Profile-NEST Quick Start Guide

VisualCAM 2020

Published: January 2020

MecSoft Corpotation

© Copyright 1998-2020

VisualCAM Profile-NEST 2020 Quick Start

by MecSoft Corporation

User Notes:

Table of Contents

Quick Start 5

Resource Guide 8

About this Guide 9

1 Useful Tips ... 9
2 About Profile-NEST ... 10
3 Using this Guide ... 10
4 Watch the Video! ... 11

Getting Ready 12

1 Running VisualCAM ... 12
2 About the VisualCAM Display ... 12
3 Load the Profile-NEST Module ... 13
4 Load the Part Model ... 16
5 Machining Strategy ... 18
6 Main Programming Steps ... 19
7 Operational Sequence ... 19

The Machine Setup 21

1 Define the Machine Tool ... 21
2 Select the Post Processor ... 21
3 Define Sheet to Nest ... 24

The Stock 27

1 Create Stock Geometry ... 27
2 Align Part and Stock ... 29
3 Specify Material ... 31

The Profiles to Nest 34

1 Profile Part 1 ... 34

Profile 1 Control Geometry tab ... 36
Profile 1 Nesting Parameters tab ... 38
Profile 1 Tool tab ... 40
Profile 1 Feeds & Speeds tab ... 46
Profile 1 Clearance Plane tab ... 47
Profile 1 Cut Parameters tab ... 49
Profile 1 Cut Levels tab ... 51
Profile 1 Entry/Exit tab ... 52

© MecSoft Corporation

Profile 1 Advanced Cut Parameters tab ... 55
Profile 1 Cornering Parameters tab ... 56
Profile 1 Sorting tab ... 59
Profile 1 Generate ... 60
2 Profile Parts 2,3 & 4 ... 62
Copy & Paste Profile Operations ... 62
Edit Profile Operations ... 65
The Nested Sheets 70
1 Set Nesting Parameters ... 70
2 Reviewing the Nest ... 72
3 Editing the Nest ... 75
4 Simulating the Nest ... 78
5 Generate Reports ... 84
Nesting Reports ... 84
Machining Reports ... 85
Shop Documentation ... 87
6 Post G-Code ... 89
Where to go for more help 91
Index 93

Quick Start

VISUALCAM2020

Prefer Printed Documentation? Click Here!

Quick Start Guides for each VisualCAM module are available in both PDF and Video format. Refer to the following information to access these resources:

What's New

What's New in VisualCAD/CAM 2020

Watch the What's New in 2020 Webinar!

The Complete Video Play List

Here is a link to the complete 2020 Video Play List

How to Access the Quick Start Guide Documents

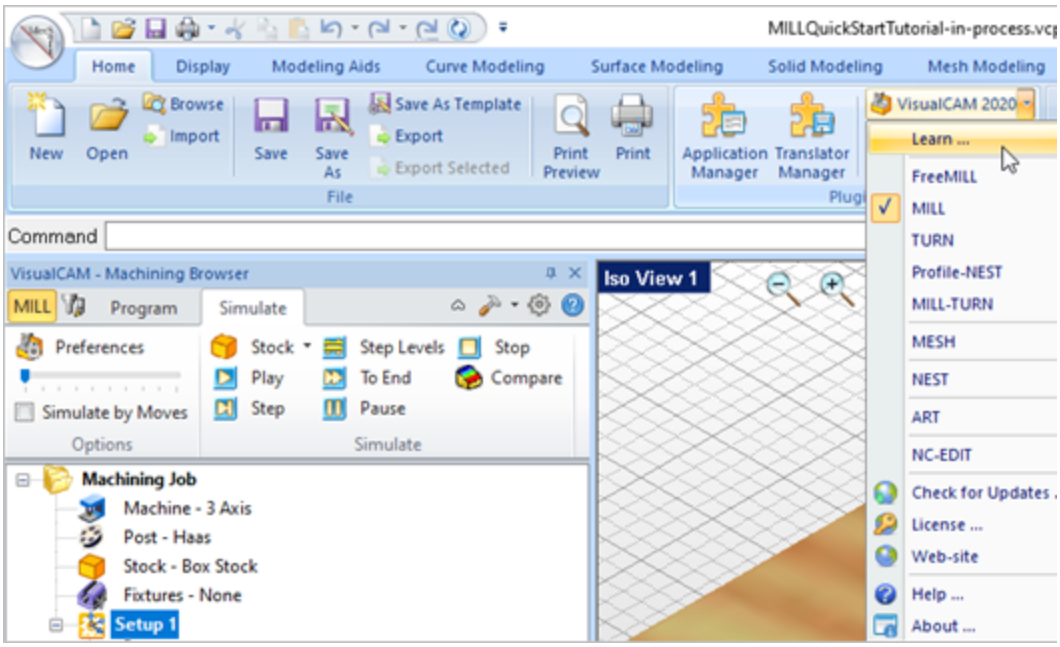
To help you quickly get started in working with each module, select one of the Help buttons located on the VisualCAM Learning Resources dialog.

You will find:

- Quick Start Guides
- What's New documents
- Online Help links

The Quick Start Guides will help you step through an example tutorial which will illustrate how to use the module. To access the Learning Resources dialog:

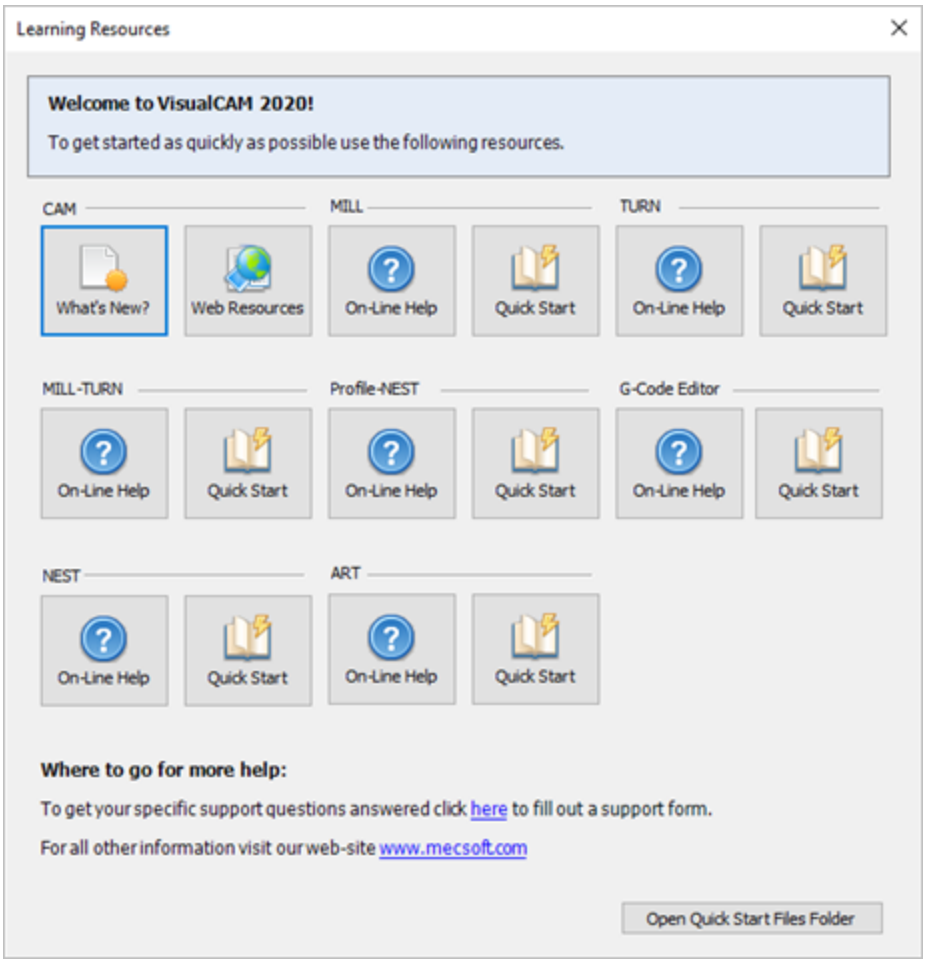
1. From the VisualCAD Home Ribbon Bar, drop down the Main menu and select Learn ...



To access the Learning Resources dilog in VisualCAM

2. Select a document from the Learning Resources dialog to get started using the module of your choice.

   You can also select the Open Quick Start Files Folder button located at the bottom of the dialog to open the Quick Start folder where the source files (start and completed versions) are located.

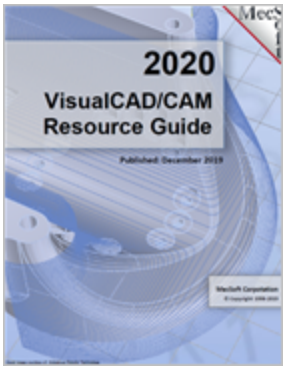


Learning Resources Dialog

Resource Guide

Download this PDF Guide for a list of the available VisualCAM Resources.

2020 VisualCAM Resource Guide



The VisualCAM 2020 Resource Guide

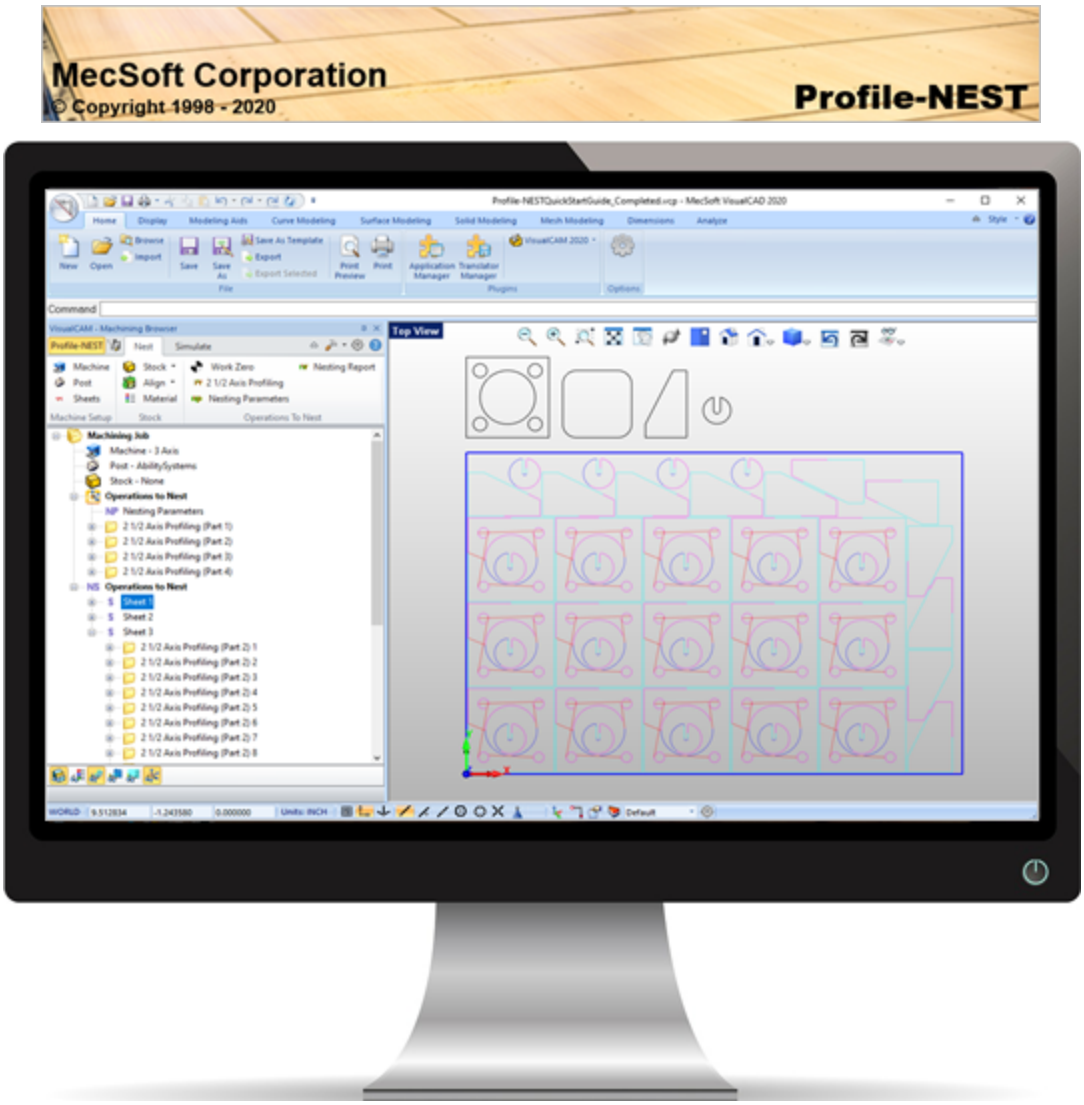
18 Pages

Lists PDF downloads and Online resources including Quick Start Guides, Reference Guides, Exercise Guides, Tutorials and More.

Click Here to download this guide!

About this Guide

VISUALCAM2020



3.1 Useful Tips

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

1. Copy the tutorial files to 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!

3.2 About Profile-NEST

The VisualCAM module offers fast sheet nesting of 2-1/2 Axis Profiling toolpath operations coupled with cutting simulation/verification capabilities running inside VisualCAD for programming CNC Mills. This integration allows for seamless generation of flat sheet nesting of profile toolpath and cut material simulation/verification within VisualCAD, for programming CNC routers.

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

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 curves, sketches, solids, surfaces and meshes. Then you can use the VisualCAM module to create 2-1/2 Axis Profile machining operations and associated nested sheets for CNC Routers and Mills. The toolpaths can then be simulated, verified, and post-processed to the controller of your choice.

3.3 Using this Guide

If you have installed VisualCADCAM 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 module.

This guide will illustrate profile machining and nesting of a few simple prismatic parts. The parts have regular and irregular outlines as well as internal cutouts that represent typical shapes that you may encounter. The guide uses 2D geometry to represent the part profiles and the sheet size.

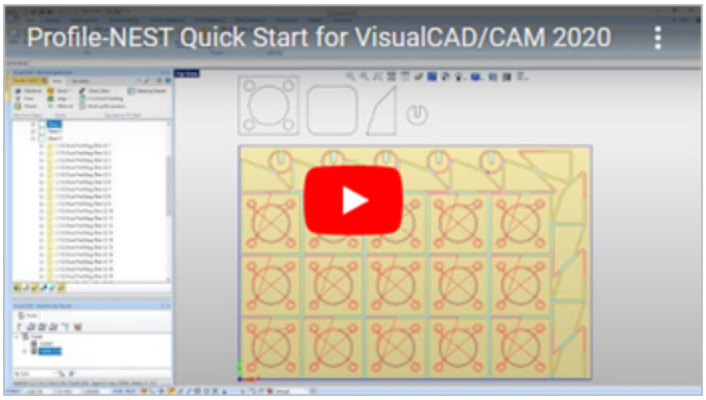
This guide has two associated VisualCAD files that you can find located in the QuickStart folder under the installation folder of VisualCAM. The first file is a completed file that contains all of the completed toolpaths and nested sheets and represents the file that you should end up with after working through the guide.

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 it to begin the guide.

© MecSoft Corporation

3.4 Watch the Video!

Want to see a video demonstration of this quick start guide? Just click on the image below.



Profile-NEST Quick Start for VisualCAD/CAM 2020

© MecSoft Corporation

Getting Ready

4.1 Running VisualCAM

Locate the VisualCADCAM 2020 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 VisualCADCAM 2020. (The name of this program group will usually be called VisualCADCAM 2020, unless you specified otherwise during setup.)

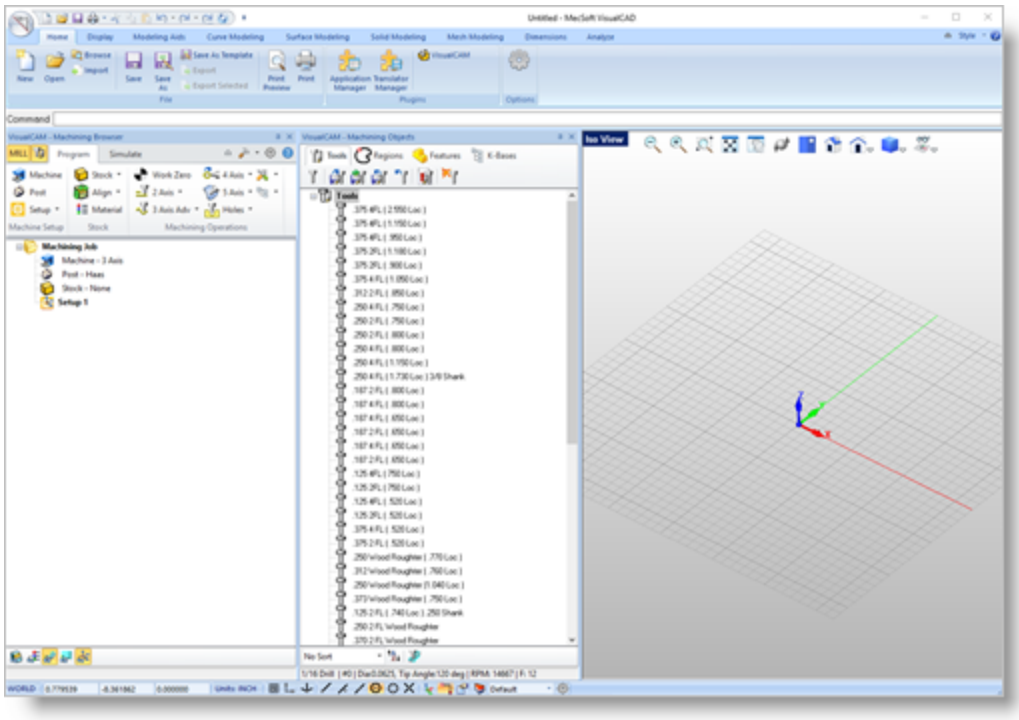
Once you locate the program group, select it and then select VisualCADCAM 2020 to launch the application.

If the installation was successful, upon launching of VisualCADCAM 2020 you should observe a menu entry called VisualCAM 2020 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.

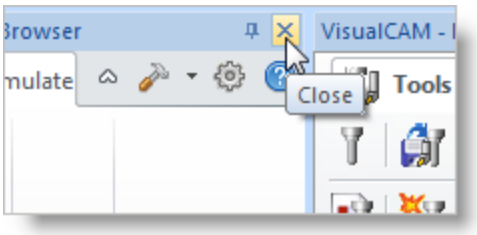
4.2 About the VisualCAM Display

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 this.

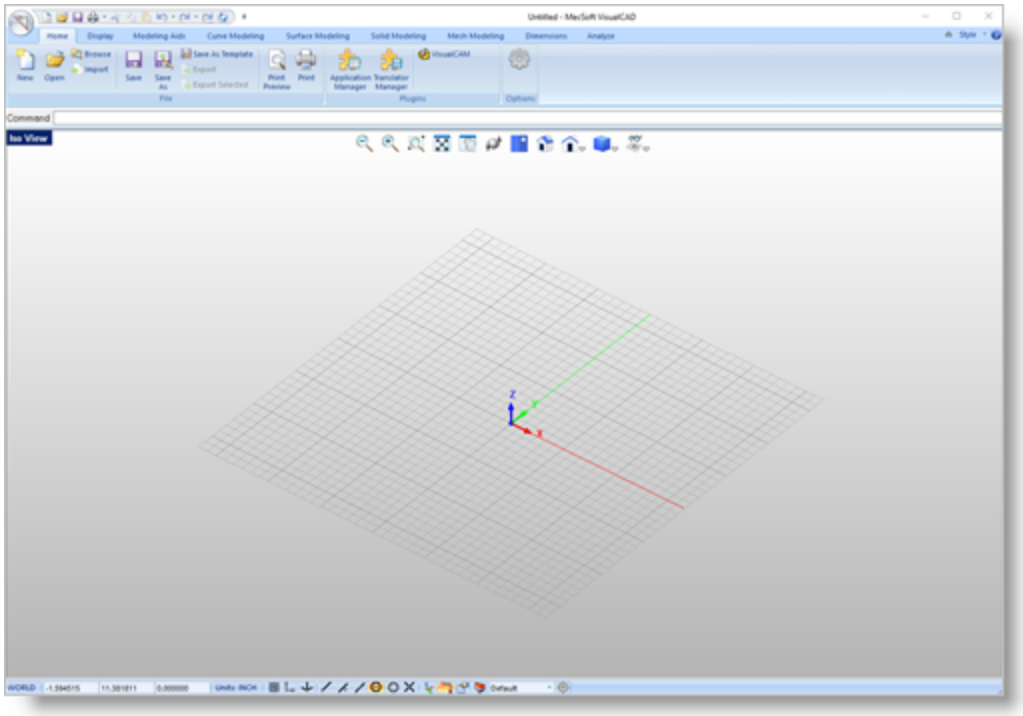


These windows on the left belong to plug-in modules that are currently loaded. For now, let's close all of them.

© MecSoft Corporation



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

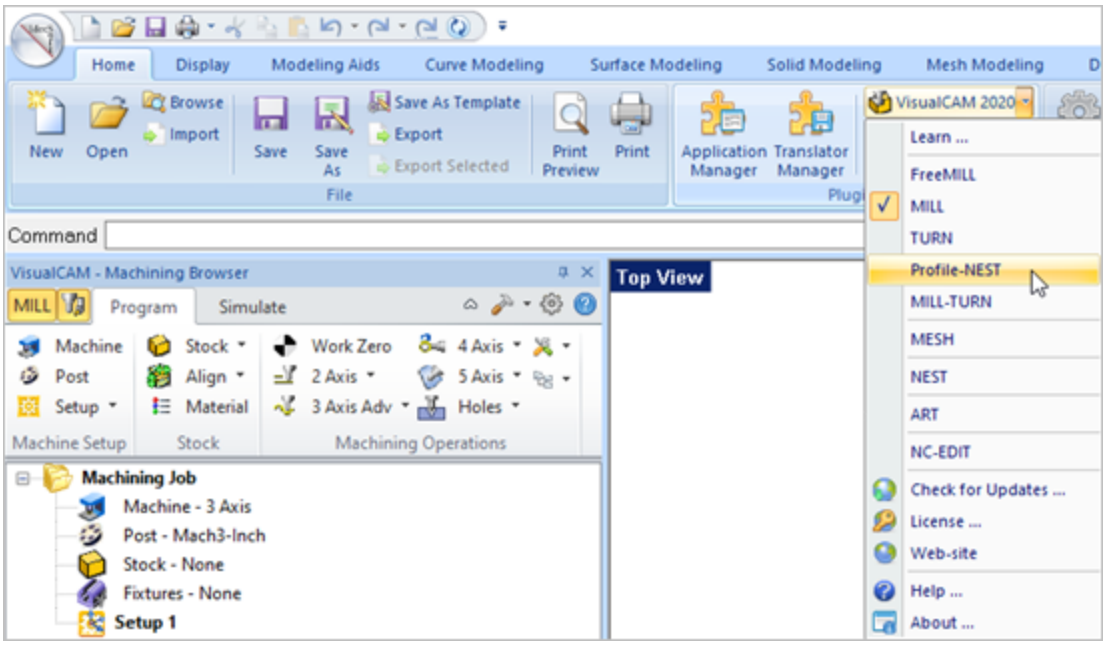


4.3 Load the Profile-NEST Module

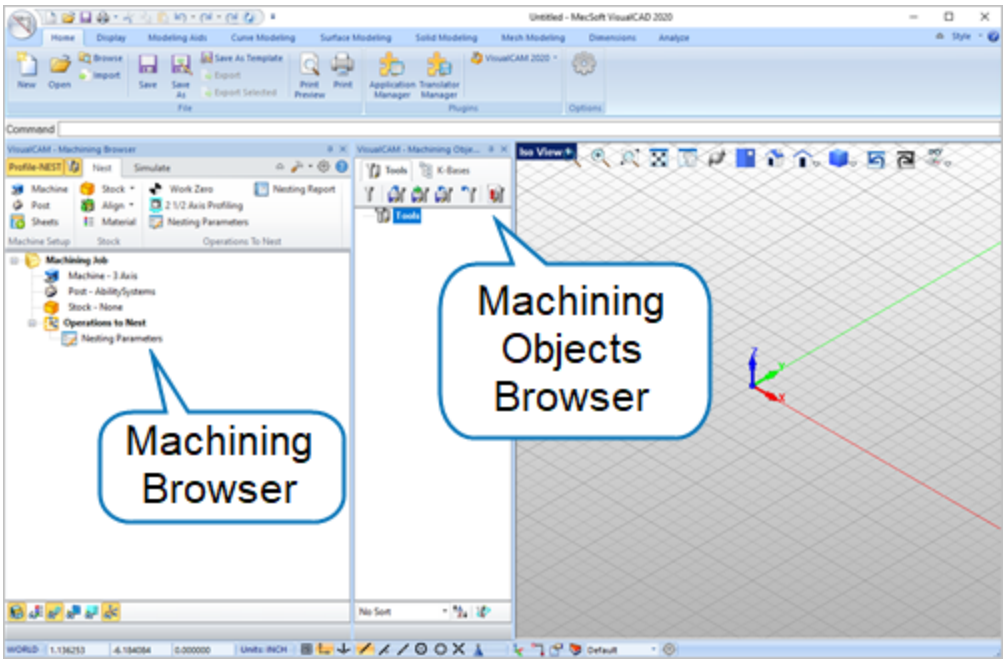
Now, let's begin by launching the VisualCAM module.

1. From the Plugins pane of VisualCAD's Home Ribbon Bar, you will see the VisualCAM 2020 main menu item.
2. Drop-down the menu and pick Profile-NEST to load the module.

© MecSoft Corporation

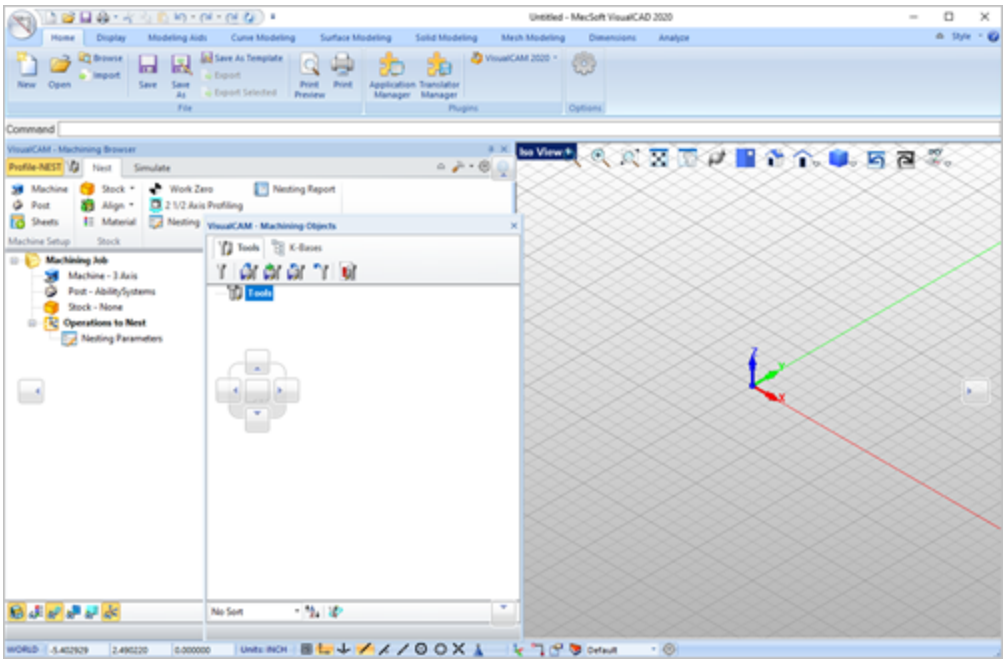


3. Docked on the left you will see the Machining Browser and the Machining Objects Browser. When you first run VisualCAM, these two browsers my be docked side by side. However, you can move them anywhere on the screen that feels comfortable for you.



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.

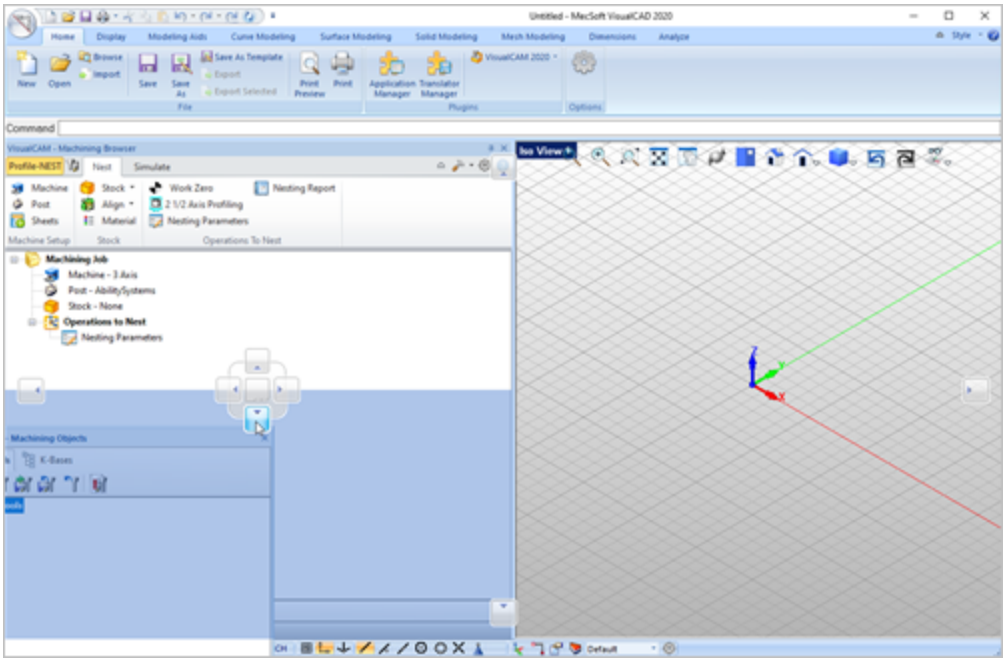
© MecSoft Corporation



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.

