

Quick Start Guide

VisualCAD/CAM 2026

Published: March 2026

MecSoft Corporation

© Copyright 1998-2026

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!.....	10
Getting Ready	11
1 Running VisualCAD/CAM.....	11
2 About the VisualCAM Display.....	11
3 Load the Profile-NEST Module.....	12
4 Load the Part Model.....	15
5 Machining Strategy.....	17
6 Main Programming Steps.....	18
7 Operational Sequence.....	18
The Machine Setup	20
1 Define the Machine Tool.....	20
2 Select the Post Processor.....	20
3 Define Sheet to Nest.....	22
The Stock	26
1 Create Stock Geometry.....	26
2 Align Part and Stock.....	28
3 Specify Material.....	30
The Profiles to Nest	33
1 Profile Part 1.....	33
Profile 1 Control Geometry tab	34
Profile 1 Nesting Parameters tab	37
Profile 1 Tool tab	39
Profile 1 Feeds & Speeds tab	45
Profile 1 Clearance Plane tab	46
Profile 1 Cut Parameters tab	46
Profile 1 Cut Levels tab	48
Profile 1 Entry/Exit tab	49

Profile 1 Advanced Cut Parameters tab	51
Profile 1 Cornering Parameters tab	52
Profile 1 Sorting tab	53
Profile 1 Generate	53
2 Profile Parts 2,3 & 4.....	55
Clone Profile Operations	55
Edit Profile Operations	57

The Nested Sheets

61

1 Set Nesting Parameters.....	61
2 Reviewing the Nest.....	64
3 Editing the Nest.....	67
4 Simulating the Nest.....	71
5 Generate Reports.....	76
Nesting Reports	76
Machining Reports	77
Shop Documentation	78
6 Post G-Code.....	80

Where to go for more help

82

Index

83

Quick Start



MILL Module 2026

[Prefer Printed Documentation? Click Here!](#)

[What's New](#) | [Quick Start Play List](#)

[Quick Start Guides](#) for each [VisualCAD/CAM](#) module are available in both PDF and Video format. Refer to the following information to access these resources:



How to Access the Quick Start Guide Documents

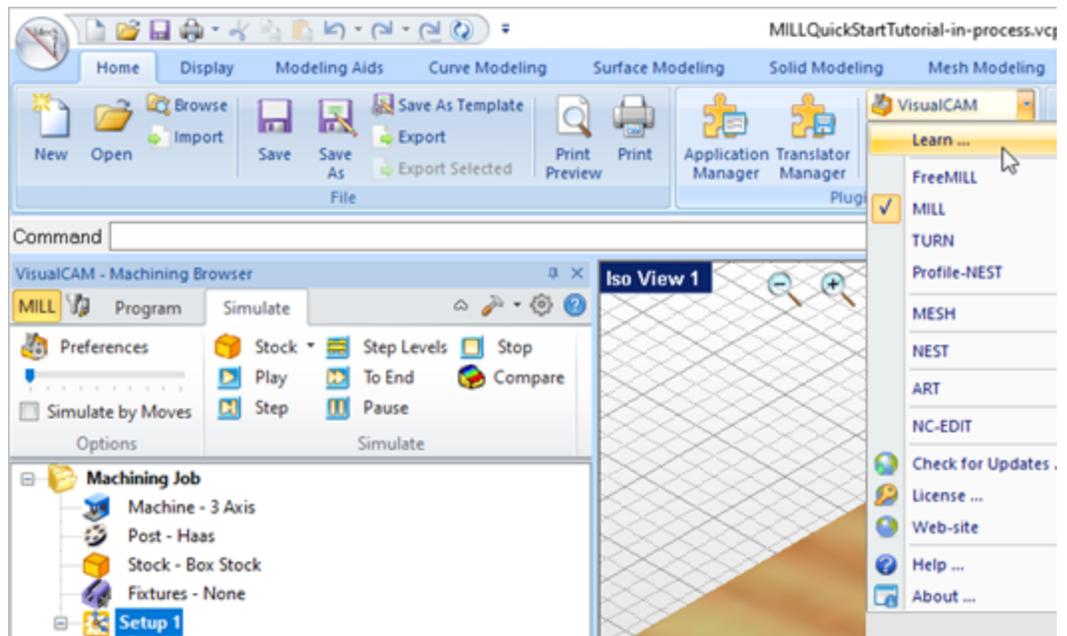
To help you quickly get started in working with each module, select one of the Help buttons located on the [VisualCAD/CAM Learning Resources](#) dialog.

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 dialog 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

Welcome to VisualCAM 2026!
To get started as quickly as possible use the following resources.

CAM **MILL** **TURN**

What's New? Web Resources On-Line Help Quick Start On-Line Help Quick Start

Profile-NEST **G-Code Editor**

On-Line Help Quick Start On-Line Help Quick Start

NEST **ART**

On-Line Help Quick Start On-Line Help Quick Start

Where to go for more help:
To get your specific support questions answered click [here](#) to fill out a support form.
For all other information visit our web-site www.mecsoft.com

Open Quick Start Files Folder

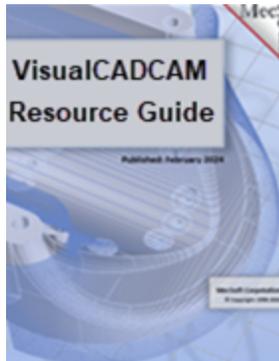
Learning Resources Dialog

Resource Guide

Download this PDF Guide for a list of the available [VisualCAD/CAM Resources](#).



2026 VisualCAD/CAM Resource Guide



The 2026 VisualCAD/CAM Resource Guide!

18 Pages

Lists PDF downloads and Online resources including [Quick Start Guides](#), [Reference Guides](#), [Exercise Guides](#), [Tutorials](#) and [More](#).

[Prefer Printed Documentation? Click Here!](#)

[What's New](#) | [Quick Start Play List](#)

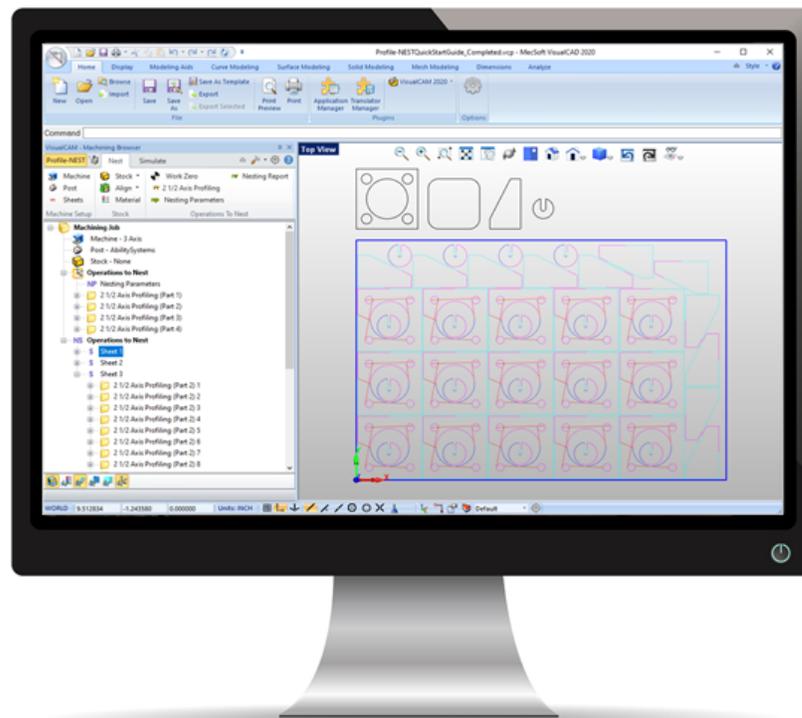
About this Guide



MILL Module 2026

[Prefer Printed Documentation? Click Here!](#)

[What's New](#) | [Quick Start Play List](#)



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 **VisualCAD/CAM** 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 **VisualCAD/CAM** 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 **VisualCAD/CAM** 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 **VisualCAD/CAM** 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 **VisualCAD/CAM**. 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.

3.4 Watch the Video!

Want to see a video demonstration of this quick start guide? Just click on the play list below and watch the Profile-NEST Quick Start Guide video.

[Here is a link to the complete 2026 Video Play List](#)

Getting Ready

4.1 Running VisualCAD/CAM

Locate the [VisualCAD/CAM 2026](#) 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 [VisualCAD/CAM 2026](#). (The name of this program group will usually be called [VisualCAD/CAM 2026](#), unless you specified otherwise during setup.)

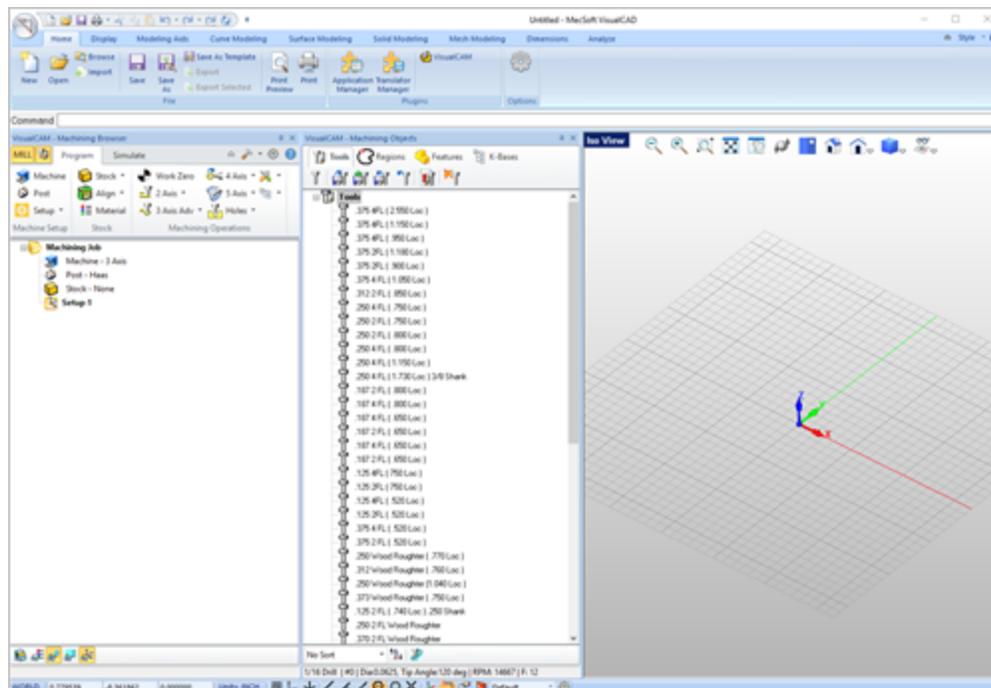
Once you locate the program group, select it and then select [VisualCAD/CAM 2026](#) to launch the application.

If the installation was successful, upon launching of [VisualCAD/CAM 2026](#) you should observe a menu entry called [VisualCAM 2026](#) on the [Home Ribbon Bar](#) menu of [VisualCAD](#).

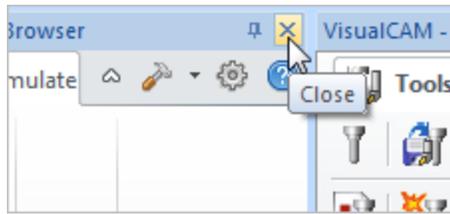
If you do not see this menu entry then please check the [On Line Help](#) document of the product (found in the installation folder) for help with trouble shooting the installation.

4.2 About the VisualCAM Display

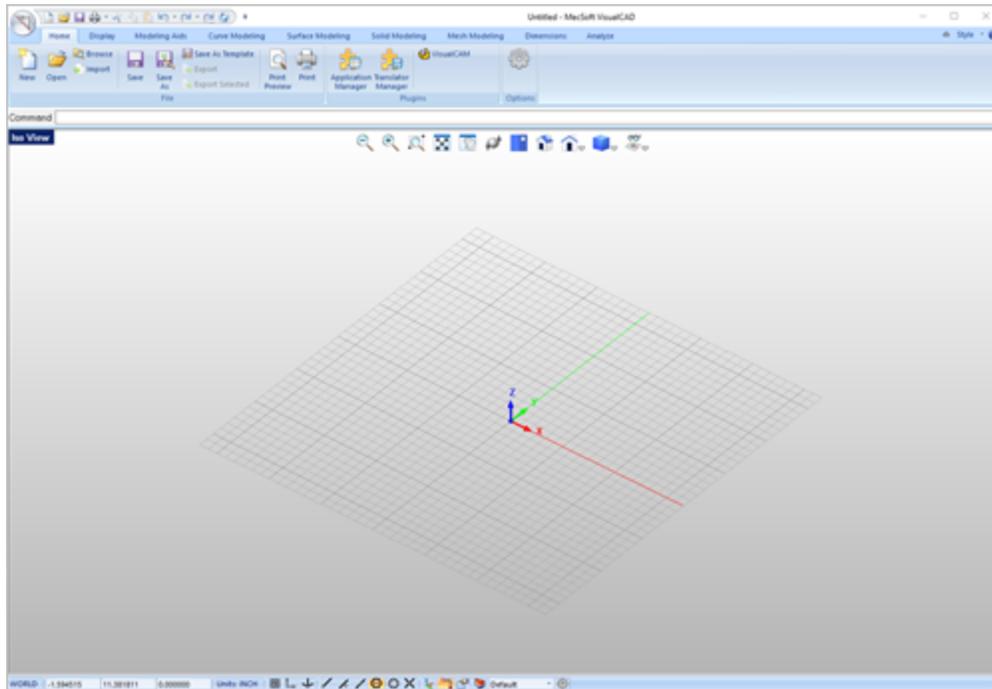
Before we begin, let's talk a bit about the [VisualCAD](#) display. When you run [VisualCAD](#) for the very first time, your screen may look this.



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



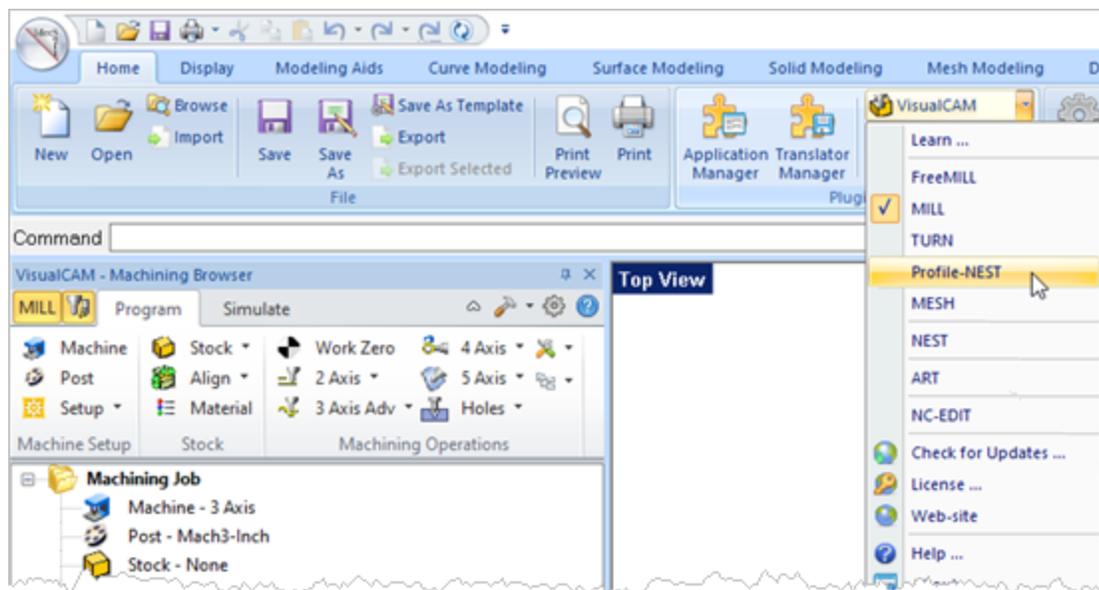
With all plug-in modules closed your screen will look like this:



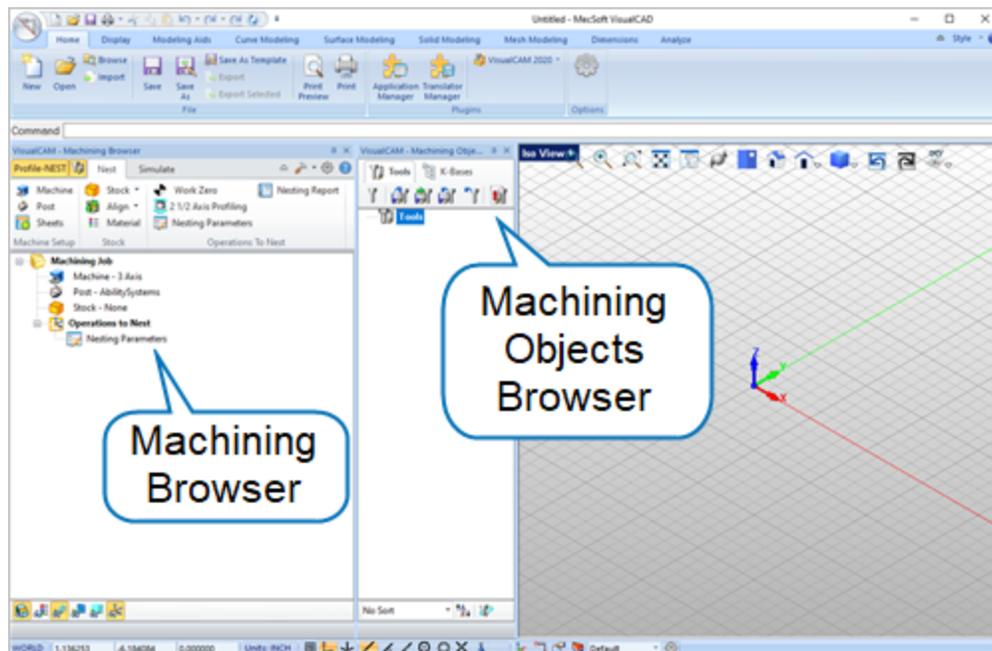
4.3 Load the Profile-NEST Module

Now, let's begin by launching the [VisualCAD/CAM](#) module.

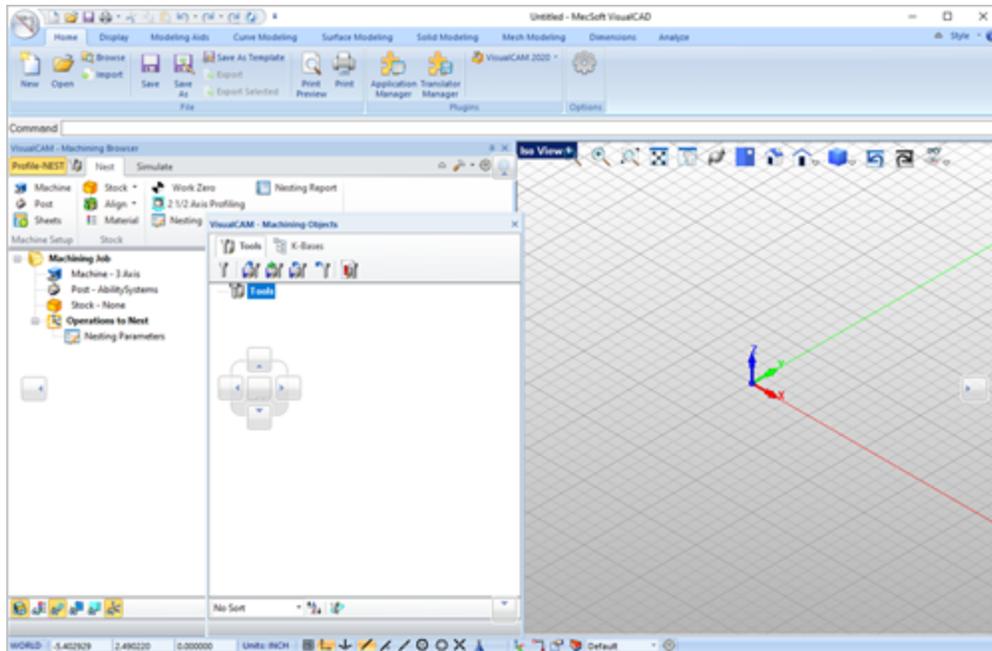
1. From the [Plugins](#) pane of [VisualCAD's Home Ribbon Bar](#), you will see the [VisualCAM 2026](#) main menu item.
2. Drop-down the menu and pick [Profile-NEST](#) to load the module.



3. Docked on the left you will see the [Machining Browser](#) and the [Machining Objects Browser](#). When you first run [VisualCAD/CAM](#), these two browsers may be docked side by side. However, you can move them anywhere on the screen that feels comfortable for you.



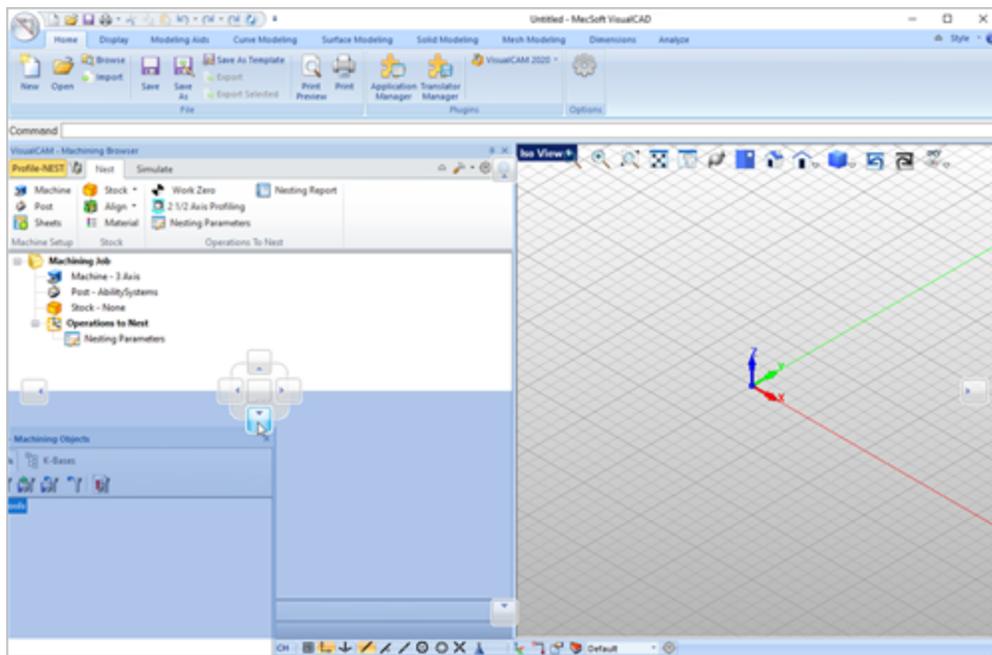
4. For example, let's move the [Machining Objects Browser](#) so that it displays under the [Machining Browser](#) on the left. Simply left-click and hold the title bar of the browser and drag it around on your screen.



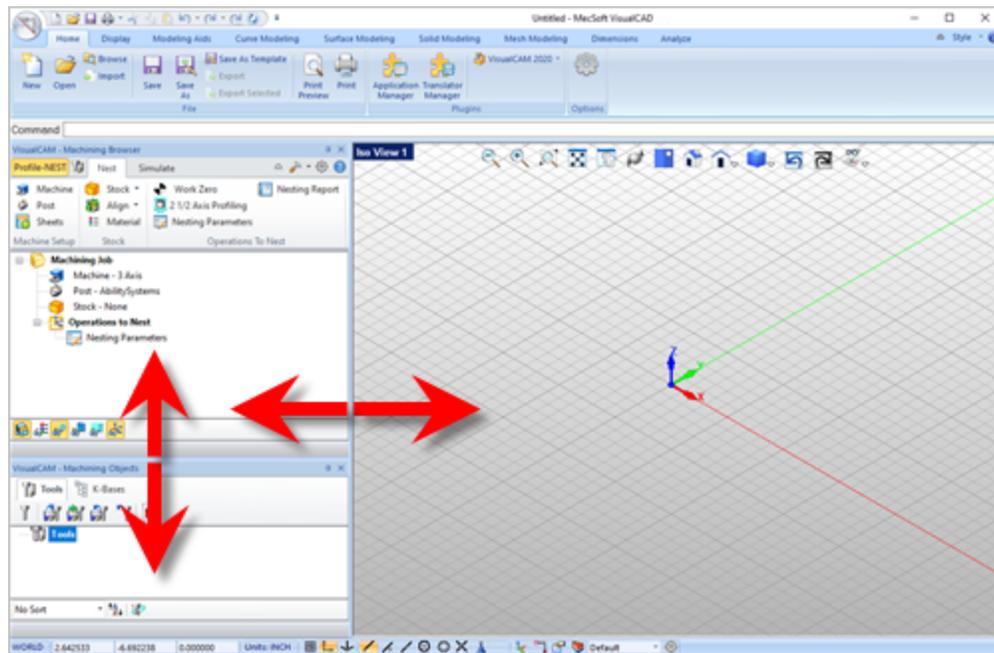
While doing so, you will see the docking widget display in the background with directional buttons allowing you to choose screen locations relative to the active window.

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

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



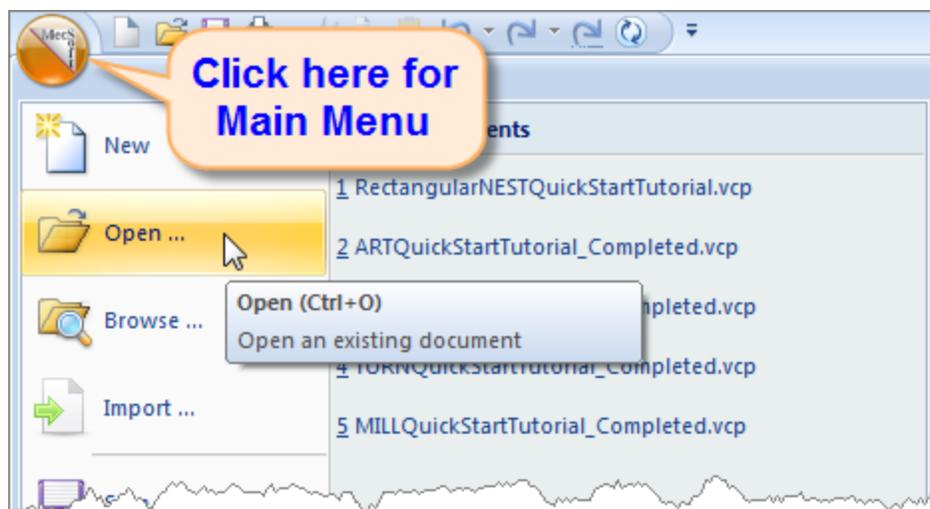
- You can also re-size the height and width of each browser making sure that all of the command icons and menus are easily accessible.



