

Questions & Answer Guide



RhinoCAM 2025 MILL

Published: 2/14/2025

MecSoft Corporation
© Copyright 1998-2025

Table of Contents

Quick Start	4
Resource Guide	7
Add a Tool Change Point	8
Add more Materials	12
Add Tool Comments	19
Align the Stock & Part	23
Assign a Stock Material	28
Control the Cut Side & Start Point	31
Copy/Edit a Toolpath	40
Create a Setup Sheet	48
Create a Tool	51
Create a Work Zero	58
Define a Box Stock	61
Define a Machine Tool	65
Define the Post-Processor	69
Define Toolpath Properties	74
Edit Toolpaths Associatively	77
Enable Cutter Compensation	84
Enable Inverse Time Feedrate in 4 Axis	87
Estimate Machining Time	89
Find Tool Related Preferences	91

Generate a Toolpath	95
Load a Tool Library	110
Load a Tool Library Automatically	112
Load the Default Tool Library	114
Orient a Part for Machining	116
Post G-Code	120
Print a Tool List	127
Save a Knowledge Base	128
Save a Tool Library	133
Save Defaults	134
Set Preferences at Start Up	138
Setup a Part	140
Simulate a Toolpath	142
Suppress a Toolpath	145
Use these help topics	147
Why are my Feed Rate values too High/Low?	148
Index	150

Quick Start



MILL Module 2025

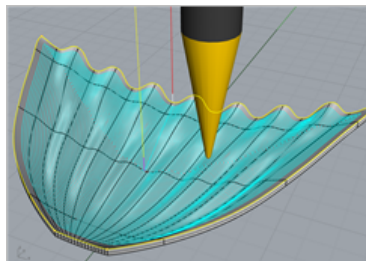
[Prefer Printed Documentation? Check Here!](#)

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

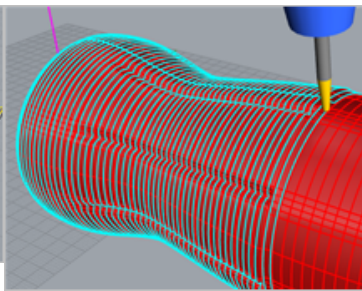
[Quick Start Guides](#) for each [RhinoCAM](#) module are available in both PDF and Video format. Refer to the following information to access these resources:



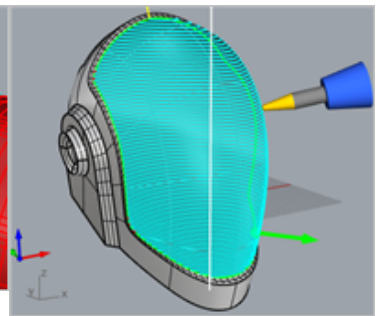
RhinoCAM Supports Sub-D Models in Rhino



3 Axis Machining with Rhino 7 Sub-D



4 Axis Machining with Rhino 7 Sub-D



5 Axis Machining with Rhino 7 Sub-D



What's New!

[What's New in RhinoCAM 2025](#)



The Complete Quick Start Video Play List

[Here is a link to the complete 2025 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 [RhinoCAM Learning Resources](#) dialog.

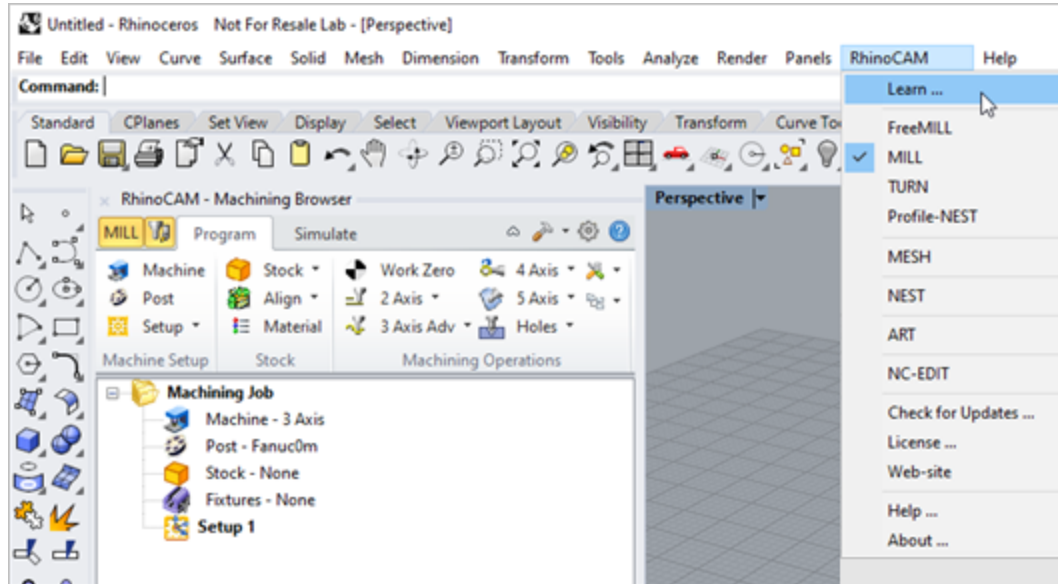
You will find:

- 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 [Rhino Main Menu](#), drop down the Main menu and select [Learn ...](#)

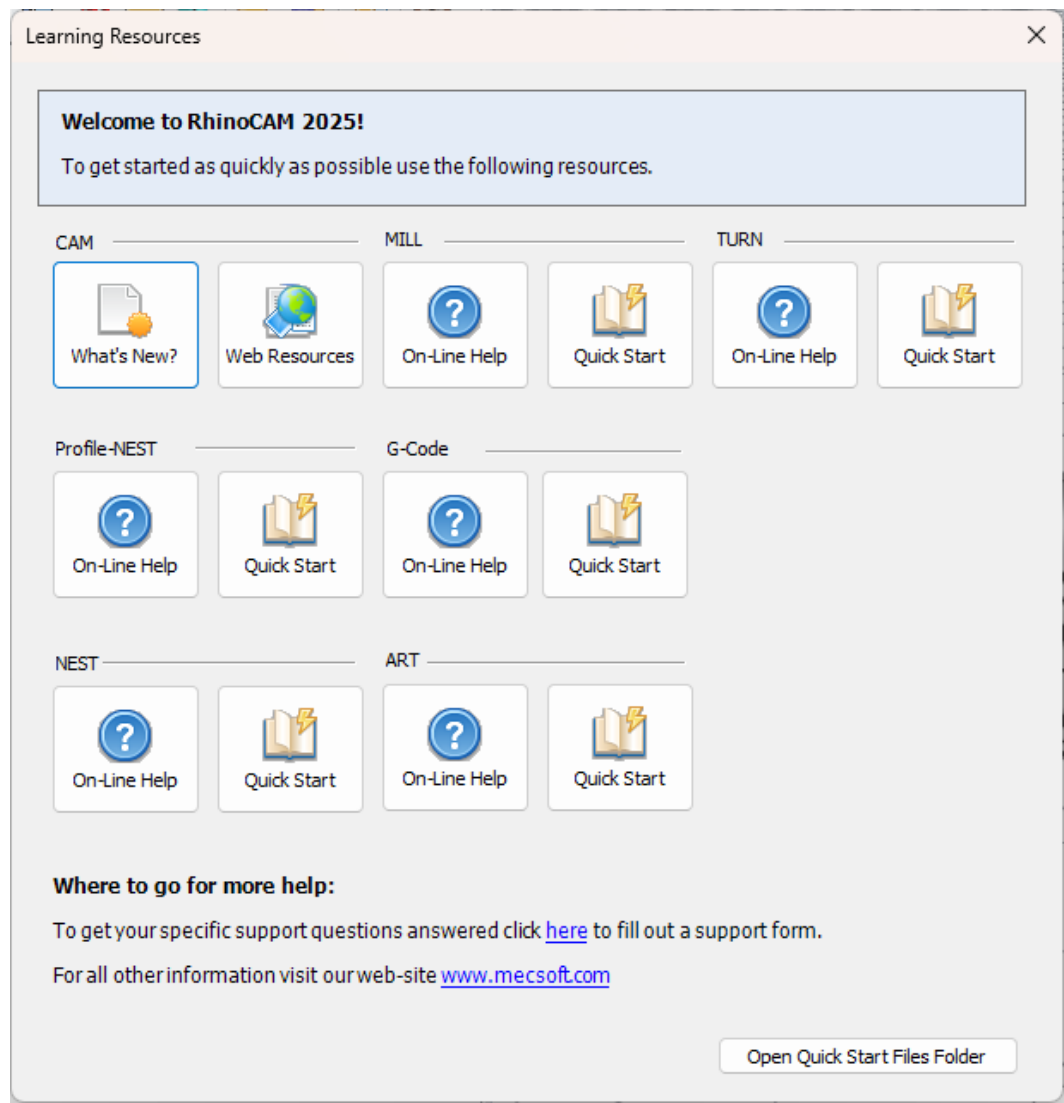


To access the Learning Resources dialog in RhinoCAM

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 [RhinoCAM Resources](#).



2025 RhinoCAM Resource Guide



The 2025 RhinoCAM Resource Guide!

18 Pages

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

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

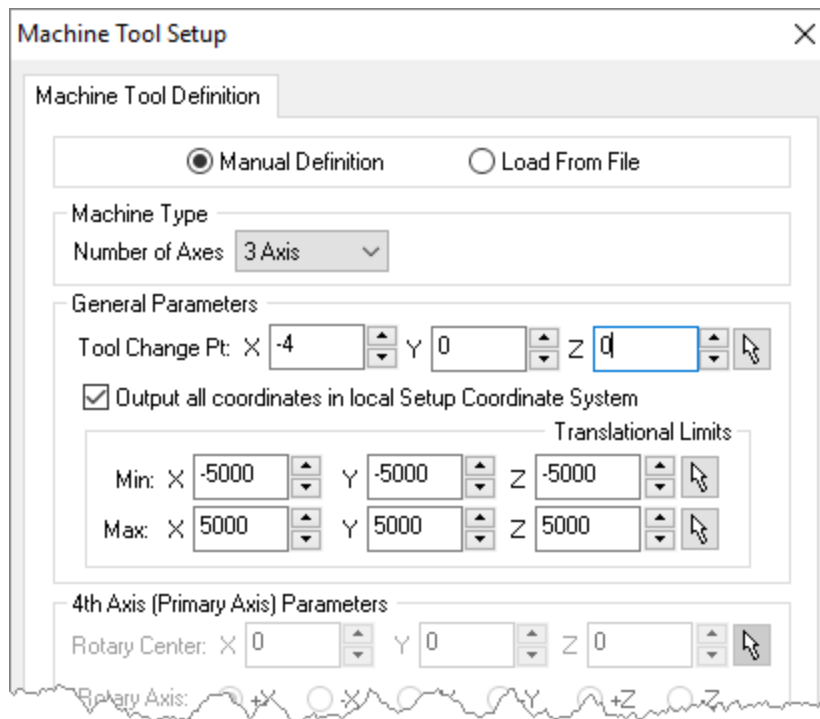
[What's New](#) | [Quick Start Play List](#)

Add a Tool Change Point

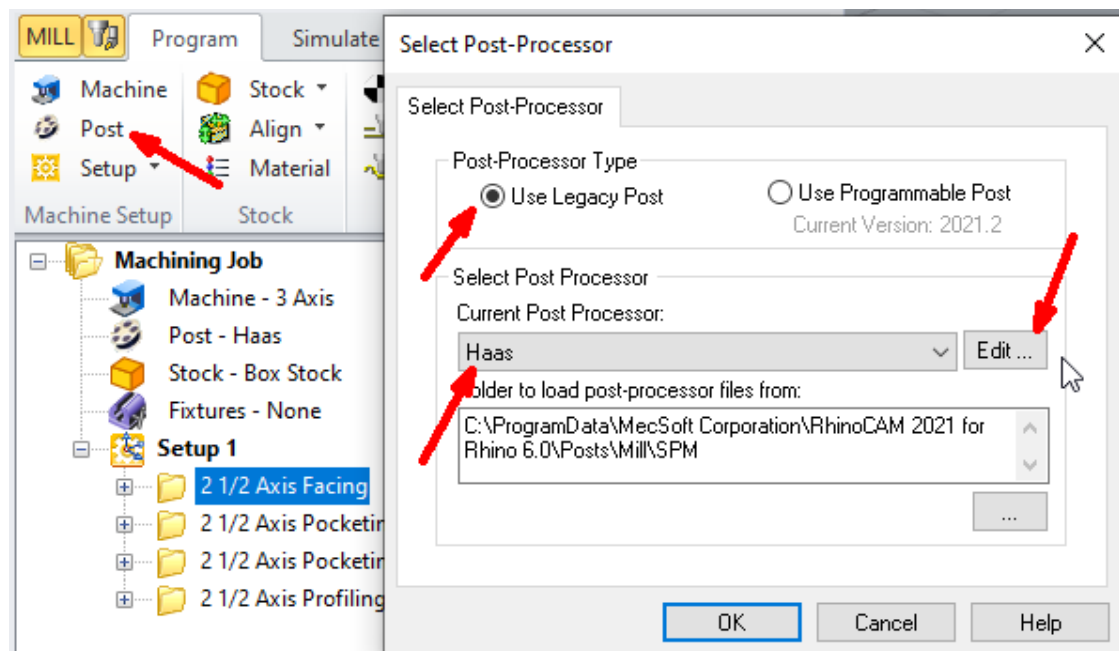
To output a [Tool Change Point](#) to your posted g-code files, please do the following:

1. From the [Machine Setup](#) dialog ([Program](#) tab > [Machine](#) > [General Parameters](#) > [Tool Change Pt](#)), enter your required tool change point coordinates.
2. For the sample code (shown at the end of this section) we entered the following values in the [Machine Setup](#) dialog:

X: -4, Y: 0 Z: 0



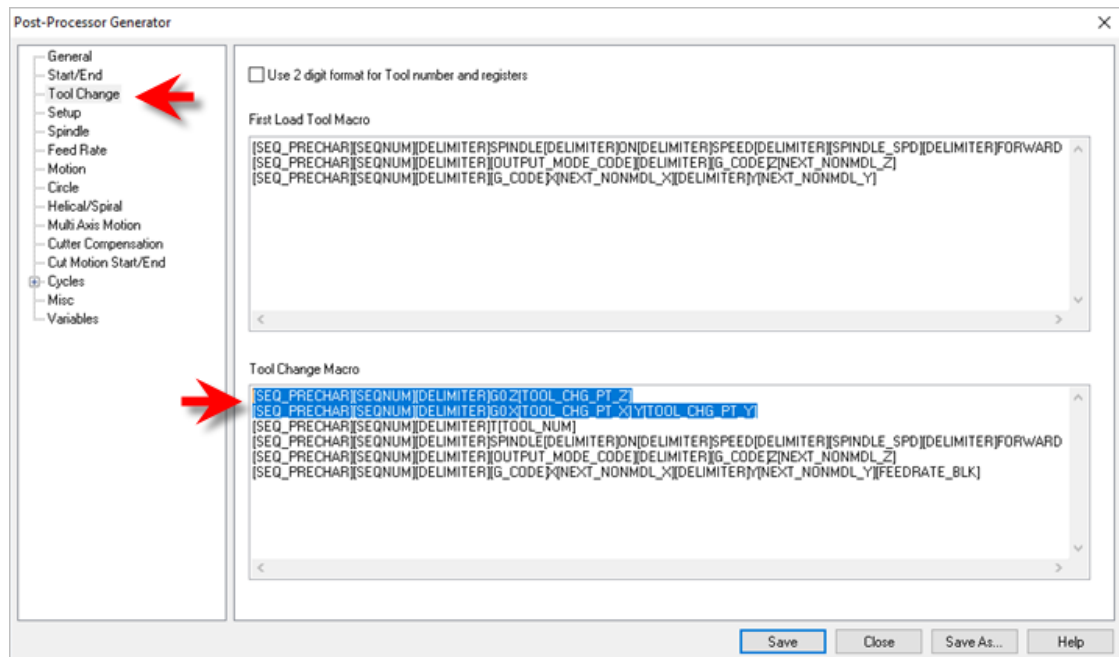
3. Edit your post processor by selecting [Program](#) tab > [Post](#) > [Edit](#).



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

4. From the **Post Process Generator** dialog, select the **Tool Change** section from the left side of the dialog.
5. In the **Tool Change Macro** block section, replace the first line of text with the following two lines of text at the top of the macro.
These two lines of text should precede the line that includes **T[TOOL_NUM]** as shown in the examples below.

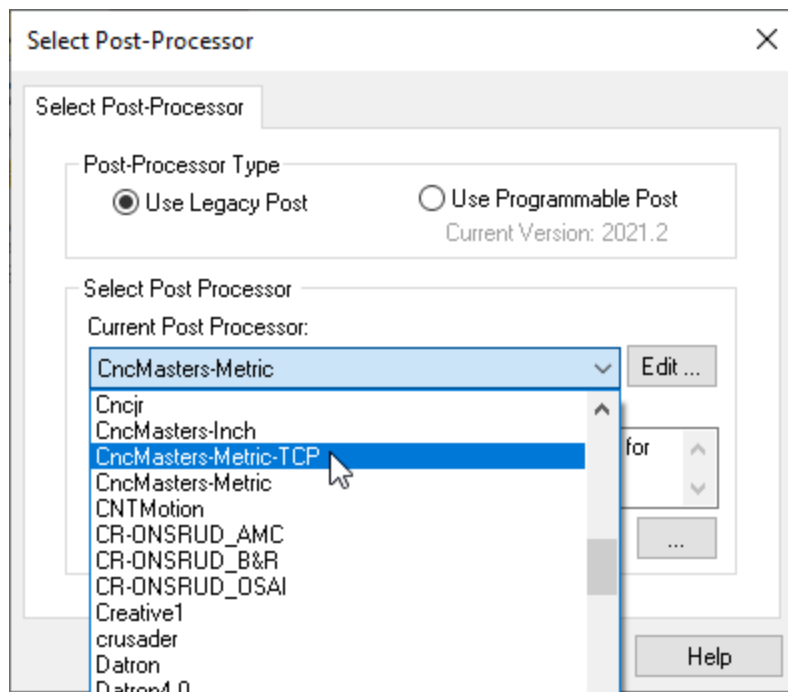
```
[SEQ_PRECHAR][SEQNUM][DELIMITER]GO Z[TOOL_CHG_PT_Z]
[SEQ_PRECHAR][SEQNUM][DELIMITER]GO X[TOOL_CHG_PT_X] Y[TOOL_CHG_PT_Y]
[SEQ_PRECHAR][SEQNUM][DELIMITER]T[TOOL_NUM]
...
...
```



6. If your controller expects to see an optional stop call BEFORE each tool change, you can add another line like below:

```
[SEQ_PRECHAR][SEQNUM][DELIMITER]G0 Z[TOOL_CHG_PT_Z]
[SEQ_PRECHAR][SEQNUM][DELIMITER]G0 X[TOOL_CHG_PT_X] Y[TOOL_CHG_PT_Y]
[SEQ_PRECHAR][SEQNUM][DELIMITER]M01
[SEQ_PRECHAR][SEQNUM][DELIMITER]T[TOOL_NUM]
...
...
```

7. From the [Post Process Generator](#) dialog, pick [Save As](#).
8. Enter a unique name for your post file ([*.spm](#)) for testing and pick [Save](#).
9. From the [Set Post-Processor Options](#) dialog, select the revised post from the [Current Post Processor](#) list.



10. **Note:** If you do not see your revised post in the list, select the "..." button to the right of the "Folder where post-processor file are located" and select the folder where you saved your revised post file to (see [Step 7](#) above) and pick **OK**.
11. You should now see your revised post in the list. Select it and pick **OK**.
12. Post a sample toolpath using the revised post.
13. Review the g-code test file and locate the first tool change lines of code.
14. Your sample test should look something like this depending on your post (based on the tool change point we used in [Step 2](#) above). Note the tool change coordinates in blue:

```
...
...
N66 ;2 1/2 Axis Profiling
N68 G0 Z0.
N70 G0 X-4. Y0.
N72 T1 M06
...
...
```

15. That's it!

Add more Materials

This topic is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. [RhinoCAM](#) has a built-in [Feeds & Speeds Calculator](#) that can suggest [Spindle Speeds](#) and [Cut Feed Rates](#) based on your stock material and active tool parameters!

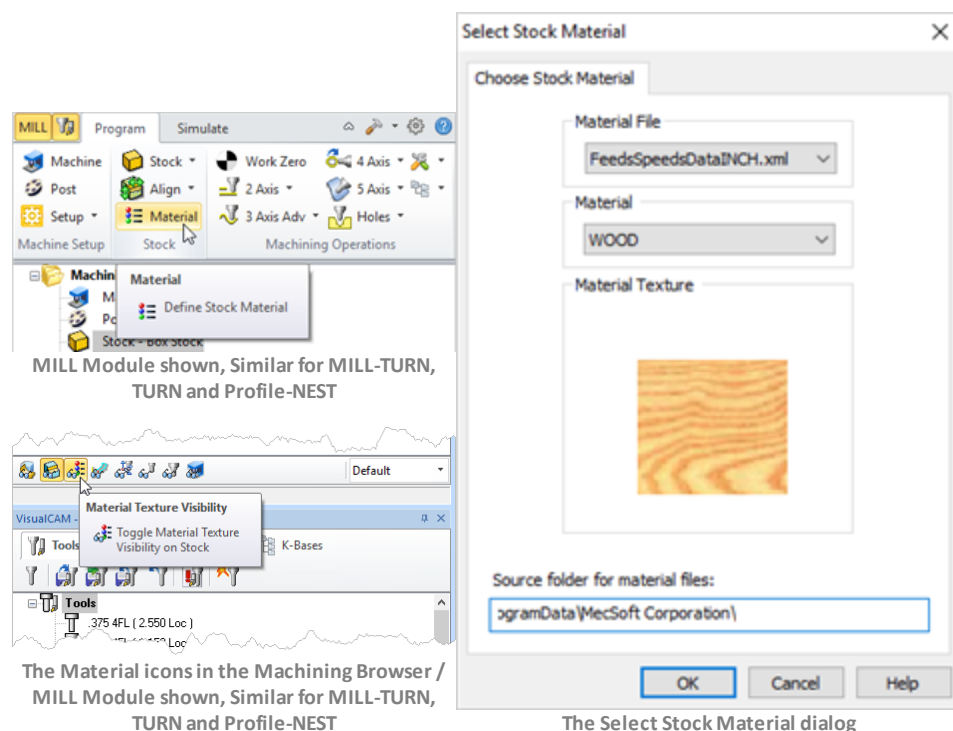
However, what if you are cutting stock material that is currently not in our [Materials Library](#)? Or what if you don't like what is currently assigned for the material of your choice in the [Materials Library](#)?

This topic will show you how to customize [RhinoCAM](#) to add and manage multiple material files as well to add your own stock materials. If you are new to [RhinoCAM](#), you can review the topic [The Feeds & Speeds Calculator](#).



How Materials are used

First let's review how material definitions are used in [RhinoCAM](#). From the [Stock](#) pane of the [Program](#) tab in the [Machining Browser](#) you will find the [Material](#) icon. Selecting it will display the Select [Stock Material](#) dialog. Here you can select a stock material from the [Material File](#). If there is an associated Material [Texture](#) file, then this texture is assigned to the stock model and rendered in the graphics window. This texture rendering on the stock can be toggled on or off using the [Material Texture Visibility](#) icon, located at the base of the [Machining Browser](#).



Also, from the [Feeds & Speeds](#) tab on the [Create/Select Tool](#) dialog you can select [Load from File](#) to display the [Feeds & Speeds Calculator](#) dialog. The [Material](#) selection in this dialog will default to the stock [Material](#) you have previously selected from the [Select Stock Material](#) dialog (shown above).



Location of Material Files

The [Material](#) selections available to you from these dialogs are retrieved from a [Feeds & Speeds XML](#) file installed with the program. There are separate files for [INCH](#) and [METRIC](#) units. The files are located in the [\Materials](#) folder of the [ProgramData](#) install path of the program.

Here is an example: [C:\ProgramData\MecSoft Corporation\Program Name & Version\Materials](#).



By default, the [Windows Operation System](#) does not display the [ProgramData](#) folder to you. If you do not see it, log in as [Administrator](#) and from the [Windows Control Panel](#), navigate to the [File Explorer Options](#) dialog, select the [View](#) tab and then select the option to [Show hidden files, folders, and drives](#) and pick [OK](#).



Maintaining Multiple Material Files

You can create your own [Material Files](#) and store them in the default folder. The CAM system on startup will load all material files in the folder specified in the dialog above as long as the units of the material files match the part units that you are working with. If the system finds multiple material files with matching units, it will populate the drop-down control in the [Materials](#) dialog allowing you to select a specific material file to suit your needs.



You need to make sure that you have the units specified correctly in your customized materials [XML](#) files or your material files will not appear in the dialog list. See item [#5](#) in the [Editing the Feeds & Speeds XML File](#) section below for the correct [Units](#) format.

Having multiple material files can help manage different materials and associated feeds & speeds data without having to cram all of the data into one file. In addition, multiple material files may be useful if you have multiple machines in the shop with different power capabilities and you want to use different feeds and speeds settings per machine. Under this scenario you could have one material file per machine saved in the default folder and load the correct material file depending on which machine you are programming toolpaths for.



Editing the Feeds & Speeds XML File

Material files can be edited to customize the data stored in these files to suit your shop needs.

! CAUTION!! - Before editing material files - Make sure you create a copy and keep them in a safe place, outside of the install path of the program. Also, make sure you maintain the [XML](#) format of these files or the [Feeds & Speeds Calculator](#) may not work properly! The file names are:

[FeedsSpeedsDataINCH.xml](#)
[FeedsSpeedsDataINCH.xml](#)

Here are the steps required to add new material definitions. Be sure to read these steps carefully:

1. Make a backup copy of the [Feeds & Speeds XML](#) files!
2. Edit the [XML](#) file to add the material definitions needed, paying close attention to the format of the [XML](#) file. This file can be edited with any [ASCII](#) text editor such as [Notepad](#) or any [XML](#) editor.
3. Each [Material](#) defined in the [XML](#) file has several records (or lines of text) associated with it. Each line of text defines information about that [Material](#) for each instance it is referred to by the [Feeds & Speeds Calculator](#). Here is a sample section for [Aluminum 2025](#).

```
<Version>1.0</Version>
<Units>Imperial</Units>
<FeedsSpeeds>
  <Material>
    <Name>ALUMINUM - 2025</Name>
    <TextureFile>ALUMINUM.bmp</TextureFile>
    <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00, 0.0200</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00, 0.0200</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
  </Material>
  ...
  ...
```

Here is what the file looks like in tabulated form:

Stock Material	Toolpath Type	Tool Material	Surface Speed	Feed/Tooth
ALUMINUM - 2024	MILLING	CARBIDE	1600.00	0.0040
	MILLING	HSS	400.00	0.0040
	MILLING	CERAMIC	400.00	0.0040
	DRILLING	CARBIDE	960.00	0.0048
	DRILLING	HSS	240.00	0.0048
	DRILLING	CERAMIC	240.00	0.0048
	TURNING	CARBIDE	1800.00	0.0200
	TURNING	CERAMIC	1800.00	0.0200
	TURNING	CERMET	1800.00	0.0200
ALUMINUM - 5050	MILLING	CARBIDE	1600.00	0.0040
	MILLING	HSS	400.00	0.0040

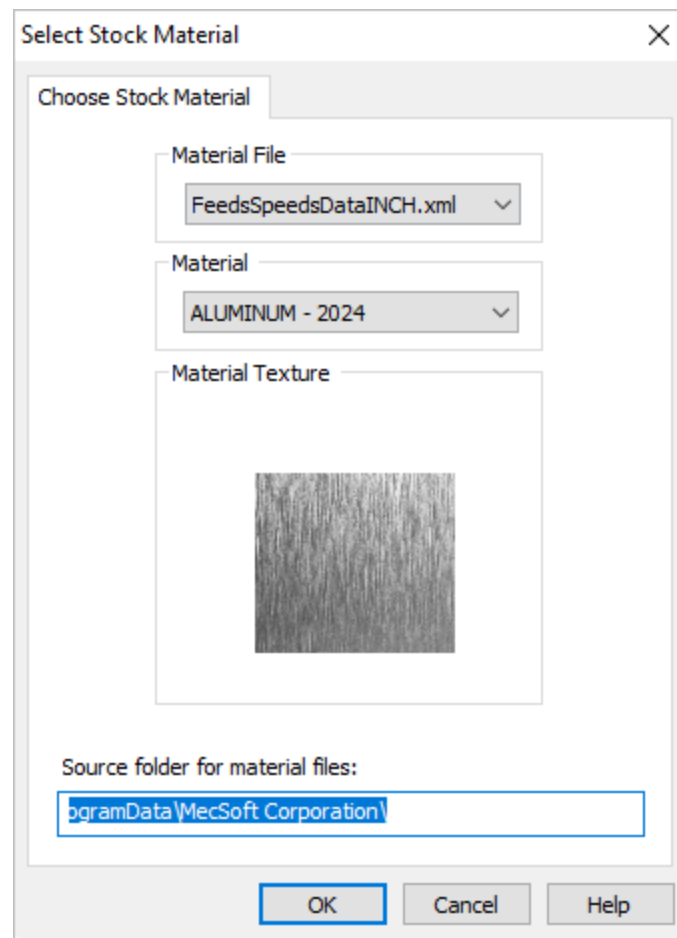
4. You will need to know the [Surface Speed](#) and [Feed/Tooth](#) specifications for each instance of the [Material](#) you are adding to the [XML](#) file. Here is the information format:

```
<FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
```

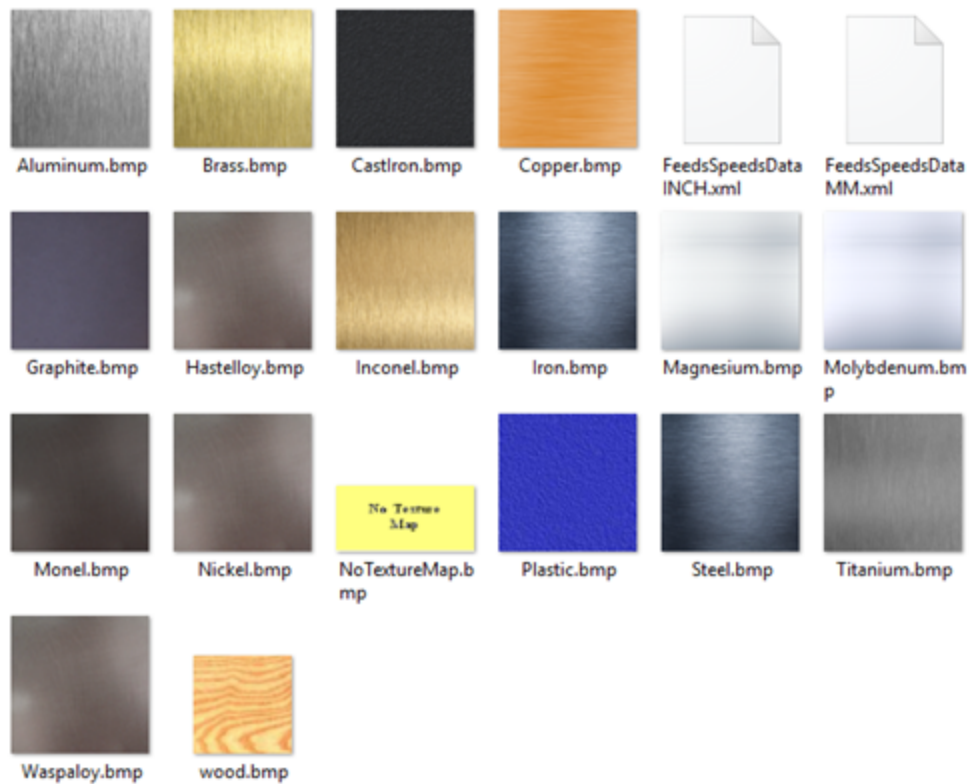
5. You need to make sure that you have the [Units](#) format specified correctly in your custom material [XML](#) files. It is defined on the 2nd line of the [Materials XML](#) file. For [Inch](#) units use `<Units>Imperial</Units>` and for [Metric](#) units use `<Units>Metric</Units>` as shown here for Inch:

```
<Version>1.0</Version>
<Units>Imperial</Units>
<FeedsSpeeds>
<Material>
...
...
```

6. There is also a bitmap image file (**.bmp*) associated with each [Material](#) type defined in the [XML](#) file. This image defines the texture that is applied to the [Stock](#) material when the [Toggle Material Texture Visibility](#) icon is enabled. The image is also displayed in the [Select Stock Material](#) dialog.



Here is the list of the material texture map files currently installed. Make sure your bitmap image file name matches the name entered into the [Materials XML](#) file or the [No Texture Map](#) icon will display instead.



Let's Review

Adding specific materials you use regularly in your shop to the [Feeds & Speeds Calculator](#) in [RhinoCAM](#) can save toolpath programming time and help ensure cut surface quality and machining time accuracy. Just be sure to review this post carefully. A summary is also listed below. If you need help, you can always contact us at MecSoft Support via the phone, web or email.

1. You can select a [Material](#) to be applied to your [Stock](#) definition.
2. The selection of materials are retrieved from an external data file (example: [FeedsSpeedsDataINCH.xml](#)) located in the install path of [RhinoCAM](#) on your computer.
3. The [XML](#) data file can be edited with any [ASCII](#) text or [XML](#) file editor.
4. The [XML](#) data file also contains information about the [CAM Module](#), [Tool Material](#), [Surface Speed](#) and [Feed/Tooth](#) for each instance of the material.
5. The stock material selected is used by the [Feeds & Speeds Calculator](#) to suggest [Surface Speeds](#) and [Cut Feed Rates](#).
6. The [Feeds & Speeds Calculator](#) is displayed when you select [Load from File](#) from the [Feeds & Speeds](#) tabs of the [Create/Select Tool](#) dialog or from each toolpath operation dialog.

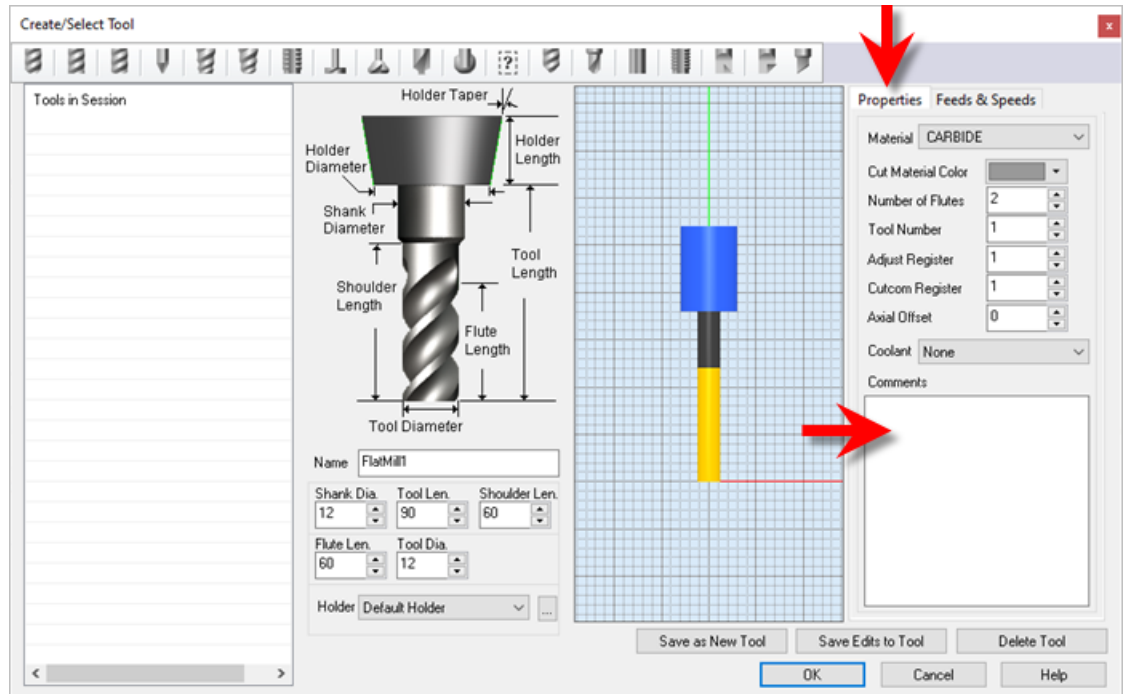
7. You can manually add more materials to the [XML](#) data file. You will need to know the recommended [Surface Speed](#) and [Feed/Tooth](#) specifications for the material you are adding based on each Tool [Material](#) type.
8. It is very **IMPORTANT** that you make a backup copy of these [XML](#) files if you plan to edit them. Also, the text format of the [XML](#) files is **VERY IMPORTANT**. If the format is incorrect, the [Feeds & Speeds Calculator](#) may not work properly! Make sure that the [Units](#) format on line 2 matches the [Units](#) you plan to use in your part files.
9. Each material also has a bitmap image texture file ([*.bmp](#)) that is located in the same folder. The material texture image is shown in the [Select Stock Material](#) dialog and is also applied as a texture map to your stock when the [Toggle Material Texture Visibility](#) icon is enabled.

Add Tool Comments

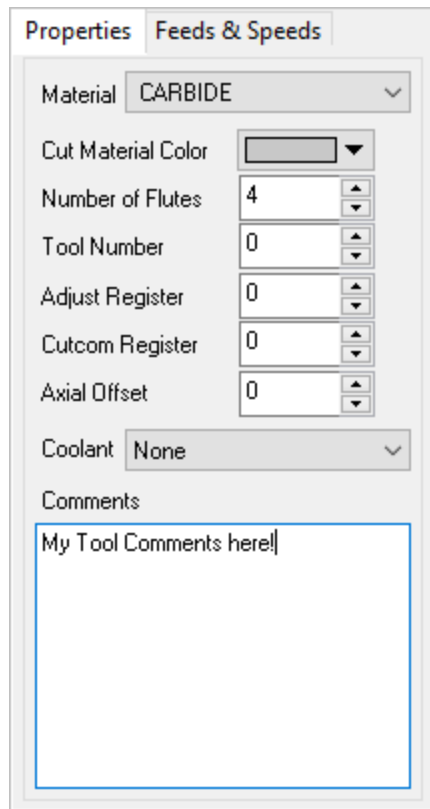
You can add comments associated with a [Tool](#). These [Comments](#) are saved with the [Tool](#) in your [Tool Library](#). They are also posted to your g-code when the tool is used.

Here are the steps to add [Comments](#) to a [Tool](#):

1. Edit the Tool using the [Create/Select Tool](#) dialog.
2. Select the [Properties](#) tab on the right.

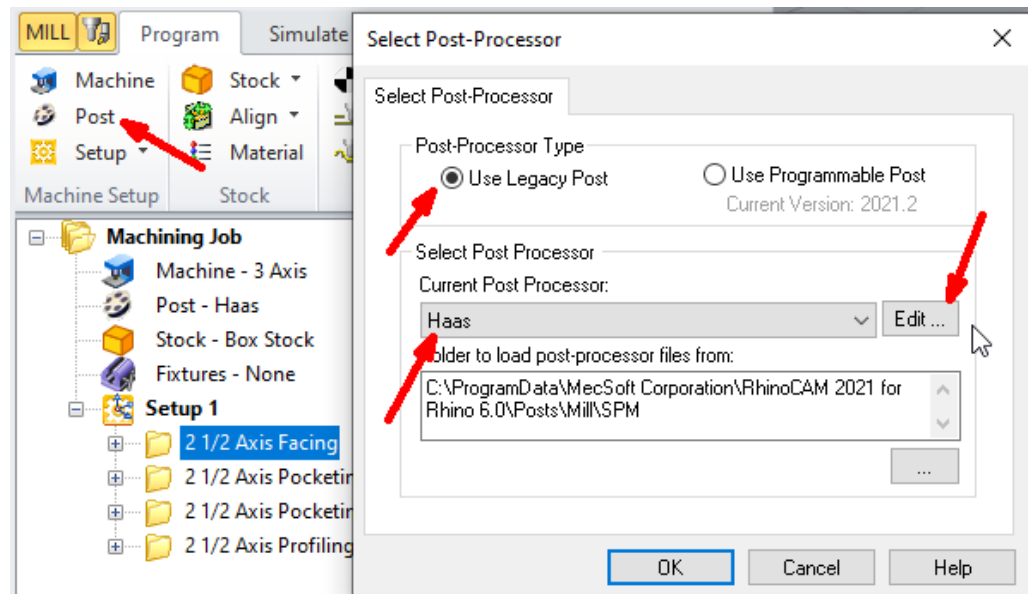


3. Add text to the [Comments](#) window.



4. Make sure [Comments](#) are enabled in your post.

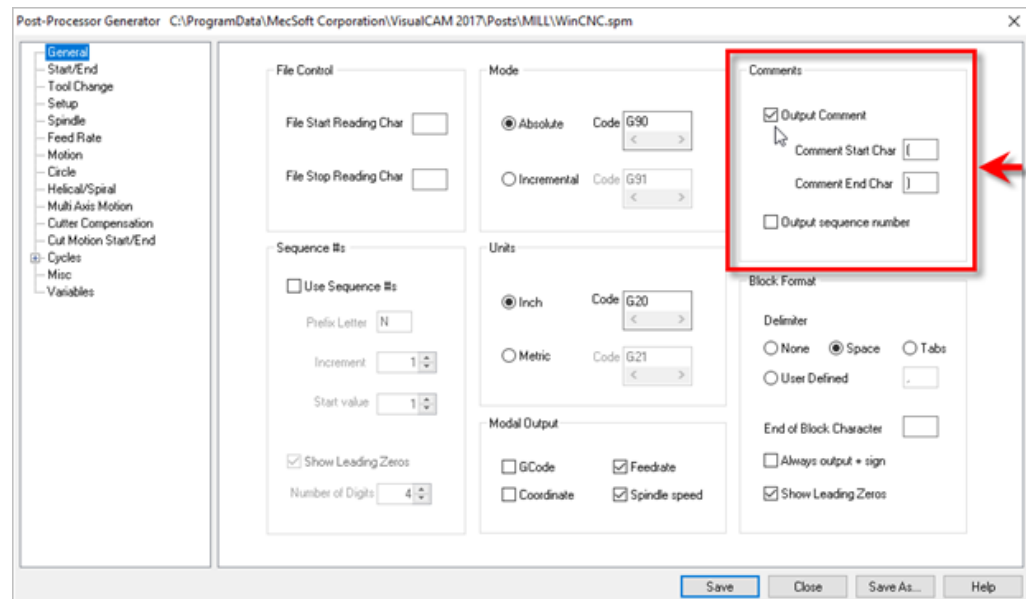
A. Click on [Post](#) ([Set Post-Processor Options](#)), then click [Edit](#).



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

B. From the [Post Processor Generator](#) dialog, select the [General](#) tab from the left.

- C. Check the box to **Output Comments**. You can also change the start and end characters to use.



- D. Then pick **Save** or **Save As**.

5. Now post your operations and see your comments:

```
...
...
G1 X0.5301 Y-0.7171 Z0.7480
G3 X0.7801 Y-0.4671 I0.0000 J0.2500 F2.6
G1 X0.7801 Y-0.2171 Z0.7480 F6.9
G0 Z0.9843
G0 X0.7801 Y-0.2171
(2 1/2 Axis Profiling)
(My Tool Comments Here!)
S18000
G0 Z0.9843
G0 X0.5301 Y-0.7097
G1 X0.5301 Y-0.7097 Z0.7480 F6.9
G1 X0.5873 Y-0.7097 Z0.7480 F3.4
G1 X0.5873 Y-0.6345 Z0.7480
G1 X0.4729 Y-0.6345 Z0.7480
...
...
```

6. If you want to post g-codes instead of comments, just place a \$ character prior to the comment in the **Create/Select Tools** dialog. Adding \$ as prefix will skip the comment start & end characters in the posted code.