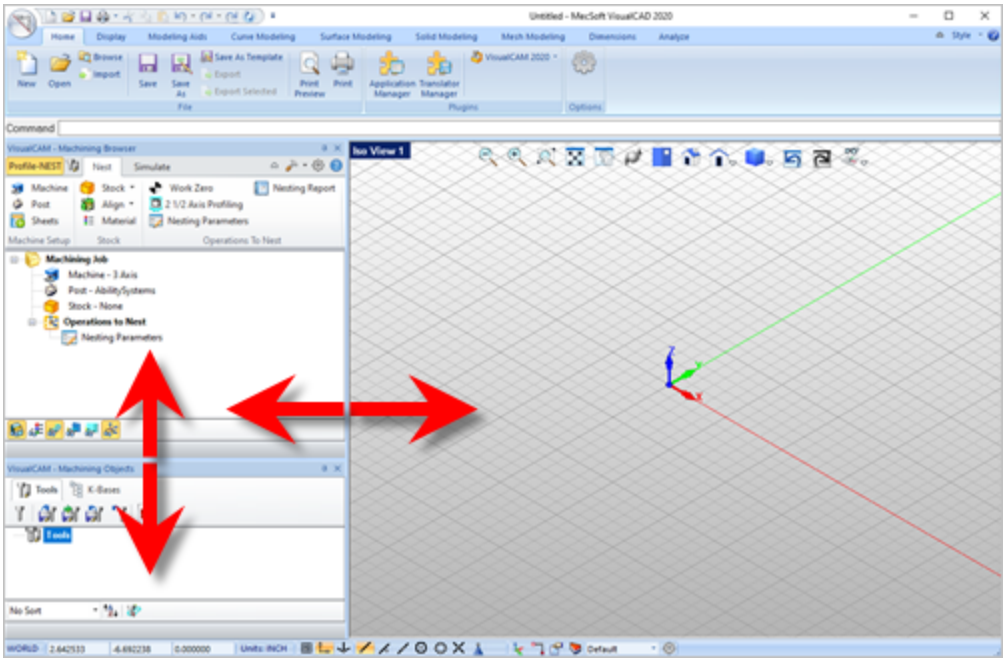
5. We'll drag the Machining Objects Browser over the base of the Machining Browser until the cursor activates the bottom directional button.

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



© MecSoft Corporation

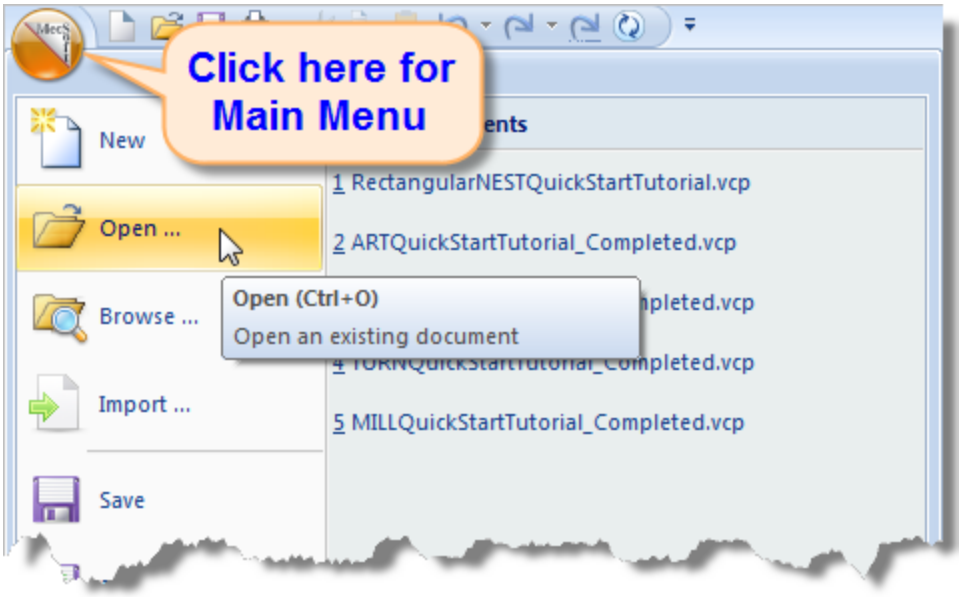
6. You can also re-size the height and width of each browser making sure that all of the command icons and menus are easily accessible.



4.4 Load 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.

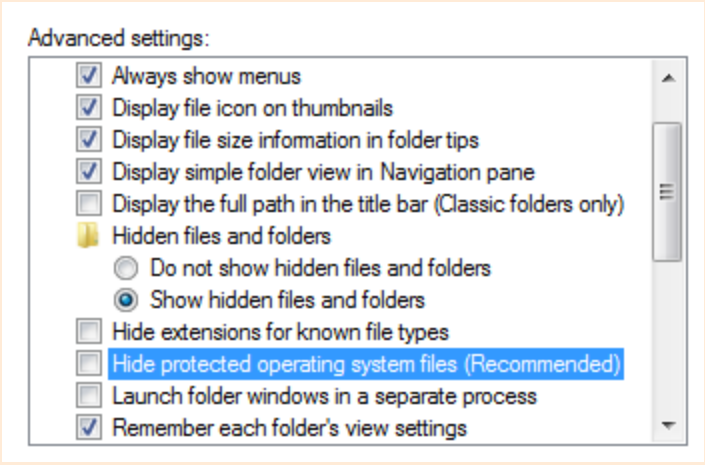


© MecSoft Corporation

2. From the Open dialog box, select the Profile-NESTQuickStartGuide.vcp file from the C:\ProgramData\MecSoft Corporation\VisualCAM 2020\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.

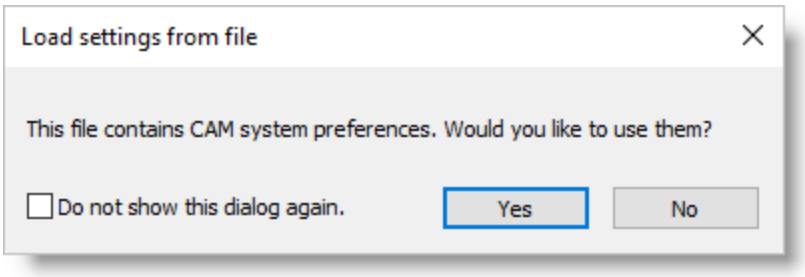
   By default, the ProgramData folder is "hidden" from view. Here are the steps to Show hidden files and folders:

   1. For Windows7/8 users: Go to Control Panel > Appearance and Personalization > Folder Options.
      For Windows10 users: Go to Control Panel > Appearance and Personalization > File Explorer Options.
   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)

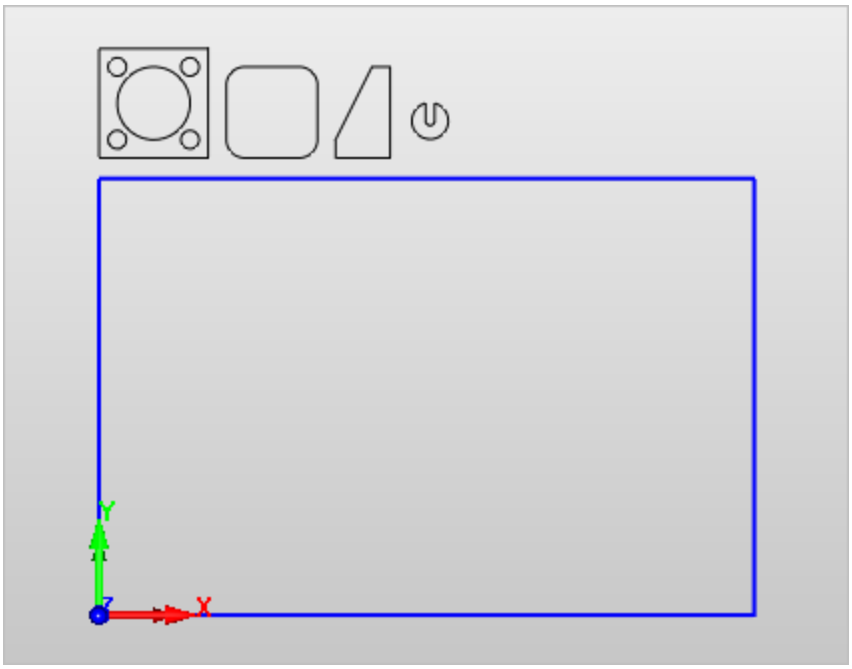
      

   3. Click Apply and OK.

When the Load Settings from File dialog appears, pick No for this file. In the future you may have older files whose CAM System Preferences you wish to use so leave the box *Do not display dialog again* unchecked for now.

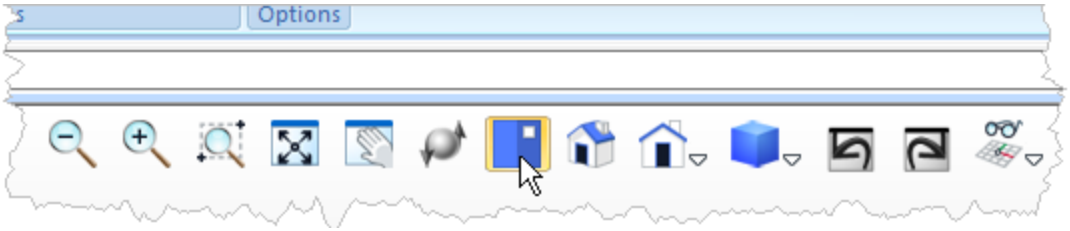


The part appears as shown below



Profile-NESTQuickStartGuide.vcp

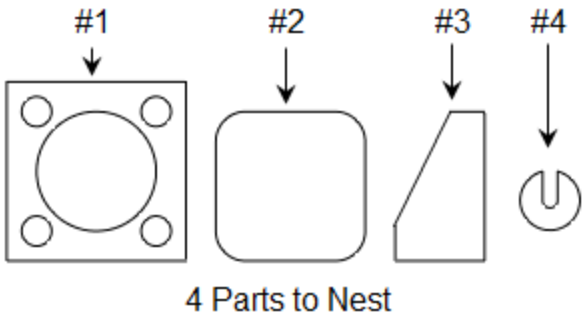
3. From the View toolbar, select the Top View to work in.



4.5 Machining Strategy

We will program 2 Axis Profiling toolpaths and nest them within a 36" x 24" x 1/4" sheet of wood or MDF stock material. There will be 4 part profiles, each with it's own 2 Axis Profiling operation and nesting parameters. The nested sheets will be created automatically based on sheet parameters and nesting parameters.

The 4 parts that we will nest are shown below.



4 Parts to Nest

© MecSoft Corporation

We will create and use a 1/4" flat end mill for all machining operations. We will also assume that the stock sheet will be held to the machine table or the spoil board using either double-sided tape or a vacuum table requiring no clamps or fixtures.

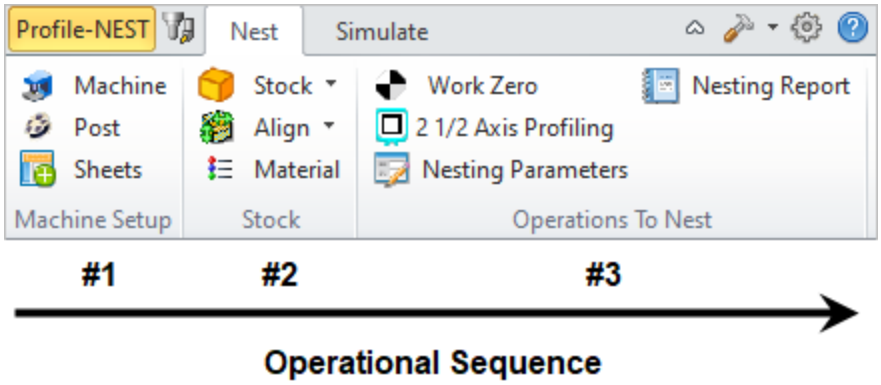
4.6 Main Programming Steps

The following steps will be followed in programming and nesting these parts. Some of these steps will have to be performed just once and others may have to be repeated to complete the process.

1. Define the Machine and Post-processor to use.
2. Define the Machining Setup including Stock Geometry and Material.
3. Create and Select a Tool to use for machining.
4. Create the profile Machining Operations including the Feeds and Speeds, the Clearance Plane and other Cutting and Nesting parameters.
5. Generate and Nest the profile toolpaths.
6. Simulate the toolpaths.
7. Post Process the toolpaths.
8. Generate Shop Documentation.

4.7 Operational Sequence

You will notice that the Nest tab of the Machining Browser is divided into groups of commands and menus. We will be following the Nest tab from left to right in the following order:



1. From the Machine Setup section we will be defining the Machine, Post-Processor and Sheet definition.
2. From the Stock section we will be defining our Sheet Stock, setting our Stock Alignment and setting the Stock Material.
3. From the Operations to Nest section we will not be using a Work Zero. We will proceed directly to setting our Nesting Parameters and then define our 2 Axis Profiling Operations

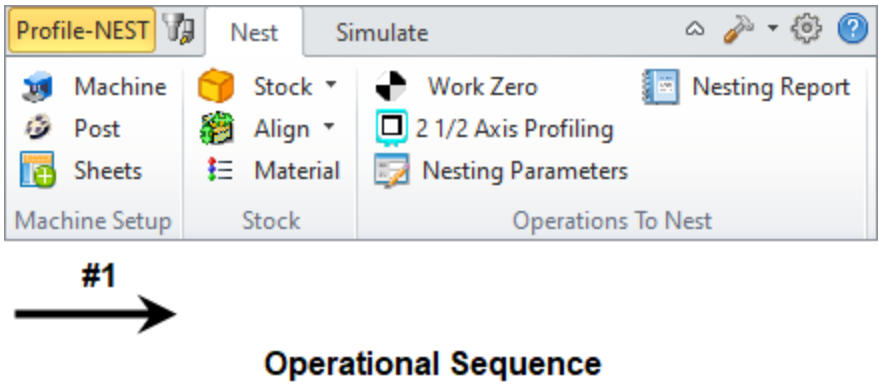
© MecSoft Corporation

that will be included in our nest. Once we are satisfied with the nest we will come back and use the Nesting Report function.

© MecSoft Corporation

The Machine Setup

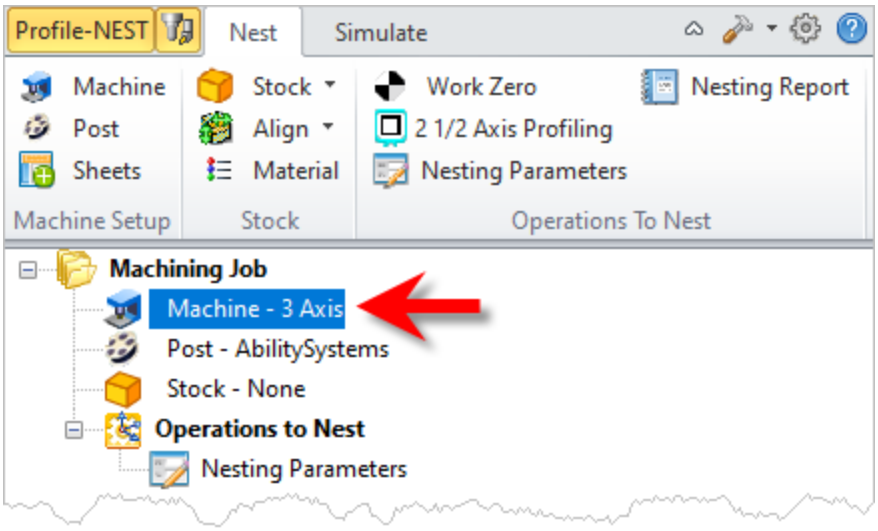
We will begin with the Machine Setup section of the Nest tab of the Machining Browser.



5.1 Define the Machine Tool

In the Profile-NEST module the Machine is defined automatically and is set to 3 Axis. This definition is used for all 2 Axis and 3 Axis machining in VisualCAM.

1. The Machining Job tree lists the Machine definition as 3 Axis. This is correct for all 2 Axis machining and nesting operations.

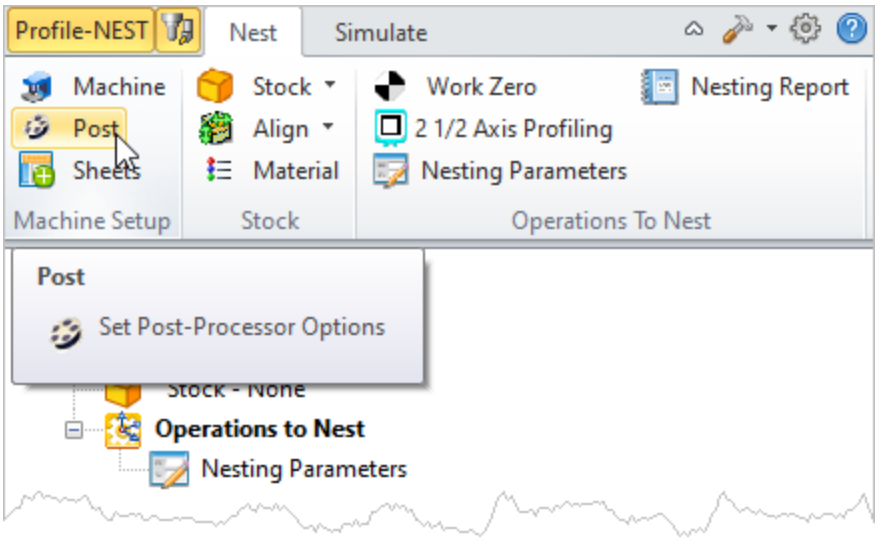


5.2 Select the Post Processor

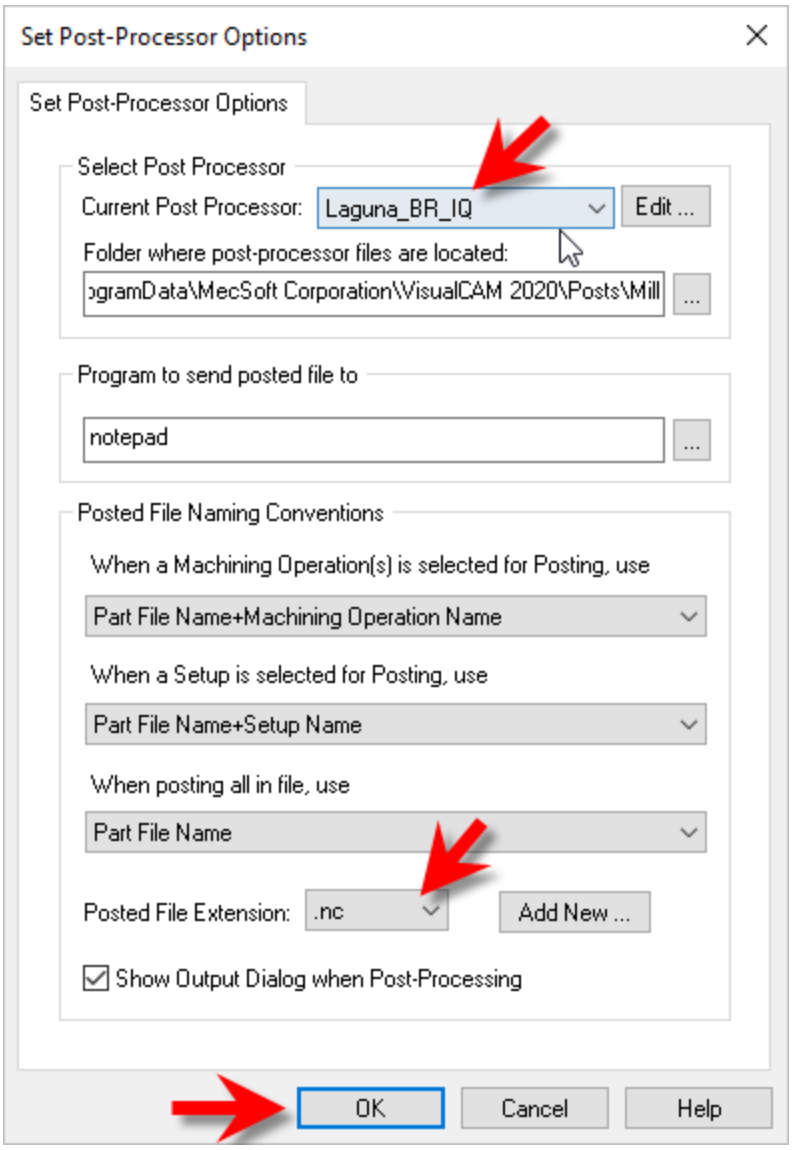
Next, we'll define the Post Processor.

1. From the Profile-NEST tab select Post to display the dialog.

© MecSoft Corporation



2. For the Current Post Processor, select Laguna_BR-IQ 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.

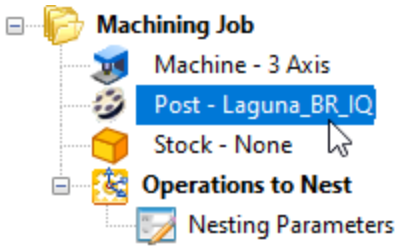


By default, post processor files are located under C:\ProgramData\MecSoft Corporation\VisualCAM 2020\Posts\MILL\

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

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

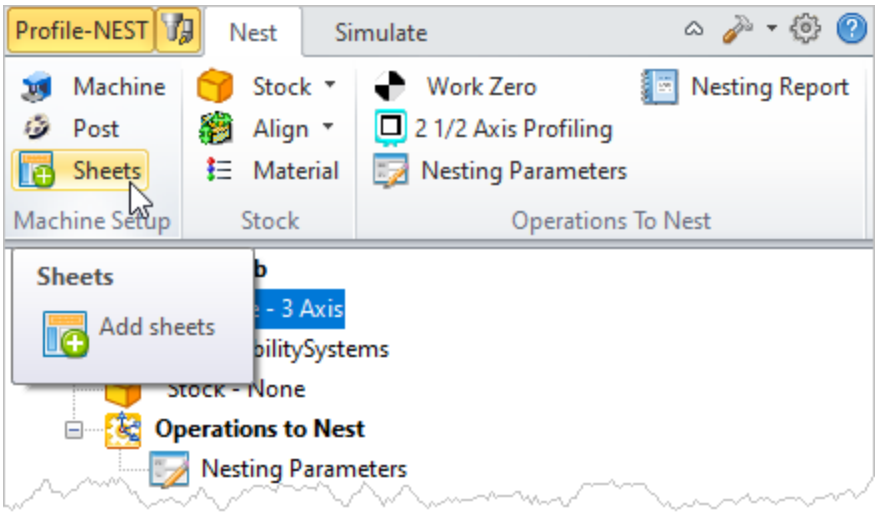
© MecSoft Corporation



5.3 Define Sheet to Nest

In this step we'll define the sheet to use in the nesting process.

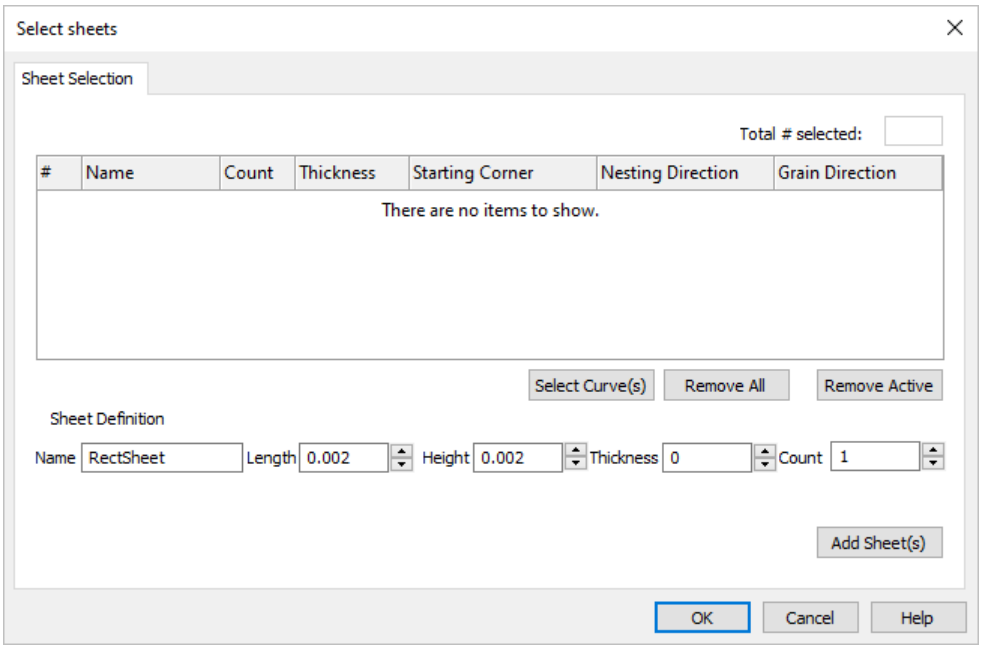
1. From the Program tab select Sheets to display the dialog.



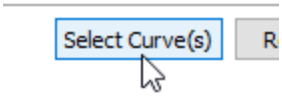
This dialog is used to define one or more sheets to use for nesting. Sheets can be defined from existing curves defining the sheet boundary or by using the Sheet Definition portion of the dialog.

We have sheet curve already defined in our drawing.

© MecSoft Corporation

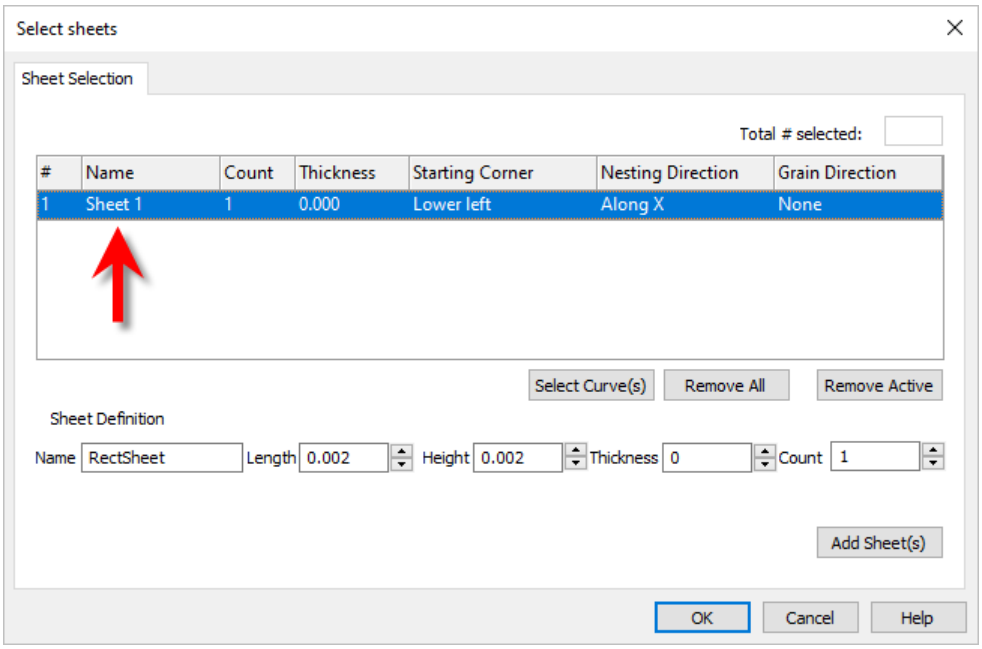


2. From the dialog pick the Select Curve(s) button.

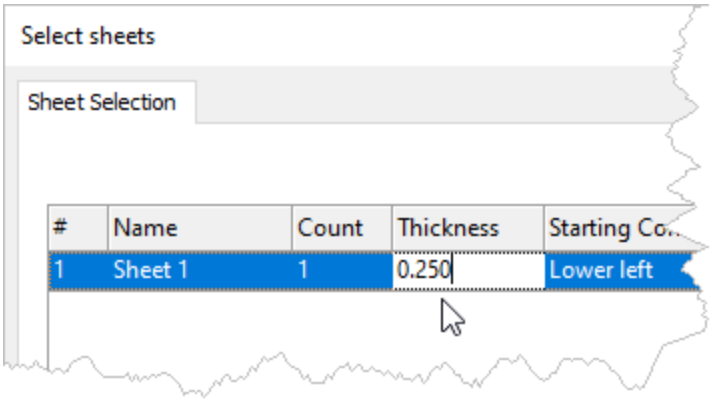


3. The dialog will minimize and prompt you to select objects. Window select the large rectangle shown on the drawing and then right-click or press the <Enter> key to complete the selection.

   The dialog reappears with Sheet 1 listed

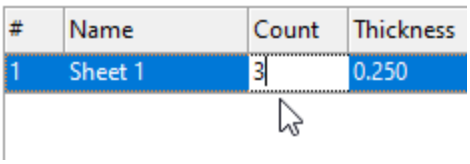


4. Now left-click on the Thickness column and change the sheet thickness to 0.250 to match our stock thickness and then pick <Enter>.



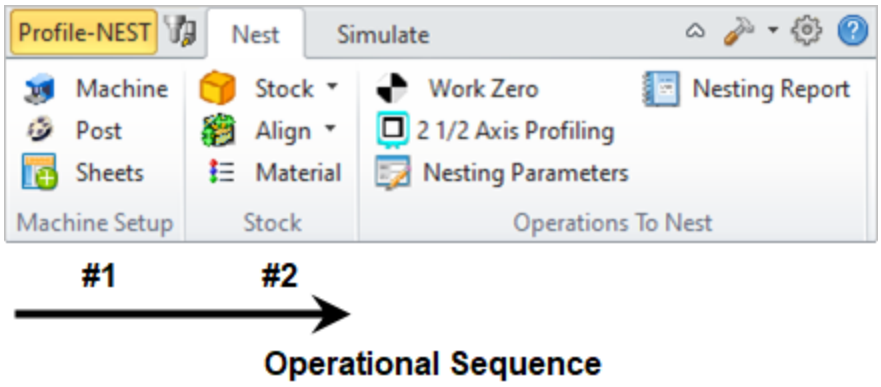
The sheet geometry is also highlighted on the drawing:

5. Now set the sheet Count to 3. We will have three sheets in our nest and then pick the OK button to accept the sheet definition and close the dialog.



The Stock

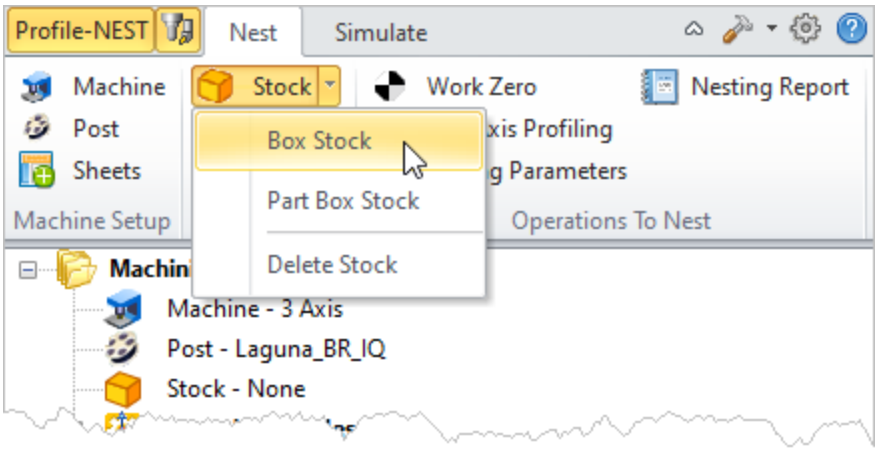
Next we will proceed to the Stock section of the Nest tab of the Machining Browser.



