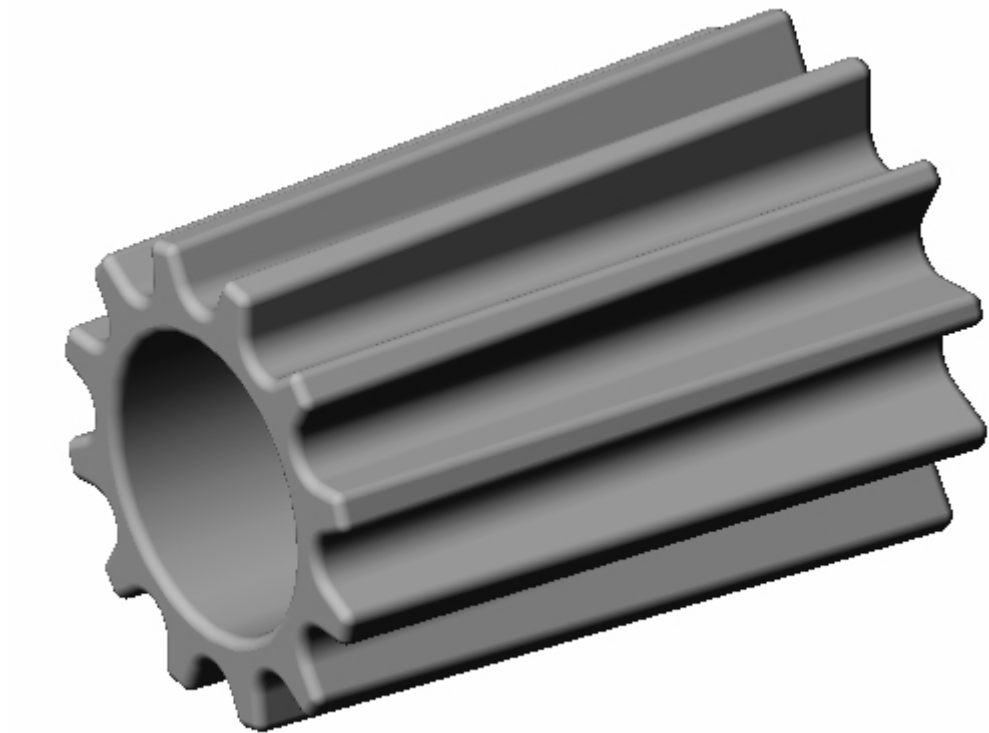


## Tutorial 6: Machining a Gear

---



Using 4 Axis Roughing and Finishing

## Introduction

This tutorial will illustrate machining the gear using 4 axis-milling operations. This tutorial will introduce the usage of several 4-axis operations such as 4 axis roughing and finishing.

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 Gear

- We will machine the gear completely using 4 axis-machining operations.
- The part itself will be machined out of a cylindrical blank.
- The stock will be held to the machine table using a rotary chuck.
- The part will be machined using 0.25" and 0.125" ball end mills.

## Main Programming Steps

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

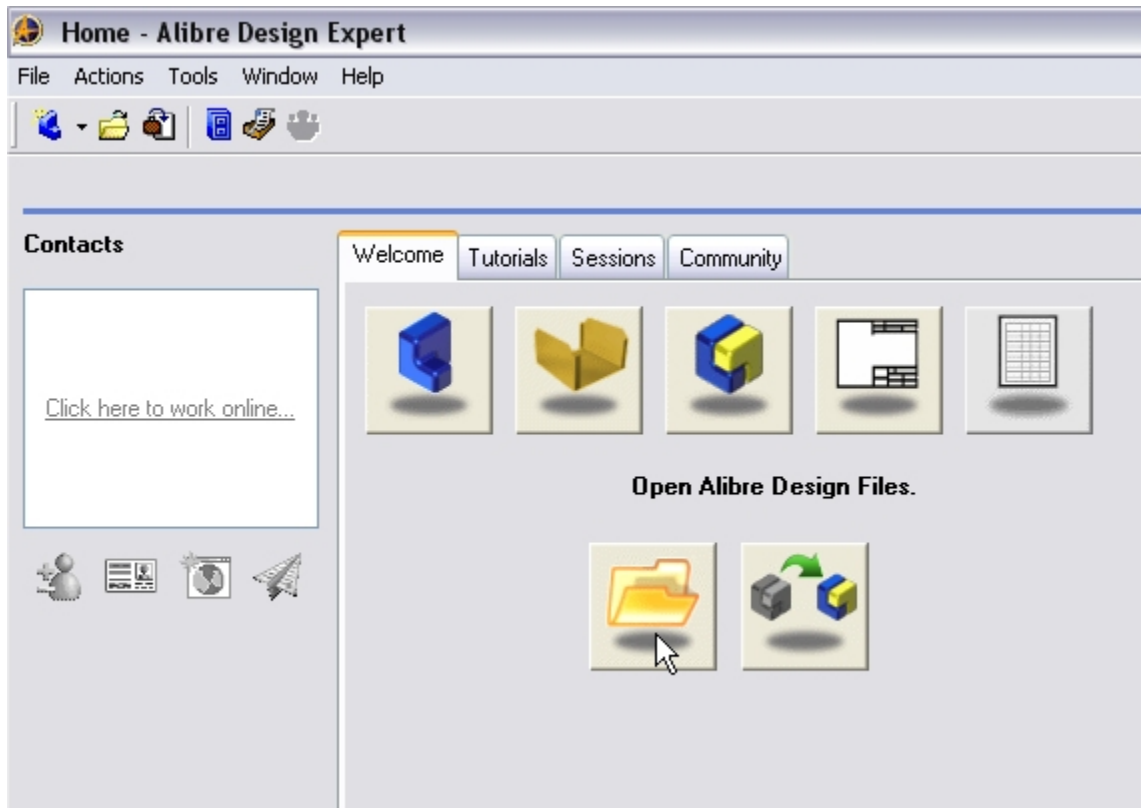
- Create the Stock geometry
- Set the Machine Zero point or Locate geometry with respect to the machine coordinates.
- Set the rotary axis and rotary center.
- Create / Select the tool used for machining
- Set the feeds and speeds
- Set the Clearance Plane for the non-cutting transfer moves of the cutter
- 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.

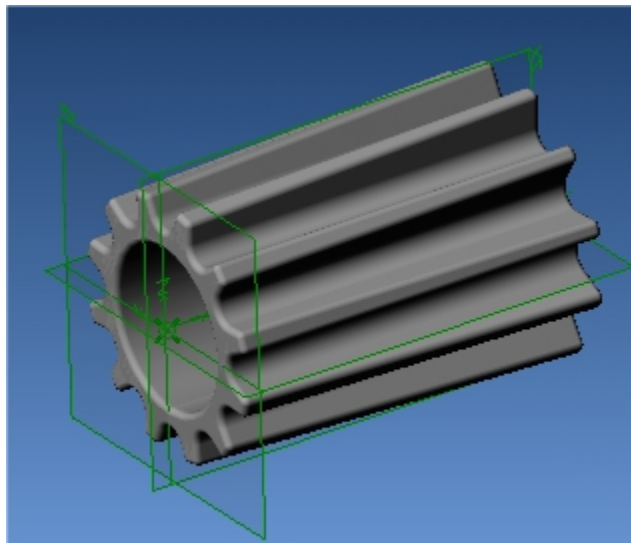
## Loading the Part Model

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.



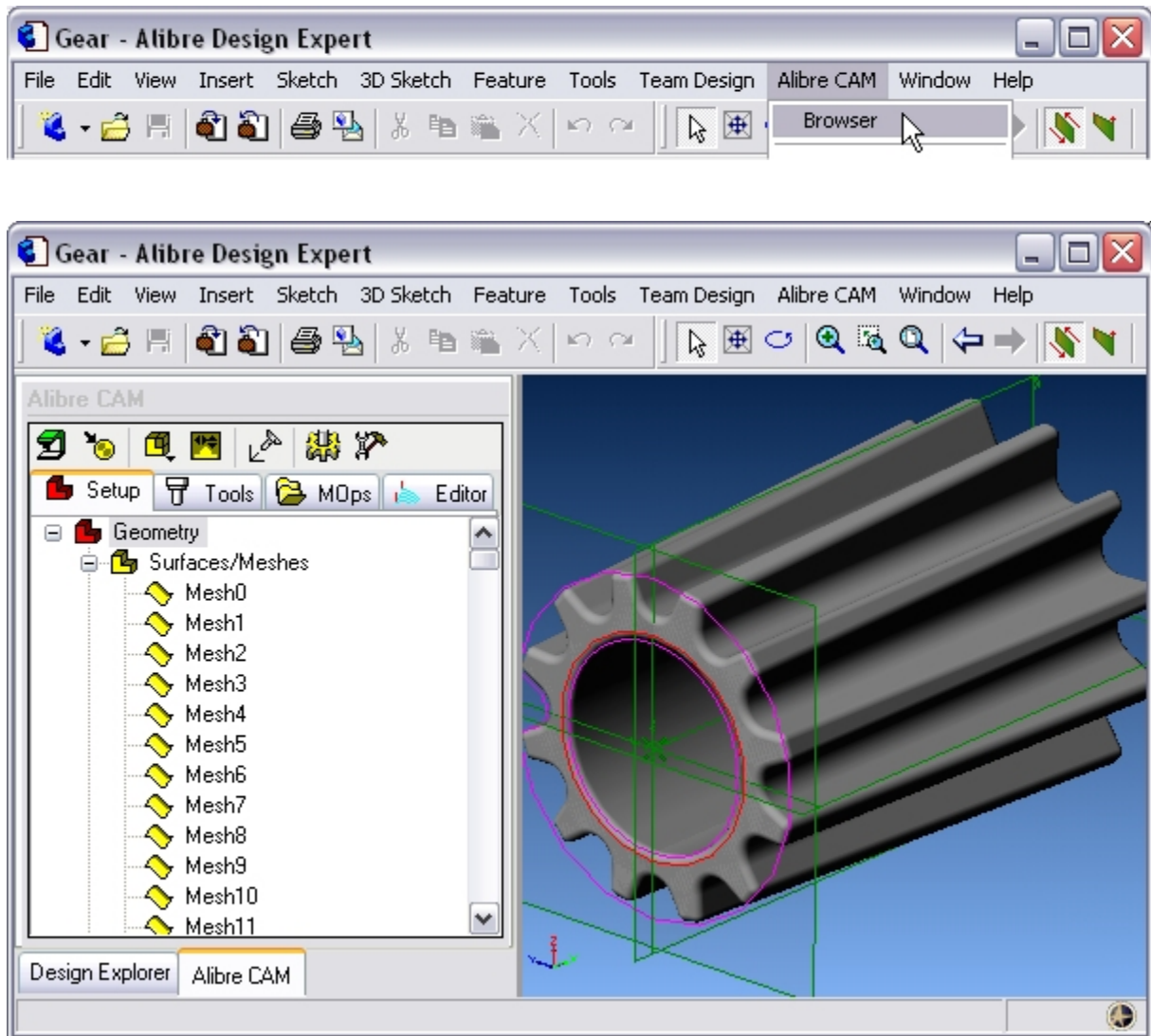
2. From the **Open** dialog box, select the **Gear.AD\_PRT** file from the **Tutorials** folder in the Alibre CAM installation folder. (Default location C:\Program Files\MecSoft Corporation\Alibre CAM\Tutorials)
3. The loaded part appears as shown below.



