Tutorial 1: Machining a Gasket
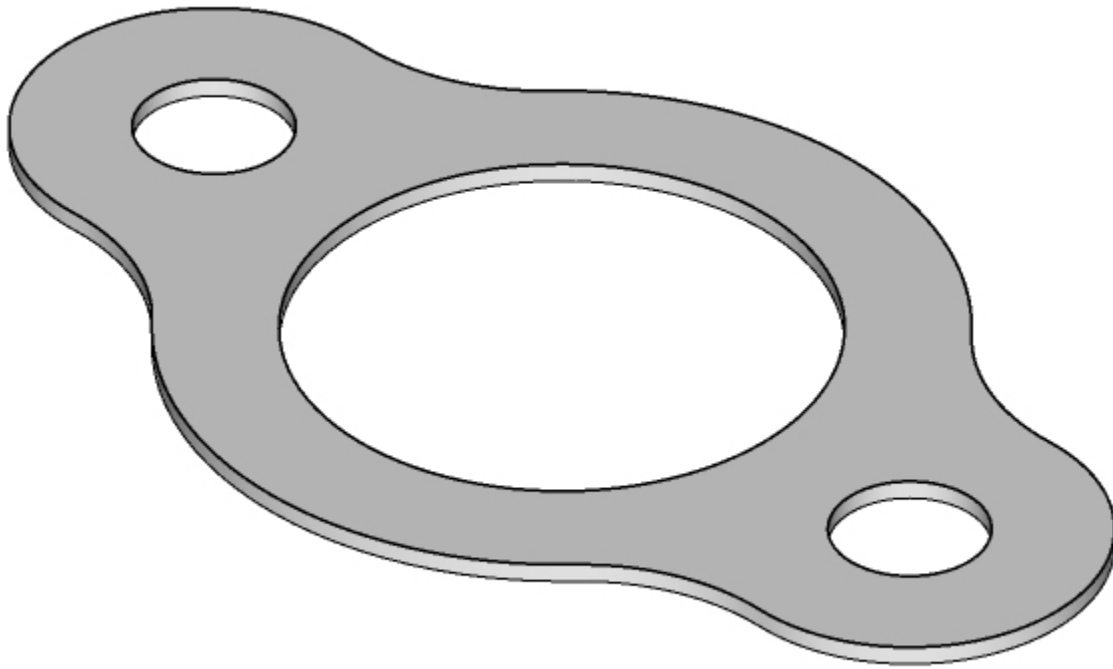
Using 2 ½ Axis Profiling

Introduction

This tutorial will illustrate machining of a simple prismatic part such as this gasket using 2-1/2 milling operations. Even though we have created a 3-D representation of the gasket, it will be seen later on that we can machine this using just 2-D curves (sketches).

The reason we are able to do this is because of the prismatic nature of this model.

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 Gasket

- We will machine the Gasket by using a 2-½ axis machining operation called Profiling.
- The part will be machined out of a 8 ½ x 5 x ¼ inch poplar wood sheet using a ½ inch Flat End Mill.
- 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 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.

New Part: Creates a new part. Though you can create your part with any units, Alibre CAM only accepts parts for which the units are in Inches or Millimeters. You can easily switch to these units when necessary by going to File > Properties > Units and selecting the applicable item under the Display Units section.
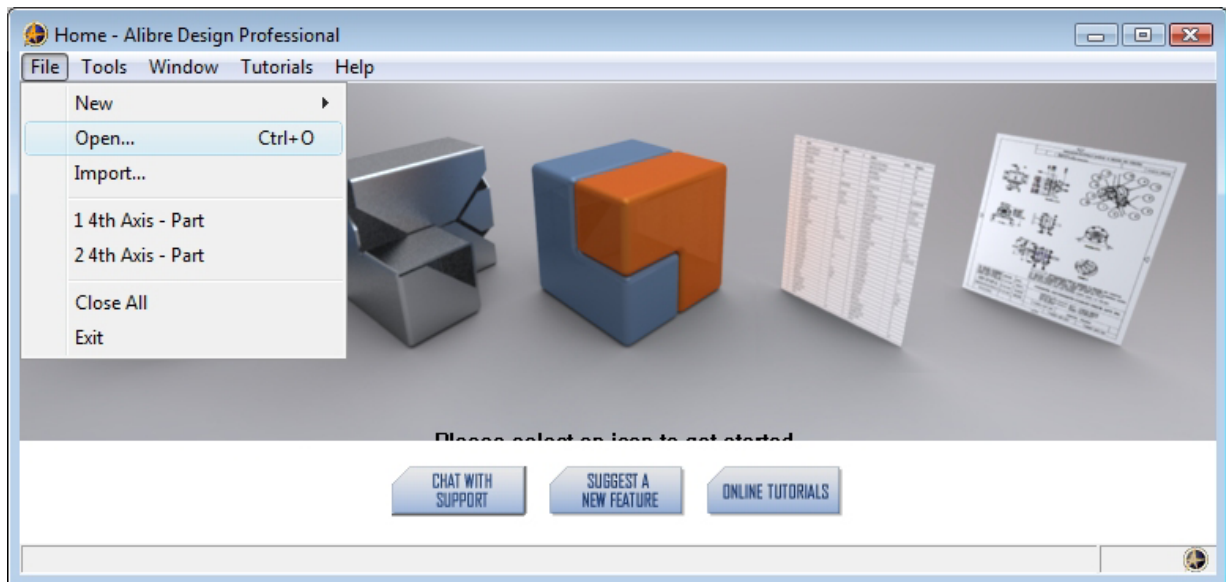**Open:** Loads Alibre Design parts or assemblies
**Import:** Imports files created outside of Alibre Design.
Save: Saves the current file.
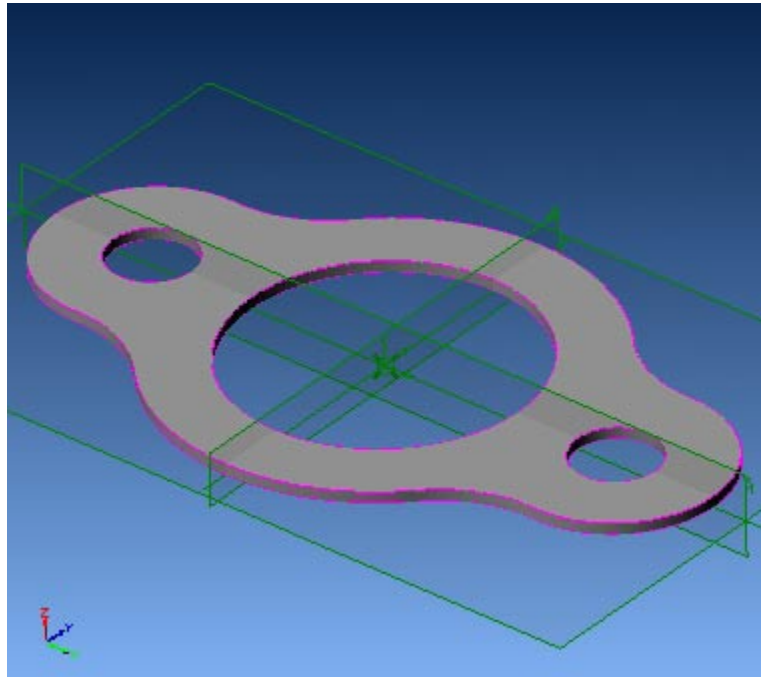
Getting Started with Alibre CAM

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 **Gasket.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)
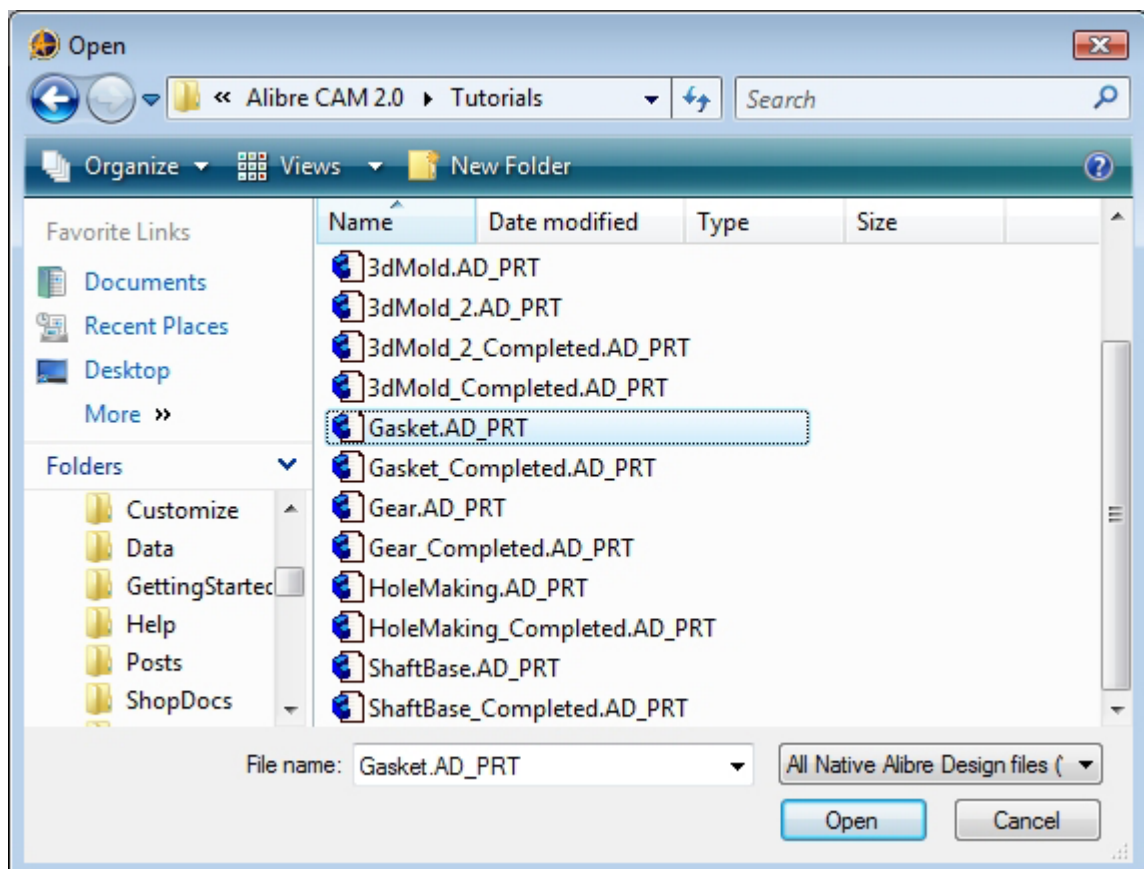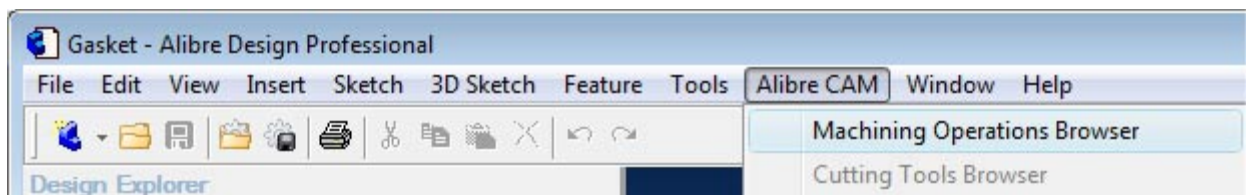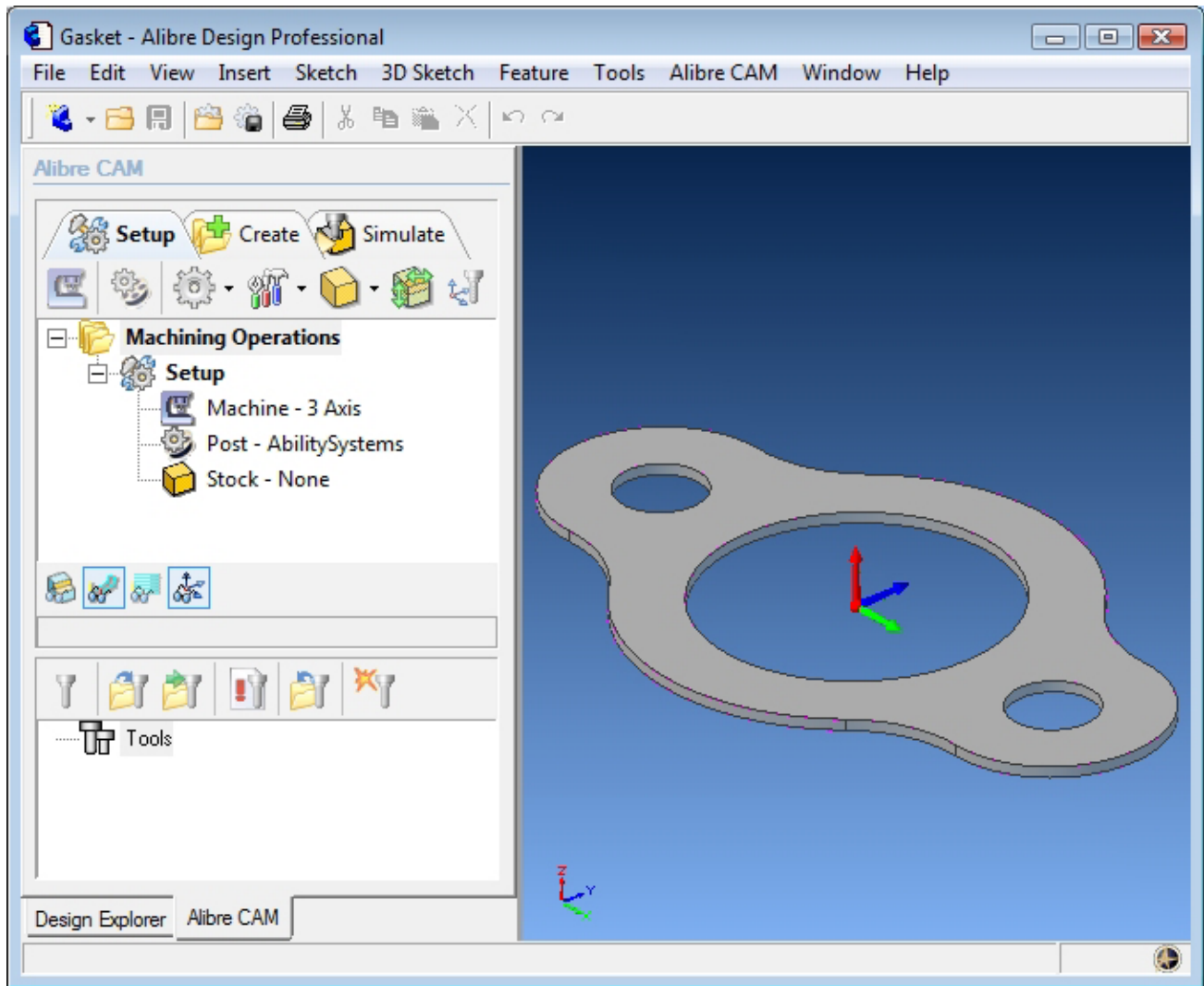
3. The loaded part appears as shown below.

