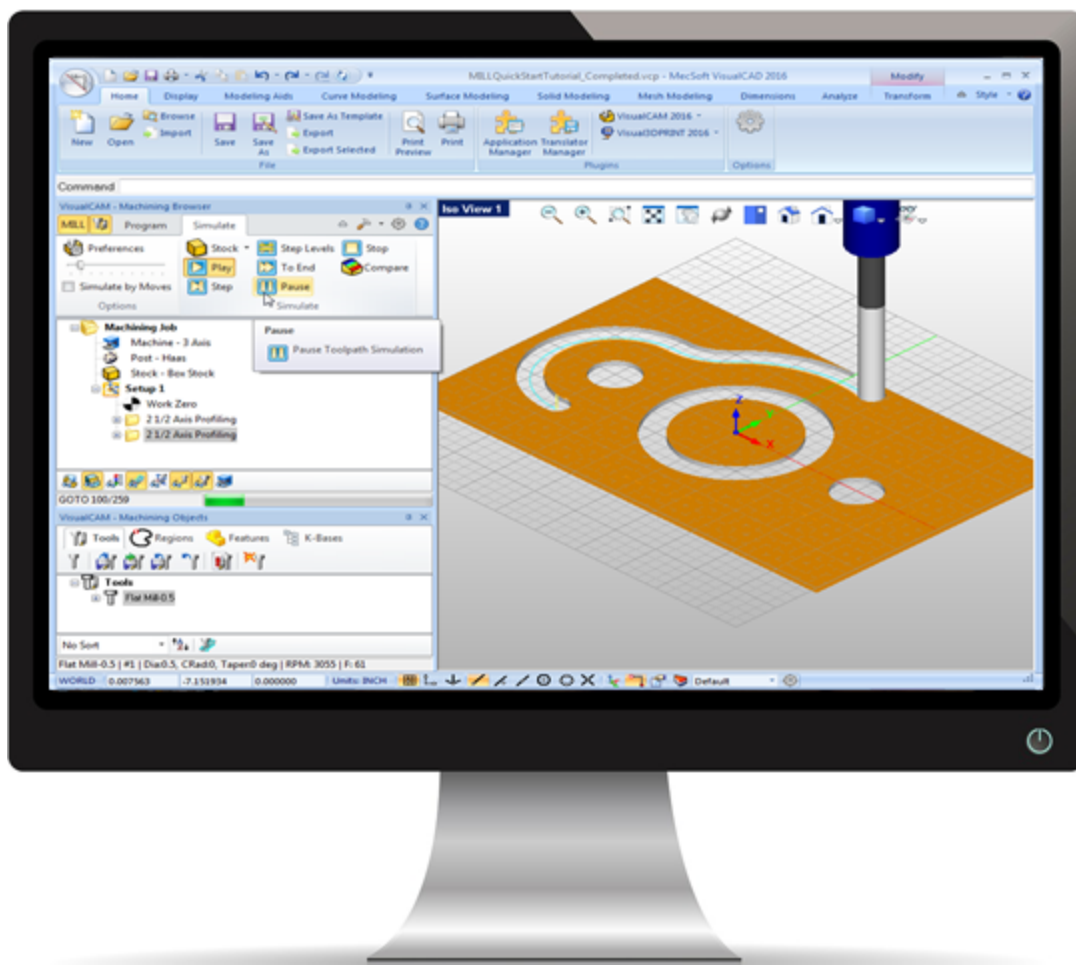


# VisualCAM© 2016 MILL Quick Start Guide

© 2016 MecSoft Corporation



# Table of Contents

<b>About this Guide</b>	<b>4</b>
1 About the MILL Module.....	4
2 Using this Guide.....	4
3 Useful Tips.....	5
<b>Getting Ready</b>	<b>6</b>
1 Running VisualCAM 2016.....	6
2 About the VisualCAM Display.....	6
3 Launching the MILL Module.....	7
4 Loading the Part Model.....	11
5 Machining Strategy.....	13
6 Main Programming Steps.....	13
7 Define the Machine Tool.....	13
8 Select the Post Processor.....	15
<b>The Setup</b>	<b>18</b>
1 Define the Machining Setup - Skip this section if in Standard or Expert Configuration.....	18
2 Create Stock Geometry.....	18
3 Align Part and Stock.....	22
4 Specify Material.....	24
5 Set Work Zero.....	26
<b>Create Tools</b>	<b>31</b>
<b>Create Machining Operations</b>	<b>35</b>
1 Select Machining Features/Regions.....	36
2 Select Cutting Tool.....	39
3 Set Feeds and Speeds.....	40
4 Clearance Plane.....	41
5 Specify Cut Parameters.....	43
6 Cut Levels.....	44
7 Entry/Exit Parameters.....	45
8 Simulate Toolpath.....	48
<b>Machining the Outer Area</b>	<b>52</b>

---

<b>Post Processing</b>	<b>58</b>
<b>Generating Reports</b>	<b>61</b>
1 Information Report.....	61
2 Shop Documentation.....	62
<b>Where to go for more help</b>	<b>66</b>
<b>Index</b>	<b>67</b>

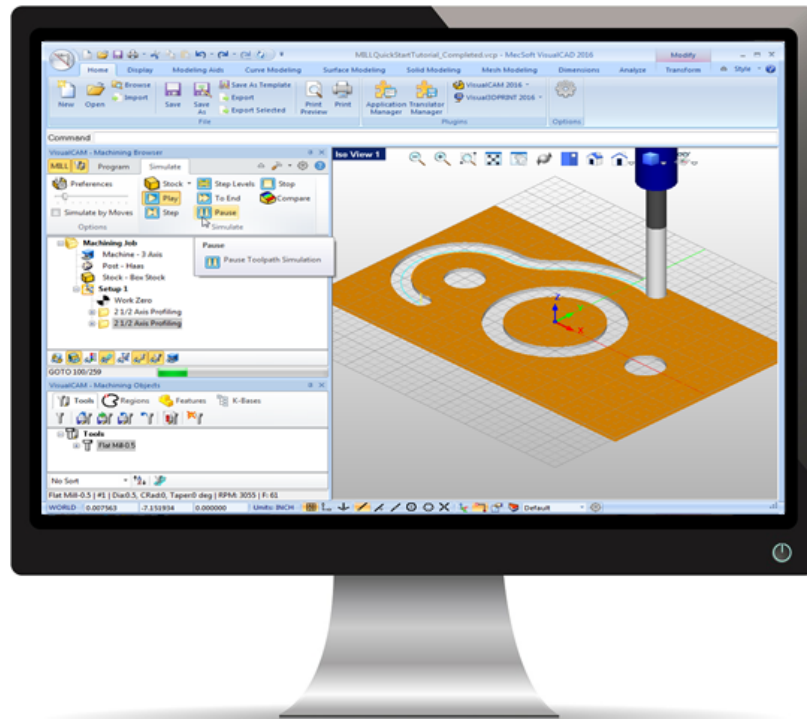
## About this Guide

### 1.1 About the MILL Module

The [VisualCAM 2016 MILL](#) module offers fast gouge free solids/surface model machining technology coupled with cutting simulation/verification capabilities for programming CNC Mills, running inside [VisualCAD](#). This integration allows for seamless generation of toolpath and cut material simulation/verification inside [VisualCAD](#), for programming milling machines that support 3, 4 and 5 axis continuous machining.

The module also comes with numerous post-processors to output the programmed G-code to some of the most popular machines in the market. A simple and well thought out user interface makes this system one of the most intuitive and easy to use milling systems in the market.

You can work with the native [VisualCAD](#) data as well as use any of the data types that can be imported into [VisualCAD](#) such as solids, surfaces and meshes. Then you can use the [VisualCAM 2016 MILL](#) module with its wide selection of tools and toolpath strategies to create machining operations and associated toolpaths for CNC Mills. These toolpaths can then be simulated and verified, and finally post-processed to the controller of your choice.



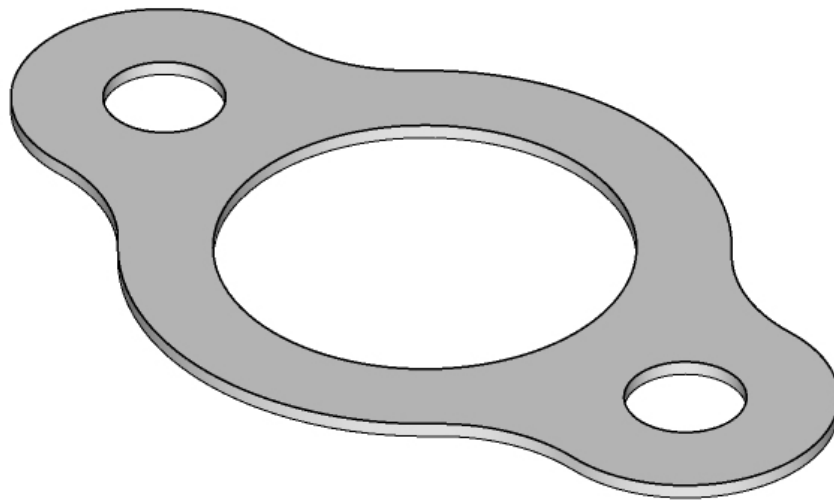
VisualCAM's MILL Module Quick Started Guide

### 1.2 Using this Guide

If you have installed [VisualCAD/CAM3DPRINT 2016](#) successfully on your computer and are now looking at the blank screen of [VisualCAD](#) and wondering what to do next, this is the guide for you. This guide will explain how to get started in using the [VisualCAM 2016 MILL](#) module to program a simple part through an example.

This guide will illustrate machining of a simple prismatic part such as this gasket using 2-1/2 milling operations. Even though we have created a 3D representation of the gasket, it will become apparent as we go that we can machine this using just 2D curves. The reason we are able to do this is because of the prismatic nature of this model, which means that the curves can be treated as the edges of vertical walls in the geometry.

This guide has two associated [VisualCAD](#) files that you can find located in the [QuickStart](#) folder under the installation folder of this guide. The first file is a completed file that contains all of the completed toolpaths and machining operations and represents the file that you should end up with after working through the tutorial. The other file is a starter file that contains only the geometry. Use the completed file as a reference. Copy the starter file and use this file to begin each tutorial.



MILLQuickStartTutorial.vcp

## 1.3 Useful Tips

Here are some useful tips that will help you use this guide effectively.

1. Copy the tutorial part files in a location other than the installation folder to make sure you have read/write privileges to the files.
2. Once you start working with the tutorial file, save your work periodically!
3. Don't stress out too much if you are having trouble with the tutorial. Call us or send us email and we can help you out.
4. Most of all have fun!

## Getting Ready

### 2.1 Running VisualCAM 2016

Locate the [VisualCADCAM3DPRINT 2016](#) shortcut on your desktop and double click to launch the application.

Alternatively you can also click on the Windows [Start](#) button and select [All Programs](#). Go to the program group containing [VisualCADCAM3DPRINT 2016](#). (The name of this program group will usually be called , unless you specified otherwise during setup.)

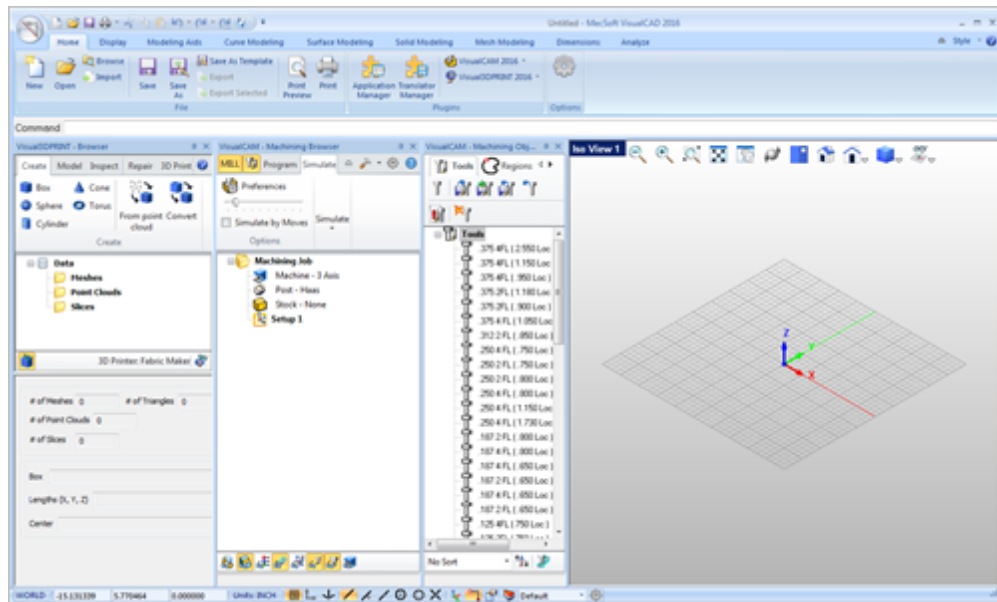
Once you locate the program group, select it and then select [VisualCADCAM3DPRINT 2016](#) to launch the application.

If the installation was successful, upon launching of [VisualCADCAM3DPRINT 2016](#) you should observe a menu entry called [VisualCAM 2016](#) on the [Home Ribbon Bar](#) menu of [VisualCAD](#).

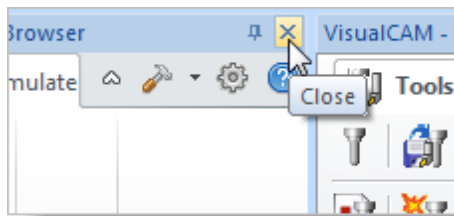
If you do not see this menu entry then please check the [On Line Help](#) document of the product (found in the installation folder) for help with trouble shooting the installation.

### 2.2 About the VisualCAM Display

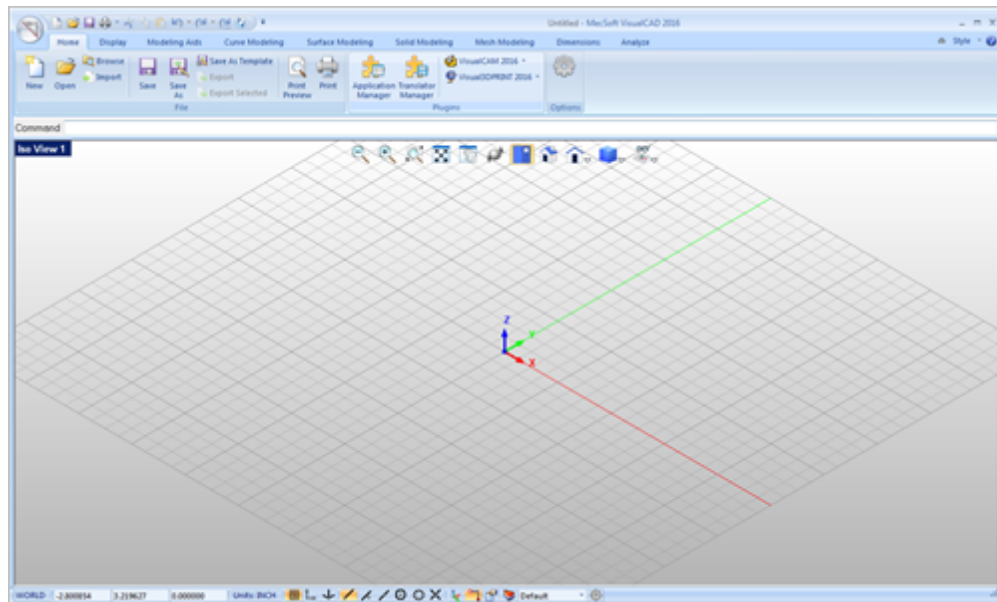
Before we begin, let's talk a bit about the [VisualCAD](#) display. When you run [VisualCAD](#) for the very first time, your screen may look like this.



These windows on the left belong to plug-in modules that are currently loaded. For now, let's close all of them.



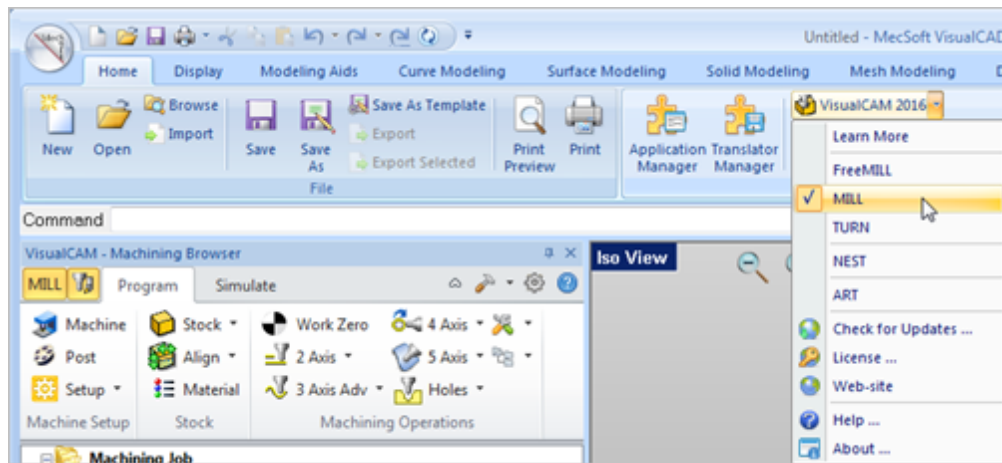
With all plug-in modules closed your screen will look like this:



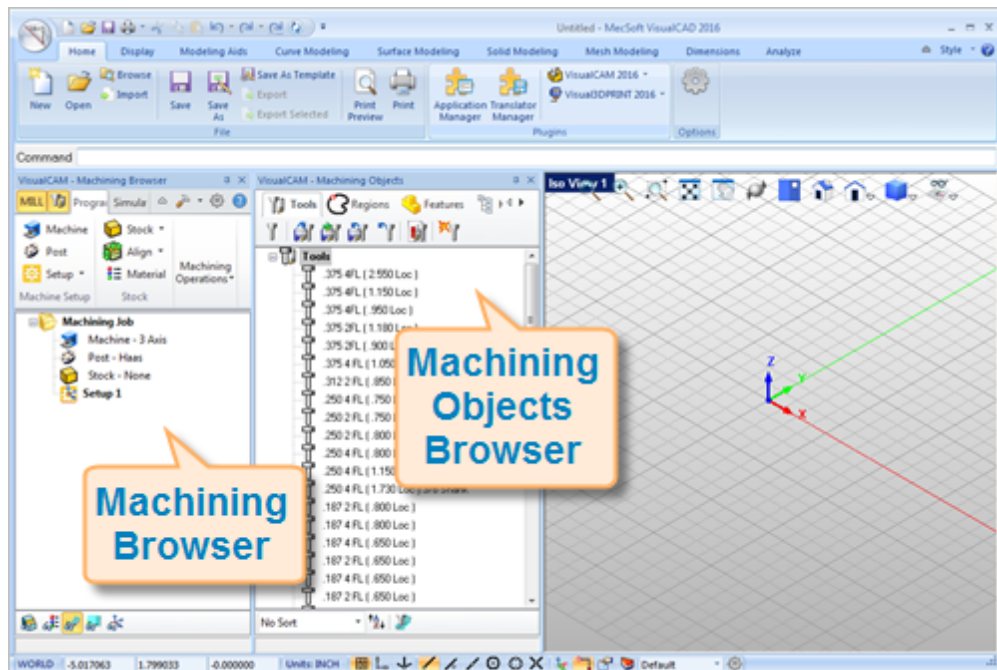
## 2.3 Launching the MILL Module

Now, let's begin by launching the [VisualCAM 2016 MILL](#) module.

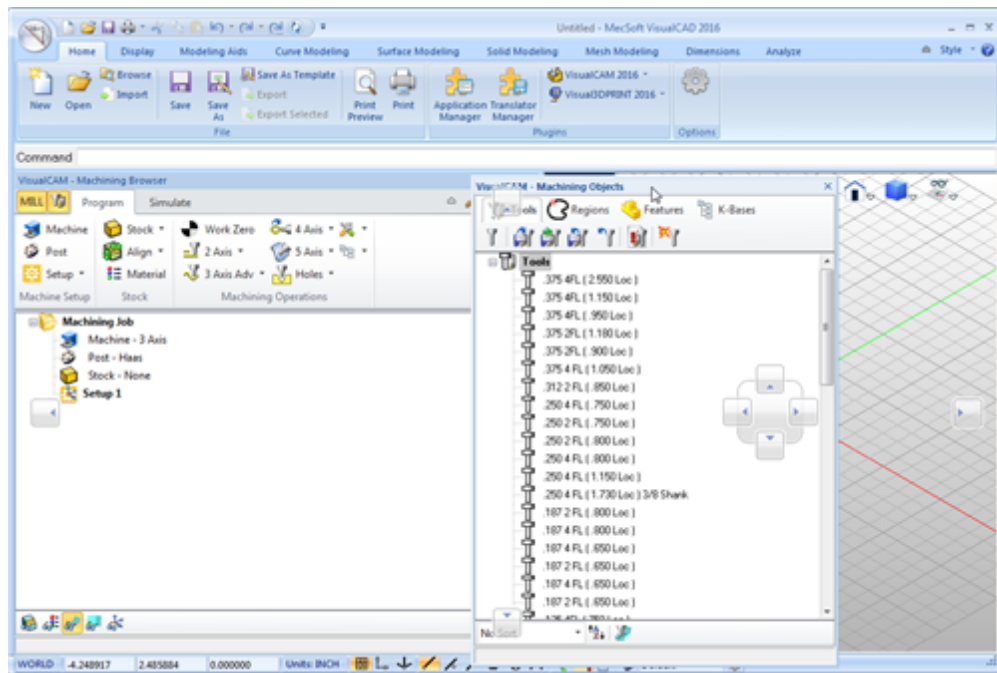
1. From the [Plugins](#) pane of [VisualCAD's Home Ribbon Bar](#), you will see the [VisualCAM 2016](#) main menu item.
2. Drop-down the menu and pick [MILL](#) to load the [MILL](#) module.



3. Docked on the left you will see the **Machining Browser** and the **Machining Objects Browser**. When you first run VisualCAM 2016, these two browsers may be docked side by side. However, you can move them anywhere on the screen that feels comfortable for you.



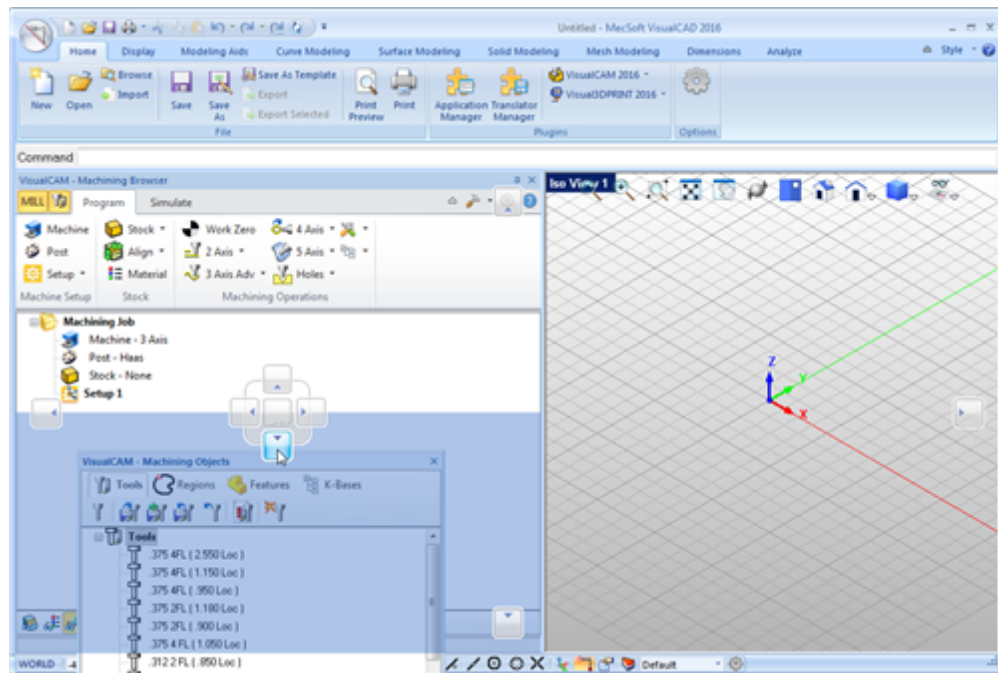
4. For example, let's move the **Machining Objects Browser** so that it displays under the **Machining Browser** on the left. Simply left-click and hold the title bar of the browser and drag it around on your screen.



