The Cutting Tools Workbook

RhinoCAM 2026

Published: March 2026

MecSoft Corpotation

© Copyright 1998-2026

Table of Contents

© 2026 MecSoft Corporation

Quick Start

RhinoCAM® MecSoft

CAM Module 2026

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

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

What's New!

What's New in RhinoCAM 2026

The Complete Quick Start Video Play List

Here is a link to the complete 2026 Video Play List

How to Access the Quick Start Guide Documents

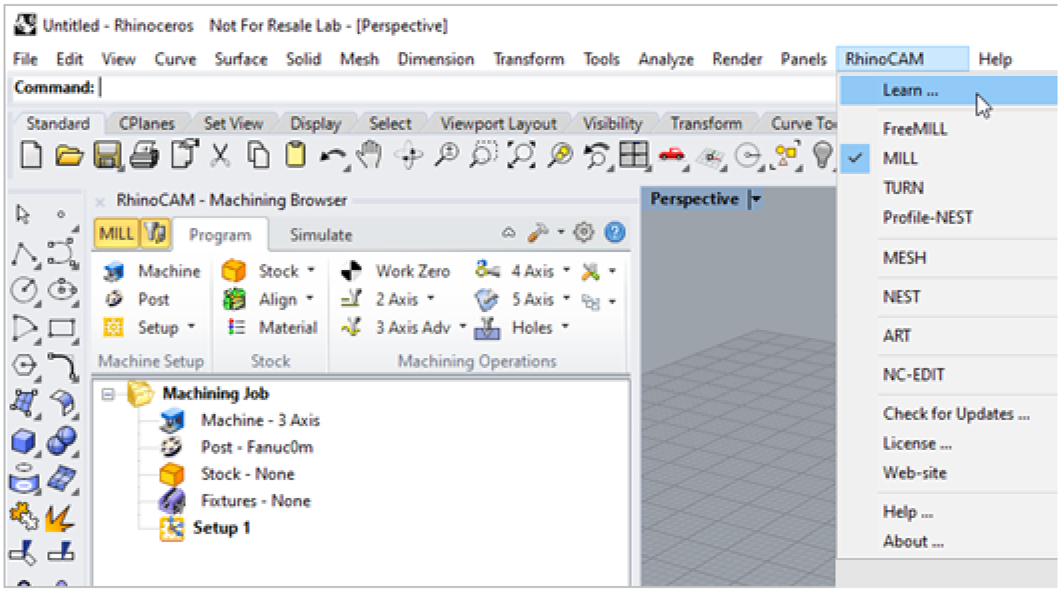
To help you quickly get started in working with each module, select one of the Help buttons located on the RhinoCAM Learning Resources dialog.

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:

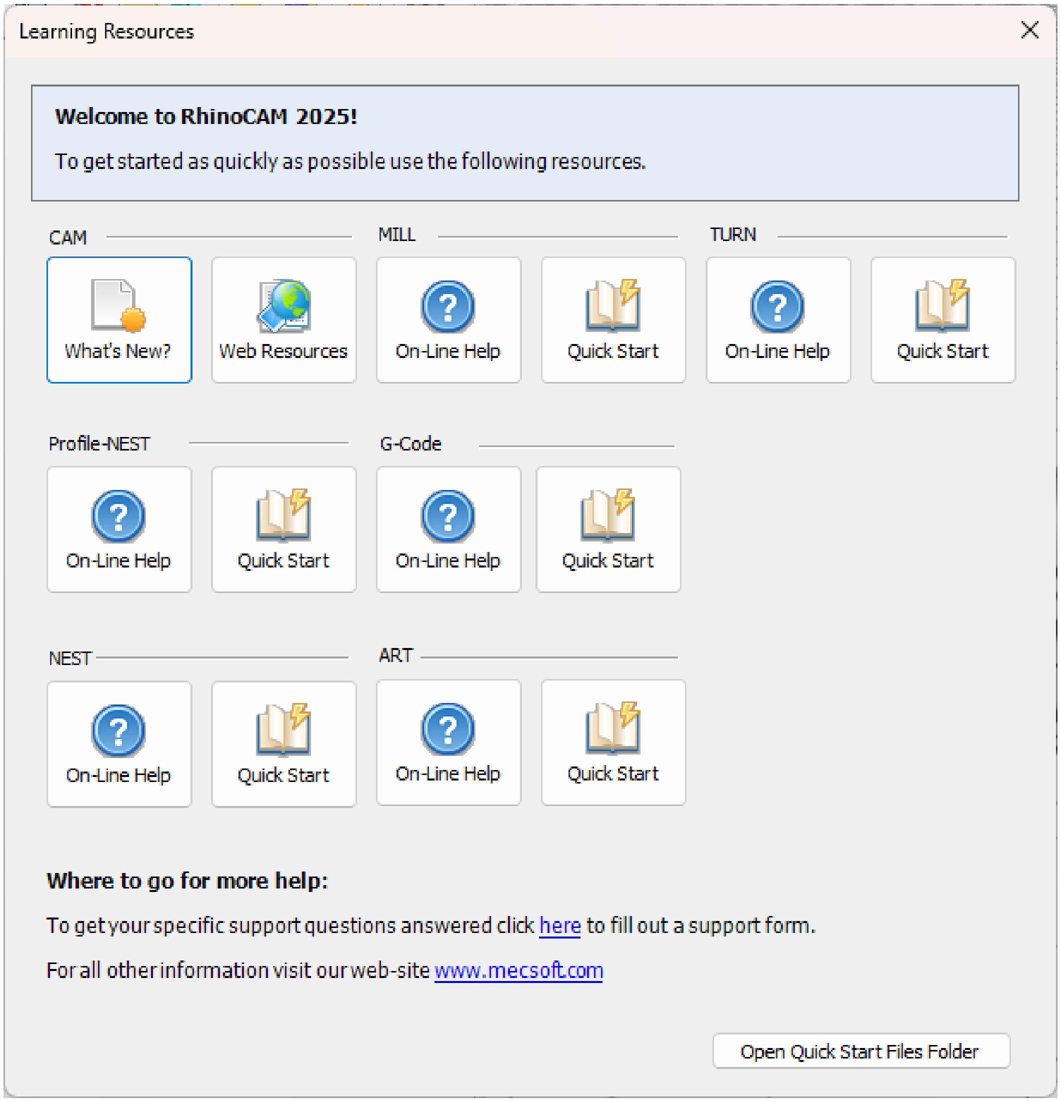
1. From the Rhino Main Menu, drop down the Main menu and select Learn ...



To access the Learning Resources dilog 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.

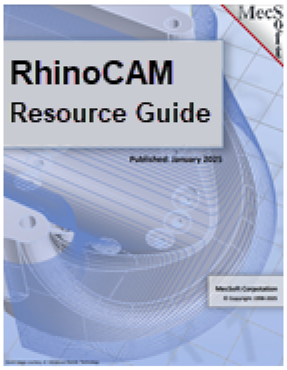


Learning Resources Dialog

Resource Guide

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

2026 RhinoCAM Resource Guide



The 2026 RhinoCAM Resource Guide!

18 Pages

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

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

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.

© 2026 MecSoft Corporation

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

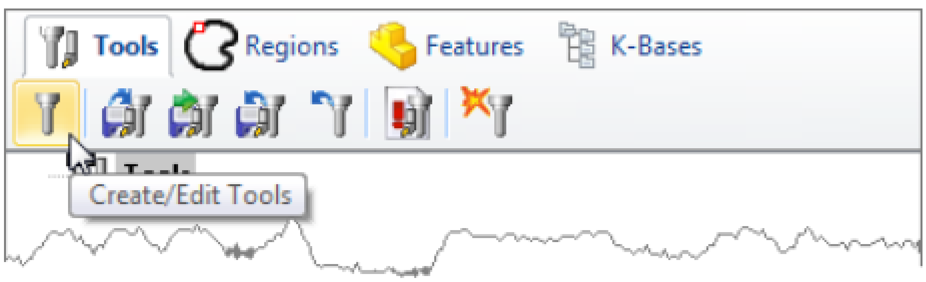
4.1 After Installing RhinoCAM MILL

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 it's 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.

