

VisualCAM© 2015 for Geomagic MILL Quick Start

© 2015 MecSoft Corporation

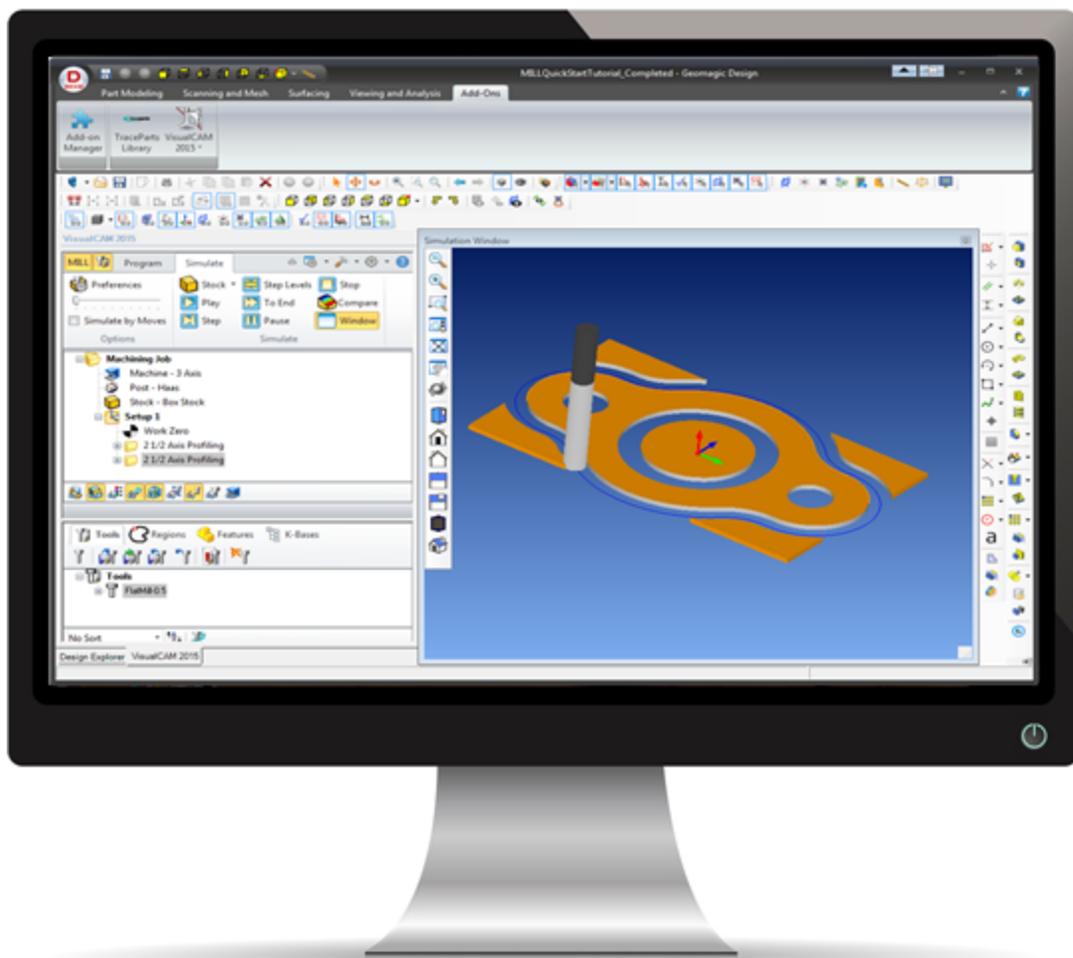


Table of Contents

About this Guide	4
1 About the MILL Module.....	4
2 Using this Guide.....	4
3 Useful Tips.....	5
Getting Ready	6
1 The VisualCAM 2015 for Geomagic Security Key.....	6
2 Running VisualCAM 2015 for Geomagic.....	6
3 Machining Strategy.....	7
4 Main Programming Steps.....	7
5 Loading the Part Model.....	7
6 Launching the MILL Module.....	10
7 Define the Machine Tool.....	12
8 Select the Post Processor.....	14
The Setup	17
1 Define the Machining Setup - Skip this section if in Standard or Expert Configuration.....	17
2 Create Stock Geometry.....	17
3 Align Part and Stock.....	20
4 Specify Material.....	22
5 Set Work Zero.....	25
Create Tools	28
Create Machining Operations	32
1 Select Machining Features/Regions.....	33
2 Select Cutting Tool.....	36
3 Set Feeds and Speeds.....	37
4 Clearance Plane.....	38
5 Specify Cut Parameters.....	40
6 Cut Levels.....	41
7 Entry/Exit Parameters.....	42
8 Simulate Toolpath.....	45
Machining the Outer Area	48

Post Processing	53
Generating Reports	56
1 Generating Shop Documentation.....	57
Where to go for more help	61
Index	62

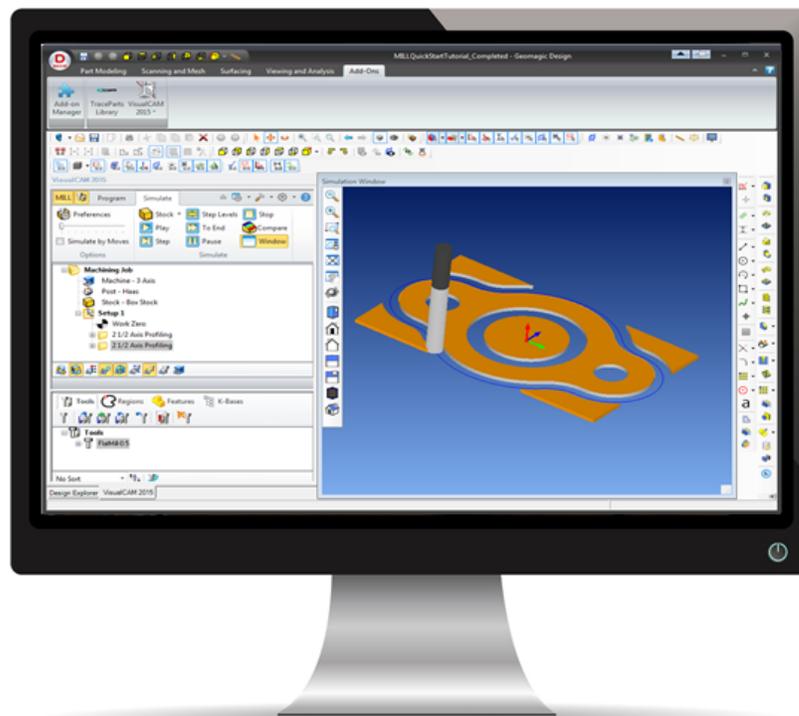
About this Guide

1.1 About the MILL Module

The [VisualCAM 2015 for Geomagic MILL](#) module offers fast gouge free solids/surface model machining technology coupled with cutting simulation/verification capabilities for programming CNC Mills, running inside [Geomagic](#). This integration allows for seamless generation of toolpath and cut material simulation/verification inside [Geomagic](#), for programming milling machines that support 3 and 4 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 [Geomagic](#) data as well as use any of the data types that can be imported into [Geomagic](#) such as solids, surfaces and meshes. Then you can use the [VisualCAM 2015 for Geomagic 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 for Geomagic MILL Module Quick Started Guide

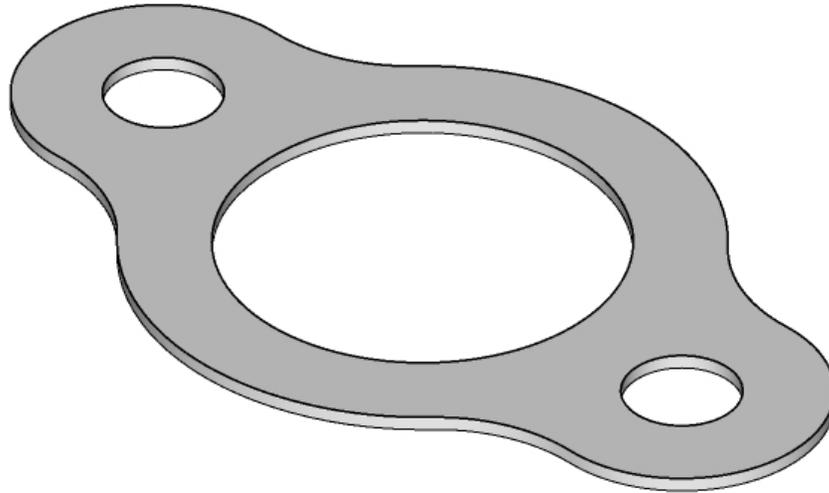
1.2 Using this Guide

If you have installed [VisualCAM 2015 for Geomagic](#) successfully on your computer and are now looking at the blank screen of [Geomagic](#) and wondering what to do next, this is the guide for you. This guide will explain how to get started in using the [VisualCAM 2015 for Geomagic](#)

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 [Geomagic](#) 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.AD_PRT

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 The VisualCAM 2015 for Geomagic Security Key

In order for [VisualCAM 2015 for Geomagic](#) to function fully it requires a security device, called a hardware key or a dongle.



Without the security device, [VisualCAM 2015 for Geomagic](#) will operate only in [Demo mode](#). When [VisualCAM 2015 for Geomagic](#) is operating in demo mode you will still be able to run [VisualCAM 2015 for Geomagic](#) but will not be able to do the following:

- Save [CAM](#) data to [Geomagic](#) files
- [Post-process](#) created tool paths

In order for [VisualCAM 2015 for Geomagic](#) to operate fully the security key needs to be properly installed on your computer or network. Please follow the instructions that came with your installation program on how to do this.

2.2 Running VisualCAM 2015 for Geomagic

Locate the [Geomagic Design](#) 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 [Geomagic Design](#). (The name of this program group will usually be called [Geomagic Design](#), unless you specified otherwise during setup.)

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

If the installation was successful, upon launching of [Geomagic Design](#) and opening a [Part](#) document you should observe a ribbon bar entry called [VisualCAM 2015](#) in the [Add-ons](#) tab of the [Geomagic Design](#) ribbon bar.

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.

! [VisualCAM 2015 for Geomagic](#) only operates in the [Part](#) mode of [Geomagic](#). If you are running in the [Drawing](#) or [Assembly](#) mode of [Geomagic](#) you will not

see [VisualCAM's](#) menu bar loaded in the [Geomagic](#) ribbon bar.

2.3 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.

2.4 Main Programming Steps

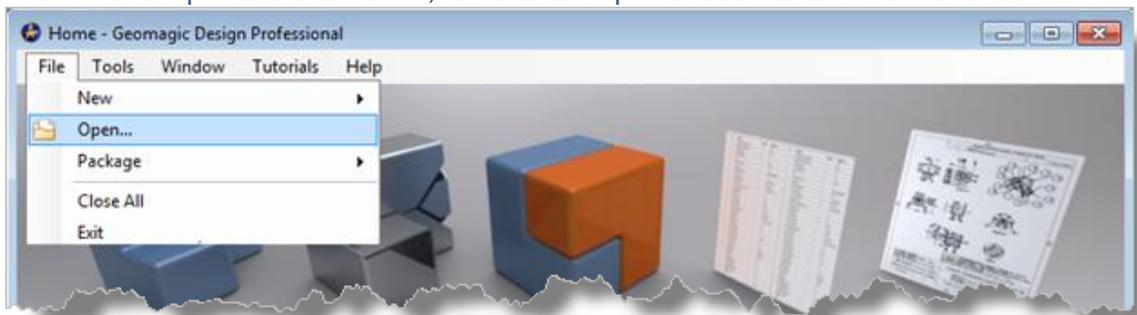
The following steps will be followed in machining this model. Some of these steps will have to be performed just once and others may have to be repeated to complete the machining.

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.

2.5 Loading the Part Model

The [Part](#) typically is the geometry that represents the final manufactured product.

1. Select [File / Open](#) from the [Menu](#), or click the [Open](#) icon from the [Standard](#) bar.

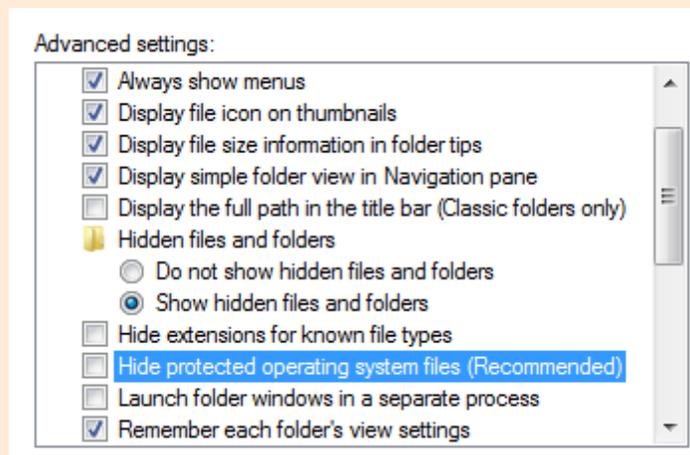


2. From the [Open](#) dialog box, select the [MILLQuickStartTutorial.ad_prt](#) file from the [C:\ProgramData\MecSoft Corporation\VisualCAM 2015 for Geomagic\QuickStart\](#) folder. As mentioned before, it is advisable to make a copy of this part at a suitable

alternative folder so that you have write privileges to modify the part.

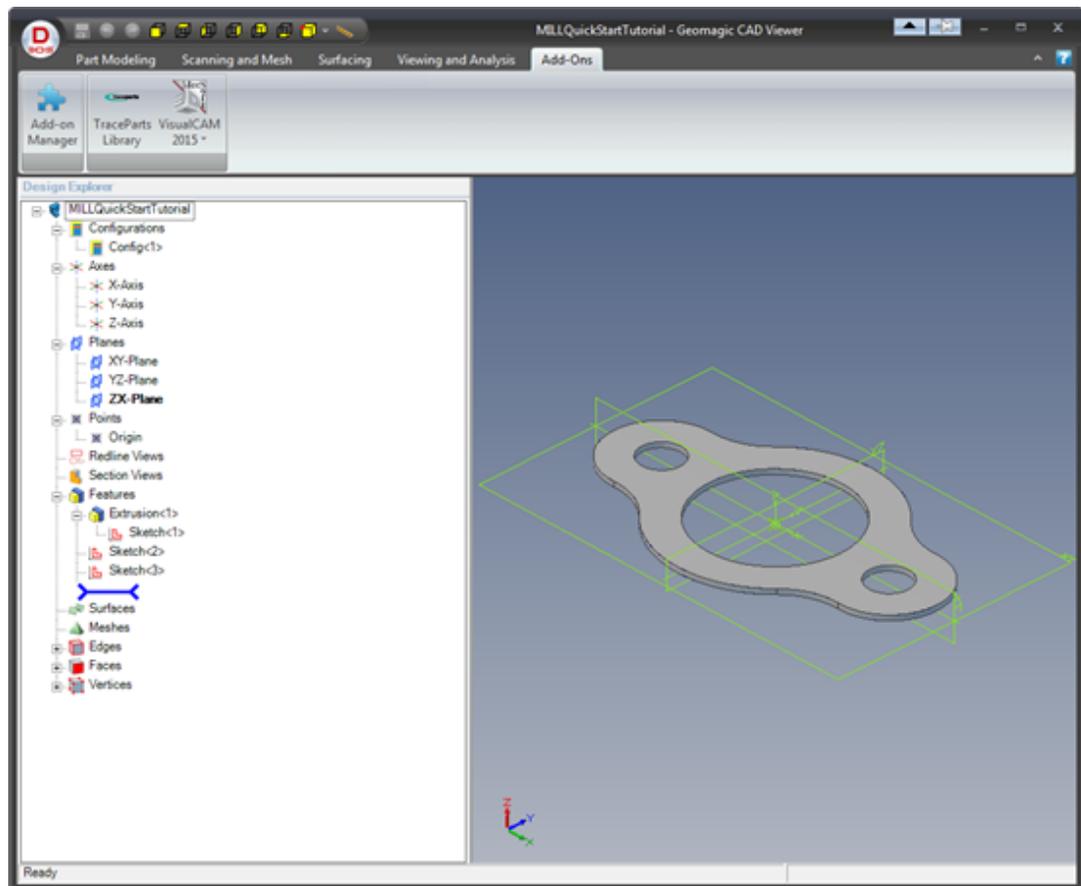
! By default, the **ProgramData** folder is "hidden" from view. Here are the steps to Show hidden files and folders:

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)**



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

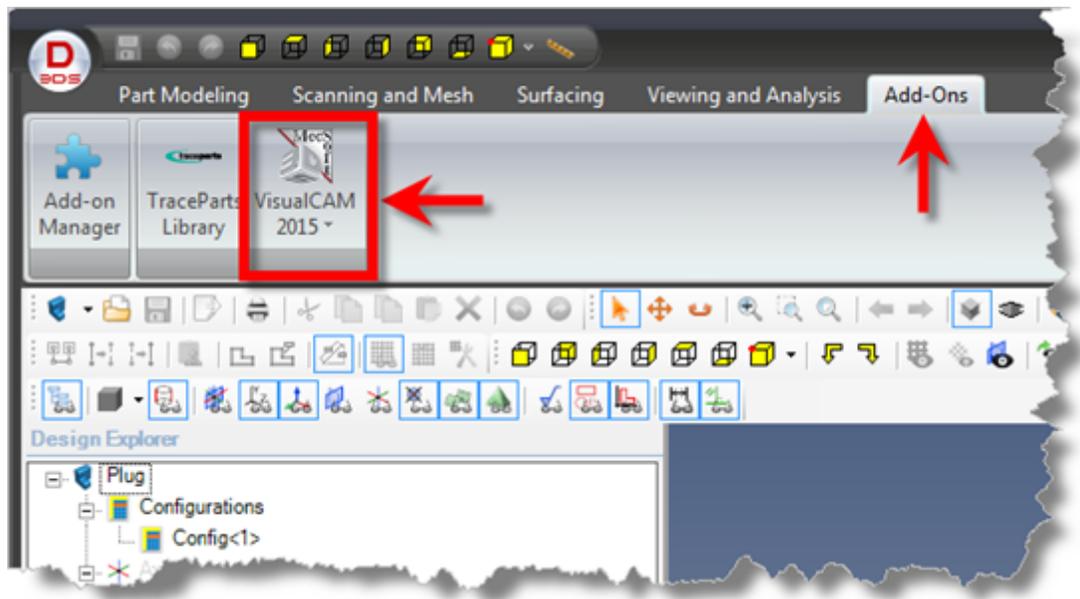
The part appears as shown below



MILLQuickStartTutorial.SLDPRT

 You can import 2D drawings, Solid, Surface and Mesh models that are supported in Geomagic.

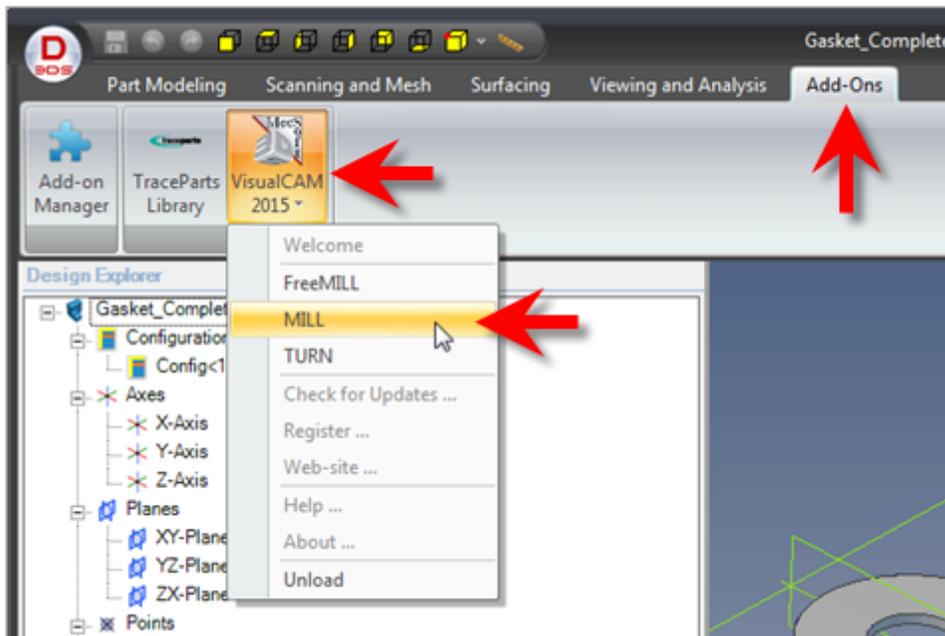
2. The VisualCAM 2015 menu item is added to the Add-Ons ribbon bar in Geomagic Design.



