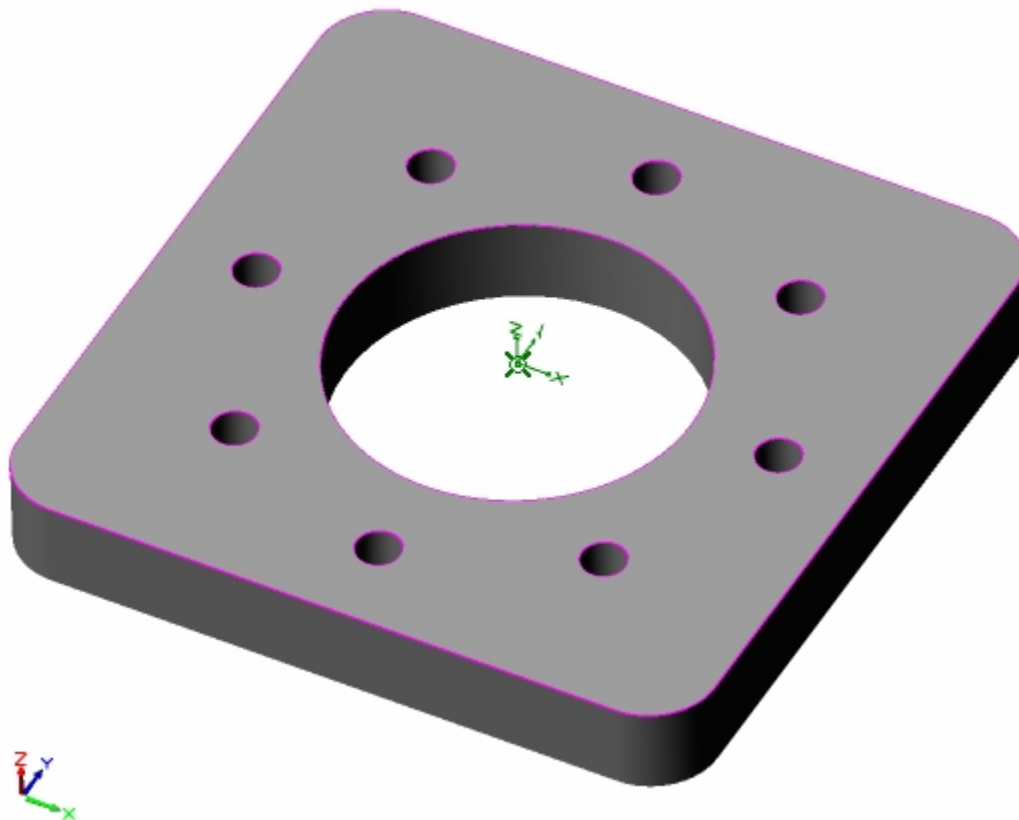


## Tutorial 5: Hole Making

---



Using Drilling Toolpaths

## Introduction

This tutorial will illustrate machining holes using Drilling operation. Even though we have created a 3-D representation of the part, it will be seen later on that we can machine this using just 2-D curves.

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 part

- We will machine part by using a Hole machining operation called Drilling.
- The part will be machined out of an 8 x 8 x 1 inch poplar wood sheet using a 0.5" Standard Drill bit.
- The wooden sheet will be held to the machine table or the spoil sheet on the table using clamps.

## Main Programming Steps

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

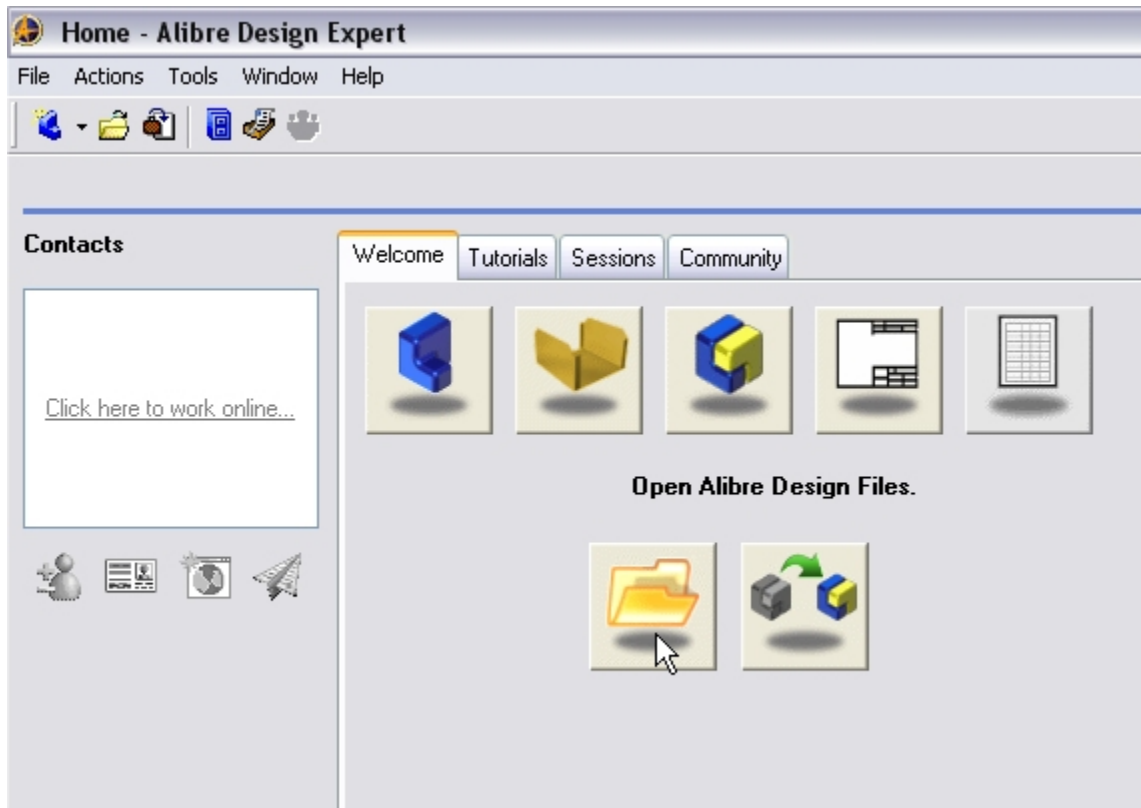
- Create the Stock geometry
- Set the Machine Zero point or locate geometry with respect to the machine coordinates
- Create / Select the tool used for machining
- Set the feeds and speeds
- Set the Clearance Plane for the non-cutting transfer moves of the cutter
- Select the machining regions for containing the cutter to specific areas
- 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.

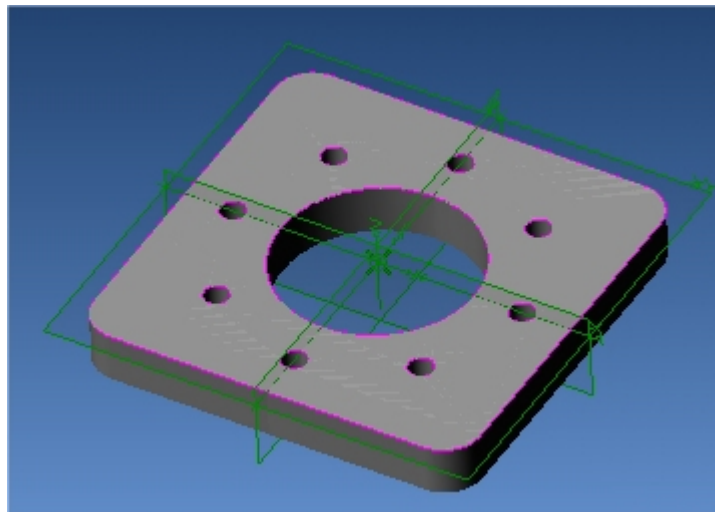
## Loading the Part Model

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.



