

The Cutting Tools Workbook

RhinoCAM 2025

Published: February 2025

MecSoft Corporation

© Copyright 1998-2024

Table of Contents

Quick Start	5
Resource Guide	8
About this Guide	9
1 About the MILL Module	9
2 About the TURN Module	9
3 Using this Guide.....	10
Getting Ready	11
1 After Installing RhinoCAM MILL.....	11
2 Locate the Tools Tab.....	11
3 The Create/Select Tools Dialog.....	11
Creating Tools	13
1 Create a Tool.....	13
2 Create a Tool Holder.....	19
3 Create a Tool Library.....	22
4 The Feeds & Speeds Calculator.....	24
5 Use The Pre Installed Tool Libraries.....	29
6 Add your Existing Tools to a Library.....	32
7 More about the Tools Tab.....	33
8 More about the Create/Select Tools dialog.....	37
Tool Related Answers	40
1 Where can I find Tool related Preferences?.....	40
2 How can I Print a Tool List?.....	43
3 How can I add a Custom Tool?.....	44
4 Why are my Feed Rate values too High/Low?.....	46
5 What about Tapping Feed Rates?.....	48
6 How can I Optimize Machining Time Estimates?.....	48
More Advanced Topics	49
1 How can I add Tool Comments?.....	49
2 How can add a Tool Change Point?.....	52
3 How can I add more Materials?.....	57
4 How can I enable Cutter Compensation?.....	64

Reference	67
1 MILL Holder Worksheet.....	67
2 Mill Tool Worksheets.....	68
Ball Mill Worksheet	69
Flat Mill Worksheet	70
Corner Radius Mill Worksheet	71
Vee Mill Worksheet	72
Chamfer Mill Worksheet	73
Taper Mill Worksheet	74
Thread Mill Worksheet	75
Face Mill Worksheet	76
Dovetail Mill Worksheet	77
Fillet Mill Worksheet	78
Lollipop Mill Worksheet	79
User Defined Mill Worksheet	80
Drag Knife Worksheet	81
Saw Tool Worksheet	82
Laser Tool Worksheet	83
3 Drill Tool Worksheets.....	84
Drill Worksheet	85
Center Drill Worksheet	86
Reamer Worksheet	87
Tap Worksheet	88
Bore Worksheet	89
Reverse Bore Worksheet	90
4 Turn Tool Worksheets.....	91
Diamond Insert Worksheet	92
Triangular Insert Worksheet	93
Round Insert Worksheet	94
Trigon Insert Worksheet	95
Parallelogram Insert Worksheet	96
Groove Insert Worksheet	97
Thread Insert Worksheet	98
Part Off Insert Worksheet	99
5 Feeds & Speeds Data.....	100
Feeds & Speeds Data (INCH)	100
Feeds & Speeds Data (METRIC)	101
6 Default Tool Libraries.....	103
Default English Tools Library	103
Default Metric Tools Library	106
Where to go for more help	112
Index	113

Quick Start



CAM Module 2025

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

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



What's New!

[What's New in 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

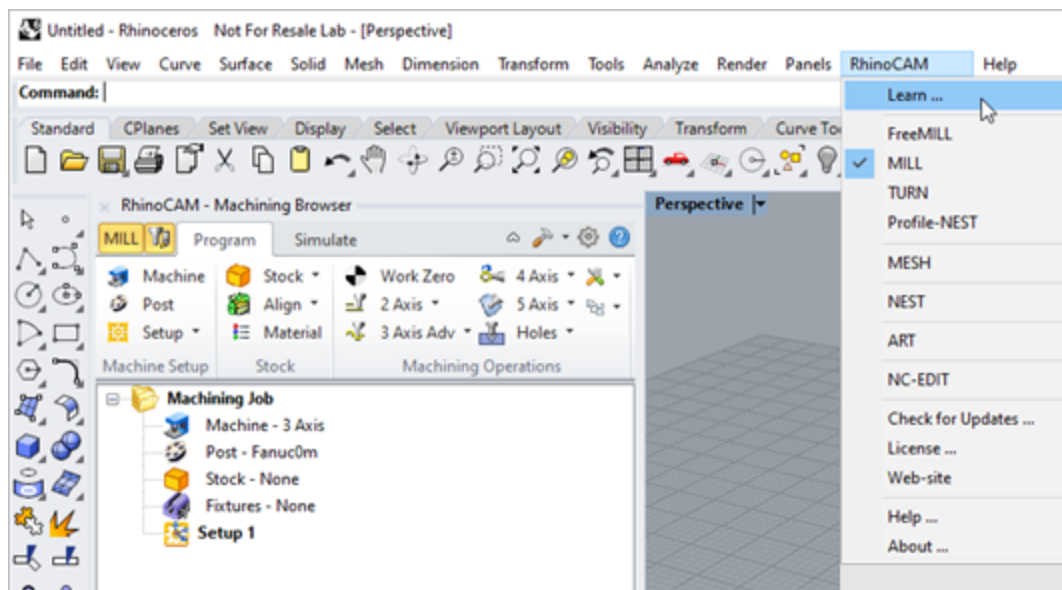
To help you quickly get started in working with each module, select one of the Help buttons located on the [RhinoCAM Learning Resources](#) dialog.

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 ...](#)

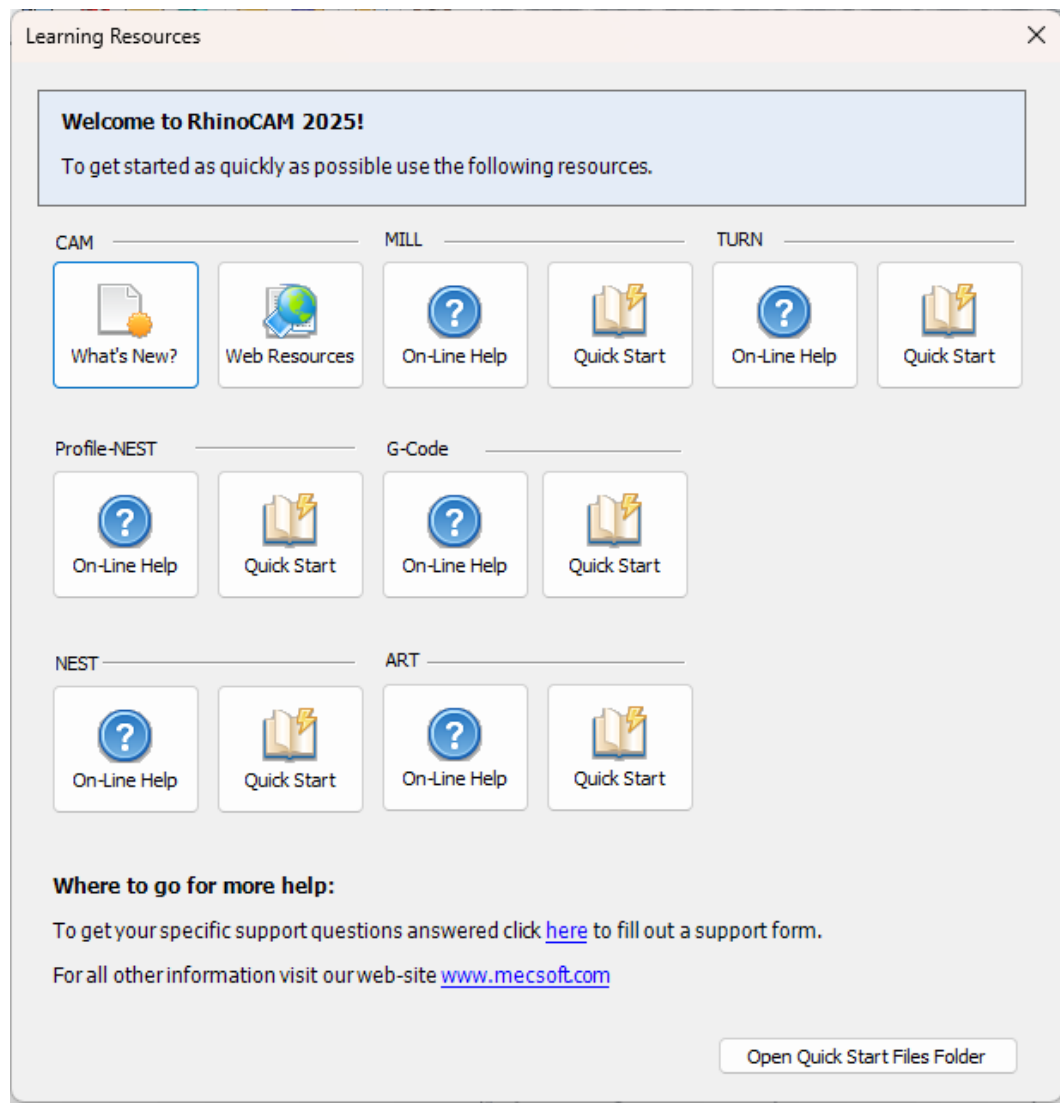


To access the Learning Resources dialog in RhinoCAM

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.



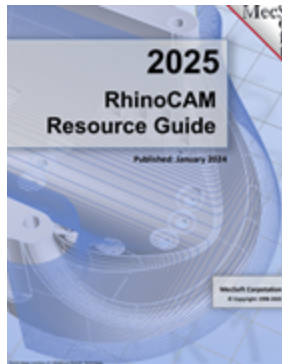
Learning Resources Dialog

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!](#)

About this Guide

RhinoCAM is a plug-in that is completely integrated within **Rhino**. To become familiar with the **MILL** module we will assume that you have previously completed one or more of [these guides](#). If you have not, please do so before reviewing this workbook.

This guide will show you how to create tools and setup your first tool library in the **RhinoCAM MILL** module. It is recommended that you do not attempt the [Advanced Topics](#) section of the guide until you are familiar working with tools and tool libraries in the **RhinoCAM MILL** module.

3.1 About the MILL Module

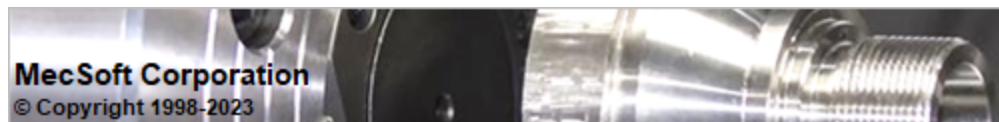


The **MILL** module offers fast gouge free solids/surface model machining technology coupled with cutting simulation/verification capabilities running inside **Rhino** for programming CNC Mills. This integration allows for seamless generation of toolpath and cut material simulation/verification within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

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

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

3.2 About the TURN Module



The **RhinoCAM-TURN** Module is a unique turning product that is used for offline programming of 2 Axis CNC Lathes. This product is a module of **RhinoCAM**, which is completely integrated inside of **Rhino**. This integration allows for the seamless generation of toolpaths and cut material simulation/verifications inside of **Rhino** for programming turning machines. **Rhino** refers to both **Rhinoceros 6** or **Rhino 7**.

The **RhinoCAM-TURN** module also comes with numerous post-processors to output the programmed G-code to some of the most popular CNC machine controllers on the market today.

Additionally, the [RhinoCAM-TURN](#) module also includes a post-processor generator that has the capability to configure entirely new post-processors. These features make programming CNC Turning easy and affordable.

The [RhinoCAM-TURN](#) module supports turning and hole making operations such as [Turn Roughing](#), [Turn Finishing](#), [Groove Roughing](#), [Groove Finishing](#), [Threading](#), [Parting](#) and [Hole](#) making operations such as [Drilling](#), [Tapping](#), [Boring](#), [Reverse Boring](#) and [Threading](#).

3.3 Using this Guide

The [Tool Library Workbook](#) will help you create your first [Tool Library](#) in [RhinoCAM MILL](#). If you have existing tool inventory in your shop, the [Tool Worksheets](#) located in the [Reference](#) section of this guide can be printed out and used to record the information and dimensions that [RhinoCAM MILL](#) uses to define tools. The Advanced Topics section is full of useful tips and procedures that you can use to get the most from your [RhinoCAM MILL](#) software.

Here is a short list of things you can do with this guide:

1. How to create [Tools](#) in [RhinoCAM MILL](#).
2. How to create, save and load a [Tool Library](#).
3. Help you gather information about your existing tool inventory and add them to a library.
4. How to use the [Feeds & Speeds Calculator](#).
5. How to perform other [Advanced](#) tool related tasks in [RhinoCAM MILL](#).

Getting Ready

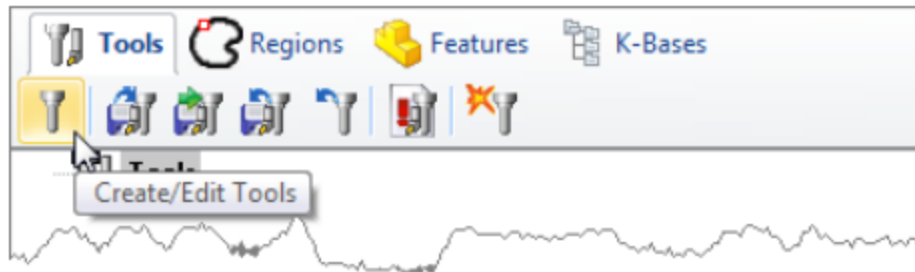
4.1 After Installing RhinoCAM MILL

After you install the [RhinoCAM](#) module, determine if you have existing tools for your CNC machine to work with. If so, go to the [Reference](#) section of this guide and print out the [Tool Worksheets](#). Use them to gather the correct information that [RhinoCAM](#) will need to add them to its [Tool Library](#). See [Add your Existing Tools to a Library](#) for more information.

4.2 Locate the Tools Tab

After installing [RhinoCAM](#) you can locate the [Tools](#) tab by referring to the image below. It is located below the [Machining Browser](#) and at the top of the [Machining Objects Browser](#).

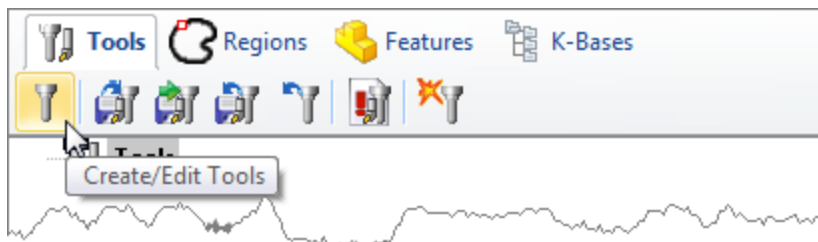
- After installing [RhinoCAM](#) load the [MILL](#) or [TURN](#) modules and locate the [Tools](#) tab shown below.



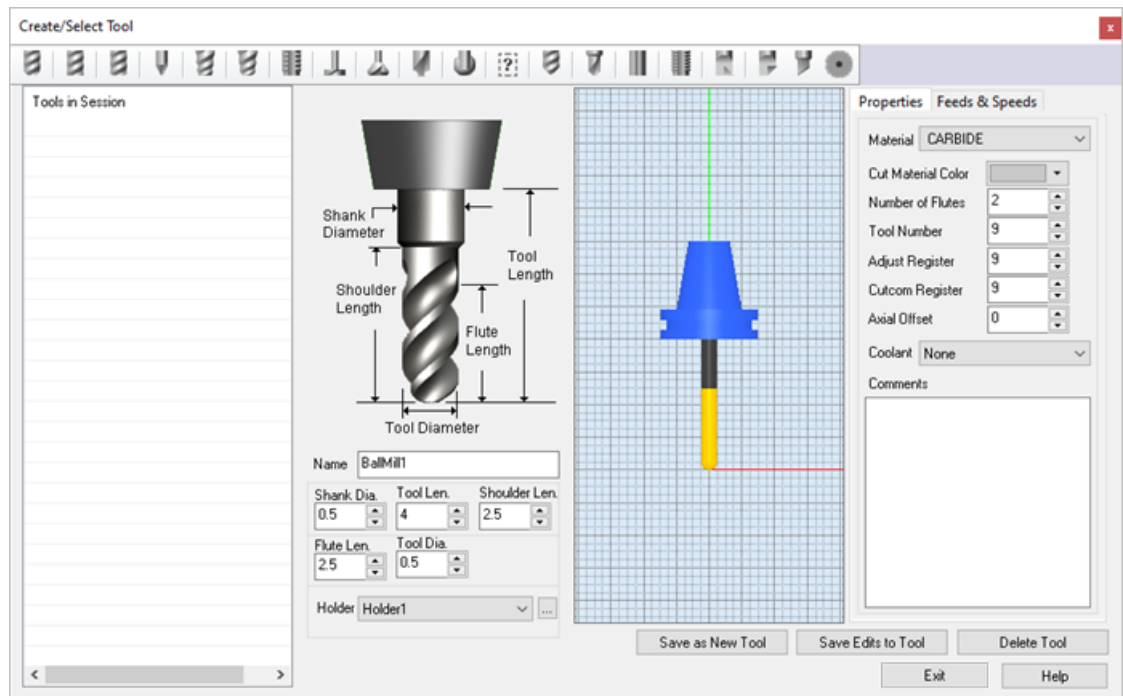
4.3 The Create/Select Tools Dialog

To display the [Create/Edit Tools](#) dialog:

1. Select the [Tools](#) tab under [Machining Objects Browser](#)
2. Select the [Create/Edit Tools](#). icon.



3. The [Create/Edit Tools](#) dialog will display.



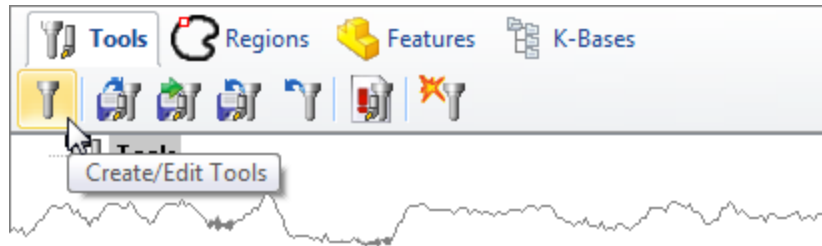
Dialog Box: Create/Select Tools

Creating Tools

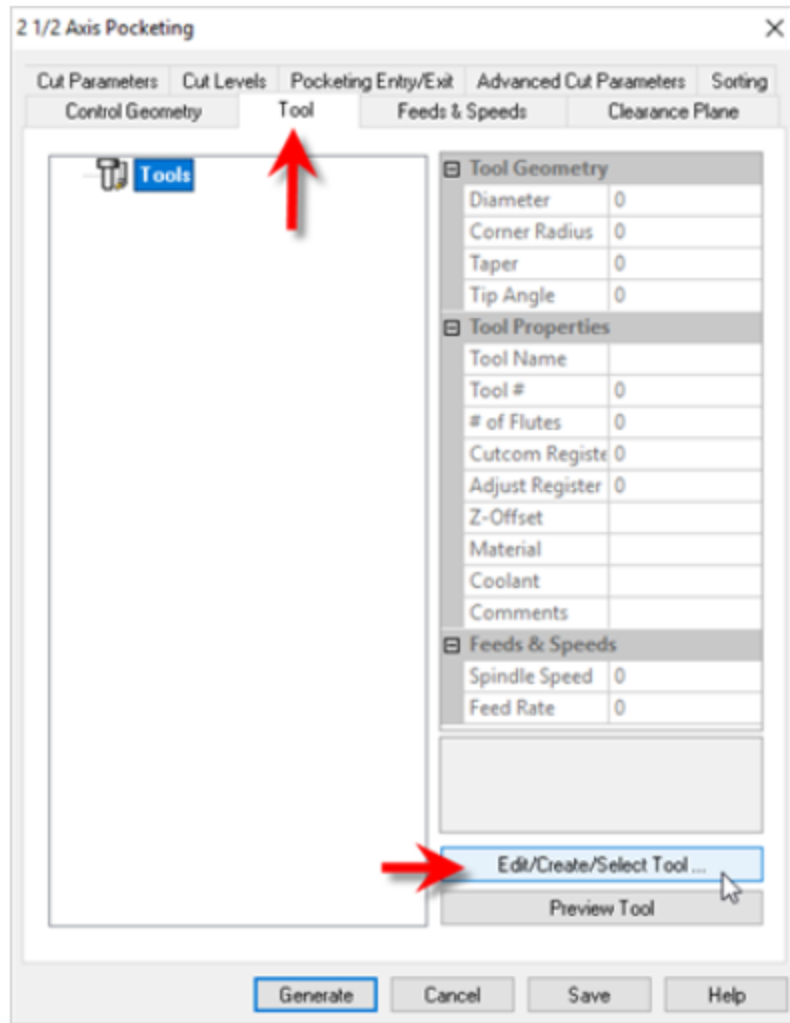
5.1 Create a Tool

You can create a tool anytime before or during the creation of a toolpath operation. Here is the basic procedure:

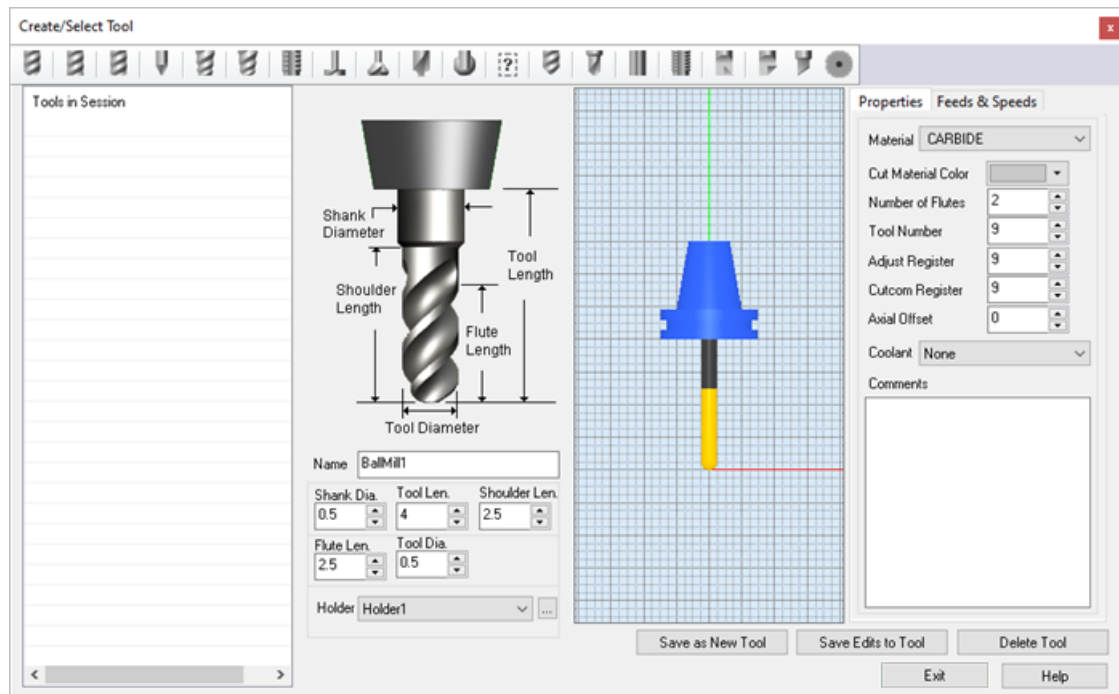
1. Select the [Create/Edit Tools](#) button from the [Tools](#) tab of the [Machining Objects Browser](#) or from the [Tools](#) tab of the toolpath operation you are in the middle of creating.



or

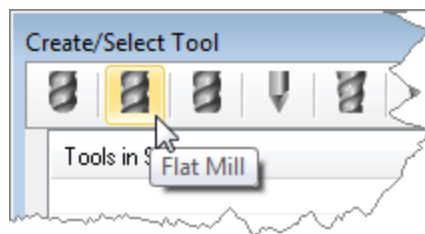


2. This will display the [Create/Edit Tools](#) dialog shown below. Notice that you can enter expressions in the parameter fields. See [Tool Related Preferences](#) to see how to enable/disable expressions in dialogs.



Dialog Box: Create/Select Tools

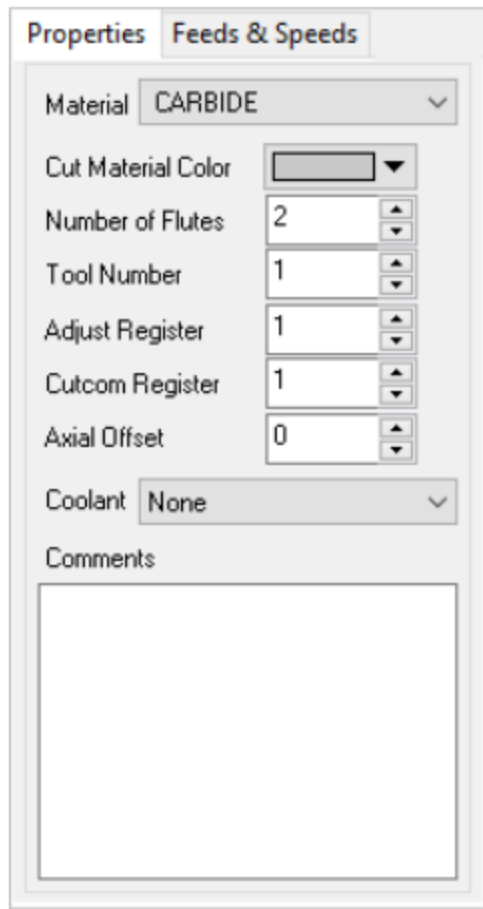
3. Select one of the **Tool Type** icons from the top of the dialog.



4. In the **Name** field enter a name for the **Tool**. A default name is provided but you should consider giving more descriptive names for your tools.
5. Enter the physical dimensions of the tool you would like to create. Default dimensions are provided. As you change the dimensions, the **Preview** of the **Tool** will update in the dialog.

However, it is very important that you enter the accurate dimensions of the actual tool you plan to use.