6.1 Create Stock Geometry

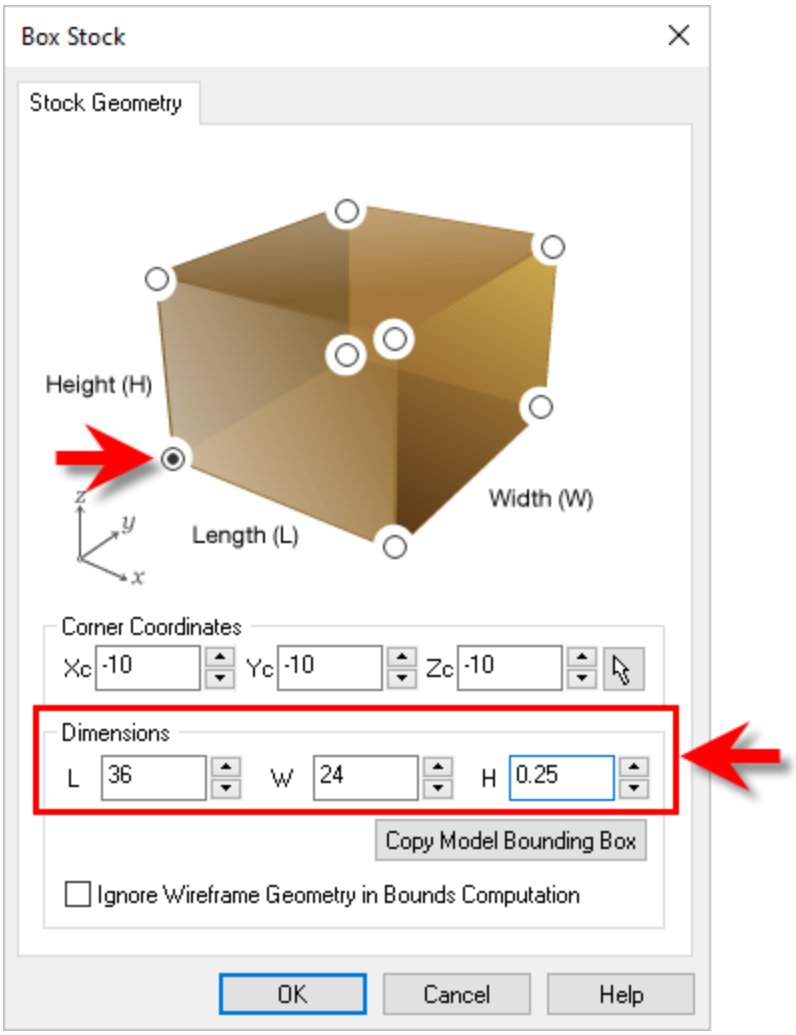
In this step we'll define the stock from which to cut the nested profiles.

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 36, Width W to 24 and Height H to 0.25.
Note that the stock dimensions you enter are measured from the corner of the bounding box selected in this dialog.

© MecSoft Corporation

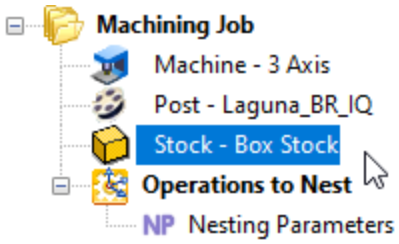


The dimensions of the stock are interpreted in relation to the corner selected in the dialog box above. For example if the corner of the box is selected as the Bottom South West corner (as shown in the dialog above), the Length (L) is interpreted to be along the +X axis, the Width (W) along the +Y axis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

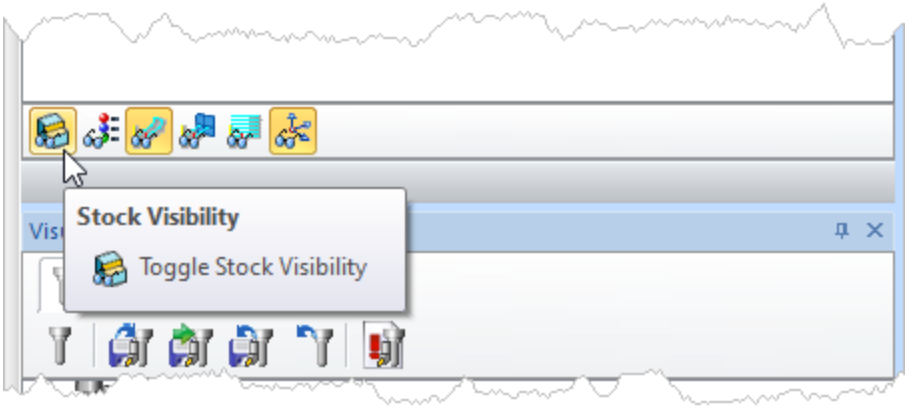
The direction of the dimensions will change depending on the corner selected. For example if the Top South West corner is selected, then the Height (H) is interpreted to be along the –Z axis and so the stock will extend below the corner.

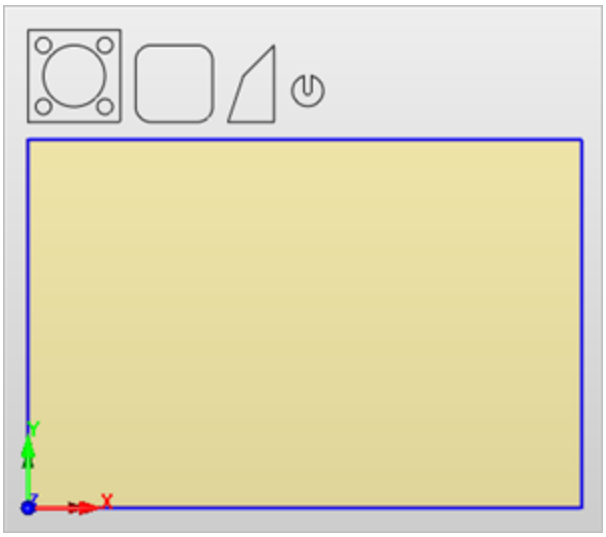
3. Pick OK and notice that the Stock type now appears under Machining Job in the Machining Browser.

© MecSoft Corporation



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





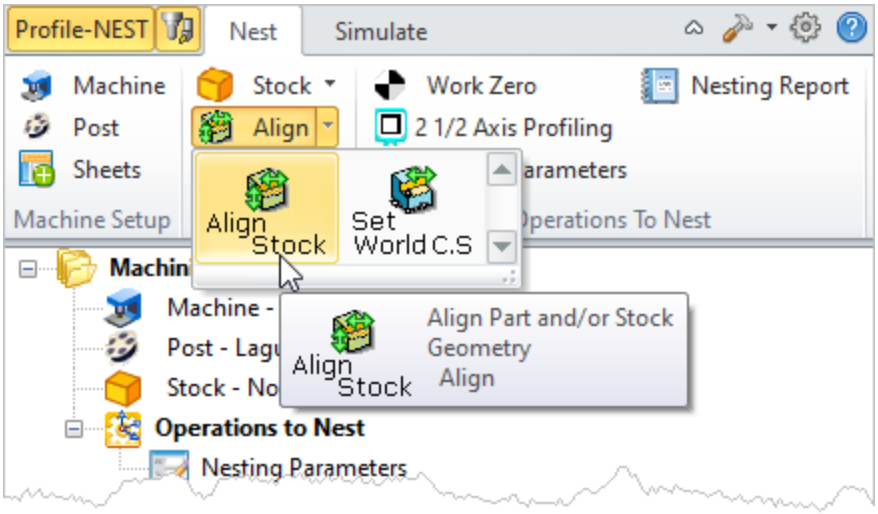
6.2 Align Part and Stock

Once the stock model is created you can move it in alignment with the sheet geometry.

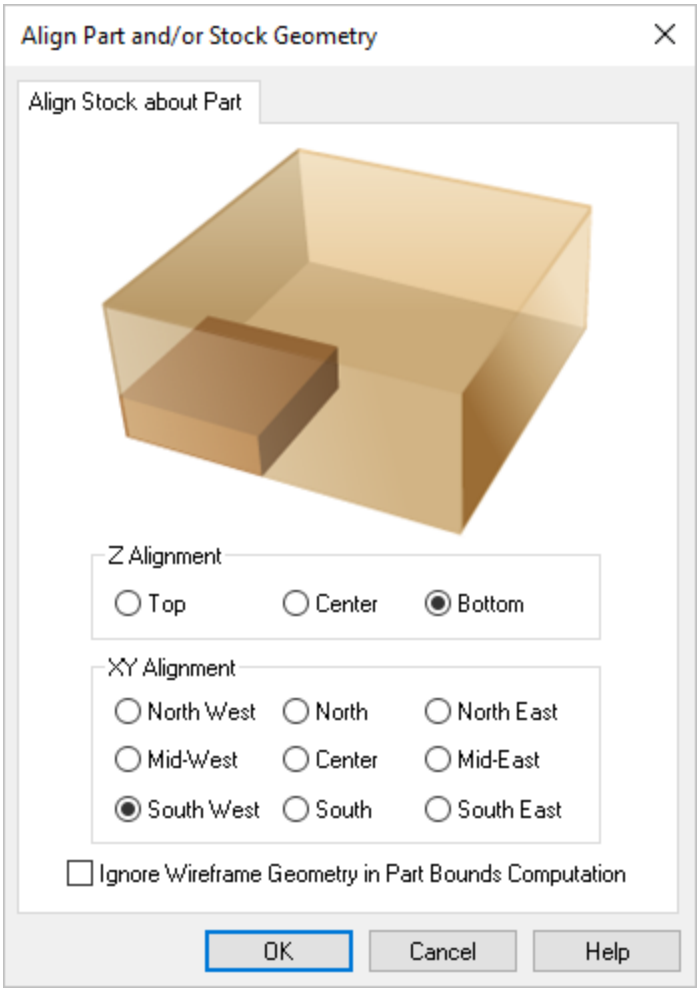
1. First from the View toolbar select the view.

© MecSoft Corporation

2. From the Program tab select Align and then Align Stock from the menu to display the dialog. Notice that we are working our way from left to right in the Program tab.

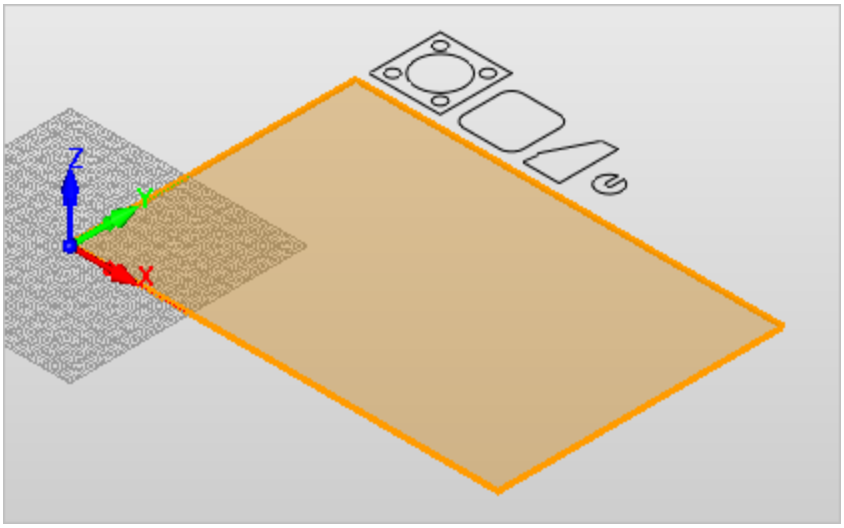


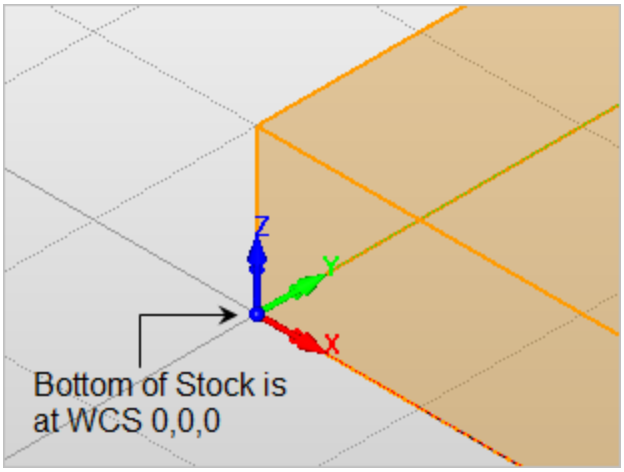
3. For Z Alignment select Bottom and for XY Alignment select Center and then pick OK.



© MecSoft Corporation

The bottom of the stock is now aligned to the bottom right corner of the stock sheet geometry with the bottom corner of the stock being located at 0,0,0 coordinate of the WCS (Work Coordinate System).

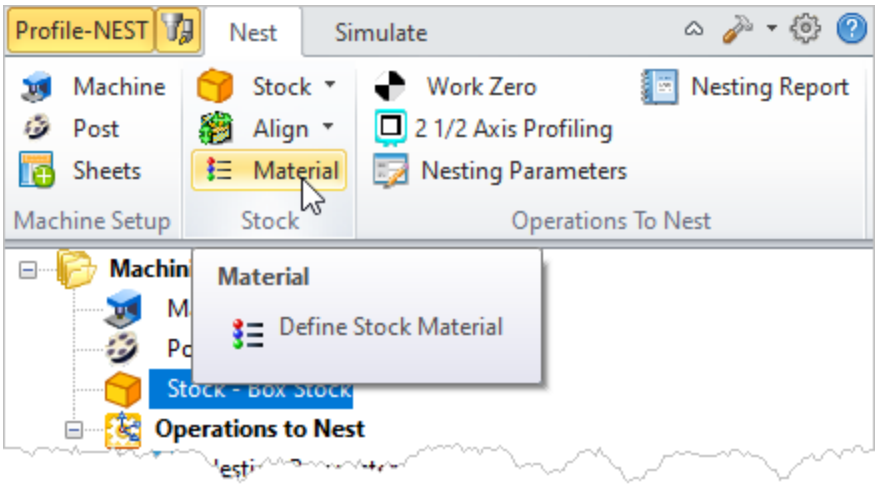




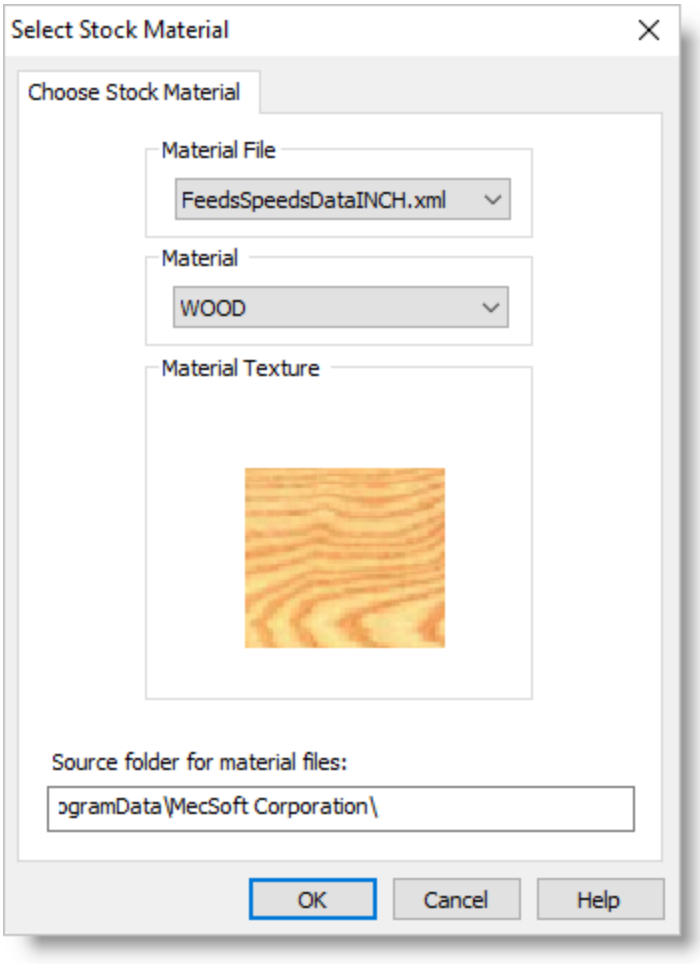
6.3 Specify Material

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

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

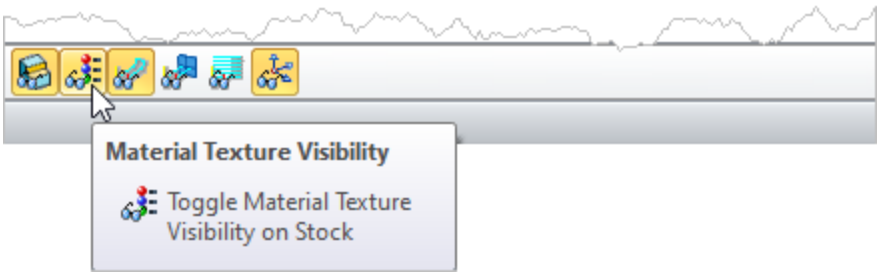


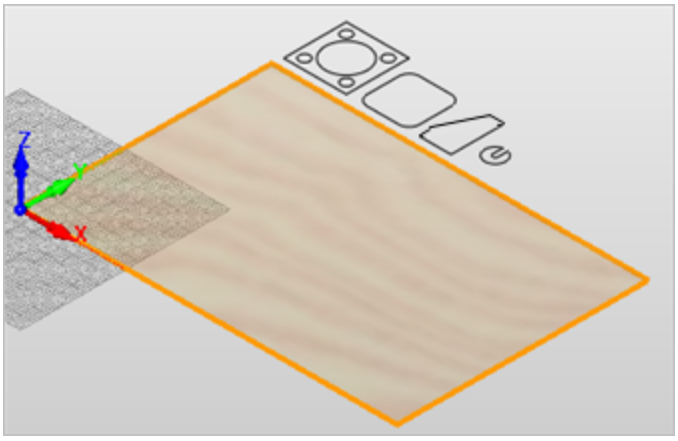
2. For Material, select Wood from the list of available materials and then pick OK.



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.

© MecSoft Corporation

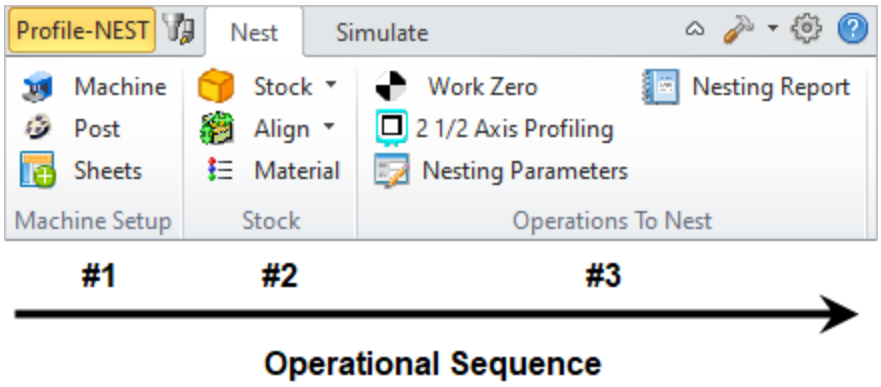




© MecSoft Corporation

The Profiles to Nest

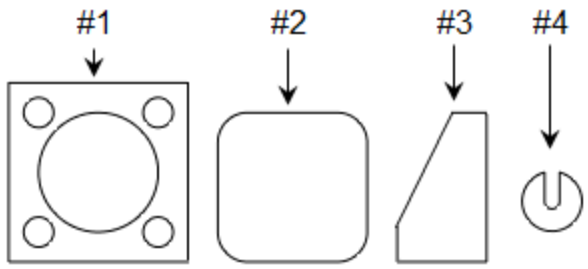
Next we will proceed to the Operations to Nest section of the Nest tab of the Machining Browser.



7.1 Profile Part 1

In Prole-NEST toolpaths are nested NOT part geometry. So every part in your Profile-NEST should have its own 2 Axis Profile toolpath operation. This allows each toolpath to be individually controlled during the nesting process. If you want multiple parts to always be nested as a group, then you can include them in the same 2 Axis Profile operation.

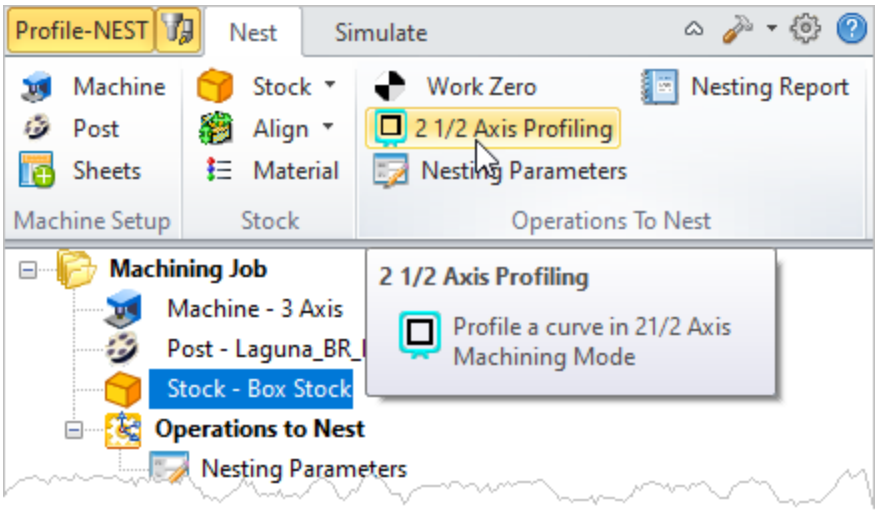
In this step we will create a 2 Axis Profile toolpath operation for Part 1 shown below.



4 Parts to Nest

1. From the Nest tab in the Machining Browser select 2-1/2 Axis Profile to display the toolpath operation dialog.

© MecSoft Corporation

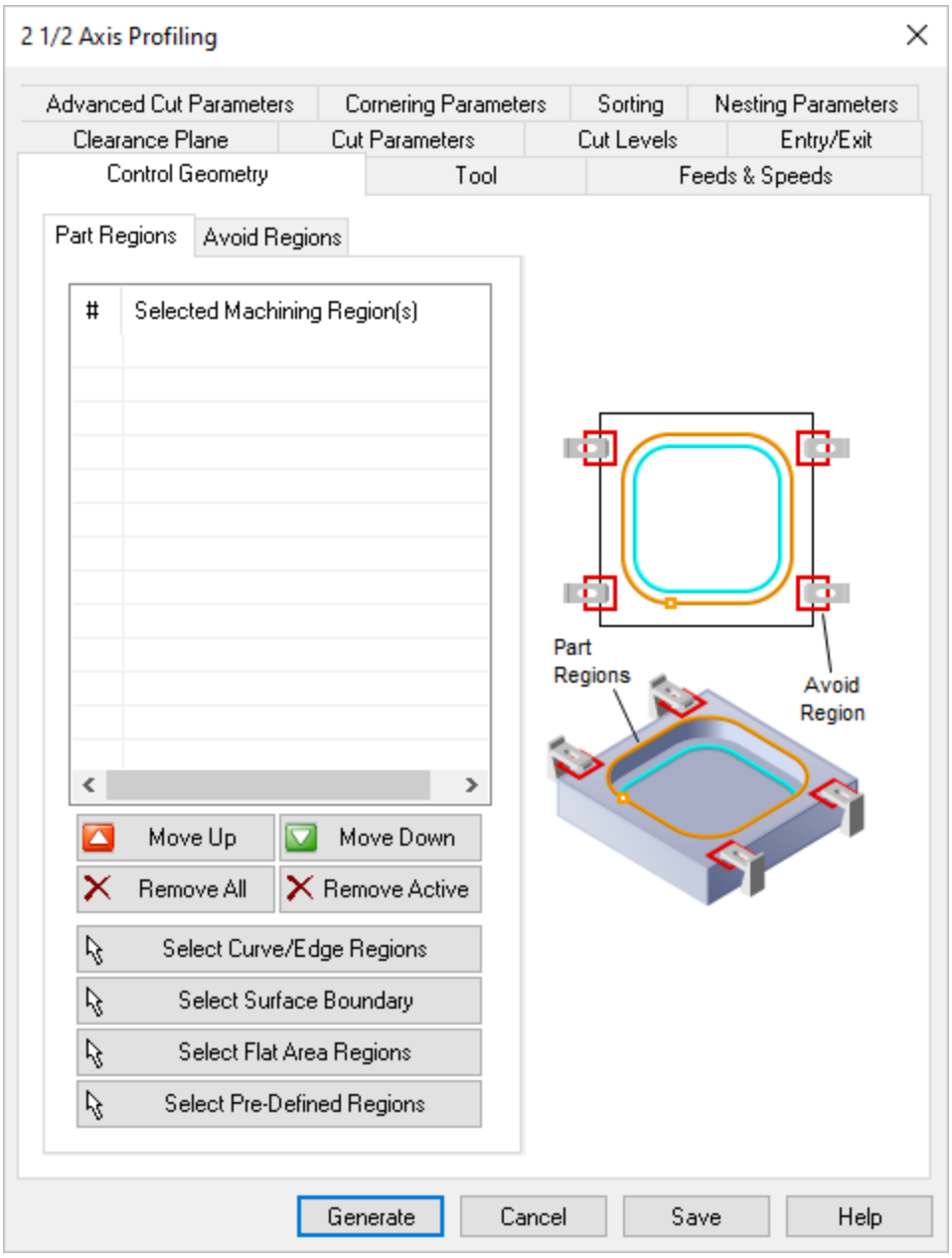


2. The 2-1/2 Axis Profile operation dialog will display with the Control Geometry selected by default.

   If you are familiar with using the MILL module you will recognize this dialog. If you are a new user, each tab in this dialog controls a different set of parameters. All parameters in this dialog relate to the 2 Axis Profile toolpath operation that you are creating right now.

   The good news is that EVERY parameter has a default selection so we will only have to make a few adjustments.

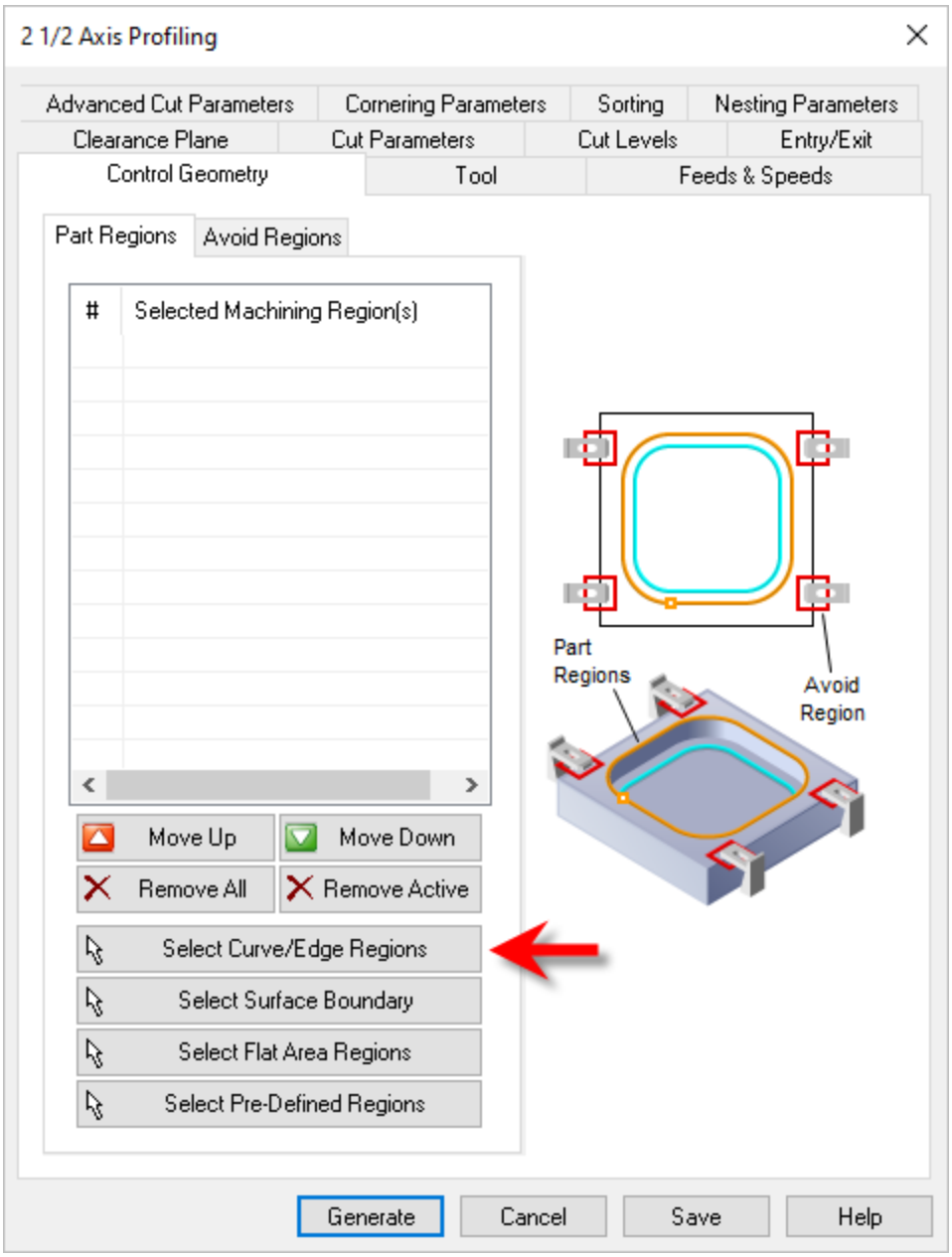
© MecSoft Corporation



7.1.1 Profile 1 Control Geometry tab

The Control Geometry tab is where you define the part geometry to profile.

1. By default the last geometry selected populates this form. From the Control Geometry tab select Remove All to remove the previous selection.
2. Then pick the Select Curve/Edge Regions button.