Properties Feeds & Speeds

Material CARBIDE

Cut Material Color

Number of Flutes 4

Tool Number 0

Adjust Register 0

Cutcom Register 0

Axial Offset 0

Coolant None

Comments

\$My g-code Here!

...

...

G1 X0.4655 Y-0.7171 Z0.7480

G1 X0.5301 Y-0.7171 Z0.7480

G3 X0.7801 Y-0.4671 I0.0000 J0.2500 F2.6

G1 X0.7801 Y-0.2171 Z0.7480 F6.9

G0 Z0.9843

G0 X0.7801 Y-0.2171

(2 1/2 Axis Profiling)

My g-code Here!

S18000

G0 Z0.9843

G0 X0.5301 Y-0.7097

G1 X0.5301 Y-0.7097 Z0.7480 F6.9

G1 X0.5873 Y-0.7097 Z0.7480 F3.4

G1 X0.5873 Y-0.6345 Z0.7480

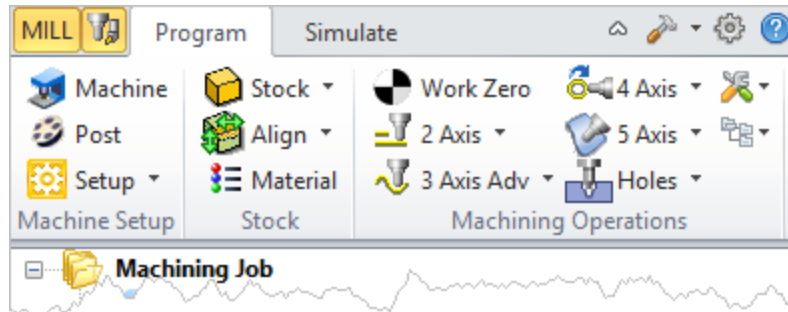
...

...

Align the Stock & Part

If your [Stock](#) is not aligned correction with the [Part](#) model you can change it.

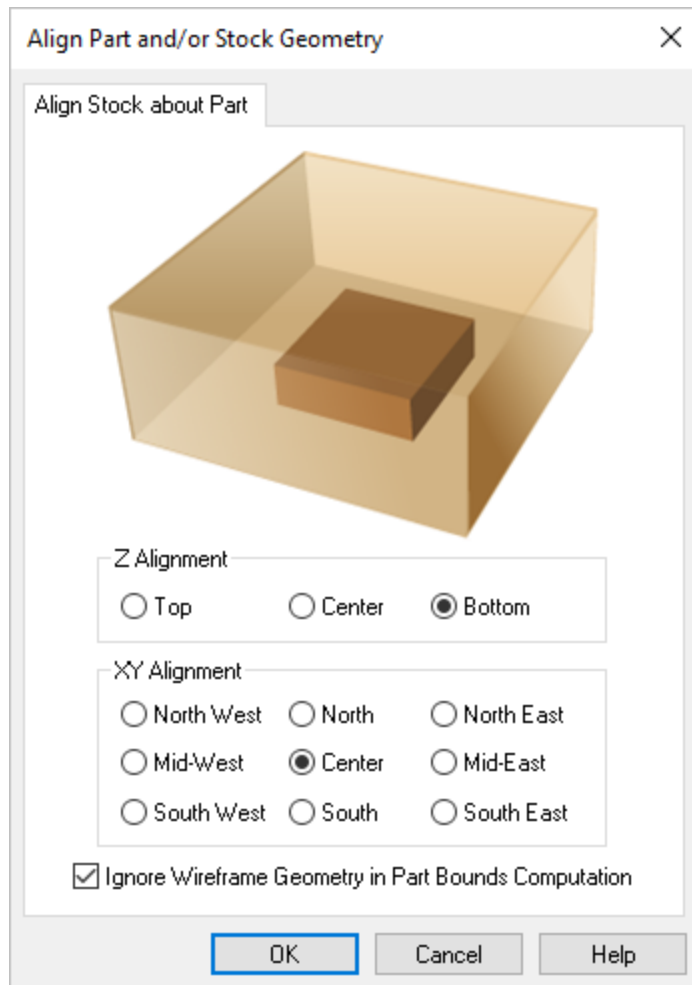
1. Select the [Program](#) tab.



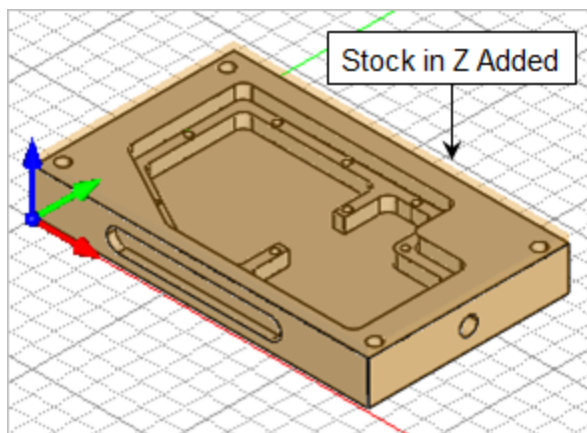
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

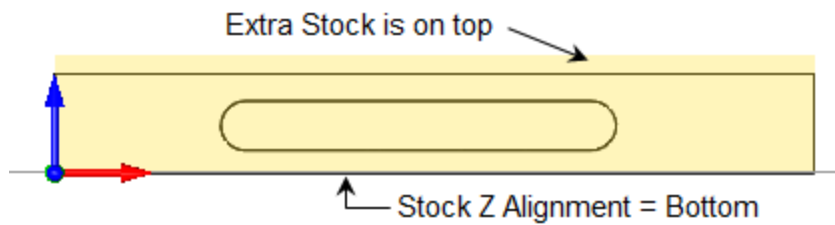
2.  Select the [Align](#) menu.

3.  Select [Align Stock](#) from the menu to display the dialog.



4. First select the **Z Alignment** by select **Top**, **Center** or **Bottom**. For our example we will select **Bottom**.
5. Then select an **XY Alignment** option. Nine different locations are available. For our example we will select **Center**.
6. The **Stock** displayed on the screen will move then you make selections from this dialog.

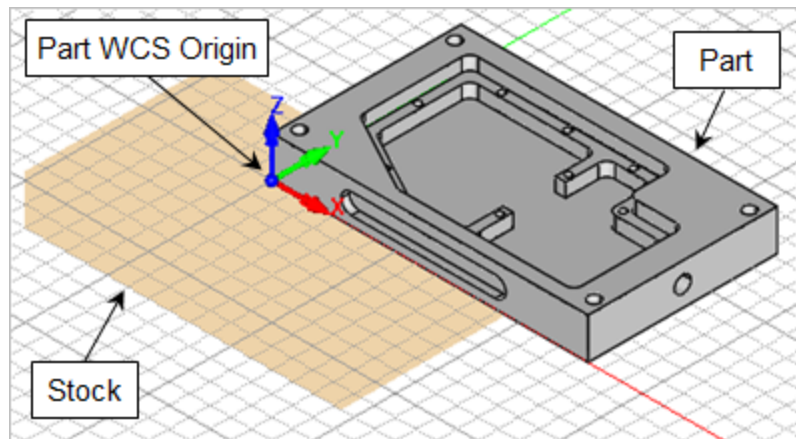




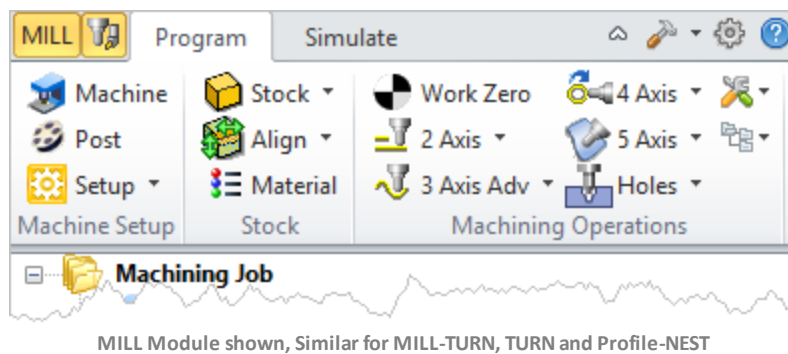
7. Again if you have curves or other non-part wireframe geometry displayed, you can check the box in the dialog to ignore them when calculating the alignment.
8. Pick **OK** to close the dialog.

Alternatively you can move the actual part model in relation to the **Stock** if desired. **IMPORTANT:** This option will physically move the location of your part geometry. This command is useful when you have opened or imported part geometry that is NOT in the correct location for machining.

1. Open the part that you wish to align.
2. Define the **Stock** dimensions and location. See [How to Define a Box Stock](#) for more information. Our example part looks like this:



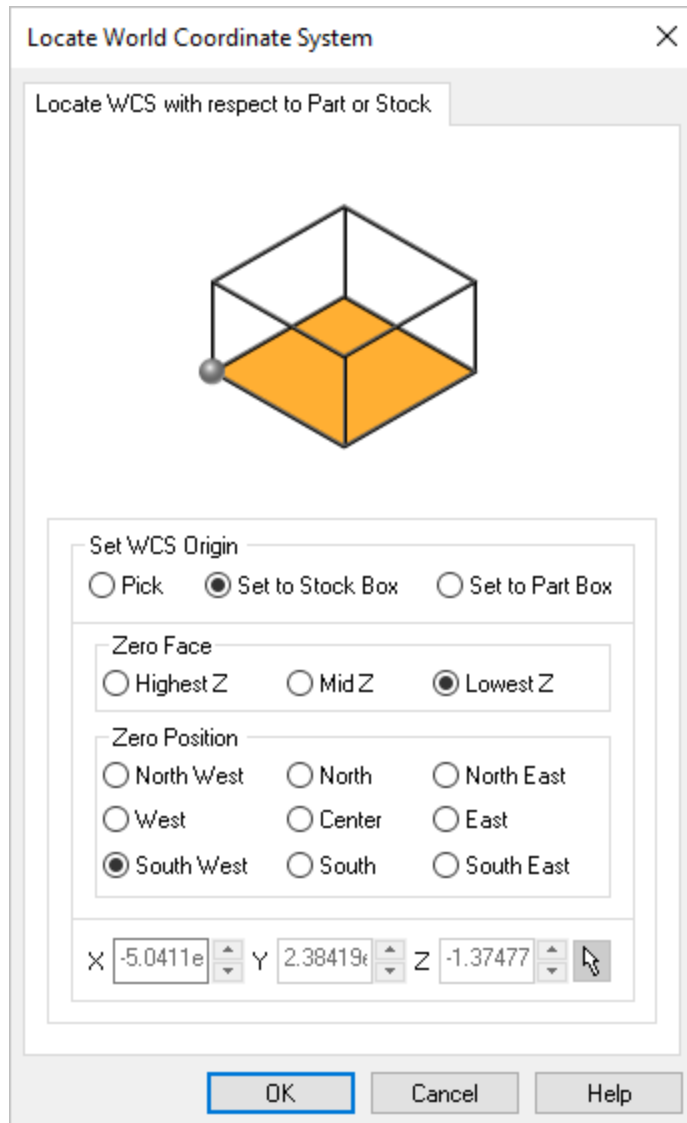
3. Select the **Program** tab.



4.  Select the **Align** menu.



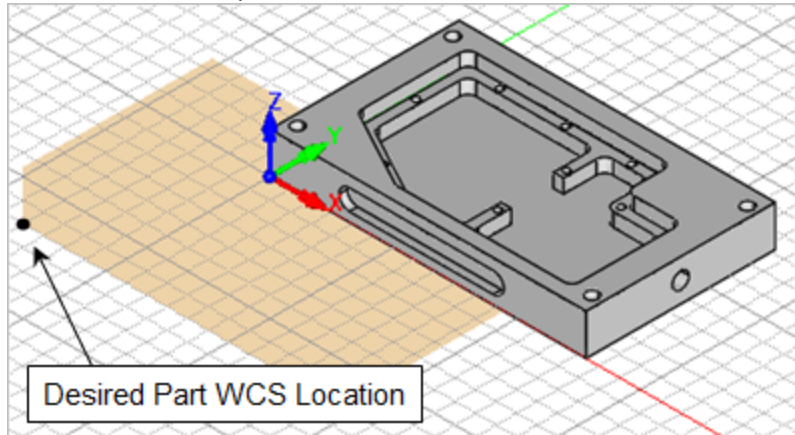
5. Select [Set World C.S.](#) from the menu to display the dialog.



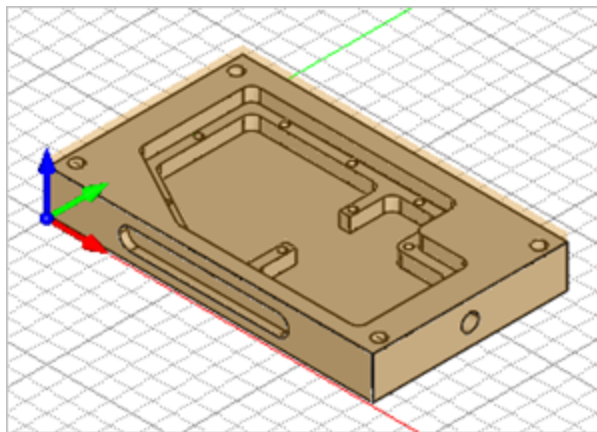
Remember that this dialog will physically MOVE your part geometry!

6. First select an option from the [Set WCS Origin](#) section. This sets the new [WCS Origin](#). In this example we select [Set to Stock Box](#). We want to move the part in relation to the currently defined [Box Stock](#).
7. Then define the [ZERO Face](#). This aligns the part in Z. In this example we select [Lowest Z](#) so the [WCS Origin](#) of the part will move in Z to the bottom of the [Stock Box](#).
8. Then define the [Zero Position](#). There are nine cardinal directions to choose from. Each is in relation to the current [Box Stock](#). In this example we select [South West](#) so the [WCS Origin](#) of the part will move to this position on the [Stock](#).

This is our desired part location:



9. When you pick **OK**, the part **WCS** origin will move the location you specified in the dialog.



Related Online Help Topics

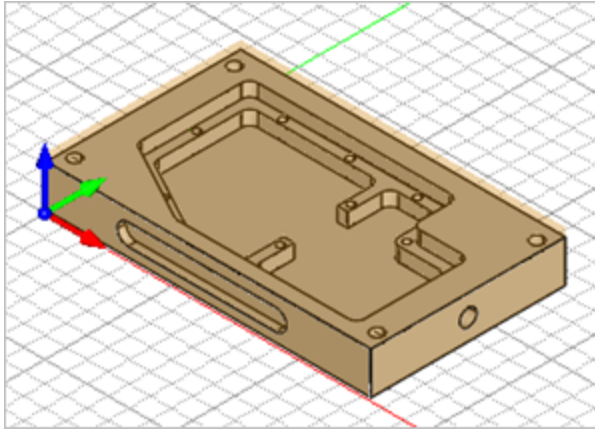
User Interface

Align Stock

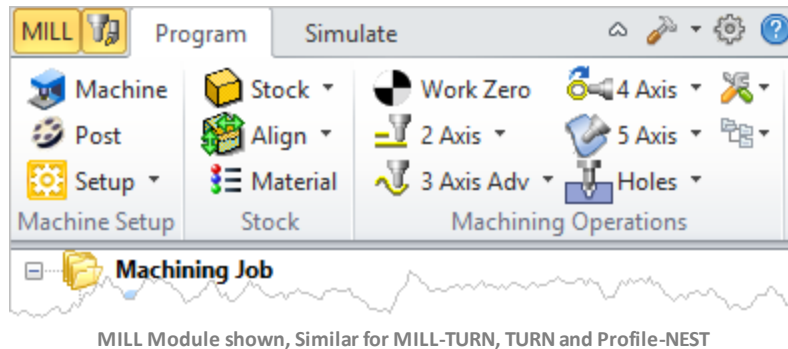
Assign a Stock Material

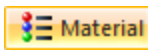
You can optionally assign a [Material](#) definition to your [Stock](#). The [Material](#) definition is used for display purposes. It will also be automatically fed into the [Feeds & Speeds Calculator](#) when it is displayed.

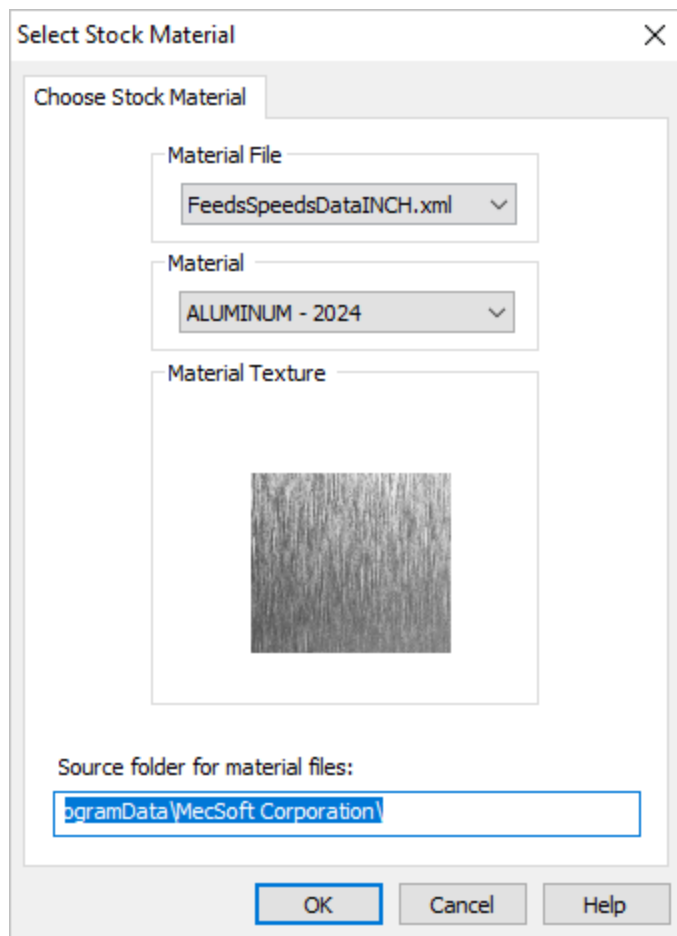
1. Currently we have a part loaded and stock defined and toggled on.



2. Select the [Program](#) tab.

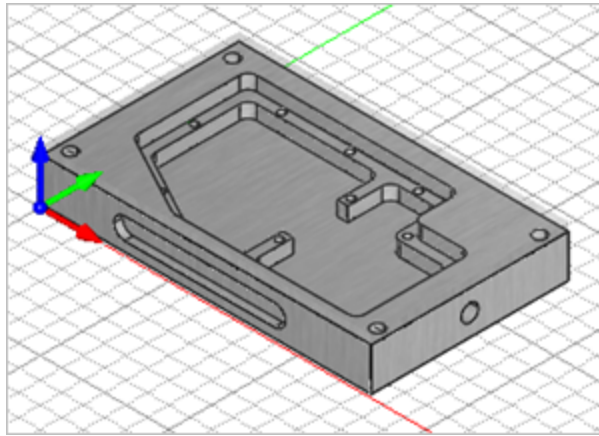


3.  Select [Material](#) to display the dialog.

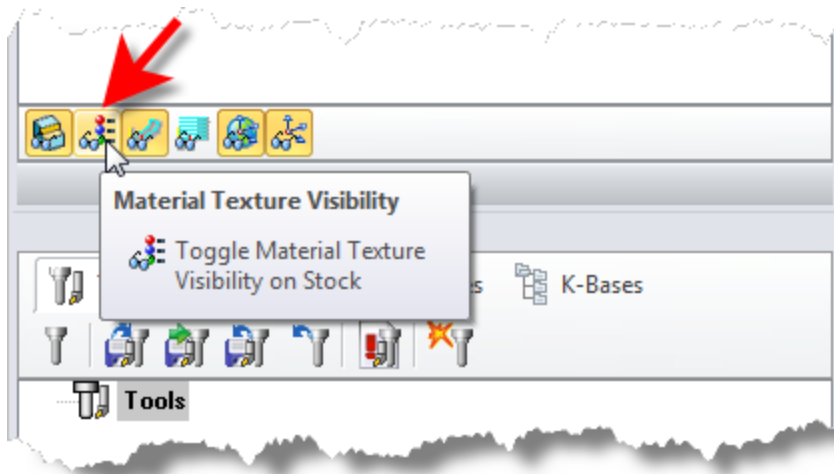


Dialog Box: Select Stock Material

4. Select a [Material](#) from the selection menu in the dialog. A preview of the [Material](#) texture will display in the dialog.
5. [Materials](#) are stored in a [Material](#) file. The default [Material File](#) is pre-selected from the menu.
6. The folder location where the [Material File](#) is located is indicated in the field at the bottom of the dialog. This location cannot be changed.
7. In the future you can add material definitions to the [Material File](#) or create your own material file.
8. Pick [OK](#) to close the dialog.



9.  To see your selected material display on the stock, select the **Toggle Material Texture Visibility** icon located at the bottom of the **Machining Browser** - second icon from the left.



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST



Related Online Help Topics

User Interface

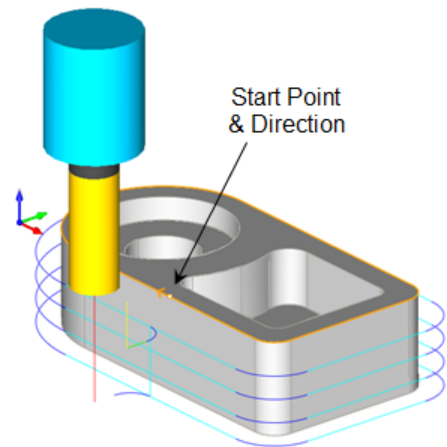
Material

Control the Cut Side & Start Point

Our support staff speaks with users on a daily basis and new users have many questions. One of the questions asked often is:

How can I control the cut side and the cut start point of my 2 Axis toolpaths?

This article addresses this question in detail. To get many more of your questions answered be sure to get the [Question & Answer Guide](#) available for each of MecSoft's CAM desktop plugins. As an [Annual Maintenance Subscription \(AMS\)](#) subscriber, this guide and other training materials are available to you as part of your annual subscription. To learn more about AMS or to become an AMS subscriber just give us a call at (949) 654-8163 option 1 for [Sales](#) or contact sales@mecsoft.com today.



Each curve has a [Start](#) point, [Direction](#) and an [End](#) point. The [Right Hand](#) side of the curve is determined by the curve direction. These definitions are shown in the illustrations below. In [2 Axis](#) machining methods the [Cut Start Point](#) is defined by the [Start Point](#) of the selected curve regions

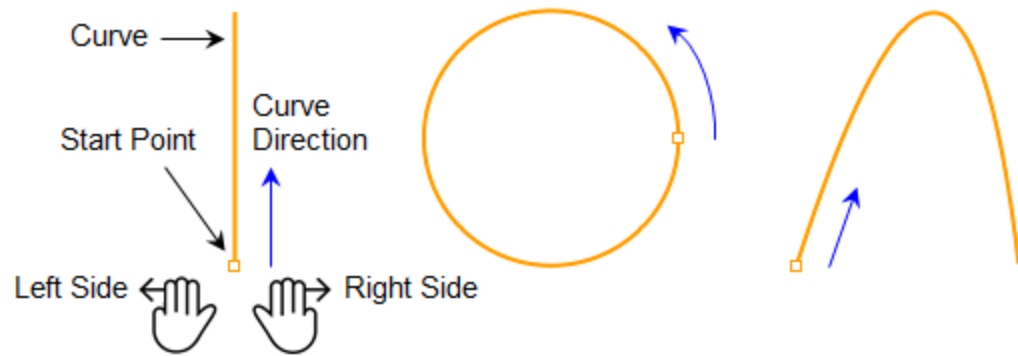
So controlling the curve [Start Point](#) is critical in controlling the [Cut Start Point](#) of the [2 Axis](#) toolpath.



Understanding Curve Geometry

To answer this question, we must first talk about curve geometry (i.e., line, arc or spline). Collectively we will call them curves. Each curve has a [Start](#) point, a [Direction](#) and an [End](#) point. If you position yourself at the start point and face the direction the curve is traveling, then your right and left hand will govern the right and left side of the curve.

These definitions are shown in the illustration below. In [2-1/2 Axis](#) machining methods the [Cut Start Point](#) is defined by the [Start Point](#) of the curve region. Thus, controlling the curve [Start Point](#) is critical in controlling the [Cut Start Point](#) of the toolpath. For closed curves, [Inside](#) and [Outside](#) can be used to control the side to cut. If the curve is a surface edge of a 3D solid model, you can also allow the program to determine the correct side to cut based on the part's topology.



The Start Point, Direction and Cut Side of Curves Geometry

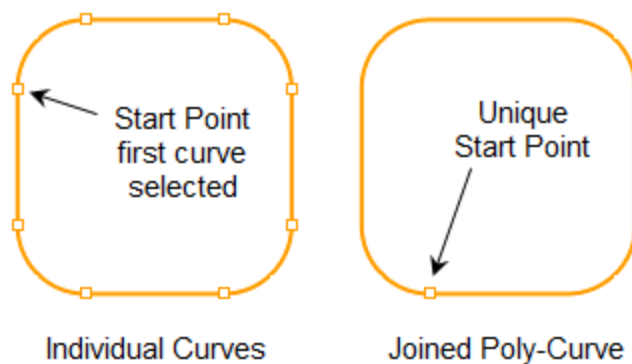


When Machining Multiple Curves

When more than one disconnected curve is selected for one machining operation each is obviously treated as a separate curve region, each with a unique cut start point.

💡 When multiple curves that are located end-to-end are selected for one machining operation it is highly recommended that you first merge or join the curves into one poly-curve. Doing this first will define a single cut start point for the entire poly-curve.

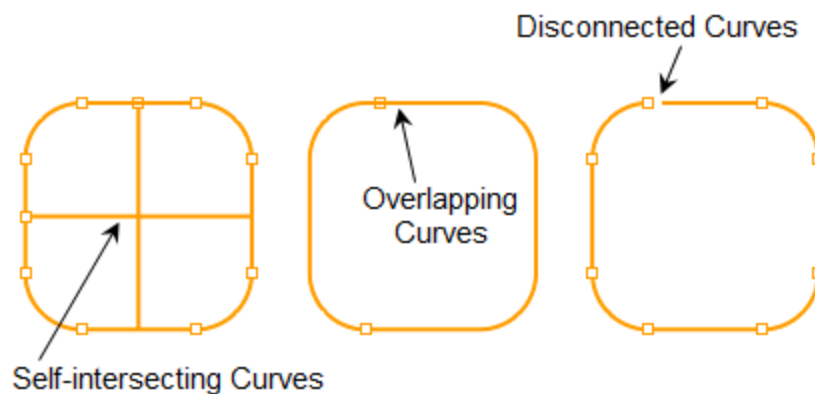
💡 NOTE: If the curves are left disconnected, the start point of the first curve selected for the machining region will serve as the cut start point. You can move a curve to the top of the [Selected Machining Region\(s\)](#) list and its start point will become the cut start point for the operation.



Error - open loops found!

If you get an "open loops found!" error message when generating an operation, this means that the machining regions you have selected are not joined when the operation is expecting them to be. [2-1/2 Axis Pocketing](#) is one operation that expects closed curves. This is why it is always good practice to join closed curves into one poly-curve

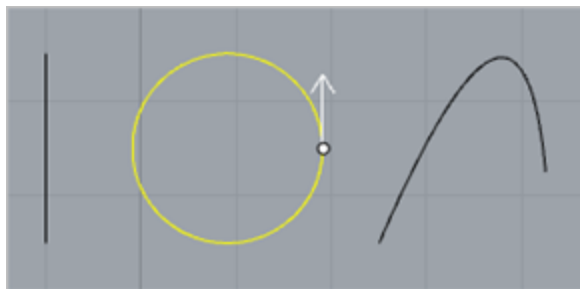
before being used in a machining operation. The following illustrates open loop conditions:



How to Identify Start Points in CAD

You can identify and display curve start point and direction arrow indicators using your CAD system tools and commands.

1. At the command prompt type the command `crvseam` and press `<Enter>`.
2. Select the curve to identify and press `<Enter>` again.
3. Start point and arrow indicators will appear..



How to Change a Curve's Start Point in CAD

You can use your CAD tools to move the **Start Point** of a curve. See the section above on [How to Identify Start Points](#) before continuing.

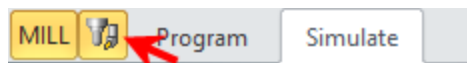
1. At the command prompt type the command `crvseam` and press `<Enter>`.
2. Select the curve to modify and press `<Enter>` again. Then select a point on the curve that you want to move the start point to..



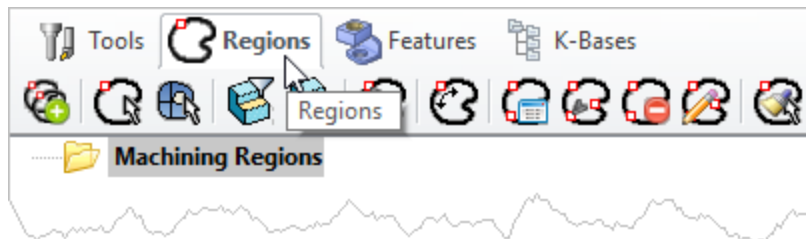
How to Change the Start Point in CAM

In **RhinoCAM** you can use **Pre-Defined Regions** to control the **Cut Start Point**, **Direction** and other aspects of your machining regions. This is a convenient method because it works on both curves and surface edges.

1.  To the left of the **Program** tab, select the **Tools Machining Objects** icon to make sure the **Machining Objects Browser** is displayed.

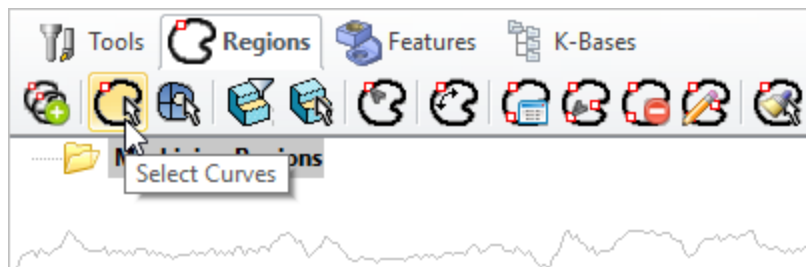


2. Select the **Regions** tab.



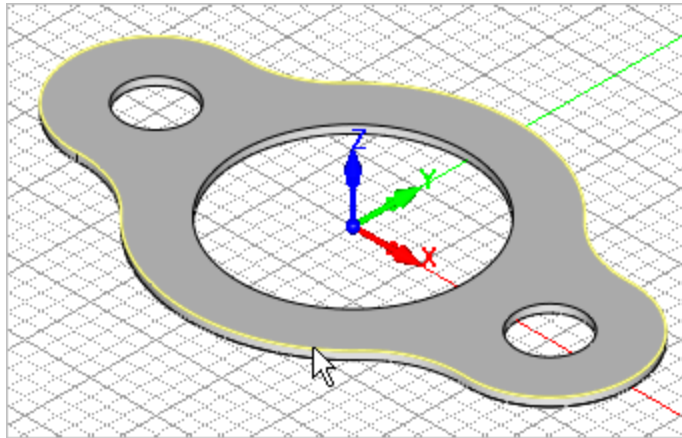
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

3.  Pick the **Select Curves** icon.

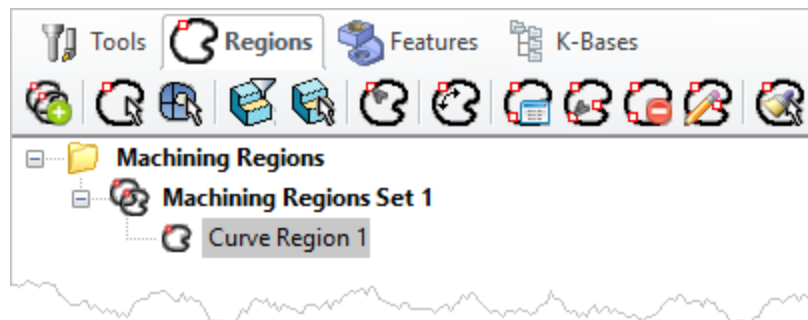


MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

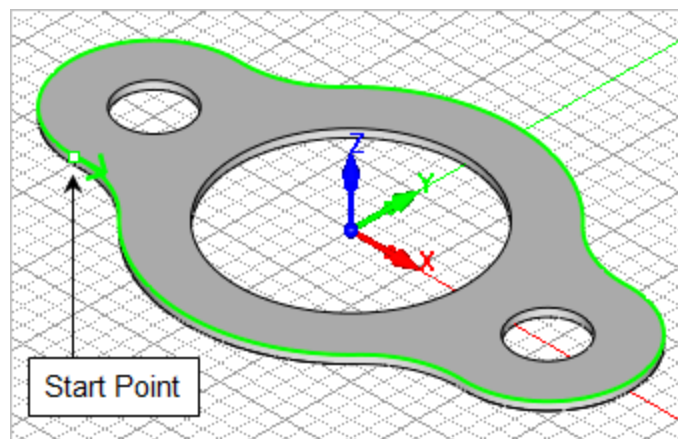
4. Select the curves or surface edges that you want to create a pre-defined region from and then right-click or press **<Enter>**. The example below we select the top outer surface edge.



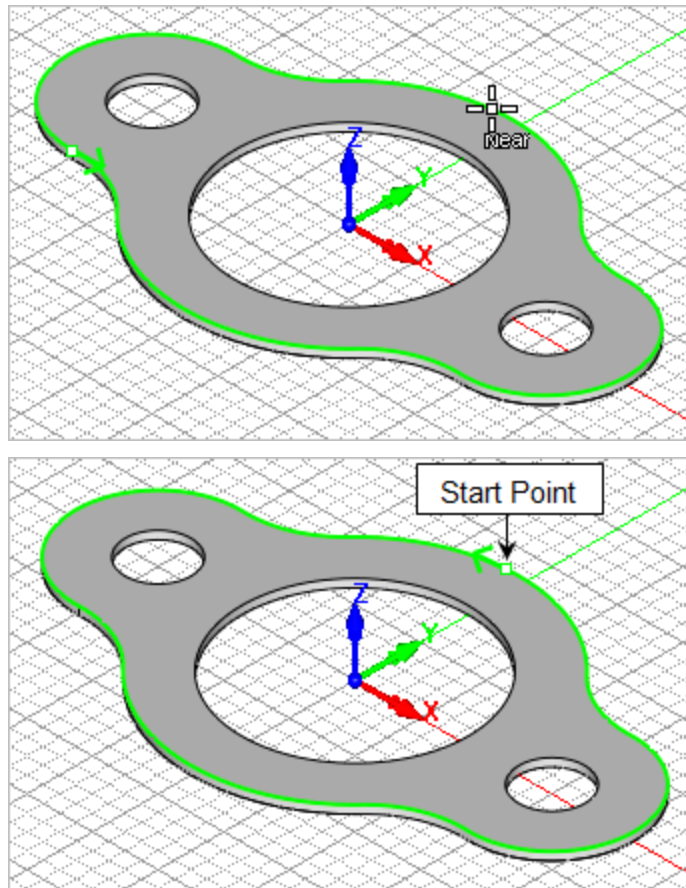
5. A **Machining Region Set** is created and added to the **Machining Regions** list with each curve/edge listed. In this example only one region is created. It is selected by default and the regions is displayed on the part with the start point and direction indicated.



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST



6.  With the curve region selected from the **Machining Regions** list, pick the **Select Start Point** icon to enable start point editing.
7. Now select a new start point anywhere along the pre-defined region. The start point will move to that location.



8.  Notice that the [Select Start Point](#) icon is still enabled. You can select another location for the start point if desired.
9. Take a moment to familiarize yourself with the other cool Region commands on the toolbar and use them to save time and control your toolpaths.

-  [Create Machining Region Set](#)
-  [Select Curve](#)
-  [Select Surface Edge Areas](#)
-  [Flat Areas Selection Filter](#)
-  [Select Flat Areas](#)
-  [Select Start Point](#)
-  [Reverse Cut Direction](#)



Automatic Bridge Points on Selections



Manual Bridge Points on Selections



Delete All Bridge Points in Selections



Edit Bridge Point in Selections



Clone Region

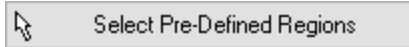


How to Select Pre-Defined Regions for a Toolpath

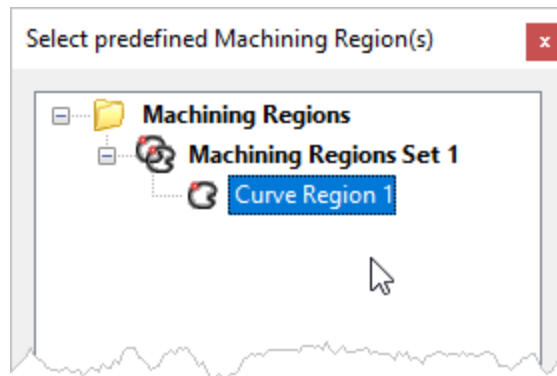
Once you have created a [Pre-Defined Region](#) with its [Start Point](#) defined you can use them as machining regions in any toolpath operation.

1. Create a toolpath operation as you normally would. See [How to Generate a Toolpath](#) for more information.

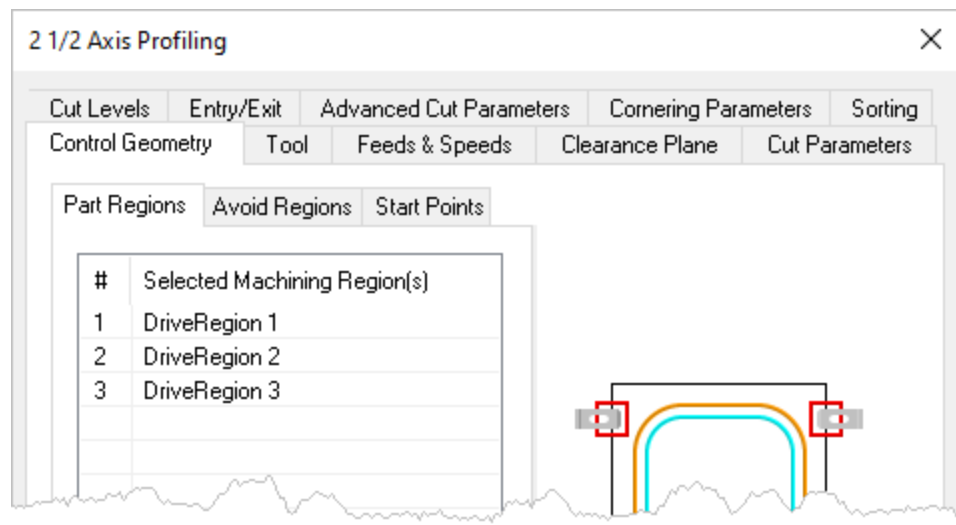
2. Select the [Control Geometry](#) tab.

3.  Pick the [Select Pre-Defined Regions](#) button to display the dialog.

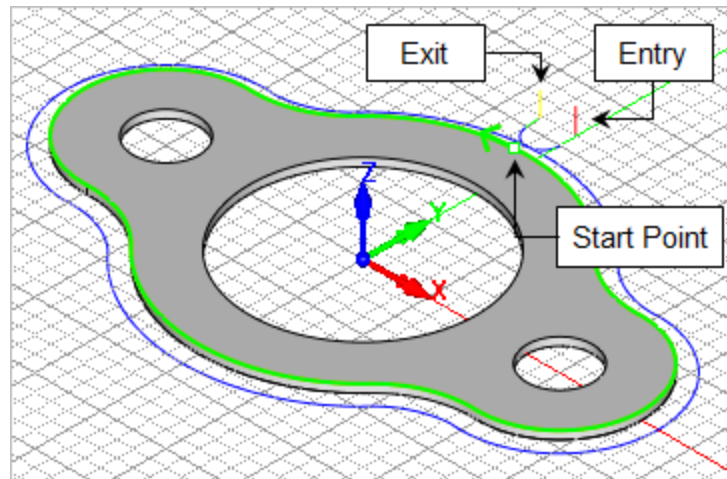
4. Select the region(s) for the toolpath operation and then pick [OK](#) to close the dialog. You can select one or more regions or a region set.



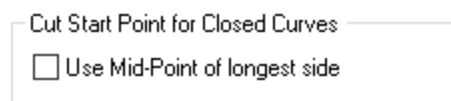
5. The [Pre-Defined Region](#) is added to the list of [Selected Machining Region\(s\)](#). It is listed as a [Curve Region](#) and can mixed with other curve or drive regions in the list.



6. Now **Generate** the toolpath as you normally would and the **Start Point** of the **Pre-Defined Region** controls the **Cut Start Point** of the **2 Axis** toolpath. In this example, it is a **Profiling** toolpath.



7. **NOTE:** If you are generating a **2 Axis Profiling** toolpath and your entry **IS NOT** being located at the curve start point, go to the **Cut Parameters** tab and make sure the option called **Use Mid-Point of Longest Side** is not checked. If it is, uncheck it and **Generate** the operation again.



Cut Start Point Controls in other Methods

Here is a list of the toolpaths that allow for **Cut Start Point** control.

1. All **2 Axis** toolpath methods.

2. [2 Axis Roughing](#) and [2 Axis Pocketing](#) have an additional [Start Points](#) sub-tab on the [Control Geometry](#) tab where you can define [Cut Start Points](#).
3. [3 Axis Horizontal Roughing](#) and [3 Axis Horizontal Re-Roughing](#) also have an additional [Start Points](#) sub-tab on the [Control Geometry](#) tab where you can define [Cut Start Points](#).
4. Other 3 Axis methods have various parameters that control the toolpath so be sure to review all of the controls on the [Cut Parameter](#) tabs for each operation type.

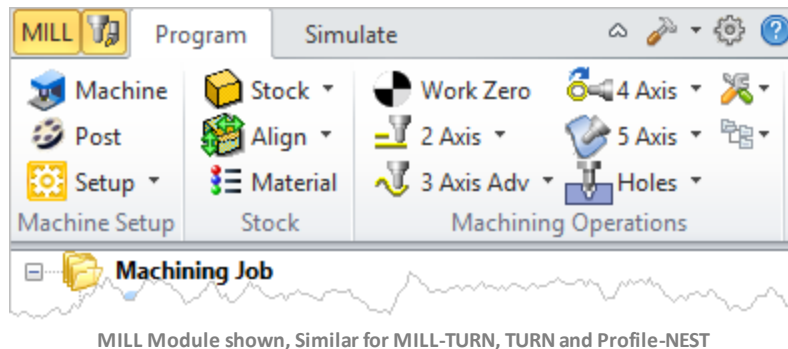
Copy/Edit a Toolpath

Toolpaths can be combined to machine complex objects. In our [How to Generate a Toolpath](#) example, we showed you how to generate a single **2 Axis Pocketing** toolpath. However, you might have noticed that example only cut the bottom portion pocket. There is also a pocket above it that has different geometry.