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.

- From VisualCAD's Main Menu, select Open.

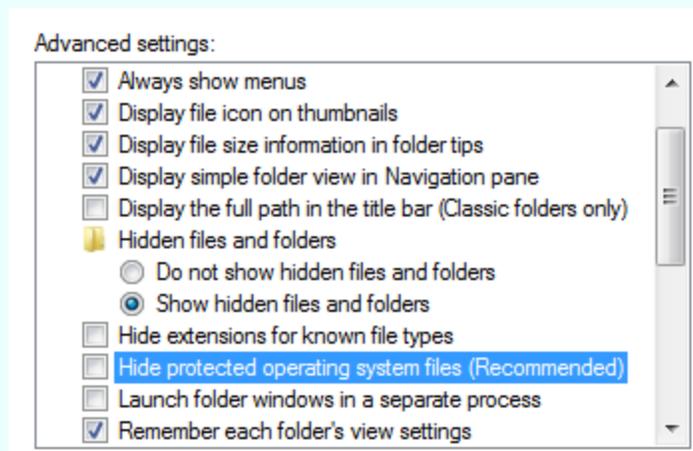


- From the Open dialog box, select the Profile-NESTQuickStartGuide.vcp file from the C:\ProgramData\MecSoft Corporation\VisualCAM 2020\QuickStart\ folder. As mentioned

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.

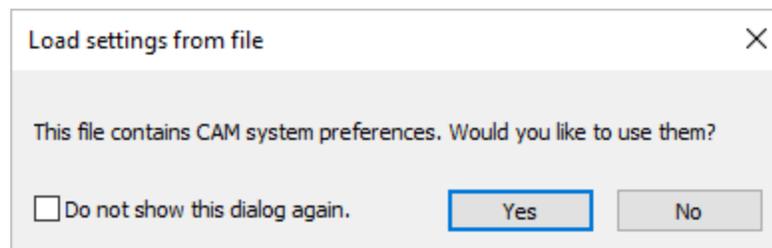
 By default, the `ProgramData` folder is "hidden" from view. Here are the steps to Show hidden files and folders:

1. For **Windows 8** users: Go to **Control Panel > Appearance and Personalization > Folder Options**.
For **Windows 11** 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)

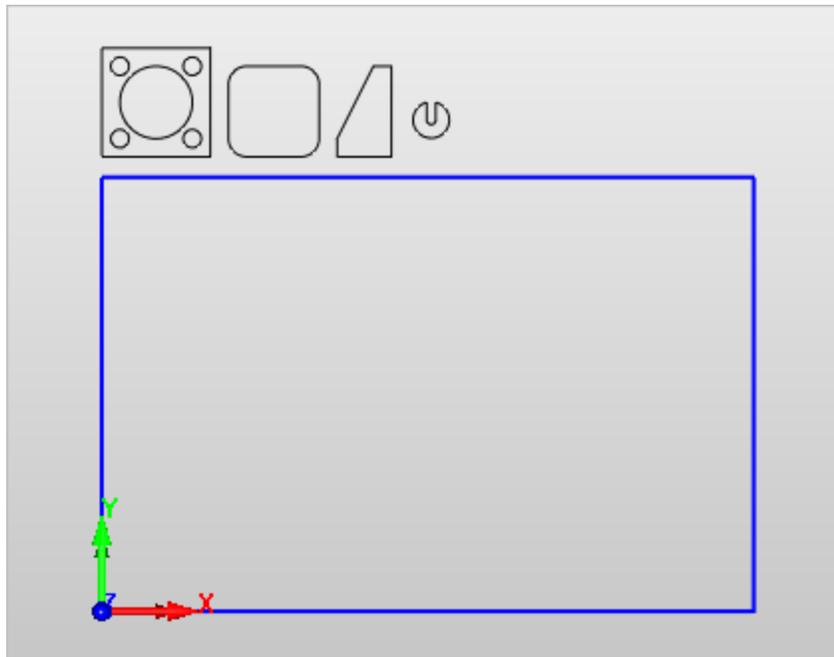


3. Click **Apply** and **OK**.

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.

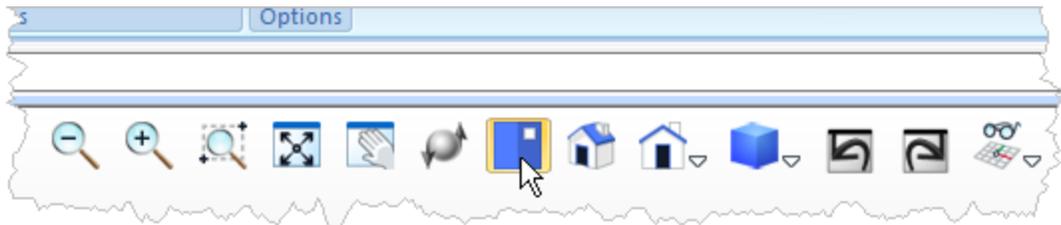


The part appears as shown below



Profile-NESTQuickStartGuide.vcp

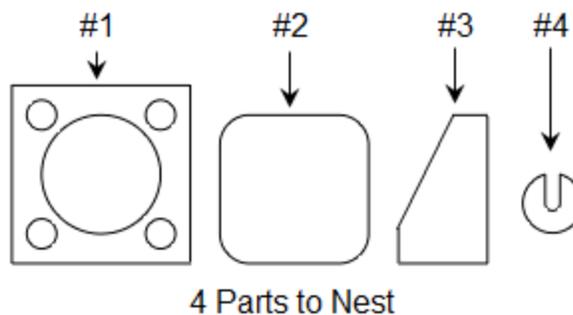
3. From the **View** toolbar, select the **Top View** to work in.



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.



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.

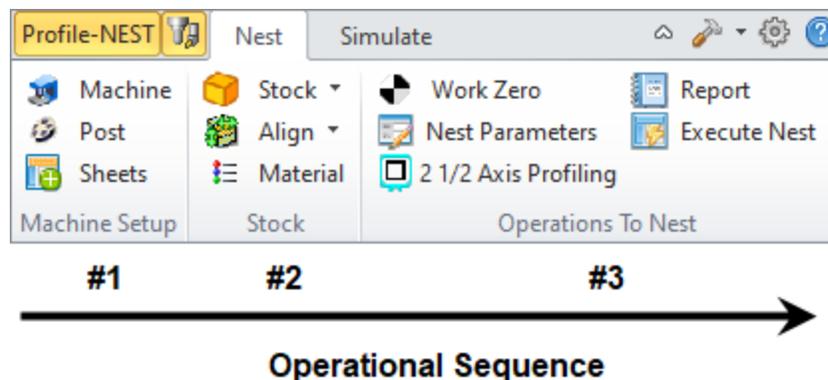
4.6 Main Programming Steps

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:

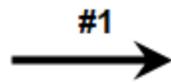
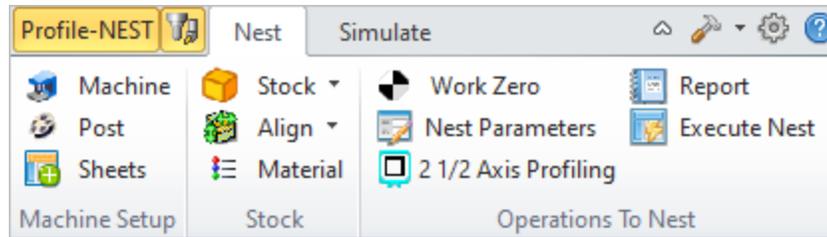


1. From the [Machine Setup](#) section we will be defining the [Machine](#), [Post-Processor](#) and [Sheet](#) definition.
2. From the [Stock](#) section we will be defining our [Sheet Stock](#), setting our [Stock Alignment](#) and setting the [Stock Material](#).
3. From the [Operations to Nest](#) 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](#)

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

The Machine Setup

We will begin with the [Machine Setup](#) section of the [Nest](#) tab of the [Machining Browser](#).

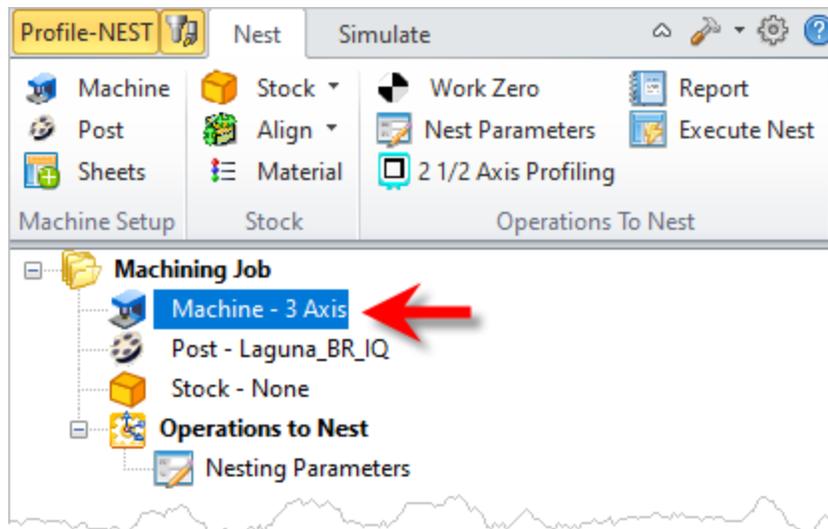


Operational Sequence

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 [VisualCAD/CAM](#).

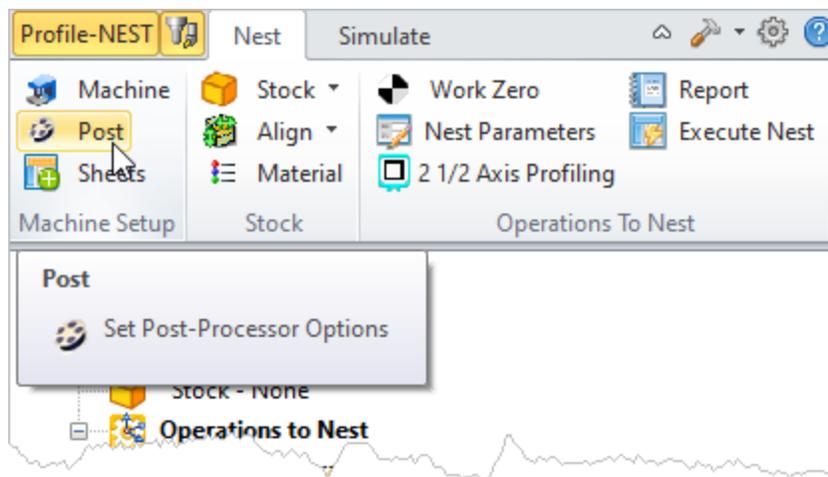
1. The [Machining Job](#) tree lists the [Machine](#) definition as [3 Axis](#). This is correct for all [2 Axis](#) machining and nesting operations.



5.2 Select the Post Processor

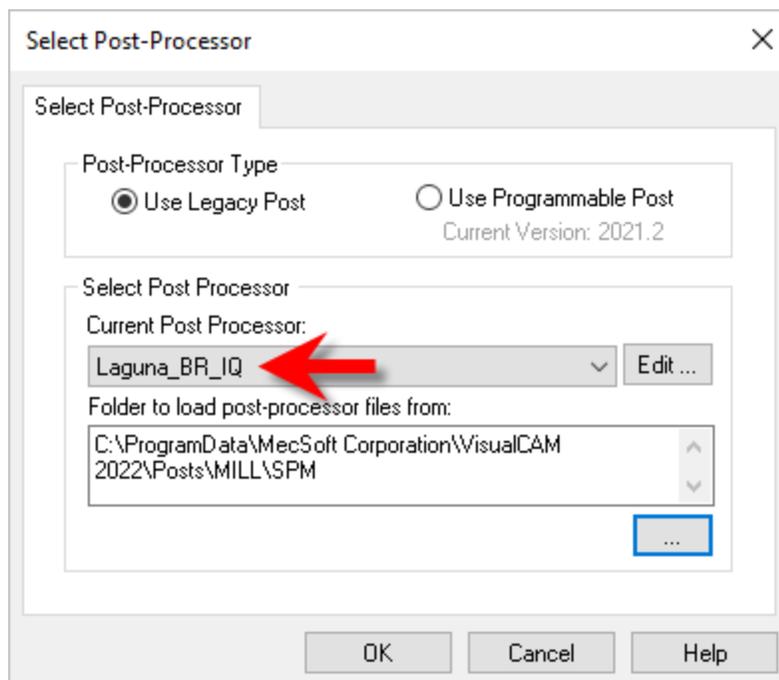
Next, we'll define the [Post Processor](#).

1. From the [Profile-NEST](#) tab select [Post](#) to display the dialog.



Post menu selection, Nest tab, Machining Browser

2. For the **Current Post Processor**, select **Laguna_BR_IQ** from the list of available posts.

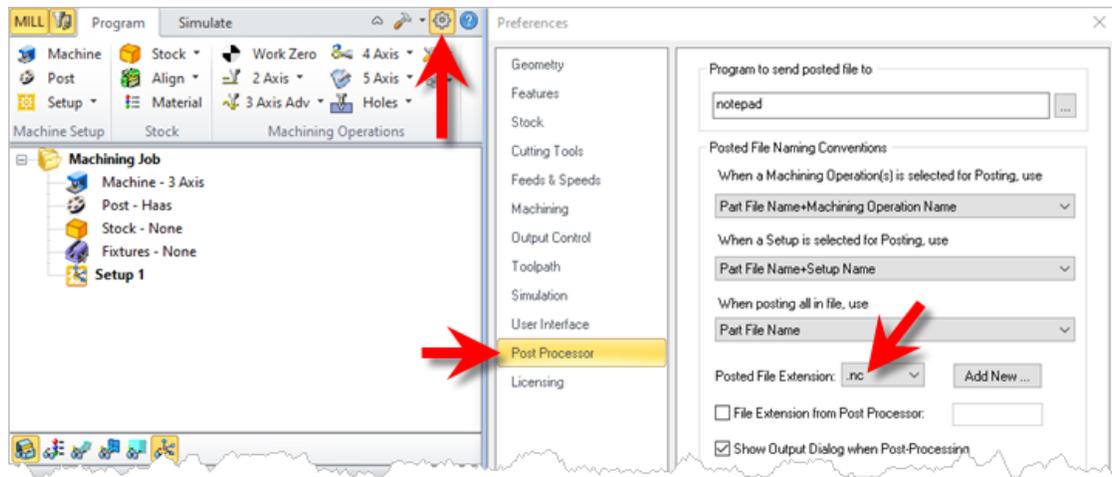


! By default, post processor files are located under

<C:\ProgramData\MecSoft Corporation\VisualCAM 2026\Posts\MILL\SPM>

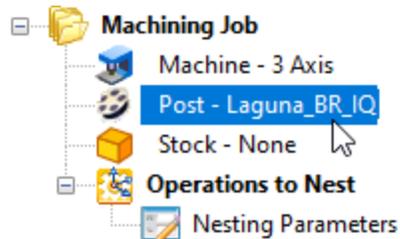
The program to send the posted output data to is set to notepad.

3. Now let's have a look at the **Post** related Preferences. Pick the **CAM Preferences** icon at the top left of the **Program** tab and then select the **Post-Processor** tab as shown below.



For **Post File Extension** select **.nc** from the dropdown list. If you need a different extension, pick the **Add New** button and enter your file extension and pick **OK**. The posted file extension looks like this: **my-gcode-file.nc**

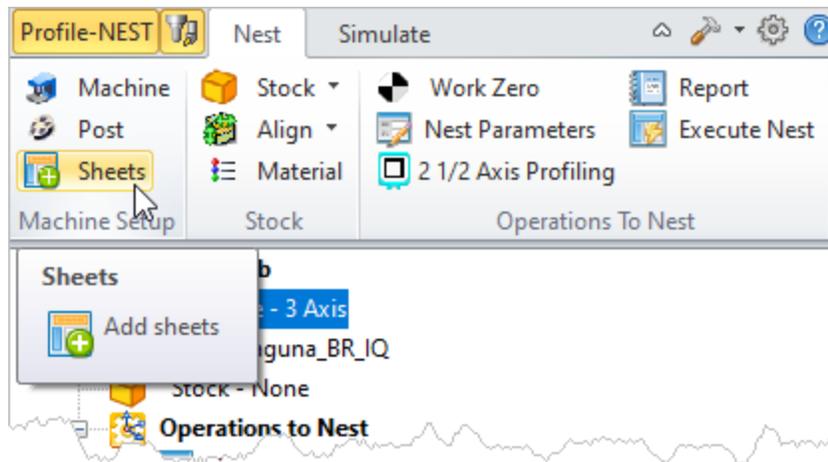
- Pick **OK** and notice that the **Post** type now appears under **Machining Job** in the **Machining Browser**.



5.3 Define Sheet to Nest

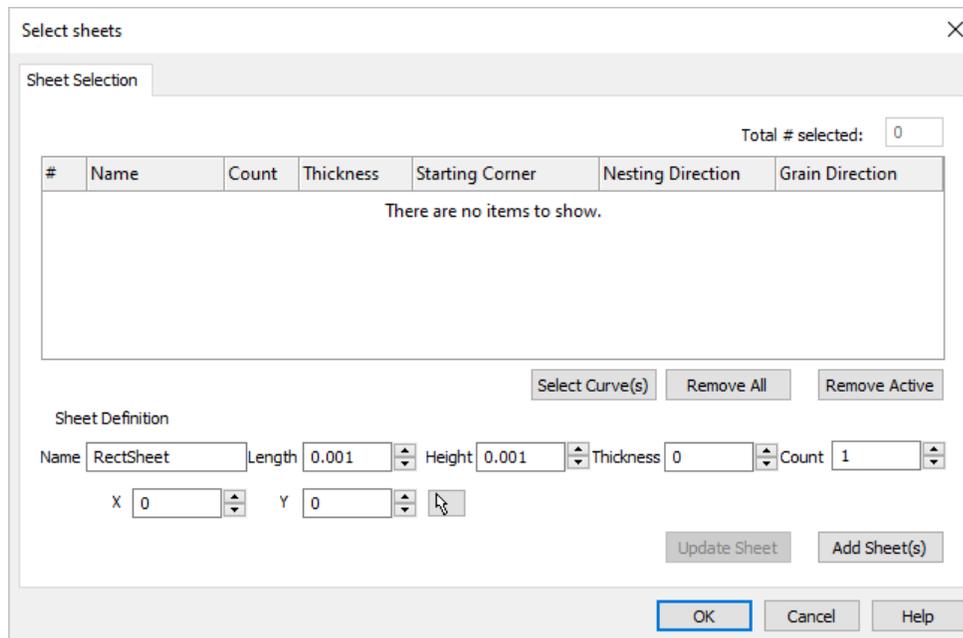
In this step we'll define the sheet to use in the nesting process.

- From the **Program** tab select **Sheets** to display the dialog.

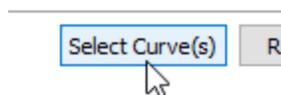


This dialog is used to define one or more sheets to use for nesting. Sheets can be defined from existing curves defining the sheet boundary or by using the [Sheet Definition](#) portion of the dialog.

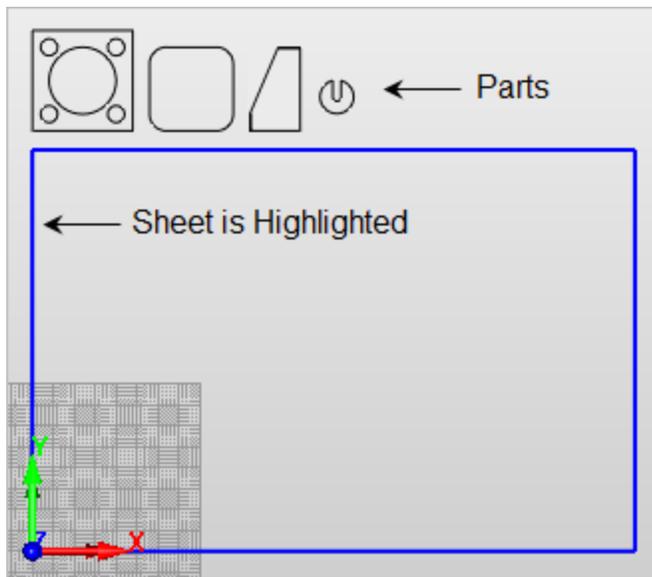
We have sheet curve already defined in our drawing.



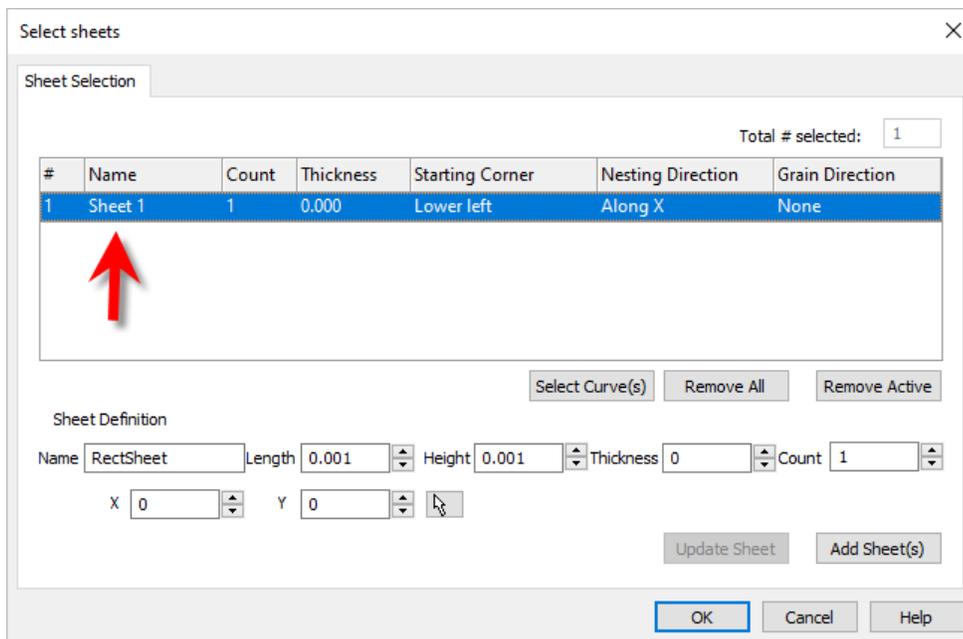
- From the dialog pick the [Select Curve\(s\)](#) button.



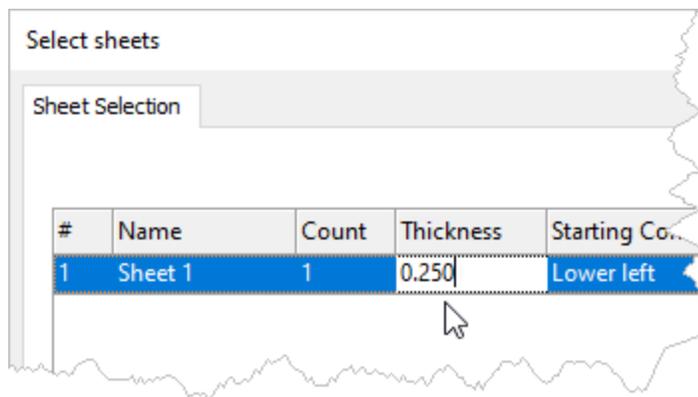
- The dialog will minimize and prompt you to select objects. Select the large rectangle shown on the drawing and right-click or simply press the [<Enter>](#) key to complete the selection.



The dialog reappears with [Sheet 1](#) listed



- Now left-click on the [Thickness](#) column and change the sheet thickness to [0.250](#) to match our stock thickness and then pick [<Enter>](#).



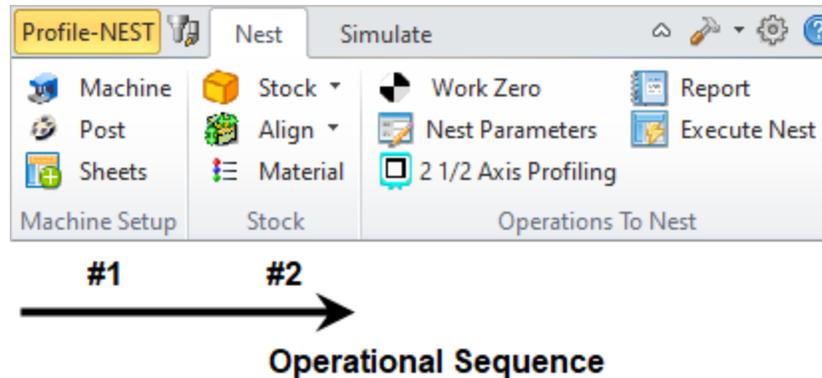
The sheet geometry is also highlighted on the drawing:

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.

#	Name	Count	Thickness
1	Sheet 1	3	0.250

The Stock

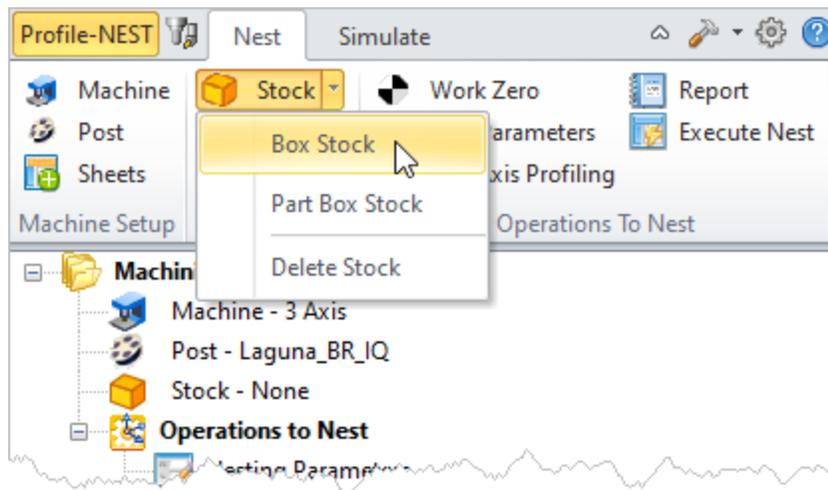
Next we will proceed to the [Stock](#) section of the [Nest](#) tab of the [Machining Browser](#).