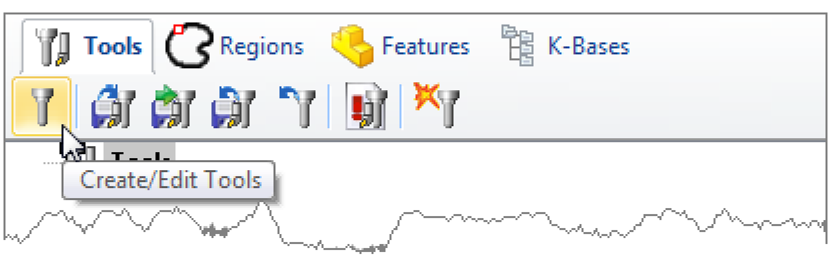
- After installing RhinoCAM load the MILL or TURN modules and locate the Tools tab shown below.



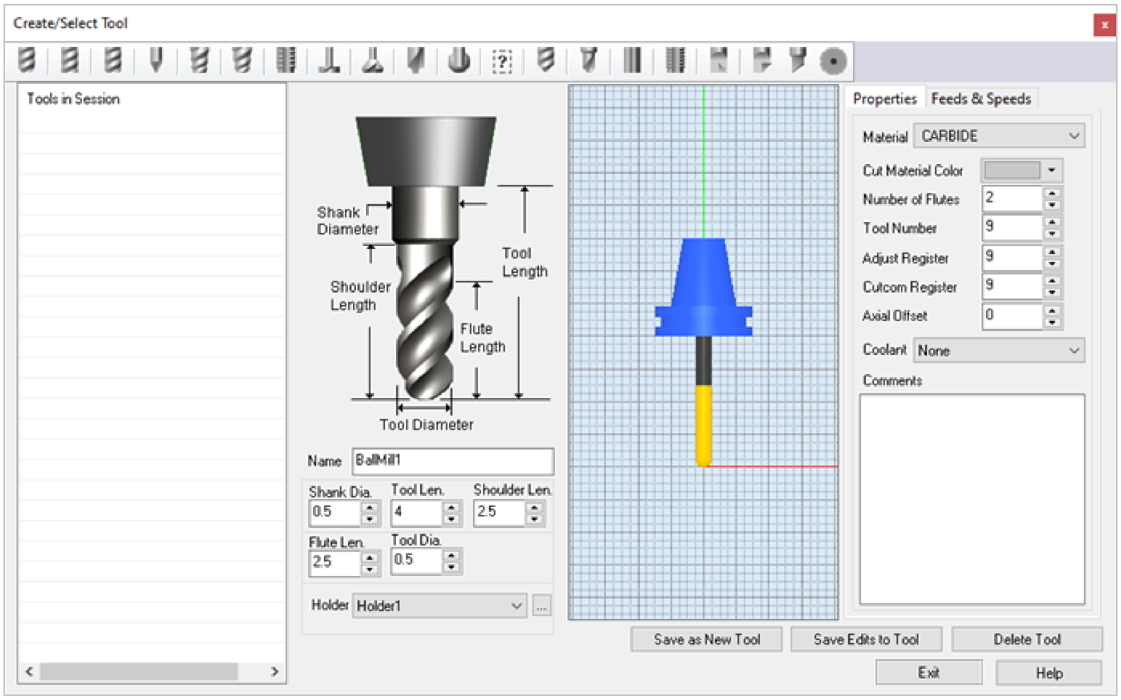
4.3 The Create/Select Tools Dialog

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.