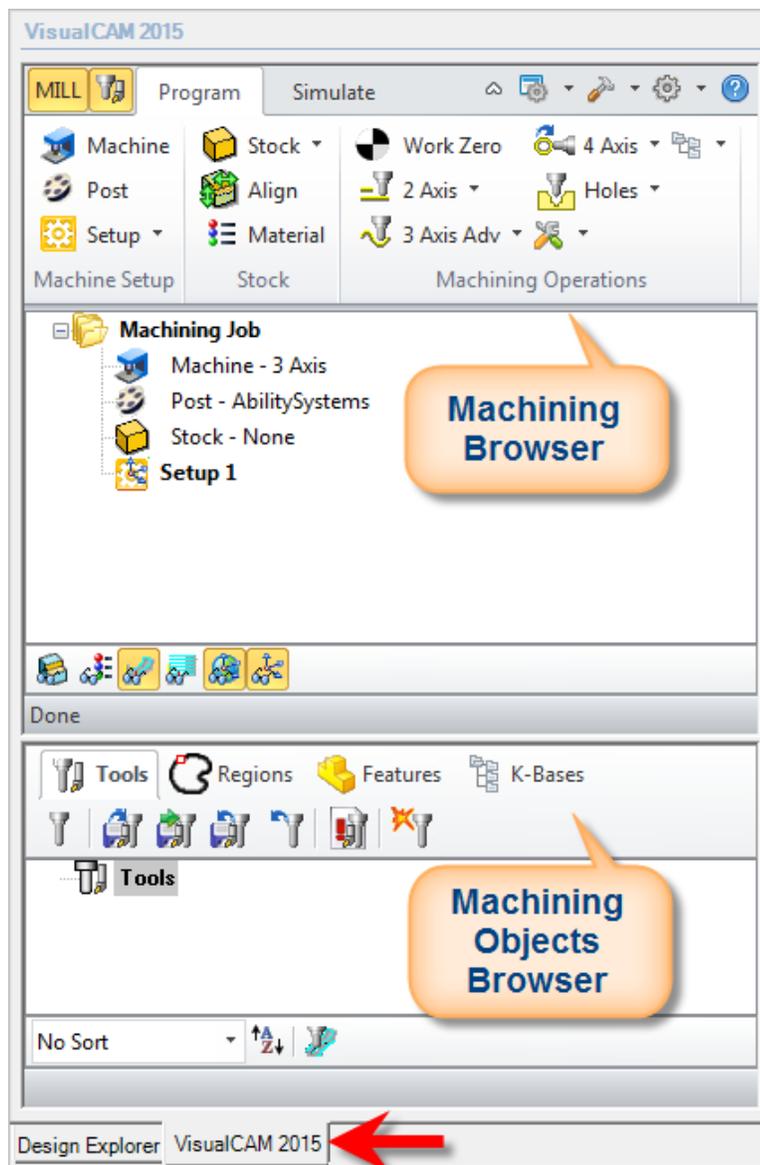
2.6 Launching the MILL Module

Follow the procedure below to launch the [VisualCAM 2015 for Geomagic TURN](#) module:

1. Select **MILL** from the **VisualCAM 2015** menu located on the **Add-Ons** ribbon bar in Geomagic Design.

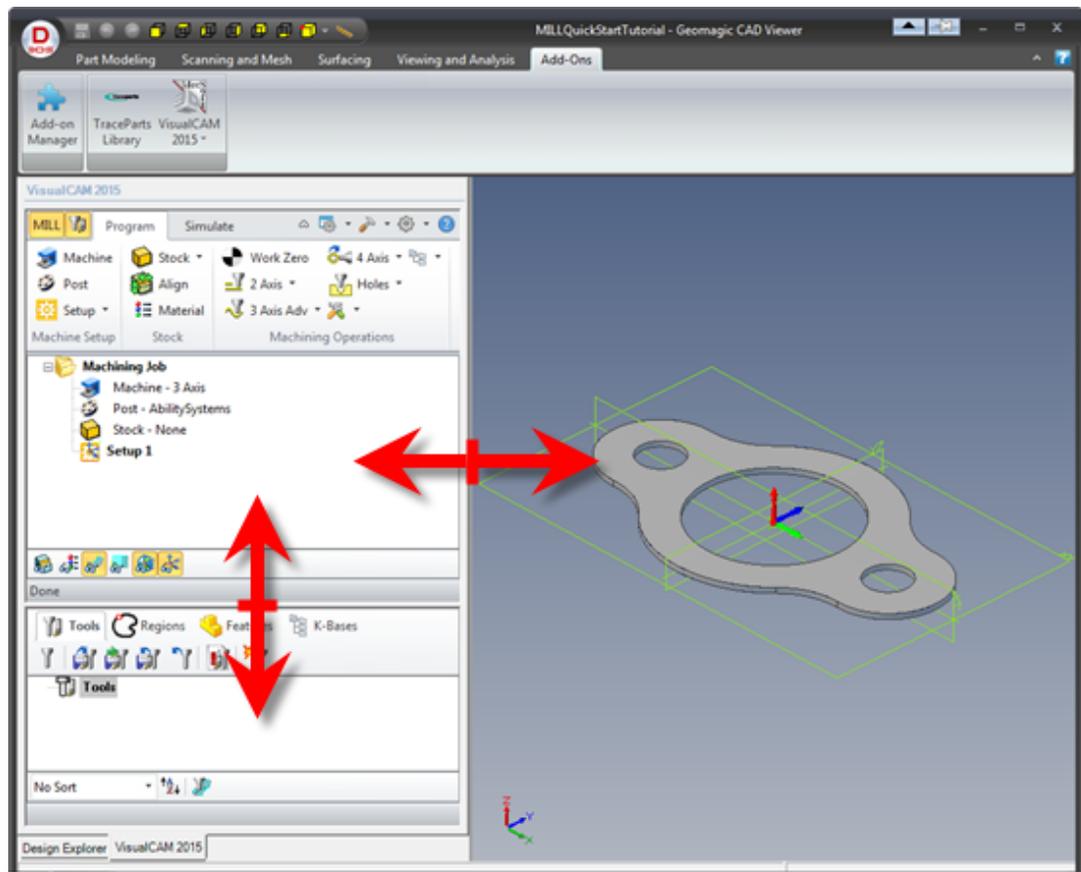


2. You will see two dialogs, one called the [Machining Browser](#) and the other called the [Machining Objects Browser](#) displayed on a separate tab labeled **VisualCAM 2015** next to the **Design Explorer** tab



The Machining Browser is also referred to as the Machining Operations Browser or the MOps Browser. The Machining Objects Browser is also referred to as the MObs Browser.

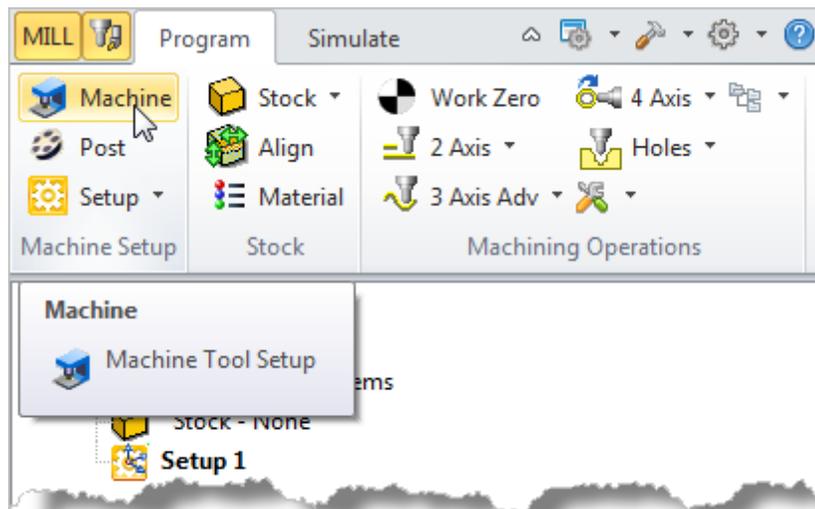
3. You can also re-size the height and width of each browser making sure that all of the command icons and menus are easily accessible.



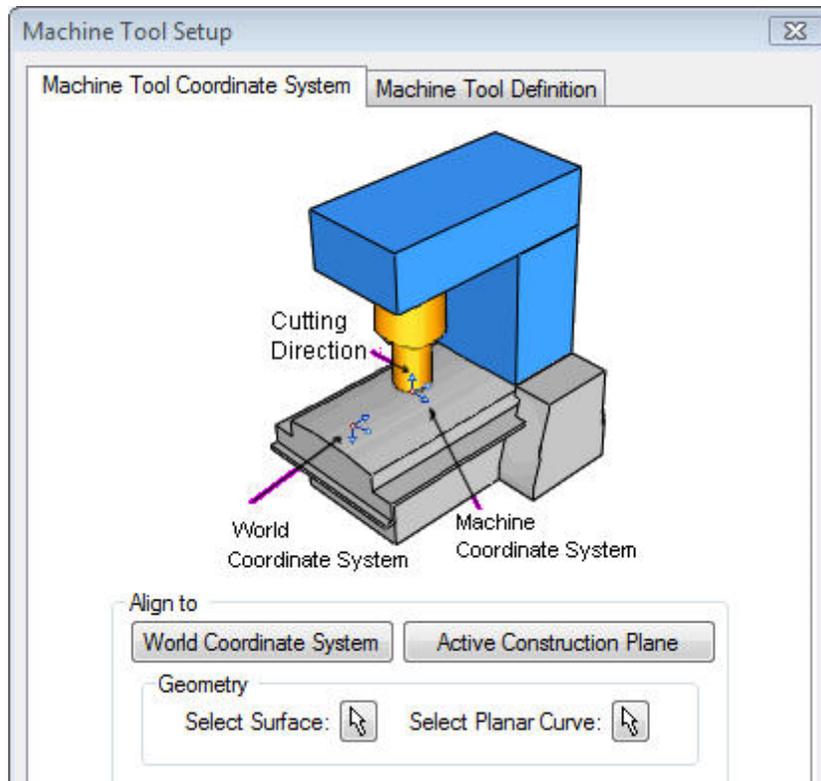
2.7 Define the Machine Tool

The first step in the machining process is to define the machine tool to use.

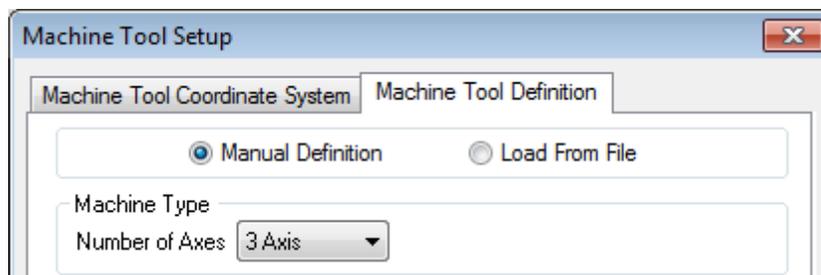
1. From the **Program** tab from the **Machining Browser** select **Machine**.



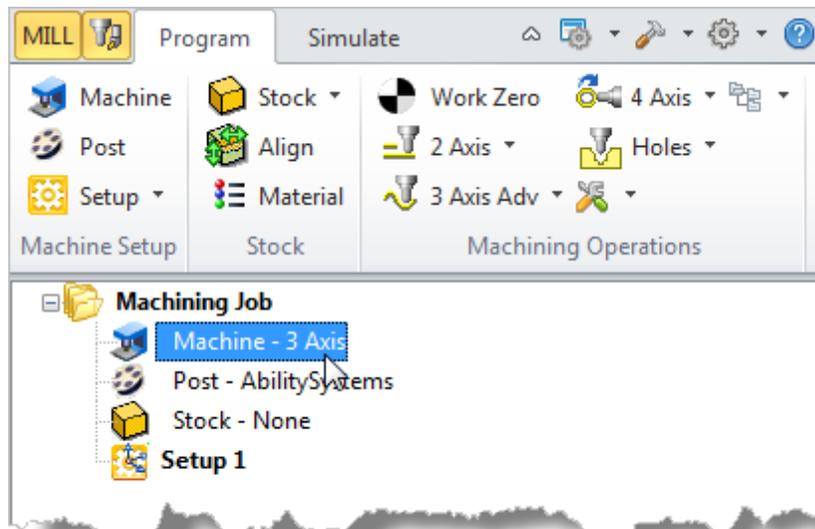
2. Under the **Machine Tool Coordinate System** tab, select the **World Coordinate System** button to align the **Machine Coordinate System (MCS)** to the **World Coordinate System (WCS)**.



3. Now select the **Machine Tool Definition** tab and under **Machine Type**, set **Number of Axes** to **3 Axis**.



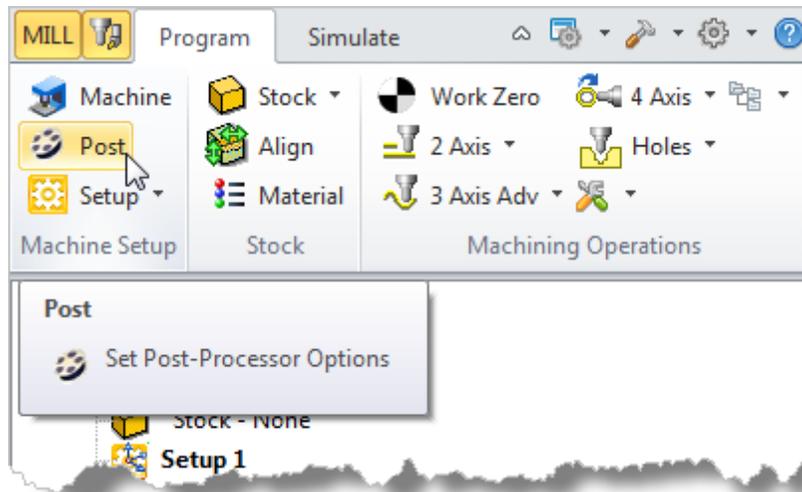
4. Click **OK**. The machine type now appears under **Machining Job** in the **Machining Browser**.



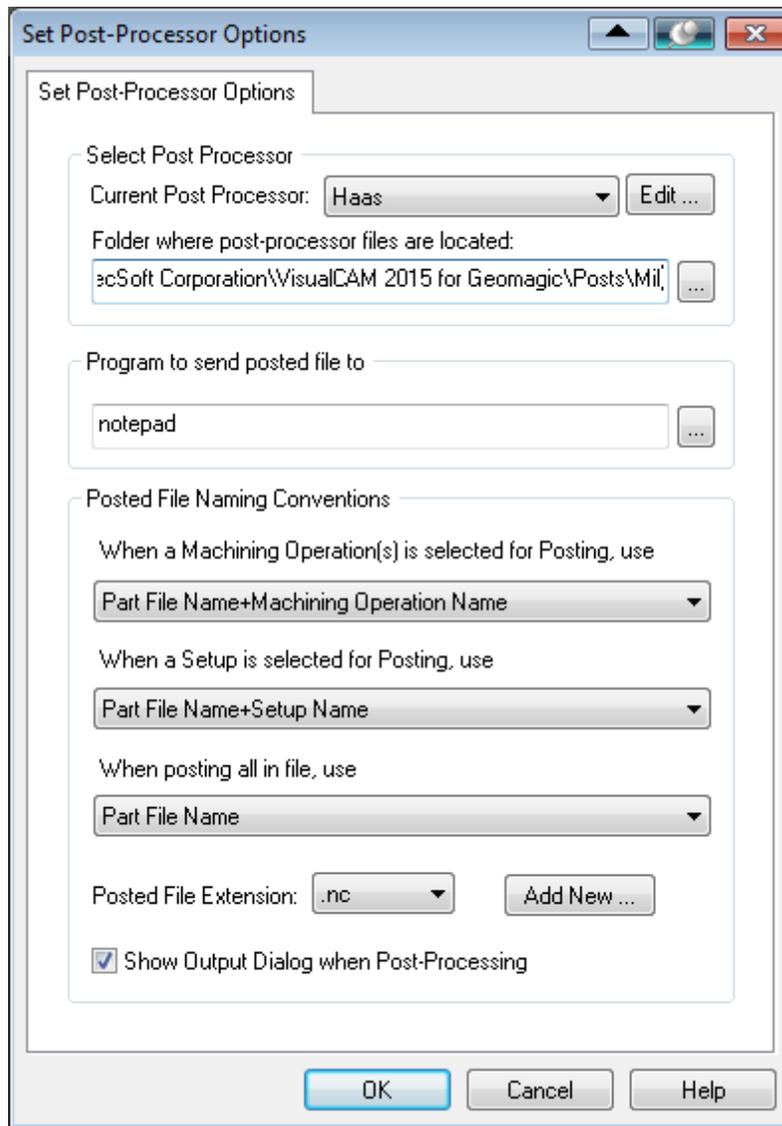
2.8 Select the Post Processor

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

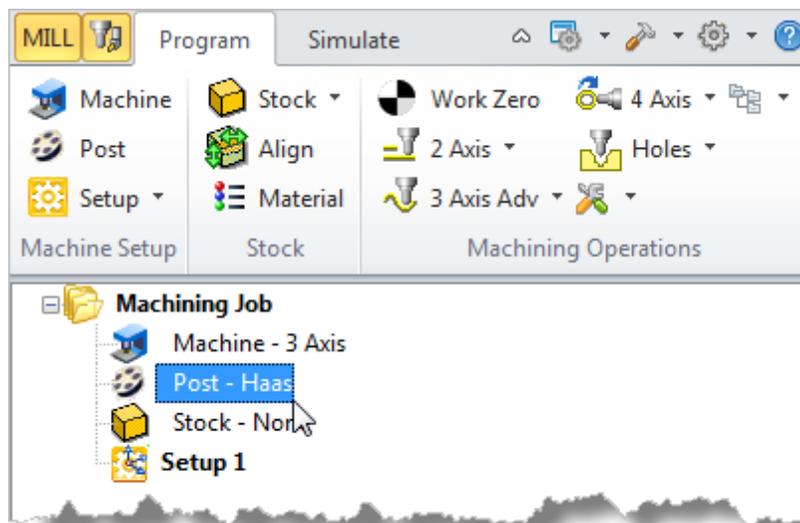
1. From the [Program](#) tab select [Post](#) to display the dialog.



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 2015 for Geomagic\Posts\MILL\

The program to send the posted output is set to notepad.

The Setup

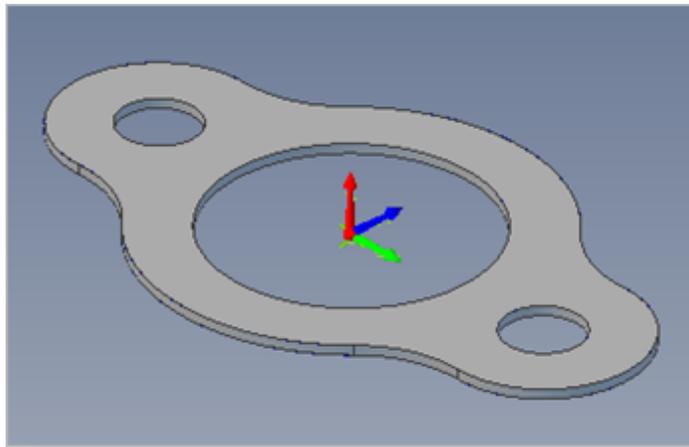
3.1 Define the Machining Setup - Skip this section if in Standard or Expert Configuration

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](#) ([Word Coordinate System](#)) so this step is not required for this part.