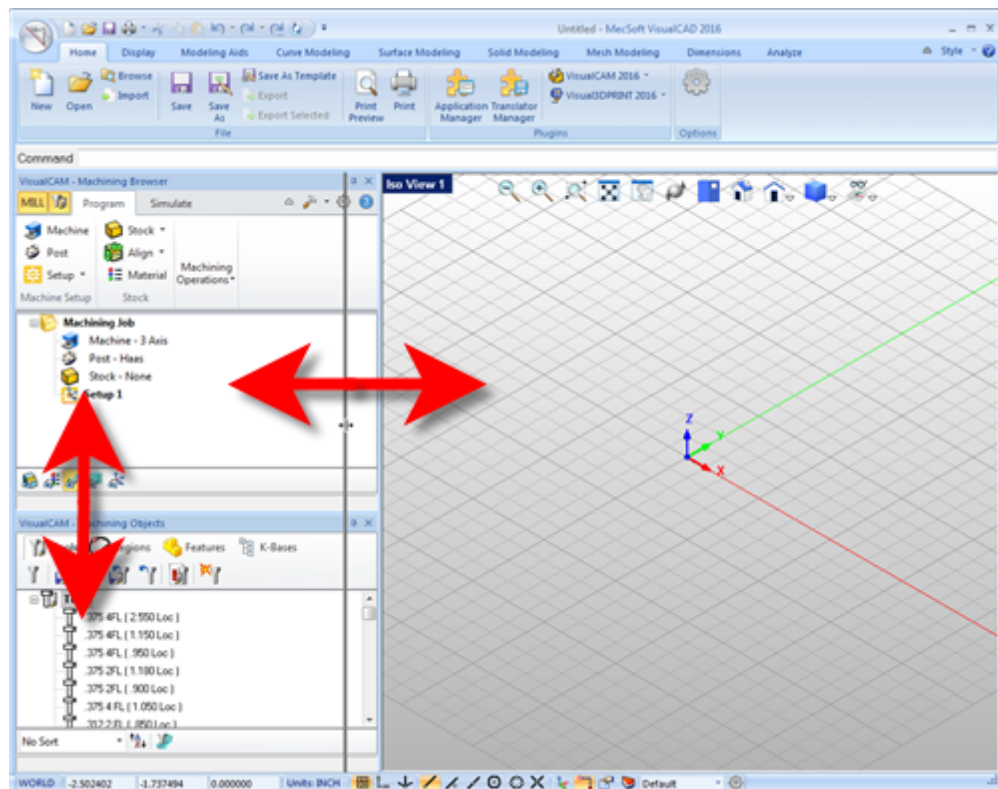
While doing so, you will see the docking widget display in the background with directional buttons allowing you to choose screen locations relative to the active window.

5. We'll drag the **Machining Objects Browser** over the base of the **Machining Browser** until the cursor activates the bottom directional button.

When the preview of the new location displays, let go of the right-mouse button and the browser will move to that location.



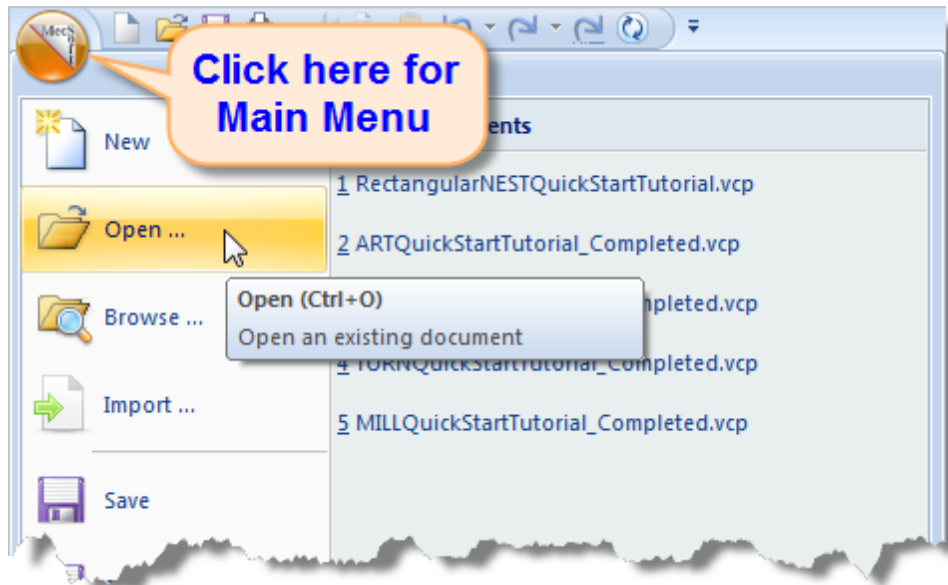
6. You can also resize the height and width of each browser making sure that all of the command icons and menus are easily accessible.



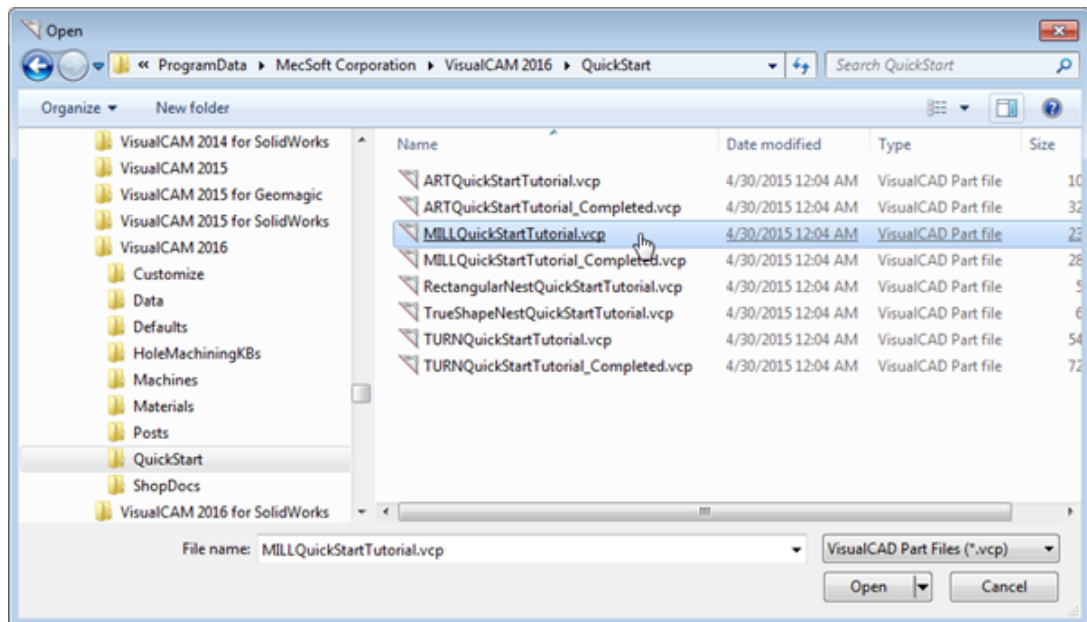
## 2.4 Loading the Part Model

“Part” refers to the geometry that represents the final manufactured product. You can create parts within VisualCAD or import geometry created in another CAD system.

1. From VisualCAD's Main Menu, select Open.



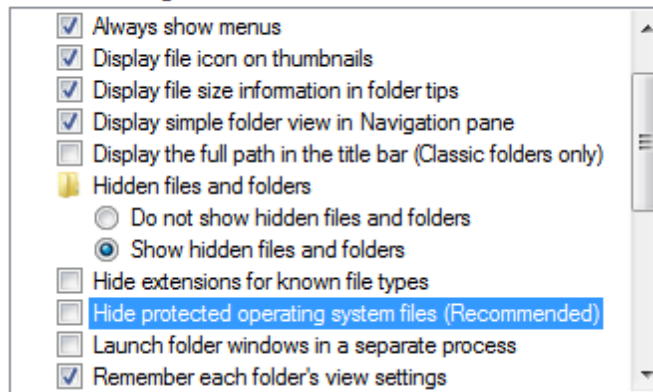
2. From the Open dialog box, select the MILLQuickStartTutorial.vcp file from the C:\ProgramData\MecSoft Corporation\VisualCAM 2016\QuickStart\ folder. As mentioned before, it is advisable to make a copy of this part at a suitable alternative folder so that you have write privileges to modify the part.



! By default, the **ProgramData** folder is "hidden" from view. Here are the steps to Show hidden files and folders:

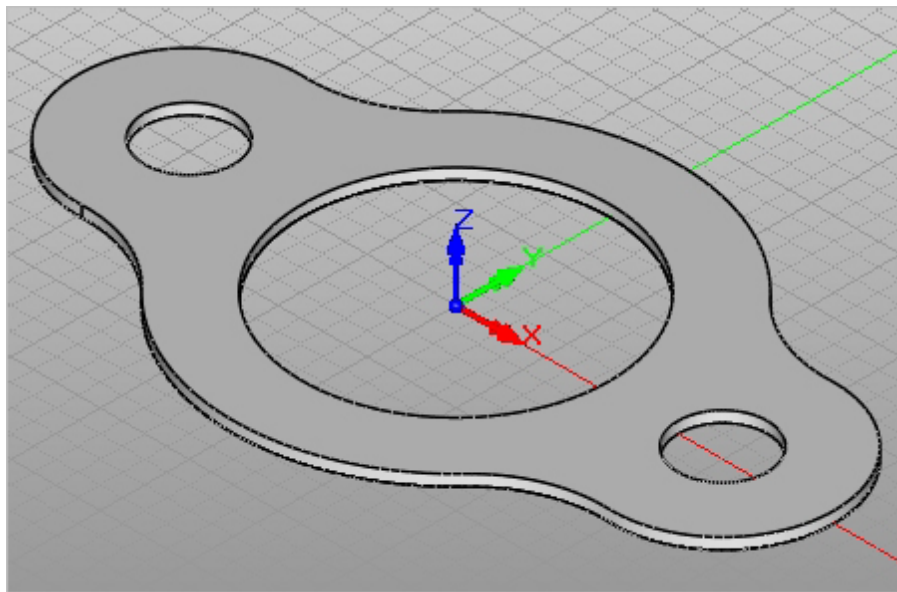
1. For **Windows7/8** users: Go to **Control Panel > Appearance and Personalization > Folder Options** (Windows XP users can locate folder options under **Control Panel**).
2. Select **View** tab and under advanced settings select **Show Hidden files and folders**, clear the check boxes for:
  - Hide extensions for known file types
  - Hide protected operating system files (Recommended)

Advanced settings:



3. Click **Apply** and **OK**.

The part appears as shown below



MILLQuickStartTutorial.vcp

**!** You can import solid models, Stereo-Lithography (both [ASCII](#) and binary) format files. Surfaces and Solids can be imported from [IGES](#), [STEP](#), [Rhino \(\\*.3dm\)](#), [Parasolids \(\\*.x\\_t, \\*.X\\_b\)](#), [SAT](#) and [DXF / DWG](#) files. Faceted (triangulated) models can be imported from [STL](#), [VRML](#), [Raw Triangle](#), or [Rhino Mesh](#).

3. From the [View](#) toolbar, select the [Isometric View](#) to work in.



## 2.5 Machining Strategy

Based on the type of geometry of this part, we will machine this model out of a 10 x 6 x 1/8 inch poplar wood sheet. Since the part is relatively thin and prismatic, we will machine this out by using only a single type of machining operation - 2-½ axis machining method called [Profiling](#). We will also use just a single 0.5 inch flat end mill for performing all machining. We will also assume that the wooden sheet will be held to the machine table or the spoil sheet on the table using double-sided tape or a vacuum table requiring no clamps or fixtures.

## 2.6 Main Programming Steps

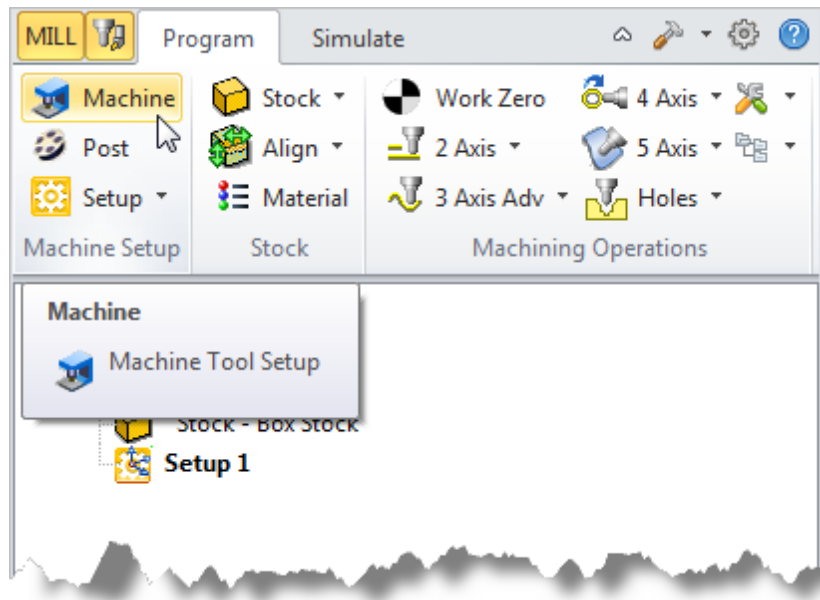
The following steps will be followed in machining this model. Some of these steps will have to be performed just once and others may have to be repeated to complete the machining.

1. [Define](#) the [Machine](#) and [Post-processor](#) to use.
2. Define the [Machining Setup](#) including [Stock Geometry](#), [Material](#) and [Work Zero](#).
3. [Create](#) and [Select](#) a [Tool](#) to use for machining.
4. Create the [Machining Operations](#) including the [Feeds and Speeds](#), the [Clearance Plane](#) and other [Cutting Parameters](#).
5. [Generate](#) the toolpaths.
6. [Simulate](#) the toolpaths.
7. [Post Process](#) the toolpaths.
8. [Generate](#) Shop documentation.

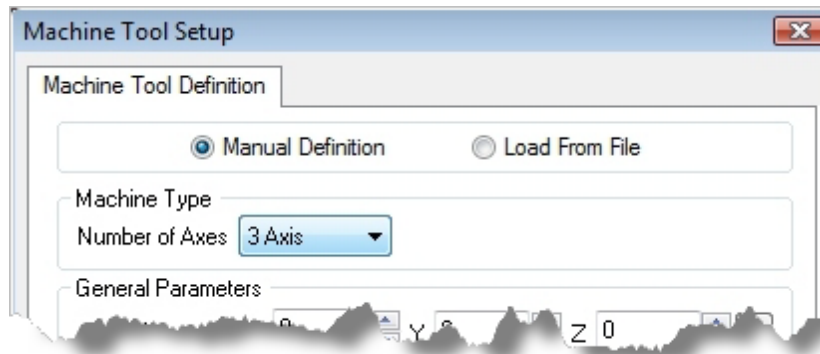
## 2.7 Define the Machine Tool

Let's start by defining the [Machine](#) to use for this job.

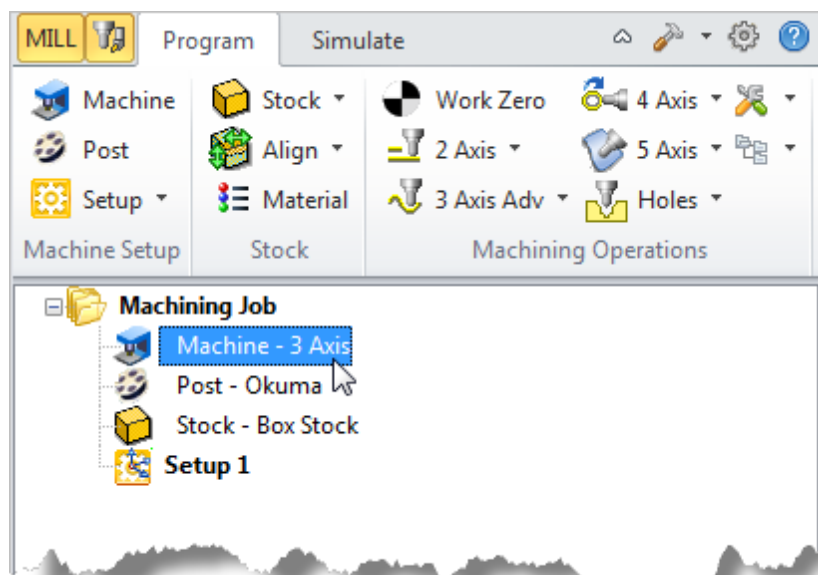
1. From the [Program](#) tab select [Machine](#) to display the dialog box.



2. Under **Machine Type**, set the **Number of Axes** to 3 Axis.



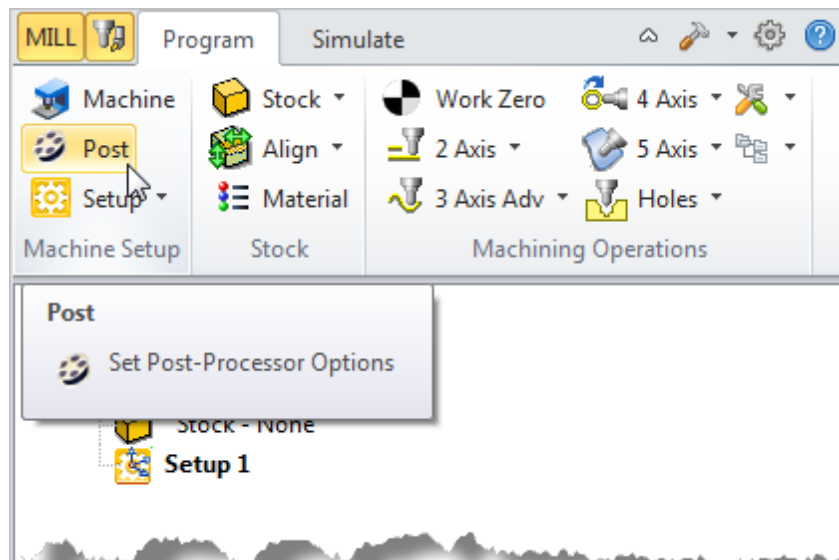
3. Pick **OK** and notice that the **Machine** type now appears under **Machining Job** in the Machining Browser.



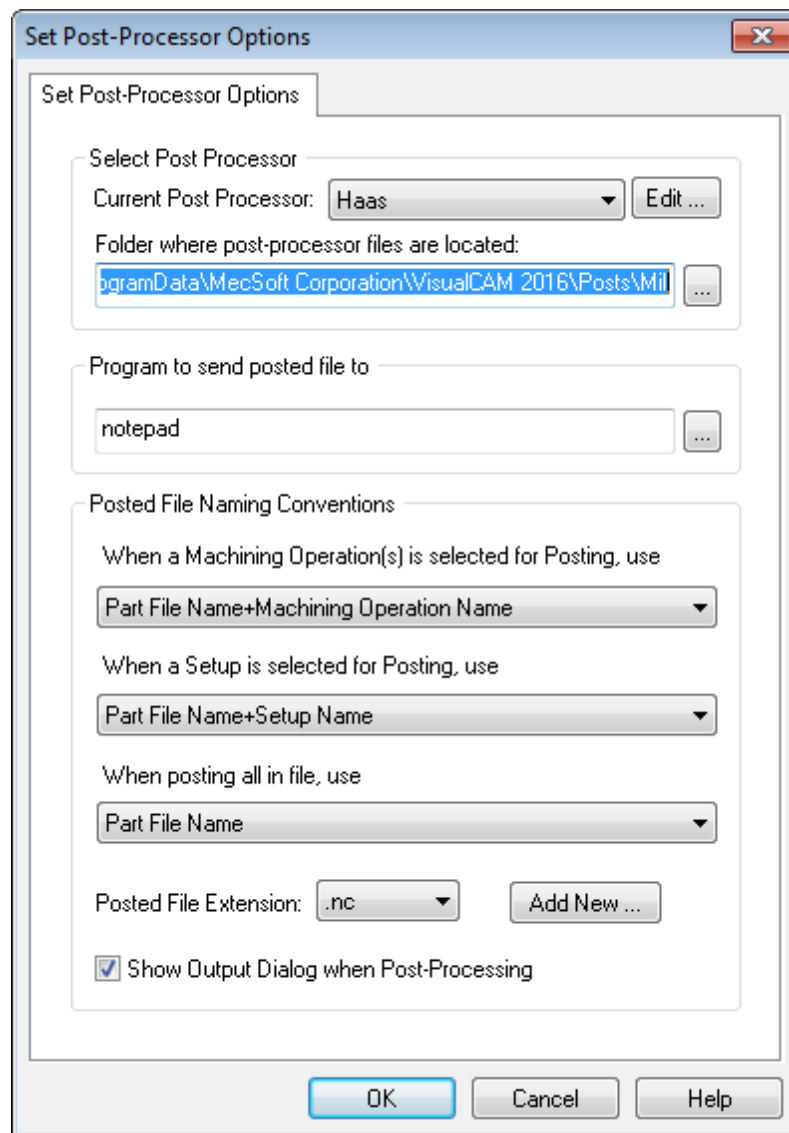
## 2.8 Select the Post Processor

Next, we'll define the [Post Processor](#).

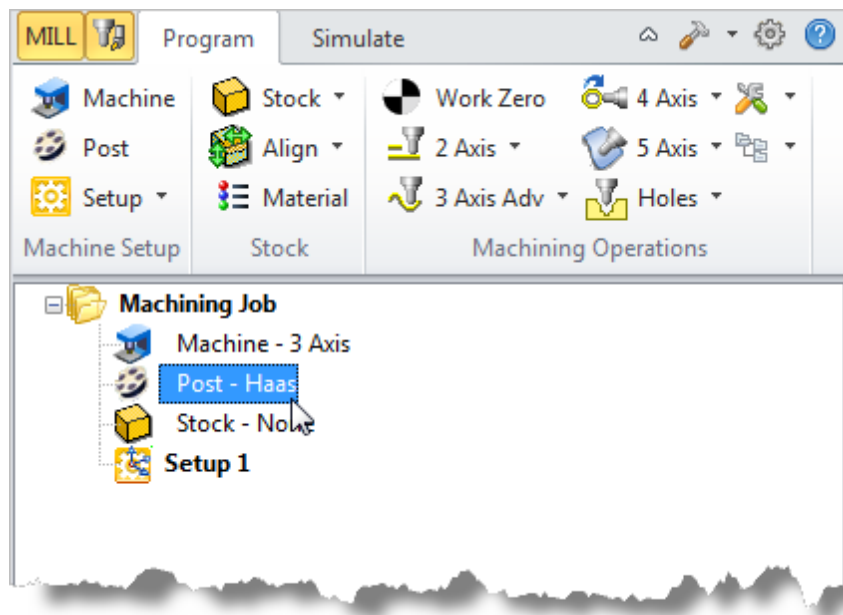
1. From the [Program](#) tab select [Post](#) to display the dialog.



