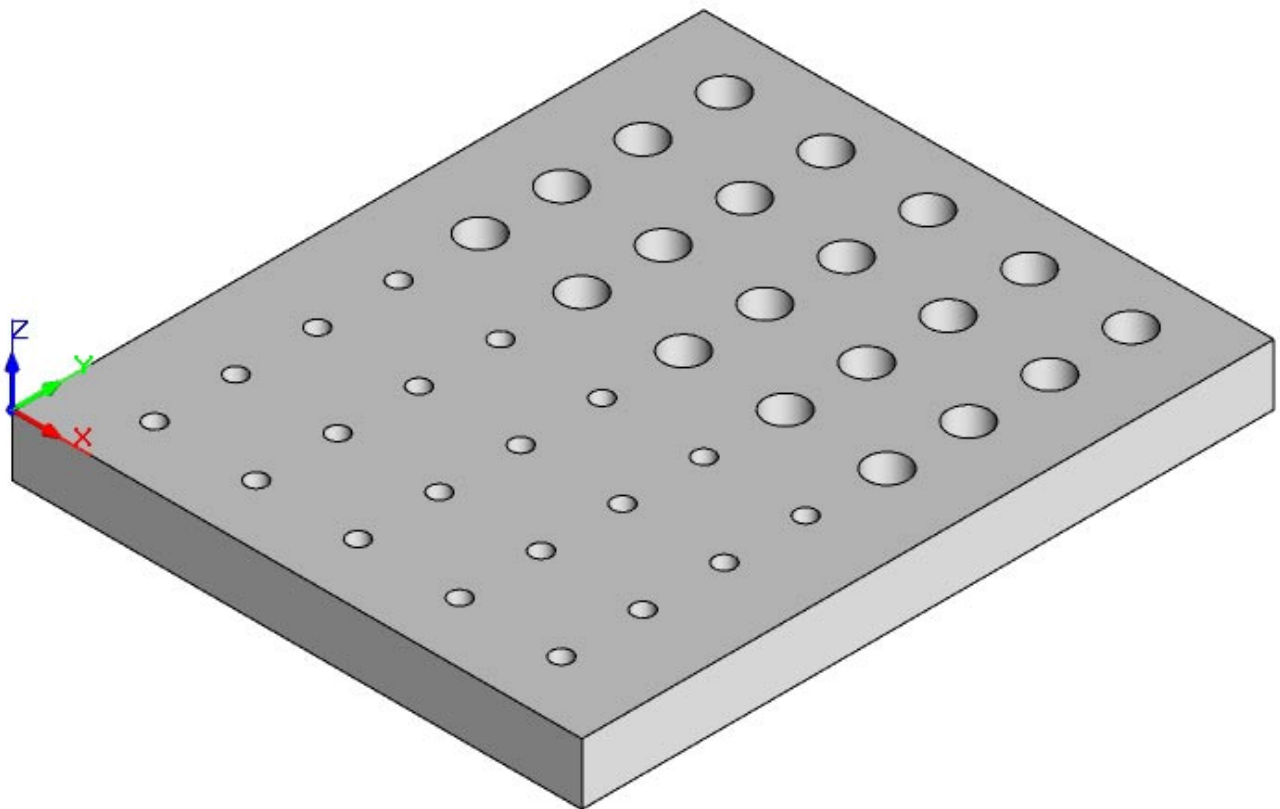


## Tutorial 9: Hole Making

---



## Introduction

This tutorial will illustrate machining holes using Drilling operations. 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 2D 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 part

- We will machine part by using a Hole machining operation called Drilling.
- The part will be machined out of a 7.25 x 8.5 x 0.75 inch poplar wood sheet using a 0.5" and 0.25" 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 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.

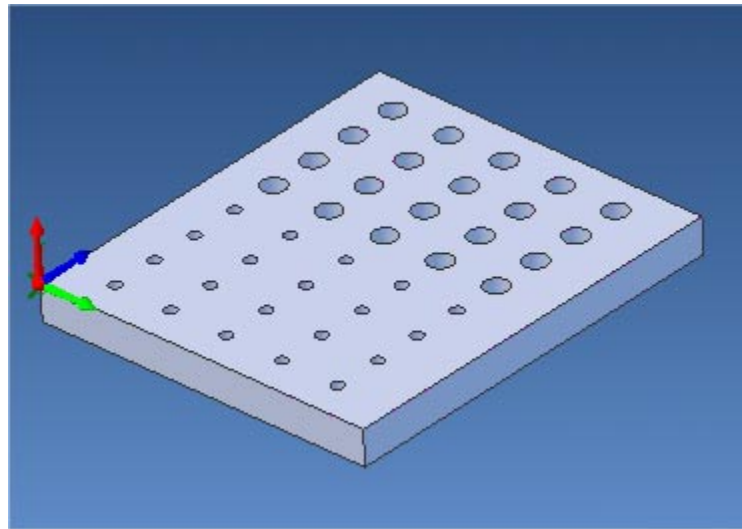
## Loading the Part Model

## 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 **Bitholder.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)

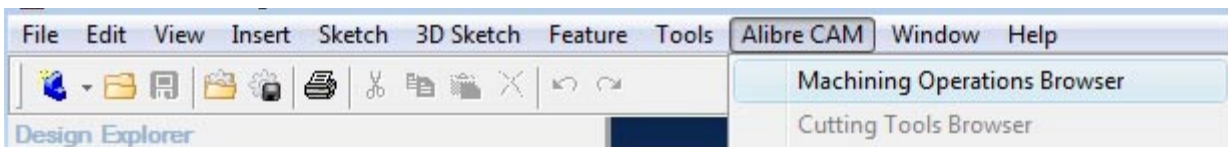
The loaded part appears as shown below.

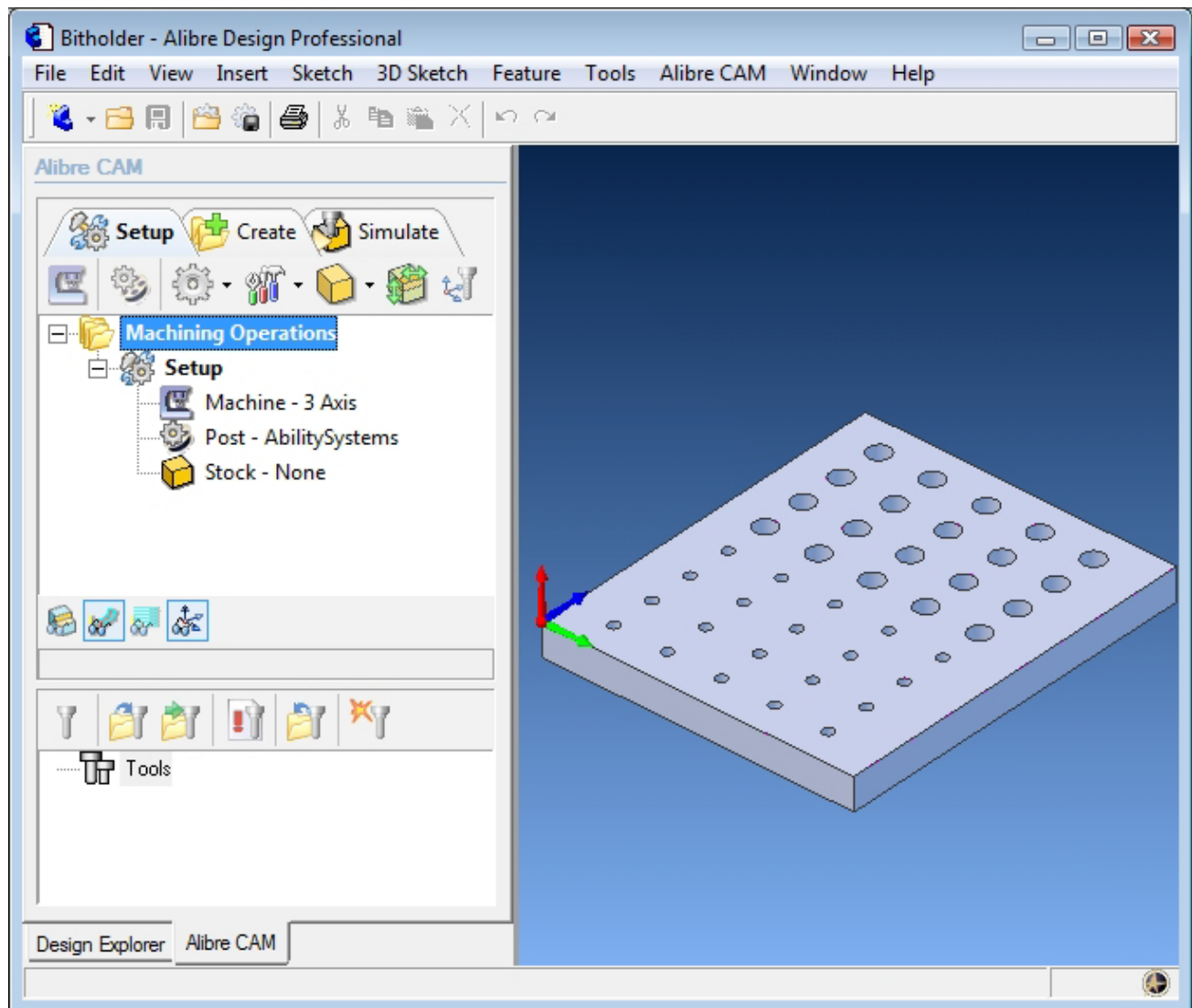


**Note:** You must work in shaded mode in order to be able to visualize toolpaths created in Alibre CAM. It is suggested for best visual performance with Alibre CAM to work with only one view port open and the view port operating in shaded mode.