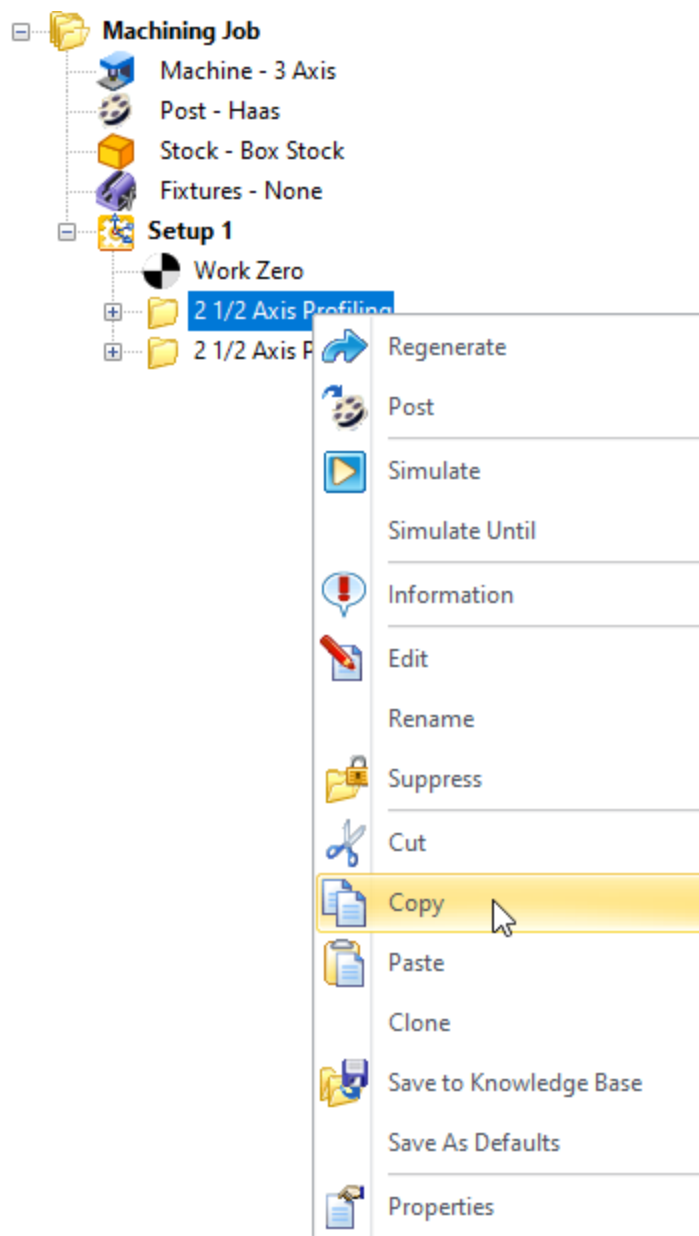
Note that all roughing and finishing toolpaths can be generated in any order. Re-Roughing requires a previous Roughing toolpath to be generated and simulated first. However when completed, it is a good practice to have each toolpath appear in the **Machining Job** in the order you plan to machine them. You can change the order of the toolpaths after you generate them.

In the example below we create the **2 Axis Pocketing** toolpath for the top level of the pocket from a copy of the previous toolpath. We will then move it up in the **Machining Job**.

1. First read [How to Generate a Toolpath](#) topic and then proceed.
2. Select the **Program** tab.

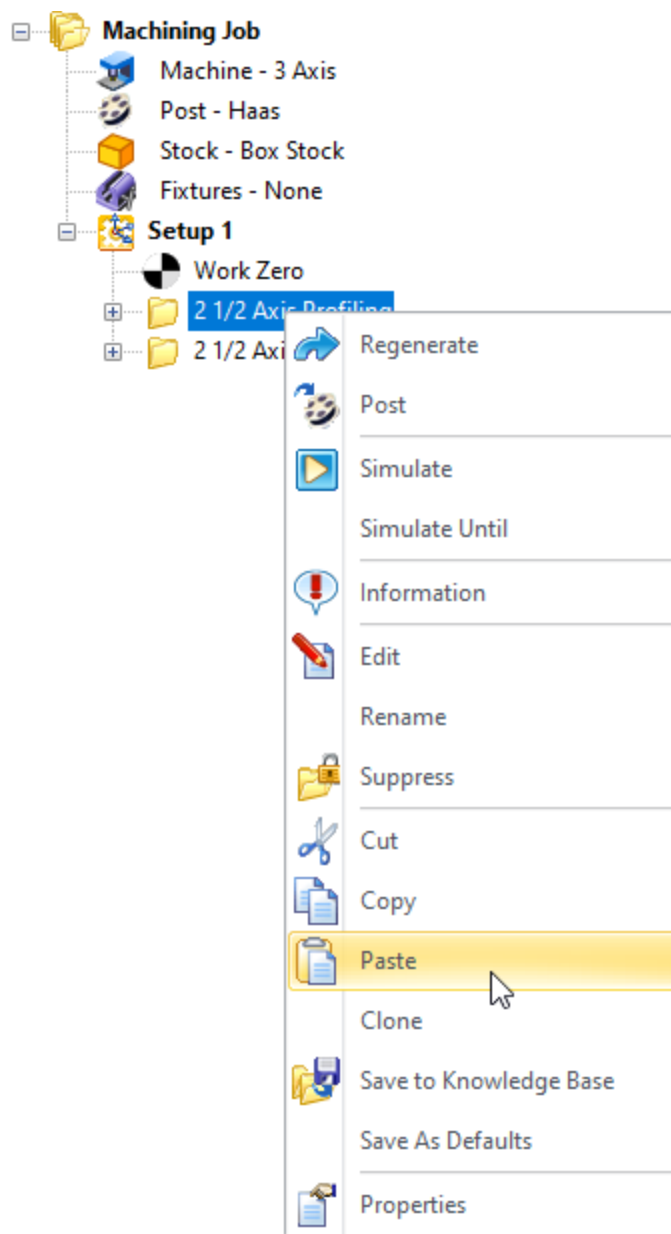


3. Select a toolpath from the **Machining Job**, right-click and select **Copy** from the menu.

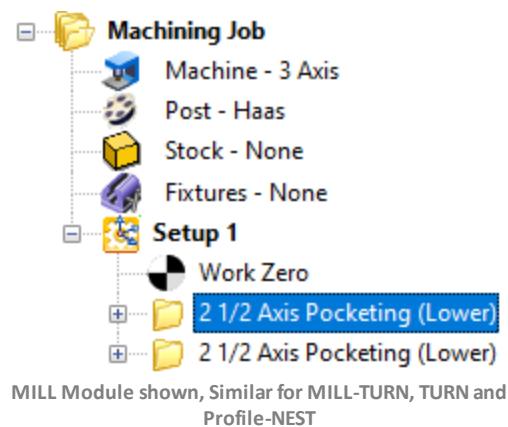


Machining Browser: Copy an Operation - Premium Configuration Shown

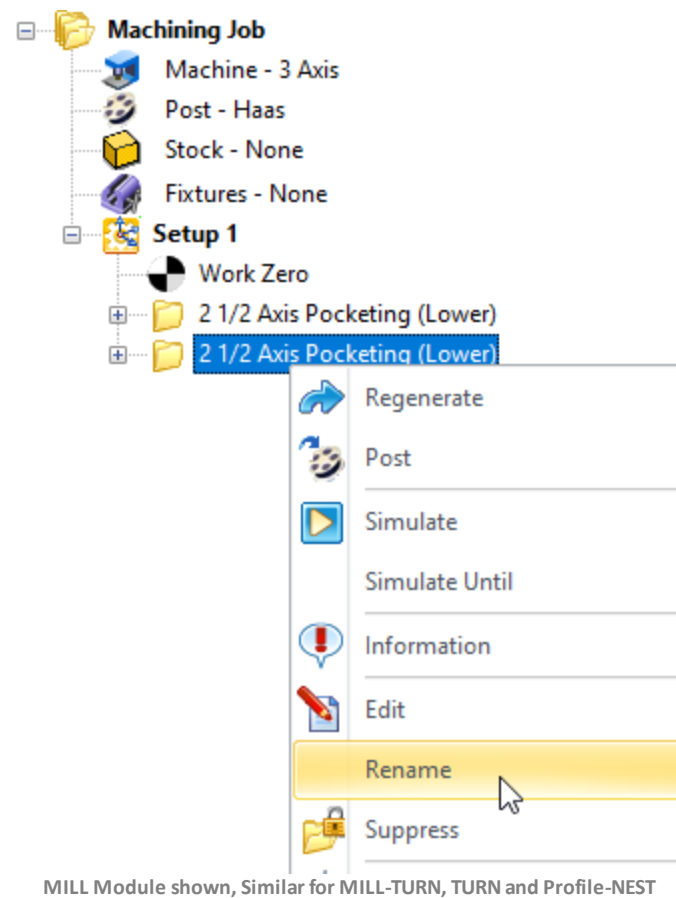
4. Now right-click on the toolpath again and select [Paste](#).



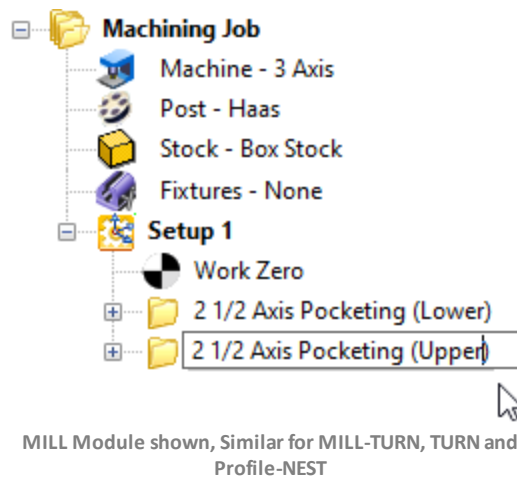
5. A copy of the toolpath will appear in the [Machining Job](#) below the original.



6. Now select the second operation, right-click and select [Rename](#).

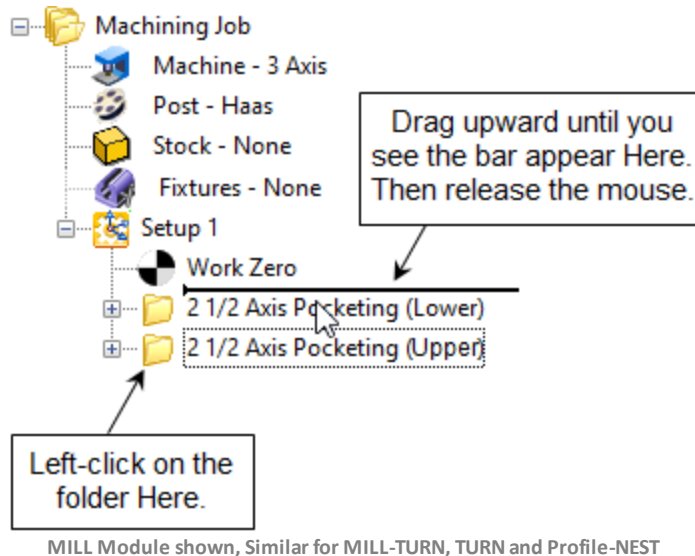


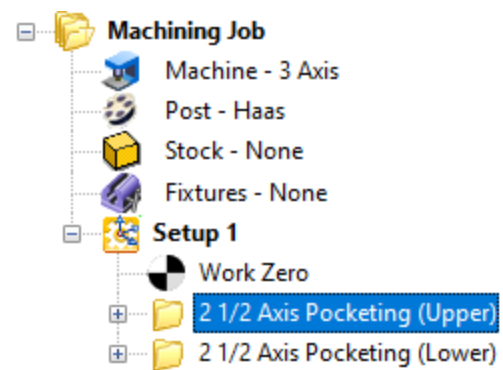
7. The name of the operation will be activated for editing. Change the name as desired and then select anywhere outside of the name to accept the edit. In this example, we changed the name to [2 1/2 Axis Pocketing \(Upper\)](#).



8. Now left-click on the folder to the left of the copied operation and Drag it up until you see a horizontal bar appear. This line indicates to you where in the **Machining Job** tree the operation will go if you release the left-mouse button.

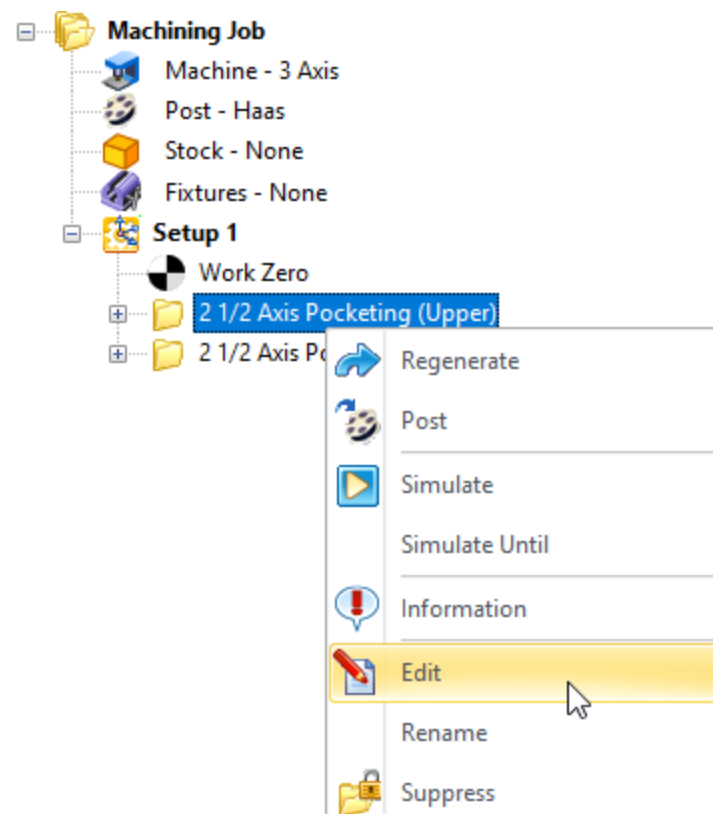
When you see the horizontal bar appear above the operation where you want to move to, release the mouse and the operation will move to that location.



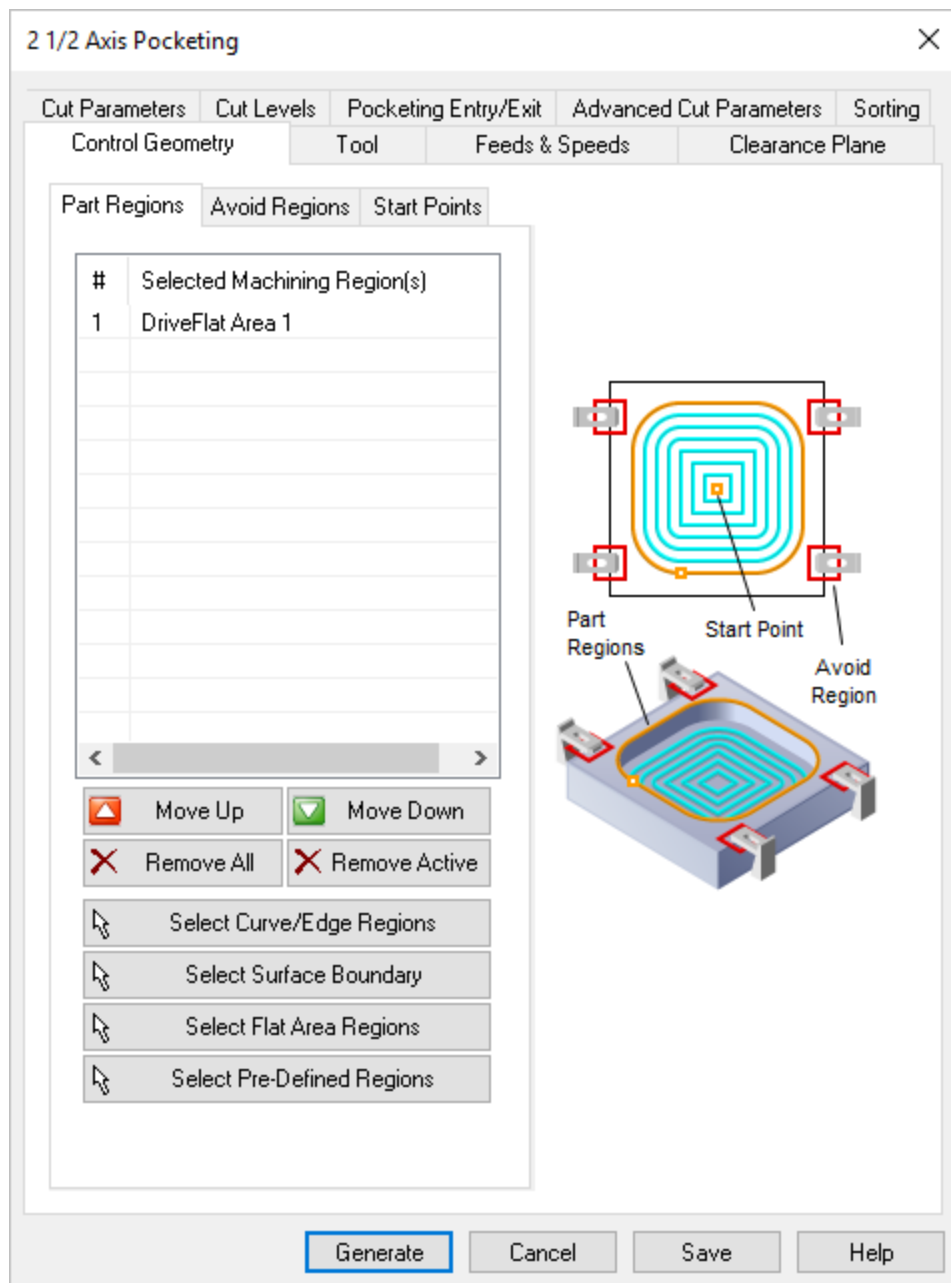


MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

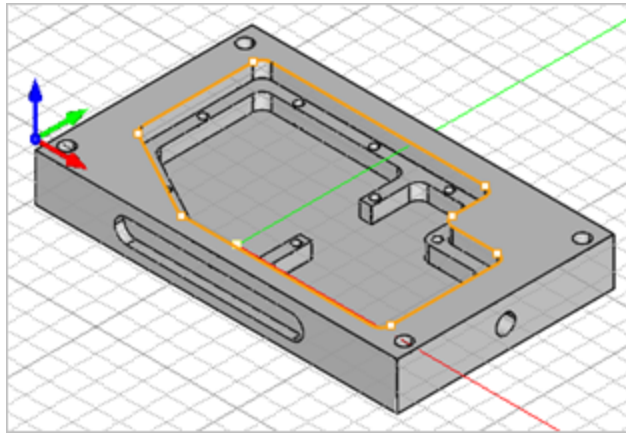
9. Now right-click on the operation you just moved and select **Edit** to display the toolpaths operation dialog.



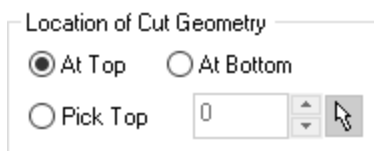
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST



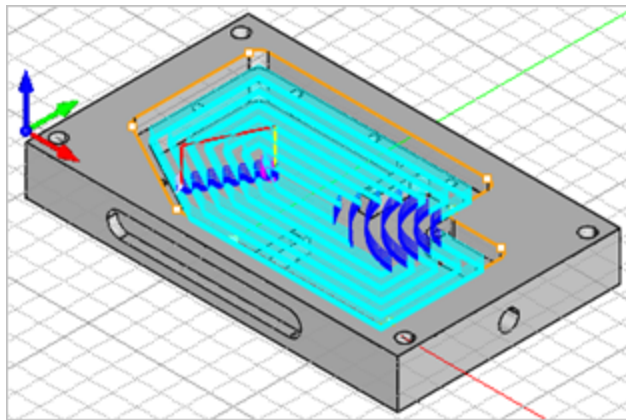
10.  Remove All The remaining procedure is similar to [How to Generate a Toolpath](#) so we will not show each dialog. From the **Control Geometry** tab select the **Remove All** button to remove the machining regions from this list.
11.  Select Curve/Edge Regions Now pick the **Select Curve/Edge Regions** button and the dialog will minimize.
12. Select the face edges around the top perimeter of the upper pocket. Then right-click or press **<Enter>** to continue.



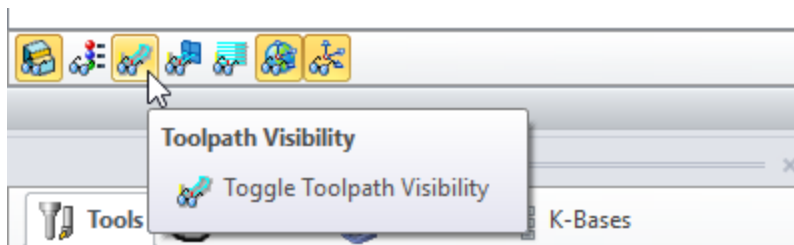
13. Pick the [Cut Levels](#) tab and change the [Location of Cut Geometry](#) to [At Top](#).



14. [Generate](#) The remaining tabs and parameters can remain the so we pick [Generate](#) to calculate the toolpath and display it on the screen.



15.  If you do not see the toolpath displayed, select the [Toggle Toolpath Visibility](#) icon located at the bottom of the [Machining Browser](#).



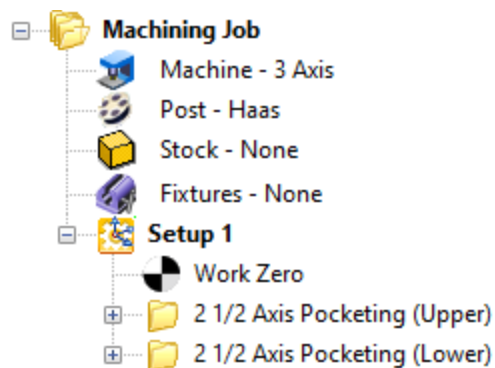
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

Create a Setup Sheet

You can generate a [Setup Sheet](#) for one or more toolpath operations. There are several templates available to choose from. Each provides various degrees of information for the selected operations.

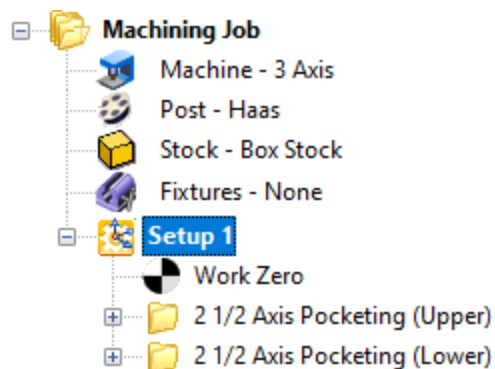
To generate a [Setup Sheet](#):

1. Create and adjust the toolpath operations that you want to estimate.
2. Make sure the toolpaths have generated cleanly. Each toolpath when generated is listed under a [Setup](#) in the [Machining Job](#). If the operation is flagged it means that it needs to be regenerated.



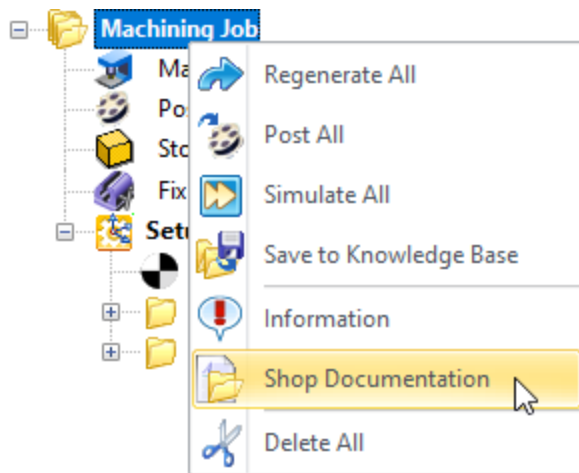
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

3. Select the [Setup](#) that you want to create a [Setup Sheet](#) for. You can also select the [Machining Job](#).



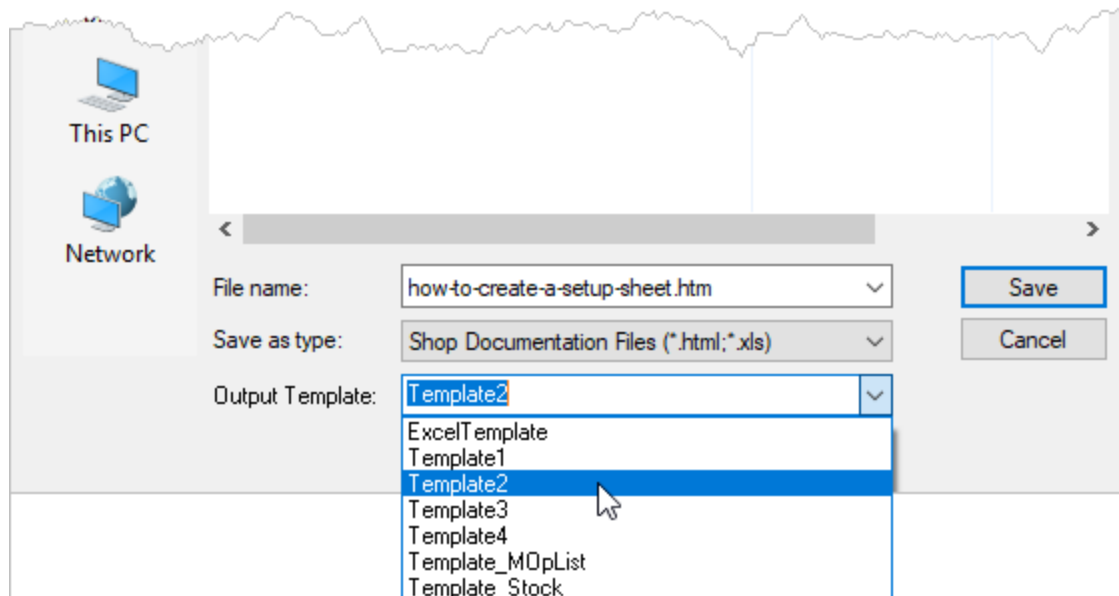
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

4. Right-click and select [Shop Documentation](#) to display the dialog.

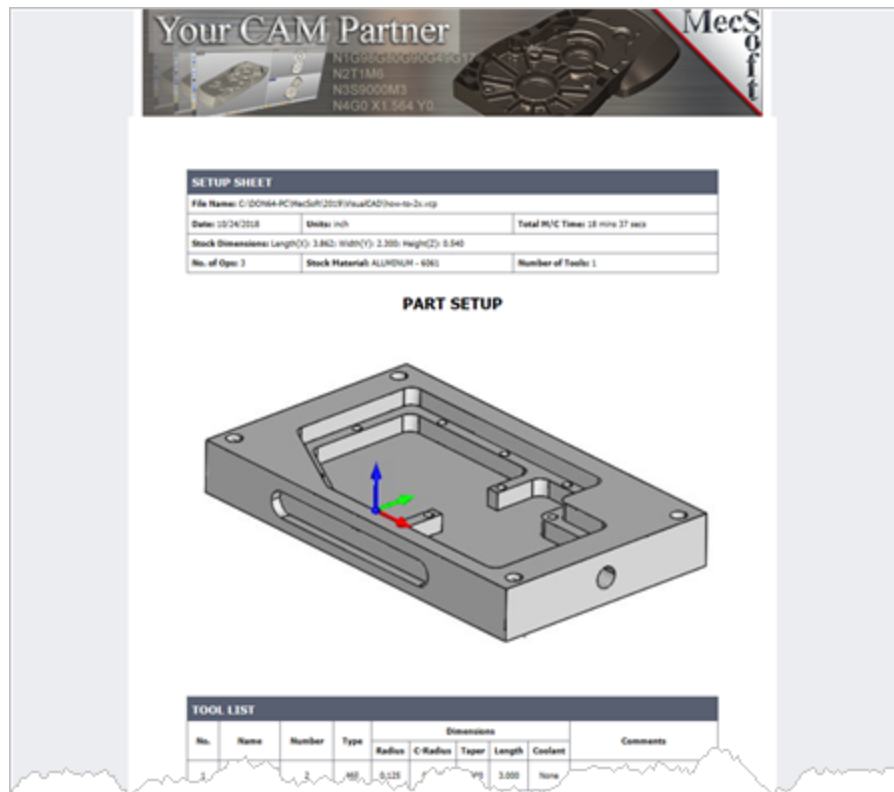


MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

- At the bottom of the dialog, select the [Output Template](#) to use. The [Excel Template](#) will display the [Setup Sheet](#) in a spreadsheet. All other template selections will display the [Setup Sheet](#) as an [HTML](#) page.



- Enter a name for the [Setup Sheet](#) and pick [Save](#).
- The [Setup Sheet](#) is generated and displayed. HTML pages are displayed in your default web browser.

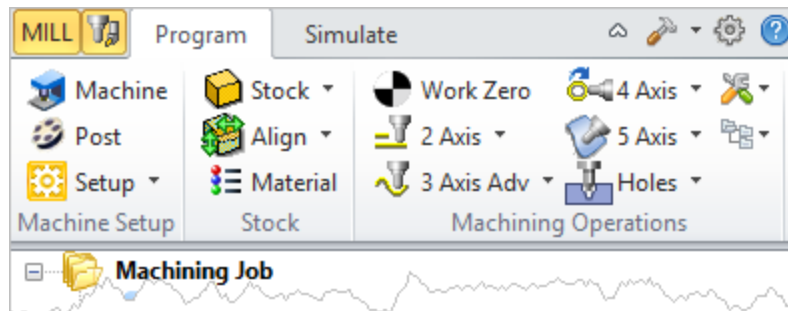


Create a Tool

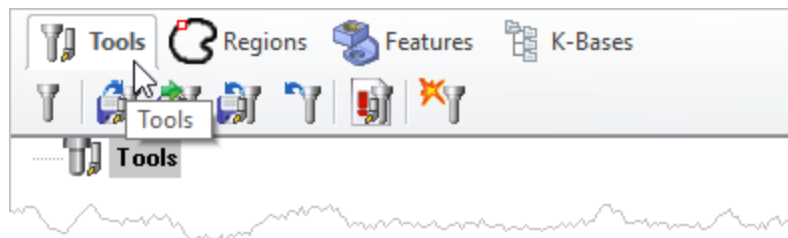
Cutting tools can be created anytime before during or after your create a toolpath. We have an extensive guide just on cutting tools called [The Cutting Tools Workbook](#) that not only teaches you about cutting tools but also provides worksheets for documenting the parameters you will need from your existing tool bin to create those tools in [RhinoCAM](#).

To create a cutting tool:

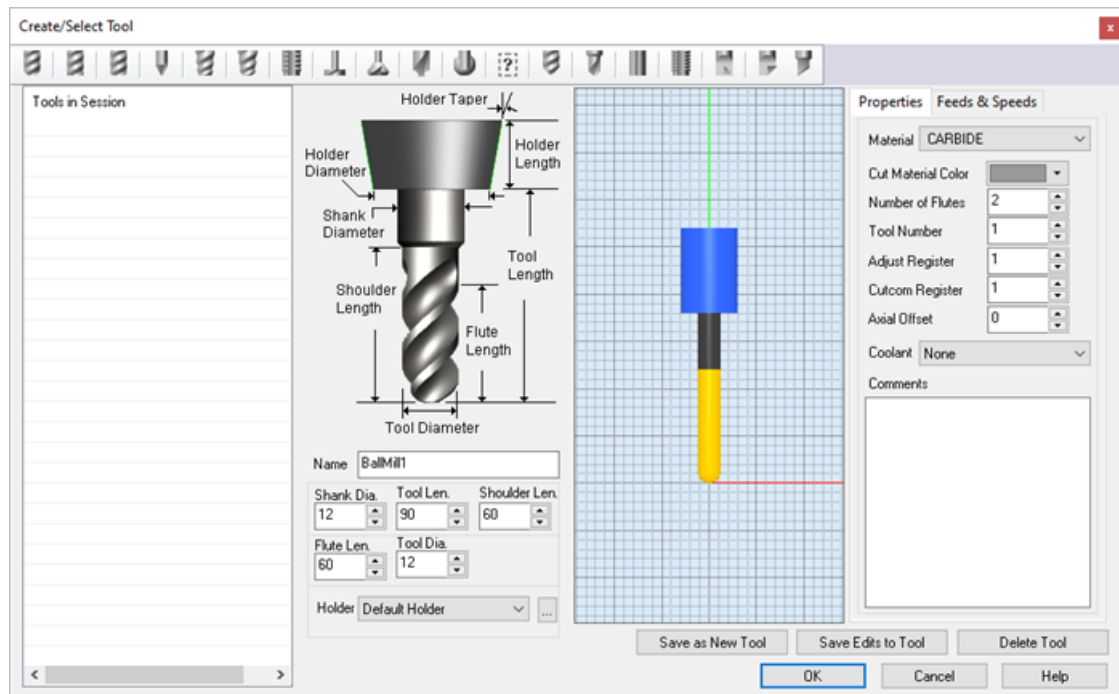
1.  To the left of the **Program** tab, select the **Tools Machining Objects** icon to make sure the **Machining Objects Browser** is displayed.



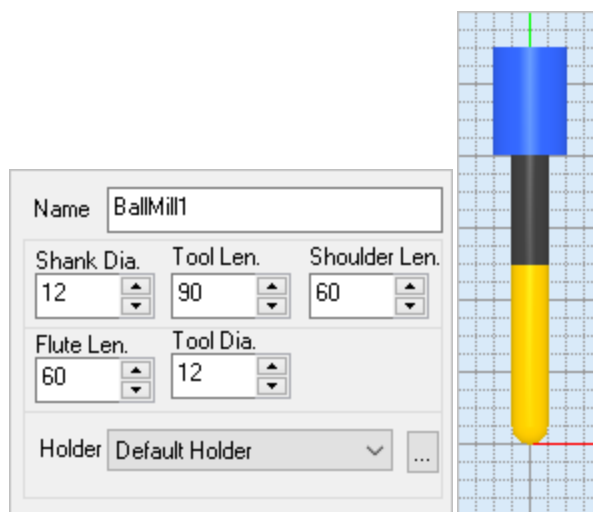
2. Select the **Tools** tab from the **Machining Objects Browser**.



3.  Select the **Create/Edit Tools** icon (first icon on the left) to display the **Create/Select Tool** dialog.

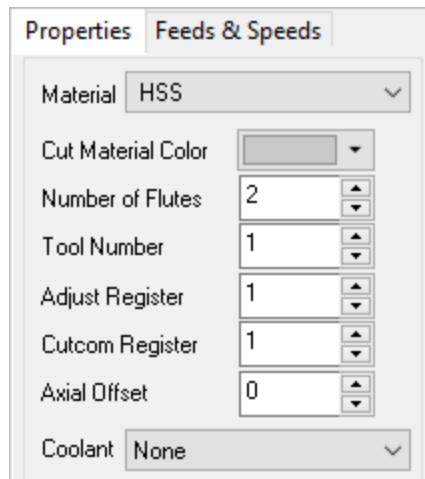


4.  The top toolbar lists the supported cutting tools. Select the icon of the tool type you want to create. The dialog will populate with the Tool type's parameters. For this example we select the Ball Mill tool icon.
5. Edit the **Name** field to label your tool with a unique identifying name. Tools are stored by their name.
6. Edit the dimension fields to define the tool's physical characteristics including the tool holder, such as diameter length, etc. A preview of the tool will display in the dialog.



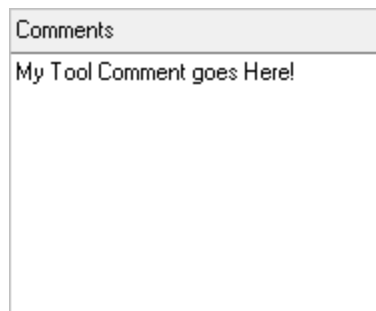
7. Move to the right side of the dialog and select the **Properties** tab.

8. Here, make selections and enter values to define the tool's properties. Tool [Material](#), [Number of Flutes](#), [Tool Number](#), etc. are defined here.



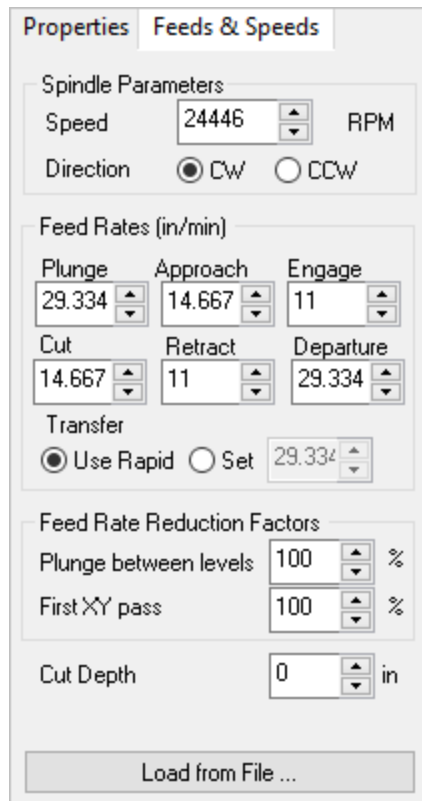
Properties	
Material	HSS
Cut Material Color	
Number of Flutes	2
Tool Number	1
Adjust Register	1
Cutcom Register	1
Axial Offset	0
Coolant	None

9. [Coolant](#) type can be assigned to each tool also.
10. If you want a comment to be posted in the G-Code file when the tool is used, enter it in the [Comments](#) field.



Comments
My Tool Comment goes Here!

11. Now select the [Feeds & Speeds](#) tab. Here you can assign all of the [Spindle Parameters](#) and [Feed Rates](#) that you want this tool to use.



The image shows the 'Feeds & Speeds' tab of the RhinoCAM 2025 Properties dialog. It is organized into several sections: 'Spindle Parameters' with Speed (24446 RPM) and Direction (CW selected); 'Feed Rates (in/min)' with sub-sections for Plunge (29.334), Approach (14.667), Engage (11), Cut (14.667), Retract (11), and Departure (29.334); a 'Transfer' section with 'Use Rapid' selected and a Set value of 29.334; 'Feed Rate Reduction Factors' for Plunge between levels (100%) and First XY pass (100%); and a 'Cut Depth' field set to 0 in. A 'Load from File ...' button is at the bottom.

Spindle Parameters	
Speed	24446 RPM
Direction	☒ CW ☐ CCW

Feed Rates (in/min)		
Plunge	Approach	Engage
29.334	14.667	11
Cut	Retract	Departure
14.667	11	29.334

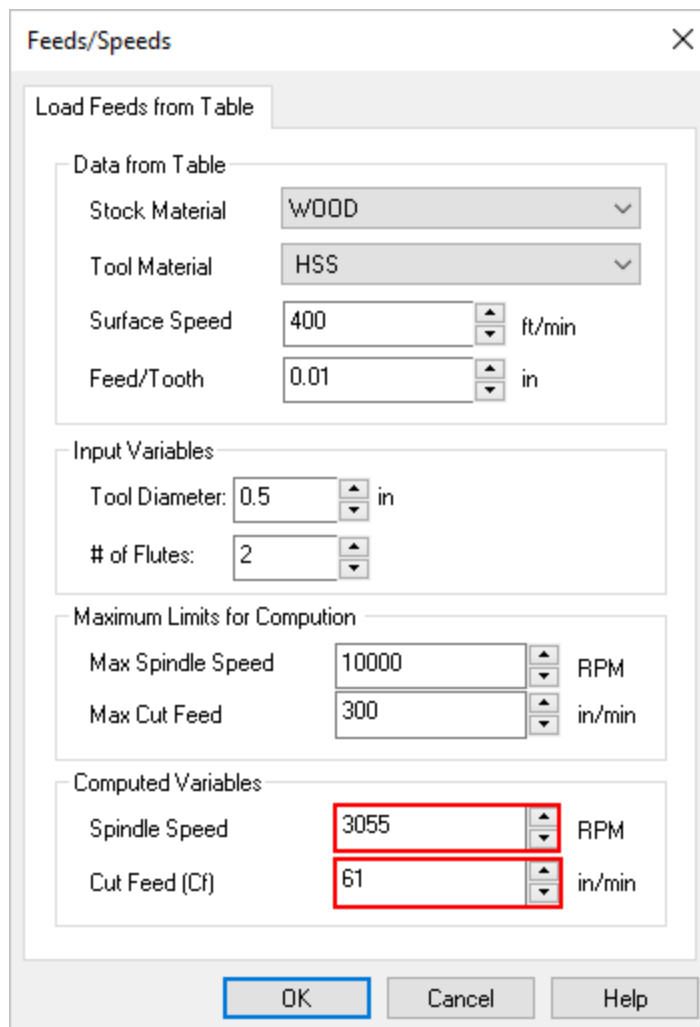
Transfer	
☒ Use Rapid ☐ Set	29.334

Feed Rate Reduction Factors	
Plunge between levels	100 %
First XY pass	100 %

Cut Depth	
0	in

Load from File ...

12. If you are unsure what values to use, you can select the [Load from File](#) button to display the [Feeds and Speeds Calculator](#) dialog.



The image shows a 'Feeds/Speeds' dialog box with a close button (X) in the top right corner. It has a tab labeled 'Load Feeds from Table'. Inside the dialog, there are four sections: 'Data from Table' with 'Stock Material' set to 'WOOD' and 'Tool Material' set to 'HSS'; 'Surface Speed' set to 400 ft/min and 'Feed/Tooth' set to 0.01 in; 'Input Variables' with 'Tool Diameter' set to 0.5 in and '# of Flutes' set to 2; and 'Maximum Limits for Computation' with 'Max Spindle Speed' set to 10000 RPM and 'Max Cut Feed' set to 300 in/min. The 'Computed Variables' section at the bottom shows 'Spindle Speed' as 3055 RPM and 'Cut Feed (Cf)' as 61 in/min, both of which are highlighted with red rectangular boxes. At the bottom of the dialog are three buttons: 'OK' (highlighted with a blue border), 'Cancel', and 'Help'.

13. From here you will see that information about the **Tool** and **Stock** material are displayed. See [How to Assign a Stock Material](#) for more information.
14. This dialog is a reference calculator that you can use to suggest spindle speed and cut feed rates based on multiple parameters and variables. You will notice that changing parameters and input variables in this dialog will update the **Computed Variables** section at the bottom of the dialog including **Spindle Speed** and **Cut Feed**.

NOTE: This dialog is only used as a reference to calculate a suggested value for **Spindle Speed** and **Cut Feed**. **ALWAYS** use values for these two parameters that are within recommendations suggested by your machine tool vendor and your first-hand experience with YOUR machine.

15. Pick **OK** when these two parameter values are to your liking for this tool.
16. You will notice that the values for these two parameter values **Spindle Speed** and **Cut Feed** are then fed into the **Feeds & Speeds** tab of the **Create/Edit Tool** dialog.

Properties Feeds & Speeds

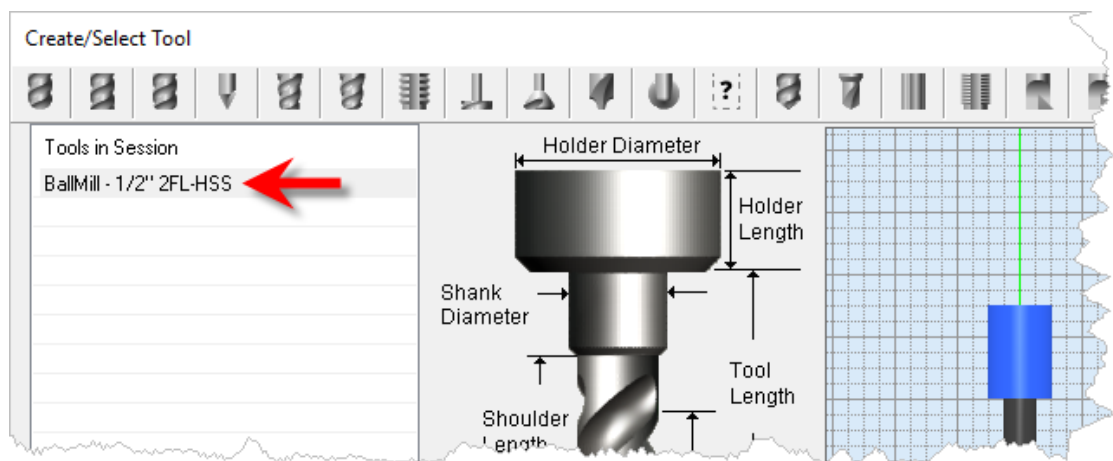
Spindle Parameters
 Speed **3055** RPM
 Direction ☒ CW ☐ CCW

Feed Rates (in/min)
 Plunge 30.5 Approach 45.75 Engage 45.75
 Cut **61** Retract 76.25 Departure 91.5
 Transfer
☒ Use Rapid ☐ Set 122

Feed Rate Reduction Factors
 Plunge between levels 100 %
 First XY pass 100 %
 Cut Depth 0 in

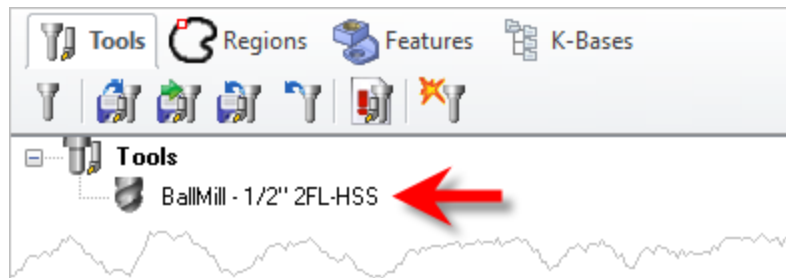
Load from File ...