2. For the [Current Post Processor](#), select [Haas](#) from the list of available posts.
3. Then set the [Posted File Extension](#) to [.nc](#). Other file extensions are available depending on your machine requirements.



4. Pick **OK** and notice that the **Post** type now appears under **Machining Job** in the **Machining Browser**.




By default, post processor files are located under  
<C:\ProgramData\MecSoft Corporation\VisualCAM 2016\Posts\MILL\>  
The program to send the posted output data to is set to notepad.

## The Setup

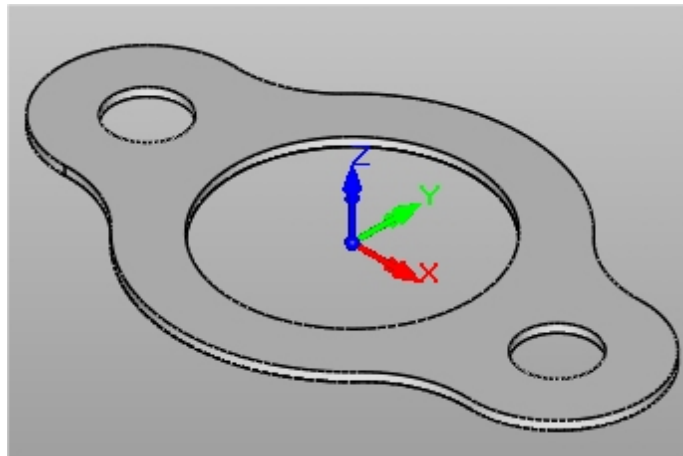
### 3.1 Define the Machining Setup - Skip this section if in Standard or Expert Configuration

Now let's define the [Machining Setup](#). The [Machining Setup](#) allows you to orient the [Machine Coordinate System](#) such that the part is aligned in the exactly same way as it would be fixtured on the machine tool for cutting.

 This functionality is available only in the [Professional](#) and [Premium](#) configurations of the product. In the other configurations ([Standard](#) and [Expert](#)) you will have to use the [CAD](#) tools to orient the part geometry so that it is in the correct orientation for machining.

If there is no [Setup1](#) listed under your [Machining Job](#), the system automatically creates one when a work zero or an operation is generated.

However, by default, the [MCS](#) ([Machine Coordinate System](#)) is already aligned with the [WCS](#) ([World Coordinate System](#)) so this step is not required for this part.

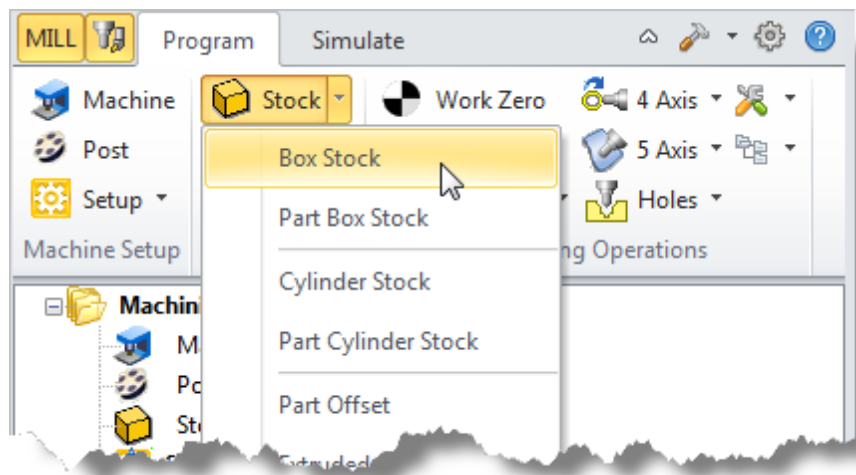


However, in production you can have multiple setups and assign different machining orientations for each.

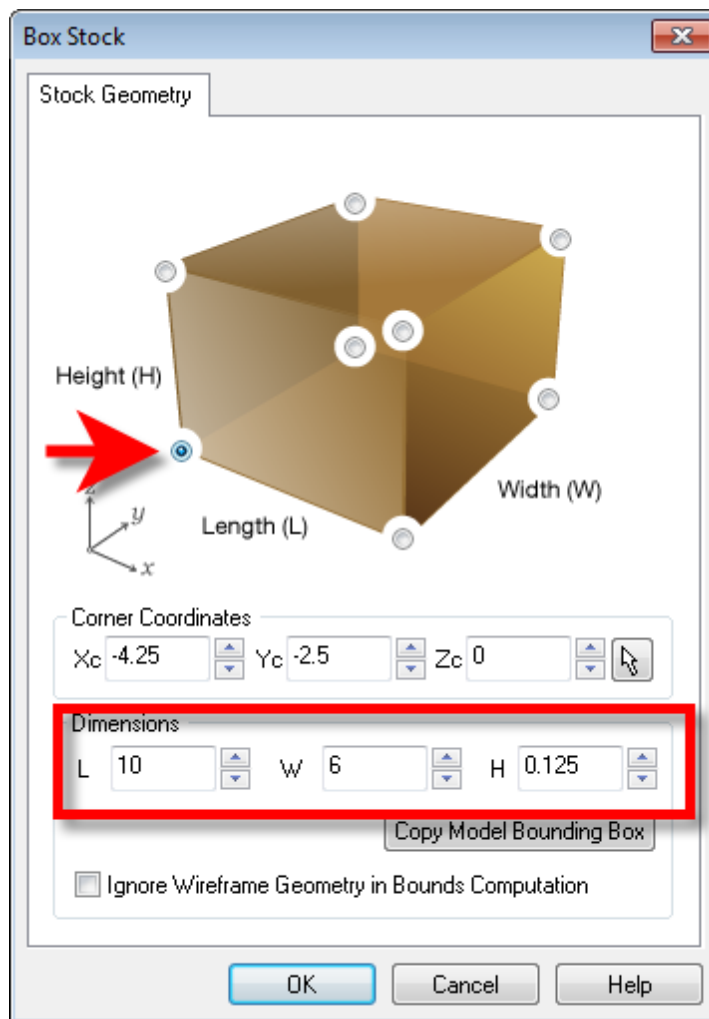
### 3.2 Create Stock Geometry


In this step we'll define the raw stock from which to cut the part.

1. From the [Program](#) Tab select [Stock](#) and then select [Box Stock](#) from the menu to display the dialog.

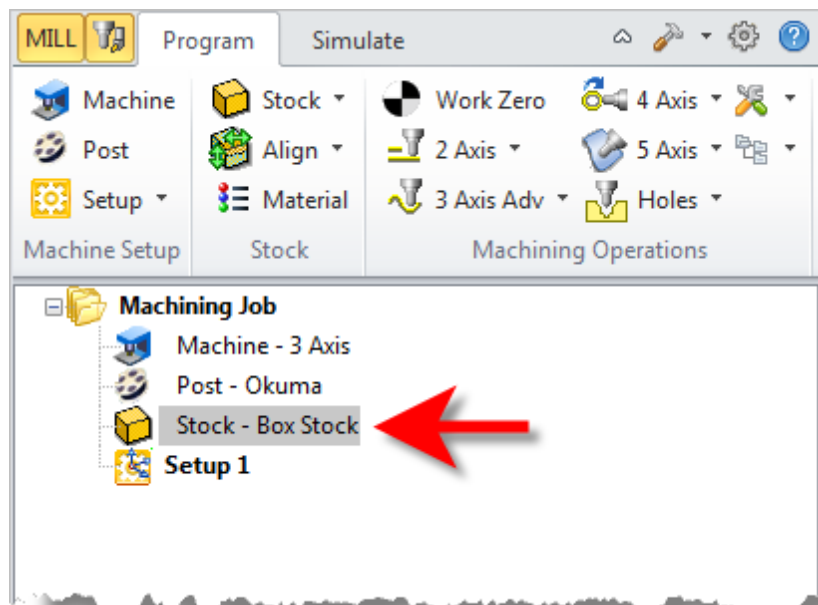


2. Under **Dimensions**, set the **Length L** to 10.0, **Width W** to 6.0 and **Height H** to 0.125. Note that the stock dimensions you enter are measured from the corner of the bounding box selected in this dialog.



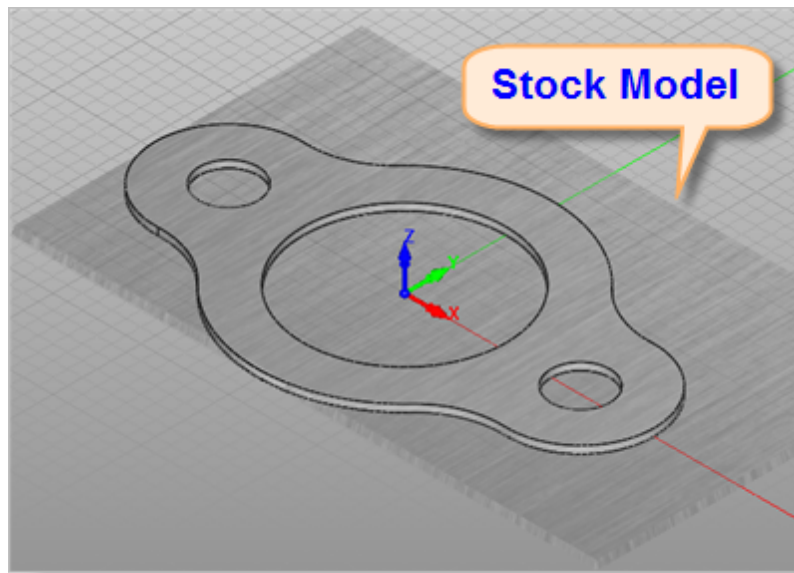
 The dimensions of the stock are interpreted in relation to the corner selected in the dialog box above. For example if the corner of the box is selected as the **Bottom South West** corner (as shown in the picture above), the **Length (L)** is interpreted to be along the **+X** axis, the **Width (W)** along the **+Y** axis and the **Height (H)** along the **+Z** axis. The direction of the dimensions will change depending on the corner selected. For example if the **Top South West** corner is selected, then the **Height (H)** is interpreted to be along the **-Z** axis and so the stock will extend below the corner.

3. Pick **OK** and notice that the **Stock** type now appears under **Machining Job** in the **Machining Browser**.



4. If the stock does not display on the screen, select the [Stock Visibility](#) icon located at the base of the [Machining Browser](#).

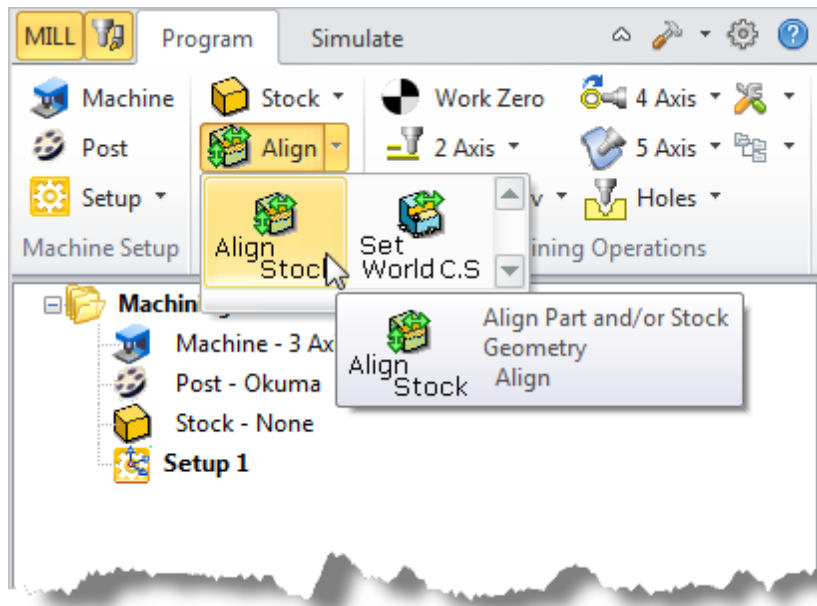




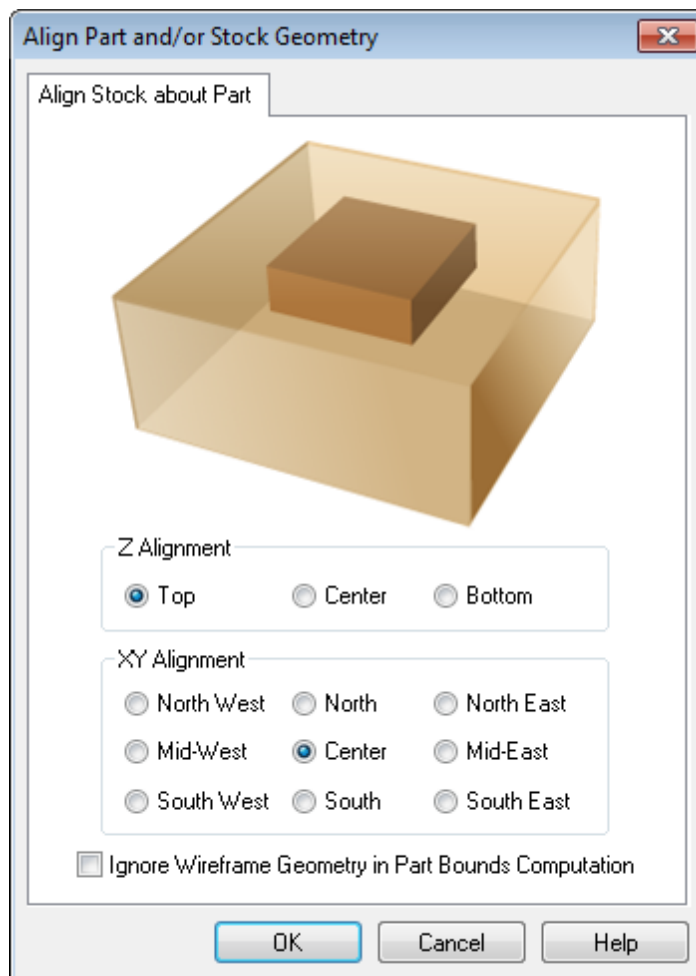
### 3.3 Align Part and Stock

Once the stock model is created you can move it in alignment with the part if needed.

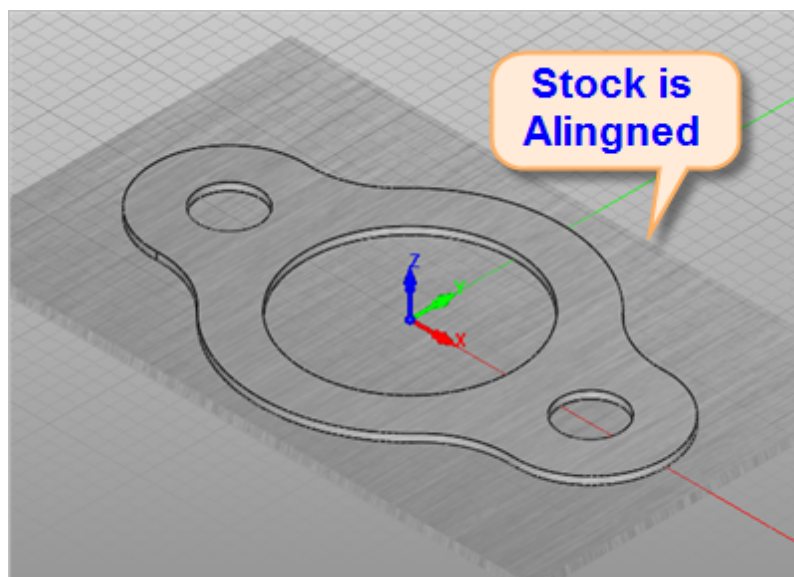
1. From the **Program** Tab select **Align** and then **Align Stock** from the menu to display the dialog.



2. For **Z Alignment** select **Top** and for **XY Alignment** select **Center** and then pick **OK**.



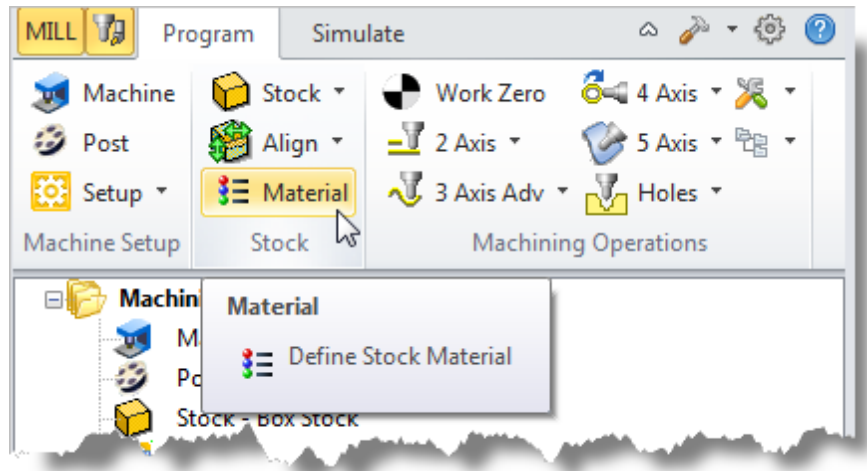
The stock is now aligned to the center of the part in *XY* and the Top of the part in *Z*.



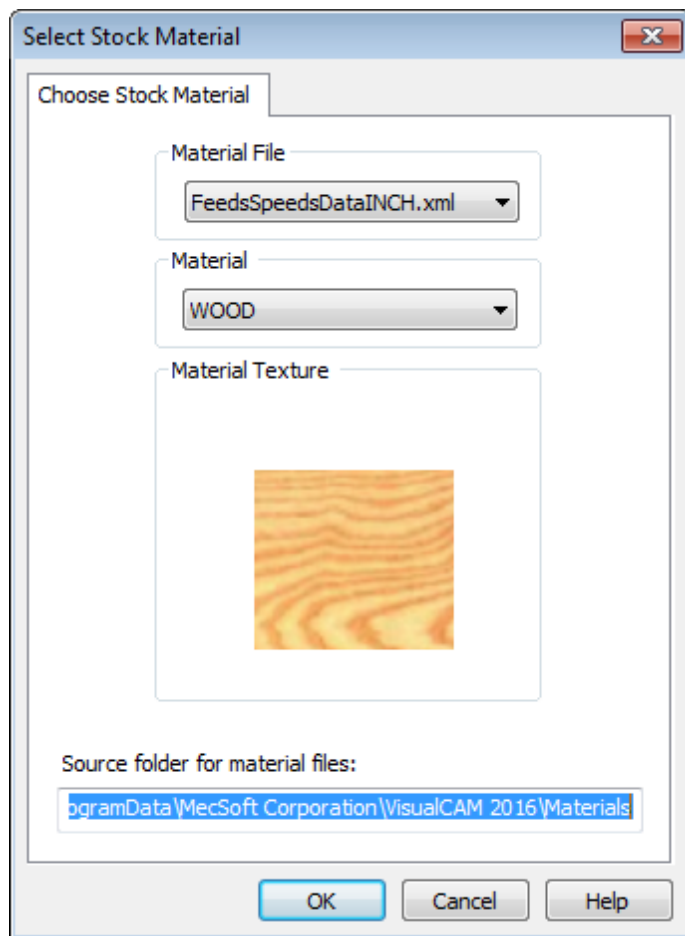
### 3.4 Specify Material

Next, we'll set the material for the stock geometry.

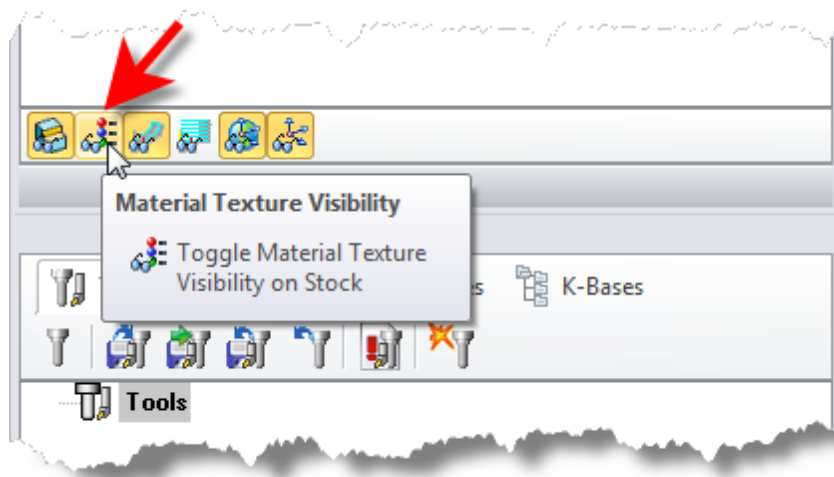
1. From the **Program** tab select **Material** to display the dialog box.

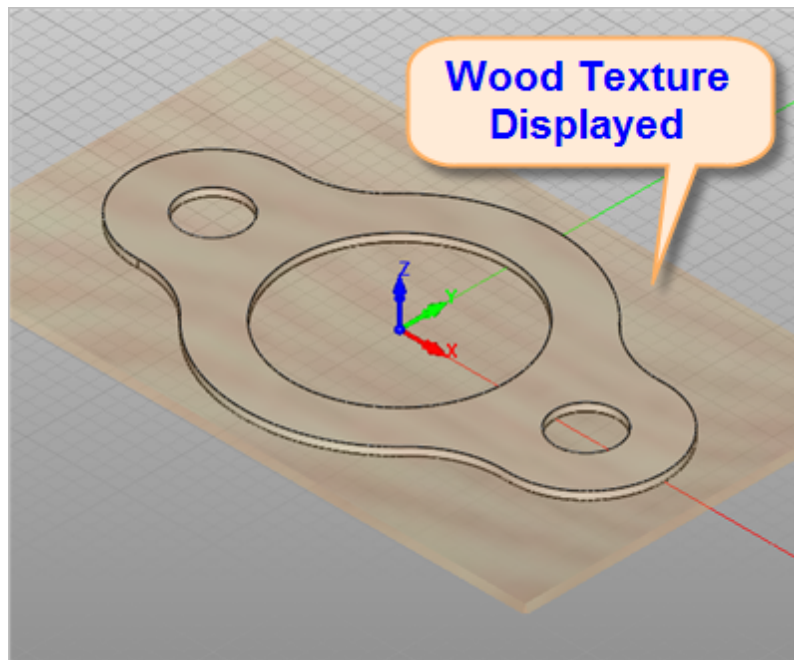


2. For **Material**, select **Wood** from the list of available materials and then pick **OK**.



3. If the material texture does not display on the stock, select the [Material Texture Visibility](#) icon located at the base of the [Machining Browser](#).