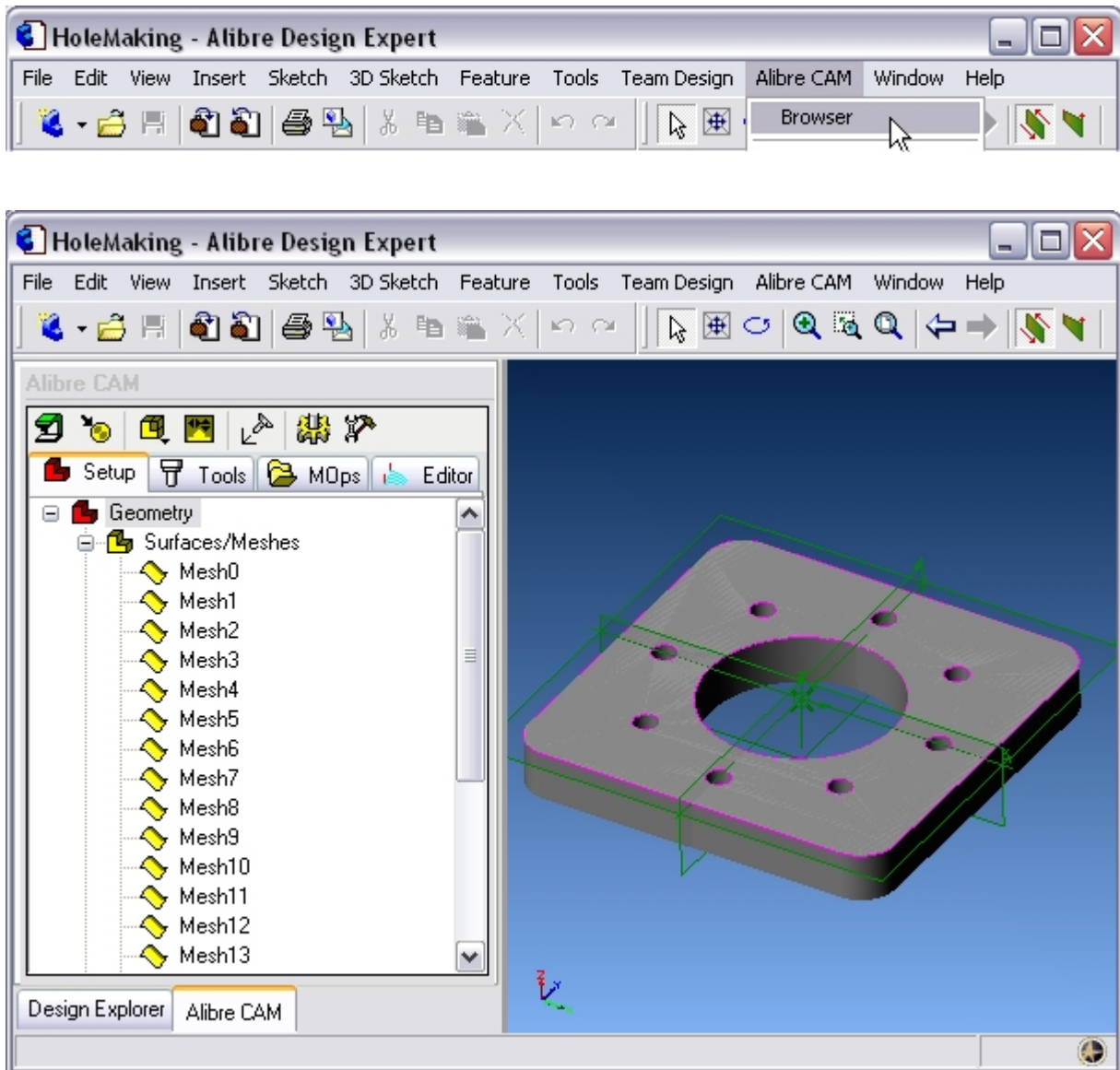
2. From the **Open** dialog box, select the **HoleMaking.AD\_PRT** file from the **Tutorials** folder in the Alibre CAM installation folder. (Default location C:\Program Files\MecSoft Corporation\Alibre CAM\Tutorials)
3. The loaded part appears as shown below.



**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

1. 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.

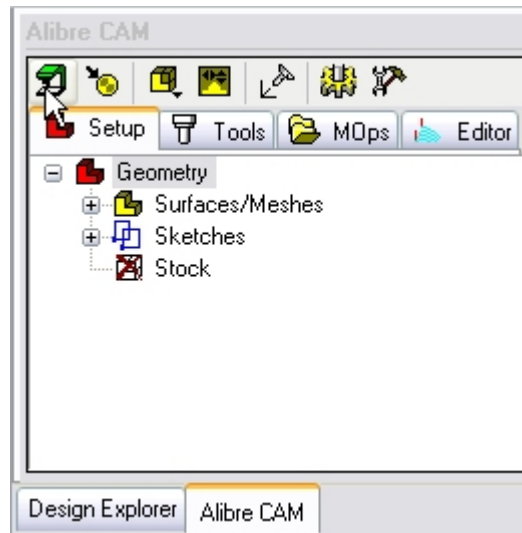
## Preparing the Part for Machining

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**.

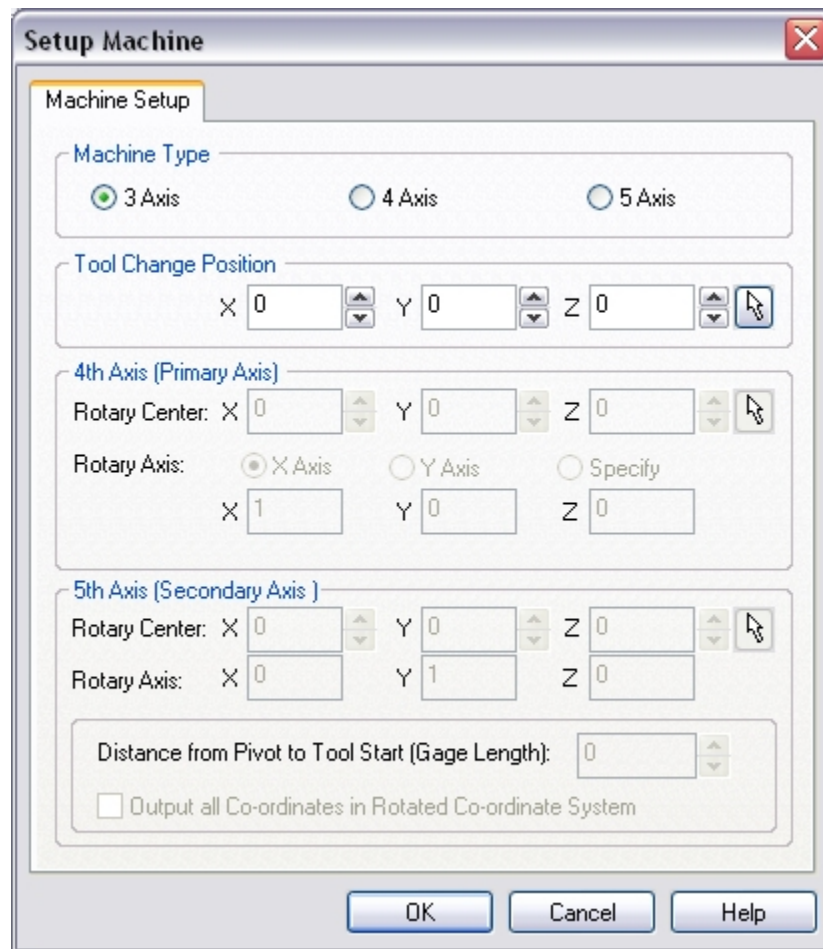
The Setup tab allows the user can specify Machine Setup, Stock Geometry, Machine Coordinate System (Machine Zero) & Alibre CAM Preferences.