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.

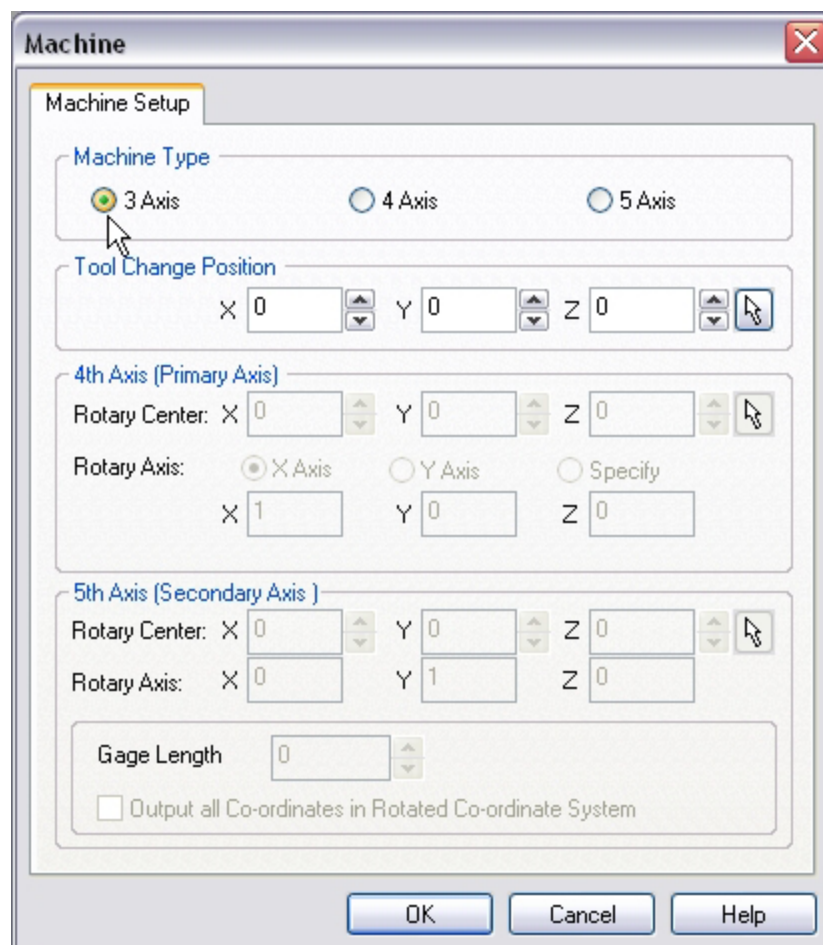## Preparing the Part for Machining

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

Setup Machine

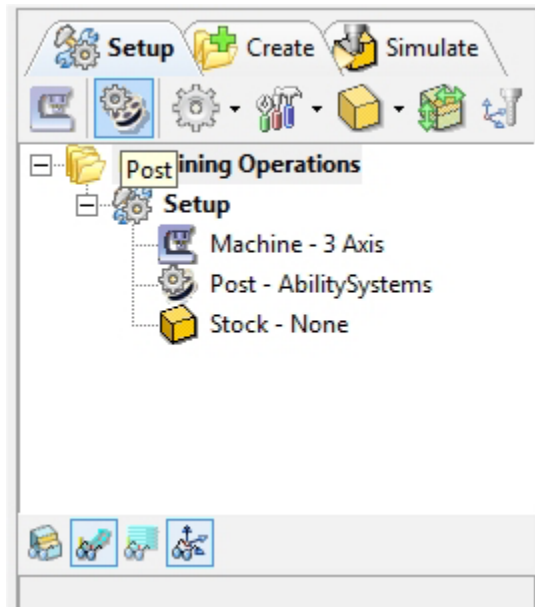1. Click on the Setup tab located under the MOPs Browser and select Setup Machine
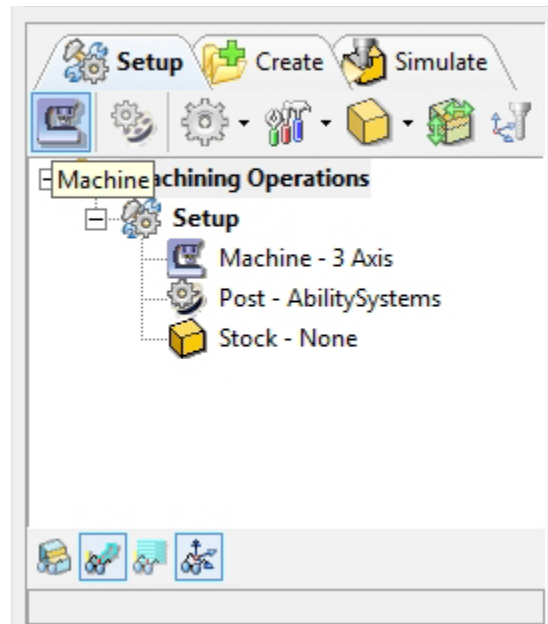
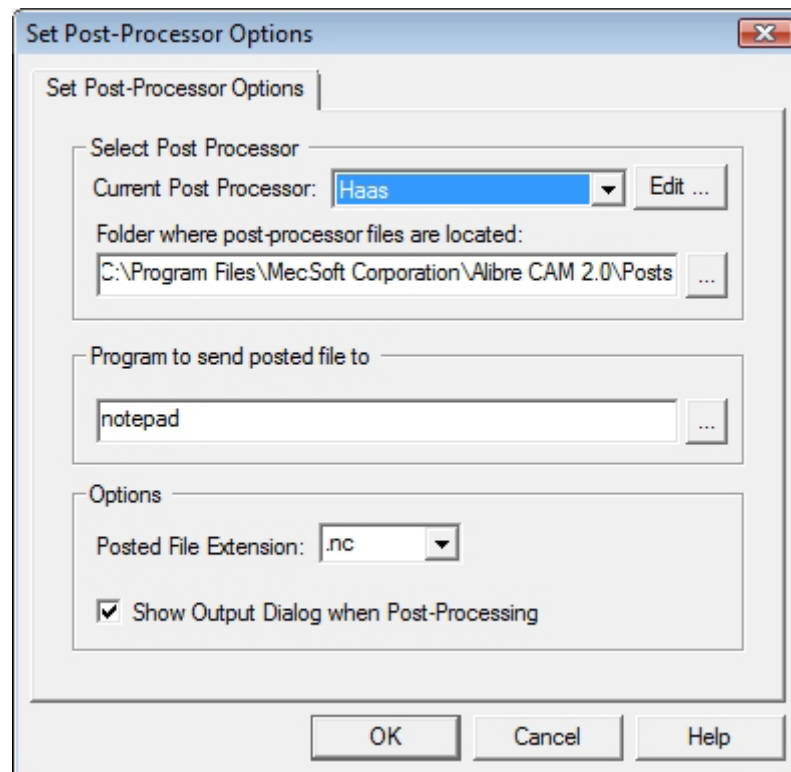2. Set the Machine Type to 3 Axis. Click Ok Close the Setup Machine Dialog.

Set Post Options

3. Select Set Post Options from the Setup tab.

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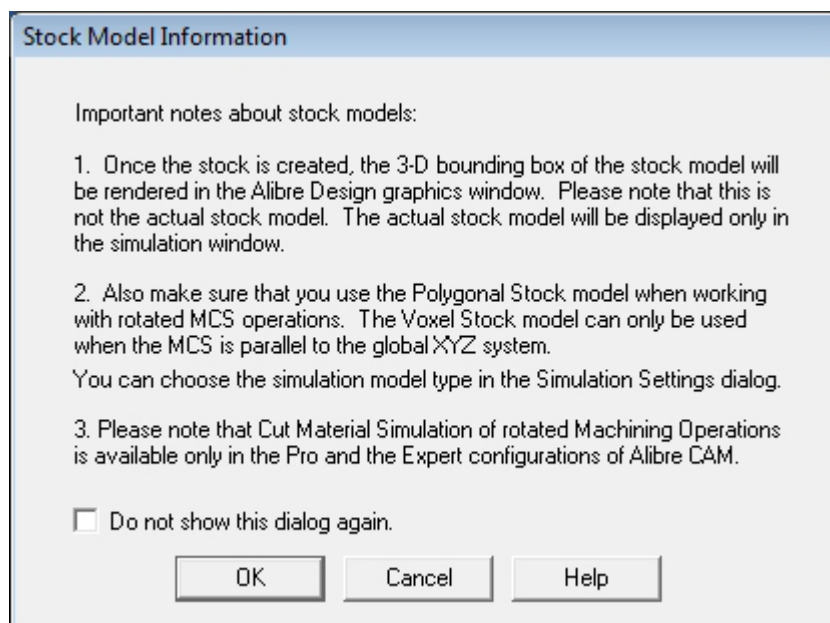
Note: By default post processor files are located under
C:\Program Files\MecSoft Corporation\Alibre CAM 2.0\Posts
The program to send the posted output is set to notepad. This would output the G code to a notepad.

Create/Load Stock

5. Select Create / Load Stock from the Setup tab and choose Box Stock
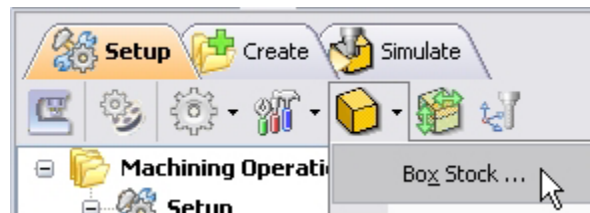
The stock model information dialog may be displayed when a stock geometry is created.