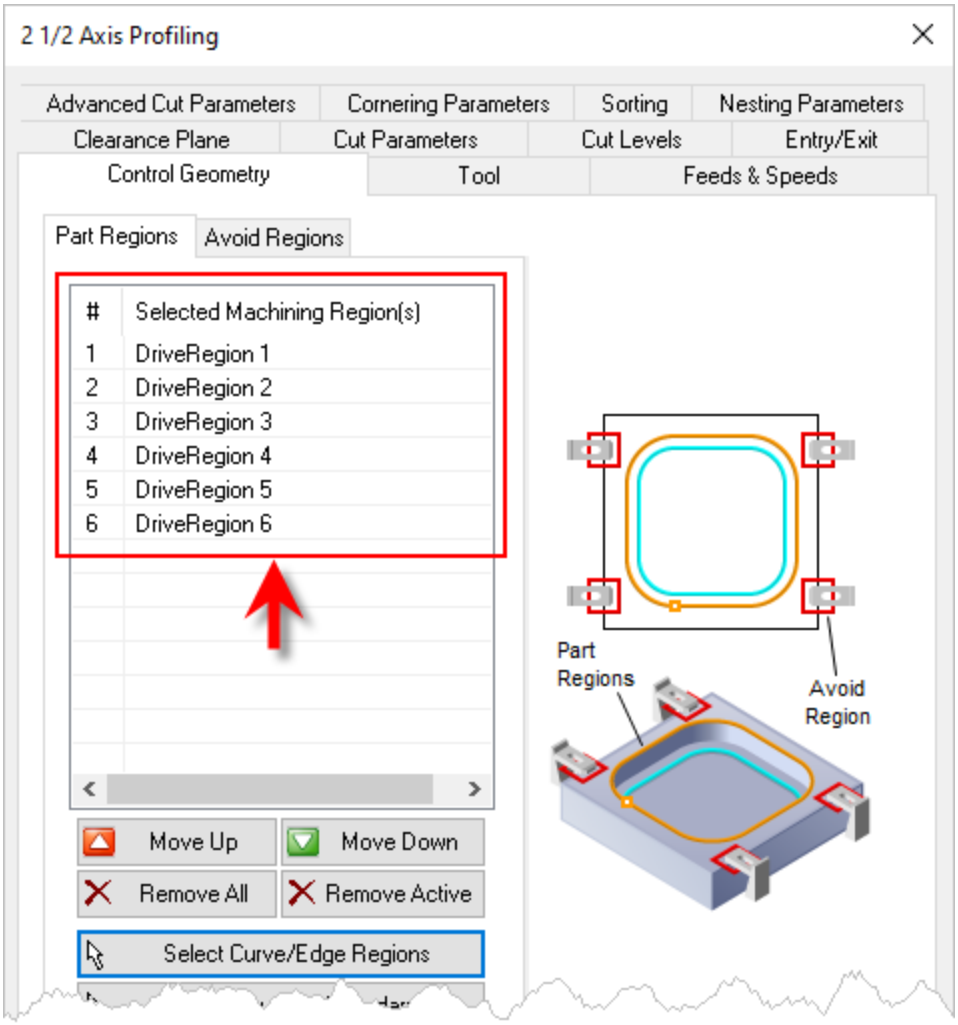
3. The dialog will minimize and you will be prompted to select a curve or surface edge. We only have 2D curve geometry and that's OK.

   First change to the Top View.

Then window select all of the curves that make up Part #1 and then right-click or simply press the <Enter> key to complete the selection.

© MecSoft Corporation

4. The Control Geometry tab of the 2-1/2 Axis Profile operation dialog reappears with the geometry from Part 1 listed as "Selected Machining Regions" under the Part Regions sub tab as shown below.



5. You will also see that while this dialog is displayed, the curves selected for the operation are highlighted on the part. You will also notice that each curve has it's start point identified. The cut motions will begin at each start point.

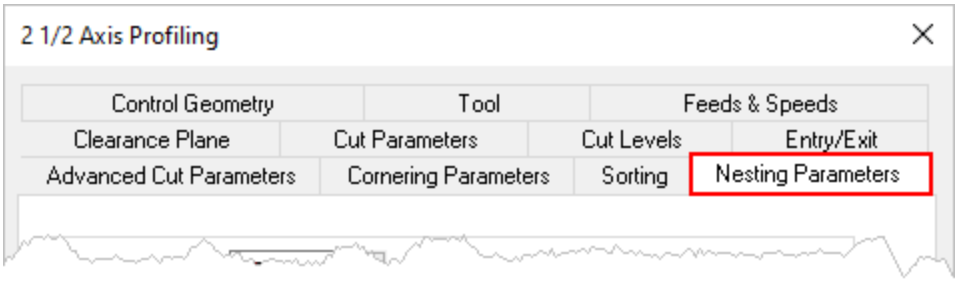
   Note: See our tech blog article How to Control the Cut Side and Start Point for more reference material on this topic.

7.1.2 Profile 1 Nesting Parameters tab

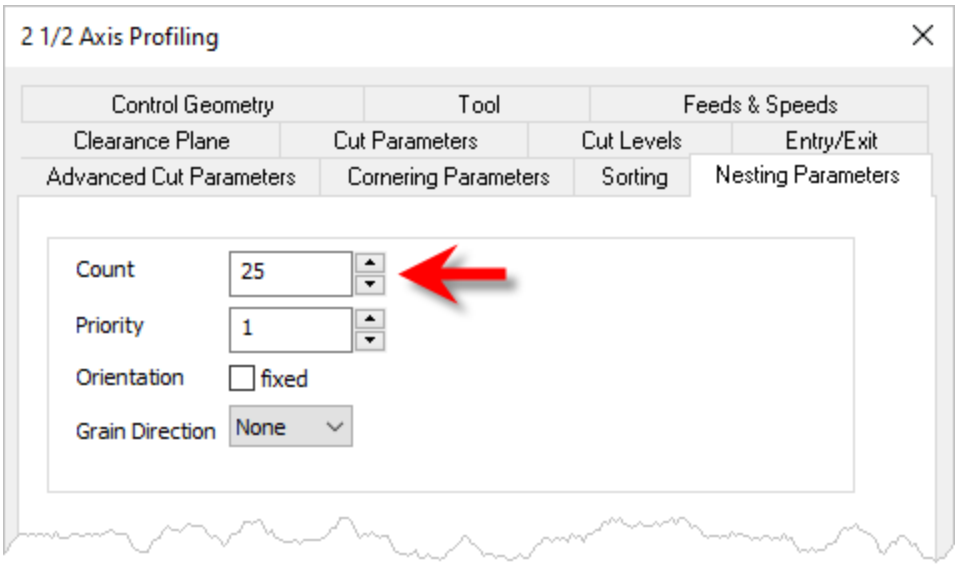
In this step we will move to the Nesting Parameters tab and set a few nesting parameters that will apply to this profile operation only.

© MecSoft Corporation

1. With the 2-1/2 Axis Profile operation dialog still displayed, select the Nesting Parameters tab.



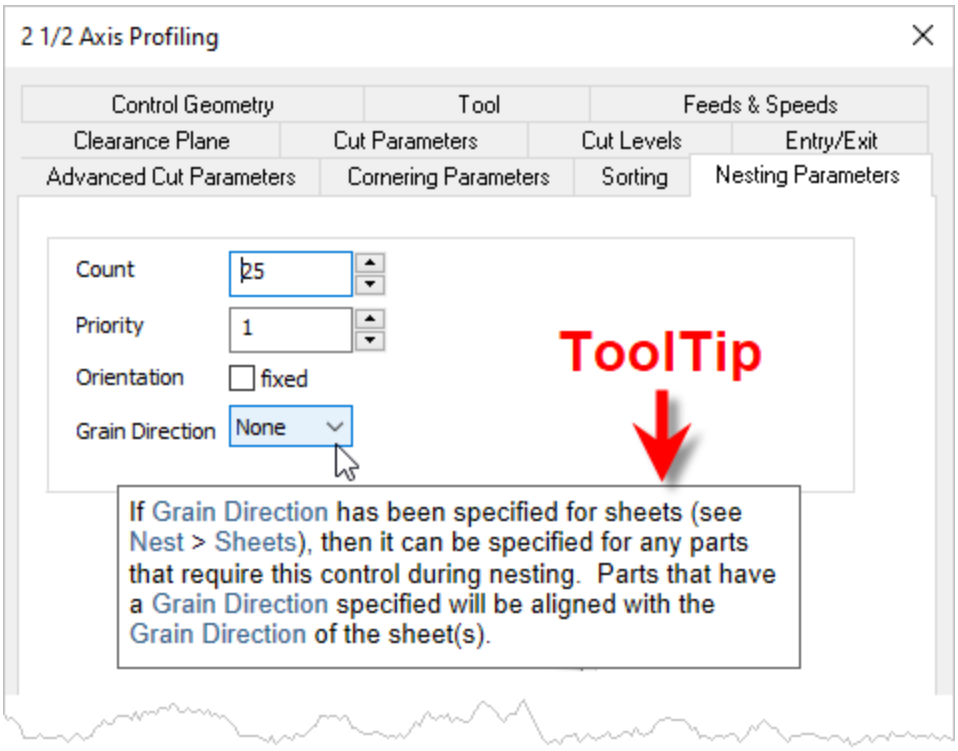
2. New locate the Count parameter and set this to 25. There will be 25 profiles of Part 1 in our resulting nested sheets.



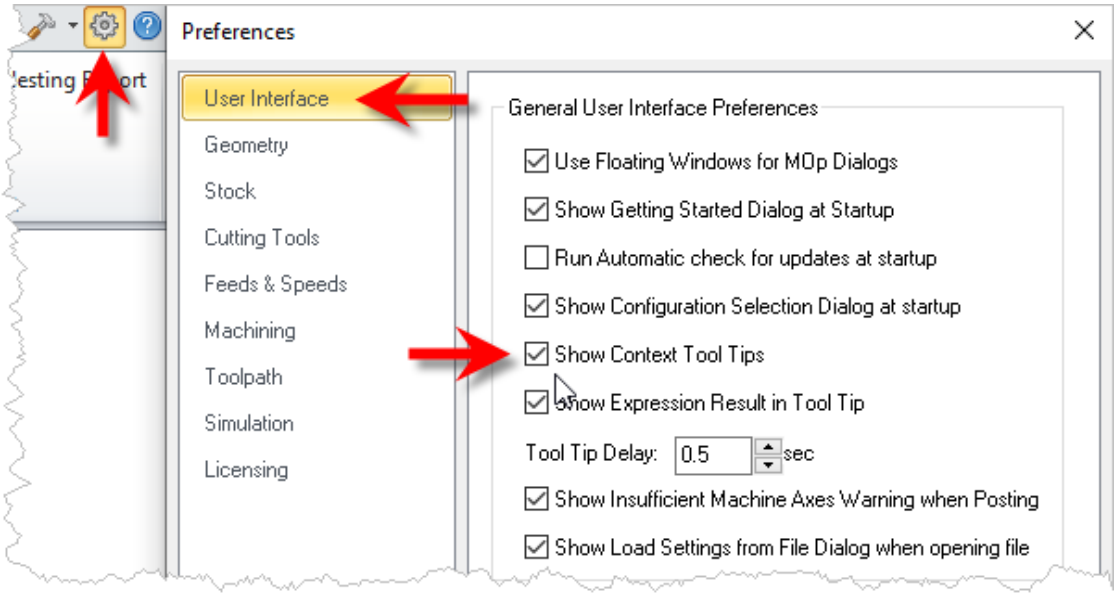
3. We will accept the default values for the remaining nesting parameters.

   However, you can take a moment to review the pop-up Context ToolTips by moving the mouse over each of the parameters as shown below.

© MecSoft Corporation



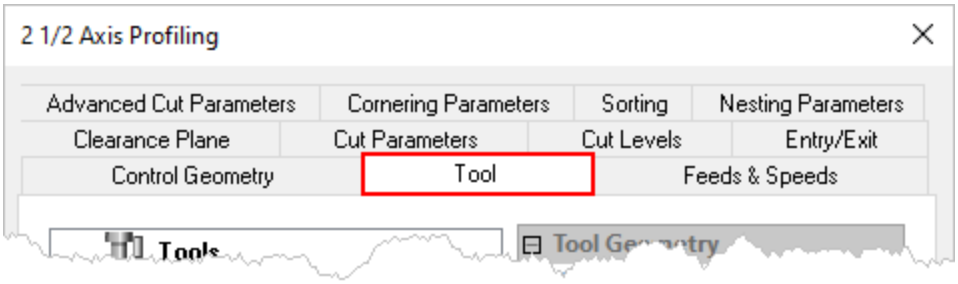
4. If the ToolTips are not displaying for you, go to the User Interface tab of the CAM Preferences to enable the display of Context ToolTips.



7.1.3 Profile 1 Tool tab

In this step we will move to the Tool tab and create a 1/4 inch (0.25”) Flat End Mill to use for nesting this operation.

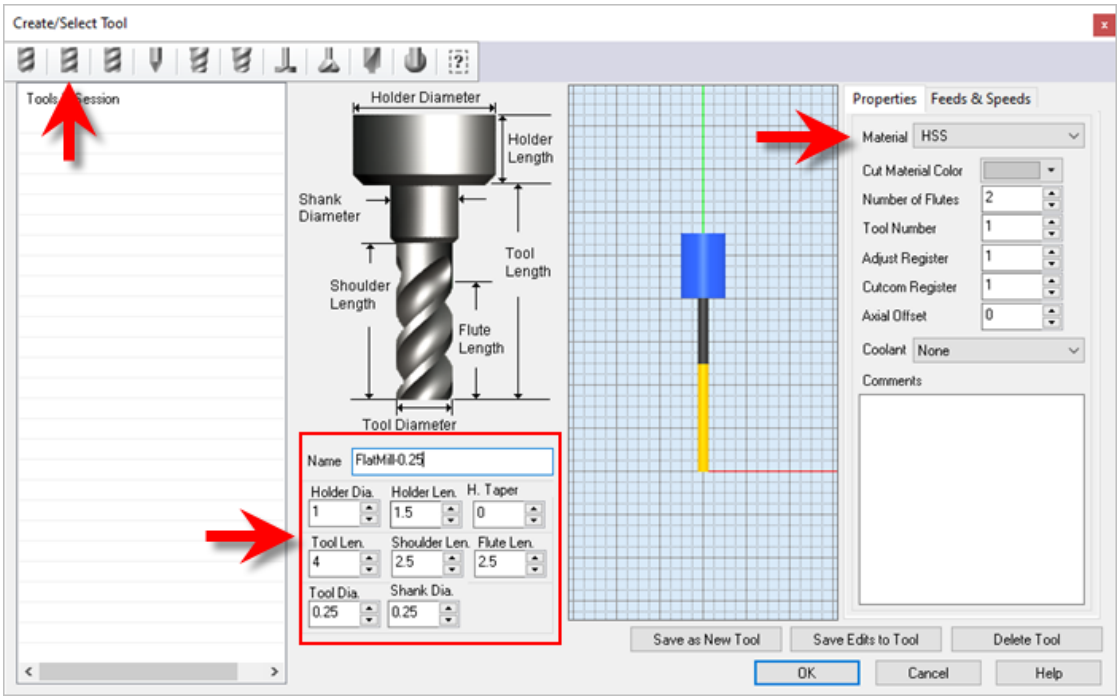
1. With the 2-1/2 Axis Profile operation dialog still displayed, select the Tool tab.



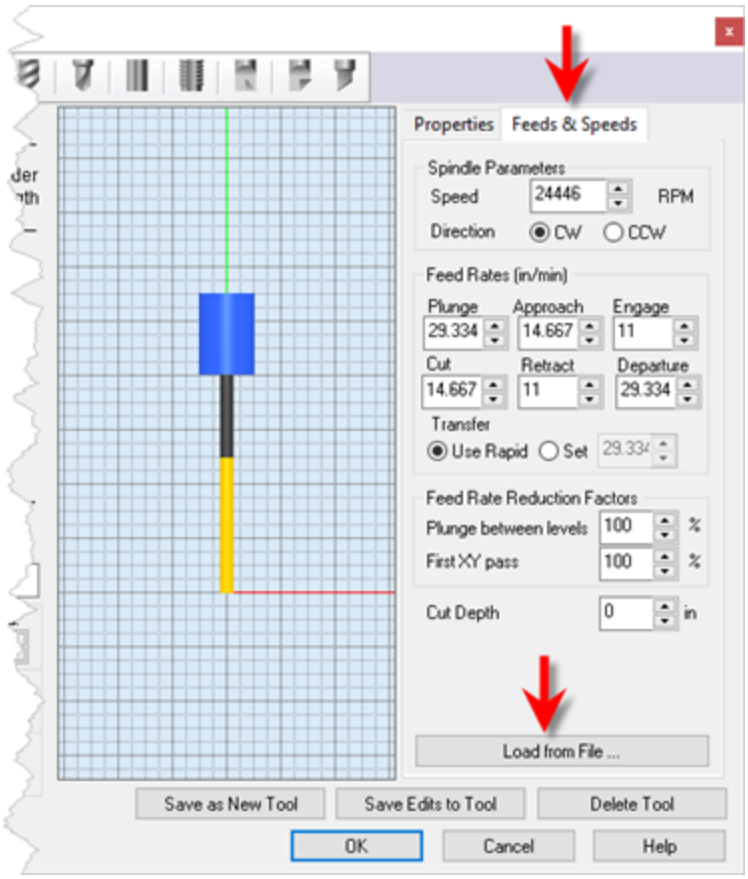
2. You see that there are no tools listed on the left. Let's create one right from this dialog. Near the bottom of the dialog pick the Edit/Create/Select Tool ... button.

2. This will display the Create/Select Tool dialog. First, select the Flat Mill icon from the Tool Type menu at the top of the dialog.

3. Set tool Name to FlatMill-0.25 and Tool Diameter to 0.25. Under the Properties tab set Material to HSS and Tool Number to 1.



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

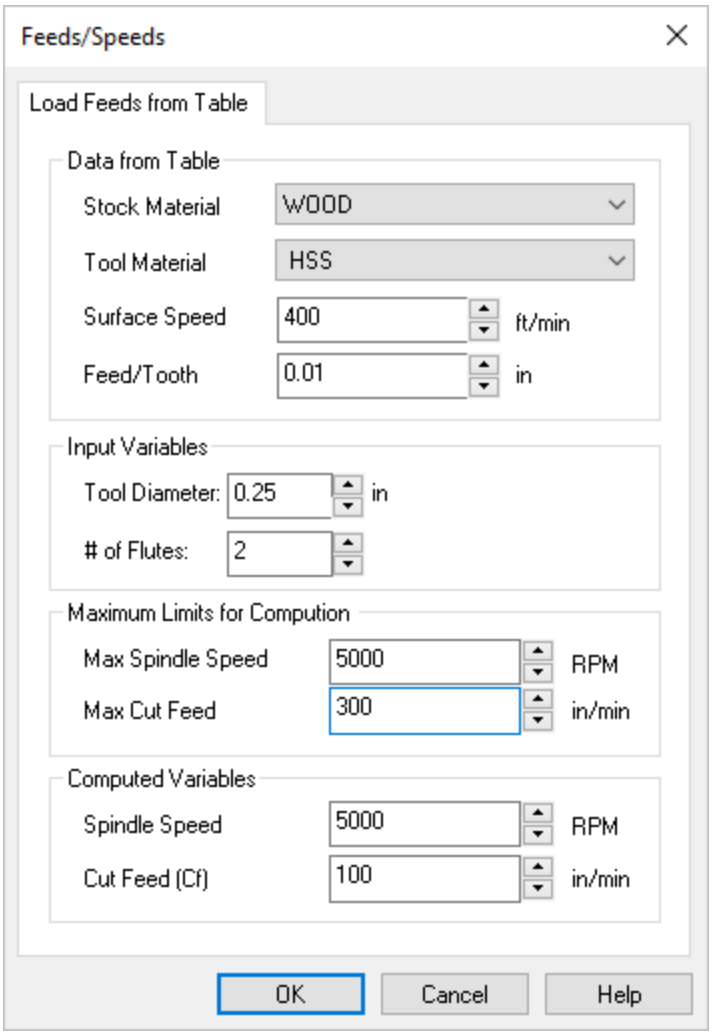


5. From the dialog that displays, set the following parameters:

   Stock Material : Wood
   Tool Material : HSS
   Max Spindle Speed: 5000
   Max Cut Feed : 300
   Tool Diameter : 0.25

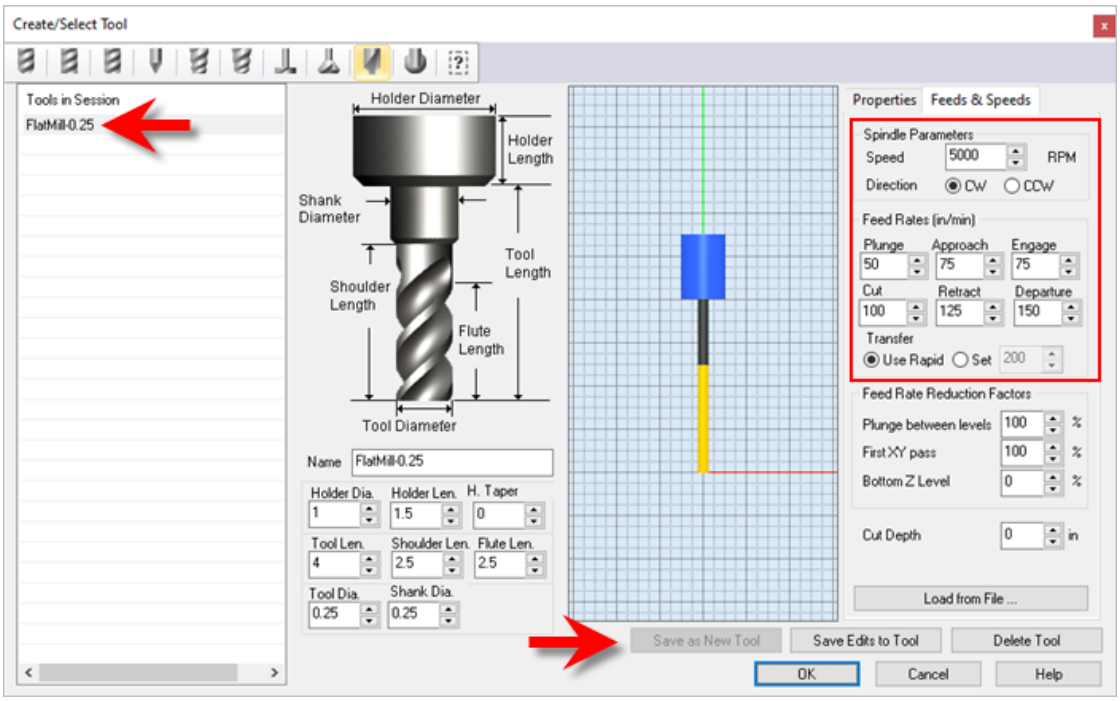
   The Computer Variables will be:

   Spindle Speed : 5000
   Cut Feed : 100



6. Now pick OK and the computed cut feedrate and spindle speed are transferred to the Feeds and Speeds tab of the Create/Select Tool dialog for this tool.

© MecSoft Corporation

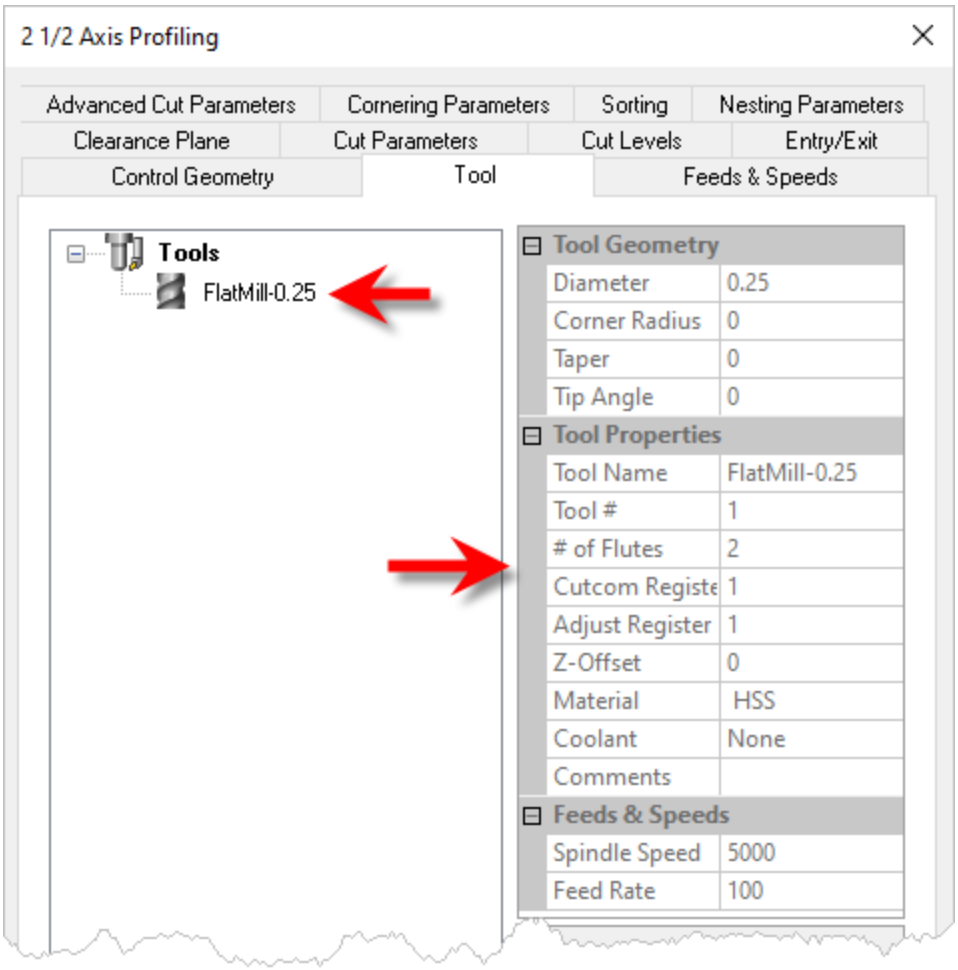


7. Pick Save as New Tool to save the tool. The tool is now created and listed under Tools in Session on the left side of the dialog.
8. Pick OK to close the dialog.

> You can edit the tool properties and pick Save Edits to Tool to save changes to this tool. To edit and save this as a New Tool, you must first enter a different tool Name.

The created tool is now appears in the Tools list within the Tool tab of the 2-1/2 Axis Profile operation dialog. It is also listed under the Tools tab in Machining Objects browser located at the bottom of the Machining Browser.

© MecSoft Corporation



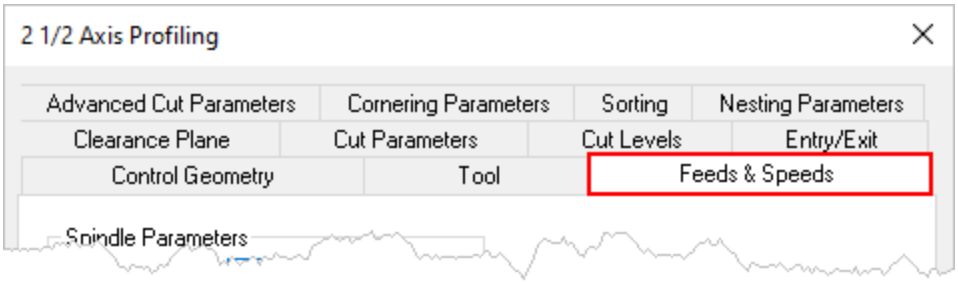
In the future you can save your tools to a Tool Library. To save Tools to a library, click Save Tool Library under the Tools tab in the Machining Objects Browser and specify a folder location and file name in the Save as dialog box. Two Tool Library file formats are supported (*.vkb and *.csv). The native Tool Library file format for VisualCAM is *.vkb.

© MecSoft Corporation

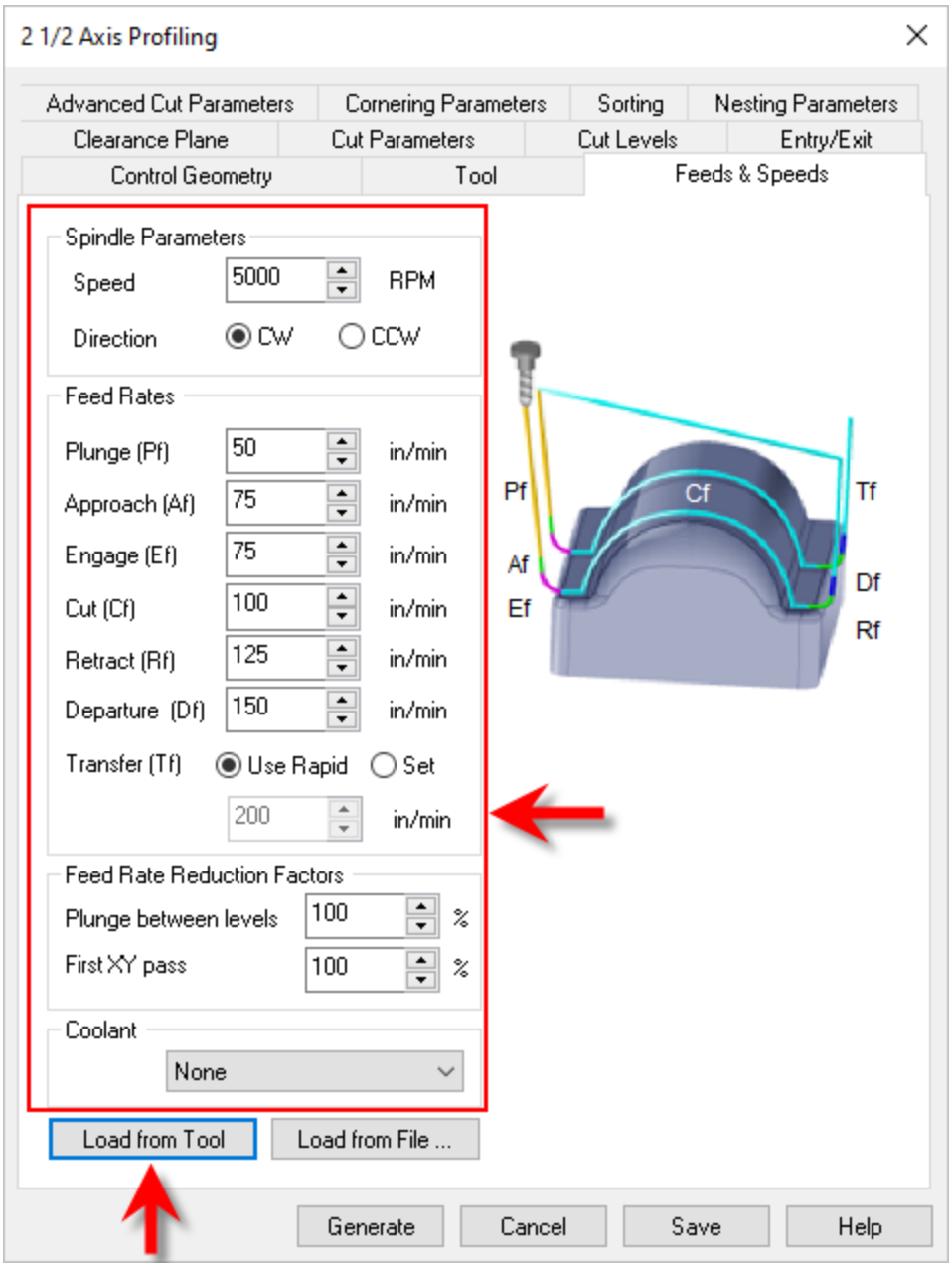
7.1.4 Profile 1 Feeds & Speeds tab

In this step we will move to the Feeds & Speeds tab to set the spindle speed and cut feeds for this operation.