**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

2. Select Alibre CAM from the menu bar and click Browser.



The Alibre CAM browser is now loaded and is docked over the Design Explorer. You can toggle between the CAM 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.

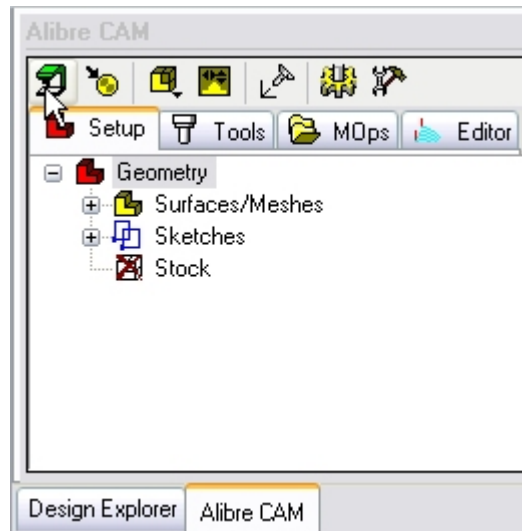
## Preparing the Part for Machining

Selecting this tab displays the Setup Toolbar at the top of the Browser window. Additionally it displays the three types of geometry that are relevant to Alibre CAM: **Surfaces/Meshes**, **Sketches** and **Stock**.

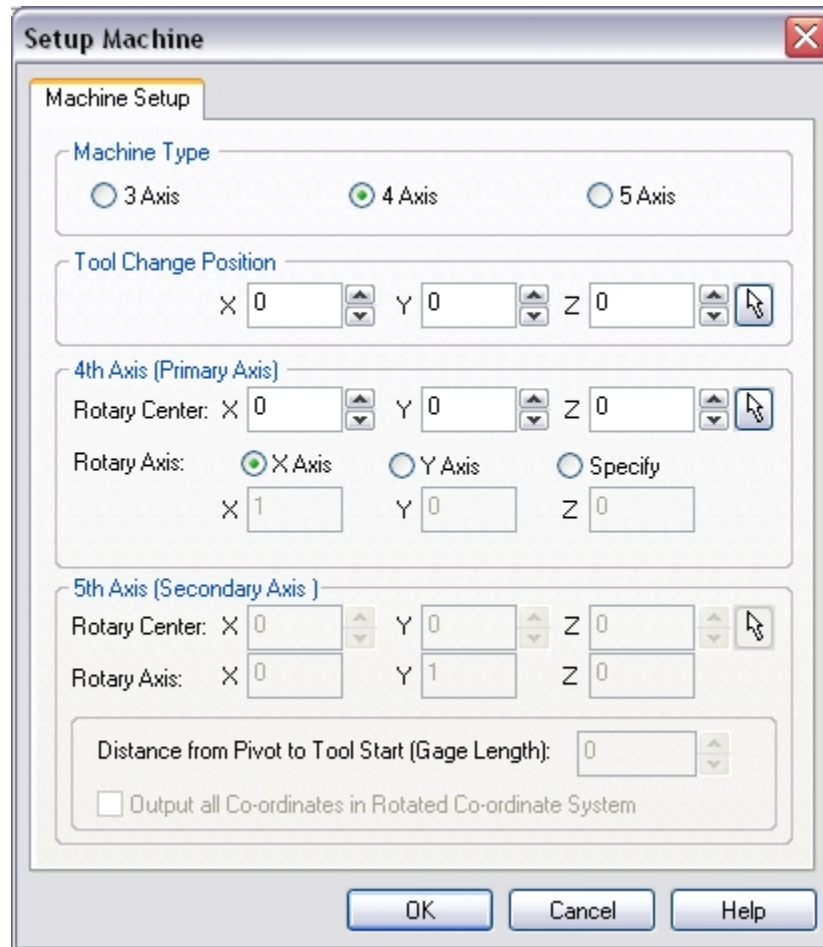
The Setup tab allows you to specify Machine Setup, Stock Geometry, Machine Coordinate System (Machine Zero) & Alibre CAM Preferences.

## Setup Machine

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



2. Set the Machine Type to **4 Axis**
3. Rotary Axis to **X (A) Axis** and the Rotary Center to 0,0,0 in X Y and Z.



The image shows a 'Setup Machine' dialog box with a 'Machine Setup' tab. It contains several sections for configuring machine parameters. The 'Machine Type' section has three radio buttons: '3 Axis', '4 Axis' (selected), and '5 Axis'. The 'Tool Change Position' section has three input fields for X, Y, and Z, all set to 0. The '4th Axis (Primary Axis)' section has a 'Rotary Center' section with X, Y, and Z input fields (all 0) and a 'Rotary Axis' section with three radio buttons: 'X Axis' (selected), 'Y Axis', and 'Specify'. Below these are input fields for X (1), Y (0), and Z (0). The '5th Axis (Secondary Axis)' section has a 'Rotary Center' section with X, Y, and Z input fields (all 0) and a 'Rotary Axis' section with three radio buttons: 'X' (selected), 'Y', and 'Z'. Below these are input fields for X (0), Y (1), and Z (0). At the bottom, there is a 'Distance from Pivot to Tool Start (Gage Length)' input field set to 0, and a checkbox labeled 'Output all Co-ordinates in Rotated Co-ordinate System' which is unchecked. At the very bottom are 'OK', 'Cancel', and 'Help' buttons.

**Setup Machine**

**Machine Setup**

**Machine Type**

☐ 3 Axis ☒ 4 Axis ☐ 5 Axis

**Tool Change Position**

X 0 Y 0 Z 0

**4th Axis (Primary Axis)**

Rotary Center: X 0 Y 0 Z 0

Rotary Axis: ☒ X Axis ☐ Y Axis ☐ Specify

X 1 Y 0 Z 0

**5th Axis (Secondary Axis)**

Rotary Center: X 0 Y 0 Z 0

Rotary Axis: X 0 Y 1 Z 0

Distance from Pivot to Tool Start (Gage Length): 0

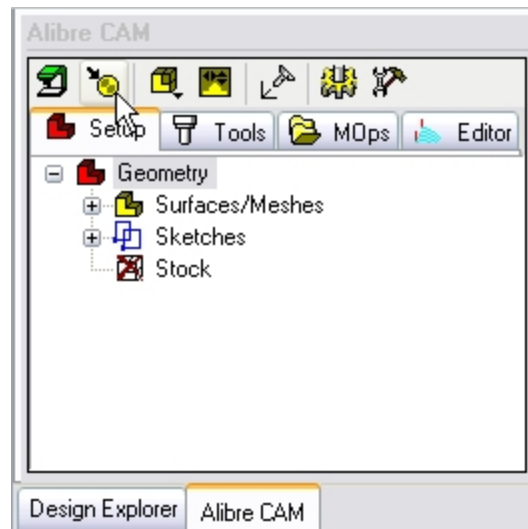
☐ Output all Co-ordinates in Rotated Co-ordinate System

OK Cancel Help

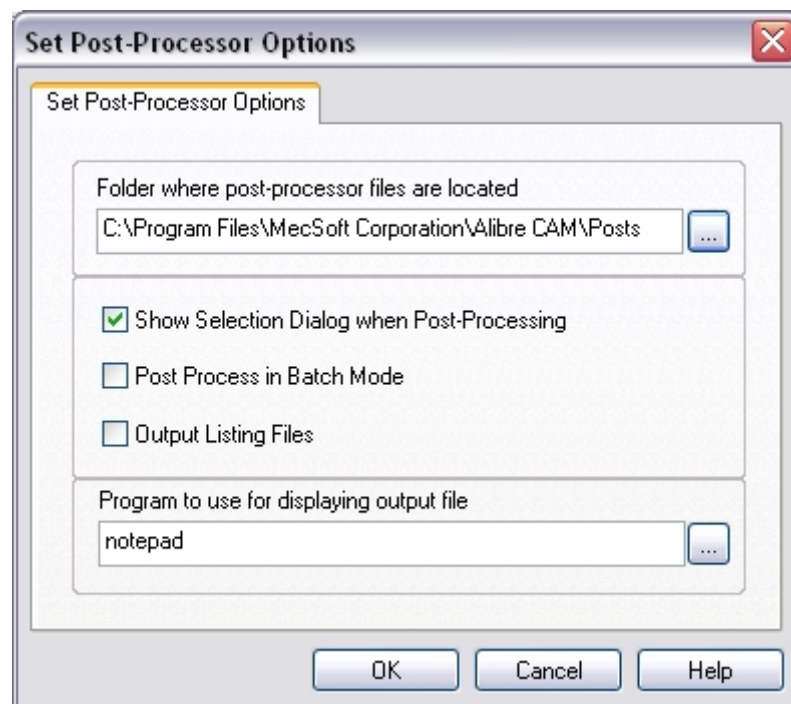
- Click OK to close the Setup Machine Dialog.