17. You will also see that a percentages of the **Cut Feed** is used to populate the other **Feed Rate** value in the **Feed Rates** section. These percentages can be defined in the **Feeds & Speeds** section of the **CAM Preferences** dialog.
18. You can also adjust the **Feed Rate Reduction Factors** if desired.
19. When you are satisfied with your tool parameters, select the **Save as New Tool** button. The new tool will be listed in the **Tools in Session** list on the left side of the dialog.

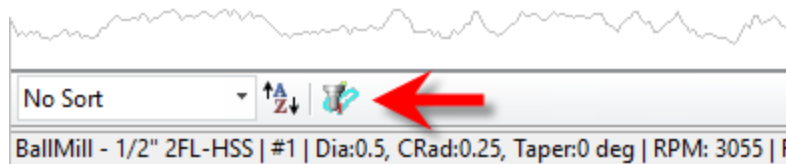


20. If you make changes after saving the tool be sure to select the **Save Edits to Tool** button to save your changes.

21. Now pick **OK** to close the dialog. You will see that the tool is then listed in the **Tools** tab list of tools.



22.  If you do not see your tool listed make sure the icon to **List only tools used in machining operations** located at the bottom of the tools list is NOT enabled.



23. You can also create a tool from the **Tools** tab of each of the different toolpath operation dialogs. From the **Tools** tab, just select the **Edit/Create/Select Tool ...** button to top display the **Create/Select Tool** dialog again. See [How to Generate a Toolpath](#) for more information.



Related Online Help Topics

User Interface

Tool

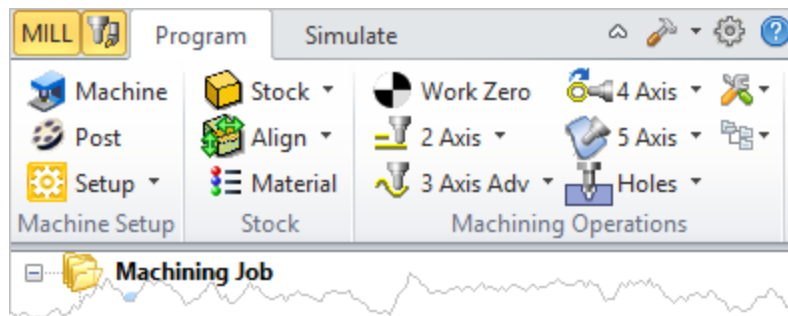
Create Edit Tools

Create a Work Zero

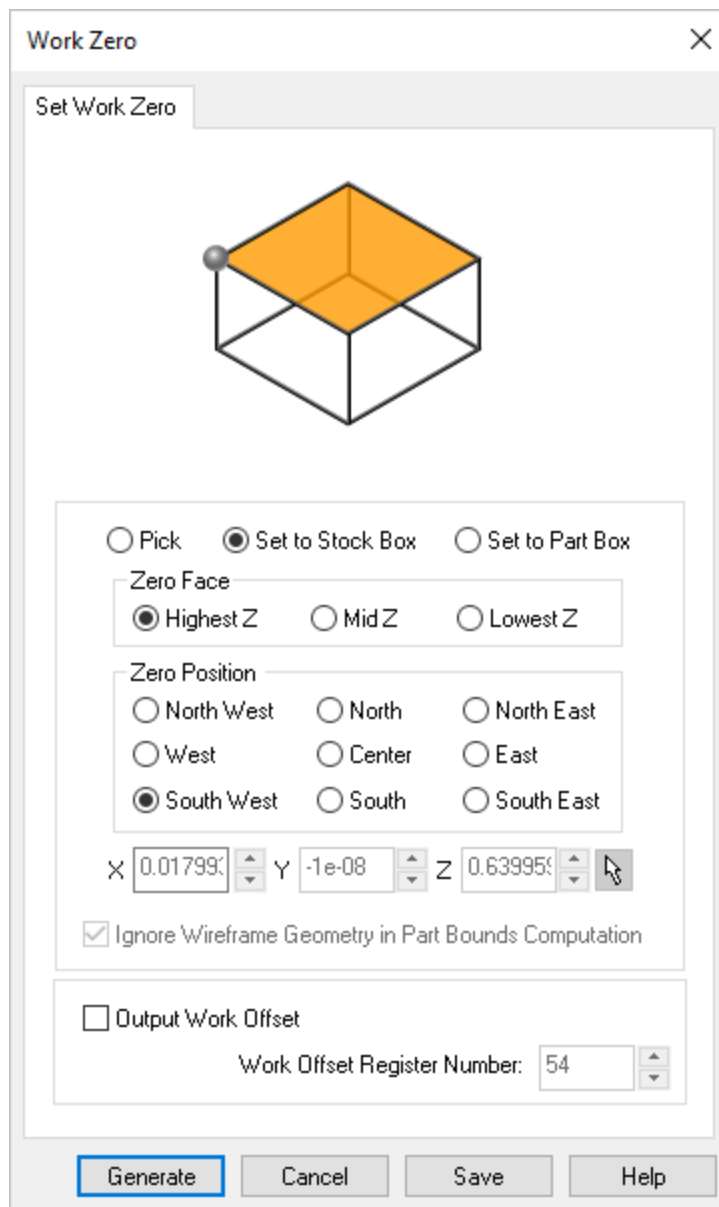
When you zero your CNC machine for a job, you can tell [RhinoCAM](#) where this [Work Zero](#) location is. Your G-Code coordinates will then be calculated from this location. You can also have multiple [Work Zero](#) locations defined in your part. If no [Work Zero](#) is defined, then coordinates are measured from the [WCS](#) origin of the part.

To Create a [Work Zero](#):

1. Select the [Program](#) tab.



2.  Select [Work Zero](#) from the menu to display the dialog.

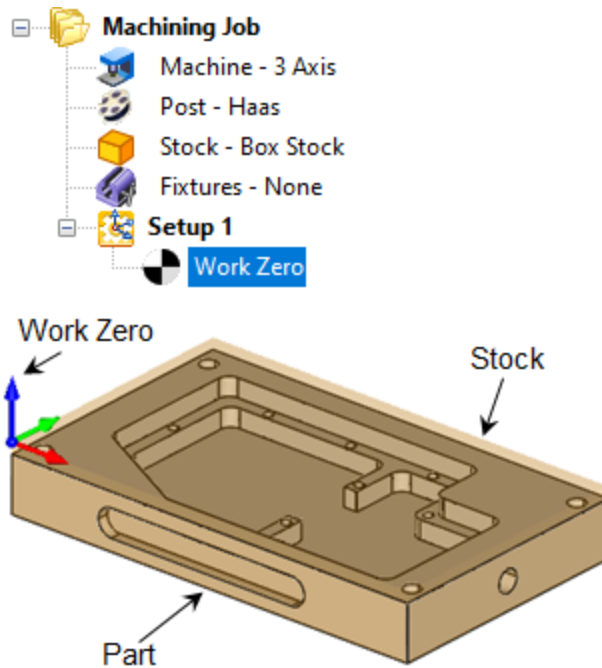
The image shows a software dialog box titled "Work Zero" with a close button (X) in the top right corner. The dialog has a tab labeled "Set Work Zero". Below the tab is a 3D wireframe cube with an orange top face and a grey sphere at its top-left-front corner. The main area contains three radio button options: "Pick", "Set to Stock Box" (which is selected), and "Set to Part Box". Below these are two grouped sections. The "Zero Face" section has three radio buttons: "Highest Z" (selected), "Mid Z", and "Lowest Z". The "Zero Position" section has a 3x3 grid of radio buttons for "North West", "North", "North East", "West", "Center", "East", "South West" (selected), "South", and "South East". Below these sections are three input fields for coordinates: X (0.01799), Y (-1e-08), and Z (0.63995), each with up/down arrows and a mouse cursor icon. A checkbox labeled "Ignore Wireframe Geometry in Part Bounds Computation" is checked. At the bottom of the main area is a checkbox for "Output Work Offset" which is unchecked, followed by a "Work Offset Register Number:" label and a spinner box containing the value "54". The bottom of the dialog features four buttons: "Generate" (highlighted with a blue border), "Cancel", "Save", and "Help".

Dialog Box: Work Zero

3. You have the option to [Pick](#) the [Work Zero](#) location. Select [Pick](#) and then select the pick button to select a point on the part. Its coordinates are populated into the dialog.
4. Optionally you can select [Set to Stock Box](#) and then select from the three available options [Highest Z](#), [Mid Z](#) and [Lowest Z](#).
5. Then select the [Zero Position](#). The nine locations to choose from.
6. Optionally you can also output the [Work Offset Register](#). To do so, check the box to [Output Work Offset](#) and then accept the 54 value or change it. This value will be posted to your G-Code file as [G54](#).

- Pick **OK** and you will see the **Work Zero** added to the **Machining Job**.

In the example shown below the **Work Zero** is set to the **Stock Box**, **Highest Z**, **South West**.



Related Online Help Topics

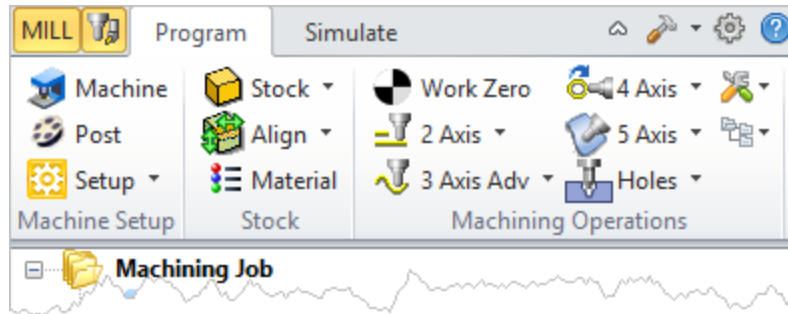
User Interface

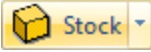
Work Zero

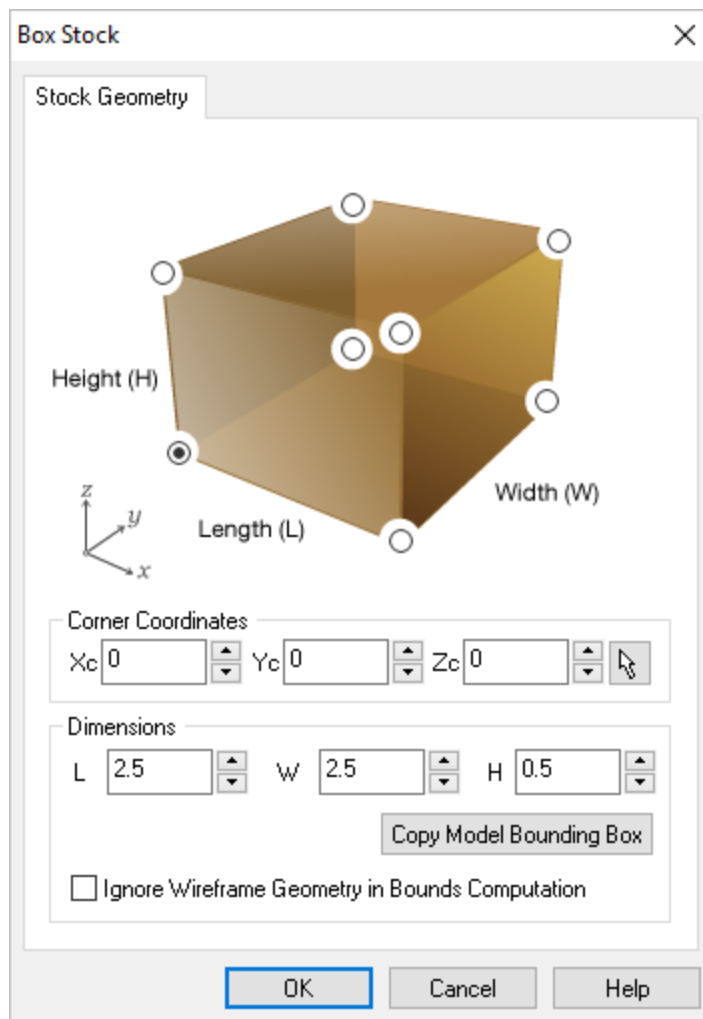
Define a Box Stock

You MUST define the [Stock](#) before creating a toolpath. There are different stock menu selections to choose from depending on your software configuration. Here are the steps to create a simple box stock.

1. Select the [Program](#) tab.

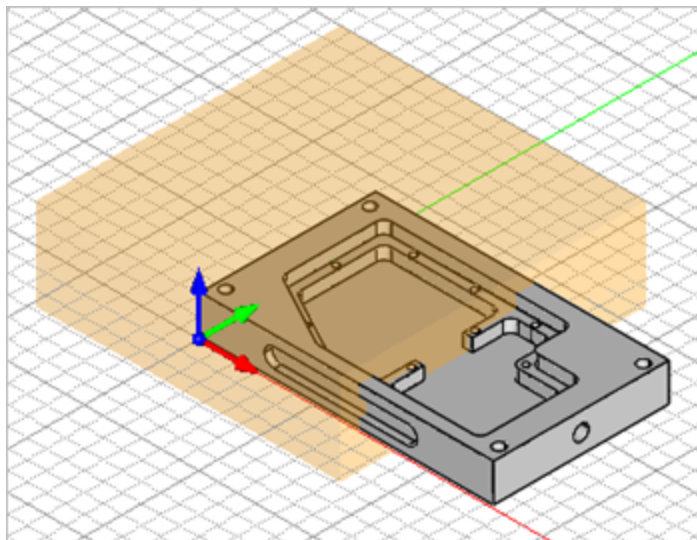


2.  From the [Program](#) tab, select the [Stock](#) menu.
3. Select [Box Stock](#) to display the dialog.

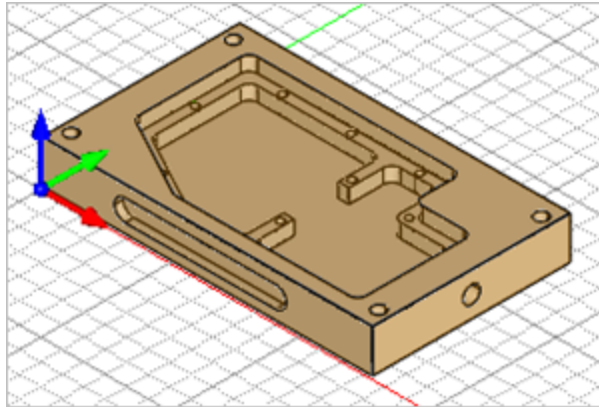


Dialog Box: Box Stock

4. The stock box will display on the part when this dialog is displayed.



5. [Copy Model Bounding Box](#) To create a stock with the exact dimensions of your part, select the [Copy Model Bounding Box](#) button.



6. If you have curves or other non-part wireframe geometry displayed, you can check the box in the dialog to [ignore wireframe geometry in Bounds Calculation](#).
7. Alternately you can enter the [Length](#), [Width](#) and [Height](#) dimensions of the stock you want to use.

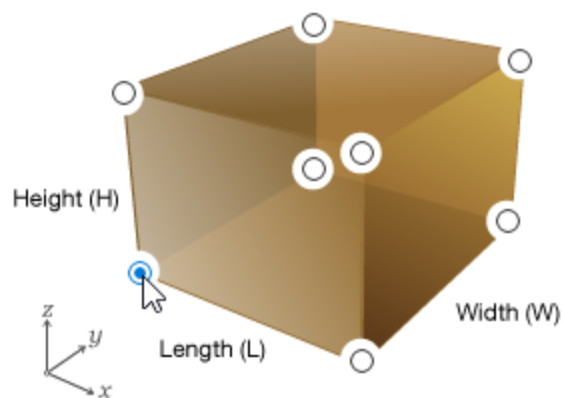
Dimensions

L	3.844	W	2.3	H	0.6
---	-------	---	-----	---	-----

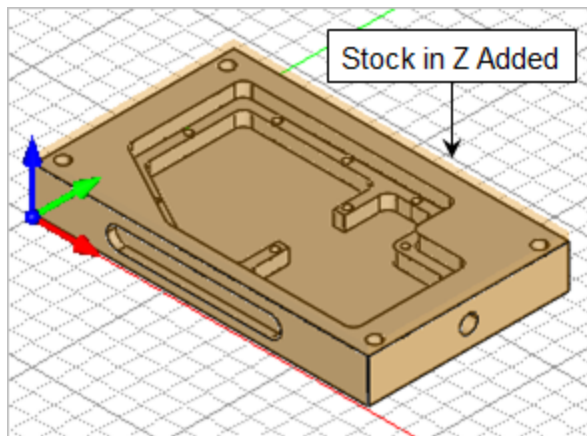
[Copy Model Bounding Box](#)

☒ Ignore Wireframe Geometry in Bounds Computation

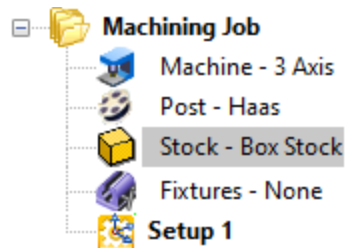
8. Select the coordinate location to measure the stock dimensions from. By default stock dimensions are measured from the bottom left corner of the part.



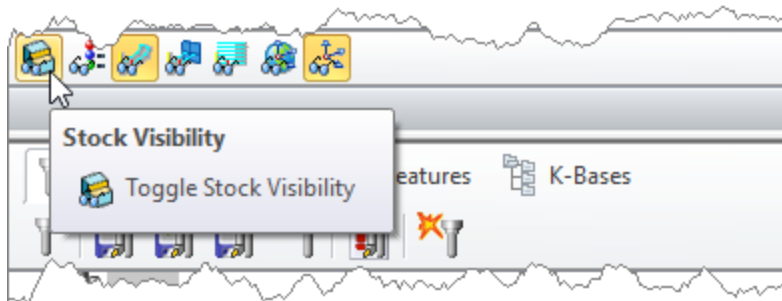
9. You can also set the [Corner Coordinates](#) if desired.
10. Making changes in the dialog will update the stock preview on the part.



11. Pick **OK** to close the dialog.
12. You will see that the **Stock** is now defined under the **Machining Job**.



13.  The stock box will display on the screen. If you do not see the stock displayed, pick the **Toggle Stock Visibility** icon located at the bottom of the **Machining Browser** - first icon from the left.



Related Online Help Topics

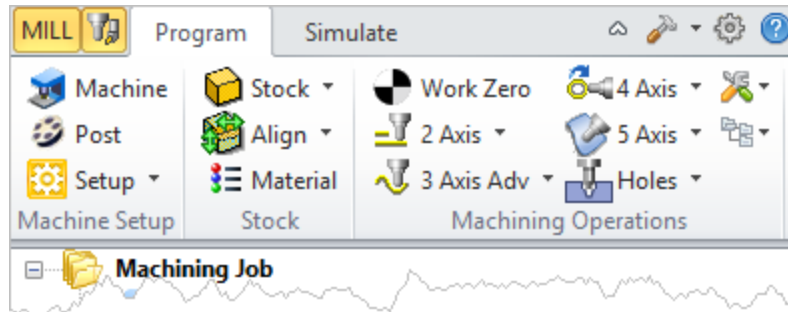
User Interface

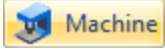
Box Stock

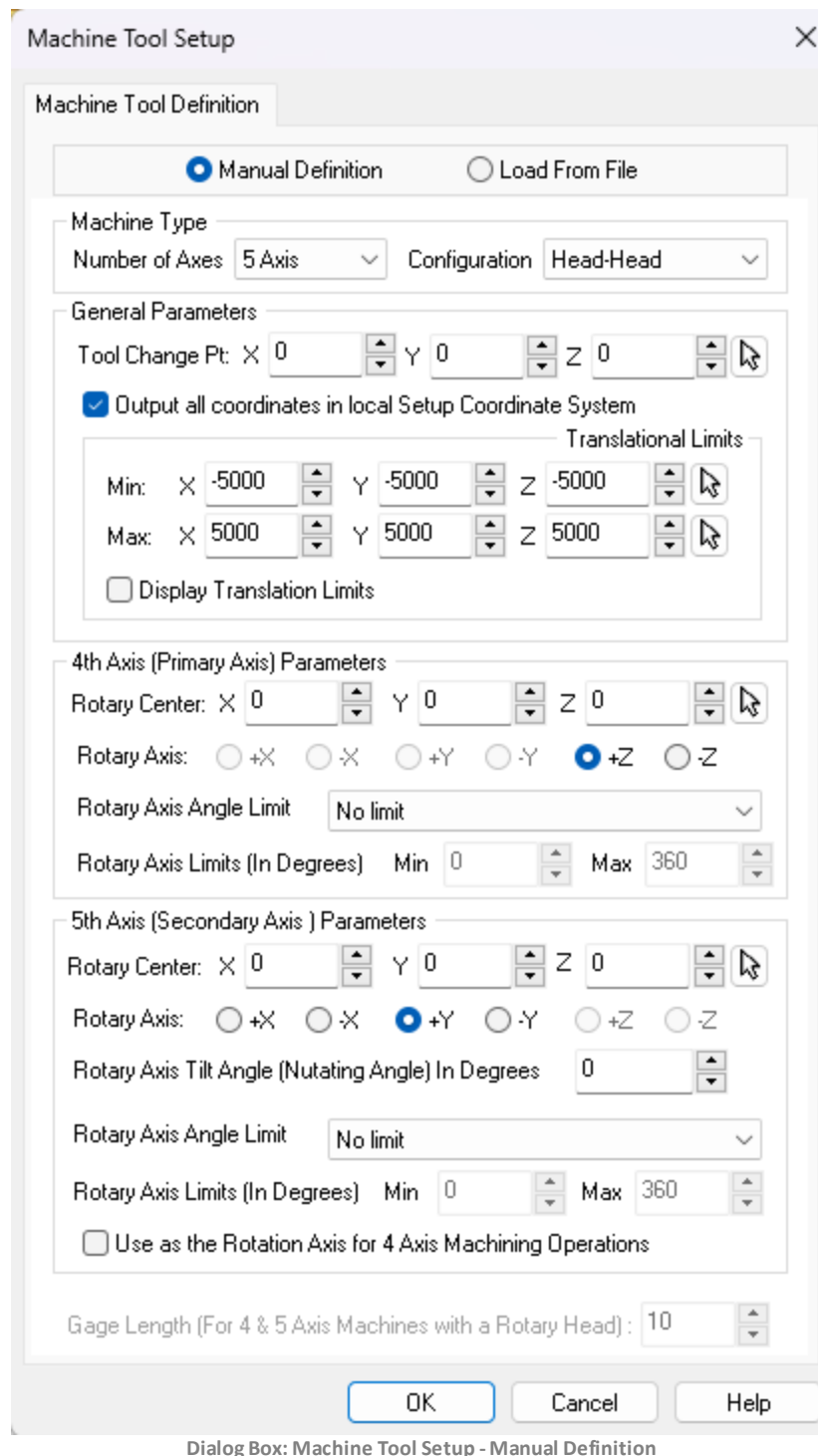
Define a Machine Tool

Before creating a toolpath make sure you have the [Machine Tool](#) definition set to the correct [Number of Axis](#) and the axis directions defined.

1. Select the [Program](#) tab.



2.  From the [Program](#) tab, select [Machine](#) to display the dialog.



3. From the **Machine Tool Definition** tab select **Manual Definition**.
4. For **Machine Type**, select the **Number of Axis** from the selection menu.
5. There are other **General Parameters** that you can set such as a **Tool Change Point** and **Translational Limits**.

6. Check the box to [Output all coordinates in local Setup Coordinate System](#). This is typically check by default.
7. If [Machine Type](#) is set to [4 Axis](#), check the [4th Axis \(Primary\) Parameters](#) section of the dialog and make sure the [Rotation Center](#) and [Rotary Axis](#) direction are set correctly. An axis indicator arrow (**Green**) will display on the part showing you this location and direction. You can also set the [Rotary Axis Limit](#) here.

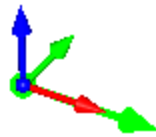
4th Axis (Primary Axis) Parameters

Rotary Center: X Y Z

Rotary Axis: ☒ +X ☐ -X ☐ +Y ☐ -Y ☐ +Z ☐ -Z

Rotary Axis Angle Limit

Rotary Axis Limits (In Degrees) Min Max



8. If [Machine Type](#) is set to [5 Axis](#), check the additional [5th Axis \(Secondary Axis\) Parameters](#) section of the dialog and make sure the [Rotary Center](#) and [Rotary Axis](#) direction are set correctly. An axis indicator arrow (**Red**) will display on the part showing you this location and direction.

5th Axis (Secondary Axis) Parameters

Rotary Center: X Y Z

Rotary Axis: ☐ +X ☐ -X ☒ +Y ☐ -Y ☐ +Z ☐ -Z

Rotary Axis Tilt Angle (Nutting Angle) In Degrees

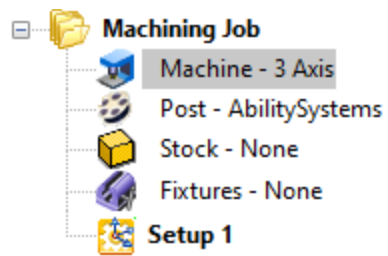
Rotary Axis Angle Limit

Rotary Axis Limits (In Degrees) Min Max



9. If [Machine Type](#) is set to [5 Axis](#) and the parameter to [Output all coordinates in local Setup Coordinate System](#) is NOT checked, make sure to set the value for [Gage Length](#) (For machines with a [Rotary Head](#)). This represents the distance from the head's rotary axis to the spindle face. This length value much match your machine in order for toolpath coordinates to be output correctly.
10. Pick [OK](#) to close the [Machine Tool Setup](#) dialog.

11. You will find that the [Machine](#) is defined under the [Machining Job](#).



Related Online Help Topics

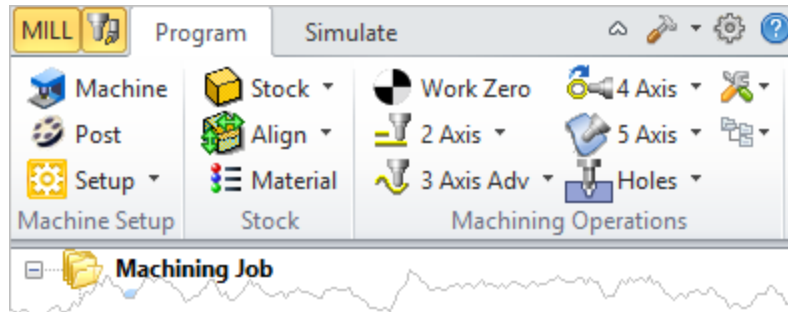
User Interface

Machine Setup

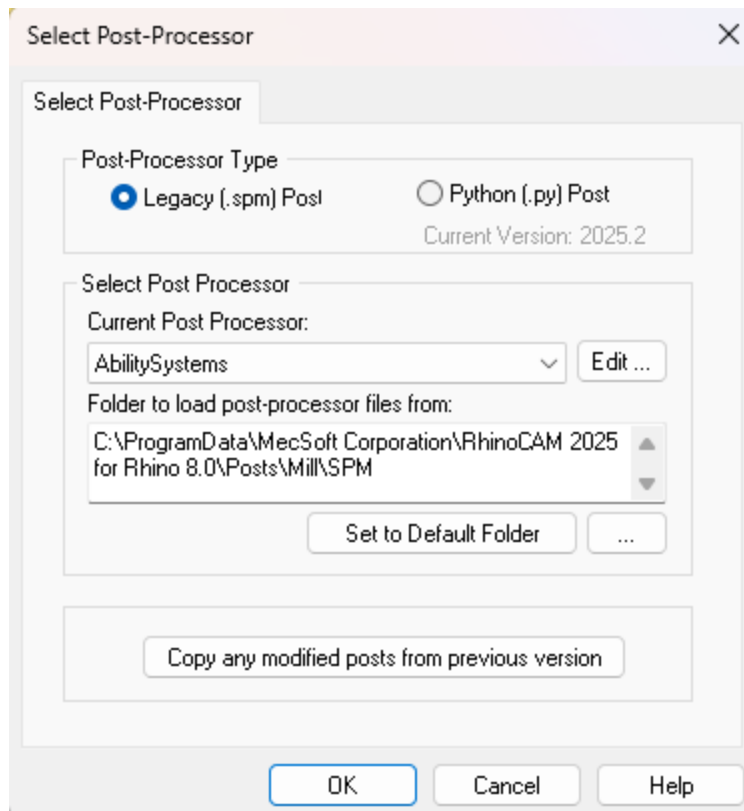
Define the Post-Processor

RhinoCAM installs with over 300 pre-defined post-processors for the most popular CNC machine controllers.

1. Select the **Program** tab.

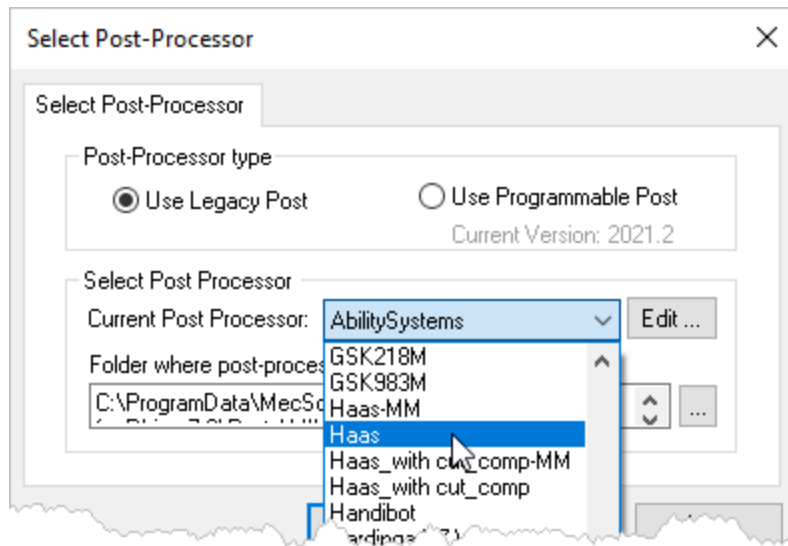


2.  From the **Program** tab, select **Post** to display the Set **Post-Processor Options** dialog.

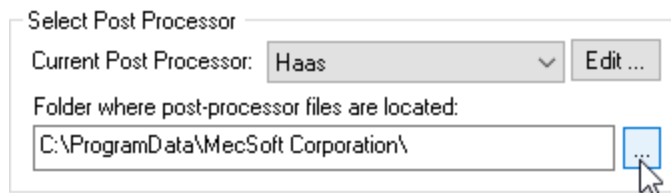


Dialog Box: Set Post-Processor Options

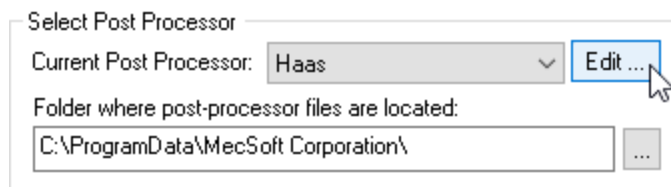
3. Select a **Post** from the **Current Post Processor** selection menu.

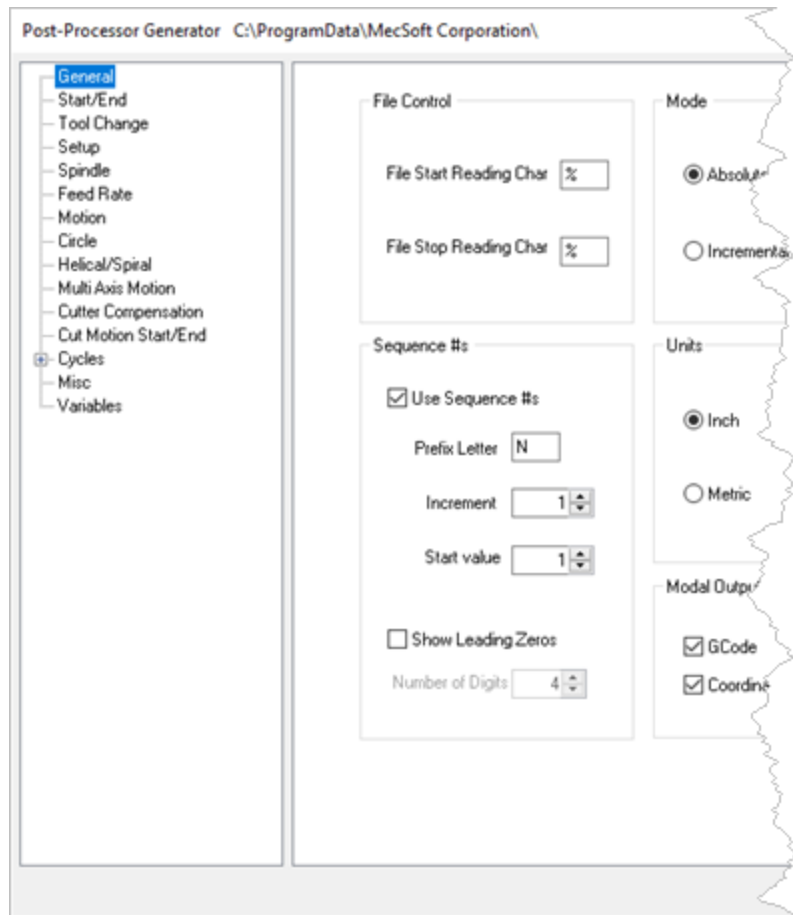


4. Each post in the menu has a corresponding post definition file (example: [Haas.spm](#)). The location of these files is displayed in the field directly below the menu called [Folder where post-processor files are located](#).
5. Make sure this points to the correct folder where your post definition file is located. To change the folder location pick the [...](#) button.

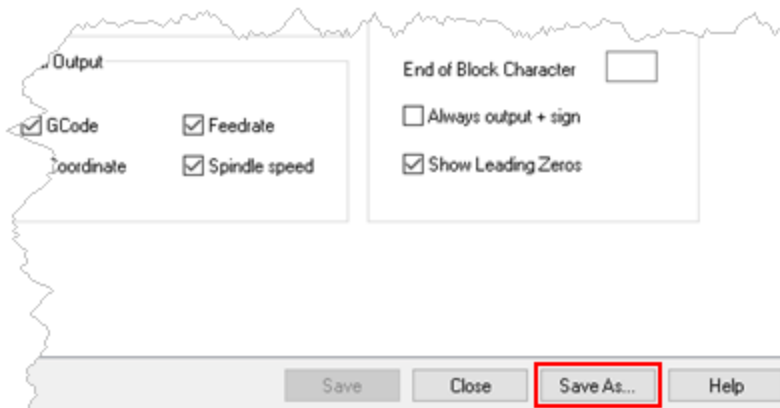


6. [RhinoCAM](#) also comes with a [Post-Processor Generator \(PPG\)](#) that allows you to edit each post definition file to suit your needs. Select the [Edit ...](#) button to the right of the [Current Post Processor](#) selection to load that post into the [PPG](#).

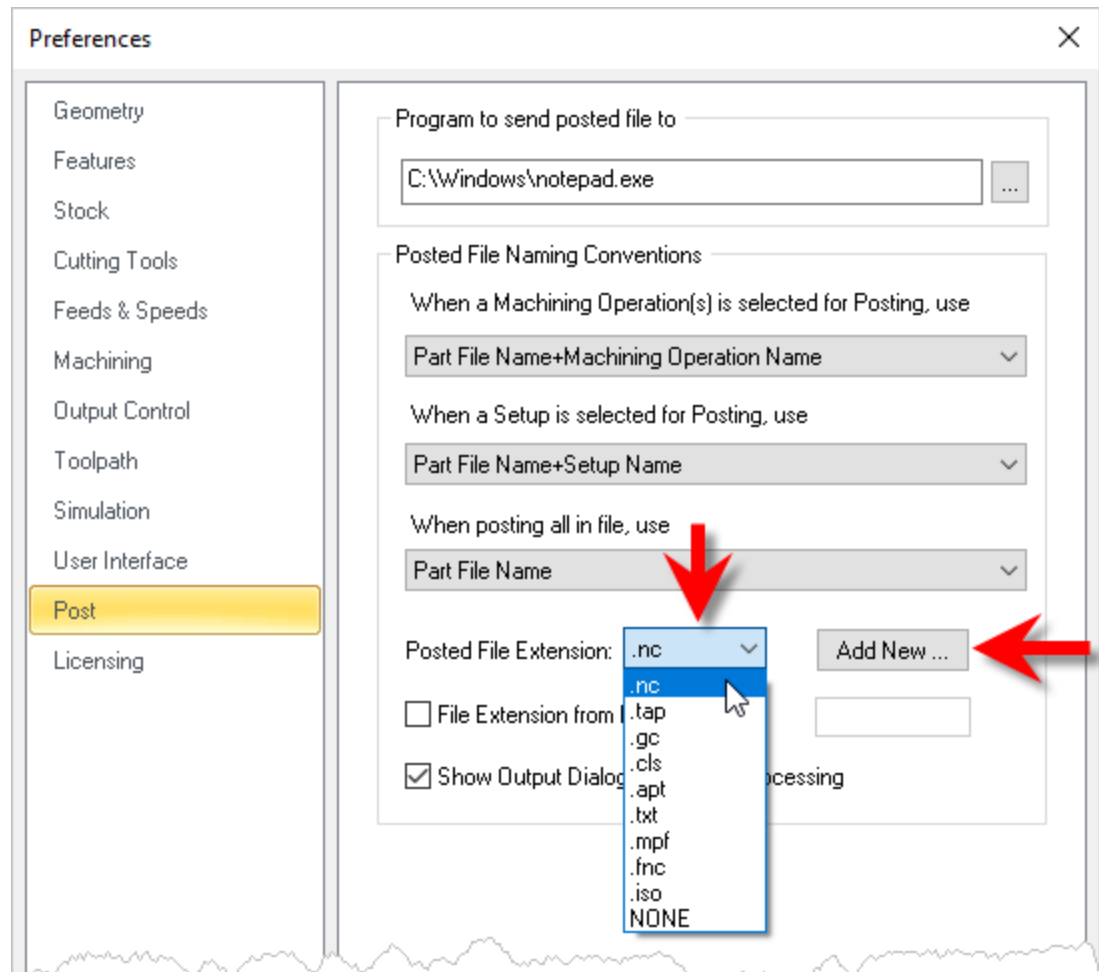




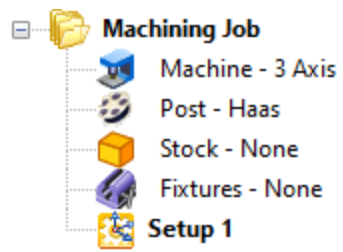
7. **IMPORTANT:** If you edit a post definition file included with **RhinoCAM** make sure you use the **Save As** button from the **PPG** dialog and save your edited post using a unique file name different than the pre-installed post name (example "Haas-REV1.spm") and save the post definition file to a folder that is OUTSIDE of the **RhinoCAM** install path. This will ensure that you will always have access to your modified posts even if **RhinoCAM** is upgraded or uninstalled.



8. Then, from the CAM Preferences dialog, select the Post tab and then drop down the [Posted File Extension](#) list and select the file extension that is required by your cnc controller software (example .nc). Your machine tool manual will tell you what file extension is required. If your file extension is not on the list, pick the [Add New ...](#) button to add it to the menu.



9. If you use a [G-Code Editor](#) program, you can tell [RhinoCAM](#) to launch it when displaying your G-Code files. By default, [notepad](#) is used to display G-Code files. See [How to Post G-Code](#) for more information.
10. You can also set file naming conventions to use when posting your G-Code files.
11. When done pick [OK](#) to close the menu.
12. You will see that the Post is now defined under the [Machining Job](#).



Related Online Help Topics

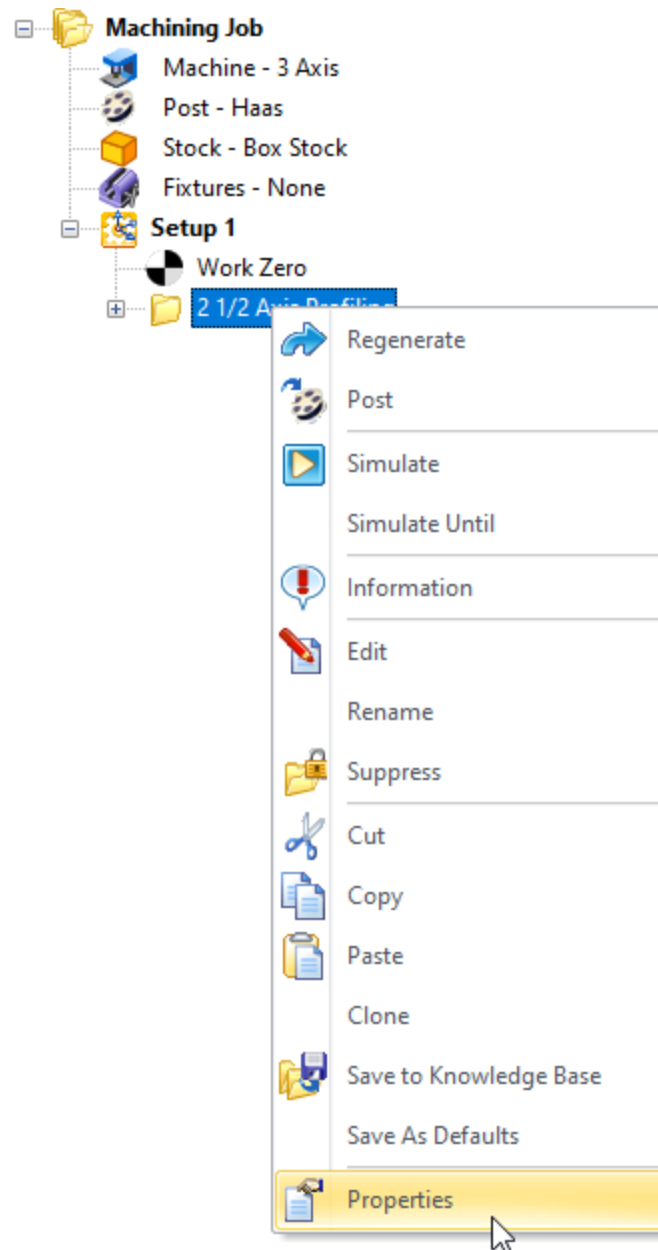
User Interface

Set Post Options

Define Toolpath Properties

You can set the properties of a [Operation](#) by selecting it in the [Machining Browser](#) window, clicking on the right mouse button and selecting the [Properties](#) menu item.