## 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.

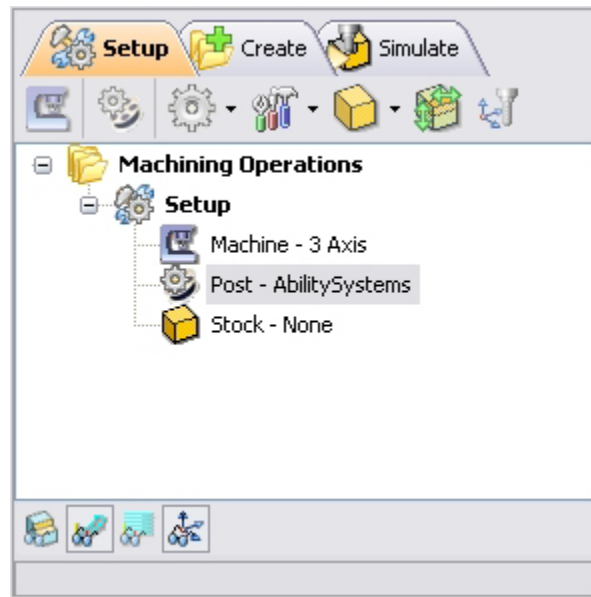
## Preparing the part for Machining

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

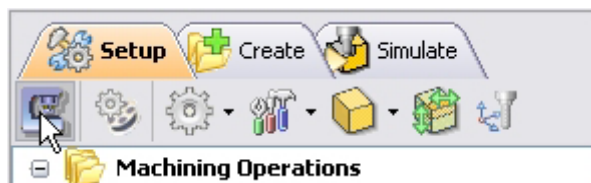
### Setup Tab

## Getting Started with Alibre CAM

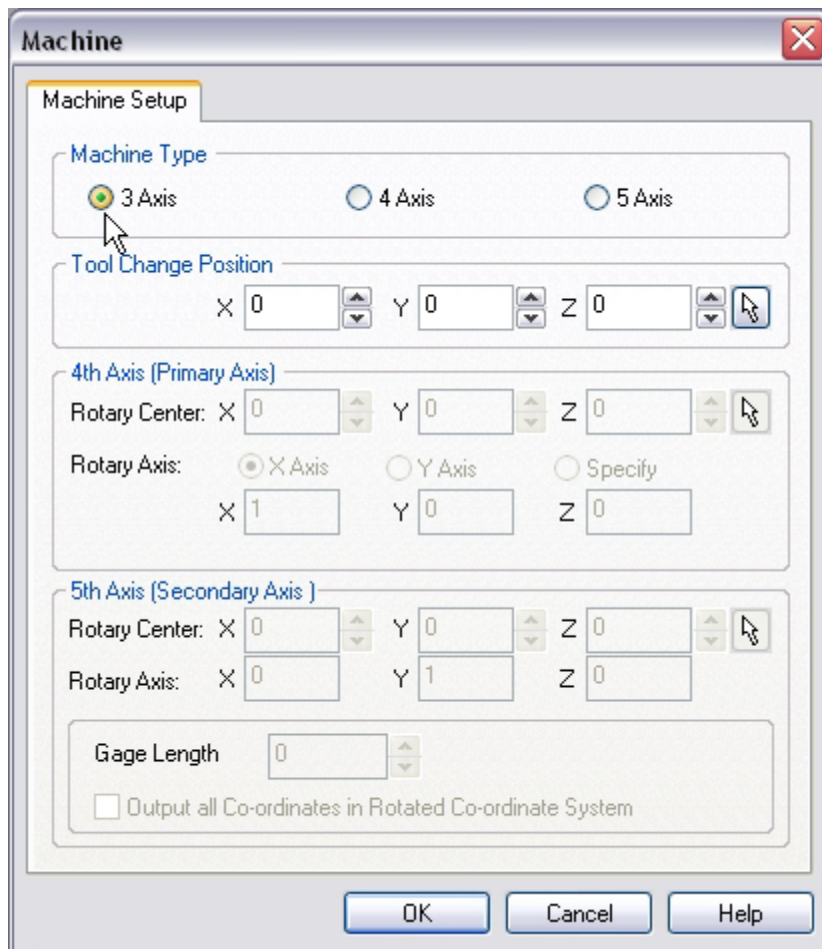
1. Go to the Alibre CAM MOps browser and click on the Setup tab



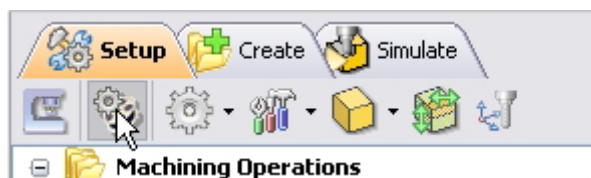
2. Select Machine Setup from the setup tab.



3. Set the Machine type to 3 axis



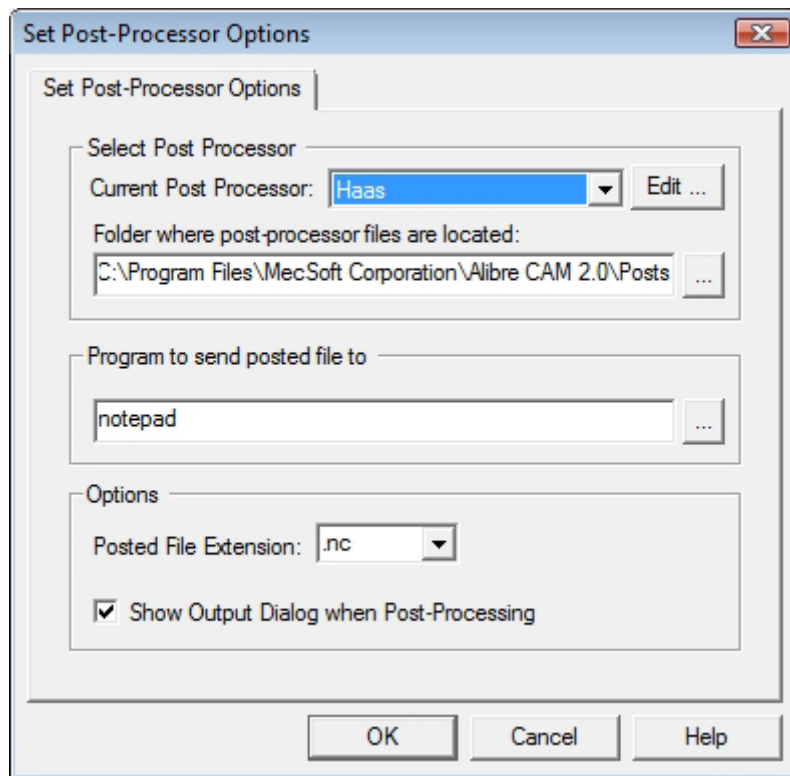
4. Select **Post** from the setup tab to specify the post processor options



5. 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.*

## Getting Started with Alibre CAM

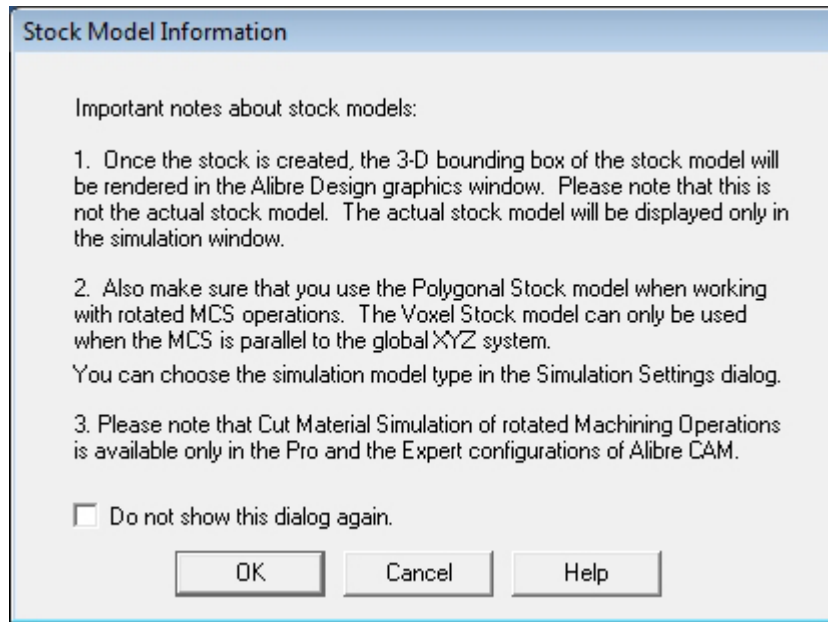


## Create Stock Geometry

1. Select Create/Load stock from the setup tab and create a Part Box Stock



The stock model information dialog may be displayed when a stock geometry is created.