## Set Post Options

- Select Set Post Options from the Setup tab.



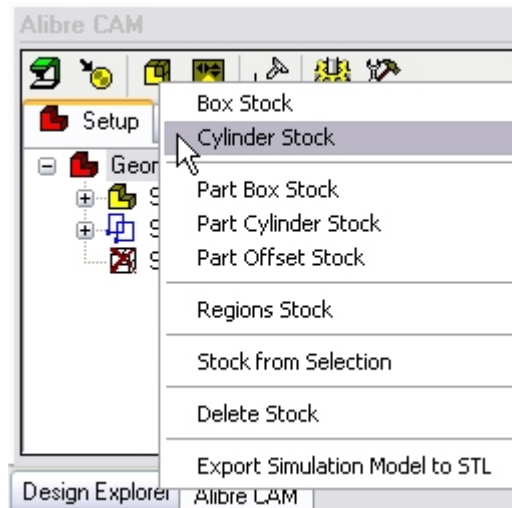
6. Make sure to use the following settings.



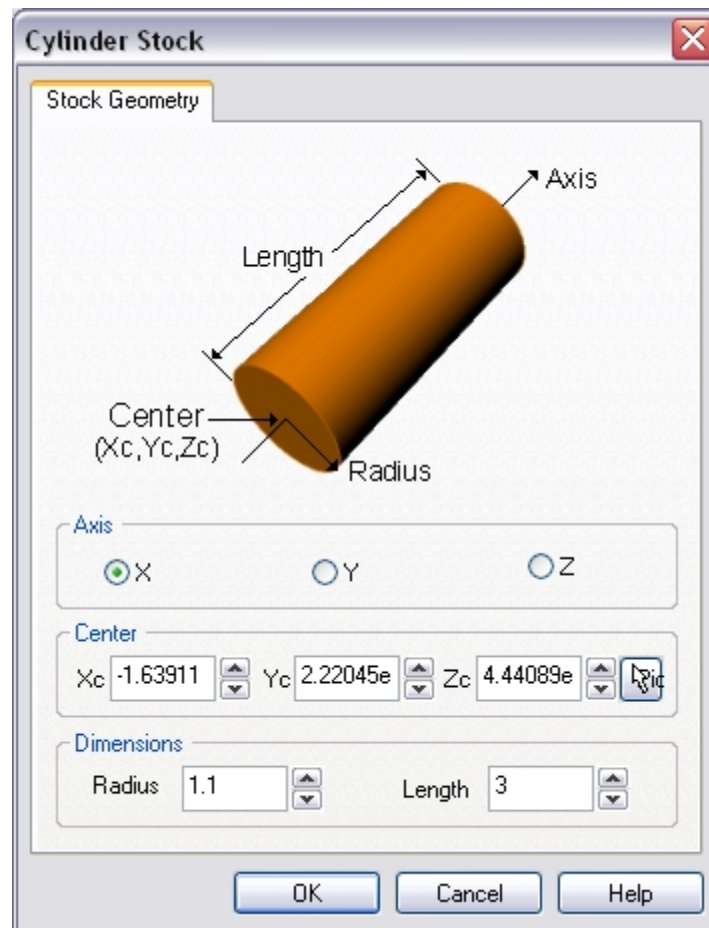
- a. Folder where post processor files are located is set to the Posts folder in Alibre CAM installation folder.
- b. Select – Show Selection Dialog when Post Processing
- c. Program to use for displaying output file can be set to notepad or WordPad.
- d. Click OK to accept the settings.

## Create/Load Stock

7. Select Create / Load Stock from the Setup tab and choose Cylinder Stock.



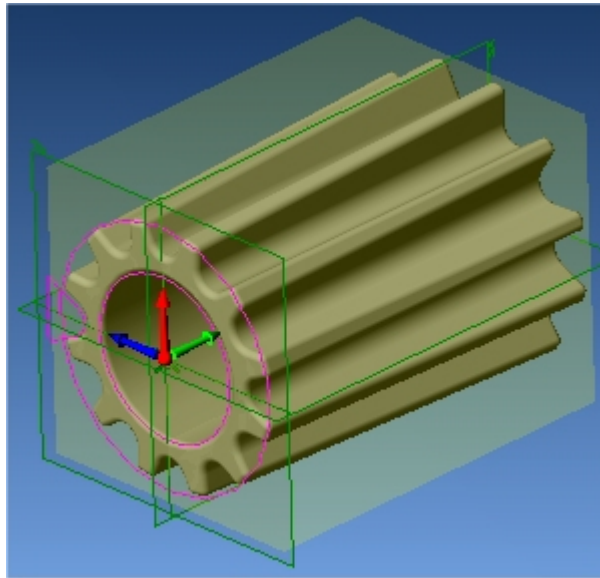
This brings up the Cylinder Stock parameters.



8. Use the following Dimensions. Radius = 1.1, Length = 3. Click OK to create the stock geometry.



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

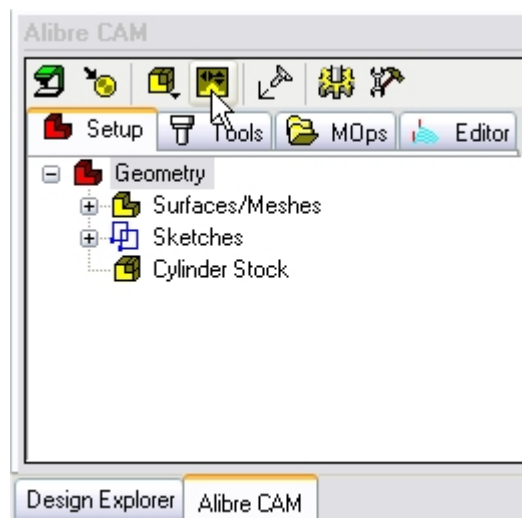


**Note:** If the Stock Geometry is not visible, Switch to the MOps tab and Make sure Show Stock is selected.

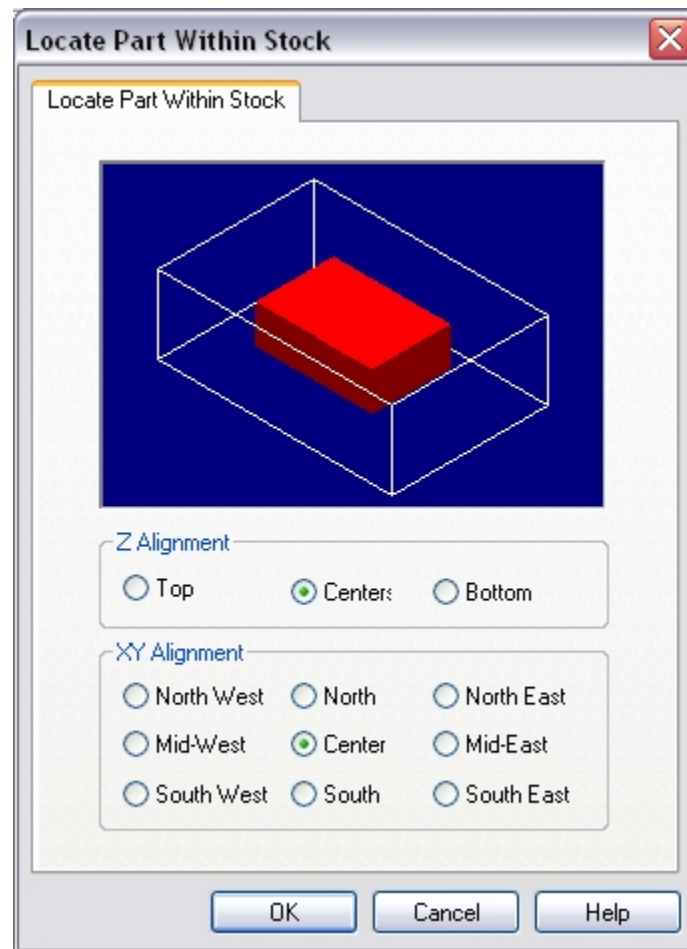
## Locate Part Within Stock

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).

10. Select Locate Part With Stock from the Setup tab.



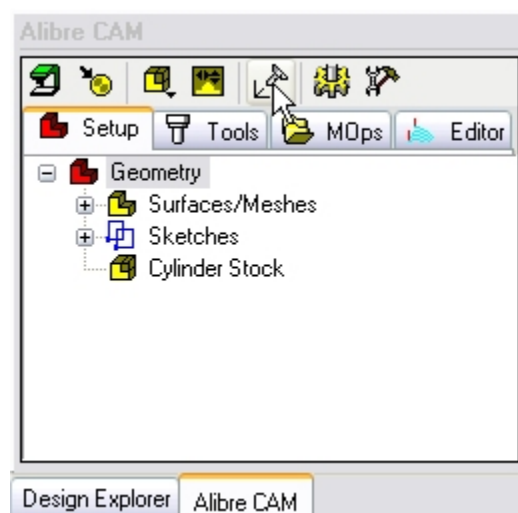
11. Set the Z alignment to Center and XY alignment to Center. (This would align the stock to the center of the part in Z and center in XY)