1. With the 2-1/2 Axis Profile operation dialog still displayed, select the Feeds & Speeds tab.



2. Now select the Load from Tool button. This will load the feeds and speeds parameters that we have set for the tool into this dialog to use for this operation.



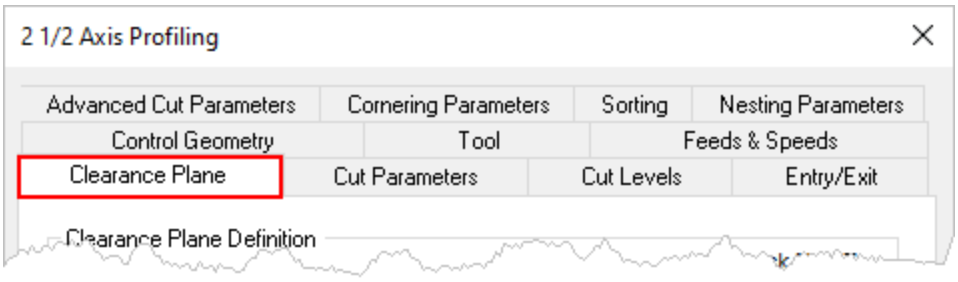
3. You can override the feeds and speeds values here in this dialog and they will apply to just this operation.

7.1.5 Profile 1 Clearance Plane tab

In this step we will move to the Clearance Plane tab. This tab is used to set the Clearance and Cut Transfer parameters for this Profile operation.

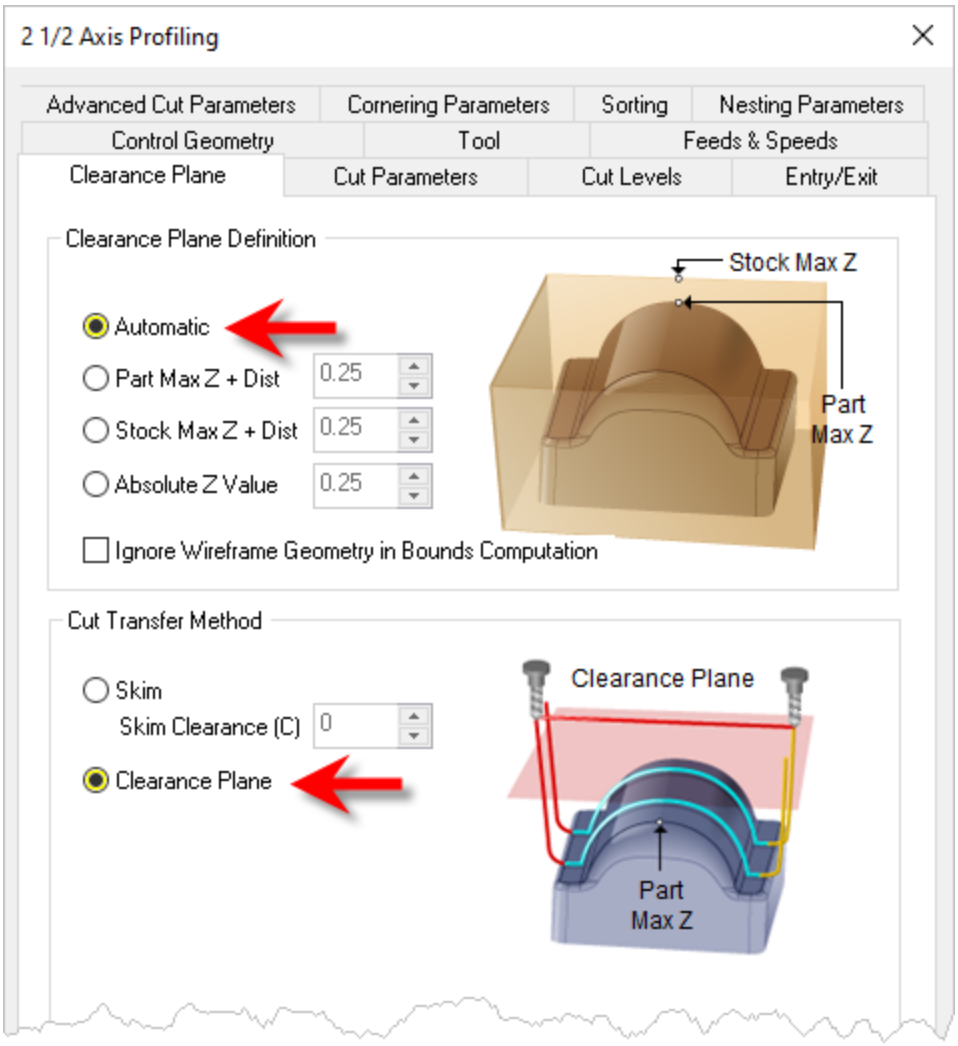
1. With the 2-1/2 Axis Profile operation dialog still displayed, select the Clearance Plane tab.

© MecSoft Corporation



2. For Clearance Plane Definition select Automatic. The system with determine a safe distance to use for the clearance plane.
3. For Cut Transfer Method, select Clearance Plane. The cutting tool will retract to the Clearance Plane to perform all cut transfer moves.

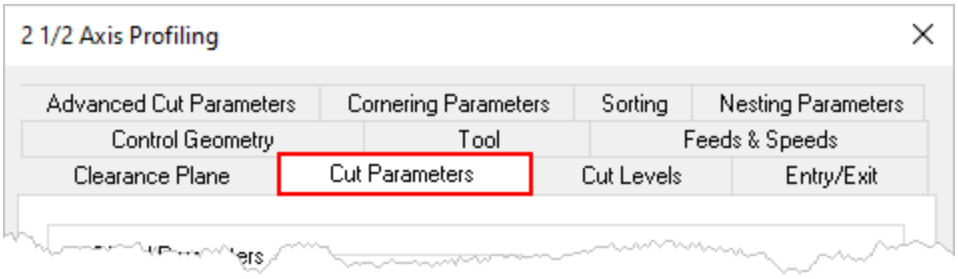
   The selections are shown in the dialog below.



7.1.6 Profile 1 Cut Parameters tab

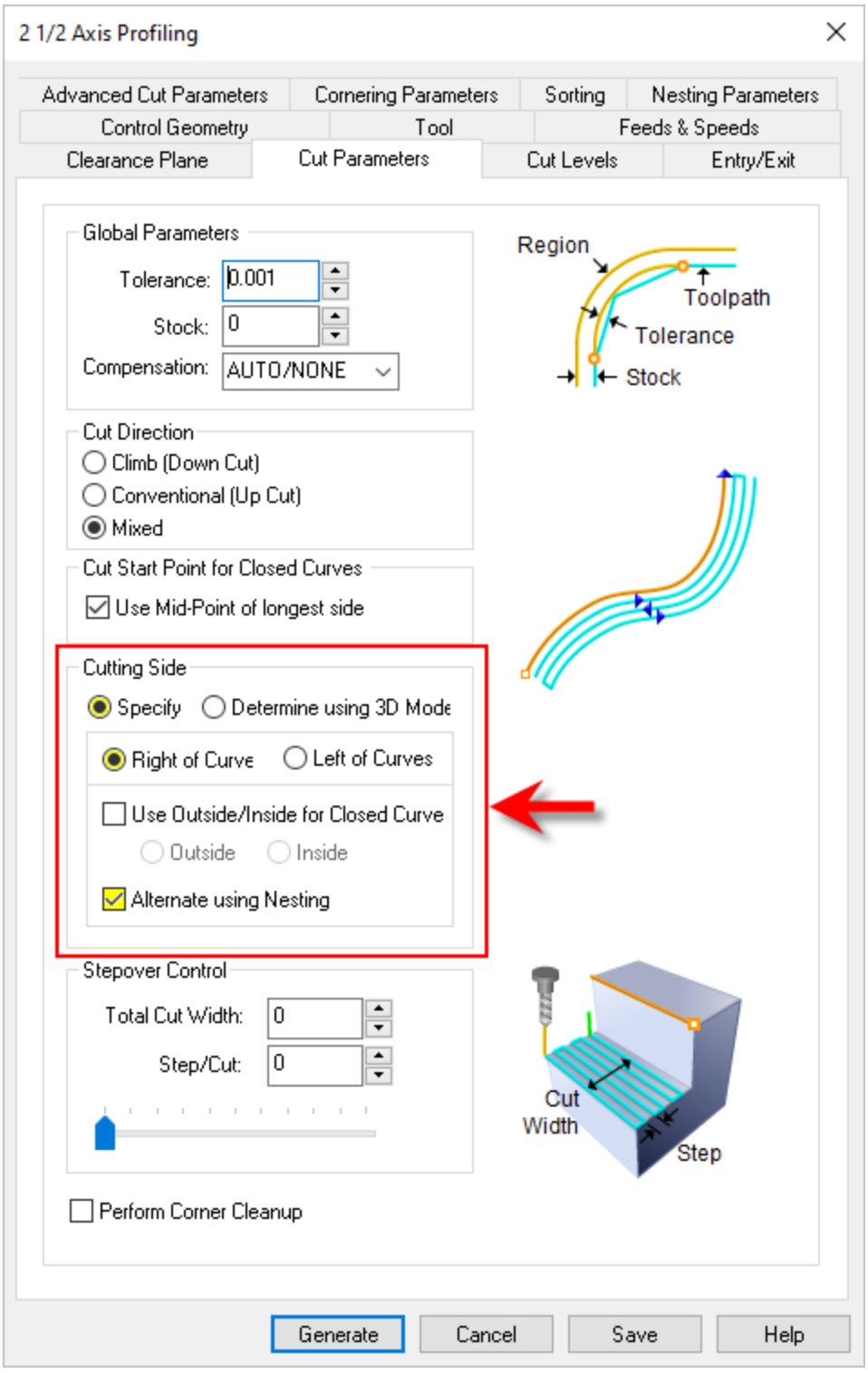
In this step we will move to the Cut Parameters tab and set the cutting parameters for this Profile operation.

1. With the 2-1/2 Axis Profile operation dialog still displayed, select the Cut Parameters tab.



2. You see that the dialog has default selections made. We will make some changes to the Cutting Side section of the dialog.
3. For Cutting Side select the Specify option and then select Right of Curve.
4. Then also check the box called Alternate using Nesting. You will notice that Part 1 has interior cutouts. With this option checked, the system will automatically determine the cutting side for all interior cutouts.

© MecSoft Corporation

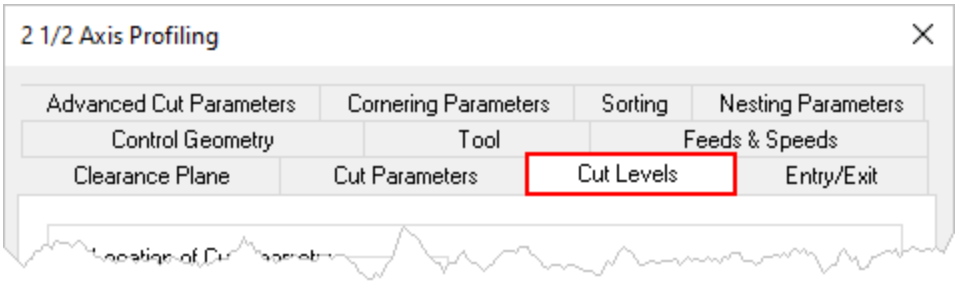


5. You can leave the remaining cut parameters at their default selections.

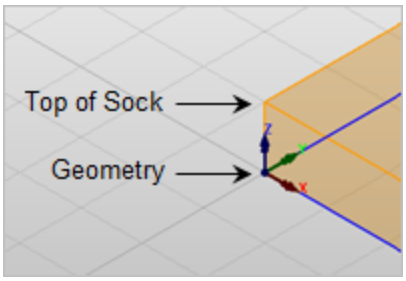
7.1.7 Profile 1 Cut Levels tab

In this step we will move to the Cut Levels tab and set our depth parameters. **Note**: The previous Cut Parameters tab affects XY tool motions. This Cut Levels tab affects all Z level tool motions for this operation.

1. With the 2-1/2 Axis Profile operation dialog still displayed, select the Cut Levels tab.

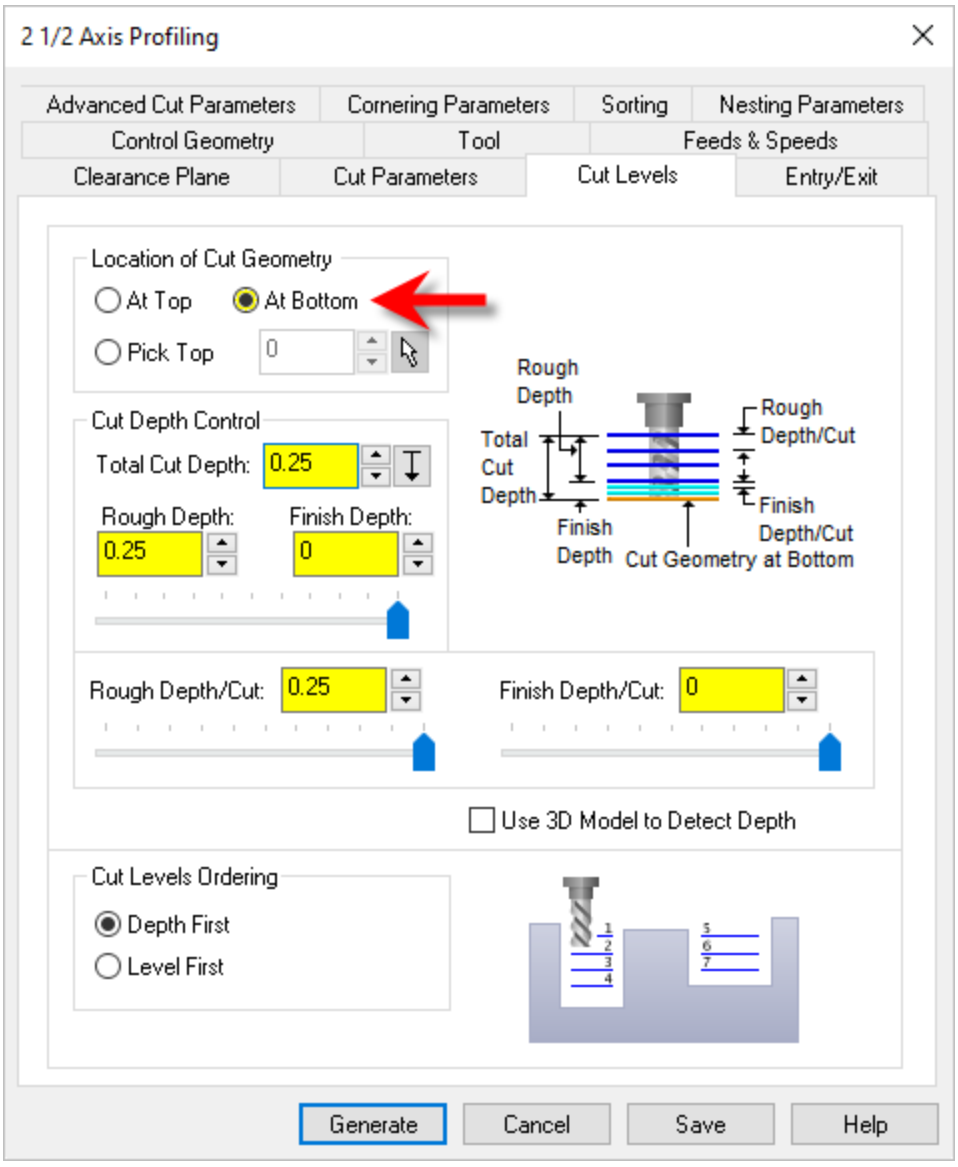


2. Fist, under Location of Cut Geometry select At Bottom as shown in the dialog below. This means that our Control Geometry for Part 1 is located at the bottom of the cut. For this example the bottom of the cut is located at Z0.



3. Now since we previously set our Sheet Thickness to 0.25, we need to make sure we set the Total Cut Depth to 0.25 also. These two values must match or Nesting will return an error when you Execute Nest. We have highlighted each of the fields to set in the dialog below.

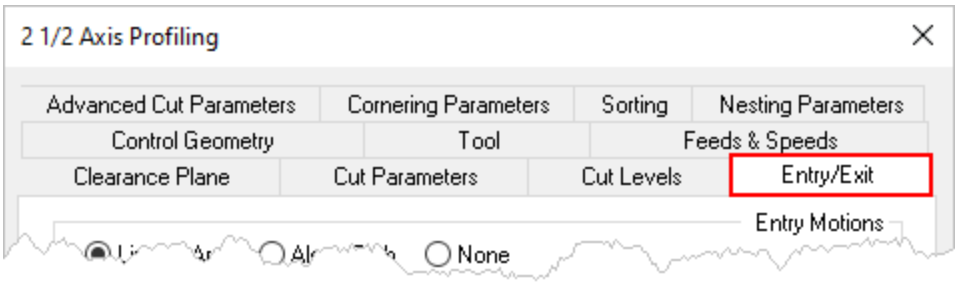
© MecSoft Corporation



7.1.8 Profile 1 Entry/Exit tab

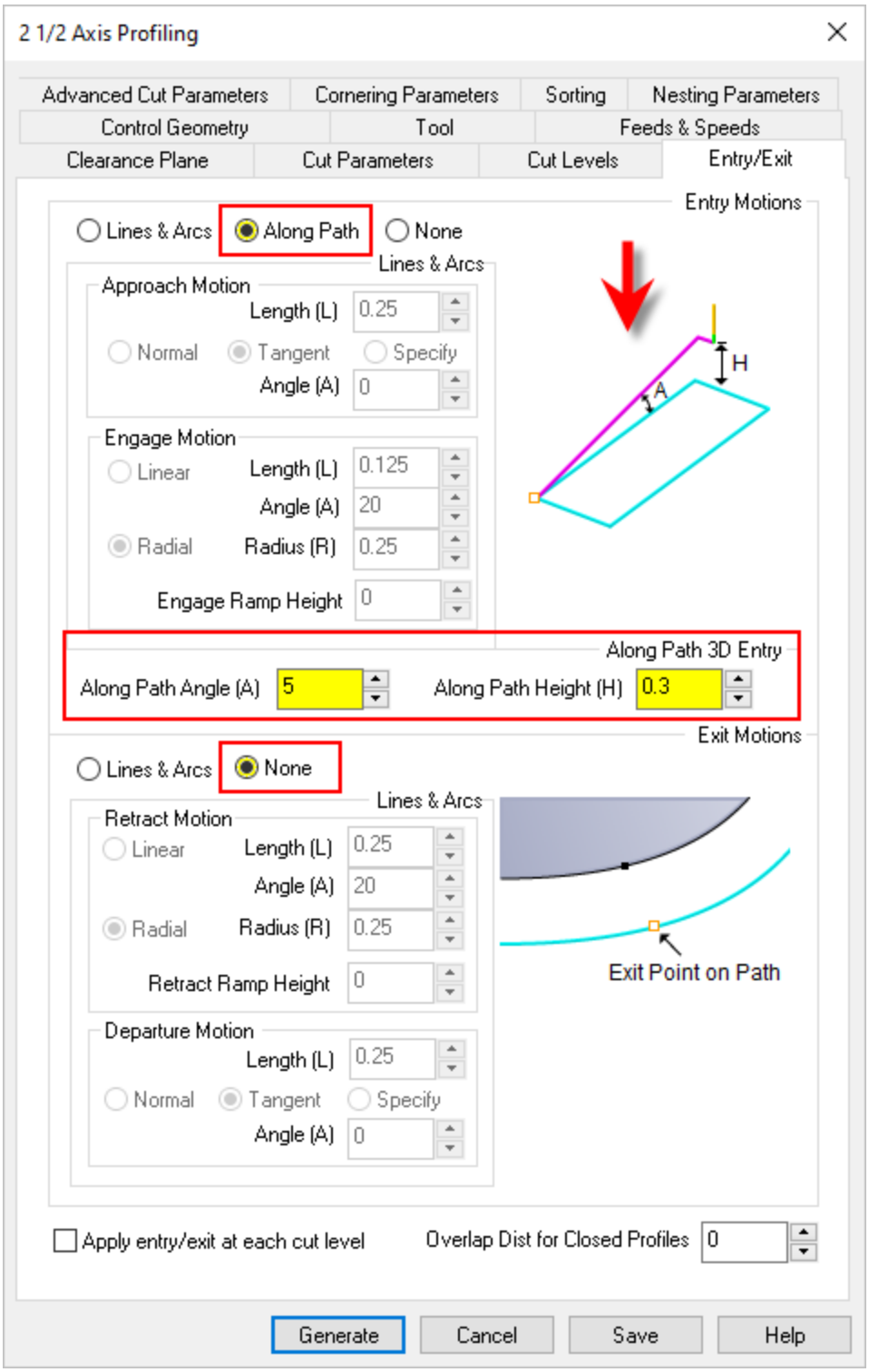
In this step we will move to the Entry/Exit tab and set parameters to control how the tool will entry and exit the cutting path. These include four types of motions: Approach, Engage, Retract and Departure.

1. With the 2-1/2 Axis Profile operation dialog still displayed, select the Entry/Exit tab.



2. We want the tool to ramp into the cut at an angle. To do this under the Entry Motions section (upper portion) of the dialog select Along Path. The image in the dialog will show you what each entry type looks like and the parameters to go with them. We have highlighted the options and values to use in the dialog below.
3. With Along Path selected, enter 5 for the Along Path Angle (A) and 0.3 for the Along Path Height (H). Each entry will begin at 0.3 above the Control Geometry and enter at a 5 degree angle.

© MecSoft Corporation

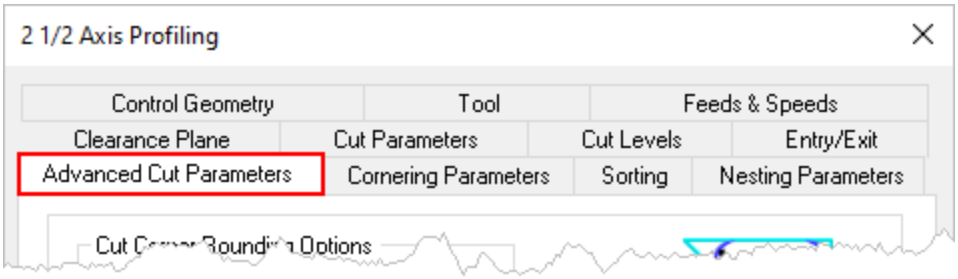


4. In the Exit Motions section (bottom portion) of the dialog, select None. The tool will retract straight upward to the Clearance Plane at the end of each cut.

7.1.9 Profile 1 Advanced Cut Parameters tab

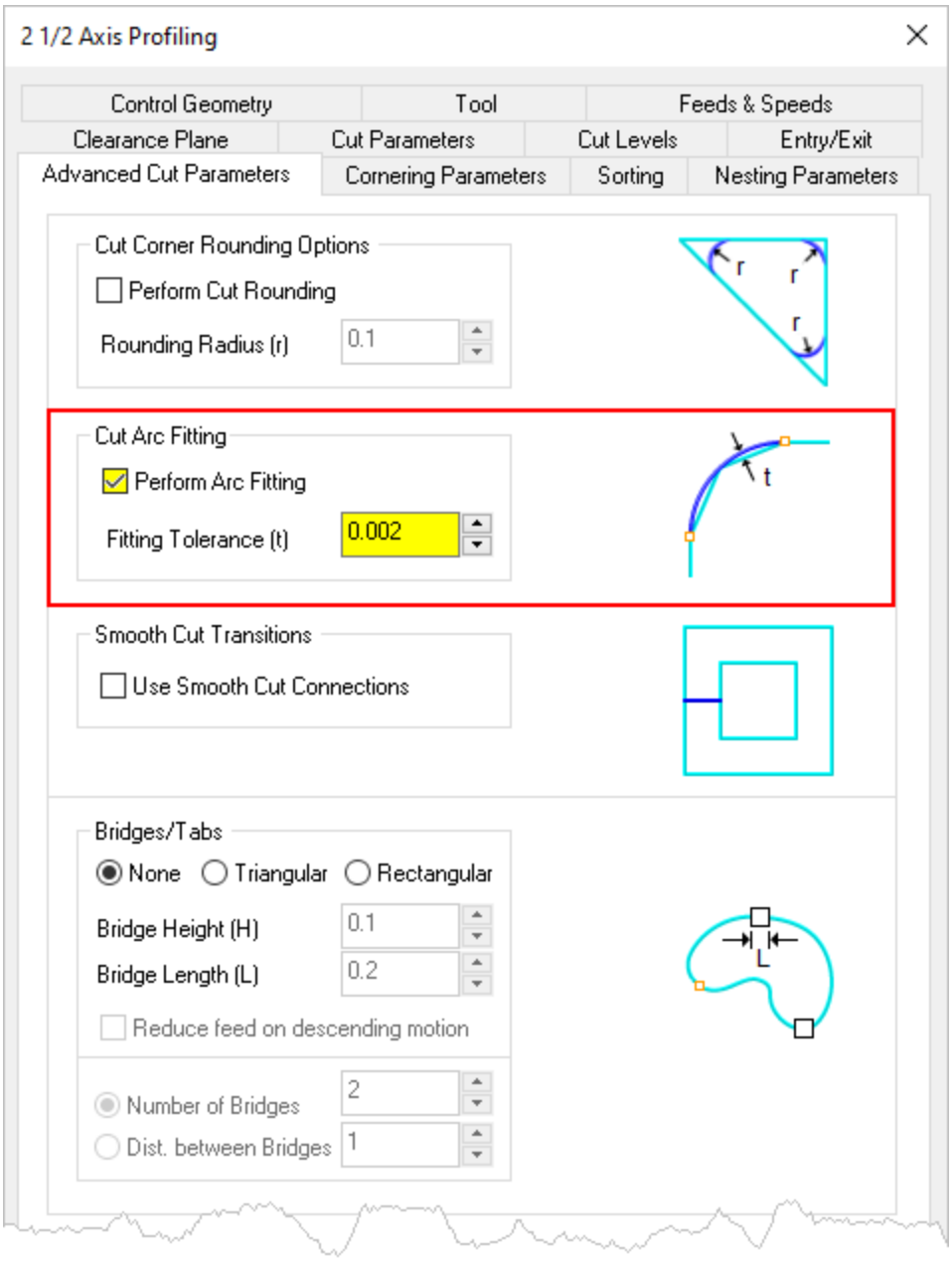
In this step we will move to the Advanced Cut Parameters tab and set Arc Fitting parameters.

1. With the 2-1/2 Axis Profile operation dialog still displayed, select the Advanced Cut Parameters tab.



2. Under the Cut Arc Fitting section, check the box to Perform Arc Fitting and then enter a value of 0.002 for Fitting Tolerance (t). Refer to the image in the dialog for how this value is calculated.

© MecSoft Corporation

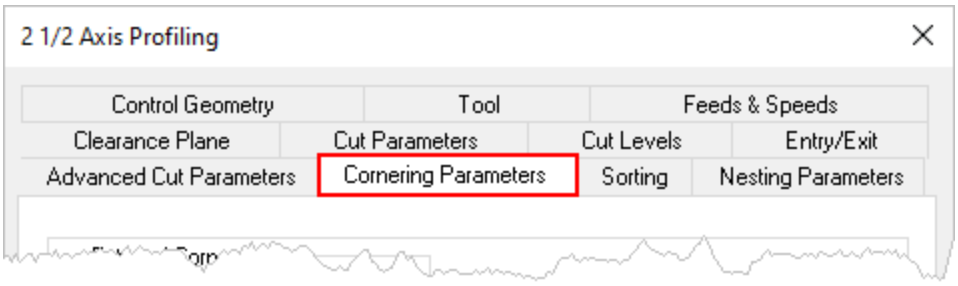


3. In the future you can also use this tab to automatically place Bridges & Tabs along the profile path to help keep your parts attached and stable during cutting.

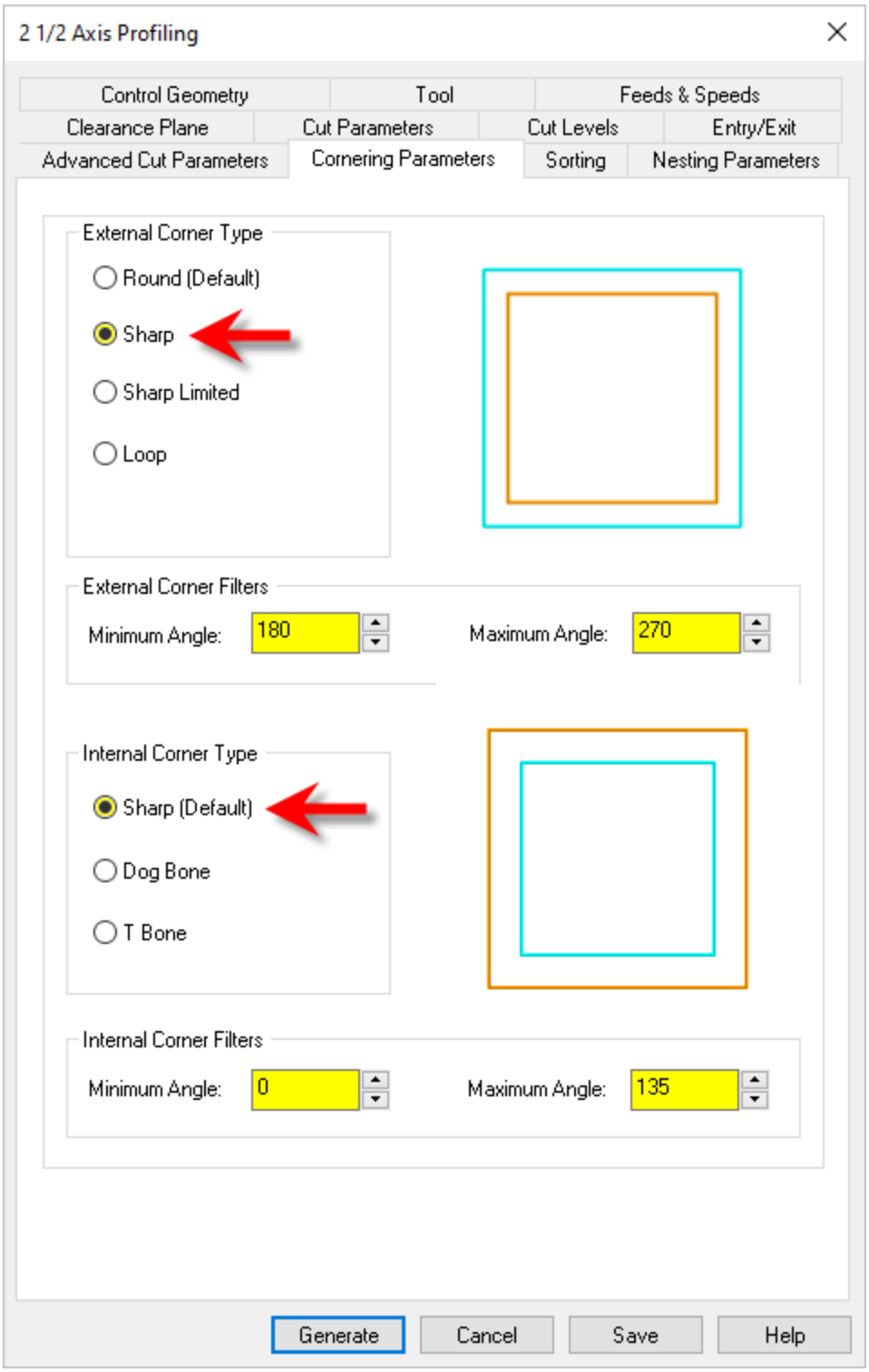
7.1.10 Profile 1 Cornering Parameters tab

In this step we will move to the Cornering Parameters tab and set parameters that control how the cutter will respond to both inner and outer corners.