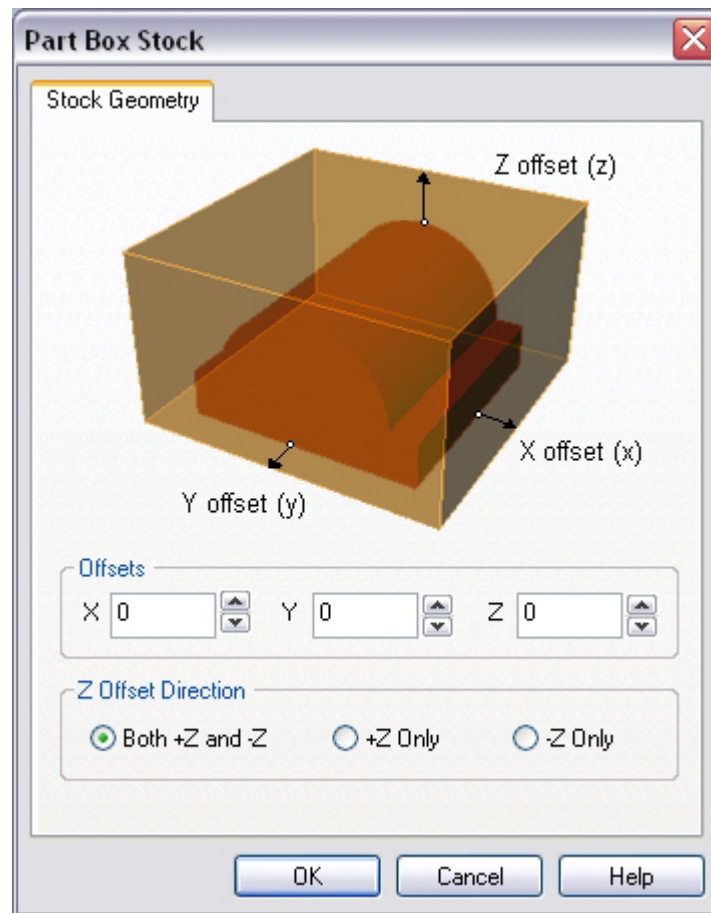
Click OK

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.

2. This brings up the Part Box Stock Parameters.

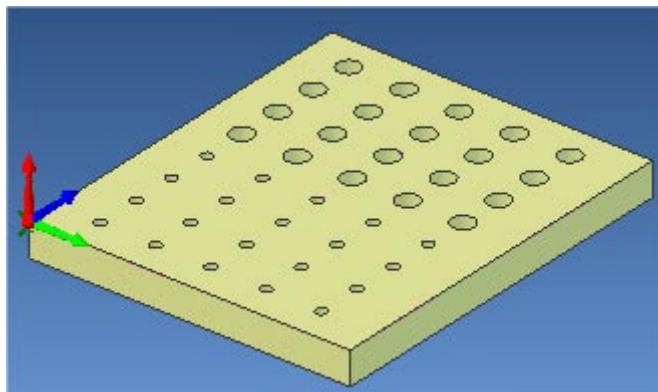




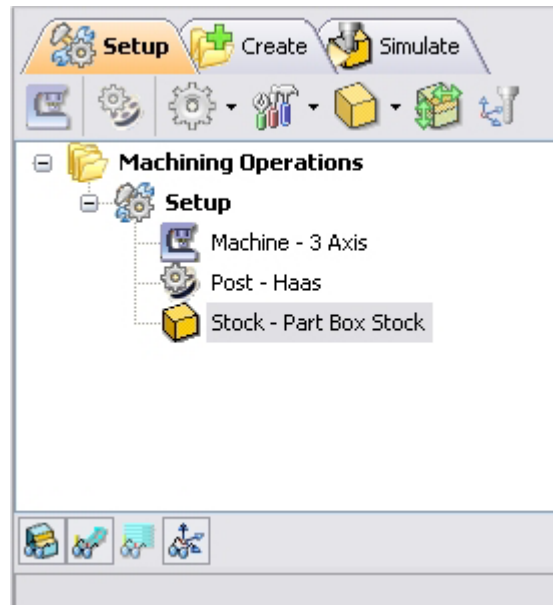
The system calculates the bounding box of the part model as the XYZ extents of geometry of the part model. The user can then define offsets in any of the three coordinate directions to apply to the computed bounding box.

3. Set the offsets for X, Y and Z =0 and click OK.

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



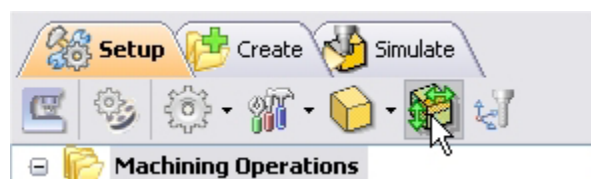
4. The setup tab now displays the following information: Machine Type, Post Processor, and Stock type as show below.



## Align Part and Stock

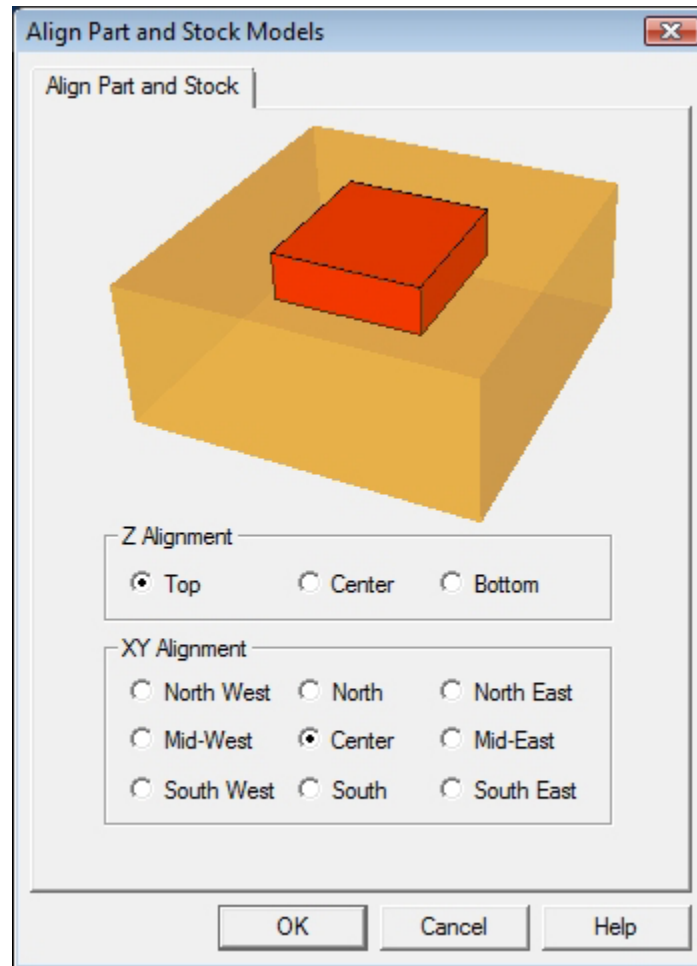
Once the stock model is created, user can move the stock geometry relative to the part geometry and use the stock box to specify the machine zero (home position).

1. Select Align Part and Stock from the Setup tab



2. 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)