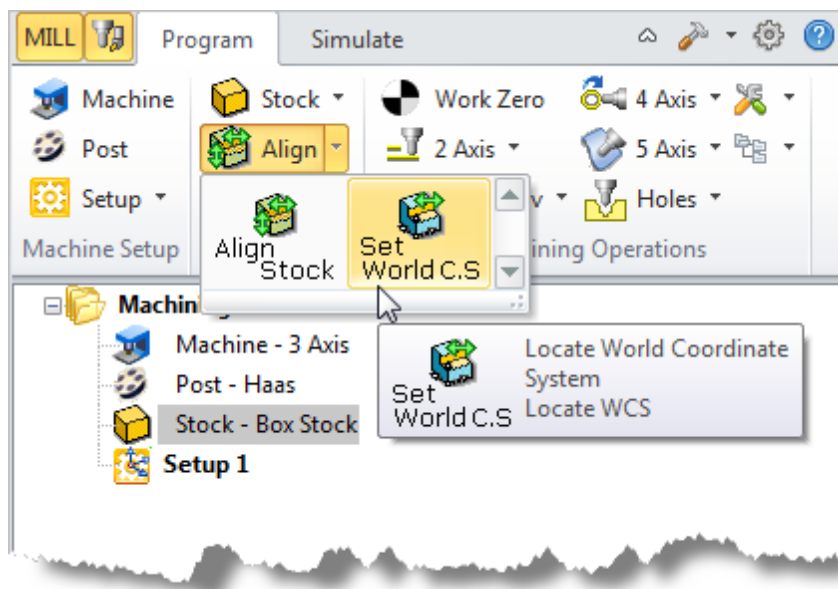




### 3.5 Set Work Zero

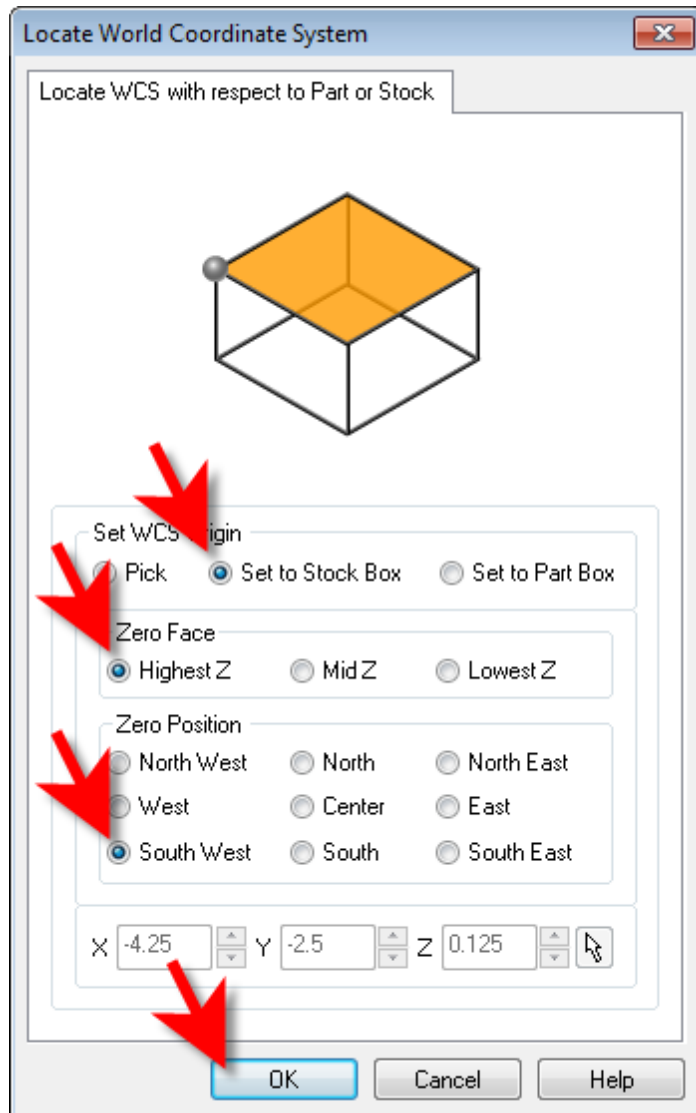
Now that the stock is aligned to the part geometry, in this step, we will establish the work coordinate origin. This location defines the zero point from which all tool path points are interpreted by the controller.

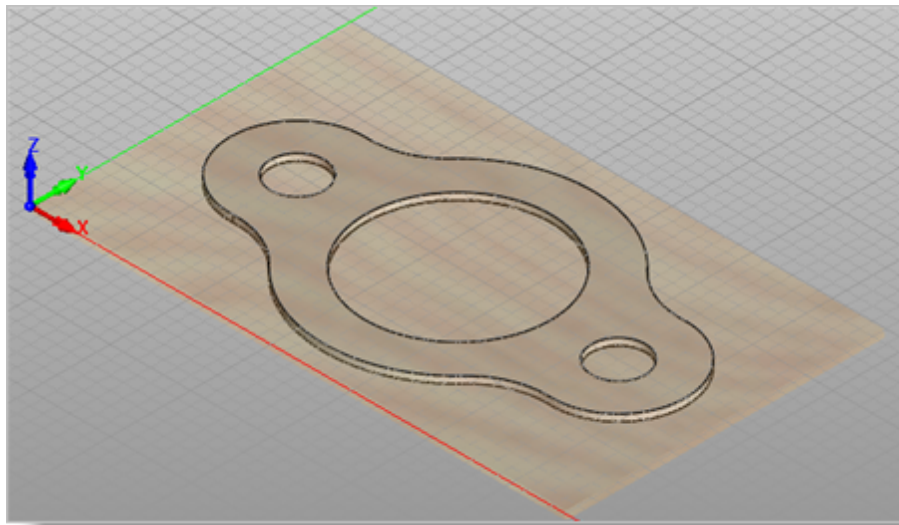
1. From the **Program** Tab select **Align** and then **Set World CS**.



2. Then select **Set to Stock Box**.

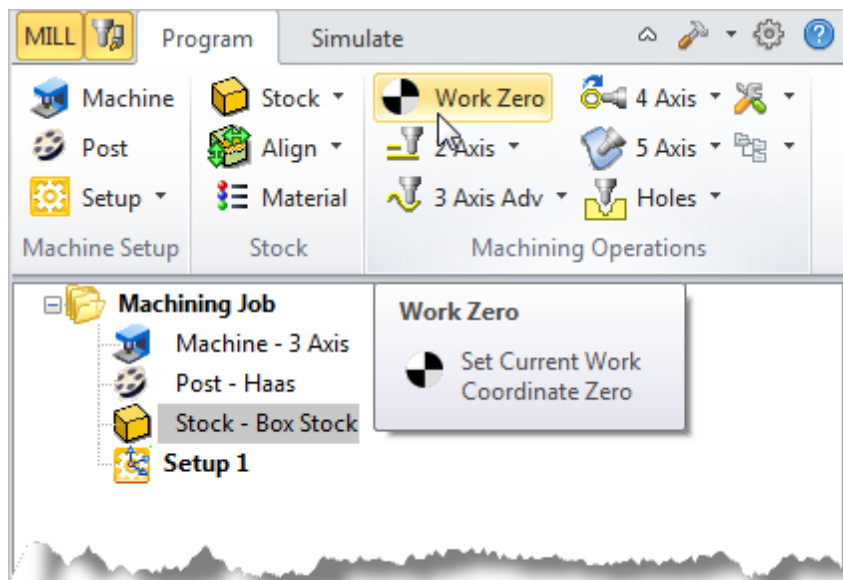
3. Then set **Zero Face** to **Highest Z** and **Zero Position** to **South West** corner. This sets the machine home to the top of the stock material and the southwest corner of the stock geometry.
4. Pick **Generate** and the part and stock geometry are now transformed to the **World Coordinate Origin (WCS)**.



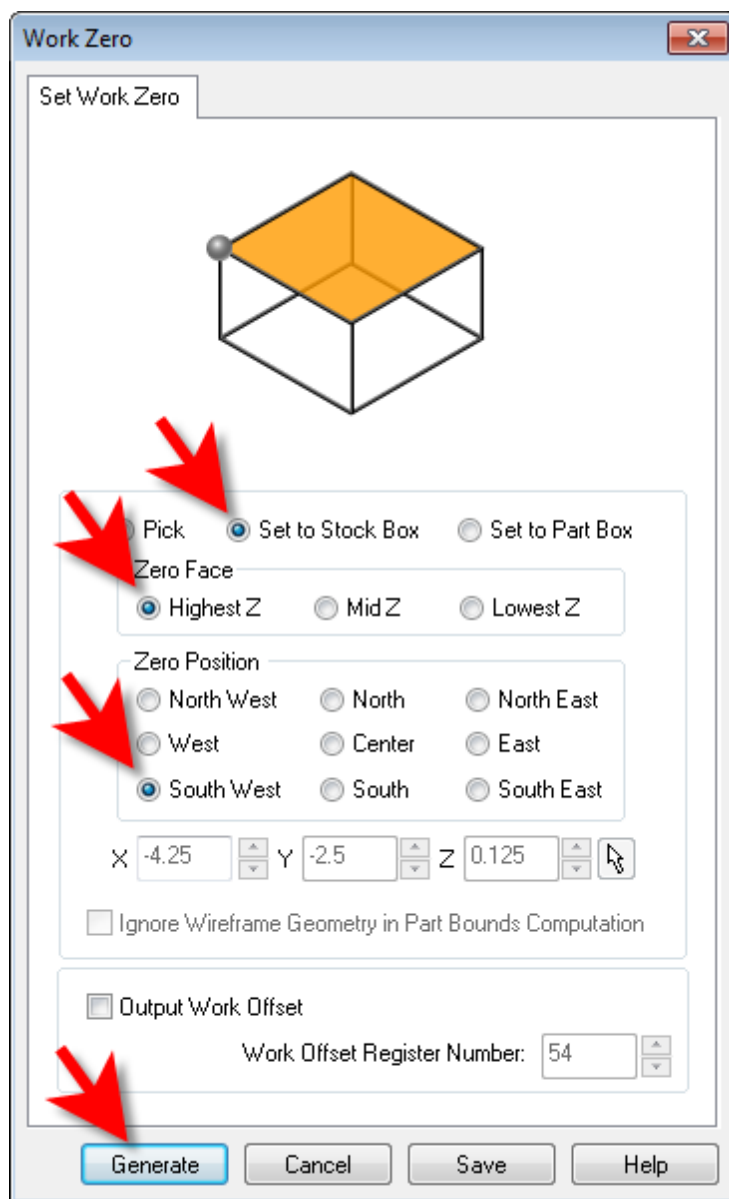


Alternatively you can use **Work Zero** to set the work coordinate origin. Instead of moving the part and stock to the **WCS** origin, this moves the machine coordinate system origin to the specified location.

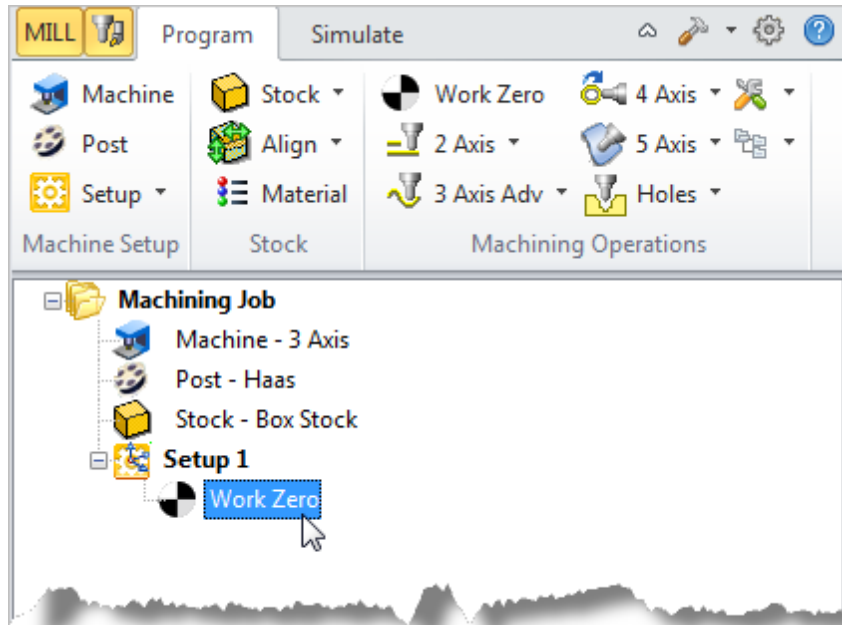
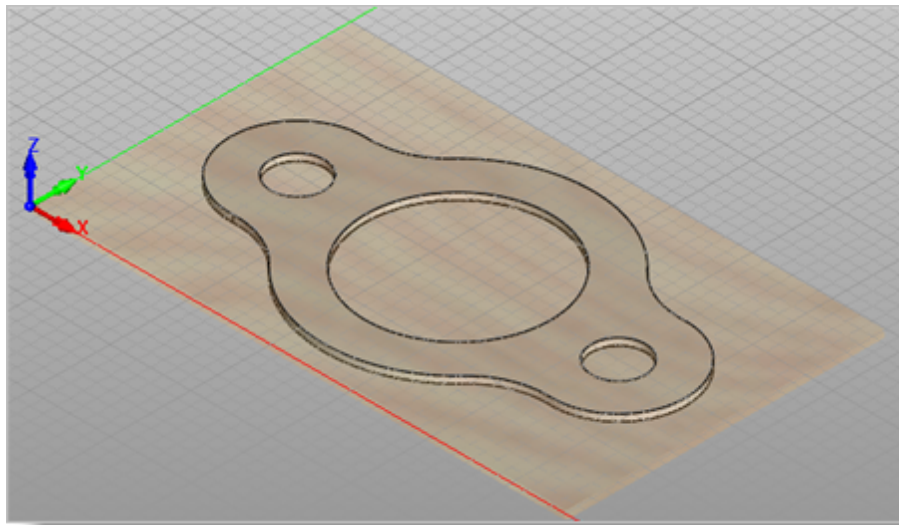
1. From the **Program** Tab select **Work Zero** to display the dialog.



5. Then select **Set to Stock Box**.
6. Then set **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 stock geometry.



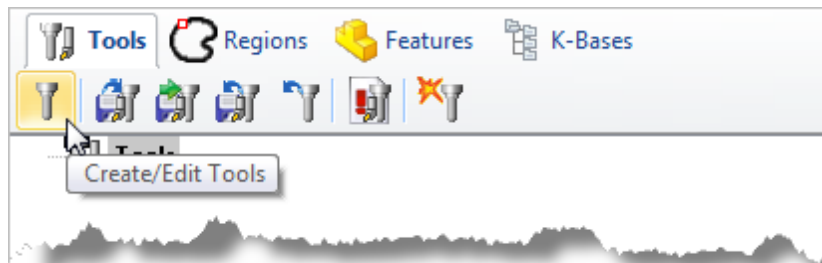
7. Pick **Generate** and notice that the **MCS** is translated and that the **Work Zero** now appears under **Setup 1** in the **Machining Browser**..



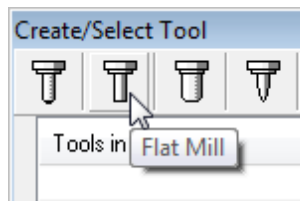
## Create Tools

To machine the above part we will now create a ½ inch (0.5") Flat End Mill.

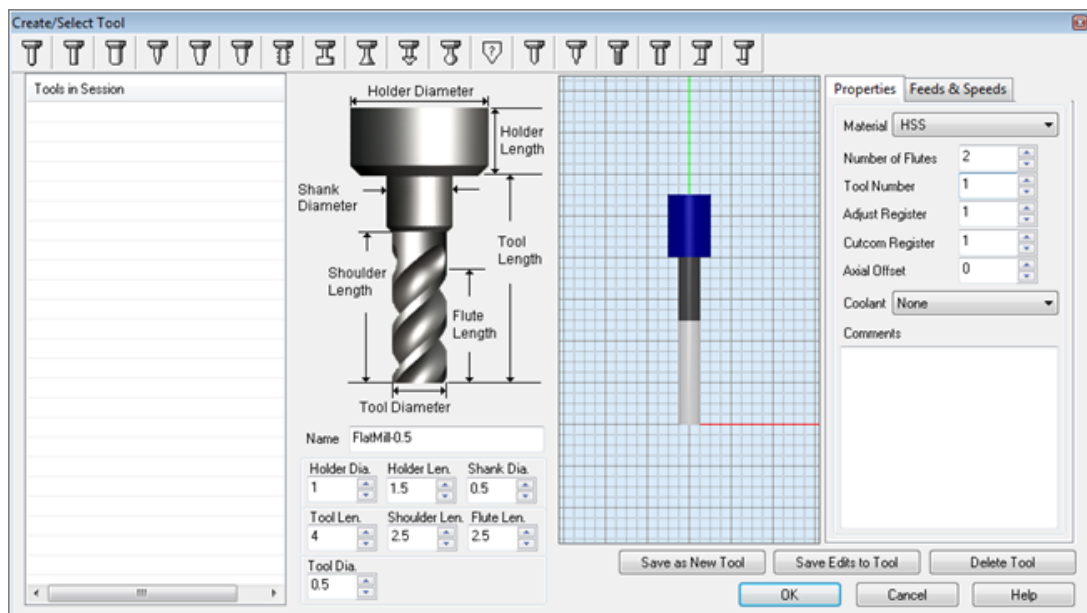
1. Select the **Tools** tab under **Machining Objects Browser** and click **Create/Edit Tools**.



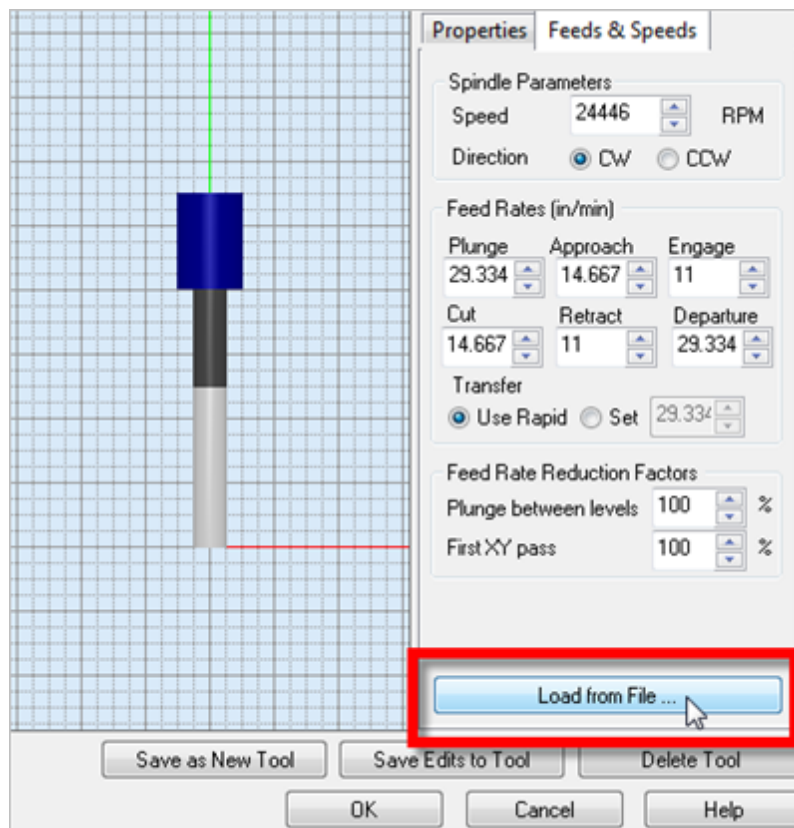
2. Select the **Tool Type** to **Flat Mill**.



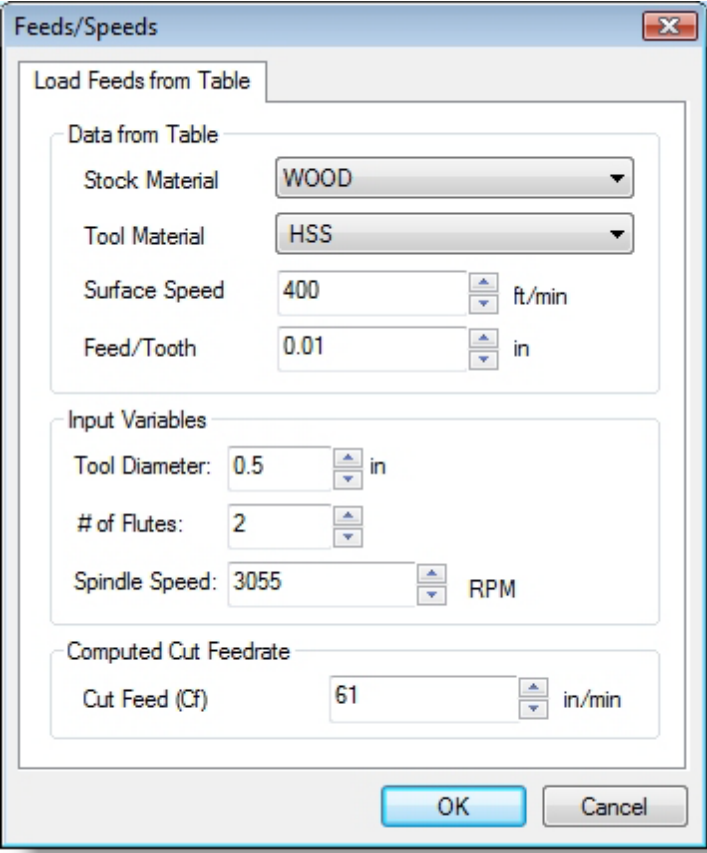
- Set tool Name as **FlatMill-0.5**, Tool Diameter = **0.5**. Under the **Properties** tab set **Material** to **HSS** and **Tool Number** = **1**.



4. Switch to **Feeds and Speeds** tab and click **Load from File**.



5. Set **Stock Material** to **Wood** and **Tool Material** to **HSS**.

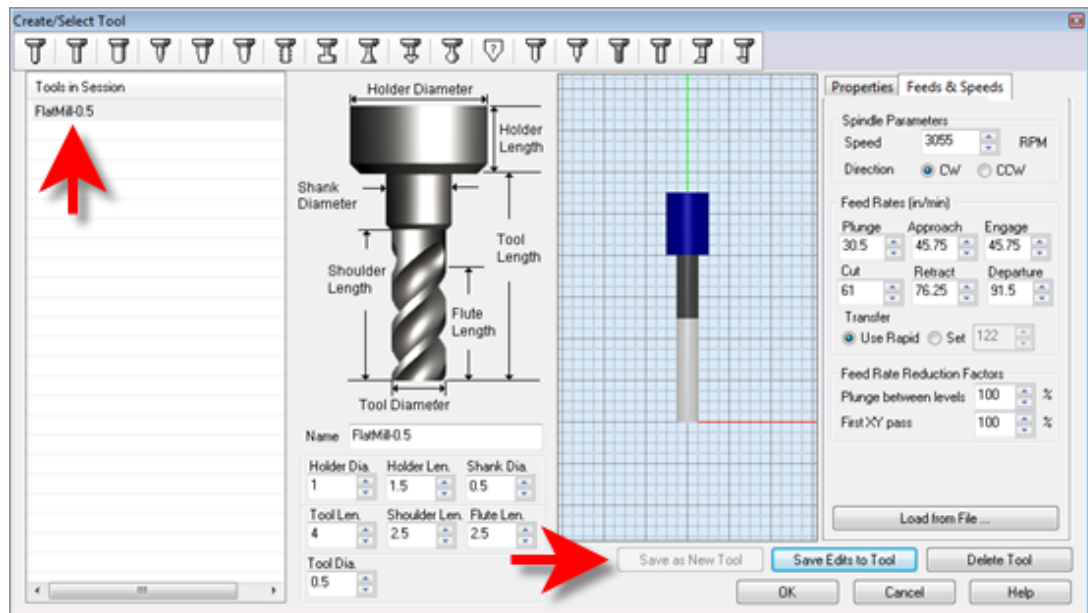


The image shows a software dialog box titled "Feeds/Speeds". It has a tab labeled "Load Feeds from Table". Inside the dialog, there are three main sections: "Data from Table", "Input Variables", and "Computed Cut Feedrate".