6. Under the **Properties** tab (on the right of the dialog) select the **Material** of the **Tool** and then enter the tool properties requested for the tool. The **Tool Number** must be unique to this tool and the **Adjust Register** and **Cutcom Register** are typically assigned the same value of the **Tool Number**. The **Number of Flutes** will affect the **Spindle Speed** and computed **Cut Feed Rate**. We'll get to those in a moment. For a definition of each parameter you can pick the **Help** button located at the bottom of the dialog. If you enter text in the **Comments** window they will be posted to your g-code file prior to the tool number being called.



The image shows a software dialog box titled 'Feeds & Speeds' with a 'Properties' tab. The 'Material' dropdown menu is set to 'CARBIDE'. Below it is a 'Cut Material Color' selection box. The 'Number of Flutes' is set to 2, 'Tool Number' is 1, 'Adjust Register' is 1, 'Cutcom Register' is 1, and 'Axial Offset' is 0. The 'Coolant' dropdown menu is set to 'None'. At the bottom, there is a 'Comments' text area.

7. Now select the [Feeds & Speeds](#) tab. Here you can manually set the [Spindle Parameters](#) and [Feed Rates](#) for this tool. If you do not know what [Feed Rate](#) values to use, select the [Load from File](#) button. It displays what we refer to as the [Feeds & Speeds Calculator](#). It will suggest values based on the [Stock Material](#), [Tool Material](#) and other [Tool](#) parameters.

Properties **Feeds & Speeds**

Spindle Parameters

Speed 14667 RPM

Direction ☒ CW ☐ CCW

Feed Rates (in/min)

Plunge 23.468 Approach 11.734 Engage 8.8

Cut 11.734 Retract 8.8 Departure 23.468

Transfer

☒ Use Rapid ☐ Set 23.468

Feed Rate Reduction Factors

Plunge between levels 100 %

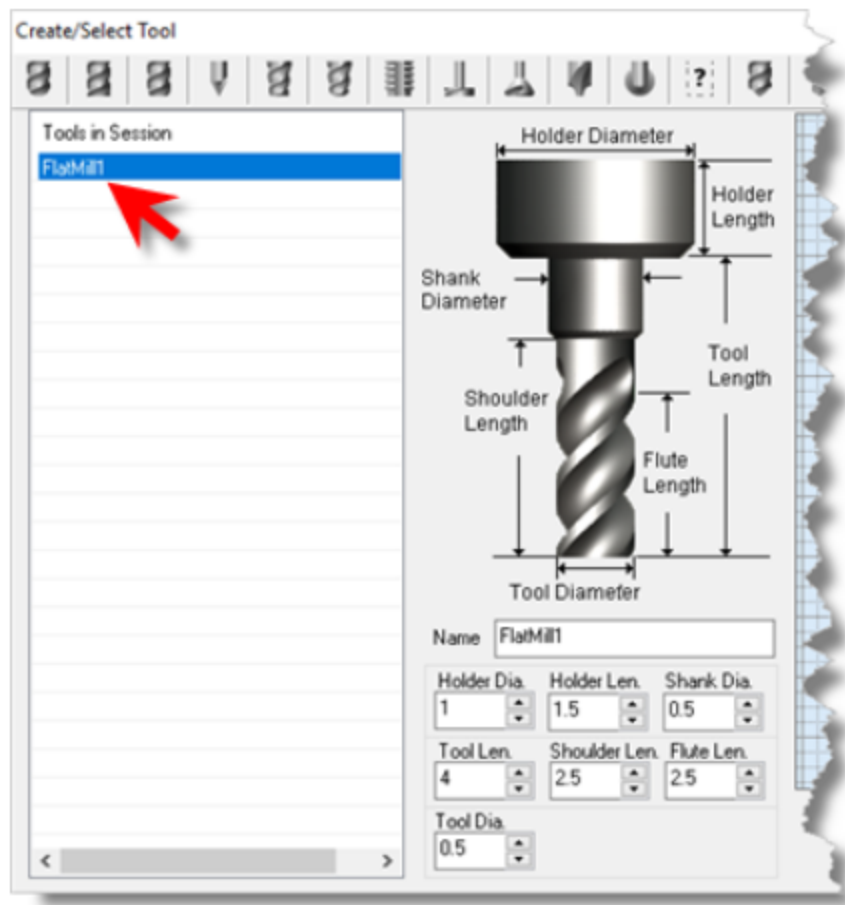
First XY pass 100 %

Bottom Z Level 100 %

Cut Depth 0 in

Load from File ...

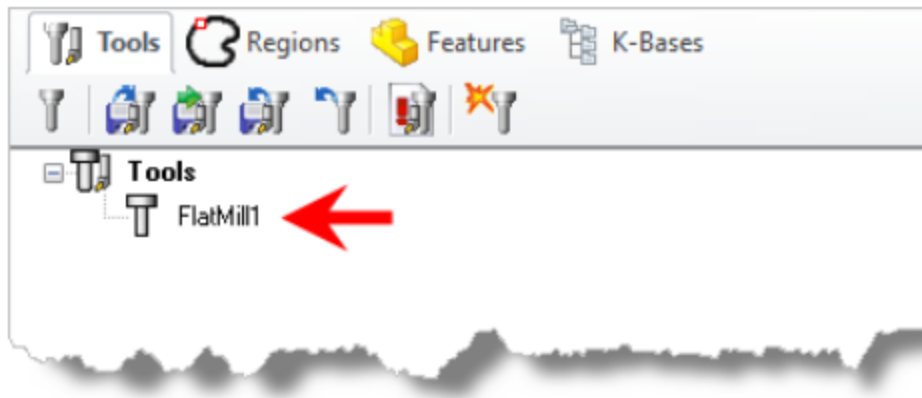
- When you are done defining your tool's parameters, select the [Save as New Tool](#) button. The new tool is created and added to the Tools in Session list located on the left side of the dialog. This means that your tool is defined in your current session of [RhinoCAM](#) ONLY. When you save your part file, the tool will be saved along with it.



9. **IMPORTANT:** After your tool is listed in the [Tools in Session](#) list, if you make ANY changes to the tool parameters, you must select the [Save Edits to Tool](#) button for your revisions to be saved.

! You can edit the tool properties and click [Save Edits to Tool](#) to save the changes. You can create additional tools by assigning a different name and specify the tool parameters.

10. When done, pick the [OK](#) button to close the [Create/Edit Tools](#) dialog.



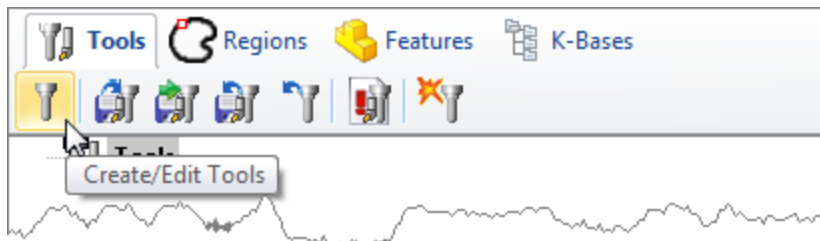
! To save **Tools** to a library, Click **Save Tool Library** under the **Tools** tab in the **Machining Objects Browser** and specify a folder location and file name in the **Save as** dialog box. Saving a **Tool** library as a **Knowledge base** file (*.vkb) saves feeds and speeds with tool properties.

See [Create a Tool Library](#) for instructions on how to do this.

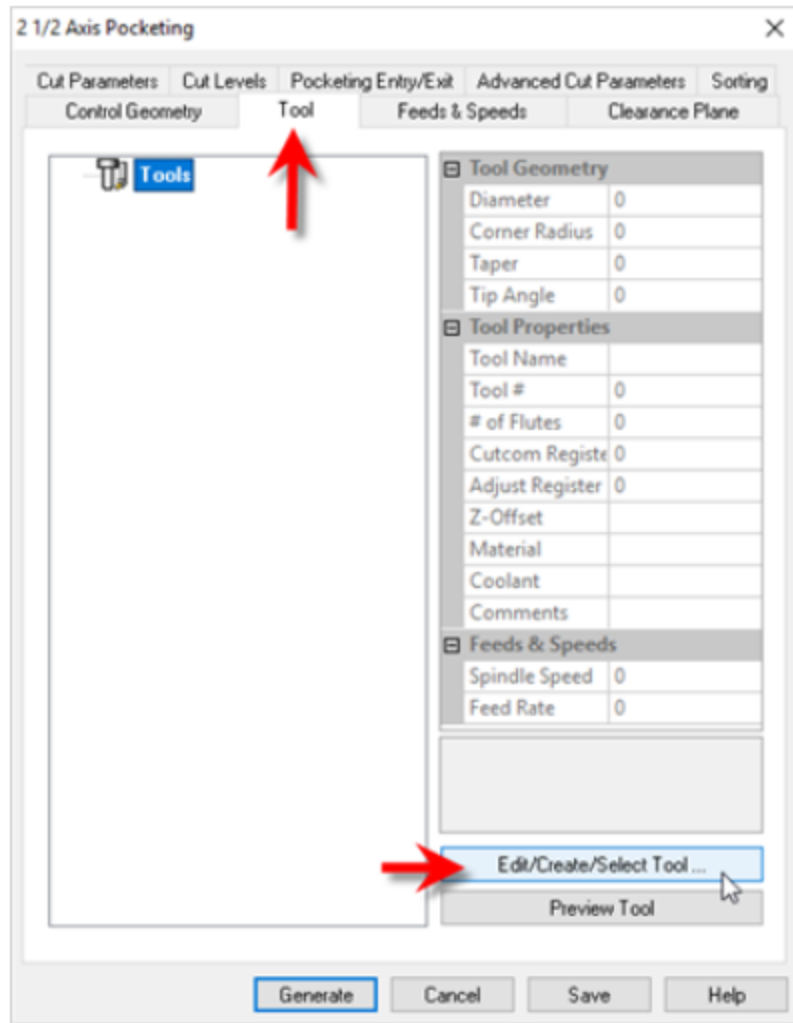
5.2 Create a Tool Holder

You can create a **Tool Holder** anytime before or during the creation of a toolpath operation. Here are the basic steps.

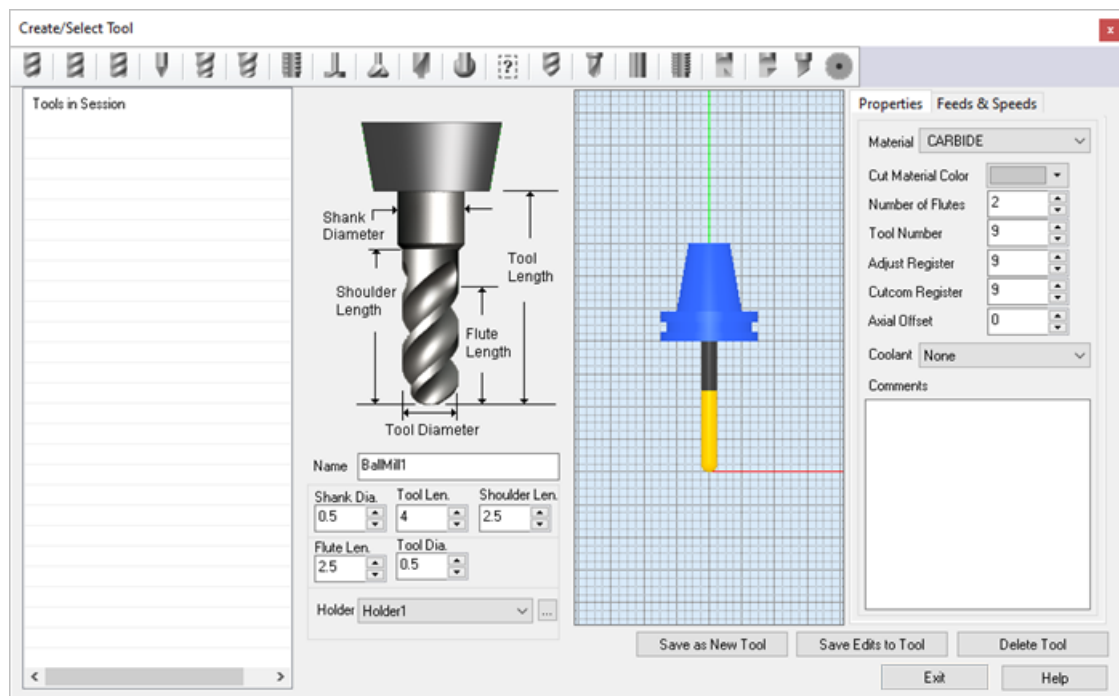
1. Select the **Create/Edit Tools** button from the **Tools** tab of the **Machining Objects Browser** or from the **Tools** tab of the toolpath operation you are in the middle of creating.



or

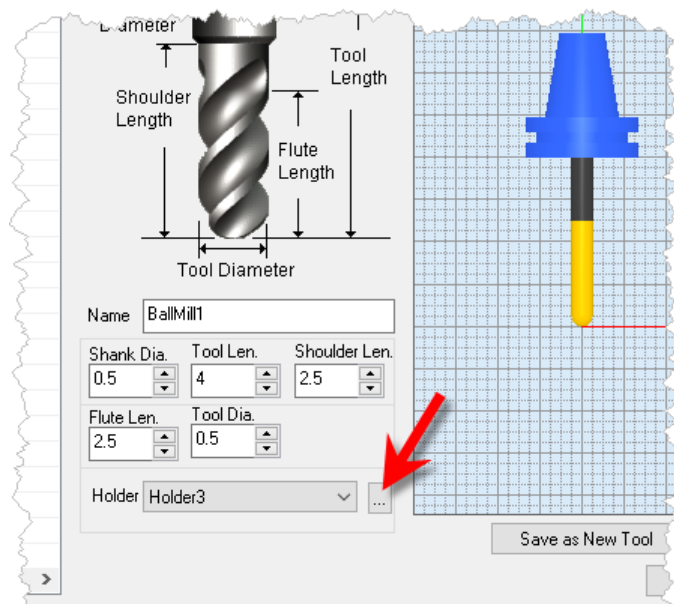


2. This will display the [Create/Edit Tools](#) dialog shown below. Notice that you can enter expressions in the parameter fields. See [Tool Related Preferences](#) to see how to enable/disable expressions in dialogs.



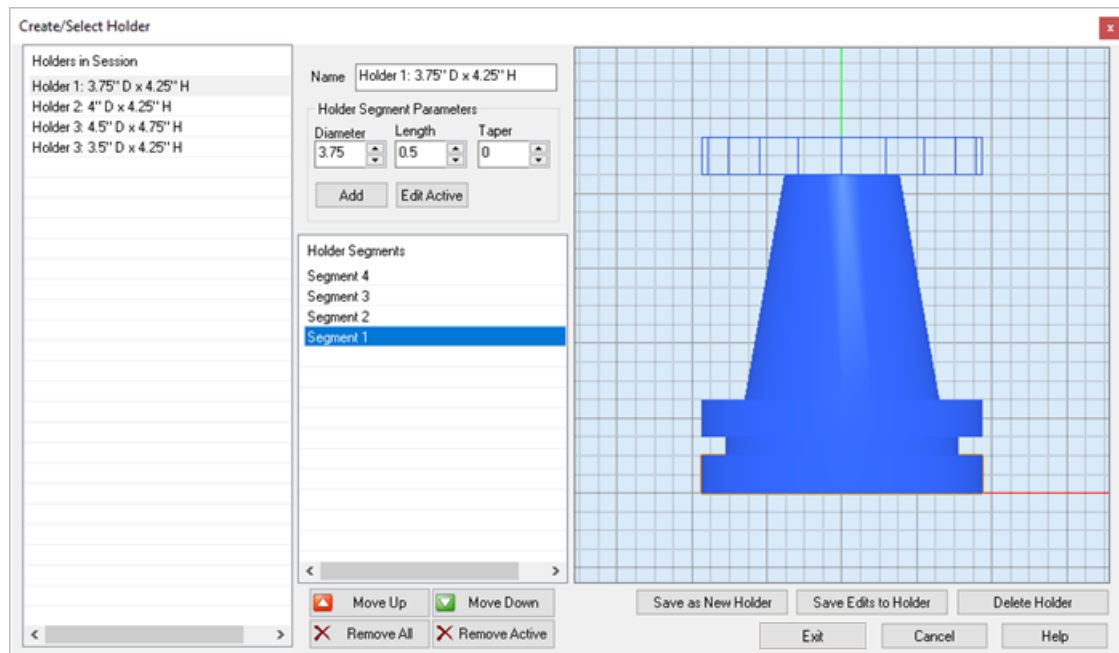
Dialog Box: Create/Select Tools

- To create or edit your holders, pick the ... button to display the [Create/Select Holder](#) dialog. Holders are save with your tool library.



Location of the Holders menu

- Use the [Create/Edit Tool Holder](#) dialog to define your Tool Holders.



Dialog Box: Create/Edit Tool Holders

5. In the **Name** field enter a name for the **Tool Holder**. The name must be unique to the holder being defined.
6. Tool Holders are created in segments. Each segment has three dimensions. These are: **Diameter**, **Length** and **Taper**.
7. When you are satisfied with your segment dimensions, select the **Add** button and the segment will be added to the **Holder Segments** list.
8. To edit a segment, first select it from the list and then pick the **Edit Active** button.
9. Select a segment and then use the **Move Up** and **Move Down** buttons to arrange the segments in the proper order that defines the holder being created.
10. A preview of the active **Tool Holder** is displayed on the right.
11. Use the buttons under the preview window to **Save**, **Edit** or **Delete** the the active **Tool Holder**.
12. When your holder is defined, select the **Exit** button to return to the **Create/Select Tool** dialog.
13. To use a Holder, first select a **Tool** or create one using the **Create/Select Tool** dialog.
14. Then drop down the **Holder** menu and select a **Holder** for that Tool.

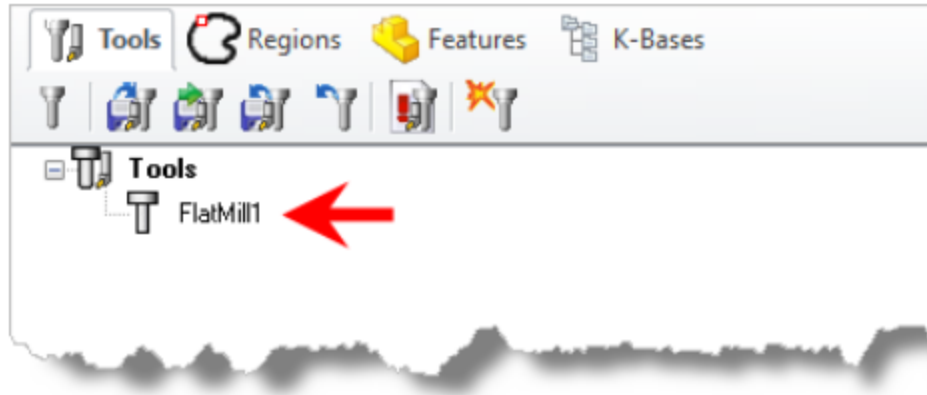
5.3 Create a Tool Library

When you create tools they are only saved with the part file you are working on. To reuse tools efficiently in **RhinoCAM**, from one job to the next, you should create a **Tool Library** that contains all of the tools in your CNC shop's inventory.

! RhinoCAM automatically installs a [Tool Library](#) that you can customize with your tools. It is available in both [English](#) and [Metric](#) versions. See the topic [Use The Pre-Installed Tool Library](#) for more information.

The steps below will walk your through creating your own tool library.

1. Complete the steps in the topic [Create a Tool](#) to create to create your first tool. It should be listed under the [Tools](#) tab:



2. From the [Tools](#) tab menu select the [Save Tool Library](#) icon from the menu:



3. From the [Save As](#) dialog that displays, change the [Save as Type](#) selector to [*.vkb](#). Both formats ([*.vkb](#) or [*.csv](#)) allow the [Feeds & Speeds](#) parameters to be saved along with each tool definition but the [*.vkb](#) format is recommended.

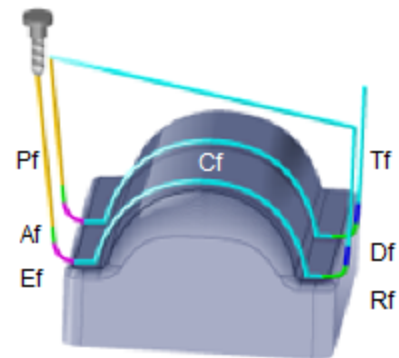
! Select a folder location that is unique to your company or computer and NOT located within the install path of [RhinoCAM](#). For example this can be a network drive or even just your [Desktop](#).

4. Now enter a name for your [Tool Library](#) (i.e., [My_First_Tool_Library.vkb](#)) and pick [Save](#).
5. See the topic [Tool Related Preferences](#) to learn how to make your [Tool Library](#) load automatically every time a new part is created in [Rhino](#).

5.4 The Feeds & Speeds Calculator

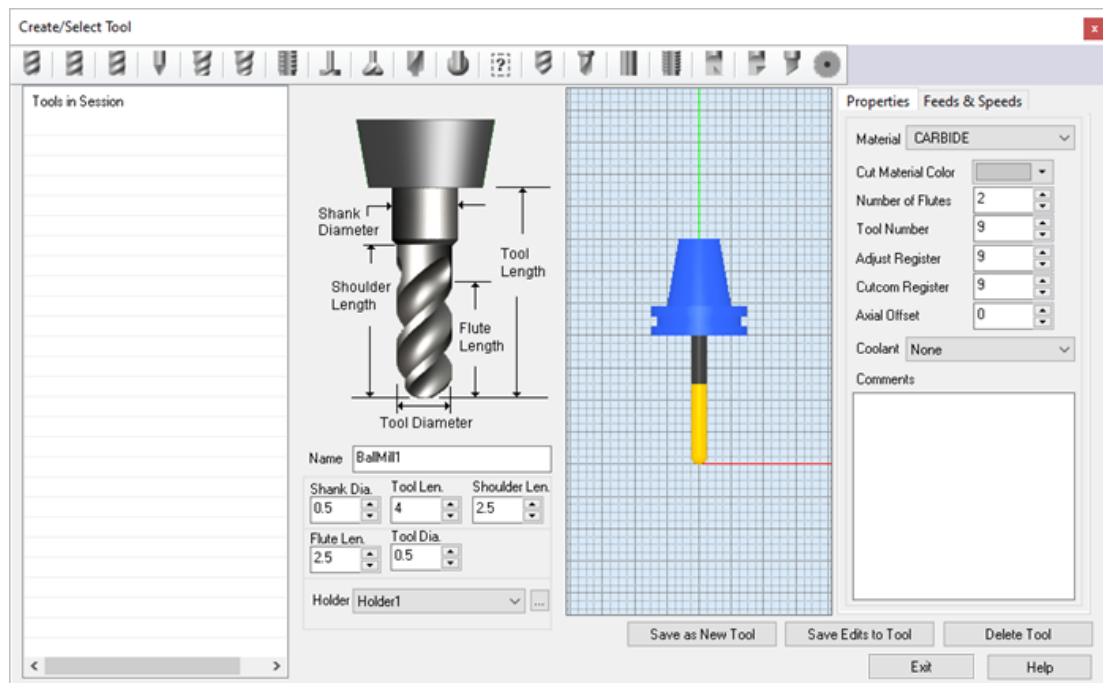
The **MILL Module** has a built-in **Feeds & Speeds Calculator**. That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a **Cut Feed** is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including **Plunge**, **Approach**, **Engage**, **Retract** and **Departure**! The percentages of the **Cut Feed** to assign are all controlled from the **CAM Preferences** dialog.

Of course you can override any of these calculated values at any time, for any tool and for any operation.



Feeds & Speeds Associated with a Tool

In the **MILL Module**, feeds & speeds can be defined and associated with a specific tool. This allows you the flexibility to have different tool definitions based on the type of material being cut (i.e., steel, wood, acrylic, etc.) or the operation type (**Pocketing**, **Facing**, etc.). The **Create/Select Tool** dialog includes a **Feeds & Speeds** tab where these values are defined. When the tool is saved, the feeds & speeds values are saved with it.

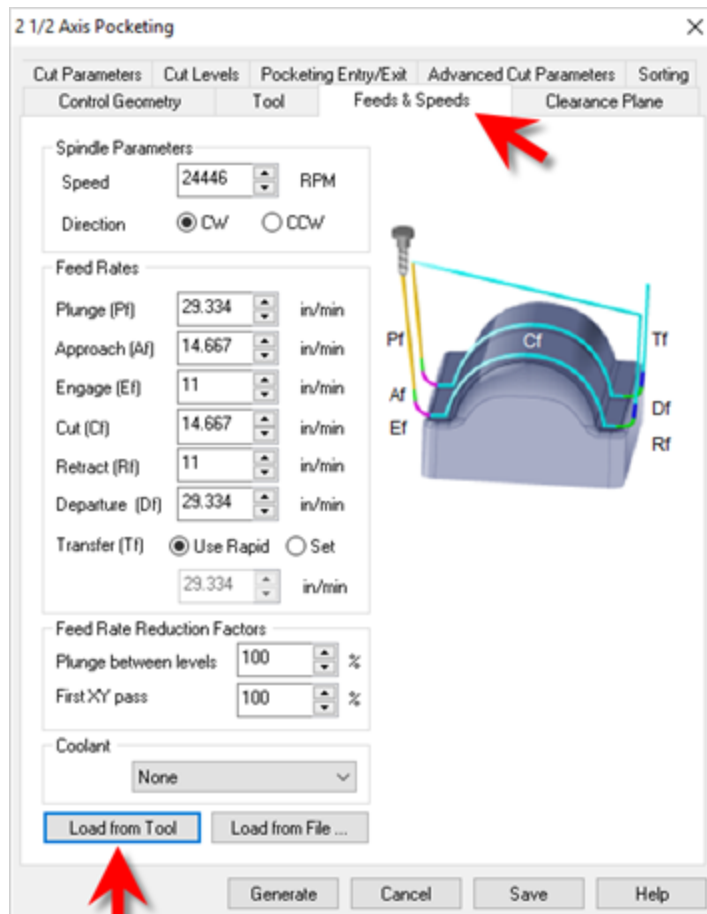


Dialog Box: Create/Select Tools

Feeds & Speeds Associated with an Operation

Each toolpath operation type also has a **Feeds & Speeds** tab in its dialog. This allows you the flexibility to assign feeds & speeds values specific to that operation. The **2½ Axis**

Pocketing operation dialog is shown here.