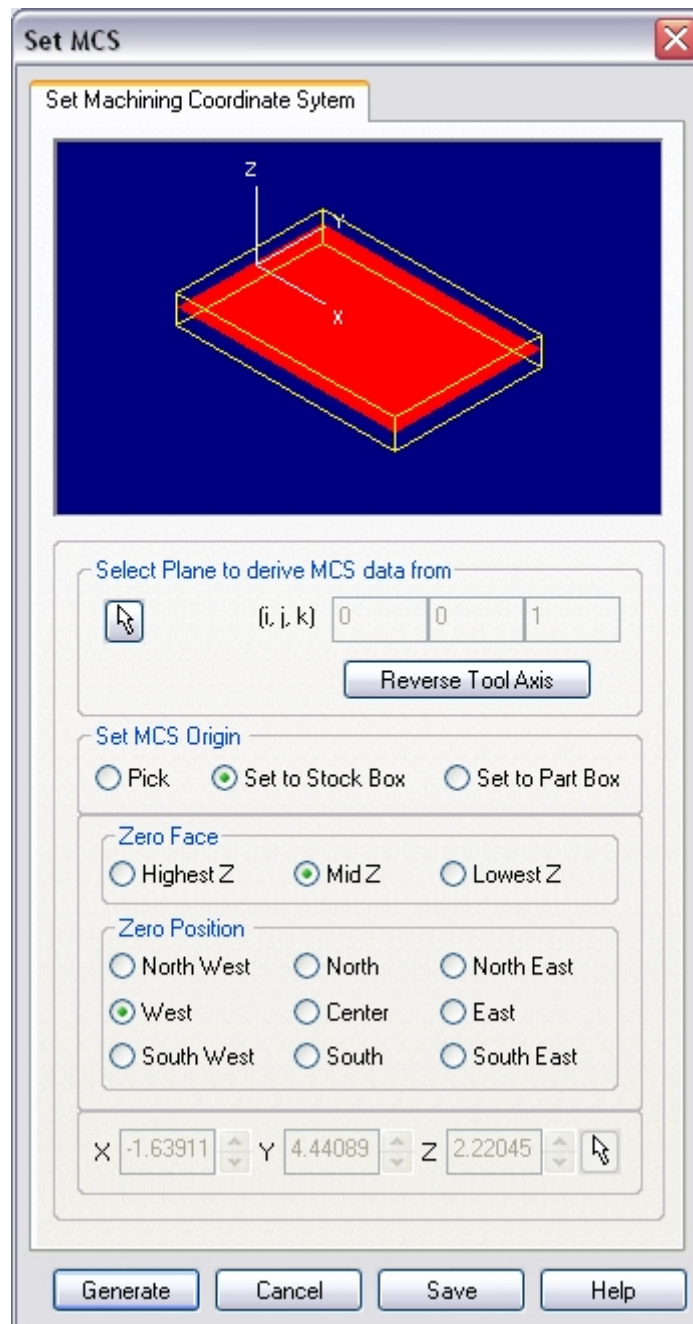
## Set MCS

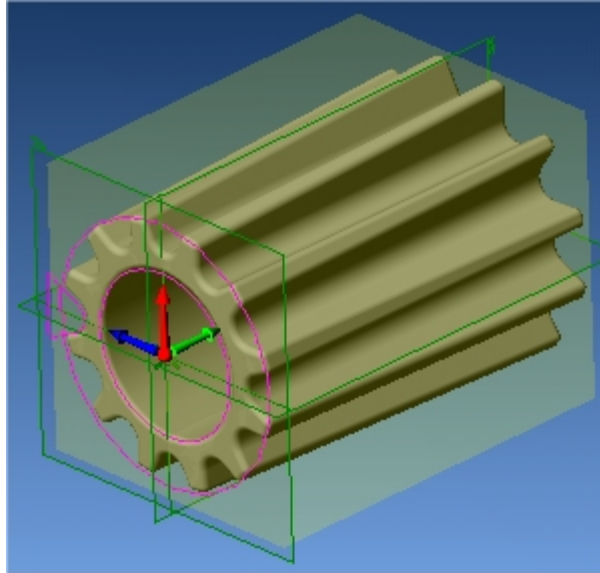
The next step is to locate the machining coordinate system (MCS). The MCS is represented as a triad with 3 arrows, the green arrow indicating X axis, blue the Y axis and red the Z axis. It is always a good practice to set the Machine Zero before generating a toolpath.

12. Select Set MCS from the Setup tab.



13. Set the MCS Origin to Stock Box, Zero Face to Mid Z and Zero Position to West.





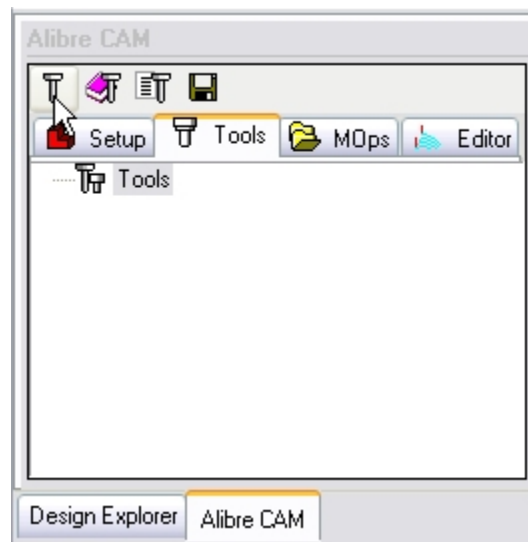
## Defining Cutting Tools

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.

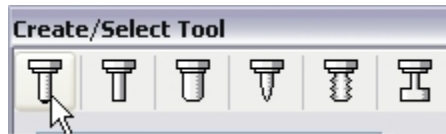
## Create Tools

To machine the above part we will now create 2 tools.

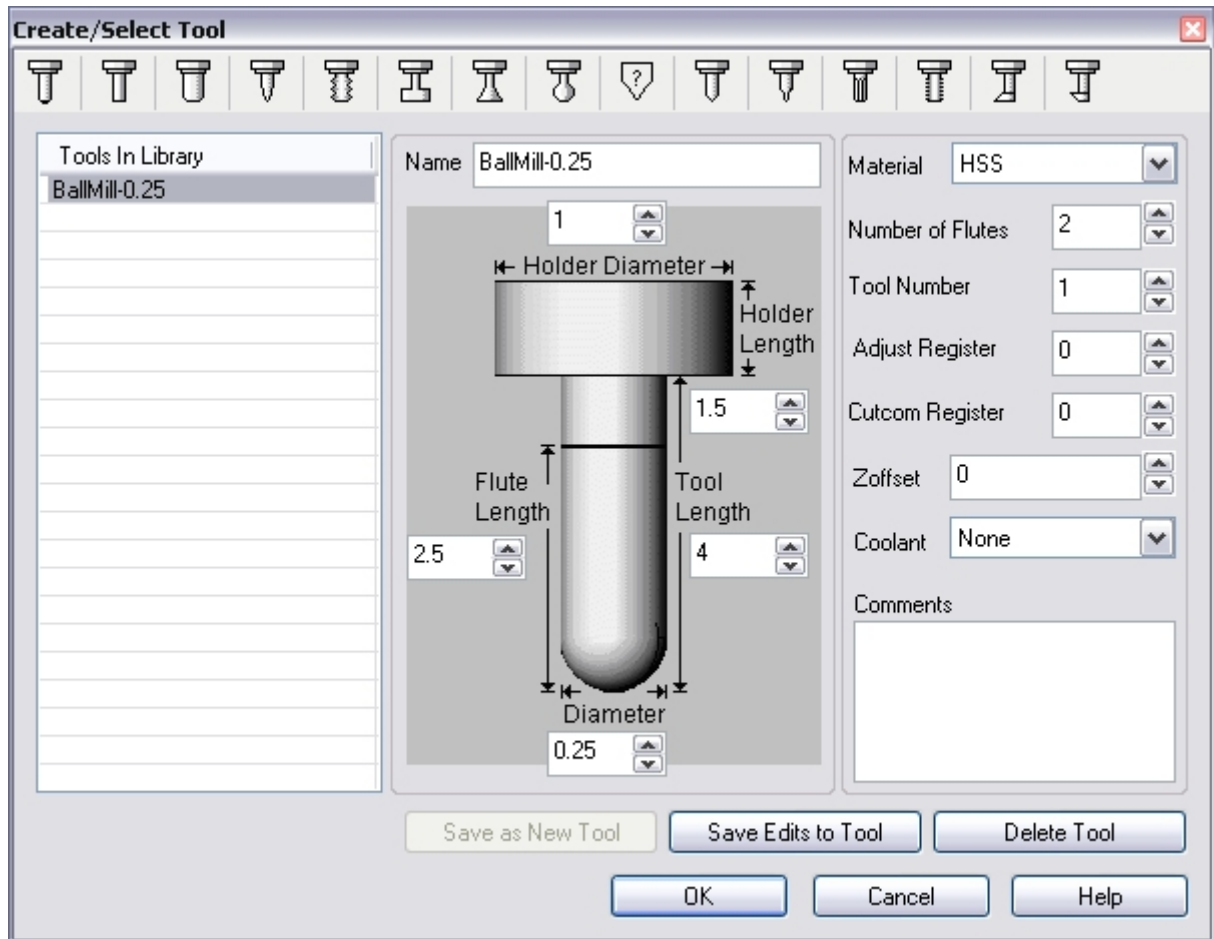
1. Select Create / Select Tools from the Tools tab.



2. Set the Tool Type to Ball Mill.

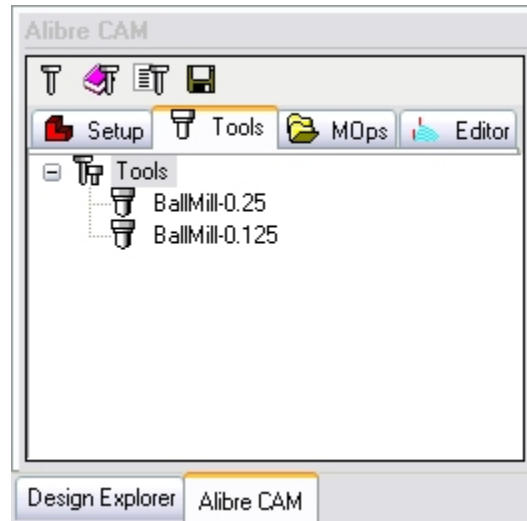


- Set the tool name as **BallMill-0.25**, Tool Diameter = **0.25**, Tool Number = **1**.



- Create a 2<sup>nd</sup> Ball End Mill. Set the tool name as **BallMill-0.125**, Tool Diameter = **0.125**, Tool Number = **2**.
- Click Save as New Tool. Click OK to close the Create/Select Tool Dialog.

The tools are now created and listed in the Tools tab under the Alibre CAM browser.