- Data from Table:** Contains two dropdown menus: "Stock Material" set to "WOOD" and "Tool Material" set to "HSS". Below these are two input fields: "Surface Speed" with the value "400" and units "ft/min", and "Feed/Tooth" with the value "0.01" and units "in".
- Input Variables:** Contains three input fields: "Tool Diameter" with the value "0.5" and units "in", "# of Flutes" with the value "2", and "Spindle Speed" with the value "3055" and units "RPM".
- Computed Cut Feedrate:** Contains one input field: "Cut Feed (Cf)" with the value "61" and units "in/min".

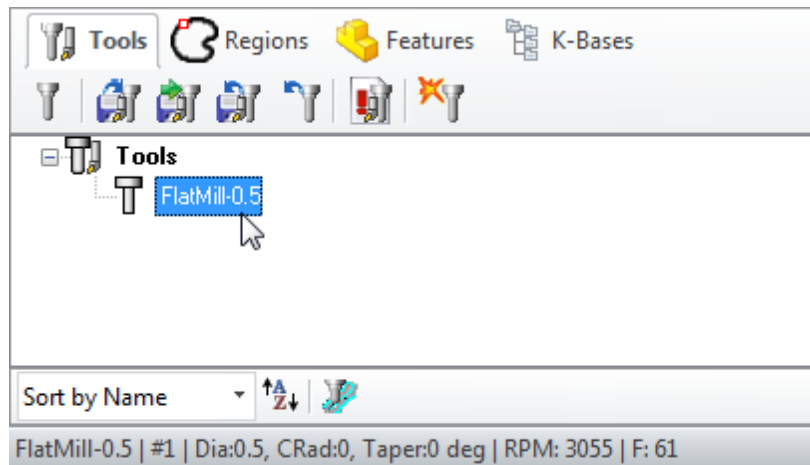
At the bottom of the dialog are two buttons: "OK" and "Cancel".

6. Click **OK** and the computed cut feedrate and spindle speed are transferred to the **Feeds and Speeds** tab of the tool dialog.
7. Click **Save as New Tool** to save the tool. The tool is now created and listed under **Tools in Session** on the left side of the **Create/Select Tool** dialog. Click **OK** to close the dialog.



! 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 tool is now listed under **Tools** tab in **Machining Objects** browser.



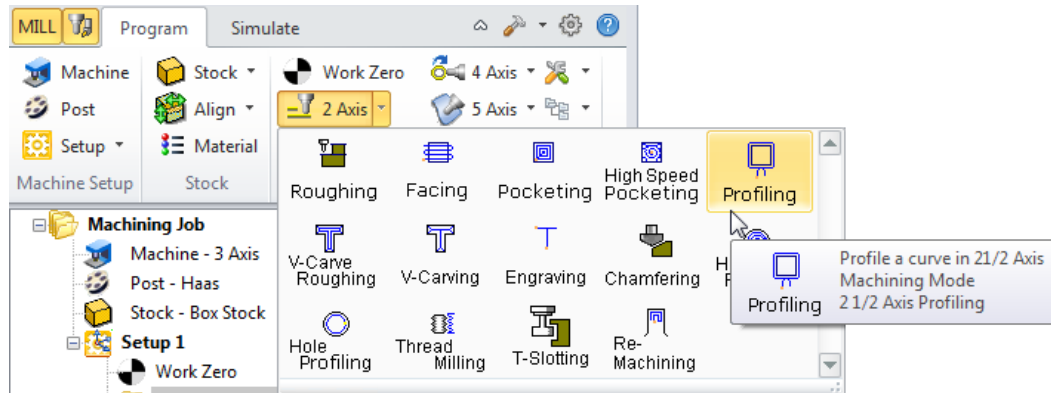
! To save Tools to a library, Click **Save Tool library** under the **Tools** tab in the **Machining Objects Browser** and specify a folder location and file name in the **Save as** dialog box. Saving a **Tool** library as a **Knowledge base** file (\*.vkb) saves feeds and speeds with tool properties.

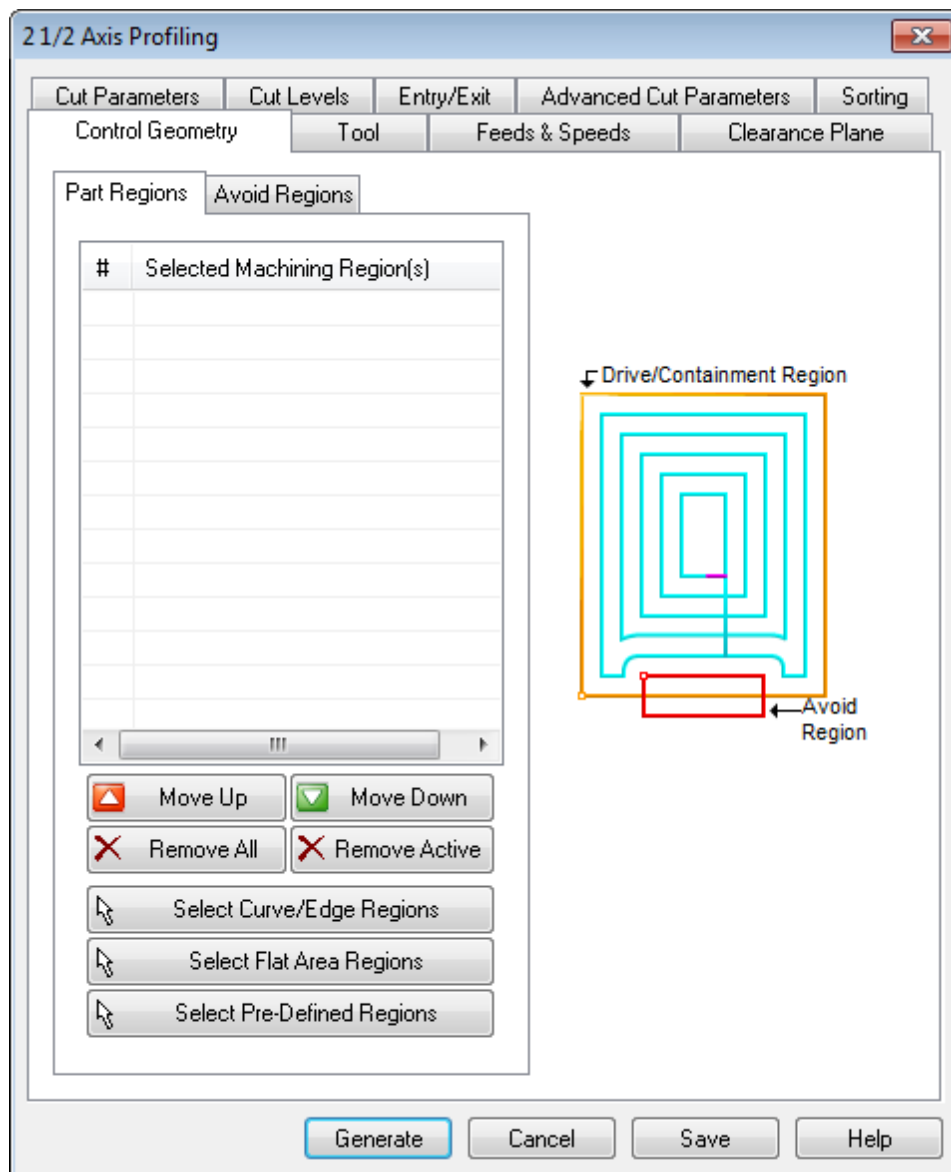
## Create Machining Operations

Now we're ready to create our first Profiling operation.

1. From the **Program** Tab select **2 Axis** and then **Profiling** from the menu of **2 Axis** operations.

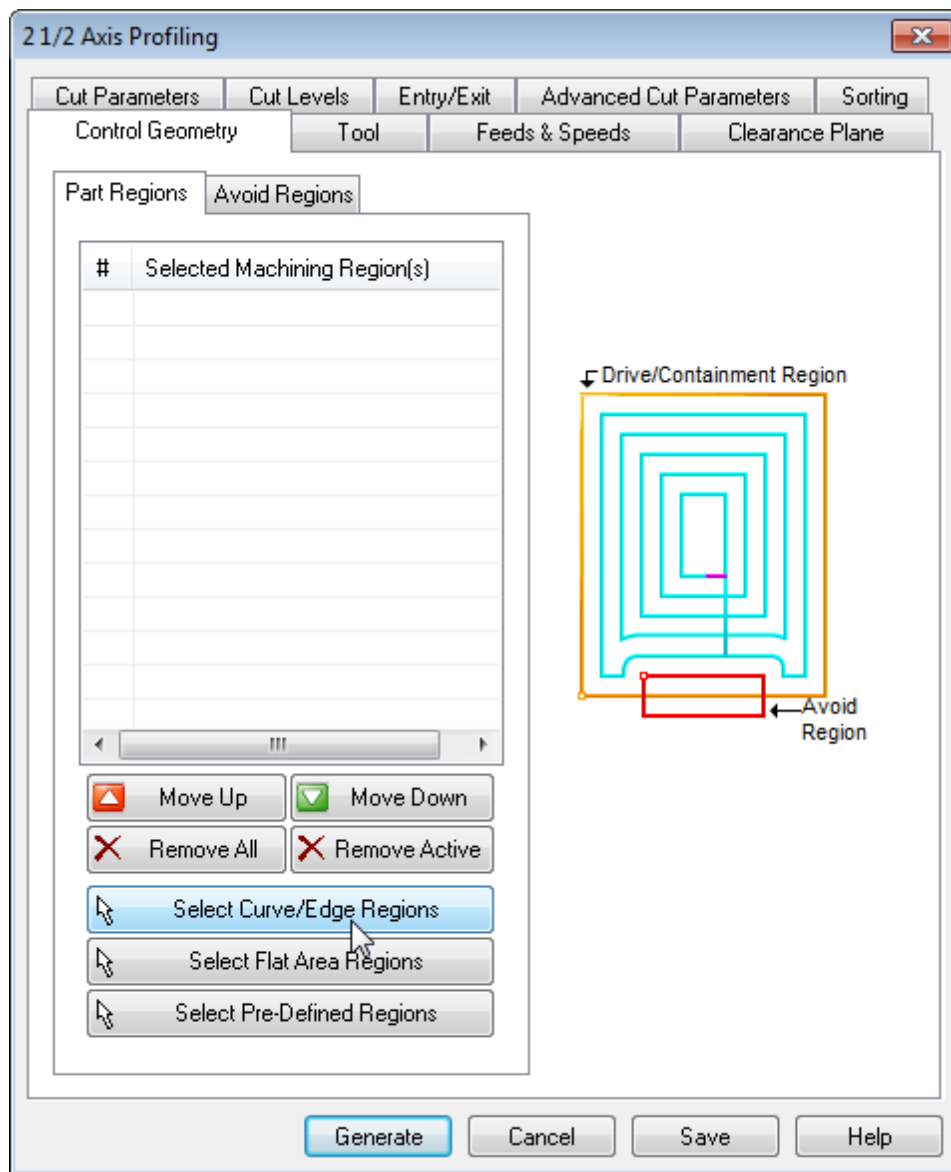
This will display the **2½ Axis Profiling** operations dialog. We will go over the steps for creating the profile operations for the inner features of the **Gasket**.





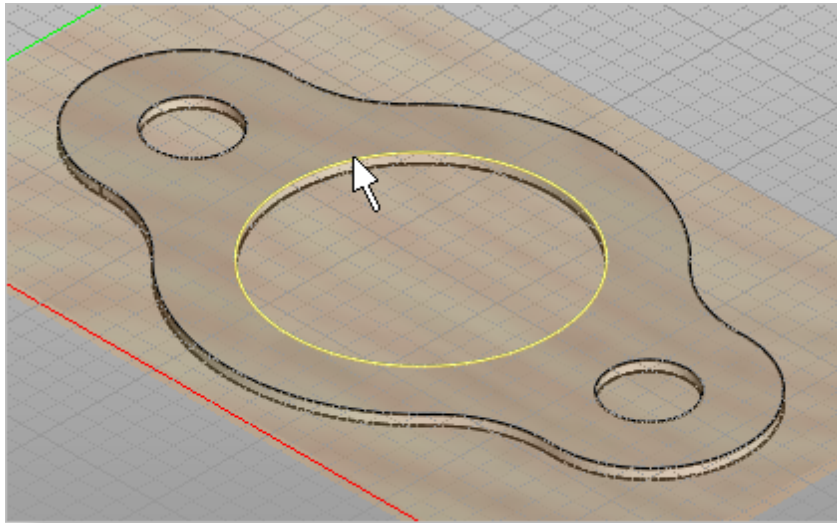
## 5.1 Select Machining Features/Regions

1. Under the [Control Geometry](#) tab pick [Select Curve/Edge Regions](#).

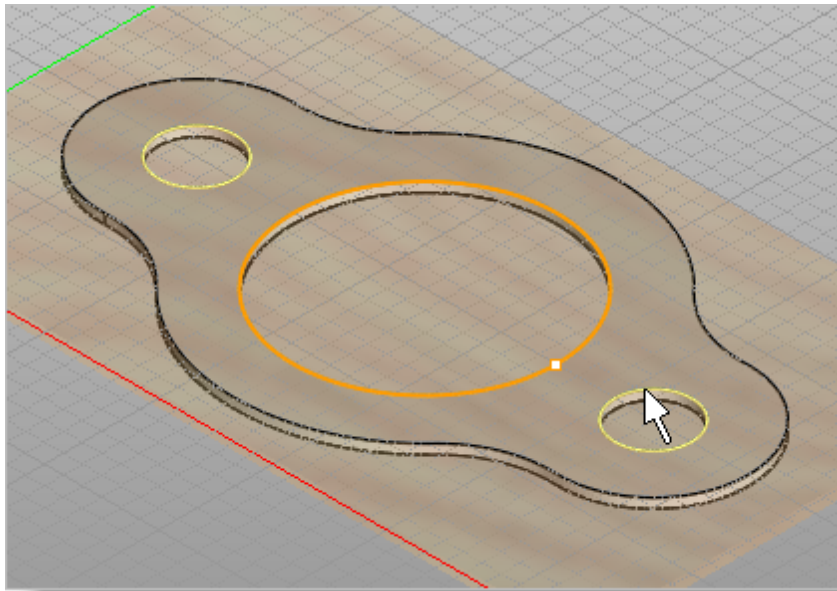


The Profiling operation dialog is now minimized and allows selection of features to machine. We will now select the surface edges of the 3 inside hole features.

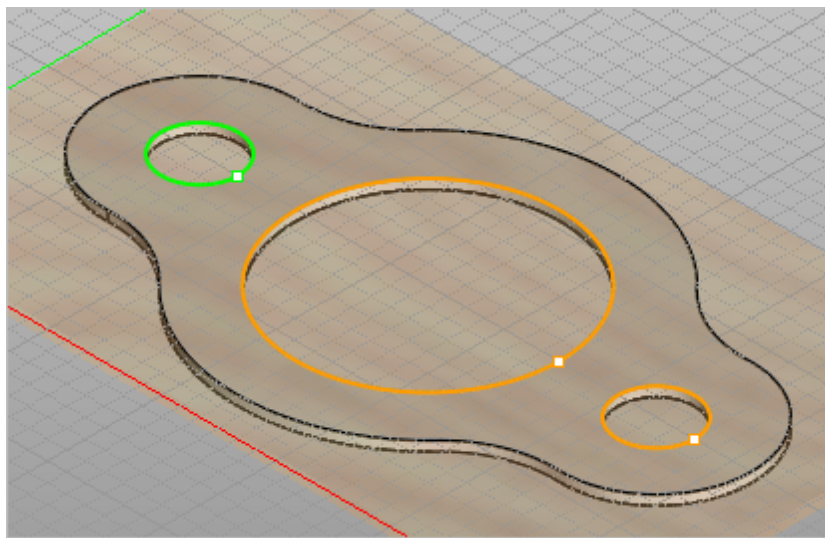
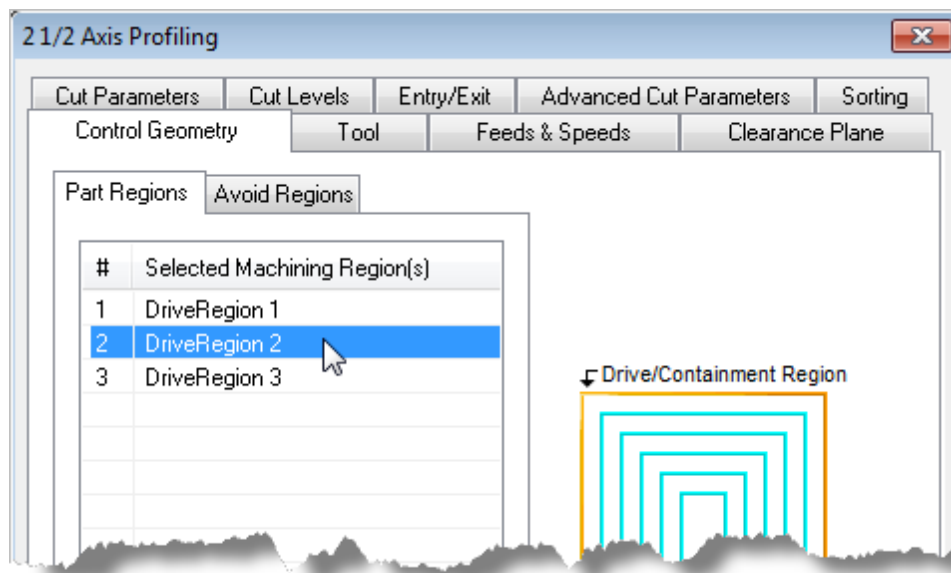
2. Select the first hole by clicking near the upper surface edge as shown below.



3. Repeat to select the edges of the two smaller holes.



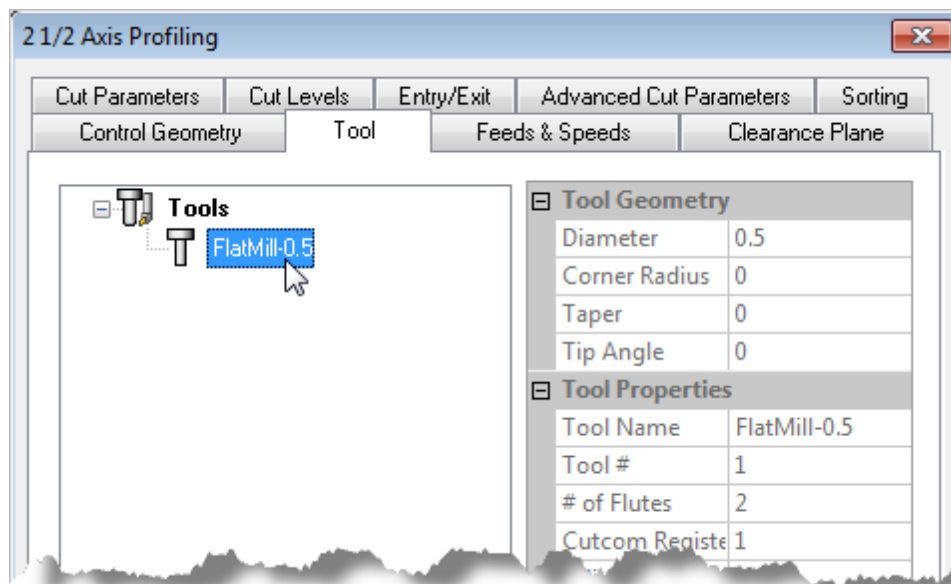
4. Press **Enter** or **right-click** to end the selection.
5. The **2½ Axis Profiling** dialog comes back up displaying the selected **Drive Regions**. They are also highlighted on the part.
6. Notice that selecting a **Drive Region** from the list highlights the corresponding surface edge curve on the part.



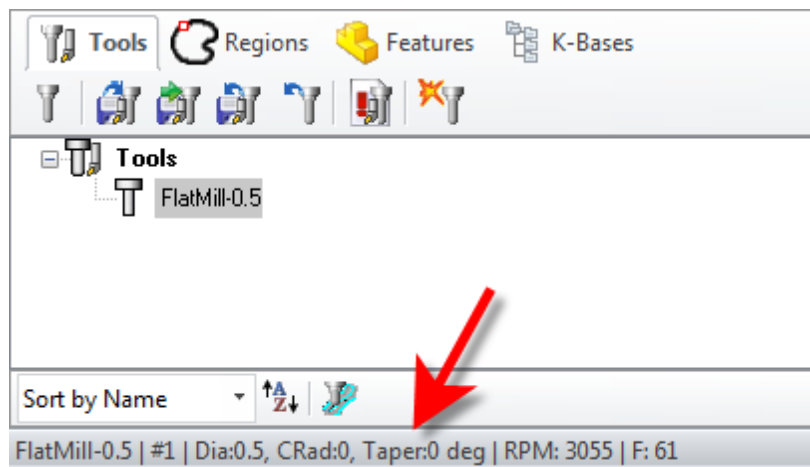
## 5.2 Select Cutting Tool

Now we'll set the **Tool** for our operation:

1. Now we switch to the **Tool** tab of the dialog.
2. Select **Flat Mill-0.5** under **Tools**. The **0.5" Flat End Mill** is now selected as the active tool.



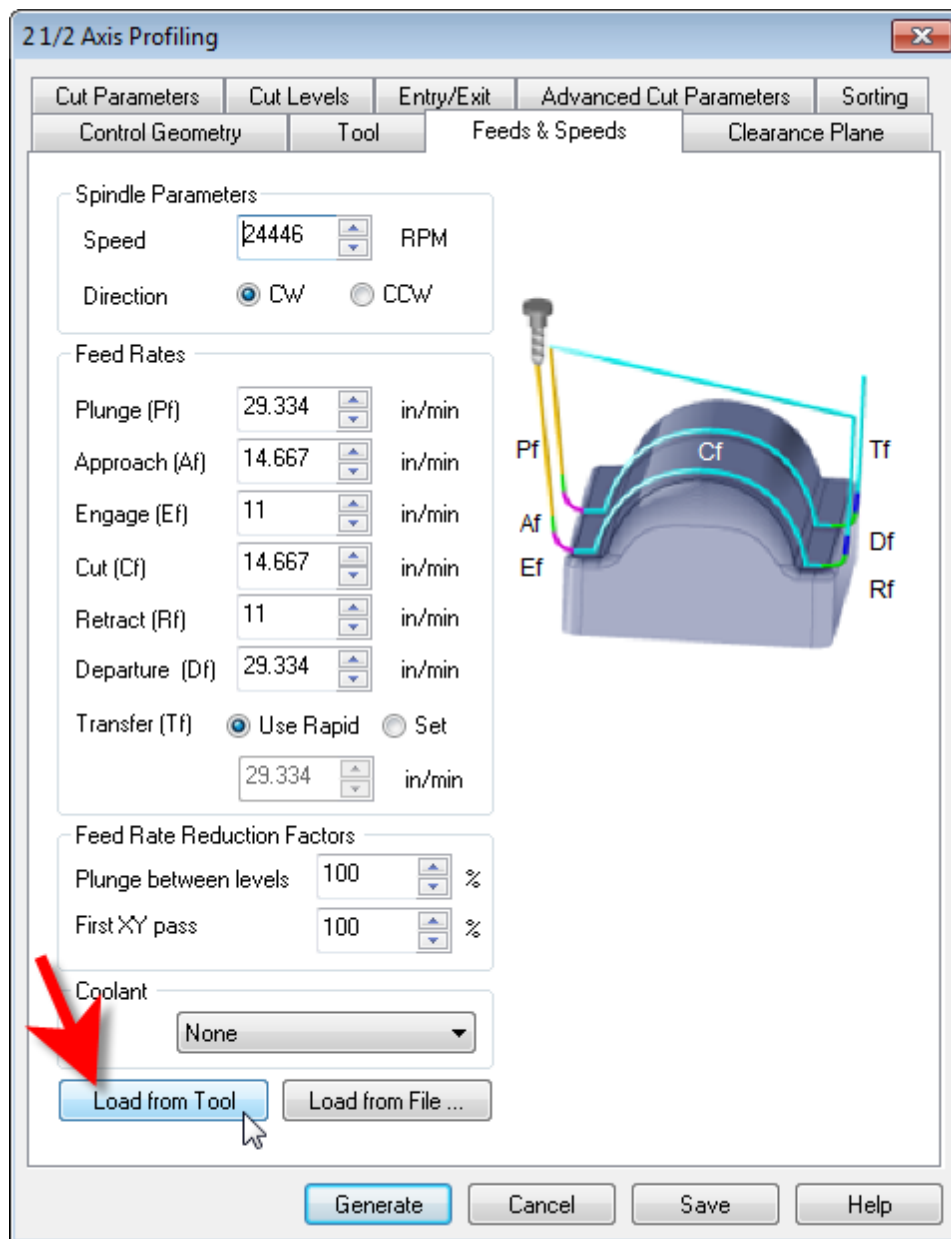
Note that the **Tool** parameters of the currently active tool are always displayed in the status bar at the bottom of the **Machining Objects Browser**.