Each **Feeds & Speeds** tab also contains a button called **Load from Tool**. This allows you to load the feeds & speeds values that you have previously associated with active tool. The **Active Tool** simply refers to the tool that you have selected from the **Tools** tab of the dialog.

As you can see, just between the tool and the operation type, you have a wide range of flexibility for defining your feeds & speeds strategy.

The Feeds & Speeds Calculator

You may have noticed that both the **Feeds & Speeds** tab of the **Create/Select Tool** dialog and the **Feeds & Speeds** tab of the operation dialog have a button called **Load from File ...**. Selecting this button will display the **Feeds & Speeds Calculator** dialog. This dialog allows you to load feeds & speeds values that are calculated from information stored in an external **Materials XML** file.

How does it Work?

There are two parts to the calculator.

Data from Table

This information is extracted from the default [Materials XML](#) table file and is based on the following parameters:

1. The [Stock Material](#) is selected from the [Materials](#) dialog located on the [Program](#) tab.
2. The [Tool Material](#) is selected from the [Properties](#) tab of the [Create/Select Tools](#) dialog.

The values for [Surface Speed](#) (measured in units per minute) and [Feed per Tooth](#) (measured in units) are retrieved from the [XML](#) file and displayed in the dialog. [Units](#) refers to the part file's current Units setting.

Input Variables

The values for [Tool Diameter](#) and [# of Flutes](#) are automatically loaded based on the tool selected for the operation (i.e., the active tool). Based on these parameters, the program computes a [Spindle Speed](#) value measured in [RPM](#) ([Rotations per Minute](#)).

Computed Cut Feedrate

A [Cut Feed](#) value is also calculated for you (measured in [Units/Minute](#)) based on [Stock Material](#), [Tool Material](#), [Tool Diameter](#), and [# of Flutes](#). Changing the [Spindle Speed](#) updates the [Cut Feed](#) and vice versa.

When you pick OK from this dialog, the computed [Feeds & Speeds](#) values are submitted to either the active tool or the active toolpath operation.



Formulas Used for Cut Feed Computation

Units in Inches	
Spindle Speed (RPM)	$\text{Surface Speed (SFM)} \times 12 / (\pi \times \text{Tool Diameter(in)})$
Cut Feed (IPM)	$\text{Feed Per Tooth(IPT)} \times \text{\# of Flutes} \times \text{Spindle Speed(RPM)}$
SFM: Surface Feet Per Minute IPT: Inches Per Tooth RPM: Revolutions per Minute IPM: Inches Per Minute	
Units in Millimeters	
Spindle Speed (RPM)	$\text{Surface Speed (SMM)} * 1000 / (\pi \times \text{Tool Diameter (mm)})$
Cut Feed (IPM)	$\text{Feed Per Tooth(MMPT)} \times \text{\# of Flutes} \times \text{Spindle Speed(RPM)}$

SMM: Surface Meters Per Minute

MMPT: Millimeters Per Tooth

RPM: Revolutions per Minute

MMPM: Millimeters Per Minute



Let's Review:

1. You can associate unique feeds & speeds values with each tool or with each operation.
2. You can invoke the [Feeds & Speeds Calculator](#) from either the [Create/Select Tool](#) dialog or from any toolpath operation dialog.
3. The [Feeds & Speeds Calculator](#) extracts data from an external [XML](#) file and combines it with your stock material and tool parameters to calculate suggested [Spindle Speed](#) and [Cut Feed](#) values.
4. Changing parameters such as [Tool Diameter](#), [Surface Speed](#), [Material](#), etc. will calculate new [Spindle Speed](#) and [Cut Feed](#) values automatically.
5. A percentage of the calculated [Cut Feed](#) can be assigned for [Plunge](#), [Approach](#), [Engage](#), [Retract](#) and [Departure](#) motions. These percentages are set in the [CAM Preferences](#) Dialog.
6. You can override [ANY](#) or [ALL](#) of the suggested feeds & speeds values at any time!

More information about the Load from File button:

This loads the [Feeds & Speeds](#) values from the [Feeds & Speeds Table](#) file. This will display the [Load Feeds from Table](#) dialog box to make your selections.



Dialog Box: Load Feeds from Table

Selecting [OK](#) from this dialog transfers the spindle speed and cut feedrate to the [Feeds & Speeds](#) tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under [Feeds & Speeds Preferences](#).

The image shows a 'Feeds/Speeds' dialog box with a 'Load Feeds from Table' tab. It contains several sections for configuring machining parameters:

- Data from Table:**
 - Stock Material: ALUMINUM - 2024
 - Tool Material: CARBIDE
 - Surface Speed: 1600 ft/min
 - Feed/Tooth: 0.004 in
- Input Variables:**
 - Tool Diameter: 0.5 in
 - # of Flutes: 2
- Maximum Limits for Computation:**
 - Max Spindle Speed: 14000 RPM
 - Max Cut Feed: 200 in/min
- Computed Variables:**
 - Spindle Speed: 12223 RPM
 - Cut Feed (Cf): 97 in/min

At the bottom are 'OK', 'Cancel', and 'Help' buttons.

Dialog Box: Load Feeds from Table



Data from Table

Stock Material

Selecting a [Stock Material](#) and [Tool Material](#) displays the [Surface Speed](#) and [Feed/Tooth](#). This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a [Stock Material](#) and [Tool Material](#) displays the [Surface Speed](#) and [Feed/Tooth](#). This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a [Stock Material](#) and [Tool Material](#) displays the [Surface Speed](#) and [Feed/Tooth](#). This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a [Stock Material](#) and [Tool Material](#) displays the [Surface Speed](#) and [Feed/Tooth](#). This information is contained in a feeds and speeds data file which can be edited to add newer materials.



Input Variables

The input variables - [Work Diameter](#) is automatically loaded from the Stock Radius. Based on this parameter and the [Variables Limits](#) parameters, the program computes [Spindle Speed](#) and [Cut Feedrate \(Cf\)](#). measured in [Unites/Revolution](#). Changing the spindle speed modifies the cut feedrate.



Maximum Limits for Computation

Here you can set the [Max Spindle Speed](#) and [Max Cut Feed \(Cf\)](#) values. Once these two values are set, the [Spindle Speed](#) and [Cut Feed](#) calculated by this dialog will not exceed these values even if you attempt to enter higher values into the [Computed Variables](#) fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.



Computed Variables

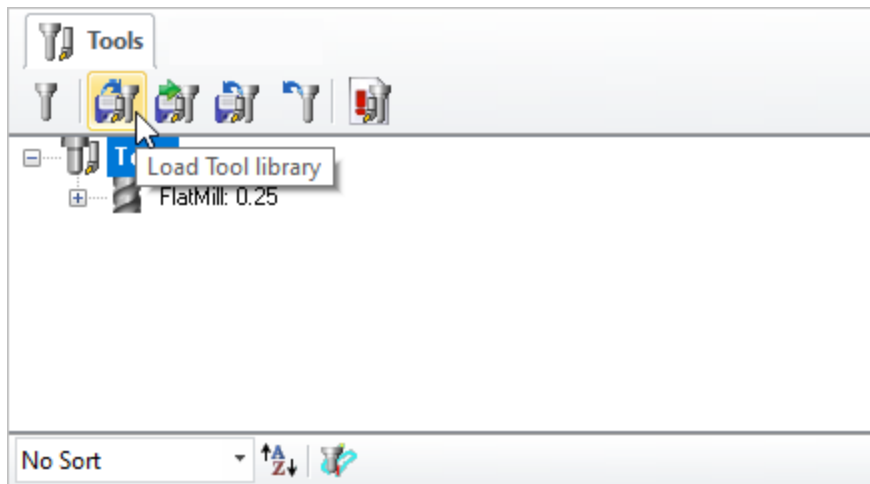
The variables for [Spindle Speed](#) and [Cut Feed \(Cf\)](#) are computed for you based on the selections made in this dialog but will not exceed the values set in the [Maximum Limits for Computation](#) section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a [Feeds & Speeds Calculator](#), you cannot override both values. To do so, you must edit the operation or tool parameters manually.

5.5 Use The Pre Installed Tool Libraries

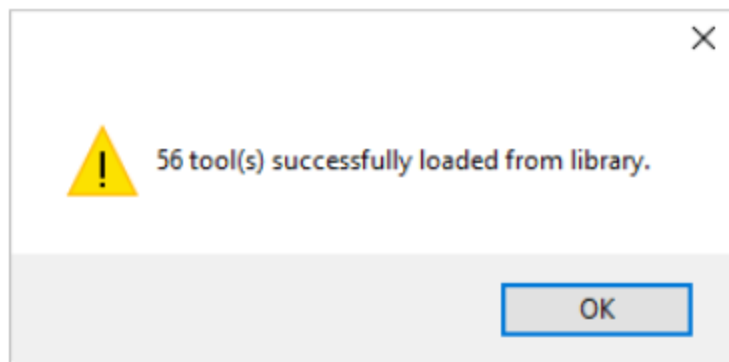
[RhinoCAM MILL](#). comes with two [Tool Libraries](#) ([INCH](#) and [METRIC](#)) each containing over 50 [Tools](#) that you can use and modify to create your own customized [Tool Library](#). To see a complete list of the tools in these default libraries, visit the [Default Tool Libraries](#) section of this guide.

How to load the default Tool Library:

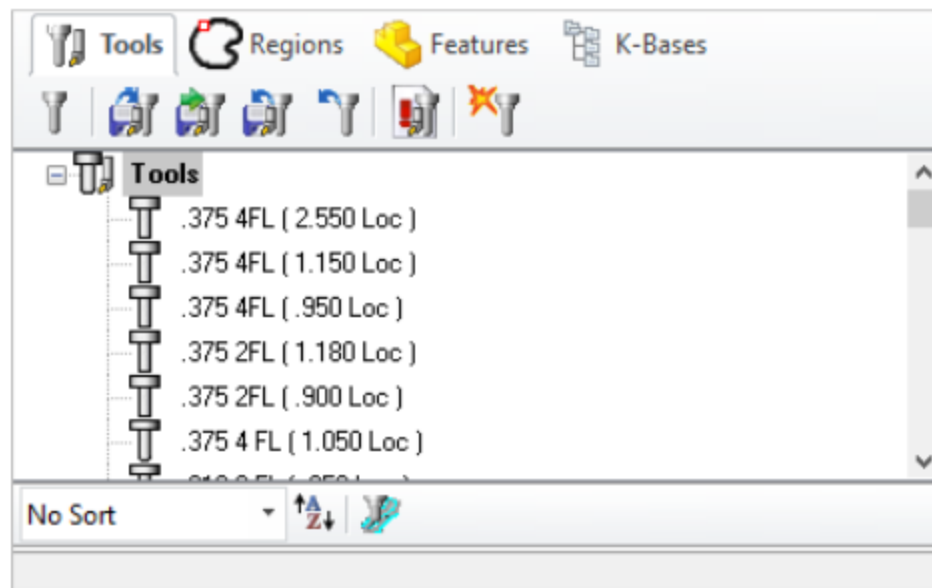
1. Go to the [Tools](#) tab and select the [Load Tool Library](#) icon.



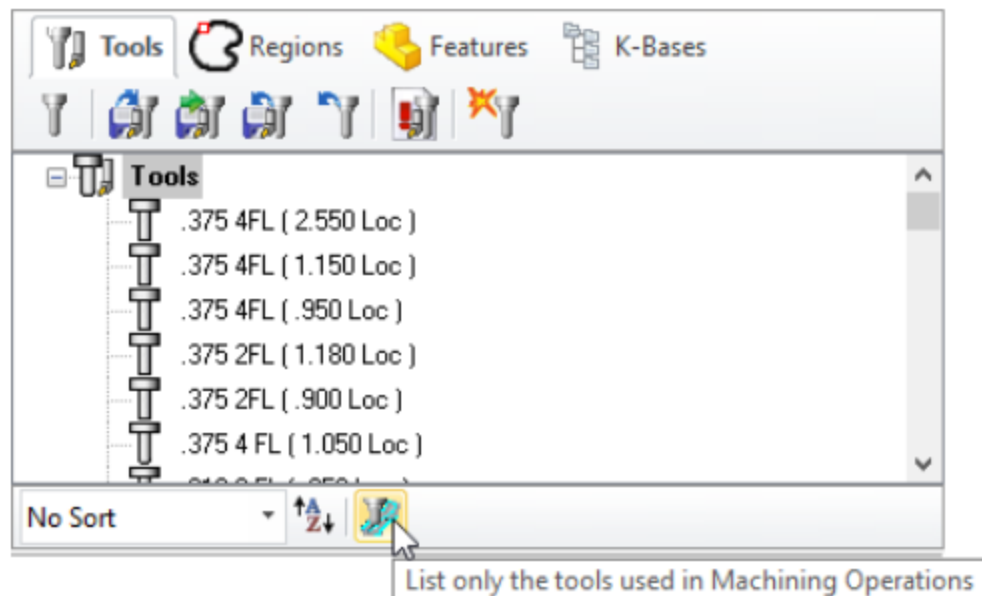
2. Navigate to the `/Data` folder of the install path in RhinoCAM.
By default it is located at: `C:\ProgramData\MecSoft Corporation\RhinoCAM 20xx for Rhino x\Data`
3. In the **File Open** dialog, change the file type to **Tool Library Text Files (*.csv)**.
4. Select the `DefaultEnglishTools.csv` or the `DefaultMetricTools.csv` file and then pick **Open**.
5. The message will inform you of the total number of tools that were loaded.



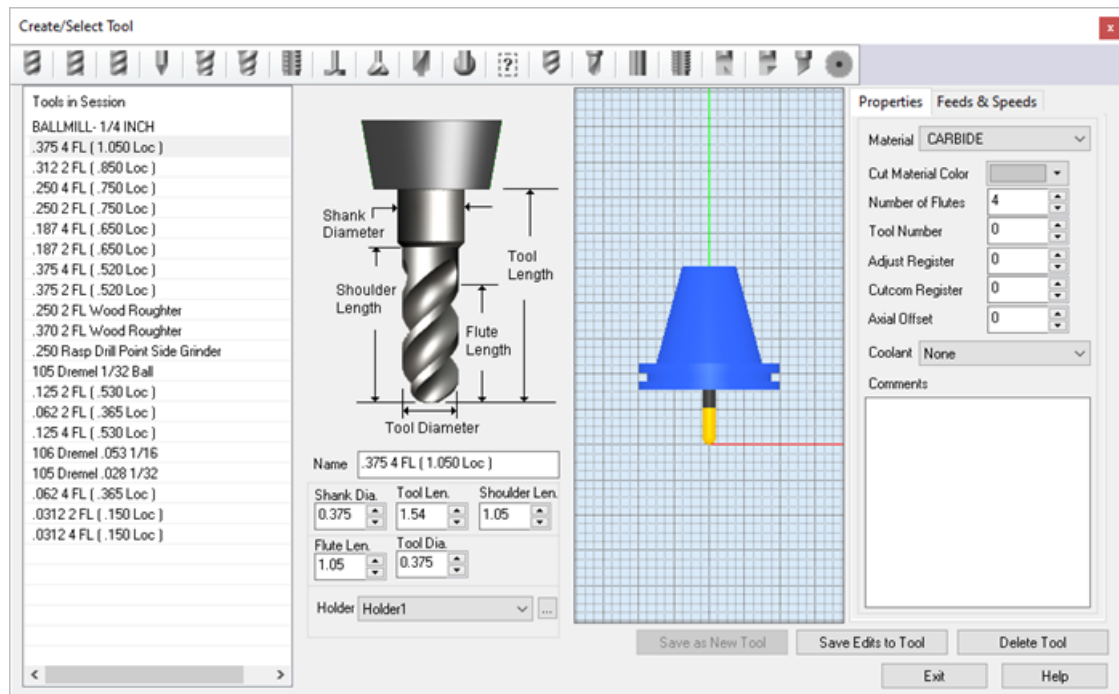
6. The tools in the library will also be listed in the **Tools** tab.



7. If you DO NOT see the tools listed, make sure the icon for [List tool only used in operations](#) IS NOT toggled on.



8. You can now edit each of these tools as needed to start your own Tool Library!



Dialog Box: Create/Select Tools

5.6 Add your Existing Tools to a Library

If you have existing tool inventory in your shop, the [Tool Worksheets](#) located in the [Reference](#) section of this guide can be printed out and used to record the information and dimensions that [RhinoCAM MILL](#) uses to define tools.

Here are the basic steps to get started:

1. Go to the [Tool Worksheets](#) located in the [Reference](#) section of this guide and print out the [Worksheet](#) for each tool type.
2. Fill out a worksheet for each of the tools in your existing shop's inventory. Each worksheet lists the key dimensions that are needed to define each tool in [RhinoCAM](#). Here is an example worksheet:

Vee Mill

Holder Taper, Holder Diameter, Holder Length, Shank Diameter, Shoulder Length, Flute Length, Taper Angle, Tool Diameter

Name: _____

Holder Dia.	Holder Len.	H. Taper
Shank Dia.	Tool Len.	Shoulder Len.
Tool Dia.	Taper Angle	

Material: _____

Cut Material Color: _____

Number of Flutes:

Tool Number:

Adjust Register:

Cutcom Register:

Axial Offset:

Coolant: None

Comments: _____

Spindle Parameters

Speed: RPM

Direction: ☒ CW ☐ CCW

Feed Rates (in/min)

Plunge: Approach: Engage:

Cut: Retract: Departure:

Transfer: ☒ Use Rapid ☐ Set

Feed Rate Reduction Factors

Plunge between levels: %

First XY pass: %

Bottom Z Level: %

Cut Depth: in

Load from File ...

This is a worksheet to help you document tool parameters. This IS NOT a RhinoCAM menu!

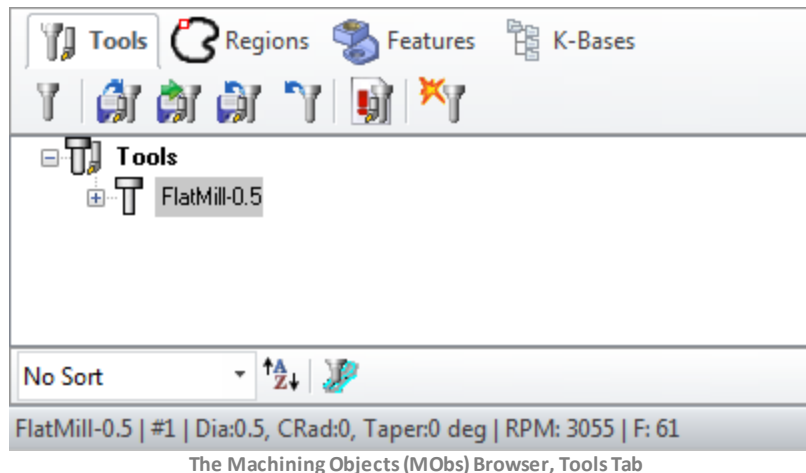
3. With worksheets in hand, Create a Tool or load and/or create a [Tool Library](#). See [Create a Tool](#) and [Create a Tool Library](#) in this guide for more information about how do this.
4. **Note:** The tool spec sheet from your tool vendor will have a lot of information. Pay close attention to which dimensions you need to write down. For example, if your tool has an insert (i.e., a boring tool), you only need to document the total [Tool Diameter](#).
5. When done entering all of your existing tools be sure to [Save your Tool Library](#) and use a unique file name and location outside of [RhinoCAM](#) install path.

5.7 More about the Tools Tab

Here is additional information about the [Tools](#) tab:



[The Machining Objects \(Mobs\) Browser, Tools Tab](#)



RhinoCAM supports 2 types of tool library file format ***.vkb** and ***.csv** (*.vkb is recommended).



Tools Tab Functions

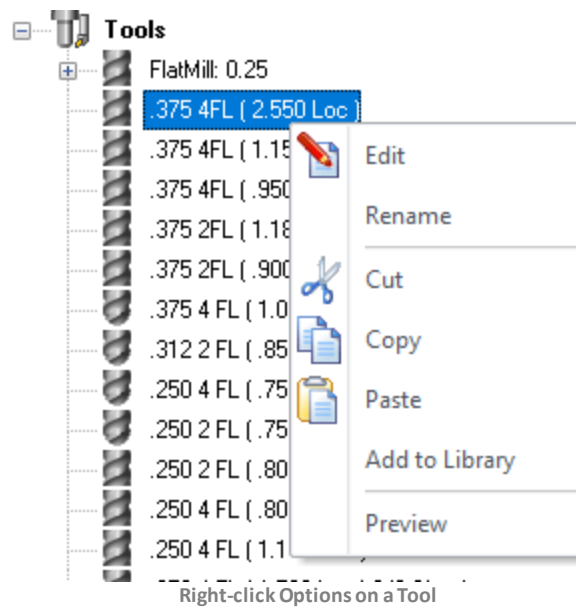
Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Tools Tab Functions					
	✓	✓	✓	✓	✓
	Create/Edit Tools: This button brings up the tool dialog that enables the creation and saving of tools. All milling, drilling and user defined tools can be created here. Refer to Tool section for a detailed description on creating tools and defining tool parameters.				
	✓	✓	✓	✓	✓
	Load Tool Library: The load tool library button enables the loading of a previously saved tool library. Refer to the following section for additional information - Load Tool Library				
	✓	✓	✓	✓	✓
	Select Tools from Library: The select tool library button enables you to select tools from a previously saved tool library. Refer to the following section for additional information - Select Tools from Library				
	✓	✓	✓	✓	✓
	Save Tool Library: This button enables the created tools to be saved in a tool library file. The file can be saved in the desired				

	directory and read in when required. Refer to the following section for additional information - Save Tool Library				
	✓	✓	✓	✓	✓
	Unload Tool Library: This button will unload the current Tool Library .				
	✓	✓	✓	✓	✓
	List Tools: The button brings up all the tool properties associated with the tools currently recorded in the current MILL session. Refer to the following section for additional information - List Tools				
				✓	✓
	Compute Tool Holder Collisions*: Determines tool holder collision with the part geometry. Refer to the following section for additional information - Compute Tool Holder Collisions				



Right-click Options on Tools

You can right-click on a [Tool](#) listed in the [Mobs Browser](#) to perform various functions. These are listed below:



Edit

Displays the [Create/Edit Tool](#) dialog allowing you to edit the [Tool](#) parameters.

Rename

Allows you to [Rename](#) the selected tool.



Cut / Copy / Paste



These options allow you to **Cut** or **Copy** the selected **Tool** to the **Windows Clipboard** and then **Paste** it back to the Tools list to create a new tool using the previous tool as a template.



Add to Library

This allows you to **Add** the selected **Tool** to an exiting **Tool Library *.csv** data file.

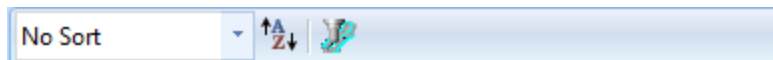
Preview

This will display a **Preview** of the selected **Tool** in the **Graphics Window** similar to how the **Tool** displays during **Simulation**. The **Tool** will display at the origin of the **MCS** for the current operation.



Tools Toolbar Functions

The following **Tool Sorting** rules (when set) will apply to both the **Tools** tab of the **Machining Objects Browser** and the **Create/Select Tools** dialog.



Sorting Selector: This allows you to sort the tool list. You can select **No Sort** or sort by **Name**, **Number**, **Type** and **Diameter**.



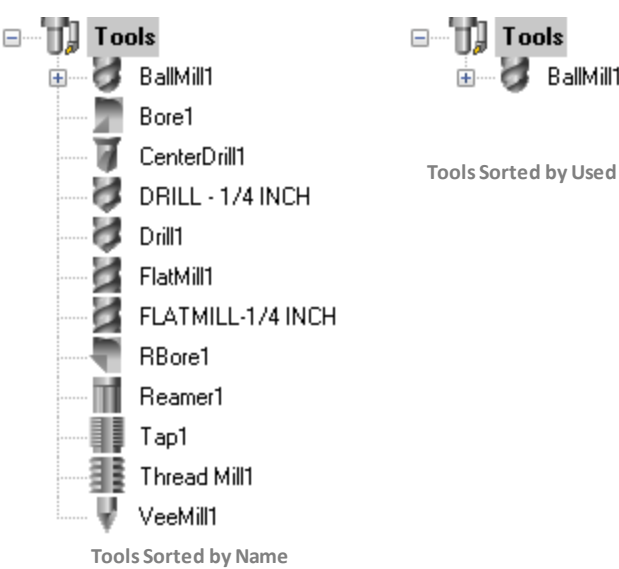
Sort in Ascending/Descending Order: This icon acts like a toggle to switch between Ascending and Descending sort order.



List on the Tool used in Machining Operations: Toggle this icon to list **ONLY** the tools currently assigned to an operation. **Note:** You must **Generate** an operation for the assigned tool to be listed.

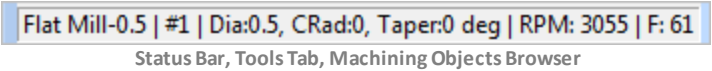


If you do not see any of your tools listed, check to make sure this icon is toggled OFF. If no operations are using tools yet and this icon is ON, then no tools will be listed!



 Tools Status Bar

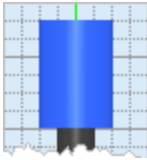
The status bar displays the currently selected tool, tool tip radius & angle, spindle speed and cut feedrate.

