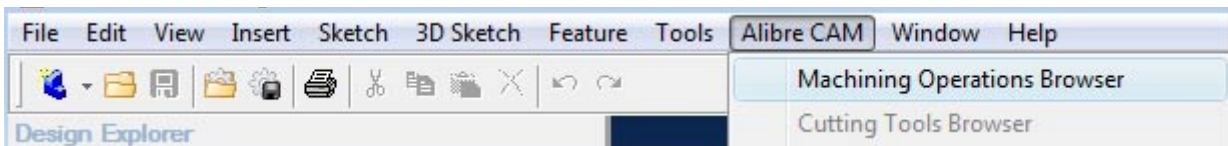
The Alibre CAM browser (MOPs and Cutting Tools) is now loaded and is docked over the Design Explorer. You can toggle between the MOPs browser and the Design Explorer from the tabs at the bottom of the window. It can be hidden by un-checking Browser on the Alibre CAM menu bar. To re-display the hidden 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. The loaded part has the stock model defined and includes a 2 ½ axis Facing Operation with a 0.5" Flat End Mill. The Machining Operation information is listed in the MOps browser as shown below.
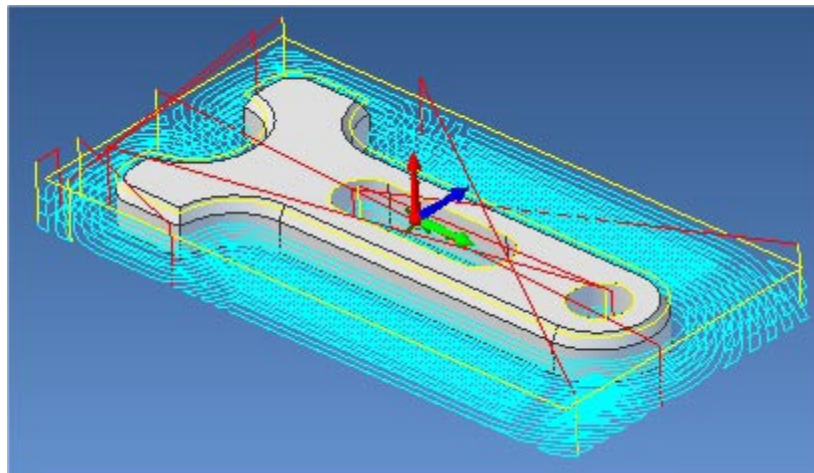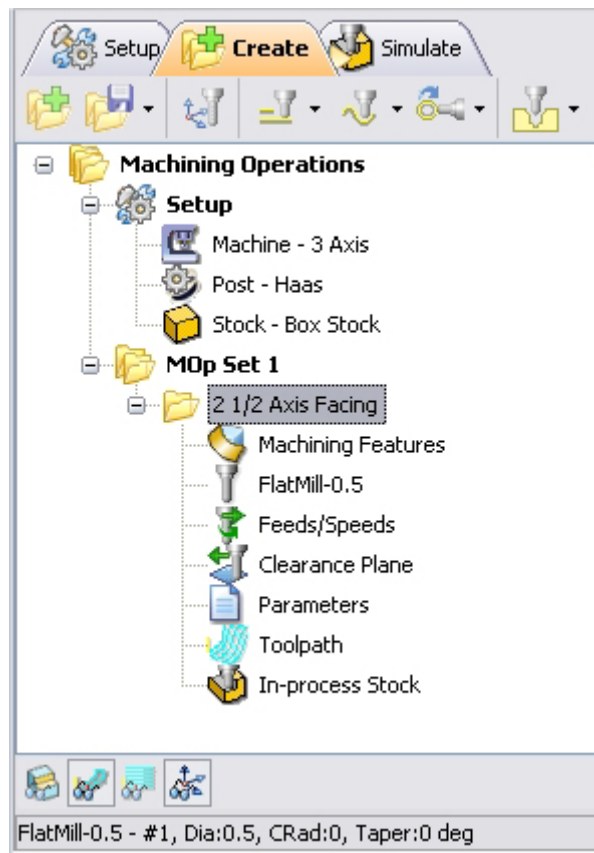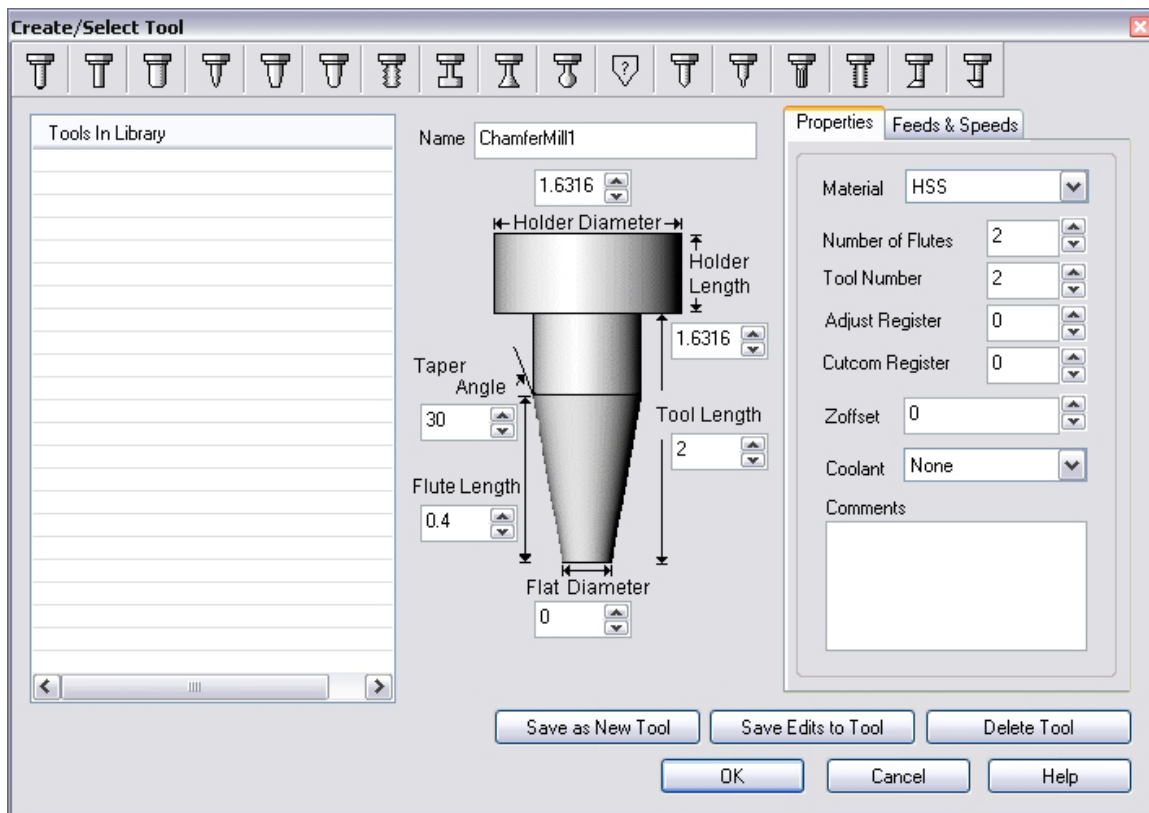
Create Tools