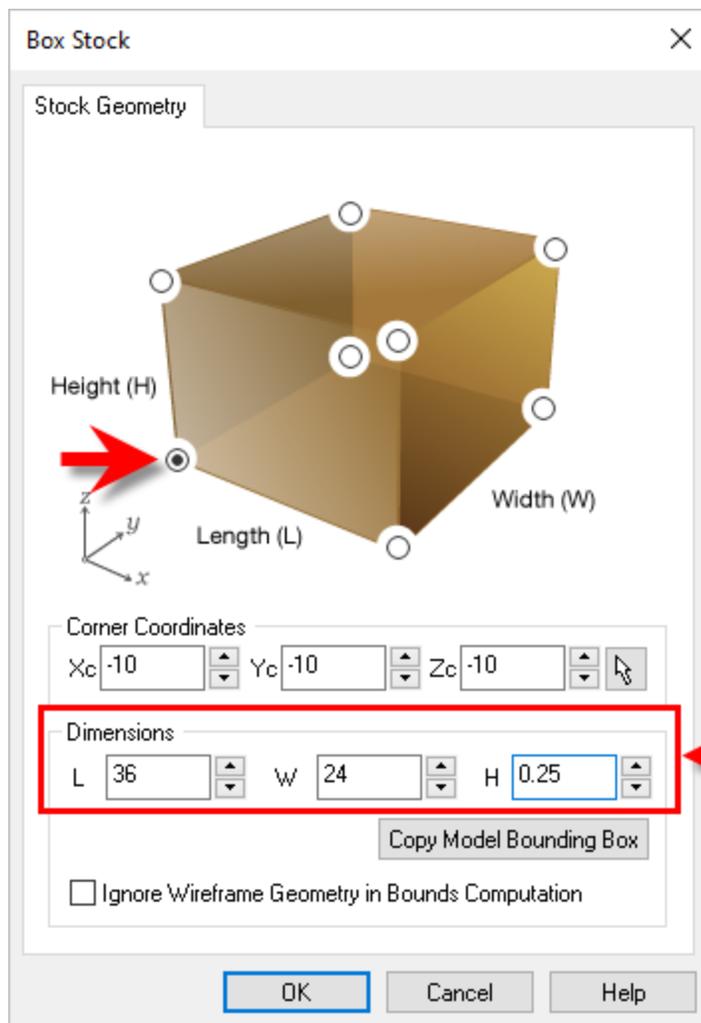
6.1 Create Stock Geometry

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.



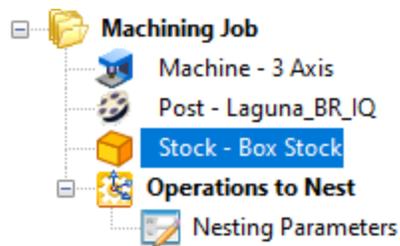
2. Under [Dimensions](#), set the [Length L](#) to 36, [Width W](#) to 24 and [Height H](#) to 0.25. Note that the stock dimensions you enter are measured from the corner of the bounding box selected in this dialog.



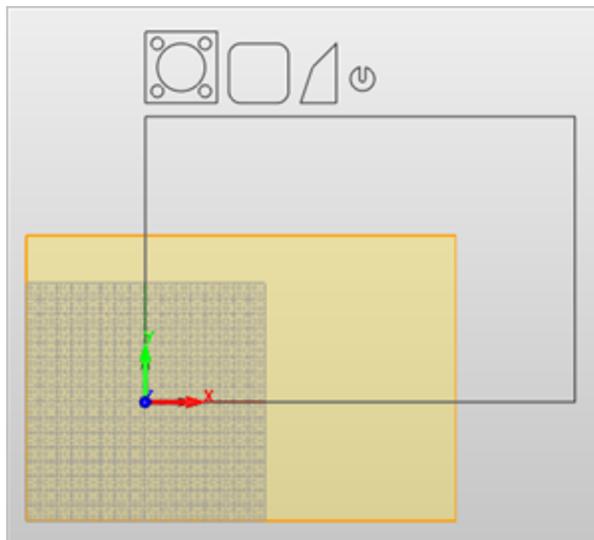
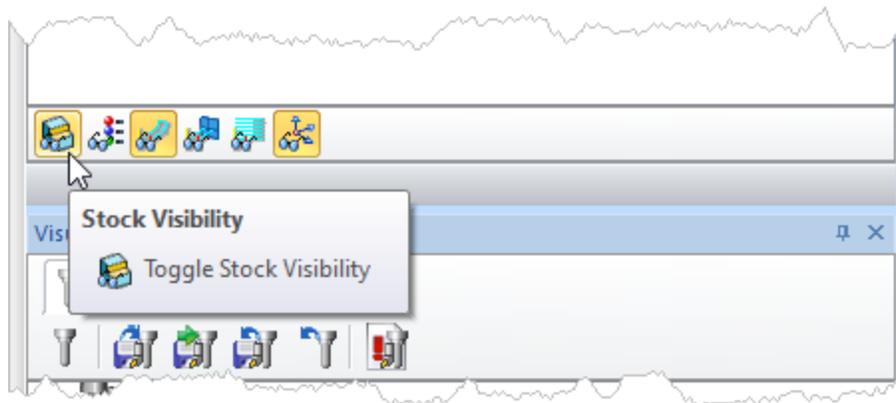
 The dimensions of the stock are interpreted in relation to the corner selected in the dialog box above. For example if the corner of the box is selected as the **Bottom South West** corner (as shown in the dialog above), the **Length (L)** is interpreted to be along the **+X** axis, the **Width (W)**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



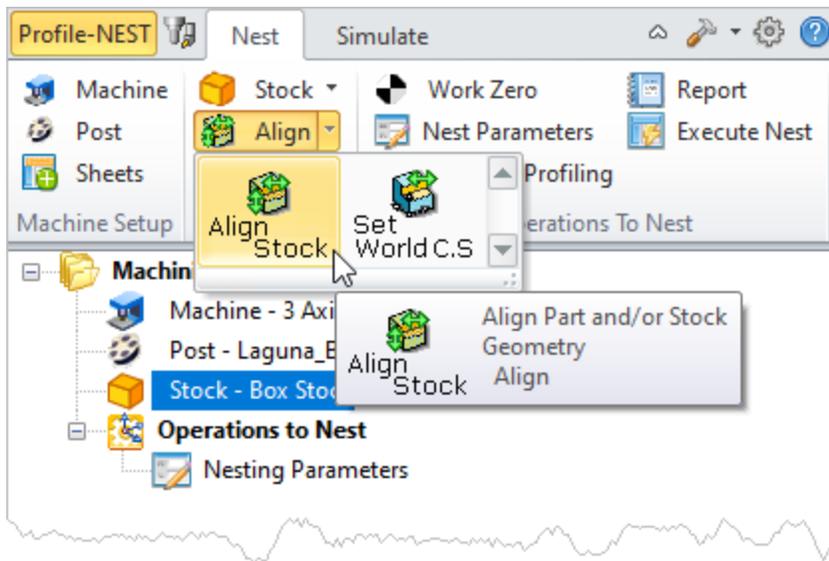
4. If the stock does not display on the screen, select the [Stock Visibility](#) icon located at the base of the [Machining Browser](#).



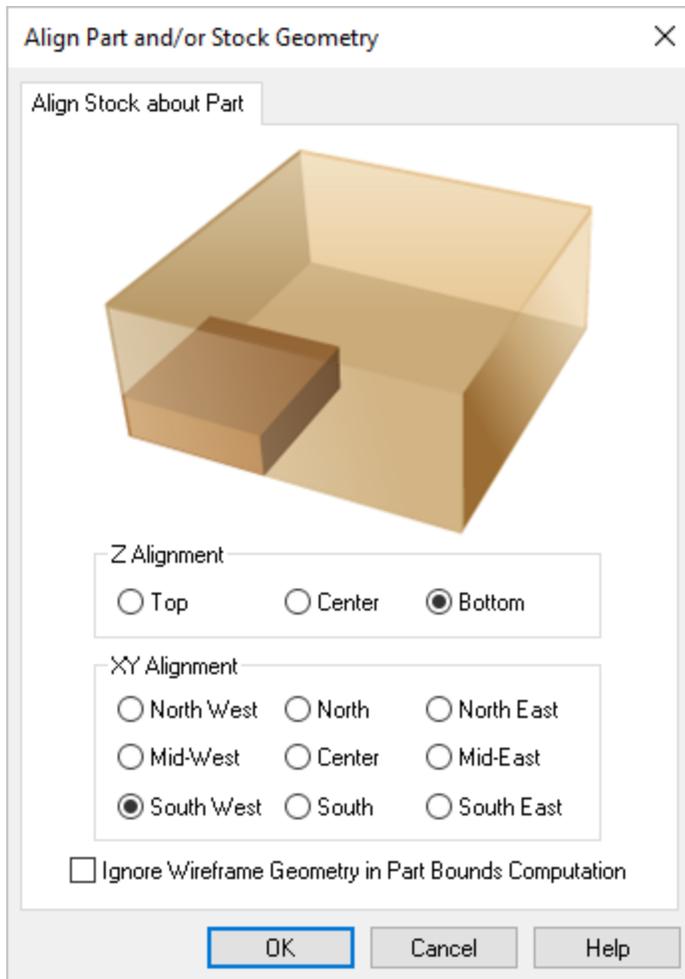
6.2 Align Part and Stock

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

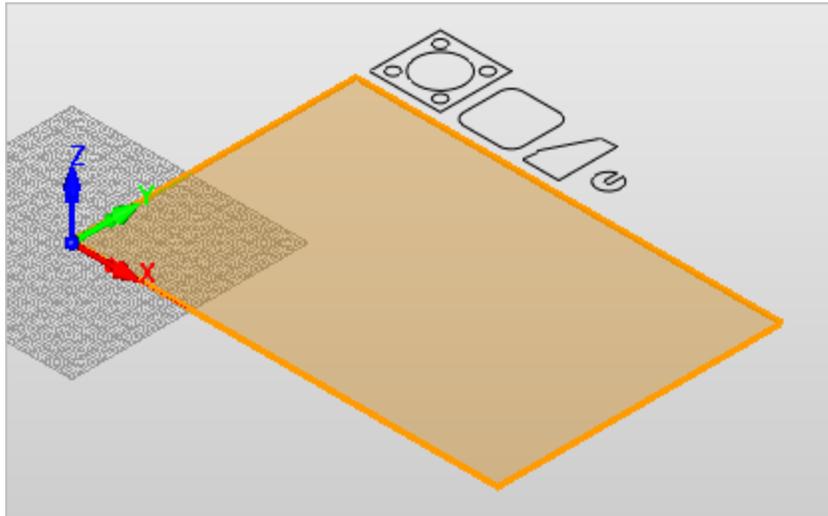
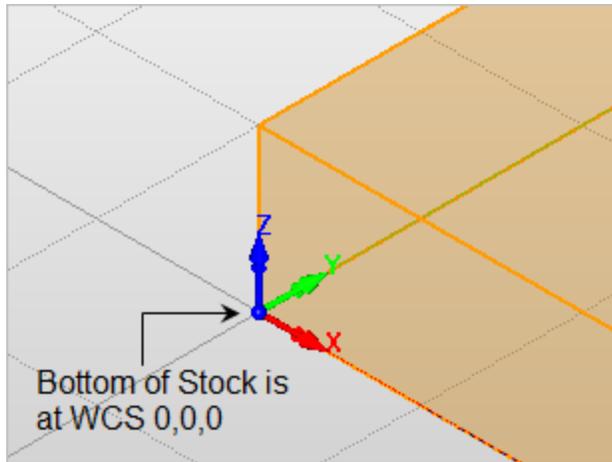
1. First from the [View](#) toolbar select the [Isometric](#) view.
2. From the [Program](#) tab select [Align](#) and then [Align Stock](#) from the menu to display the dialog. Notice that we are working our way from left to right in the [Program](#) tab.



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



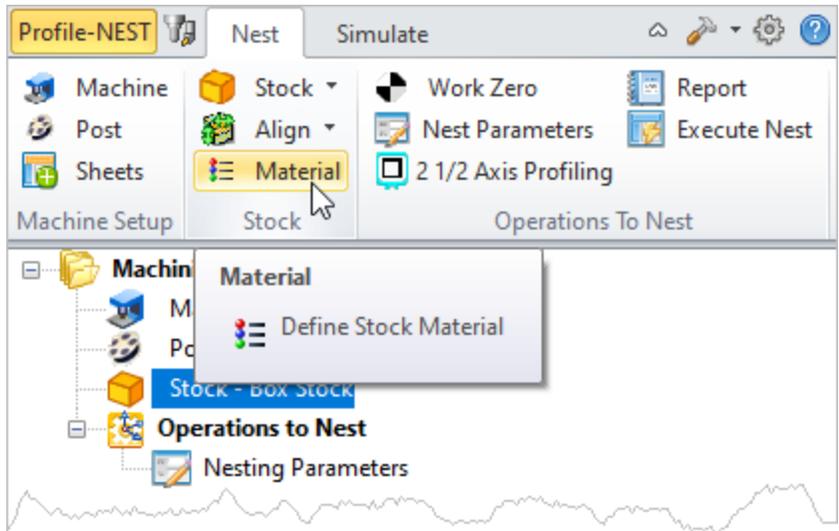
The bottom of the stock is now aligned to the bottom right corner of the stock sheet geometry with the bottom corner of the stock being located at **0,0,0** coordinate of the **WCS** (**Work Coordinate System**).



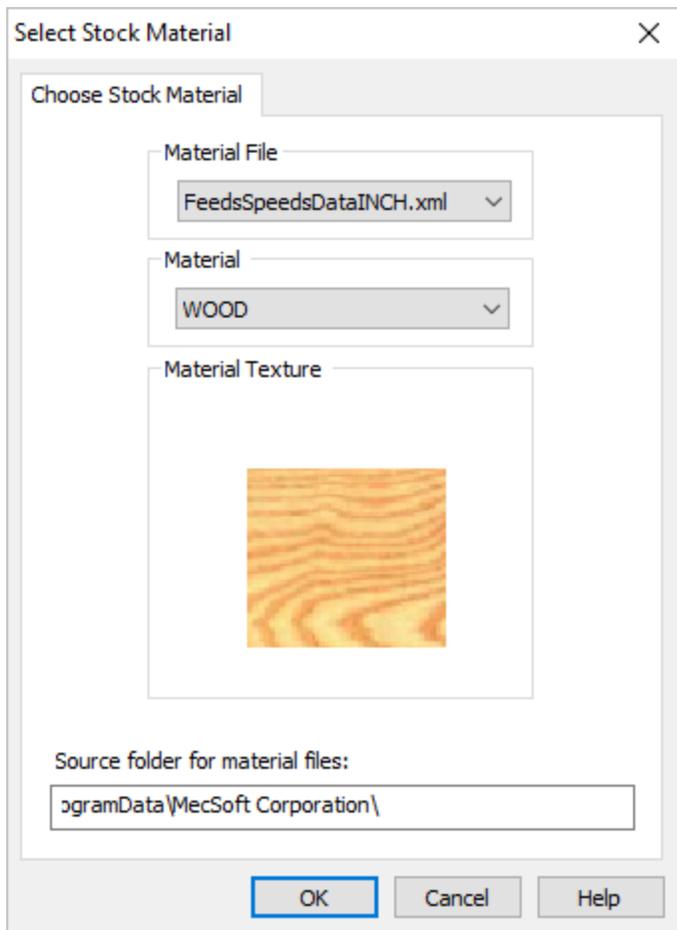
6.3 Specify Material

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

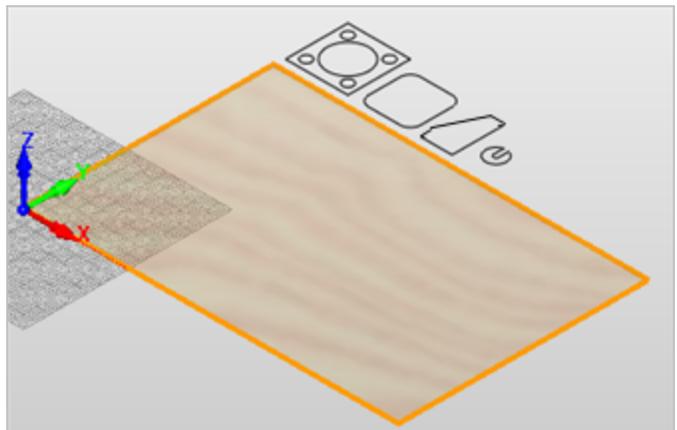
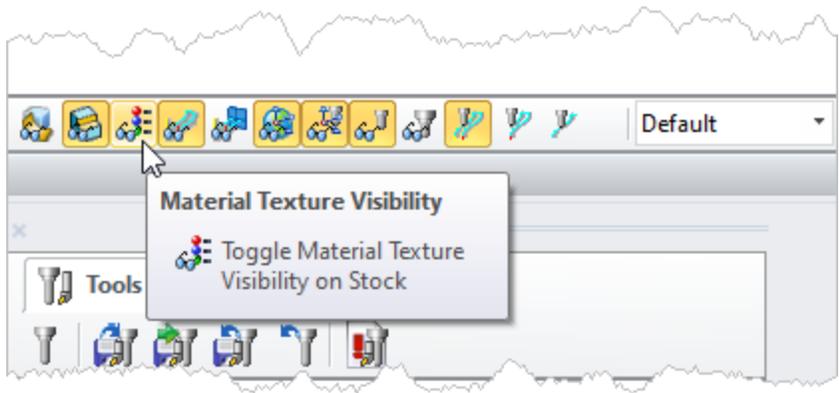
1. From the **Program** tab select **Material** to display the dialog box.



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

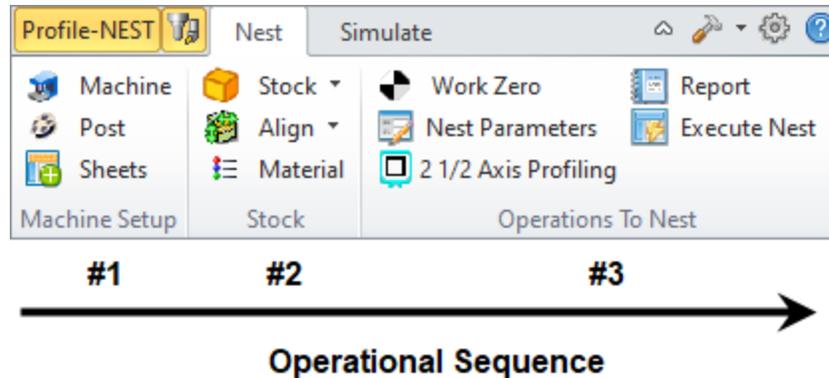


3. If the material texture does not display on the stock, select the **Material Texture Visibility** icon  located at the base of the **Machining Browser**.



The Profiles to Nest

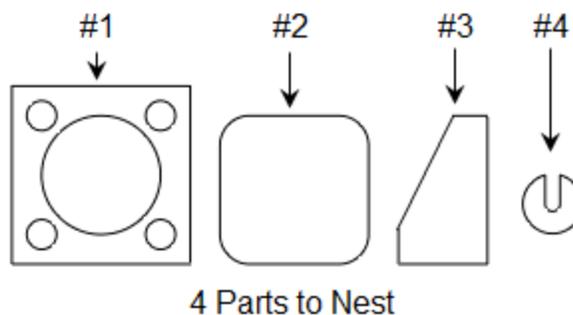
Next we will proceed to the [Operations to Nest](#) section of the [Nest](#) tab of the [Machining Browser](#).



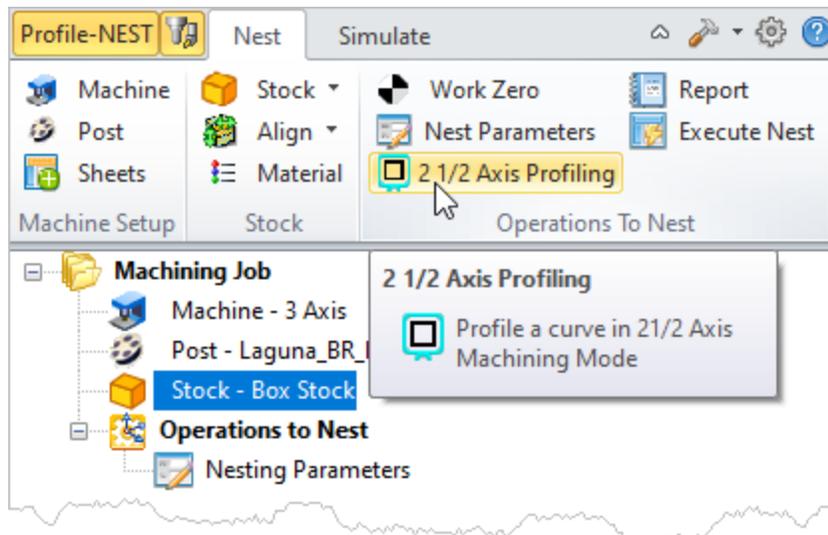
7.1 Profile Part 1

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.



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



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

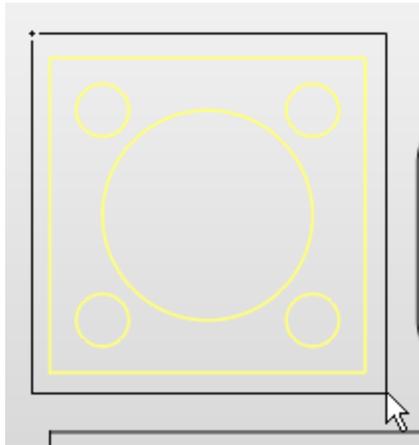
If you are familiar with using the **MILL** module you will recognize this dialog. If you are a new user, each tab in this dialog controls a different set of parameters. All parameters in this dialog relate to the **2 Axis Profile** toolpath operation that you are creating right now.

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

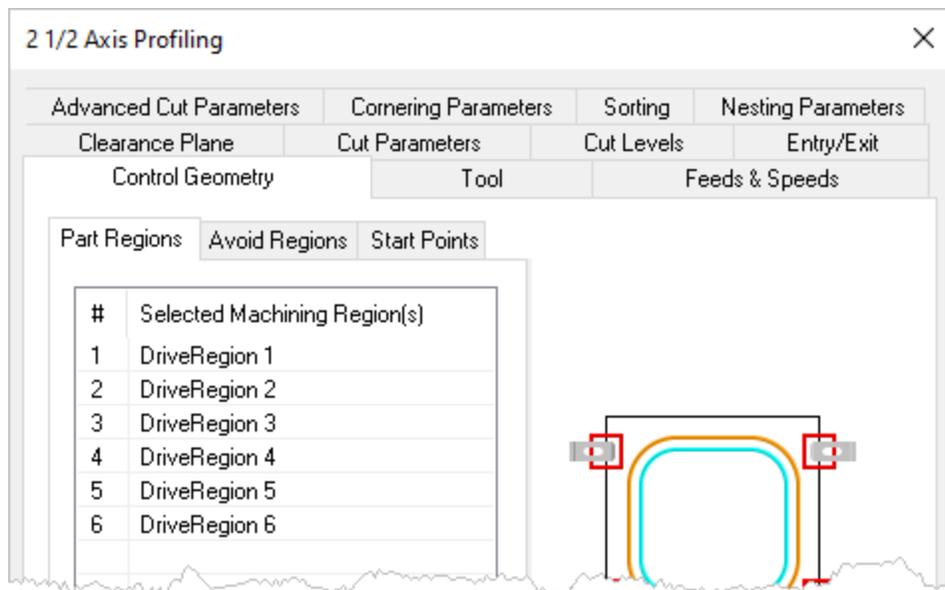
7.1.1 Profile 1 Control Geometry tab

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.



- 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.



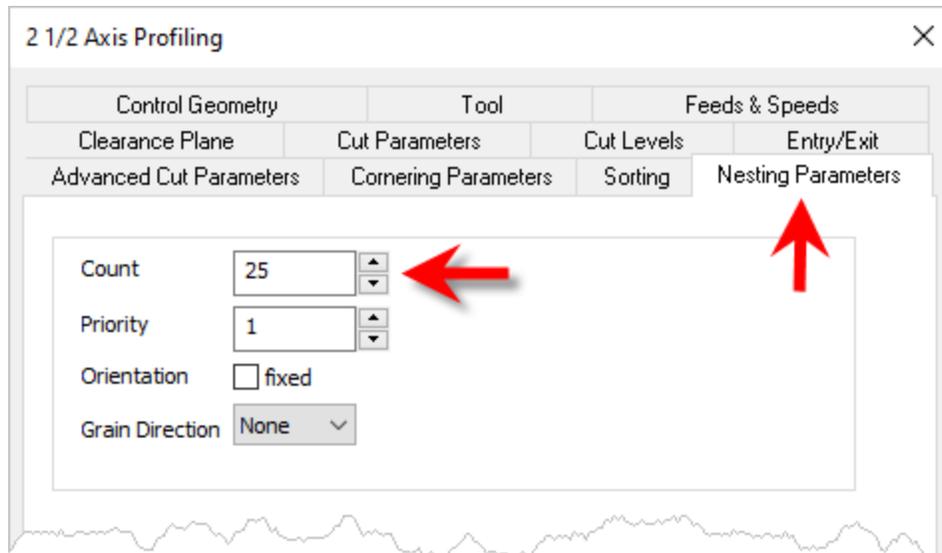
- You will also see that while this dialog is displayed, the curves selected for the operation are highlighted on the part. You will also notice that each curve has its start point identified. The cut motions will begin at each start point.