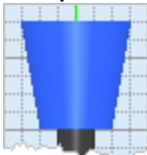
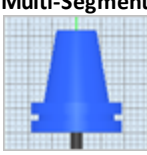


5.8 More about the Create/Select Tools dialog

Here is some additional information specific to the [Create/Select Tool](#) dialog:

 Tool Holder Types

Tool Holder Types	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Vertical 	✓	✓	✓	✓	✓

Tapered 	✓	✓	✓	✓	✓
Multi-Segment 	✓	✓	✓	✓	✓



Tool Types by Configuration

Tool Types		Configuration				
		Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Ball Mills		✓	✓	✓	✓	✓
Flat Mills		✓	✓	✓	✓	✓
Corner Radius Mills		✓	✓	✓	✓	✓
Vee Mills		✓	✓	✓	✓	✓
Chamfer Mills			✓	✓	✓	✓
Taper Mills			✓	✓	✓	✓
Thread Mills			✓	✓	✓	✓
Face Mills			✓	✓	✓	✓
Dovetail Cutters			✓	✓	✓	✓
Fillet Mills			✓	✓	✓	✓
Lollipop Cutters			✓	✓	✓	✓
User Defined Cutters			✓	✓	✓	✓

Drills		✓	✓	✓	✓	✓
Center Drills			✓	✓	✓	✓
Reamers			✓	✓	✓	✓
Taps			✓	✓	✓	✓
Bore Tool			✓	✓	✓	✓
Reverse Bore Tool			✓	✓	✓	✓
Knife Tool			✓	✓	✓	✓
Saw Tool			✓	✓	✓	✓
Plasma Tool			✓	✓	✓	✓

Tool Related Answers

It is recommended that you do not attempt some of these more [Advanced Topics](#) until you are familiar working with tools and tool libraries in the [RhinoCAM MILL](#) module.

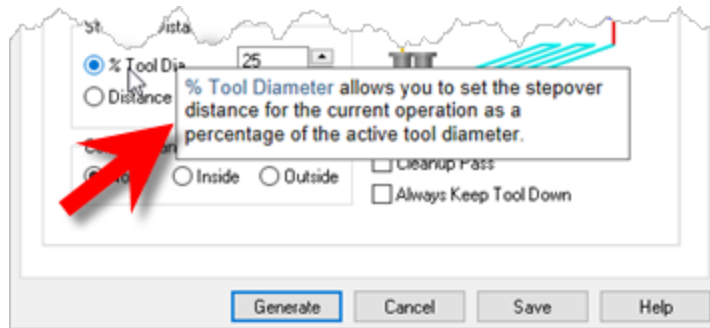
6.1 Where can I find Tool related Preferences?

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:

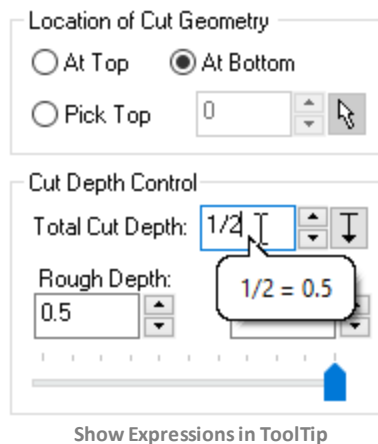
Show context ToolTips

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.

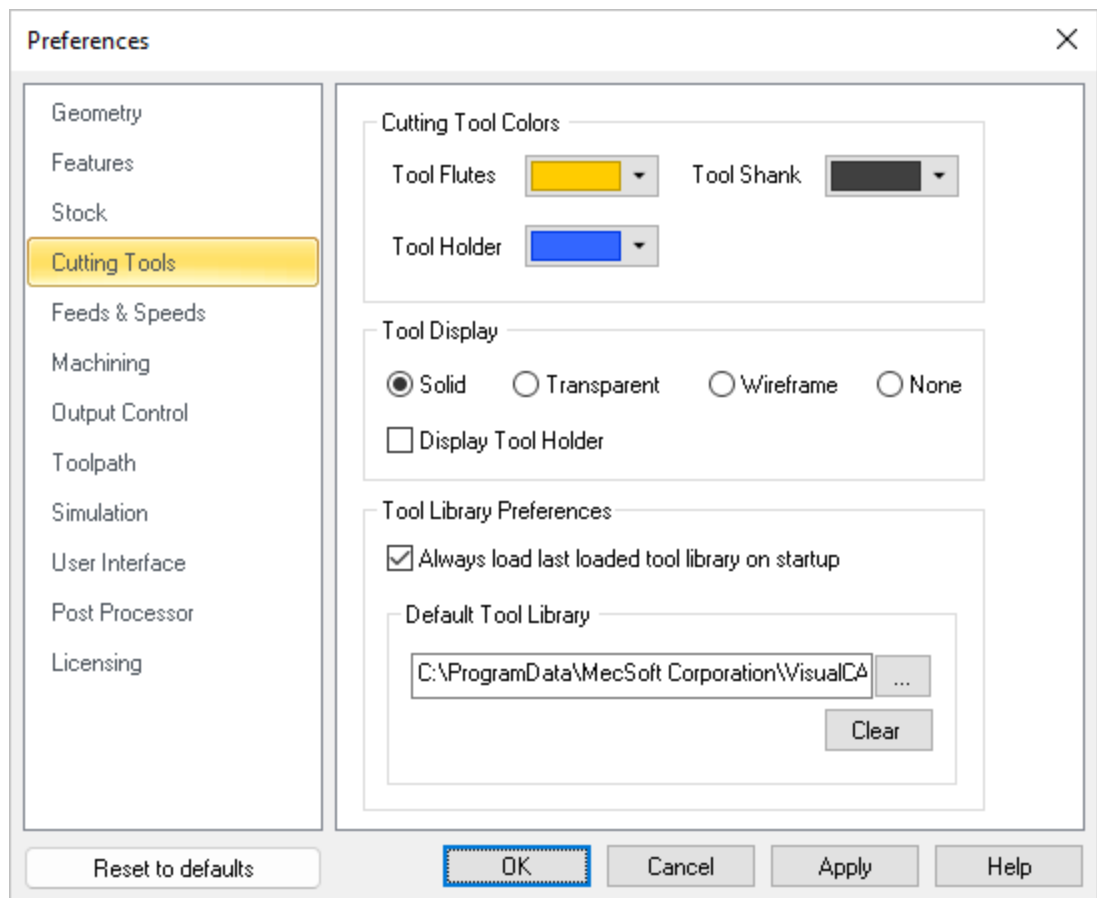


Show Expressions in Tooltip

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.

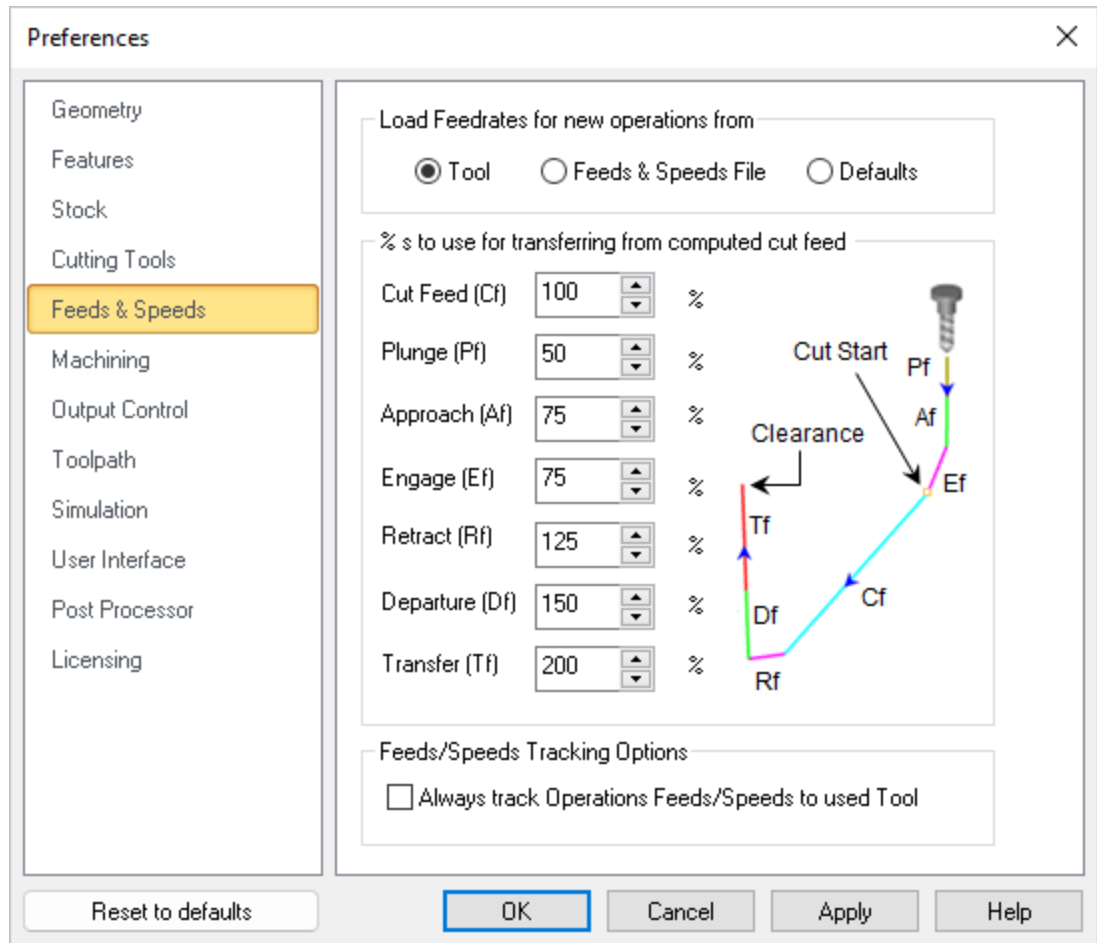


CAM Preferences > Cutting Tools

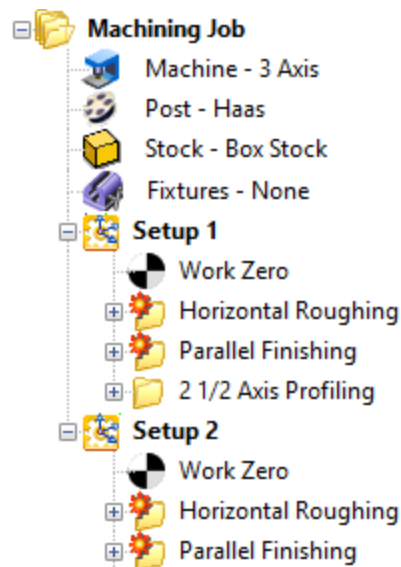
Note: Menu selections on the left may change depending on module and configuration

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 the Colors used when the Tool is displayed on the screen.

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.



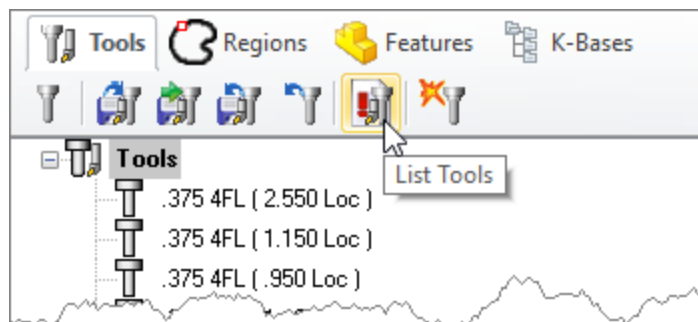
8. You can also set the % of the computed **Cut Feed** to use for the various types of transfer motions. For example, in the **Feeds & Speeds Calculator** (displayed when you select **Load from File** from either the **Create/Select Tools** dialog or from the **Feeds & Speeds** tab of any operation type) 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.



6.2 How can I 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:



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

Cutting Tools Information

Tools Information

Name	Tool Type	Diameter	Corner Radius	Taper	Flute Length	Tool Length	Tool #	Tool Material	Spindle RPM	Cut Feed	Adjust R	Cutcom R	Comments
375 4FL (2.550 Loc)	MM	0.375 mm	0 mm	0 deg	2.55 mm	4.23 mm	0	Carbide	25872	87.62	0	0	
375 4FL (1.150 Loc)	MM	0.375 mm	0 mm	0 deg	1.15 mm	3.05 mm	0	Carbide	25872	87.62	0	0	
375 4FL (.950 Loc)	MM	0.375 mm	0 mm	0 deg	0.95 mm	1.5 mm	0	Carbide	25872	87.62	0	0	
375 2FL (1.180 Loc)	MM	0.375 mm	0 mm	0 deg	1.18 mm	3.04 mm	0	Carbide	25872	87.62	0	0	
375 2FL (.900 Loc)	MM	0.375 mm	0 mm	0 deg	0.9 mm	2.53 mm	0	Carbide	25872	87.62	0	0	
375 4 FL (1.050 Loc)	MM	0.375 mm	0.1875 mm	0 deg	1.05 mm	1.54 mm	0	HSS	25872	87.62	0	0	
312 2 FL (.850 Loc)	MM	0.375 mm	0.1875 mm	0 deg	0.85 mm	2 mm	0	Carbide	25872	87.62	0	0	
250 4 FL (.750 Loc)	MM	0.25 mm	0.125 mm	0 deg	0.75 mm	2.5 mm	0	Carbide	25872	87.62	0	0	
250 2 FL (.750 Loc)	MM	0.25 mm	0.125 mm	0 deg	0.75 mm	2.5 mm	0	Carbide	25872	87.62	0	0	
250 2 FL (.800 Loc)	MM	0.25 mm	0 mm	0 deg	0.8 mm	2.5 mm	0	Carbide	25872	87.62	0	0	
250 4 FL (.800 Loc)	MM	0.25 mm	0 mm	0 deg	0.8 mm	2.5 mm	0	Carbide	25872	87.62	0	0	
250 4 FL (1.150 Loc)	MM	0.25 mm	0 mm	0 deg	1.15 mm	3.04 mm	0	Carbide	25872	87.62	0	0	
250 4 FL (1.750 Loc) 3/...	MM	0.25 mm	0 mm	0 deg	1.73 mm	3.54 mm	0	HSS	25872	87.62	0	0	
187 2 FL (.800 Loc)	MM	0.187 mm	0 mm	0 deg	0.8 mm	2.54 mm	0	Carbide	25872	87.62	0	0	
187 4 FL (.800 Loc)	MM	0.187 mm	0 mm	0 deg	0.8 mm	2.5 mm	0	Carbide	25872	87.62	0	0	
187 4 FL (.650 Loc)	MM	0.187 mm	0 mm	0 deg	0.65 mm	2 mm	0	Carbide	25872	87.62	0	0	
187 2 FL (.650 Loc)	MM	0.187 mm	0 mm	0 deg	0.65 mm	2 mm	0	Carbide	25872	87.62	0	0	
187 4 FL (.650 Loc)	MM	0.1875 mm	0.09375 mm	0 deg	0.65 mm	2 mm	0	Carbide	25872	87.62	0	0	
187 2 FL (.650 Loc)	MM	0.1875 mm	0.09375 mm	0 deg	0.65 mm	2 mm	0	Carbide	25872	87.62	0	0	
125 4FL (750 Loc)	MM	0.125 mm	0 mm	0 deg	0.75 mm	2.3 mm	0	Carbide	25872	87.62	0	0	
125 2FL (750 Loc)	MM	0.125 mm	0 mm	0 deg	0.75 mm	2.3 mm	0	Carbide	25872	87.62	0	0	
125 4FL (520 Loc)	MM	0.125 mm	0 mm	0 deg	0.52 mm	2.5 mm	0	Carbide	25872	87.62	0	0	
125 2FL (520 Loc)	MM	0.125 mm	0 mm	0 deg	0.52 mm	2.5 mm	0	Carbide	25872	87.62	0	0	
375 4 FL (520 Loc)	MM	0.125 mm	0.0625 mm	0 deg	0.52 mm	1.5 mm	0	Carbide	25872	87.62	0	0	
375 2 FL (520 Loc)	MM	0.125 mm	0.0625 mm	0 deg	0.52 mm	1.5 mm	0	Carbide	25872	87.62	0	0	
250 Wood Raughter (.7...	MM	0.25 mm	0 mm	0 deg	0.77 mm	2 mm	0	Carbide	25872	87.62	0	0	

Print

OK Cancel Help

6.3 How can I add a Custom Tool?

Available in:

Xpress

Standard

Expert

Professional

Premium

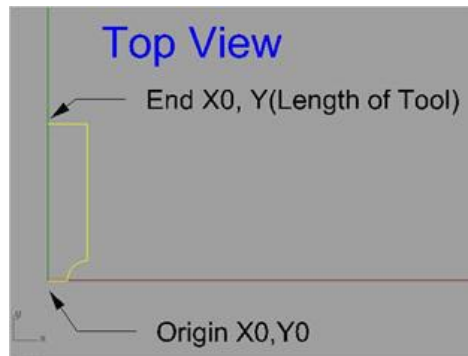
The **MILL** module allows creation of special purpose tools like form tools. These can be defined under user defined tool in the create/select tool dialog.



User Defined Tools can be used in **Drill** operations to allow multi-function tools to be defined as user defined tools and used in drilling operations. See **User Defined Tools** for more information.



Steps to create a user defined tool

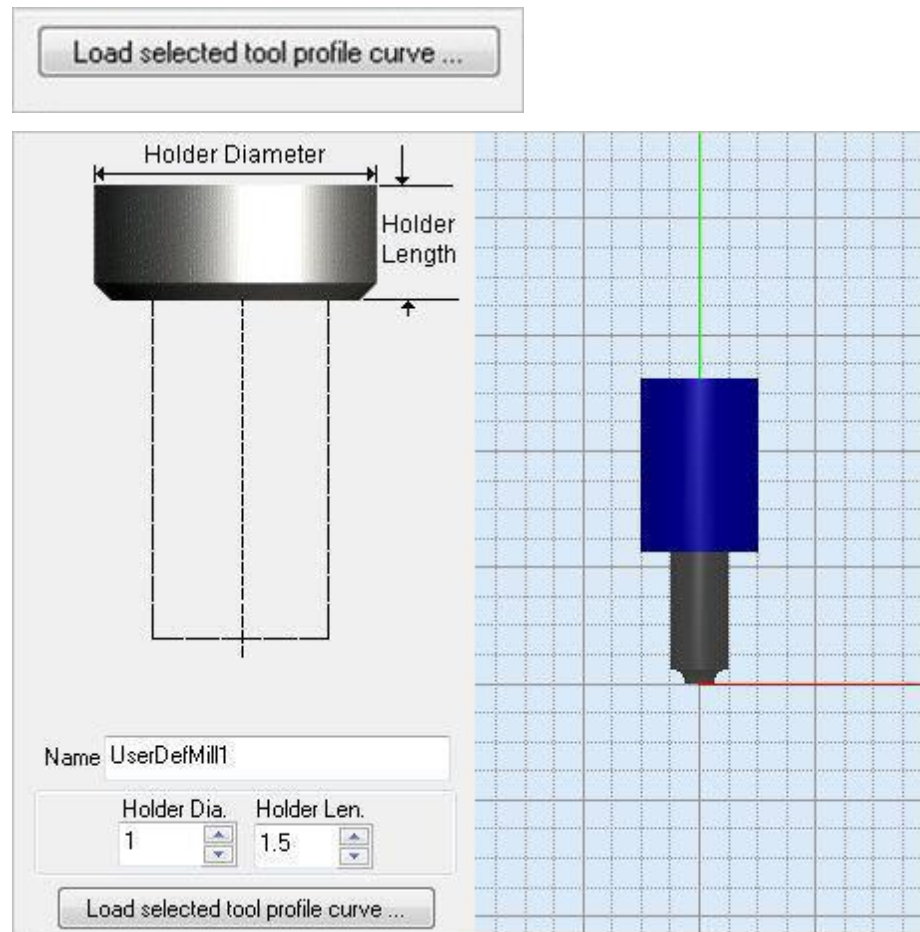


Steps to create a user defined tool:

1. Draw half the tool profile from the top view (XY plane of the world coordinate system) as shown in the picture above and then join the curves to form a single curve.
2. Make sure one end of the curve (tool tip) is at origin (0,0) and the other end at X0, Y<value>.
3. From the **Tools** tab under the **Machining Objects Browser**, click **Create/Select Tool** and select **User Defined Tool**.



4. Click **Load selected tool profile curve**.



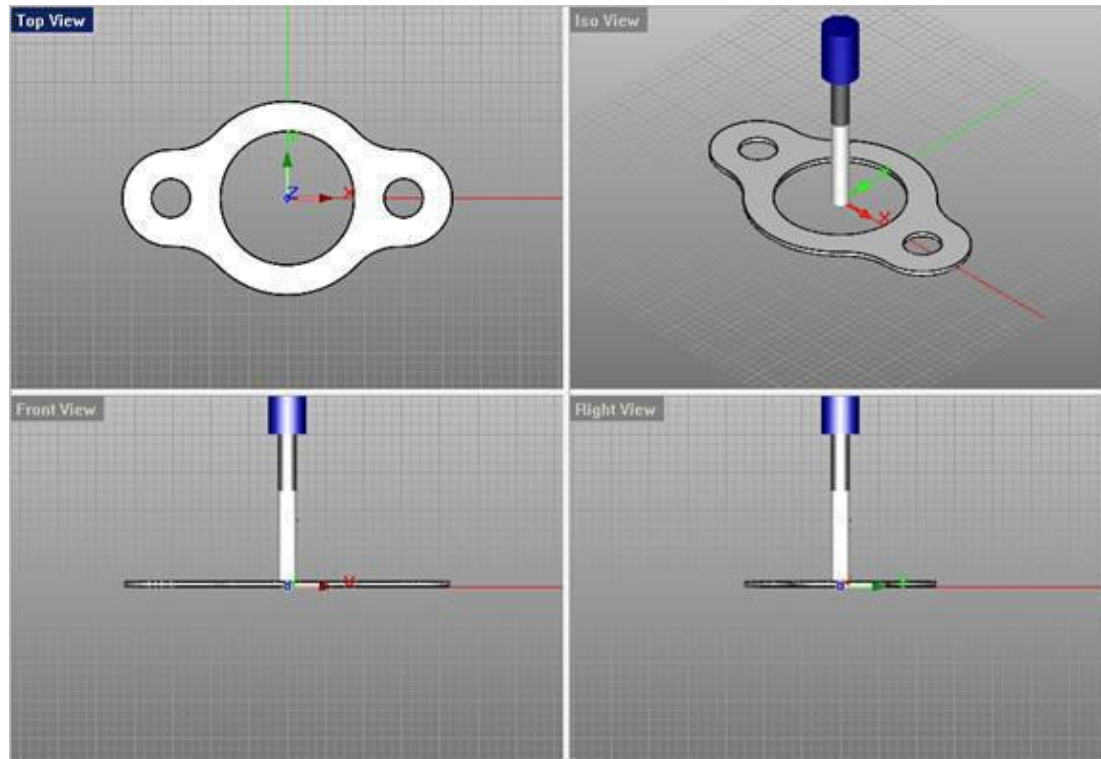
Click Load selected tool profile curve

5. Specify the **Holder Diameter**, **Holder Length**, **Properties**, **Feed & Speeds** and Click **Save as New Tool**.



Preview your Tool

[Preview Tool](#) allows you to preview the highlighted tool in the workspace as seen below. The tool is previewed at the [WCS](#) origin.

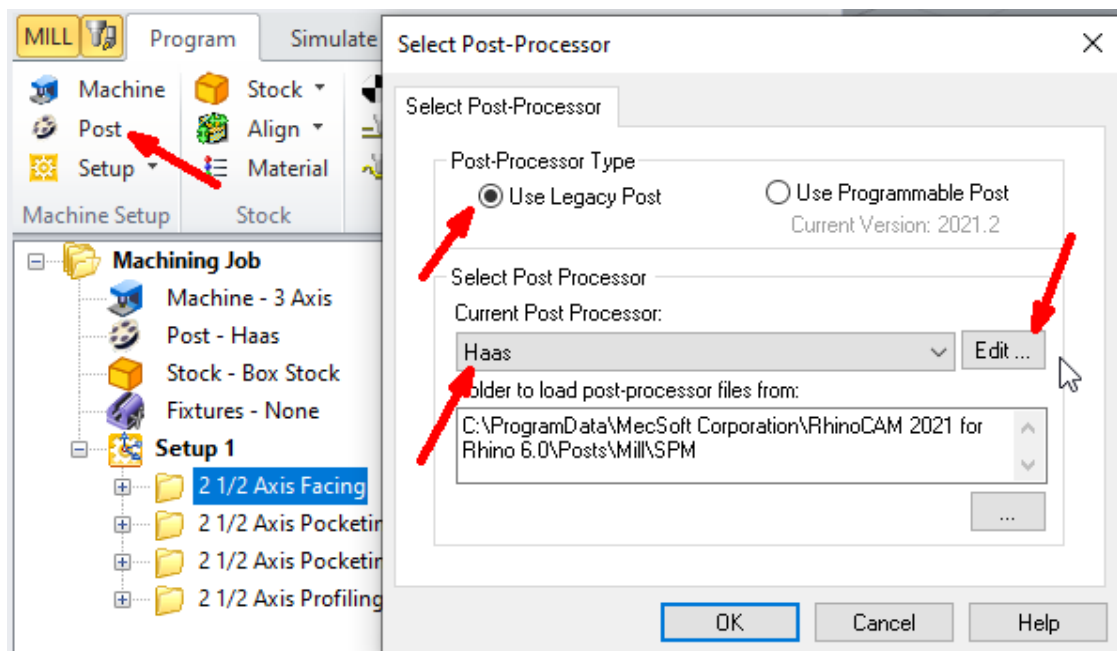


Preview your Tool

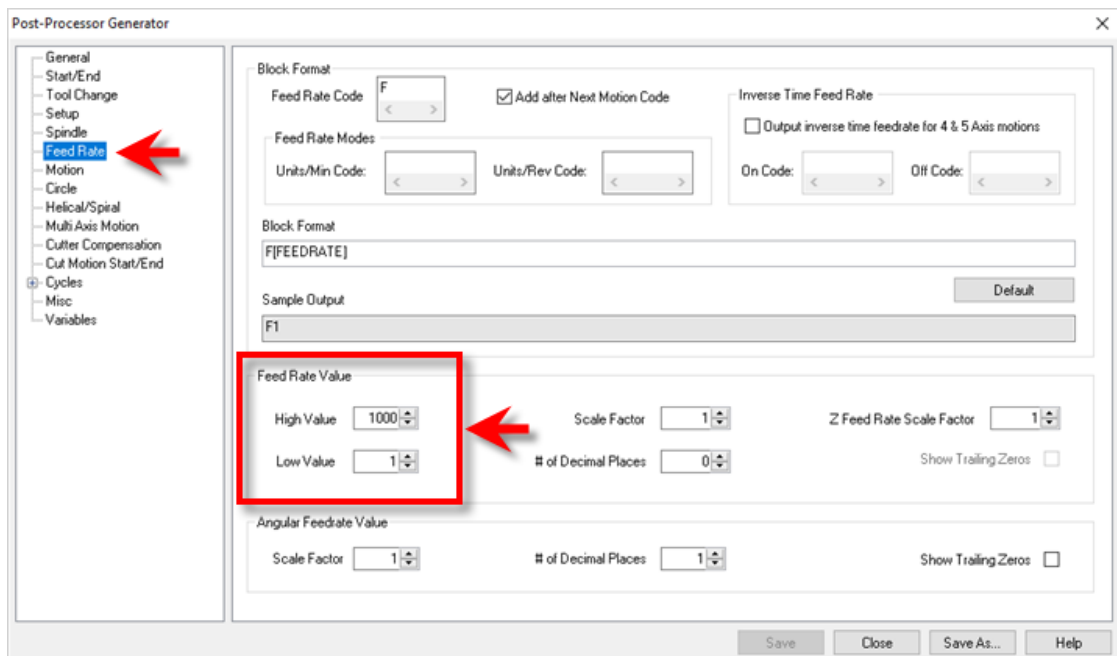
6.4 Why are my Feed Rate values too High/Low?