To machine the chamfer, 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 Chamfer.
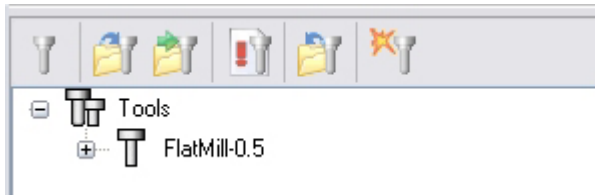
2. Set the tool name as **ChamferMill1**, Taper Angle = **30,** Flute Length = **1**, and Tool Length = **2**. Under the Properties tab, set Tool Number = **2**.

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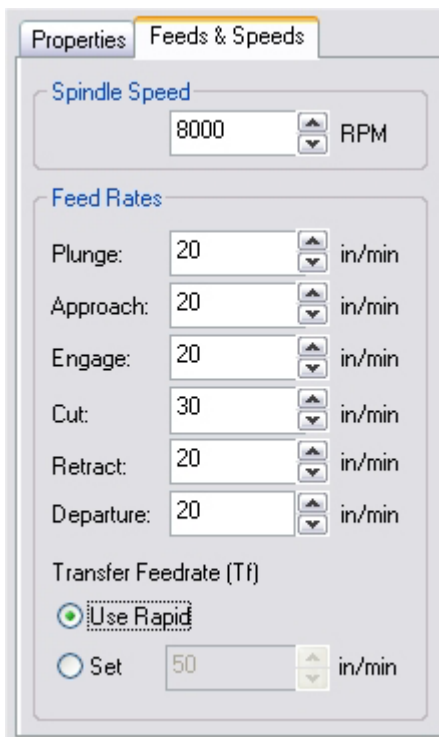
5. Click **Save as New Tool**. The tool is now created and listed under Tools in Library. Click OK to close the dialog.