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

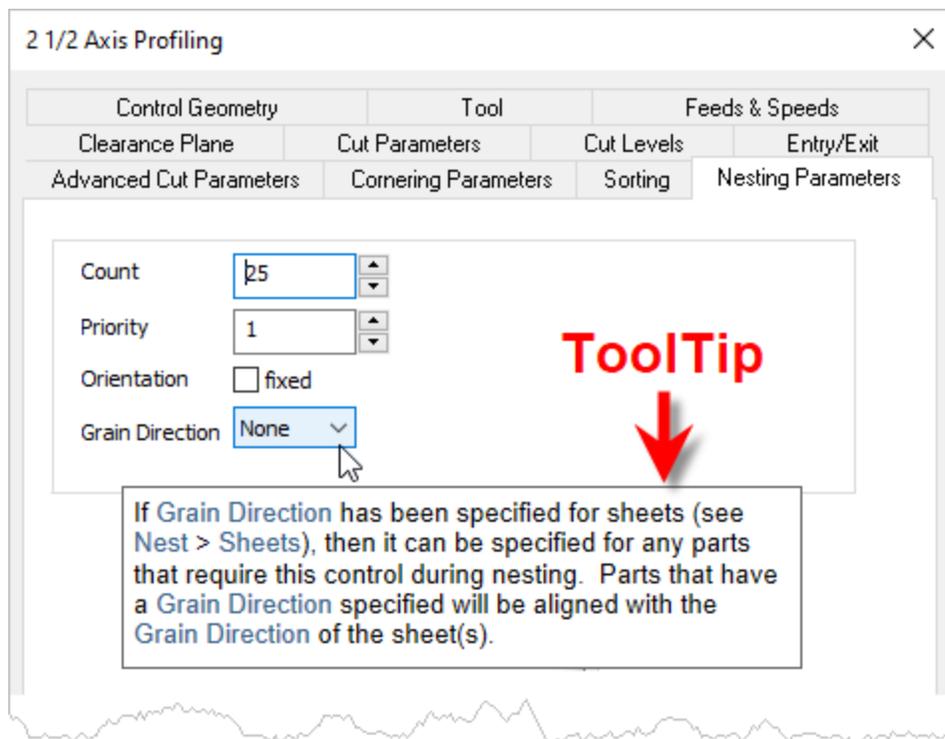
In this step we will move to the [Nesting Parameters](#) tab and set a few nesting parameters that will apply to this profile operation only.

1. With the [2-1/2 Axis Profile](#) operation dialog still displayed, select the [Nesting Parameters](#) tab.
2. Now locate the [Count](#) parameter and set this to [25](#). There will be [25](#) profiles of [Part 1](#) in our resulting nested sheets.

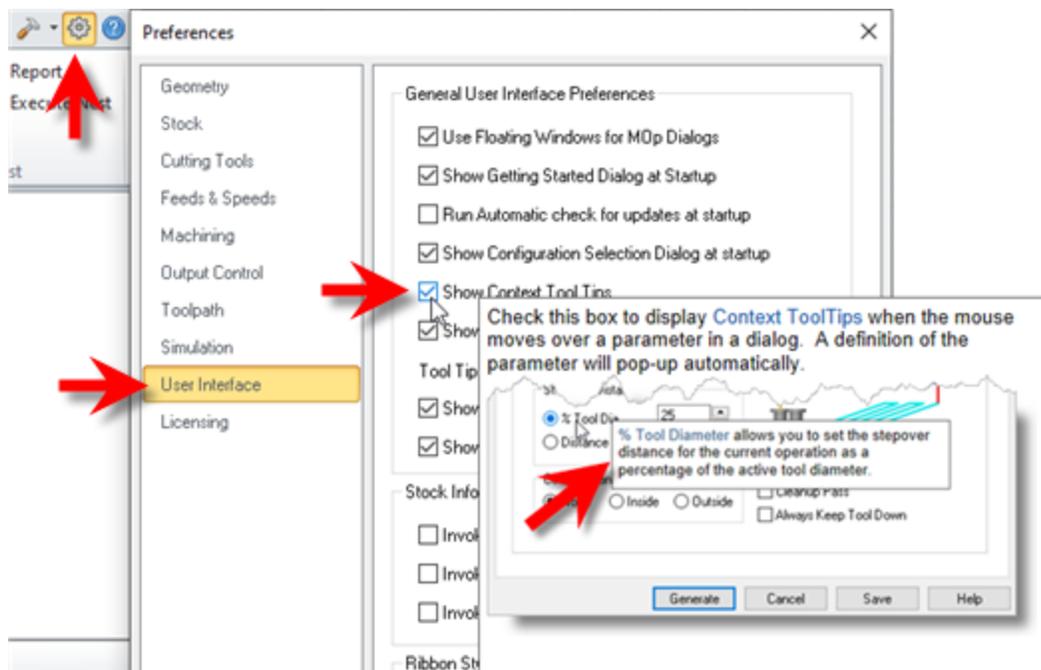


3. We will accept the default values for the remaining nesting parameters.

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.



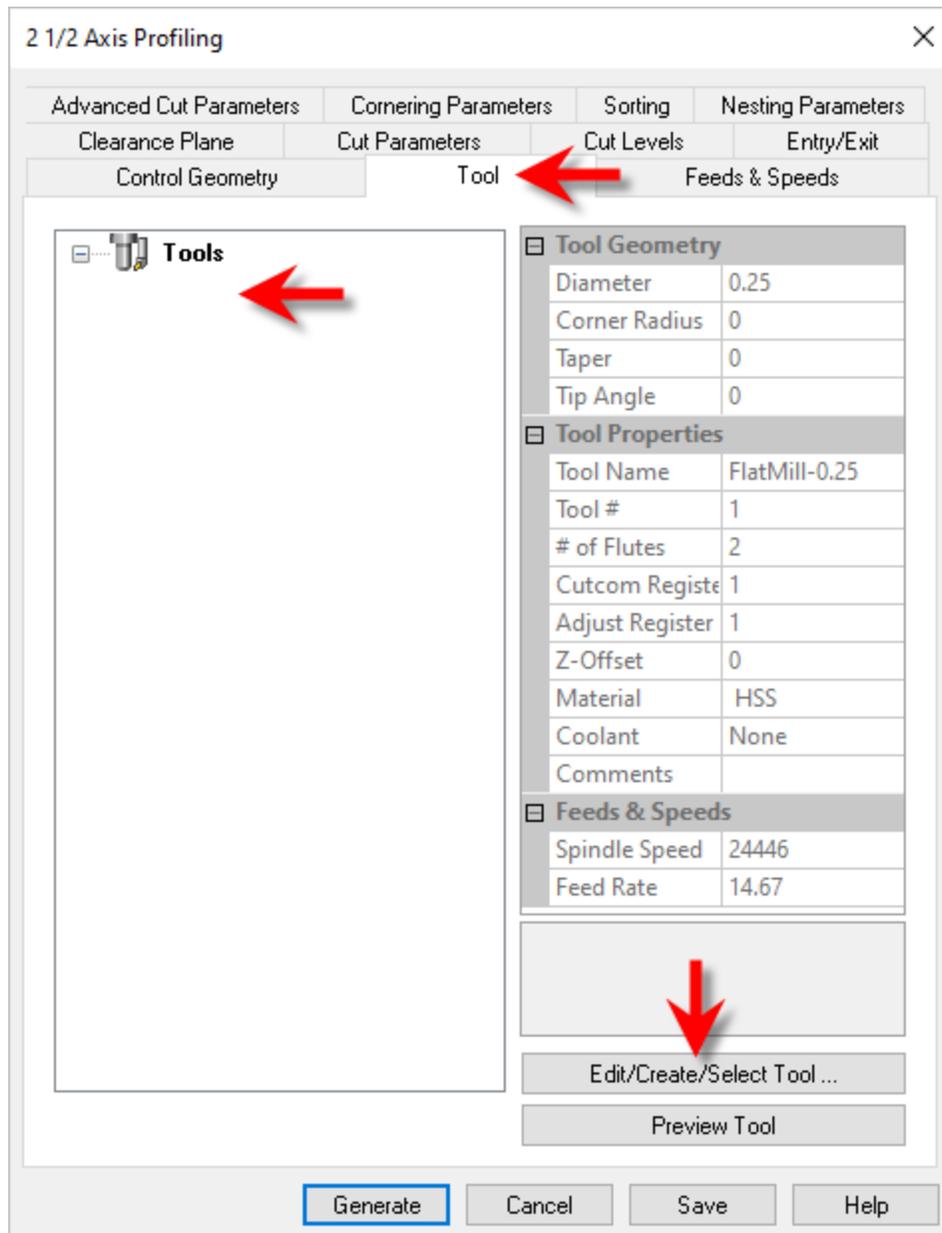
4. If the ToolTips are not displaying for you, go to the User Interface tab of the CAM Preferences to enable the display of Context ToolTips.



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.

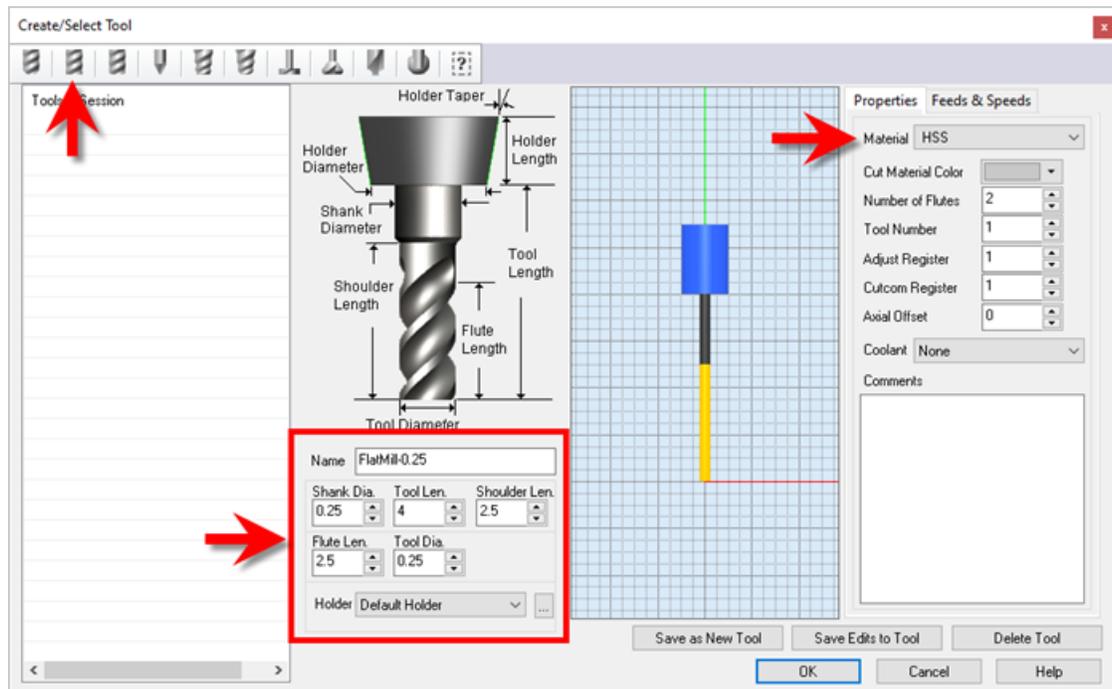
1. With the **2-1/2 Axis Profile** operation dialog still displayed, select the **Tool** tab.



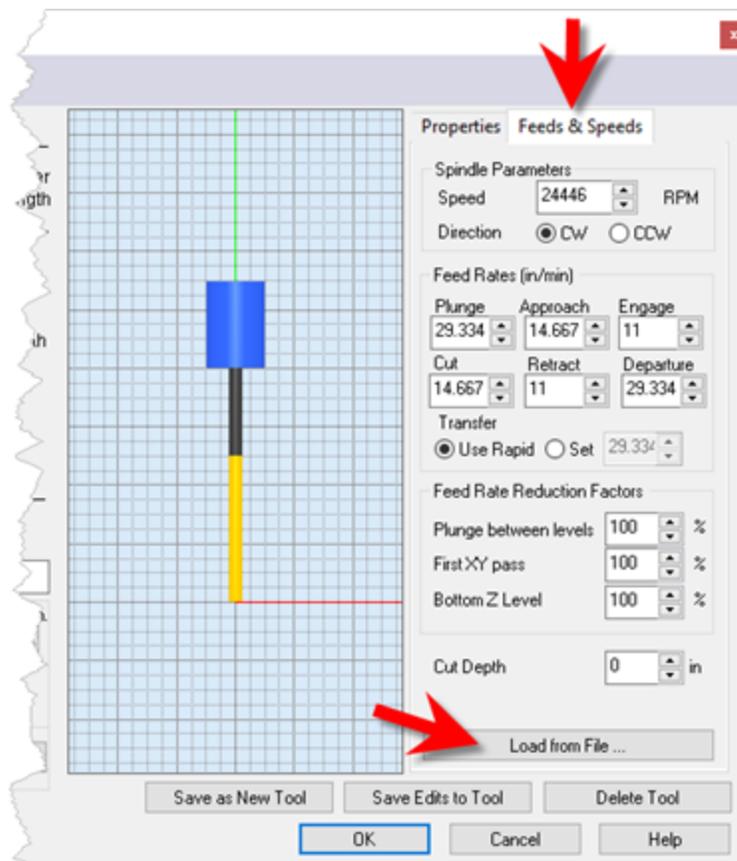
2. You see that there are no tools listed on the left. Let's create one right from this dialog. Near the bottom of the dialog pick the **Edit/Create/Select Tool ...** button.
2. This will display the **Create/Select Tool** dialog. First, select the **Flat Mill** icon from the **Tool Type** menu at the top of the dialog.



- Set tool **Name** to **FlatMill-0.25**, and fill in the tool dimensions and holder as shown in the dialog below. Under the **Properties** tab set **Material** to **HSS** and **Tool Number**, **Adjust Register** and **Cutcom Register** to **1**.



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

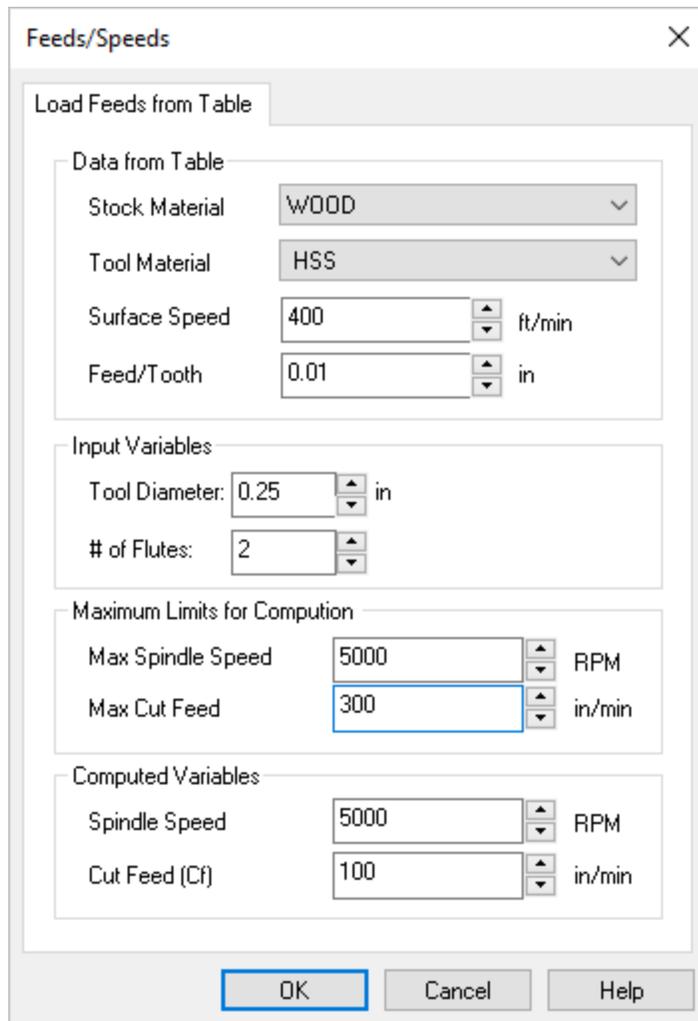


5. From the dialog that displays, set the following parameters:

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

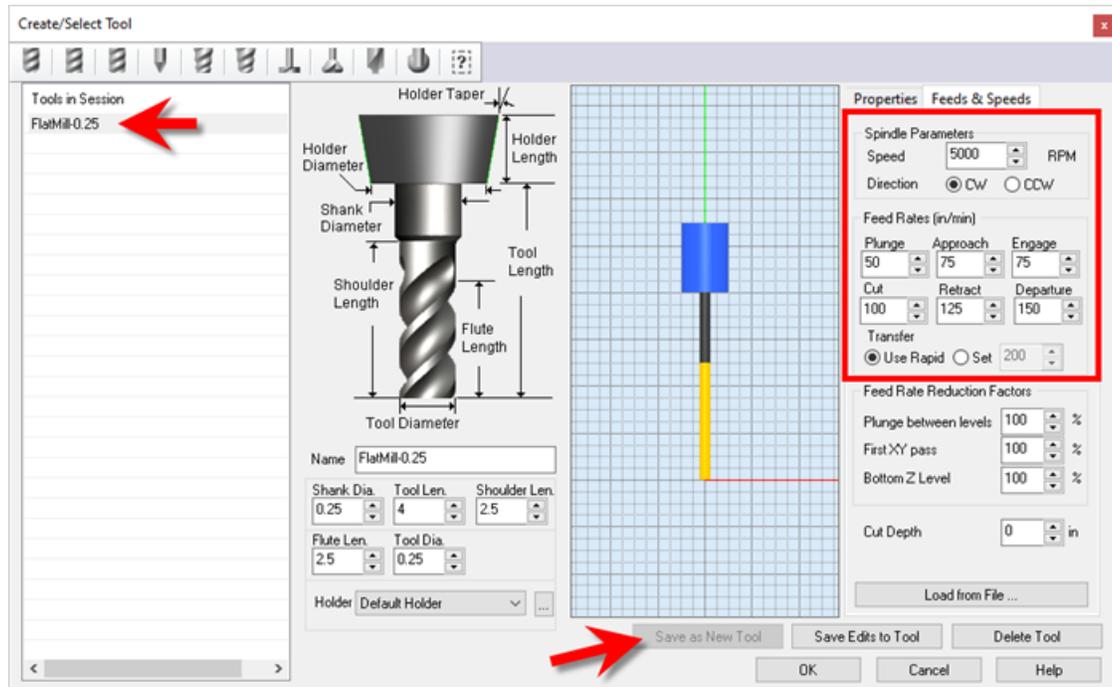


The image shows a 'Feeds/Speeds' dialog box with a close button (X) in the top right corner. It is divided into several sections:

- Load Feeds from Table**: This section contains a 'Data from Table' group box with four fields: 'Stock Material' (WOOD), 'Tool Material' (HSS), 'Surface Speed' (400 ft/min), and 'Feed/Tooth' (0.01 in).
- Input Variables**: This section contains two fields: 'Tool Diameter' (0.25 in) and '# of Flutes' (2).
- Maximum Limits for Computation**: This section contains two fields: 'Max Spindle Speed' (5000 RPM) and 'Max Cut Feed' (300 in/min).
- Computed Variables**: This section contains two fields: 'Spindle Speed' (5000 RPM) and 'Cut Feed (Cf)' (100 in/min).

At the bottom of the dialog box are three buttons: 'OK', 'Cancel', and 'Help'. The 'OK' button is highlighted with a blue border.

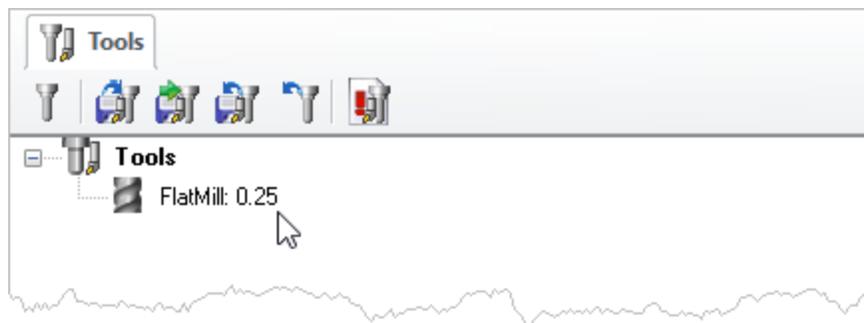
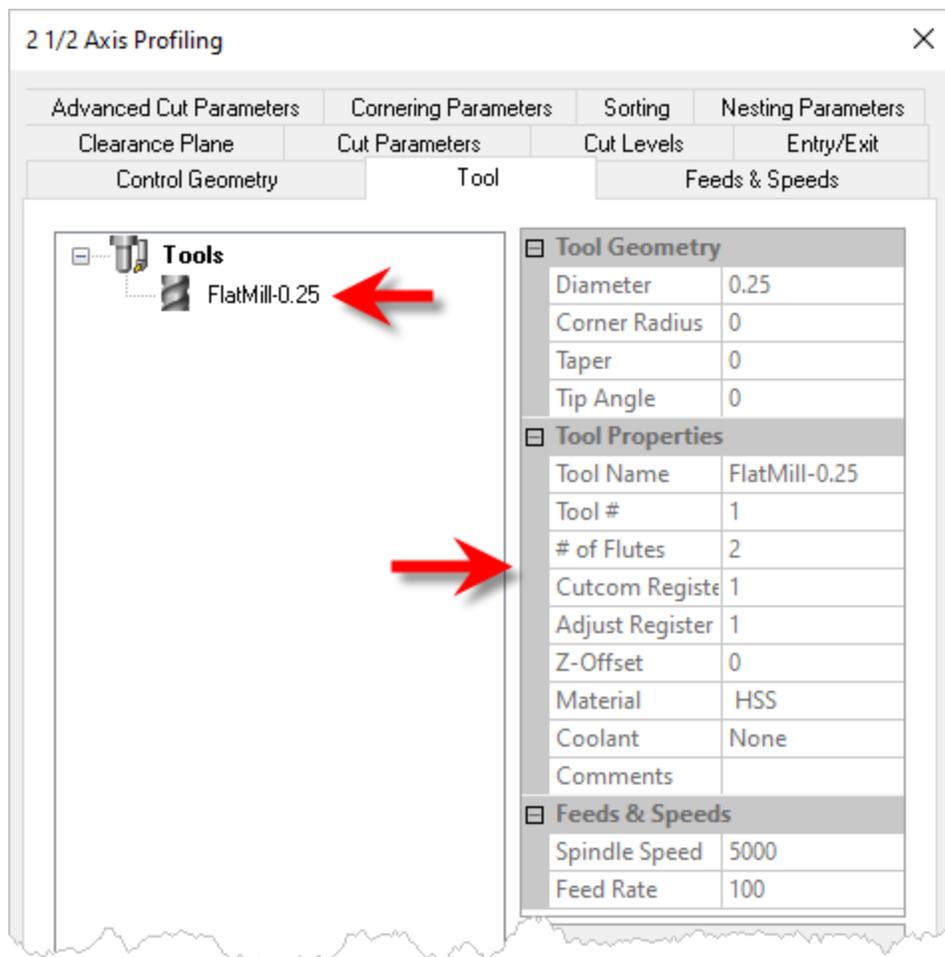
- Now pick **OK** and the computed cut feedrate and spindle speed are transferred to the **Feeds and Speeds** tab of the **Create/Select Tool** dialog for this tool.



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**.

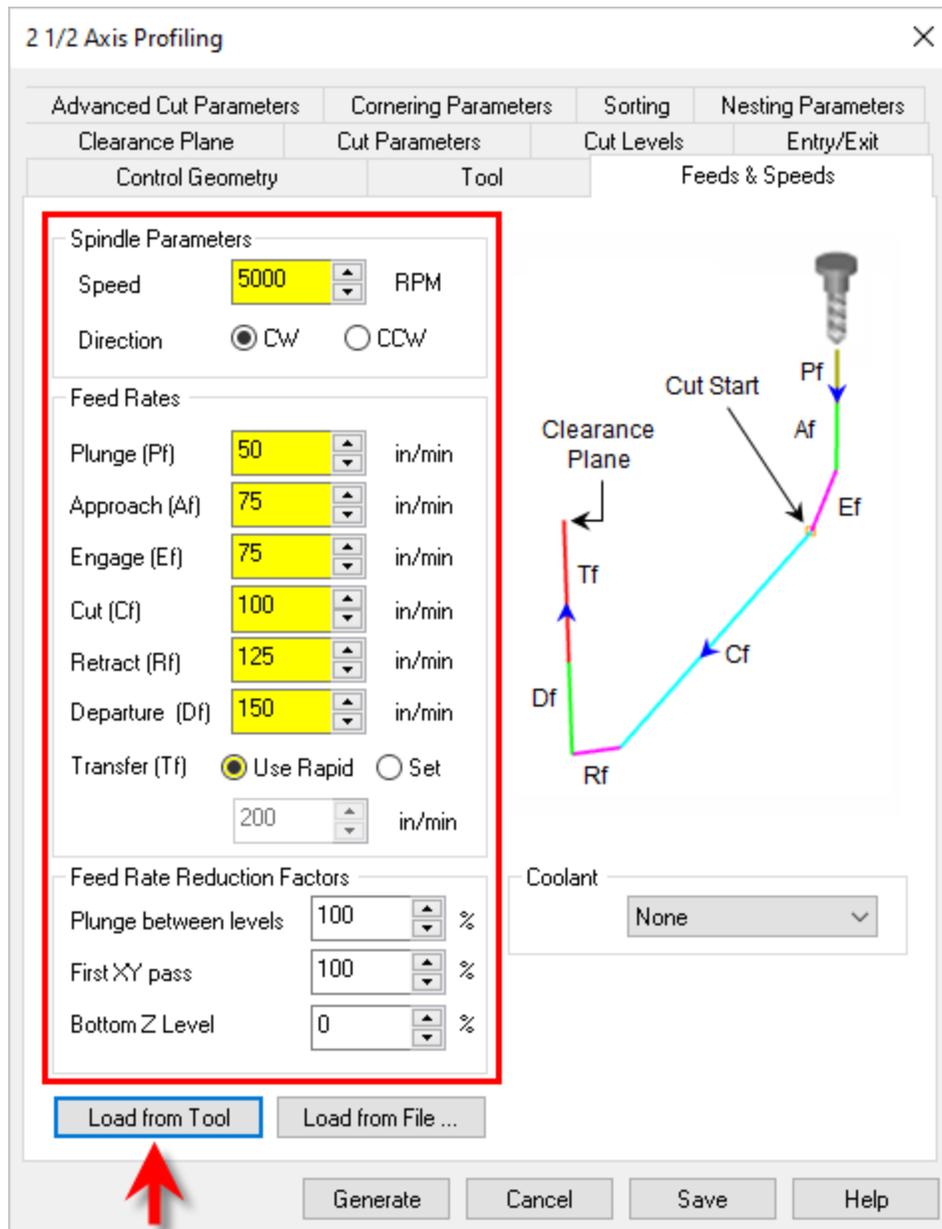


! In the future you can save your tools to a [Tool Library](#). To save [Tools](#) to a library, click [Save Tool Library](#) under the [Tools](#) tab in the [Machining Objects Browser](#) and specify a folder location and file name in the [Save as](#) dialog box. Two [Tool Library](#) file formats are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

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.