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](#).



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](#).



4. Then pick [Save](#).

6.5 What about Tapping Feed Rates?

For tap operations, the feedrate can be computed in different ways. This depends on what is expected by the controller.

Spindle Speed x Thread Pitch

Thread Pitch

1/Thread Pitch

Cut Feedrate

The post needs to be setup with the appropriate variable to output the feedrate. The **Thread Pitch** is defined under the **Tool** definition and **Spindle Speed** is set under **Feeds/Speed** tab of the **Tap** operation.

Use the following macro's in the post to output the feedrate

[CYCL_IPR] – Spindle Speed x Thread Pitch

[CYCL_TPI] – Thread Pitch

[CYCL_1/TPI] – 1/Thread Pitch

[CUT_FEED] – Cut Feedrate

6.6 How can I Optimize Machining Time Estimates?



Optimize Machining Time Estimates!

In any MecSoft CAM product you can get an Information report about a selected toolpath operation, a **Setup** or all operations in the **Machining Job**. This report contains some very useful information that includes the **Tool #s** used, the **Cut Feed**, the **# of GOTO** motions and most importantly, the estimated **Machining Time**.

[Read the full article...](#)

	Machine Time
	3 hr 32 min
	10 hr 20 min
	13 hr 52 min

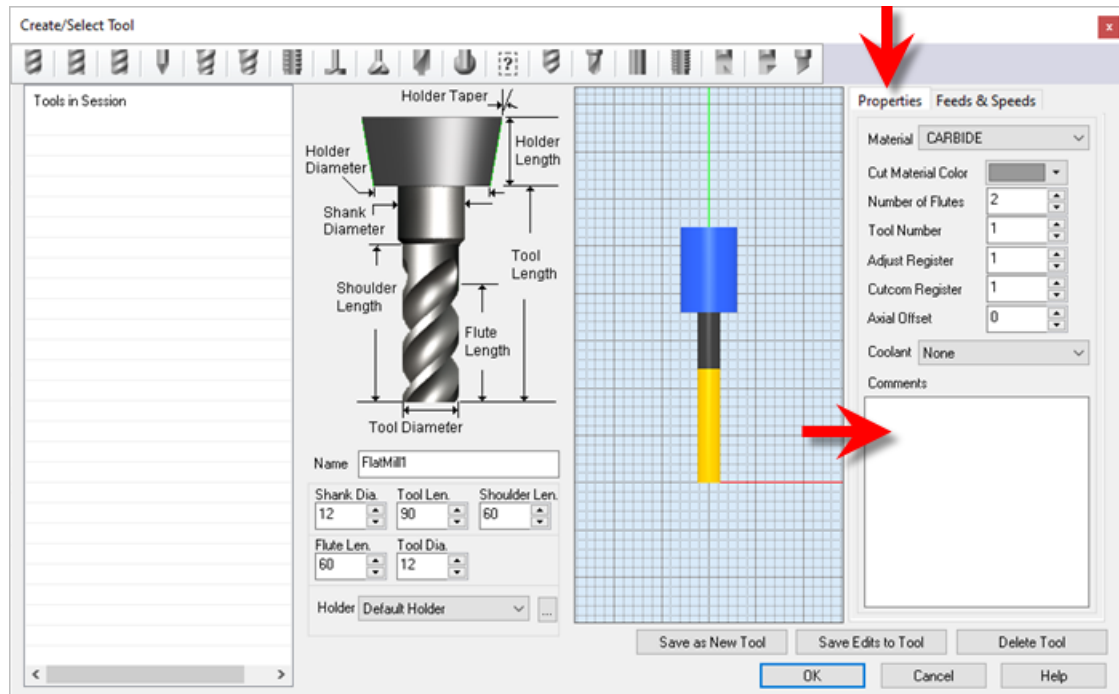
More Advanced Topics

7.1 How can I add Tool Comments?

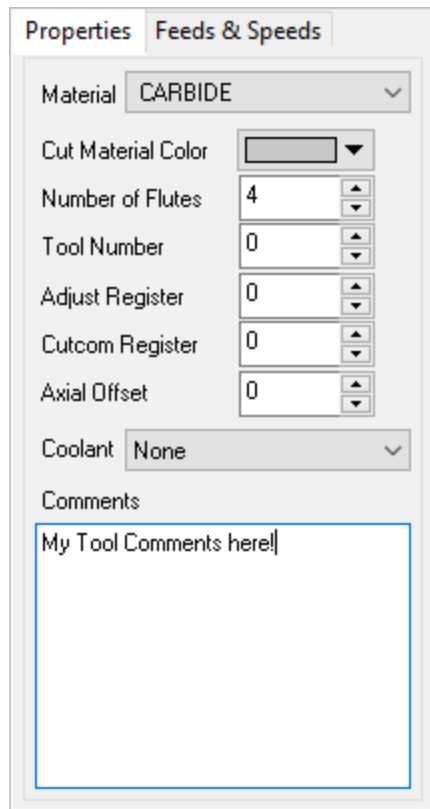
You can add comments associated with a [Tool](#). These [Comments](#) are saved with the [Tool](#) in your [Tool Library](#). They are also posted to your g-code when the tool is used.

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.

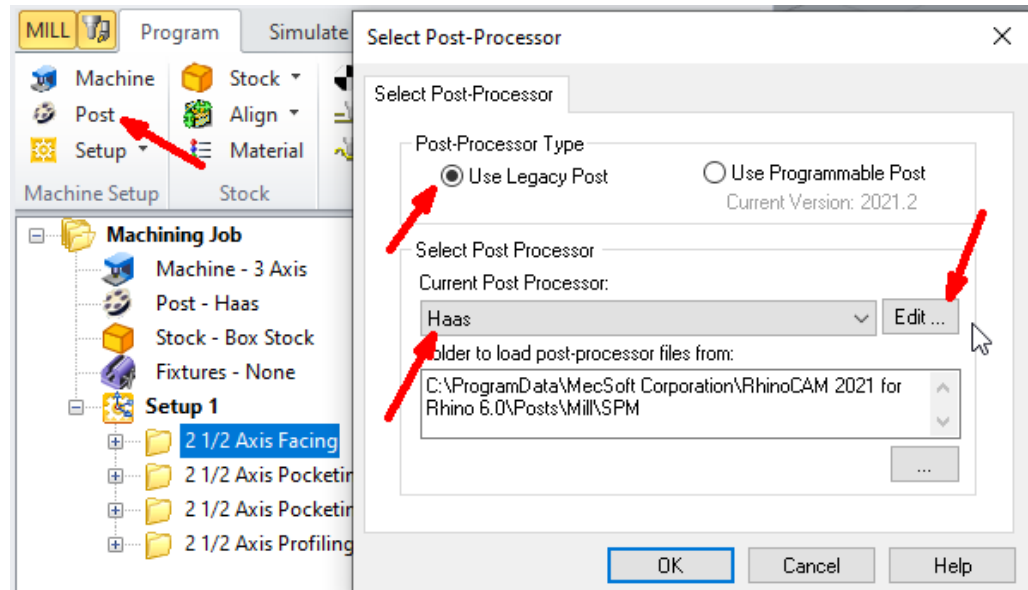


3. Add text to the [Comments](#) window.



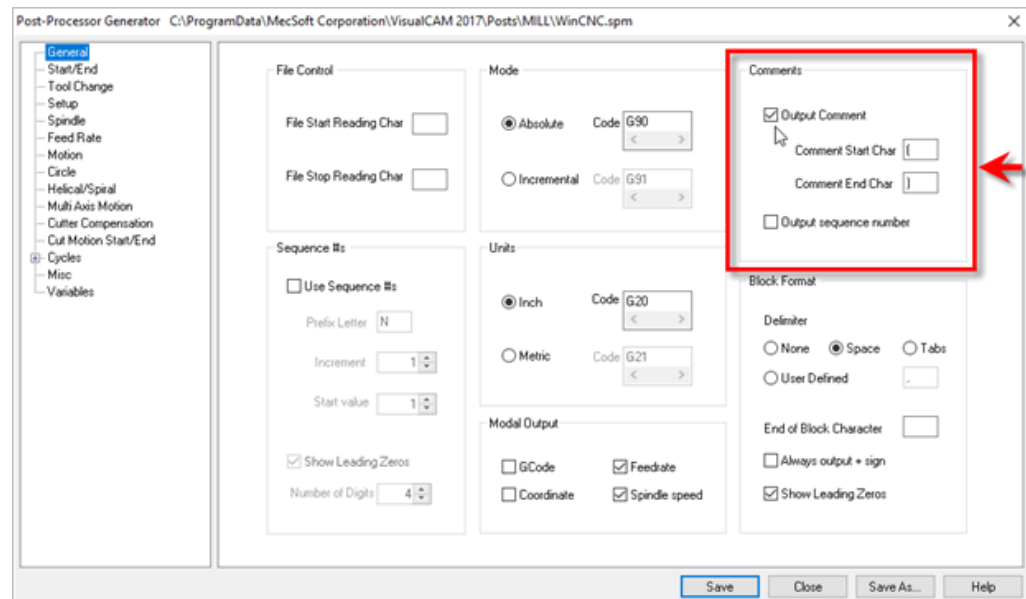
4. Make sure [Comments](#) are enabled in your post.

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



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.

The screenshot shows the 'Feeds & Speeds' tab in the RhinoCAM-MILL software. The 'Material' is set to 'CARBIDE'. The 'Cut Material Color' is a grey swatch. The 'Number of Flutes' is 4. The 'Tool Number' is 0. The 'Adjust Register' is 0. The 'Cutcom Register' is 0. The 'Axial Offset' is 0. The 'Coolant' is set to 'None'. The 'Comments' text area contains '\$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
...
...

```

7.2 How can add a Tool Change Point?

Implementing a [Tool Change Point](#) can be useful. For example in 2 and 3 Axis, you may want to change tools manually between operations (i.e., your CNC machine does not have an automatic tool changer). Also in 4 Axis you may want to ensure the tool is moved to a save location prior to a table rotation. To output a [Tool Change Point](#) to your posted g-code files, please do the following:



For 2 and 3 Axis Output

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

Machine Tool Setup

Machine Tool Definition

☒ Manual Definition ☐ Load From File

Machine Type

Number of Axes: 3 Axis

General Parameters

Tool Change Pt: X: -4 Y: 0 Z: 0

☒ Output all coordinates in local Setup Coordinate System

Translational Limits

Min: X: -5000 Y: -5000 Z: -5000

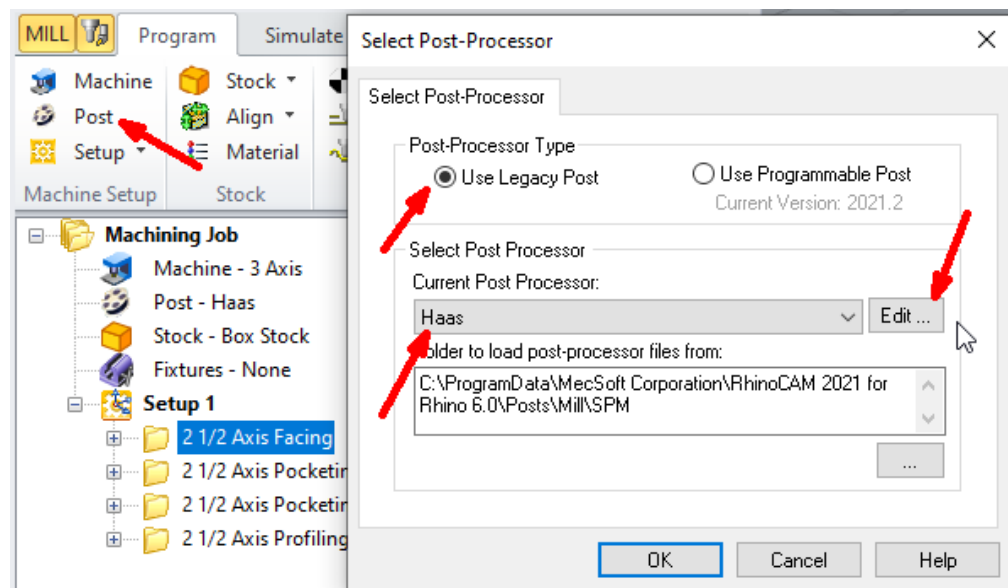
Max: X: 5000 Y: 5000 Z: 5000

4th Axis (Primary Axis) Parameters

Rotary Center: X: 0 Y: 0 Z: 0

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

3. Edit your post processor by selecting **Program** tab > **Post** > **Edit**.

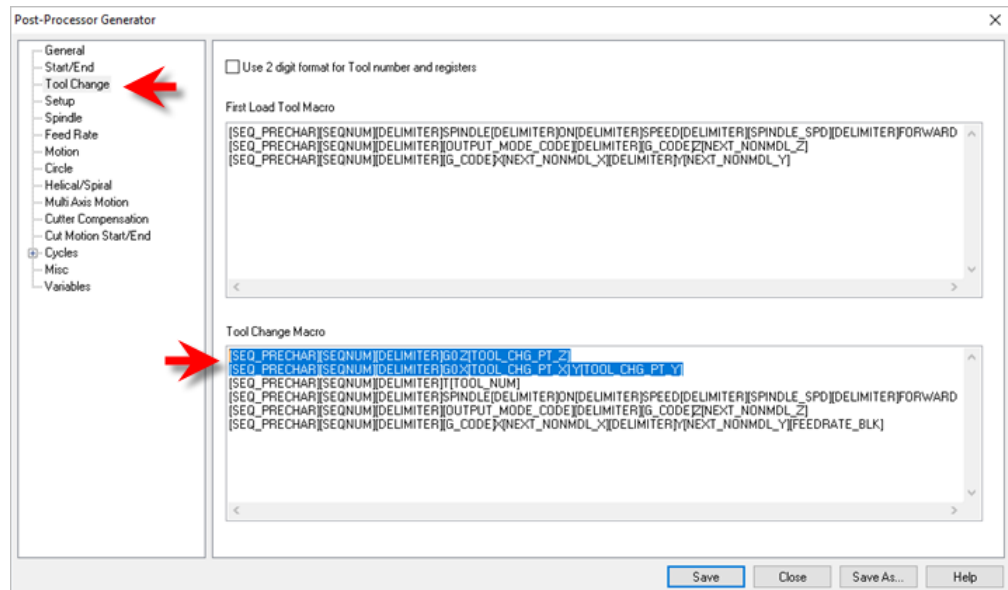


4. From the **Post Process Generator** dialog, select the **Tool Change** section from the left side of the dialog.
5. In the **Tool Change Macro** block section, replace the first line of text with the following two lines of text at the top of the macro. These two lines of text should precede the line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

```
[SEQ_PRECHAR][SEQNUM][DELIMITER]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]T[TOOL_NUM]
```

...

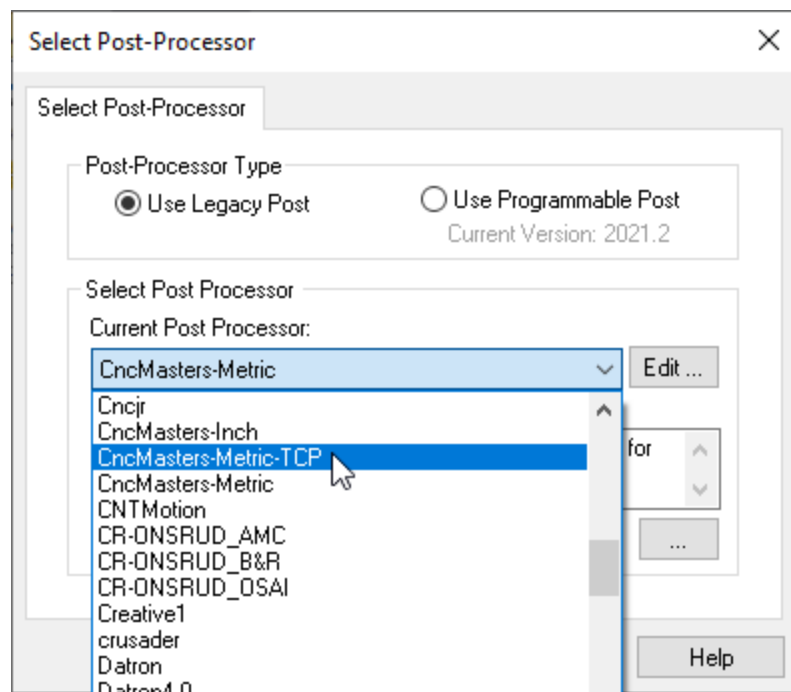
...



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]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]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:

...

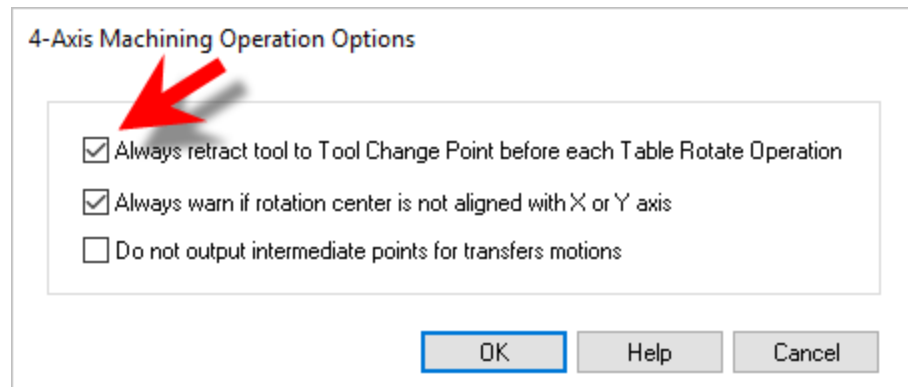
...
N66 ;2 1/2 Axis Profiling
N68 G0 Z0.
N70 G0 X-4. Y0.
N72 T1 M06
...
...

15. That's it!

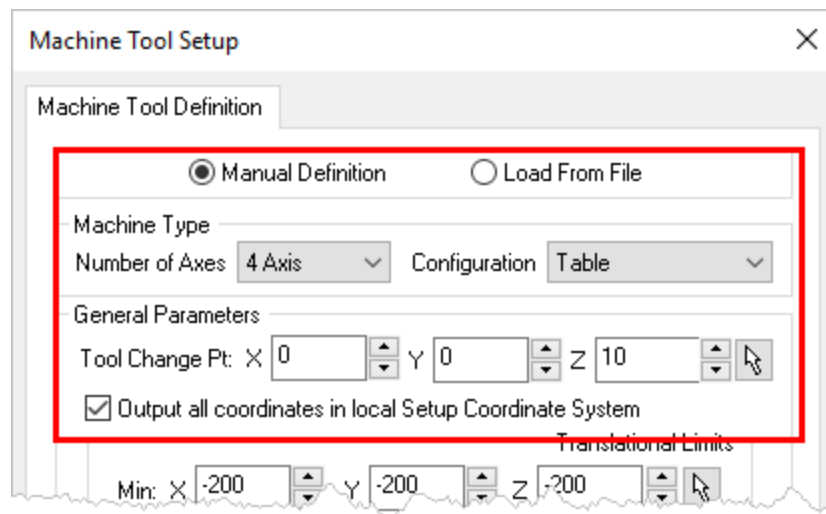


For 4 Axis Output

1. From the **Program** tab select **4 Axis**.
2. From the **4 Axis** menu select **4 Axis Options**.
3. From the **4 Axis Operation Options** dialog check the box to **Always retract tool to Tool Change Point before each Table Rotate Operation**.



4. Now from the **Program** tab select **Machine** and then **Manual Definition**.
5. For the **Machine Type** select **4 Axis**.
6. Under **General Parameters**, enter the X, Y and Z coordinate values for the **Tool Change Point**.



7. Then check the box to [Output all coordinates in local Setup Coordinate System](#) and then pick **OK** to close the dialog.
8. Post the [4 Axis](#) toolpath operation and verify that the [Tool Change Point](#) is being posted before the table rotation angle similar to this:

```
...
...
(Setup 2)
N6263 Z10.
N6264 X0.Y0.
(Horizontal Roughing)
N6265 A180.F300.
...
...
```

7.3 How can I 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!

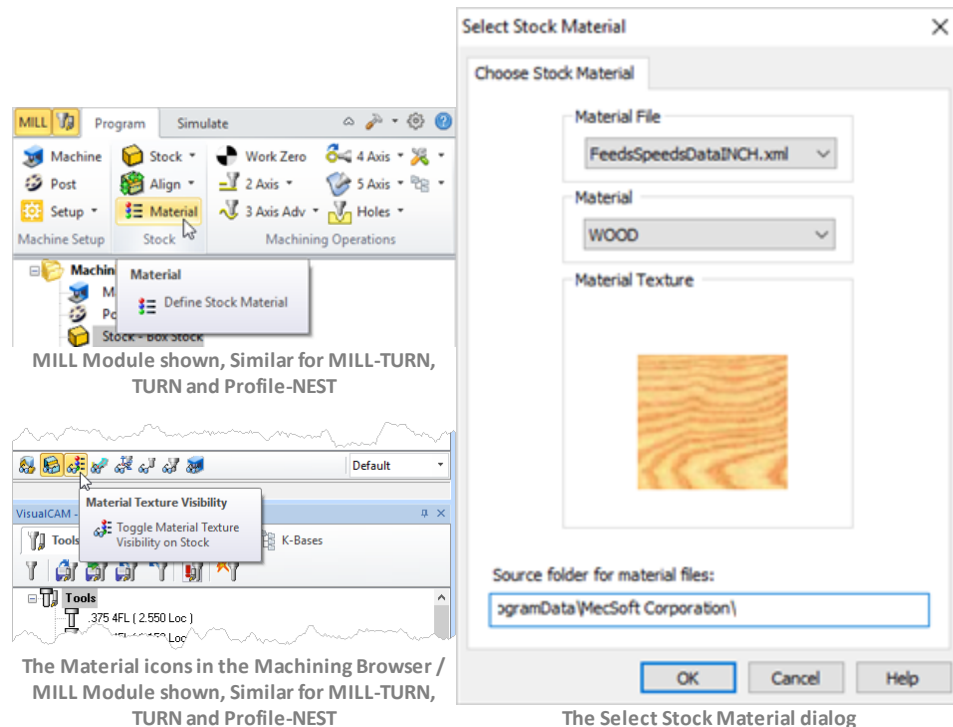
However, what if you are cutting stock material that is currently not in our [Materials Library](#)? Or what if you don't like what is currently assigned for the material of your choice in the [Materials Library](#)?

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](#).