1. With the 2-1/2 Axis Profile operation dialog still displayed, select the Cornering Parameters tab.



2. For the External Corner Type section select the Sharp option. This will force the cutter to extend past the corner before changing directions (i.e., the cutter will not ride on the corner). Then set Minimum Angle to 180 and Maximum Angle to 270 as shown in the dialog below.



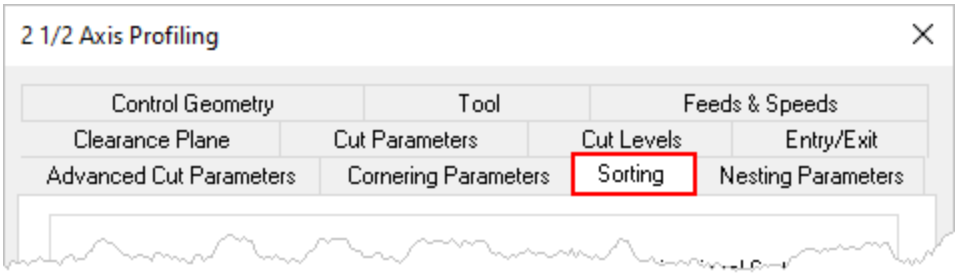
3. Then for Internal Corner Type, select Sharp. This is the default selection and you can accept the default values for Minimum and Maximum Angle also.

© MecSoft Corporation

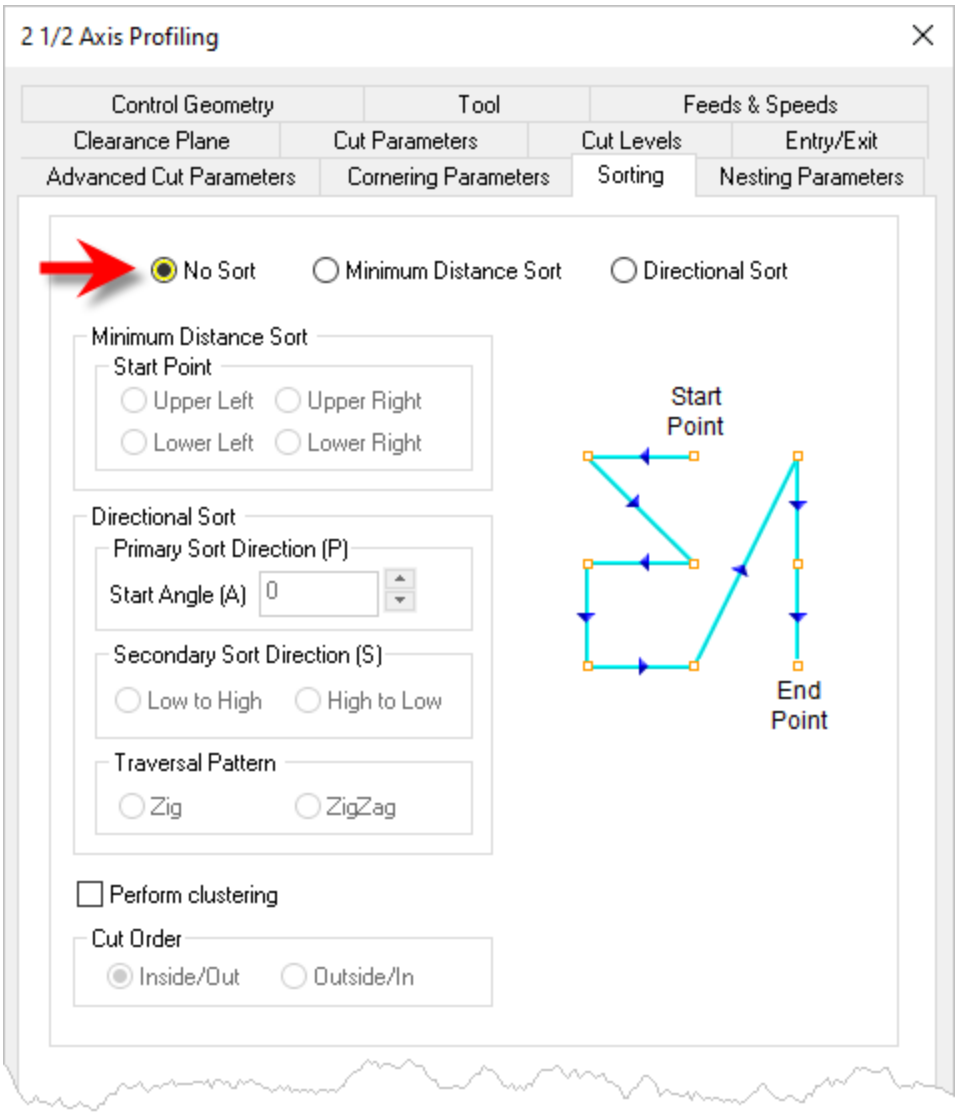
7.1.11 Profile 1 Sorting tab

In this step we will move to the Sorting tab. While in this guide we will not perform any sorting, it is important that know about this tab. If you have multiple sets of geometry in this profile operation, for example a scattered hole pattern, you can use these parameters to control the order of the cutting within this operation.

1. With the 2-1/2 Axis Profile operation dialog still displayed, select the Sorting tab.



2. Select No Sort.

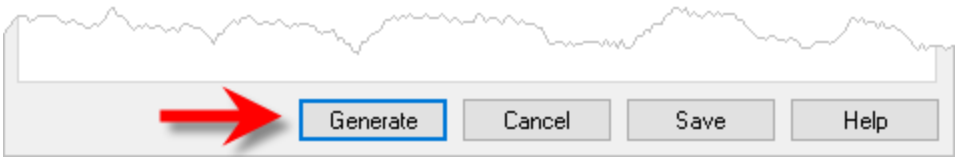


3. Then for Internal Corner Type, select Sharp. This is the default selection and you can accept the default values for Minimum and Maximum Angle also.

7.1.12 Profile 1 Generate

Now that we have covered all of the tabs in our Profile operation for Part 1 we can generate the toolpath.

1. With the 2-1/2 Axis Profile operation dialog still displayed, select the Generate button at the bottom of the dialog.

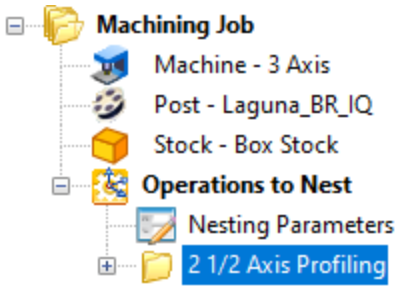


© MecSoft Corporation

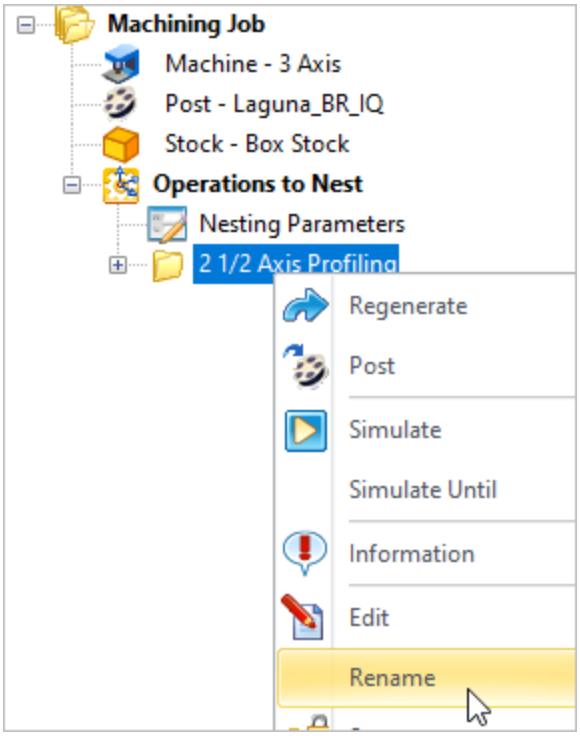
2. You will see that the 2-1/2 Axis Profile toolpath operation was created for Part #1.

3. You will also see that the toolpath has the 5 degree ramp entry, sharp corners, arc motions and nested cut side.

4. You will also see that the Profile operation was added to the Machining Job Tree.

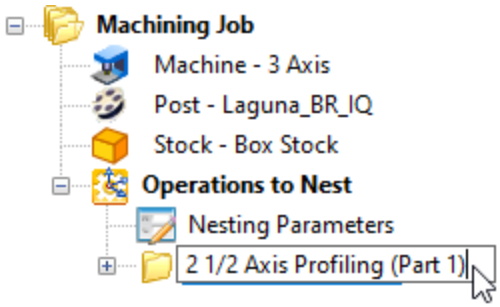


5. Let's rename the operation. Left-click on the operation name in the Machining Job to edit it. You can also left-click on the operation name and select Rename from the menu of operations.



6. Rename the operation as 2-1/2 Axis Profiling (Part 1) as shown below.

© MecSoft Corporation



7.2 Profile Parts 2,3 & 4

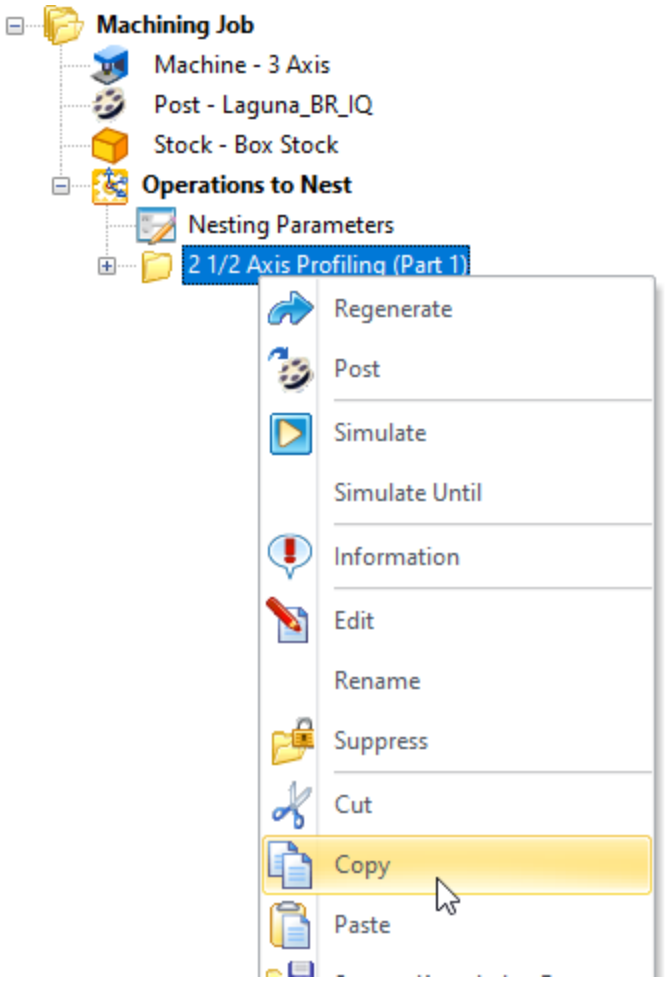
Now that we have a profile operation completed for Part 1, it will be very easy to create profile operations for Part 2,3 & 4. We'll just copy and paste our first operation and then edit the new operations.

7.2.1 Copy & Paste Profile Operations

In this step we will Copy & Paste our Profile operation to make 3 additional copies.

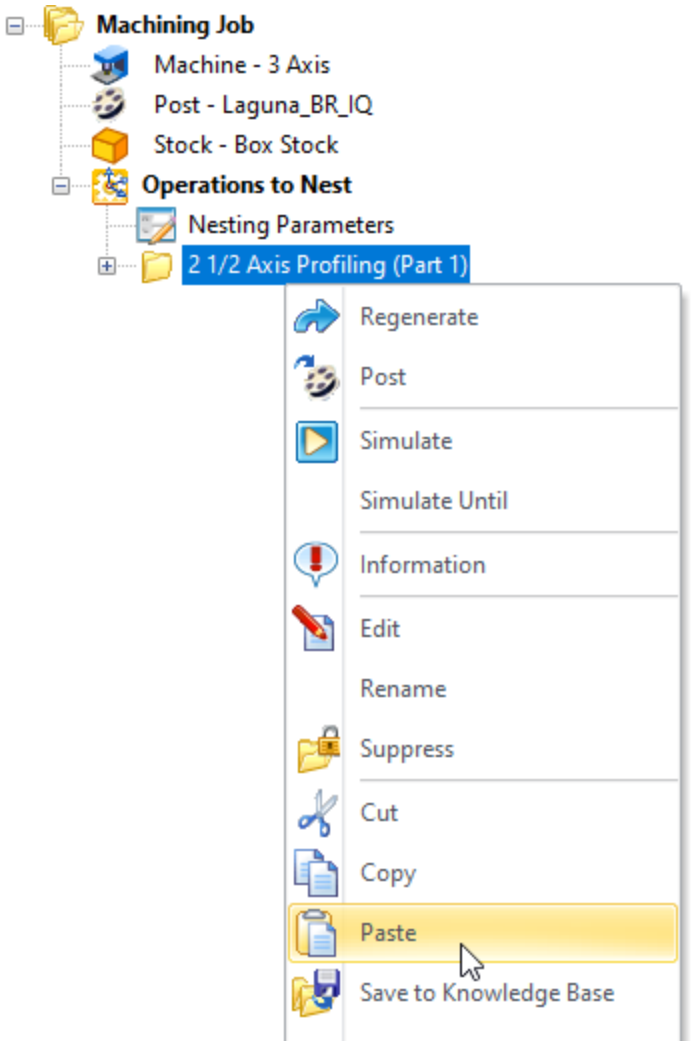
1. First make sure the 2 1/2 Axis Profiling (Part 1) operation is select from the Machining Job tree. Then right-click on the selected operation and select Copy from the menu. This is shown below.

© MecSoft Corporation

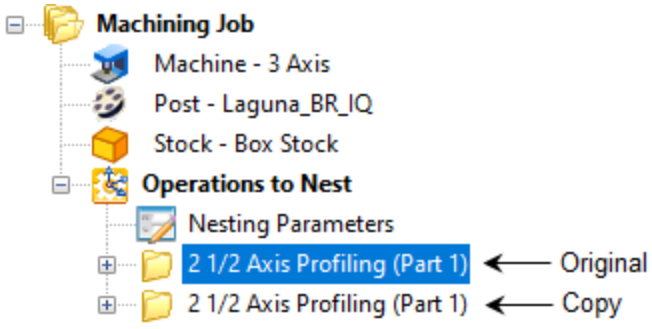


2. Now right-click on the selected operation again and select Paste.

© MecSoft Corporation

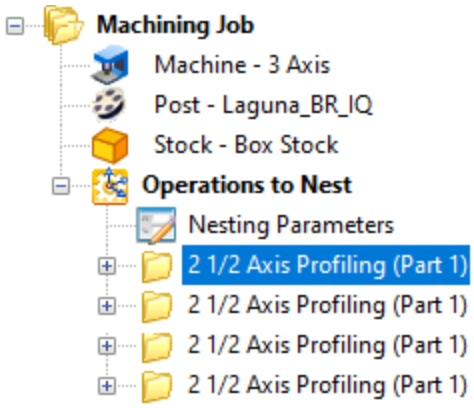


3. You will see that a second exact copy of the operation was added below the original.

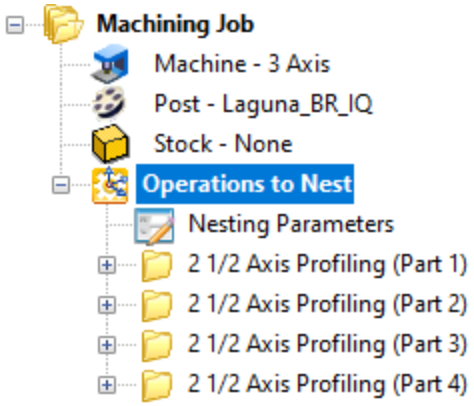


4. Now perform Step 2 & Step 3 (above) again two more times until you have a total of 4 operations listed in your Machining Job.

© MecSoft Corporation



5. Now rename the 3 copied operations for Part 2, 3 and 4 so that they appear as shown below. If you forgot how to rename an operation go back and revisit topic Profile 1 Generate to learn how.

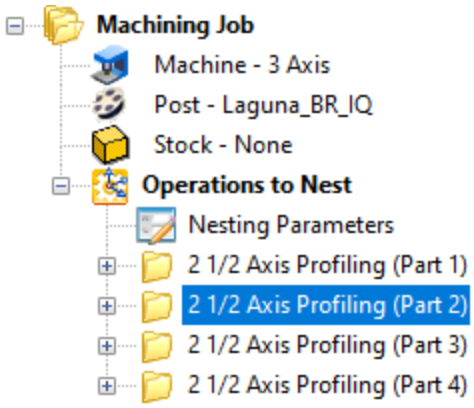


7.2.2 Edit Profile Operations

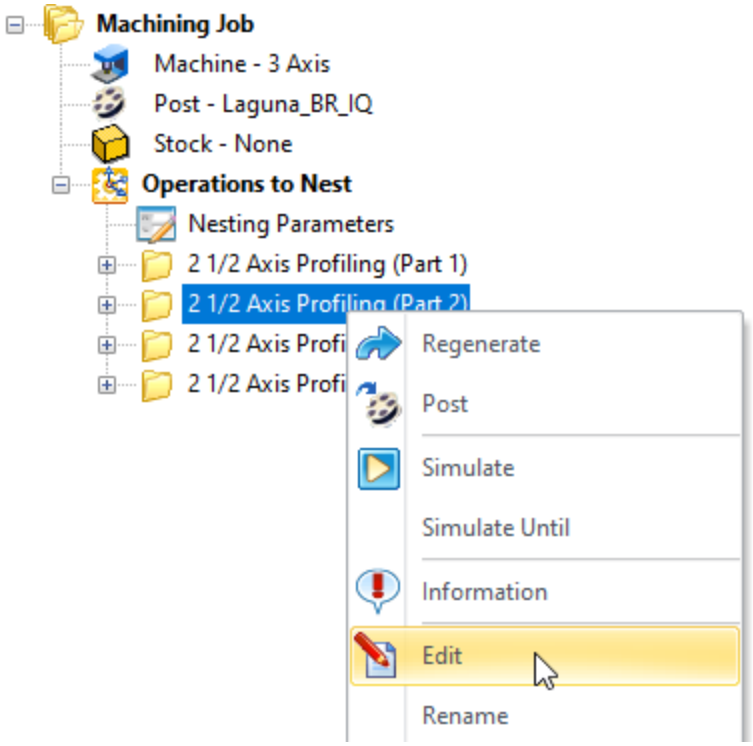
In this step we will Edit our Profile operations for Part 2, Part 3 and Part 4 by replacing the Control Geometry.

1. First select the Profile operation for Part 2. This will be 2 1/2 Axis Profiling (Part 2) under the Operations to Nest portion of the Machining Job.

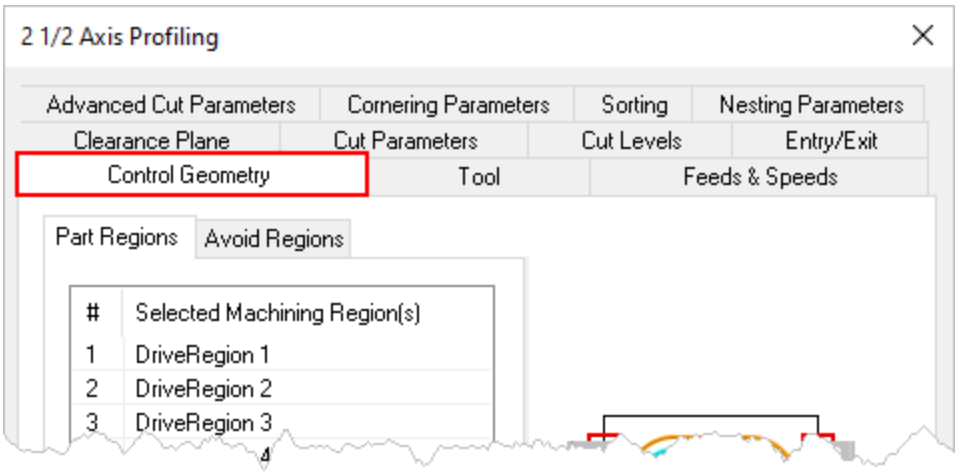
© MecSoft Corporation



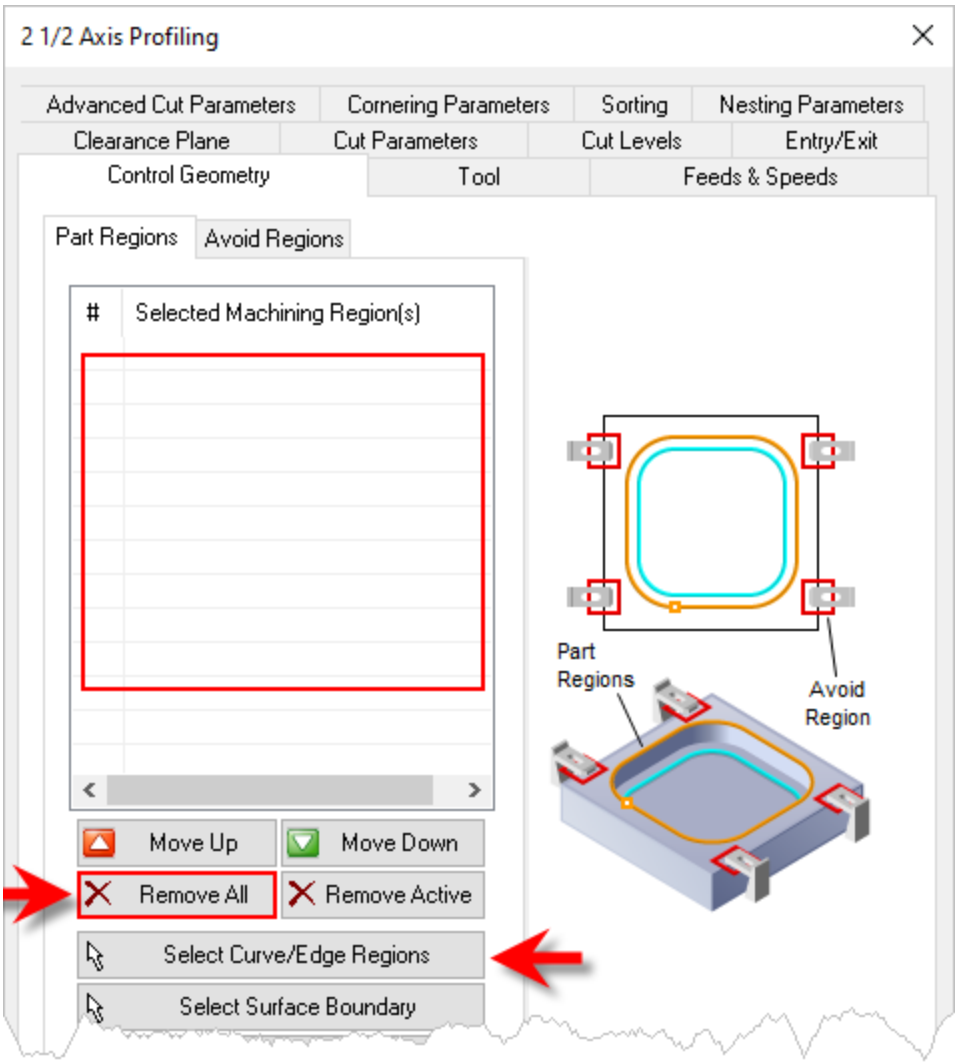
2. Now right-click on the selected operation again and select Edit.



3. The Profiling operation dialog will display with the Control Geometry tab selected by default.

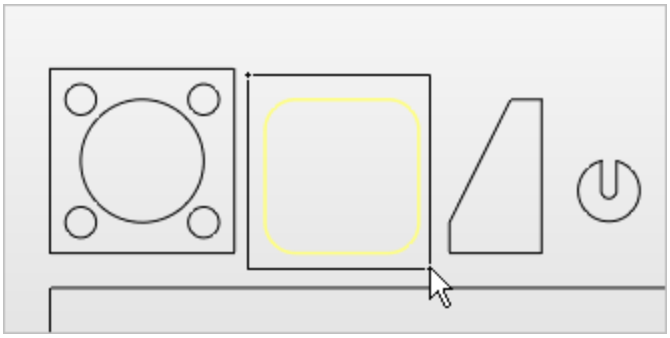


4. We don't want to use the same Part Regions so select the Remove All button to remove them from the list.

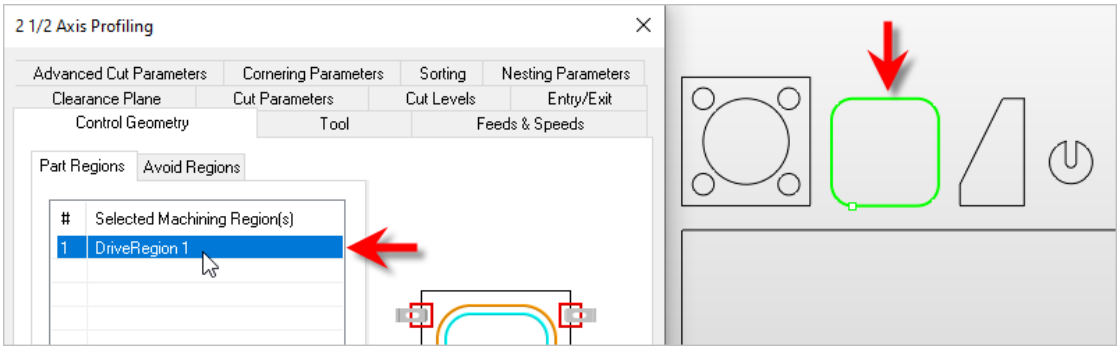


© MecSoft Corporation

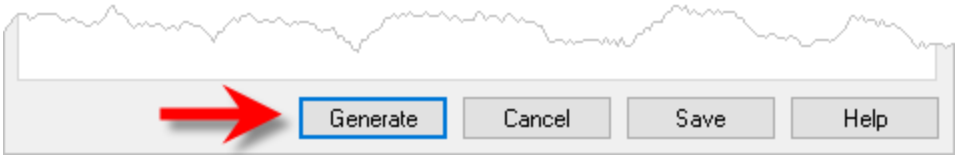
5. Now pick the Select Curve/Edge Regions button. The dialog will minimize and prompt you to select curve or edge regions. Window select the curve profile of Part 2 and then right-click or press <Enter> to complete the selection.



6. The dialog reappears with the selected profile geometry listed in the Machining Regions list.



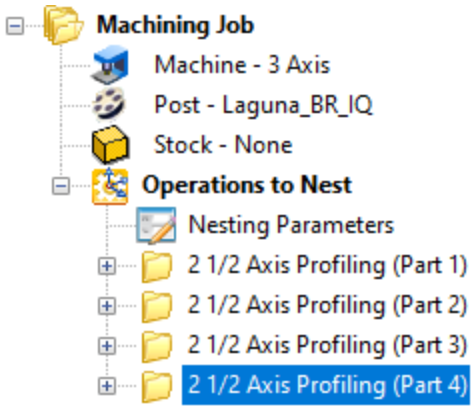
7. Now pick Generate to create the profile toolpath for Part 2.

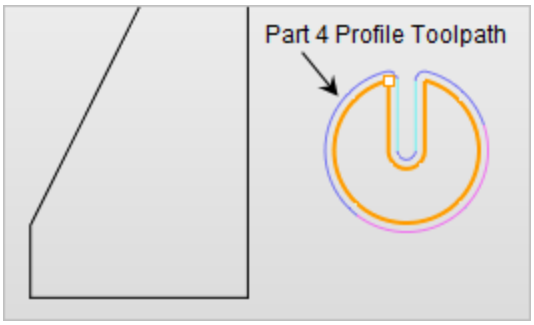


8. Now repeat the above Steps 1 through Step 7 again to generate a Profile toolpath for Part 3 and Part 4.

9. Once all Profile operations are edited and generated, you will see that selecting each operation folder in the Machining Job will display the toolpath for that part.