### 5.3 Set Feeds and Speeds

Now we'll set the **Speeds and Feeds** for our operation:

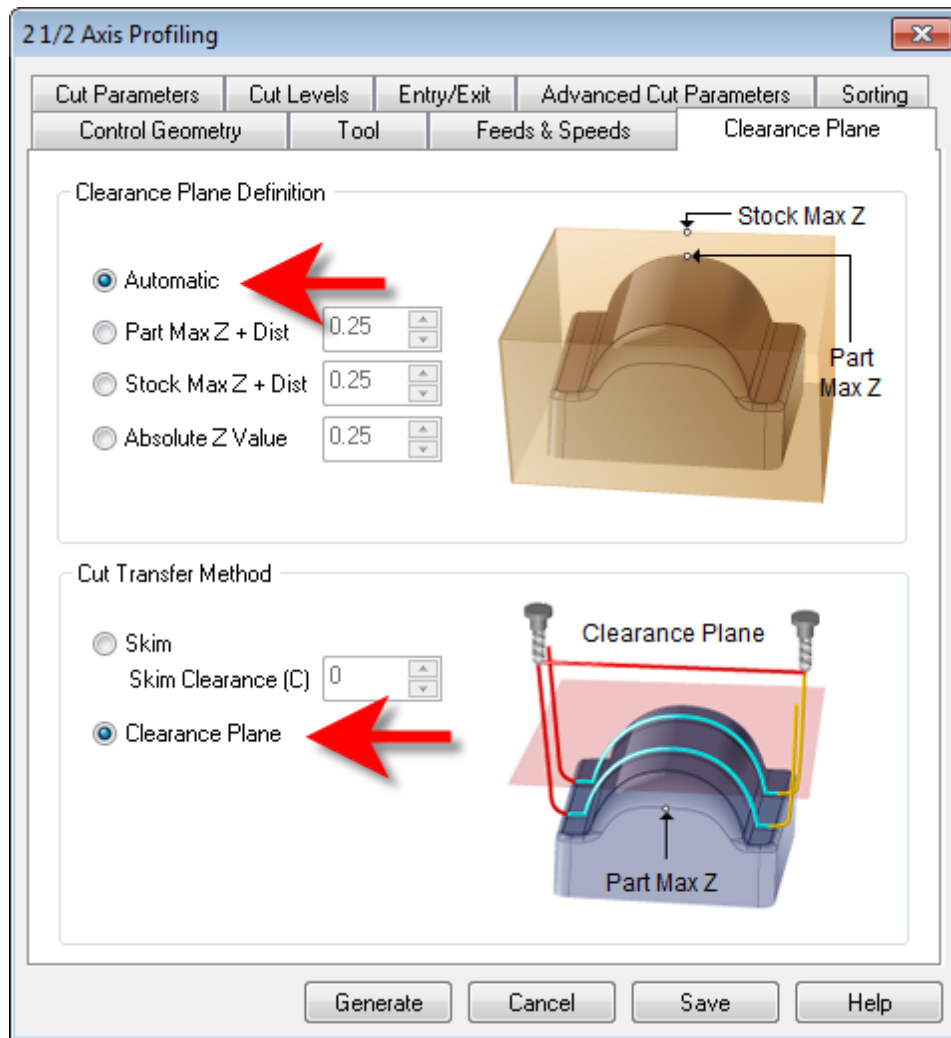
1. Switch to the **Feeds & Speeds** tab of the **2 1/2 Axis Profiling** dialog.
2. Select the **Load from Tool** button. **VisualCAM 2016** will retrieve the feeds and speeds parameters that were set when the tool was defined and associate them with the current operation.



## 5.4 Clearance Plane

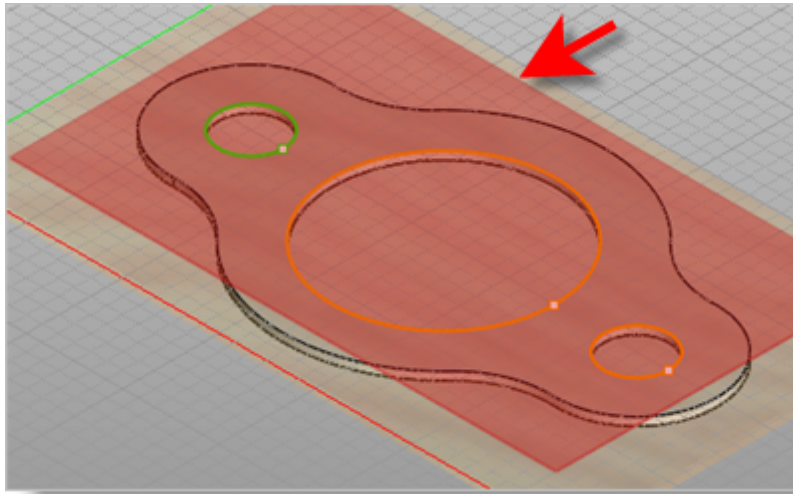
Now we'll set the [Clearance](#) geometry for our operation:

1. We'll switch to the [Clearance Plane](#) tab of the [2 1/2 Axis Profiling](#) dialog.
2. Set the [Clearance Plane Definition](#) to [Automatic](#) and [Cut Transfer Method](#) to [Clearance Plane](#).



In the **Automatic** mode, VisualCAM 2016 will determine a safe Z height for locating the clearance plane. Setting the **Cut Transfer Method** to **Clearance Plane** will force all transfer moves to be performed in this determined clearance plane.

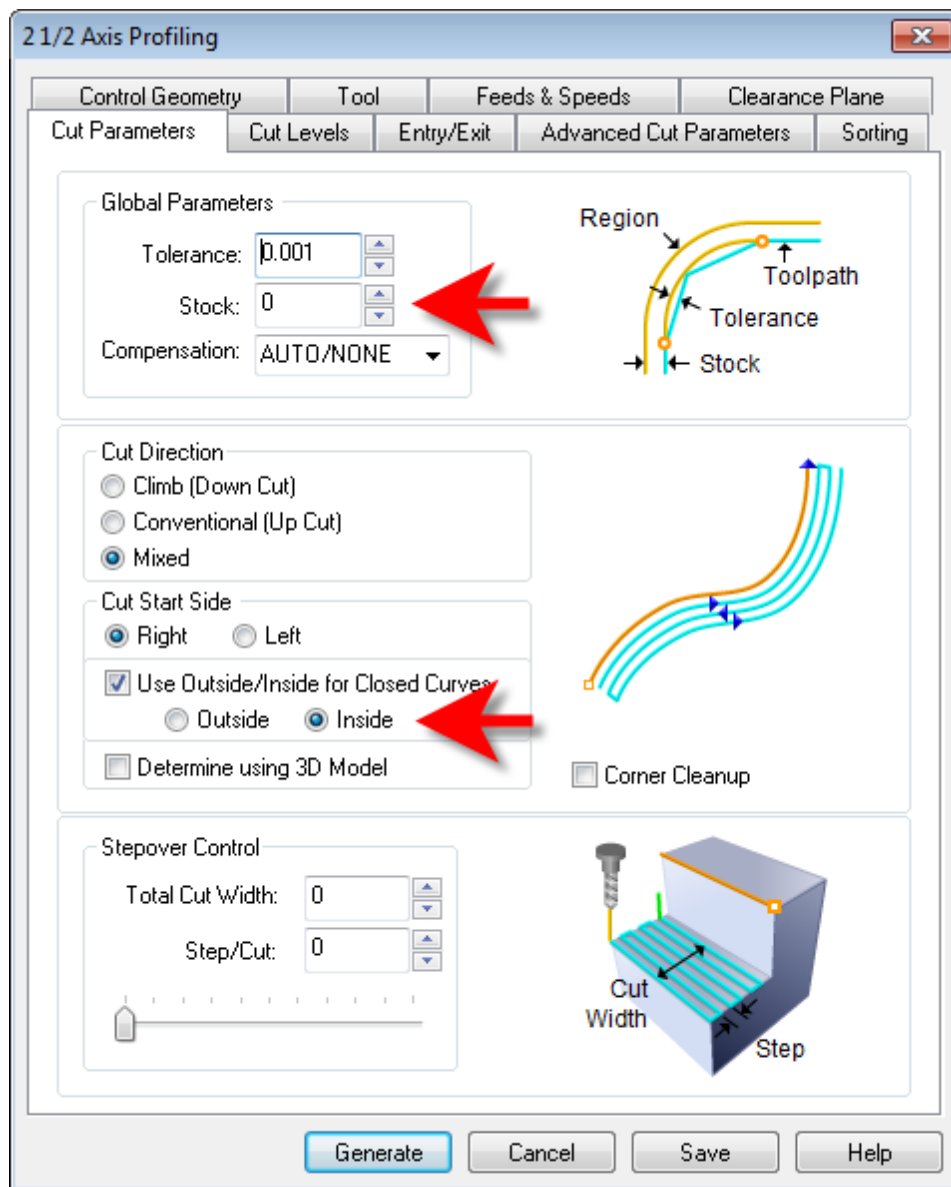
When this tab of the dialog is active, the clearance plane is shown on the graphics screen.



## 5.5 Specify Cut Parameters

Now we'll set the [Cut Parameters](#) for our operation:

1. Switch to the [Cut Parameters](#) tab of the [2½ Axis Profiling](#) dialog to control the cutting.
2. Set the [Stock](#) = 0. This means that we will not be leaving any thickness on the part after machining.
3. Under the [Cut Start Side](#) section check the box next to [Use Outside/Inside for Closed Curves](#) and then select [Inside](#).

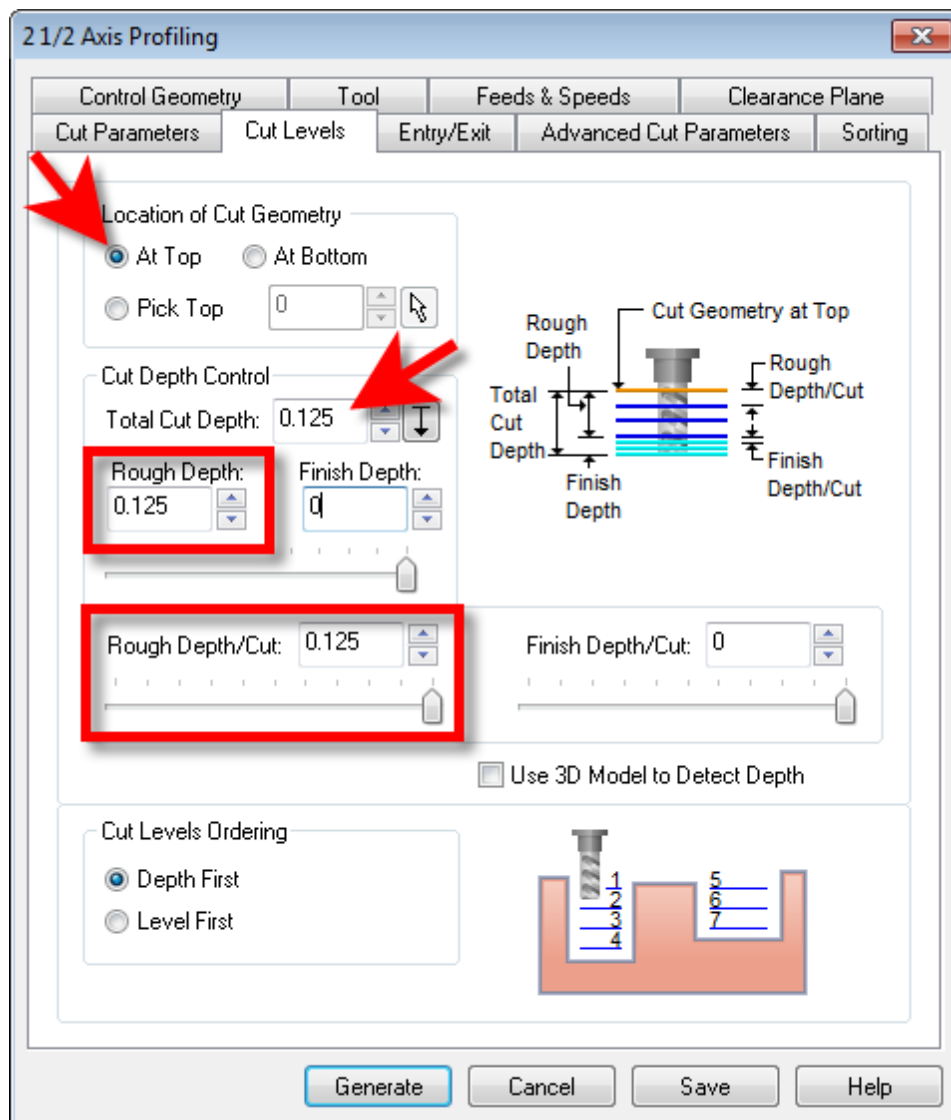


Alternately you could use the [Determine using 3D Model](#) option. In this case [VisualCAM 2016](#) would use the 3D model to determine which side of the curve to place the cutter for machining.

## 5.6 Cut Levels

Now we'll set the [Cut Level](#) for our operation:

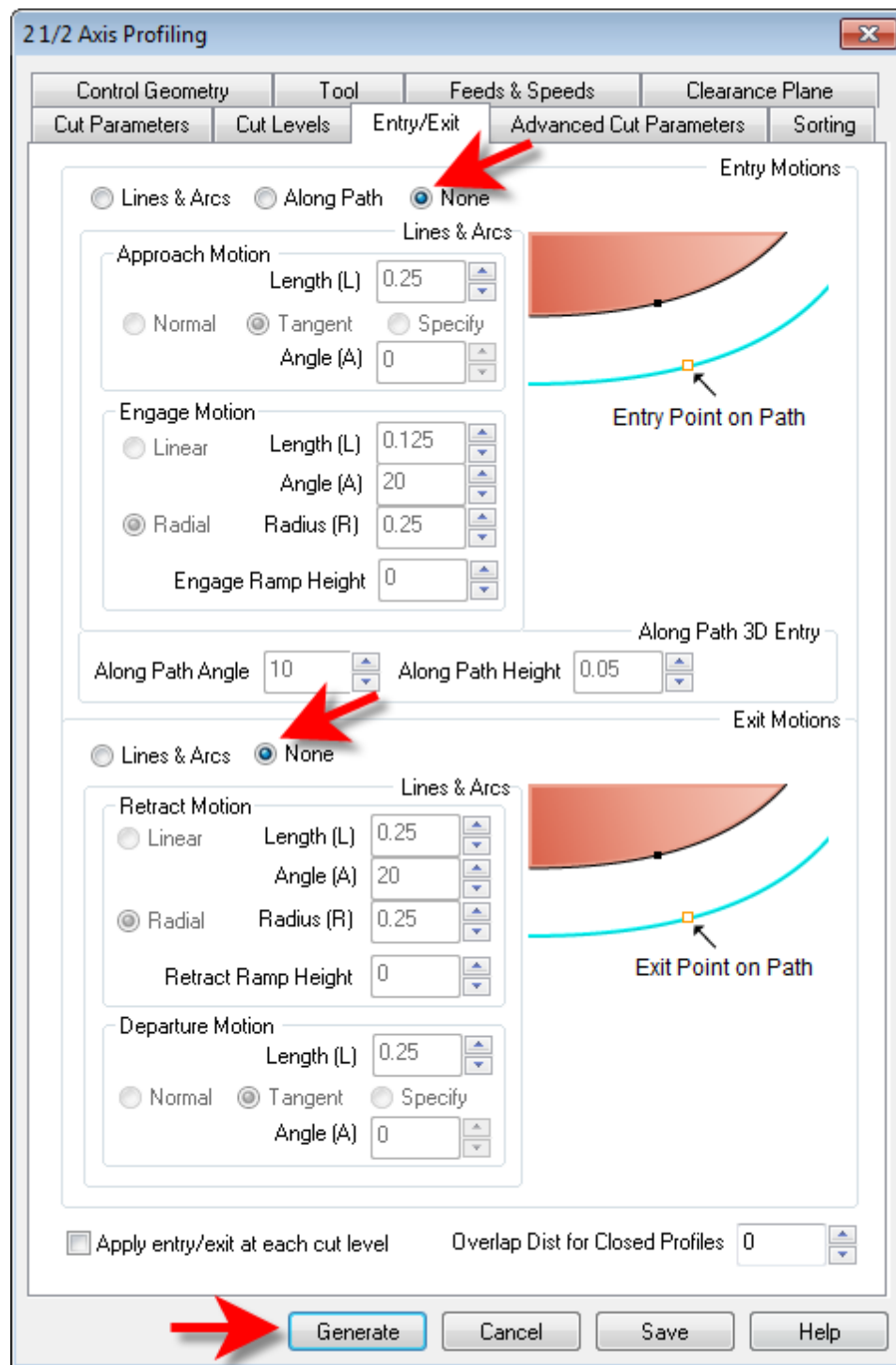
1. Select the [Cut Levels](#) tab of the [2 1/2 Axis Profiling](#) dialog.
2. Set [Location of Cut Geometry](#) to [At Top](#).
3. For [Total Cut Depth](#), enter [0.125](#). The cut depth is always set as an absolute value.
4. This automatically sets the [Rough Depth](#) and [Rough Depth/Cut](#) to [0.125](#).



## 5.7 Entry/Exit Parameters

Next we'll set [Entry](#) and [Exit](#) parameters for our operation:

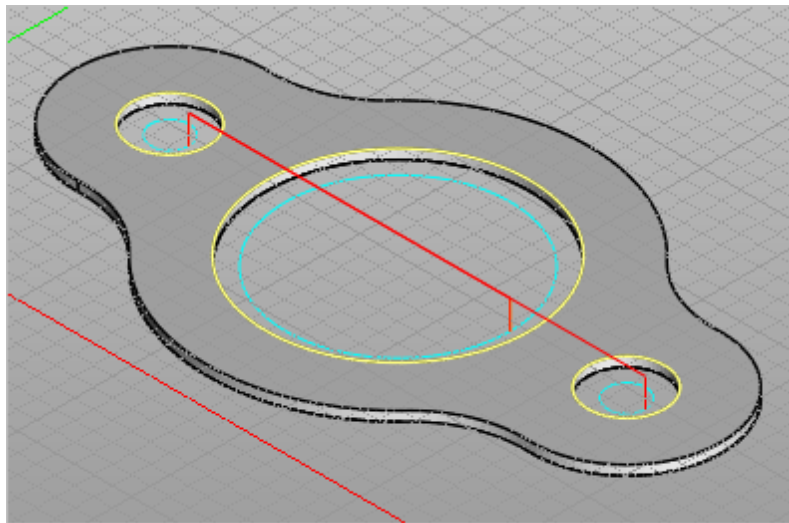
1. Next, we elect the [Entry/Exit](#) tab.
2. [Entry/Exit](#) parameters control how the cutter will engage material as it begins cutting and how it leaves the material as it completes cutting.
3. Set [Entry Motions](#) and [Exit Motions](#) to [None](#).
4. Now pick [Generate](#).



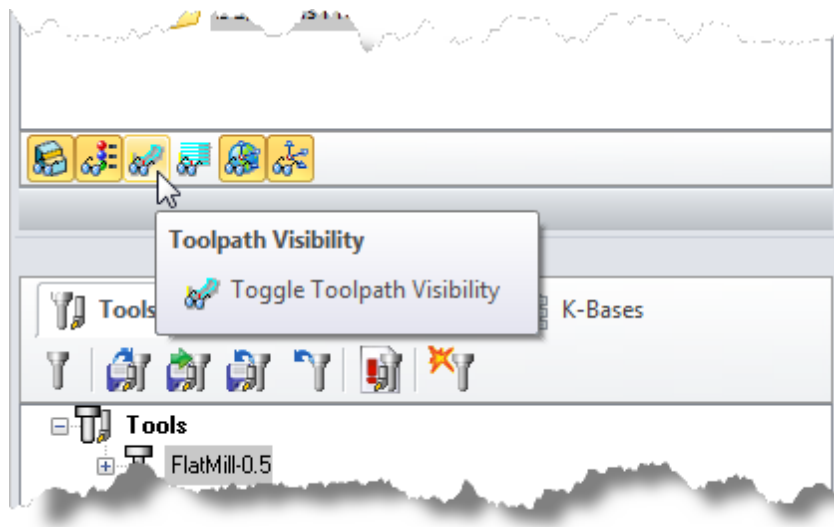
5. The **2½ Axis Profile** toolpath is generated and the operation is listed under **Setup 1** in the **Machining Browser**.



6. The toolpath is also displayed in the graphics screen.



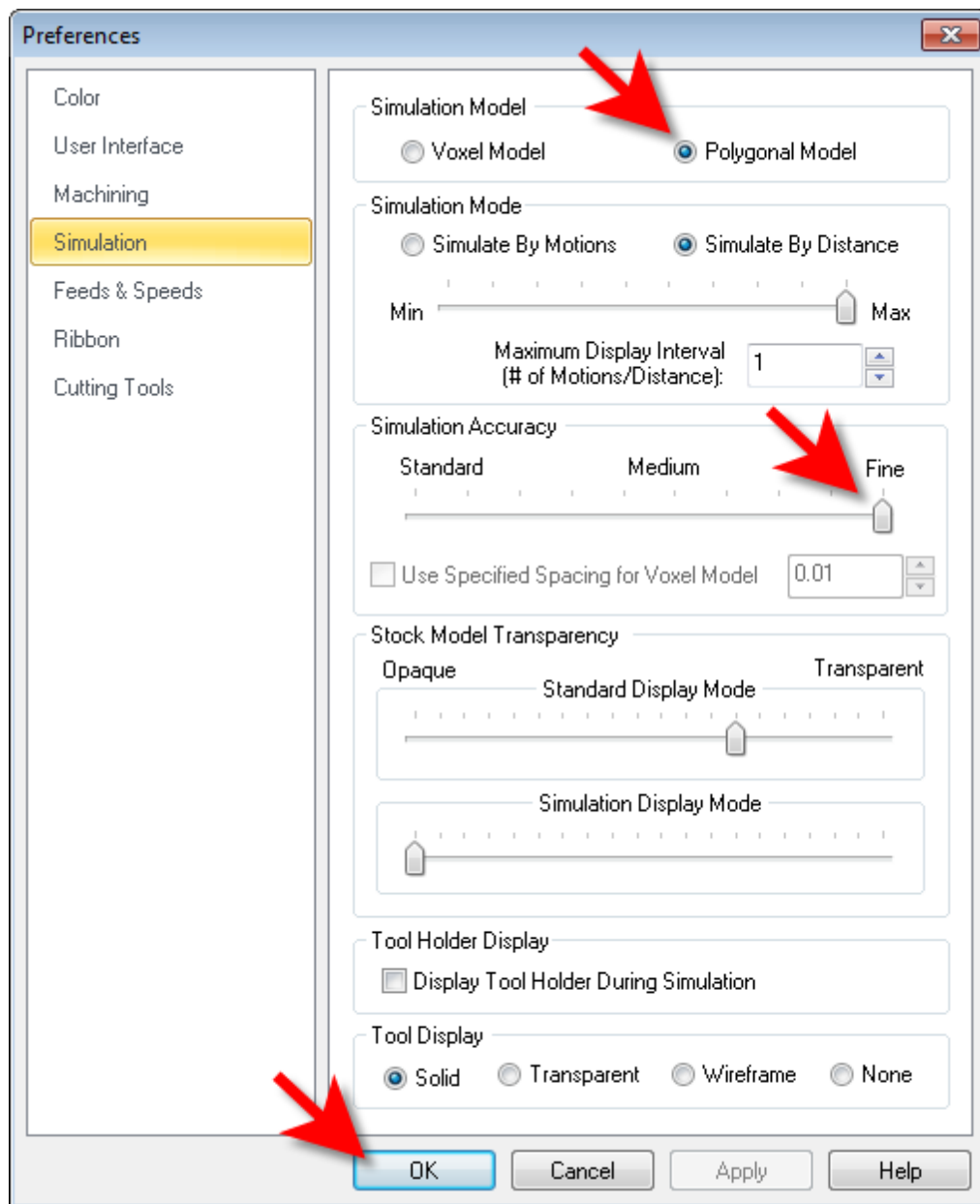
- Note that the display of the toolpath in the graphics screen can be turned on/off by selecting the **Toolpath Visibility** icon located at the base of the **Machining Browser**.