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.
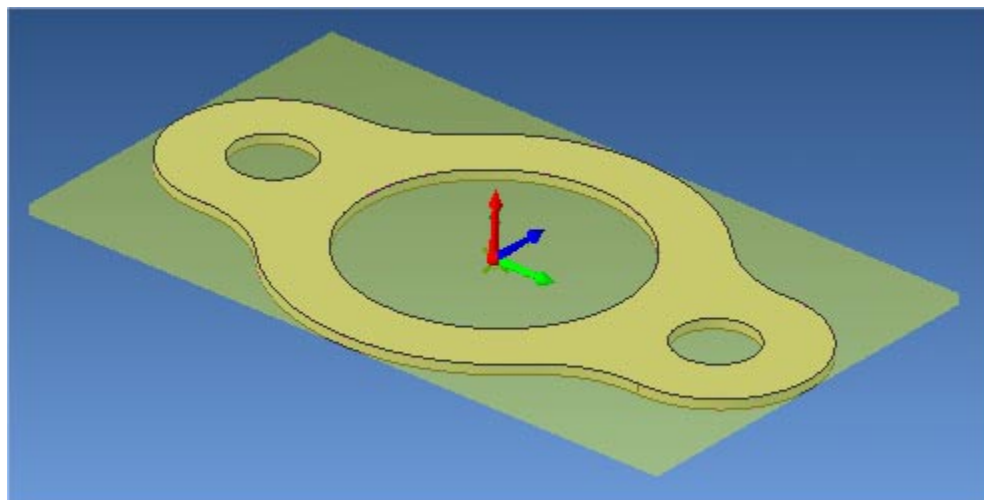
6. This brings up the Box Stock parameters. Set the Length (L) = 8.50, Width W = 5.00 and Height (H) = 0.125. Leave the other parameters as default and Click OK.
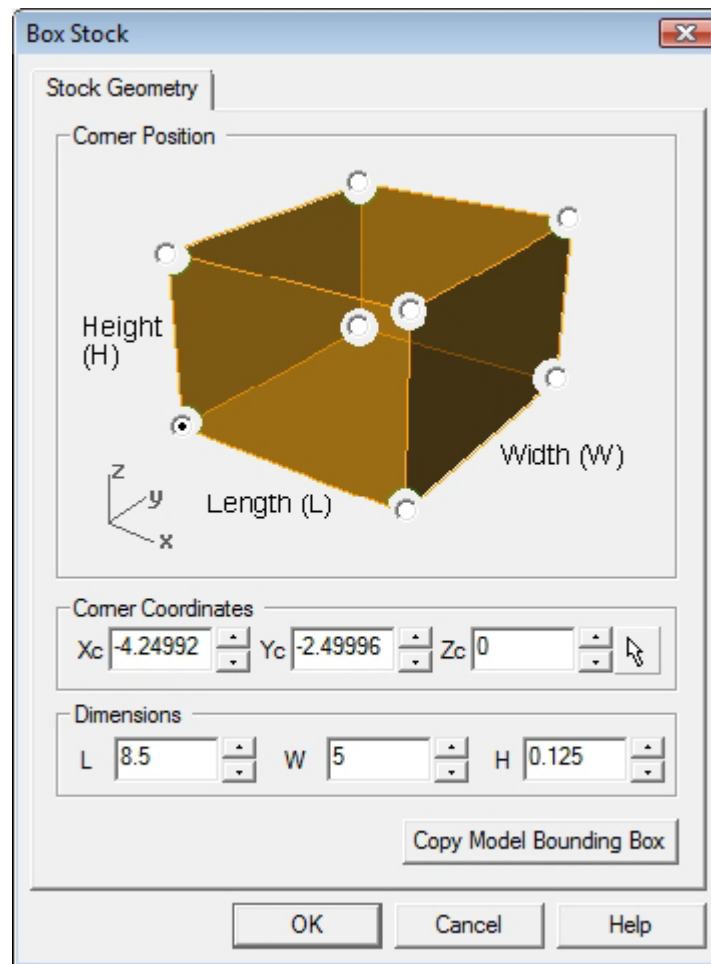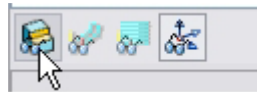
7. The stock geometry is now created, and a semi-transparent stock box is displayed on top of the part geometry.
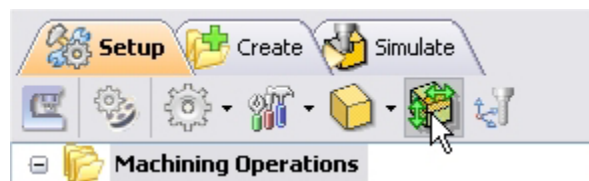
Note: If the stock geometry is not visible, click Stock Visibility located on the status bar in the MOPs browser.
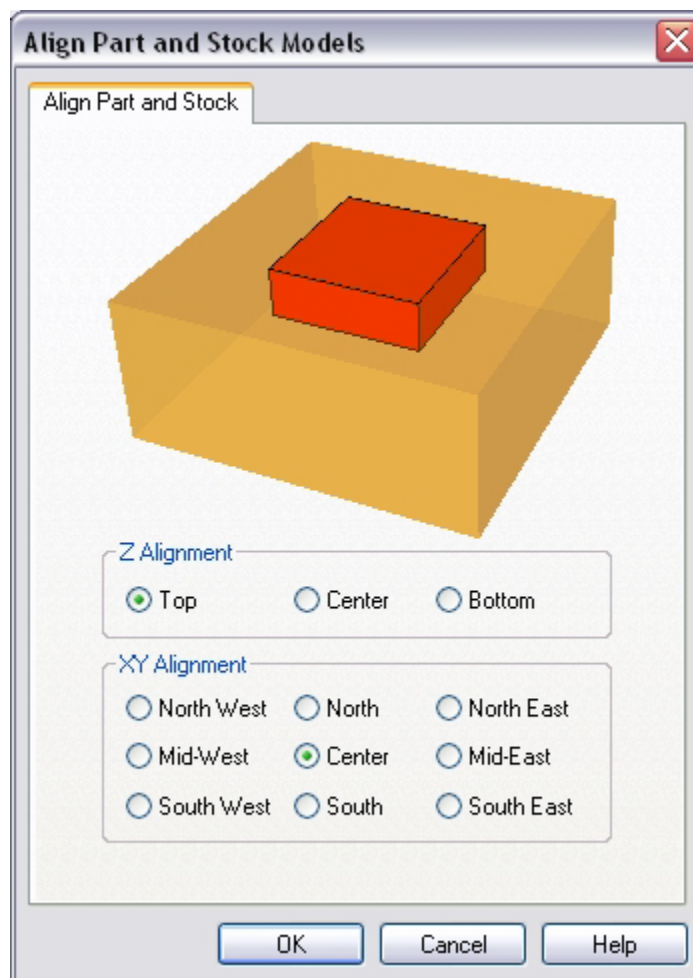

Align Part and Stock

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).

8. Select Align Part and Stock from the Setup tab

9. 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 know as machine zero or tool touch off point) for the part/stock geometry.

The MCS is represented as a triad with 3 arrows, the Green arrow indicating X axis, Blue the Y axis and Red the Z axis. It is always a good practice to set the machine zero before generating a toolpath.
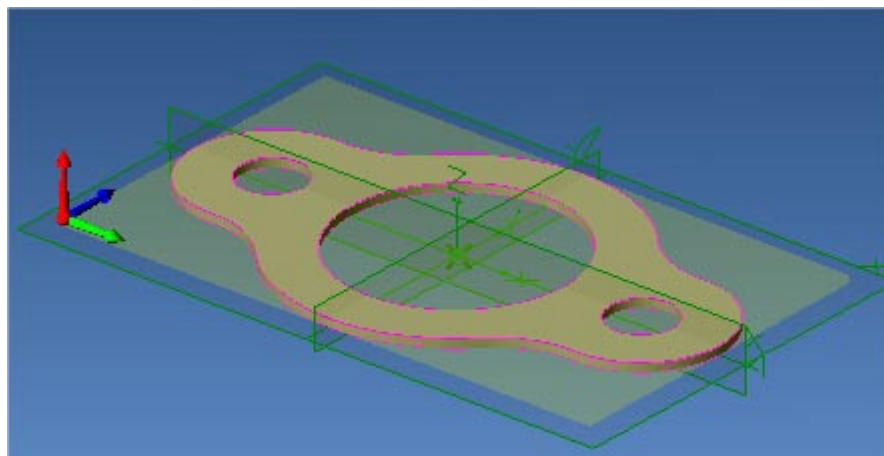
1. Select Set MCS from the Setup tab.

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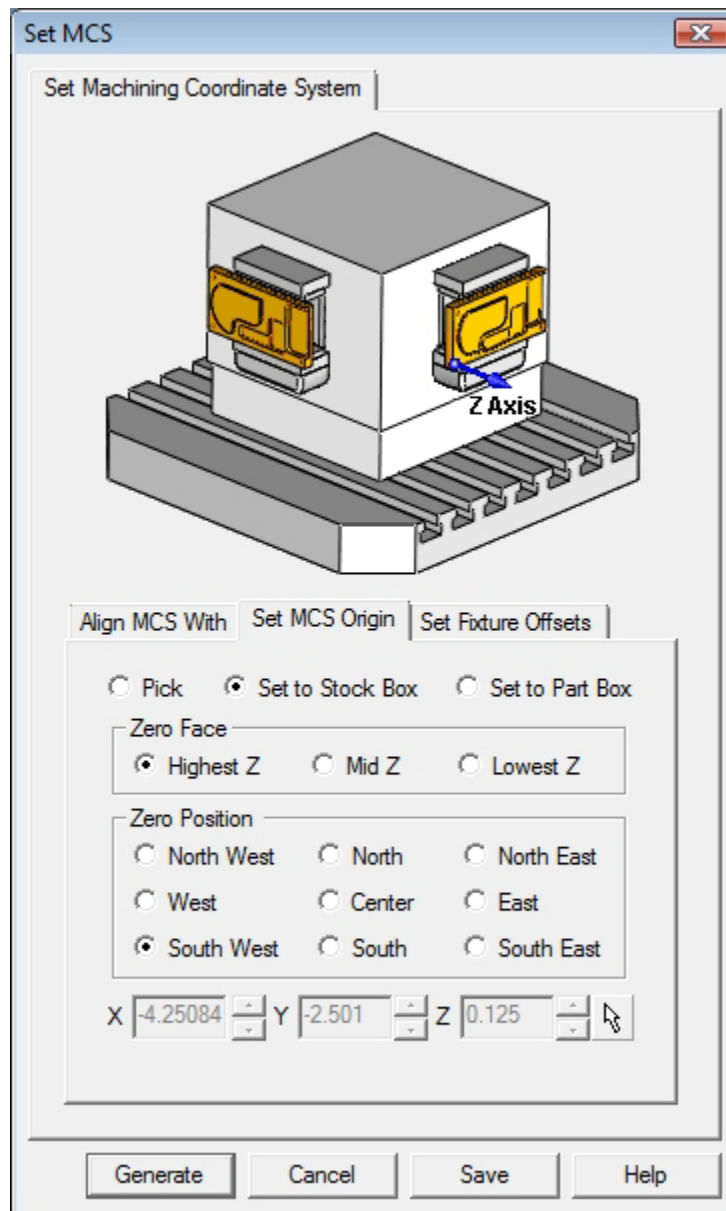
(This sets the machine home to Southwest corner and top of the stock material).

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 Gasket-Rev1.