Select Properties from the Right-click Menu

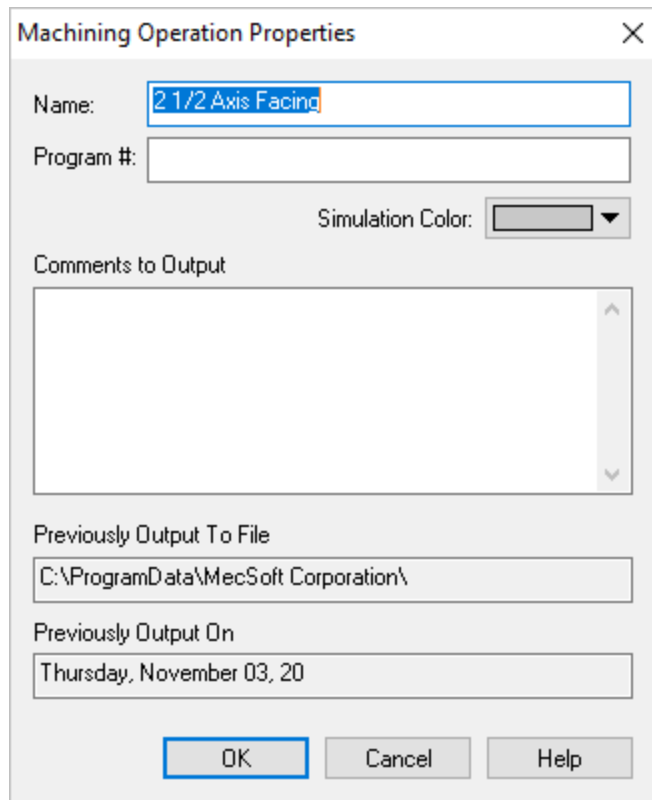


Select Properties from the Right-click Menu

Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

Dialog Box: Machining Operation Properties

This will bring up the dialog that is shown below.

The image shows a 'Machining Operation Properties' dialog box. It has a title bar with a close button. Inside, there are several fields: 'Name' with the text '2 1/2 Axis Facing', 'Program #' which is empty, 'Simulation Color' with a grey color swatch and a dropdown arrow, 'Comments to Output' with a large empty text area and a vertical scrollbar, 'Previously Output To File' with the path 'C:\ProgramData\MecSoft Corporation\' and 'Previously Output On' with the date 'Thursday, November 03, 20'. At the bottom are three buttons: 'OK', 'Cancel', and 'Help'.

Dialog Box: Machining Operation Properties

Name

Change the [Name](#) of the [Machining Operation](#).

Program

Specify [Program #](#) for the operation. This program number will be output during post processing of the operation.

Simulation Color

This allows you to specify a unique color for this operation during [Simulation](#) display. Refer to the [Simulate tab Status Bar](#) for setting the simulation to display by Mop (i.e., machining operation type).

Comments to Output

You can also include commands that will be saved with the operation. These comments will also be output during post-processing of the operation. This might be a good place to put in comments or instructions for the machine tool operator.



This can be used to put in add comments or instructions for the machine tool operator!

Previously Output To File

This refers to the name of the external post-processed file that this particular operation was output to.

Previously Output On

This refers to the last time the operation was post-processed and the time the post-processing was performed.

Edit Toolpaths Associatively

[Machining Operations](#) can be edited by using the [Machining Browser](#). Each machining operation is represented as a folder in the browser. In the expanded state of this folder icon, seven icons representing different objects that make up the operation are displayed. The first five can be associatively edited.

The Machining Operation Tree Icons

The following icons are displayed under a machining operation's folder and represent the different objects that make up the operation. The first five ([Machining Features](#), [Tool](#), [Feeds/Speeds](#), [Clearance Geometry](#), [Parameters](#), [Toolpath Viewer/Editor](#) and [G-Code Editor](#)) can be associatively edited.

 Machining Features

 Tool

 Feeds/Speeds

 Clearance Geometry

 Parameters

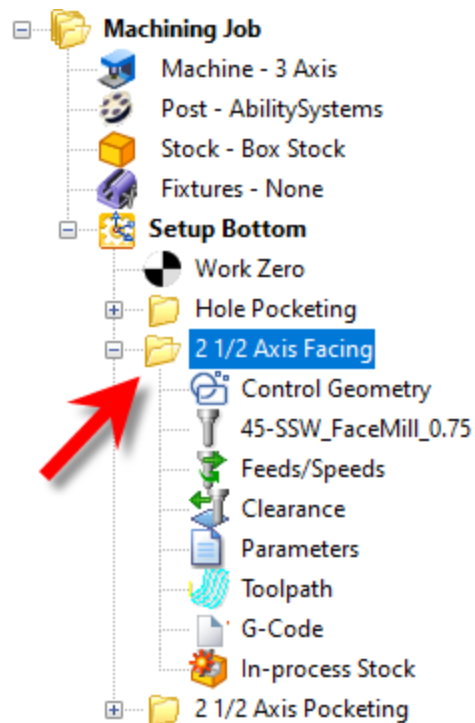
 Toolpath

 G-Code Editor

 In-process Stock

Double Click to Open a Machining Operations Dialog for Editing

[Double clicking](#) on the operation folder (or name) will open the operation's properties dialog with all tabs displayed for editing.



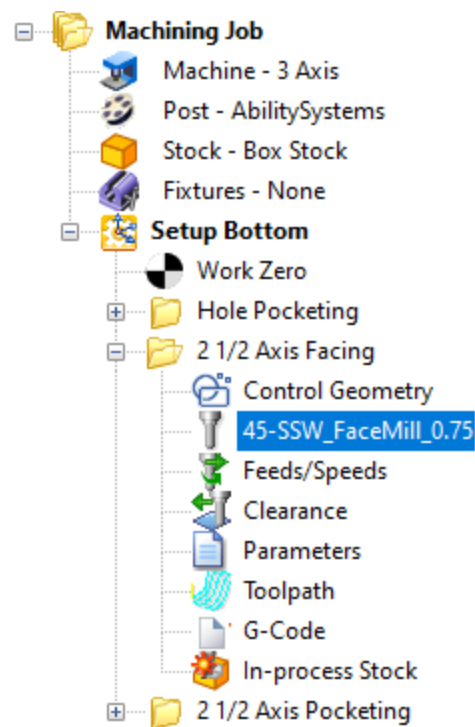
Double Click to Open the G-Code Editor module
 Note: MILL Module shown, Similar for TURN and Profile-NEST



Right or Double Click one of an Operation's Icons to Edit its Properties

Right mouse click or double clicking a specific icon, for example the **Tool** icon would bring up the **Tool Creation** dialog, upon which you can substitute the current tool with another or edit the parameters of the current tool.

Click on this Icon	Displays the Operation's
	Control Geometry tab
	Tool tab
	Feeds/Speeds tab
	Clearance tab
	Cut Parameters tab(s)
	Toolpath in the Toolpath Viewer
	Launch the G-Code Editor with the latest posted g-code loaded.
	If flagged, simulate the operation.



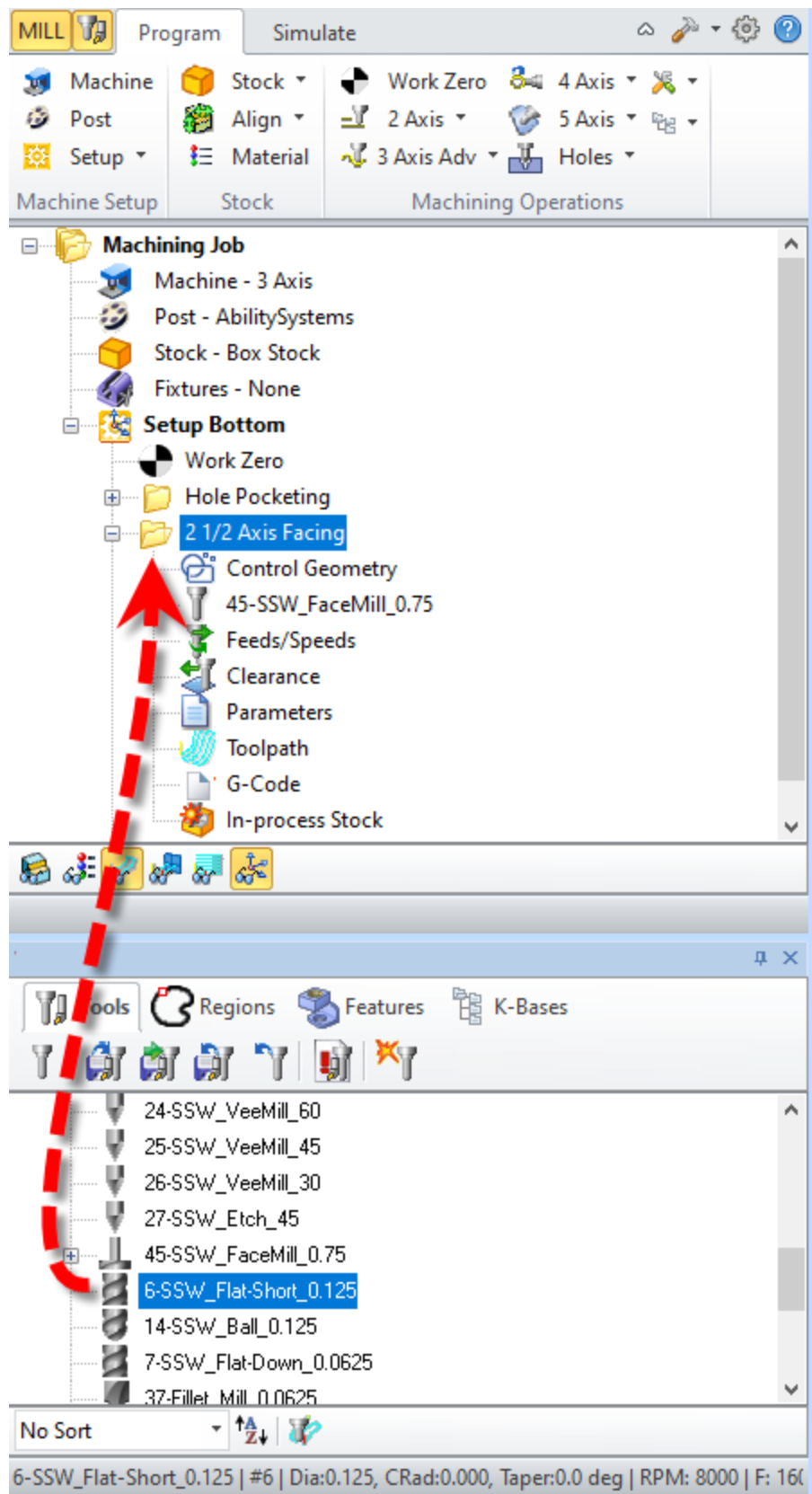
Right or Double Click one of an Operation's Icons to Edit its Properties

Note: MILL Module shown, Similar for TURN and Profile-NEST



Drag & Drop a Tool from the Object Browser to an Operation

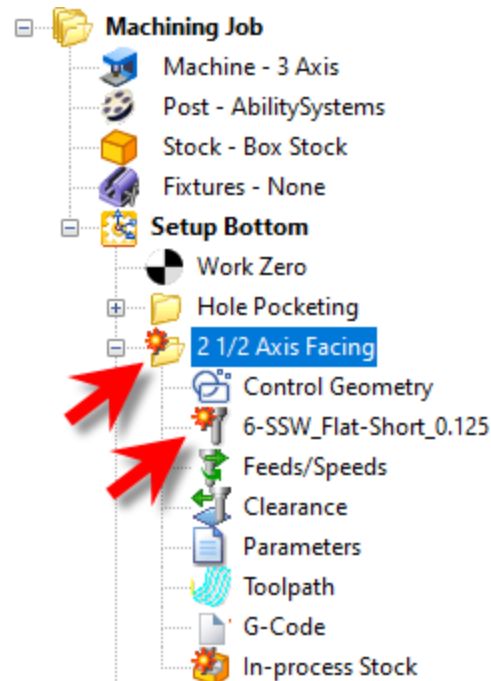
The tool can also be edited by dragging and dropping a tool from [Tools](#) tab to the [Machining Browser](#).



Display of Operations that need Regenerating after Editing

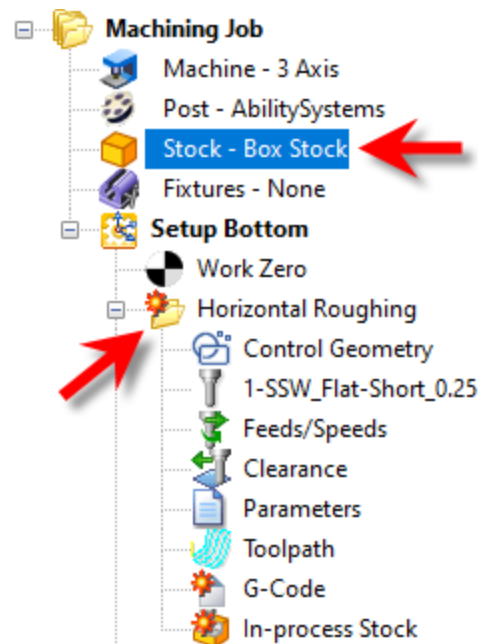
If any of the objects that make up the operation were to be edited after the toolpath was initially generated, the operation will be flagged dirty (i.e., needing regeneration). This condition is indicated by adding a red marker  to the operation folder. Also, the object that necessitated this condition  is also displayed with a red marker.

An example of this is shown below. In this case the tool used in the operation was edited after the machining operation was created and so is shown differently, as is the operation.



Display of Operations that need Regenerating after Editing
Note: MILL Module shown, Similar for TURN and Profile-NEST

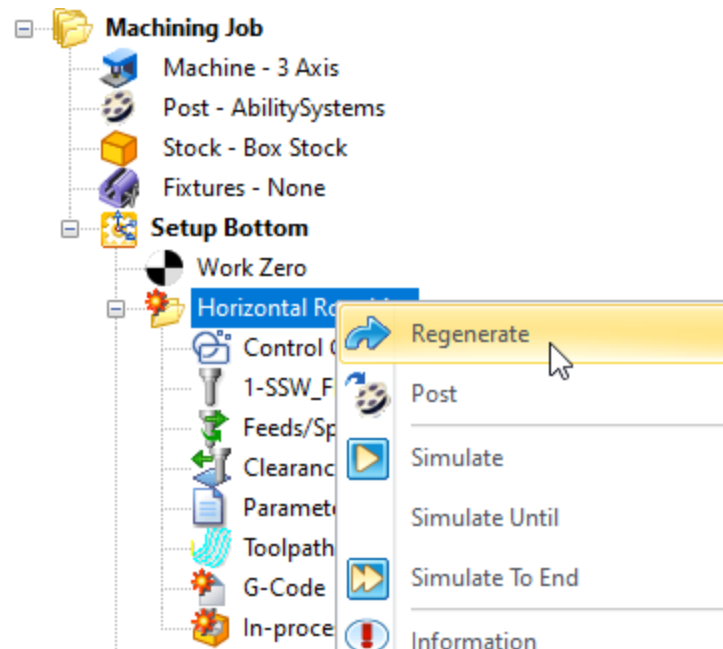
Mops will be flagged if any associated parameters outside of the operation are edited. For example, if the Stock is modified, any **Roughing** operations dependent on that **Stock** will be flagged for regeneration.



Note: MILL Module shown, Similar for TURN and Profile-NEST

Regenerating "Flagged" Machining Operations

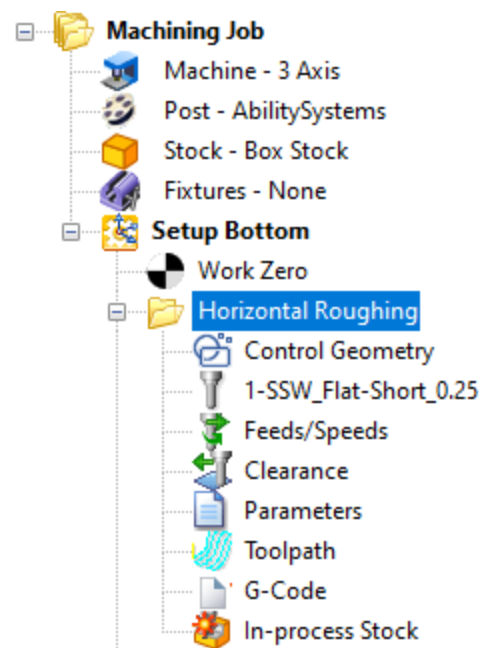
In order to regenerate the operation that is flagged with a red marker, you would have to select the operation, right click and select [Regenerate](#).



Right-click on an Operation and Select Regenerate

Note: MILL Module shown, Similar for TURN and Profile-NEST

The toolpath is now generated with the modified settings.



Machining Operation is Regenerated

Note: MILL Module shown, Similar for TURN and Profile-NEST

Enable Cutter Compensation

All toolpaths except engraving are automatically compensated for the tool geometry. [Cutter compensation](#) is used typically to compensate for the difference in the dimensions of the actual cutter used in machining and the cutter used for programming in [RhinoCAM](#). For example, if the cutter used in programming is 0.25 inches and due to tool wear the actual cutter is only 0.24 inches in size, you can compensate for this at the controller rather than having to re-program the operation in [RhinoCAM](#).

[Cutter compensation](#) is used extensively in production (high volume) machining where the machine operator can compensate for tool wear before having to stop and replace the tool or insert.

In order to do this the user needs to do the following:

1. Turn cutter compensation on in the operation to [Auto/ON](#) or [CONTROL/ON](#).
2. Specify the cutter compensation value and the compensation register in the controller (the controller needs to be capable of doing this).
3. Please make sure the post processor is configured to output cutter compensation. This is defined under the [Cutter Compensation](#) section in the post processor generator. Most controllers expect an X & Y motion on the same line as cutter compensation.

Cutter Compensation Left

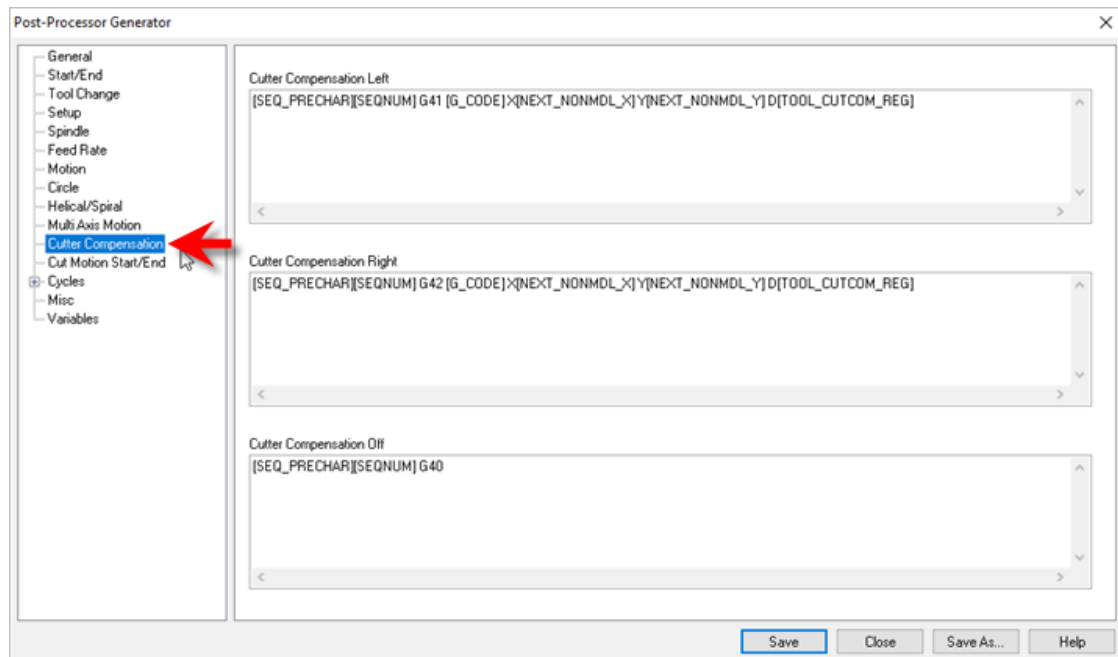
[\[SEQ_PRECHAR\]\[SEQNUM\]](#) G41 [\[G_CODE\]](#) X[\[NEXT_NONMDL_X\]](#) Y[\[NEXT_NONMDL_Y\]](#) D[\[TOOL_CUTCOM_REG\]](#)

Cutter Compensation Right

[\[SEQ_PRECHAR\]\[SEQNUM\]](#) G42 [\[G_CODE\]](#) X[\[NEXT_NONMDL_X\]](#) Y[\[NEXT_NONMDL_Y\]](#) D[\[TOOL_CUTCOM_REG\]](#)

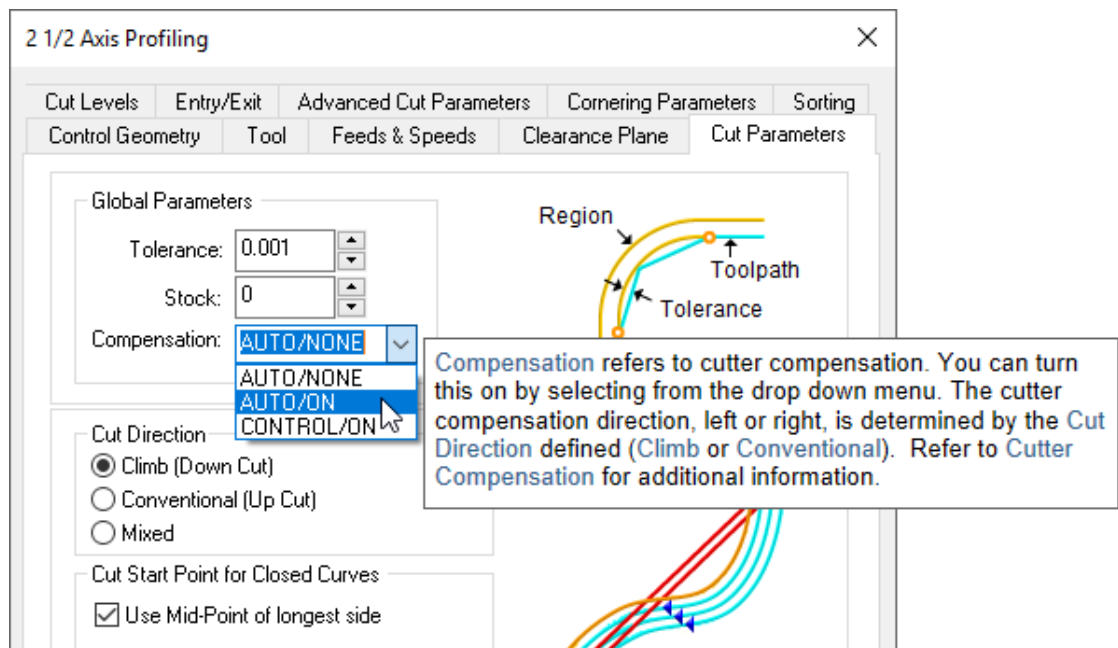
Cutter Compensation Off

[\[SEQ_PRECHAR\]\[SEQNUM\]](#) G40



A few things to watch out for:

1. **Cutter compensation** makes sense only in 2-1/2 axis operations. If you are using roughing (pocketing & facing) the compensation will be turned on only in the final passes.
2. Make sure you are using **Climb** or **Conventional** cut traversal in any of the methods that you want to turn compensation on.

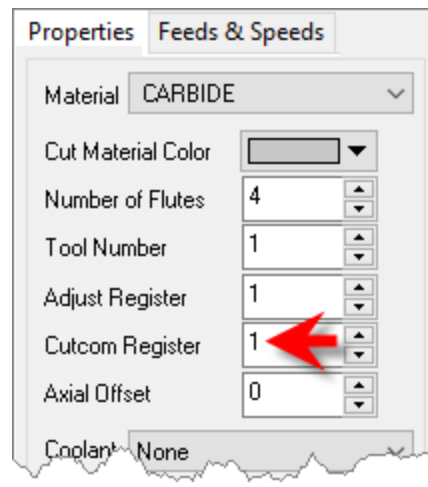


3. Make sure you have a linear motion for the controller to turn on the compensation for. If your first motion is an arc the controller will not be able to turn on the compensation.

Thus, in [2-1/2 axis profiling](#), make sure there is a linear entry motion for the controller to be able to turn compensation on & linear exit to turn off compensation.

If you are looking to compensate for the full tool diameter, set [Stock](#) = [-0.125](#) under the cut parameters tab. ([0.125](#) being the radius of the tool). This would generate the toolpath ON the curve. This would invalidate the simulation as the tool tip stays on the drive geometry.

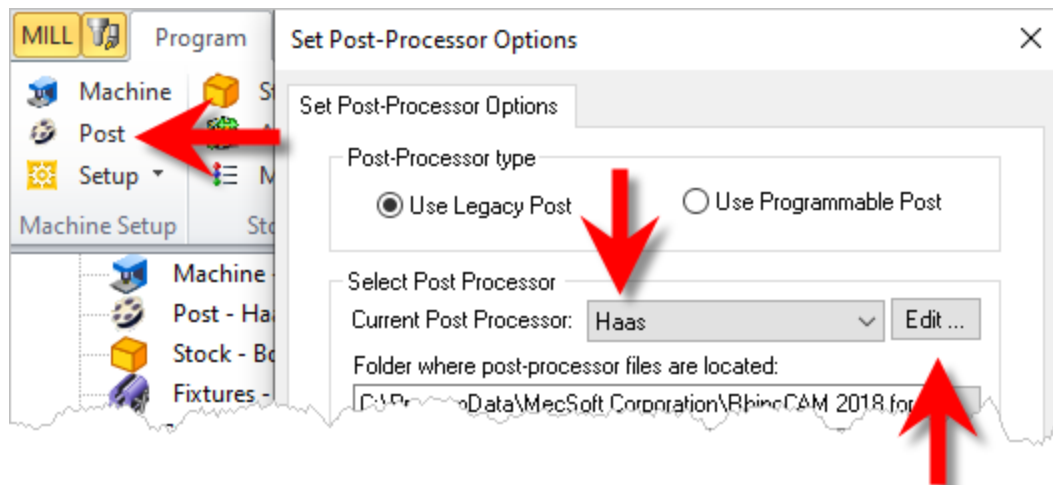
Note: The [Cutcom Register](#) is set under the [Create/Select Tool](#) definition dialog.



Enable Inverse Time Feedrate in 4 Axis

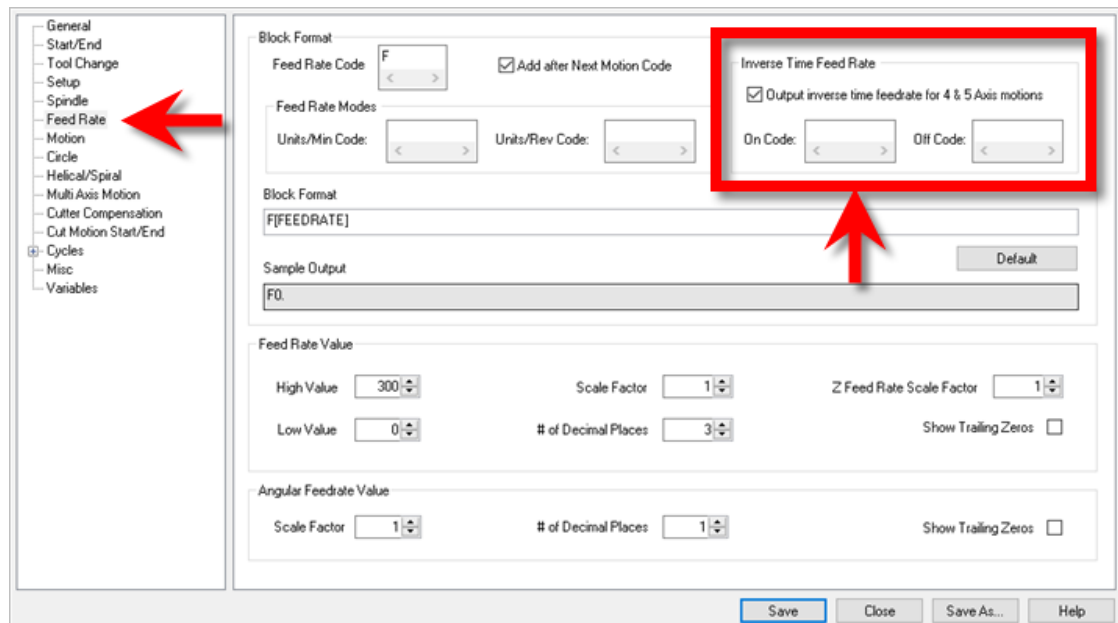
You do not need to edit your toolpath operations to output the **Inverse Time Feedrate** format! It is done automatically once it is enabled in your post processor. Follow these steps to enable the **Inverse Time Feedrate** format in your post:

1. From the **Program** tab under **Machine Setup**, select the **Post** icon to display the **Set Post-Processor Options** dialog.
2. For **Current Post Processor**, select your post.
3. Pick the **Edit** button to the right. This will display the **Post-Process Generator** dialog.



Launch the Post-Process Generator

4. From the left side pane, select the **Feedrate** section.



Post-Process Generator / Feedrate / Inverse Time

5. On the right side pane, under **Block Format**, check the box next to "Output inverse time feedrate for 4 & 5 Axis motions".
6. For **On Code**, enter **G93**.
7. For **Off Code**, enter **G94**.
8. Pick **Save As** to save your (*.spm) post.

 Anytime you make an edit to your post (*.spm) file, it is recommended that you select **Save As** to save your post file to a unique name and to a unique folder where you can find it (such as your **Desktop** or shared **Network Drive**).

9. With your revised post set as your **Current Post Processor**, pick **OK** to close the **Set Post-Processor Options** dialog.

 If you do not see your revised post in the **Current Post Processor** list, select the "... " button to the right of "**Folder where post processor file are located**" and navigate to the folder where you save your post (*.spm) file. It should then appear in the list.

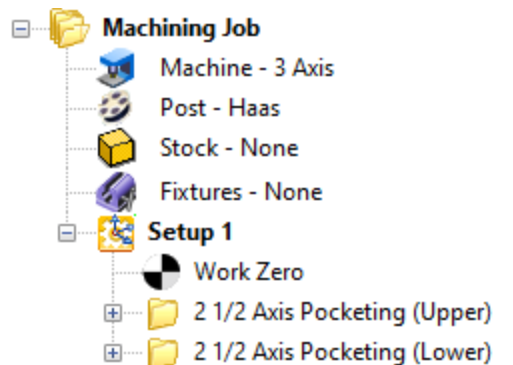
10. Test the revised post with a sample **4** or **5 Axis** toolpath operation.

Estimate Machining Time

You can calculate the estimated machining time for one or more toolpath operations. The time is calculated based on the [Cut Feed](#) and [Transfer Feed](#) rates assigned to each operation. Understand that the machining time is an estimate. It does not take into account any physical or limiting factors of your CNC machine or controller settings. See our blog article [Optimize Machining Time Estimates!](#) for additional reading.

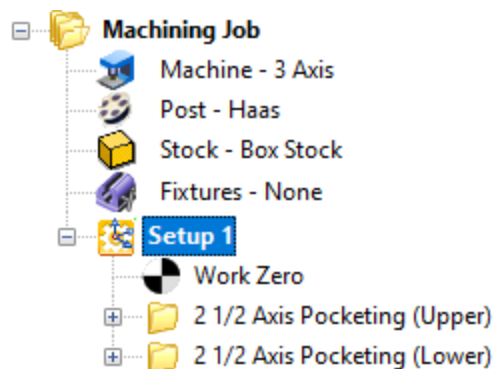
To calculate the estimated machining time:

1. Create and adjust the toolpath operations that you want to estimate.
2. Make sure the toolpath has generated cleanly. Each toolpath when generated is listed under a [Setup](#) in the [Machining Job](#). If the operation is flagged it means that it needs to be regenerated.



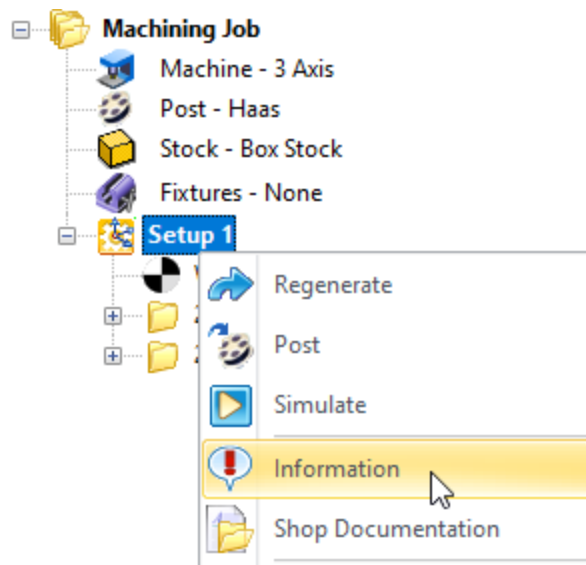
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

3. Select the operation that you want to estimate. You can select multiple operations by pressing the **<Ctrl>** key while selecting. You can also select an entire [Setup](#) or the entire [Machining Job](#).



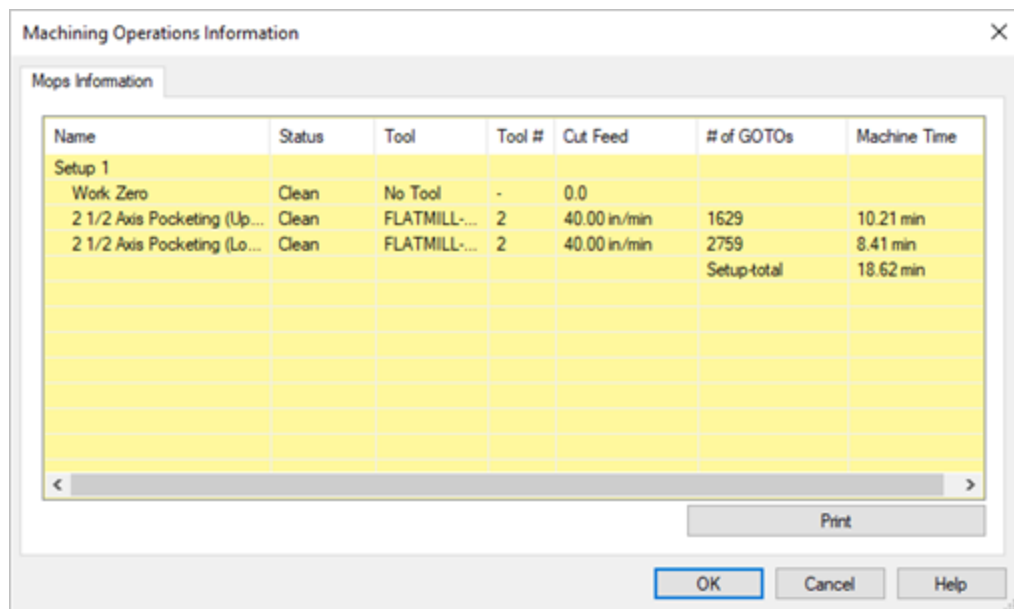
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

4. Right-click on the selected operation(s) and select [Information](#) to display the dialog.



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

5. Each selected operation is listed. The estimated machining time is shown in the right side column along with the total time for all listed operations.



6. Select the [Print](#) button to print the report if desired.
7. Pick [OK](#) to close the information dialog.

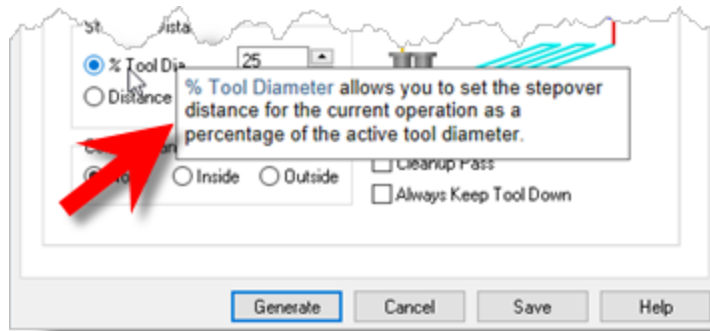
Find Tool Related Preferences

Listed below are the [CAM Preferences](#) that are related to [Tools](#) and [Tool Libraries](#):

1. Locate the [CAM Preferences](#) icon to the right of the [Program](#) tab and select it.
2. Select the [User Interface](#) item from the left. Here are a couple of [Preferences](#) that will help you with the tools related dialogs:

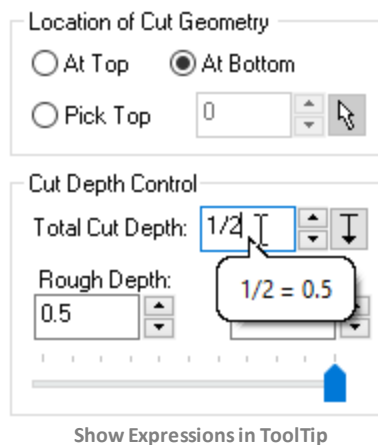
Show context ToolTips

Check this box to display [Context ToolTips](#) when the mouse moves over a parameter in a dialog. A definition of the parameter will pop-up automatically. **Note** that [Context ToolTips](#) may not be available for ALL dialogs. You can also set the [ToolTip Delay](#) in seconds. This is the amount of time it takes to display the [Context ToolTip](#) when the mouse activate it.

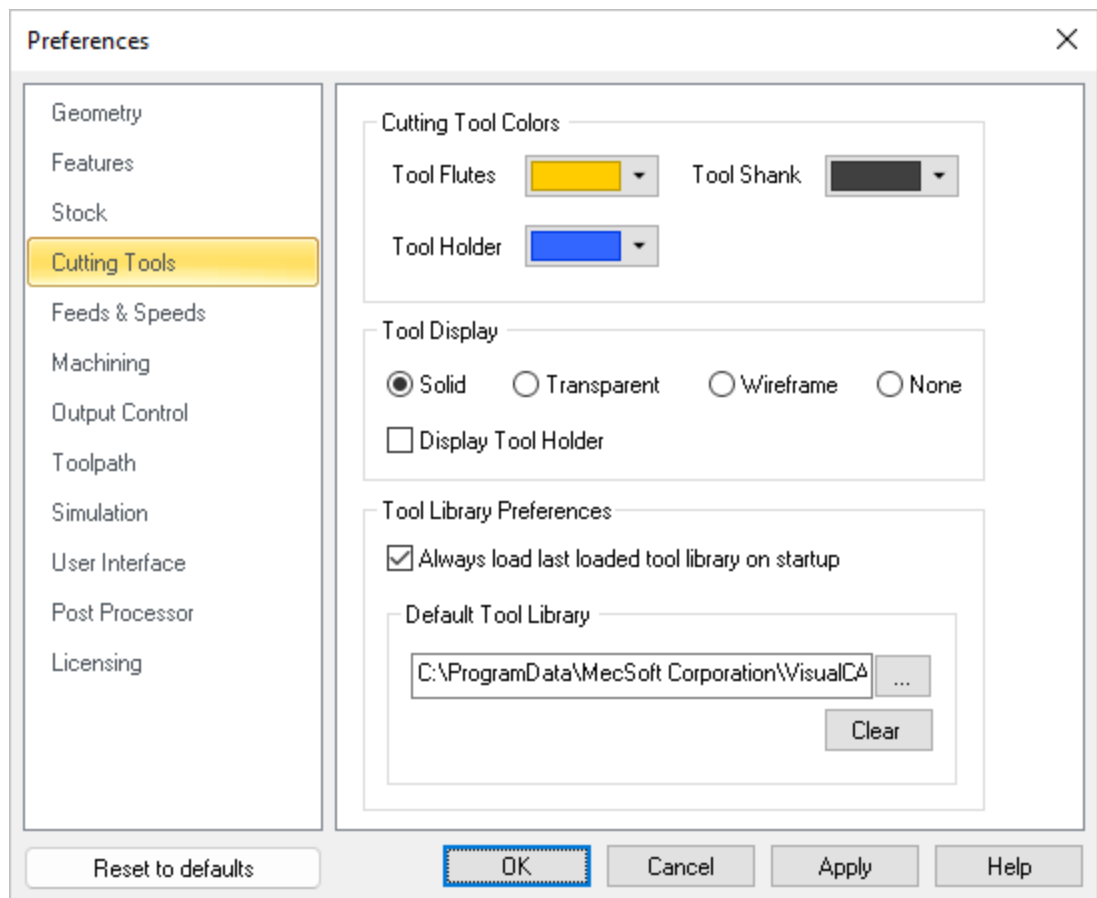


Show Expressions in Tooltip

You can enter expressions in any dialog field that expects a numerical value and the value will be computed and entered automatically. Check this box to pop-up the results of any expressions in a [ToolTip](#) balloon. An example is shown below.



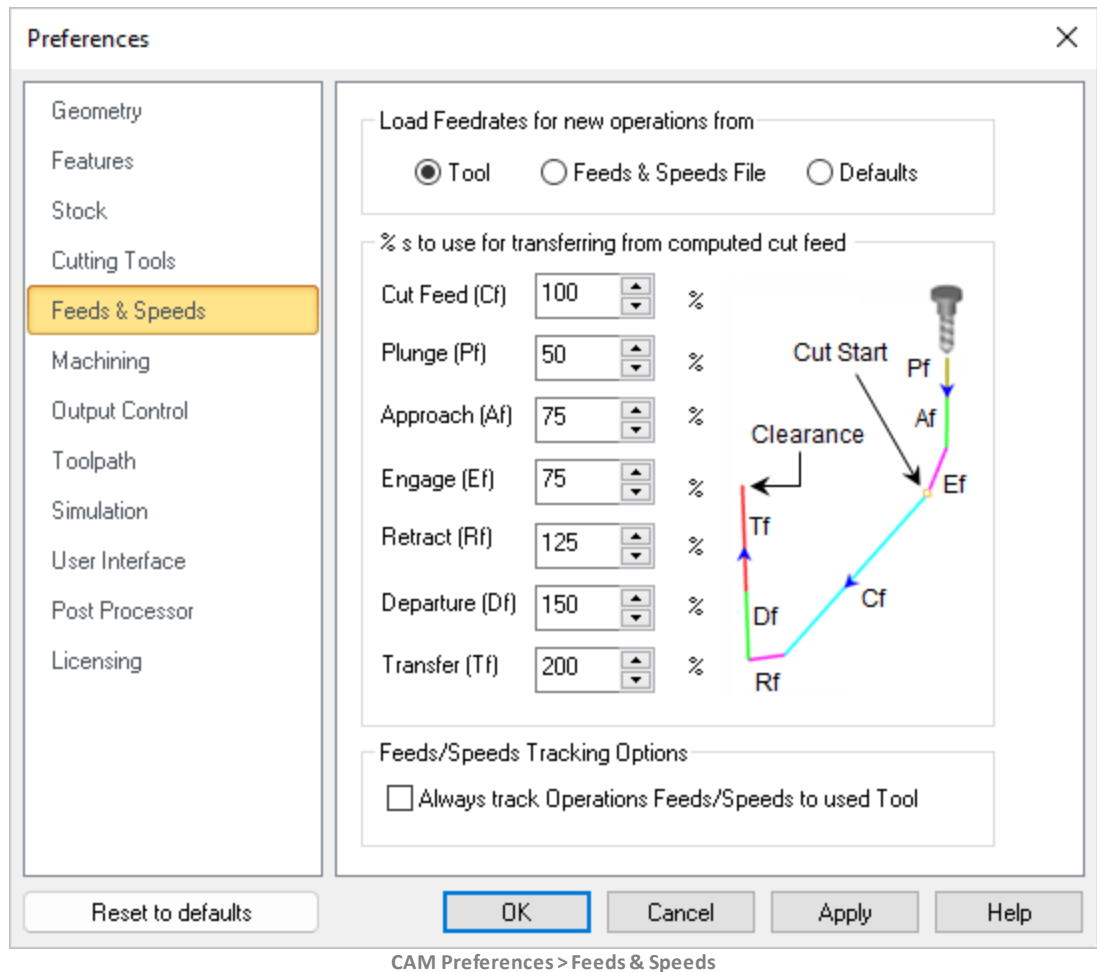
3. Select the [Cutting Tools](#) item from the left.