## 5.8 Simulate Toolpath

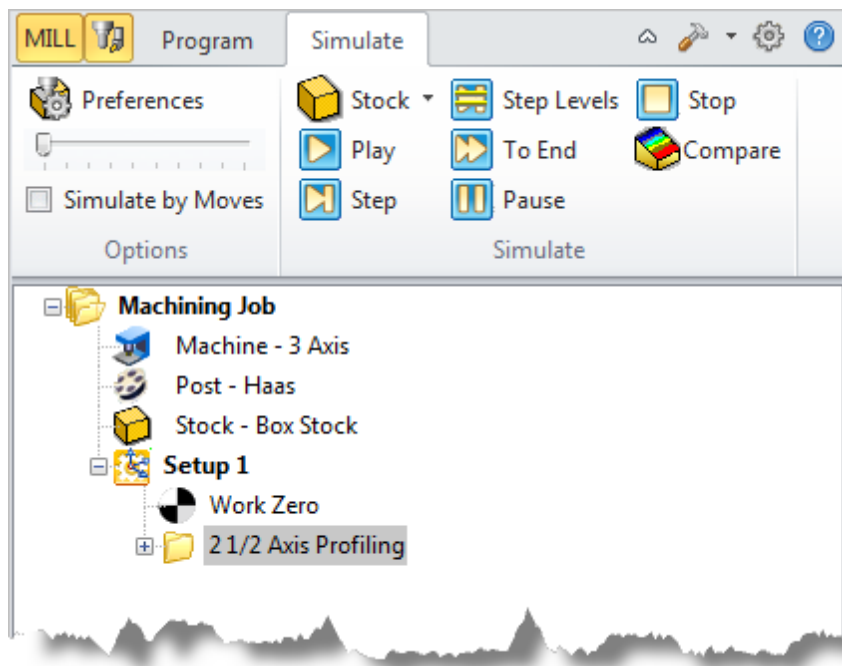
The new toolpath can now be [Simulated](#) to display the in-process stock model.

1. Switch to the [Simulate](#) tab at the top of the [Machining Browser](#).
2. Select [Preferences](#) and set the [Simulation Model](#) to [Polygonal](#) and the [Simulation Accuracy](#) to [Fine](#) and then pick [OK](#).

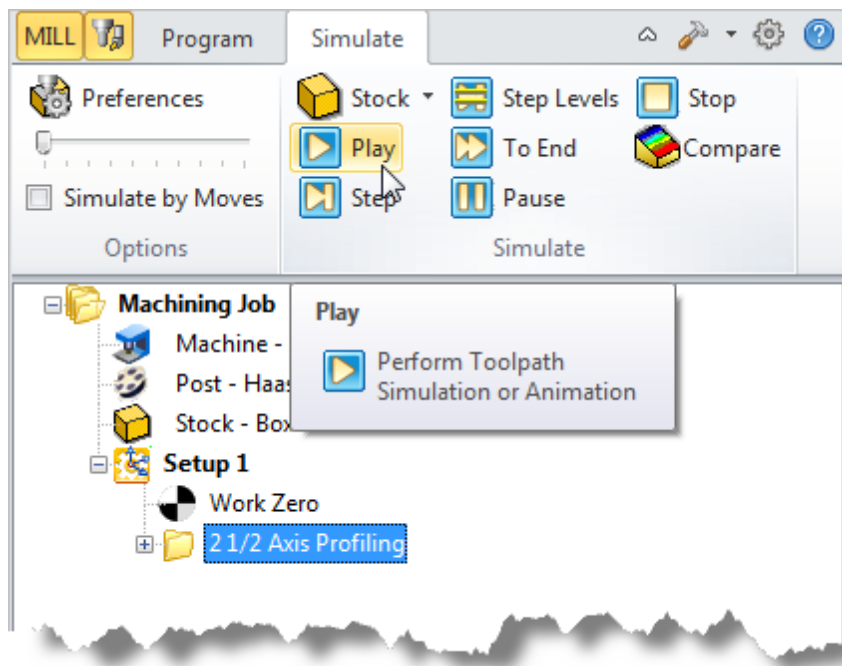


Set Simulation Preferences

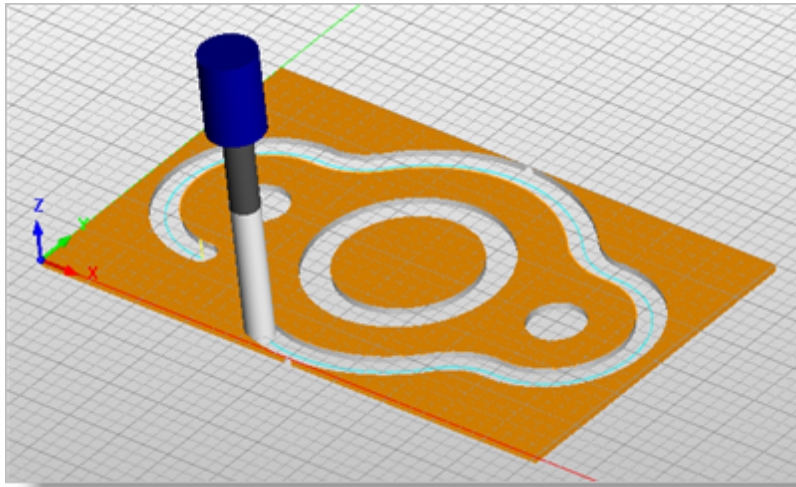
3. Then uncheck **Simulate by Moves** and adjust the slider to the left to slow down the simulation speed.



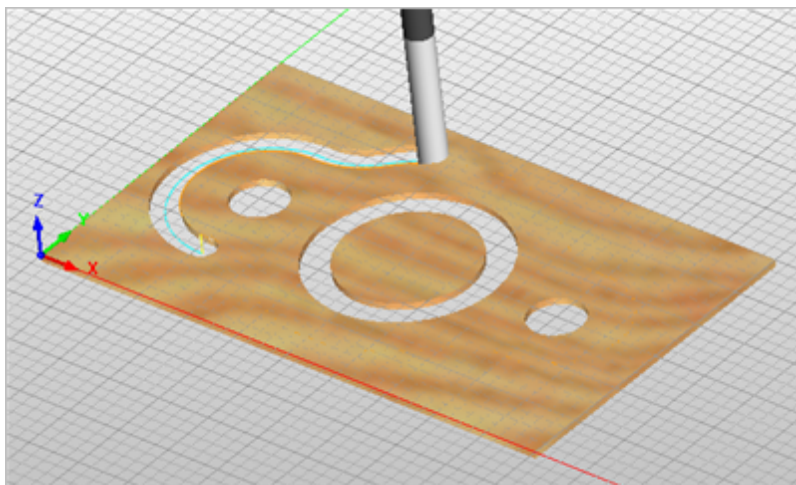
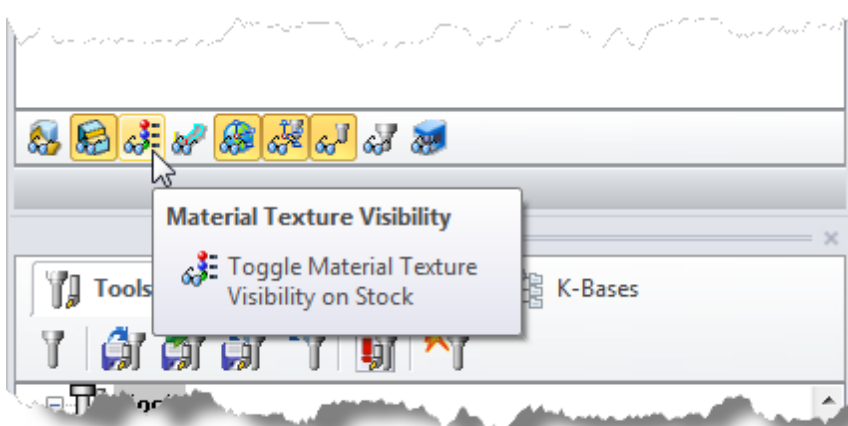
4. Select the **2½ Axis Profiling** operation we just created and then pick **Play** to start the simulation.



5. You can stop the simulation at anytime by selecting the **Pause** button in the simulation ribbon bar. Subsequent to pausing the simulation, you can either choose to continue the simulation by selecting the **Play** button again or exit the simulation by selecting the **Stop** button.



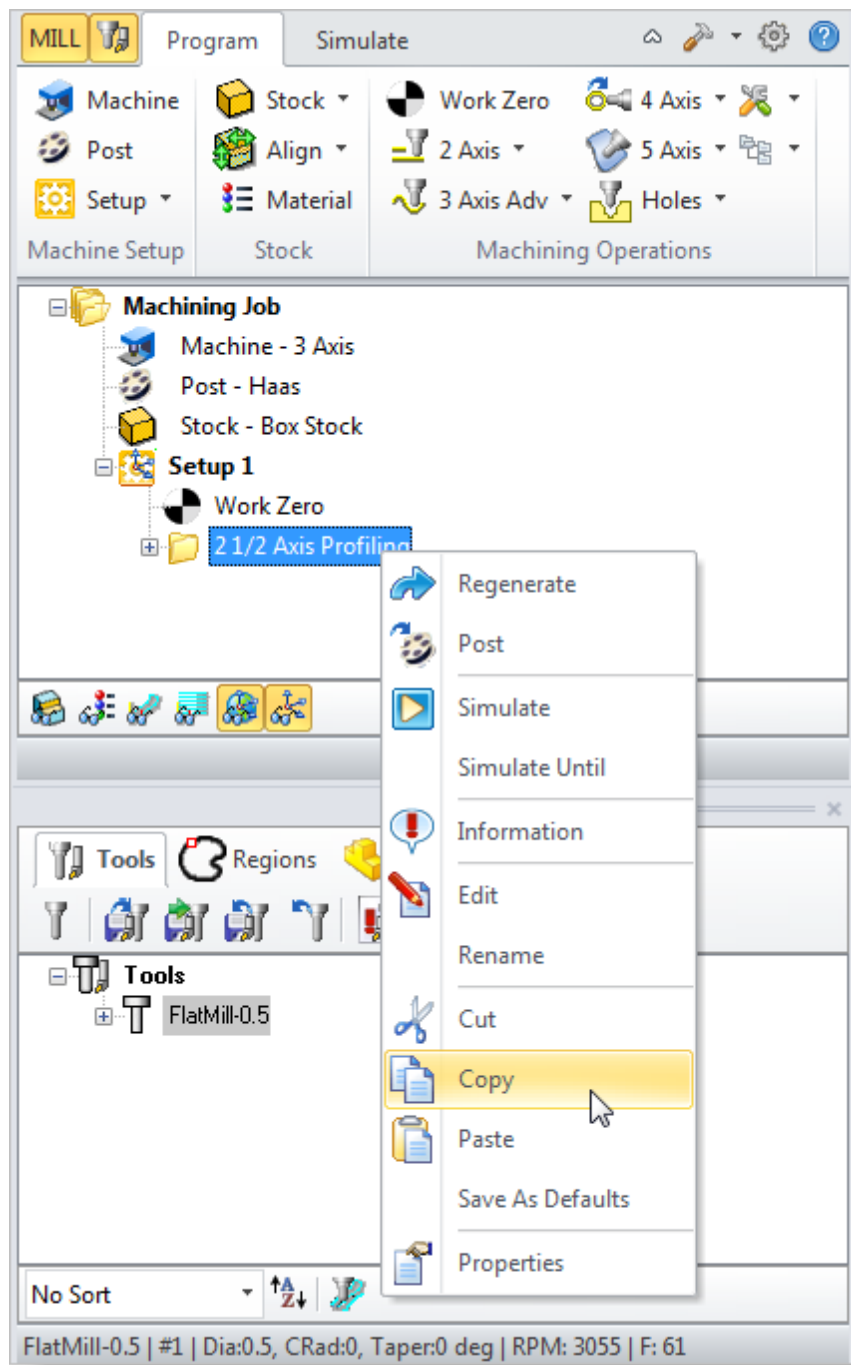
6. To view the cut model with textures applied, select the **Material Texture Visibility** icon located at the base of the **Machining Browser**.



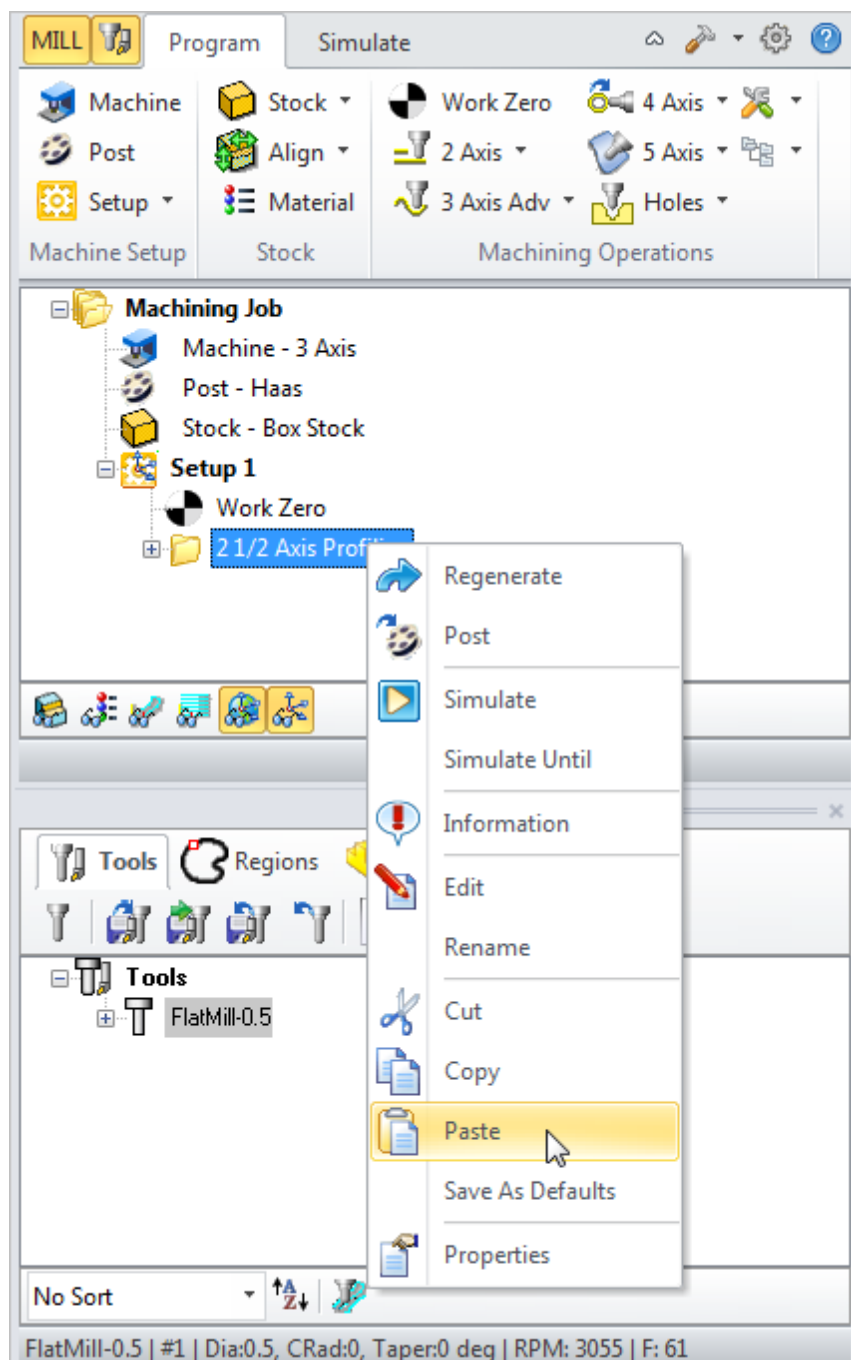
## Machining the Outer Area

Now we will turn our attention to machining the outer profile of the part. Again, we will create a simple profile toolpath, this time around the outer periphery of the part.

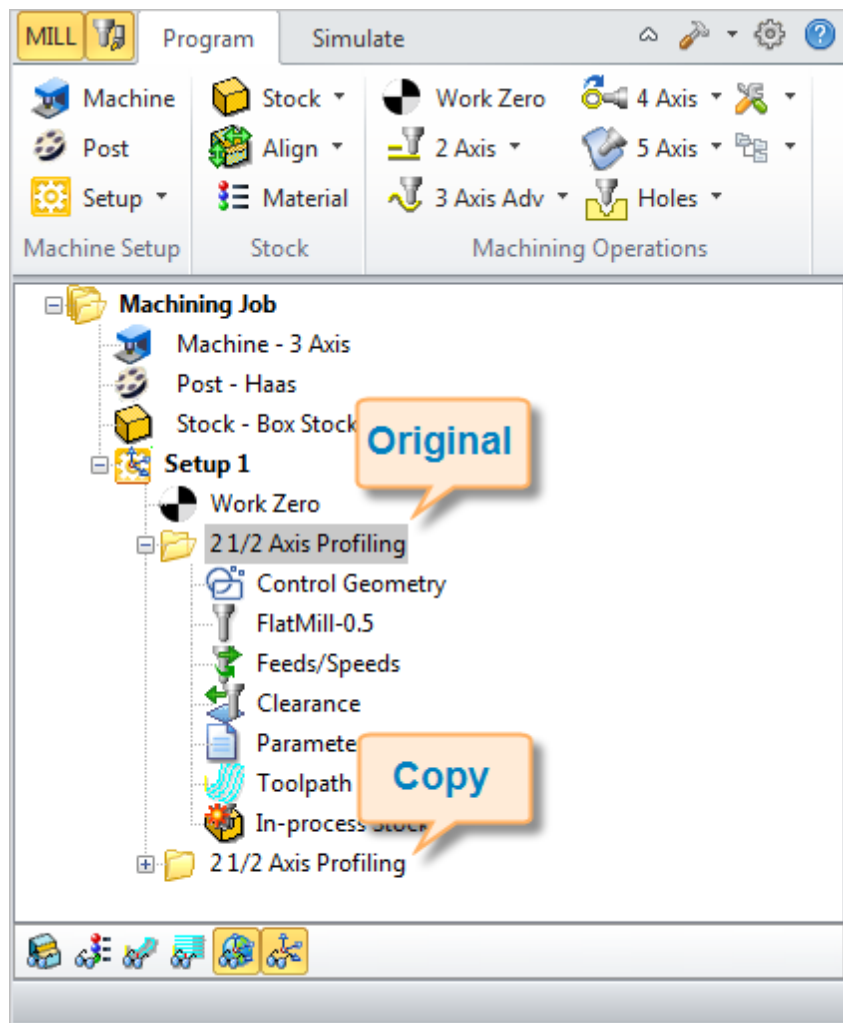
1. Switch to [Program](#) tab in the [Machining Browser](#).
2. Select the [2½ Axis Profiling](#) operation we just created.
3. [Right-click](#) on the selected operation and select [Copy](#).



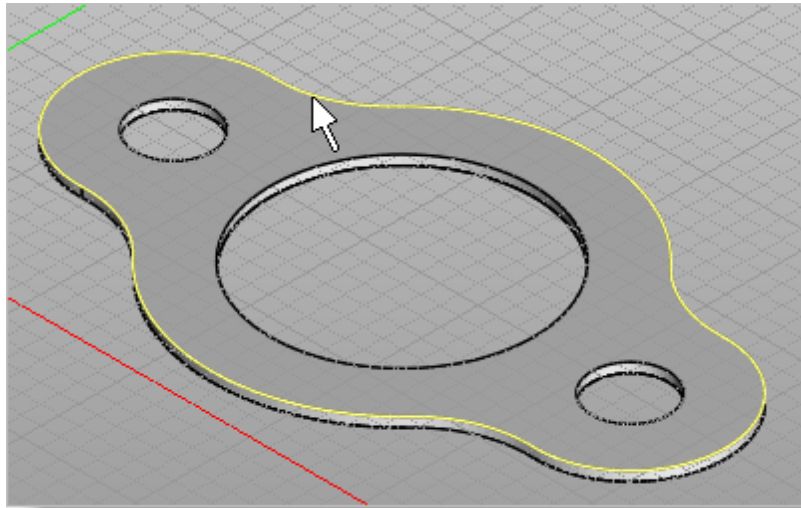
4. Now [Right-click](#) again and select [Paste](#).



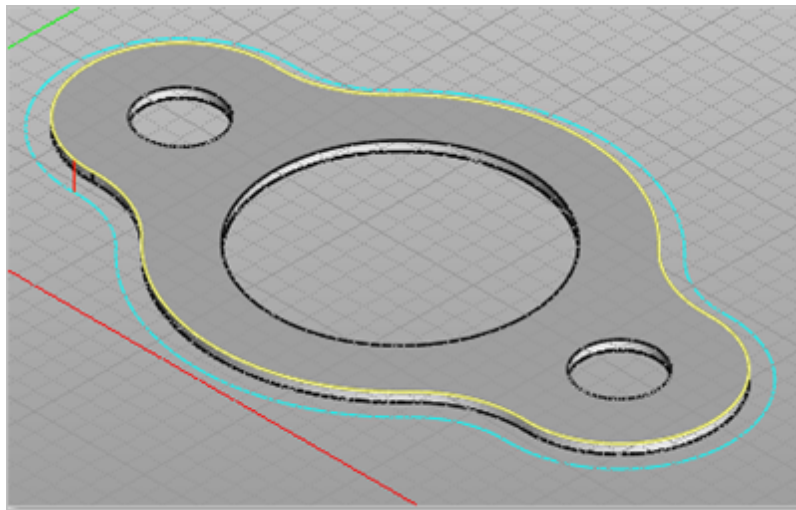
5. This creates a copy of the operation and places it below the original in the [Machining Browser](#).