## Getting Started with Alibre CAM



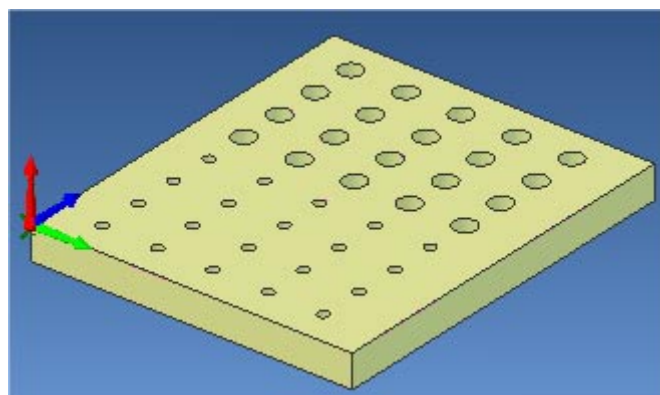
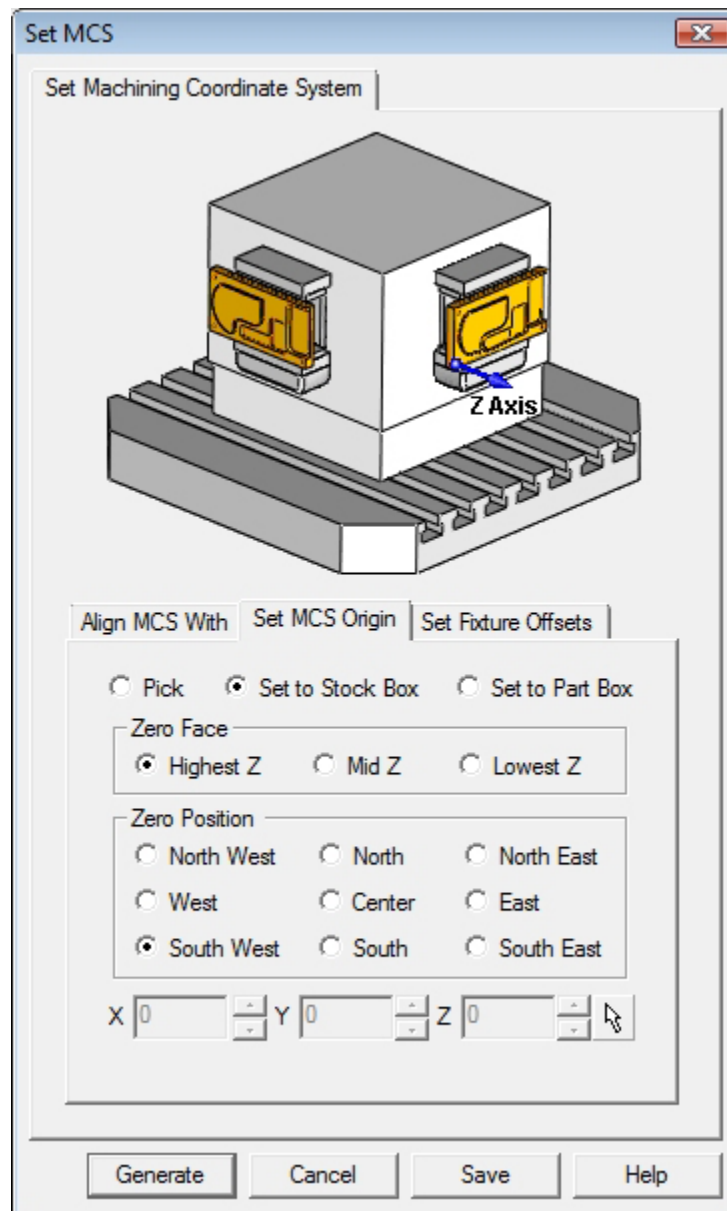
## Set Machine Coordinate System (MCS)

The steps below help you determine the machine home (also known as machine zero or tool touch off point) for the part/stock geometry.

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).

## Getting Started with Alibre CAM

*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 Bitholder-Rev1.

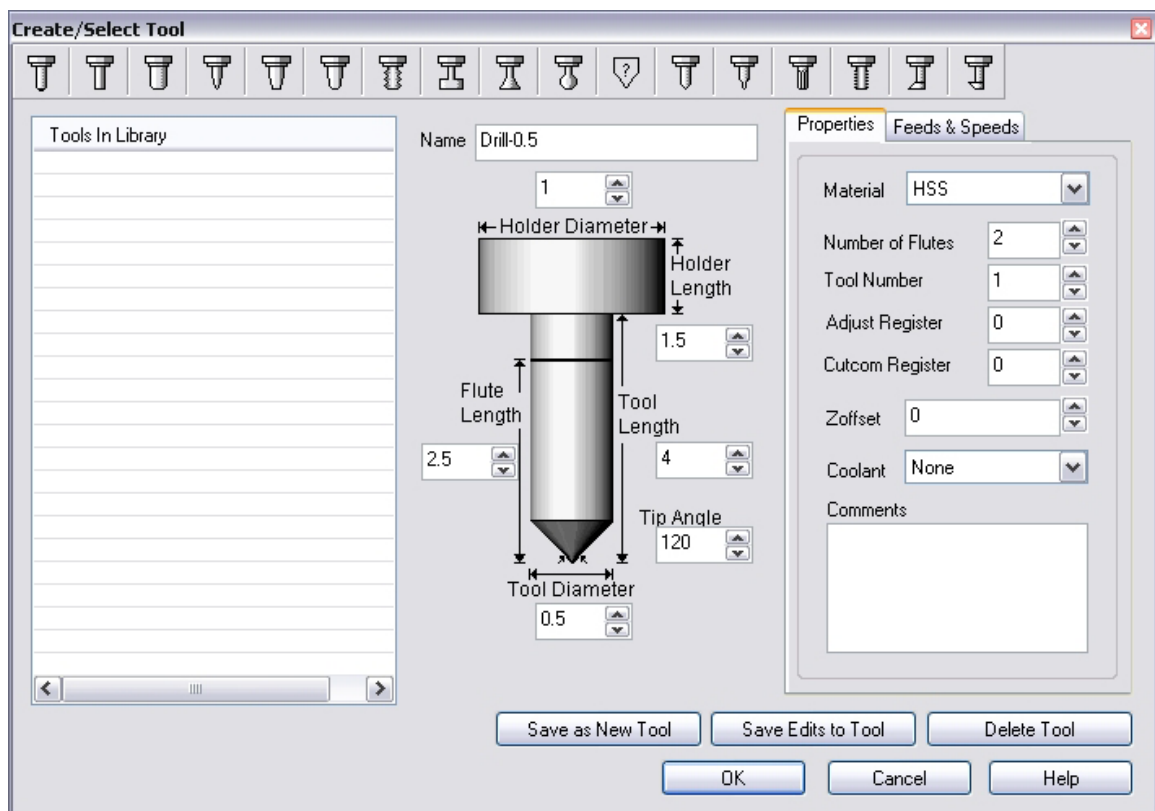
## Create Tools

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

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 Drill Tool.



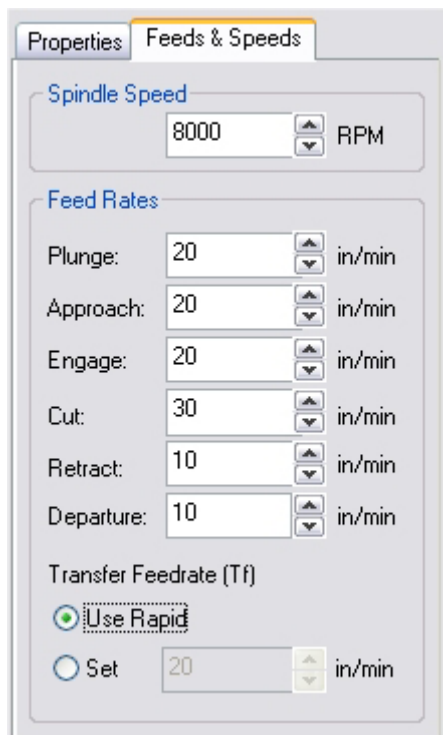
2. Set the tool name as **Drill-0.5** and Tool Diameter = **0.5**. Under the Properties tab, set Tool Number = **1**.



## 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.



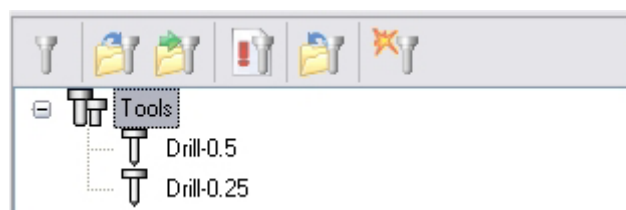
The screenshot shows a dialog box with two tabs: 'Properties' and 'Feeds & Speeds'. The 'Feeds & Speeds' tab is active. It contains the following settings:

- Spindle Speed:** 8000 RPM
- Feed Rates:**
  - Plunge: 20 in/min
  - Approach: 20 in/min
  - Engage: 20 in/min
  - Cut: 30 in/min
  - Retract: 10 in/min
  - Departure: 10 in/min
- Transfer Feedrate (Tf):**
  - ☒ Use Rapid
  - ☐ Set 20 in/min

5. Click **Save as New Tool**. The tool is now created and listed under Tools in Library.

*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.*

6. Create a second tool of type Drill. Set the Tool Name as **Drill-0.25**, Tool Number = **2**. Use the same settings for the other parameters and click Save as New Tool. Click OK to close the dialog.



The created tools are now listed under the Cutting Tools browser.

## Getting Started with Alibre CAM

### Create Machining Operations

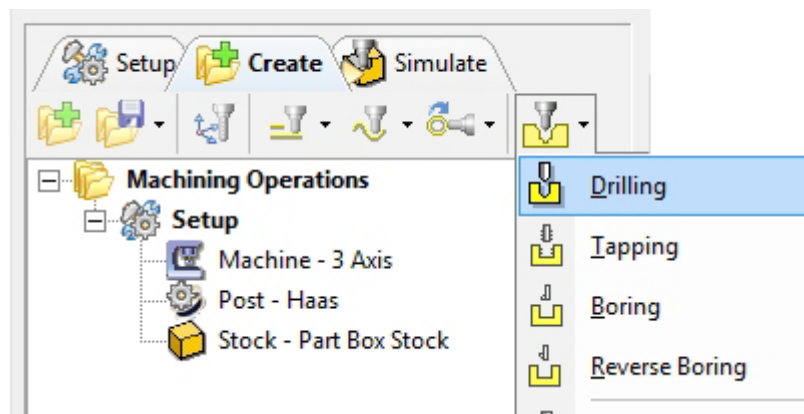
In this process, we will create a 2.5 axis profiling operation.