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.
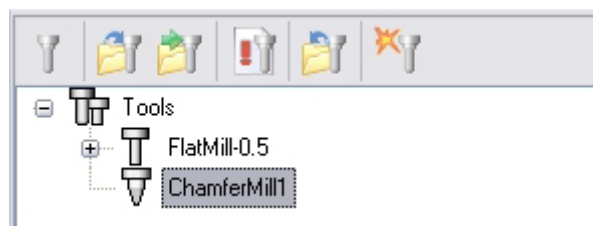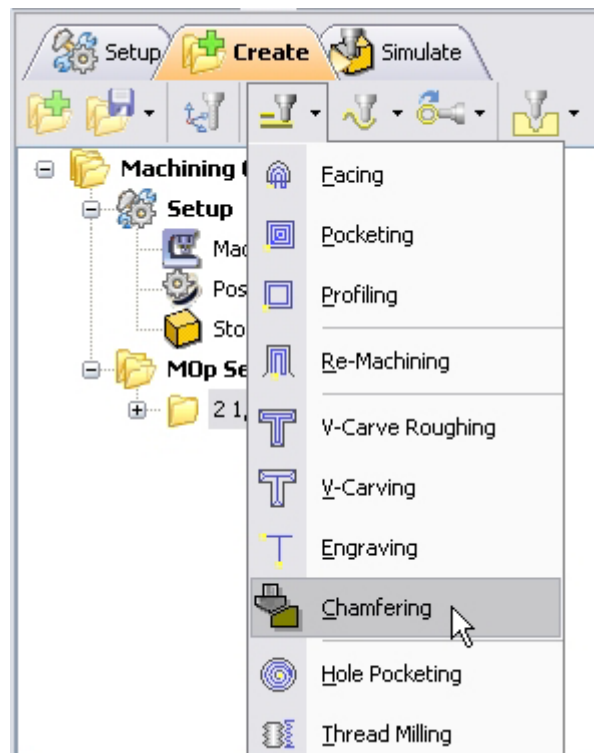

The created tools are now listed under the Tools browser.

Create Machining Operations

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

Chamfering
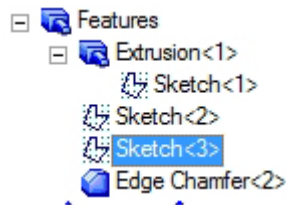
2. Select 2.5 Axis Mill and choose Chamfering.

This brings up the Chamfering Operations dialog. We will go over the steps for creating the toolpath.
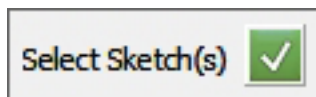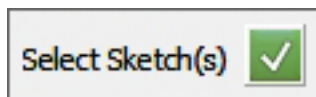
Select Machining Features/Regions