© MecSoft Corporation





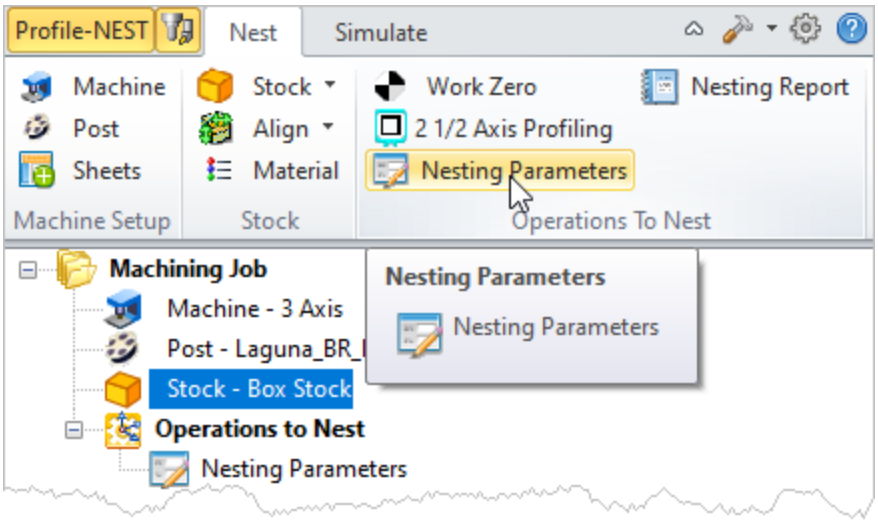
© MecSoft Corporation

The Nested Sheets

8.1 Set Nesting Parameters

In this step we'll define the general nesting parameters that will apply to all nested profiles and sheets.

1. From the Program tab select Nesting Parameters to display the dialog.

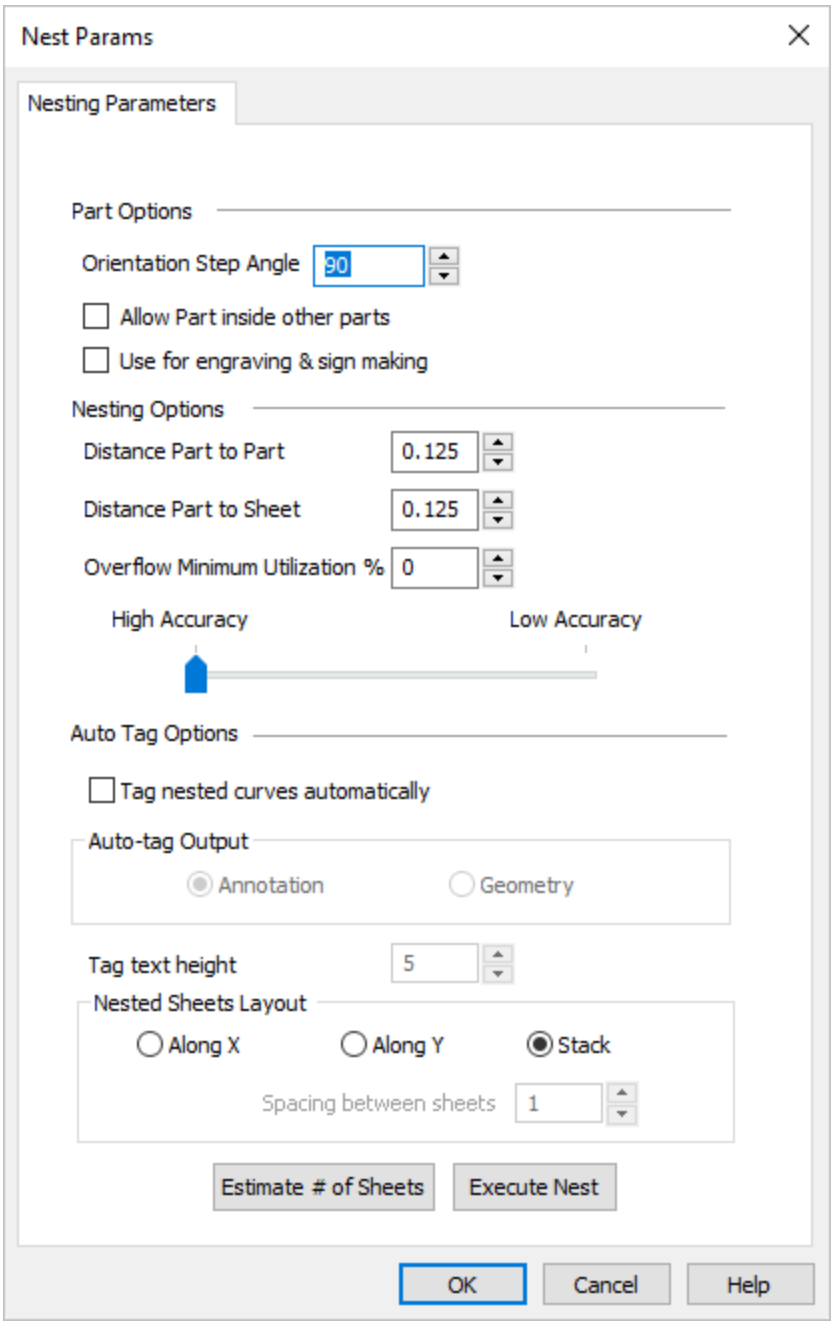


2. From the Nest Parameters dialog make the following adjustments and then pick OK to close the dialog.

   Orientation Step Angle : 90
   Allow part inside other parts : Checked
   Distance Part to Part : 0.125
   Distance Part to Sheet : 0.125
   Accuracy : High
   Tag nested curves automatically : Unchecked

   Note: The distance parameters are measured from the center of the tool (i.e., on the resulting toolpath) NOT from the profile geometry. This will ensure that adequate space is allowed for the tool diameter to pass between parts and the sheet edges.

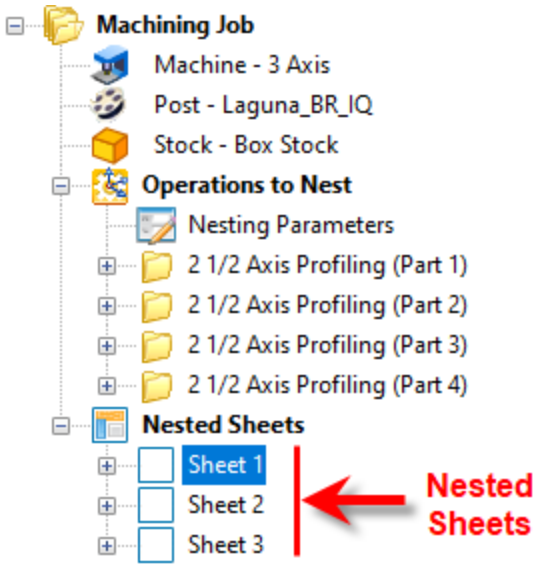
© MecSoft Corporation



For this guide we will not be using the Auto Tag options.

3. Select the Estimate # of Sheets button.
4. From the Estimate # of Sheets dialog pick the Update Sheet Count button and pick OK.
5. Now from the Nest Parameters dialog select the Execute Nest button and then pick OK to close the dialog.
6. You will see that Nested Sheets were created and added to the Machining Job.

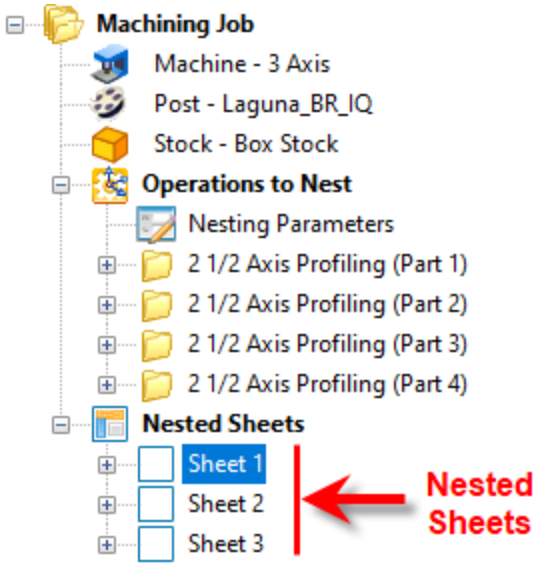
© MecSoft Corporation



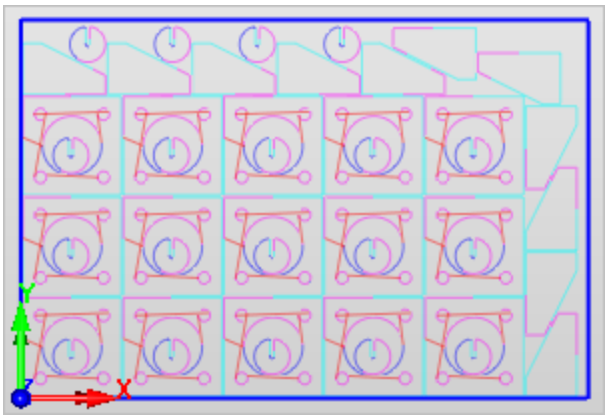
8.2 Reviewing the Nest

While the profile operations were being generated for Parts 1-4 the nested sheets were also created and updated automatically. In this section we will have a look at the resulting nested sheets.

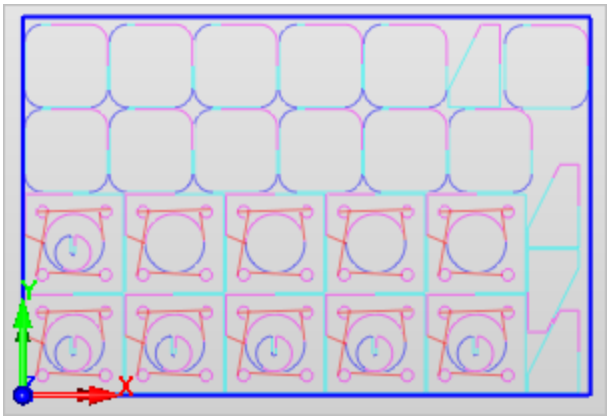
1. The resulting nested sheets are listed below the Operations to Nest in the Machining Job. Expand the nested sheets section to see all of your sheets.



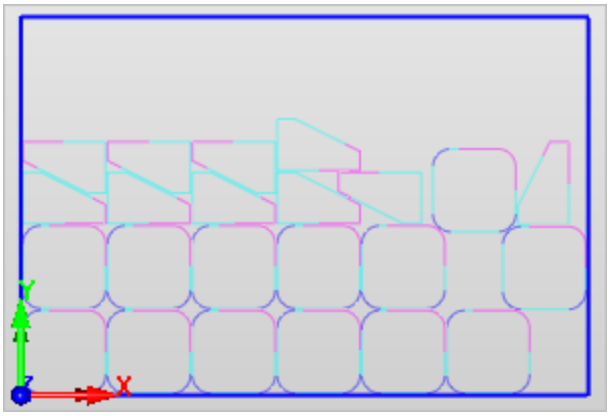
© MecSoft Corporation



Sheet 1



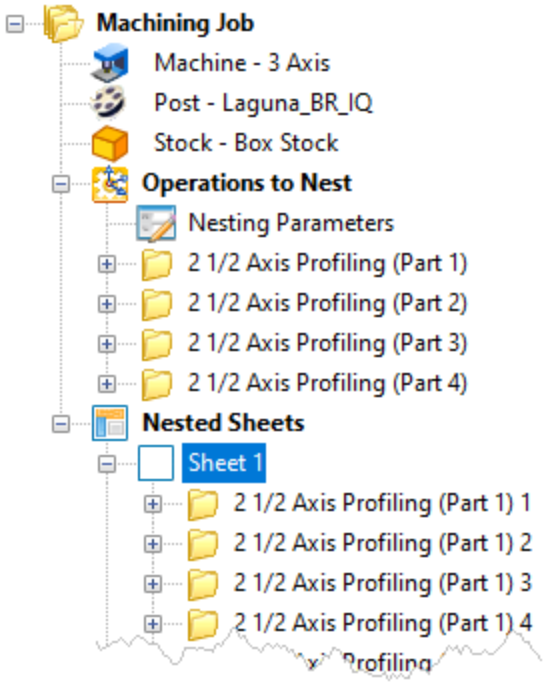
Sheet 2



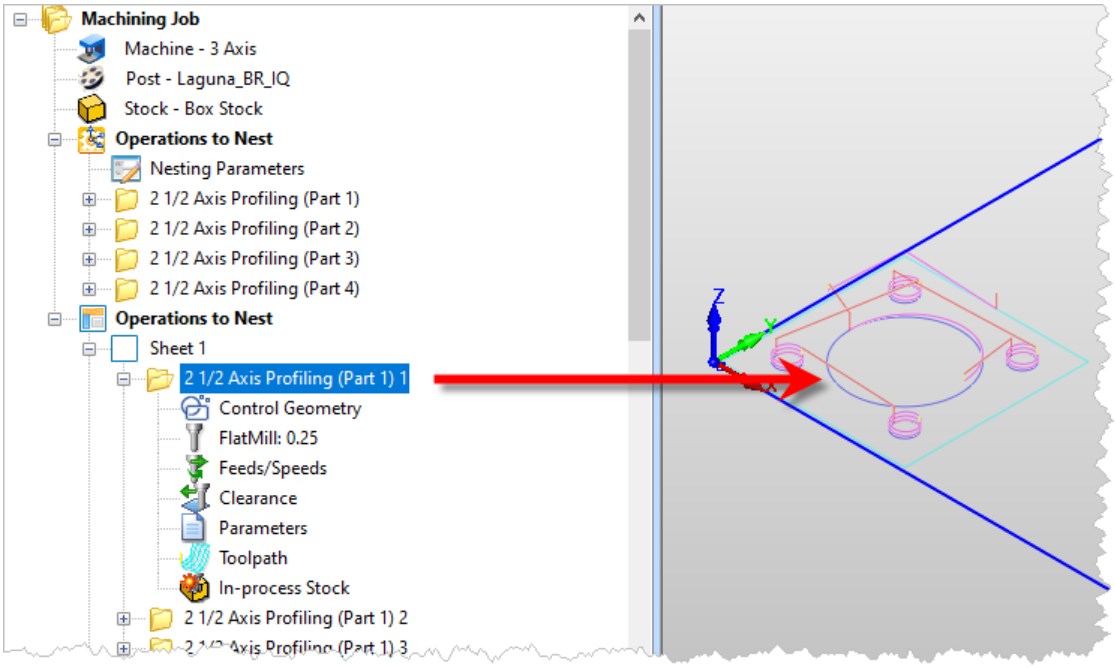
Sheet 3

2. You can also expand each sheet to see all of the Profile operations contained within it.

© MecSoft Corporation



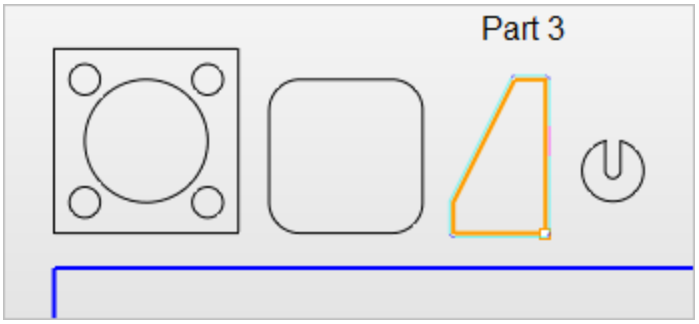
3. You can also expand each Profile operation folder within each Sheet to gain access to that Profile's operation parameters. Each Profile is a self-contained toolpath operation just like the original operation.



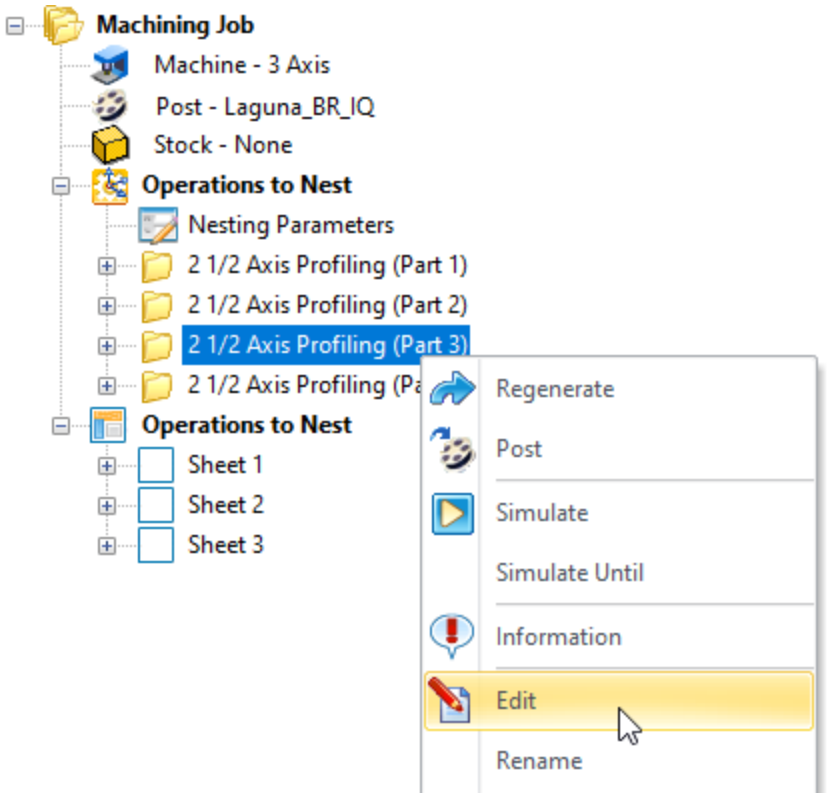
8.3 Editing the Nest

In this section we will discuss ways of editing and updating your nested sheets. It is important to remember that the original Profile operations under Operations to Nest and the resulting nested sheets are associated.

1. You can edit your nest by editing the original Profiling operations located under the Operations to Nest section of the Machining Job. For example, we discovered that Part 3 needs to maintain it's exact orientation as the original Profile.

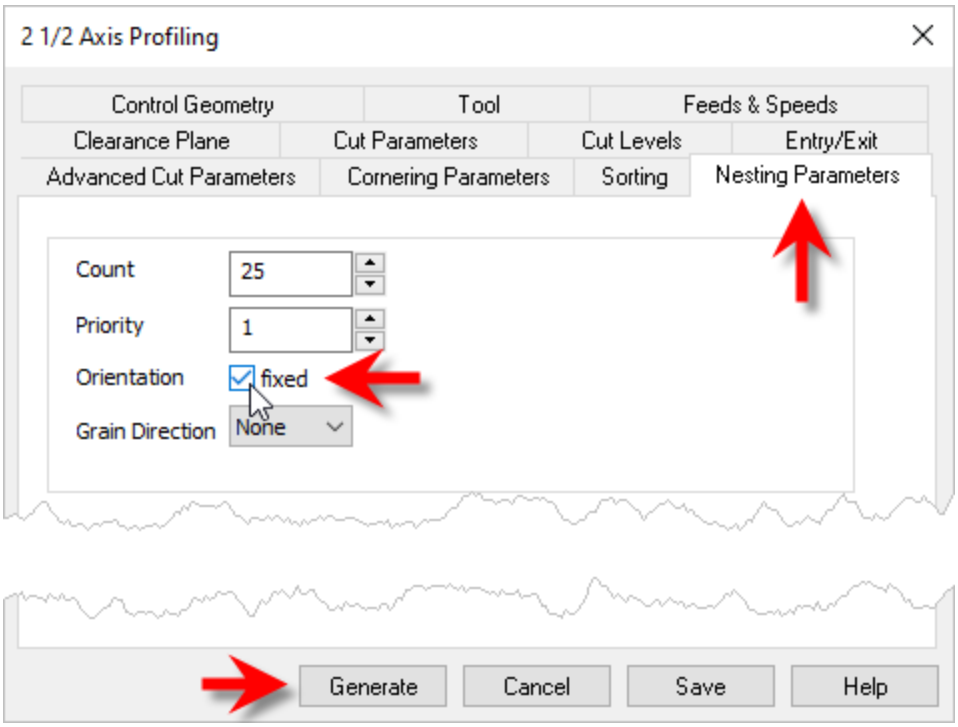


2. You can "*fix*" the orientation of one of your profiles by editing that profile's nesting parameters. Select Part 3 from the Operations to Nest section of the Machining Job and then right-click and select Edit from the menu.

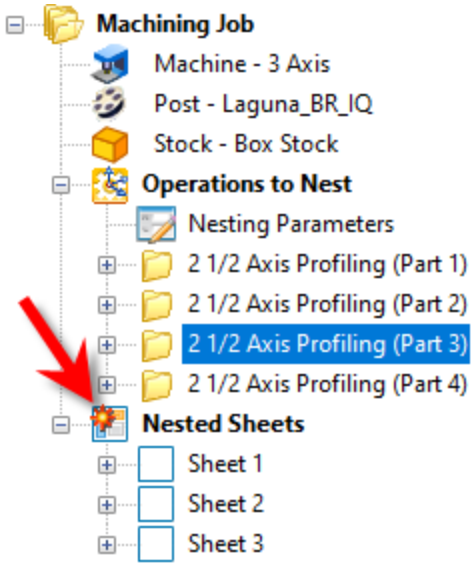


3. From the Profile operation dialog for Part 3, go to the Nesting Parameters tab and check the box next to Fixed and then pick Generate from the bottom of the dialog.

© MecSoft Corporation



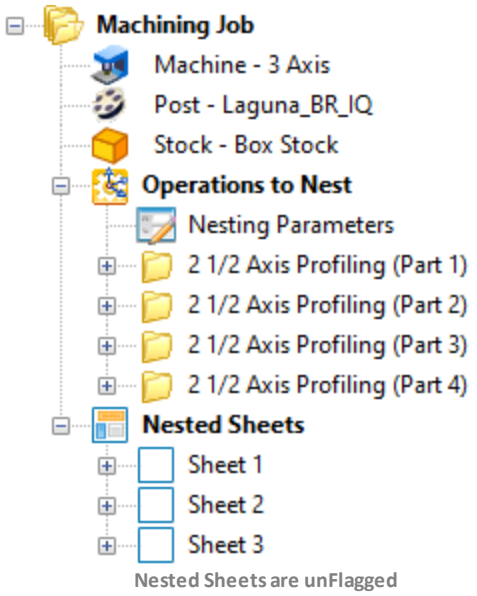
4. Pick OK from the Execute Nest message dialog. You will notice that the Nested Sheets was flagged. This means that the nest needs to be executed again.



Nested Sheets are Flagged

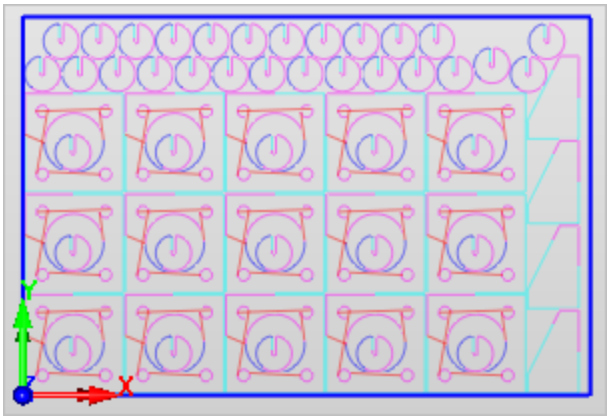
5. Select Nesting Parameters from the Nest tab and then pick the Execute Nest button. See Set Nesting Parameters if you do not remember where the Execute Nest button is located.
6. The flag on Nested Sheets is now gone:

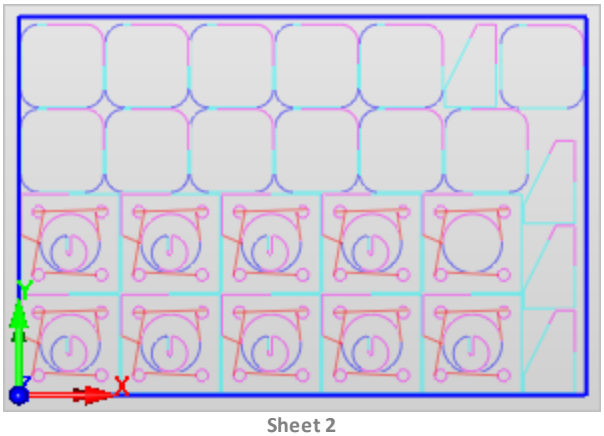
© MecSoft Corporation

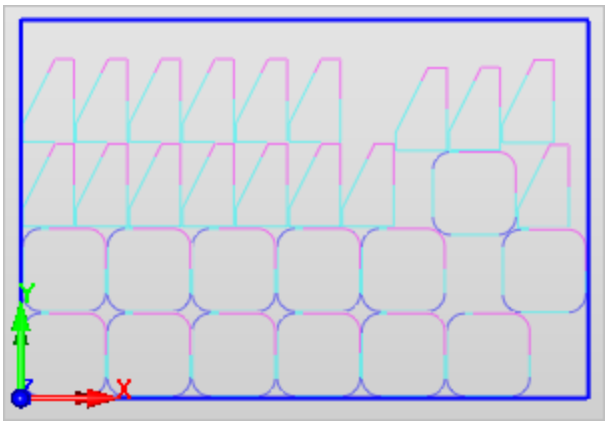


Nested Sheets are unFlagged

7. Select each nested sheet and now see that Part 3 is fixed in the same orientation as the original profile.

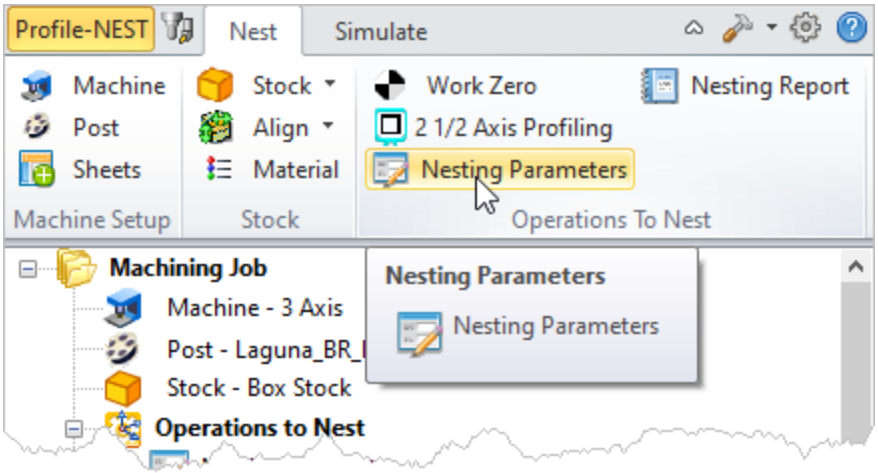

Sheet 1


Sheet 2



Sheet 3

8. Any edits you make to the original Profile operations (including any parameter from any tab) will propagate to the resulting nested sheets.
9. You can also edit the resulting nested sheets my making changes to the Nesting Parameters dialog located on the Nest tab of the Profile-NEST Browser.



8.4 Simulating the Nest

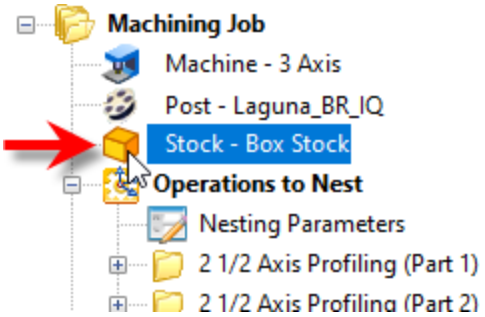
You can perform a cut material simulation of the original Profiling toolpaths or the nested sheets. It is recommended that you first simulate the original toolpaths to verify their individual parameters so that you can edit and regenerate them as needed, making sure to update the nested sheets also. Once you are satisfied, you can then simulate each sheet to verify the nesting parameters.

To Simulate the Original Profiling Toolpaths

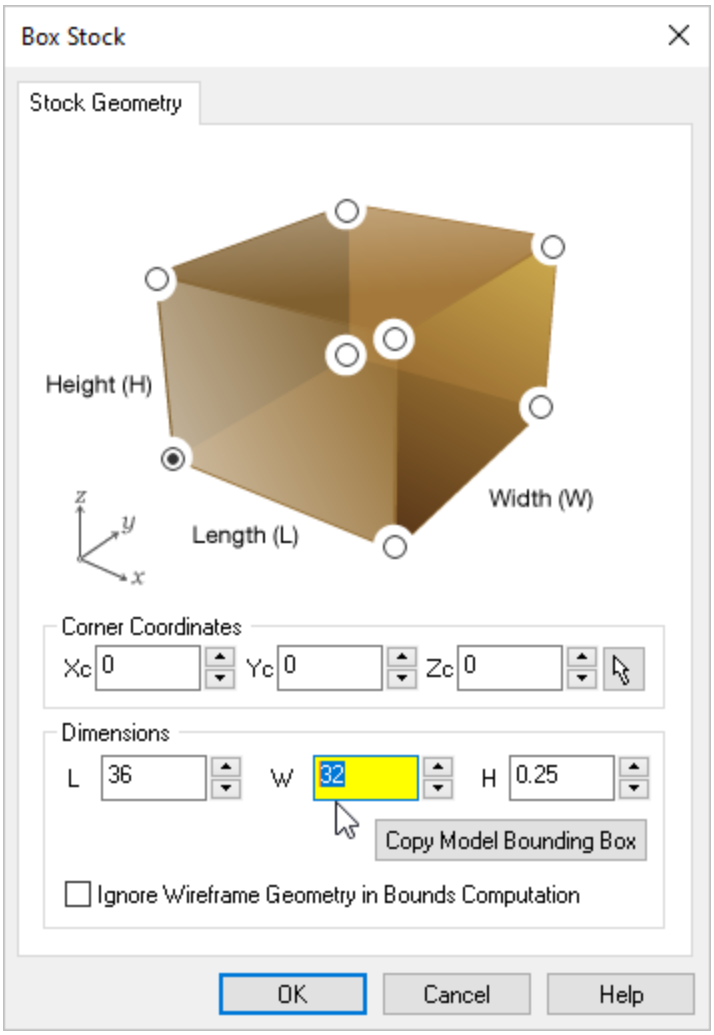
First adjust the Stock size so that it encompasses the original Profiling operations.

© MecSoft Corporation

1. From the Machining Job, double-left-click on the Stock icon to display the Box Stock dialog.

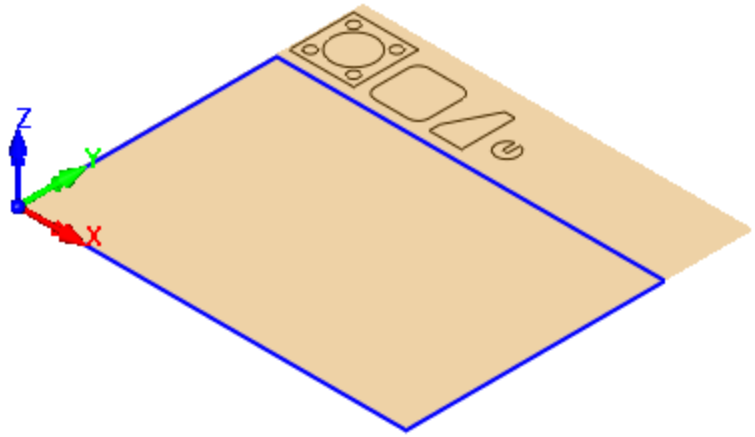


2. From the dialog, change the W dimension to 32 and then pick OK.

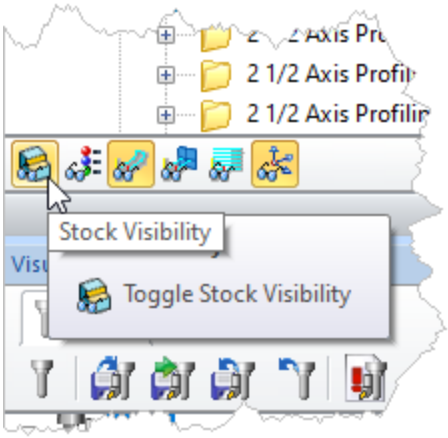


You will see that the Stock displayed on the screen is large enough to encompass the original profile operations.