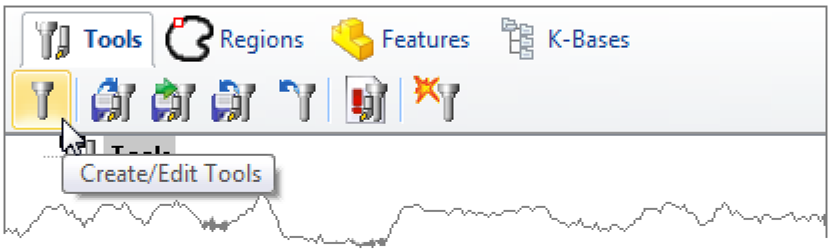
Dialog Box: Create/Select Tools

Creating Tools

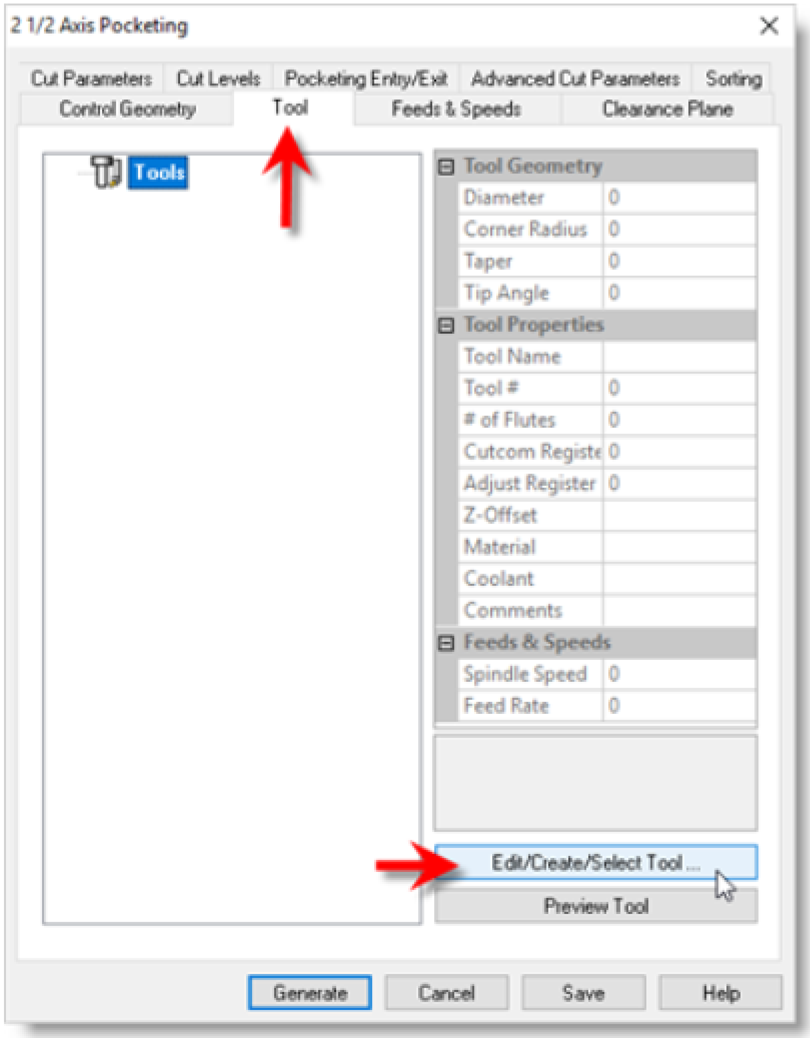
5.1 Create a Tool

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

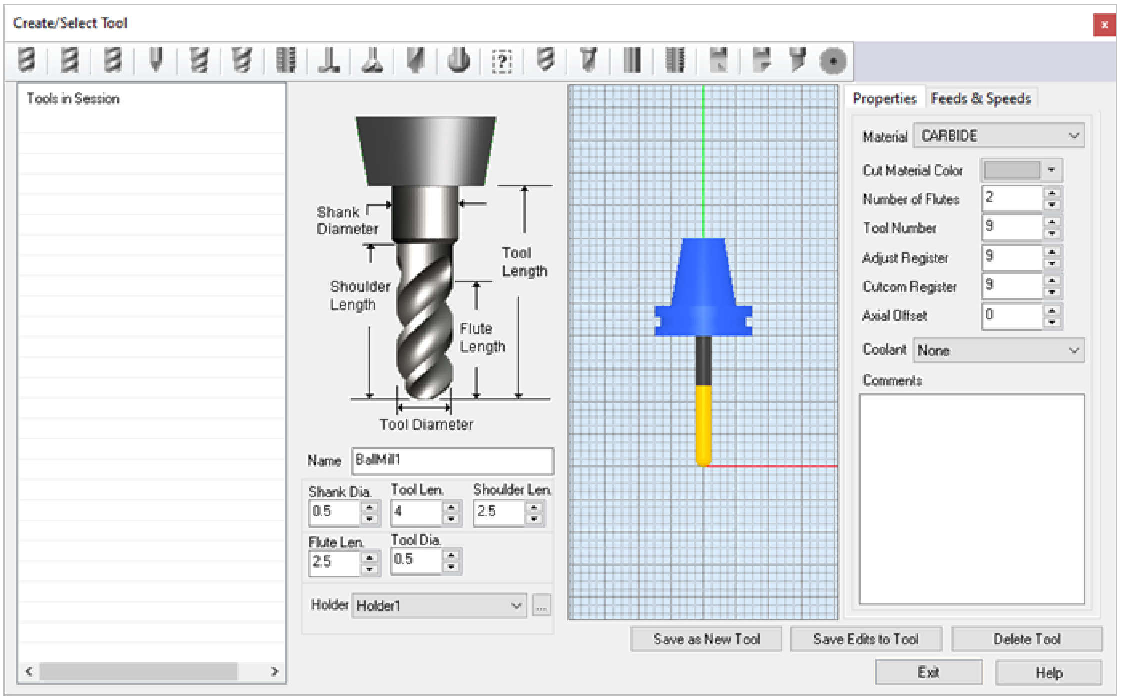
1. Select the Create/Edit Tools button from the Tools tab of the Machining Objects Browser or from the Tools tab of the toolpath operation you are in the middle of creating.