## Setup Machine

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

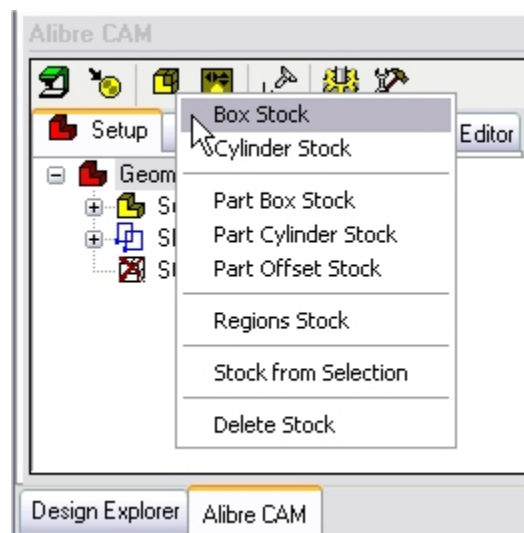


2. Set the Machine Type to 3 Axis. Click OK to close the Setup Machine Dialog.

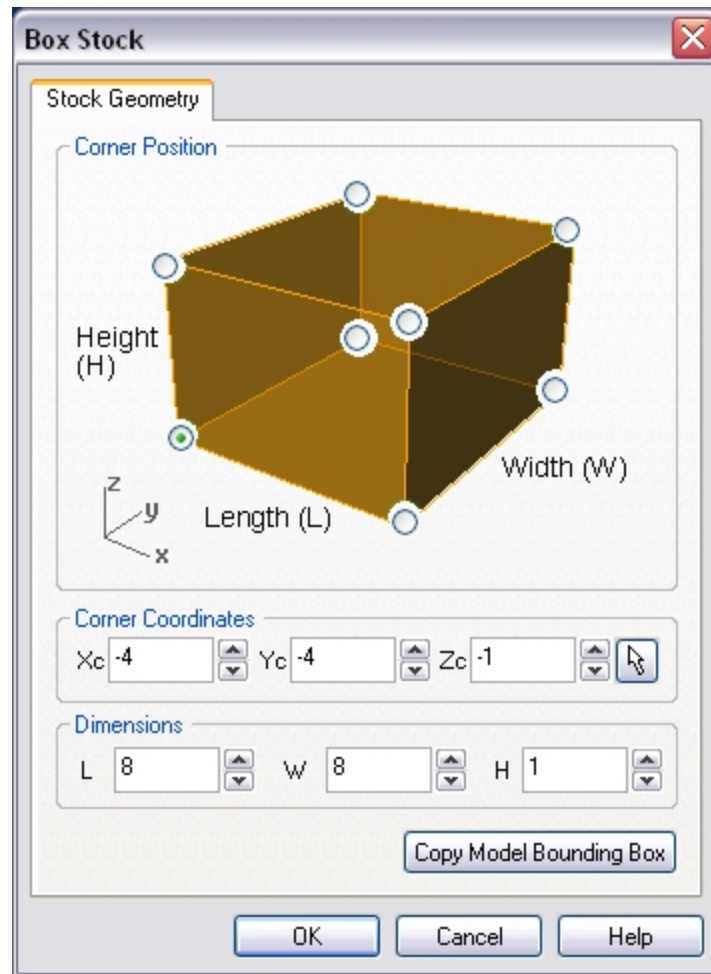


## Create/Load Stock

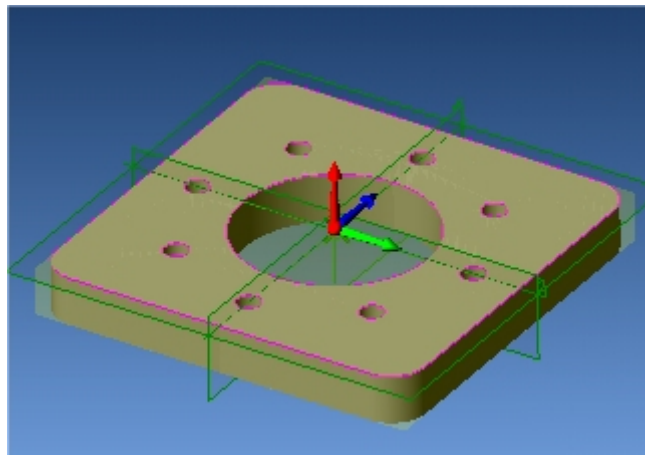
3. Select Create / Load Stock from the Setup tab and choose Box Stock.



This brings up the Box Stock parameters.



4. Use the following Dimensions. Length = 8, Width = 8 and Height = 1. Click OK to create the stock geometry.
5. The stock geometry is now created, and a semi-transparent stock box is displayed on top of the part geometry.

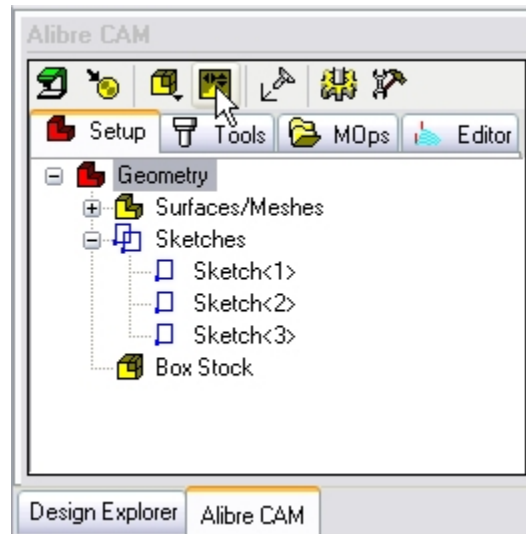


**Note:** If the Stock Geometry is not visible, Switch to the MOps tab and Make sure Show Stock is selected.

## Locate Part Within the Stock

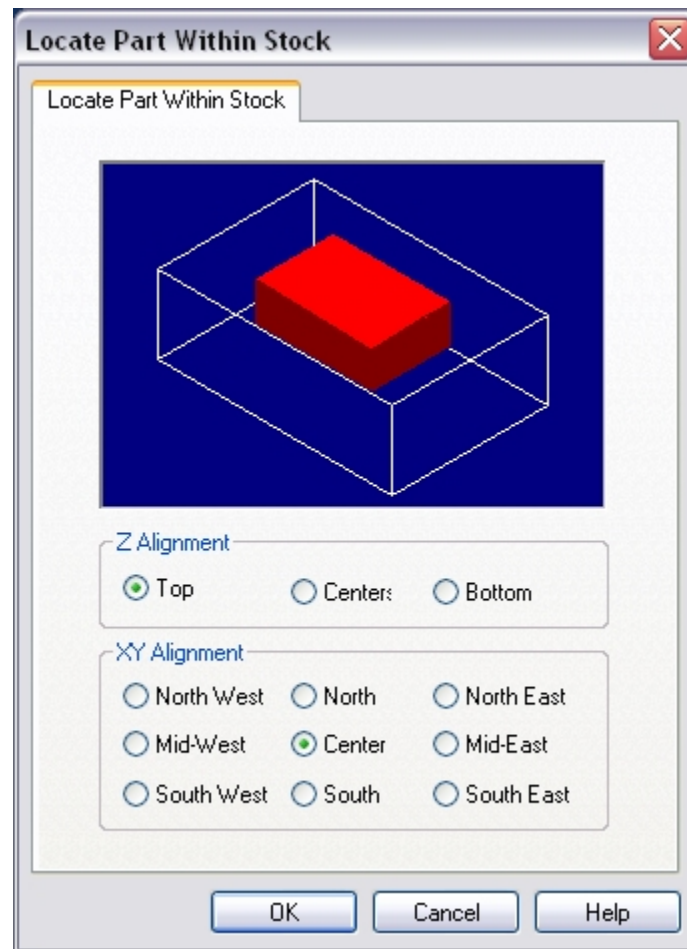
Once the stock model is created, you can move the stock geometry relative to the part geometry and use the stock box to specify the Machine Zero (home position).

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

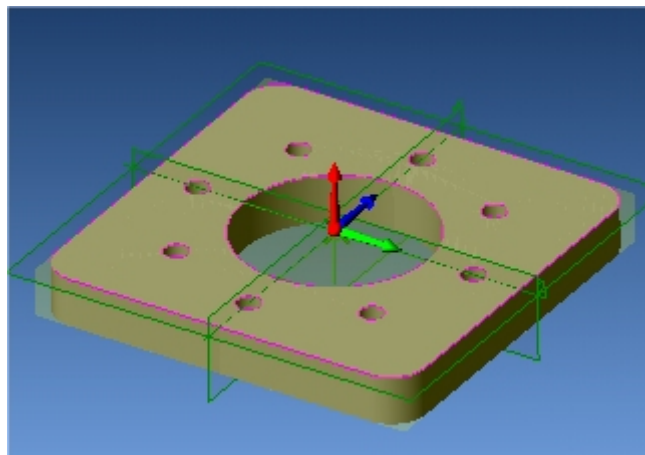


7. Set the Z alignment to Top and XY alignment to Center. (This would align the stock to the top of the part in Z and center in XY)





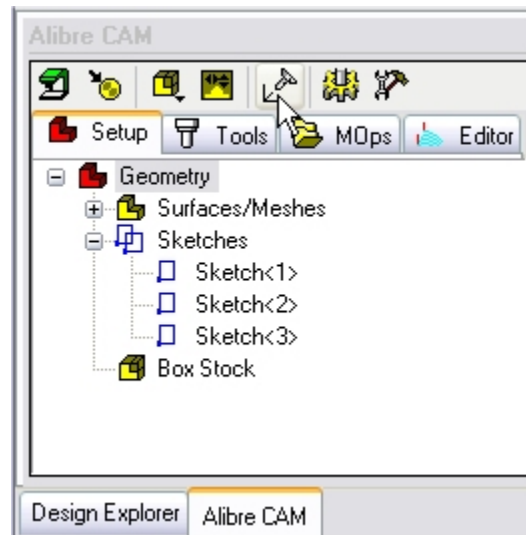
The stock geometry is now aligned to the part and displayed as shown below.