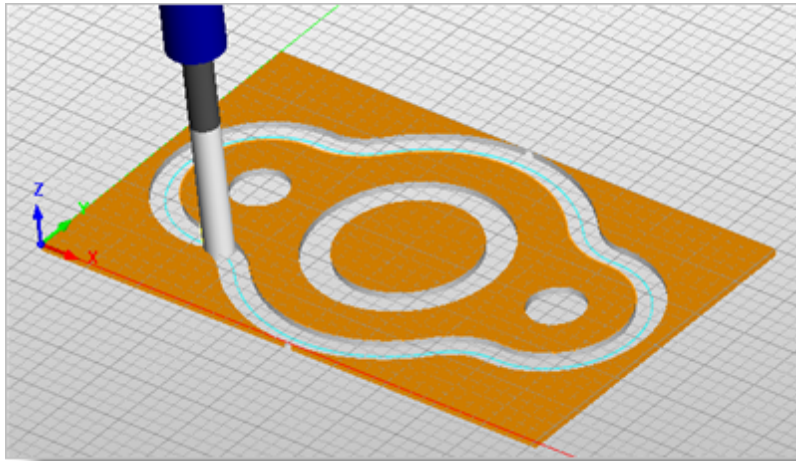
6. Now [right-click](#) on the second operation and pick [Edit](#) to adjust its parameters.
7. Pick [Remove All](#) under the [Control Geometry](#) tab.
8. Pick [Select Drive/Containment Regions](#).
9. Select the top outer surface edge and then right-click or press enter to complete the selection.



10. Switch to the **Cut Parameters** tab and change the **Cut Start Side** to **Outside**.
11. We'll accept all of the remaining parameters and pick **Generate**.
12. The new **2 ½ Profiling** toolpath is generated and displayed on the graphics screen.



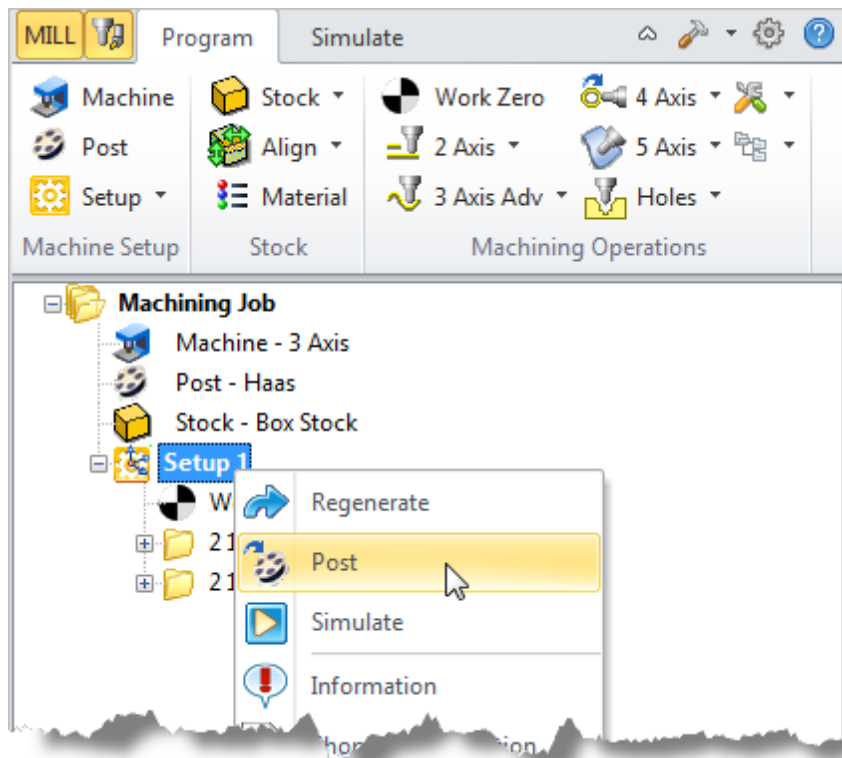
13. Now we'll select the new **2 ½ Axis Profiling** operation we just created, select the **Simulation** tab and then pick **Play**.




## Post Processing

Now with the toolpaths complete we're ready to post-process to an output text file containing G-codes that can then be sent to the machine tool to actually machine the part.

1. Select **Setup 1** from the **Machining Browser**, right-click and select **Post**. This will post-process all operations created under the **Setup**.



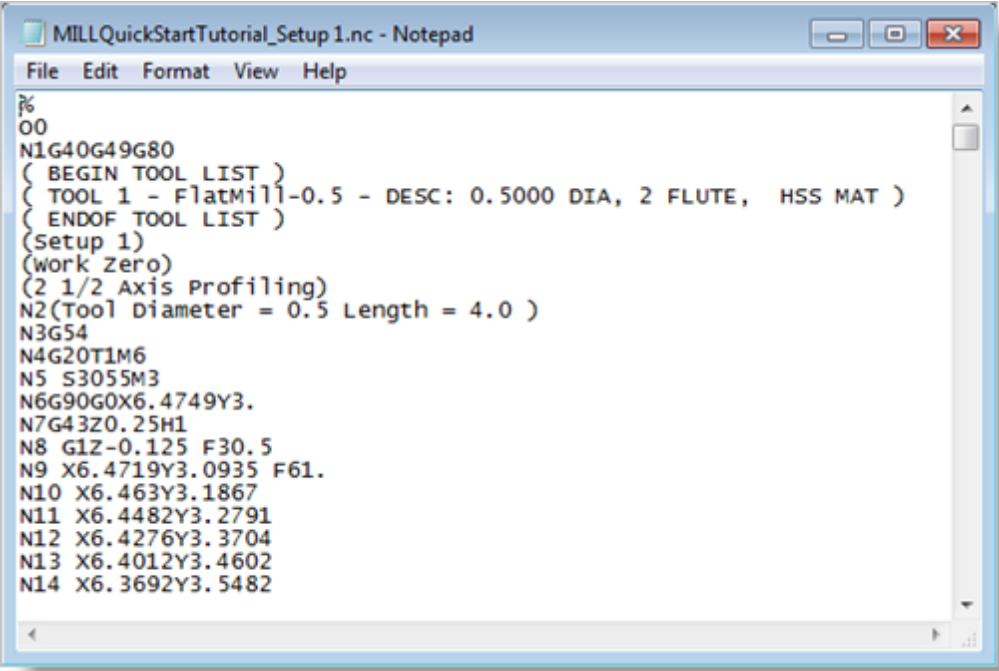
2. By default, the **Part** file name and the **Setup** name are appended for the G-code file name. Also by default, the posted G-code file is saved to the folder where the part file is located.

 The output file names can be controlled by setting the Posted **File Naming Conventions** sections of the **Set Post-Processor Options** dialog. Refer to the [Select the Post Processor](#) step for displaying this dialog.



**!** 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.

3. Now pick **Post** and the G-code file is displayed in **Notepad** where it can be viewed or edited manually.



```
%
O0
N1G40G49G80
( BEGIN TOOL LIST )
( TOOL 1 - FlatMill-0.5 - DESC: 0.5000 DIA, 2 FLUTE, HSS MAT )
( END OF TOOL LIST )
(Setup 1)
(work Zero)
(2 1/2 Axis Profiling)
N2(Tool Diameter = 0.5 Length = 4.0 )
N3G54
N4G20T1M6
N5 S3055M3
N6G90G0X6.4749Y3.
N7G43Z0.25H1
N8 G1Z-0.125 F30.5
N9 X6.4719Y3.0935 F61.
N10 X6.463Y3.1867
N11 X6.4482Y3.2791
N12 X6.4276Y3.3704
N13 X6.4012Y3.4602
N14 X6.3692Y3.5482
```

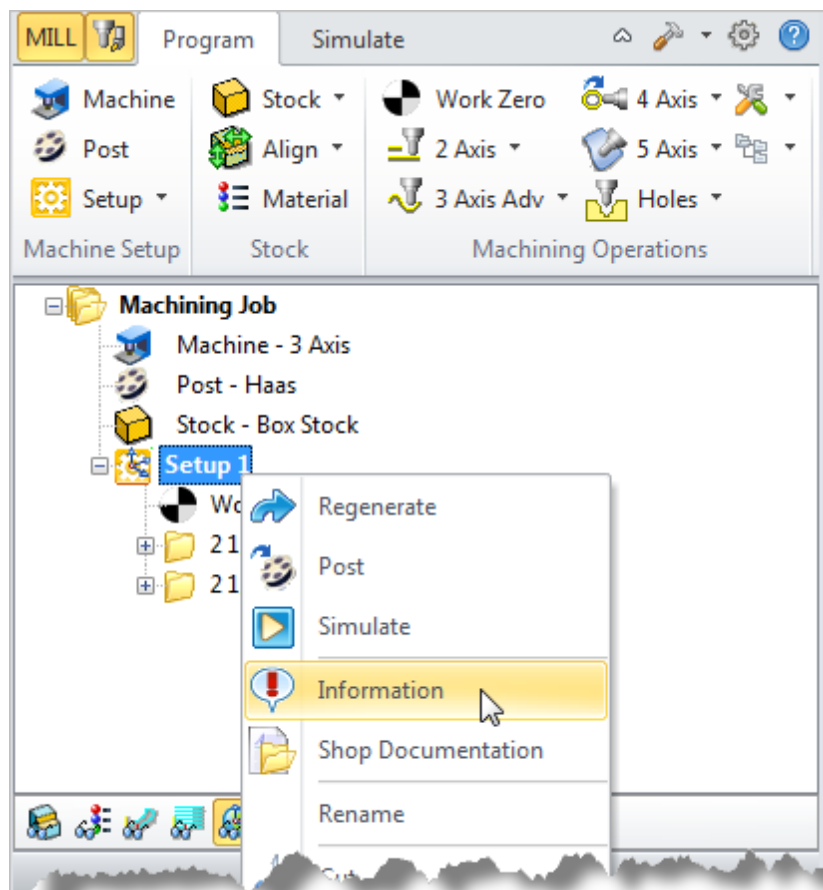
4. Now close Notepad.

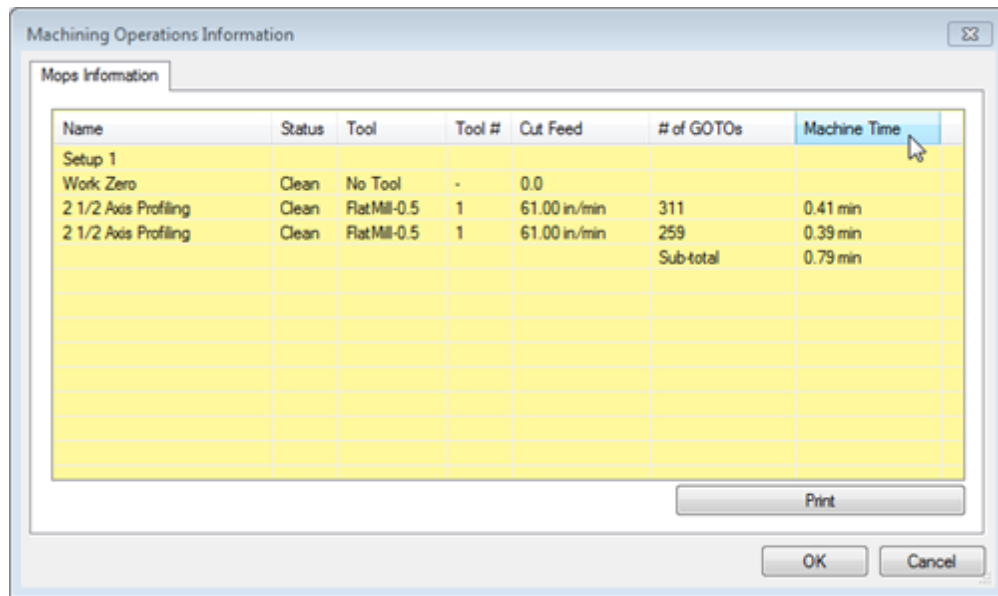
## Generating Reports

### 8.1 Information Report

At any time, you can create a [Report](#) of your [Machining Operations](#).

1. Switch to [Program](#) tab in the [Machining Browser](#).
2. Select [Setup 1](#).
3. Right-click and select [Information](#) to display and [Print](#) the report.





This dialog provides an estimate of the machining time required for the operations in the **Setup**.

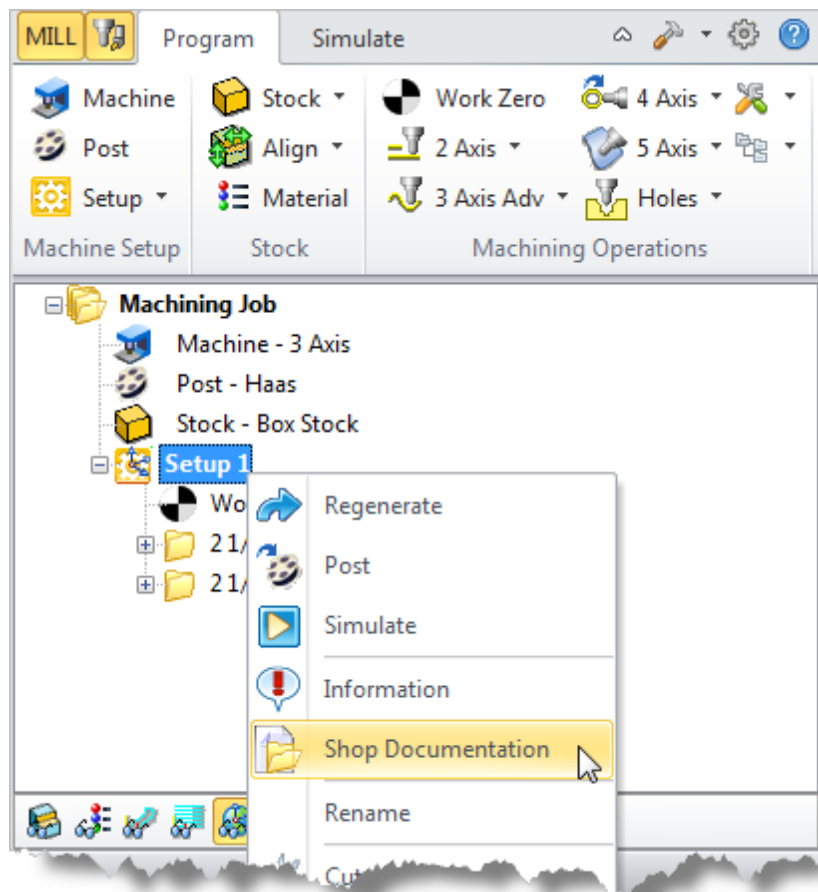
You can perform the same right-click sequence on the [Machining Job](#) to determine the estimated machining time for all [Setups](#).

4. Now pick **OK** to close the **Information** dialog.

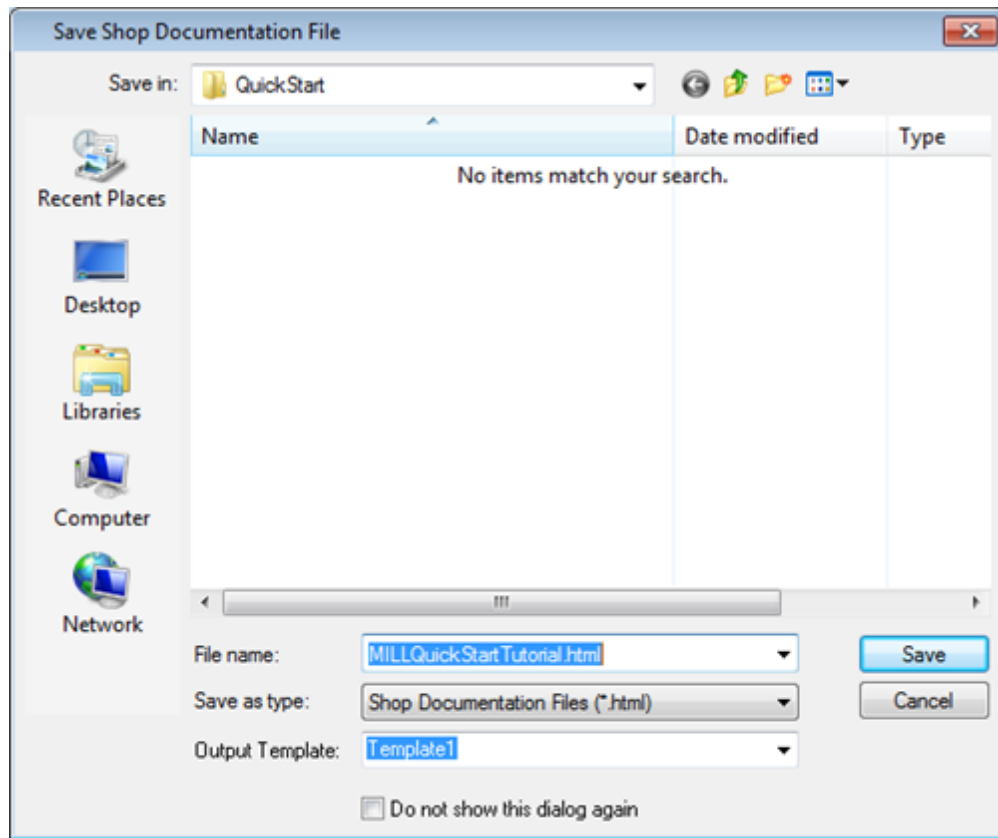
## 8.2 Shop Documentation

You can also create a [Setup Sheet](#) by generating a [Shop Document](#). This is typically used to instruct machine operators on how to setup and machine the part on the CNC machine.

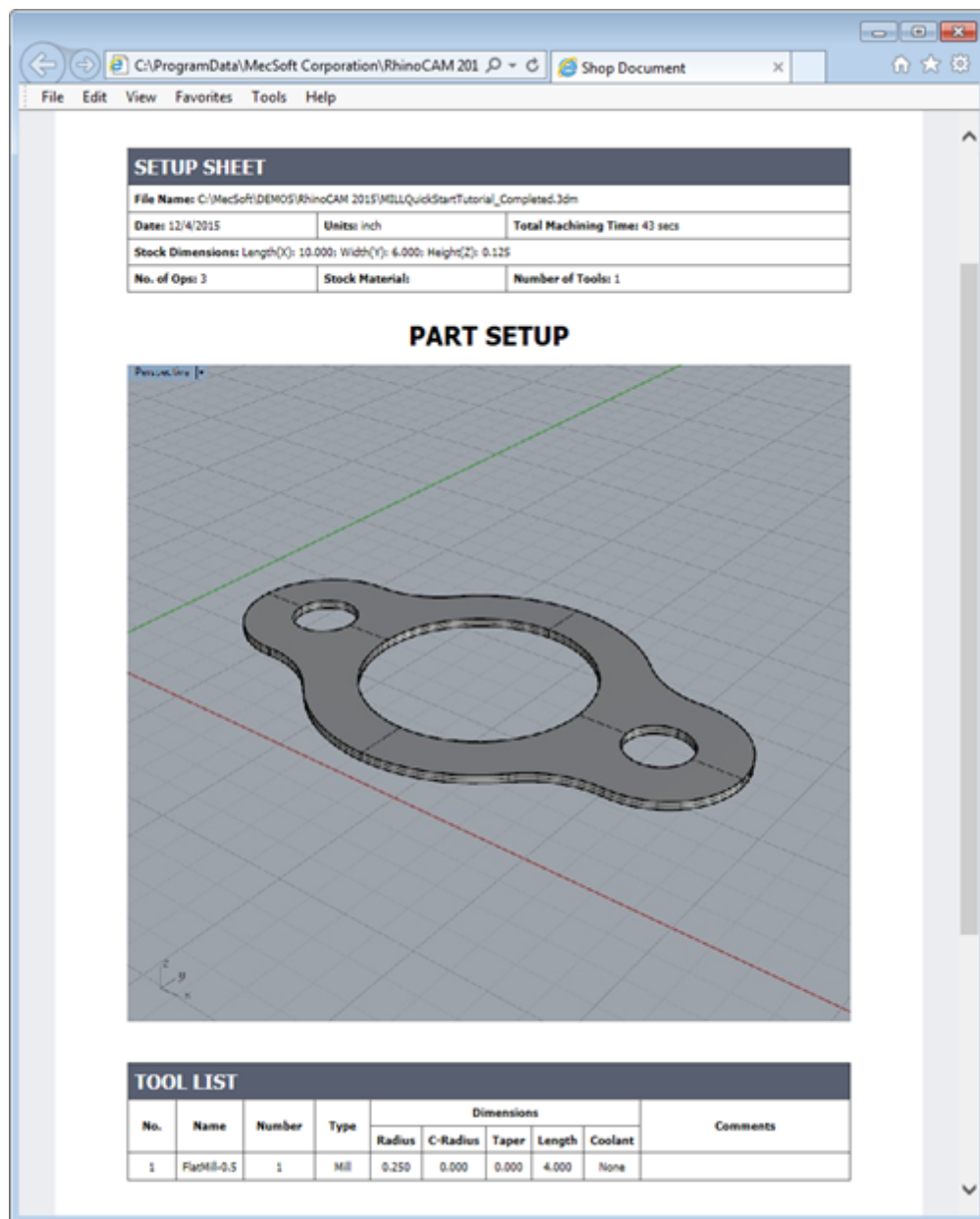
1. Select [Setup1](#).
2. Right-click and select [Shop Documentation](#).



3. Select [Template1](#) and click [Save](#).



4. This creates an [HTML](#) based [Shop Document](#) that can be viewed in a web browser. You can select from one of the 2 HTML templates that are shipped with the product and generate shop documentation. Once you have selected the template, a shop documentation html file will be created and saved. This file can then be printed and/or viewed in a web browser such as [Internet Explorer](#).



5. You can perform the same right-click sequence on the [Machining Job](#) to generate [Shop Documentation](#) for all [Setups](#).

## Where to go for more help

If you need additional help please use the following resources:

- The on-line help distributed with the product is a great resource to find reference information on the various functions available
- Apart from the on-line help system you can download other tutorials and projects from [MecSoft Corporation's](http://www.mecsoft.com) web site at [www.mecsoft.com](http://www.mecsoft.com). This will help you get started with using [VisualCAM 2016](#) .
- If you need additional help, or if you have any questions regarding [VisualCAM 2016](#), you may contact us via e-mail at [support@mecsoft.com](mailto:support@mecsoft.com)
- [MecSoft](#) offers Online training as well as personalized full day training sessions. Please look up our website or email us at [sales@mecsoft.com](mailto:sales@mecsoft.com) for further details
- Please do continue to visit our home page to learn about the latest updates to [VisualCAM 2016](#) and any other help material.

# Index

## - A -

About the MILL Module 4  
Align Part and Stock 22

## - C -

Clearance Geometry 41  
Create Machining Operations 35  
Create Stock Geometry 18  
Create Tools 31  
Cut Levels 44

## - D -

Define the Machine Tool to use 13  
Define the Machining Setup - Skip this section if in  
Standard or Expert Configuration 18

## - G -

Generating Reports 61

## - L -

Launching the VisualCAM 2016 MILL Module 7  
Loading the Part Model 11

## - M -

Machining Strategy 13  
Machining the Outer Area 52  
Main Programming Steps 13

## - P -

Post Processing 58

## - R -

Running VisualCAM 2016 6

## - S -

Select Cutting Tool 39  
Select Machining Features/Regions 36  
Select the Post Processor to use 15  
Set Feeds and Speeds 40  
Set Work Zero 26  
Simulate Toolpath 48  
Specify Cut Parameters 43  
Specify Material 24

## - U -

Useful Tips 5  
Using this Guide and Associated Part Files 4

## - W -

Where to go for more help 66