or

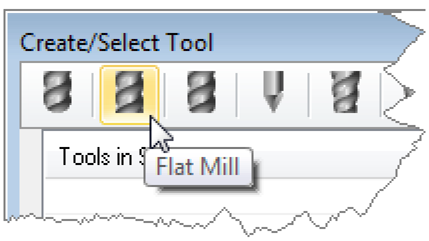


2. This will display the Create/Edit Tools dialog shown below. Notice that you can enter expressions in the parameter fields. See Tool Related Preferences to see how to enable/disable expressions in dialogs.



Dialog Box: Create/Select Tools

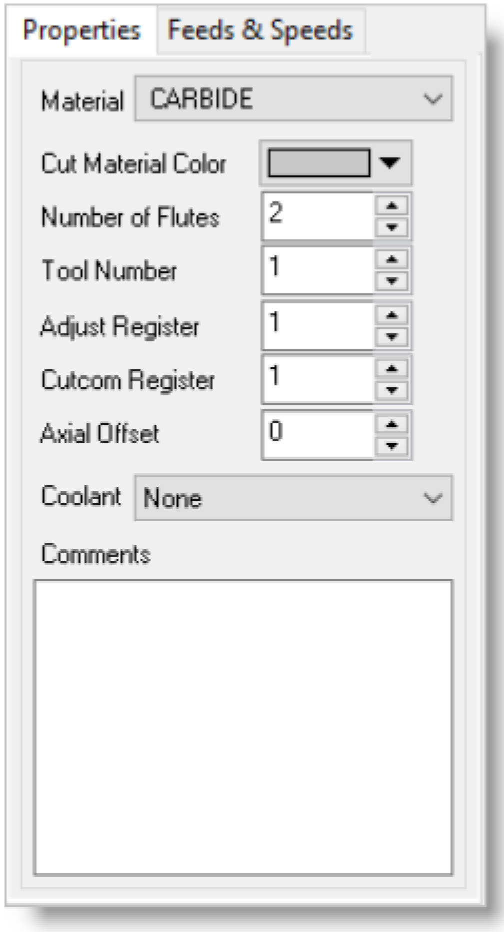
3. Select one of the Tool Type icons from the top of the dialog.



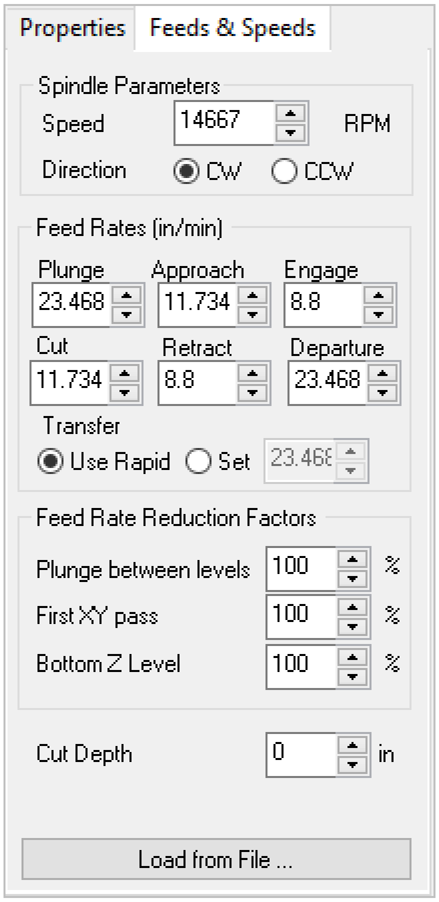
4. In the Name field enter a name for the Tool. A default name is provided but you should consider giving more descriptive names for your tools.
5. Enter the physical dimensions of the tool you would like to create. Default dimensions are provided. As you change the dimensions, the Preview of the Tool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