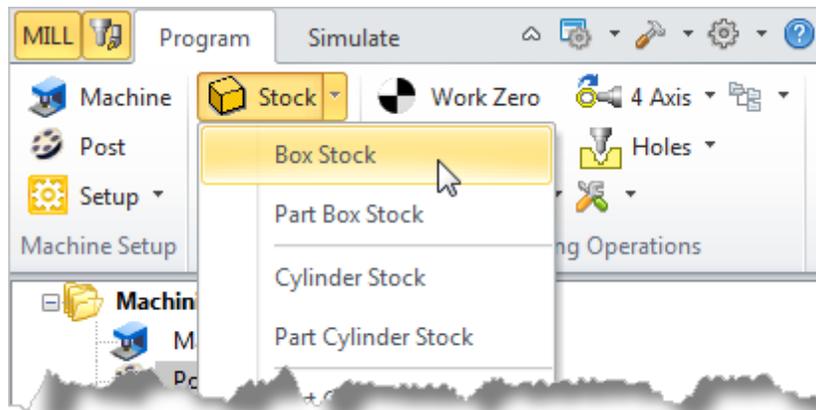


However, in production you can have multiple setups and assign different machining orientations for each.

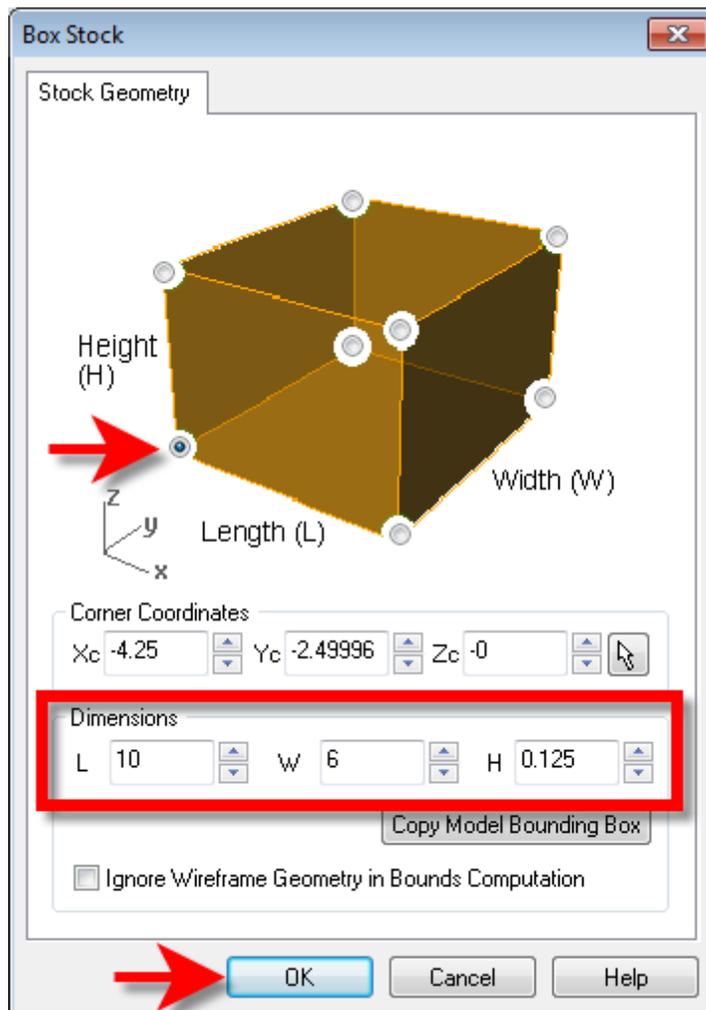
3.2 Create Stock Geometry

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.

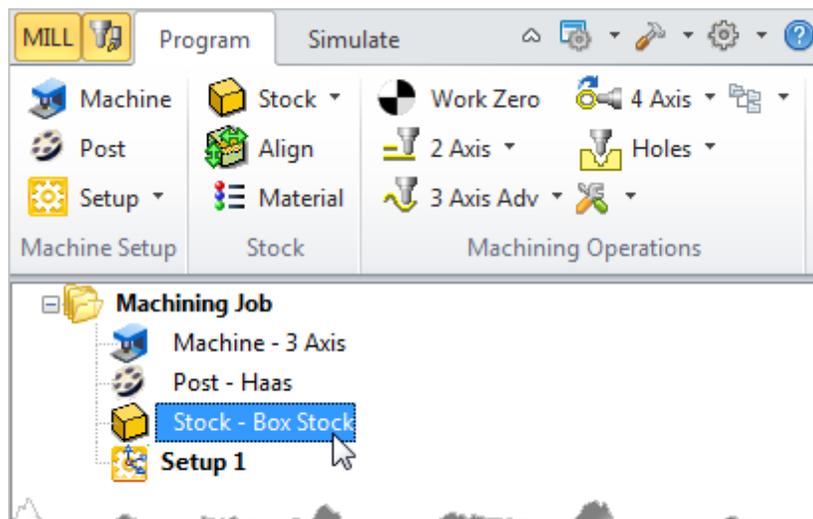


2. Under **Dimensions**, set the **Length L** to 10.0, **Width W** to 6.0 and **Height H** to 0.125. Note that the stock dimensions you enter are measured from the corner of the bounding box selected in this dialog.

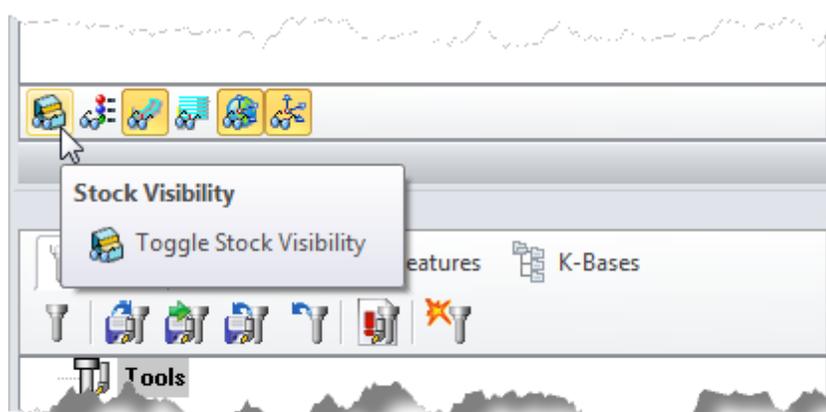


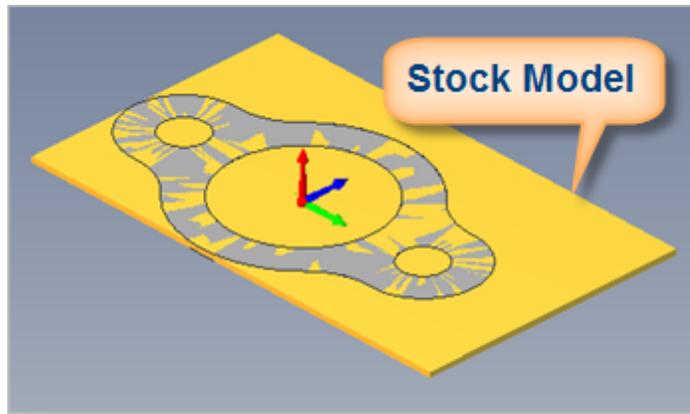
 The dimensions of the stock are interpreted in relation to the corner selected in the dialog box above. For example if the corner of the box is selected as the **Bottom South West** corner (as shown in the picture above), the **Length (L)** is interpreted to be along the **+X** axis, the **Width (W)** 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.

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



- If the stock does not display on the screen, select the **Stock Visibility** icon located at the base of the **Machining Browser**.

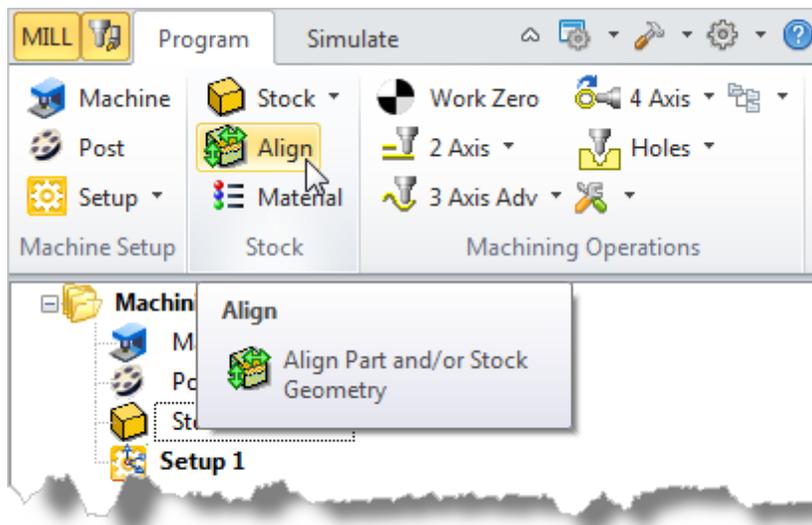




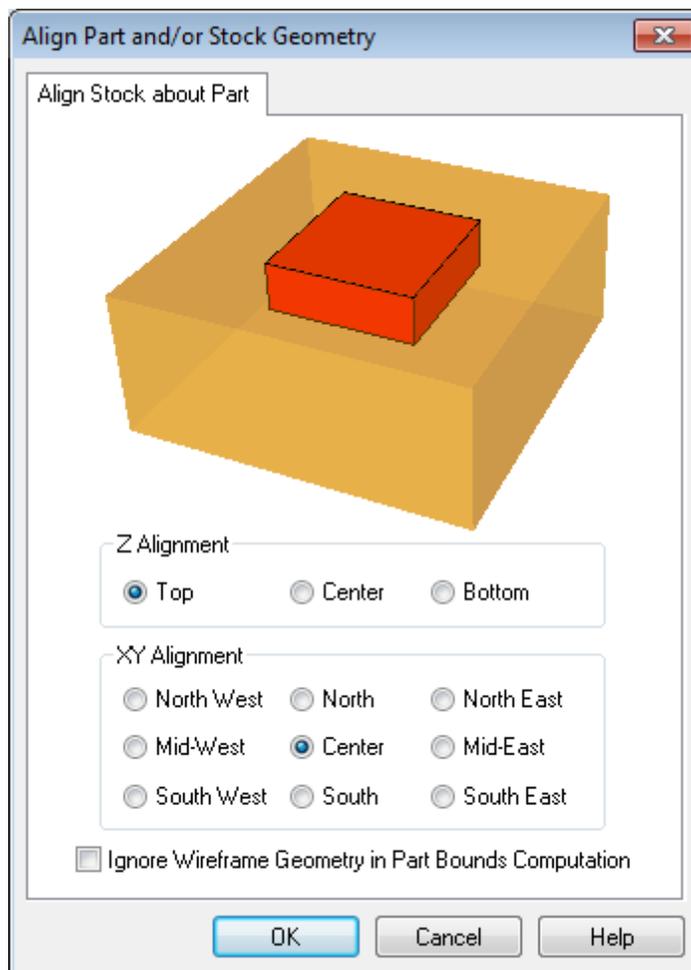
3.3 Align Part and Stock

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

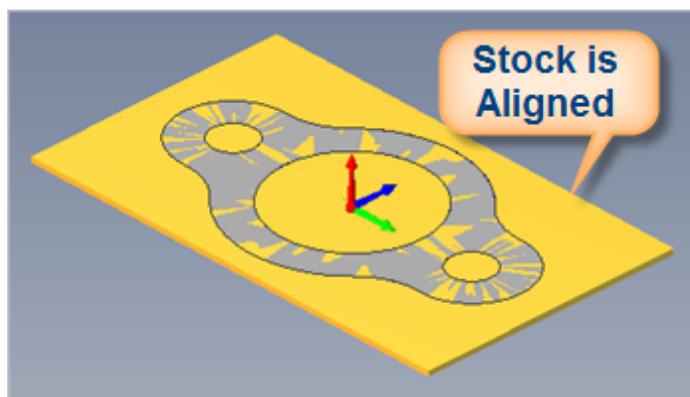
1. From the **Program** Tab select **Align** and then **Align Stock** from the menu to display the dialog.



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



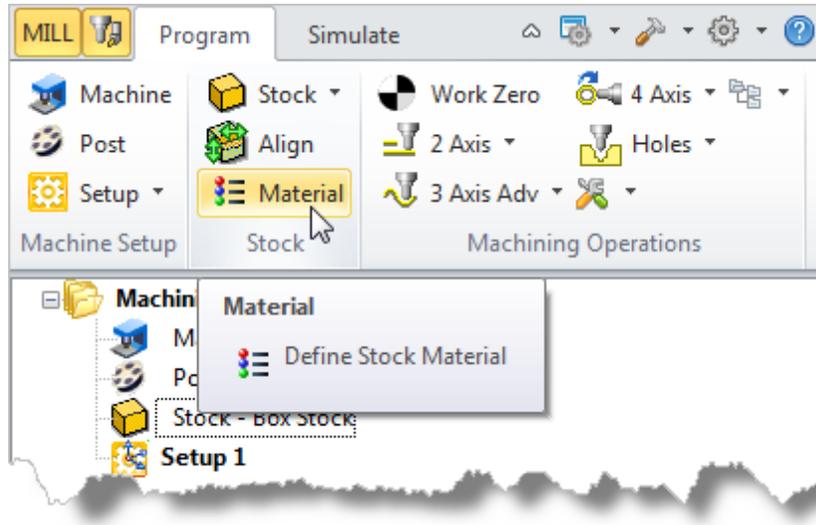
The stock is now aligned to the center of the part in XY and the Top of the part in Z.



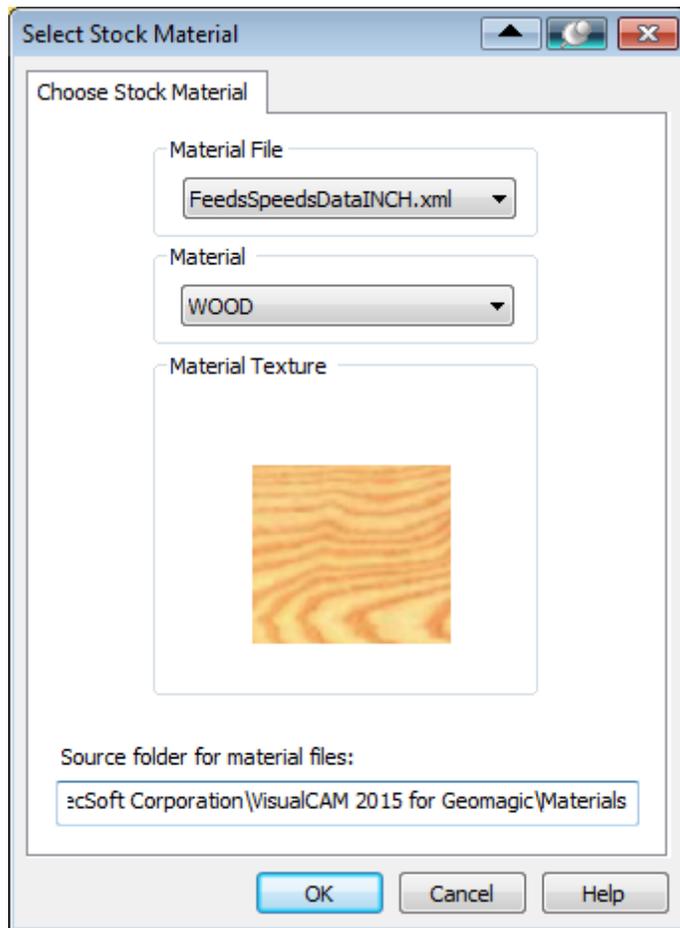
3.4 Specify Material

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

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

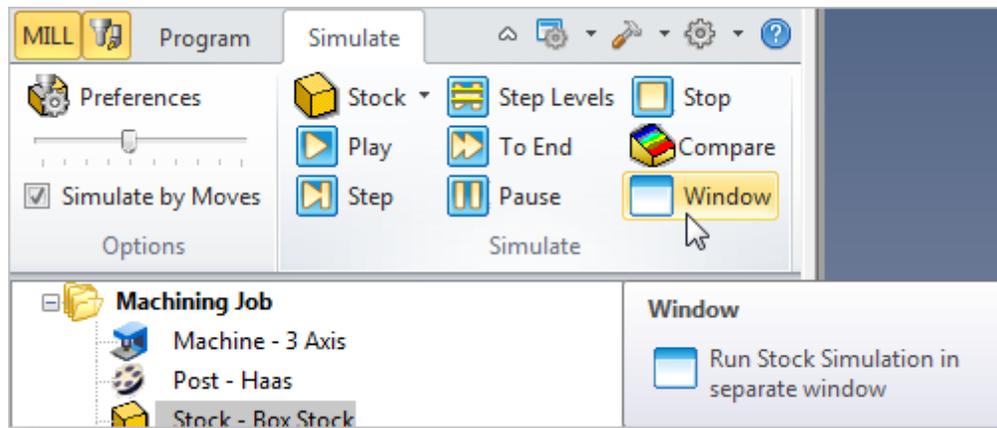


2. For **Material**, select **Wood** from the list of available materials and then pick **OK**.

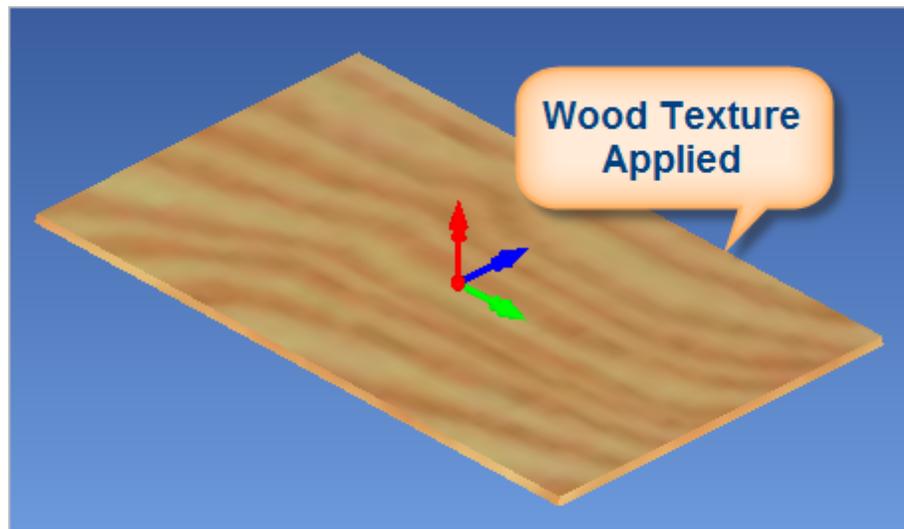
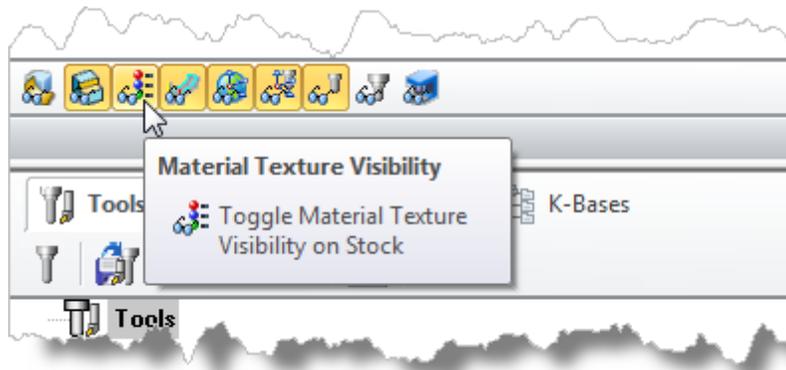


The material texture is now assigned to the [Stock](#) geometry. The texture is applied when the stock model is displayed in the [Simulation](#) window.

Note: [VisualCAM 2015 for Geomagic](#) uses a separate simulation window to perform and render the cut material simulation. To bring up this window, select the [Simulation](#) tab from the [Machining Browser](#) and select the [Window](#) icon as shown below:



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**.