3. Go to the **Machining Features/ Regions** tab and click remove all if any Machining regions are listed.

4. Click **Select Containment Regions**.

5. Switch to Design Explorer and select **Sketch3**.

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.
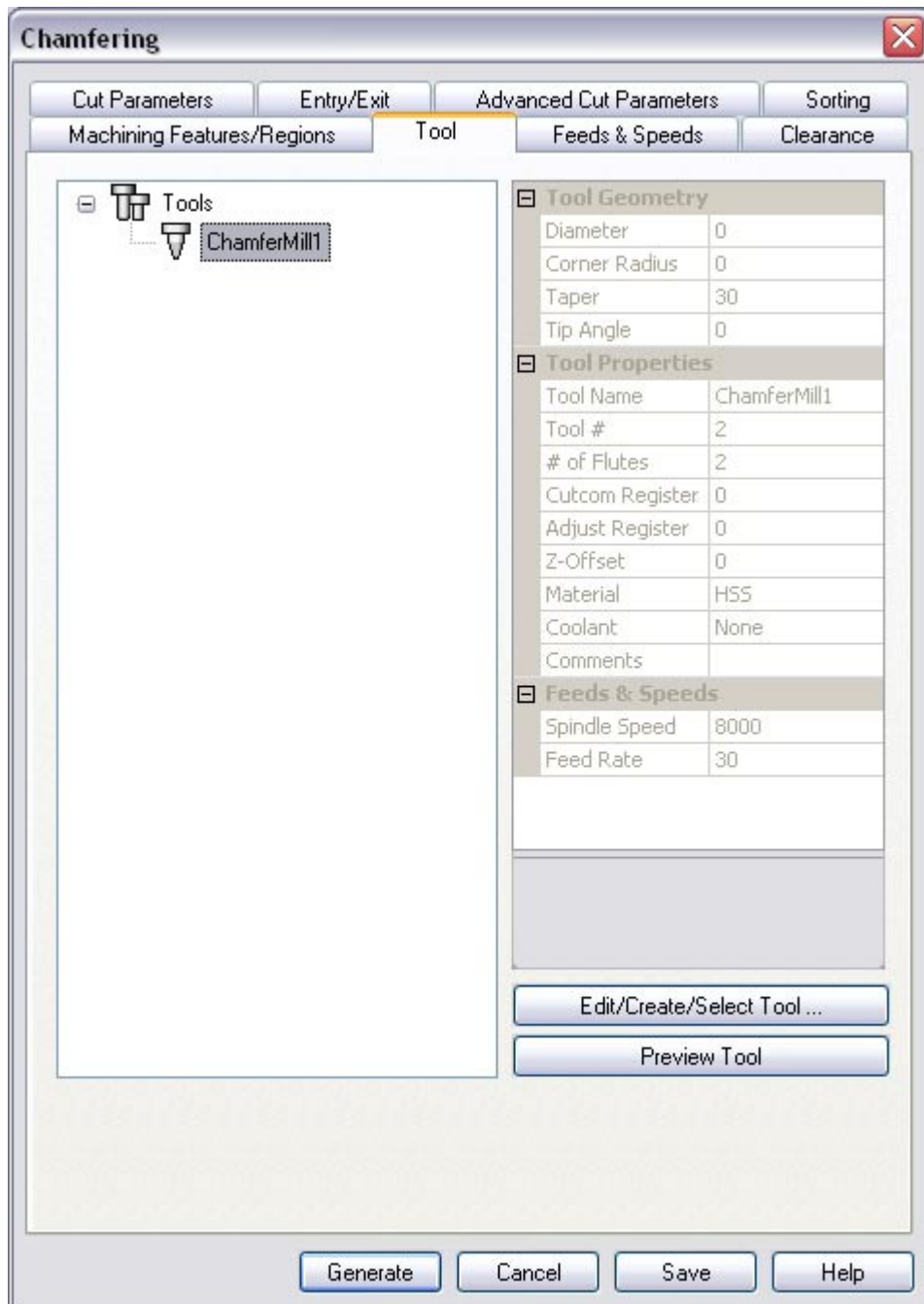
6. Click [Select Sketch(s) ✓] to complete the selection.

7. The selected region is now displayed under Machining Regions.
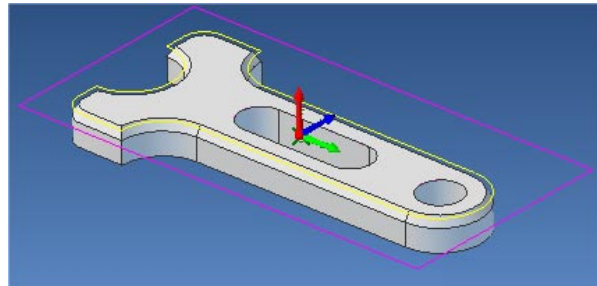
Selecting the Tool