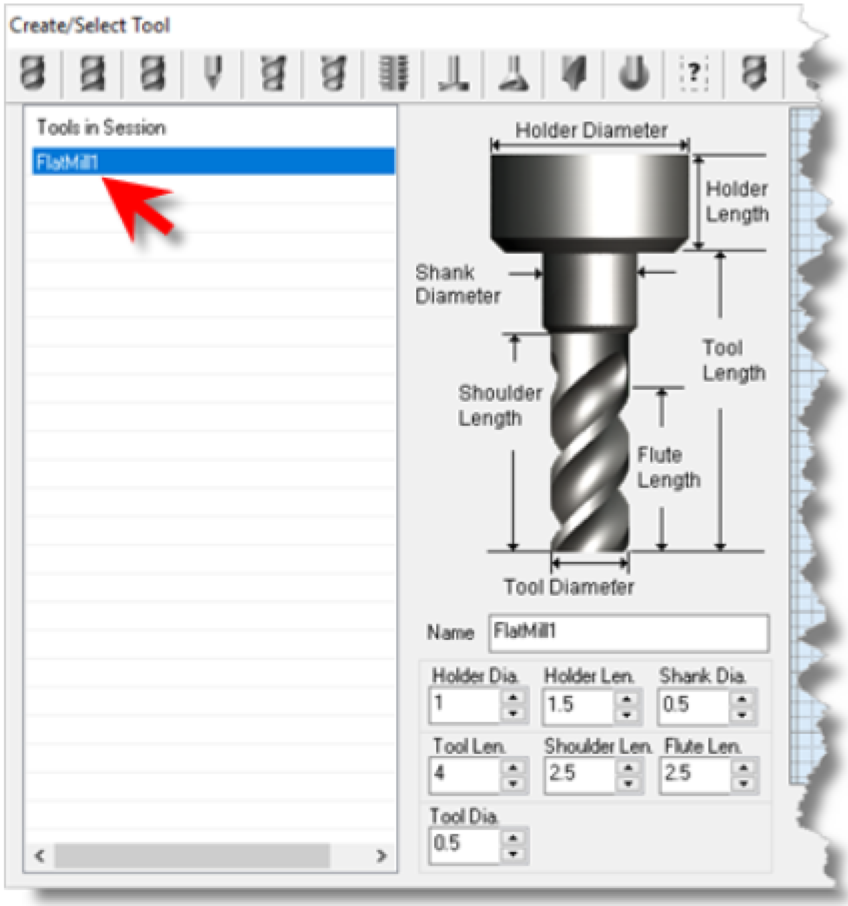
© 2026 MecSoft Corporation



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.



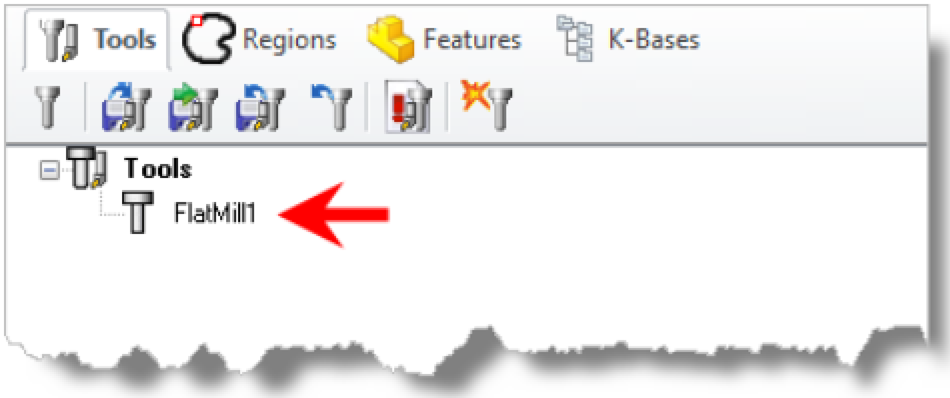
8. When you are done defining your tool's parameters, select the Save as New Tool button. The new tool is created and added to the Tools in Session list located on the left side of the dialog. This means that your tool is defined in your current session of RhinoCAM ONLY. When you save your part file, 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.