## Creating the 4<sup>th</sup> Axis Roughing Toolpath

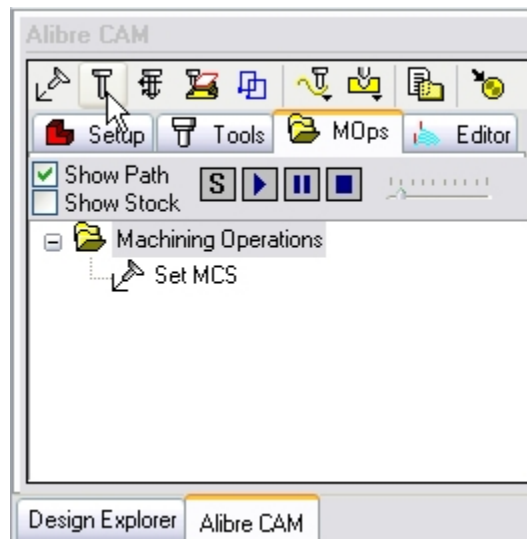
We will machine the gear using 2 different machining operations – 4 axis Roughing and 4 axis Finishing.

The first step in machining the gear will be a roughing operation. In this cut method, the tool cuts the stock in successive levels. The spacing between these levels is specified by the user.

This type of machining is very efficient for removing large volumes of material and is typically performed with a large tool. Roughing is typically followed by semi-finishing or finishing toolpaths.

### Select Tool

1. Select Create/Load Tool from the Mops tab and select Ball Mill-0.25

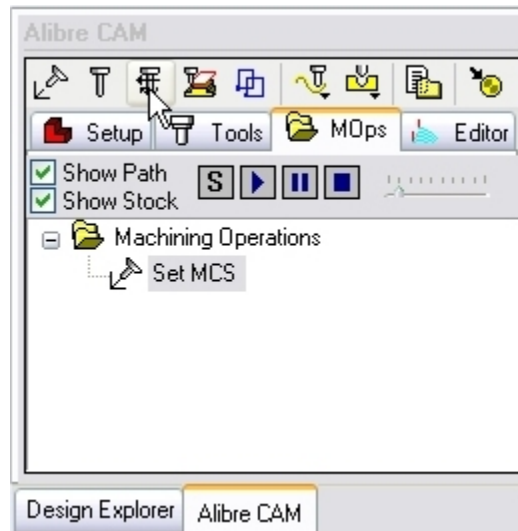


2. Select BallMill-0.25 from Tools in Library and click OK to set this as the active tool.

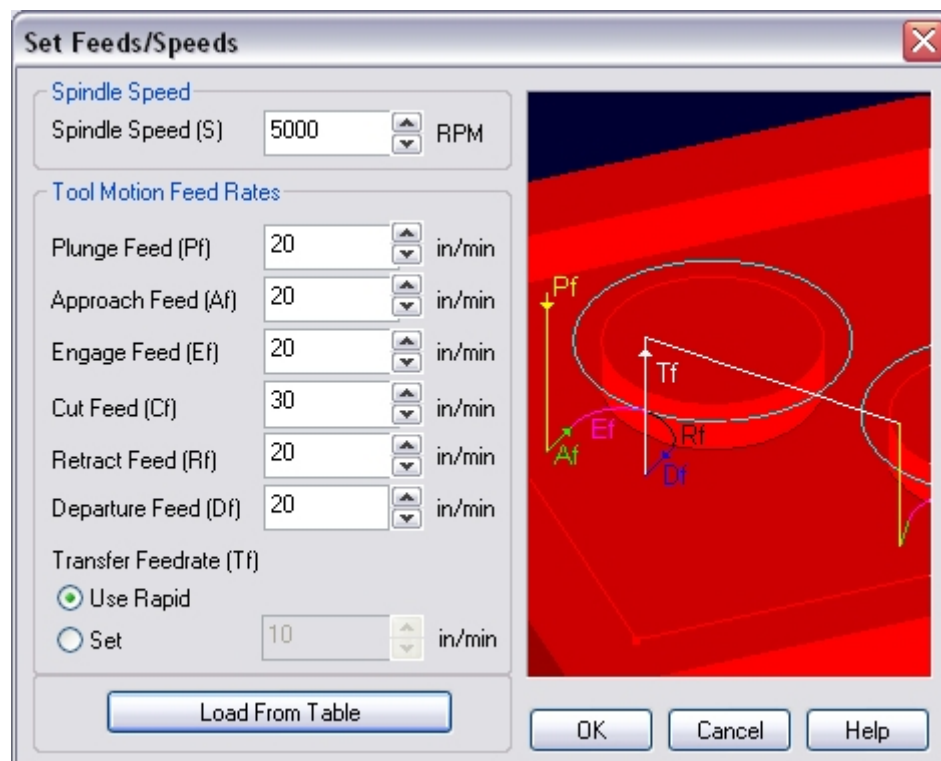
## Set Feeds / Speeds

You can set toolpath feeds and speeds and customize these settings for later use. You can do this using the Set Feeds/Speeds dialog. This dialog can be invOKed from the **Set Feeds/Speeds** button found in the **MOPs** tab toolbar.

4. Select Set Feeds/Speeds from the MOPs tab.



5. Set the Spindle Speed = 5000 rpm, Plunge, Approach, Engage Feed = 20 in/min, Cut Feed = 30 in/min, Retract & Departure Feed = 20 in/min.



6. Click OK to accept the settings.

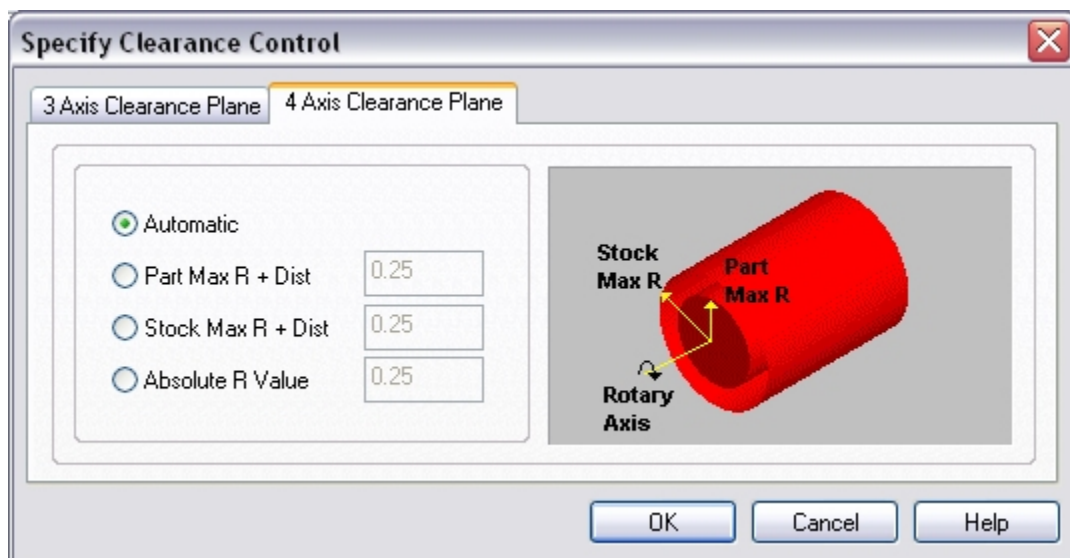
## Clearance Control

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.

1. Select Clearance Control from the MOps tab.



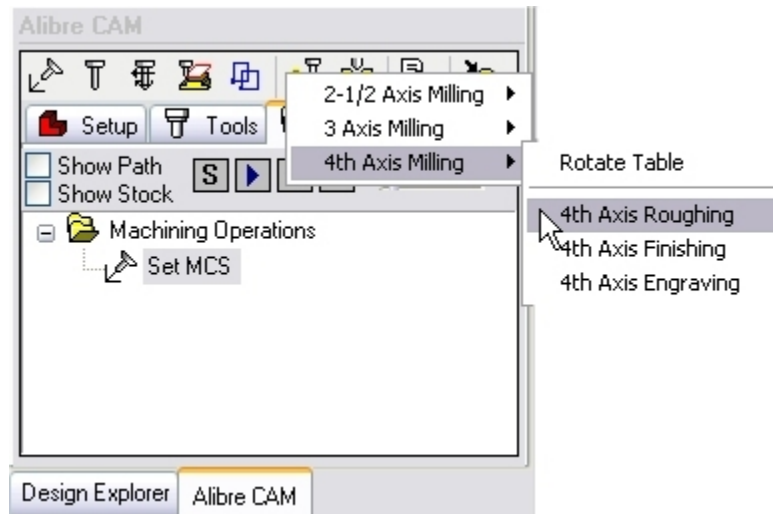
2. Set the Clearance Plane to Automatic under the 4 Axis Clearance Plane tab.