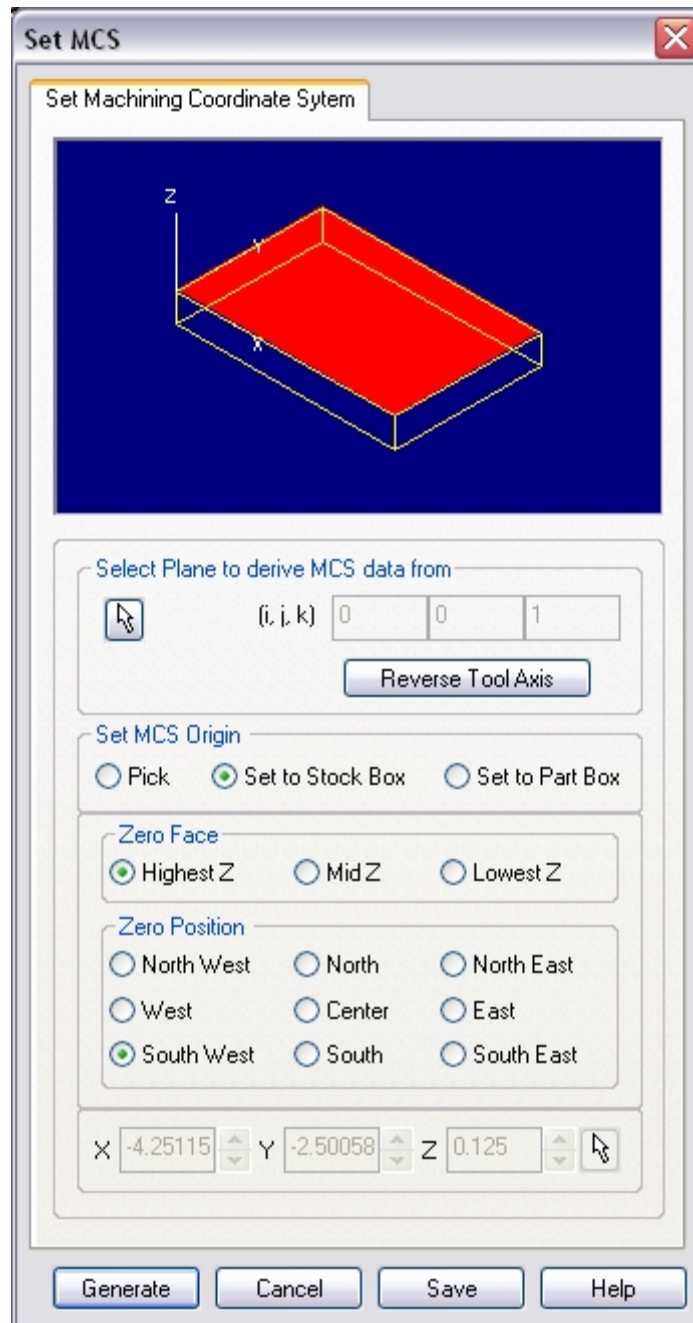
## Set MCS

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.

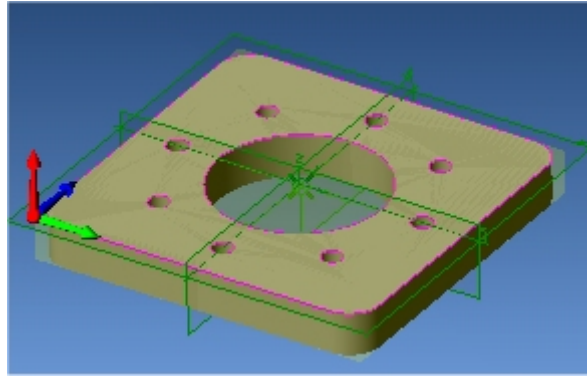
8. Select Set MCS from the Setup tab.



9. Set the MCS Origin to Stock Box, Zero Face to Highest Z and Zero Position to Southwest Corner.



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



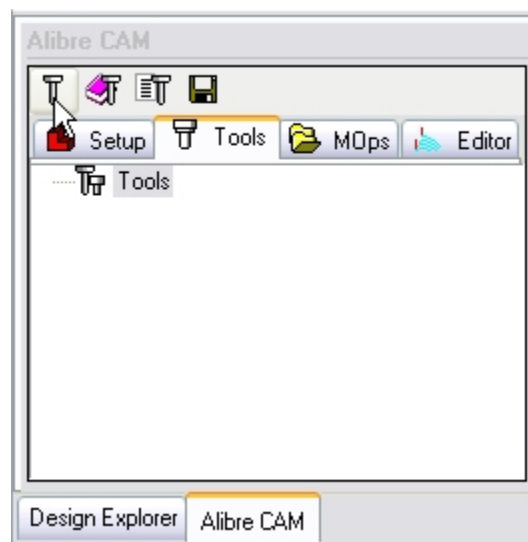
## Defining Cutting Tools

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") Standard Drill.

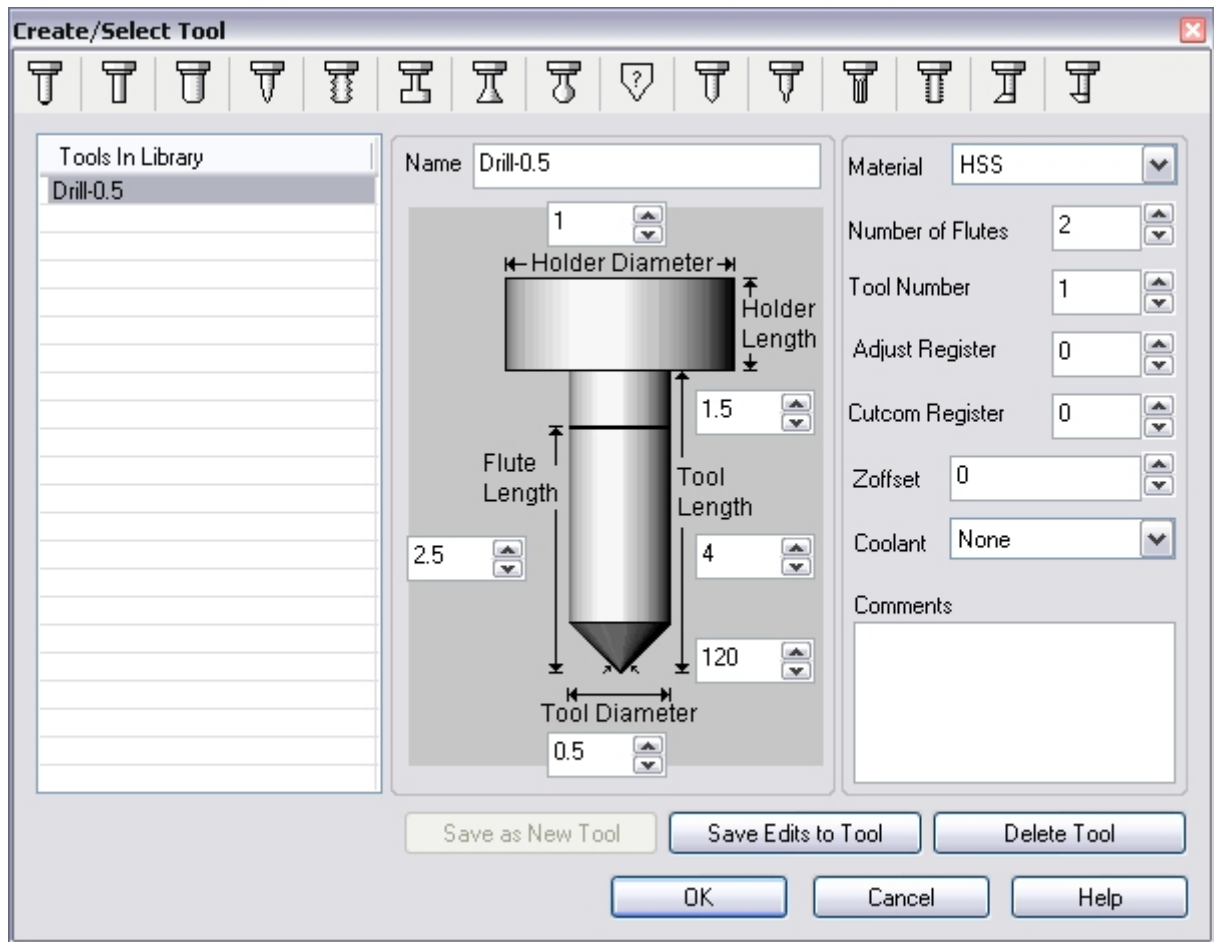
1. Select Create / Select Tools from the Tools tab



2. Set the Tool Type to Standard Drill

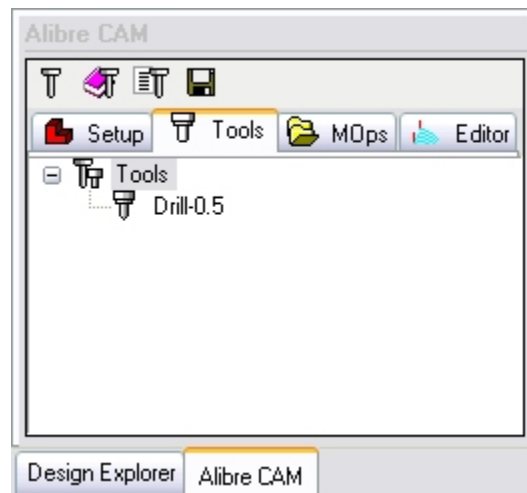


3. Set the tool name as **Drill-0.5**, Tool Diameter = **0.5**, Tool Number = **1**.



4. Click Save as New Tool. Click OK to close the Create/Select Tool Dialog.

The tool is now created and listed in the Tools tab under the Alibre CAM browser.