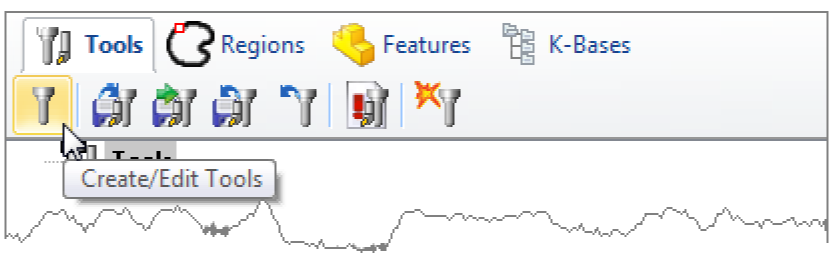
To save Tools to a library, Click Save Tool Library under the Tools tab in the Machining Objects Browser and specify a folder location and file name in the Save as dialog box. Saving a Tool library as a Knowledge base file (*.vkb) saves feeds and speeds with tool properties.

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

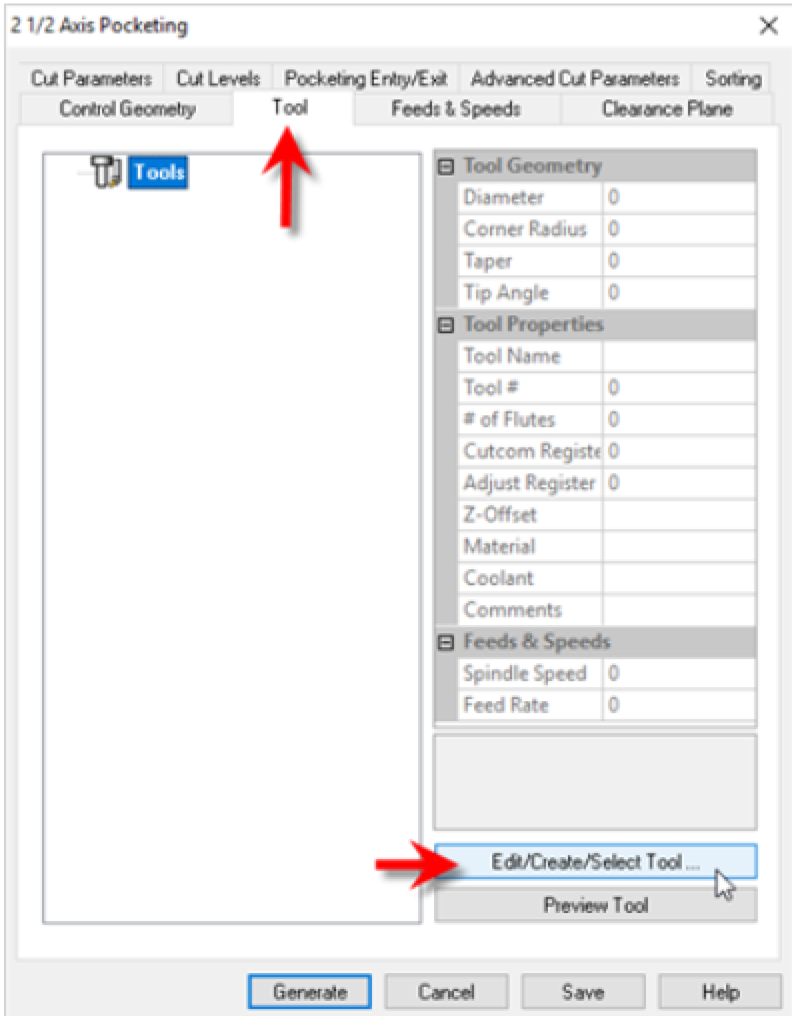
5.2 Create a Tool Holder

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.