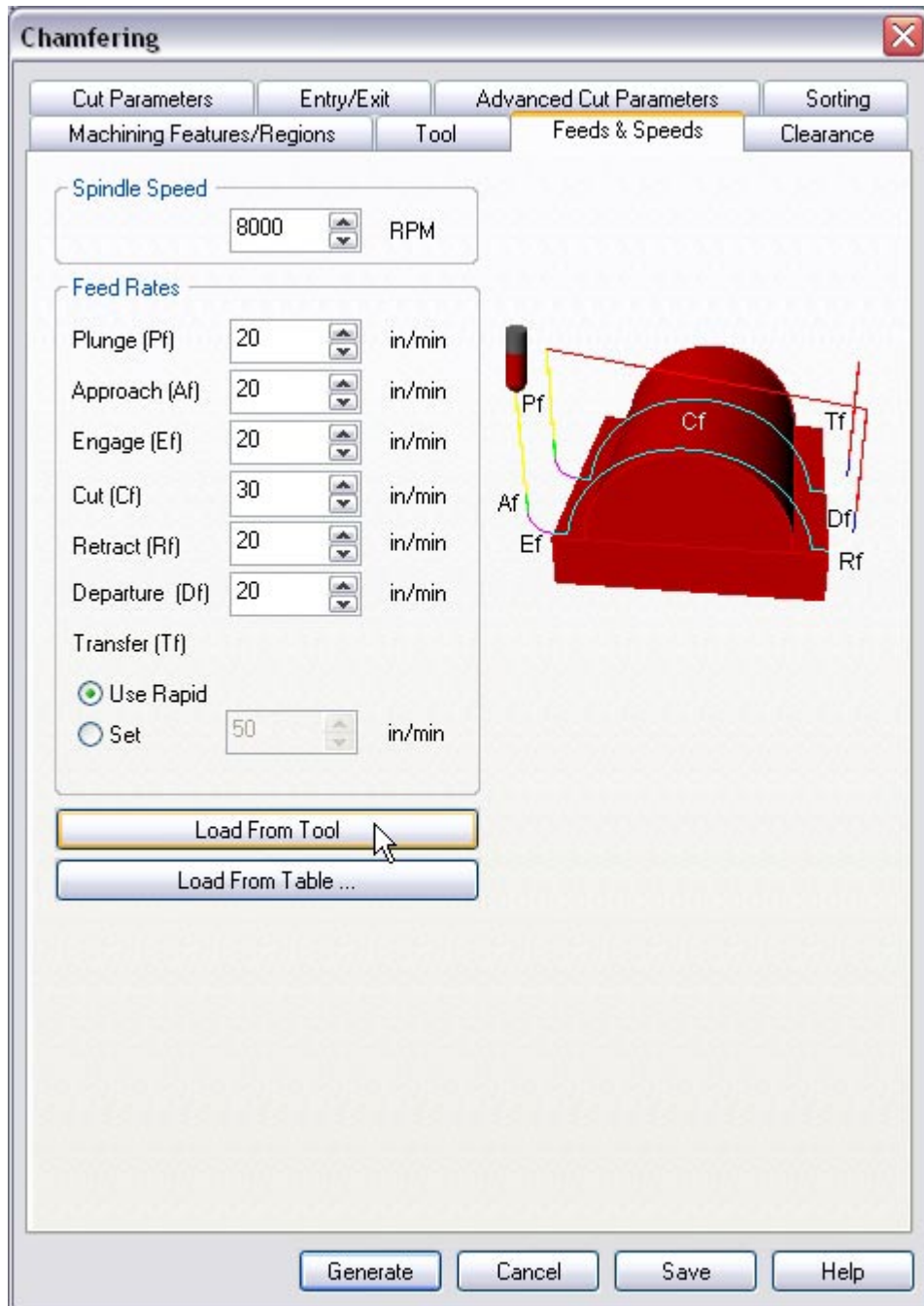8. Switch to the Tools tab inside the Chamfering operation.

9. Select ChamferMill1. ChamferMill1 is now selected as the active tool, and the Tool parameters are displayed to the right of the Tools window.
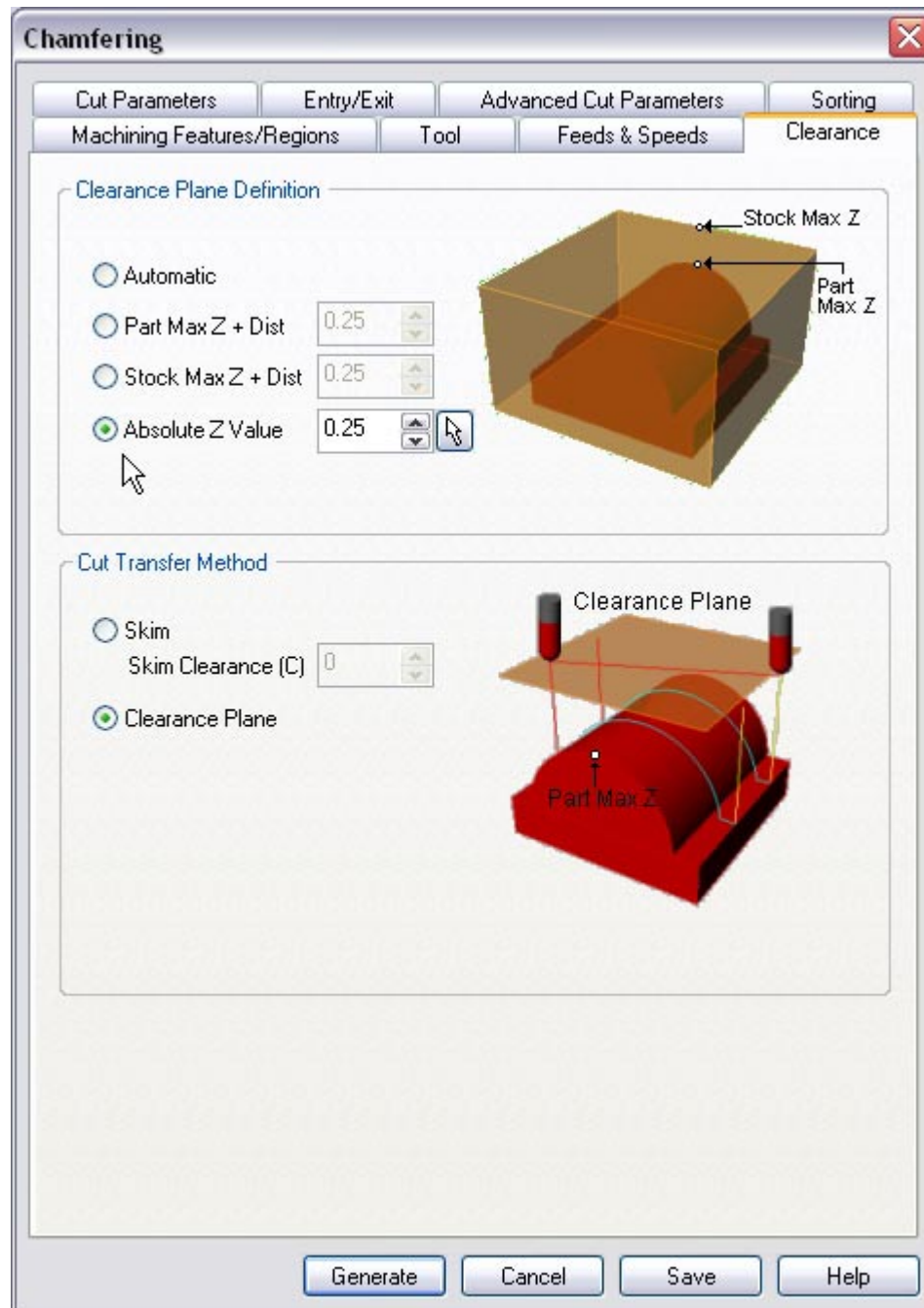
Set Feeds and Speeds