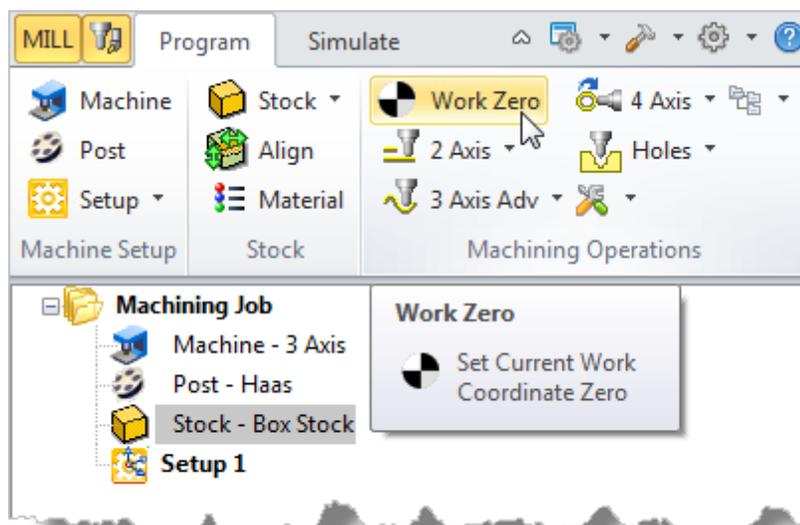
4. Now select the Simulation Window icon  again to close the Simulation Window.

3.5 Set Work Zero

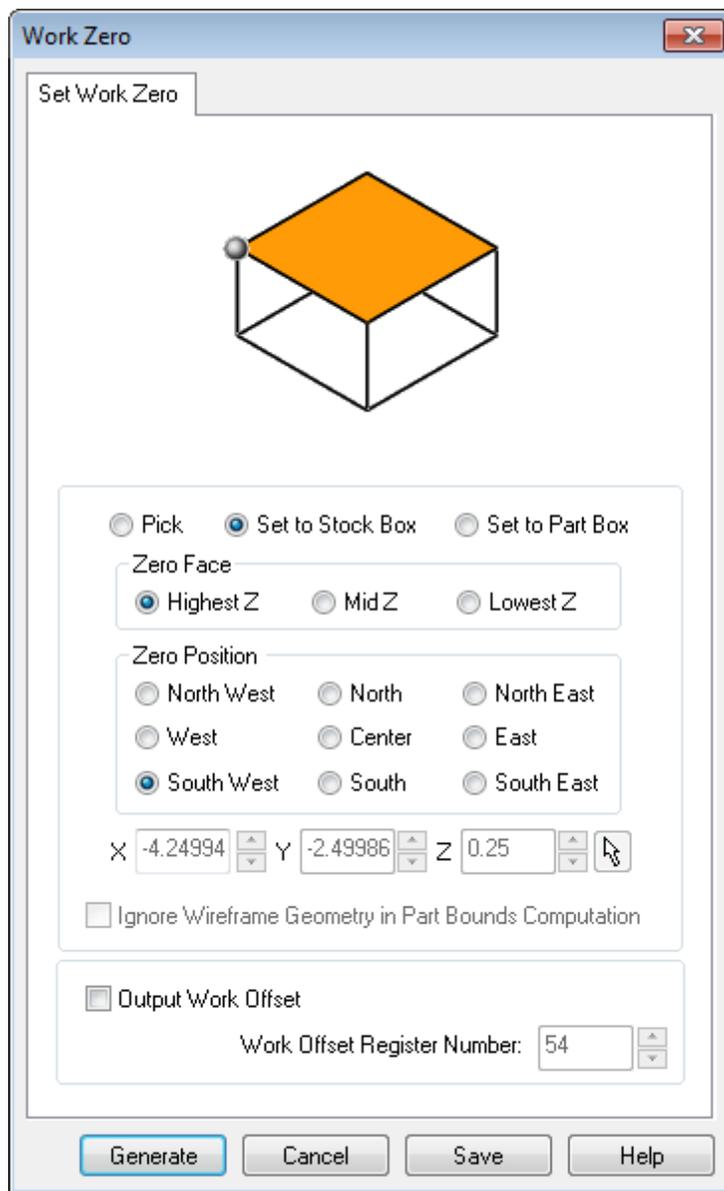
Now that the stock is aligned to the part geometry, in this step, we will establish the work coordinate origin also referred to as the *Work Zero*. The *Work Zero* translates the *MCS* origin from the *Setup* to the desired location. This can be set to any location on the part or stock geometry.

! The *Work Zero* defines the zero point with respect to which all toolpath points are interpreted by the controller. This would normally be the same as the tool touch off point on the actual work-piece on your machine. So care should be taken to make sure that this *Work Zero* point defined in VisualCAM*Work Zero* matches the tool zero point used on the actual work piece located on the table of your machine.

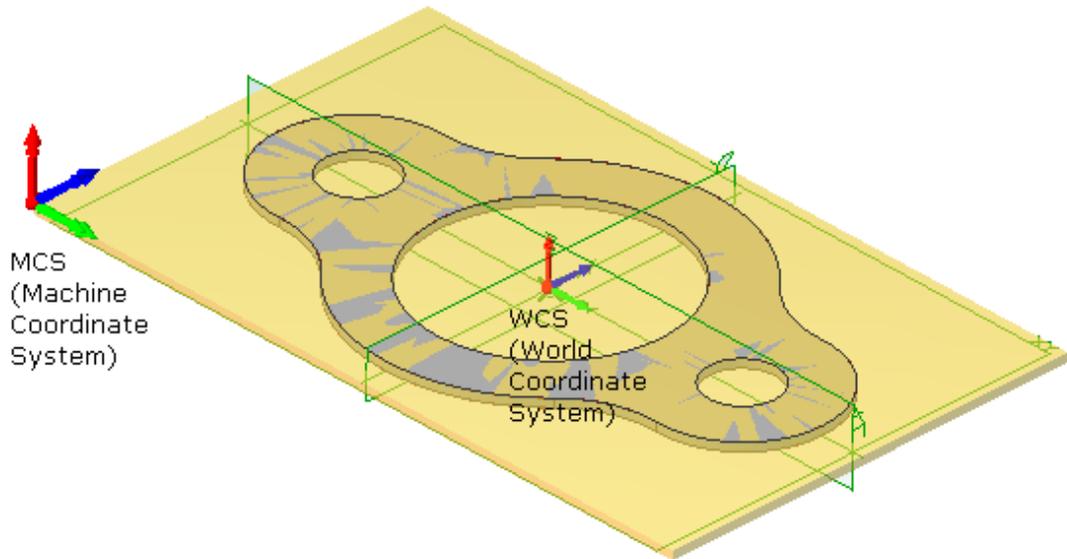
1. Select *Work Zero* from the *Program* tab of the *Machining Browser* to display the *Work Zero* dialog.



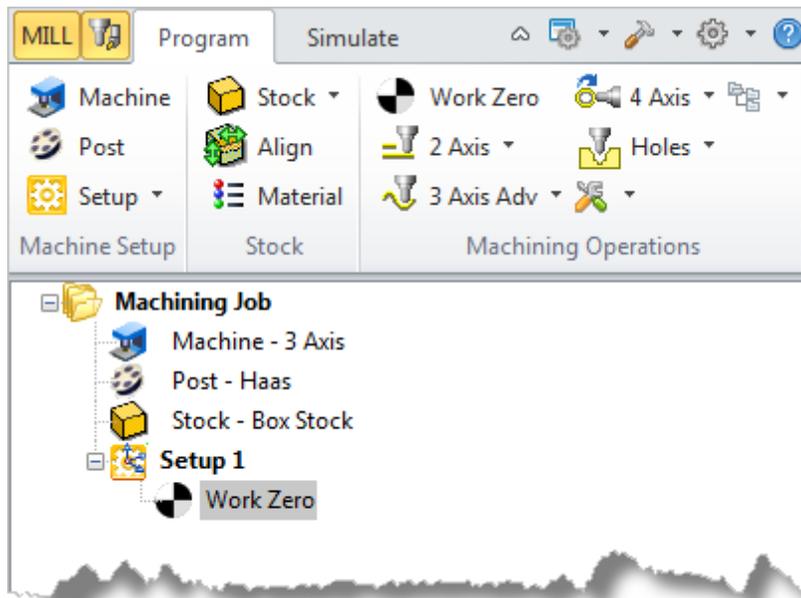
2. Select *Set to Stock Box* and then set *Zero Face* to *Highest Z*, *Zero Position* to *South West* corner and then pick *Generate*. This sets the machine home to the top of the stock material and the southwest corner of the stock geometry.



3. The **Work Zero** is now translated to the **Southwest** corner and top of stock material.



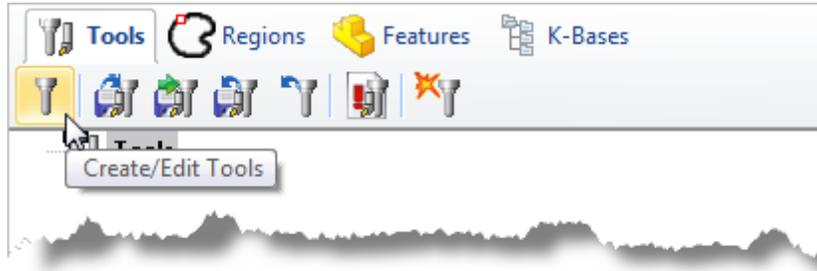
- 4. Also, the Work Zero now appears under the Machining Job in the Machining Browser.



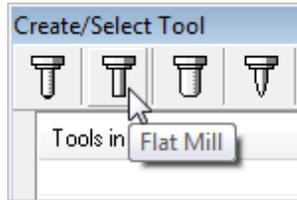
Create Tools

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

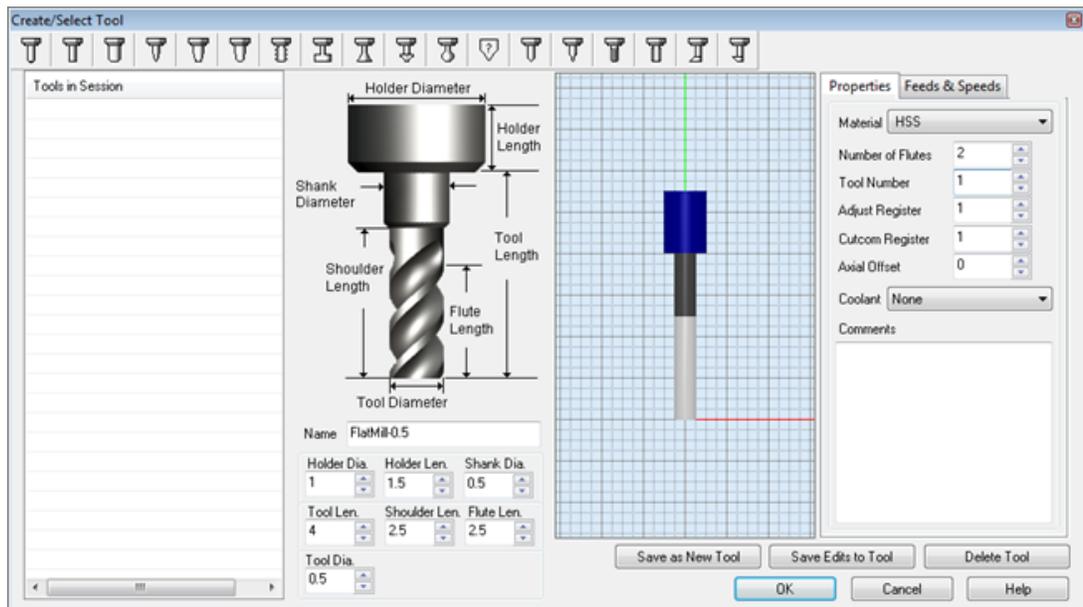
1. Select the **Tools** tab under **Machining Objects Browser** and click **Create/Edit Tools**.



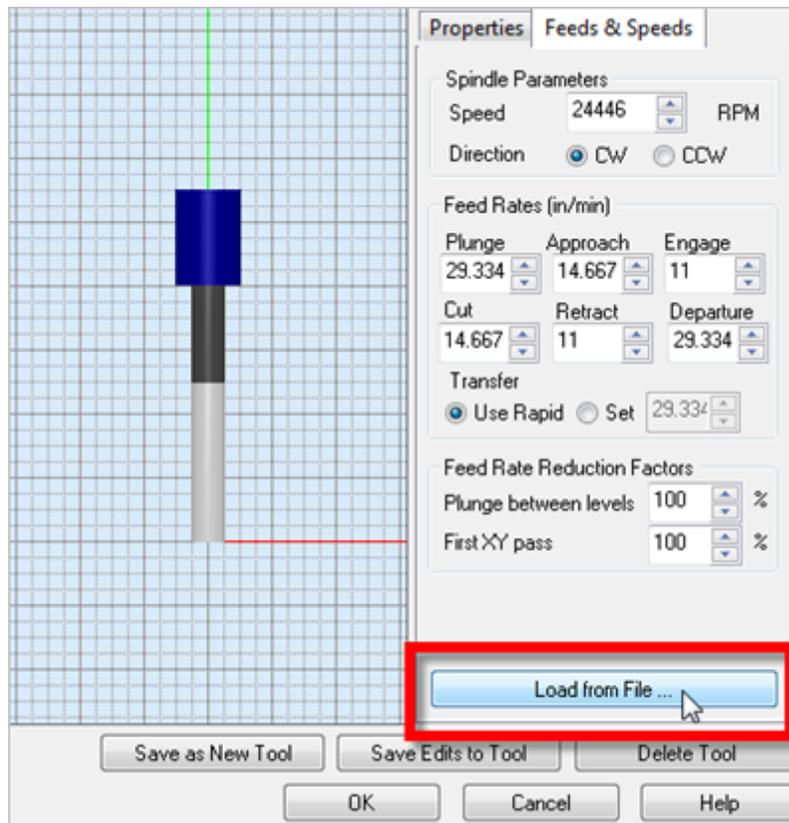
2. Select the Tool Type to Flat Mill.



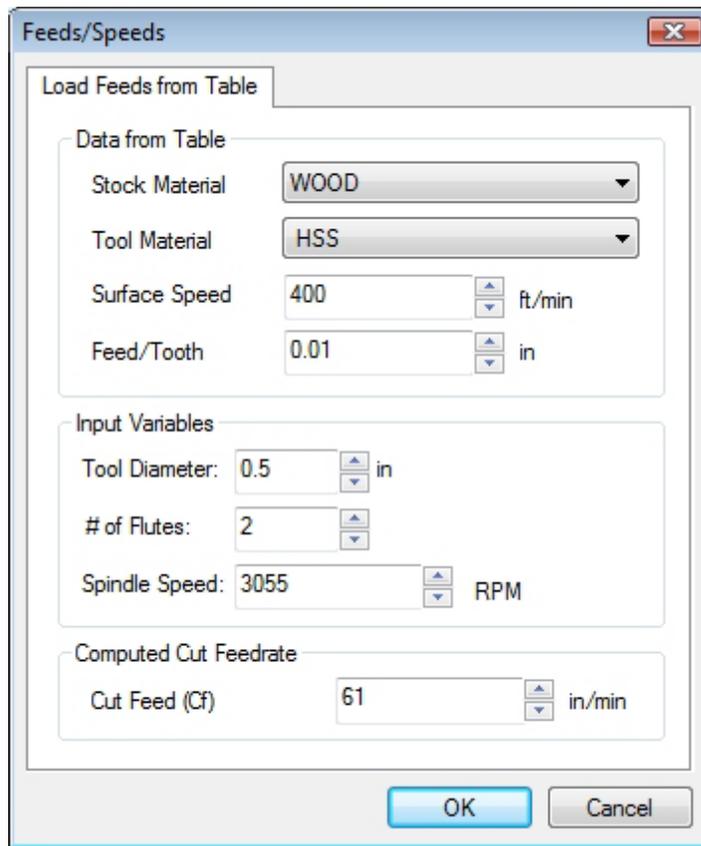
3. Set tool Name as FlatMill-0.5, Tool Diameter = 0.5. Under the Properties tab set Material to HSS and Tool Number = 1.



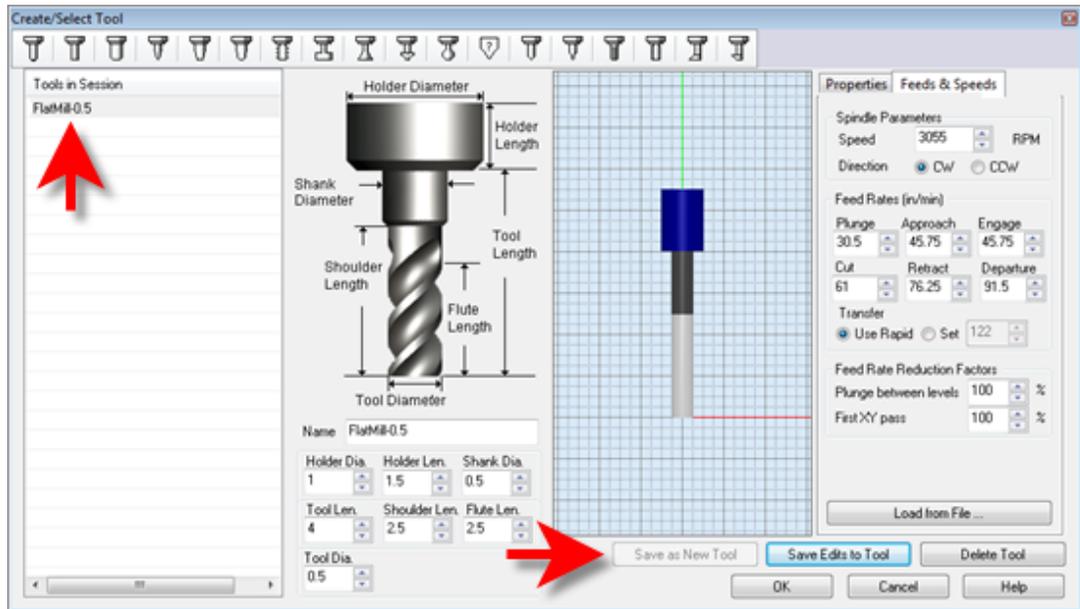
4. Switch to **Feeds and Speeds** tab and click **Load from File**.



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

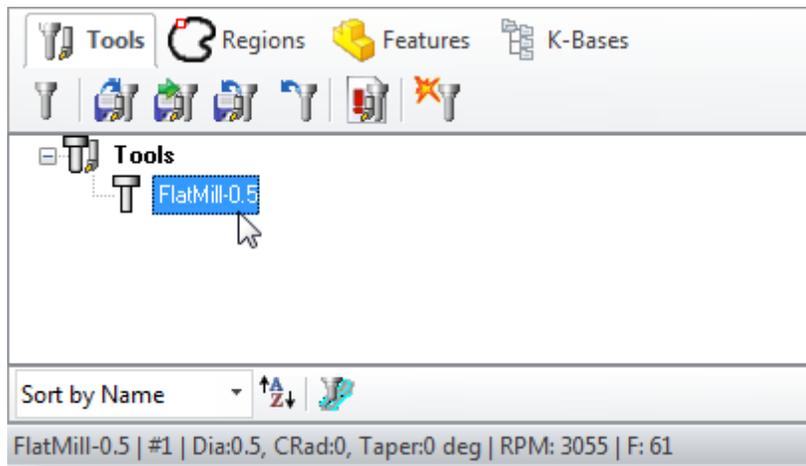


6. Click **OK** and the computed cut feedrate and spindle speed are transferred to the **Feeds and Speeds** tab of the tool dialog.
7. Click **Save as New Tool** to save the tool. The tool is now created and listed under **Tools in Session** on the left side of the **Create/Select Tool** dialog. Click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



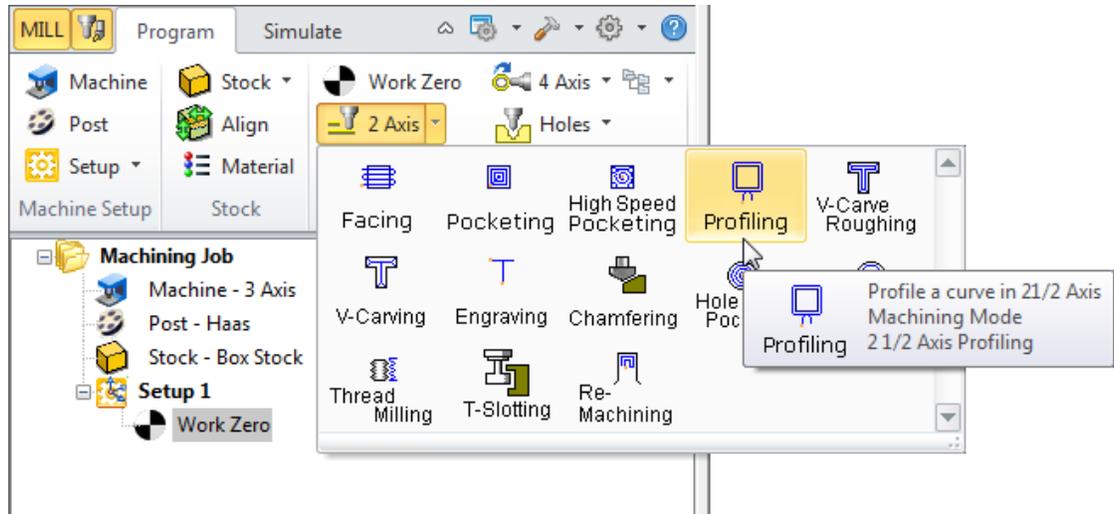
! To save Tools to a library, Click **Save Tool library** under the **Tools** tab in the **Machining Objects Browser** and specify a folder location and file name in the **Save as** dialog box. Saving a **Tool** library as a **Knowledge base** file (*.vkb) saves feeds and speeds with tool properties.