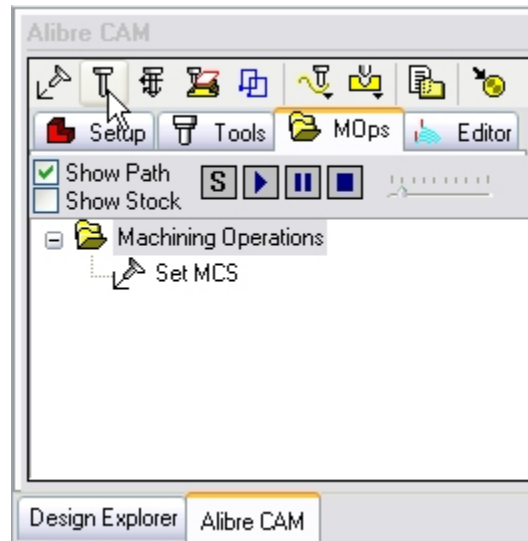


## Creating the Drilling Toolpath

"MOps" stands for Machining Operations. All machining operations you create are listed here, in order of creation.

### Select Tool

1. Select Create/Load Tool from the Mops tab.

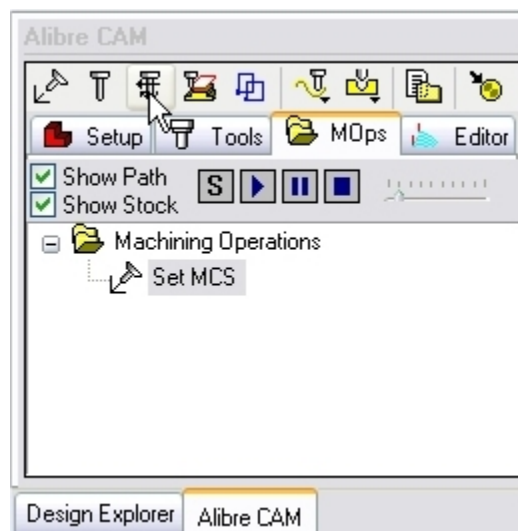


2. Select Drill-0.5 from Tools in Library and Click OK to set this as the active tool.

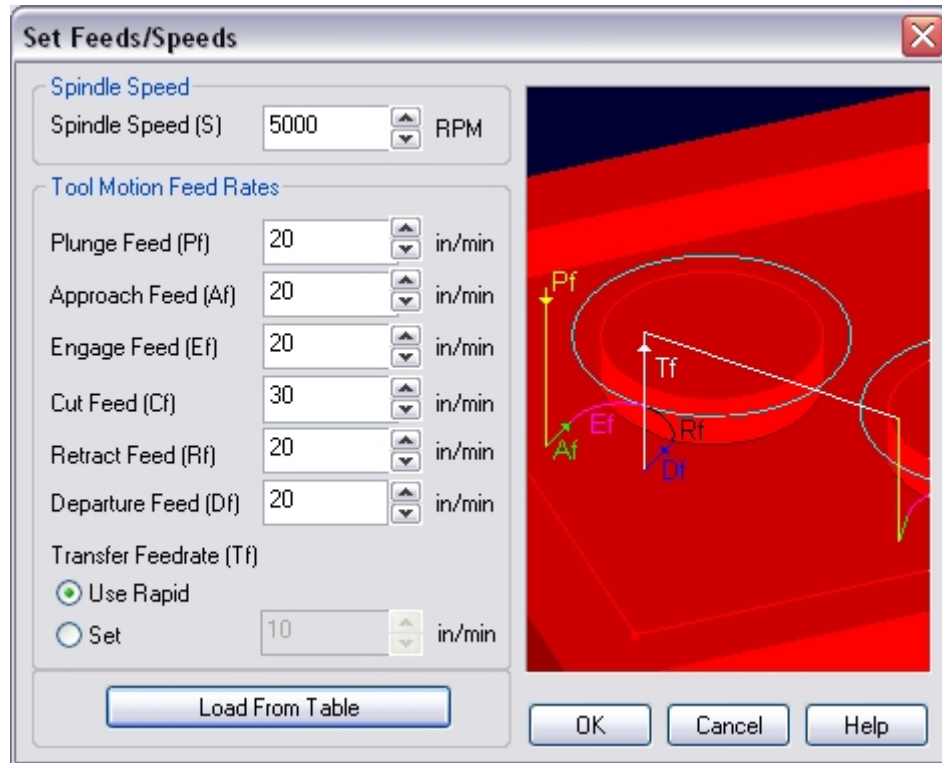
### Set Feeds / Speeds

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.

3. Select Set Feeds/Speeds from the MOps tab.



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



5. Click OK to accept the settings.

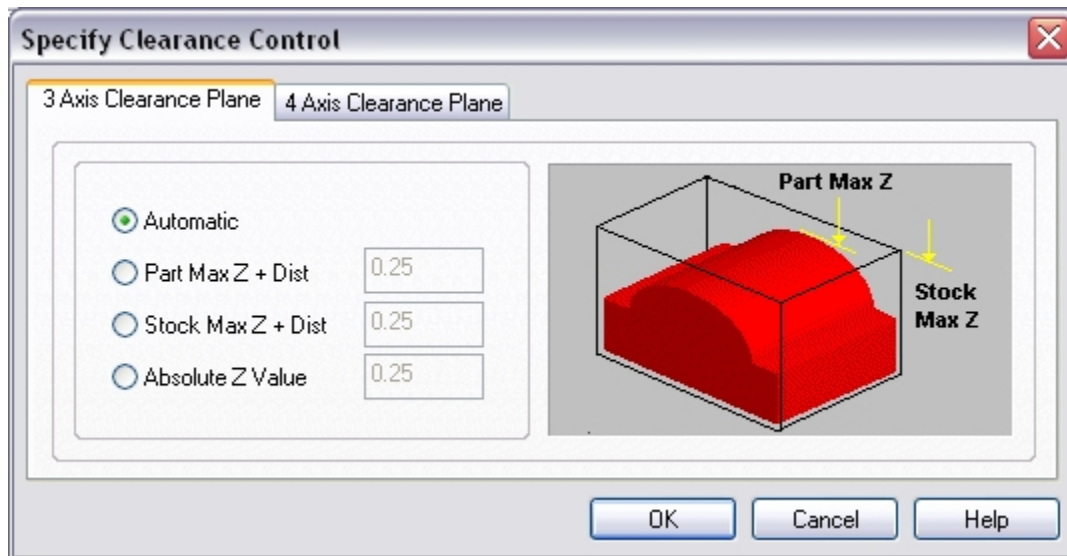
## Clearance Control

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.

6. Select Clearance Control from the MOps tab.



7. Set the Clearance Plane to Automatic.

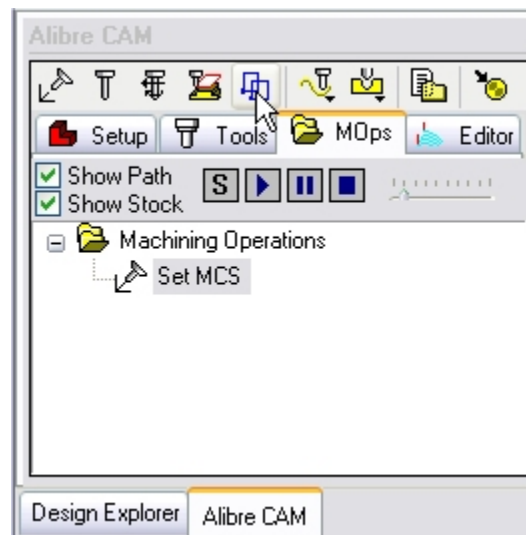


## Select Regions

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.

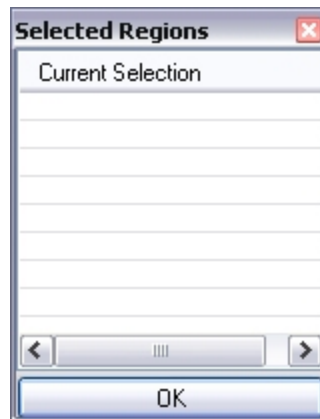
Regions must be selected before they can be used in an operation. Creating a region does not make it active; you must use one of the **Select Regions** tools before creating the toolpath.