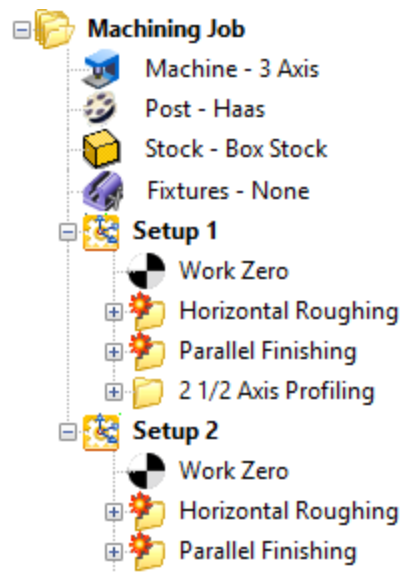
CAM Preferences > Cutting Tools

Note: Menu selections on the left may change depending on module and configuration

4. You can select the [Default Tool Library](#) to load for new part files. You can also check the box to [Load the last loaded tool library on startup](#). This will ensure that your [Tool Library](#) loads every time the program runs.
5. You can also set [Tool](#) related colors on this dialog.
6. Now select the [Feeds & Speeds](#) section from the left.
7. Here you can decide if you want default [Feeds & Speeds](#) loaded from the [Tool](#) for new operations. If you set this to [Tool](#) and define your [Speeds & Feeds](#) for each of your tools, you can be sure those [Feeds & Speeds](#) are being used when a new operation is created that uses that tool.



8. You can also set the % of the computed **Cut Feed** to use for the various types of transfer motions. For example, in the **Feeds & Speeds Calculator** (displayed when you select **Load from File** from either the **Create/Select Tools** dialog or from the **Feeds & Speeds** tab of any operation type) a **Cut Feed** value is calculated and suggested. These percentages listed in this **CAM Preferences** dialog will determine how much of that **Cut Feed** value is used for each of they remaining tool motion types.
9. You can also check the box to **Always track Operations Feeds/Speeds to used Tool**. When this is checked, changing the feeds/speeds parameters of a tool and saving the edits, will automatically update the feeds/speeds on each operation that utilizes this tool in addition to marking the operations as dirty.



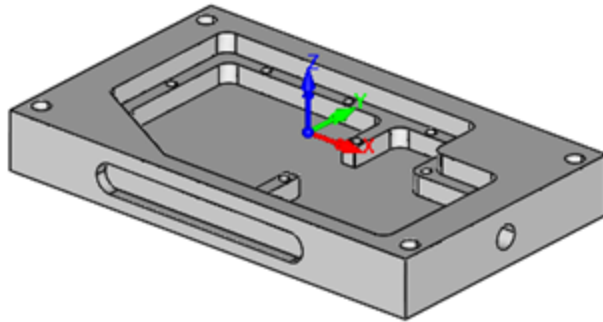
Generate a Toolpath

RhinoCAM includes many toolpath operation types to choose from depending on your software configuration. Also each operation type has default parameters predefined to allow you to quickly create a toolpath and view the results.

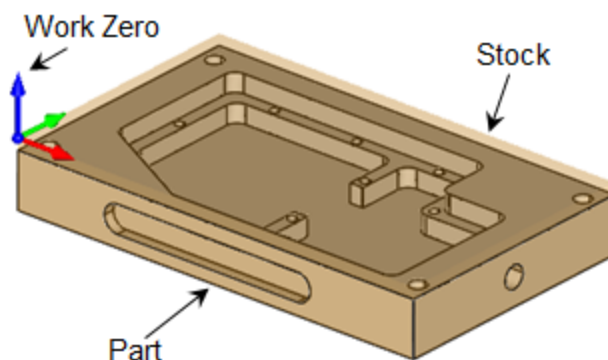
Here are the basic steps to create a toolpath:

1. Open a part or drawing file or create the geometry that you wish to create a toolpath for. **2 Axis** operations can be created with 2D or 3D geometry. **3 Axis** operations require surface and/or mesh geometry. Some **4 Axis** operations require 3D solids or surfaces only. All and **5 Axis** operations require 3D solids or surfaces only. You must have visible geometry in your part file to create a toolpath.

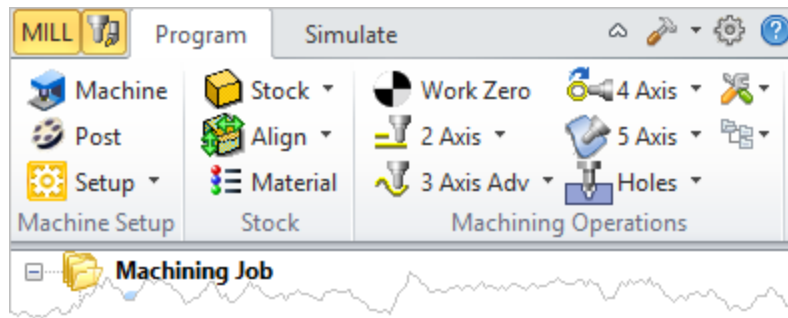
In this example we will generate a **2 Axis Pocketing** toolpath to machine the bottom pocket in this 3D solid model.



2. Perform the steps needed to setup your part. These include defining your **Machine**, **Post** and **Stock**. You must first have geometry and **Stock** defined to create a toolpath.

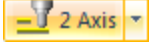


3. Select the **Program** tab.

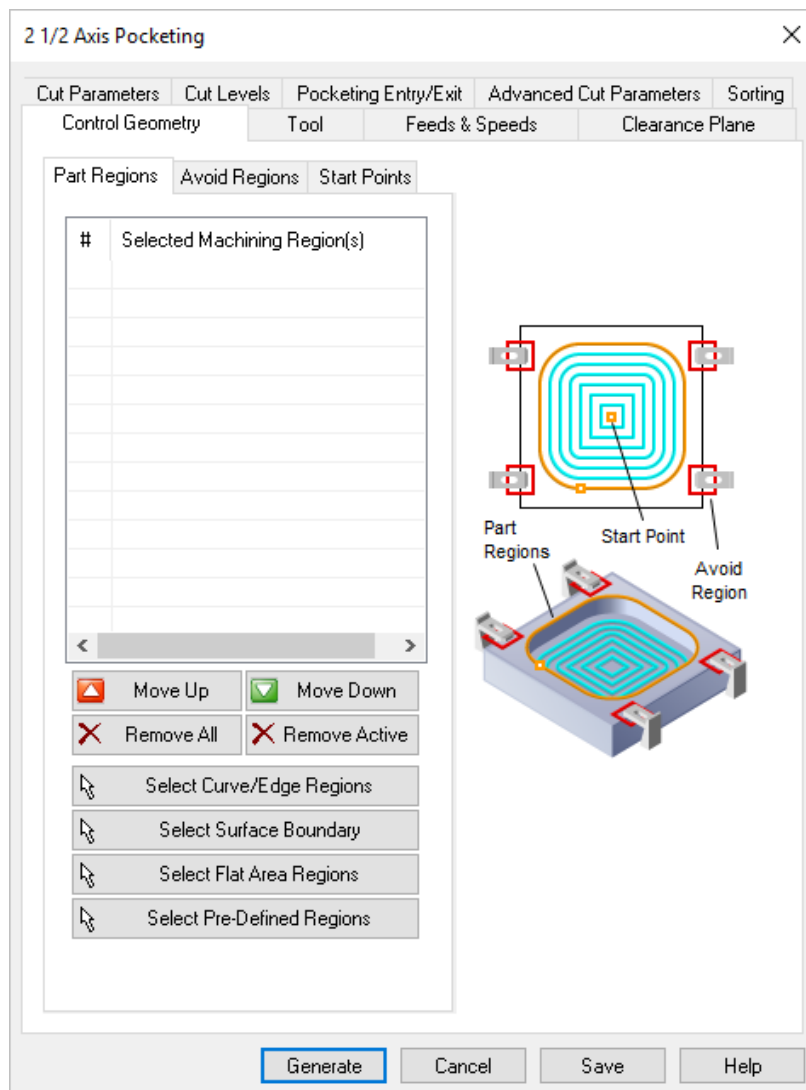


MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

4. Decide what type of toolpath to create and select the operation type from the menus on the **Program** tab. Depending on your software configuration, you can select from the **2 Axis**, **3 Axis**, **4 Axis** or **5 Axis** menus. The operation dialog for that toolpath type will display.

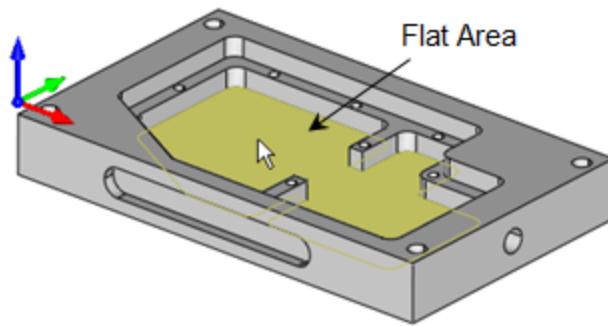
5.  For this example we will generate a **2 Axis Pocketing** toolpath so select the **2 Axis** menu.

6.  Select **Pocketing** from the menu to display the dialog.

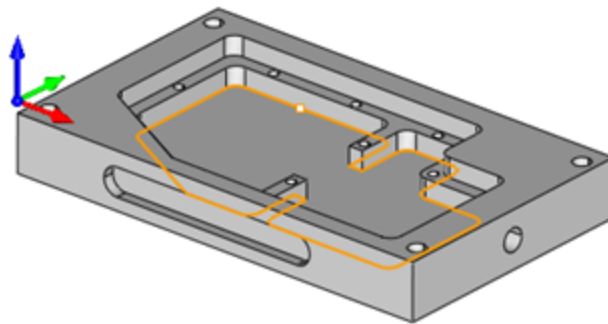
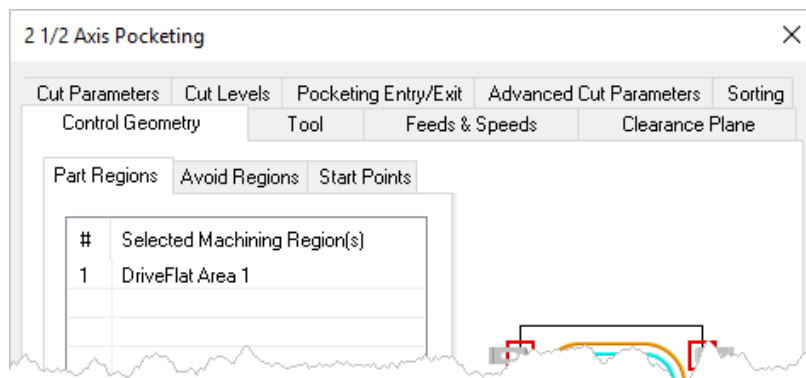


7. The dialog has several tabs at the top. The **Control Geometry** tab is displayed first. Use it to select your machining regions. All **2 Axis** operations require that you select machining regions. In **3 Axis** and **4 Axis** operations machining regions are used to if you want to contain the toolpath but are not required for all 3 and **4 Axis** operations. **5 Axis** operations do require machining regions.
8. The buttons on this tab are used to select your machining regions. Here we will pick the **Select Flat Area Regions** button.

9. The dialog minimizes and you are prompted to select a flat area. We select the bottom of the inner pocket and then right-click or press **<Enter>**.

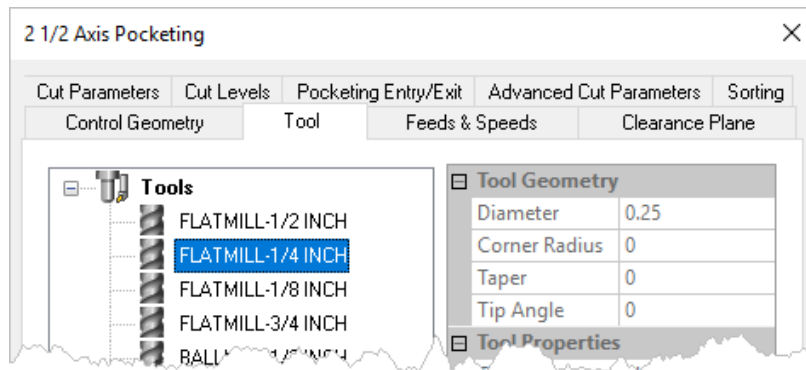


10. The dialog reappears with the selected machining regions listed. They are also highlighted on the part.

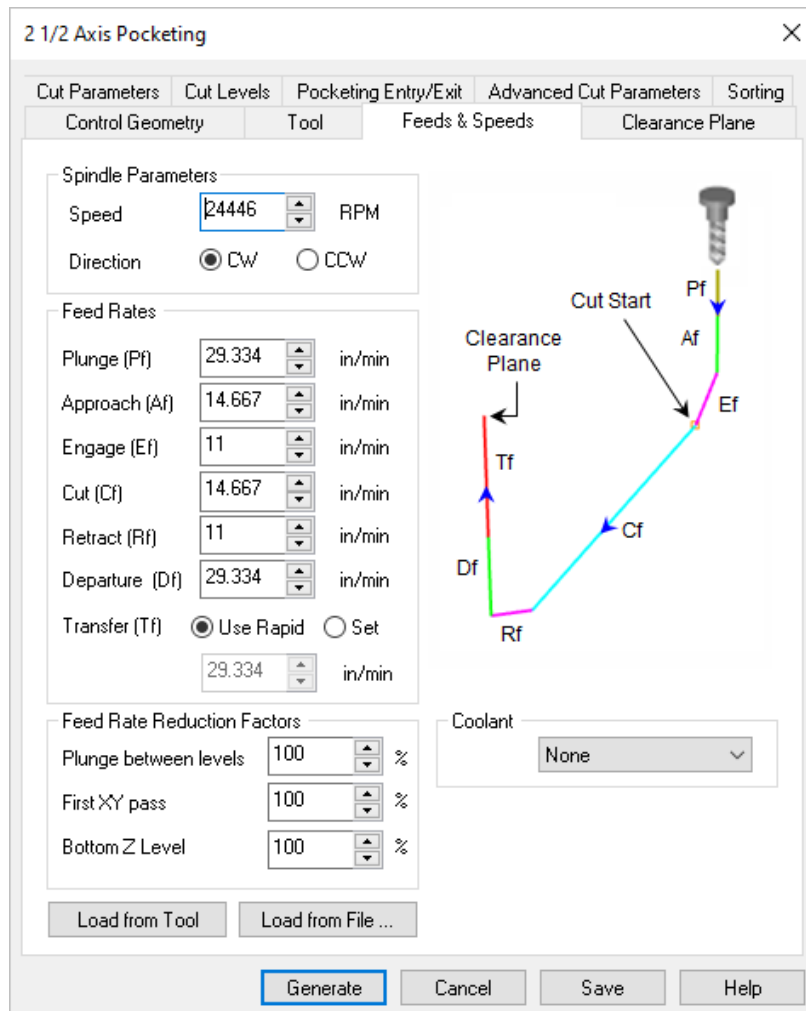


There are other ways to define and select machining regions. You can pick Help from the dialog to learn about each button option.

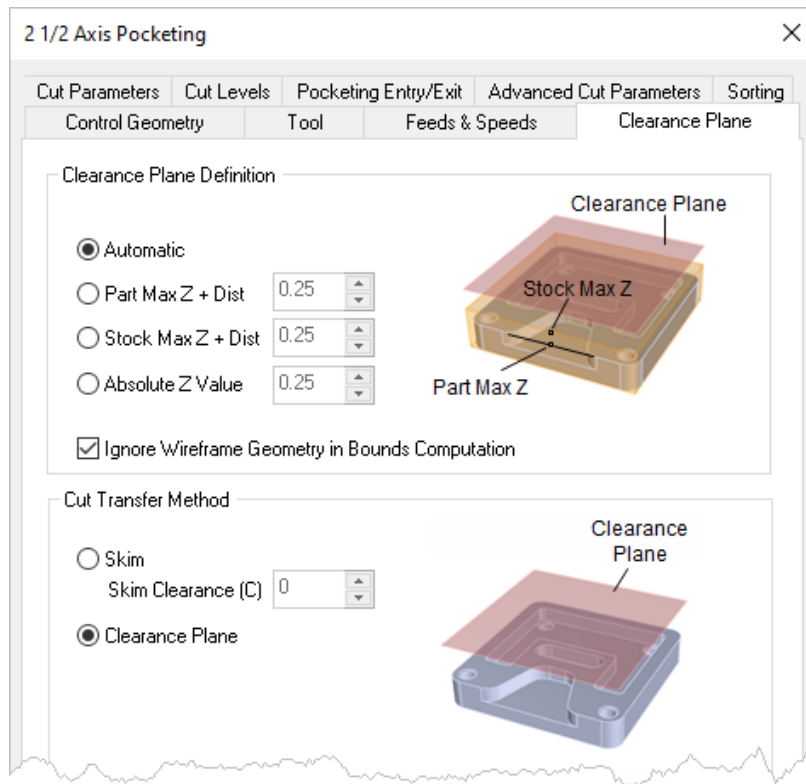
11. Select the **Tools** tab and select a tool from the **Tools** list or select the **Edit/Create/Select Tool ...** button to create a tool. You must select a tool to create a toolpath.



12. Select the **Feeds & Speeds** tab and set the **Spindle Speed** and **Feed Rates** for the operation and use the **Load from Tool ...** or **Load from File ...** buttons. The **Load from File ...** button displays the **Feeds & Speeds Calculator** to assist you with selecting feeds and speeds values for the operation.

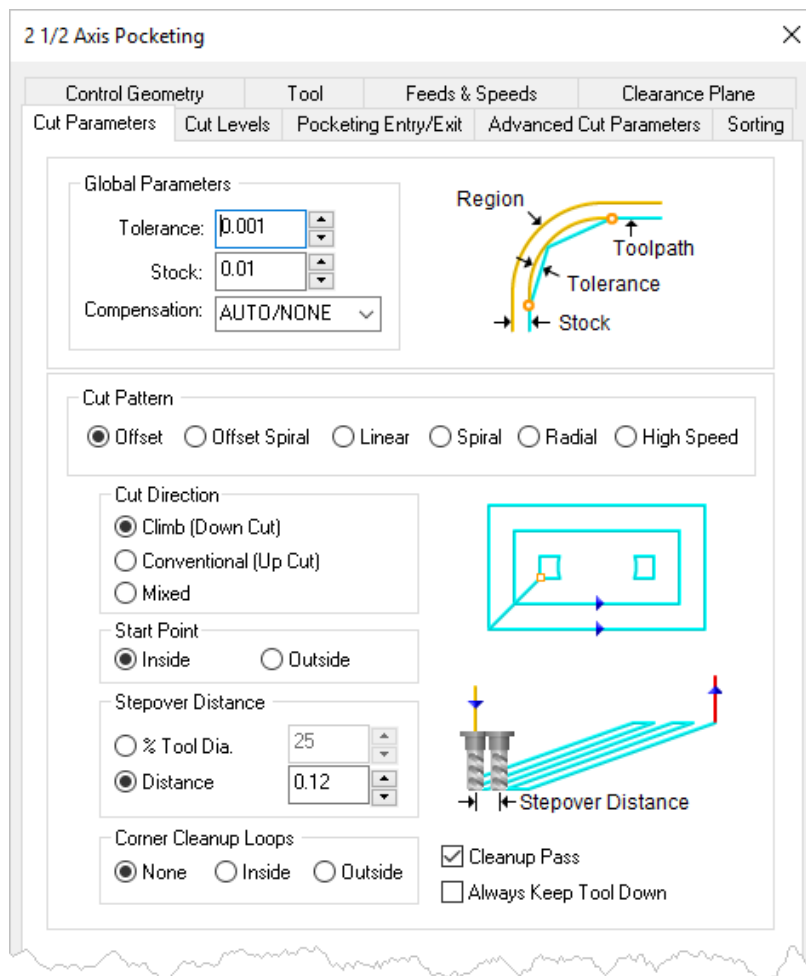


13. Select the **Clearance Plane** tab and adjust if desired.

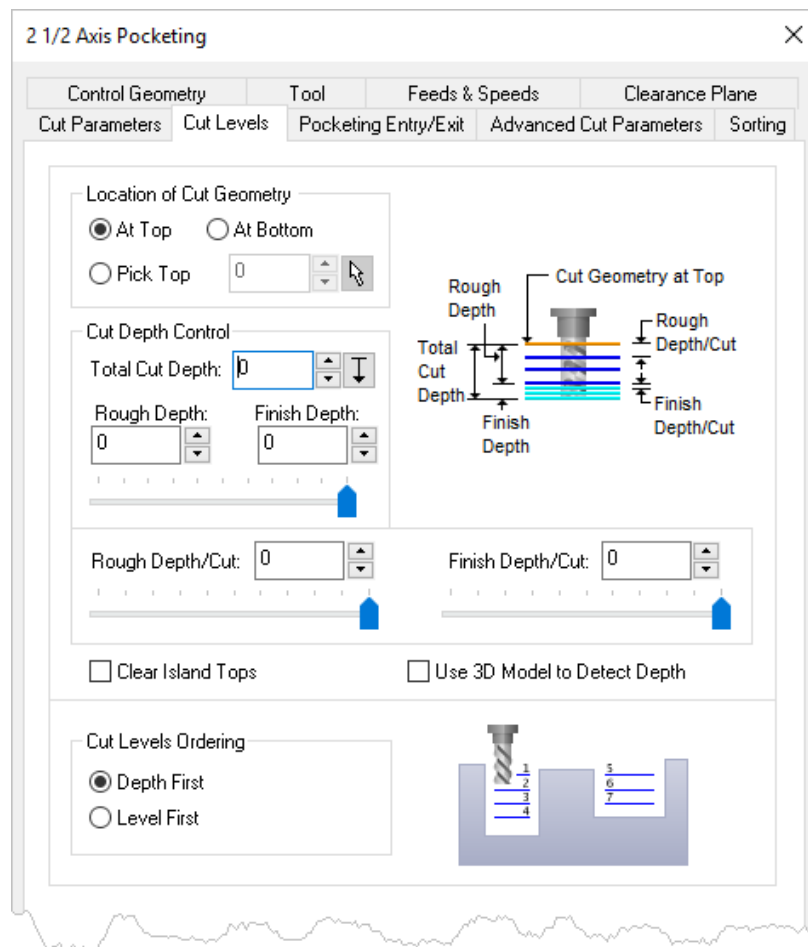


14. Now we will select the [Cut Parameters](#) tab of the dialog and make the following adjustments. Stock will leave 0.01 remaining on the side walls. The cut pattern will offset the perimeter of the pocket. The cut will start in the middle of the pocket, offset outward at a stepover 0.01 and perform a cleanup pass at each cut level.

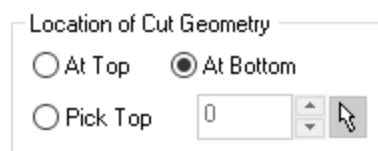
[Tolerance](#) : 0.001
[Stock](#) : 0.01
[Cut Pattern](#) : Offset
[Cut Direction](#) : Climb
[Start Point](#): Inside
[Stepover Distance](#) : 0.12
[Corner Cleanup Loops](#) : None
[Cleanup Pass](#) : Checked



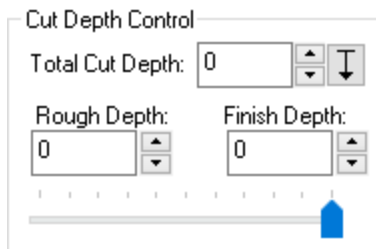
15. Next we select the **Cut Levels** tab. All **2 Axis** operations require your to adjust the **Cut Levels** tab to control the depth of cut. In all **3, 4 and 5 Axis** operations your part surfaces will limit the maximum depth of cut.



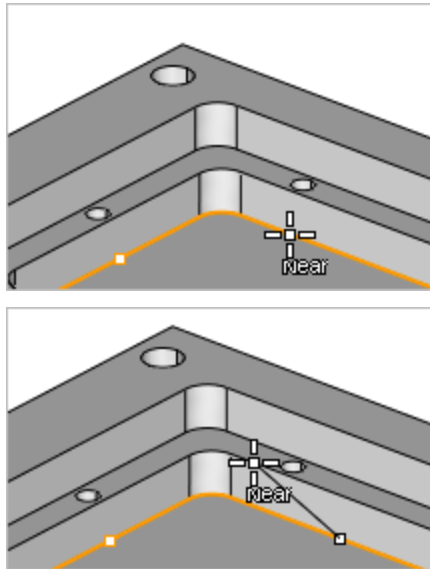
16. First make a selection to define the [Location of Cut Geometry](#). In our example we select [At Bottom](#) because the machining regions we selected lie at the bottom of the pocket. If they lie at the top we would select [At Top](#). If they do not lie either at the top or the bottom of the pocket you can use the [Pick Top](#) option to define the top of the cut.



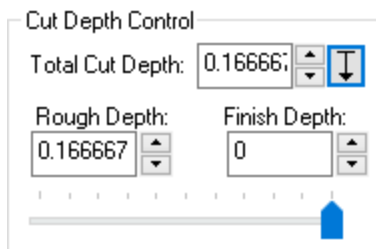
17. Next we move to the [Cut Depth Control](#) section. If you know how deep the pocket should be just enter the depth in the [Total Cut Depth](#) field. This is always an absolute value (i.e., no negative numbers).



18.  In our case we do not know the exact depth without manually measuring the geometry so we will select the [Pick](#) button.
19. The dialog minimizes and prompts us to select a start point and an end point to define the [Total Cut Depth](#). We select the top and bottom floor of the pocket and then right-click or press [Enter](#).

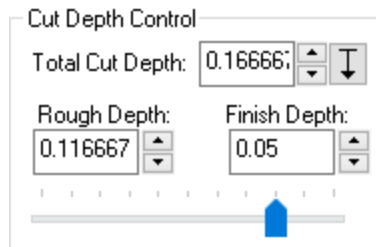


20. The dialog reappears with the Depth of Cut calculated for us.

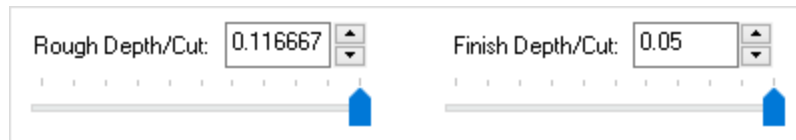


21. Notice that the value in the [Rough Depth](#) field is now the same as the [Total Cut Depth](#). You have the option to split the [Total Cut Depth](#) into a [Rough Depth](#) and a [Finish Depth](#). We will enter [0.05](#) into the Finish Depth field and then press the [Tab](#) key (or simply activate another field).

22. We see that the **Rough Depth** value is now reduced by 0.05. The **Rough Depth** and **Finish Depth** values will always equal the **Total Cut Depth** value. You can also use the slider to split these values up.



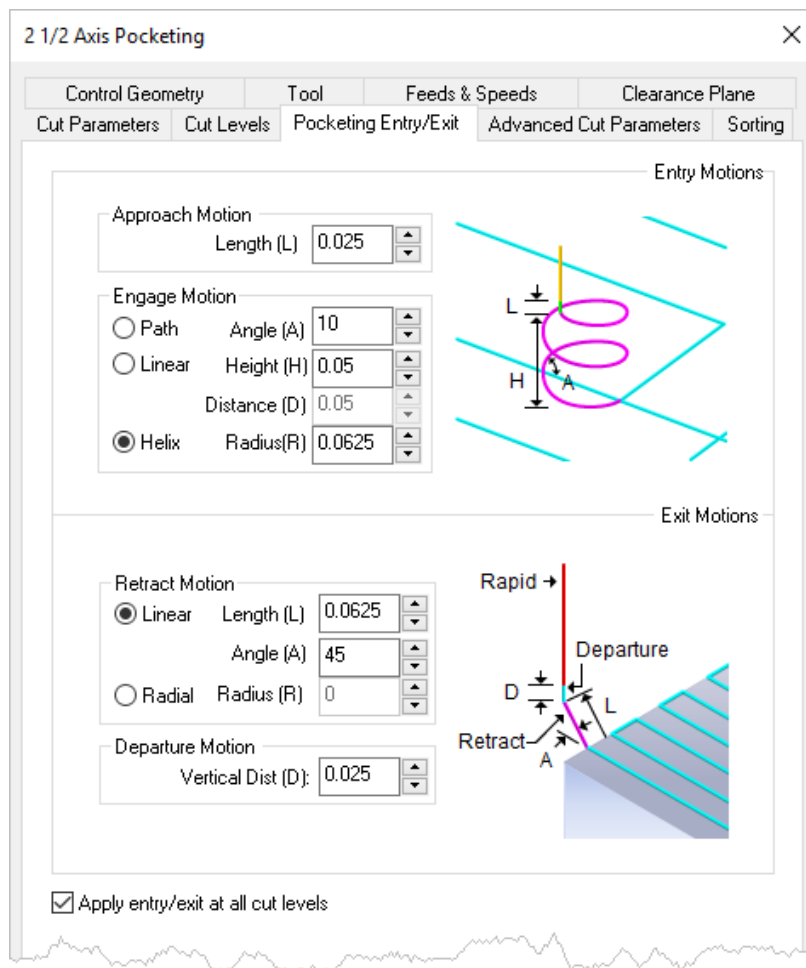
23. You will also notice now that the **Rough Depth/Cut** and **Finish Depth/Cut** fields are populated with the same values. If we generate the pocket toolpath right now, there would be one rough level cut and one finish level cut.



24. We can add additional cut levels by entering 0.03 in the **Rough Depth/Cut** field and entering 0.01 in the **Finish Depth/Cut** field. This will produce 4 rough cut levels at the top of the pocket and 5 finish cut levels at the bottom of the pocket.



25. With our cut levels defined we select the **Pocket Entry/Exit** tab of the dialog.



26. Now we select the **Pocketing Entry/Exit** tab to control how the cutter will enter and exit the part and make the following selections:

Approach Motion Length (L) : Default (0.025)

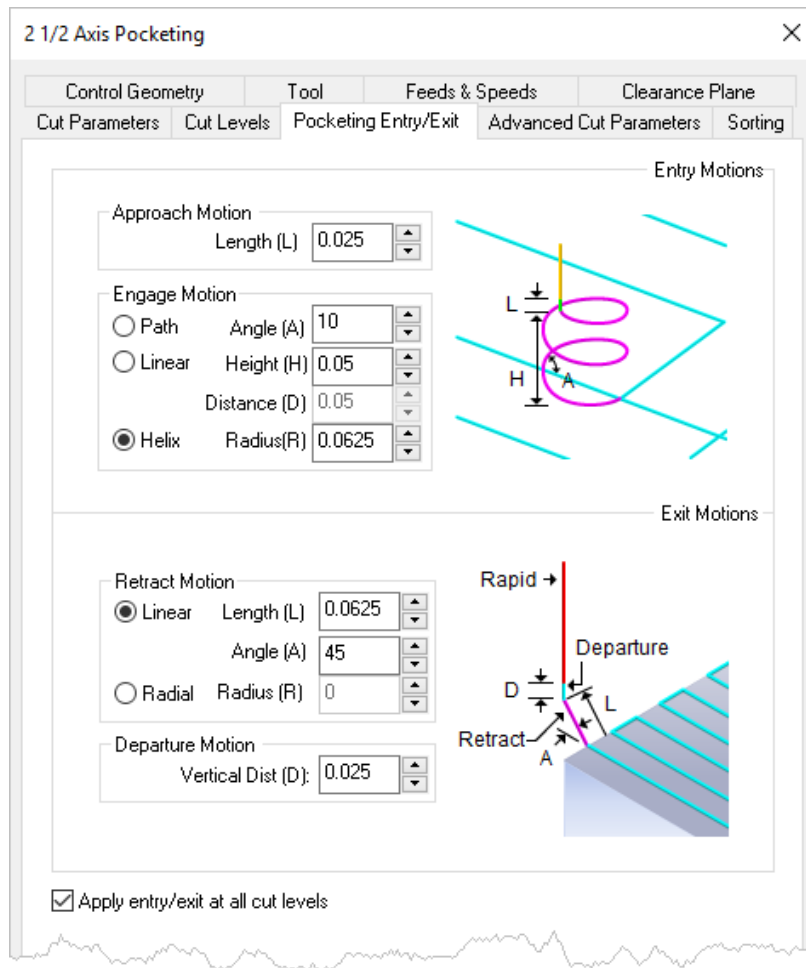
Engage Motion : Helix

Helix Radius (R): Default (0.0625)

Retract Motion : Linear, Length: 0.0625, Angle (A): 45

Departure Motion : Vertical Distance (D): Default (0.025)

Apply entry/exit at each cut level : Checked

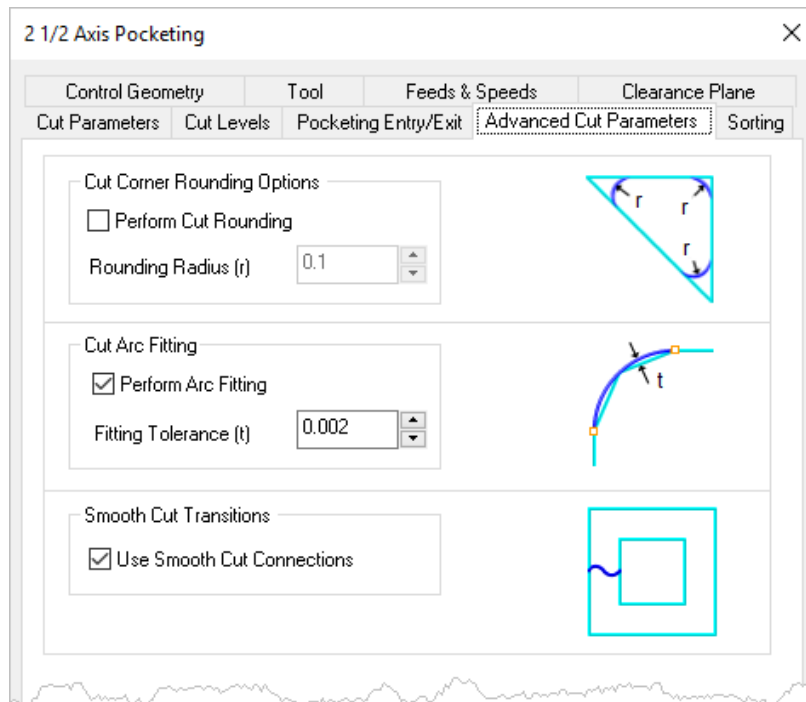


27. Now we will select the [Advanced Cut Parameters](#) tab and make the following selections. The parameters available on this tab will depend of the type of toolpath operation and your software configuration.

[Perform Arc Fitting](#) : Checked

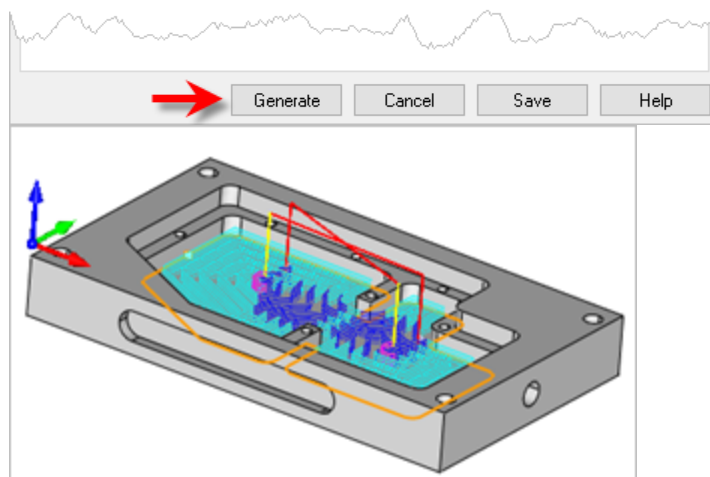
[Fitting Tolerance \(t\)](#) : 0.002

[Use Smooth Cut Connections](#) : Checked

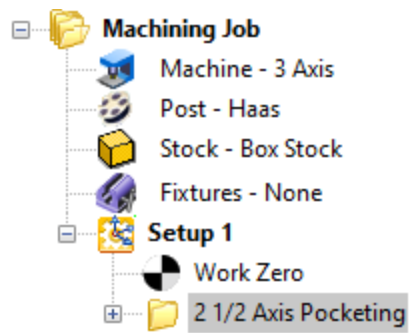


With these selections full and partial arc motions will be generated within the Fitting Tolerance (t) we have specified. There will also be smooth cut connections between stepover paths. These advanced parameters will reduce the torque stress on your CNC machine and also reduce tool deflection, prolonging tool life.

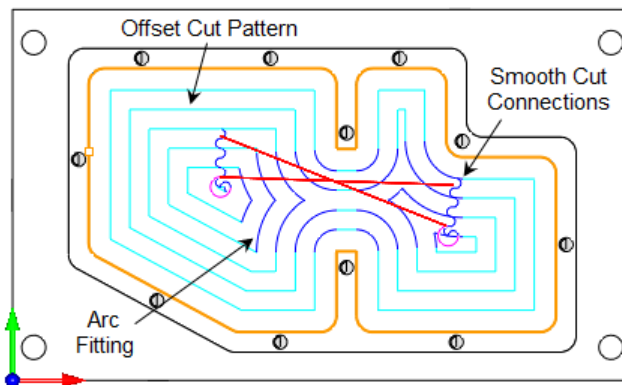
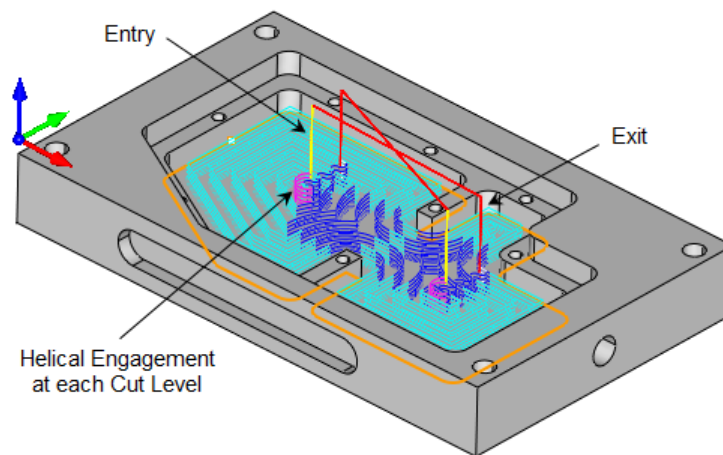
28. Since we only have one pocket we do not need to look at the [Sorting](#) tab.
29. Now were ready! Select the [Generate](#) button at the bottom of the dialog to generate the [2 Axis Pocketing](#) toolpath and display it on the screen.

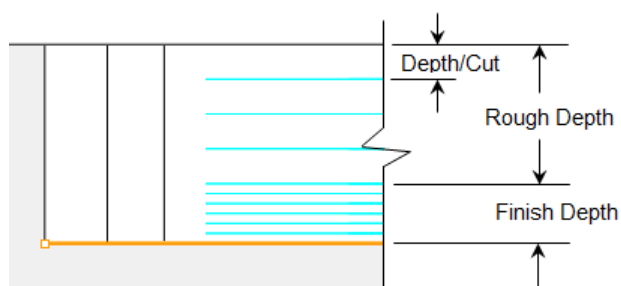


30. It is also listed in the Machining Job.



31.  If you do not see the toolpath displayed, select the [Toggle Toolpath Visibility](#) icon located at the bottom of the [Machining Browser](#).
32. Let's have a closer look at the toolpath.

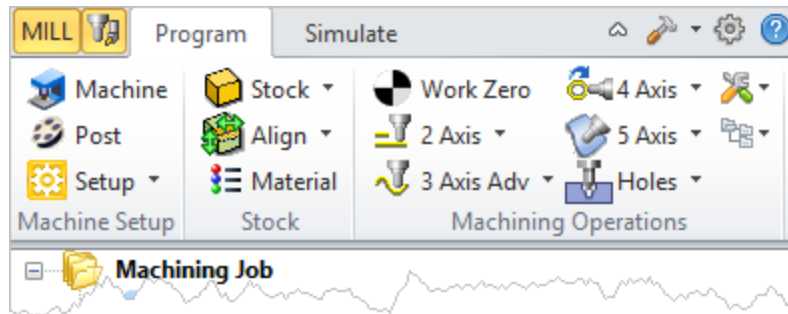




Load a Tool Library

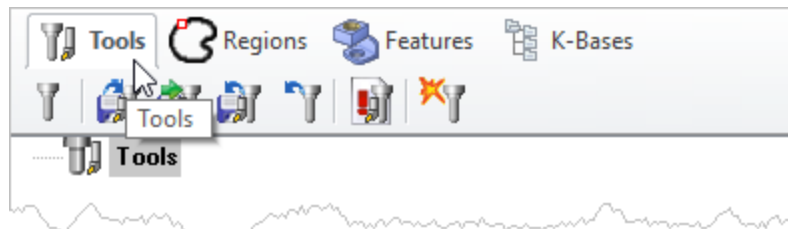
You can have multiple tool libraries saved. This is useful when archiving tools by type or by stock material, etc. You can then load a tool library at any time.

1.  To the left of the [Program](#) tab, select the [Tools Machining Objects](#) icon to make sure the [Machining Objects Browser](#) is displayed.



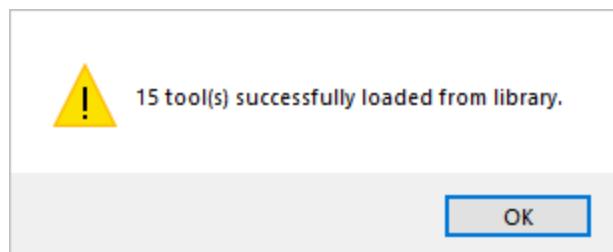
MILL Module Shown, Similar for TURN, MILL-TURN and Profile-NEST Modules

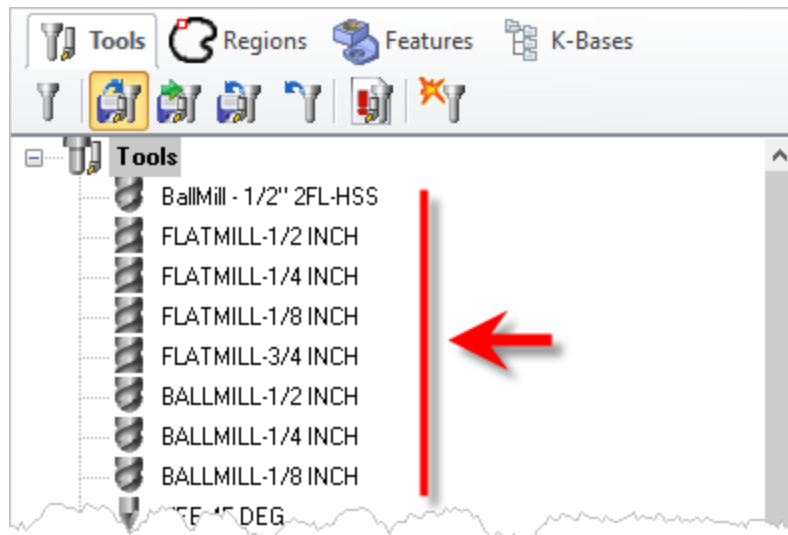
2. Select the [Tools](#) tab from the [Machining Objects Browser](#).



MILL Module Shown, Similar for TURN, MILL-TURN and Profile-NEST Modules

3.  Select the [Load Tool Library](#) icon to display the [Open](#) dialog.
4. Navigate to the folder where your tool library file is saved. Select the file and pick [Open](#).
5. You will see a message alerting you that tools are being loaded from a library. Pick [OK](#) and all tools in the library are loaded into the current part file and the tools are listed in the [Tools](#) list of the [Tools](#) tab.