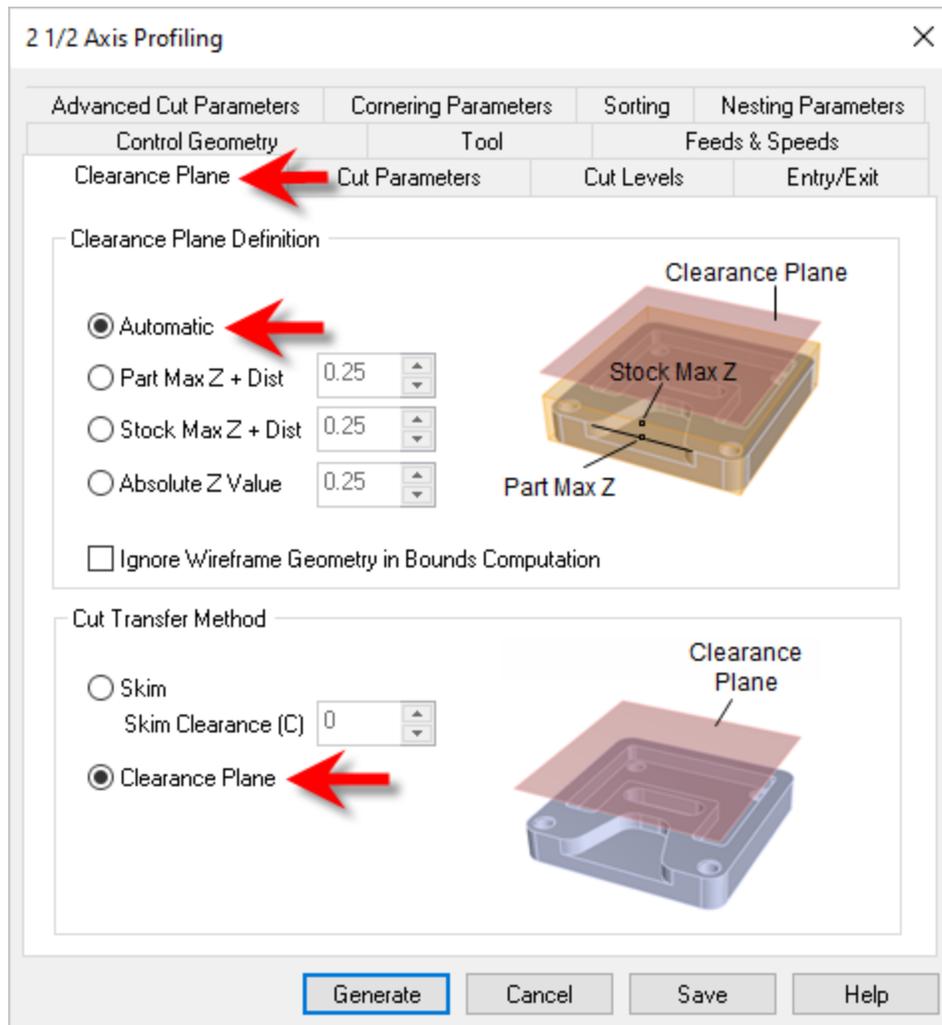


2. Now select the [Load from Tool](#) button. This will load the feeds and speeds parameters that we have set for the tool into this dialog to use for this operation.
3. You can override the feeds and speeds values here in this dialog and they will apply to just this operation.

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.

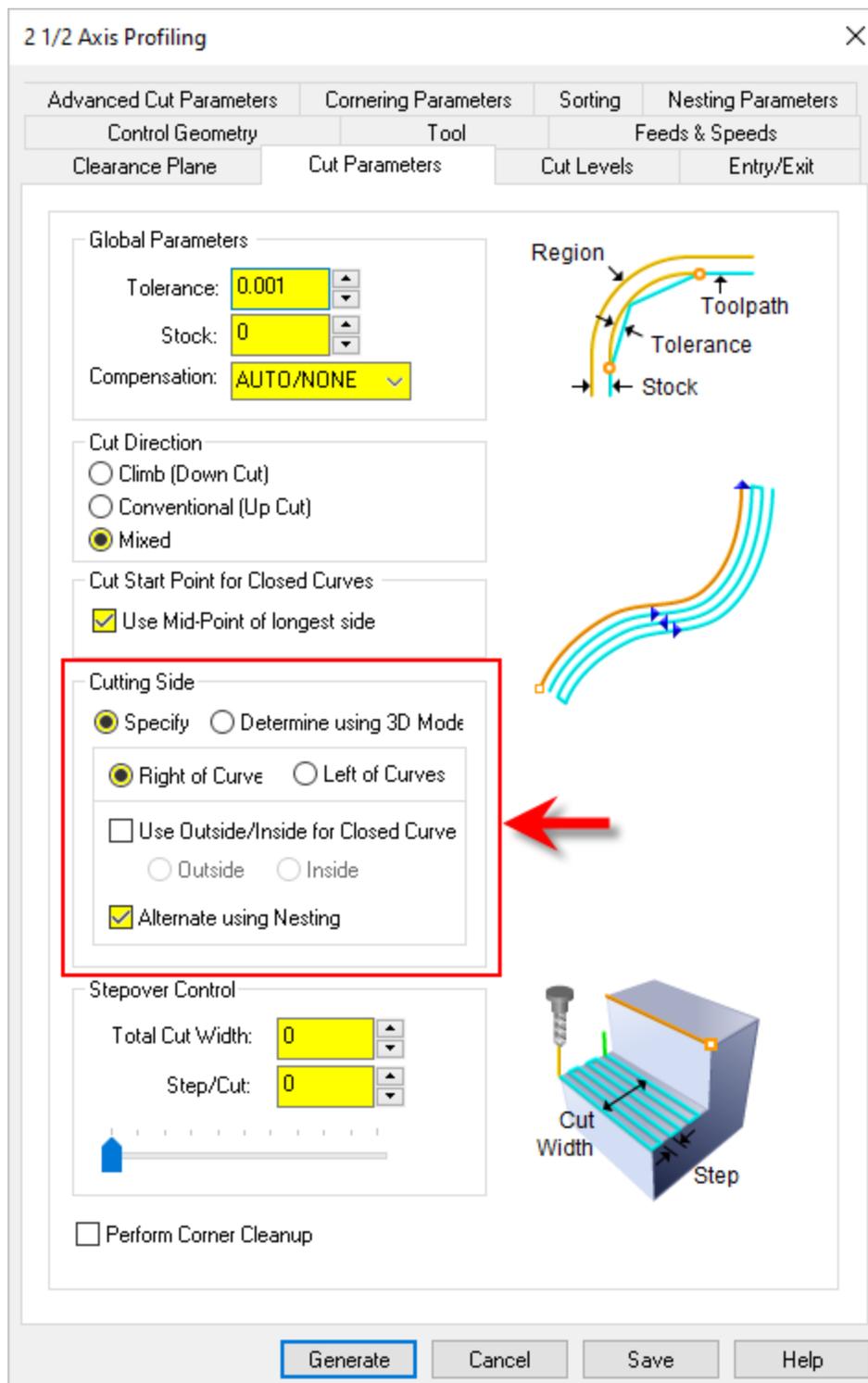


2. For [Clearance Plane Definition](#) select [Automatic](#). The system will determine a safe distance to use for the clearance plane.
3. For [Cut Transfer Method](#), select [Clearance Plane](#). The cutting tool will retract to the [Clearance Plane](#) to perform all cut transfer moves. The selections are shown in the dialog above.

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.



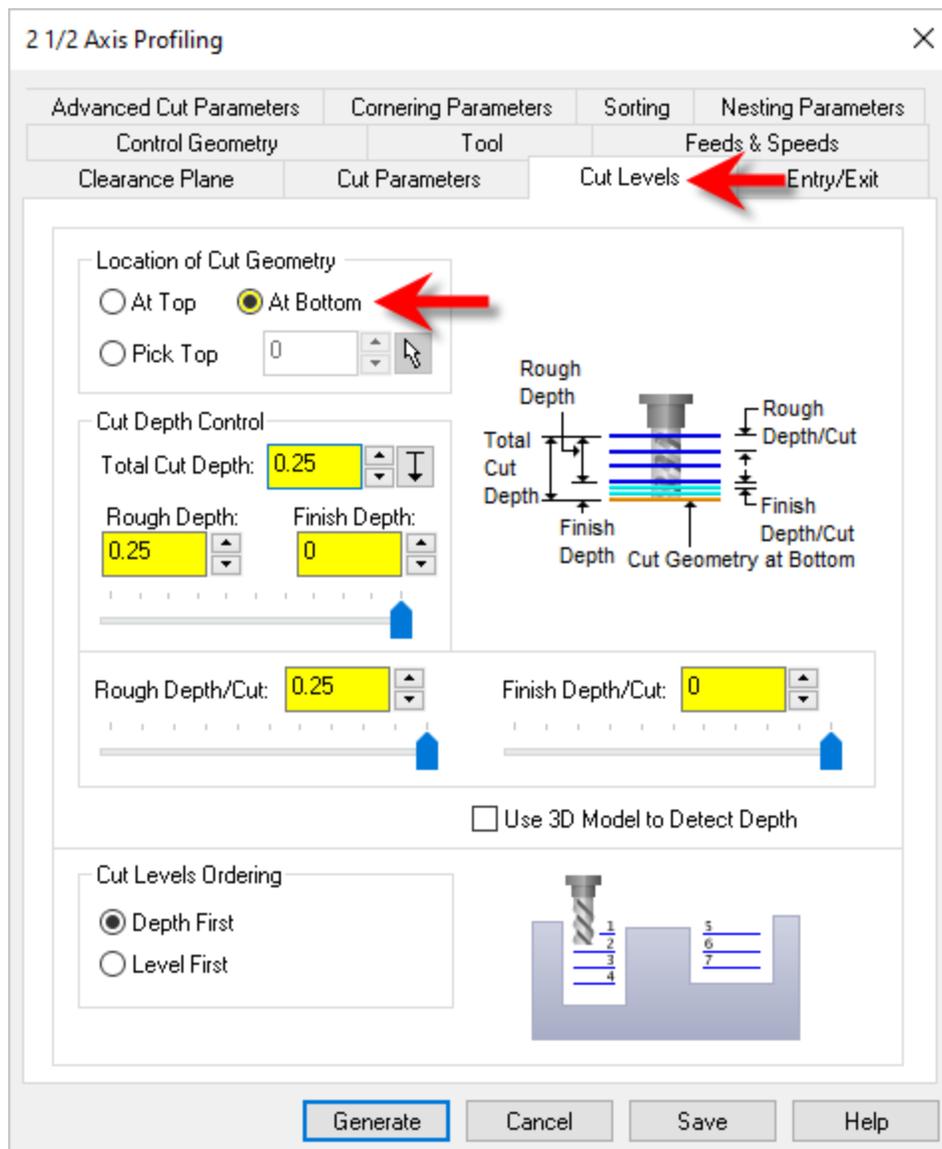
2. You see that the dialog has default selections made. We will make some changes to the **Cutting Side** section of the dialog.
3. For **Cutting Side** select the **Specify** option and then select **Right of Curve**.

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

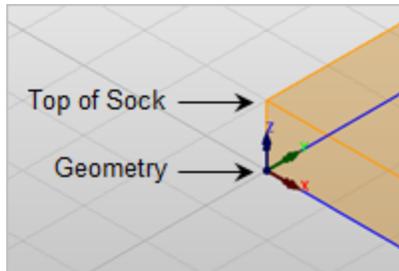
7.1.7 Profile 1 Cut Levels tab

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.

- With the [2-1/2 Axis Profile](#) operation dialog still displayed, select the [Cut Levels](#) tab.



2. First, 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.

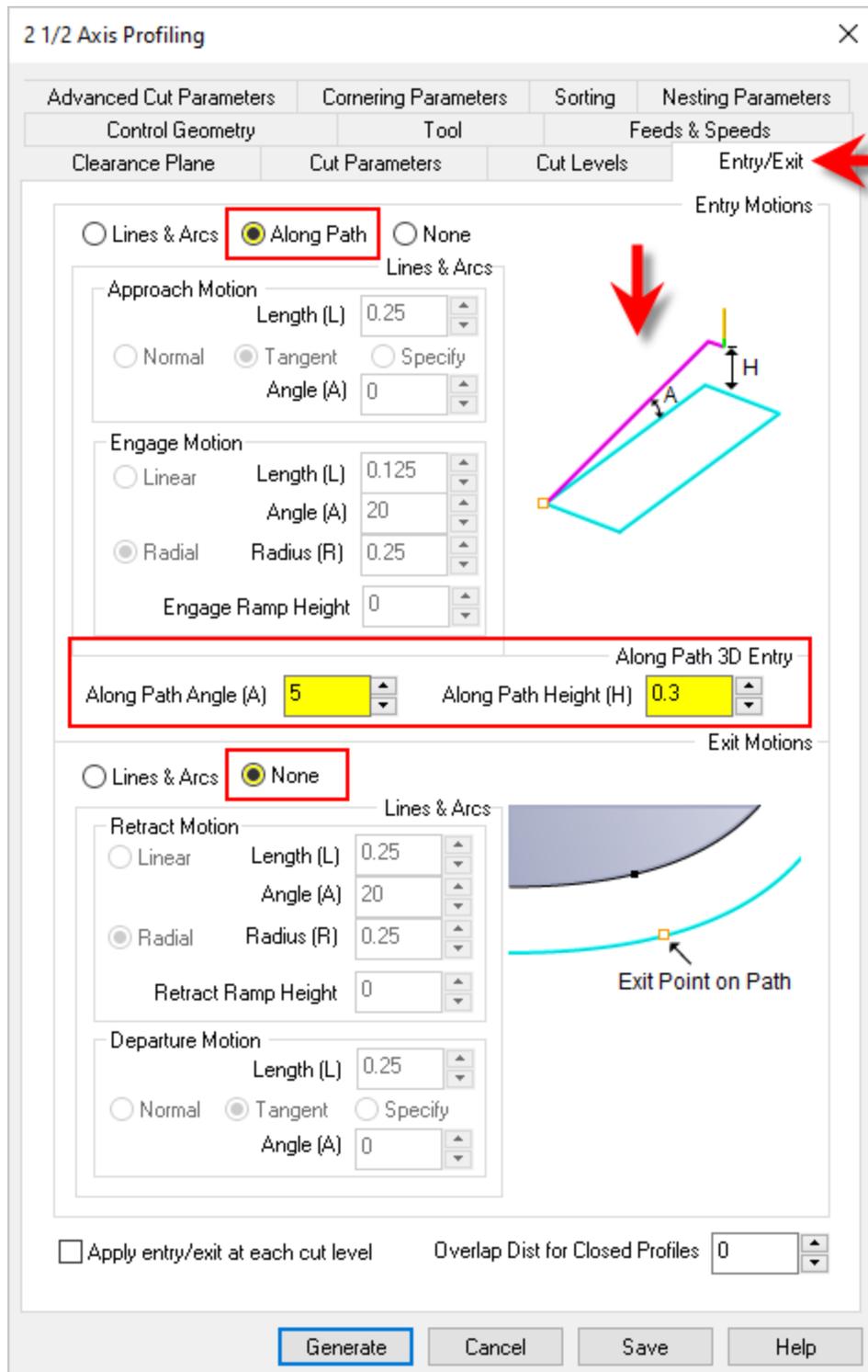


3. Now since we previously set our [Sheet Thickness](#) to 0.25, we need to make sure we set the Total [Cut Depth](#) to 0.25 also. We have highlighted each of the fields to set in the dialog above.

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.



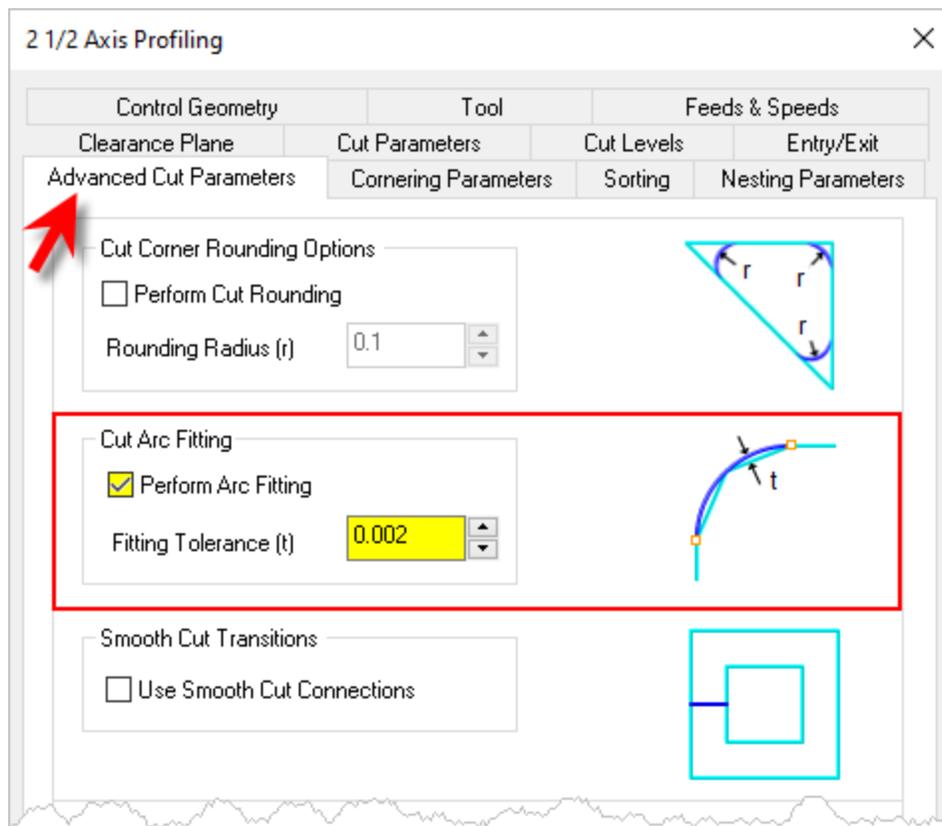
- We want the tool to ramp into the cut at an angle. To do this under the **Entry Motions** section (upper portion) of the dialog select **Along Path**. The image in the dialog will show you what each entry type looks like and the parameters to go with them. We have highlighted the options and values to use in the dialog above.

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.
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.

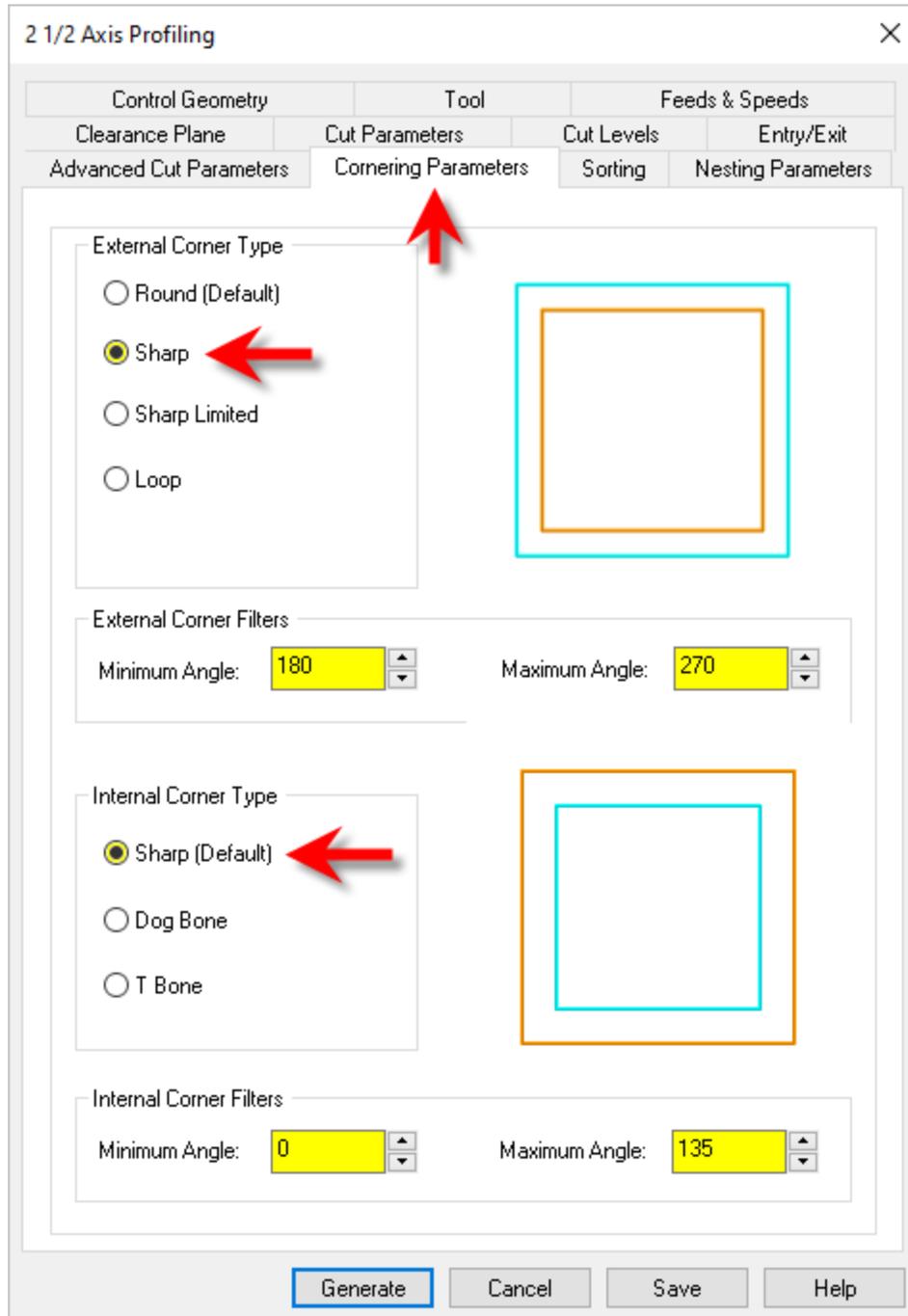


2. Under the [Cut Arc Fitting](#) section, check the box to [Perform Arc Fitting](#) and then enter a value of **0.002** for [Fitting Tolerance \(t\)](#). Refer to the image in the dialog for how this value is calculated. As a rule, the arc [Fitting Tolerance \(t\)](#) should be at least **2 times** the [Global Tolerance](#) that you set on the [Cut Parameters](#) tab.
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.

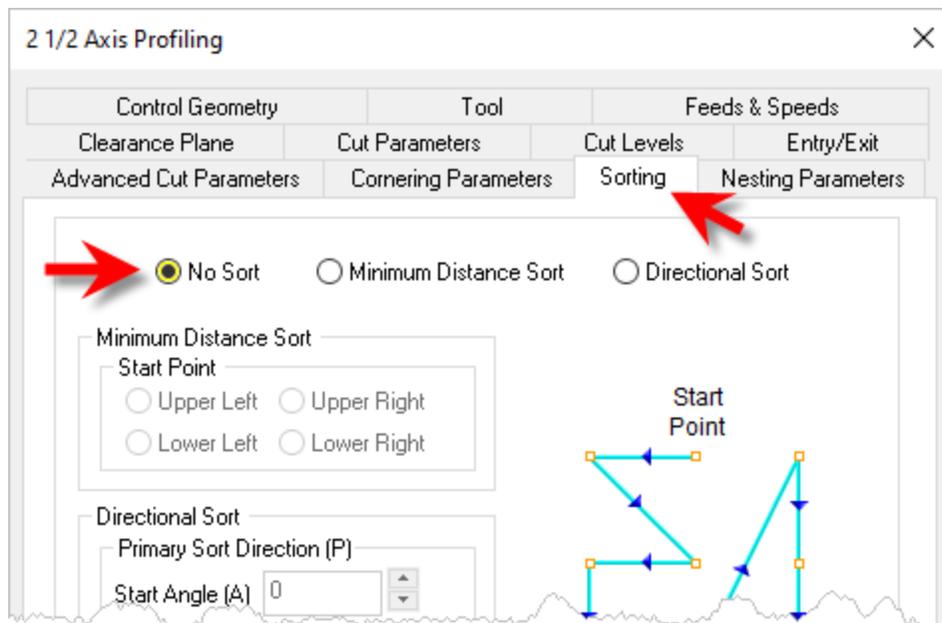


2. For the **External Corner Type** select the **Sharp** option. This will force the cutter to extend past the corner before changing directions (i.e., the cutter will not ride on the corner). Then set **Minimum Angle** to **180** and **Maximum Angle** to **270** as shown in the dialog above.
3. Then for **Internal Corner Type**, select **Sharp**. This is the default selection and you can accept the default values for **Minimum** and **Maximum Angle** also.

7.1.11 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 you 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.

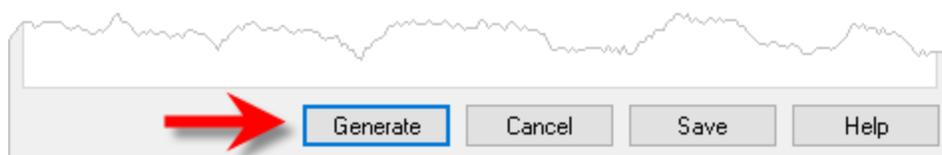


2. Select **No Sort**.

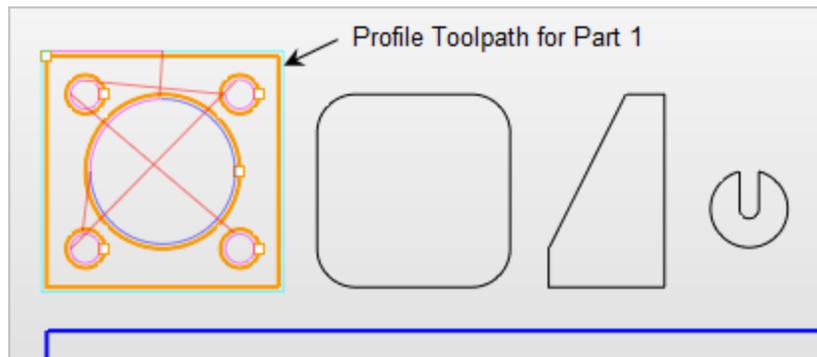
7.1.12 Profile 1 Generate

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

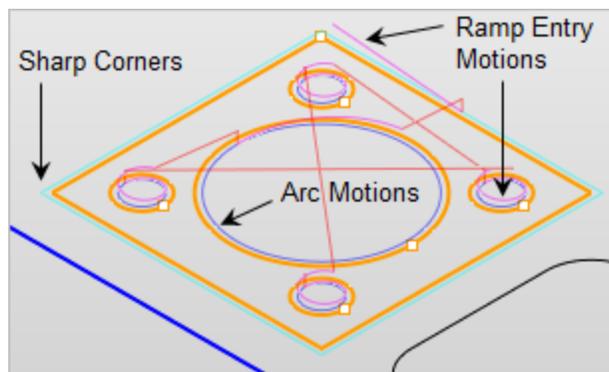
1. With the **2-1/2 Axis Profile** operation dialog still displayed, select the **Generate** button at the bottom of the dialog.