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



### Hole Machining

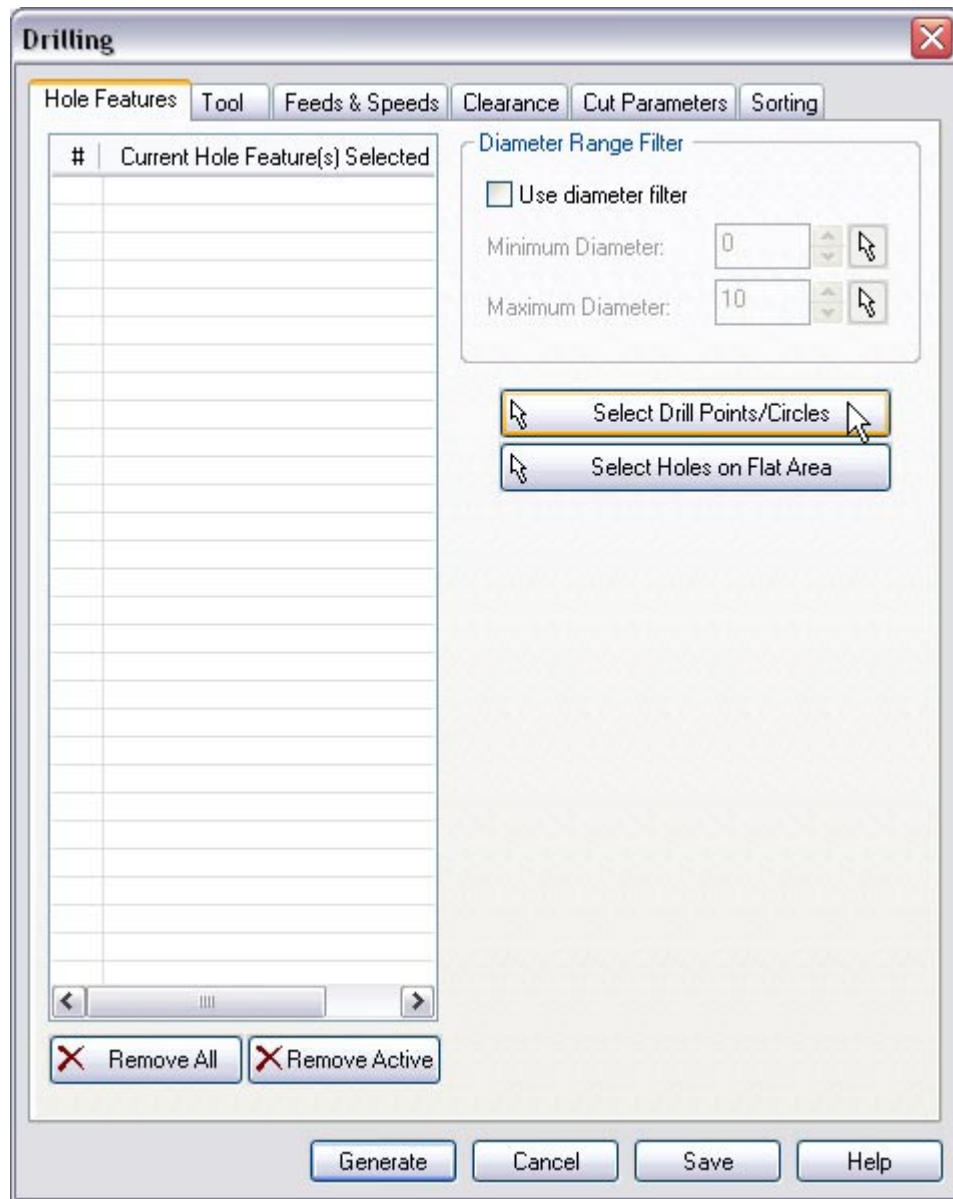
2. Select Hole Machining and choose Drilling.



This brings up the Drilling Operations dialog. We will go over the steps for creating the toolpath.

## Select Hole Features

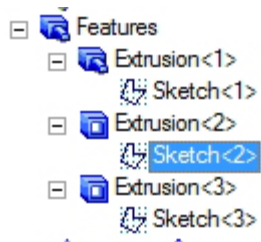
3. Go to the Hole Features tab and click **Select Drill Points/Circles**. The drilling operation dialog is now minimized and allows selection of the sketch geometry.



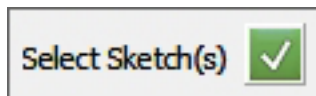
4. Switch to Design Explorer and select **Sketch2**. This selects the circles of diameter 0.5"

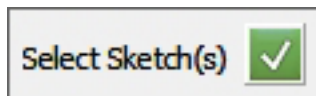


## Getting Started with Alibre CAM

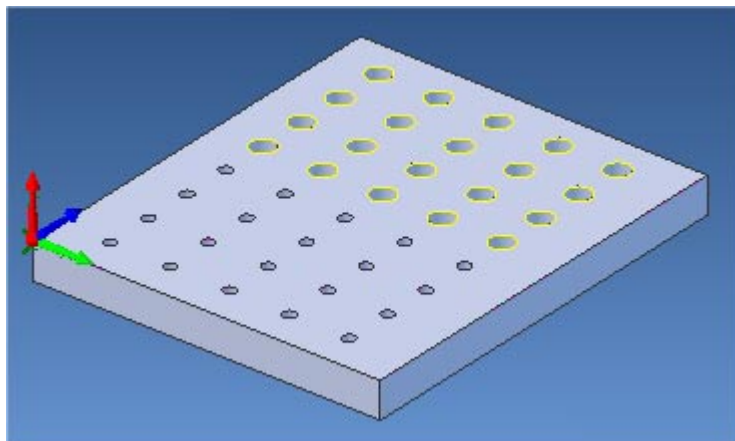
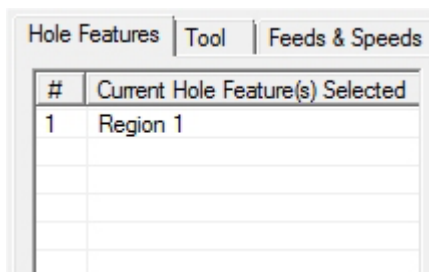


Note: You can also select the sketch from Alibre Design View. When select containment region is selected, Alibre CAM turns on Sketch selection filter to allow the user to select sketches. Hold the Shift key down to select multiple sketches.



5. Click  to complete the selection.

The Drilling operation dialog comes back up displaying the selected region. The selected regions are also highlighted on the part.



## Selecting the Tool

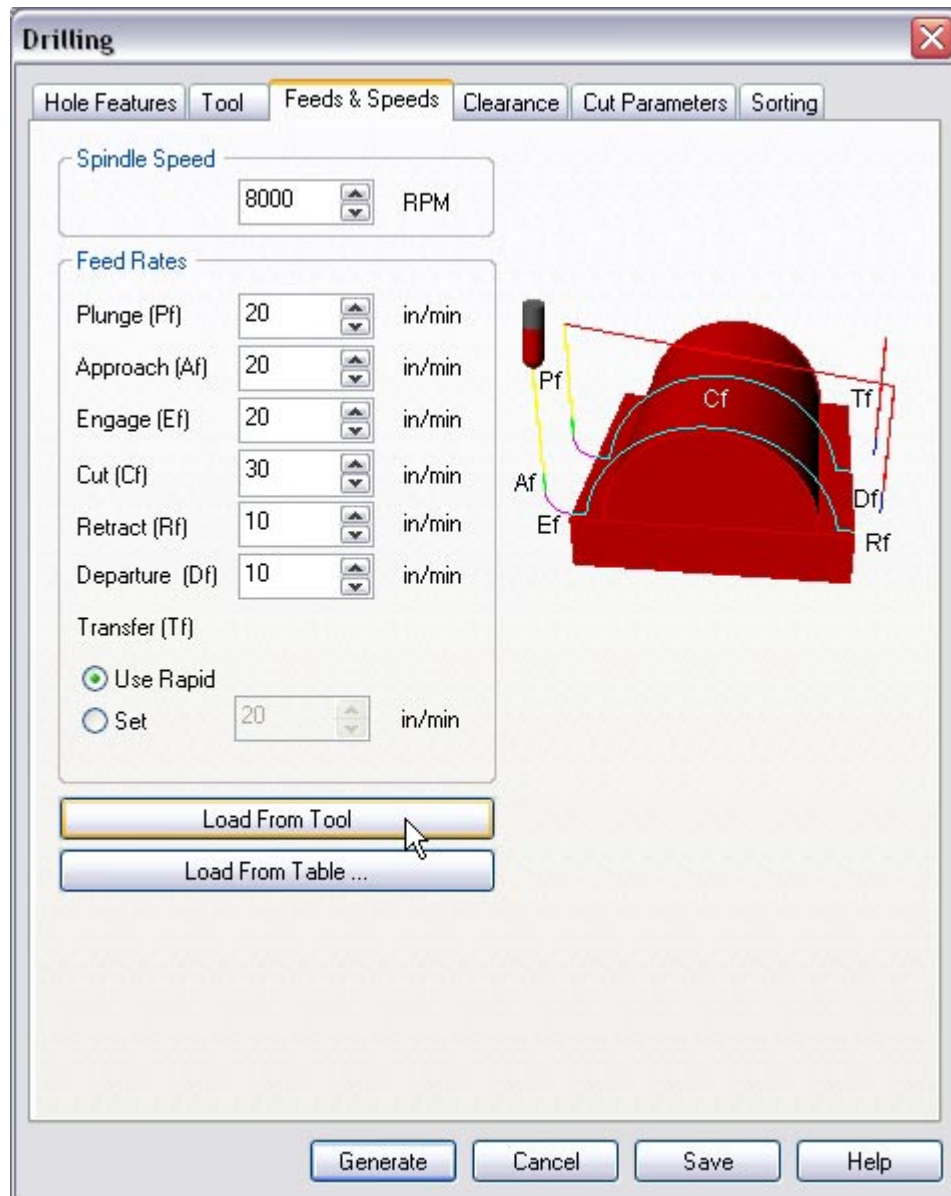
6. Switch to the Tools tab inside the Drilling operation.