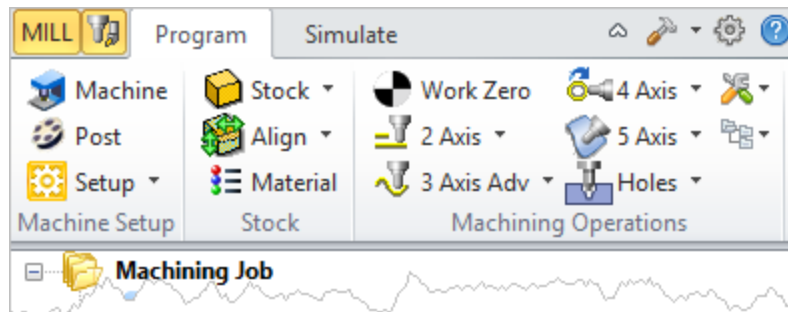


6. If you have tools already created in this part file they are not removed. The tools in the library are just added to your [Tools](#) list.

Load a Tool Library Automatically

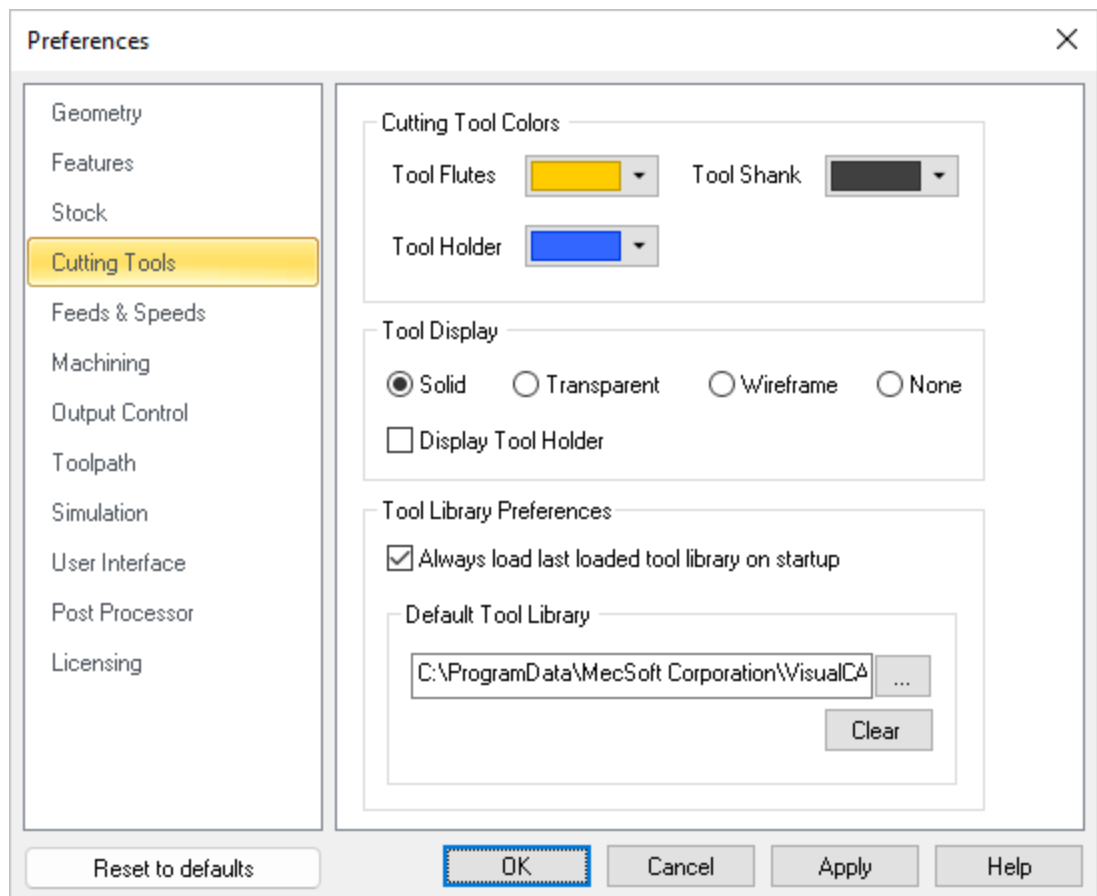
If you have a [Tool Library](#) file saved and you want this tool library to load each time you create a new part file:

1. First make sure you have a [Tool Library](#) saved. See [How to Save a Tool Library](#) for more information.
2.  Select the [CAM Preferences](#) icon located at the top-right of the Machining [Objects Browser](#) (i.e., to the far right of the [Program](#) tab) to display the dialog.



MILL Module Shown, Similar for TURN, MILL-TURN and Profile-NEST Modules

3. From the left side of the dialog select [Cutting Tools](#).



CAM Preferences > Cutting Tools

Note: Menu selections on the left may change depending on module and configuration

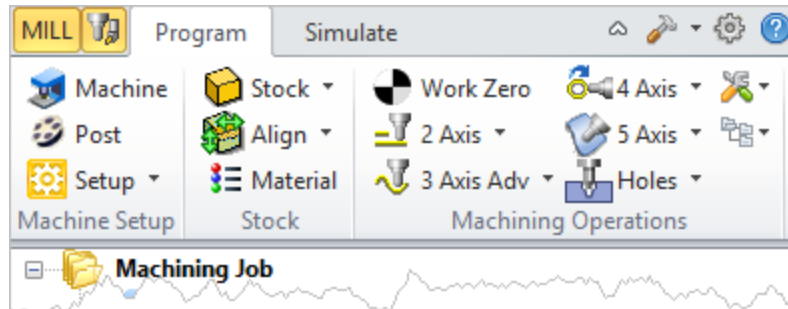
4. Check the box to [Always load last loaded tool library on start up](#).
5. From the [Default Tool Library](#) section of the dialog, pick the ... button to display the [Open](#) dialog.
6. Navigate to the folder where your tool library file is saved. Select the file and pick [Open](#).
7. The folder path and file name are then listed in the [Default Tool Library](#) field.
8. Pick [OK](#) to close the [CAM Preferences](#) dialog.
9. This [Tool Library](#) will now load automatically when you create/open a new part file.

Load the Default Tool Library

RhinoCAM includes a pre-defined tool library of both [Inch](#) and [Metric](#) tools that you can use to start your own tool library.

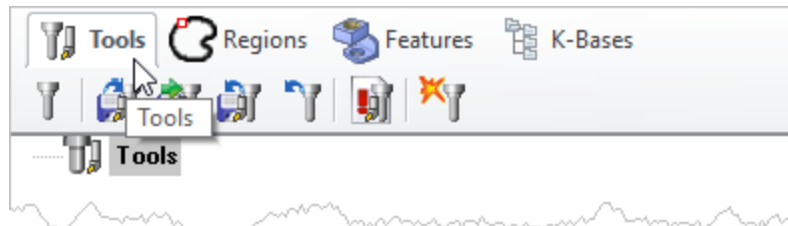
To load this tool library:

1.  To the left of the [Program](#) tab, select the [Tools Machining Objects](#) icon to make sure the [Machining Objects Browser](#) is displayed.



MILL Module Shown, Similar for TURN, MILL-TURN and Profile-NEST Modules

2. Select the [Tools](#) tab from the [Machining Objects Browser](#).



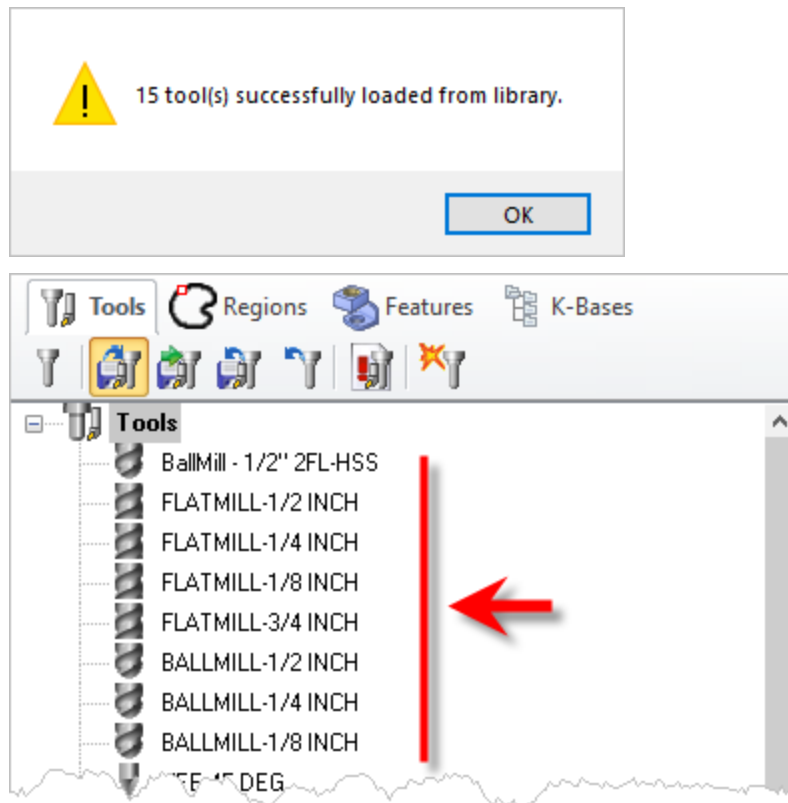
MILL Module Shown, Similar for TURN, MILL-TURN and Profile-NEST Modules

3.  Select the [Load Tool Library](#) icon to display the [Open](#) dialog.
4. From the [Open](#) dialog set the file type to [Tool Library Text Files \(*.csv\)](#).
5. Navigate to [C:\ProgramData\MecSoft Corporation\RhinoCAM 2025 for Rhino 7.0 or 8\Data](#).
6. There are two tool libraries in this folder:

[DefaultEnglishTools.csv](#)

[DefaultMetricTools.csv](#)

7. Select the tool library to load and pick [Open](#).
8. You will see a message alerting you that tools are being loaded from a library. Pick [OK](#) and all tools in the library are loaded into the current part file and the tools are listed in the [Tools](#) list of the [Tools](#) tab.



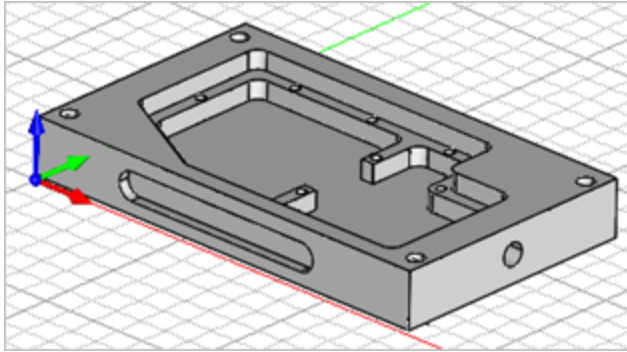
MILL Module Shown, Similar for TURN, MILL-TURN and Profile-NEST Modules

9. If you have tools already created in this part file they are not removed. The tools in the library are just added to your [Tools](#) list.

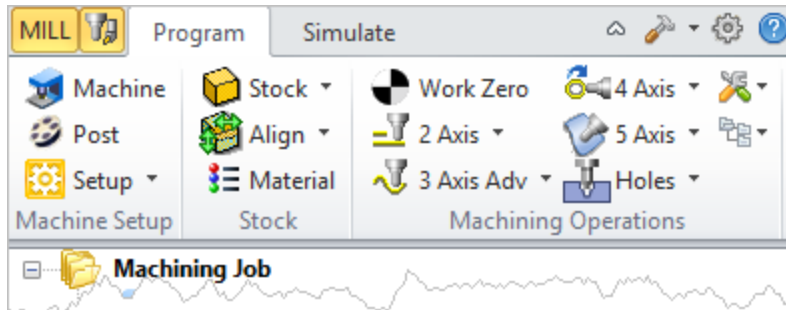
Orient a Part for Machining

RhinoCAM includes an [Orient Part](#) command to help you quickly change the part orientation for machining.

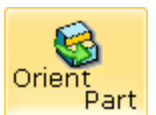
1. Open the part file you wish to orient.

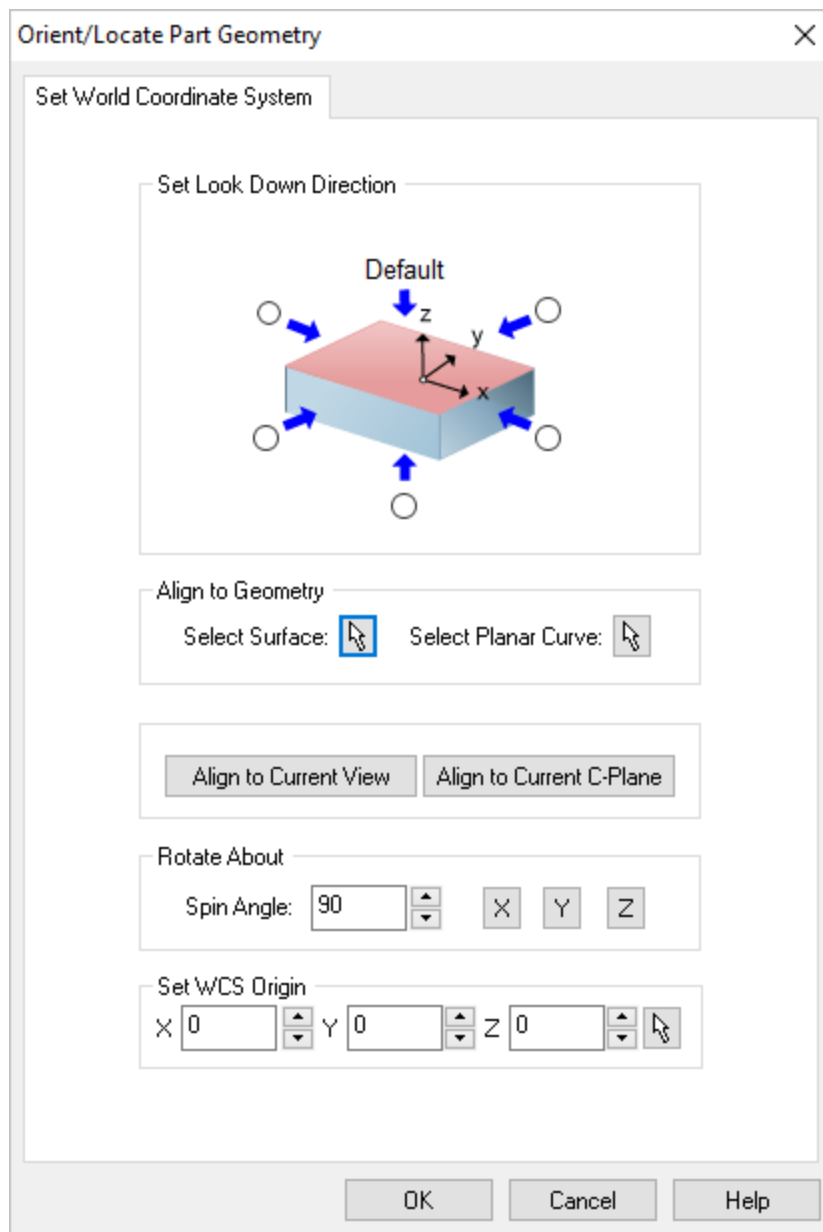


2. Select the [Program](#) tab.



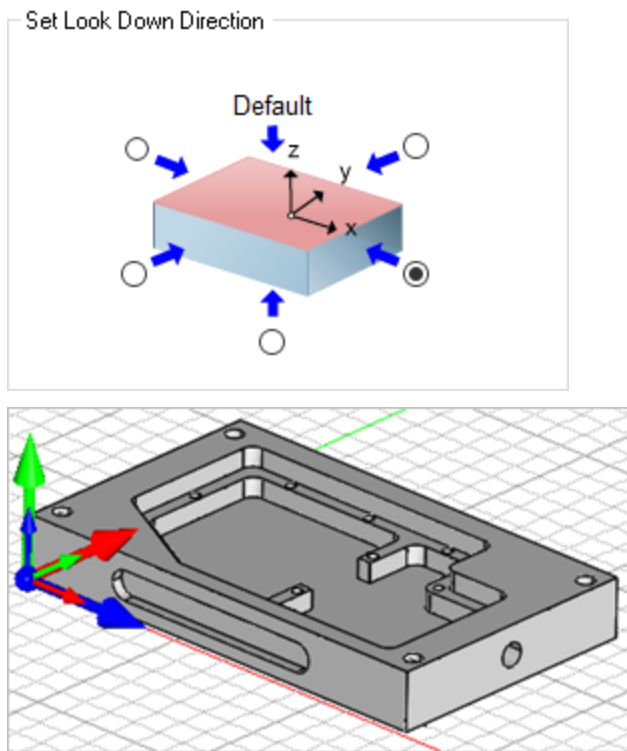
2.  From the [Program](#) tab, select the [Setup](#) menu.

3.  Select [Orient Part](#) to display the dialog.

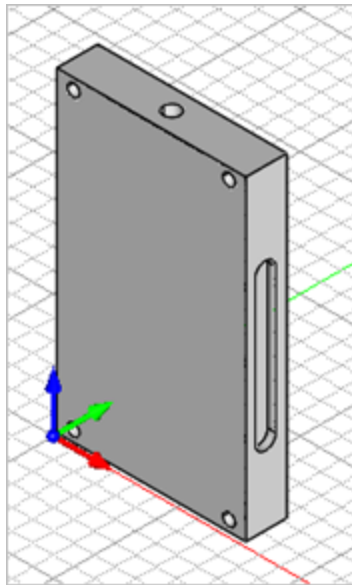


Dialog Box: Orient Geometry

4. From the [Set Look Down Direction](#) section of the dialog, select the direction you want to machine. While this dialog is displayed, the machining axis will display on the part. The **Z Axis (Blue)** should point toward the direction you want to machine.



5. Pick **OK** and the part will be re-oriented with that direction facing the positive **Z** axis.



6. There are other options on this dialog to help you orient the part. Select the **Help** button to learn about them.



[Related Online Help Topics](#)

User Interface

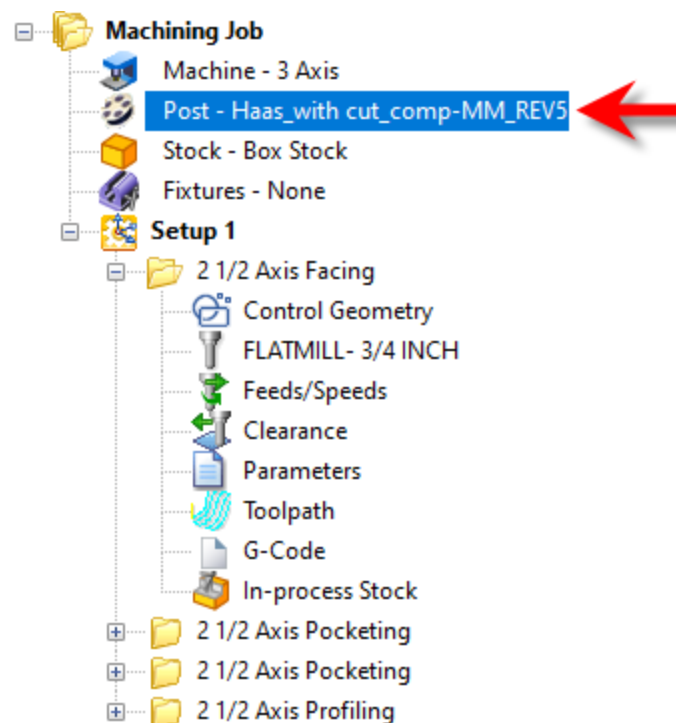
Orient Part, Setup

Post G-Code

Once machining operations are created they can be post-processed to a specific machine controller. To post-process a machining operation, select the operation in the browser, right click and select **Post**. The product comes with a set of over 300 post-processors to choose from. The current post-processor and g-code is also stored with the Part file. The current post-processor and latest G-Code is stored with the part file for better [CAM Life-cycle Management](#).

How your Post-Processor is Stored

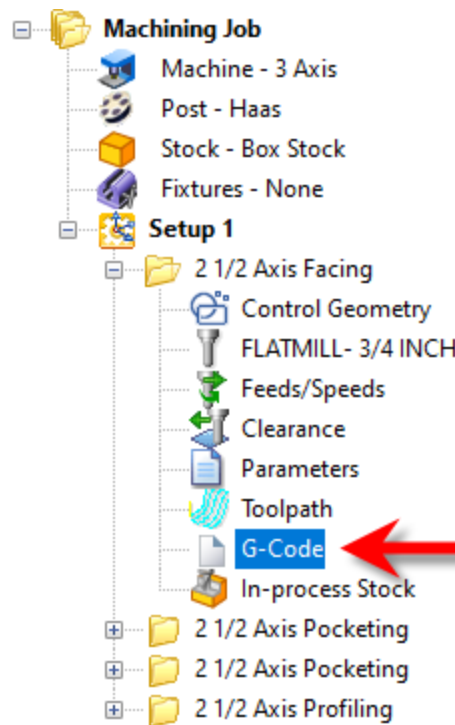
When you set the current post-processor for your [Machining Job](#) it is saved with your part file when the file is saved. This keeps the post-processor used to generate your g-code associated with your part file for better [CAM Life-cycle Management](#).



The Current Post-Processor is stored with your Part File

How G-Code is Stored

When you post-process a machining operation from the [Machining Job](#), the G-Code data is also saved with your part file when the file is saved. This keeps all cam data together for better [CAM Life-cycle Management](#). If you see that the G-Code icon is flagged, it means that the latest G-Code has not been captured. Regenerate the operation and the flag will be removed.

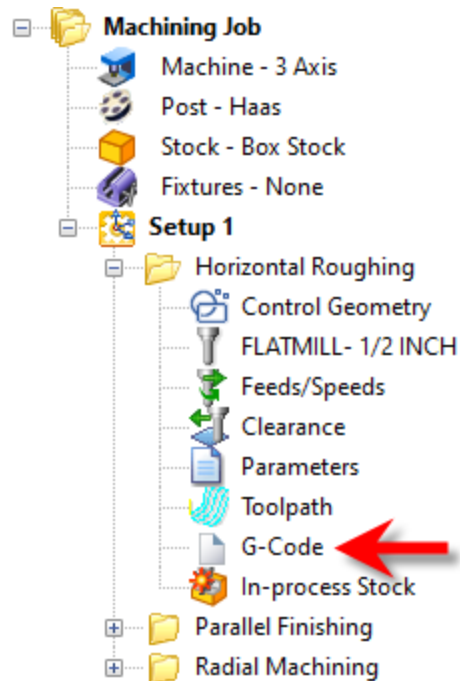


The G-Code Item within the MOp Folder



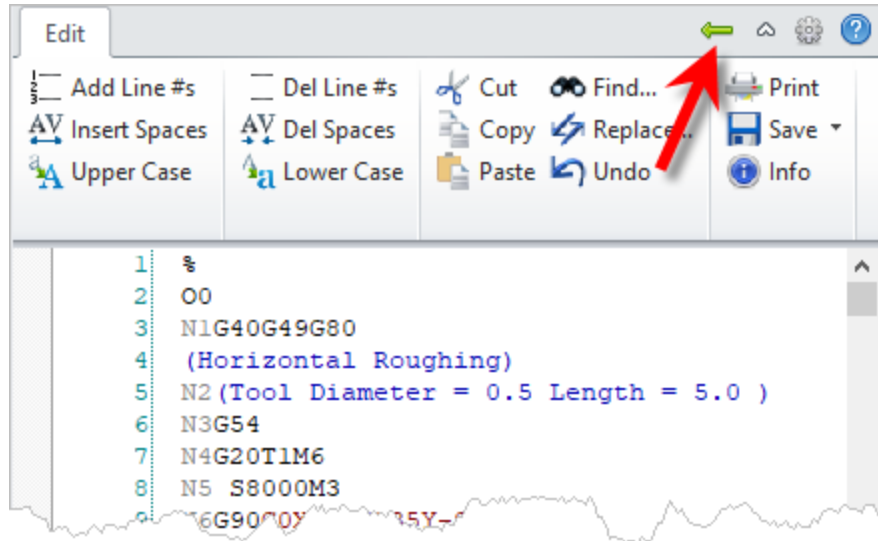
Editing G-Code from the MILL module

You can view or edit your G-Code from an operation that you have generated by simply clicking on the [G-Code](#) icon within the mop folder.



G-Code Browser: Tool Crib menu item

The [Machining Browser](#) will be replaced with the [Edit](#) tab of the [G-Code Editor](#) module. To return to the [MILL](#) module pick the left arrow icon at the top of the [G-Code Editor](#) browser. For documentation on using the G-Code Editor [Edit](#) tab click on the "?" help icon located at the top right side of the [G-Code Editor](#) browser.

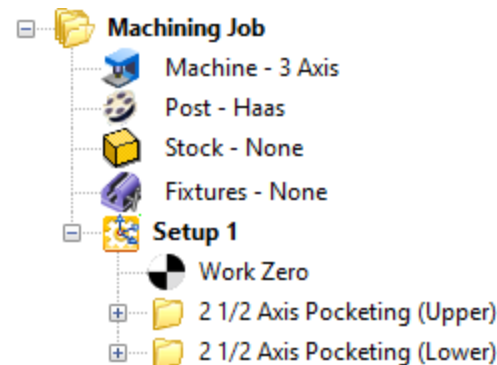


G-Code Browser: Tool Crib menu item



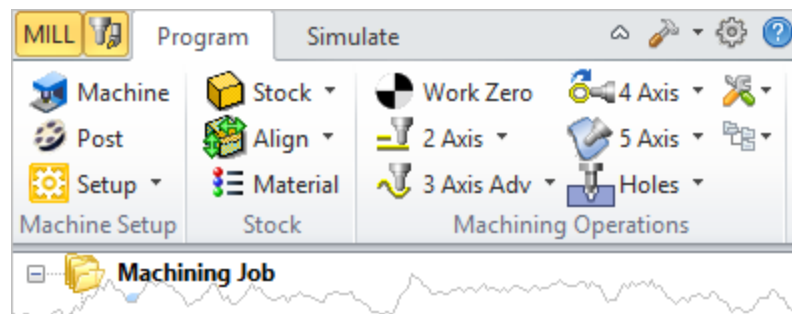
Steps to Manually Post G-Code

1. Create and adjust the toolpath operations that you want to post G-Code for.
2. Make sure the toolpaths have generated cleanly. Each toolpath when generated is listed under a [Setup](#) in the [Machining Job](#). If the operation is flagged it means that it needs to be regenerated.

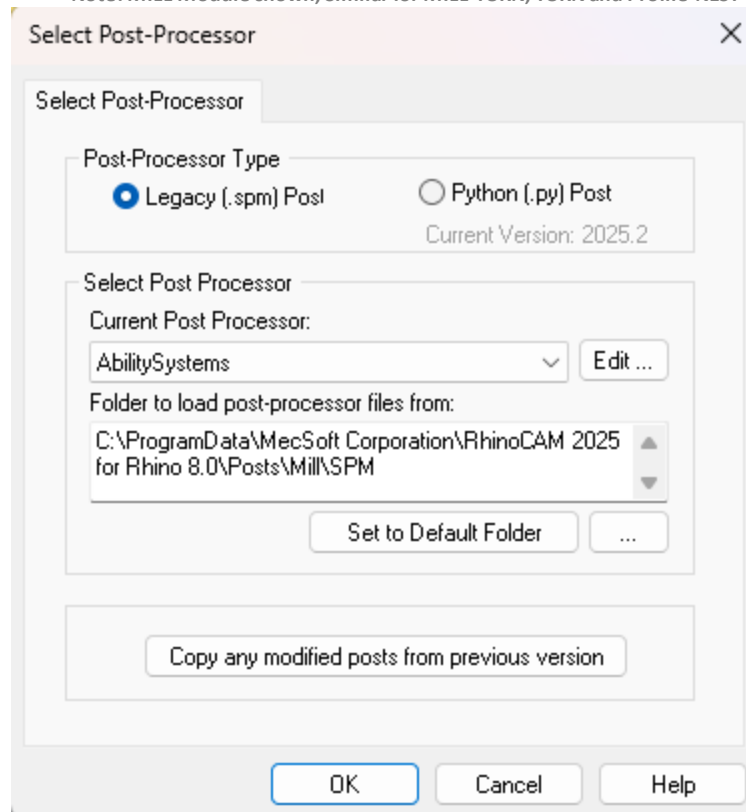


Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

3.  **Post** From the [Program](#) tab, select [Post](#) to display the dialog.

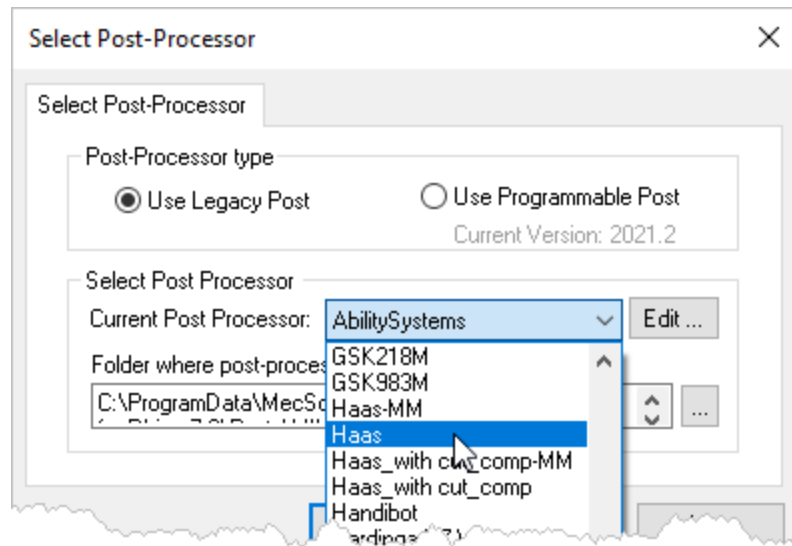


Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

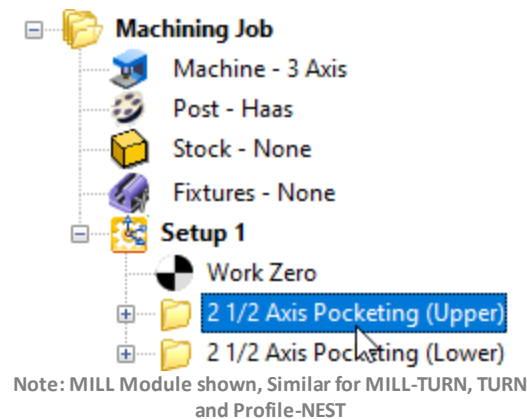


Dialog Box: Set Post-Processor Options

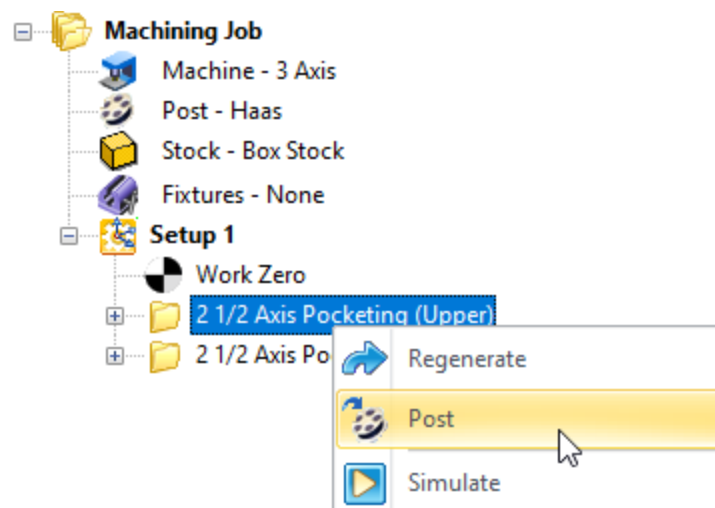
4. Select a **Post** from the **Current Post Processor** selection menu. See [How to Define the Post-Processor](#) for more information.



5. Make sure you have simulated each toolpath and are satisfied with each toolpath operation. See [How to Simulate a Toolpath](#) for more information.
6. Select the operation that you want to post. You can select multiple operations by pressing the <Ctrl> key while selecting. You can also select an entire [Setup](#) or the entire [Machining Job](#).

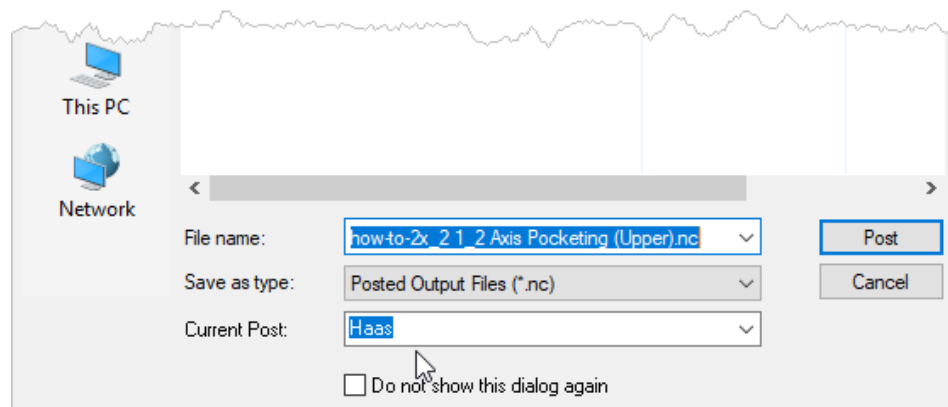


7. Right-click on the selected operation(s) and select [Post](#) to display the dialog.

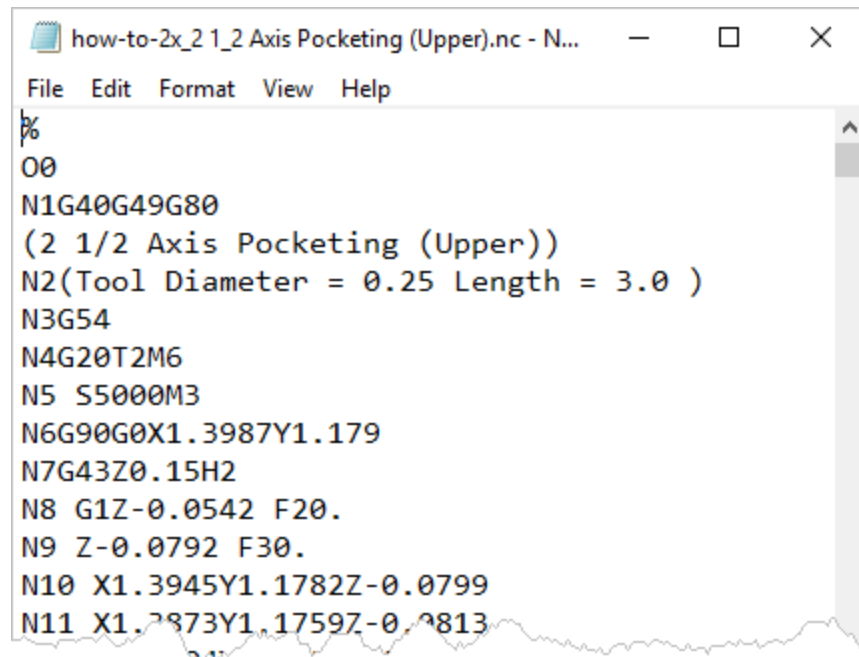


Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

8. Your **Current Post** is listed at the bottom of the dialog. You can change it here if desired.



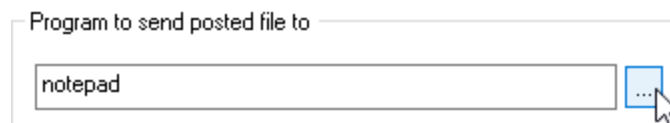
9. Navigate to the folder where you want the posted G-Code file to be saved.
10. Adjust the name of the G-Code file if desired.
11. Make sure the file extension (example: **.nc**) of the G-Code file is correct and that your CNC machine controller will read this file extension. Refer to your controller manual for this information.
12. Select the **Post** button and the G-Code file is calculated and displayed.



```

how-to-2x_2 1_2 Axis Pocketing (Upper).nc - N...
File Edit Format View Help
%
O0
N1G40G49G80
(2 1/2 Axis Pocketing (Upper))
N2(Tool Diameter = 0.25 Length = 3.0 )
N3G54
N4G20T2M6
N5 S5000M3
N6G90G0X1.3987Y1.179
N7G43Z0.15H2
N8 G1Z-0.0542 F20.
N9 Z-0.0792 F30.
N10 X1.3945Y1.1782Z-0.0799
N11 X1.3973Y1.1759Z-0.0813
  
```

13. G-Code files are [ASCII Text Files](#) and by default, are displayed in [notepad](#), a windows generic text editor.
14. You can review the G-Code and make any manual edits if needed and save the file from notepad.
15. If you use a different text and G-Code editor program, you can tell [RhinoCAM](#) to display your G-Code files in this program. To do this select [Post](#) from the [Program](#) tab to display the dialog.
16.  In the section of the dialog were it says [Program to send posted file to](#) select the ... button to the right to display the File Open dialog.

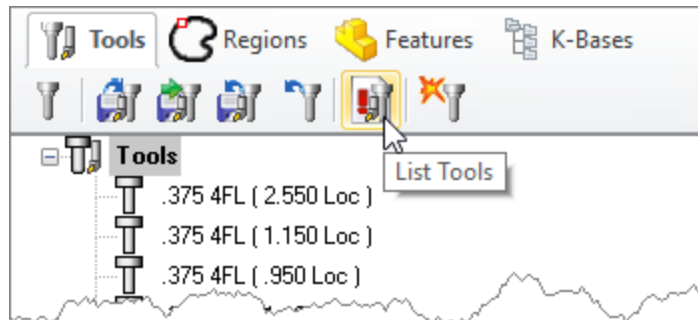


17. Locate the [.exe](#) program that you use to edit your G-Code files with and then press [Open](#).
18. Then pick [OK](#) to close the [Set Post-Processor Options](#) dialog.

Print a Tool List

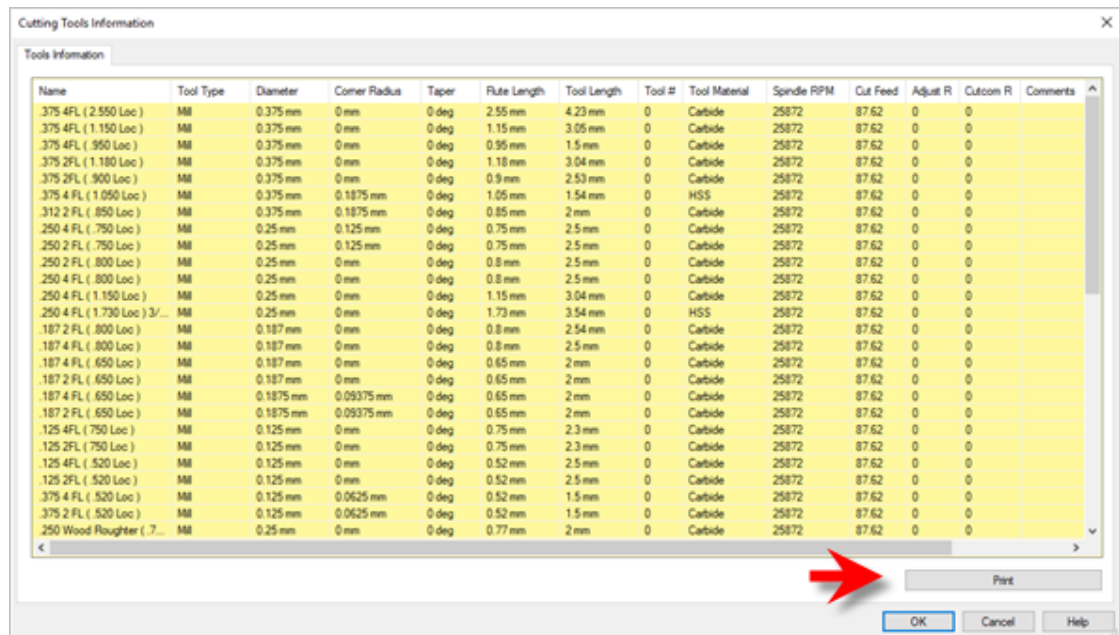
Follow the steps outlined below to print your [Tool List](#):

1. The [Tool List](#) is based on the tools currently listed under the [Tools](#) tab of the [Machining Objects Browser](#). If you want to print your tool library list, first [Load your Tool Library](#) so that all of your are listed in the [Tools](#) tab. See [Create a Tool Library](#) to learn how to Load your Tool Library.
2. From the [Tools](#) tab, select the [Tool List](#) icon:



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

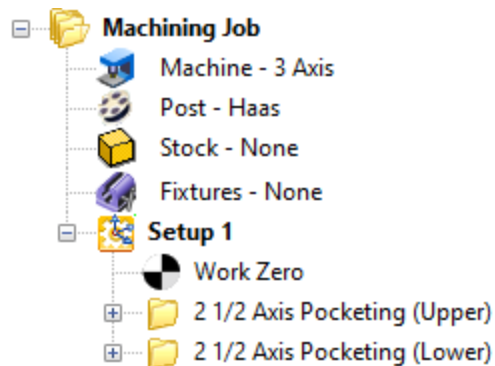
3. From the [Cutting Tools Information](#) dialog, pick the [Print](#) button.



Save a Knowledge Base

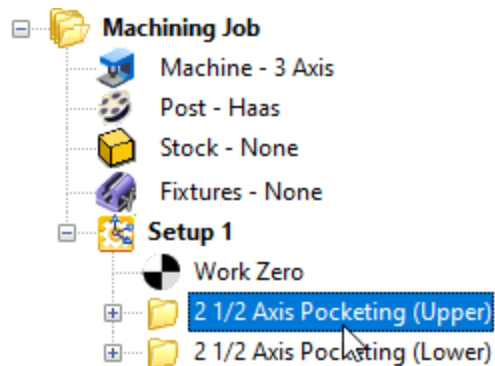
RhinoCAM includes powerful [Knowledge Base](#) functionality that makes "push button" programming a reality. You can archive an entire machining strategy specific to a certain class of parts in a [Knowledge Database](#) (also referred to as a [K-Base](#)) and then optionally assign [Geometry Selection Rules](#) that are applied automatically when toolpath operations are selected for use from the [Knowledge Base](#).

1. Create and adjust the toolpath operations that you want to include in your knowledge base.
2. Make sure the toolpath has generated cleanly. Each toolpath when generated is listed under a [Setup](#) in the [Machining Job](#). If the operation is flagged it means that it needs to be regenerated.



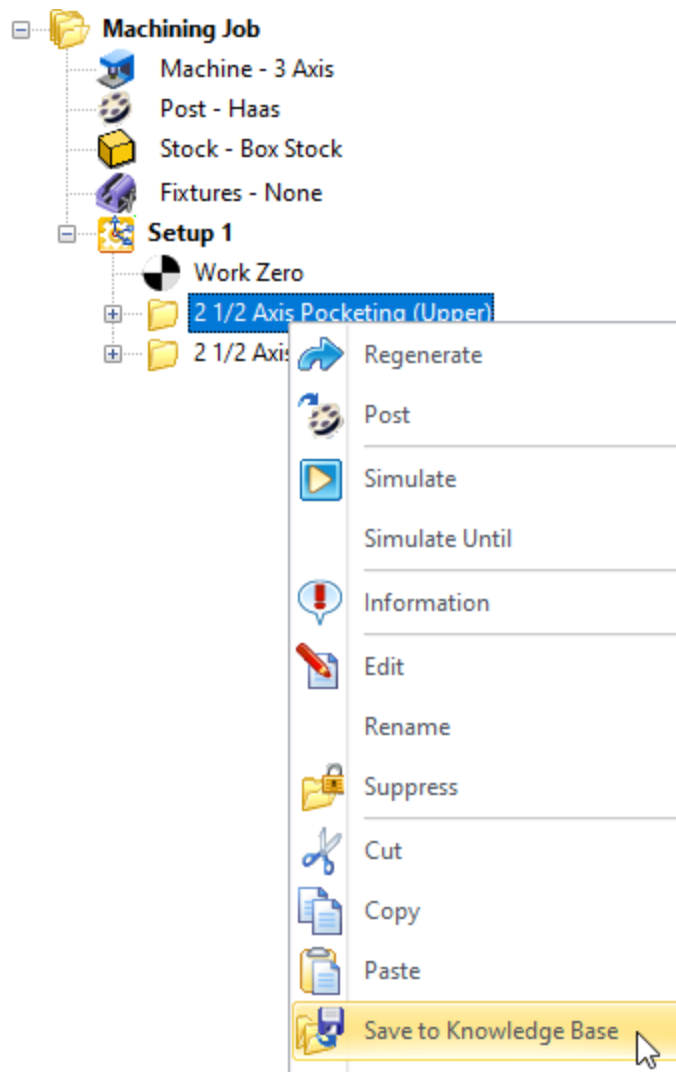
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

3. Make sure you have simulated each toolpath and are satisfied with each toolpath operation.
4. Select the operation that you want to save to the [Knowledge Base](#) for. You can select multiple operations by pressing the <Ctrl> key while selecting.