Defining Cutting Tools

Alibre CAM supports numerous types of milling and drilling tools. For each tool you can specify standard APT parameters: diameter, corner radius, flute length and tool length. Flute length is the cutting length, and Tool Length is the total length of the tool up to the tool holder. All defined tools can be viewed in the Tools tab of the Browser. You can also save a set of tools to an external folder that can be loaded in other files.
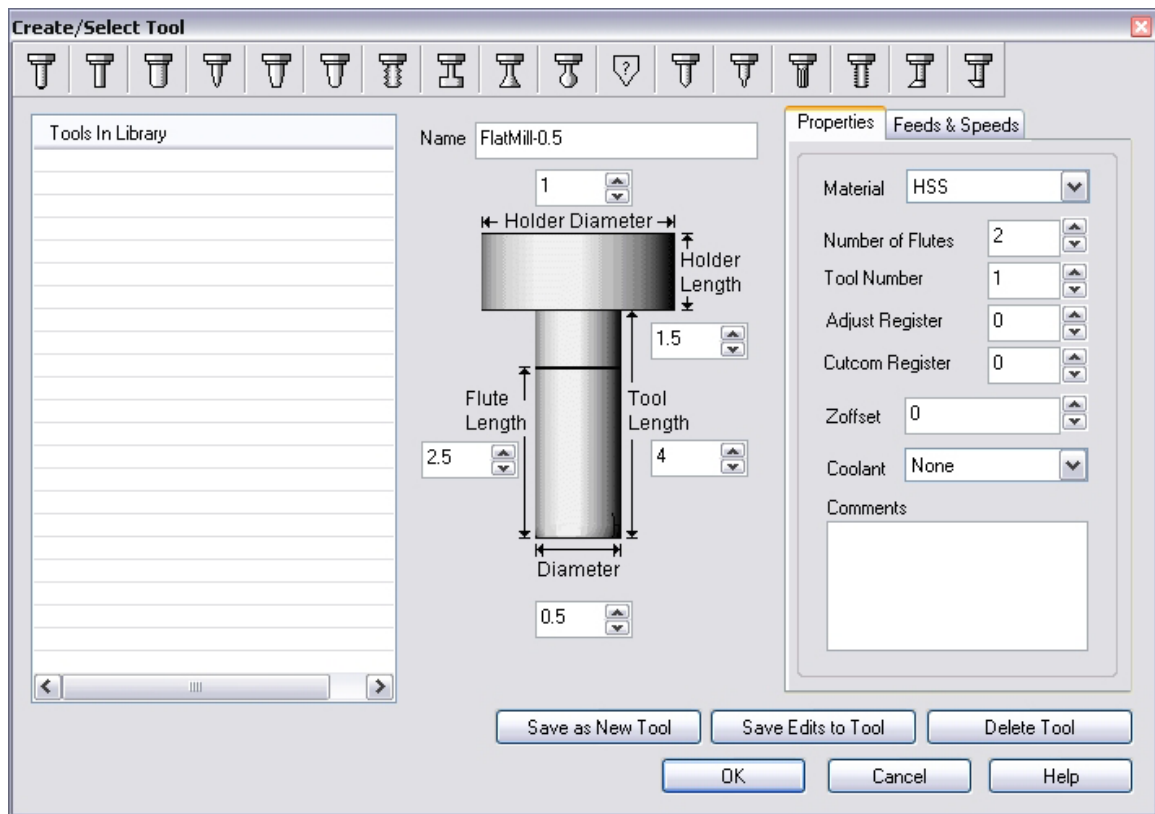
Create Tools

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**.
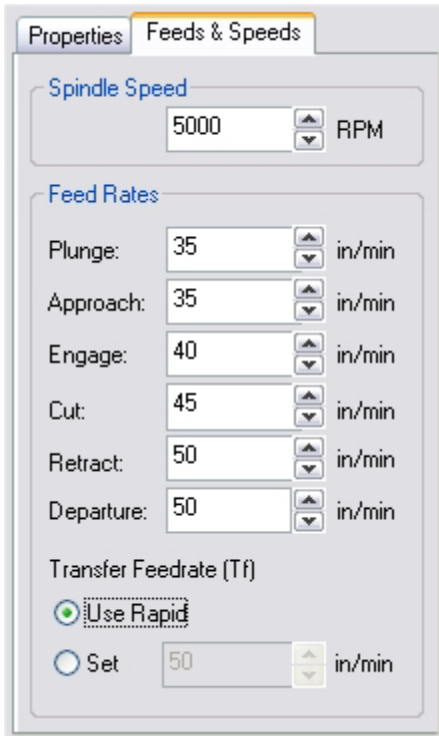
Setting Feeds and Speeds

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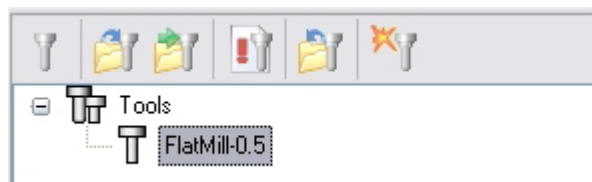
The created tools are now listed under the Alibre CAM -Tools browser.

Create Machining Operations

"MOps" stands for <u>M</u>achining <u>O</u>peration<u>s</u>. All machining operations you create are listed here, in order of creation.
We will machine the Gasket in 2 steps using the 2 ½ Axis Profiling. Inner areas will be profiled first and then the outer area.
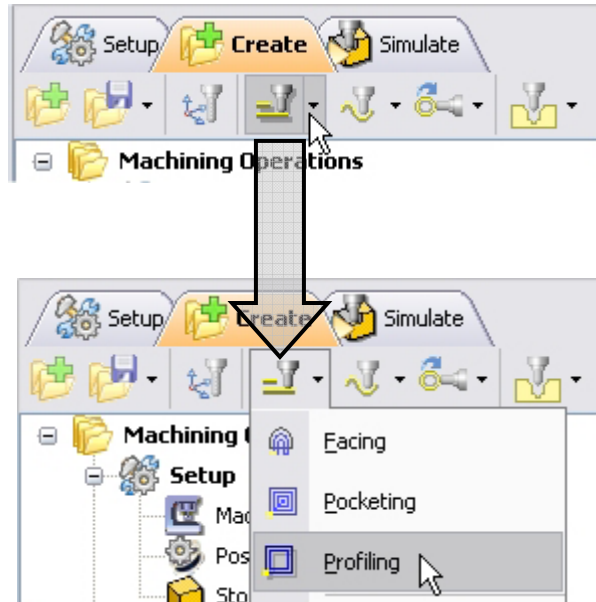
In this process we will create a 2.5 axis profiling operation.

1. Switch to the Create Operations tab in Alibre CAM-Mops browser.
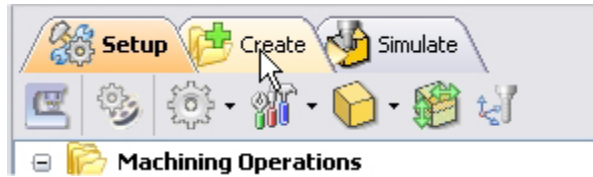
2 ½ Axis Profiling

2. Select 2 ½ Axis Milling and choose **Profiling**

This brings up the 2 ½ Axis Profiling Operations dialog. We will go over the steps for creating the profile operations for the inner features of the Gasket.

Select Machining Features/Regions

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.

3. Go to the **Machining Features/ Regions** tab and click **Select Containment Regions.** The Profiling operation dialog is now minimized and allows selection of the sketch geometry.
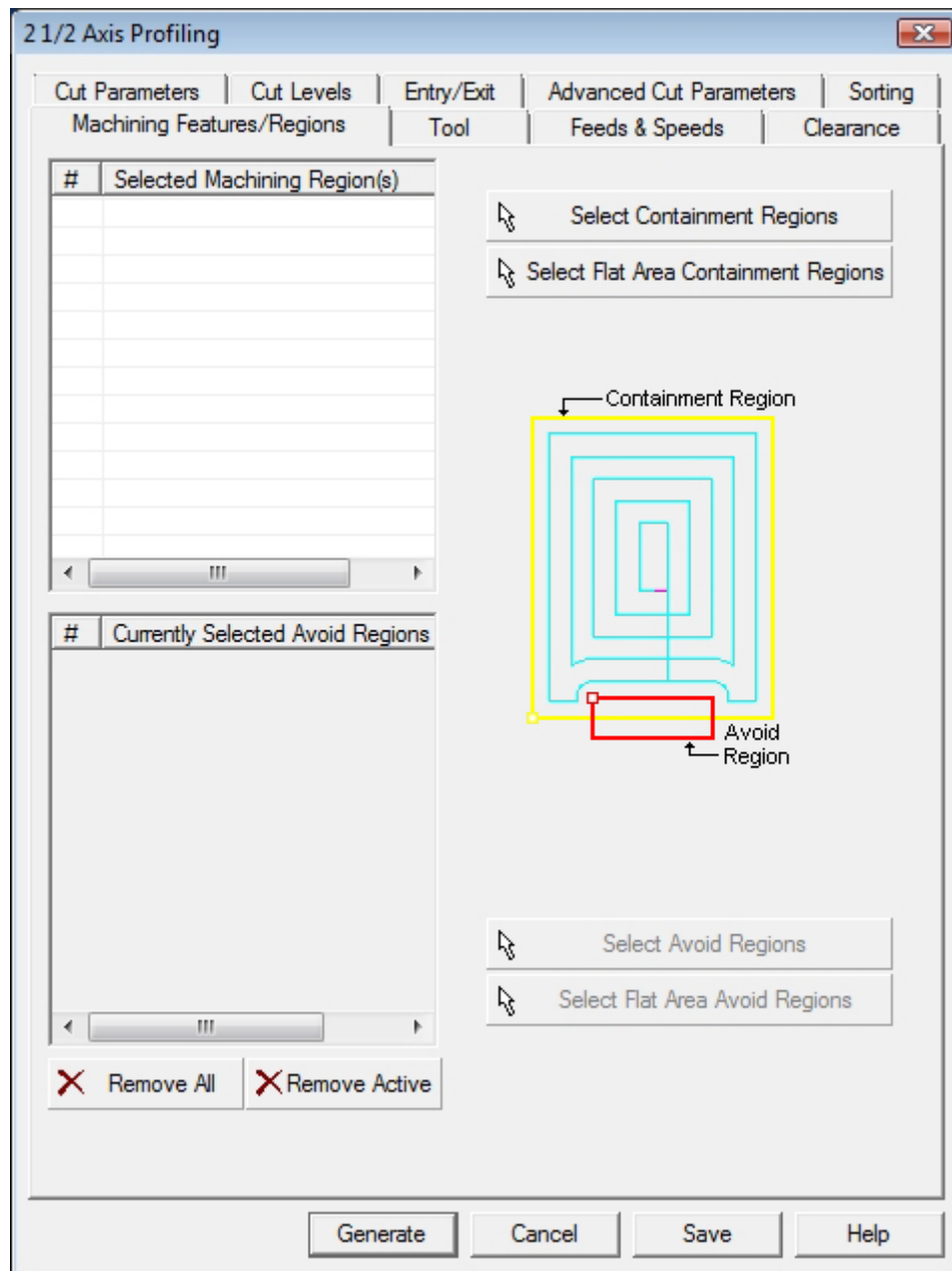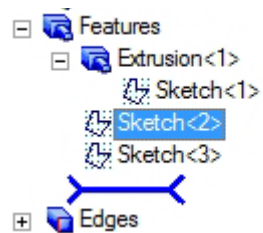
We will not select the sketch to machine the 3 inner circles on the part.
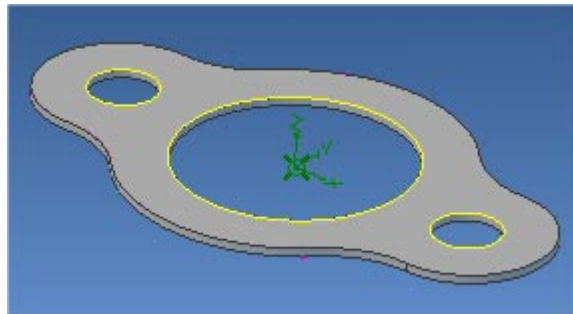
4. Switch to Design Explorer and select **Sketch2**.

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 **Select Sketch(s)** ✓ to complete the selection.