10. Click on the Feeds and Speeds tab.

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

Clearance Control

12. Switch to Clearance Tab.

13. 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

14. Switch to Cut Parameters tab.
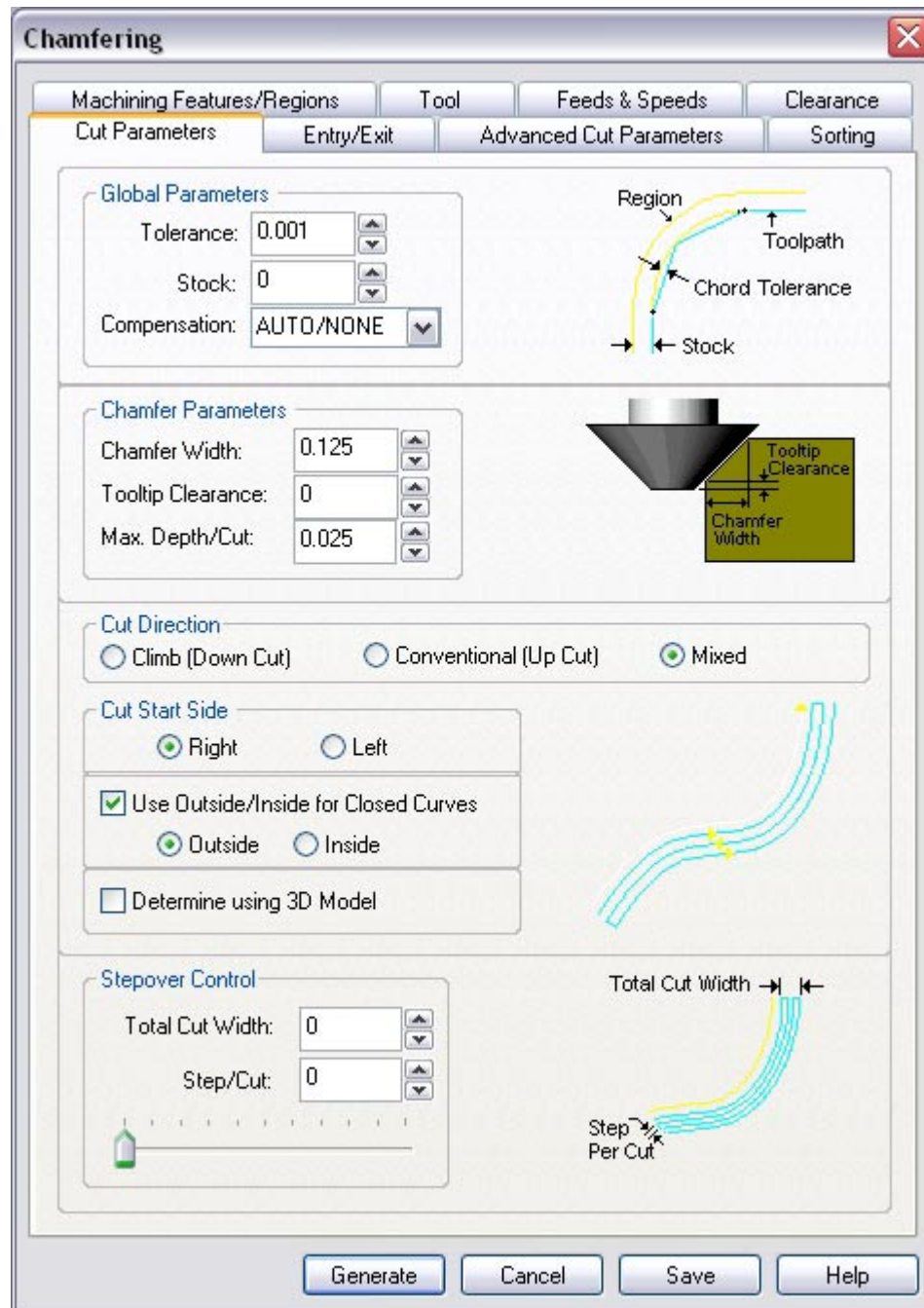
15. Set the Tolerance = **0.001**, Stock = **0**, Chamfer Parameters use Chamfer Width = **0.125**, Max Depth/Cut = **0.025** Cut Side = **Outside**. Click **Generate**. The Chamfering
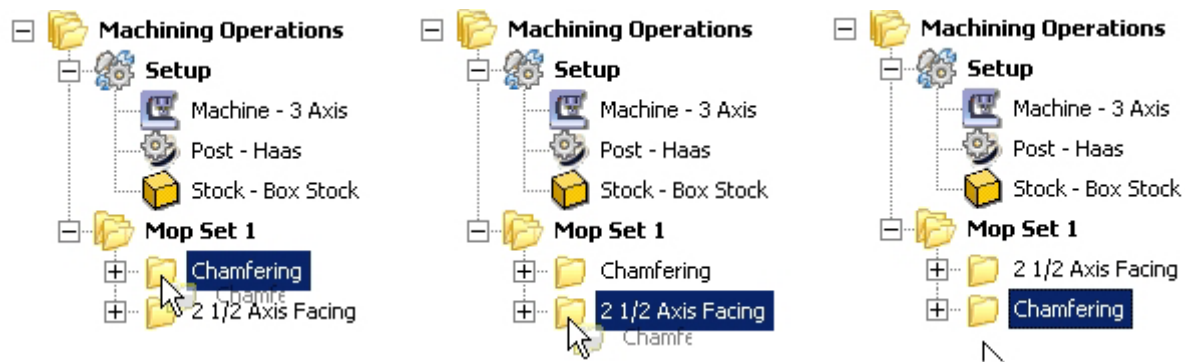
toolpath is now generated and the Operation is listed under the Alibre CAM MOps browser.