1. Click Select Regions from the MOps tab.



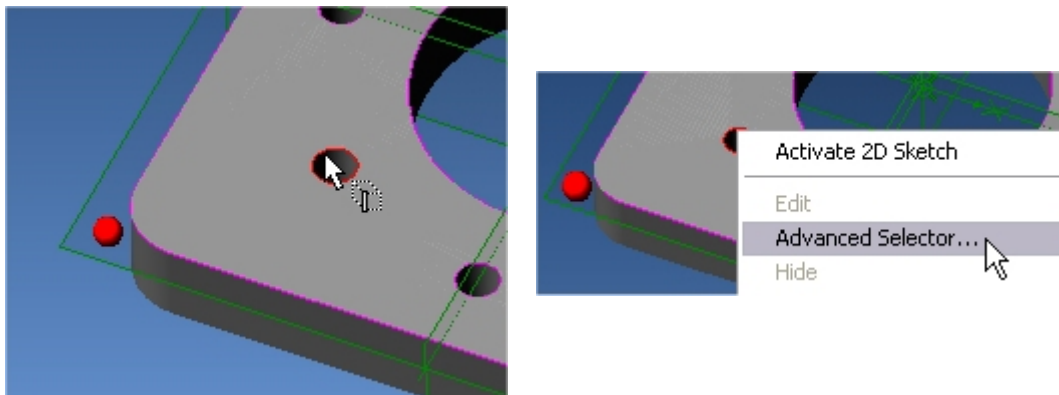
This brings up the Select Regions dialog.



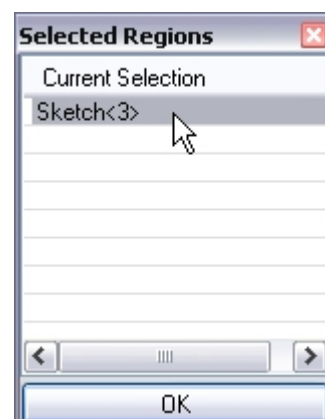
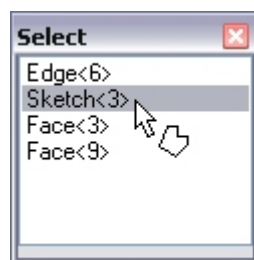


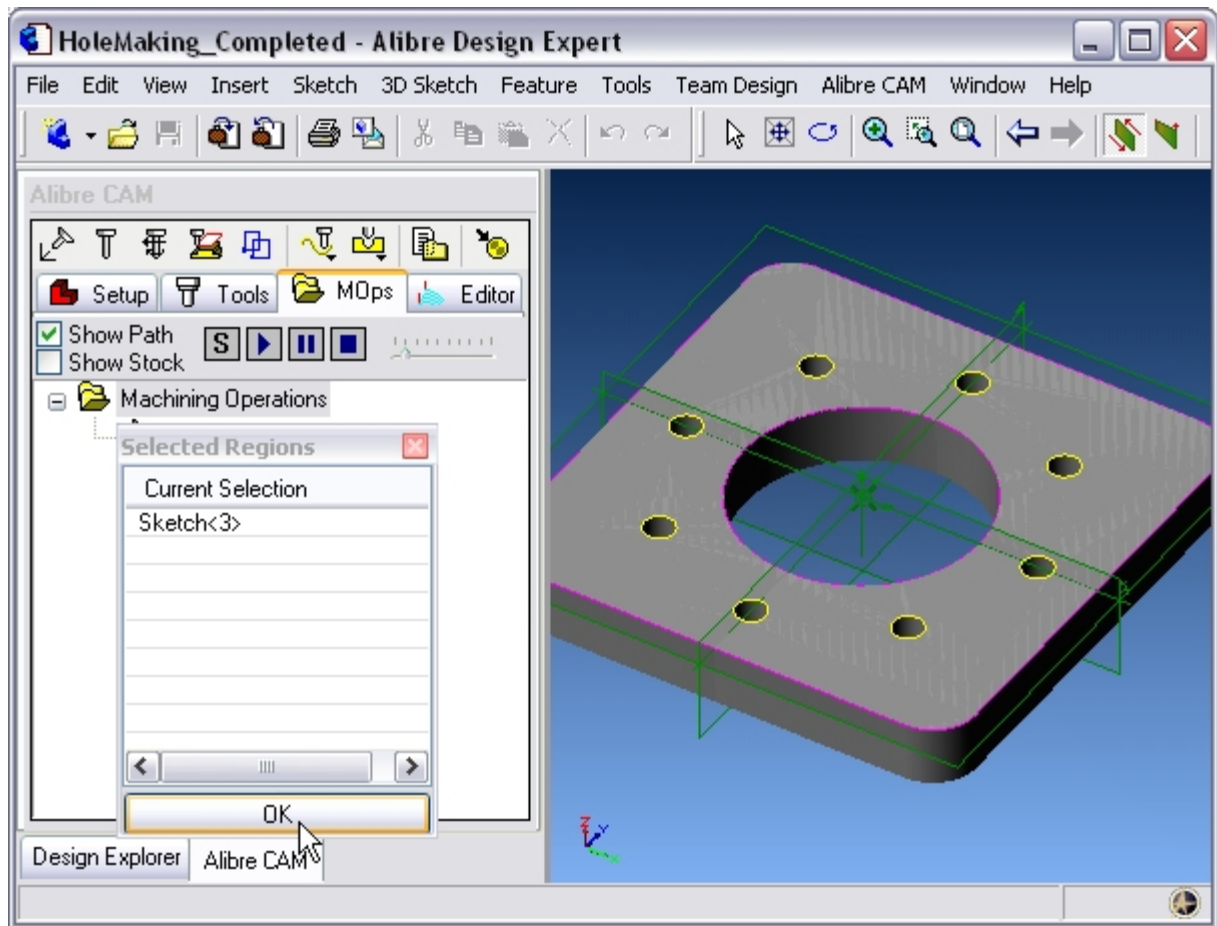
2. Select Sketch <3> as machining region.

- a. To select Sketch <3> Move the cursor over the hole, right click and choose Advanced Selector.



- b. Select Sketch<6> from the selection list. The Sketch<6> is now listed under Selected Regions.



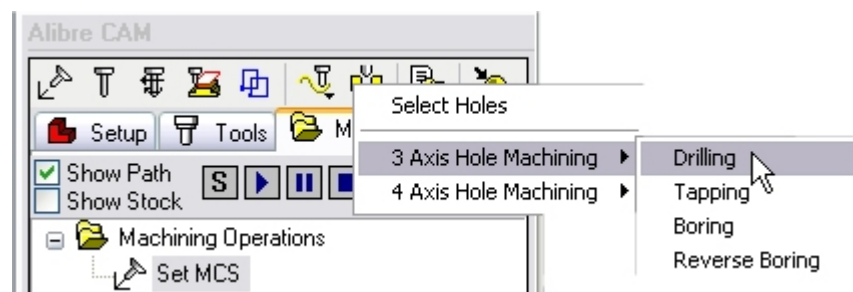


c. Click OK to accept the selected regions.

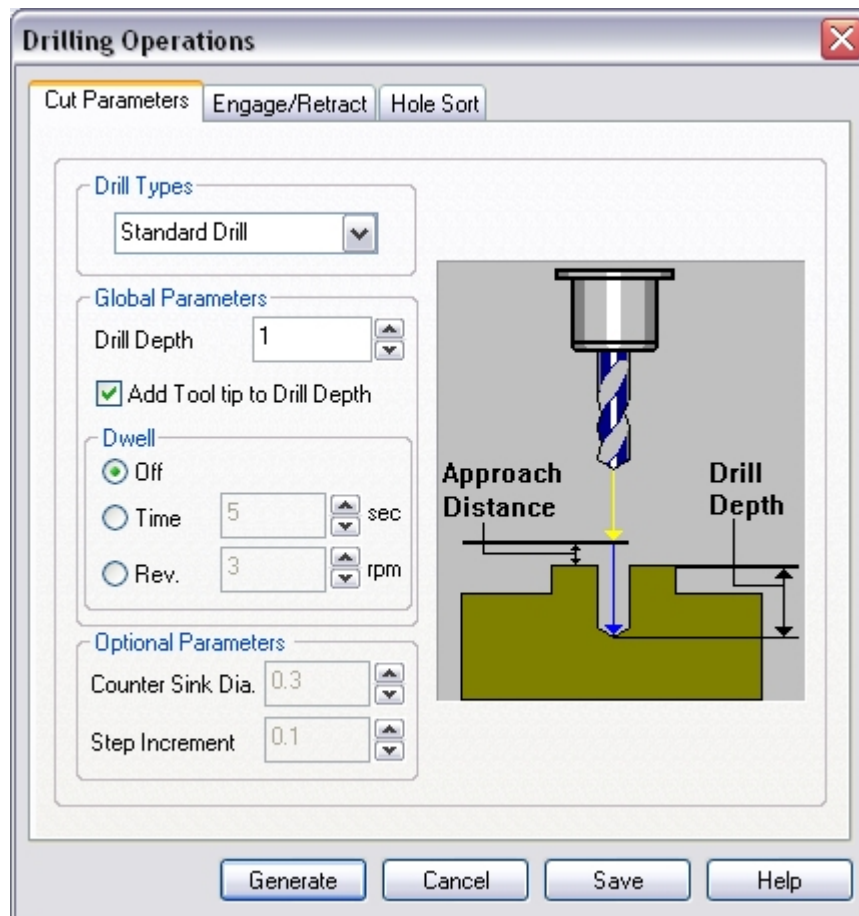
**Note:** If you are unable to select a Sketch, from the Alibre Design Menu bar select Tools->Selection Filters and ensure 2D Sketches are selected. Keeping this unchecked will not allow the user to select Sketches.