The 2 ½ Axis Profiling operation dialog comes back up displaying the selected regions (sketches). The selected regions are also highlighted on the part.
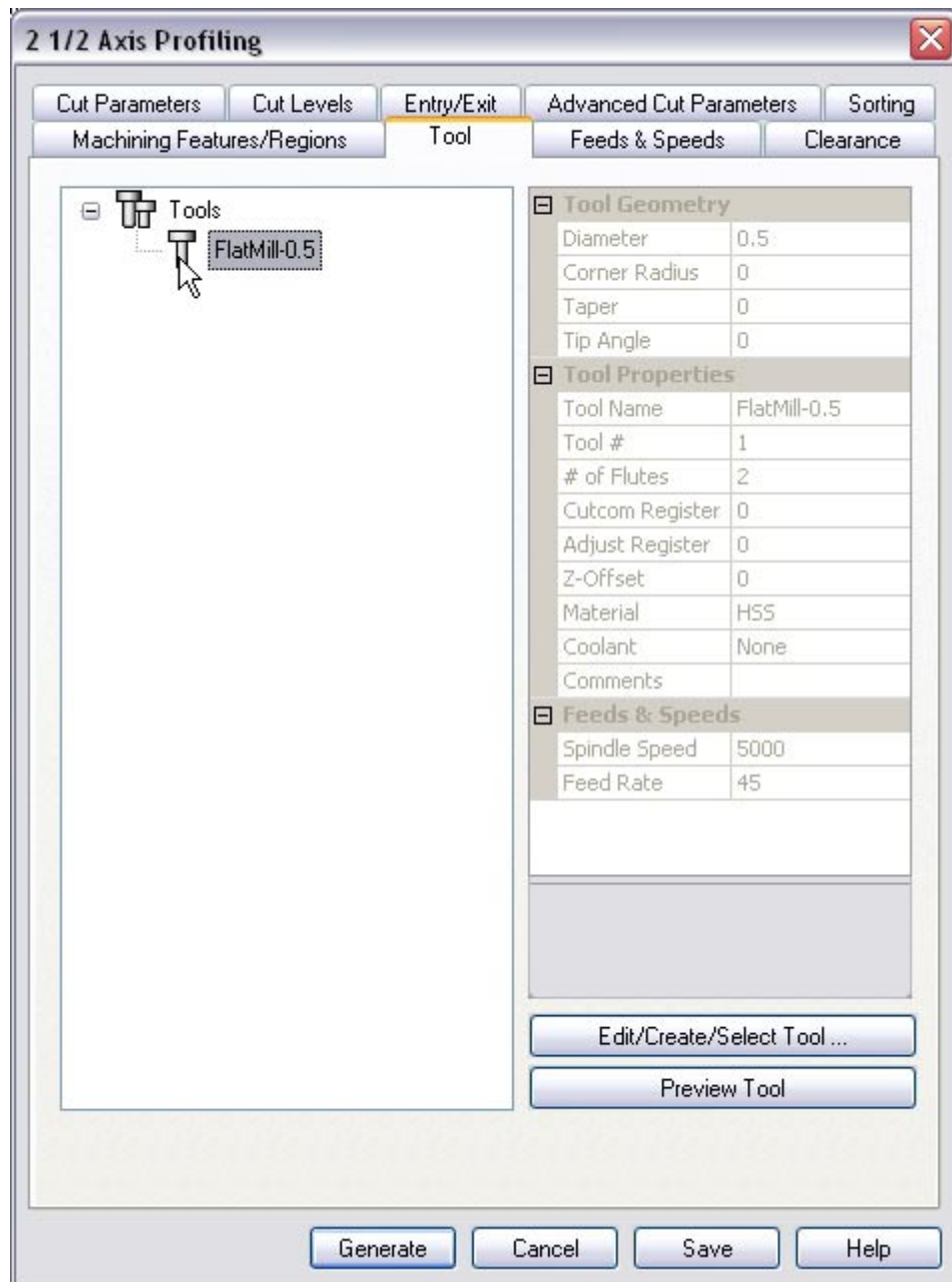

Selecting the Tool

6. Switch to the Tools tab inside the 2 ½ Axis Profiling operation.
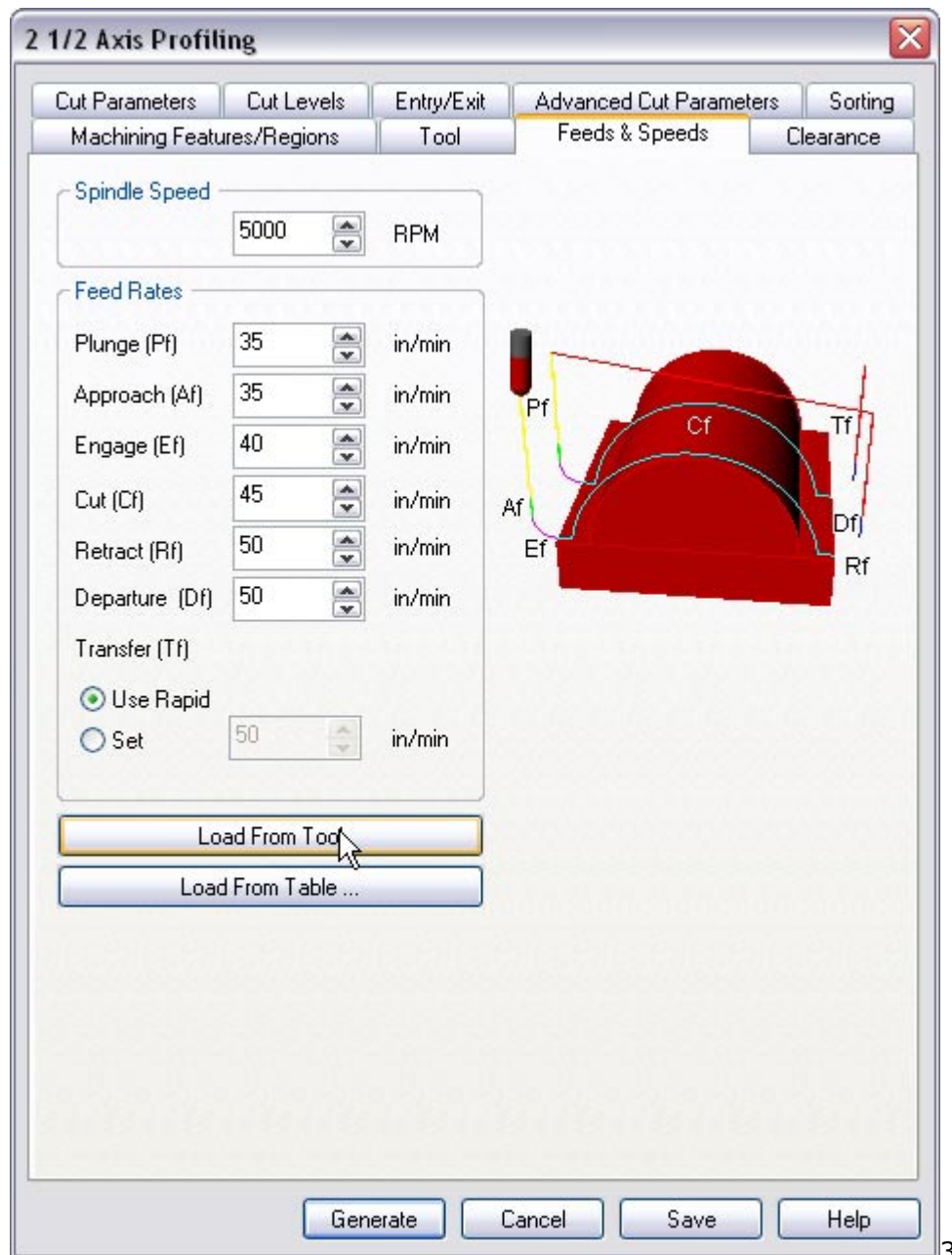
7. Select the FlatMill-0.5. The 0.5" Flat End mill is now selected as the active tool and the Tool parameters are displayed to the right of the Tools window.

Set Feeds and Speeds

8. Click on the Feeds and Speeds tab.

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

Clearance Control

The clearance plane is a horizontal plane (set Z level) from which the approach motions start and retract motions end. After retracting, the tool moves rapidly along this plane to the position of the next engage. This plane is typically a certain safe distance above the part geometry.
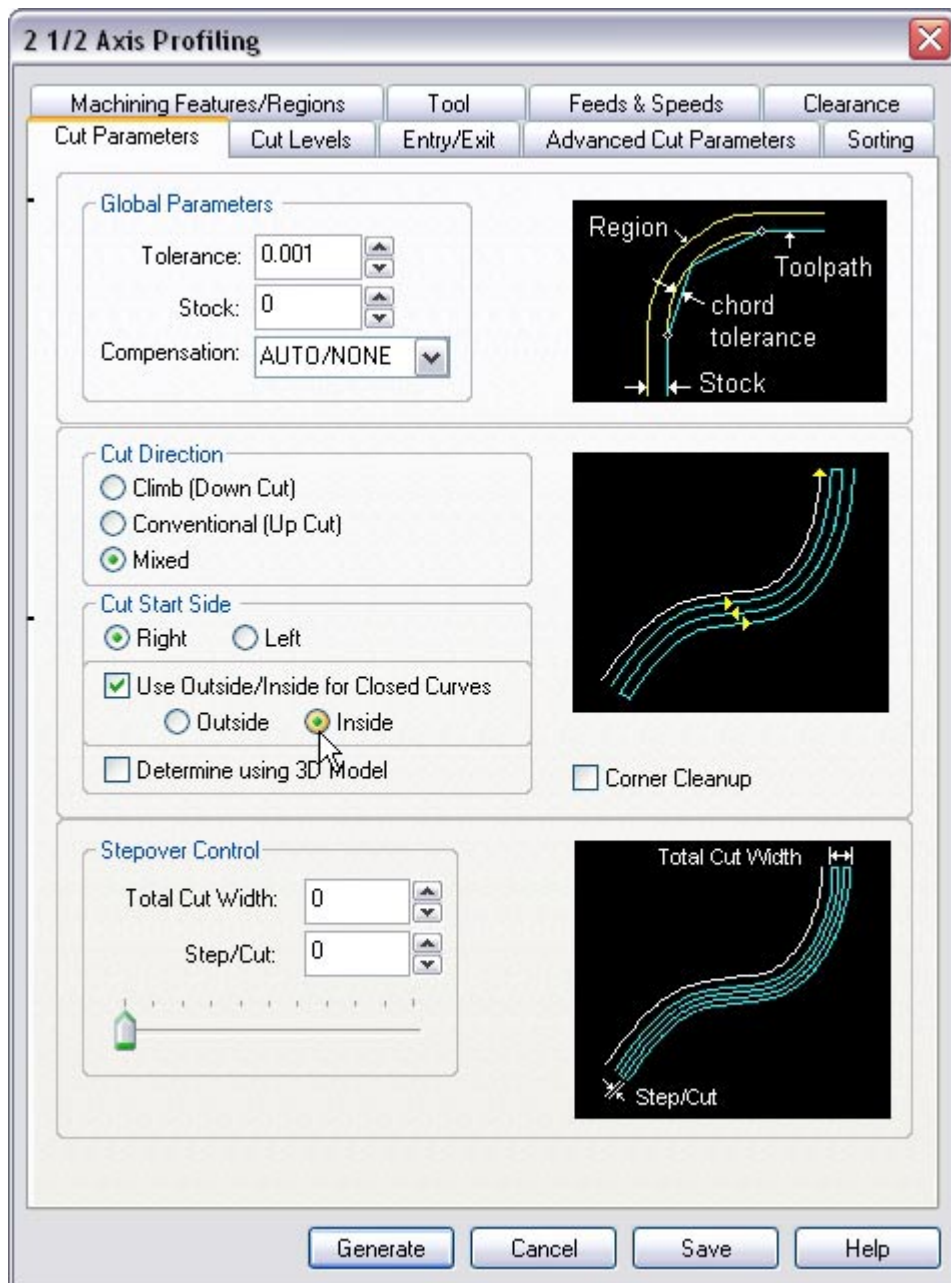
10. Switch to Clearance Tab.
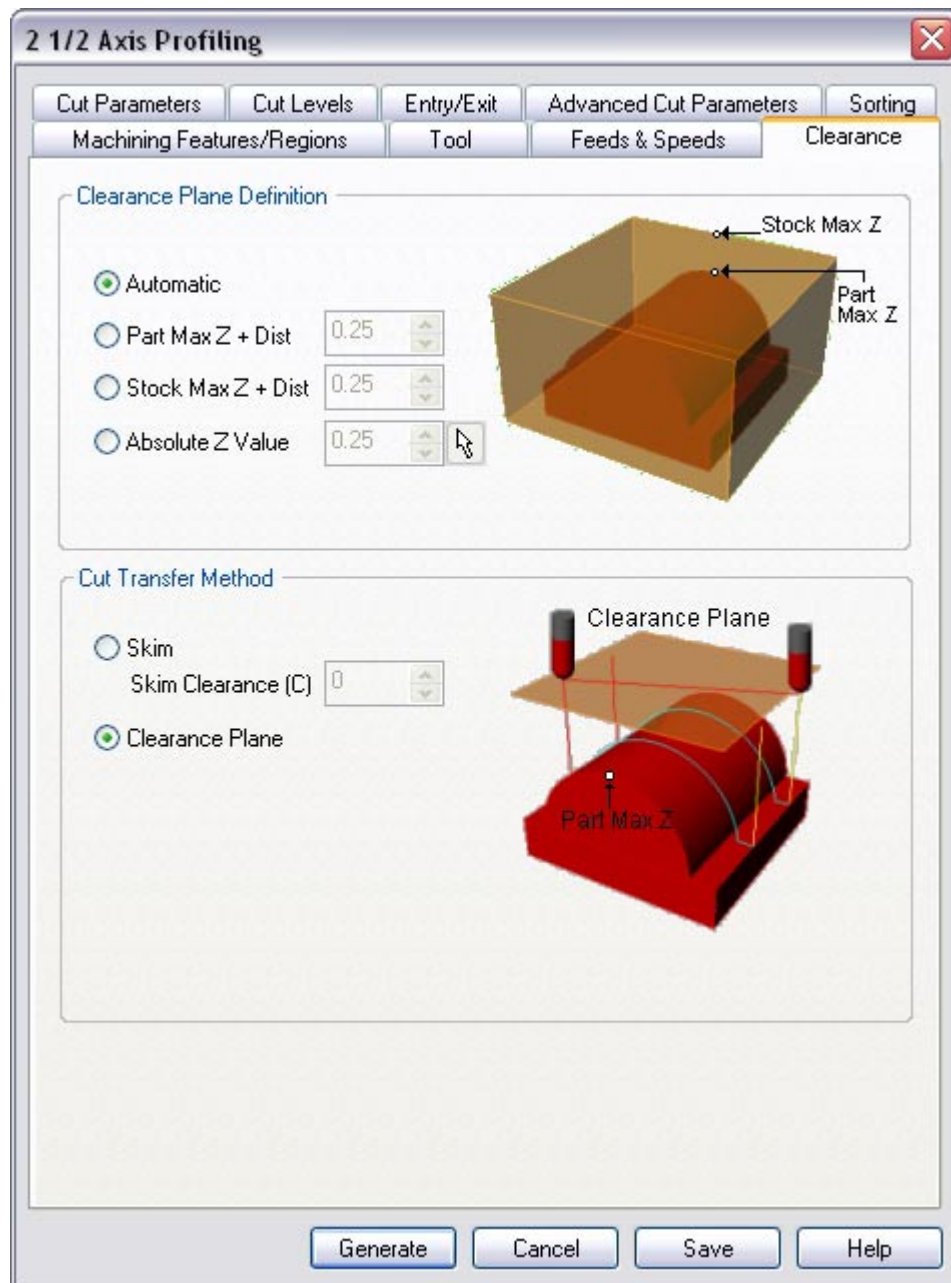
11. Set the Clearance Plane Definition to Automatic and Cut Transfer Method to Clearance Plane.