```

<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
	MILLING	CERAMIC	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:

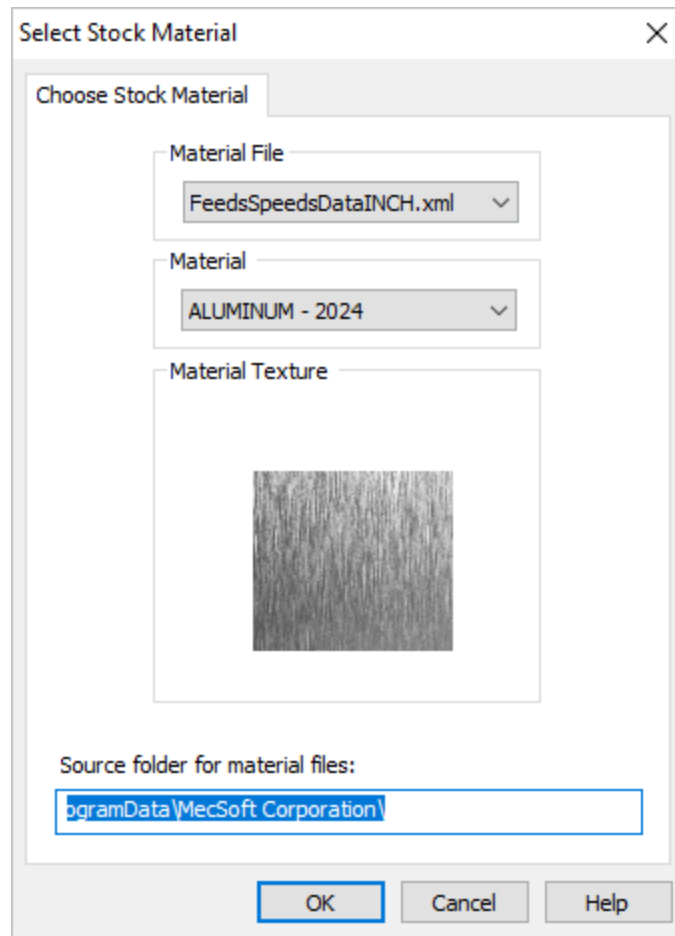
```
<FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
```

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>
```

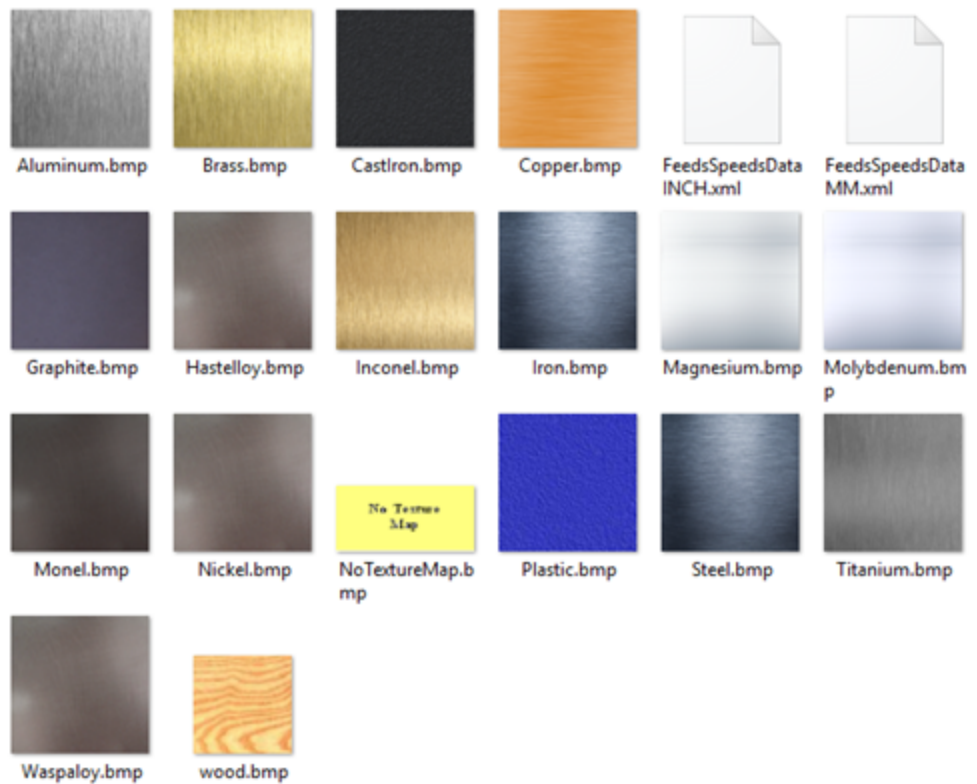
```
...
...
```

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.



Dialog Box: Select Stock Material

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.

7.4 How can I 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

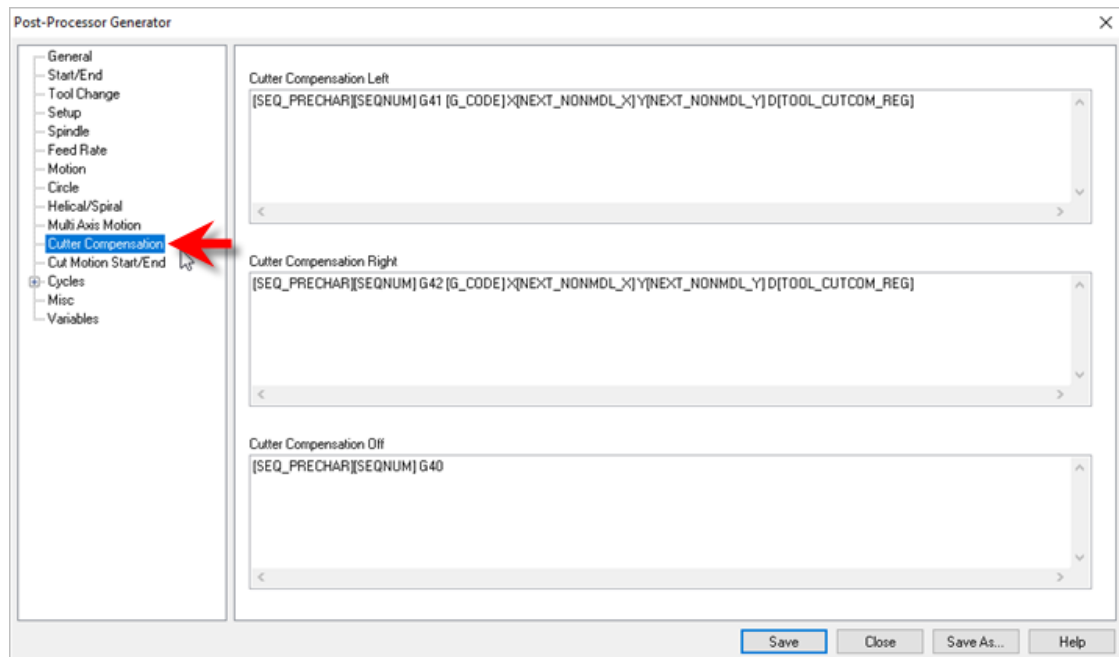
[\[SEQ_PRECHAR\]\[SEQNUM\] G41 \[G_CODE\] X\[NEXT_NONMDL_X\] Y\[NEXT_NONMDL_Y\] D\[TOOL_CUTCOM_REG\]](#)

Cutter Compensation Right

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

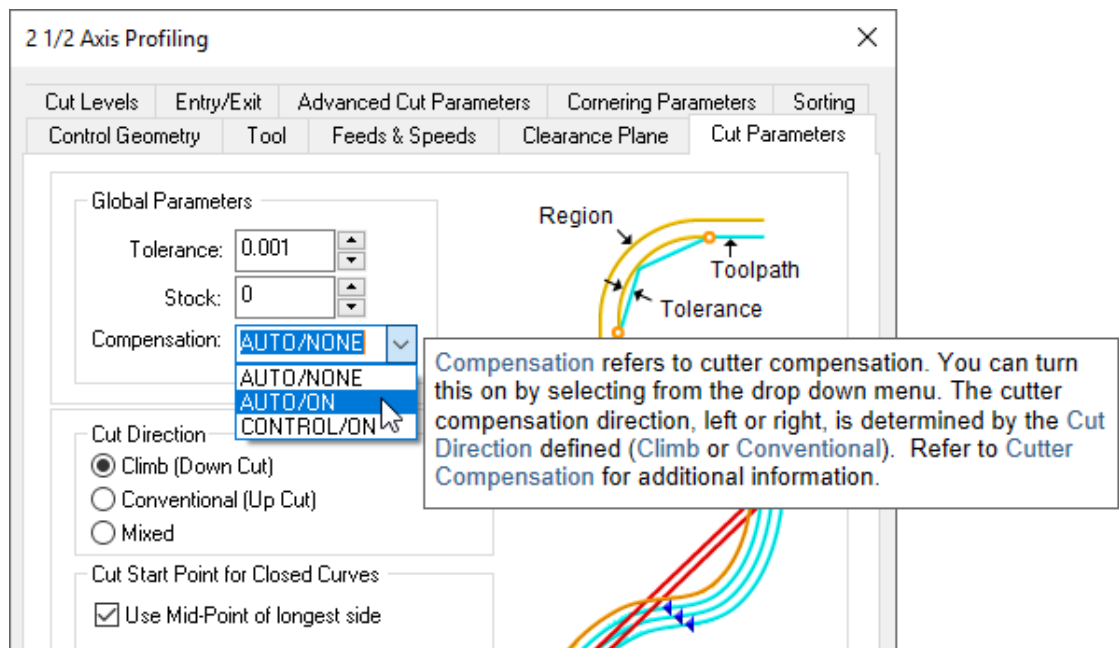
Cutter Compensation Off

[\[SEQ_PRECHAR\]\[SEQNUM\] 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.

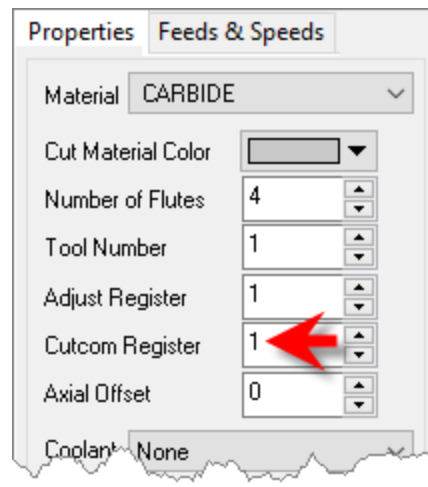


3. Make sure you have a linear motion for the controller to turn on the compensation for. If your first motion is an arc the controller will not be able to turn on the compensation.

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.



Reference

8.1 MILL Holder Worksheet

Create/Select Holder

Holders in Session

Holder1

Holder2

Holder: 2.75 x 2.2 x 12"

Holder Name

Holder Segment Parameters

Diameter

Length

Taper

Add as New

Save to Active

Holder Segments

Segment 4

Segment 3

Segment 2

Segment 1

Move Up

Move Down

Remove All

Remove Active

Save as New Holder

Save Edits to Holder

Delete Holder

Exit

Help

8.2 Mill Tool Worksheets

The following [Mill Tool Worksheets](#) are available:

1. [Ball Mill Worksheet](#)
2. [Flat Mill Worksheet](#)
3. [Corner Radius Mill Worksheet](#)
4. [Vee Mill Worksheet](#)
5. [Chamfer Mill Worksheet](#)
6. [Taper Mill Worksheet](#)
7. [Thread Mill Worksheet](#)
8. [Dovetail Mill Worksheet](#)
9. [Fillet Mill Worksheet](#)
10. [Lollipop Mill Worksheet](#)
11. [User Defined Mill Worksheet](#)
12. [Drag Knife Worksheet](#)
13. [Saw Tool Worksheet](#)
14. [Laser Tool Worksheet](#)

8.2.1 Ball Mill Worksheet

Ball End Mill

Properties Feeds & Speeds

Material:

Cut Material Color:

Number of Flutes:

Tool Number:

Adjust Register:

Cutcom Register:

Axial Offset:

Coolant:

Comments:

Spindle Parameters

Speed: RPM

Direction: ☐ CW ☐ CCW

Feed Rates (in/min)

Plunge: Approach: Engage:

Cut: Retract: Departure:

Transfer:

☐ Use Rapid ☐ Set

Feed Rate Reduction Factors

Plunge between levels: %

First XY pass: %

Bottom Z Level: %

Cut Depth: in

Name: BallMill

Shank Dia.:

Flute Len.:

Holder:

Tool Len.:

Shoulder Len.:

Tool Dia.:

8.2.2 Flat Mill Worksheet

Flat End Mill

Properties **Feeds & Speeds**

Material:

Cut Material Color:

Number of Flutes:

Tool Number:

Adjust Register:

Cutcom Register:

Axial Offset:

Coolant:

Comments:

Spindle Parameters

Speed: RPM

Direction: ☐ CW ☐ CCW

Feed Rates (in/min)

Plunge: Approach: Engage:

Retract: Departure:

Cut:

Transfer: ☐ Use Rapid ☐ Set

Feed Rate Reduction Factors

Plunge between levels: %

First XY pass: %

Bottom Z Level: %

Cut Depth: in

Name:

Shank Dia.: Tool Len.: Shoulder Len.:

Flute Len.: Tool Dia.:

Holder:

8.2.4 Vee Mill Worksheet

Vee End Mill

Properties **Feeds & Speeds**

Material:

Cut Material Color:

Number of Flutes:

Tool Number:

Adjust Register:

Cutcom Register:

Axial Offset:

Coolant:

Comments:

Spindle Parameters

Speed: RPM

Direction: ☐ CW ☐ CCW

Feed Rates (in/min)

Plunge: Approach: Engage:

Retract: Departure:

Cut:

Transfer: ☐ Use Rapid ☐ Set

Feed Rate Reduction Factors

Plunge between levels: %

First XY pass: %

Bottom Z Level: %

Cut Depth: in

[Load from File ...](#)

Shank Diameter:

Shoulder Length:

Flute Length:

Tool Length:

Taper Angle:

Tool Diameter:

Name:

Shank Dia.: Tool Len.: Shoulder Len.:

Tool Dia.: Taper Angle:

Holder:

8.2.5 Chamfer Mill Worksheet

Chamfer End Mill

Properties
Feeds & Speeds

Material

Cut Material Color

Number of Flutes

Tool Number

Adjust Register

Cutcom Register

Axial Offset

Coolant

Comments

Spindle Parameters

Speed RPM

Direction ☐ CW ☐ CCW

Feed Rates (in/min)

Plunge Approach Engage

Cut Retract Departure

Transfer ☐ Use Rapid ☐ Set

Feed Rate Reduction Factors

Plunge between levels %

First XY pass %

Bottom Z Level %

Cut Depth in

Load from File ...

Name

Shank Dia. Tool Len. Shoulder Len.

Flute Len. Flat Dia. Taper Angle

Holder

8.2.6 Taper Mill Worksheet

Tapered End Mill

Properties **Feeds & Speeds**

Material:

Cut Material Color:

Number of Flutes:

Tool Number:

Adjust Register:

Cutcom Register:

Axial Offset:

Coolant:

Comments:

Spindle Parameters

Speed: RPM

Direction: ☐ CW ☐ CCW

Feed Rates (in/min)

Plunge: Approach: Engage:

Retract: Departure:

Cut:

Transfer:

☐ Use Rapid ☐ Set

Feed Rate Reduction Factors

Plunge between levels: %

First XY pass: %

Bottom Z Level: %

Cut Depth: in

Material

Cut Material Color

Number of Flutes

Tool Number

Adjust Register

Cutcom Register

Axial Offset

Coolant

Comments

Name

Shank Dia. **Shoulder Len.**

Flute Len. **Flat Dia.**

Corner Rad. **Taper Angle**

Holder

8.2.8 Face Mill Worksheet

Face Mill

Properties Feeds & Speeds

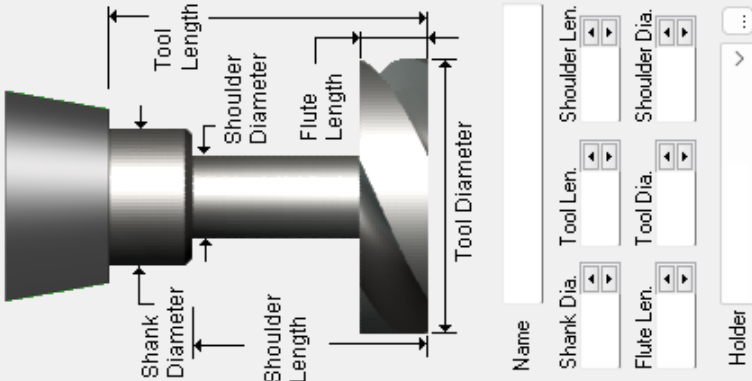
Material Spindle Parameters Speed RPM Direction ☐ CW ☐ CCW

Cut Material Color Feed Rates (in/min) Plunge Approach Engage Number of Flutes Retract Departure Tool Number Cut Transfer Coolant ☐ Use Rapid ☐ Set

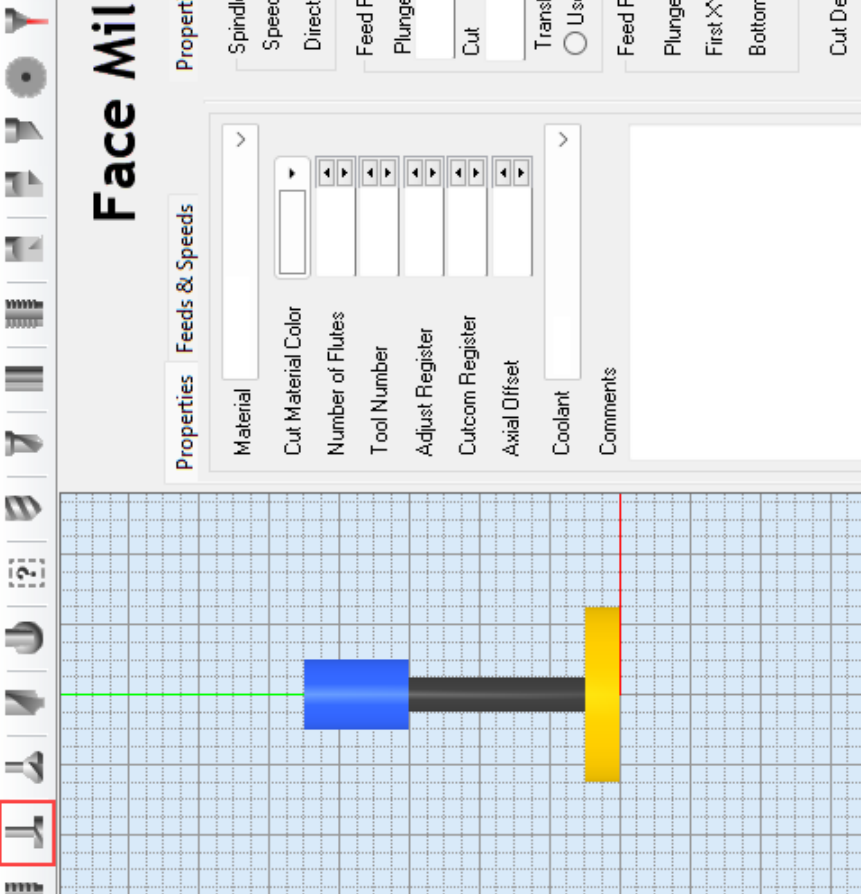
Adjust Register Feed Rate Reduction Factors Plunge between levels % First XY pass % Bottom Z Level % Axial Offset Cut Depth in

Comments

Load from File ...



Name Shank Dia. Tool Len. Shoulder Len. Flute Len. Tool Dia. Shoulder Dia. Holder



8.2.10 Fillet Mill Worksheet

Fillet Mill

Properties **Feeds & Speeds**

Material:

Cut Material Color:

Number of Flutes:

Tool Number:

Adjust Register:

Cutcom Register:

Axial Offset:

Coolant:

Comments:

Spindle Parameters

Speed: RPM

Direction: ☐ CW ☐ CCW

Feed Rates (in/min)

Plunge: Approach: Engage:

Cut: Retract: Departure:

Transfer:

☐ Use Rapid ☐ Set

Feed Rate Reduction Factors

Plunge between levels: %

First XY pass: %

Bottom Z Level: %

Cut Depth: in

Diagram

Shank Diameter:

Shoulder Length:

Flute Length:

Corner Radius:

Flat Diameter:

Tool Length:

Tool Data

Name:

Shank Dia.: Shoulder Len.:

Flute Len.: Flat Dia.:

Corner Rad.:

Holder:

8.2.11 Lollipop Mill Worksheet

Lollypop Mill

Name

Shank Dia. **Tool Len.** **Shoulder Len.**

Flute Len. **Tool Dia.** **Shoulder Dia.**

Holder

Properties
Feeds & Speeds

Material	
Cut Material Color	<input type="color"/>
Number of Flutes	
Tool Number	
Adjust Register	
Cutcom Register	
Axial Offset	
Coolant	
Comments	

Properties
Feeds & Speeds

Spindle Parameters	
Speed	RPM
Direction ☐ CW ☐ CCW	
Feed Rates (in/min)	
Plunge	Approach
Engage	Disengage
Retract	Departure
Transfer ☐ Use Rapid ☐ Set	
Feed Rate Reduction Factors	
Plunge between levels %	
First XY pass %	
Bottom Z Level %	
Cut Depth in	

[Load from File ...](#)

8.2.12 User Defined Mill Worksheet

User Defined Mill

Properties **Feeds & Speeds**

Material:

Cut Material Color:

Number of Flutes:

Tool Number:

Adjust Register:

Cutcom Register:

Axial Offset:

Coolant:

Comments:

Spindle Parameters

Speed: RPM

Direction: ☐ CW ☐ CCW

Feed Rates (in/min)

Plunge: Approach: Engage:

Cut: Retract: Departure:

Transfer:

☐ Use Rapid ☐ Set

Feed Rate Reduction Factors

Plunge between levels: %

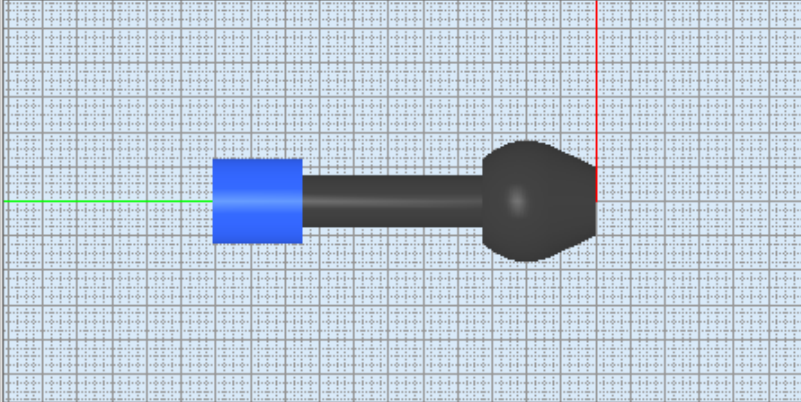
First XY pass: %

Bottom Z Level: %

Cut Depth: in

Name

Holder



8.2.14 Saw Tool Worksheet

Saw Tool

Properties Feeds & Speeds

Material: Spindle Parameters: Speed: RPM Direction: ☐ CW ☐ CCW

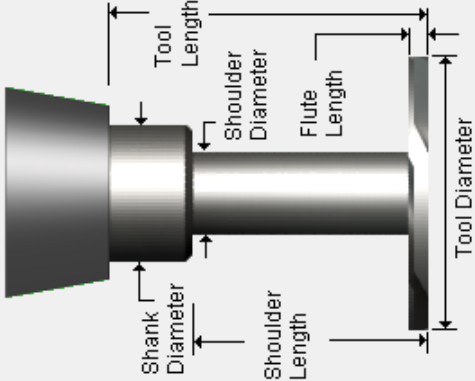
Number of Flutes: Tool Number: Adjust Register: Cutcom Register: Axial Offset: Coolant: Comments:

Feed Rates (in/min): Plunge: Approach: Engage: Retract: Departure: Transfer: ☐ Use Rapid ☐ Set

Feed Rate Reduction Factors: Plunge between levels: % First XY pass: % Bottom Z Level: %

Cut Depth: in

Load from File ...



Name:

Shank Dia.: Tool Len.: Shoulder Len.:

Flute Len.: Tool Dia.: Shoulder Dia.:

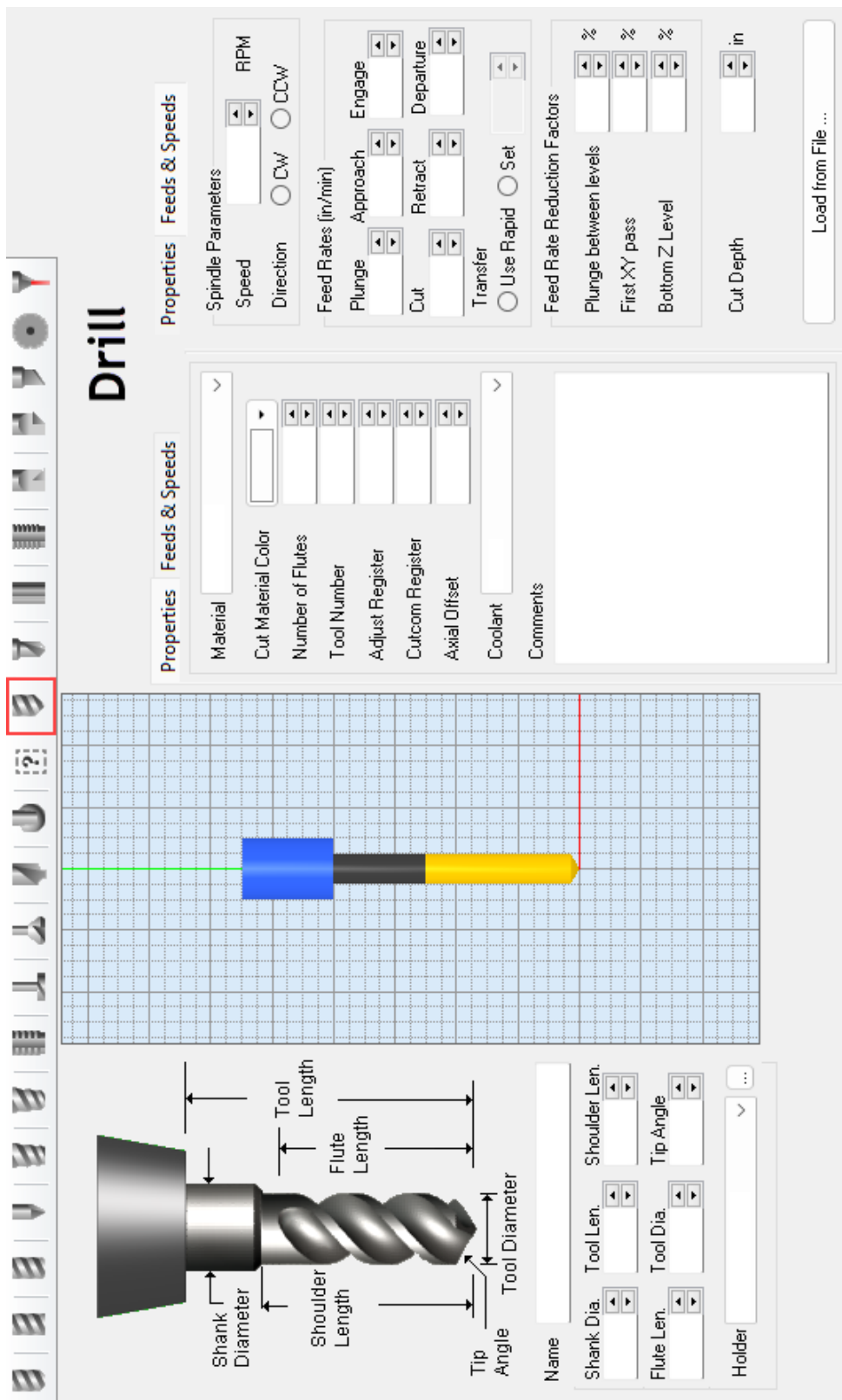
Default Orient.:

Holder:

8.3 Drill Tool Worksheets

The following [Drill Tool Worksheets](#) are available:

1. [Drill Worksheet](#)
2. [Center Drill Mill Worksheet](#)
3. [Reamer Worksheet](#)
4. [Tap Worksheet](#)
5. [Bore Worksheet](#)
6. [Reverse Bore Worksheet](#)



8.3.2 Center Drill Worksheet

Center Drill

Properties **Feeds & Speeds**

Material:

Cut Material Color:

Number of Flutes:

Tool Number:

Adjust Register:

Cutcom Register:

Axial Offset:

Coolant:

Comments:

Spindle Parameters

Speed: RPM

Direction: ☐ CW ☐ CCW

Feed Rates (in/min)

Plunge: Approach: Engage:

Retract: Departure:

Cut:

Transfer:

☐ Use Rapid ☐ Set

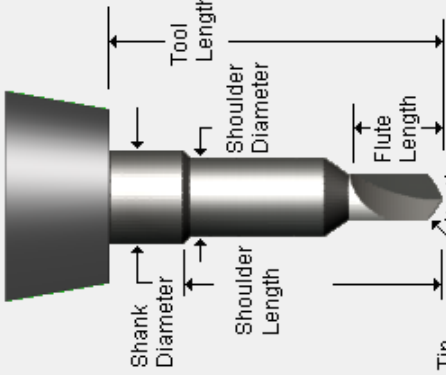
Feed Rate Reduction Factors

Plunge between levels: %

First XY pass: %

Bottom Z Level: %

Cut Depth: in



Shank Diameter:

Shoulder Length:

Shoulder Diameter:

Tool Length:

Flute Length:

Tip Diameter:

Tip Angle:

Name:

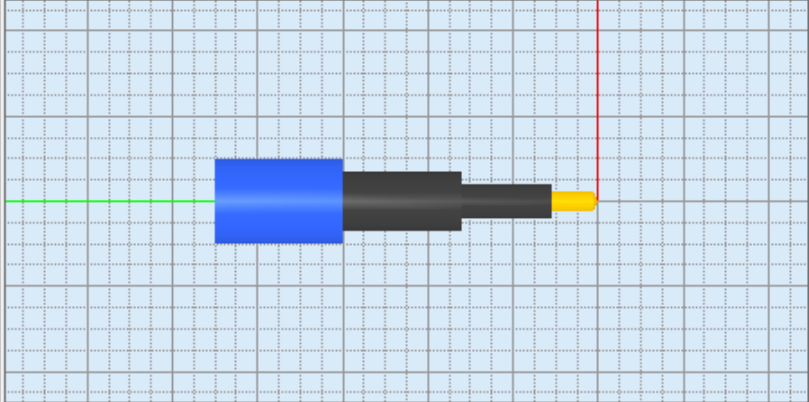
Shank Dia.: Shoulder Len.:

Flute Len.: Tip Dia.:

Tip Angle:

Shoulder Dia.:

Holder:



8.3.3 Reamer Worksheet

Reamer

Properties Feeds & Speeds

Material:

Cut Material Color:

Number of Flutes:

Tool Number:

Adjust Register:

Cutcom Register:

Axial Offset:

Coolant:

Comments:

Spindle Parameters

Speed: RPM

Direction: ☐ CW ☐ CCW

Feed Rates (in/min)

Plunge: Approach: Engage:

Cut: Retract: Departure:

Transfer: ☐ Use Rapid ☐ Set

Feed Rate Reduction Factors

Plunge between levels: %

First XY pass: %

Bottom Z Level: %

Cut Depth: in

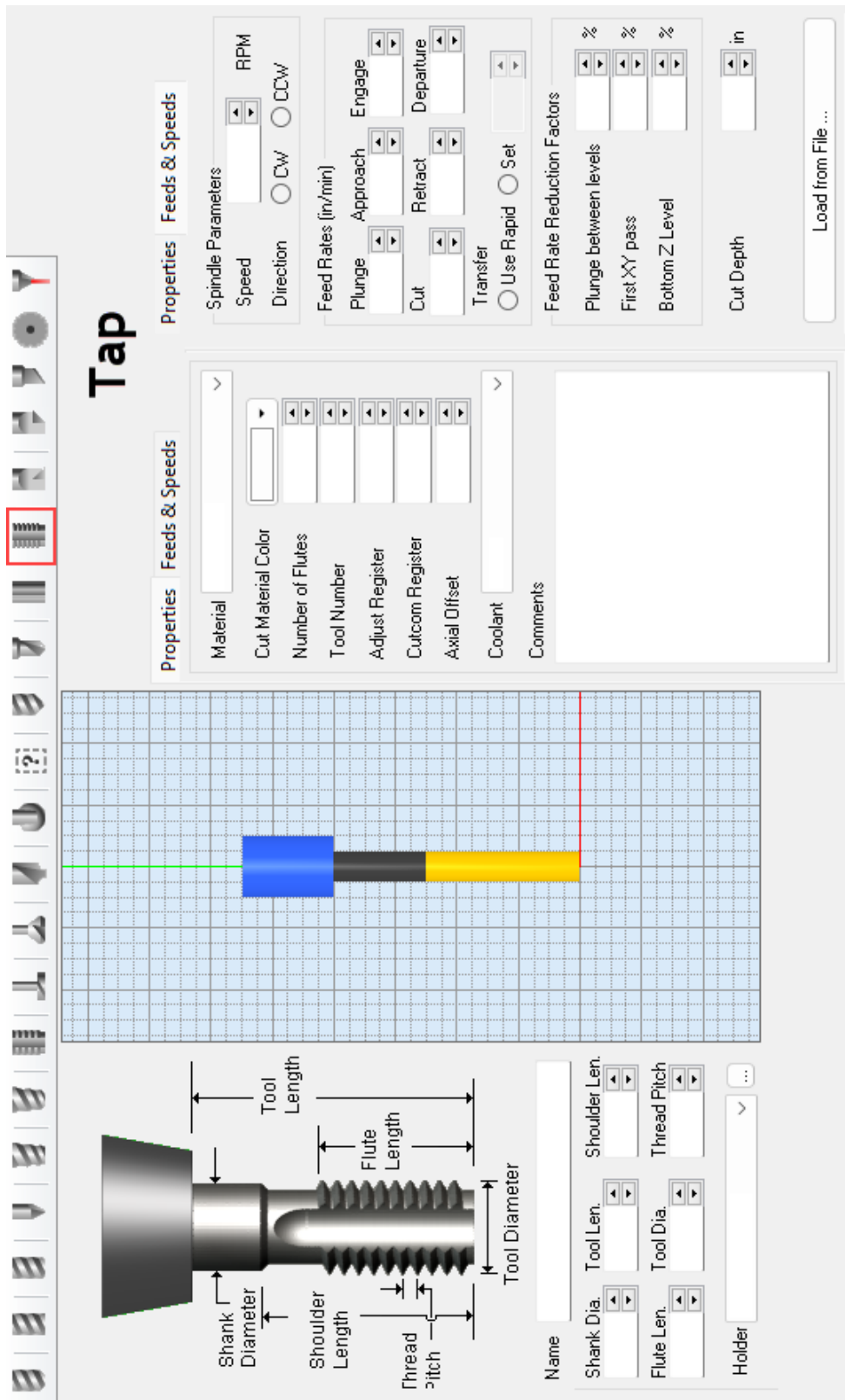
Name:

Shank Dia.: Tool Len.: Shoulder Len.:

Flute Len.: Tool Dia.:

Holder:

Load from File ...



8.3.6 Reverse Bore Worksheet

Reverse Bore

Properties **Feeds & Speeds**

Material

Cut Material Color

Number of Flutes

Tool Number

Adjust Register

Cutcom Register

Axial Offset

Coolant

Comments

Spindle Parameters

Speed RPM

Direction ☐ CW ☐ CCW

Feed Rates (in/min)

Plunge **Approach** **Engage**

Cut **Retract** **Departure**

Transfer

☐ Use Rapid ☐ Set

Feed Rate Reduction Factors

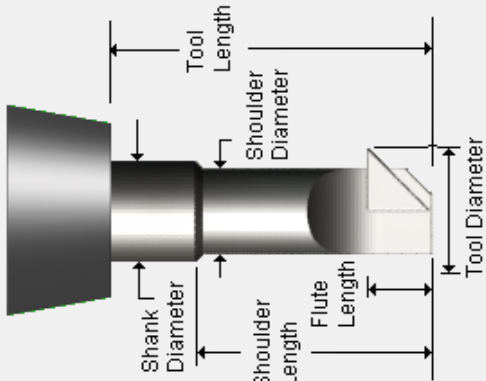
Plunge between levels %

First XY pass %

Bottom Z Level %

Cut Depth in

Load from File ...

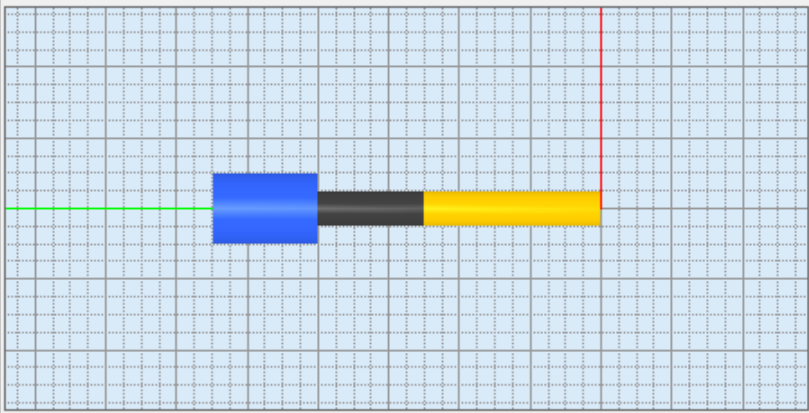


Name

Shank Dia. **Tool Len.** **Shoulder Len.**

Flute Len. **Tool Dia.** **Shoulder Dia.**

Holder



8.4 Turn Tool Worksheets

The following [TURN Tool Worksheets](#) are available:

1. [Diamond Insert Mill Worksheet](#)
2. [Triangular Insert Worksheet](#)
3. [Round Insert Worksheet](#)
4. [Trigon Insert Worksheet](#)
5. [Parallelogram Insert Worksheet](#)
6. [Groove Insert Worksheet](#)
7. [Thread Insert Worksheet](#)
8. [Part Off Insert Worksheet](#)

8.4.1 Diamond Insert Worksheet

Diamond Insert

Properties | Feeds & Speeds

Material

Cut Material Color

Number of Flutes

Tool Number

Adjust Register

Cutcom Register

Axial Offset

Coolant

Comments

Spindle Parameters

☐ Constant Surface Speed (CSS)

Speed RPM

Max Speed RPM

Direction ☐ CW ☐ CCW

Feed Rates (in/rev)

☐ IPR ☐ IPM

Plunge Approach Engage

Cut Retract Departure

Transfer ☐ Use Rapid ☐ Set

Cut Depth in

Inscribed Circle

Tip Angle

Tip Radius

Thickness

Name

Inscribed Circle Radius Tip Radius Tip Angle

Thickness Relief Angle Lead Angle

Orientation ☐ OD Forward ☐ OD Backward ☐ ID Forward ☐ ID Backward

Load from File ...

8.4.2 Triangular Insert Worksheet

Triangular Insert

Properties

Material >

Cut Material Color >

Number of Flutes >

Tool Number >

Adjust Register >

Cutcom Register >

Axial Offset >

Coolant >

Comments >

Material

Cut Material Color

Number of Flutes

Tool Number

Adjust Register

Cutcom Register

Axial Offset

Coolant

Comments

Feeds & Speeds

☐ Constant Surface Speed (CSS)

Speed > RPM

Max Speed > RPM

Direction > ☐ CW ☐ CCW

Feed Rates (in/rev)

☐ IPR ☐ RPM

Plunge > Approach > Engage >

Cut > Retract > Departure >

Transfer >

☐ Use Rapid ☐ Set

Cut Depth > in

Nose Angle

Tip Radius

Thickness

Name >

Inscribed Circle Radius > Tip Radius > Tip Angle >

Thickness > Relief Angle > Lead Angle >

Orientation

☐ OD Forward ☐ OD Backward

☐ ID Forward ☐ ID Backward

Load from File ...

8.4.3 Round Insert Worksheet

Round Insert

Properties | Feeds & Speeds

Material >

Cut Material Color >

Number of Flutes >

Tool Number >

Adjust Register >

Cutcom Register >

Axial Offset >

Coolant >

Comments >

Spindle Parameters

☐ Constant Surface Speed (CSS)

Speed RPM

Max Speed RPM

Direction CW CCW

Feed Rates (in/rev)

☐ IPR ☐ IPM

Plunge > Approach > Engage >

Cut > Retract > Departure >

Transfer >

☐ Use Rapid ☐ Set >

Cut Depth > in

Name Circular Insert1

Radius 0.05

Thickness 0.125

Load from File ...

8.4.5 Parallelogram Insert Worksheet

Parallelogram Insert



Properties **Feeds & Speeds**

Material

Cut Material Color

Number of Flutes

Tool Number

Adjust Register

Cutcom Register

Axial Offset

Coolant

Comments

Properties **Feeds & Speeds**

☐ Constant Surface Speed (CSS)

Speed RPM

Max Speed RPM

Direction ☐ CW ☐ CCW

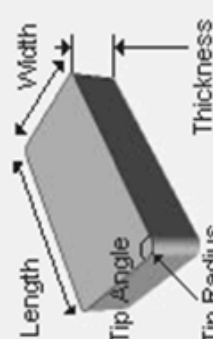
Feed Rates (in/rev) ☐ IPR ☐ IPM

Plunge Approach Engage

Cut Retract Departure

Transfer ☐ Use Rapid ☐ Set

Cut Depth in



Name

Tip Radius

Length

Thickness

Tip Angle

Width

Relief Angle

Lead Angle

Orientation

☐ OD Forward ☐ OD Backward

☐ ID Forward ☐ ID Backward

Load from File ...

Groove Insert

Groove Insert

Name

Tip Radius **Length** **Total Length**

Width **Thickness** **Total Width**

Program Point
☐ Left ☐ Center ☐ Right

Properties
Feeds & Speeds

Material >

Cut Material Color >

Number of Flutes >

Tool Number >

Adjust Register >

Cutcom Register >

Axial Offset >

Coolant >

Comments

Spindle Parameters

☐ Constant Surface Speed (CSS)

Speed > RPM

Max Speed > RPM

Direction ☐ CW ☐ CCW

Feed Rates (in/rev)

☐ IPR ☐ IPM

Plunge > **Approach** > **Engage** >

Cut > **Retract** > **Departure** >

Transfer
☐ Use Rapid ☐ Set >

Cut Depth > in

[Load from File ...](#)

8.4.7 Thread Insert Worksheet

Thread Insert

Properties
Feeds & Speeds

Material >

Cut Material Color >

Number of Flutes >

Tool Number >

Adjust Register >

Cutcom Register >

Axial Offset >

Coolant >

Comments

Properties
Feeds & Speeds

☐ Constant Surface Speed (CSS)

Speed > RPM

Max Speed > RPM

Direction > CW > CCW

Feed Rates (in/rev)

☐ IPR ☐ IPM

Plunge > Approach > Engage >

Cut > Retract > Departure >

Transfer >

☐ Use Rapid ☐ Set

Cut Depth > in

Name >

Tip Radius >

Tip Angle >

Length >

Width >

Thickness >

Load from File ...

Parting Off Insert

Parting Off Insert

Name

Tip Radius		Tip Angle	
Length		Width	
		Thickness	

Properties Feeds & Speeds

>

Material	
Cut Material Color	
Number of Flutes	
Tool Number	
Adjust Register	
Cutcom Register	
Axial Offset	
Coolant	
Comments	

Properties Feeds & Speeds

Spindle Parameters

☐ Constant Surface Speed (CSS)

Speed		RPM
Max Speed		RPM
Direction	☐ CW ☐ CCW	

Feed Rates (in/rev)

☐ IPR ☐ IPM

Plunge		Approach		Engage	
Cut		Retract		Departure	
Transfer	☐ Use Rapid ☐ Set				
Cut Depth		in			

Load from File ...

8.5 Feeds & Speeds Data

8.5.1 Feeds & Speeds Data (INCH)

For INCH units, the [Surface Speed](#) and [Feed/Tooth](#) provided in the [Feeds & Speeds Calculator](#) are derived from the file [FeedsSpeedsDataINCH.xml](#). You can customize this file to add your own materials.

This file is located at: [C:\ProgramData\MecSoft Corporation\RhinoCAM for Rhino x.x\Materials](#)

! CAUTION!! - Before editing this file - Make sure you create a copy of this file and keep it in a safe place, outside of the install path of [RhinoCAM](#). Also, make sure you maintain the XML format of this file or the [Feeds & Speeds Calculator](#) may not work properly.

Here is the tabulated format of the file. It will help you understand the XML files format for recording data.

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
	MILLING	CERAMIC	400.00	0.0040

Below is an excerpt from the [FeedsSpeedsDataINCH.xml](#) XML file:

```
<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>
<Material>
  <Name>ALUMINUM - 5050</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>
...
...

```

8.5.2 Feeds & Speeds Data (METRIC)

For METRIC units, the [Surface Speed](#) and [Feed/Tooth](#) provided in the [Feeds & Speeds Calculator](#) are derived from the file [FeedsSpeedsDataMM.xml](#). You can customize this file to add your own materials.

This file is located at: [C:\ProgramData\MecSoft Corporation\RhinoCAM for Rhino x.x\Materials](#)

! CAUTION!! - Before editing this file - Make sure you create a copy of this file and keep it in a safe place, outside of the install path of [RhinoCAM](#). Also, make sure you maintain the XML format of this file or the [Feeds & Speeds Calculator](#) may not work properly.

Here is the tabulated format of the file. It will help you understand the XML files format for recording data.

Stock Material	Toolpath Type	Tool Material	Surface Speed	Feed/Tooth
ALUMINUM - 2024	MILLING	CARBIDE	487.68	0.1016
	MILLING	HSS	121.92	0.1016
	MILLING	CERAMIC	121.92	0.1016
	DRILLING	CARBIDE	292.61	0.1219
	DRILLING	HSS	73.15	0.1219
	DRILLING	CERAMIC	73.15	0.1219
	TURNING	CARBIDE	548.64	0.5080
	TURNING	CERAMIC	548.64	0.5080
	TURNING	CERMET	548.64	0.5080
ALUMINUM - 5050	MILLING	CARBIDE	487.68	0.1016

Below is an excerpt from the *FeedsSpeedsDataMM.xml* XML file:

```
<Version>1.0</Version>
<Units>Metric</Units>
<FeedsSpeeds>
  <Material>
    <Name>ALUMINUM - 2025</Name>
    <TextureFile>ALUMINUM.bmp</TextureFile>
    <FeedsSpeedsRecord>MILLING, CARBIDE, 487.68, 0.1016</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>MILLING, HSS, 121.92, 0.1016</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>MILLING, CERAMIC, 121.92, 0.1016</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>DRILLING, CARBIDE, 292.61, 0.1219</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>DRILLING, HSS, 73.15, 0.1219</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>DRILLING, CERAMIC, 73.15, 0.1219</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>TURNING, CARBIDE, 548.64, 0.5080</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>TURNING, CERAMIC, 548.64, 0.5080</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>TURNING, CERMET, 548.64, 0.5080</FeedsSpeedsRecord>
  </Material>
  <Material>
    <Name>ALUMINUM - 5050</Name>
    <TextureFile>ALUMINUM.bmp</TextureFile>
    <FeedsSpeedsRecord>MILLING, CARBIDE, 487.68, 0.1016</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>MILLING, HSS, 121.92, 0.1016</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>MILLING, CERAMIC, 121.92, 0.1016</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>DRILLING, CARBIDE, 292.61, 0.1219</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>DRILLING, HSS, 73.15, 0.1219</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>DRILLING, CERAMIC, 73.15, 0.1219</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>TURNING, CARBIDE, 548.64, 0.5080</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>TURNING, CERAMIC, 548.64, 0.5080</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>TURNING, CERMET, 548.64, 0.5080</FeedsSpeedsRecord>
  </Material>
</Material>
```

...
...

8.6 Default Tool Libraries

8.6.1 Default English Tools Library

The following table lists each tool included in the [DefaultEnglishTools.csv](#) library file.

This file is located at: [C:\ProgramData\MecSoft Corporation\RhinoCAM for Rhino x.x\Data](#)

Name	Tool Type	Diameter	Corner Radius	Taper	Flute Length	Tool Length	Tool #	Tool Material	Spindle RPM	Cut Feed	Adjust R	Cutcom R
.375 4FL (2.550 Loc)	MIL	0.375 in	0 in	0 deg	2.55 in	4.23 in	0	Carbide	24446	14.67	0	0
.375 4FL (1.150 Loc)	MIL	0.375 in	0 in	0 deg	1.15 in	3.05 in	0	Carbide	24446	14.67	0	0
.375 4FL (.950 Loc)	MIL	0.375 in	0 in	0 deg	0.95 in	1.5 in	0	Carbide	24446	14.67	0	0
.375 2FL (1.180 Loc)	MIL	0.375 in	0 in	0 deg	1.18 in	3.04 in	0	Carbide	24446	14.67	0	0
.375 2FL (.900 Loc)	MIL	0.375 in	0 in	0 deg	0.9 in	2.53 in	0	Carbide	24446	14.67	0	0
1/8-Bore Faker	MIL	0.124 in	0 in	0 deg	0.75 in	2.3 in	0	Carbide	24446	14.67	0	0
113 Dremel .038 (.200 Lo ...	MIL	0.038 in	0 in	0 deg	0.2 in	1.8 in	0	HSS	24446	14.67	0	0
.062 2 FL (.250 Loc)	MIL	0.0625 in	0 in	0 deg	0.25 in	1.5 in	0	Carbide	24446	14.67	0	0
.750 Wood Rougher	MIL	0.75 in	0 in	0 deg	0.75 in	2 in	0	HSS	24446	14.67	0	0
.250 2 FL (.800 Loc)	MIL	0.25 in	0 in	0 deg	0.8 in	2.5 in	0	Carbide	24446	14.67	0	0
.250 4 FL (.800 Loc)	MIL	0.25 in	0 in	0 deg	0.8 in	2.5 in	0	Carbide	24446	14.67	0	0
.250 4 FL (1.150 Loc)	MIL	0.25 in	0 in	0 deg	1.15 in	3.04 in	0	Carbide	24446	14.67	0	0
.250 4 FL (1.730 Loc) 3/...	MIL	0.25 in	0 in	0 deg	1.73 in	3.54 in	0	HSS	24446	14.67	0	0
.187 2 FL (.800 Loc)	MIL	0.187 in	0 in	0 deg	0.8 in	2.54 in	0	Carbide	24446	14.67	0	0
.187 4 FL (.800 Loc)	MIL	0.187 in	0 in	0 deg	0.8 in	2.5 in	0	Carbide	24446	14.67	0	0
.187 4 FL (.650 Loc)	MIL	0.187 in	0 in	0 deg	0.65 in	2 in	0	Carbide	24446	14.67	0	0
.187 2 FL (.650 Loc)	MIL	0.187 in	0 in	0 deg	0.65 in	2 in	0	Carbide	24446	14.67	0	0
.125 2 FL (.740 Loc) 25...	MIL	0.125 in	0 in	0 deg	0.74 in	2.15 in	0	HSS	24446	14.67	0	0
.373 Wood Rougher (.7 ...	MIL	0.373 in	0 in	0 deg	0.75 in	2 in	0	HSS	24446	14.67	0	0
.125 4FL (750 Loc)	MIL	0.125 in	0 in	0 deg	0.75 in	2.3 in	0	Carbide	24446	14.67	0	0
.125 2FL (750 Loc)	MIL	0.125 in	0 in	0 deg	0.75 in	2.3 in	0	Carbide	24446	14.67	0	0
.125 4FL (520 Loc)	MIL	0.125 in	0 in	0 deg	0.52 in	2.5 in	0	Carbide	24446	14.67	0	0
.125 2FL (520 Loc)	MIL	0.125 in	0 in	0 deg	0.52 in	2.5 in	0	Carbide	24446	14.67	0	0
.250 Wood Rougher (1.0...	MIL	0.25 in	0 in	0 deg	1.04 in	2.25 in	0	HSS	24446	14.67	0	0
.312 Wood Rougher (.7 ...	MIL	0.312 in	0 in	0 deg	0.76 in	2 in	0	HSS	24446	14.67	0	0
.250 Wood Rougher (.7 ...	MIL	0.25 in	0 in	0 deg	0.77 in	2 in	0	Carbide	24446	14.67	0	0
.375 2 FL (.520 Loc)	MIL	0.125 in	0.0625 in	0 deg	0.52 in	1.5 in	0	Carbide	24446	14.67	0	0
.375 4 FL (.520 Loc)	MIL	0.125 in	0.0625 in	0 deg	0.52 in	1.5 in	0	Carbide	24446	14.67	0	0
.187 2 FL (.650 Loc)	MIL	0.1875 in	0.09375 in	0 deg	0.65 in	2 in	0	Carbide	24446	14.67	0	0
.187 4 FL (.650 Loc)	MIL	0.1875 in	0.09375 in	0 deg	0.65 in	2 in	0	Carbide	24446	14.67	0	0
.250 2 FL Wood Rougher	MIL	0.25 in	0.125 in	0 deg	0.25 in	1.5 in	0	HSS	24446	14.67	0	0
.370 2 FL Wood Rougher	MIL	0.37 in	0.185 in	0 deg	0.265 in	1.25 in	0	HSS	24446	14.67	0	0
.250 Rasp Drill Point Side ...	MIL	0.243 in	0.1215 in	0 deg	1 in	2 in	0	HSS	24446	14.67	0	0

Name	Tool Type	Diameter	Corner Radius	Taper	Flute Length	Tool Length	Tool #	Tool Material	Spindle RPM	Cut Feed	Adjust R	Outcom R
250 4 FL (.750 Loc)	Mil	0.25 in	0.125 in	0 deg	0.75 in	2.5 in	0	Carbide	24446	14.67	0	0
105 Dremel 1/32 Ball	Mil	0.028 in	0.014 in	0 deg	0.028 in	1.75 in	0	HSS	24446	14.67	0	0
.0312 4 FL (.150 Loc)	Mil	0.0312 in	0.0156 in	0 deg	0.15 in	1.5 in	0	Carbide	24446	14.67	0	0
.0312 2 FL (.150 Loc)	Mil	0.0312 in	0.0156 in	0 deg	0.15 in	1.5 in	0	Carbide	24446	14.67	0	0
.062 4 FL (.365 Loc)	Mil	0.0625 in	0.03125 in	0 deg	0.365 in	1.5 in	0	HSS	24446	14.67	0	0
.375 4 FL (1.050 Loc)	Mil	0.375 in	0.1875 in	0 deg	1.05 in	1.54 in	0	HSS	24446	14.67	0	0
105 Dremel .028 1/32	Mil	0.028 in	0.014 in	0 deg	0.03 in	1.75 in	0	HSS	24446	14.67	0	0
106 Dremel .053 1/16	Mil	0.053 in	0.0265 in	0 deg	0.06 in	1.76 in	0	HSS	24446	14.67	0	0
.125 2 FL (.530 Loc)	Mil	0.125 in	0.0625 in	0 deg	0.53 in	1.5 in	0	Carbide	24446	14.67	0	0
.062 2 FL (.365 Loc)	Mil	0.0625 in	0.03125 in	0 deg	0.365 in	1.5 in	0	Carbide	24446	14.67	0	0
.312 2 FL (.850 Loc)	Mil	0.375 in	0.1875 in	0 deg	0.85 in	2 in	0	Carbide	24446	14.67	0	0
.125 4 FL (.530 Loc)	Mil	0.125 in	0.0625 in	0 deg	0.53 in	1.5 in	0	Carbide	24446	14.67	0	0
100 Dremel 1/4 Ball (1/8...	Mil	0.028 in	0.014 in	10 deg	0.028 in	1.75 in	0	HSS	24446	14.67	0	0
114 Dremel 5/16 Ball (3/...	Mil	0.028 in	0.014 in	10 deg	0.028 in	1.75 in	0	HSS	24446	14.67	0	0
107 Dremel .087 Ball (3/...	Mil	0.087 in	0.0435 in	10 deg	0.375 in	1.75 in	0	HSS	24446	14.67	0	0
106 Dremel .049 Ball (3/...	Mil	0.049 in	0.0245 in	10 deg	0.3 in	1.75 in	0	HSS	24446	14.67	0	0
105 Dremel 1/32 Ball (3/...	Mil	0.028 in	0.014 in	10 deg	0.028 in	1.75 in	0	HSS	24446	14.67	0	0
.030 Engraver	Mil	0.03 in	0.001 in	30 deg	0.15 in	4.5 in	0	HSS	24446	14.67	0	0
1/16 Drill	Drill	0.0625 in	0 in	0 deg	2.5 in	4 in	0	HSS	14667	11.73	0	0
1/16-Drill	Drill	0.0625 in	0 in	0 deg	1.15 in	2.75 in	0	HSS	14667	11.73	0	0
1/4-Drill	Drill	0.25 in	0 in	0 deg	1 in	3 in	0	HSS	14667	11.73	0	0
1/8-Drill	Drill	0.125 in	0 in	0 deg	1 in	3 in	0	HSS	14667	11.73	0	0

8.6.2 Default Metric Tools Library

The following table lists each tool included in the [DefaultMetricTools.csv](#) library file.

This file is located at: [C:\ProgramData\MecSoft Corporation\RhinoCAM for Rhino x.x\Data](#)

Name	Tool Type	Diameter	Corner Radius	Taper	Flute Length	Tool Length	Tool #	Tool Material	Spindle RPM	Cut Feed	Adjust R	Cutcom R
4mm End Mill	Mil	4 mm	0 mm	0 deg	10 mm	14 mm	0	HSS	25872	87.62	0	0
30mm End Mill	Mil	30 mm	0 mm	0 deg	75 mm	105 mm	0	HSS	25872	87.62	0	0
25mm End Mill	Mil	25 mm	0 mm	0 deg	62.5 mm	87.5 mm	0	HSS	25872	87.62	0	0
20mm End Mill	Mil	20 mm	0 mm	0 deg	50 mm	70 mm	0	HSS	25872	87.62	0	0
16mm End Mill	Mil	16 mm	0 mm	0 deg	40 mm	56 mm	0	HSS	25872	87.62	0	0
12mm End Mill	Mil	12 mm	0 mm	0 deg	30 mm	42 mm	0	HSS	25872	87.62	0	0
10mm End Mill	Mil	10 mm	0 mm	0 deg	25 mm	35 mm	0	HSS	25872	87.62	0	0
8mm End Mill	Mil	8 mm	0 mm	0 deg	20 mm	28 mm	0	HSS	25872	87.62	0	0
6mm End Mill	Mil	6 mm	0 mm	0 deg	15 mm	21 mm	0	HSS	25872	87.62	0	0
5mm End Mill	Mil	5 mm	0 mm	0 deg	12.5 mm	17.5 mm	0	HSS	25872	87.62	0	0
3mm End Mill	Mil	3 mm	0 mm	0 deg	7.5 mm	10.5 mm	0	HSS	25872	87.62	0	0
2mm End Mill	Mil	2 mm	0 mm	0 deg	5 mm	7 mm	0	HSS	25872	87.62	0	0
1.5mm End Mill	Mil	1.5 mm	0 mm	0 deg	3 mm	5 mm	0	HSS	25872	87.62	0	0
1mm End Mill	Mil	1 mm	0 mm	0 deg	2.5 mm	3.5 mm	0	HSS	25872	87.62	0	0
0.8mm End Mill	Mil	0.8 mm	0 mm	0 deg	1.5 mm	3 mm	0	HSS	25872	87.62	0	0
0.6mm End Mill	Mil	0.6 mm	0 mm	0 deg	1.5 mm	3 mm	0	HSS	25872	87.62	0	0
0.5mm End Mill	Mil	0.5 mm	0 mm	0 deg	1.5 mm	3 mm	0	HSS	25872	87.62	0	0
50mm End Mill	Mil	50 mm	0 mm	0 deg	100 mm	130 mm	0	HSS	25872	87.62	0	0
25mm Ballnose	Mil	25 mm	12.5 mm	0 deg	62.5 mm	100 mm	0	HSS	25872	87.62	0	0
20mm Ballnose	Mil	20 mm	10 mm	0 deg	50 mm	80 mm	0	HSS	25872	87.62	0	0
18mm Ballnose	Mil	18 mm	9 mm	0 deg	45 mm	72 mm	0	HSS	25872	87.62	0	0
16mm Ballnose	Mil	16 mm	8 mm	0 deg	40 mm	64 mm	0	HSS	25872	87.62	0	0
12mm Ballnose	Mil	12 mm	6 mm	0 deg	30 mm	48 mm	0	HSS	25872	87.62	0	0
10mm Ballnose	Mil	10 mm	5 mm	0 deg	25 mm	40 mm	0	HSS	25872	87.62	0	0
8mm Ballnose	Mil	8 mm	4 mm	0 deg	20 mm	32 mm	0	HSS	25872	87.62	0	0
6mm Ballnose	Mil	6 mm	3 mm	0 deg	15 mm	24 mm	0	HSS	25872	87.62	0	0
0.5mm Ballnose	Mil	0.5 mm	0.25 mm	0 deg	1.5 mm	5 mm	0	HSS	25872	87.62	0	0
5mm Ballnose	Mil	5 mm	2.5 mm	0 deg	12.5 mm	20 mm	0	HSS	25872	87.62	0	0
4mm Ballnose	Mil	4 mm	2 mm	0 deg	10 mm	16 mm	0	HSS	25872	87.62	0	0
3mm Ballnose	Mil	3 mm	1.5 mm	0 deg	7.5 mm	12 mm	0	HSS	25872	87.62	0	0
2mm Ballnose	Mil	2 mm	1 mm	0 deg	5 mm	8 mm	0	HSS	25872	87.62	0	0

Name	Tool Type	Diameter	Corner Radius	Taper	Flute Length	Tool Length	Tool #	Tool Material	Spindle RPM	Cut Feed	Adjust R	Cutcom R
1.5mm Ballnose	Mil	1.5 mm	0.75 mm	0 deg	3 mm	5 mm	0	HSS	25872	87.62	0	0
1.2mm Ballnose	Mil	1.2 mm	0.6 mm	0 deg	4 mm	10 mm	0	HSS	25872	87.62	0	0
1mm Ballnose	Mil	1 mm	0.5 mm	0 deg	2.5 mm	5 mm	0	HSS	25872	87.62	0	0
0.8mm Ballnose	Mil	0.8 mm	0.4 mm	0 deg	2 mm	5 mm	0	HSS	25872	87.62	0	0
0.5mm Ballnose	Mil	0.5 mm	0.3 mm	0 deg	1.5 mm	5 mm	0	HSS	25872	87.62	0	0
30mm Ballnose	Mil	30 mm	15 mm	0 deg	75 mm	120 mm	0	HSS	25872	87.62	0	0
6mm R 0.5	Mil	6 mm	0.5 mm	0 deg	12.5 mm	17.5 mm	0	HSS	25872	87.62	0	0
6mm R 1.0	Mil	6 mm	1 mm	0 deg	10 mm	20 mm	0	HSS	25872	87.62	0	0
8mm R 1.0	Mil	8 mm	1 mm	0 deg	7 mm	25 mm	0	HSS	25872	87.62	0	0
8mm R 1.5	Mil	8 mm	1.5 mm	0 deg	7 mm	25 mm	0	HSS	25872	87.62	0	0
8mm R 2.0	Mil	8 mm	2 mm	0 deg	7 mm	25 mm	0	HSS	25872	87.62	0	0
10mm R 1.5	Mil	10 mm	1.5 mm	0 deg	25 mm	35 mm	0	HSS	25872	87.62	0	0
10mm R 2.0	Mil	10 mm	2 mm	0 deg	25 mm	35 mm	0	HSS	25872	87.62	0	0
12mm R 2.0	Mil	12 mm	2 mm	0 deg	25 mm	35 mm	0	HSS	25872	87.62	0	0
16mm R 2.0	Mil	16 mm	2 mm	0 deg	16 mm	50 mm	0	HSS	25872	87.62	0	0
50mm Surfacar	Mil	58 mm	4 mm	0 deg	8 mm	40 mm	0	HSS	25872	87.62	0	0
63mm Surfacar	Mil	71 mm	4 mm	0 deg	8 mm	50 mm	0	HSS	25872	87.62	0	0
80mm Surfacar	Mil	88 mm	4 mm	0 deg	8 mm	50 mm	0	HSS	25872	87.62	0	0
63 Alexa Surfacar	Mil	75 mm	6 mm	0 deg	6 mm	100 mm	0	HSS	25872	87.62	0	0
17.5° Erg R 0.1	Mil	0.2 mm	0.1 mm	17.5 deg	4.5 mm	10 mm	0	HSS	25872	87.62	0	0
22° Erg R 0.1	Mil	0.2 mm	0.1 mm	22 deg	5.1 mm	10 mm	0	HSS	25872	87.62	0	0
17.5° Erg R 0.05	Mil	0.1 mm	0.05 mm	17.5 deg	4.6 mm	15 mm	0	HSS	25872	87.62	0	0
1.0mm HSS Drill	Drill	1 mm	0 mm	0 deg	2.5 mm	20 mm	0	HSS	15523	63.08	0	0
1.5mm HSS Drill	Drill	1.5 mm	0 mm	0 deg	3.75 mm	30 mm	0	HSS	15523	63.08	0	0
2.0mm HSS Drill	Drill	2 mm	0 mm	0 deg	5 mm	40 mm	0	HSS	15523	63.08	0	0
2.4mm HSS Drill	Drill	2.4 mm	0 mm	0 deg	6 mm	48 mm	0	HSS	15523	63.08	0	0
2.5mm HSS Drill	Drill	2.5 mm	0 mm	0 deg	6.25 mm	50 mm	0	HSS	15523	63.08	0	0
3.0mm HSS Drill	Drill	3 mm	0 mm	0 deg	7.5 mm	60 mm	0	HSS	15523	63.08	0	0
3.2mm HSS Drill	Drill	3.2 mm	0 mm	0 deg	8 mm	64 mm	0	HSS	15523	63.08	0	0
3.5mm HSS Drill	Drill	3.5 mm	0 mm	0 deg	8.75 mm	70 mm	0	HSS	15523	63.08	0	0
4.0mm HSS Drill	Drill	4 mm	0 mm	0 deg	10 mm	80 mm	0	HSS	15523	63.08	0	0

Name	Tool Type	Diameter	Corner Radius	Taper	Flute Length	Tool Length	Tool #	Tool Material	Spindle RPM	Cut Feed	Adjust R	Cutcom R
4.2mm HSS Drill	Drill	4.2 mm	0 mm	0 deg	10.5 mm	84 mm	0	HSS	15523	63.08	0	0
4.5mm HSS Drill	Drill	4.5 mm	0 mm	0 deg	11.25 mm	90 mm	0	HSS	15523	63.08	0	0
5.0mm HSS Drill	Drill	5 mm	0 mm	0 deg	12.5 mm	100 mm	0	HSS	15523	63.08	0	0
5.5mm HSS Drill	Drill	5.5 mm	0 mm	0 deg	13.75 mm	110 mm	0	HSS	15523	63.08	0	0
6.0mm HSS Drill	Drill	6 mm	0 mm	0 deg	15 mm	120 mm	0	HSS	15523	63.08	0	0
6.5mm HSS Drill	Drill	6.5 mm	0 mm	0 deg	16.25 mm	110 mm	0	HSS	15523	63.08	0	0
7.0mm HSS Drill	Drill	7 mm	0 mm	0 deg	17.5 mm	119 mm	0	HSS	15523	63.08	0	0
7.5mm HSS Drill	Drill	7.5 mm	0 mm	0 deg	18.75 mm	127.5 mm	0	HSS	15523	63.08	0	0
8.0mm HSS Drill	Drill	8 mm	0 mm	0 deg	20 mm	136 mm	0	HSS	15523	63.08	0	0
8.5mm HSS Drill	Drill	8.5 mm	0 mm	0 deg	21.25 mm	144.5 mm	0	HSS	15523	63.08	0	0
9.0mm HSS Drill	Drill	9 mm	0 mm	0 deg	22.5 mm	153 mm	0	HSS	15523	63.08	0	0
9.5mm HSS Drill	Drill	9.5 mm	0 mm	0 deg	23.75 mm	161.5 mm	0	HSS	15523	63.08	0	0
10.0mm HSS Drill	Drill	10 mm	0 mm	0 deg	25 mm	170 mm	0	HSS	15523	63.08	0	0
10.2mm HSS Drill	Drill	10.2 mm	0 mm	0 deg	25.5 mm	173 mm	0	HSS	15523	63.08	0	0
10.5mm HSS Drill	Drill	10.5 mm	0 mm	0 deg	26.25 mm	178.5 mm	0	HSS	15523	63.08	0	0
11.0mm HSS Drill	Drill	11 mm	0 mm	0 deg	27.5 mm	187 mm	0	HSS	15523	63.08	0	0
11.5mm HSS Drill	Drill	11.5 mm	0 mm	0 deg	28.75 mm	195 mm	0	HSS	15523	63.08	0	0
12.0mm HSS Drill	Drill	12 mm	0 mm	0 deg	30 mm	204 mm	0	HSS	15523	63.08	0	0
12.5mm HSS Drill	Drill	12.5 mm	0 mm	0 deg	31.25 mm	212.5 mm	0	HSS	15523	63.08	0	0
13.0mm HSS Drill	Drill	13 mm	0 mm	0 deg	32.5 mm	169 mm	0	HSS	15523	63.08	0	0
13.5mm HSS Drill	Drill	13.5 mm	0 mm	0 deg	33.75 mm	175.5 mm	0	HSS	15523	63.08	0	0
14.0mm HSS Drill	Drill	14 mm	0 mm	0 deg	35 mm	182 mm	0	HSS	15523	63.08	0	0
14.5mm HSS Drill	Drill	14.5 mm	0 mm	0 deg	36.25 mm	188.5 mm	0	HSS	15523	63.08	0	0
15.0mm HSS Drill	Drill	15 mm	0 mm	0 deg	37.5 mm	195 mm	0	HSS	15523	63.08	0	0
15.5mm HSS Drill	Drill	15.5 mm	0 mm	0 deg	38.75 mm	201.5 mm	0	HSS	15523	63.08	0	0
16.0mm HSS Drill	Drill	16 mm	0 mm	0 deg	40 mm	208 mm	0	HSS	15523	63.08	0	0
17.0mm HSS Drill	Drill	17 mm	0 mm	0 deg	42.5 mm	221 mm	0	HSS	15523	63.08	0	0
18.0mm HSS Drill	Drill	18 mm	0 mm	0 deg	45 mm	234 mm	0	HSS	15523	63.08	0	0
20.0mm HSS Drill	Drill	20 mm	0 mm	0 deg	50 mm	260 mm	0	HSS	15523	63.08	0	0
22.0mm HSS Drill	Drill	22 mm	0 mm	0 deg	55 mm	286 mm	0	HSS	15523	63.08	0	0
25.0mm HSS Drill	Drill	25 mm	0 mm	0 deg	62.5 mm	325 mm	0	HSS	15523	63.08	0	0

Name	Tool Type	Diameter	Corner Radius	Taper	Flute Length	Tool Length	Tool #	Tool Material	Spindle RPM	Cut Feed	Adjust R	Cutcom R
28.0mm HSS Drill	Drill	28 mm	0 mm	0 deg	70 mm	364 mm	0	HSS	15523	63.08	0	0
30.0mm HSS Drill	Drill	30 mm	0 mm	0 deg	75 mm	390 mm	0	HSS	15523	63.08	0	0
12 mm 90° CD Drill	Drill	12 mm	0 mm	0 deg	20 mm	30 mm	0	HSS	15523	63.08	0	0
10 mm 90° CD Drill	Drill	10 mm	0 mm	0 deg	20 mm	30 mm	0	HSS	15523	63.08	0	0
8mm 120° CD Drill	Drill	8 mm	0 mm	0 deg	10 mm	30 mm	0	HSS	15523	63.08	0	0
8mm 90° CD Drill	Drill	8 mm	0 mm	0 deg	15 mm	25 mm	0	HSS	15523	63.08	0	0
6mm 90° CD Drill	Drill	6 mm	0 mm	0 deg	10 mm	20 mm	0	HSS	15523	63.08	0	0
5mm 90° CD Drill	Drill	5 mm	0 mm	0 deg	10 mm	62 mm	0	HSS	15523	63.08	0	0
4mm 90° CD Drill	Drill	4 mm	0 mm	0 deg	10 mm	52 mm	0	HSS	15523	63.08	0	0
M14 Tap	Drill	14 mm	0 mm	0 deg	35 mm	112 mm	0	HSS	15523	63.08	0	0
M16 Tap	Drill	16 mm	0 mm	0 deg	40 mm	128 mm	0	HSS	15523	63.08	0	0
M20 Tap	Drill	20 mm	0 mm	0 deg	50 mm	160 mm	0	HSS	15523	63.08	0	0
M22 Tap	Drill	22 mm	0 mm	0 deg	55 mm	160 mm	0	HSS	15523	63.08	0	0
M24 Tap	Drill	24 mm	0 mm	0 deg	60 mm	160 mm	0	HSS	15523	63.08	0	0
M30 Tap	Drill	30 mm	0 mm	0 deg	75 mm	160 mm	0	HSS	15523	63.08	0	0
M36 Tap	Drill	36 mm	0 mm	0 deg	90 mm	200 mm	0	HSS	15523	63.08	0	0
M12 Tap	Drill	12 mm	0 mm	0 deg	30 mm	96 mm	0	HSS	15523	63.08	0	0
M10 Tap	Drill	10 mm	0 mm	0 deg	25 mm	80 mm	0	HSS	15523	63.08	0	0
M8 Tap	Drill	8 mm	0 mm	0 deg	20 mm	64 mm	0	HSS	15523	63.08	0	0
M6 Tap	Drill	6 mm	0 mm	0 deg	15 mm	48 mm	0	HSS	15523	63.08	0	0
M5 Tap	Drill	5 mm	0 mm	0 deg	12.5 mm	40 mm	0	HSS	15523	63.08	0	0
M4 Tap	Drill	4 mm	0 mm	0 deg	10 mm	32 mm	0	HSS	15523	63.08	0	0
M3 Tap	Drill	3 mm	0 mm	0 deg	7.5 mm	30 mm	0	HSS	15523	63.08	0	0
5mm Boring Head	Drill	5 mm	0 mm	0 deg	12.5 mm	50 mm	0	HSS	15523	63.08	0	0
6mm Boring Head	Drill	6 mm	0 mm	0 deg	15 mm	50 mm	0	HSS	15523	63.08	0	0
8mm Boring Head	Drill	8 mm	0 mm	0 deg	20 mm	50 mm	0	HSS	15523	63.08	0	0
10mm Boring Head	Drill	10 mm	0 mm	0 deg	25 mm	50 mm	0	HSS	15523	63.08	0	0
12mm Boring Head	Drill	12 mm	0 mm	0 deg	30 mm	50 mm	0	HSS	15523	63.08	0	0
16mm Boring Head	Drill	16 mm	0 mm	0 deg	40 mm	60 mm	0	HSS	15523	63.08	0	0
18mm Boring Head	Drill	18 mm	0 mm	0 deg	45 mm	60 mm	0	HSS	15523	63.08	0	0
20mm Boring Head	Drill	20 mm	0 mm	0 deg	50 mm	80 mm	0	HSS	15523	63.08	0	0

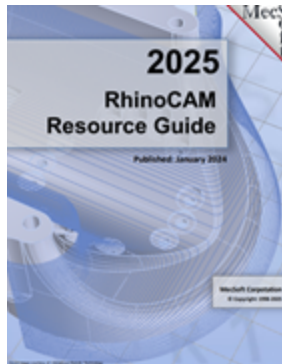
Name	Tool Type	Diameter	Corner Radius	Taper	Flute Length	Tool Length	Tool #	Tool Material	Spindle RPM	Cut Feed	Adjust R	Cutcom R
22mm Boring Head	Drill	22 mm	0 mm	0 deg	55 mm	80 mm	0	HSS	15523	63.08	0	0
24mm Boring Head	Drill	24 mm	0 mm	0 deg	60 mm	100 mm	0	HSS	15523	63.08	0	0
30mm Boring Head	Drill	30 mm	0 mm	0 deg	75 mm	100 mm	0	HSS	15523	63.08	0	0
34mm Boring Head	Drill	34 mm	0 mm	0 deg	85 mm	100 mm	0	HSS	15523	63.08	0	0
36mm Boring Head	Drill	36 mm	0 mm	0 deg	90 mm	100 mm	0	HSS	15523	63.08	0	0
40mm Boring Head	Drill	40 mm	0 mm	0 deg	100 mm	120 mm	0	HSS	15523	63.08	0	0

Where to go for more help

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!](#)

Index

- A -

About

- the MILL Module 9
- this Guide 9
- Using this Guide 10
- About Tapping Feed Rates 48
- Add a Custom Tool 44
- Add a Tool Change Point 52
- Add More Materials 57
- Add Tool Comments 49
- Add your Existing Tools to a Library 32
- After Installing RhinoCAM MILL 11

- C -

- Create a Holder 19
- Create a Tool 13
- Create a Tool Library 22
- Create/Select Tools Dialog 11

- D -

Default Tool Libraries

- INCH 103
- Metric 106

- E -

- Enabling Cutter Compensation 64

- F -

- Feeds & Speeds Calculator 24
- Feeds & Speeds Data
 - INCH 100
 - METRIC 101

- G -

- Getting Ready
 - After Installing RhinoCAM MILL 11

- H -

How To

- Add a Custom Tool 44
- Add a Tool Change Point 52
- Add More Materials 57
- Add Tool Comments 49
- Enabling Cutter Compensation 64
- Find Tool Related Preferences 40
- If Feedrate Values too High/Low 46
- Learn about Tapping Feed Rates 48
- Optimize Machining Time Estimates 48
- Print a Tool List 43

- I -

- If Feedrate Values too High/Low 46

- M -

- More about the Tools Tab 33

- O -

- Optimize Machining Time Estimates 48
- Other Tool Related Topics 40

- P -

- Pre-Installed Tool Libraries 29
- Print a Tool List 43

- T -

Tool

- Create Holder 19
- Create Tool 19
- Tool Related Preferences 40
- Tool Worksheet
 - Ball Mill Worksheet 69
 - Bore Worksheet 89
 - Center Drill Worksheet 86
 - Chamfer Mill Worksheet 73
 - Corner Radius Mill Worksheet 71
 - Dovetail Mill Worksheet 77

Tool Worksheet

Drag Knife Worksheet	81
Drill Worksheet	85
Face Mill Worksheet	76
Fillet Mill Worksheet	78
Flat Mill Worksheet	70
Laser Tool Worksheet	83
Lollipop Mill Worksheet	79
Reamer Worksheet	87
Reverse Bore Worksheet	90
Saw Tool Worksheet	82
Tap Worksheet	88
Taper Mill Worksheet	74
Thread Mill Worksheet	75
User Defined Mill Worksheet	80
Vee Mill Worksheet	72

Tool Worksheets 84

Tools Tab 11

- U -

Using this Guide 10