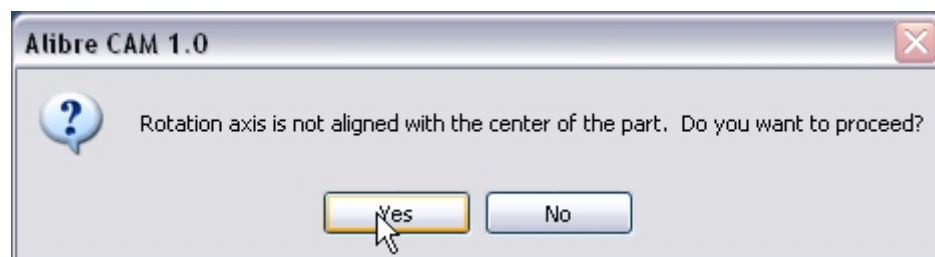
## 4<sup>th</sup> Axis Roughing

1. Select Milling Methods -> 4<sup>th</sup> Axis Milling -> 4<sup>th</sup> Axis Roughing

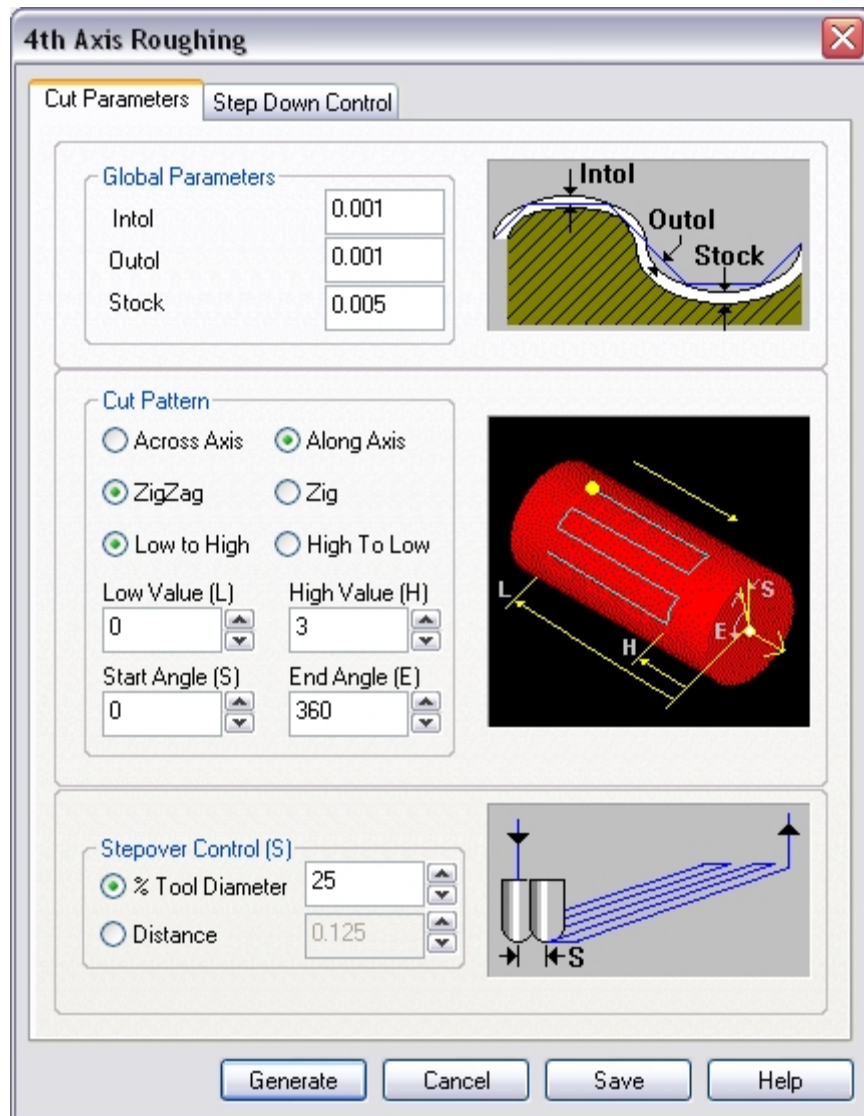




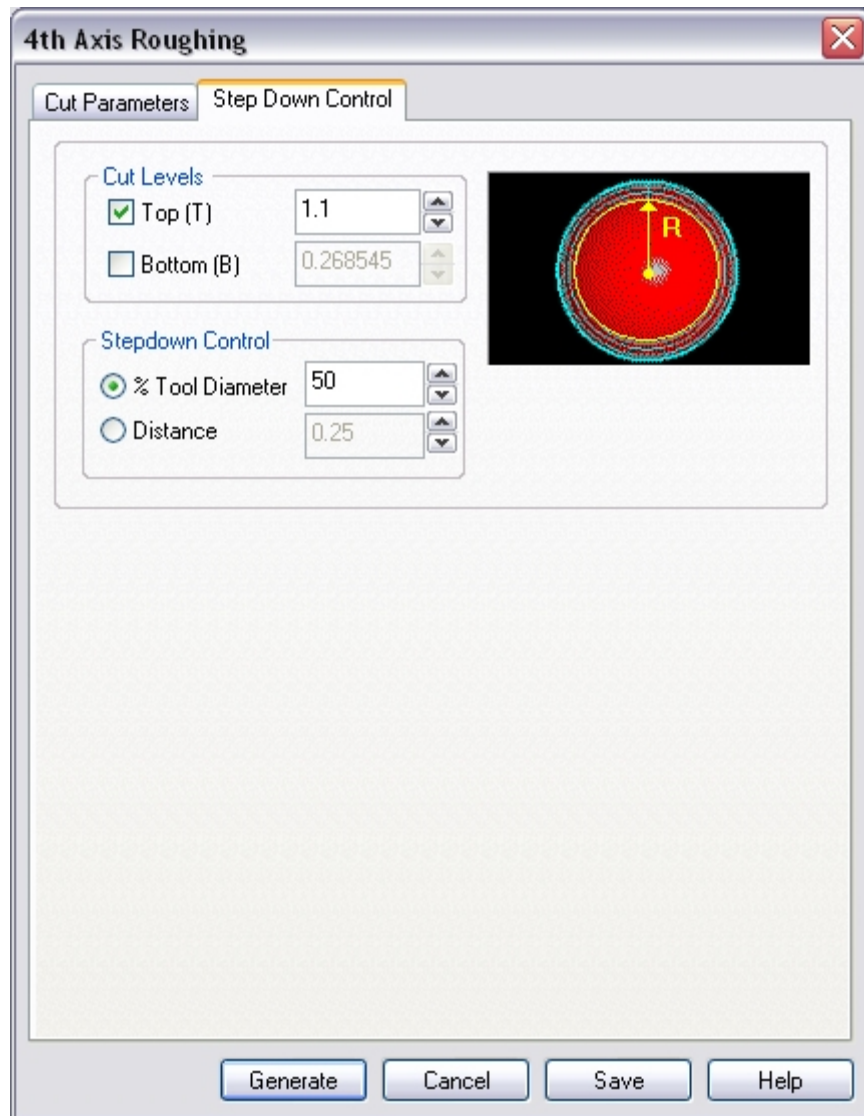
If the Machine Zero is not set at the center of the part in X, Y & Z, a warning message dialog would be displayed at all times when a 4 axis machining operation is selected. Users can override this message by clicking OK in the dialog.



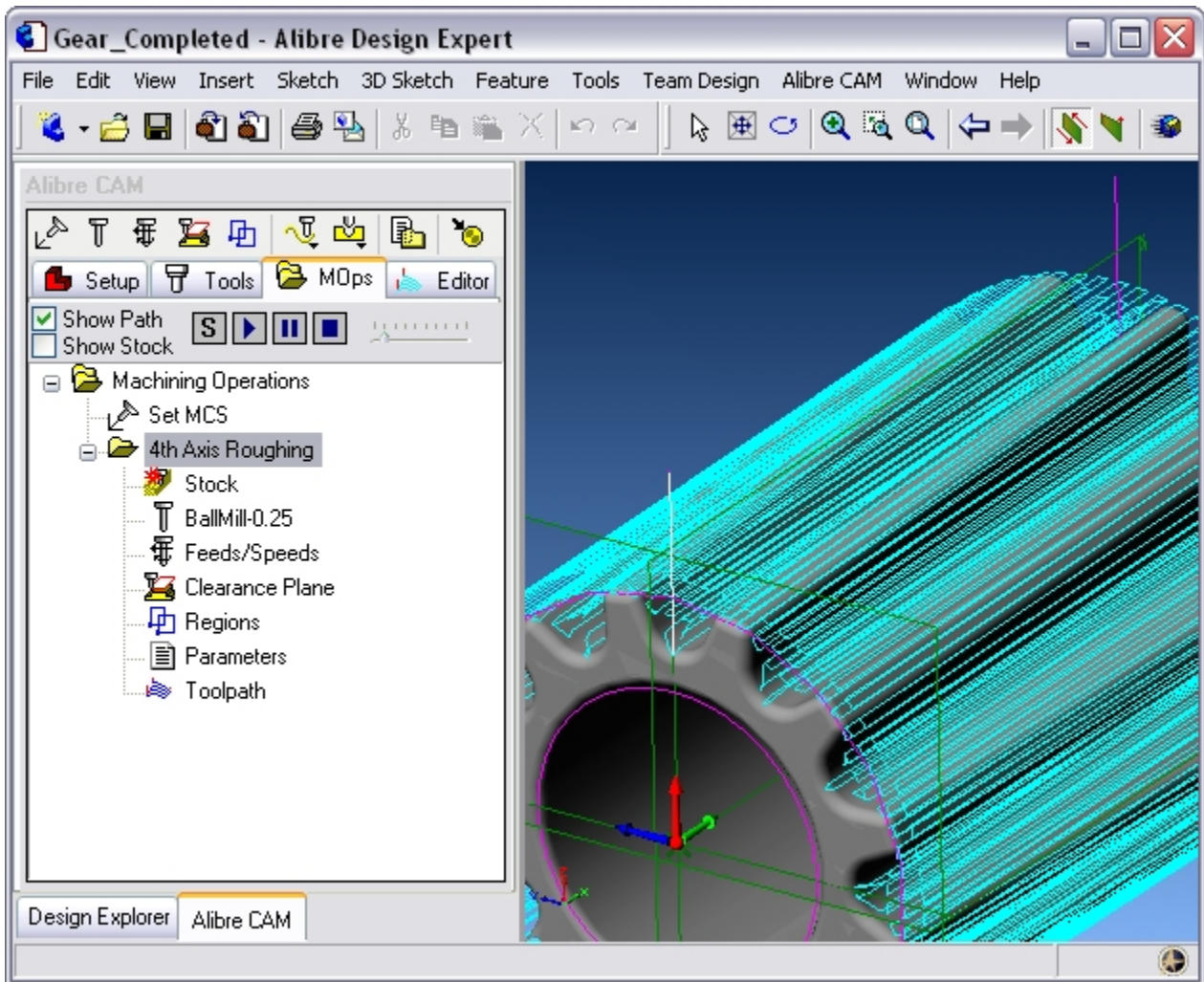
2. In the Cut Parameters tab Set the Intol and Outol = **0.001**, Stock to leave = **0.005**, Cut Pattern to **Along Axis, Zig Zag** and **Low to High**, Cut Containment Low Value = 0 and High Value = 3 (as the Machine Zero is set to the left edge of the stock/ part). Use Step Distance = 25 (% of tool diameter).