Alibre CAM will determine a safe Z height for the Entry & Exit when set to automatic. Setting Cut Transfer to Clearance Plane would apply the automatic Z clearance between transfers when the tool moves from a machining region to another.

Specifying Cut Parameters

12. Switch to Cut Parameters tab

2 1/2 Axis Profiling

Machining Features/Regions	Tool	Feeds & Speeds	Clearance
Cut Parameters	Cut Levels	Entry/Exit	Advanced Cut Parameters	Sorting

Global Parameters

Tolerance: 0.001

Stock: 0

Compensation: AUTO/NONE

Region — Toolpath — chord tolerance — Stock

Cut Direction
- ○ Climb (Down Cut)
- ○ Conventional (Up Cut)
- ● Mixed

Cut Start Side
- ● Right   ○ Left
- ☑ Use Outside/Inside for Closed Curves
  - ○ Outside   ● Inside
- ☐ Determine using 3D Model

☐ Corner Cleanup

Stepover Control

Total Cut Width: 0

Step/Cut: 0

Total Cut Width

Step/Cut

Generate | Cancel | Save | Help

13. Set the Stock = 0 and under cut start Side check Use Outside/Inside for Closed Curves and select Inside. Alternatively you can also use Determine using 3D model.

14. Select the Cut Levels Tab. Set Location of Cut Geometry At Top (the selected sketch is located at the top of the gasket) and specify the Total Cut Depth = 0.125. The cut depth is always set as an absolute value.
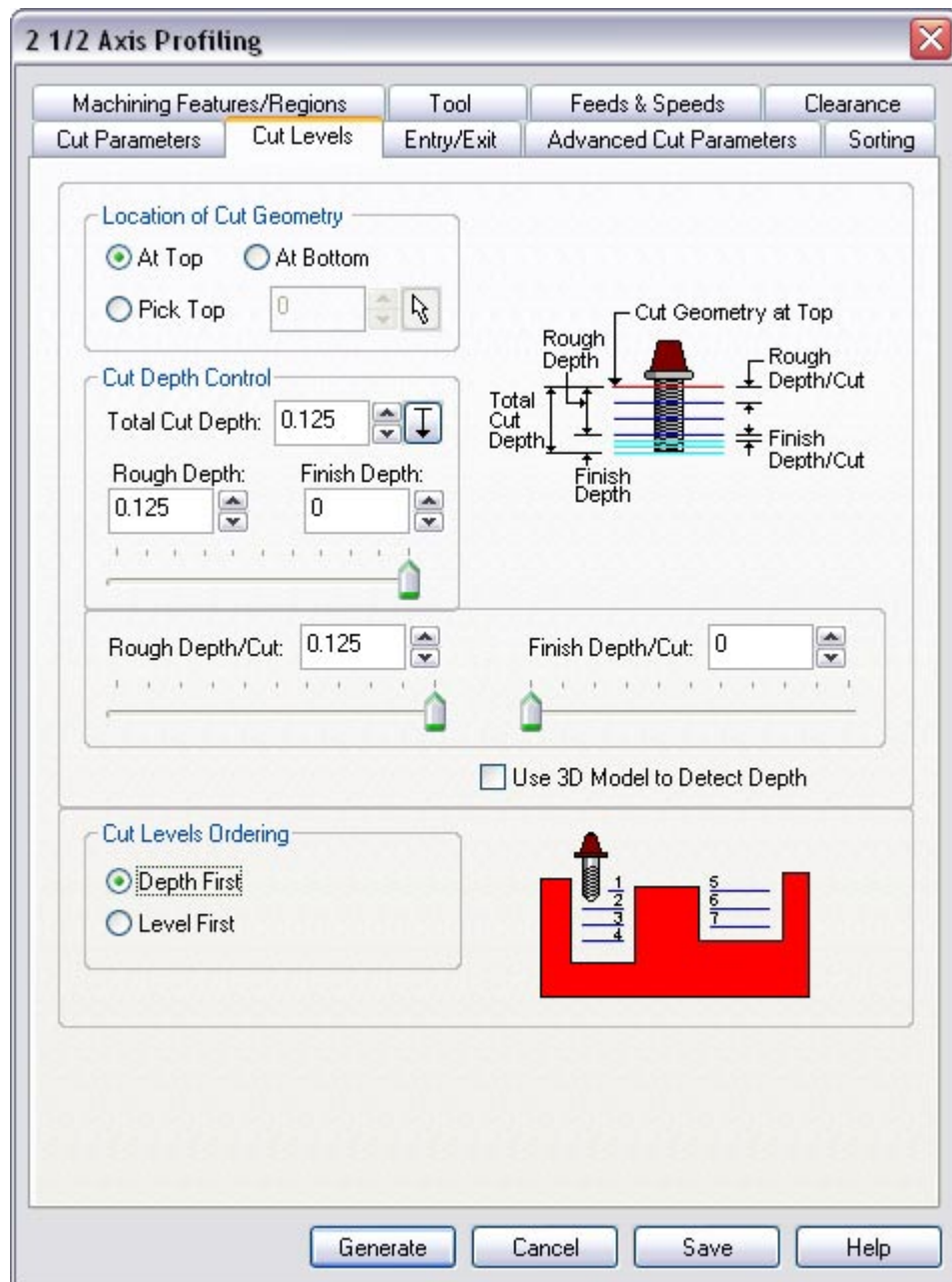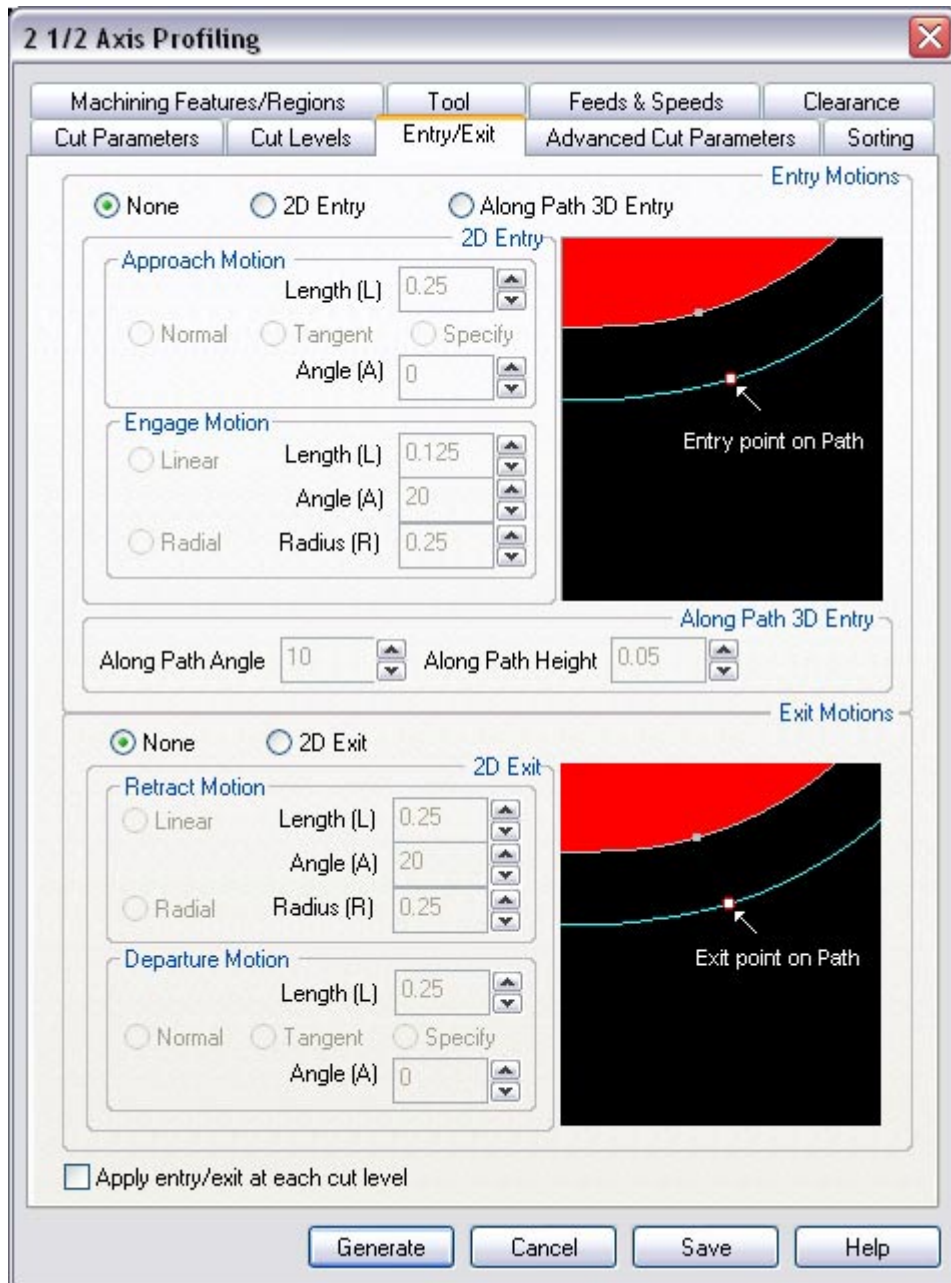
Entry/Exit