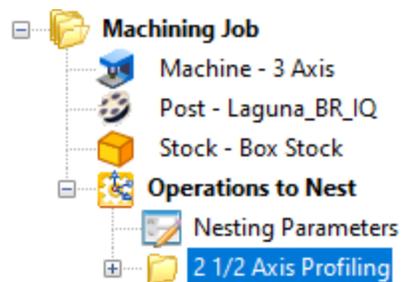
2. You will see that the 2-1/2 Axis Profile toolpath operation was created for Part #1.



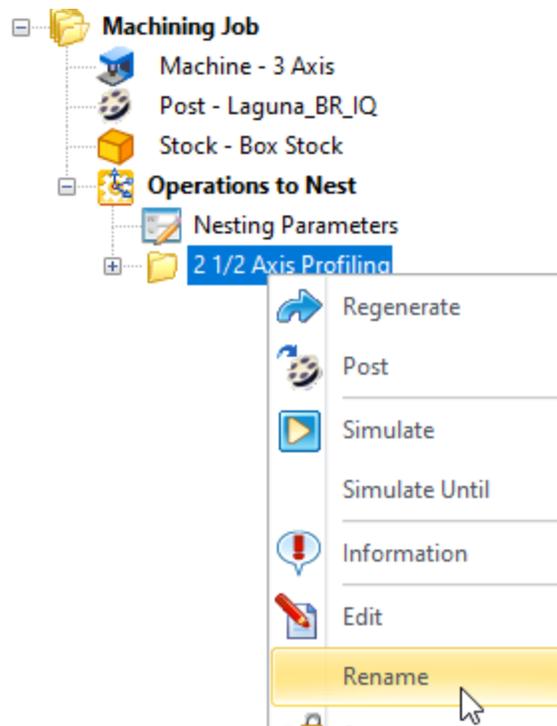
3. You will also see that the toolpath has the 5 degree ramp entry, sharp corners, arc motions and nested cut side.



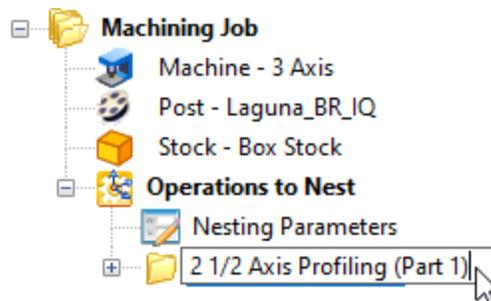
4. You will also see that the Profile operation was added to the Machining Job Tree.



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.



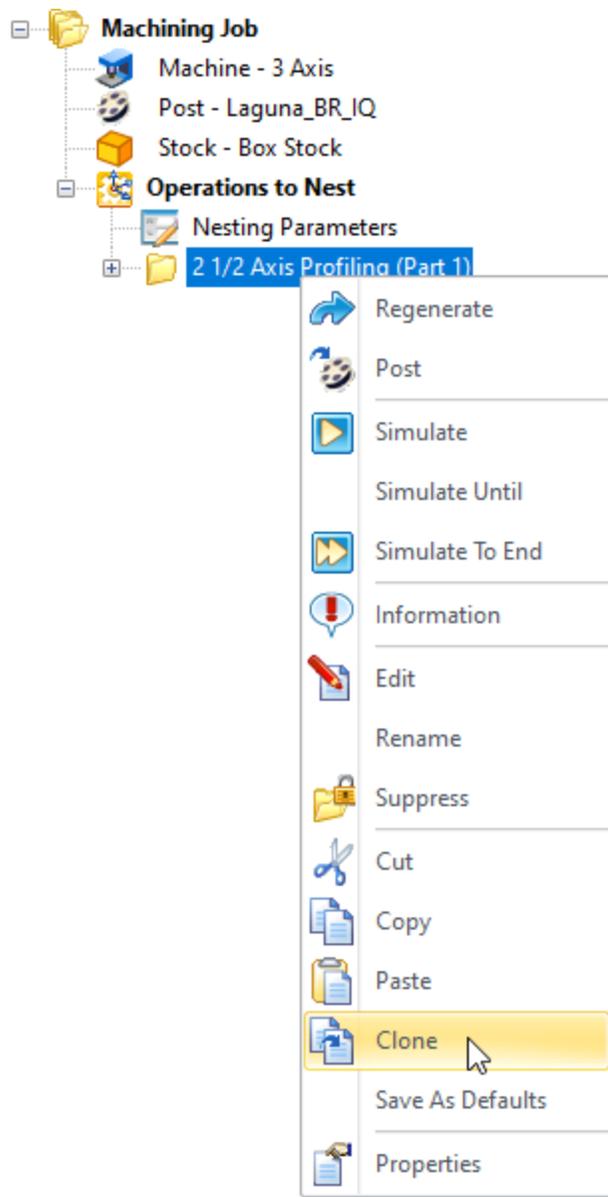
7.2 Profile Parts 2,3 & 4

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.

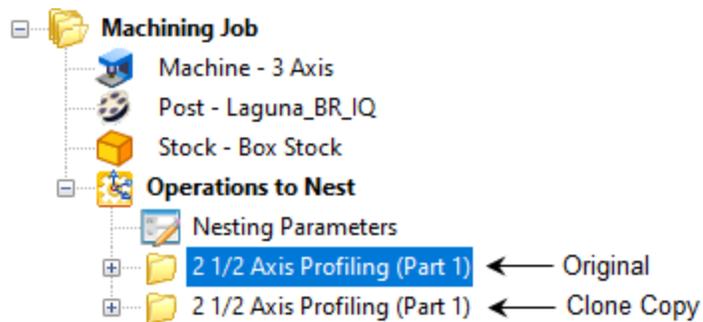
7.2.1 Clone Profile Operations

In this step we will **Clone** our **Profile** operation to make 3 additional copies.

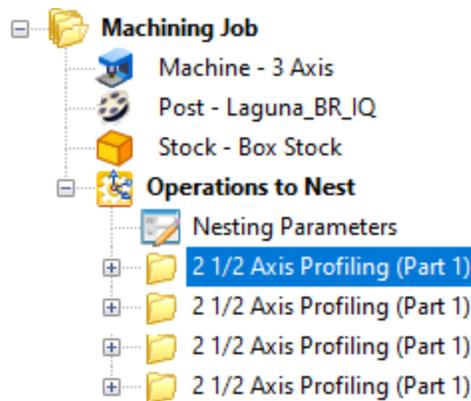
1. First make sure the **2 1/2 Axis Profiling (Part 1)** operation is selected from the **Machining Job** tree. Then right-click on the selected operation and select **Clone** from the menu. This is shown below.



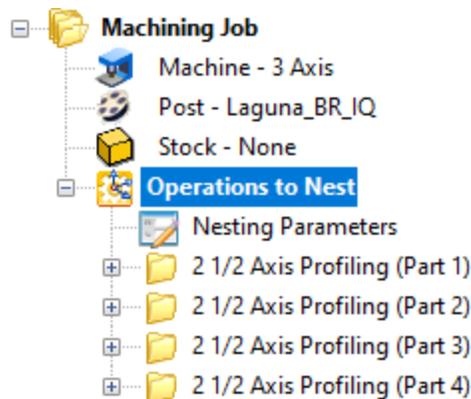
2. You will see that a second exact copy of the operation was added below the original.



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



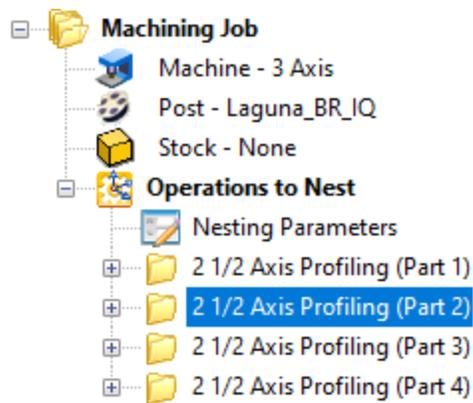
- Now rename the 3 copied operations for [Part 2](#), [3](#) and [4](#) so that they appear as shown below. If you forgot how to rename an operation go back and revisit topic [Profile 1 Generate](#) to learn how.



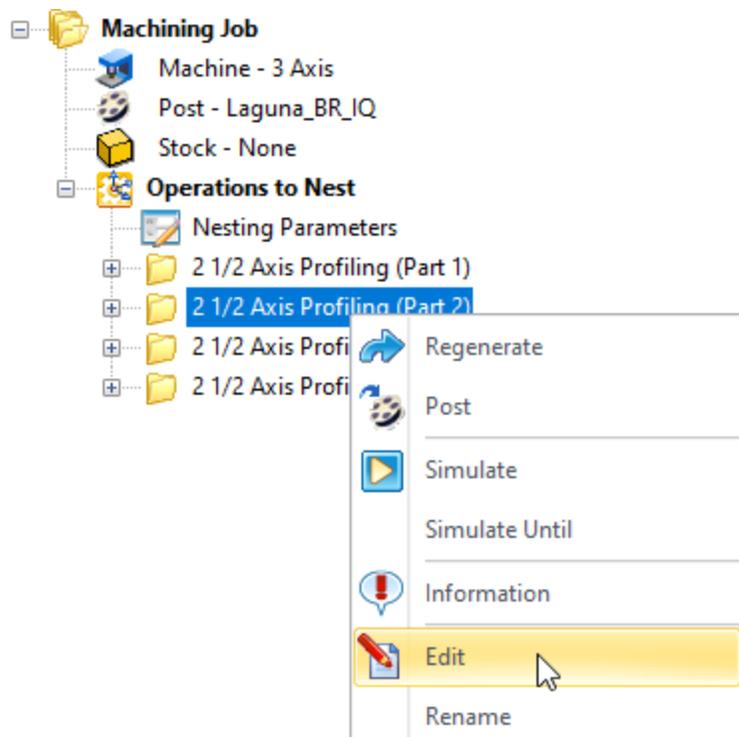
7.2.2 Edit Profile Operations

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

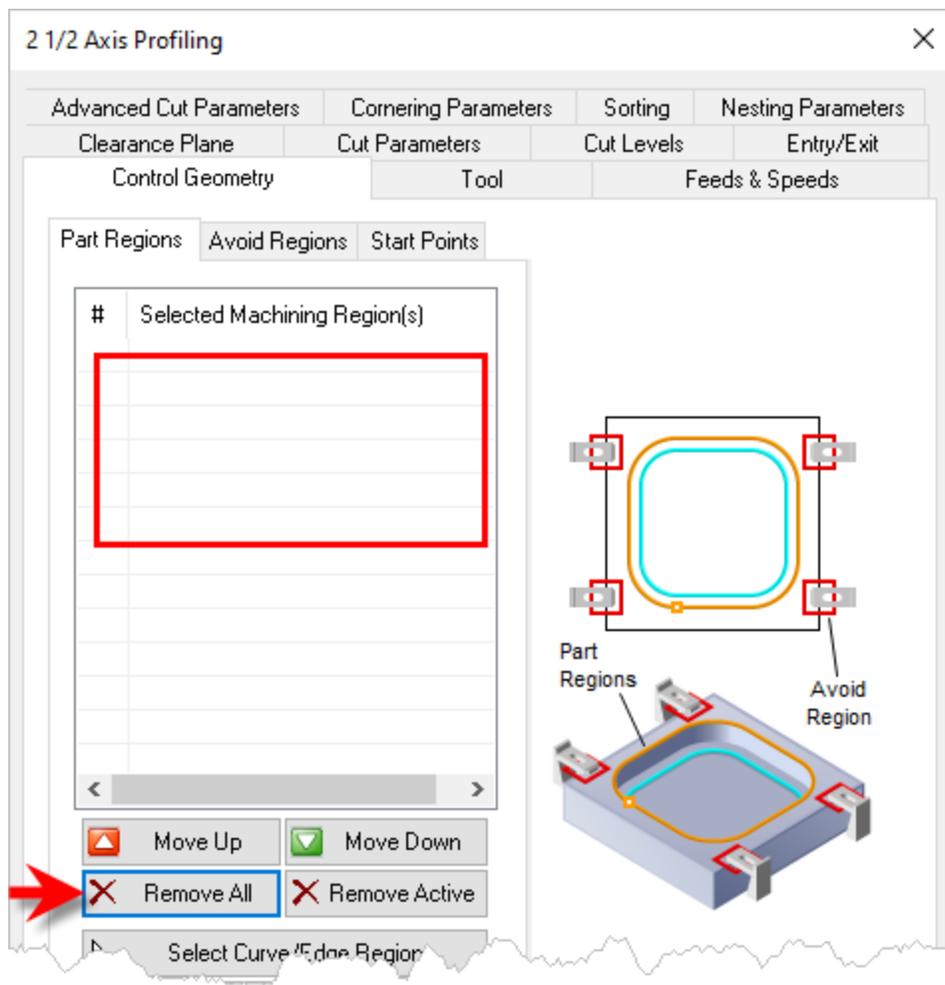
- First select the [Profile](#) operation for [Part 2](#). This will be [2 1/2 Axis Profiling \(Part 2\)](#) under the [Operations to Nest](#) portion of the [Machining Job](#).



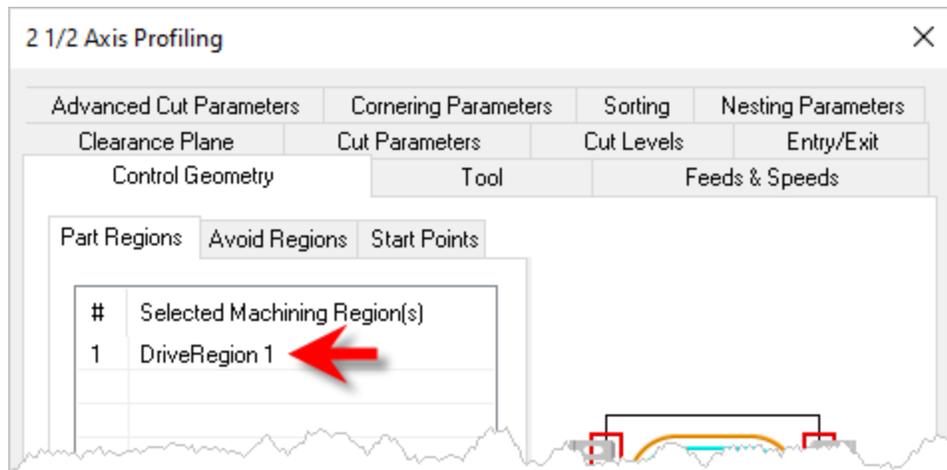
2. Now right-click on the selected operation again and select **Edit**.



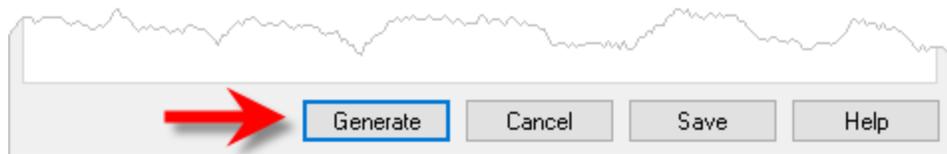
3. The Profiling operation dialog will display with the **Control Geometry** tab selected by default.



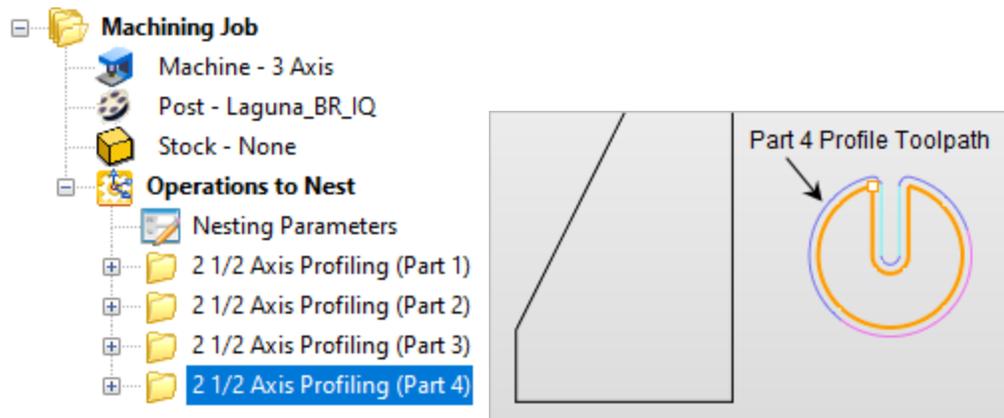
4. We don't want to use the same **Part Regions** so select the **Remove All** button to remove them from the list.
5. Now pick the **Select Curve/Edge Regions** button. The dialog will minimize and prompt you to select curve or edge regions. Now window select all of the curves that make up **Part #2** and then right-click or simply press the **<Enter>** key 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.

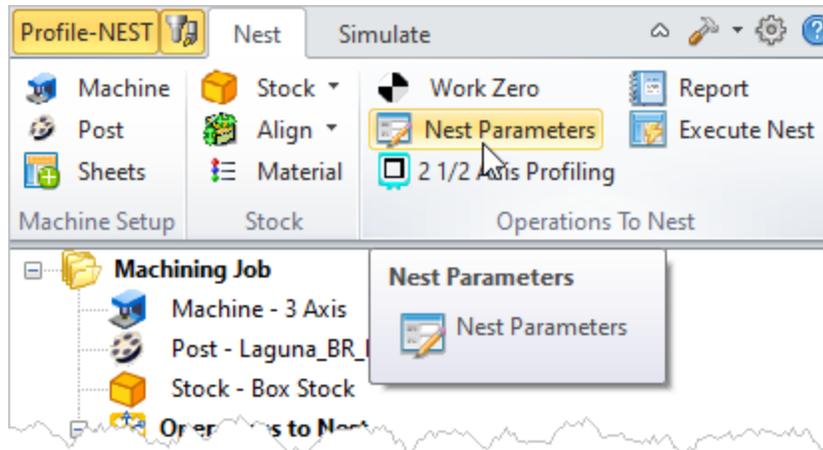


The Nested Sheets

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](#)

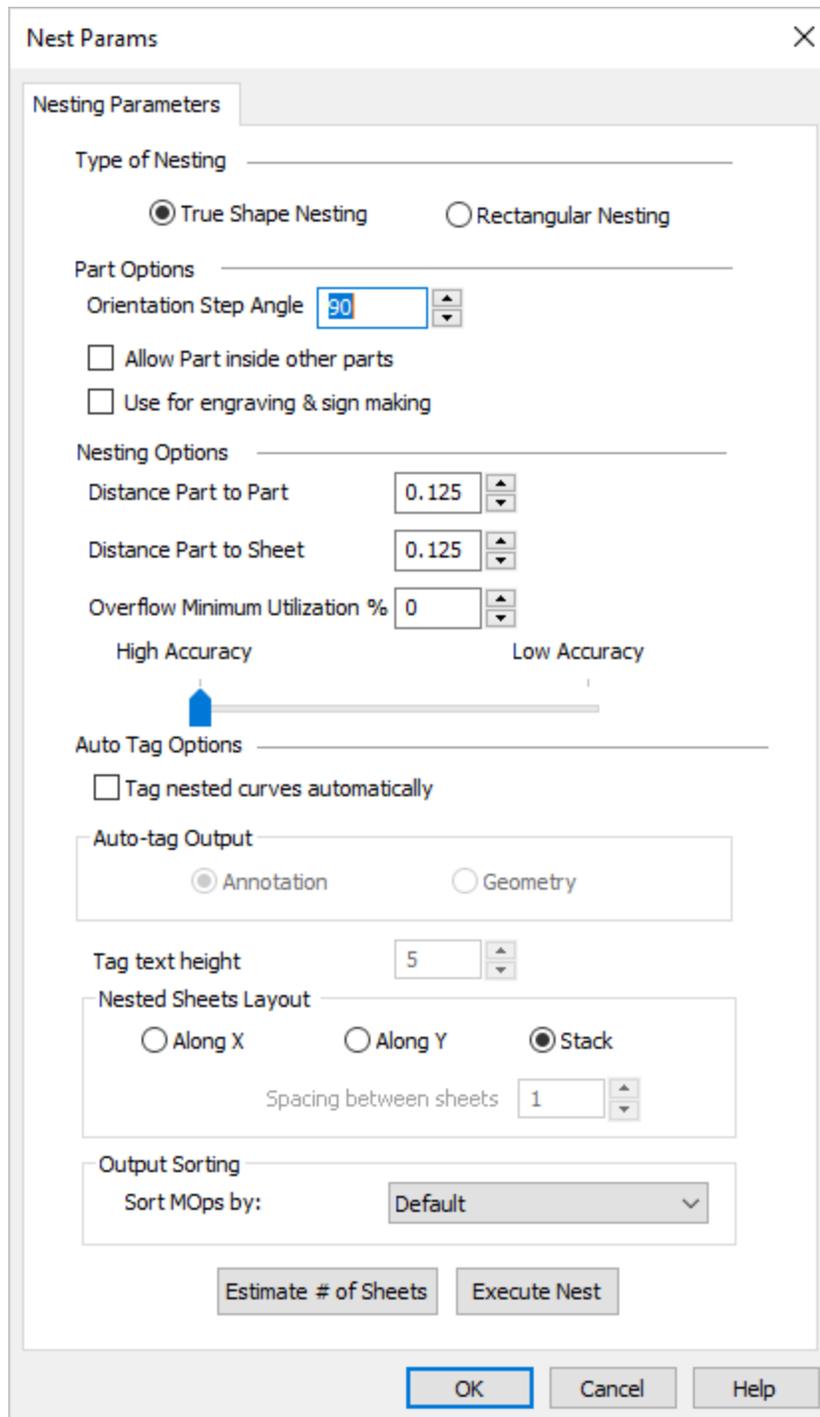
Distance Part to Part : [0.125](#)

Distance Part to Sheet : [0.125](#)

Accuracy : [High](#)

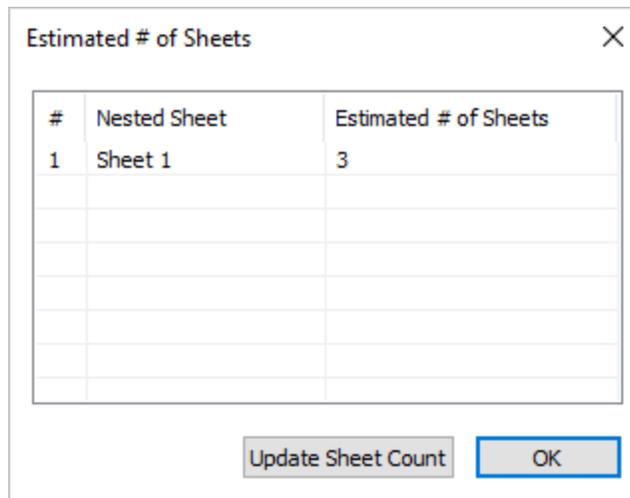
Tag nested curves automatically : [Unchecked](#)

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.



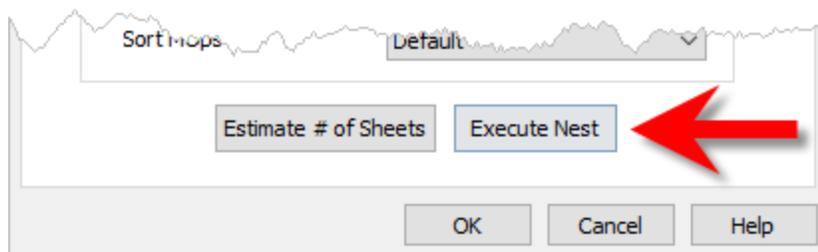
For this guide we will not be using the [Auto Tag](#) options.

3. Select the [Estimate # of Sheets](#) button.
4. From the [Estimate # of Sheets](#) dialog pick the [Update Sheet Count](#) button and pick [OK](#).

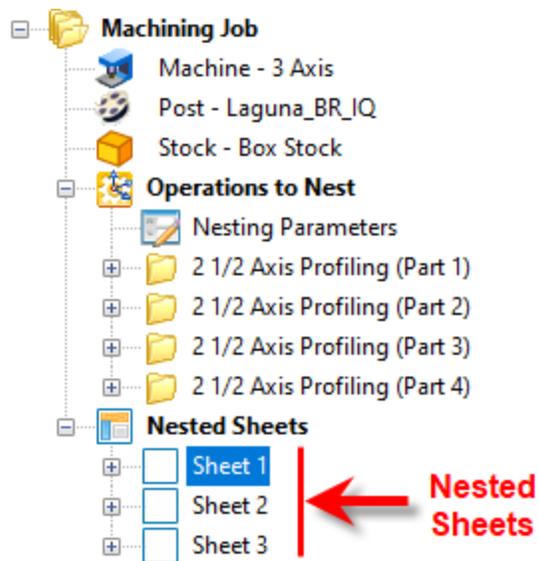


💡 On this dialog you can use the [Orientation Step Angle](#) value in conjunction with the [Estimate # of Sheets](#) button to maximize the efficiency factor (i.e., adjust the step angle to achieve a minimum number of sheets).

- Now from the [Nest Parameters](#) dialog select the [Execute Nest](#) button and then pick [OK](#) to close the dialog.



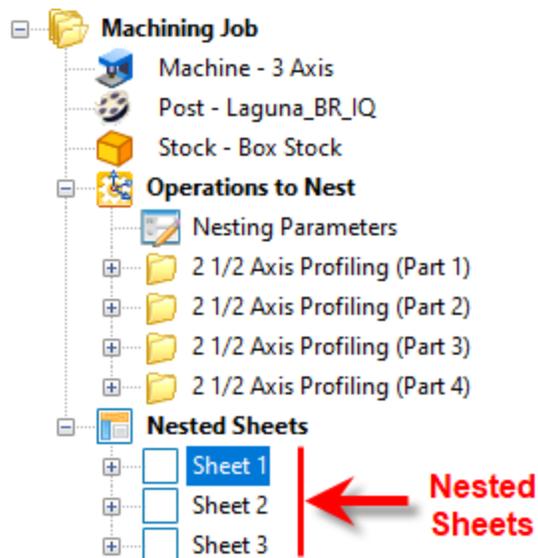
- You will see that [Nested Sheets](#) were created and added to the [Machining Job](#).