3. Switch to the Step Down Control Tab.
4. Use the Following Settings:
  - a. Cut Levels: check Top (T) and specify Top (T) = 1.1 (radius of stock material)
  - b. Step Down Control = 50 (% Tool Diameter).




5. Click **Generate**. The 4th axis Roughing toolpath is now generated, and the operation is listed under the Alibre CAM MOps browser.

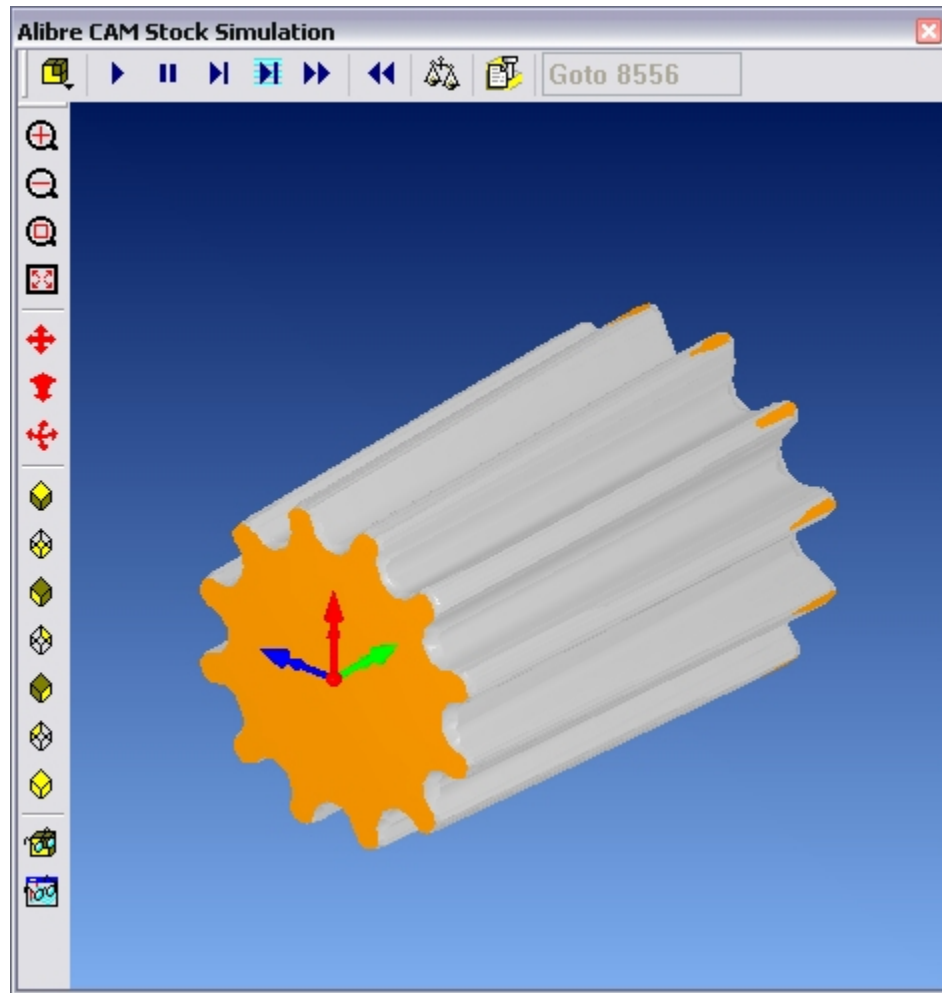


## Simulate Toolpath

**Note:** You must switch the simulation model to Polygonal model to run 4 axis simulations. Select Preferences->Simulation Preferences from the Setup Tab and switch the simulation model to Polygonal if set to Voxel.

Make sure Show Stock is selected on the MOPs Browser.

1. Select the 4<sup>th</sup> Axis Roughing 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 4<sup>th</sup> Axis Finishing Toolpath

We will now use 4th Axis Finishing operation to finish the part using a 0.125" Ball End Mill, We will use the same settings for Feeds / Speeds and Clearance Plane.

In this method, the tool is always normal to the axis of table rotation (continuous mode). The tool motions can be parallel to or normal to the rotation axis.

From the Create Operations tab, select 4 axis Milling -> 4 Axis Finishing.

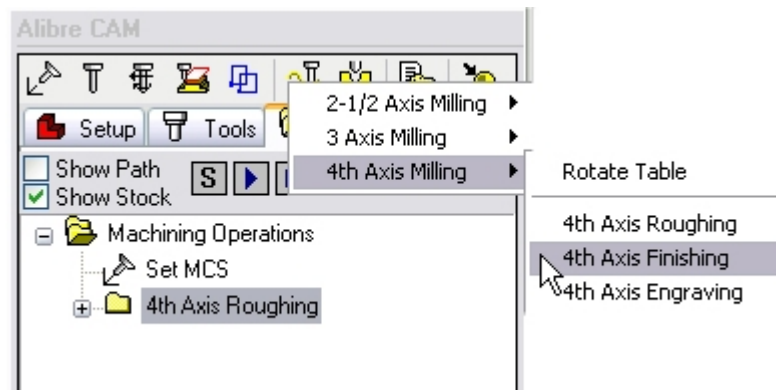
## Select Tool

From the **MOPs** tab:

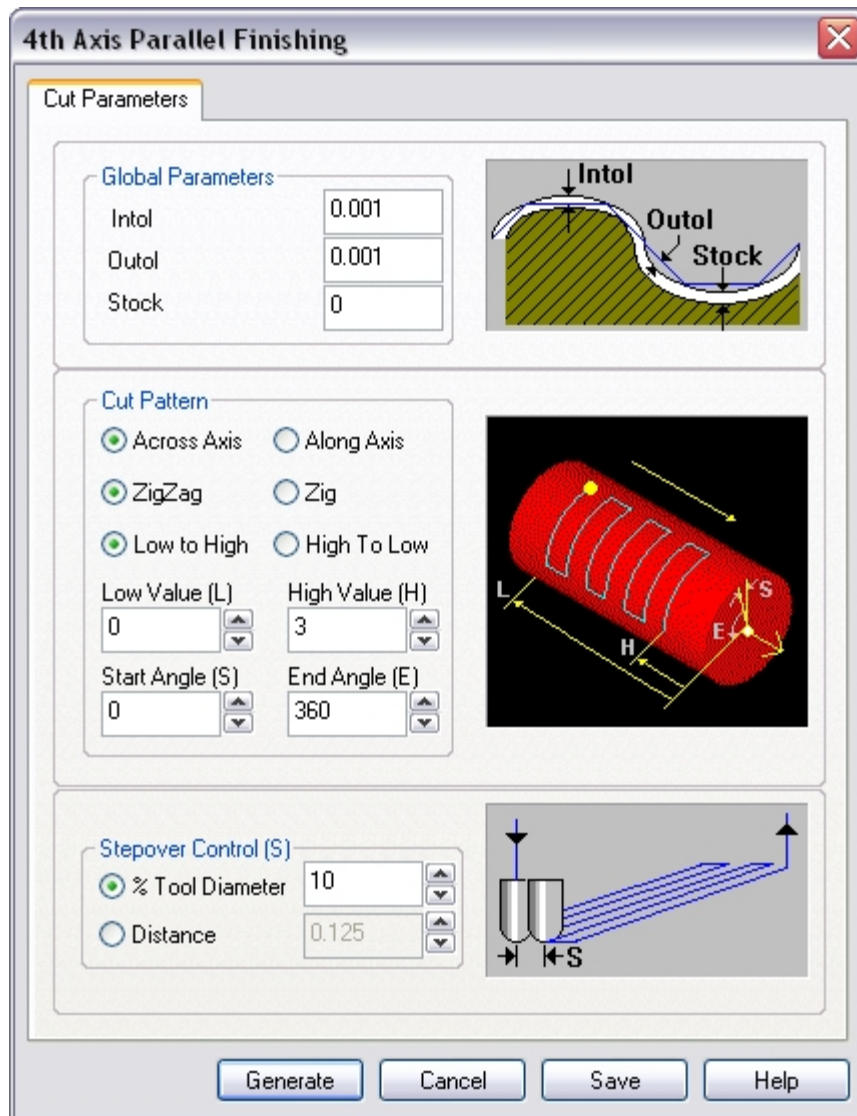
1. Open the **Select/Create Tools** window
2. Click the **Ball End Mill** tab, and select **BallMill-0.125**. Click OK to make this the active tool. (You could also double-click the tool name.)