## Drilling

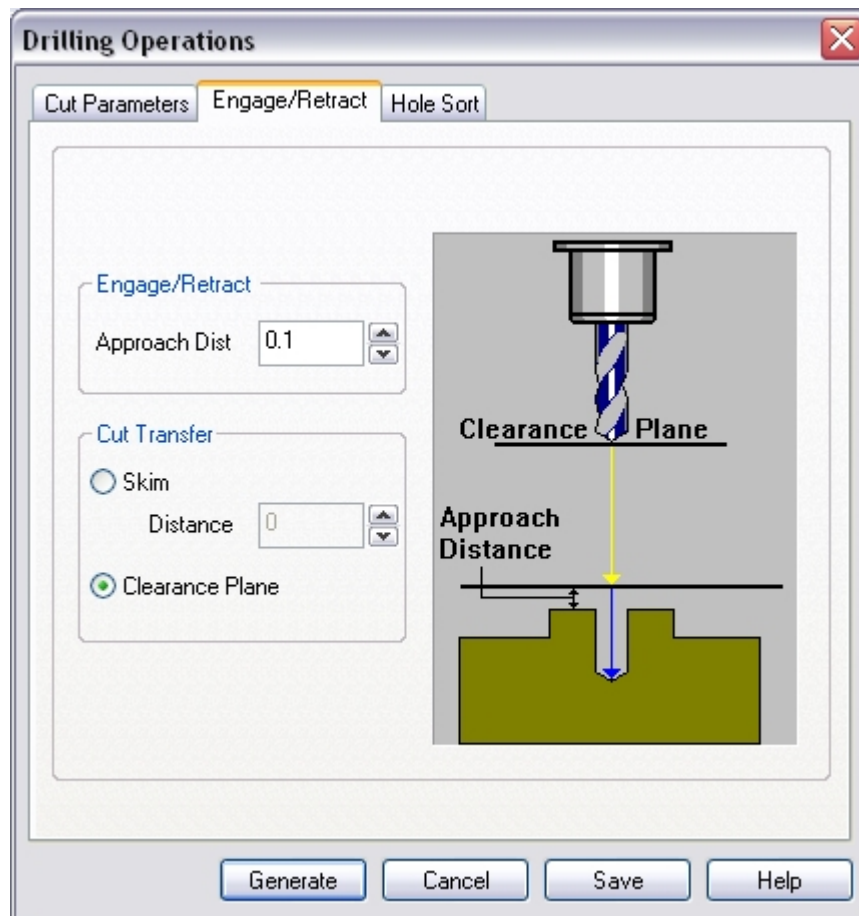
1. Select Hole Machining -> 3 axis Hole Machining -> Drilling



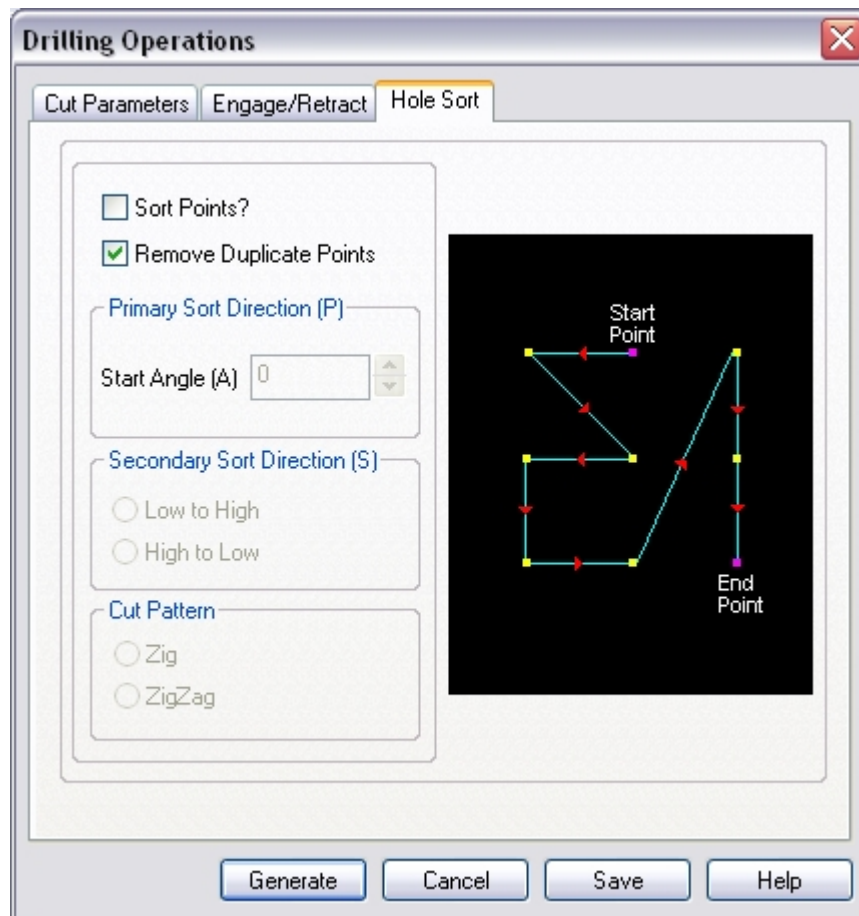
2. In the Cut Parameters tab, under drill types choose **Standard Drill**, Drill Depth = **1**, Select Add Tool tip to Drill Depth.