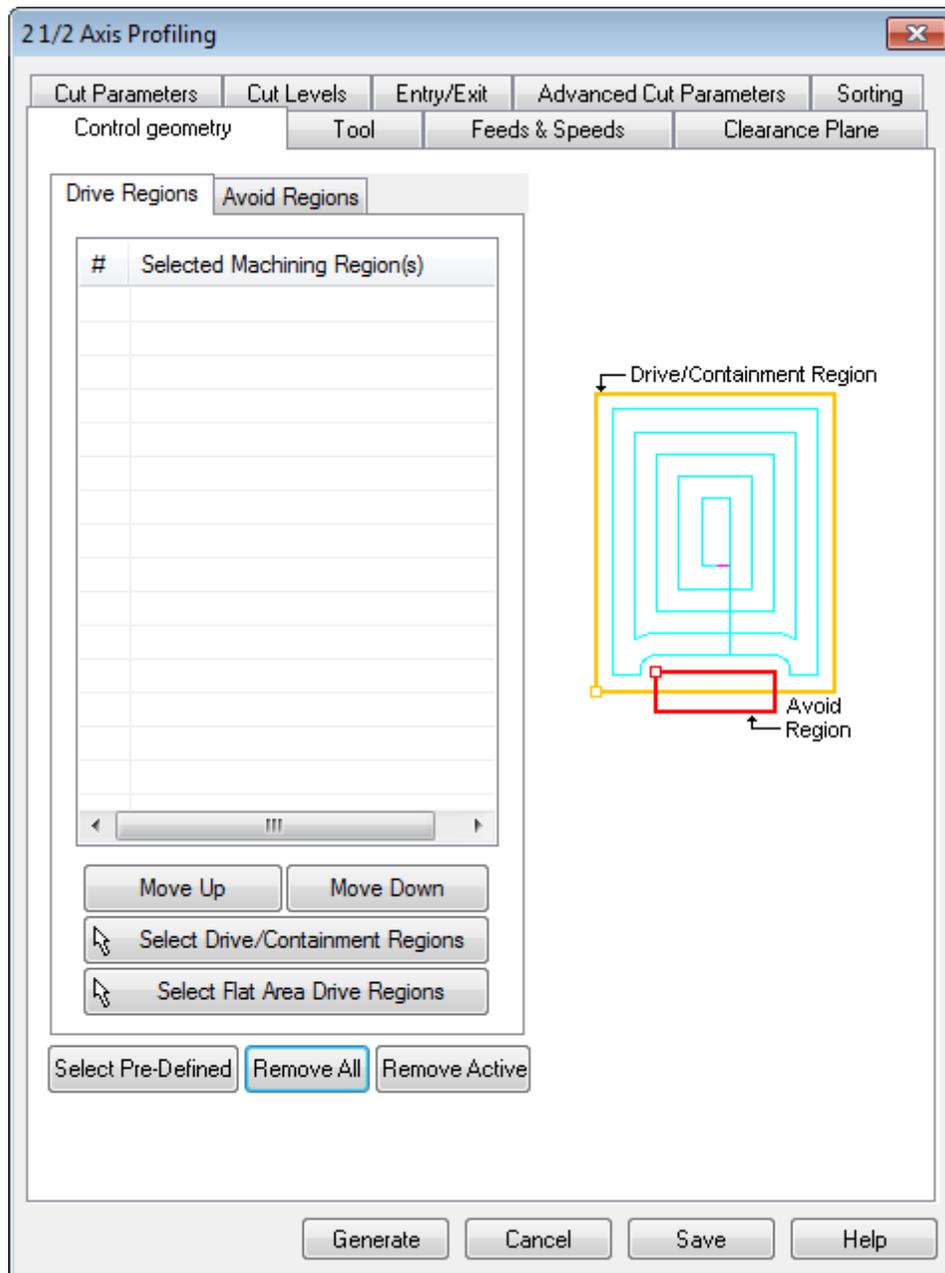
Create Machining Operations

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.

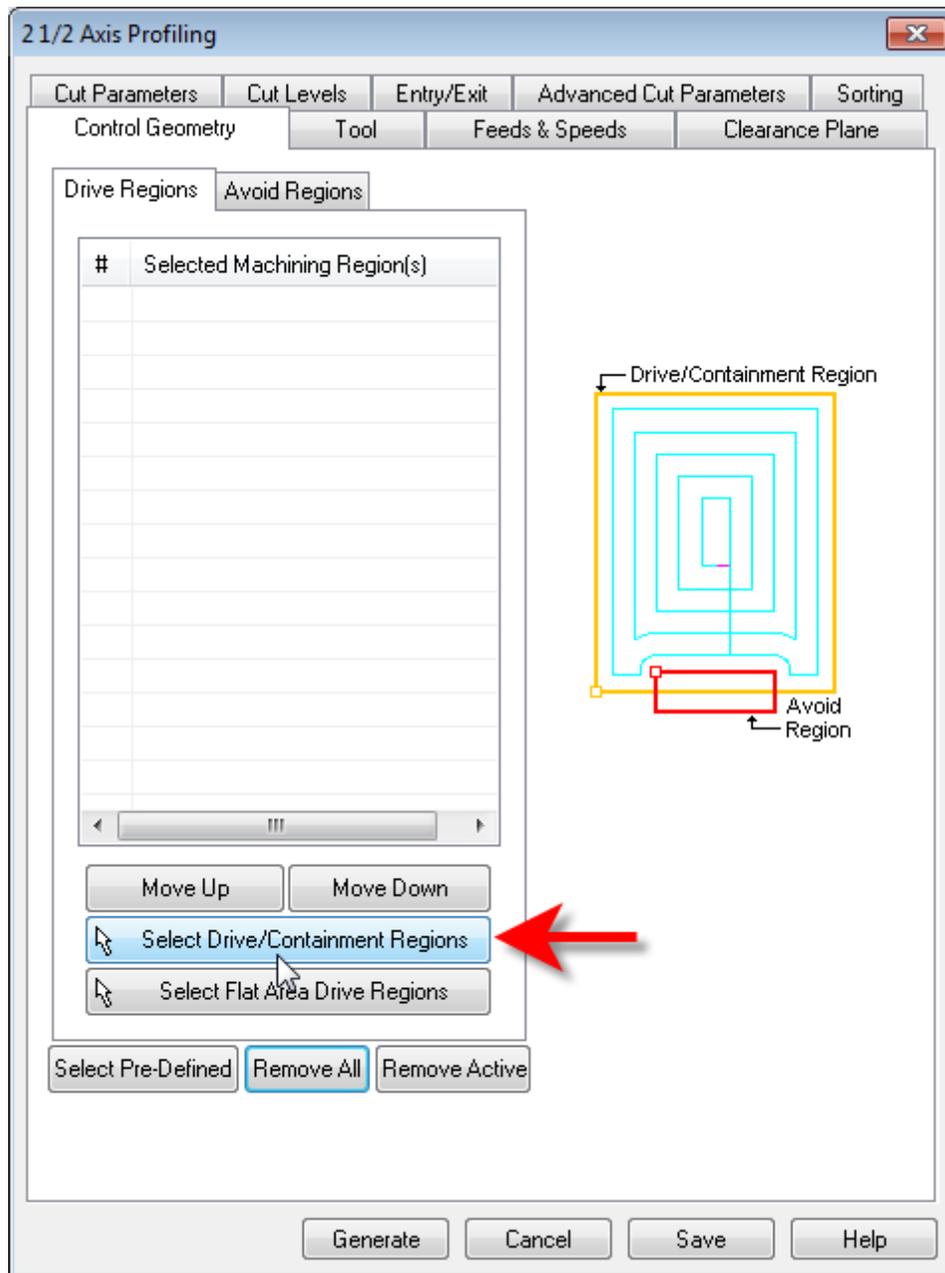
This will display the **2 ½ Axis Profiling** operations dialog. We will go over the steps for creating the profile operations for the inner features of the **Gasket**.





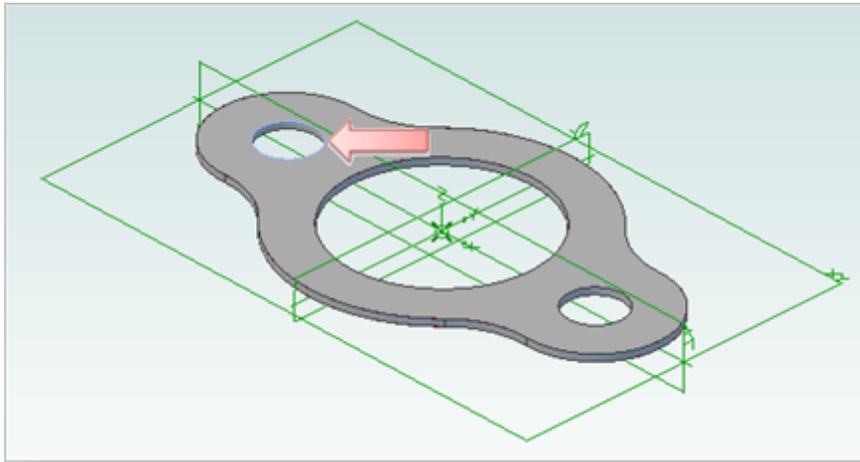
5.1 Select Machining Features/Regions

1. Under the **Control Geometry** tab pick **Select Drive /Containment Regions**.

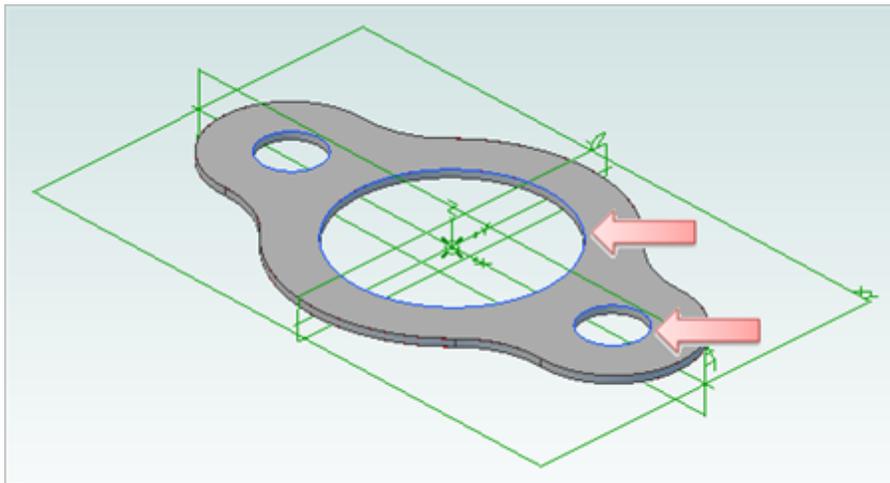


The Profiling operation dialog is now minimized and allows selection of features to machine. We will now select the surface edges of the 3 inside hole features.

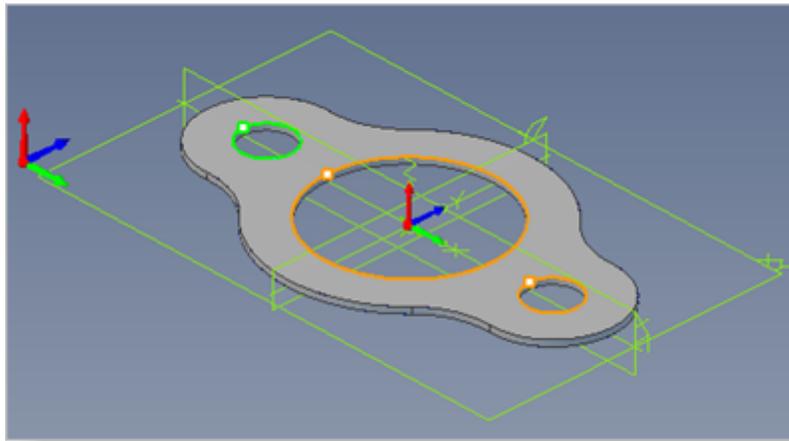
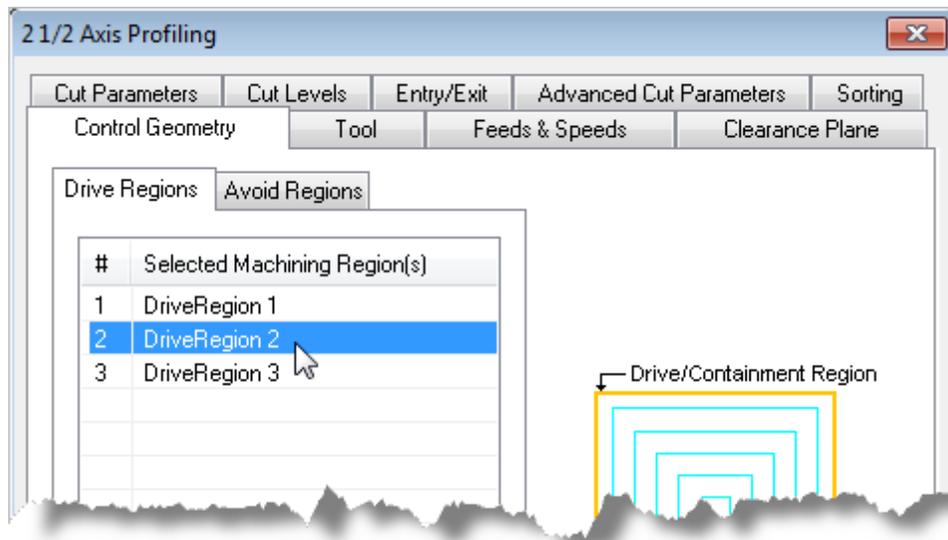
2. While pressing the <Shift> key, select the first hole by clicking near the upper surface edge as shown below.



3. Now, while the **<Shift>** key is still pressed, select the edges of the two remaining holes.



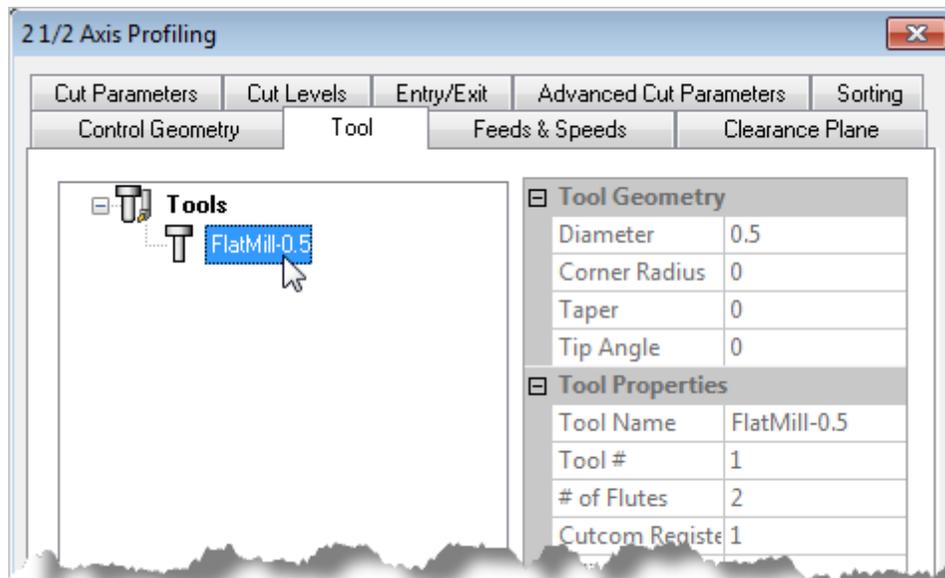
4. Now select the  icon to complete the selection.
5. The **2 ½ Axis Profiling** dialog comes back up displaying the selected **Drive Regions**. They are also highlighted on the part.
6. Notice that selecting a **Drive Region** from the list highlights the corresponding surface edge curve on the part.



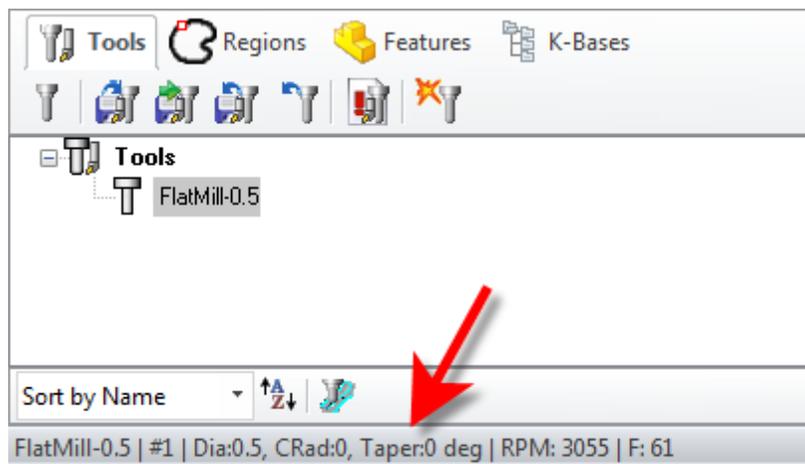
5.2 Select Cutting Tool

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.