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

Reorder a Machining Operation

If the chamfer operation is created above the 2 ½ axis Facing operation, you can reorder the MOp using the steps below.

1. Minimize all MOps inside the Mop Set1.
2. Select the Chamfering MOp and drag it over the 2 ½ axis Facing MOp. This would move the Chamfer MOp below the Facing MOp.
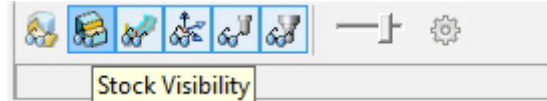

Chamfer toolpath is now generated and is displayed below.
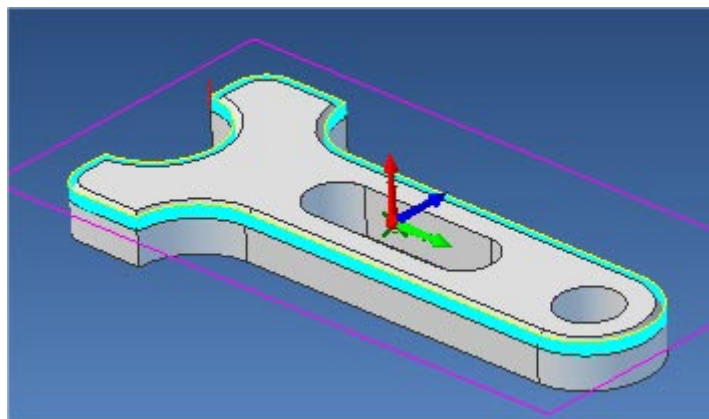
Simulate Toolpath

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

1. Select the 2 ½ Axis Facing Operation and click [ ] to launch the Alibre CAM Stock Simulation window.

2. After simulating the facing operation, close the Stock Simulation window, select the

   Chamfering Operation and click [ ] to launch the Alibre CAM Stock Simulation window.