© MecSoft Corporation

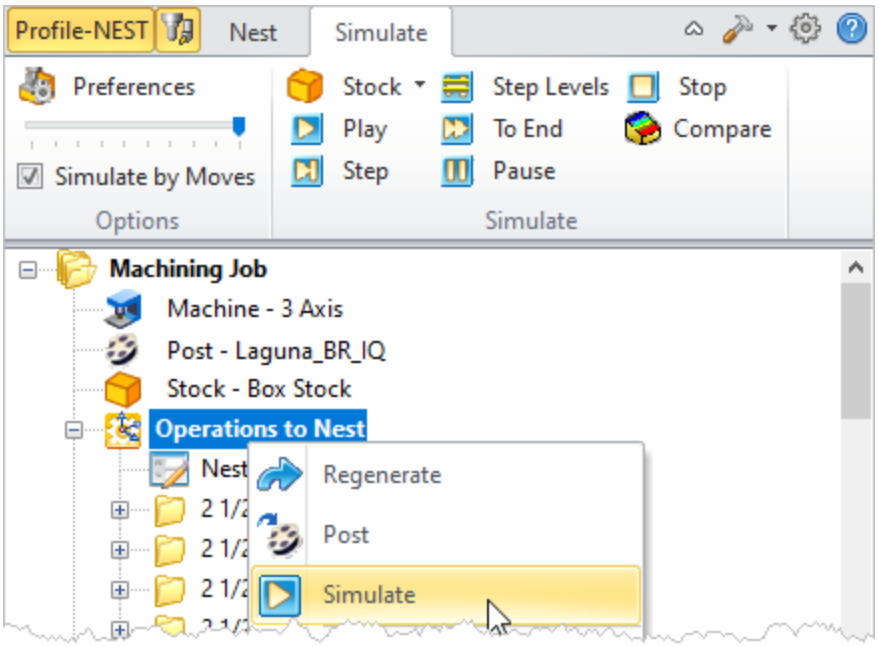


If you do not see the stock on the screen, select the Toggle Stock Visibility icon from the base of the Machining Browser:

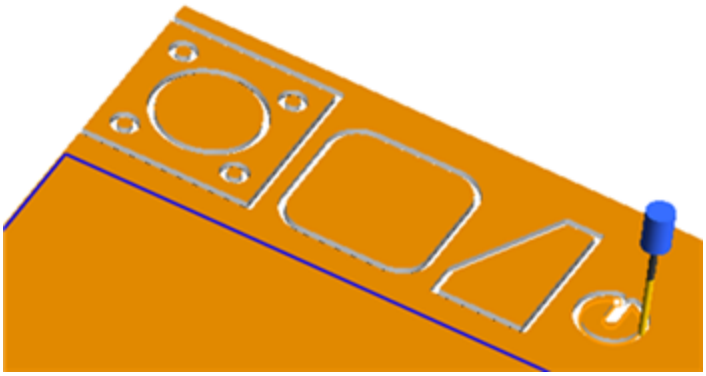


3. Select the Simulate tab.
4. Now select the Operations to Nest folder, right-click and select Simulate.

© MecSoft Corporation



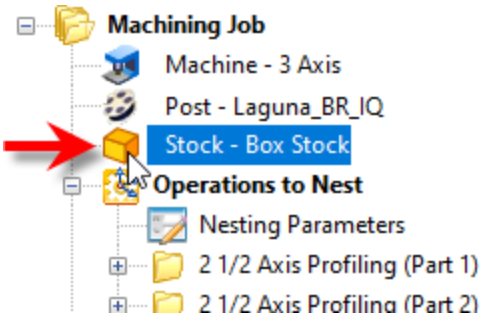
5. Verify the cut material simulation.



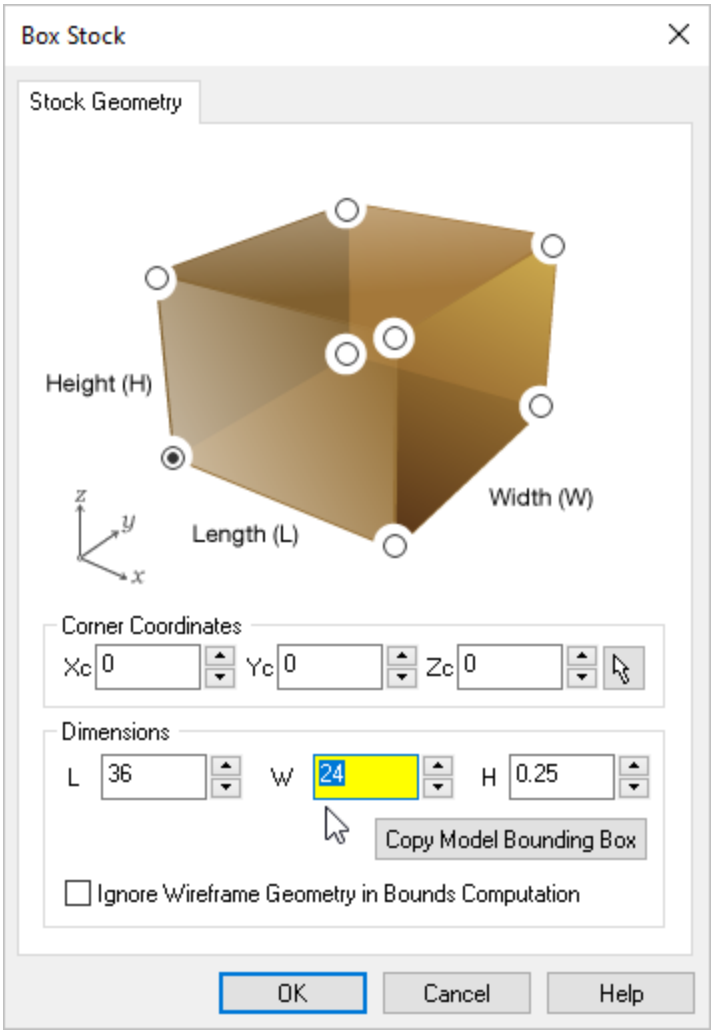
To Simulate a Nested Sheet

First adjust the Stock size back to the sheet size.

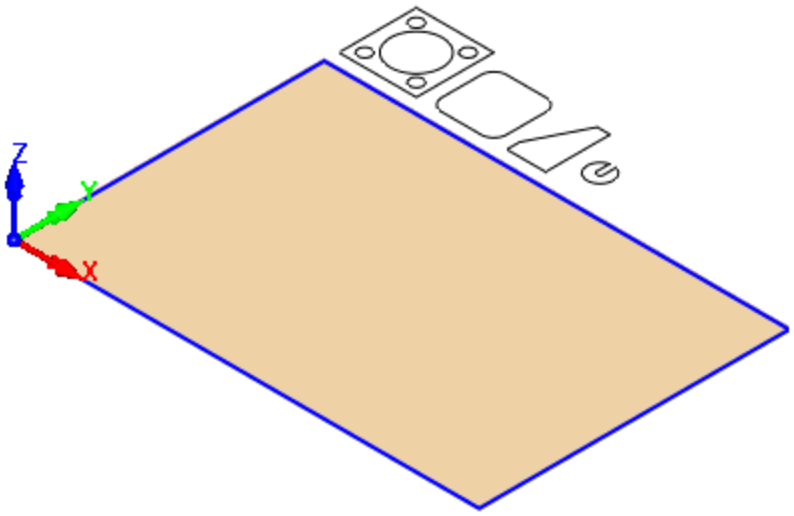
1. From the Machining Job, double-left-click on the Stock icon to display the Box Stock dialog.



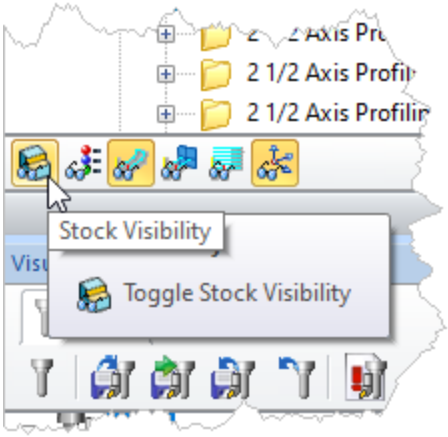
2. From the dialog, change the W dimension to 24 and then pick OK.



You will see that the Stock displayed on the screen is large enough to encompass the original profile operations.

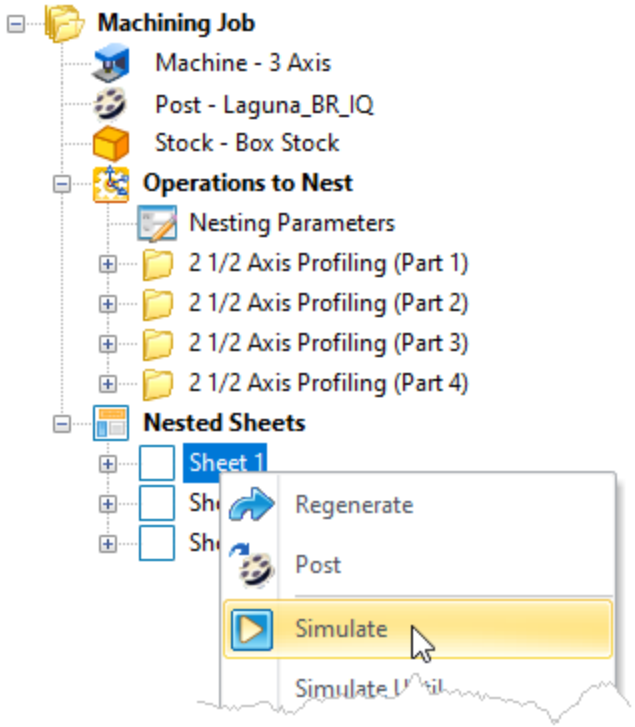


If you do not see the stock on the screen, select the Toggle Stock Visibility icon from the base of the Machining Browser:

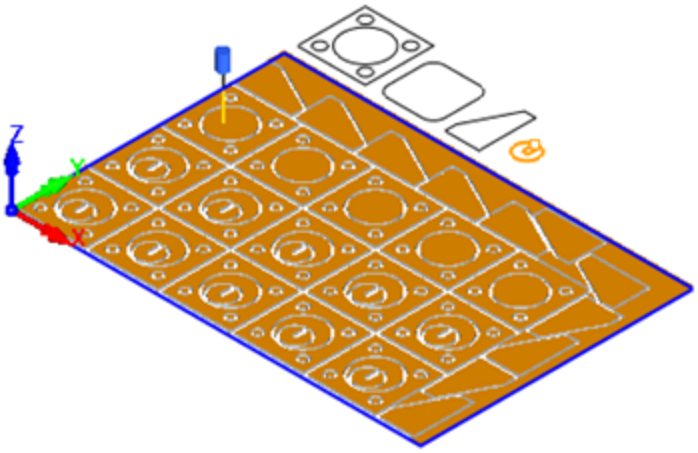


3. Select the Simulate tab.
4. Now select the Nested Sheet, right-click and select Simulate.

© MecSoft Corporation



5. Each nested operation will simulate until the entire sheet to complete. Note that the Nested Sheet folder in the Machining Job will expand and collapse as each operation is activated and simulated. This allows you to identify exactly which nested operation is currently being simulated.

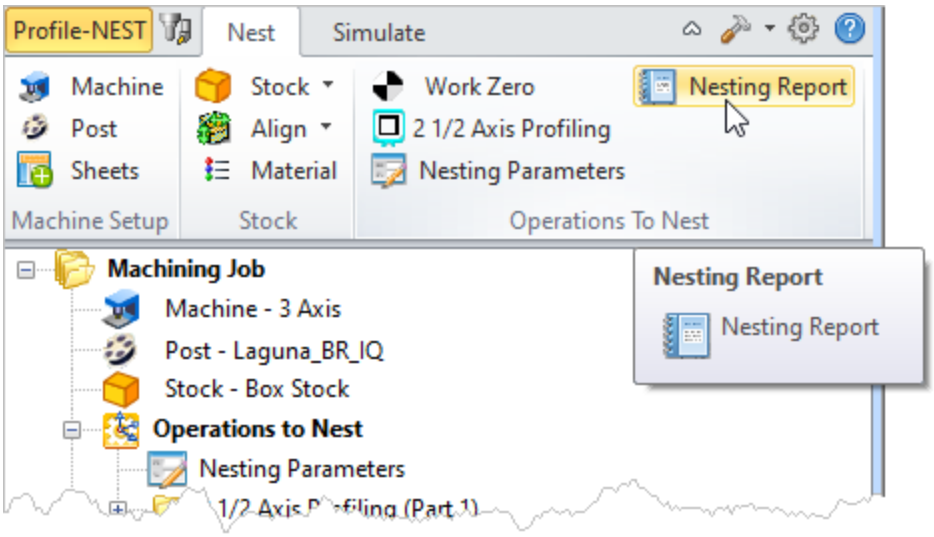


8.5 Generate Reports

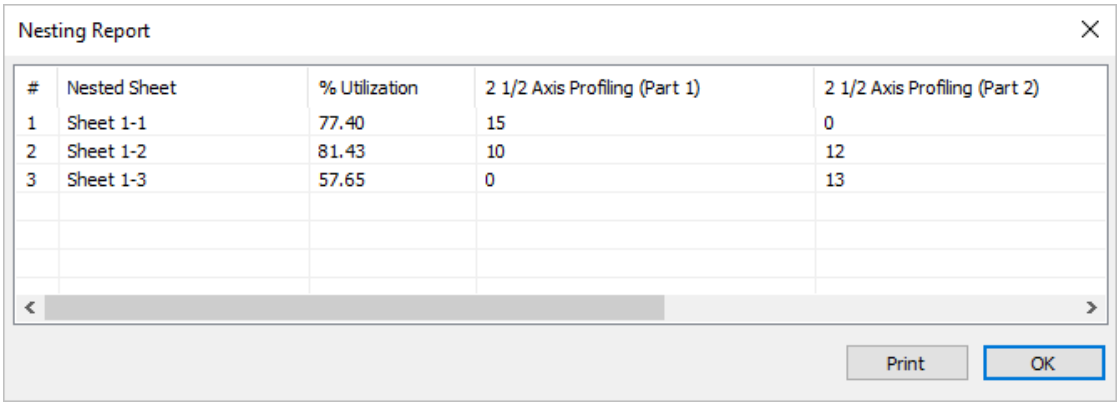
8.5.1 Nesting Reports

At any time you can access a Nesting Report of the current Profiling operations.

1. From the Nest tab select the Nesting Report icon to display the report.



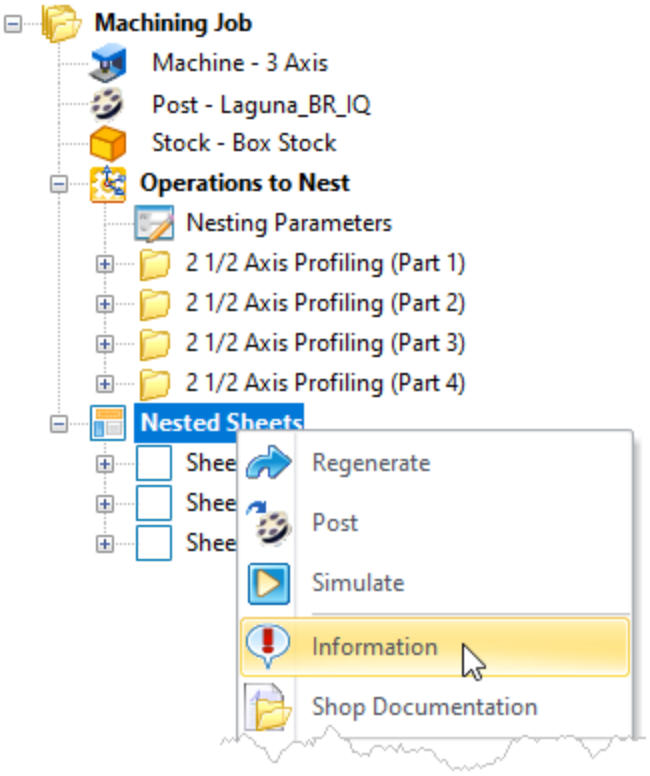
2. The report provides information about each nested sheet including the % Utilization and the total number of each Profiling operation located on each Nested Sheet. You can Print the report if desired.



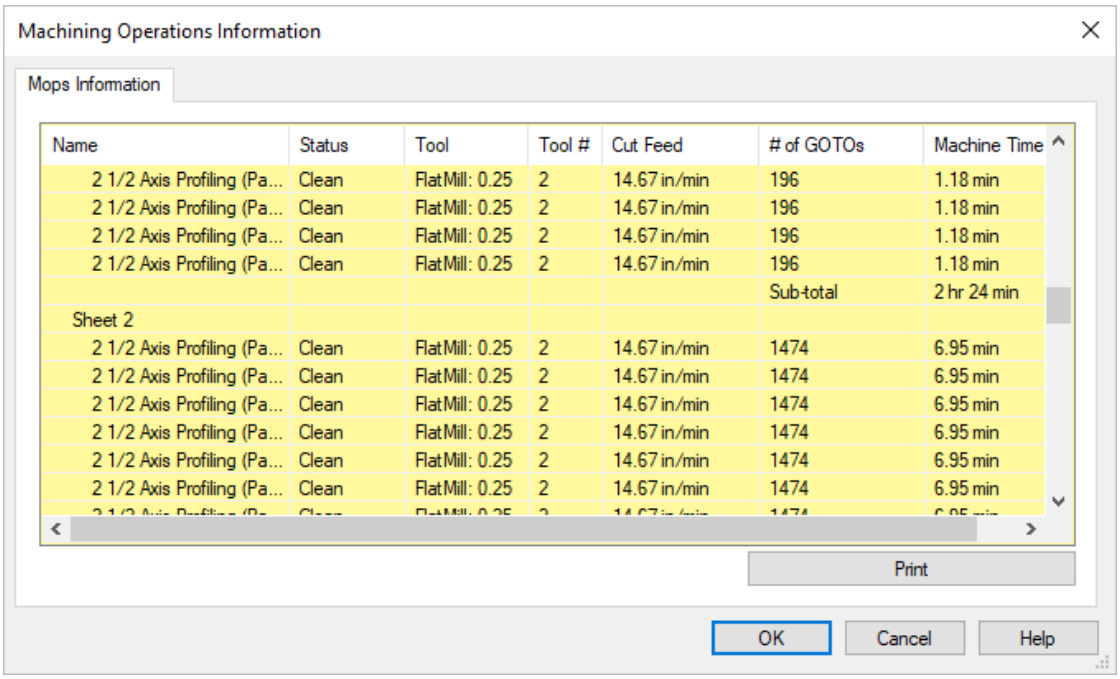
8.5.2 Machining Reports

At any time you can access a Machining Information report of any Profile operation or any Nested Sheet or all Nested Sheets.

1. From the Machining Job select the item and then right-click and select Information to generate the report.



2. The report provides information about each operation including Status, Tool Name, Tool #, Cut Feed Rate, Size and Estimated Machining Time. You can Print the report if desired.

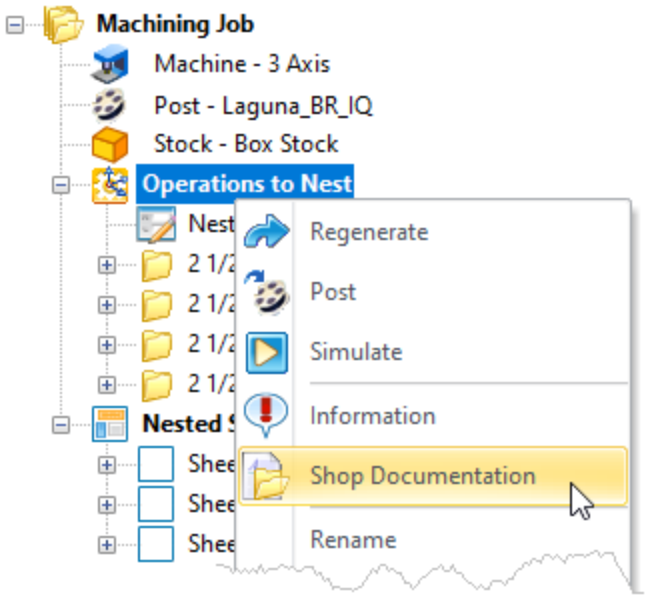


© MecSoft Corporation

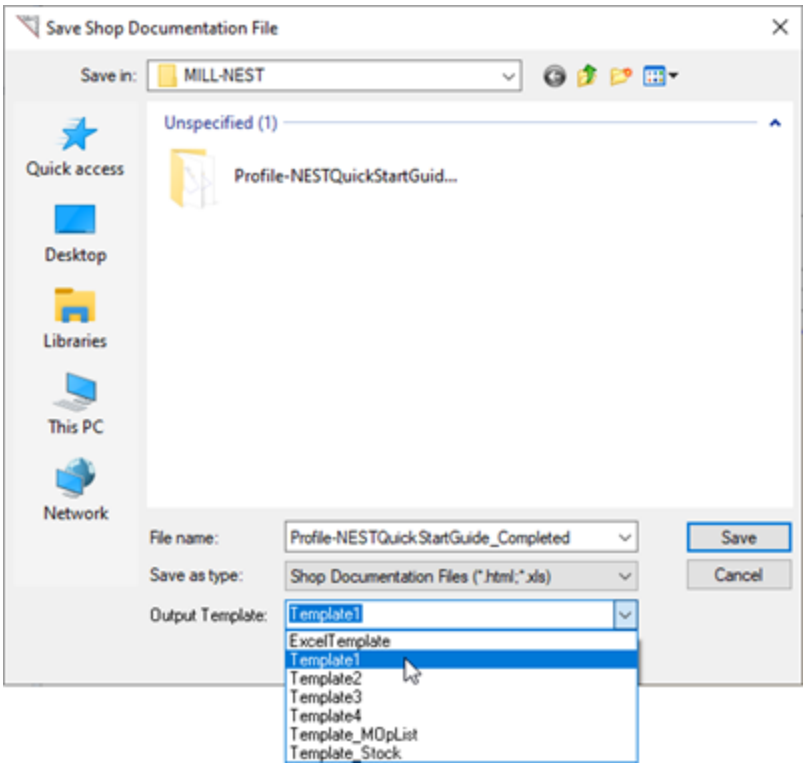
8.5.3 Shop Documentation

At any time you can create an HTML shop documentation setup sheet that contains information about the operations to nest.

1. From the Machining Job select Operations to Nest and then right-click and select Shop Documentation.



2. From the Save Shop Documentation File dialog, select the Output Template to use. By default the part file name is used as the document name.

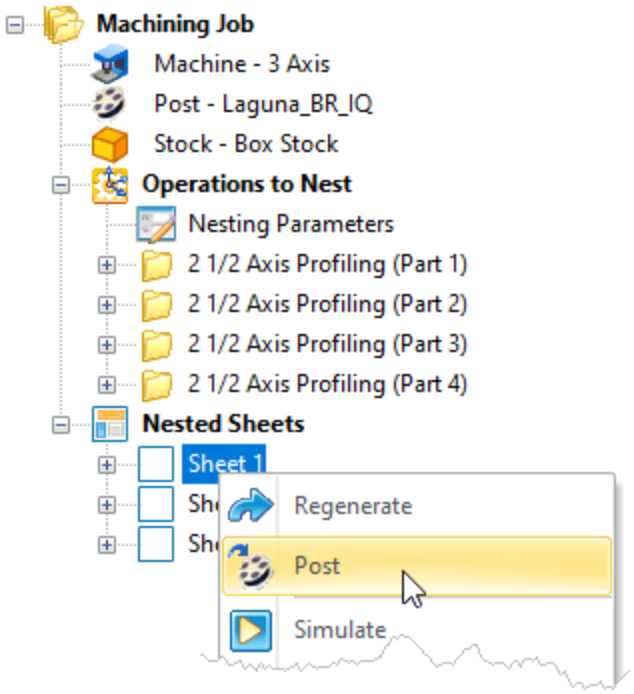


3. Pick Save to generate the Shop Documentation.

8.6 Post G-Code

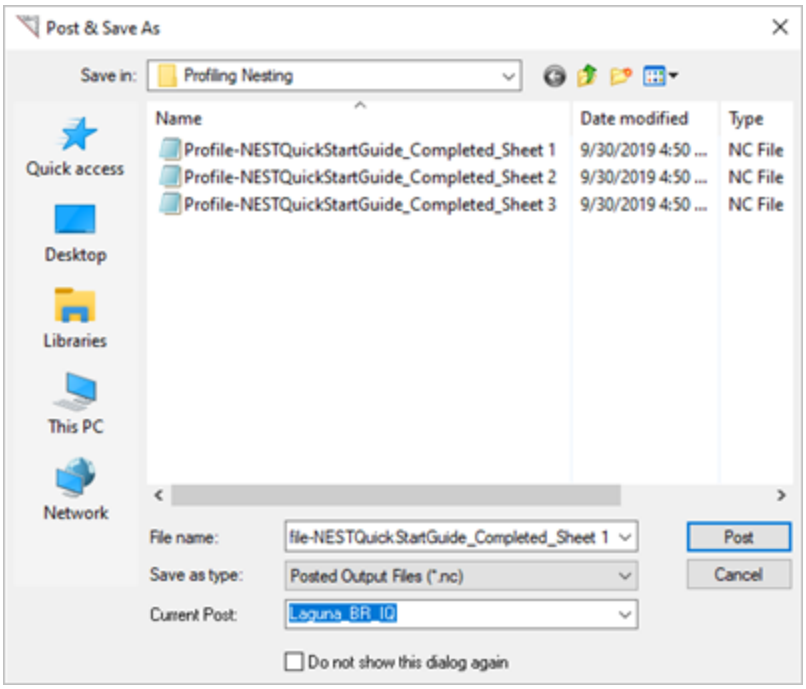
At any time you can post G-Code files from any operation located under Operations to Nest or any Sheet or operation under Nested Sheets.

1. From the Machining Job select an operation to post-process. You can select one operation, multiple operations or all Operations to Nest or one or more Nested Sheets.

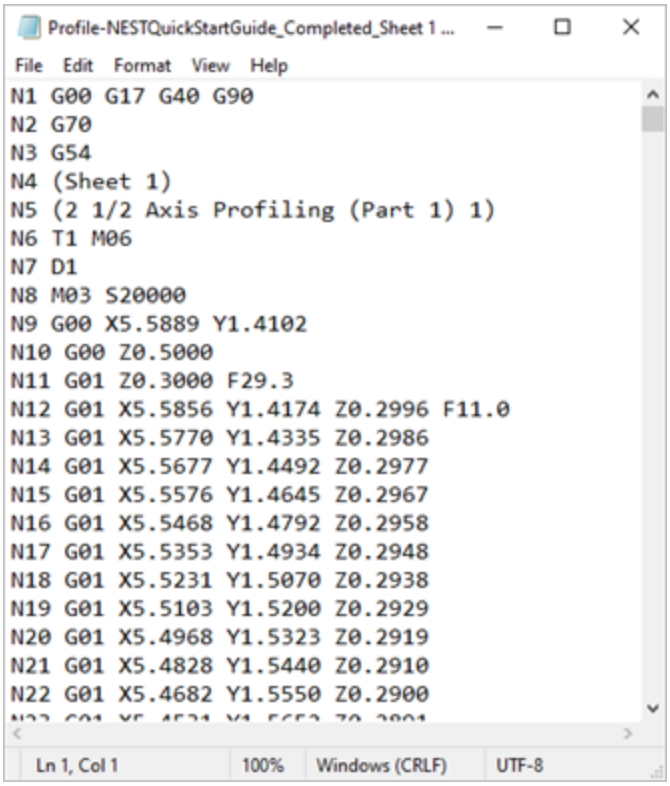


2. From the Post & Save As dialog, you can select or change the Current Post by dropping down the list and making a selection from the current list of posts.

© MecSoft Corporation



3. Pick Post to generate the G-Code file and display it in notepad.

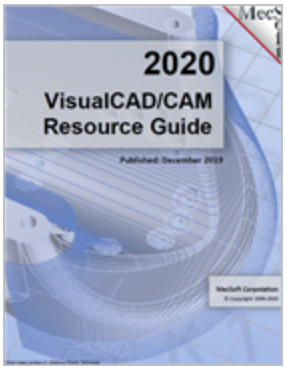


© MecSoft Corporation

Where to go for more help

Download this PDF Guide for a list of the available VisualCAM Resources.

2020 VisualCAM Resource Guide



The VisualCAM 2020 Resource Guide

18 Pages

Lists PDF downloads and Online resources including Quick Start Guides, Reference Guides, Exercise Guides, Tutorials and More.

Click Here to download this guide!

Index

- 2 -

2½ Axis Profiling
  Information Report 85
  Post G-Code 89

- A -

About
  the MILL Module 10
  the VisualCAM Display 12
  This Guide 9
  using this Guide 10
Align Part and Stock 29

- C -

Create Stock Geometry 27
Create Tools 40

- D -

Define the Machine Tool 21

- G -

Generating Reports 85
Getting Ready
  Load the Part Model 16
  Machining Strategy 18
  Main Programming Steps 19

- L -

Learning Resources 5
Load the Part Model 16
Load the VisualCAM MILL Module 13

- M -

Machine
  Define 21
Machining Strategy 18
Machining Time 85
Main Programming Steps 19

- P -

Post
  Create G-Code 89
  Select 21
Print Media Archive 8, 91

- Q -

Quick Start Guides 5

- R -

Running VisualCAM 12

- S -

Select the Post Processor to use 21
Setup Sheet 87
Shop Documentation 87
Specify Material 31
Stock
  Align with Part 29
  Create Box Stock 27
  Specify Material 31

- T -

Tool
  Create 40

- U -

Useful Tips 9
Using this Guide and Associated Part Files 10

- V -

Videos & Guides 5

© MecSoft Corporation

- W -

Where to go for more help 91

© MecSoft Corporation