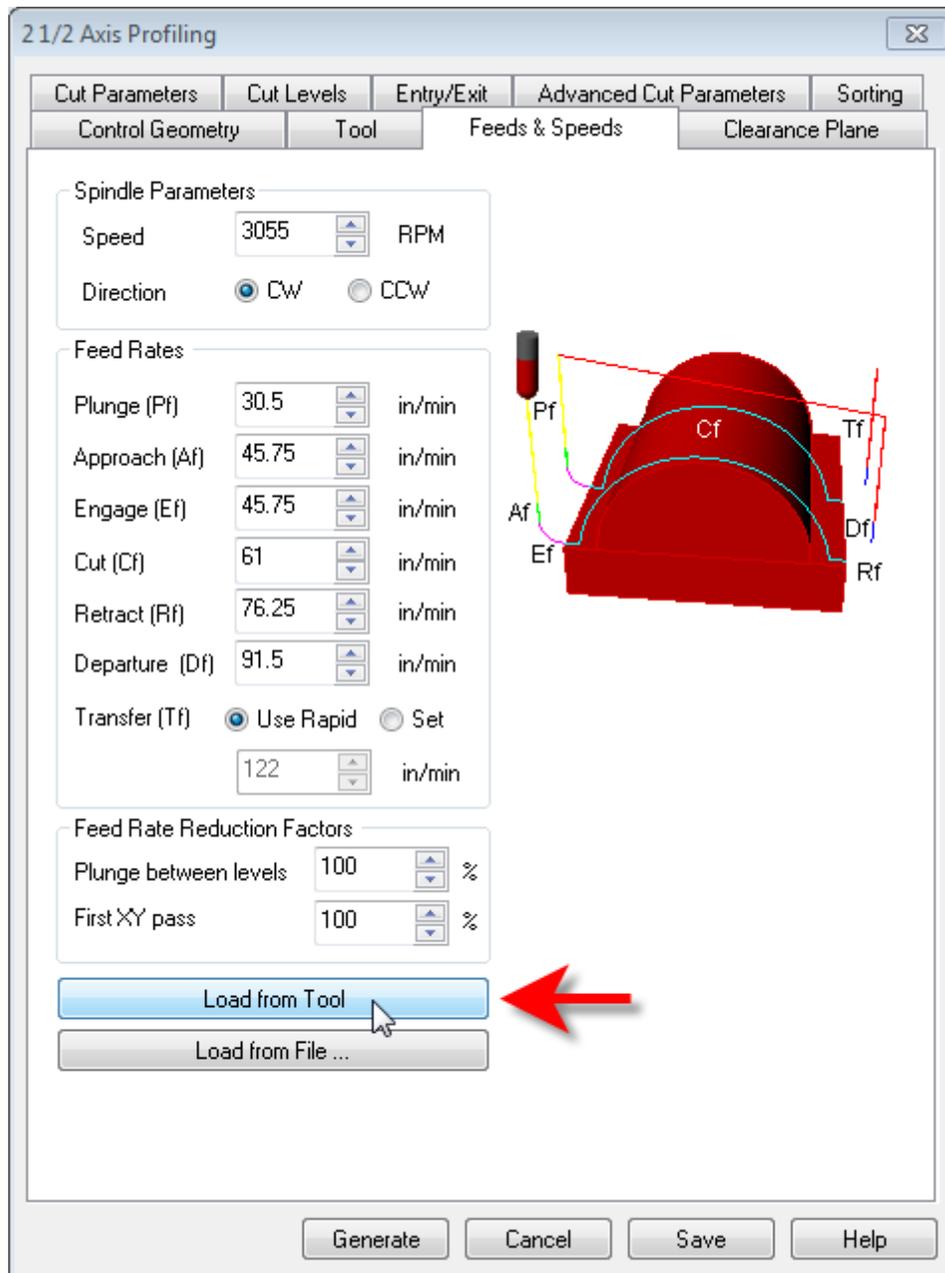
Note that the **Tool** parameters of the currently active tool are always displayed in the status bar at the bottom of the **Machining Objects Browser**.



5.3 Set Feeds and Speeds

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

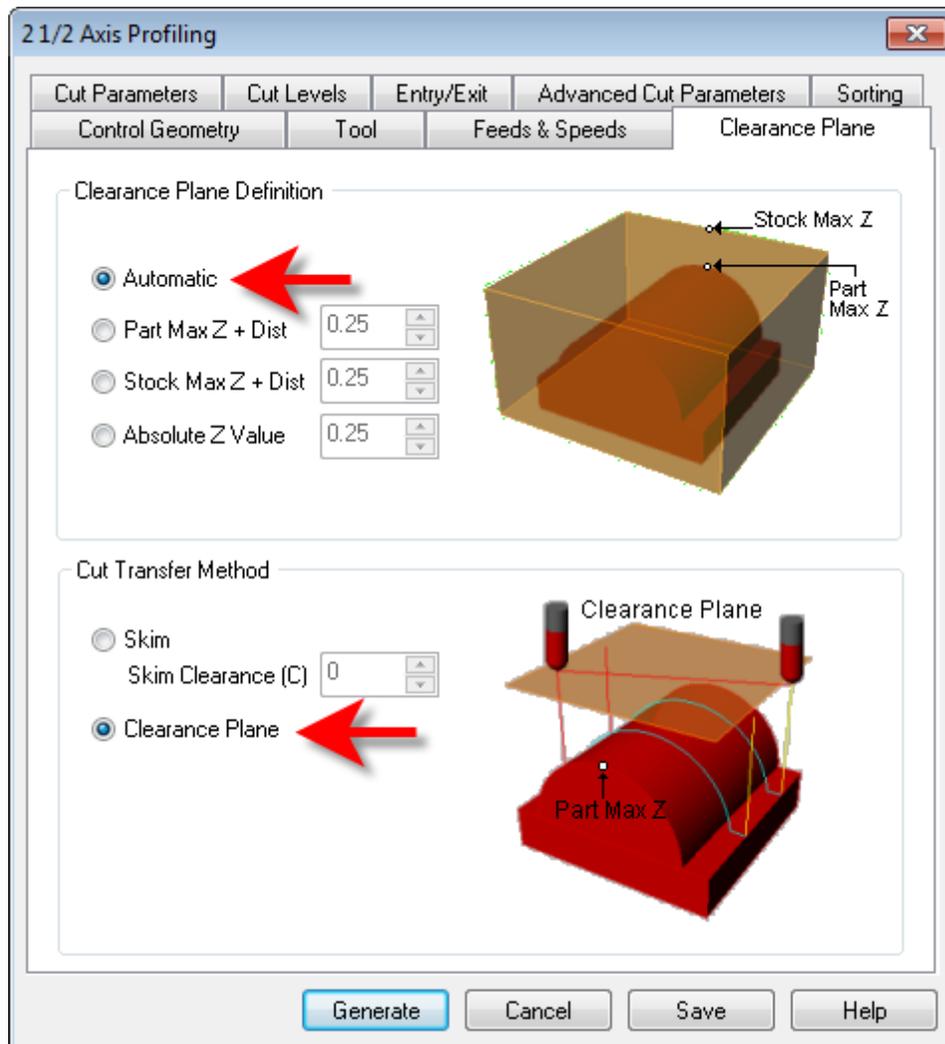
1. Switch to the **Feeds & Speeds** tab of the **2 1/2 Axis Profiling** dialog.
2. Select the **Load from Tool** button. **VisualCAM 2015 for Geomagic** will retrieve the feeds and speeds parameters that were set when the tool was defined and associate them with the current operation.



5.4 Clearance Plane

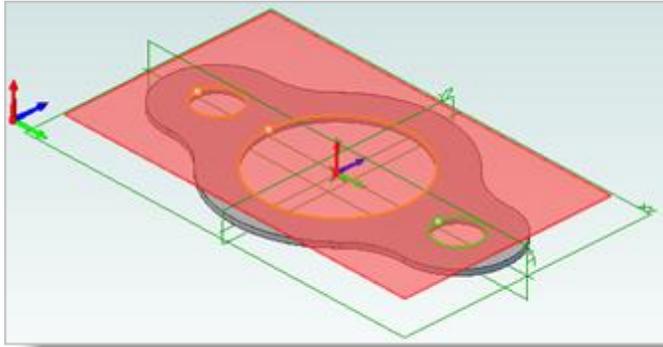
Now we'll set the [Clearance](#) geometry for our operation:

1. We'll switch to the [Clearance Plane](#) tab of the [2 ½ Axis Profiling](#) dialog.
2. Set the [Clearance Plane Definition](#) to [Automatic](#) and [Cut Transfer Method](#) to [Clearance Plane](#).



In the **Automatic** mode, VisualCAM 2015 for Geomagic 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.

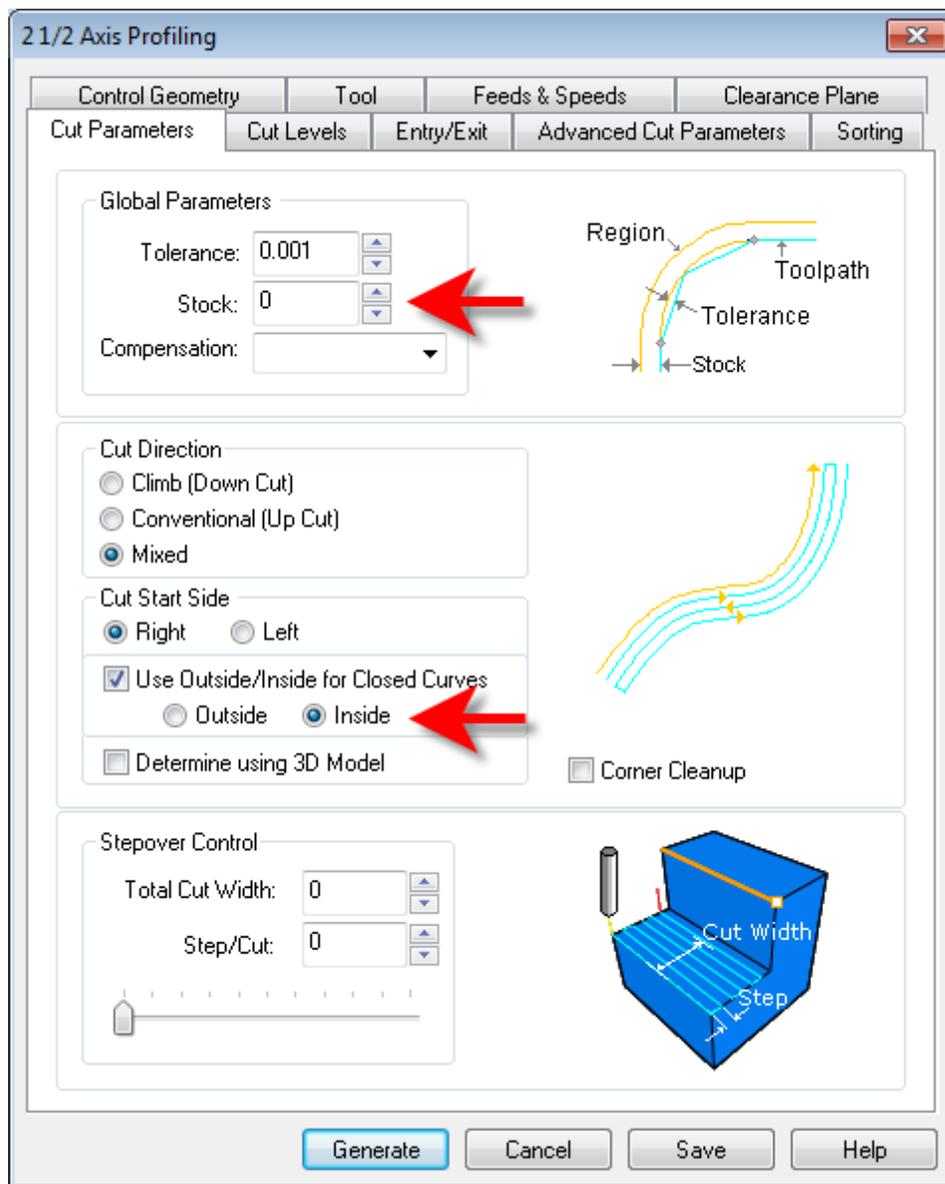
When this tab of the dialog is active, the clearance plane is shown on the graphics screen.



5.5 Specify Cut Parameters

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

1. Switch to the [Cut Parameters](#) tab of the [2 ½ 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](#).

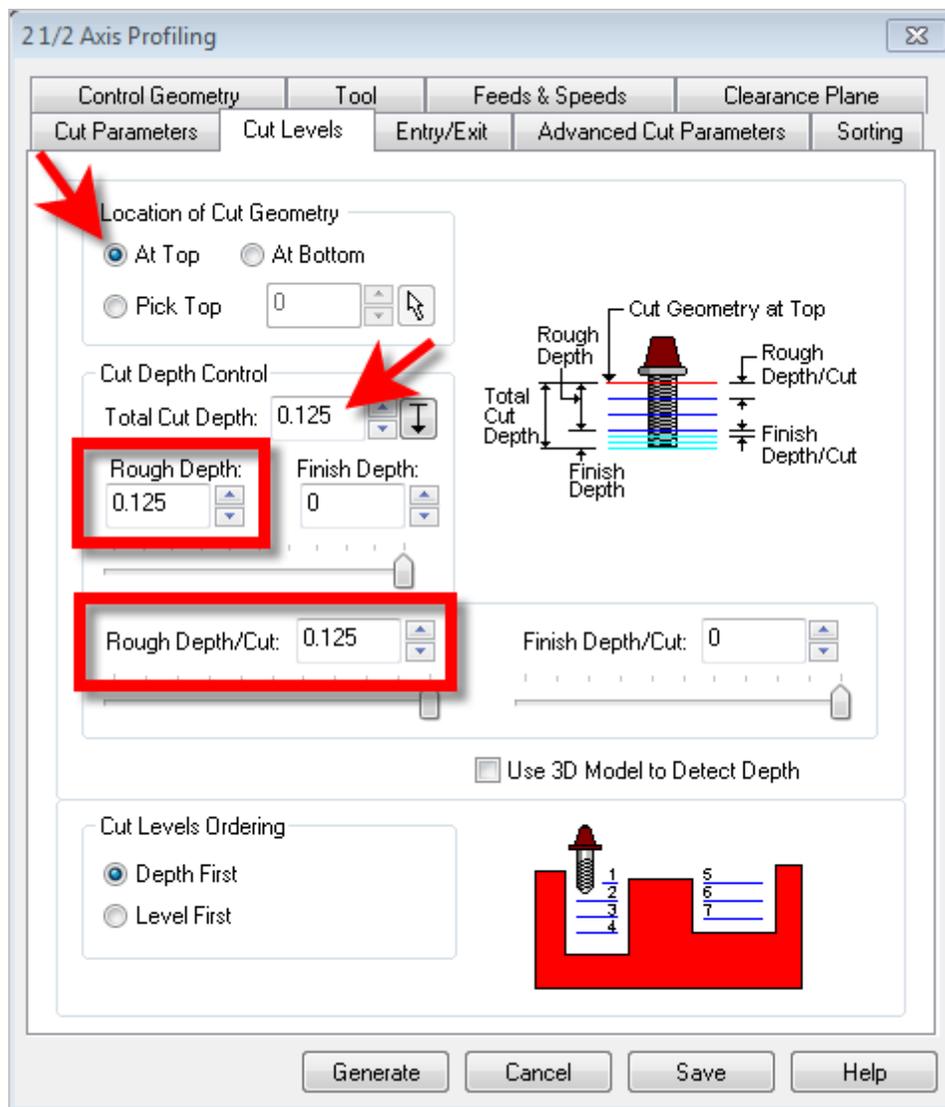


Alternately you could use the [Determine using 3D Model](#) option. In this case [VisualCAM 2015 for Geomagic](#) would use the 3D model to determine which side of the curve to place the cutter for machining.

5.6 Cut Levels

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

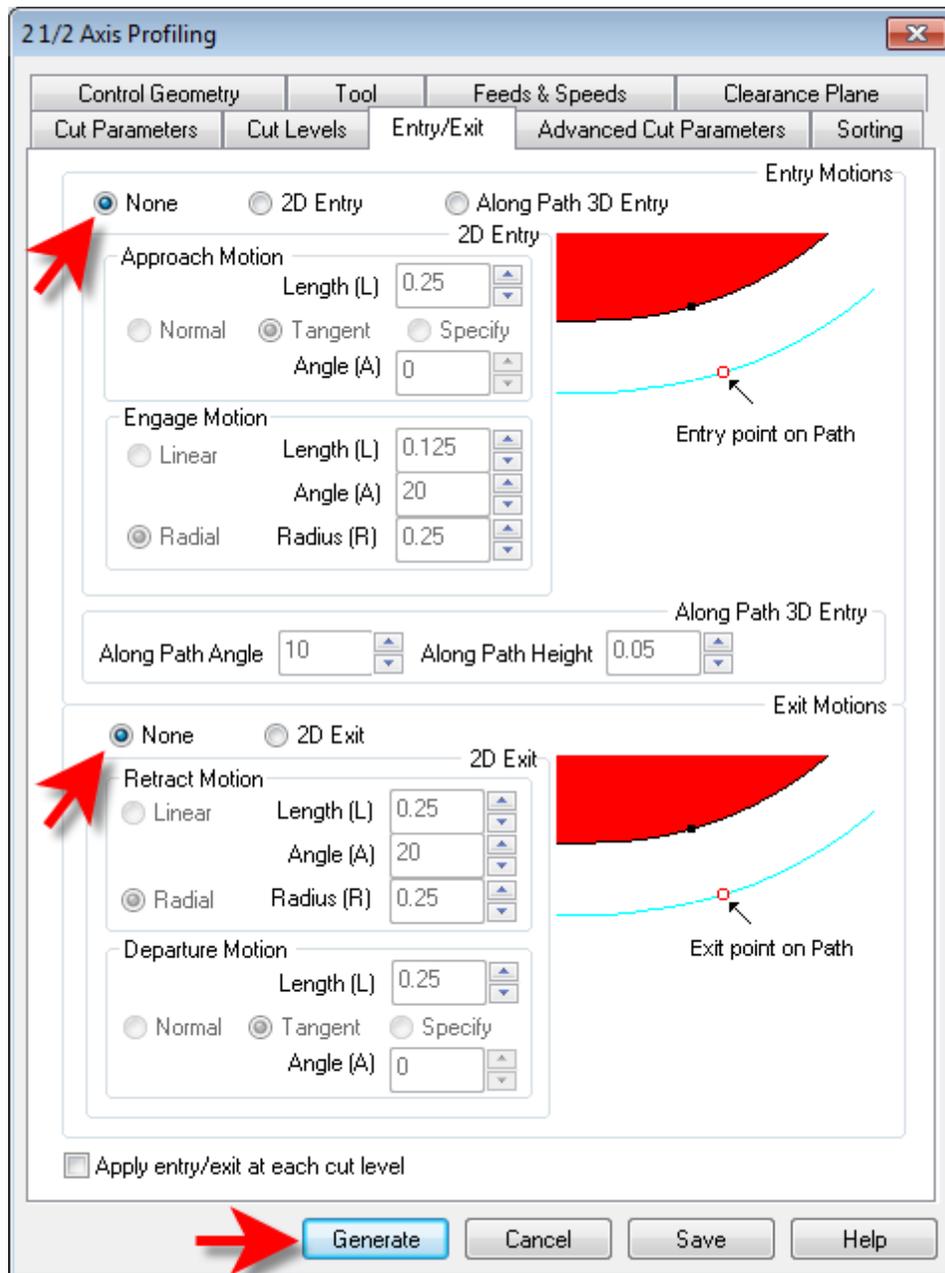
1. Select the [Cut Levels](#) tab of the [2 ½ Axis Profiling](#) dialog.
2. Set [Location of Cut Geometry](#) to [At Top](#).
3. For [Total Cut Depth](#), enter [0.125](#). The cut depth is always set as an absolute value.
4. This automatically sets the [Rough Depth](#) and [Rough Depth/Cut](#) to [0.125](#).