3. Switch to the Engage/Retract tab and set the Approach Distance = 0.1, and Cut Transfer to Clearance Plane.

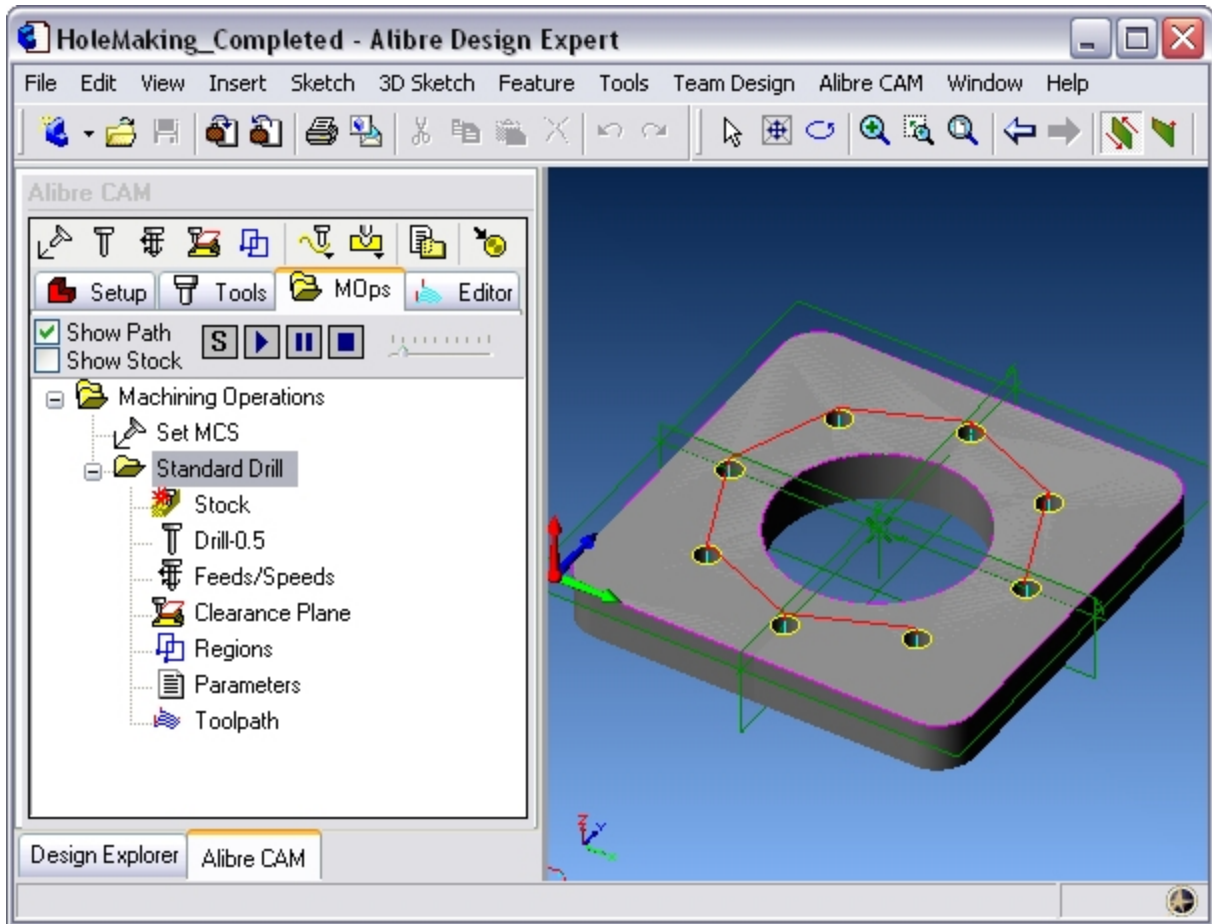


4. Switch to the Hole Sort tab and leave the Hole Sort parameters at default settings.



5. Click **Generate** to create the Standard Drill toolpath.

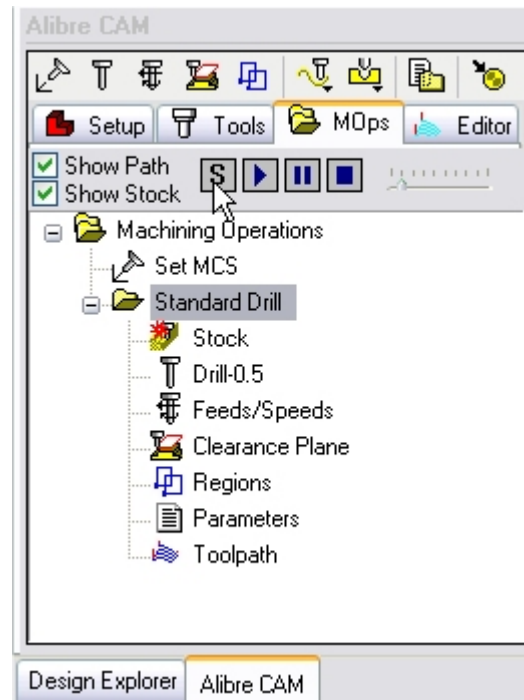
The Drilling Operation is now displayed under the MOps Browser and the Toolpath is displayed as shown below.



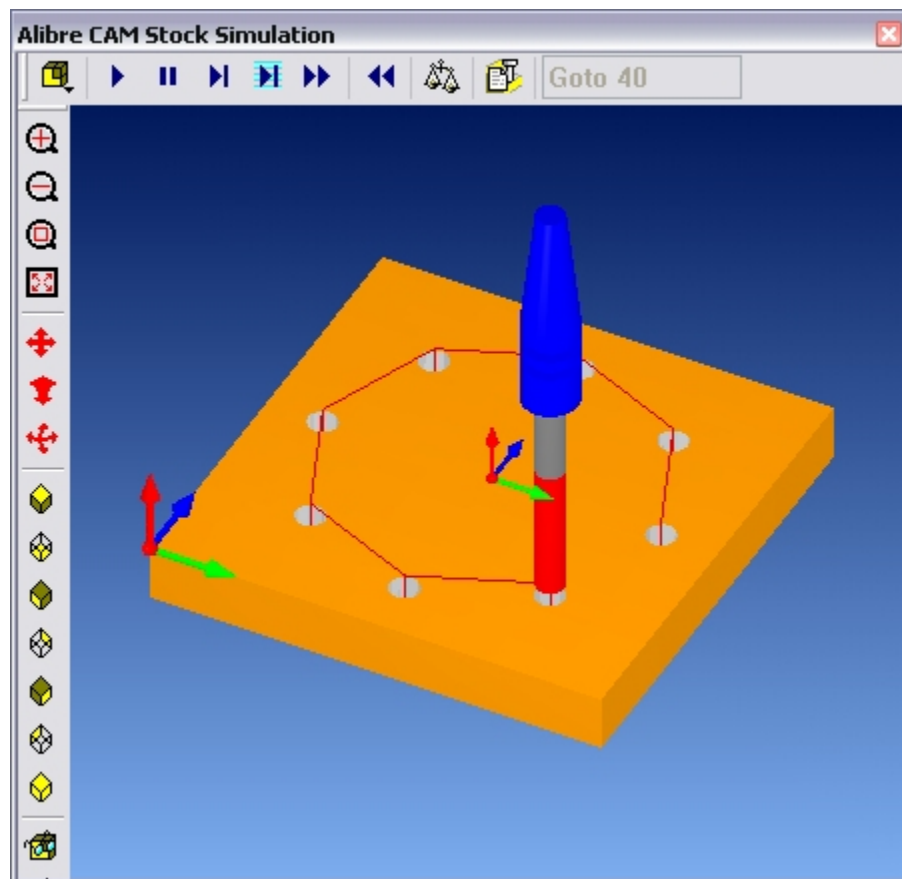
## Simulate Toolpath

Make sure Show Stock is selected on the MOps Browser.

1. Select the Standard Drill Operation and click **S** to launch the Alibre CAM Stock Simulation window.



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

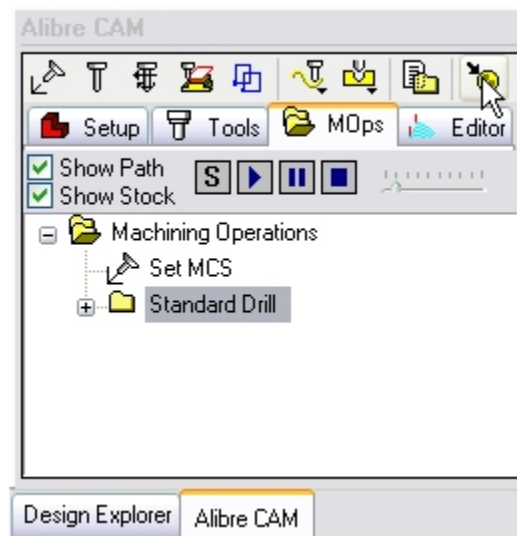


Note: You can pause/stop, step, and skip the simulation using the controls available on the Simulation window. To control the simulation speed, select Simulation Settings from the Stock Simulation window.

## 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.

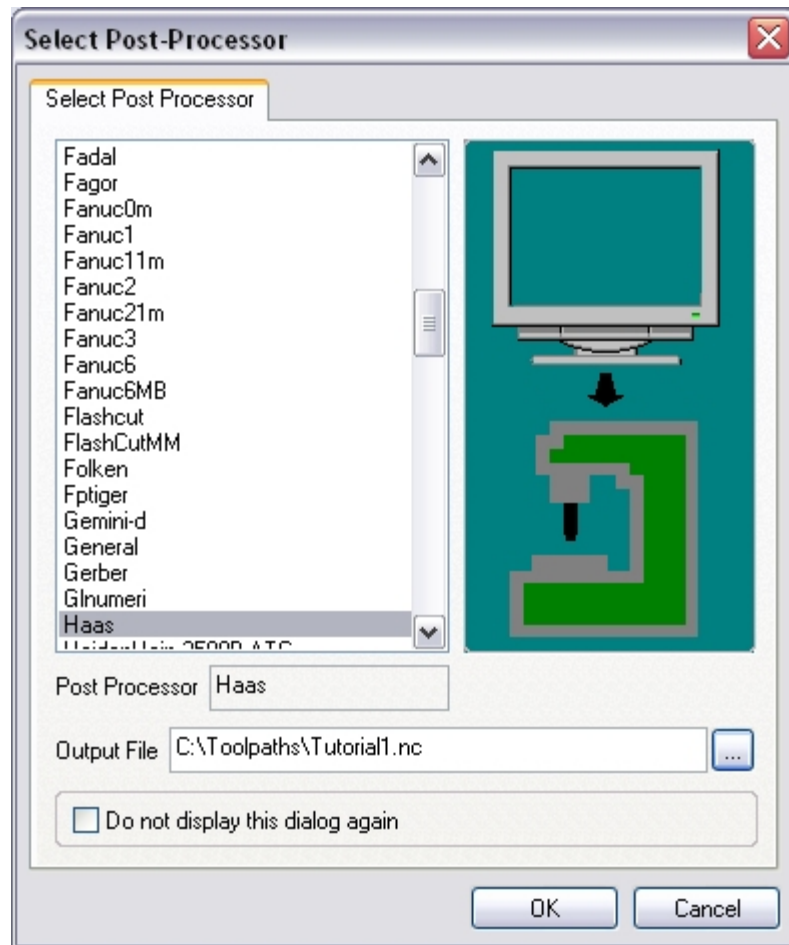
1. In the **MOps** tab, select **Standard Drill** and click **Post Process**.



This displays the Select Post Processor Dialog.

2. 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).





3. 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.

## Lista Studio srl

Borgo Belvigo 33  
36016 Thiene Vi  
ITALY  
Tel. 0445,382056  
[www.lista.it](http://www.lista.it)  
[info@lista.it](mailto:info@lista.it)