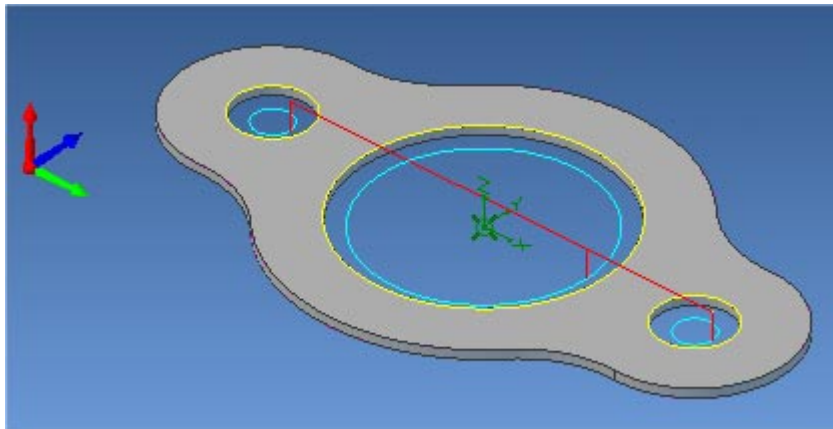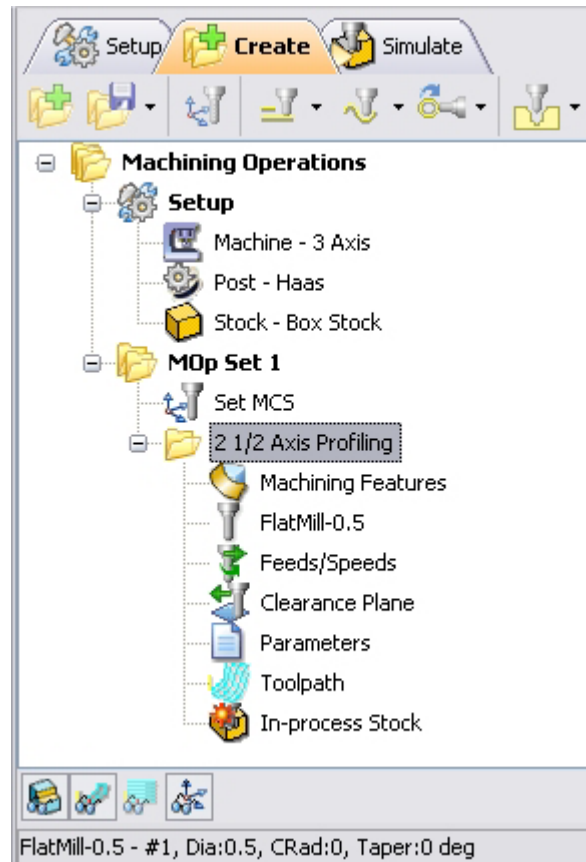15. Switch to Entry/Exit Tab and Set the Entry and Exit Type to None.

16. Click Generate. The 2½ Axis Profile 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.
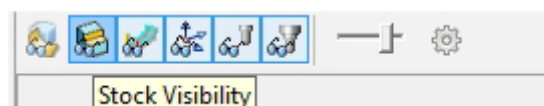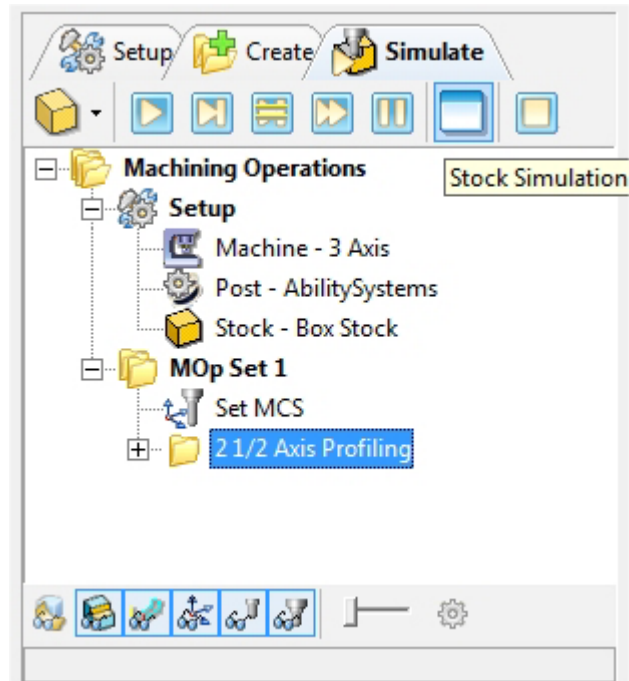
1. Right mouse click on the 2.5 axis Profile operation and rename the operation to Inner Profile.

Simulate Toolpath

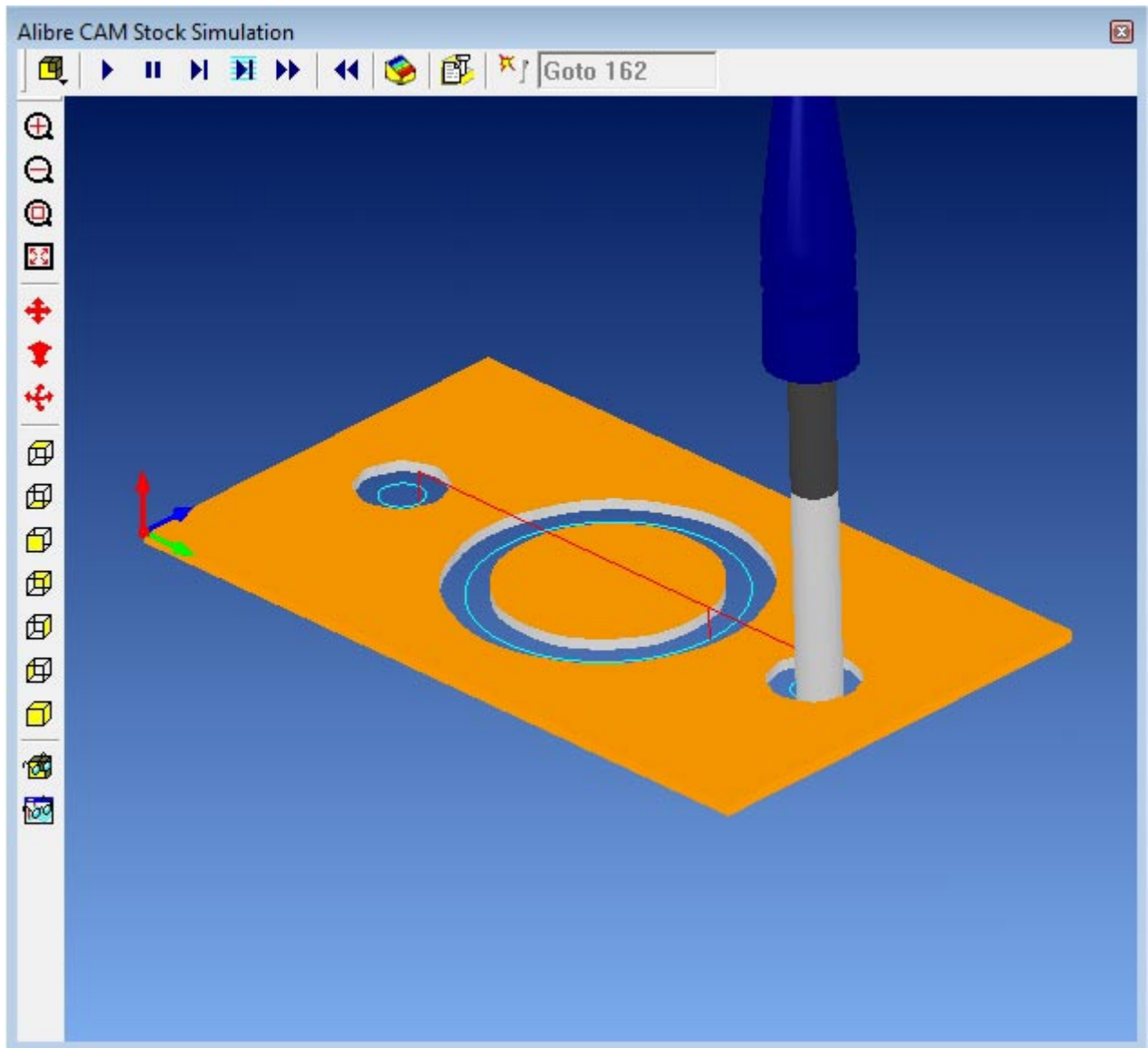Make sure to turn on Stock Visibility under the Simulate tab.

1. Switch to Simulate tab, Select the 2 ½ Axis Profiling 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.

Creating the Outer Profile Toolpath

We will not create a 2 ½ Axis Profiling toolpath to cut out the Gasket. Having created a profiling toolpath for the inner region, we will use the same Tool (FlatMill-0.5) and settings for Feeds /Speeds and Clearance plane.
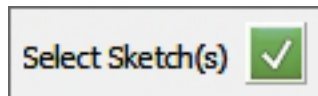
1. Switch to the **Create** Operation tab.

2. Select **Profiling** from the **2 ½ Axis** operations menu.

3. Under Machining Features/ Regions, select **Remove All**.

4. Now click on Select Curves as Regions and select **Sketch3** by switching over to the Design Explorer to select the outer profile of the Gasket as the region.
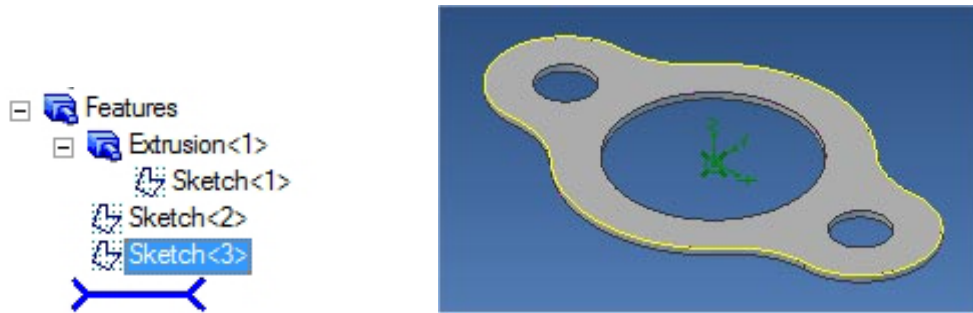
You can also select the sketch by clicking on the edge of the part from Alibre Design View.
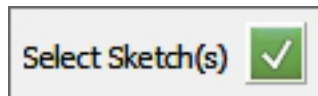
5. Click [Select Sketch(s) ✓] to complete the selection.

6. Switch to Tools tab and select **FlatMill-0.5** as the active tool.

7. Under Feeds/Speeds select **Load from Tool.**

8. Set the Clearance control to Automatic.

9. Switch to Cut Parameters tab, check **Use Outside/Inside for Closed Curves,** and set the Cut Start Side to **Outside**
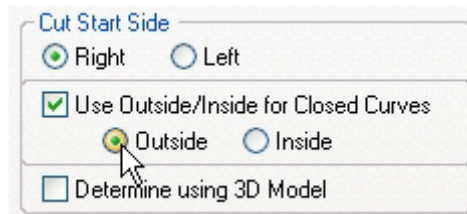
10. Go over to Cut Levels and set the Total Cut Depth = 0.125.

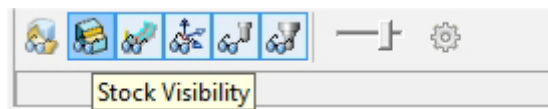11. Click Generate to Create the 2 ½ Axis Profile Toolpath.