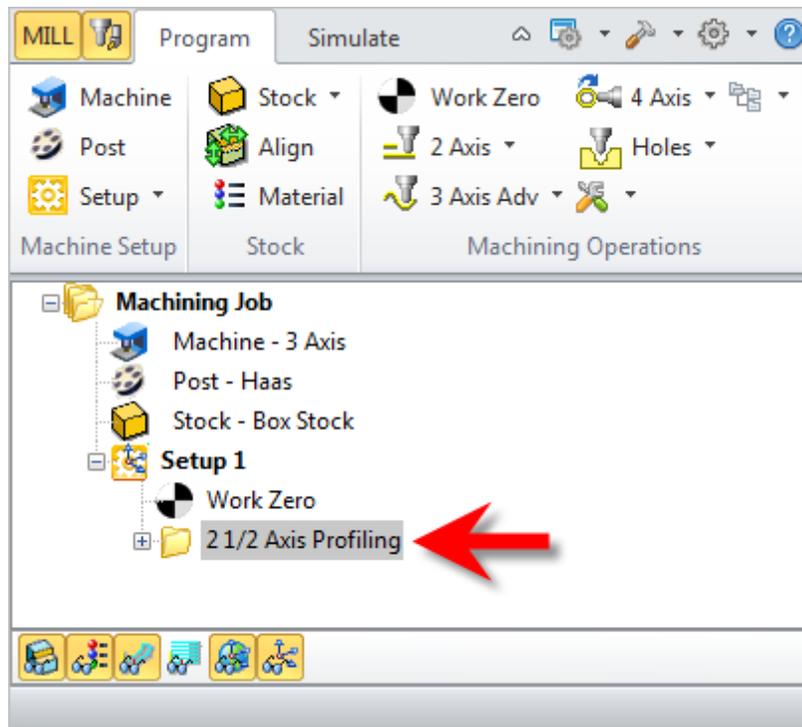
5.7 Entry/Exit Parameters

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

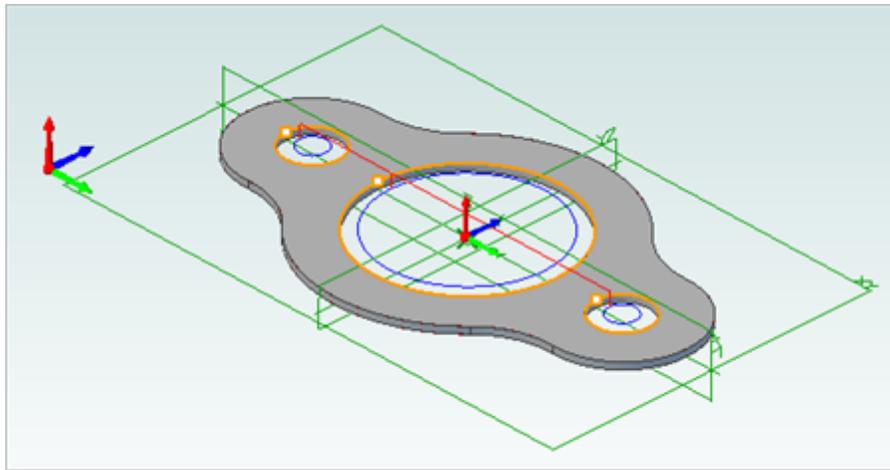
1. Next, we elect the [Entry/Exit](#) tab.
2. [Entry/Exit](#) parameters control how the cutter will engage material as it begins cutting and how it leaves the material as it completes cutting.
3. Set [Entry Motions](#) and [Exit Motions](#) to [None](#).
4. Now pick [Generate](#).



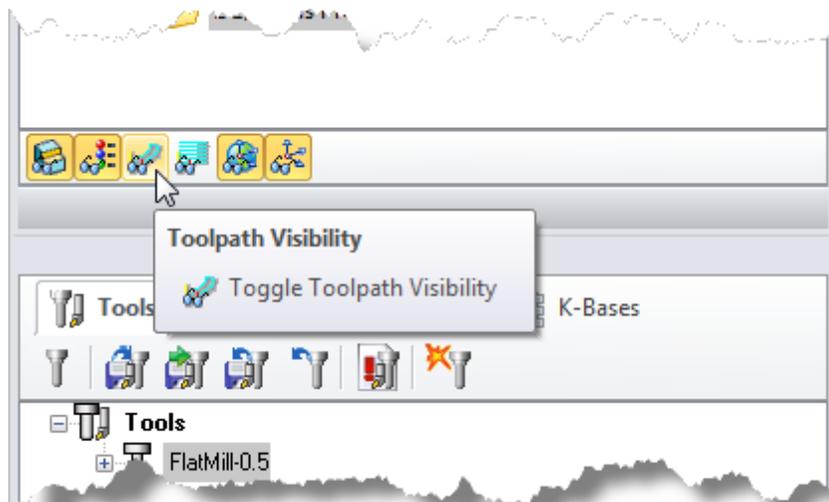
5. The 2½ Axis Profile toolpath is generated and the operation is listed under Setup 1 in the Machining Browser.



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



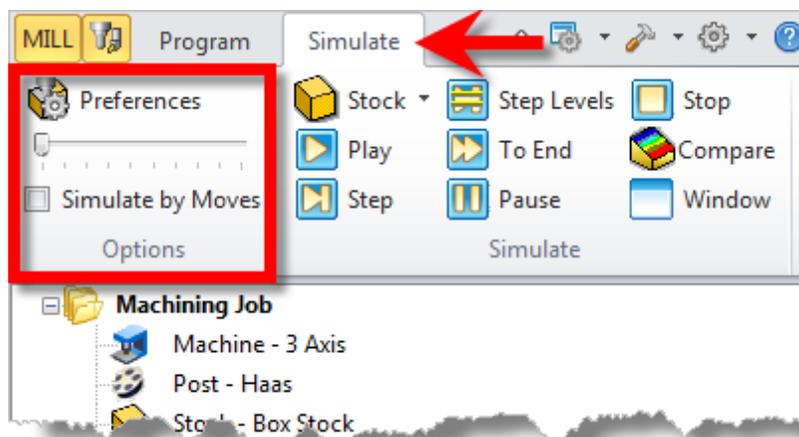
7. Note that the display of the toolpath in the graphics screen can be turned on/off by selecting the **Toolpath Visibility** icon located at the base of the **Machining Browser**.



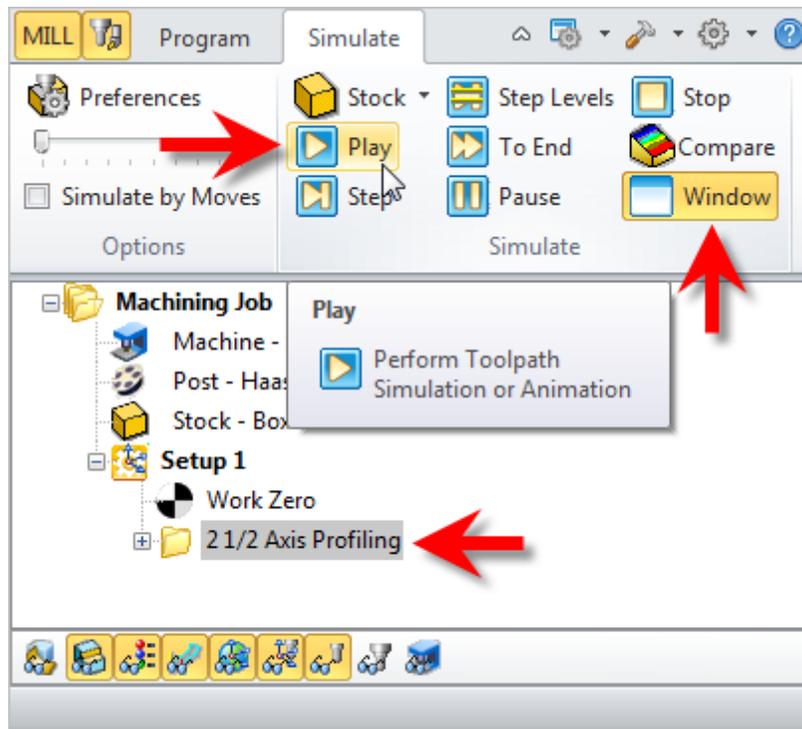
5.8 Simulate Toolpath

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

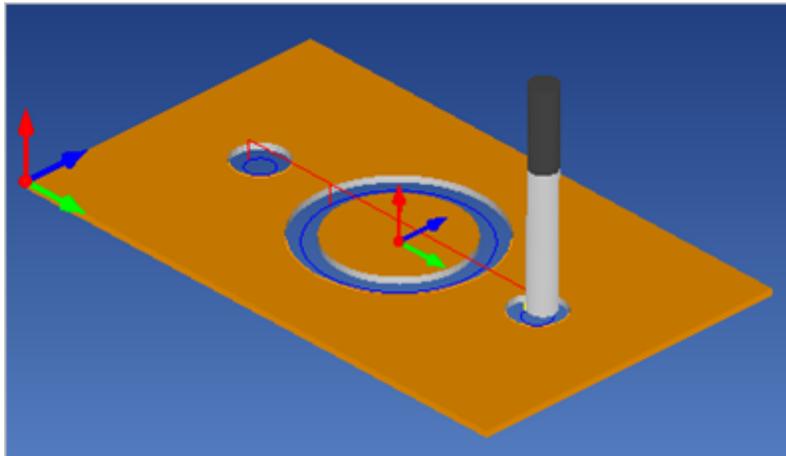
1. Switch to the [Simulate](#) tab at the top of the [Machining Browser](#).
2. Under [Preferences](#), uncheck [Simulate by Moves](#) and adjust the slider to the left to slow down the simulation speed.



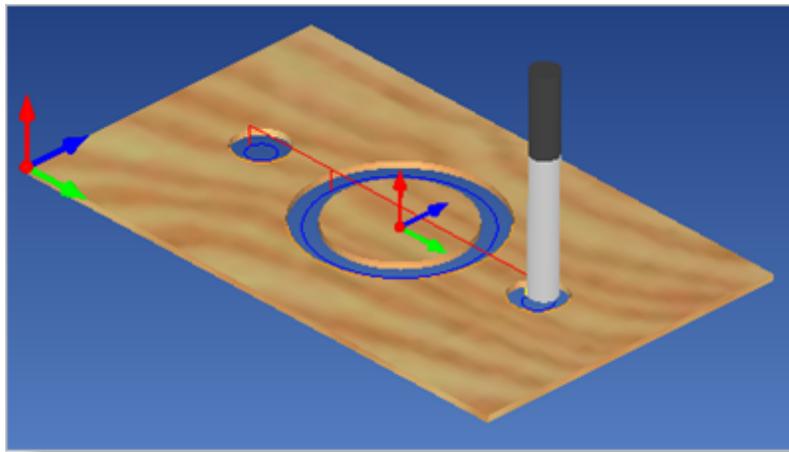
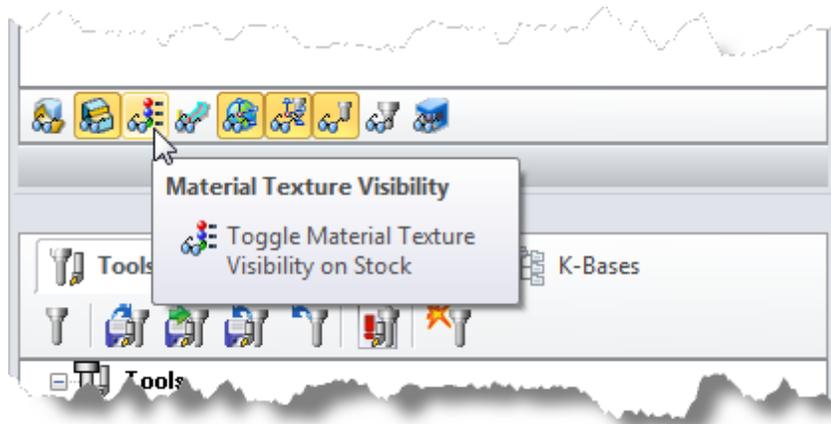
3. Select the [2 1/2 Axis Profiling](#) operation we just created, select the [Simulation Window](#) icon  and then pick [Play](#) to start the simulation.



4. 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.



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

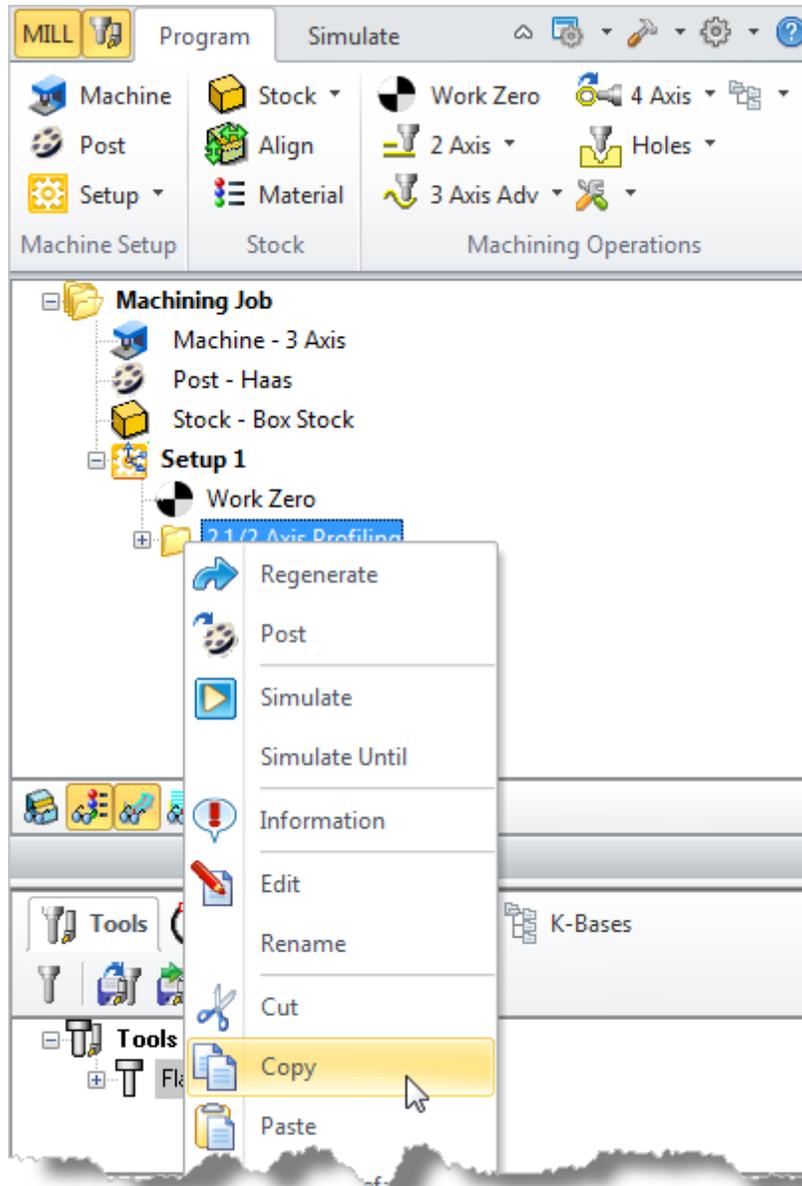


Now select the Simulation Window icon  Window again to close the Simulation Window.

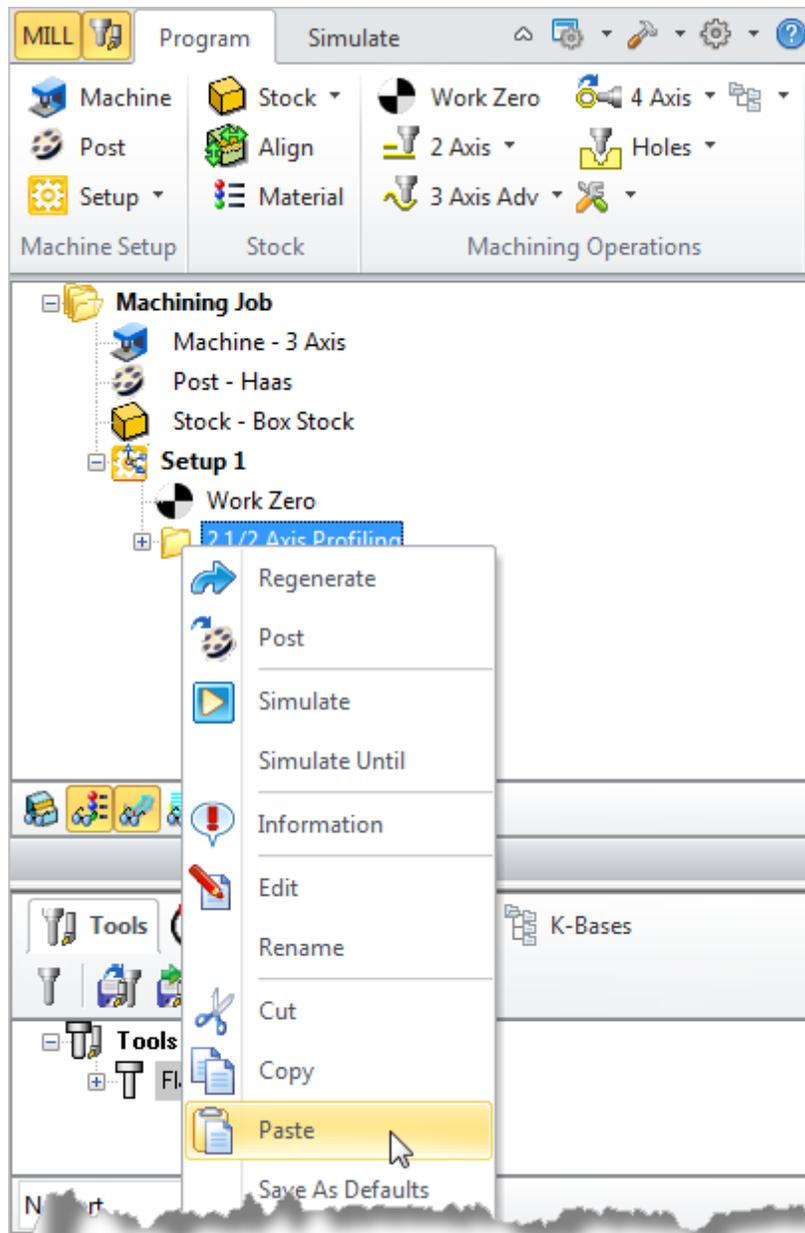
Machining the Outer Area

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.

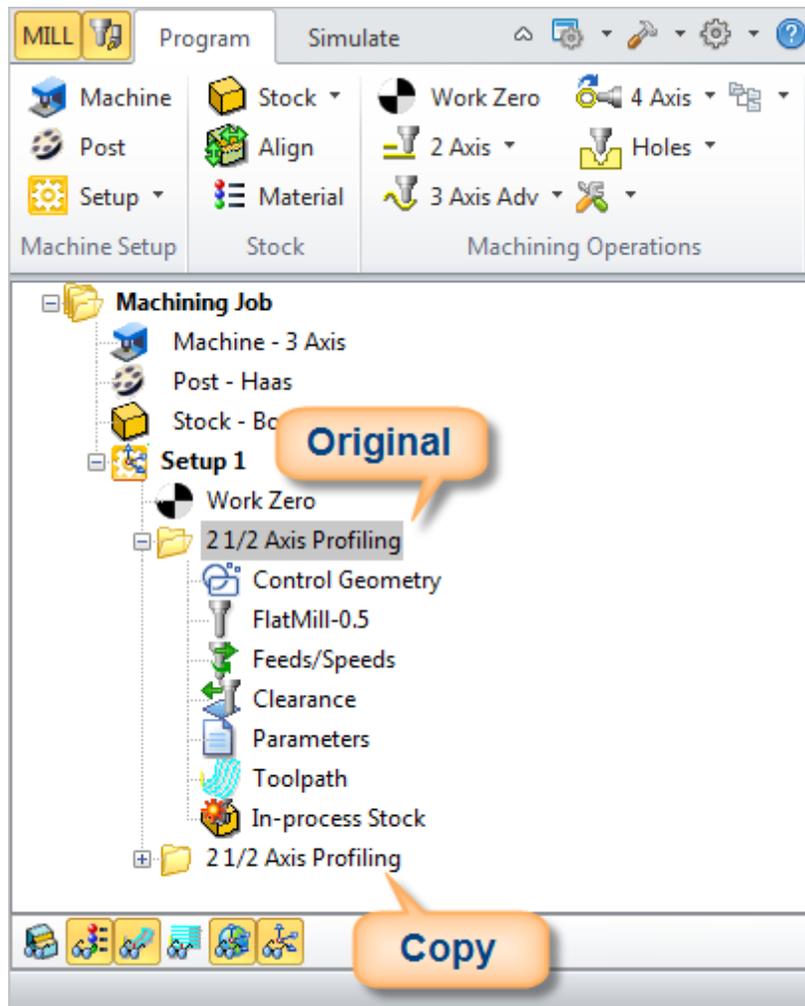
1. Switch to **Program** tab in the **Machining Browser**.
2. Select the **2 ½ Axis Profiling** operation we just created.
3. **Right-click** on the selected operation and select **Copy**.