7. Select the Drill-0.5. The 0.5" Drill is now selected as the active tool and the Tool parameters are displayed to the right of the Tools window.

### Set Feeds and Speeds

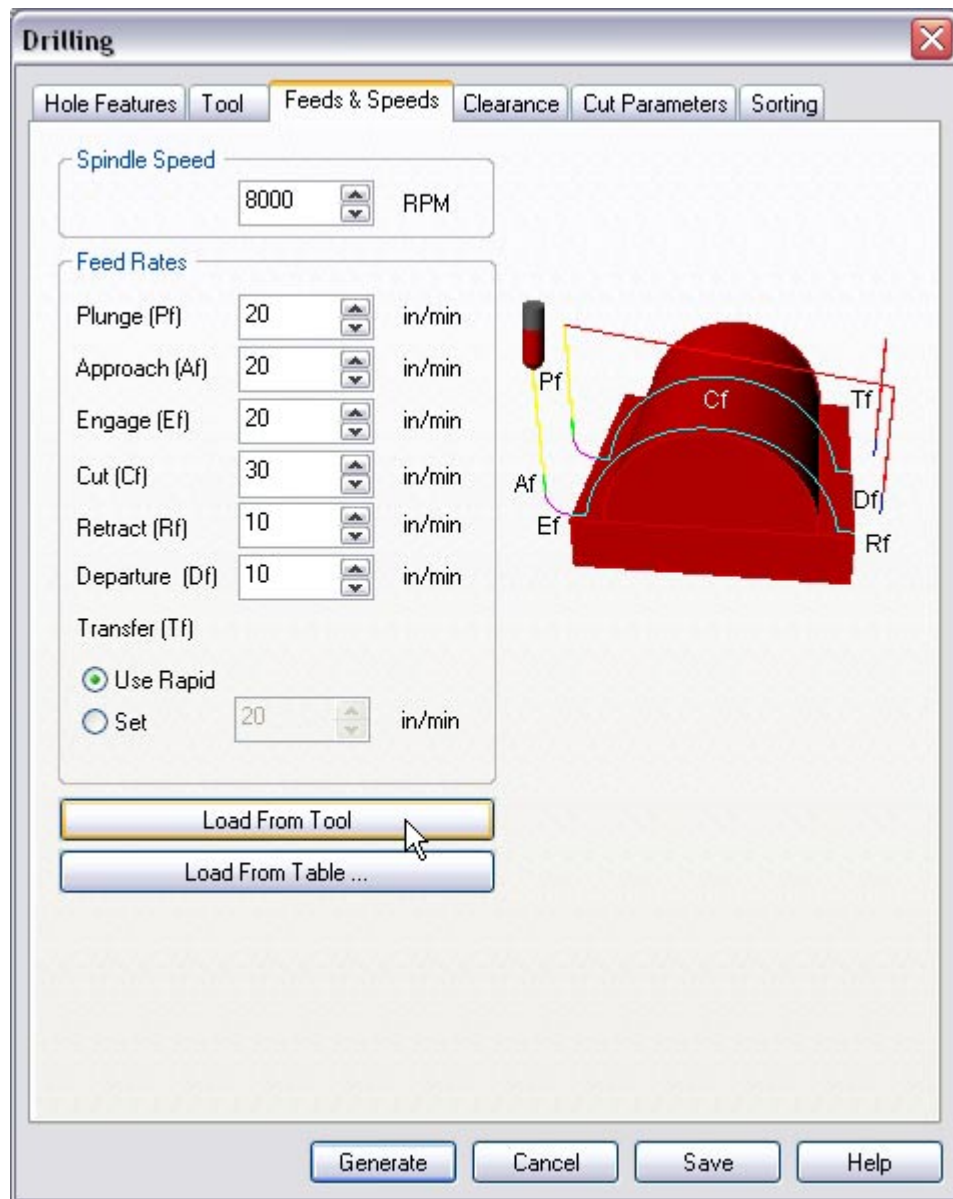
- Click on the Feeds and Speeds tab.



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

## Clearance Control

10. Switch to Clearance Tab.

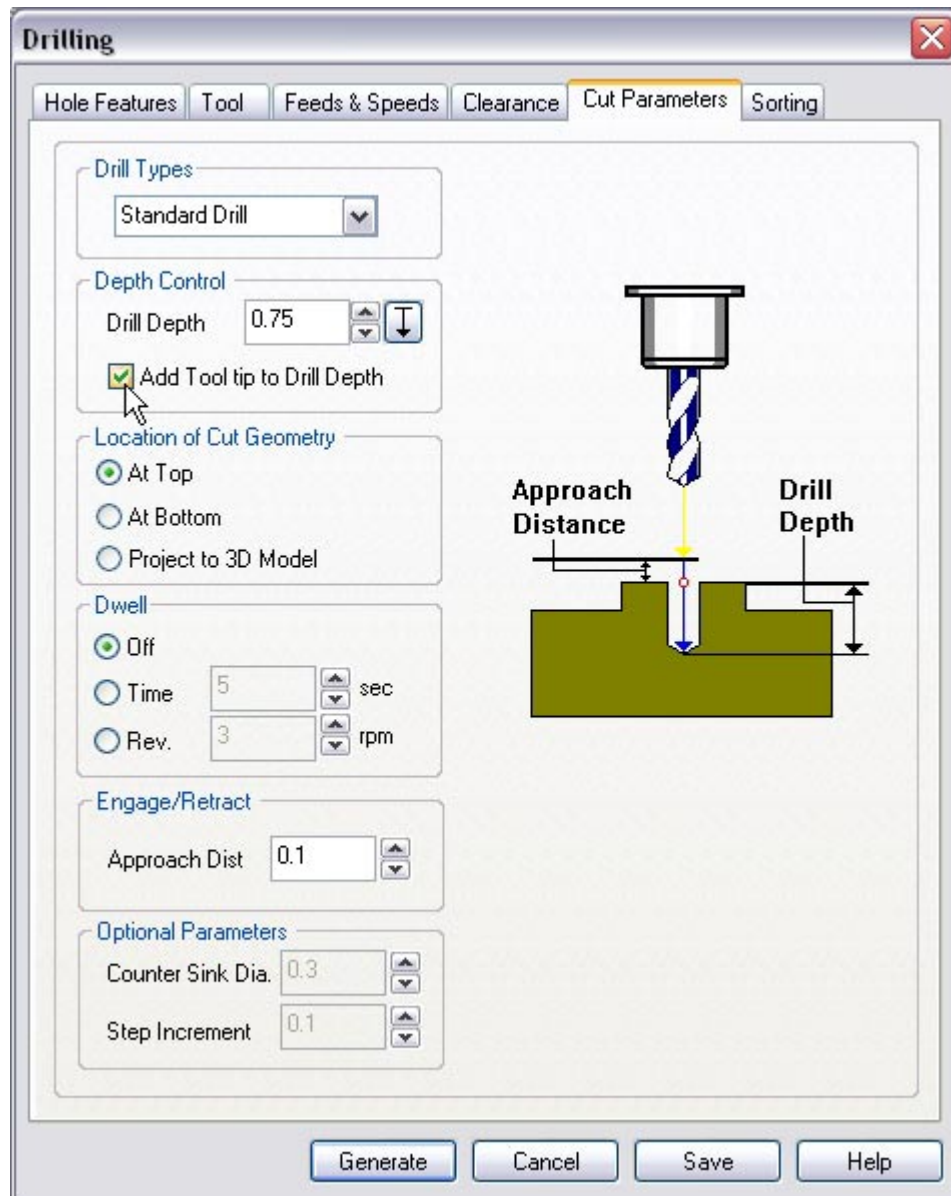


11. Set the Clearance Plane Definition to Automatic and Cut Transfer Method to Clearance Plane.

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.