## 4<sup>th</sup> Axis Finishing

1. From the MOps tab, Select Milling Methods -> 4<sup>th</sup> Axis Milling -> 4<sup>th</sup> Axis Finishing

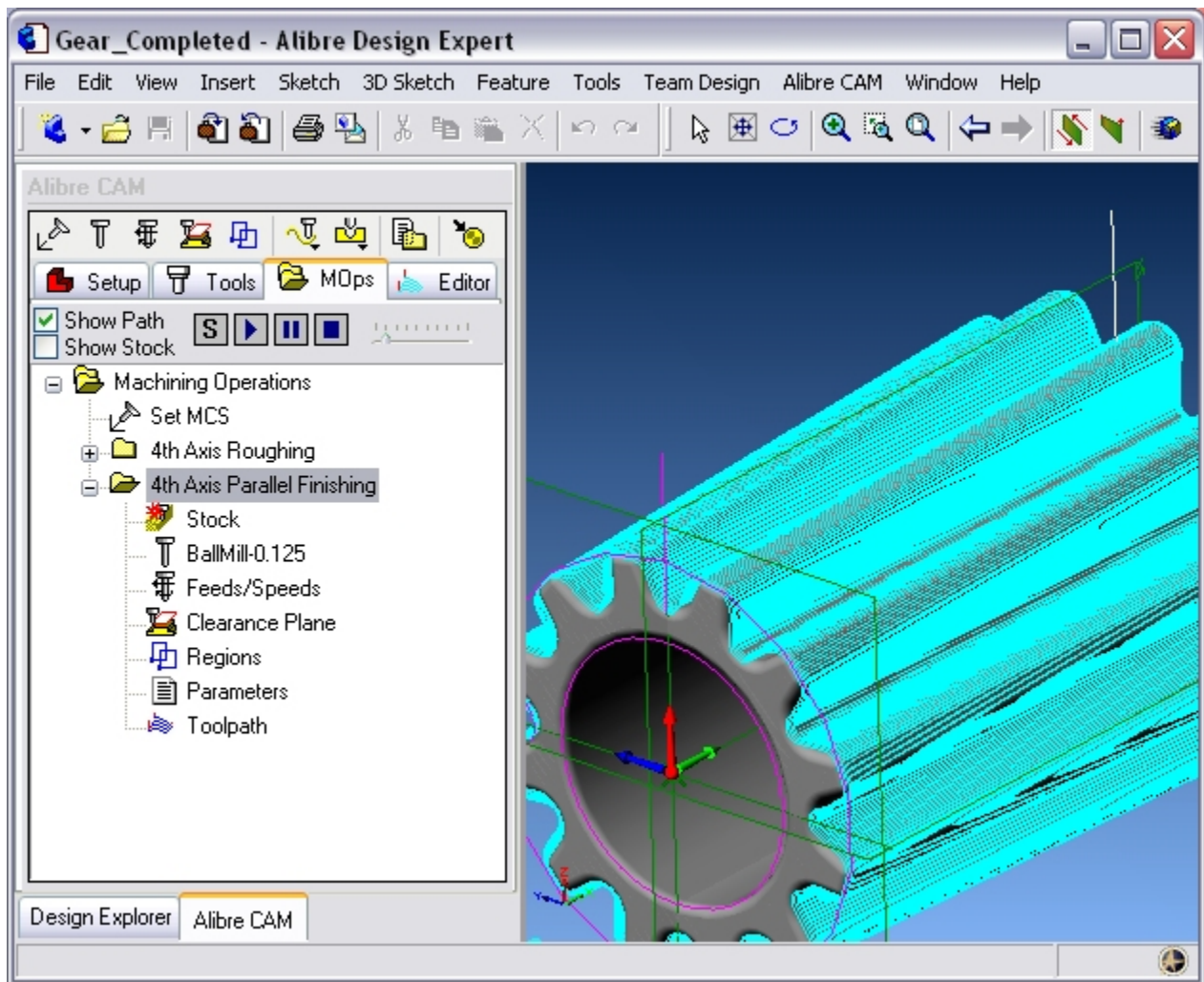


2. In the Cut Parameters tab Set the Intol and Outol = **0.001**, Stock to leave = **0**, Cut Pattern to **Across Axis, Zig Zag** and **Low to High**, Cut Containment Low Value = 0 and High Value = 3 (as the Machine Zero is set to the left edge of the stock/ part). Use Step Distance = **10** (% of tool diameter).




3. Click **Generate**. The 4th axis Finishing toolpath is now generated, and the Operation is listed under the Alibre CAM MOPs browser.

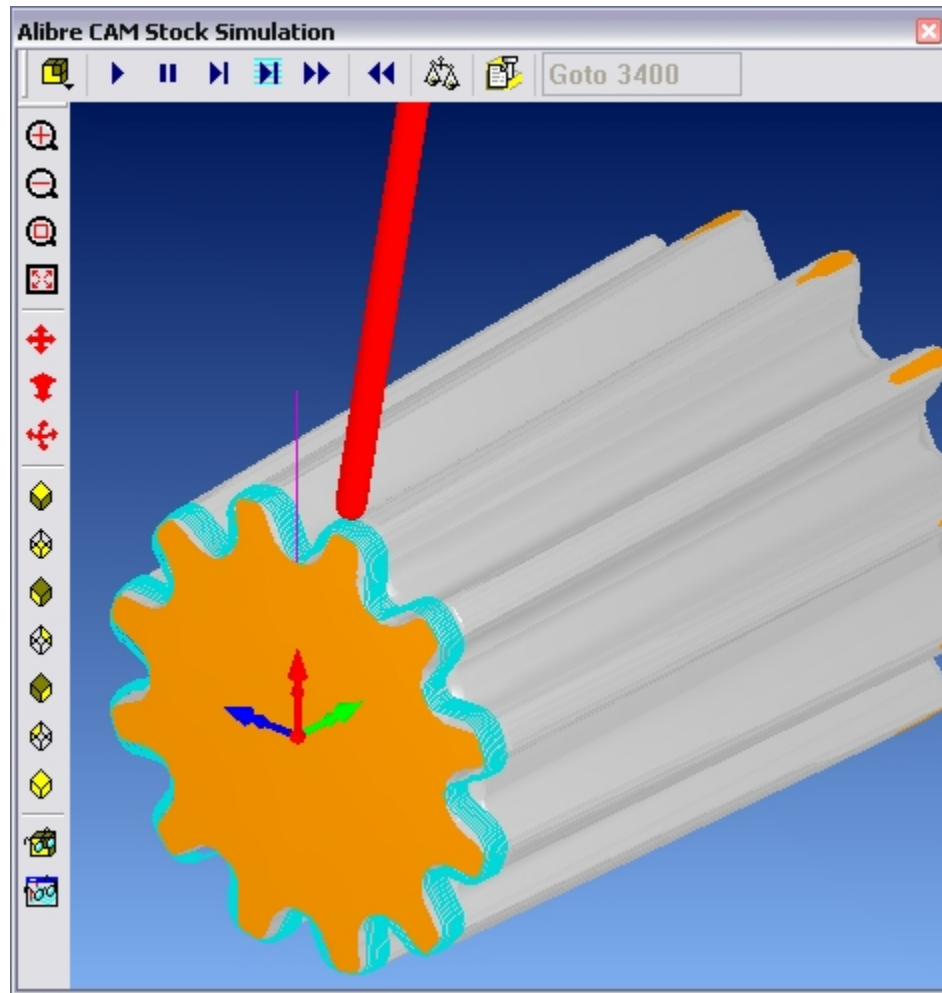




## Simulate Toolpath

1. Select the 4<sup>th</sup> Axis Finishing Operation and click  to launch the Alibre CAM Stock Simulation window.
2. Click Simulate from the Stock Simulation window to run simulation.



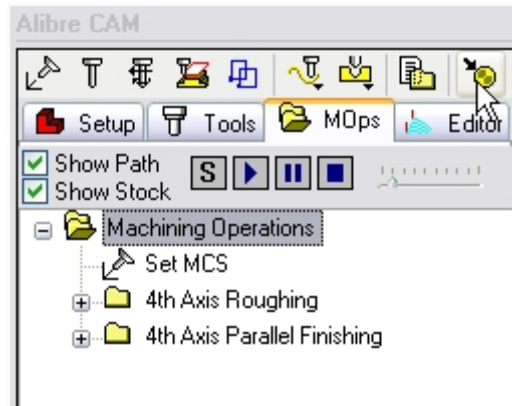


3. Once the simulation is complete, you can close the Stock Simulation window and return to the Alibre CAM browser.

## Post Processing

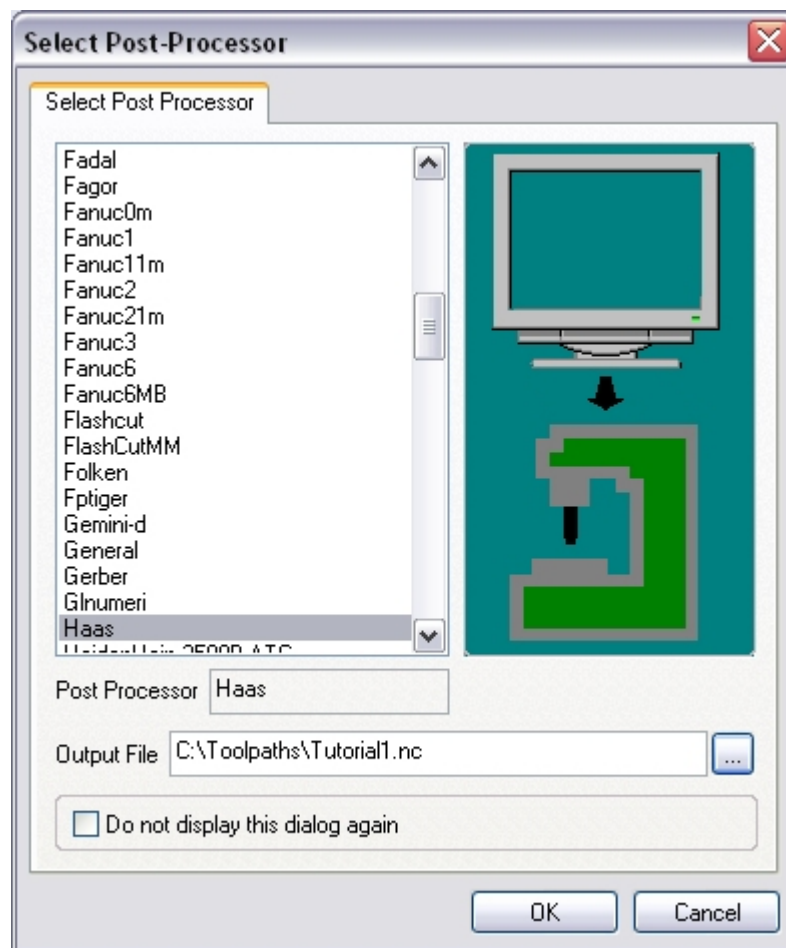
Once the toolpath has been generated, it can be post-processed to a specific machine controller. Alibre CAM Standard comes with a set of post-processors to choose from. Each post-processor is represented by an \*.spm file, all of which are located in the **Posts** folder under the Alibre CAM installation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

4. In the **MOps** tab, select **Machining Operations** (located at the root level in the Browser window) and click **Post Process**.



This displays the Select Post Processor Dialog.

5. Browse to the desired output directory and assign a file name for the output. Make sure that this file name is a valid file name and you have permissions to write to the specified file. Then double-click on the post-processor you want to use (such as Haas).



6. When complete, the post output file will open in the default text editor (Notepad by default). This file contains all the G-code for your toolpaths.