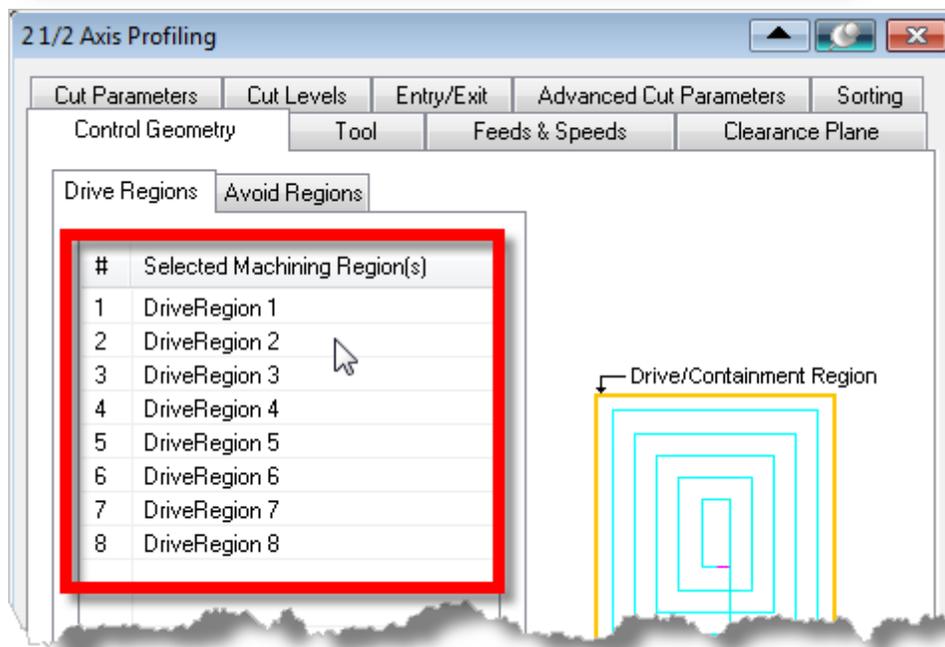
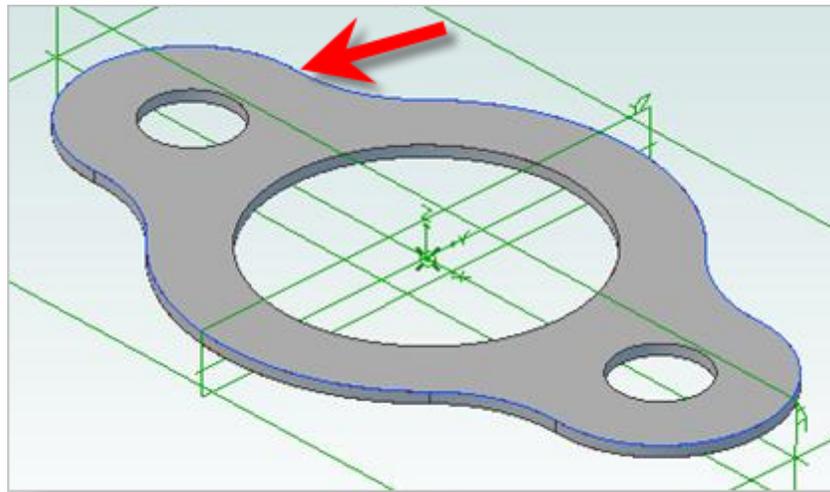
4. Now **Right-click** again and select **Paste**.



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

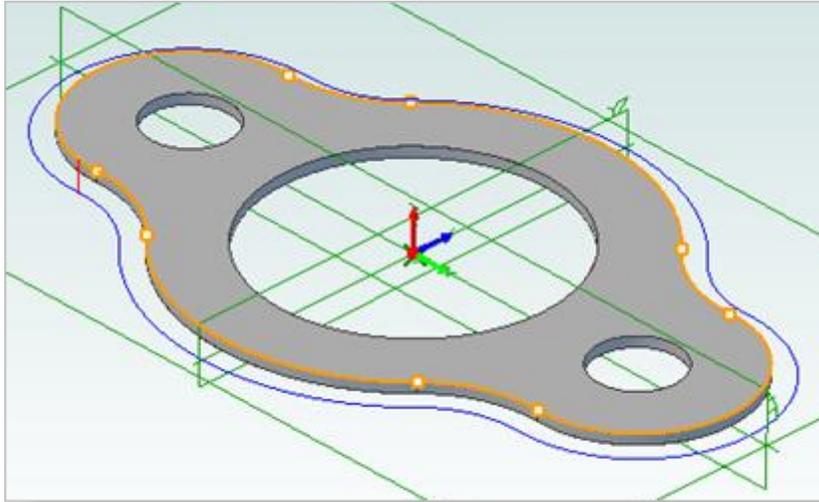


6. Now right-click on the second operation and pick **Edit** to adjust its parameters.
7. Pick **Remove All** under the **Control Geometry** tab.
8. Pick **Select Drive/Containment Regions**.
9. While pressing the <Shift> key, select the top outer surface edge and then select the icon to complete the selection. **Note:** This edge has eight (8) separate segments so be sure to select all of them as shown in the dialog below.

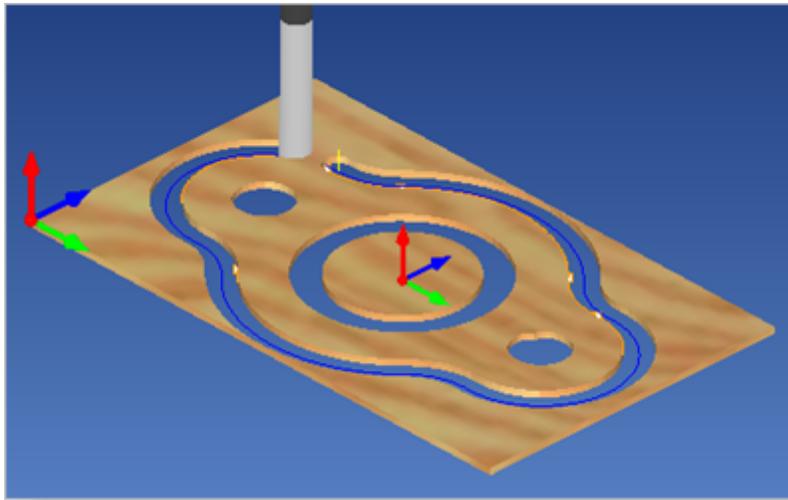


#	Selected Machining Region(s)
1	DriveRegion 1
2	DriveRegion 2
3	DriveRegion 3
4	DriveRegion 4
5	DriveRegion 5
6	DriveRegion 6
7	DriveRegion 7
8	DriveRegion 8

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, select the Simulation Window icon  and then pick Play.

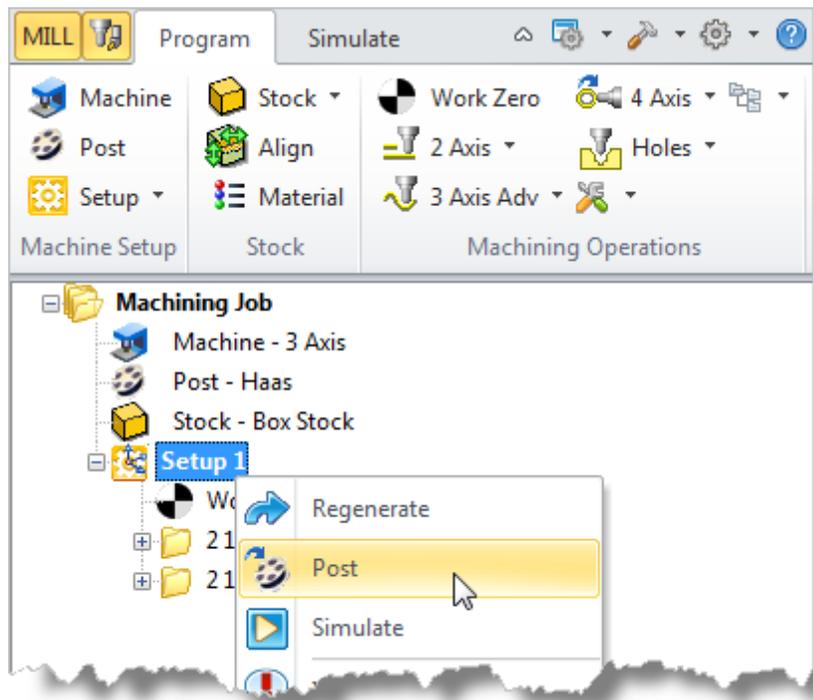


Now select the Simulation Window icon  again to close the Simulation Window.

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**.



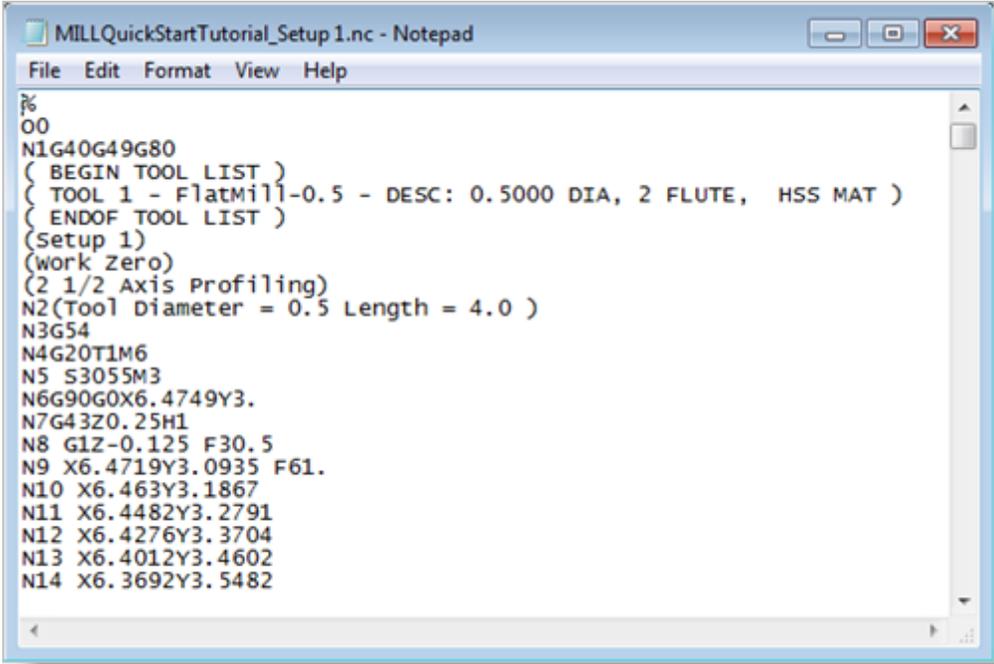
2. By default, the **Part** file name and the **Setup** name are appended for the G-code file name. Also by default, the posted G-code file is saved to the folder where the part file is located.

 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 )
( ENDOF 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

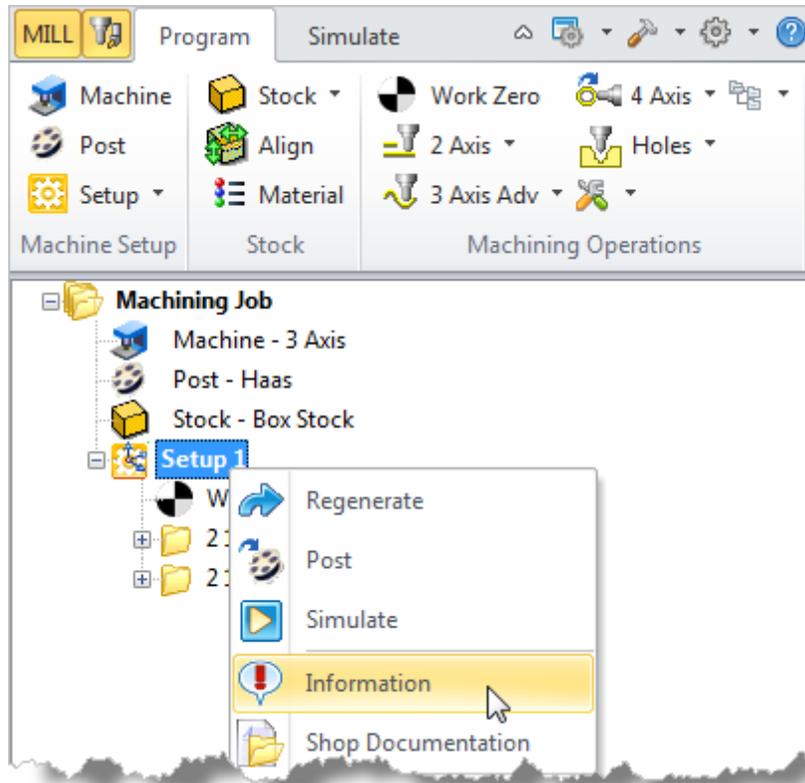
```

4. Now close Notepad.

Generating Reports

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.



Name	Status	Tool	Tool #	Cut Feed	# of GOTOs	Machine Time
Setup 1						
Work Zero	Clean	No Tool	-	0.0		
2 1/2 Axis Profiling	Clean	Flat Mill-0.5	1	61.00 in/min	311	0.41 min
2 1/2 Axis Profiling	Clean	Flat Mill-0.5	1	61.00 in/min	259	0.39 min
					Sub-total	0.79 min

This dialog provides an estimate of the machining time required for the operations in the [Setup](#).

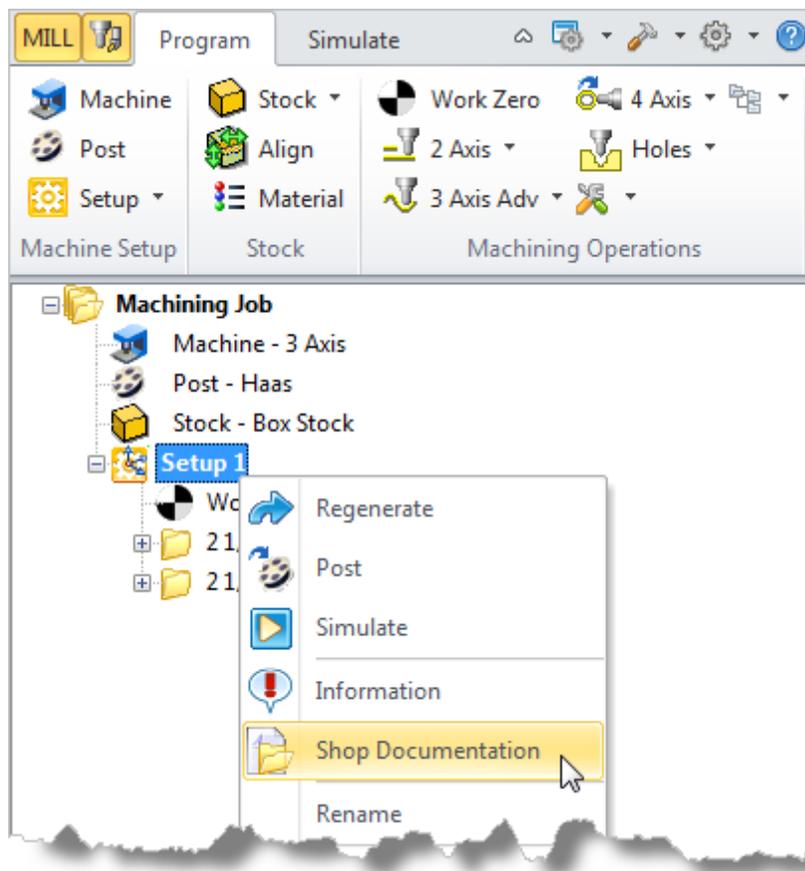
You can perform the same right-click sequence on the [Machining Job](#) to determine the estimated machining time for all [Setups](#).

4. Now pick [OK](#) to close the [Information](#) dialog.

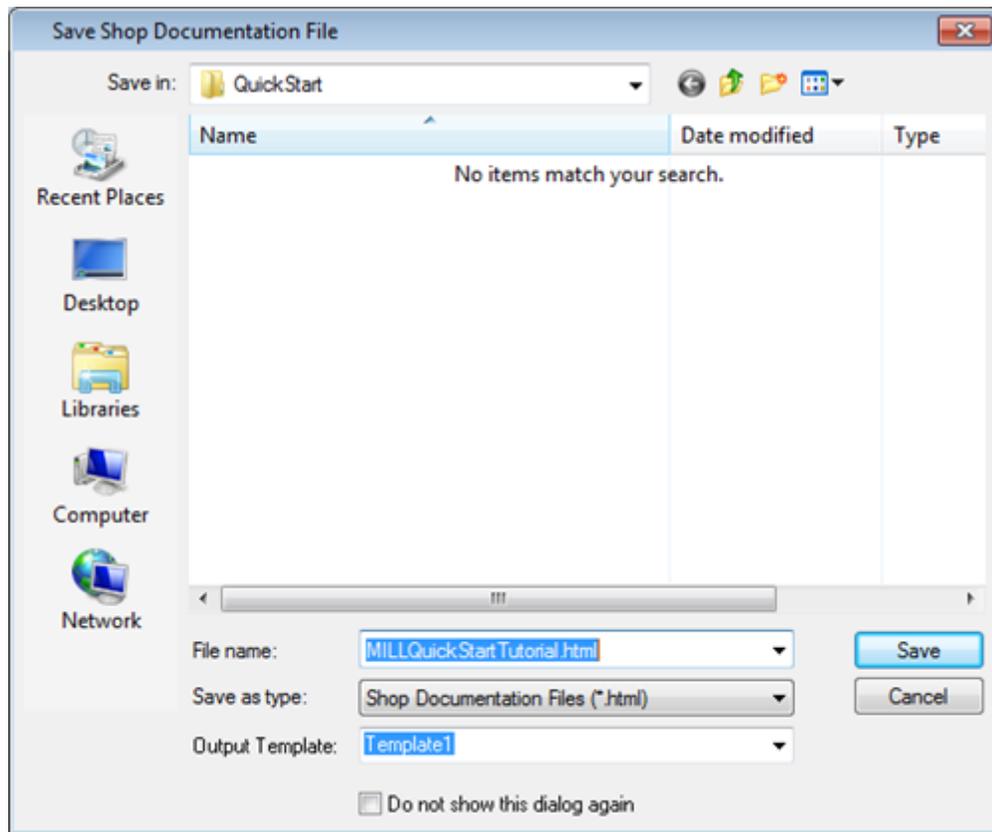
8.1 Generating Shop Documentation

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](#).

Your CAM Partner

N1G98G80G90G49G17
N2T1M6
N3S9000M3
N4G0 X1.584 Y0.

MecSoft

SETUP SHEET

File Name: C:\ProgramData\MecSoft Corporation\VisualCAM 2015 for Geomagic\QuickStart\Gasket_AD_PRT		
Date: 3/24/2015	Units: inch	Total Machining Time: 43 secs
Stock Dimensions: Length(X): 10.000; Width(Y): 6.000; Height(Z): 0.125		
No. of Ops: 3	Stock Material:	Number of Tools: 1

PART SETUP

TOOL LIST

No.	Name	Number	Type	Dimensions					Comments
				Radius	C-Radius	Taper	Length	Coolant	
1	FlatMill-0.5	1	Mill	0.250	0.000	0.000	4.000	None	

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

We have come to the end of the [Quick Start Guide](#) of the [VisualCAM 2015 for Geomagic MILL](#) module.

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](#) web site at www.mecsoft.com. This will help you get started with using [VisualCAM 2015 for Geomagic](#) .
- If you need additional help, or if you have any questions regarding [VisualCAM 2015 for Geomagic](#), you may contact us via e-mail at 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 for further details
- Please do continue to visit our home page to learn about the latest updates to [VisualCAM 2015 for Geomagic](#) and any other help material.

Index

- A -

About the MILL Module 4
Align Part and Stock 20

- C -

Clearance Geometry 38
Create Machining Operations 32
Create Stock Geometry 17
Create Tools 28
Cut Levels 41

- D -

Define the Machining Setup - Skip this section if in
Standard or Expert Configuration 17

- G -

Generating Reports 56

- M -

Machining Strategy 7
Machining the Outer Area 48
Main Programming Steps 7

- P -

Post Processing 53

- R -

Running VisualCAM 2015 for Geomagic 6

- S -

Select Cutting Tool 36
Select Machining Features/Regions 33
Select the Post Processor to use 14

Set Feeds and Speeds 37
Simulate Toolpath 45
Specify Cut Parameters 40
Specify Material 22

- T -

The VisualCAM 2015 for Geomagic Security Key 6

- U -

Useful Tips 5
Using this Guide and Associated Part Files 4

- W -

Where to go for more help 61