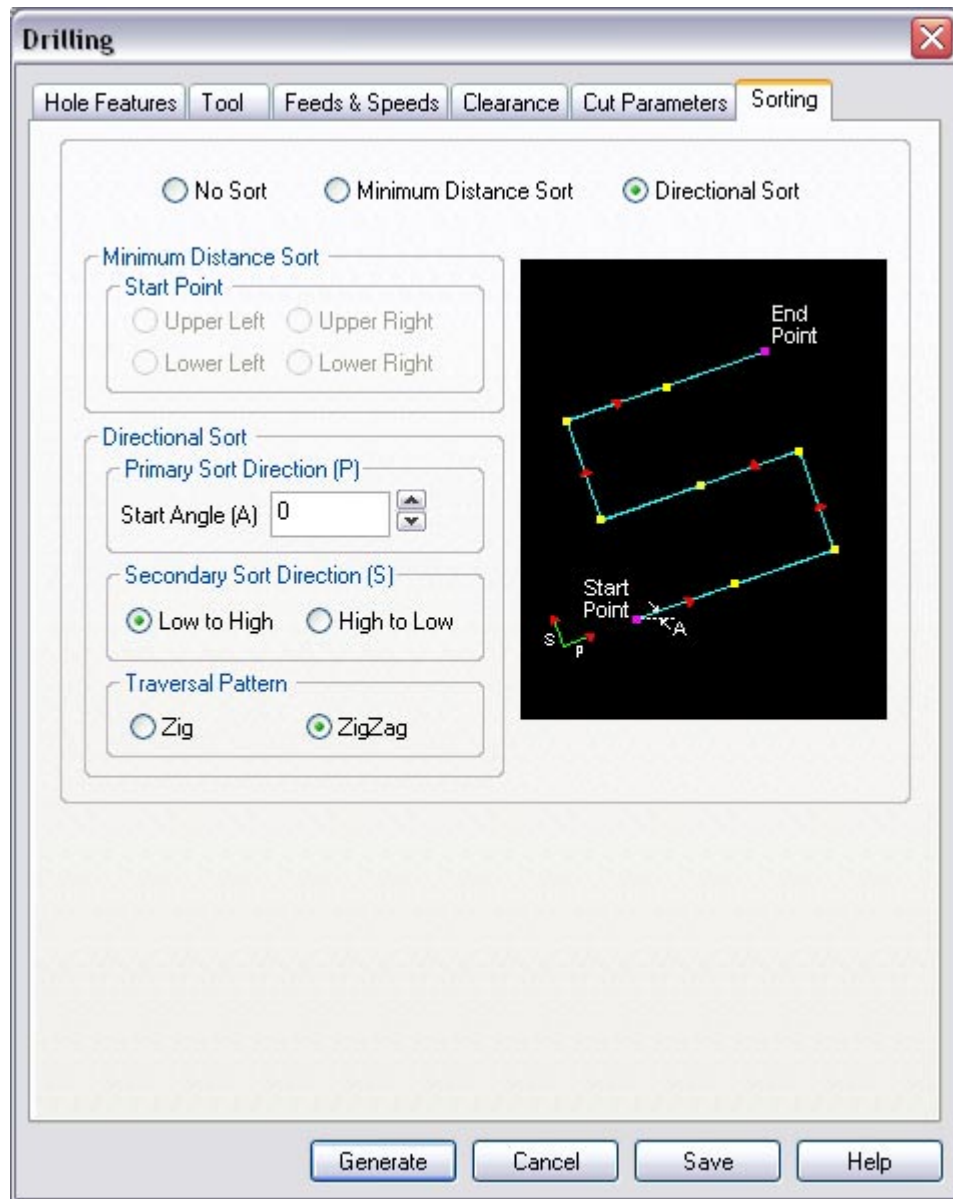


13. Set the Drill Type to Standard Drill, Drill Depth = **0.75**, Check **Add Tool Tip to Drill Depth**, Location of Cut Geometry – **At Top**, and Approach Distance = **0.1**.

*Note: Adding Tool Tip to Drill Depth adds the taper height of the drill tool to the drill depth to make it a through hole.*

## Sorting

14. Select the Sorting Tab. Use **Directional Sort**, leave the primary sort direction Start Angle = **0**, Secondary Sort direction (s) – **Low to High** and Traverse Pattern to **ZigZag**.

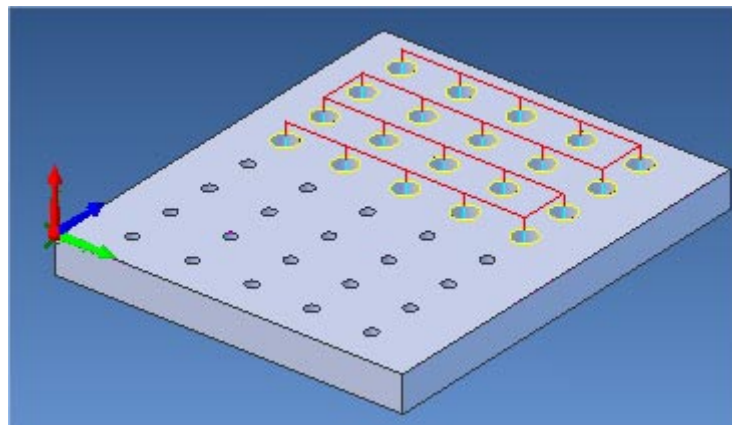
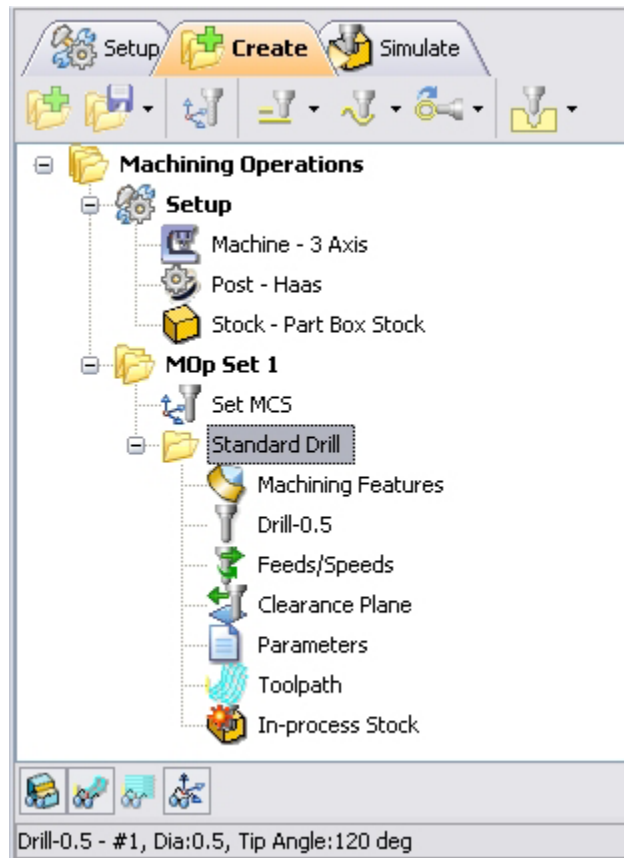


The holes will now be sorted row first starting from the lowest point moving up in Y in a ZigZag pattern.

15. Click **Generate**. The Drilling toolpath is now generated, and the Operation is listed under the Alibre CAM MOps browser.

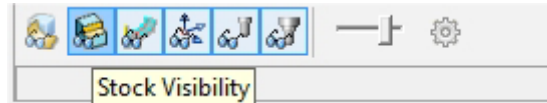
*Note: Toolpath display can be turned on/off by selecting Toolpath Visibility under the MOps browser.*