The 2 ½ Axis Profile Operation is now created and is listed in the MOps Browser.
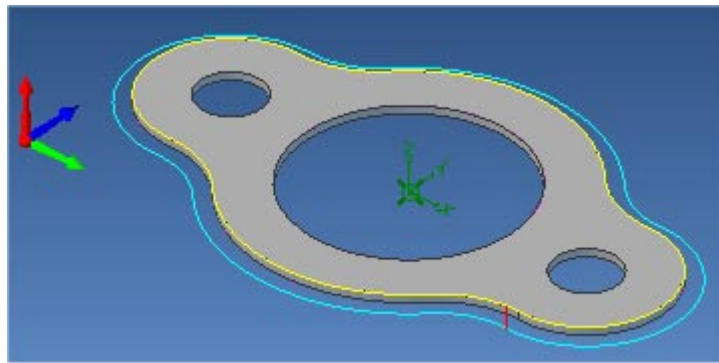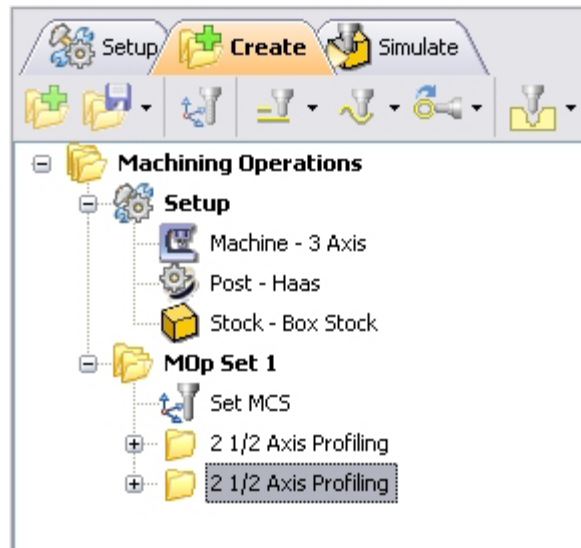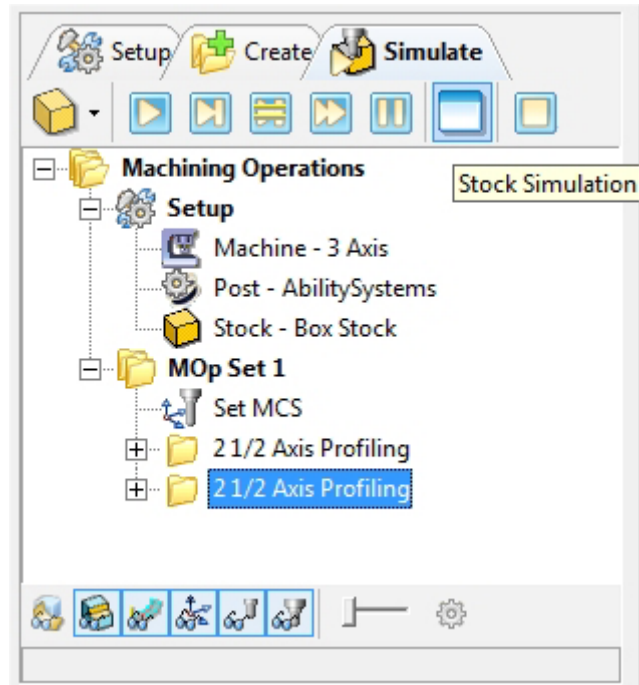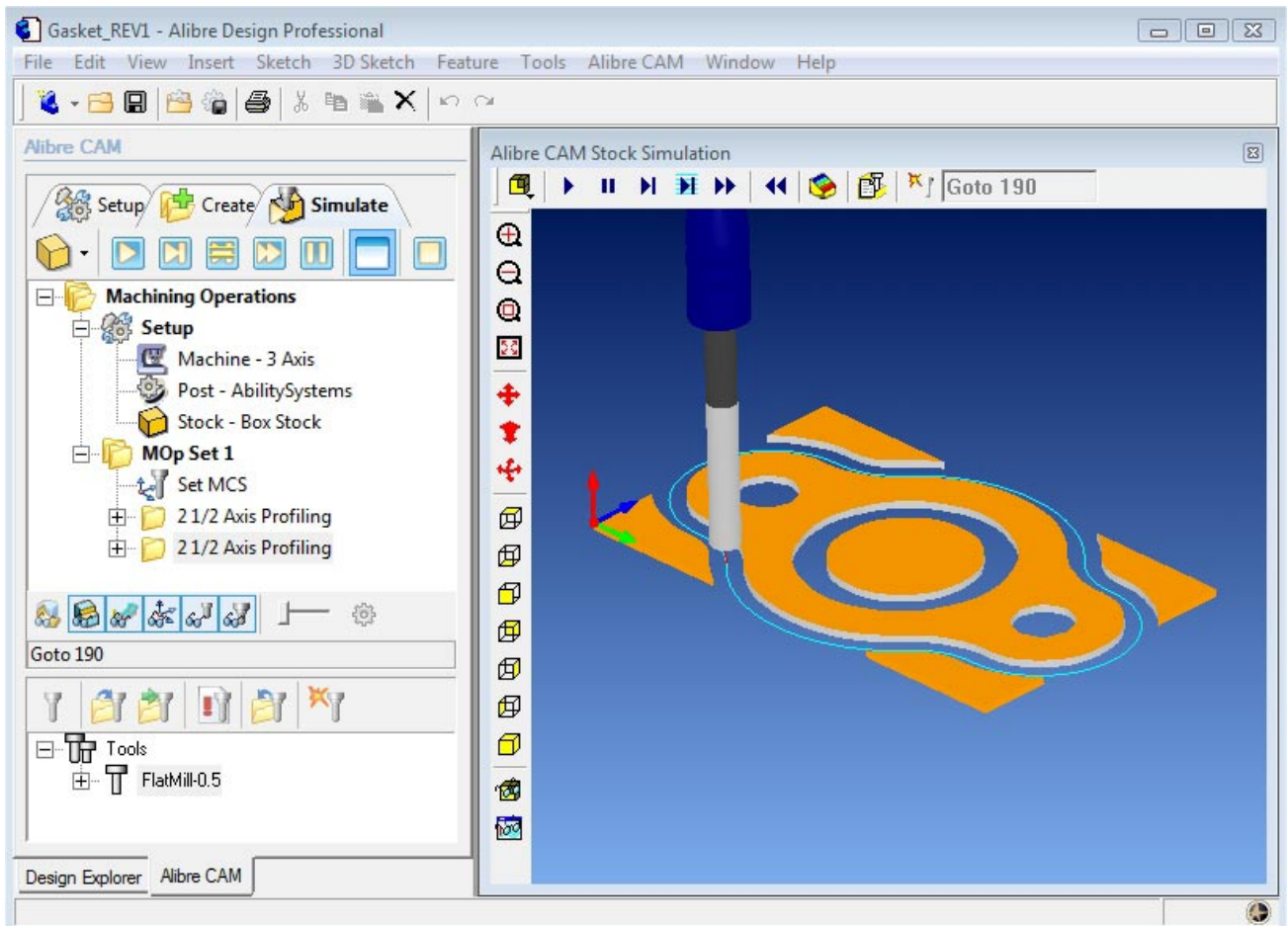
Simulate Toolpath

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

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

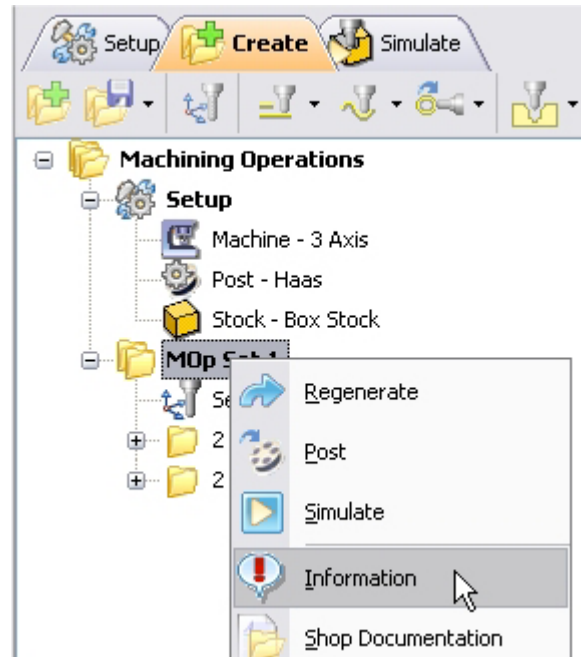2. Click Simulate from the Stock Simulation window to run simulation.

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

Reports

1. Switch to Create Operations Tab.

2. Select the MOp Set1 and right click and select Information.

This provides the estimated machining time for the operations created under MOp Set1.

Note: You can also go over to Machining Operations and right click and select information determine the estimated machining time for all the MOp Sets.

Shop Docs

Shop documentation can be generated selecting Machining Operations under the Create Operations tab. Right mouse click and **Shop Documentation**.
User can select from one of the 2 templates and generate shop documentation. This is saved as an html file and can be printed and handed over to the operator in preparation for the part to be machined on the CNC.
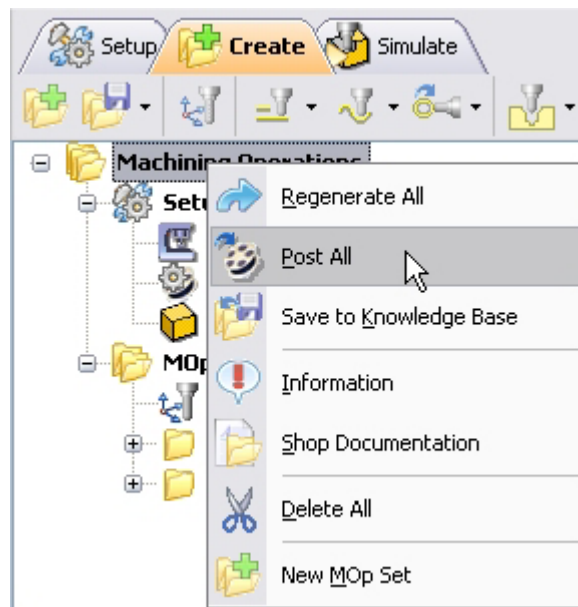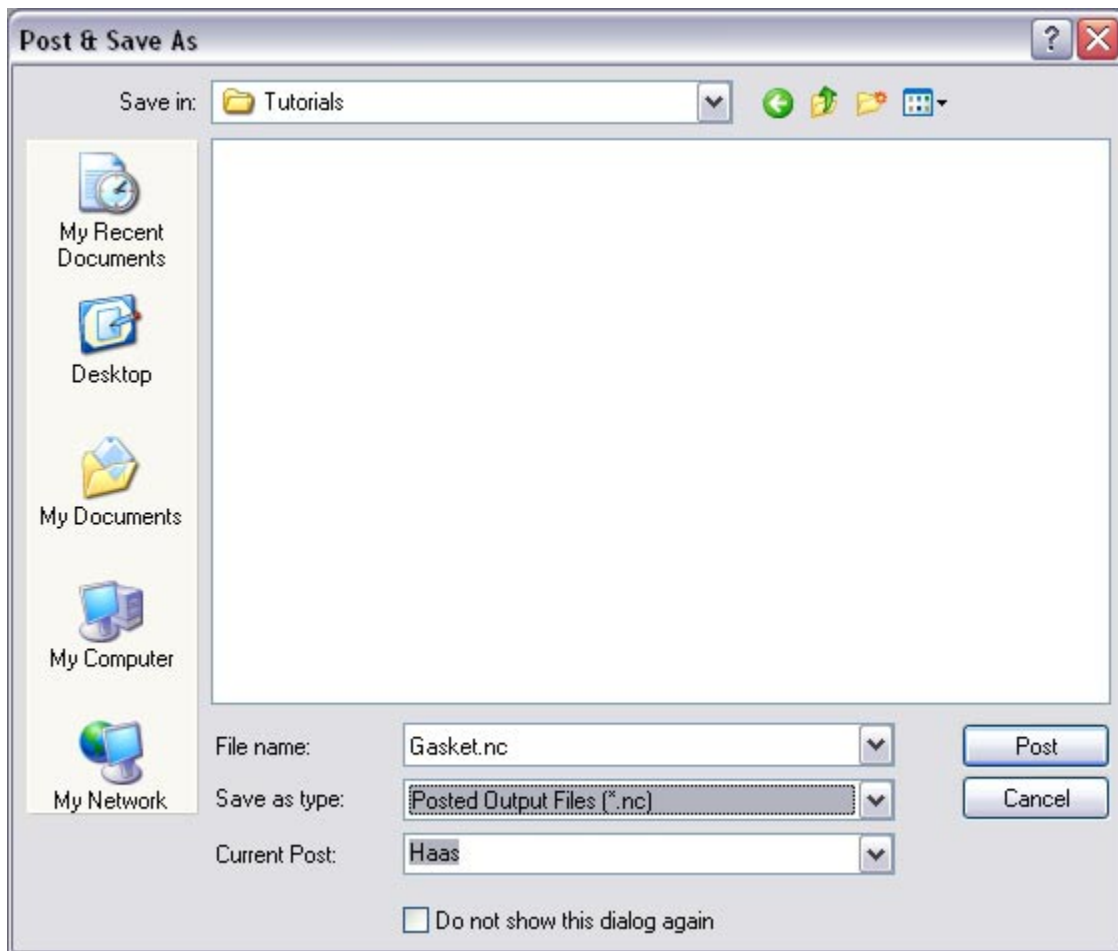
Post Processing

Once the toolpath has been generated, it can be post-processed to a specific machine controller. Alibre CAM comes with a set of post-processors to choose from. Each post-processor is represented by an *.**spm** file, all of which are located in the **Posts** folder under the Alibre CAM installation folder.

1. Select **Machining Operations** from the **Create** Operations tab and right click and select **Post Process** / **Post All**.

2. Specify the File Name as **Gasket.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