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

5. Right-click on the selected operation(s) and select [Save to Knowledge Base](#).

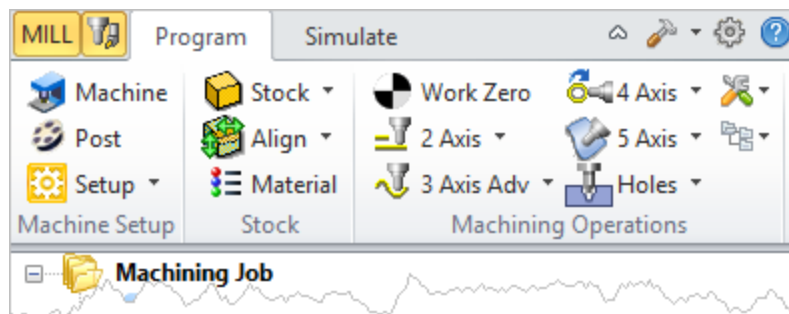


Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

6. The [Save As](#) dialog is displayed and automatically navigates to the [Defaults](#) folder for [RhinoCAM](#). Enter a name for the [Knowledge Base](#) file and pick [Save](#) (Example: [My_Knowledge_Base.vkb](#))
7. Only one [Default Knowledge Base](#) can be loaded at a time. However, you may want to use multiple different [Default Knowledge Base](#) files.

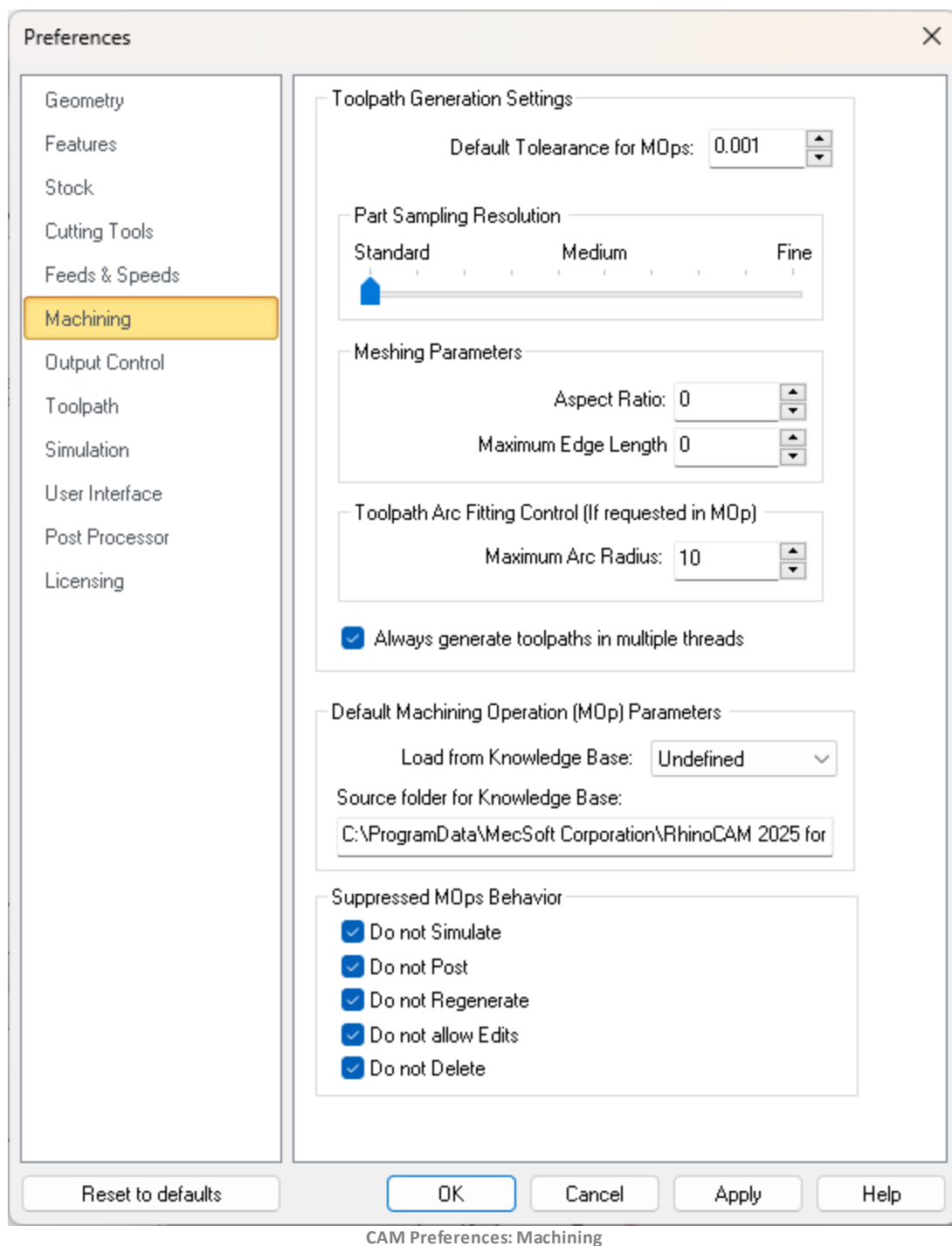


To change the [Default Knowledge Base](#) select the [CAM Preferences](#) icon from the [Machining Objects Browser](#) to display the dialog.

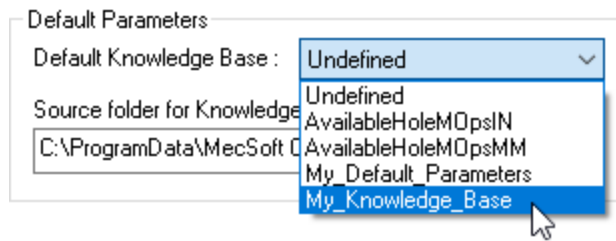


Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

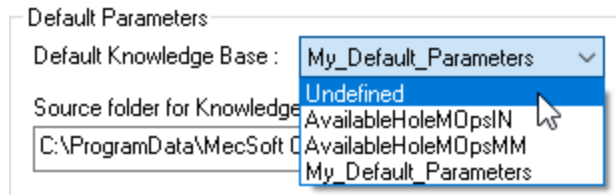
8. Select [Machining](#) from the left to display the [Machining Preferences](#).



- From the [Default Parameters](#) section select the [Default Knowledge Base](#) to use from the menu. If you have saved different [Default Knowledge Base](#) into the [Defaults](#) folder, they will be listed here.



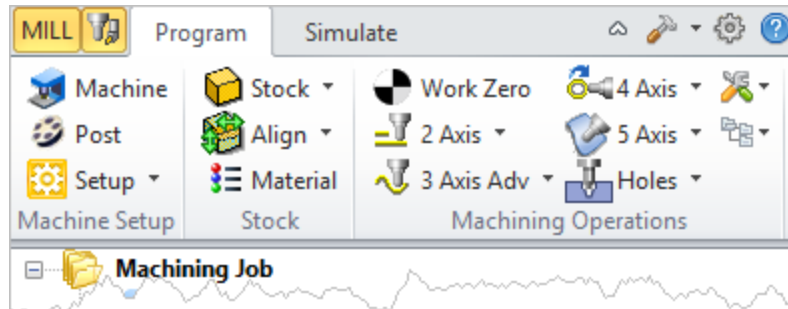
10. If you set this to **Undefined**, the *factory defaults* are used when creating an operation.



Save a Tool Library

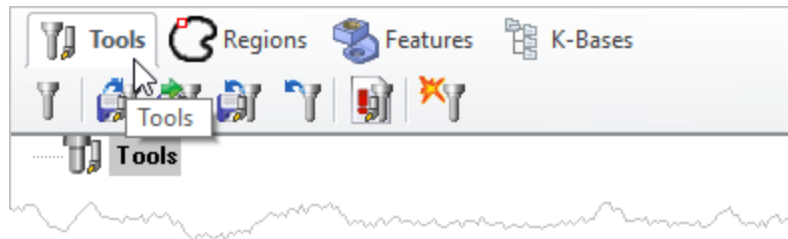
Creating a tool only assigns that tool to the part file you are editing at the time. To use your tools repeatedly for other part files you need to create a tool library.

1. Open the part file that contains tools that you want to save to a library.
2.  To the left of the [Program](#) tab, select the [Tools Machining Objects](#) icon to make sure the [Machining Objects Browser](#) is displayed.



MILL Module Shown, Similar for TURN, MILL-TURN and Profile-NEST Modules

3. Select the [Tools](#) tab from the [Machining Objects Browser](#) to see your list of tools.

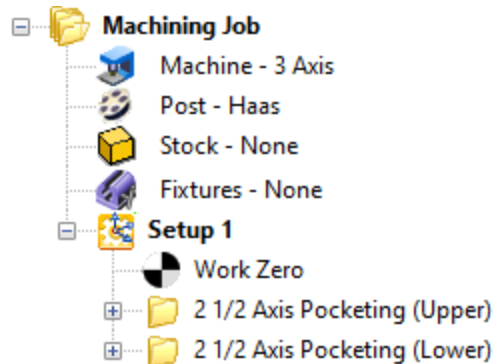


4.  From the toolbar below the [Tools](#) tab select the [Save Tool Library](#) icon.
5. From the [Save As](#) dialog that displays, enter a [File Name](#) for the tool library.
6. Select a folder where you want the tool library file saved.
7. By default tool library files are extended with a [*vkb](#) file extension.
8. Pick [Save](#) to save the file.

Save Defaults

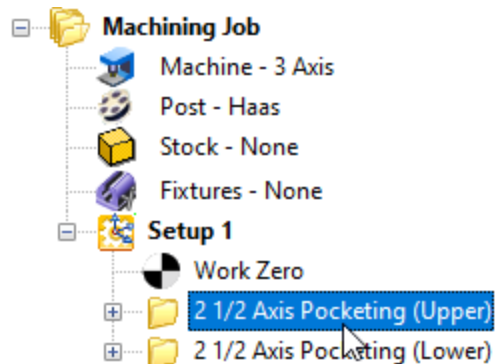
RhinoCAM can save all of your machining parameters as default settings so that they are loaded automatically each time you create the same toolpath operation. This is your first step toward toolpath automation.

1. Create and adjust the toolpath operations that you want to save defaults for.
2. Make sure the toolpath has generated cleanly. Each toolpath when generated is listed under a [Setup](#) in the [Machining Job](#). If the operation is flagged it means that it needs to be regenerated.



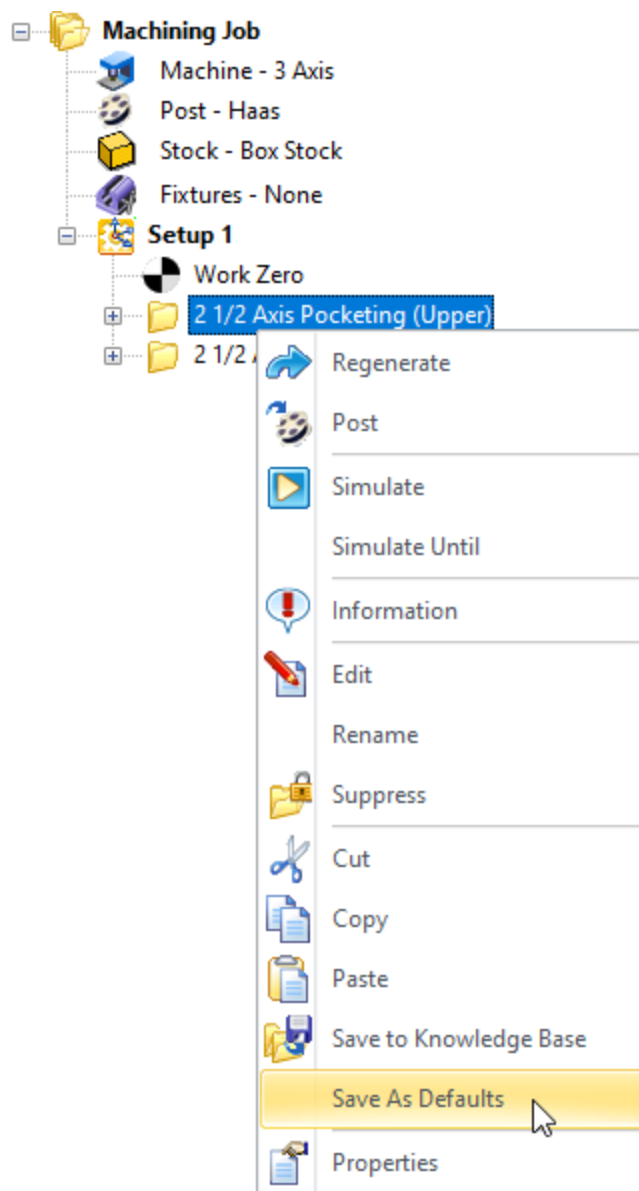
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

3. Make sure you have simulated each toolpath and are satisfied with each toolpath operation.
4. Select the operation that you want to save defaults for. You can select multiple operations by pressing the <Ctrl> key while selecting.



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

5. Right-click on the selected operation(s) and select [Save as Defaults](#).



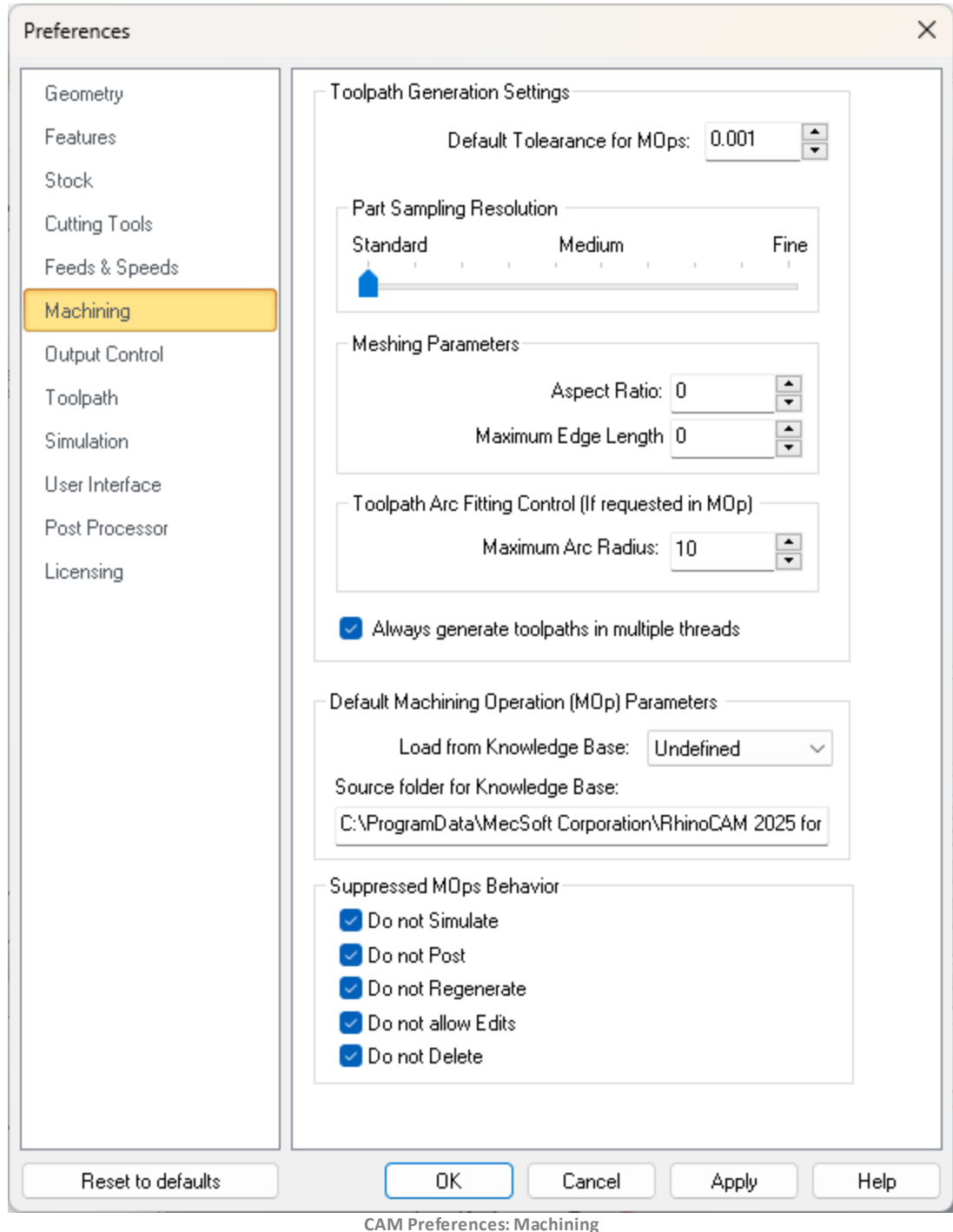
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

6. The [Save As](#) dialog is displayed and automatically navigates to the [Defaults](#) folder for [RhinoCAM](#). Enter a name for the defaults file (for example: [My_Default_Parameters.vkb](#)) and pick [Save](#).
7. The defaults are saved as the [Default Knowledge Base](#) file with a [.vkb](#) file extension and the CAM [Machining Preferences](#) are set automatically to use this [Default Knowledge Base](#) file.
8. Only one [Default Knowledge Base](#) can be loaded at a time. However, you may want to use multiple different [Default Knowledge Base](#) files.

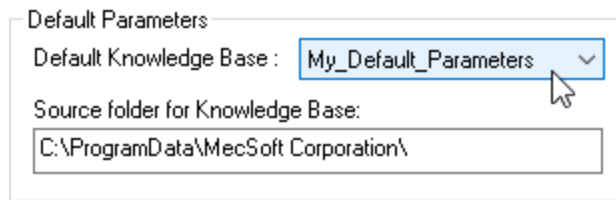


To change the [Default Knowledge Base](#) select the [CAM Preferences](#) icon from the Machining [Objects Browser](#) to display the dialog.

9. Select [Machining](#) from the left to display the [Machining Preferences](#).



10. From the [Default Parameters](#) section select the [Default Knowledge Base](#) to use from the menu. If you have saved different [Default Knowledge Base](#) into the [Defaults](#) folder, they will be listed here.



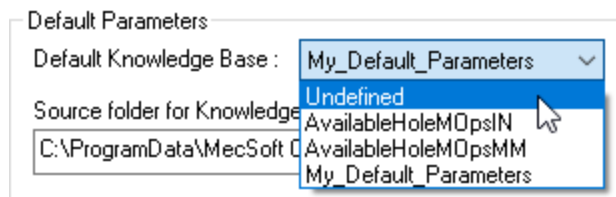
Default Parameters

Default Knowledge Base : My_Default_Parameters

Source folder for Knowledge Base:

C:\ProgramData\MecSoft Corporation\

11. If you set this to [Undefined](#), the *factory defaults* are used when creating an operation.



Default Parameters

Default Knowledge Base : Undefined

Source folder for Knowledge Base:

C:\ProgramData\MecSoft Corporation\

AvailableHoleMOpsIN

AvailableHoleMOpsMM

My_Default_Parameters

Set Preferences at Start Up

The steps below will assist you in making sure the correct preferences are loaded at start up.

Important: When you install the plug-in (near the end of the installation) the program will ask you if you want to use your existing settings. Make sure you say YES. If you said NO, or if your preferences have changed without your knowledge, then follow the steps below to recapture your preferred settings manually.

To open new files using a default template file:

1. Open an existing file that has your settings defined (i.e., post, tool library, etc.).
2. Save the file and the settings will update in the Windows registry.
3. As a check, open a new file and go to the [Program](#) tab > [Post](#) and look at the [Select Post-Processor](#) dialog.
4. Make sure your correct post is selected. If not, use the ... button to browse and select the folder where you post is and pick [OK](#) and then [OK](#) again.
5. For new files you can open the Template file. If not, that's OK, [RhinoCAM/VisualCAM](#) will load with your settings.

To load a default Tool Library at start up:

1. Check the [Tools](#) tab. If your tools are not loaded by default, go to the [CAM Preferences](#) dialog ([Machining Browser](#) > [CAM Preferences](#) (the gear icon at the right side of the browser)) and go to the [Cutting Tools](#) tab.
2. Use the [Tool Library Preferences](#) section, make sure your tool library is listed in the [Default Tool Library](#) field. If not, use the ... button to locate your tool library and pick [OK](#) and then [OK](#) again.

To load a default Knowledge Base:

1. If you have a default knowledge base, go to CAM Preferences and from the [Machining](#) tab make sure your knowledge base is selected from the [Load from Knowledge Base](#) drop-down menu. If not, highlight and copy the source folder for knowledge bases. Then open the windows file browser and navigate to that folder and copy your knowledge base file to that folder.
2. Now close and reopen the [CAM Preferences](#) tab and make sure your default KB is selected and pick [OK](#).

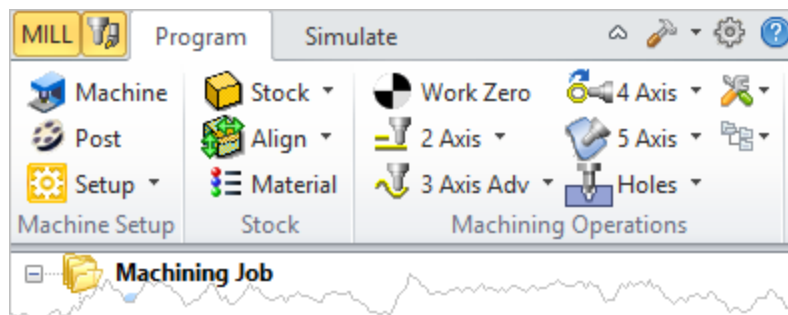
General options for handling your CAM Preferences:

1. Open or start a new file and make sure your preferences are correct.
2. Go to the CAM Preferences dialog by selecting the gear icon from the [Machining Browser](#) (top-right and to the left of the help ? icon).
3. Select the [User Interface](#) tab.
4. There 4 options under the Save Preferences section.
5. Uncheck the third checkbox and the large button that says "[Always save current preferences to registry on exit](#)" will be activated.
6. Select this button and all of your current preferences will be saved to your Windows registry.
7. Test this by closing and reopening [RhinoCAM/VisualCAM](#) and your default preferences should load by default.
8. The three check boxes are self-explanatory. Use them to set how preferences are loaded/saved.

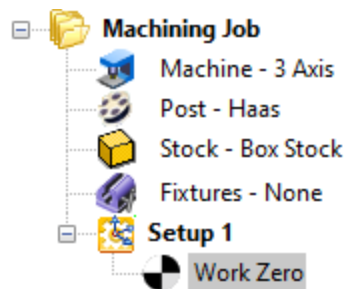
Setup a Part

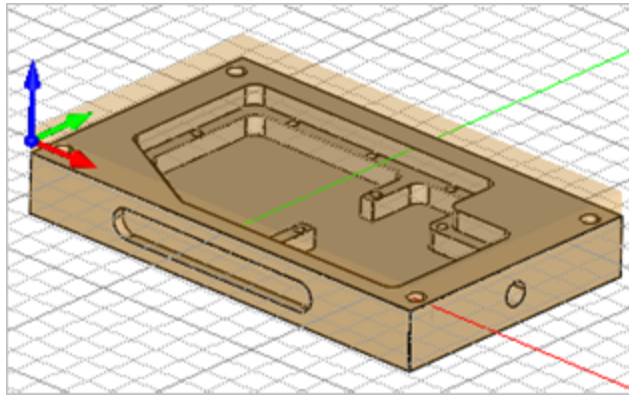
To setup a part for machining perform the following basic steps. These are similar for all types of parts.

1. Check and know the part units ([Inches/Millimeters](#)). Only these two units are supported in CAM.
2. Check the orientation of the part to make sure it can be machined. By default the [Z axis](#) is the machining direction. Typically the [X axis](#) points toward the right or along the bed of the CNC machine. The [Y axis](#) typically points to the back of the CNC machine.
3. See [How to Orient a Part for Machining](#) for a quick way to orient the part.
4. Select the [Program](#) tab.



5. **Machine** Set the [Machine](#) definition to either 3, 4 or 5 axis depending on your machine tool and part requirements.
6. **Post** Set the [Post-Processor](#) definition.
7. **Stock** Define your [stock size](#) and [material](#).
8. **Align** [Align the Stock](#) with the [Part](#) if needed.
9. **Work Zero** By default the program zero will be measured from the [WCS \(World Coordinate System\)](#). You can create an alternate [Work Zero](#) location if needed.



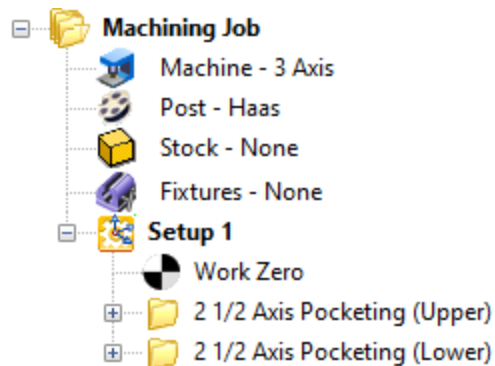


Simulate a Toolpath

RhinoCAM allows you to simulate your toolpath to see the cut material removal and in-process stock.

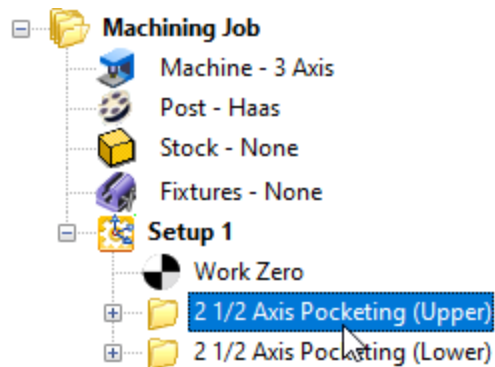
Here are the basic steps to simulate a toolpath:

1. First create a toolpath to simulate and make sure the toolpath has generated cleanly. Each toolpath when generated is listed under a [Setup](#) in the [Machining Job](#). If the operation is flagged it means that it needs to be regenerated and cannot be simulated yet.



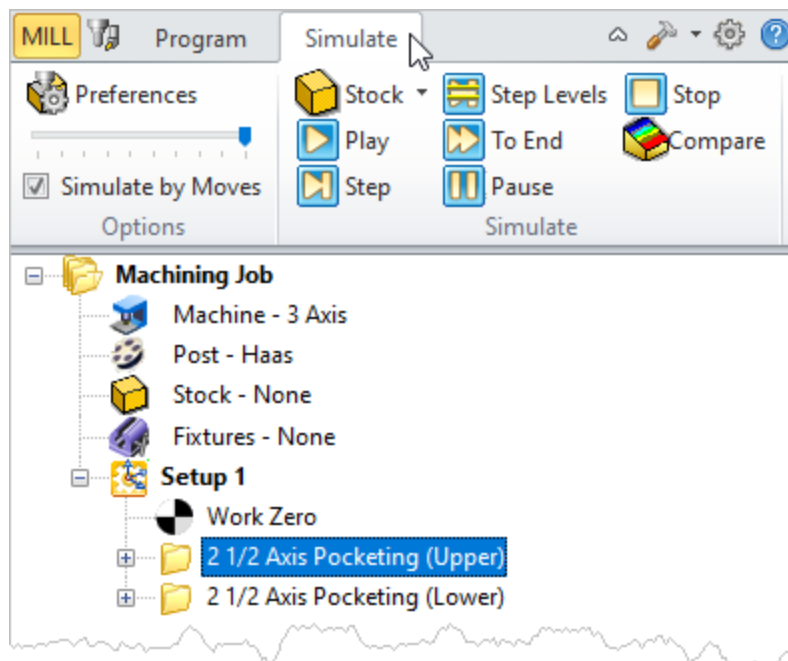
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

2. From the [Machining Job](#), select the operation that you want to simulate.



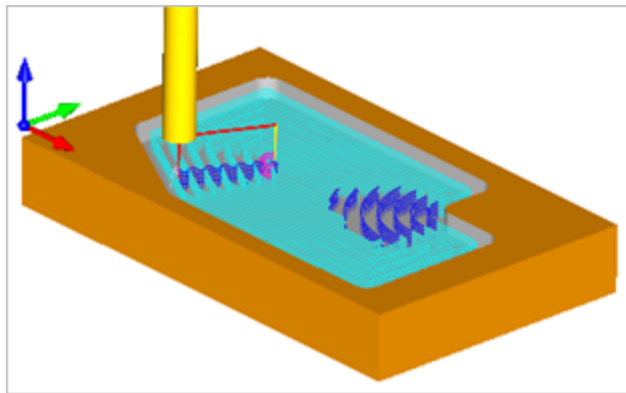
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

3. Select the [Simulate](#) tab. Its located to the right of the [Program](#) tab.

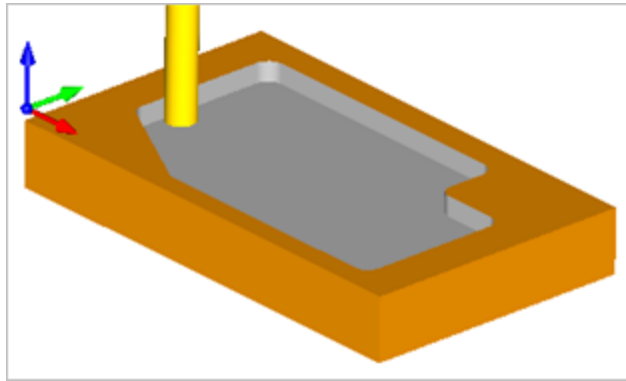


Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

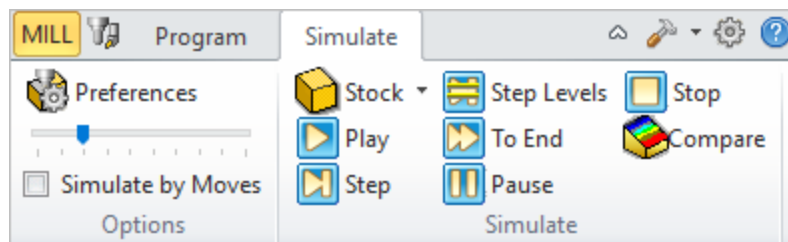
4.  Pick the [Play](#) button to run the simulation.



5.  You can toggle off the toolpath to see the resulting in-process stock model by selecting the [Toggle Toolpath Visibility](#) icon from the base of the [Machining Browser](#).



6. You can slow down the simulation by adjusting the slider located on the [Simulate](#) tab menu. You can further slow down the simulation by un-checking [Simulate by Moves](#) from the [Simulate](#) tab menu.

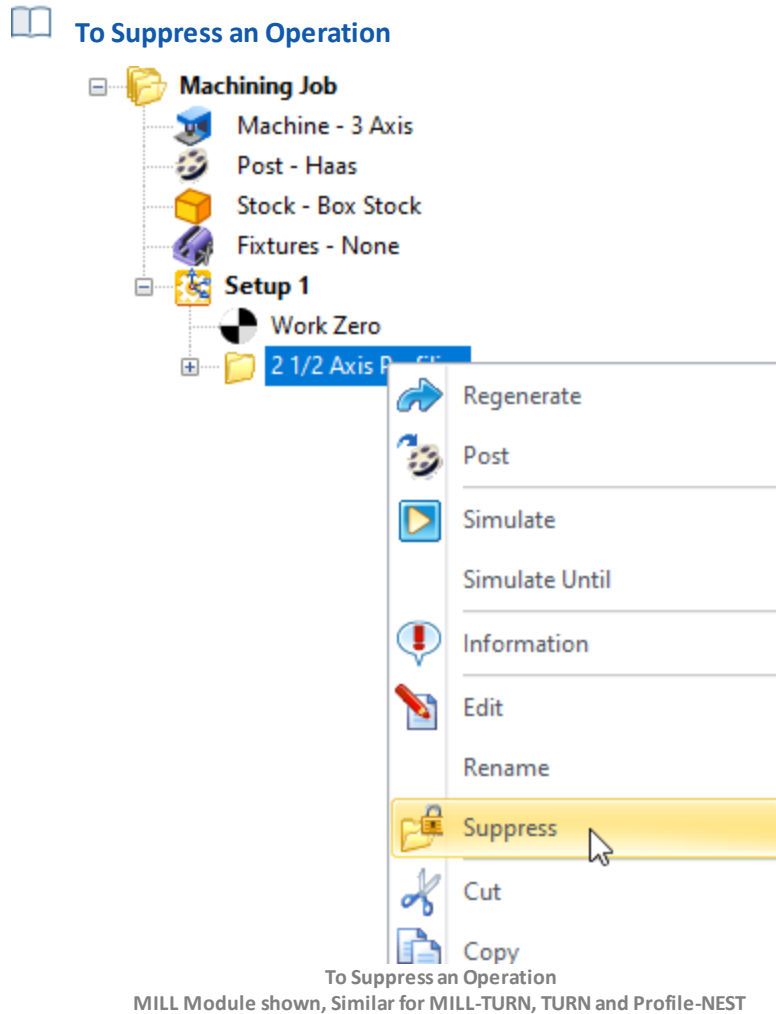


Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

7.   You can fast-forward to the end of the simulation by selecting [Pause](#) and then [To End](#).
8.  To see each cut motion one at a time, you can select the [Step](#) button.
9.  You can set simulation preferences by selecting [Preferences](#) from the [Simulate](#) tab menu.
10.  You can apply a [Stock Material](#) texture to the simulation by selecting the [Toggle Material Texture Visibility](#) icon from the base of the [Machining Browser](#).
11. If the in-process [Stock](#) does not display on the screen during the simulation, make sure you have simulated all previous operations that appear in the [Machining Job](#).

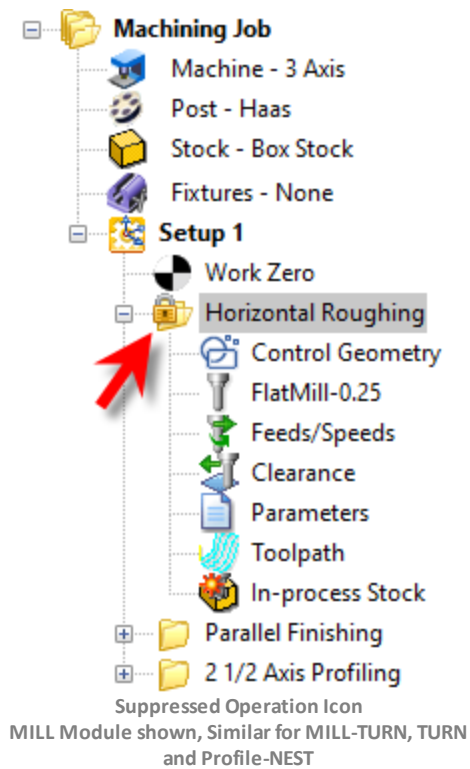
Suppress a Toolpath

You can **Suppress** a machining operation or a **Setup** in the **Machining Browser** by selecting it, right click and select **Suppress** from the context menu. Suppressed operations will not be displayed, posted or simulated. You can also right-click and **Unsuppress** an operation. **Note:** You can customize how suppressed operations are managed using the Machining Preferences dialog.



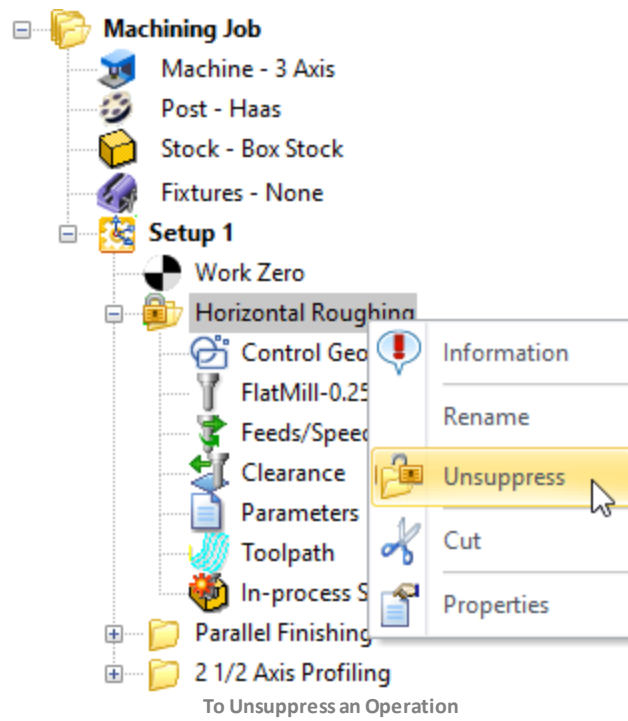
The Suppressed Operation Icon

A Suppress operation will display in the **Machining Job** with the following icon:



To Unsuppress an Operation

To **Unsuppress** an operation, right-click on it and select **Unsuppress**.



Use these help topics

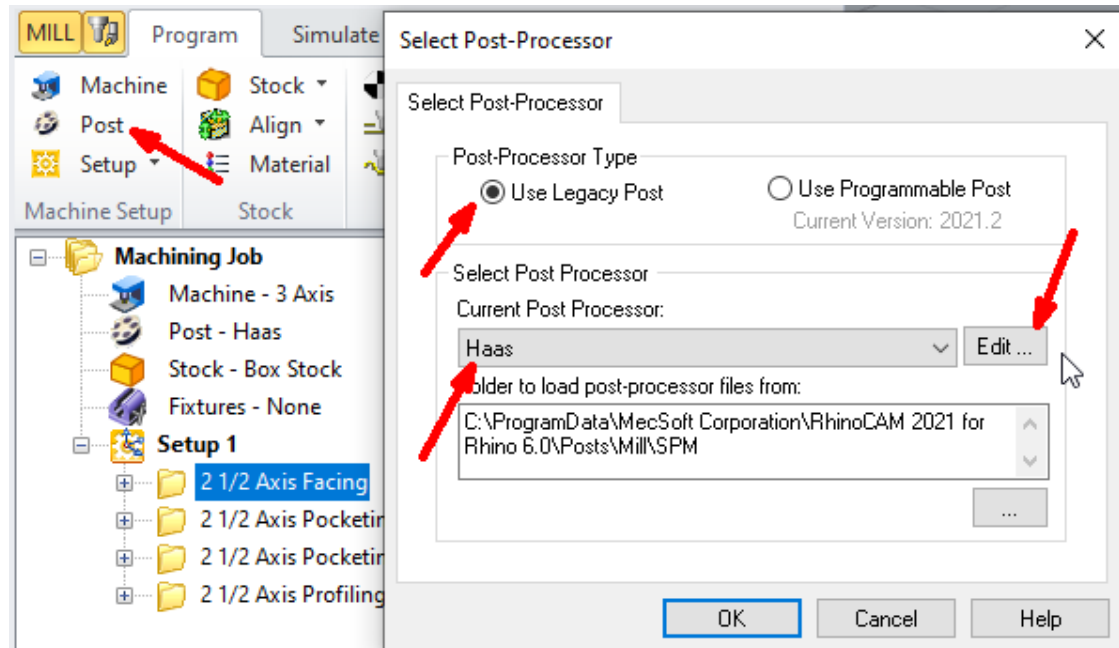
These [How To ...](#) help topics are quick answers to some of the many questions you will have as a new user starting out. They are arranged in the order that is typically used to create and work with toolpaths in [RhinoCAM](#). Here are some suggestions on how to use these topics:

1. If you are new to [RhinoCAM](#), we suggest that you first take some time to complete the [Quick Start Guide](#). It walks you through the basic procedure to create a toolpath. See the [Videos & Guides](#) topic for more information.
2. Next, take some time to read through all of these How To topics. The topics are short, providing quick information.
3. Then refer back to a specific topics later as you need to refresh your skills.
4. Remember that each dialog in [RhinoCAM](#) has a Help button that you can select to display the online help for that dialog.

Why are my Feed Rate values too High/Low?

If your **Feed Rate** values in your posted g-code are too **High** or too **Low**, you can adjust the **High/Low** parameters in the **Post Processor** following the steps outlined below:

1. Click on **Post (Set Post-Processor Options)**, then click **Edit**.



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

2. From the **Post Processor Generator** dialog, select the **Feed Rate** tab from the left.
3. Then make the necessary changes under the **Feed Rate Value** section for either the **High Value** or **Low Value**.

Post-Processor Generator

General
Start/End
Tool Change
Setup
Spindle
Feed Rate
Motion
Circle
Helical/Spiral
Multi Axis Motion
Cutter Compensation
Cut Motion Start/End
Cycles
Misc
Variables

Block Format
Feed Rate Code: F
☒ Add after Next Motion Code
Inverse Time Feed Rate
☐ Output inverse time feedrate for 4 & 5 Axis motions
On Code: Off Code:
Feed Rate Modes
Units/Min Code: Units/Rev Code:
Block Format
F[FEEDRATE]
Sample Output
F1
Default
Feed Rate Value
High Value: 1000
Low Value: 1
Scale Factor: 1
Z Feed Rate Scale Factor: 1
of Decimal Places: 0
Show Trailing Zeros: ☐
Angular Feedrate Value
Scale Factor: 1
of Decimal Places: 1
Show Trailing Zeros: ☐
Save Close Save As... Help

4. Then pick [Save](#).

Index

- A -

Add a Tool Change Point? 8
Add more Materials 12
Add Tool Comments? 19
Align the Stock & Part 23
Assign a Stock Material 28

- C -

CAM Preferences
 At Startup 138
 Knowledge Base 138
 Set Manually 138
 Tool Library 138
Control the Cut Side & Start Point 31
Copy/Edit a Toolpath 40
Create a Setup Sheet 48
Create a Tool 51
Create a Work Zero 58

- D -

Define a Box Stock 61
Define a Machine Tool 65
Define the Post-Processor 69
Define Toolpath Properties 74

- E -

Edit Toolpaths Associatively 77
Editing Machining Operations
 Associatively 77
 Properties 74
 Suppress 145
Enable Cutter Compensation 84
Enable Inverse Time Feedrate in 4 Axis 87
Estimate Machining Time 89

- F -

Find Tool Related Preferences 91

- G -

Generate a Toolpath 95

- H -

How to Use these help topics 147

- L -

Load a Tool Library 110
Load a Tool Library Automatically 112
Load the Default Tool Library 114

- M -

Machining
 Operation Properties 74

- O -

Orient a Part for Machining 116

- P -

Post G-Code 120
Print a Tool List 127

- Q -

Quick Start 4

- R -

Resource Guide 7

- S -

Save a Knowledge Base 128
Save a Tool Library 133
Save Defaults 134
Setup a Part 140
Simulate a Toolpath 142
Suppress

Suppress

 Toolpaths (MOps) 145

Suppress a Toolpath 145

- U -

Unsuppress

 Toolpaths (MOps) 145

- W -

Why are my Feed Rate values too High/Low? 148