## Getting Started with Alibre CAM

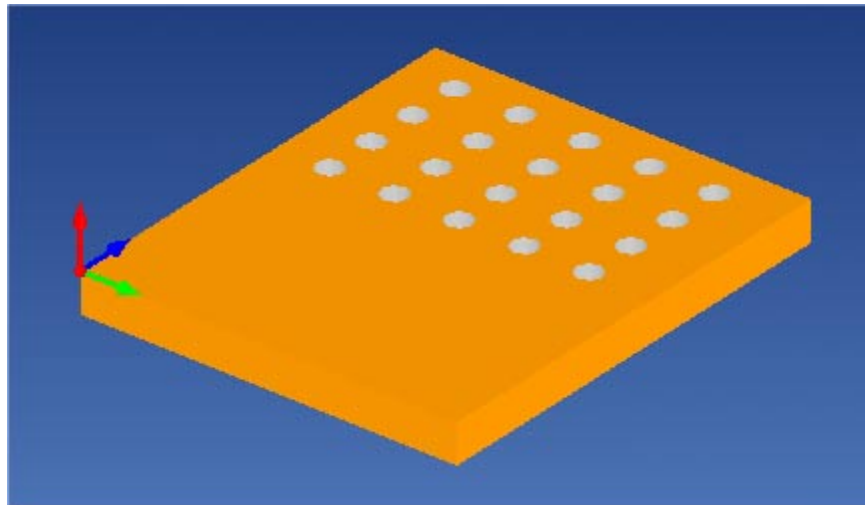


## Simulate Toolpath

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



1. Switch to Simulate tab, Select the Standard Drill 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.

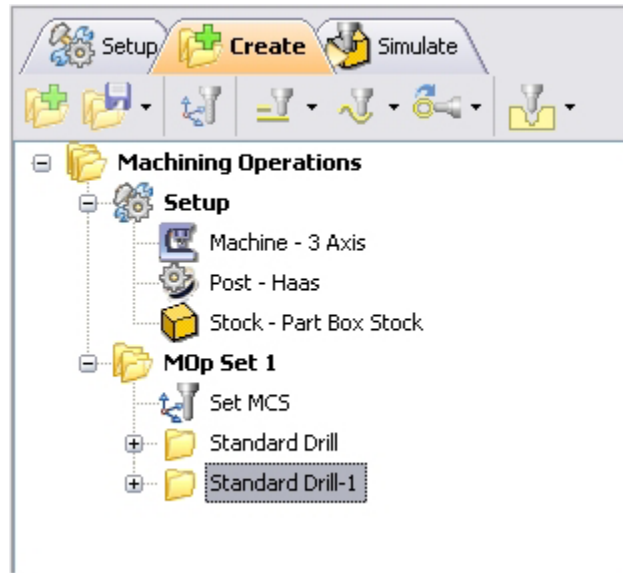
3. Once the simulation is complete, you can close the Stock Simulation window and return to the Alibre CAM browser.

## Creating the Drill operation for the 0.25" Holes

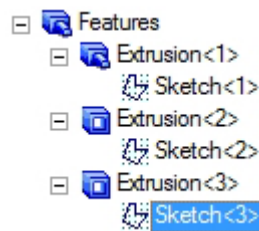
1. Switch to the Create operations tab
2. Select the Standard Drill Operation created from the previous step, right mouse click, and select **Copy**.
3. Right click and select **Paste**.
4. This would create a copy of the Drilling Operation and lists below the first pocketing operation as show below.



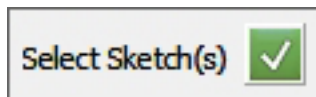
## Getting Started with Alibre CAM

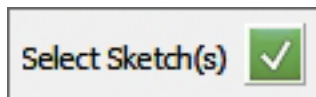


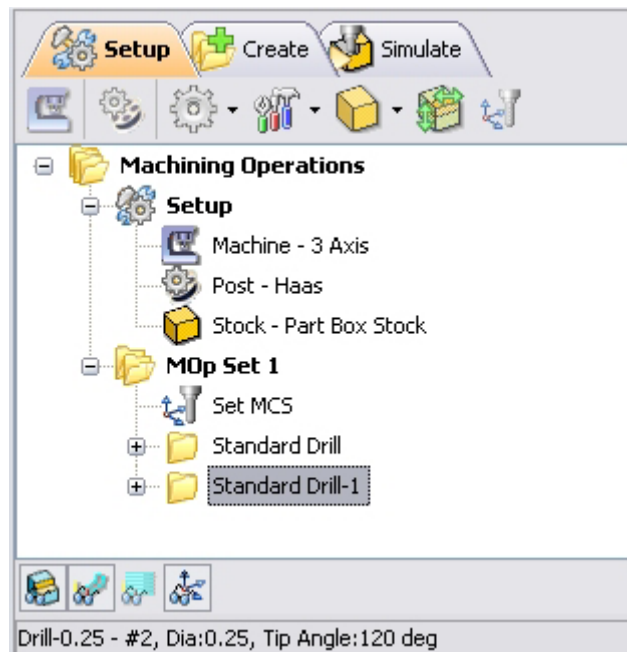
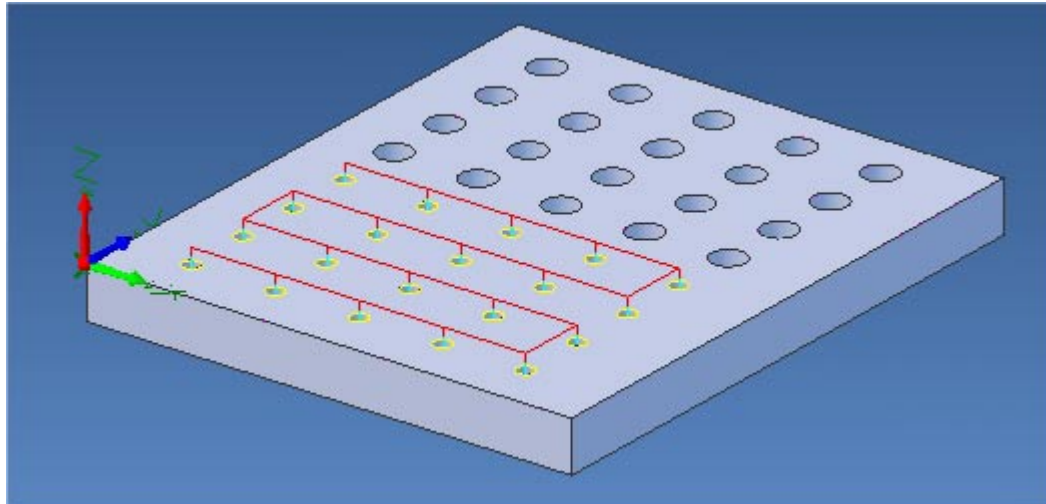
5. Double click on the Standard Drill-1 to edit the Hole Features.
6. Click **Remove All** from the Hole Features Dialog to remove all the regions from the list.
7. Now use **Select Drill Points/Circles**. The drilling operation dialog is now minimized and allows selection of the sketch geometry.
8. 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.

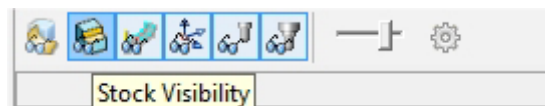



9. Click  to complete the selection.
10. Switch to the Tools tab under the Drilling Operations dialog and select **Drill-0.25**.
11. Click **Generate**. The toolpath for Drill Operation-1 is now generated.



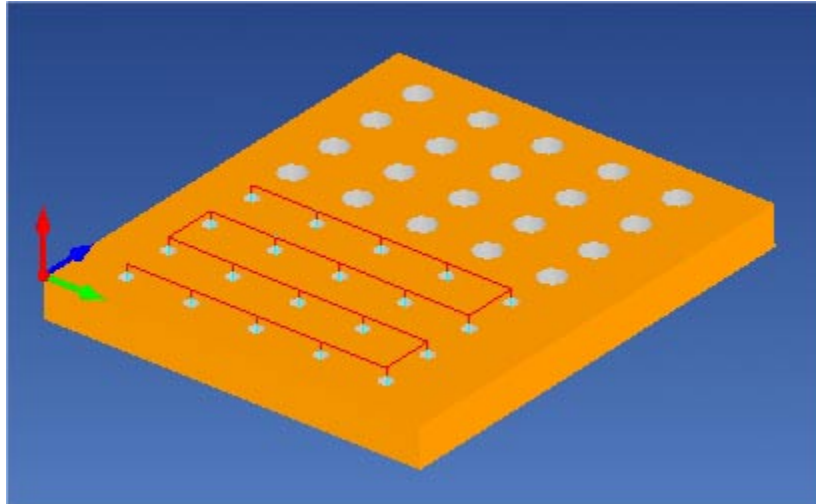
## Simulate Toolpath

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



1. Switch to Simulate tab, Select the Standard Drill-1 Operation and click  to launch the Alibre CAM Stock Simulation window.
2. Click Simulate from the Stock Simulation window to run simulation.

## Getting Started with Alibre CAM

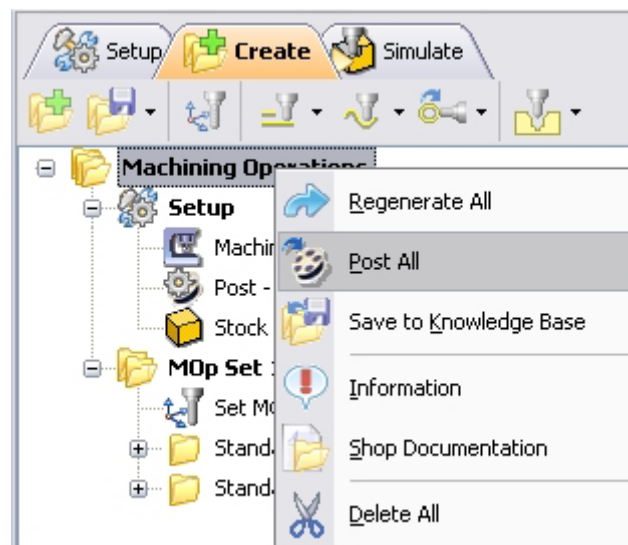


Note: You can pause/stop, step, and skip the simulation using the controls available on the Simulation window. To control the simulation speed, select Simulation Settings from the Stock Simulation window.

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 **Drilling.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](http://www.lista.it)    [info@lista.it](mailto:info@lista.it)**