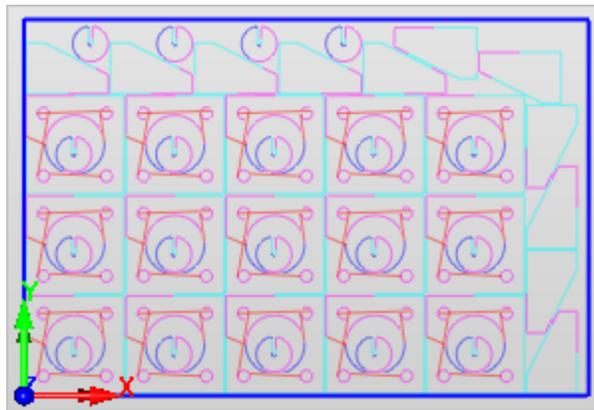


8.2 Reviewing the Nest

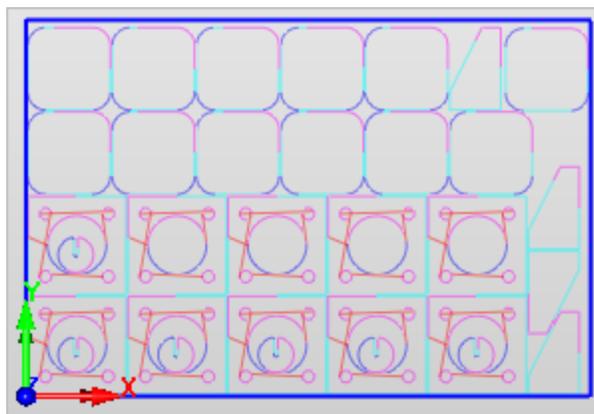
While the profile operations were being generated for [Parts 1-4](#) the nested sheets were also created and updated automatically. In this section we will have a look at the resulting nested sheets.

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.

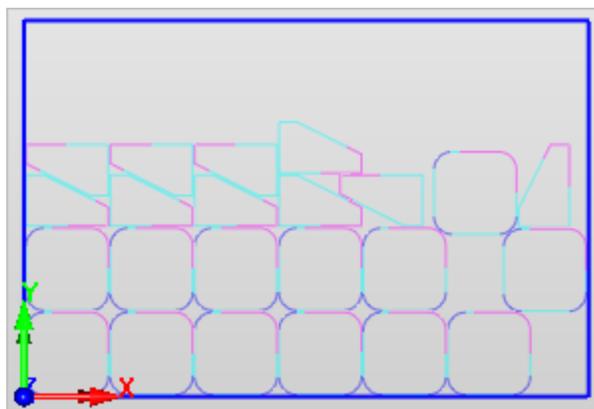




Sheet 1

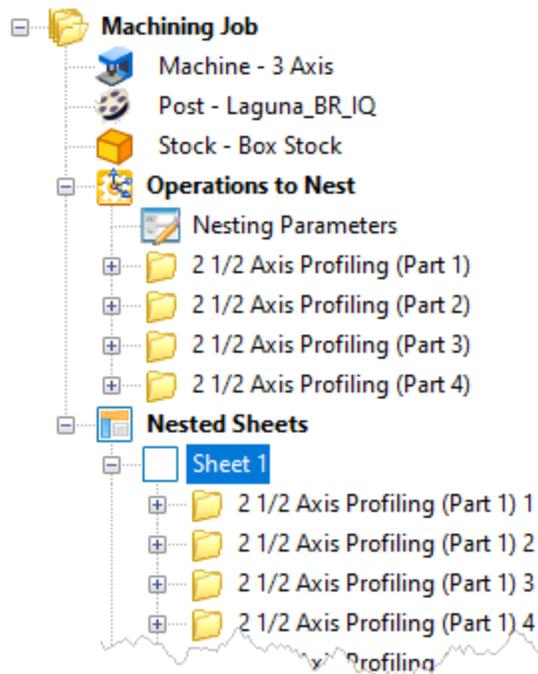


Sheet 2

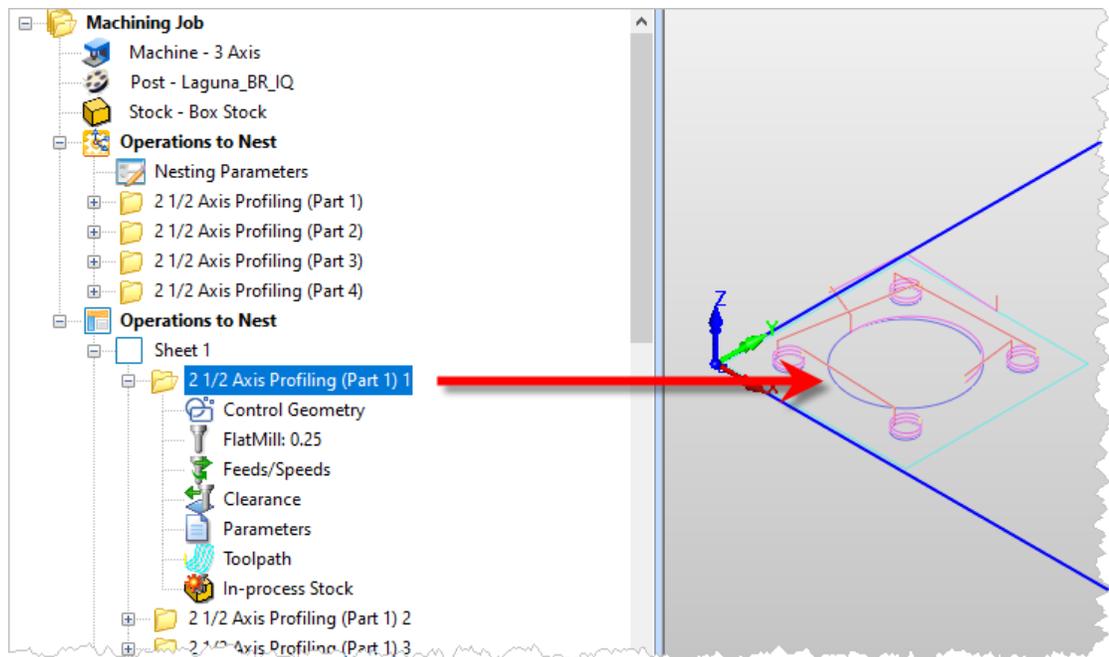


Sheet 3

2. You can also expand each sheet to see all of the [Profile](#) operations contained within it.



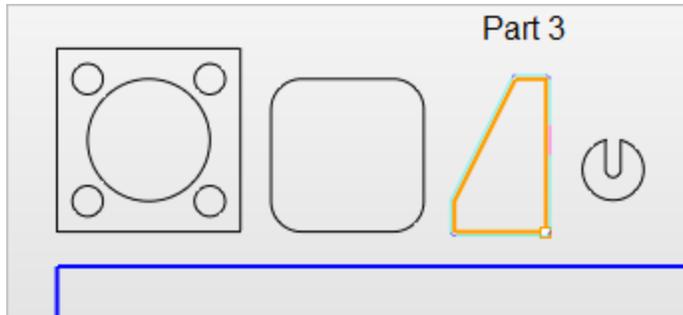
3. You can also expand each **Profile** operation folder within each **Sheet** to gain access to that **Profile's** operation parameters. Each **Profile** is a self-contained toolpath operation just like the original operation.



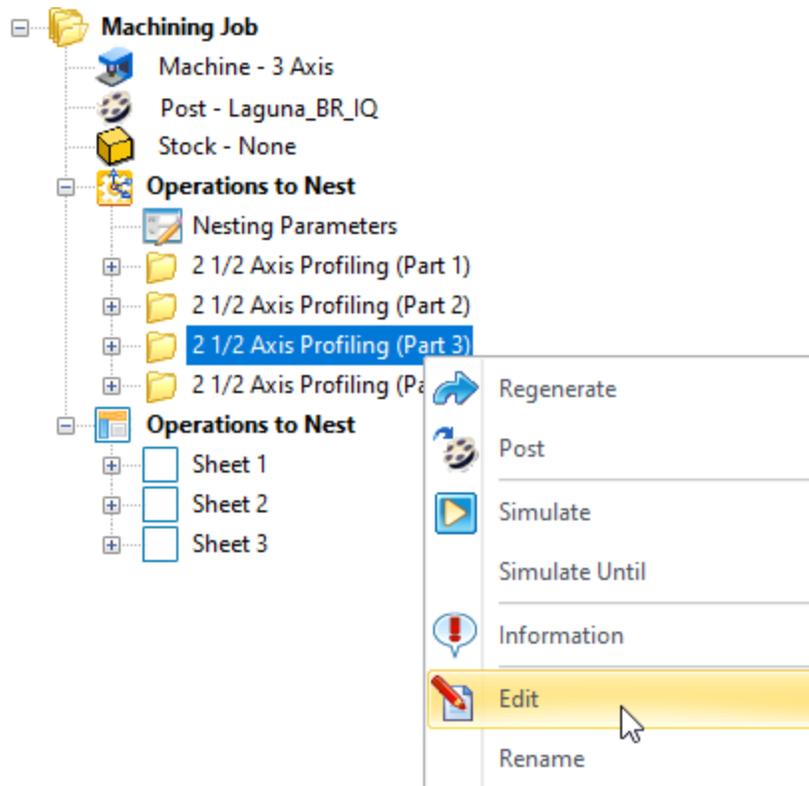
8.3 Editing the Nest

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.

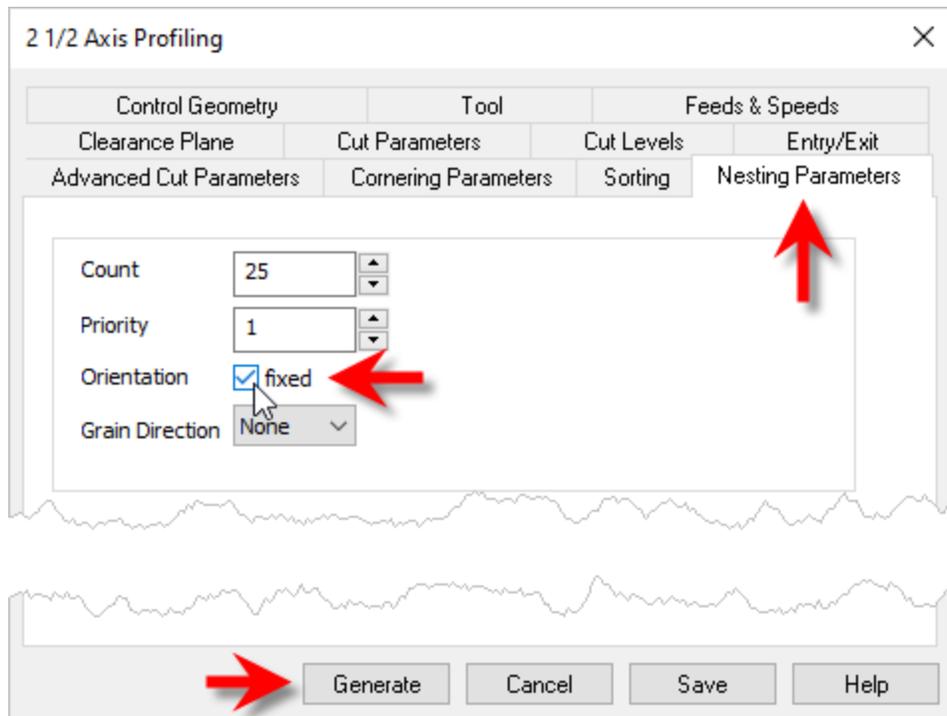
1. You can edit your nest by editing the original [Profiling](#) operations located under the [Operations to Nest](#) section of the [Machining Job](#). For example, we discovered that [Part 3](#) needs to maintain its exact orientation as the original Profile.



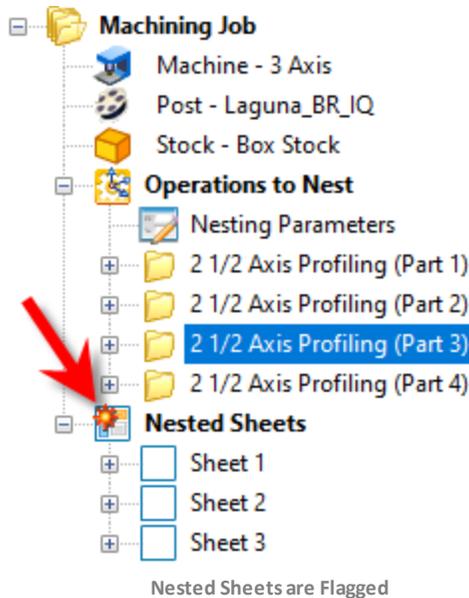
2. You can "fix" the orientation of one of your profiles by editing that profile's nesting parameters. Select [Part 3](#) from the [Operations to Nest](#) section of the [Machining Job](#) and then right-click and select [Edit](#) from the menu.



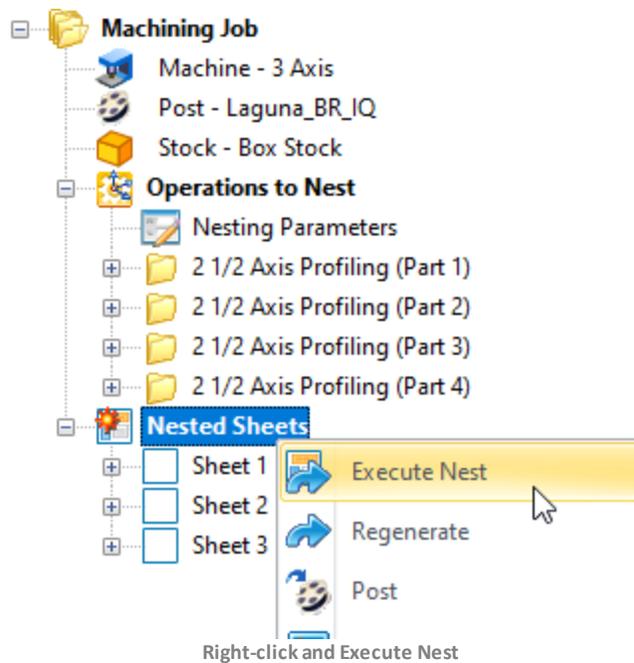
3. From the [Profile](#) operation dialog for [Part 3](#), go to the [Nesting Parameters](#) tab and check the box next to [Fixed](#) and then pick [Generate](#) from the bottom of the dialog.



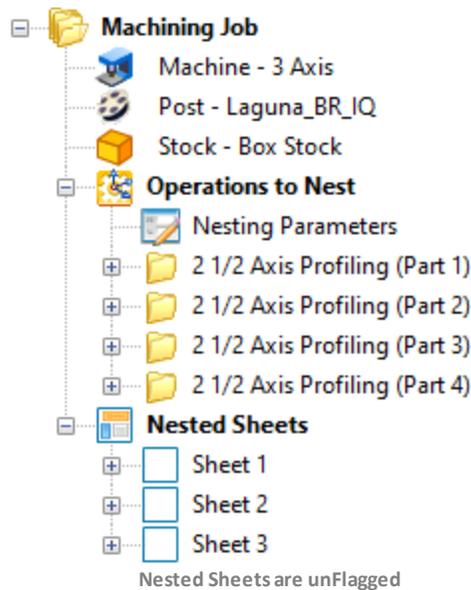
- Pick **OK** from the **Execute Nest** message dialog. You will notice that the Nested Sheets was flagged. This means that the nest needs to be executed again.



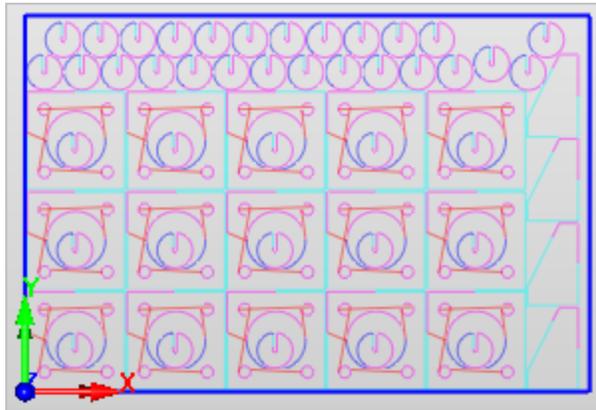
- Select **Nesting Sheets** from the **Machining Job**, right-click and select **Execute Nest**.



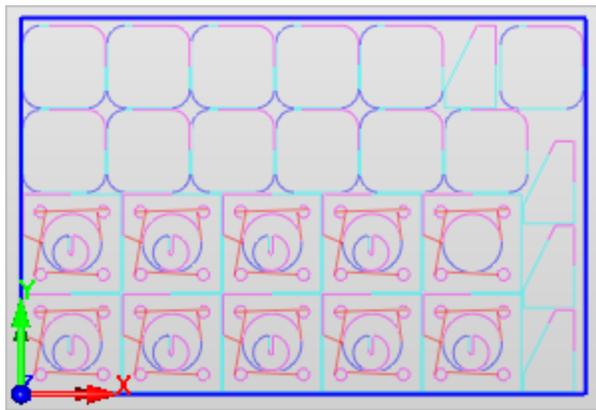
6. The flag on **Nested Sheets** is now gone:



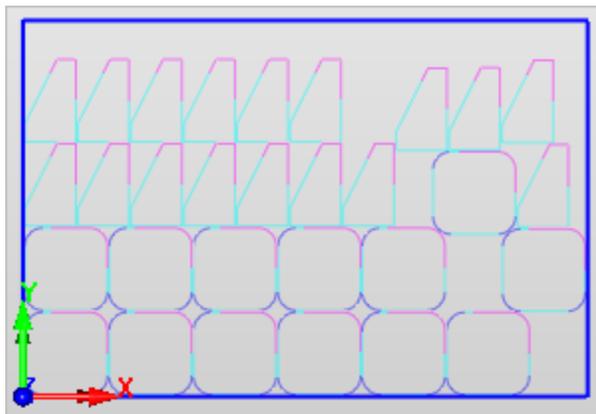
7. Select each nested sheet and now see that **Part 3** is fixed in the same orientation as the original profile.



Sheet 1

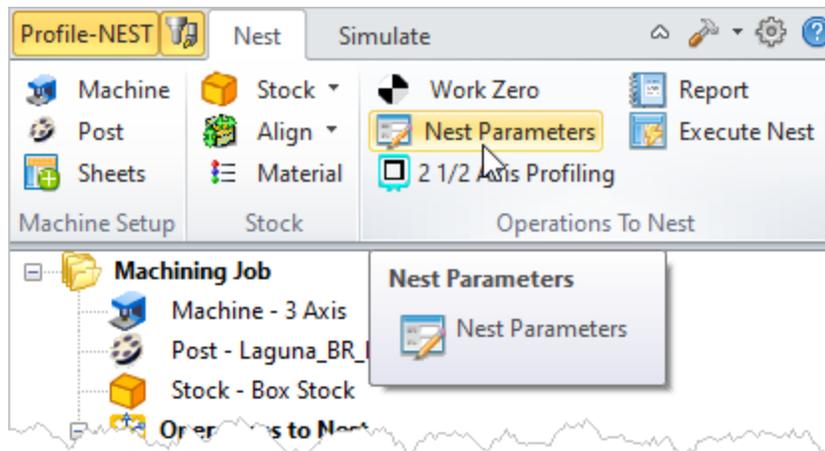


Sheet 2



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 by making changes to the [Nesting Parameters](#) dialog located on the [Nest](#) tab of the [Profile-NEST Browser](#).



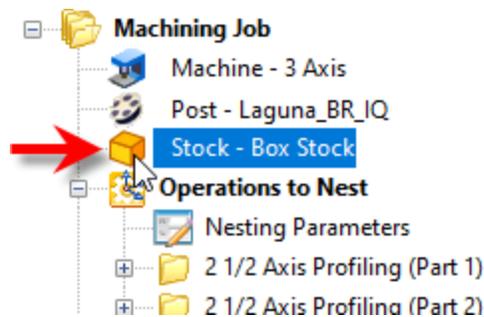
8.4 Simulating the Nest

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.

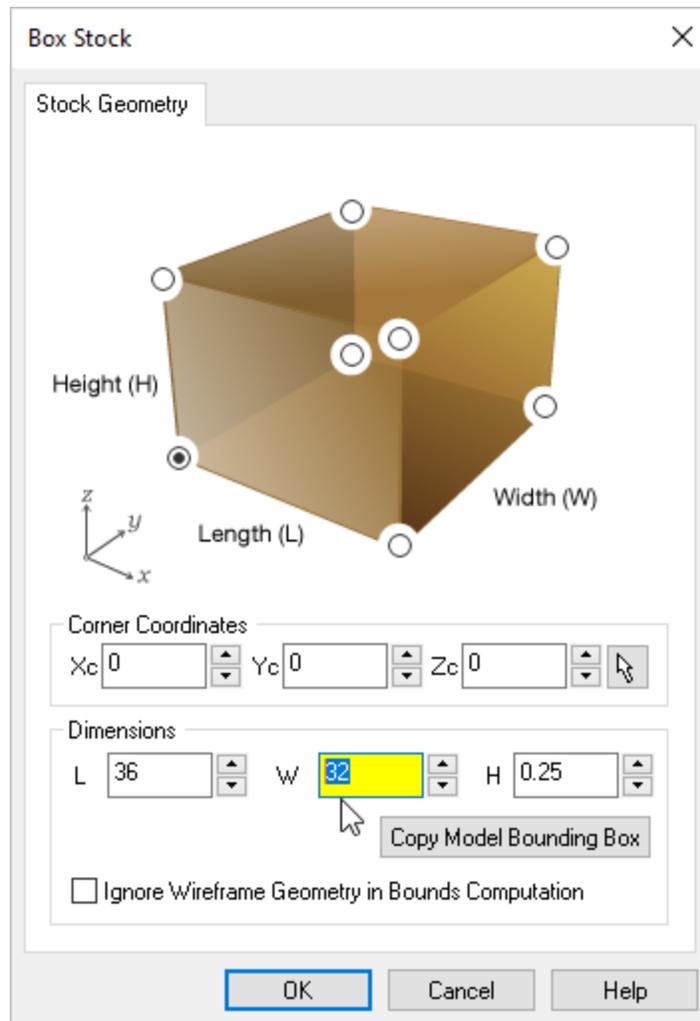
To Simulate the Original Profiling Toolpaths

First adjust the [Stock](#) size so that it encompasses the original [Profiling](#) operations.

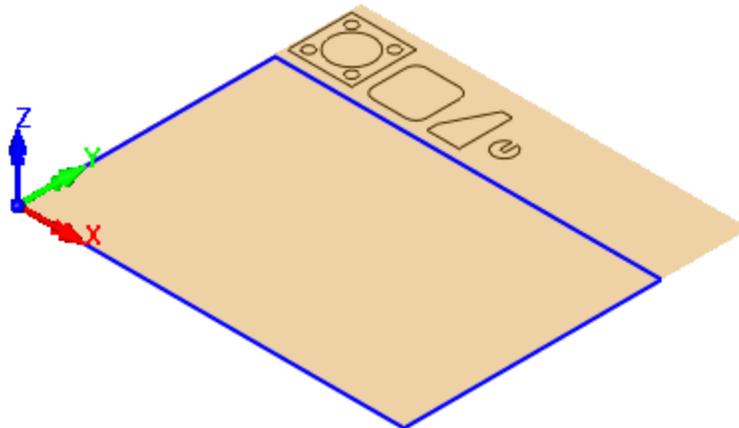
1. From the [Machining Job](#), double-left-click on the [Stock](#) icon to display the [Box Stock](#) dialog.



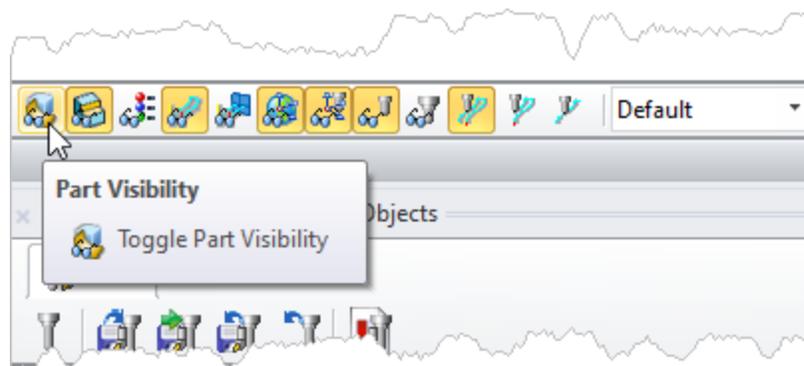
2. From the dialog, change the [W](#) dimension to [32](#) and then pick [OK](#).



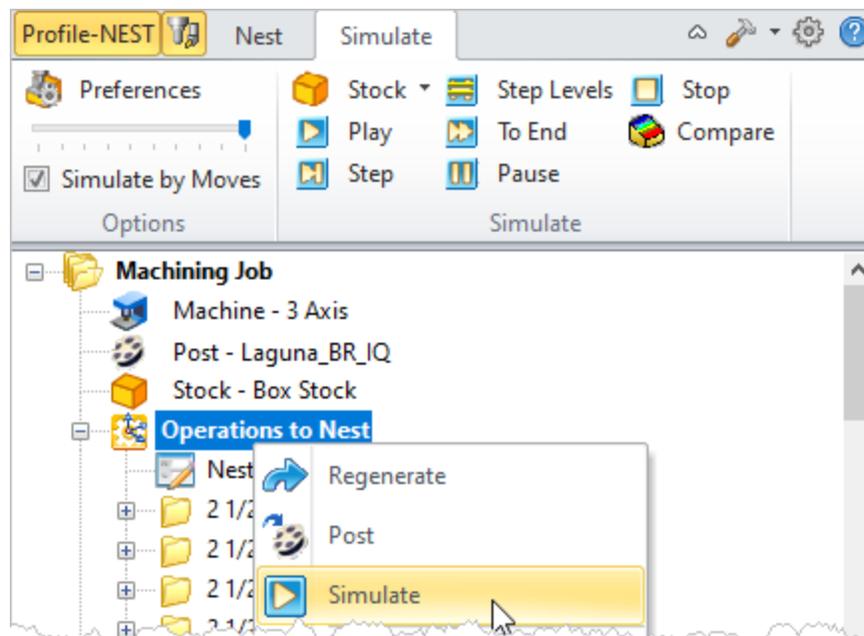
You will see that the [Stock](#) displayed on the screen is large enough to encompass the original profile operations.