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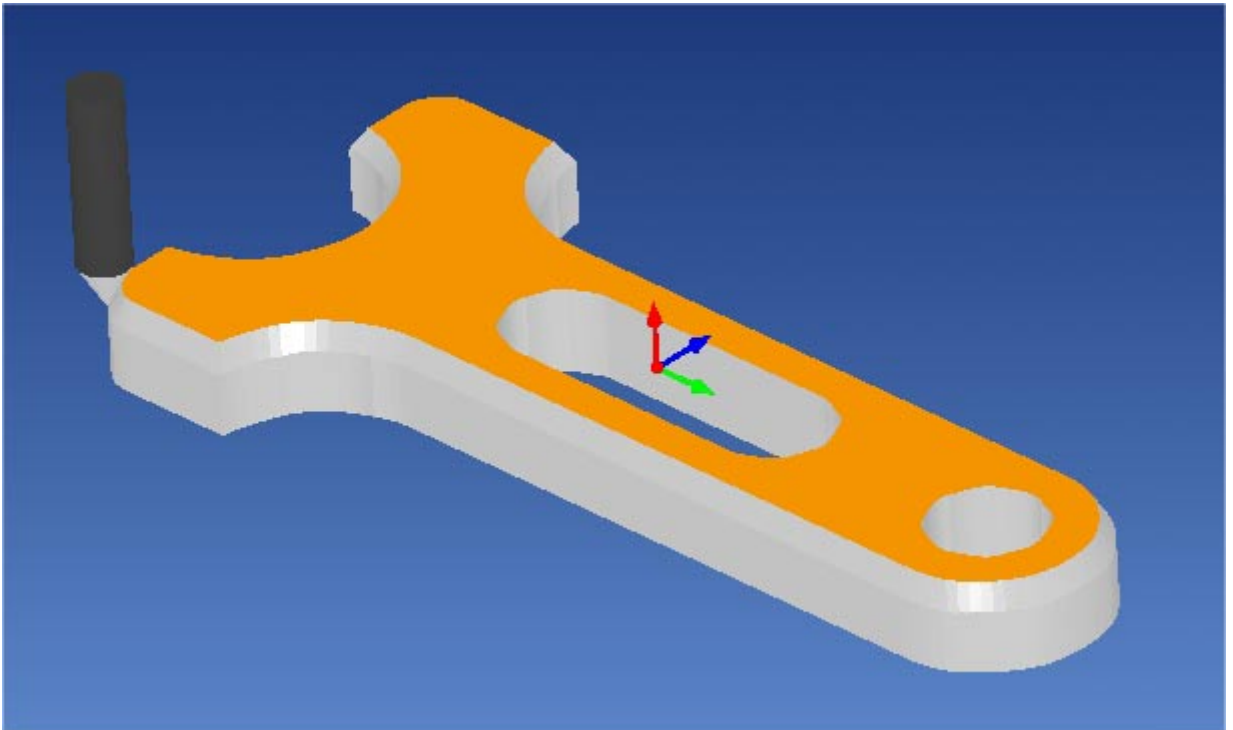

Note: You can pause/stop, step, and skip the simulation using the controls available on the Simulation window. To control the simulation speed, select Simulation Settings from the Stock Simulation window.

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

Post Processing

1. Select Machining Operations from the Create Operations tab and right click and select post process.
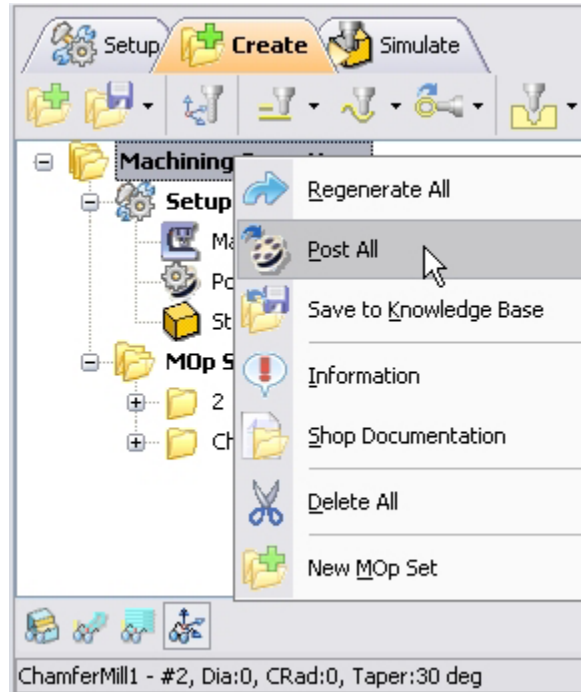
2. Specify the File Name as **Chamfering.nc** and click Save.

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