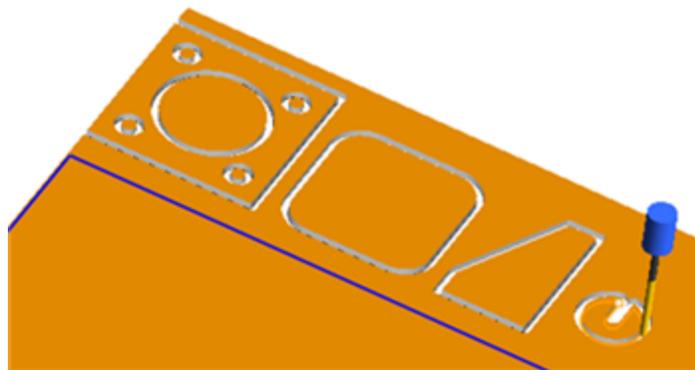
If you do not see the stock on the screen, select the [Toggle Stock Visibility](#) icon from the base of the [Machining Browser](#):



3. Select the [Simulate](#) tab.
4. Now select the [Operations to Nest](#) folder, right-click and select [Simulate](#).



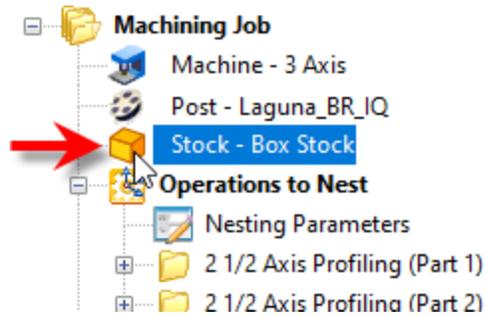
5. Verify the cut material simulation.



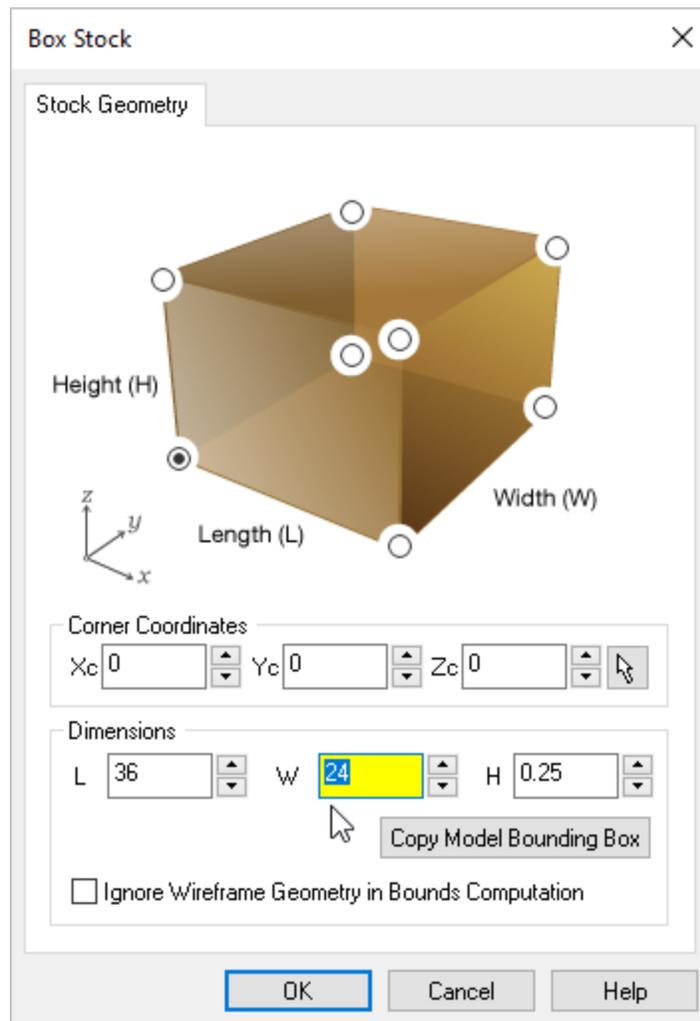
 To Simulate a Nested Sheet

First adjust the [Stock](#) size back to the sheet size.

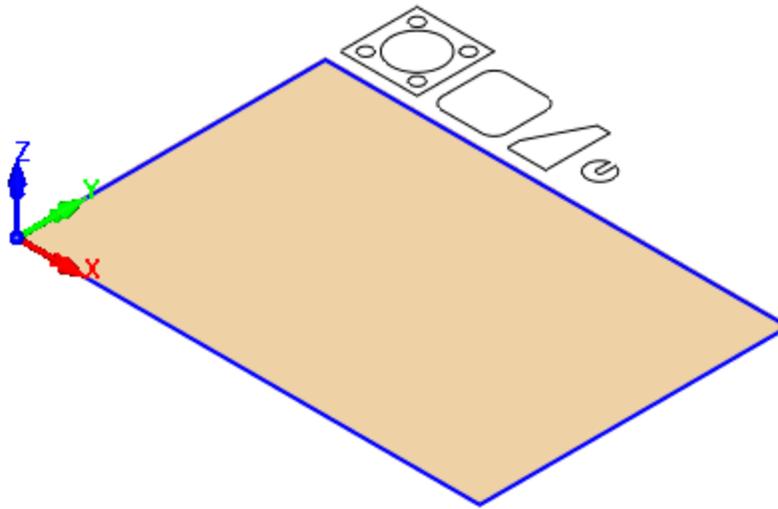
1. From the [Machining Job](#), double-left-click on the [Stock](#) icon to display the [Box Stock](#) dialog.



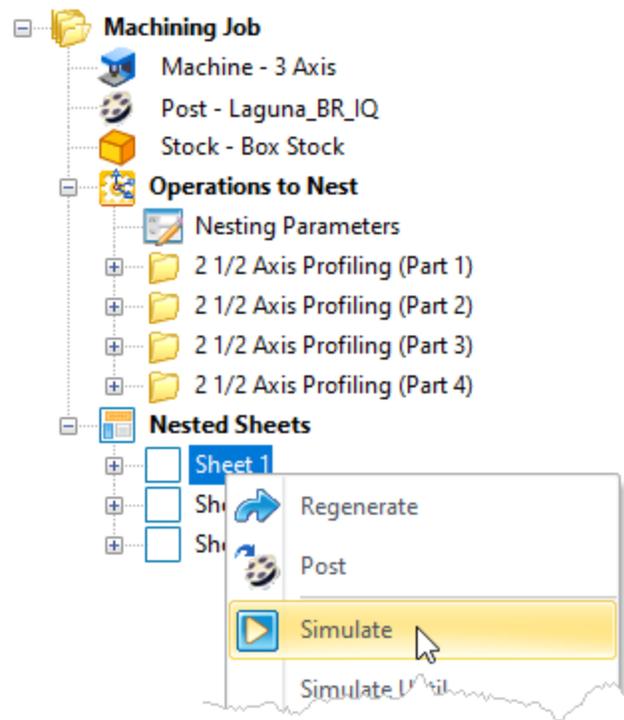
2. From the dialog, change the [W](#) dimension to [24](#) and then pick [OK](#).



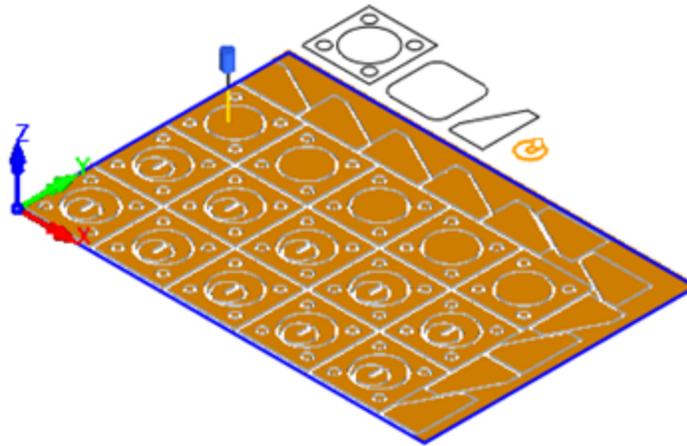
You will see that the **Stock** displayed on the screen is large enough to encompass the original profile operations.



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



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.

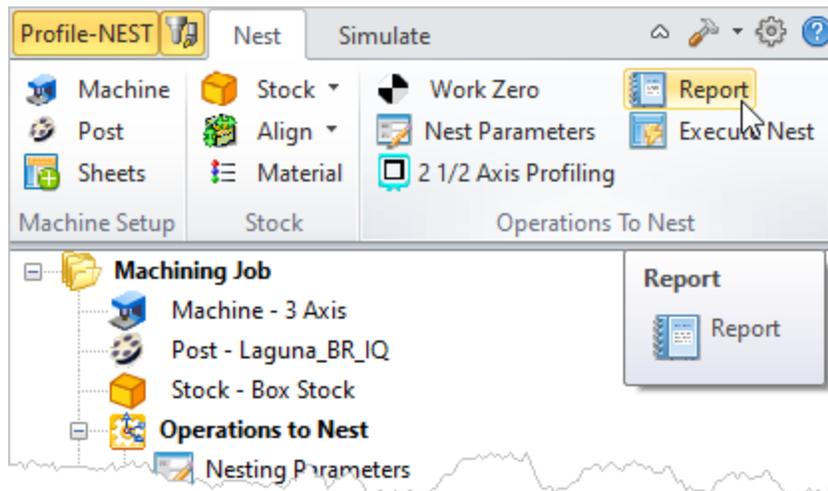


8.5 Generate Reports

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.



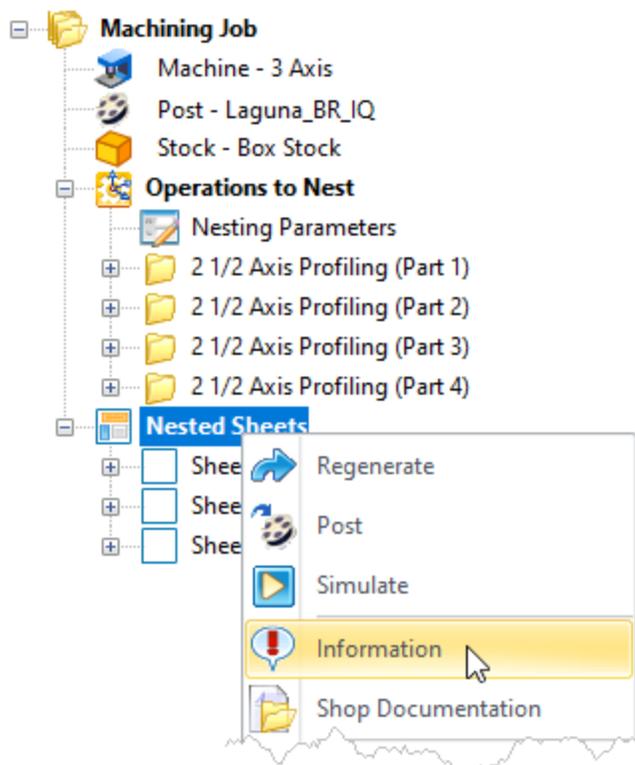
2. The report provides information about each nested sheet including the [% Utilization](#) and the total number of each [Profiling](#) operation located on each [Nested Sheet](#). You can [Print](#) the report if desired.

#	Nested Sheet	% Utilization	2 1/2 Axis Profiling (Part 1)	2 1/2 Axis Profiling (Part 2)
1	Sheet 1-1	77.40	15	0
2	Sheet 1-2	81.43	10	12
3	Sheet 1-3	57.65	0	13

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.

Machining Operations Information

Mops Information

Name	Sta...	Tool	Tool #	Cut Feed	Spindle Speed	# of GOTOs	Machining Time
2 1/2 Axis Profiling (Part 4) 29	Clean	FlatMill: 0.25	2	14.67 in/min	24446 RPM	128	0.97 min
2 1/2 Axis Profiling (Part 4) 30	Clean	FlatMill: 0.25	2	14.67 in/min	24446 RPM	128	0.97 min
2 1/2 Axis Profiling (Part 4) 31	Clean	FlatMill: 0.25	2	14.67 in/min	24446 RPM	128	0.97 min
2 1/2 Axis Profiling (Part 4) 32	Clean	FlatMill: 0.25	2	14.67 in/min	24446 RPM	128	0.97 min
						Sub-total	1 hr 49 min
Sheet 2							
2 1/2 Axis Profiling (Part 1) 1	Clean	FlatMill: 0.25	2	14.67 in/min	24446 RPM	914	5.89 min
2 1/2 Axis Profiling (Part 1) 2	Clean	FlatMill: 0.25	2	14.67 in/min	24446 RPM	914	5.89 min
2 1/2 Axis Profiling (Part 1) 3	Clean	FlatMill: 0.25	2	14.67 in/min	24446 RPM	914	5.89 min
2 1/2 Axis Profiling (Part 1) 4	Clean	FlatMill: 0.25	2	14.67 in/min	24446 RPM	914	5.89 min
2 1/2 Axis Profiling (Part 1) 5	Clean	FlatMill: 0.25	2	14.67 in/min	24446 RPM	914	5.89 min

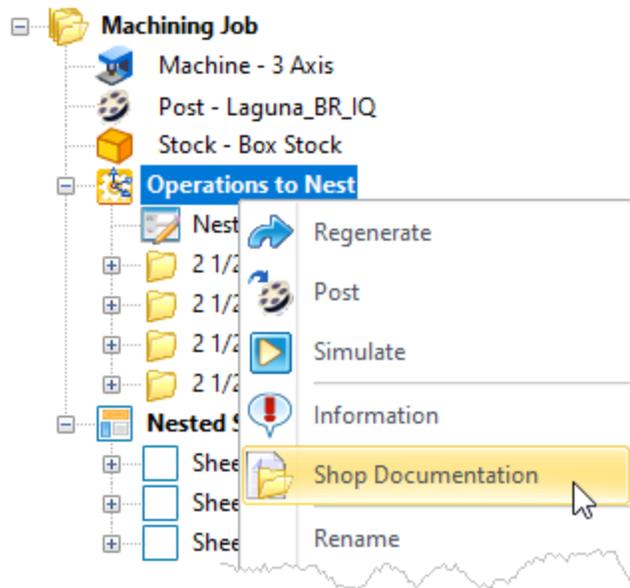
Print

OK Cancel Help

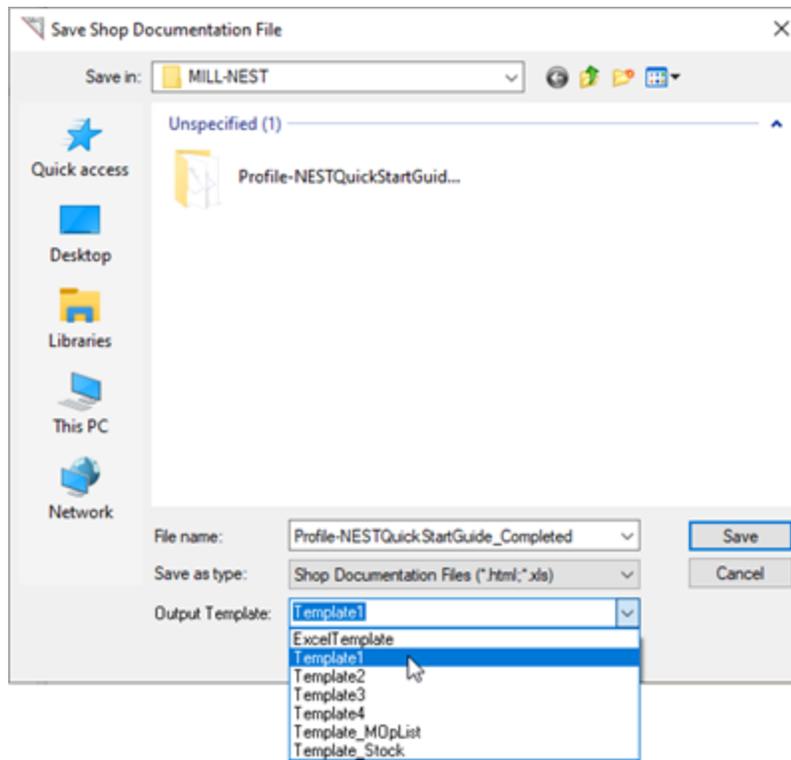
8.5.3 Shop Documentation

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

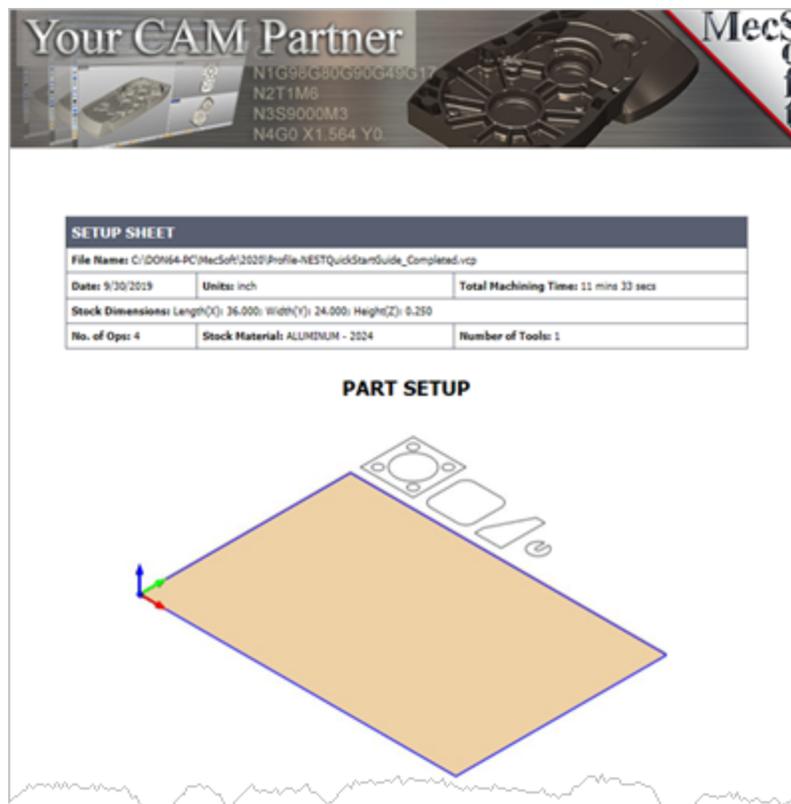
1. From the [Machining Job](#) select [Operations to Nest](#) and then right-click and select [Shop Documentation](#).



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.



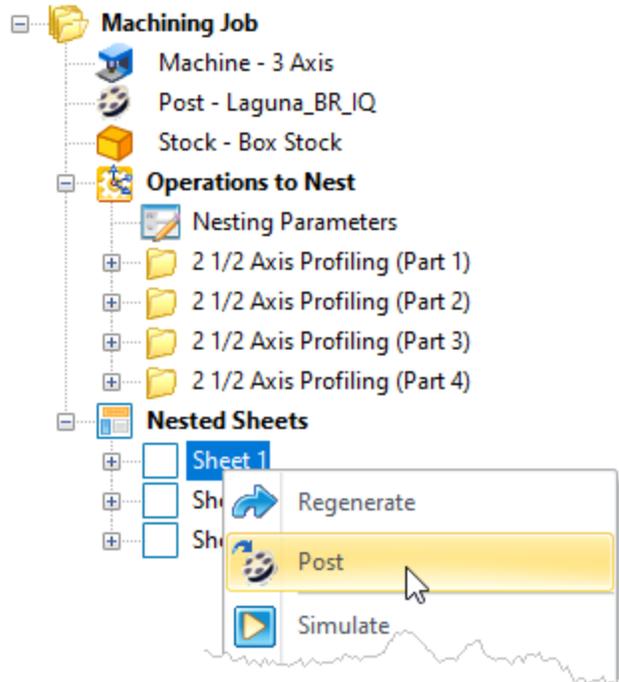
- Pick **Save** to generate the **Shop Documentation**.



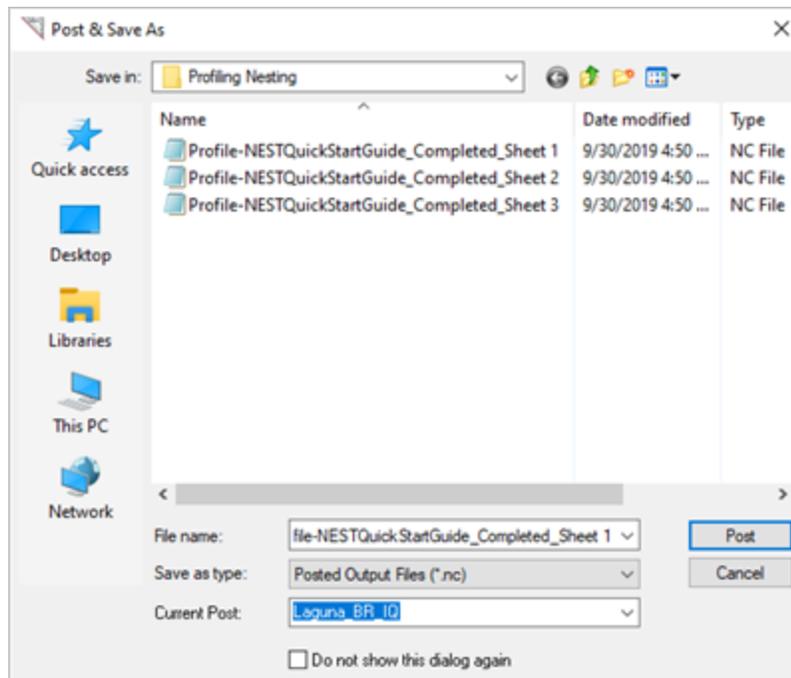
8.6 Post G-Code

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.



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

```

File Edit Format View Help
N1 G00 G17 G40 G90
N2 G70
N3 G54
N4 (Sheet 1)
N5 (2 1/2 Axis Profiling (Part 1) 1)
N6 T1 M06
N7 D1
N8 M03 S20000
N9 G00 X5.5889 Y1.4102
N10 G00 Z0.5000
N11 G01 Z0.3000 F29.3
N12 G01 X5.5856 Y1.4174 Z0.2996 F11.0
N13 G01 X5.5770 Y1.4335 Z0.2986
N14 G01 X5.5677 Y1.4492 Z0.2977
N15 G01 X5.5576 Y1.4645 Z0.2967
N16 G01 X5.5468 Y1.4792 Z0.2958
N17 G01 X5.5353 Y1.4934 Z0.2948
N18 G01 X5.5231 Y1.5070 Z0.2938
N19 G01 X5.5103 Y1.5200 Z0.2929
N20 G01 X5.4968 Y1.5323 Z0.2919
N21 G01 X5.4828 Y1.5440 Z0.2910
N22 G01 X5.4682 Y1.5550 Z0.2900

```

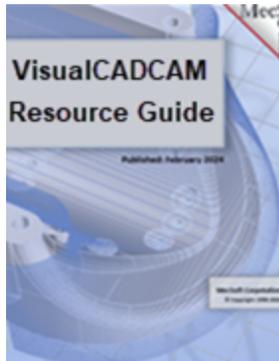
Ln 1, Col 1 100% Windows (CRLF) UTF-8

Where to go for more help

Download this PDF Guide for a list of the available [VisualCAD/CAM Resources](#).



2026 VisualCAD/CAM Resource Guide



The 2026 VisualCAD/CAM Resource Guide!

18 Pages

Lists PDF downloads and Online resources including [Quick Start Guides](#), [Reference Guides](#), [Exercise Guides](#), [Tutorials](#) and [More](#).

[Prefer Printed Documentation? Click Here!](#)

[What's New](#) | [Quick Start Play List](#)

Index

- A -

About

- the Profile-NEST Module 10
- the VisualCAM Display 11
- This Guide 9
- using this Guide 10
- Watch the Video! 10

Align Part and Stock 28

- C -

- Copy & Paste Profile Operations 55
- Cornering Parameters 52
- Create a Tool 39
- Create Stock Geometry 26
- Cut Levels 48
- Cut Parameters 46
- Cut Parameters Advanced 51

- D -

- Define Sheet to Nest 22
- Define the Machine Tool 20

- E -

- Edit Profile Operations 57
- Editing the Nest 67
- Entry/Exit 49

- F -

- Feeds & Speeds 45

- G -

- Generate Profile 1 53
- Getting Ready
 - About the VisualCAM Display 11
 - Load the Part Model 15
 - Load the Profile-NEST Module 12

- Machining Strategy 17
- Main Programming Steps 18
- Operational Sequence 18
- Running VisualCAD/CAM 11

- L -

- Load the Part Model 15
- Load the Profile-NEST Module 12

- M -

- Machine Setup 20
 - Define 20
 - Define Sheet to Nest 22
 - Select the Post Processor 20
- Machining Reports 77
- Machining Strategy 17
- Machining Time 77
- Main Programming Steps 18

- N -

Nest

- Editing the Nest 67
- Machining Reports 77
- Machining Time 77
- Nesting Reports 76
- Post Processing 80
- Reviewing the Nest 64
- Shop Documentation 78
- Simulating the Nest 71

- O -

- Operational Sequence 18

- P -

Post

- Select 20
- Post Processing 80
- Profile Part 1
 - Advanced Cut Parameters tab 51
 - Clearance Plane 46
 - Clearance Plane tab 46

Profile Part 1

Control Geometry tab	34
Cornering Parameters tab	52
Cut Levels tab	48
Cut Parameters tab	46
Entry/Exit tab	49
Feeds & Speeds tab	45
Generate	53
Nesting Parameters tab	37
Sorting tab	53
Tool tab	39

Profiles to Nest 33

Profile Part 1	33
Profile Parts 2,3 & 4	55

- W -

Where to go for more help 82

- Q -

Quick Start 5

- R -

Reports	76
Resource Guide	8
Reviewing the Nest	64
Running VisualCAD/CAM	11

- S -

Select the Post Processor	20
Set Nesting Parameters	61
Setup Sheet	78
Sheets	
Define Sheet to Nest	22
Set Nesting Parameters	61
Shop Documentation	78
Simulating the Nest	71
Specify Material	30
Stock	26
Align with Part	28
Create Box Stock	26
Specify Material	30

- U -

Useful Tips	9
Using this Guide and Associated Part Files	10