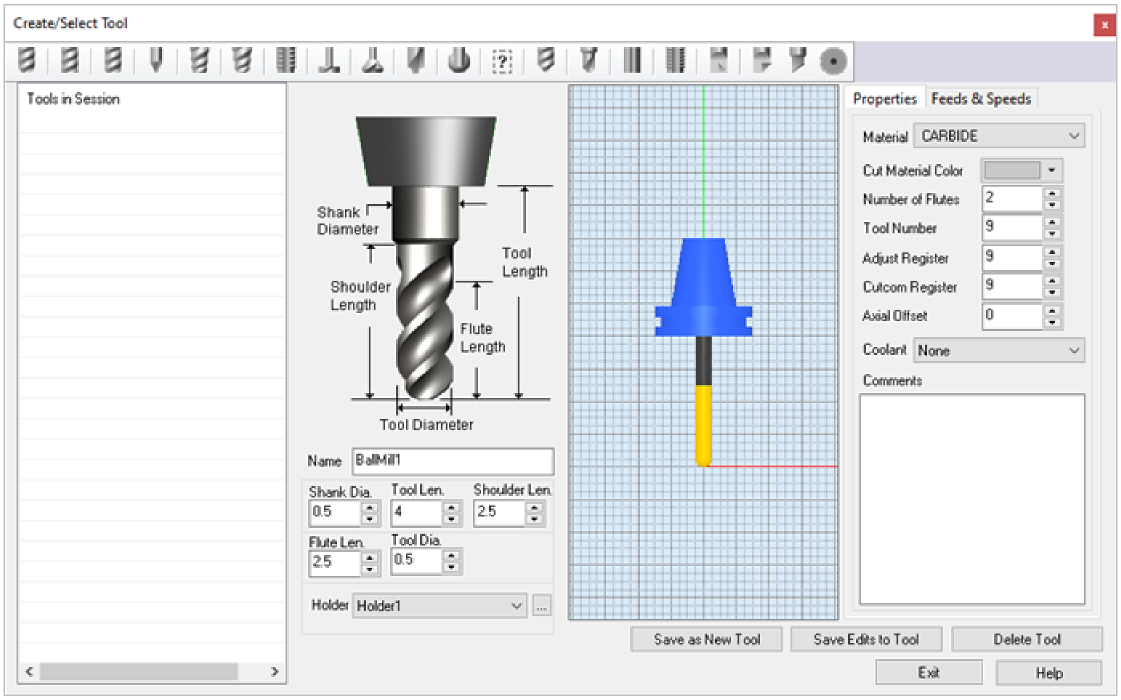


or



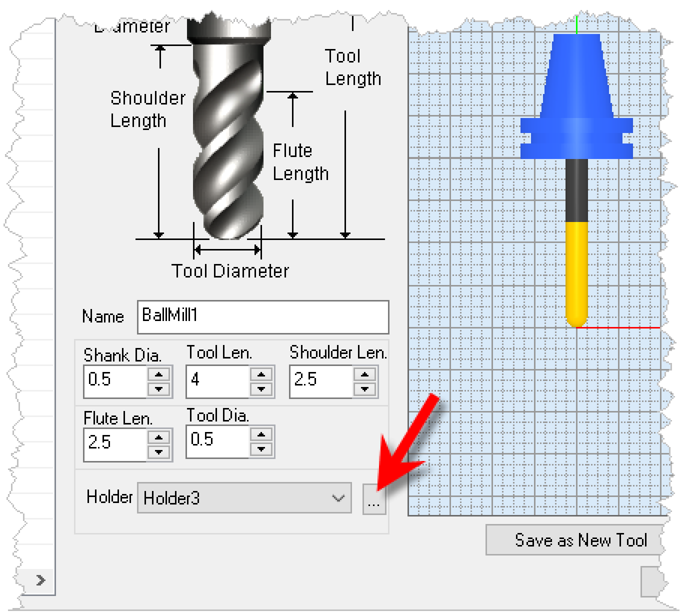
2. This will display the Create/Edit Tools dialog shown below. Notice that you can enter expressions in the parameter fields. See Tool Related Preferences to see how to enable/disable expressions in dialogs.

© 2026 MecSoft Corporation



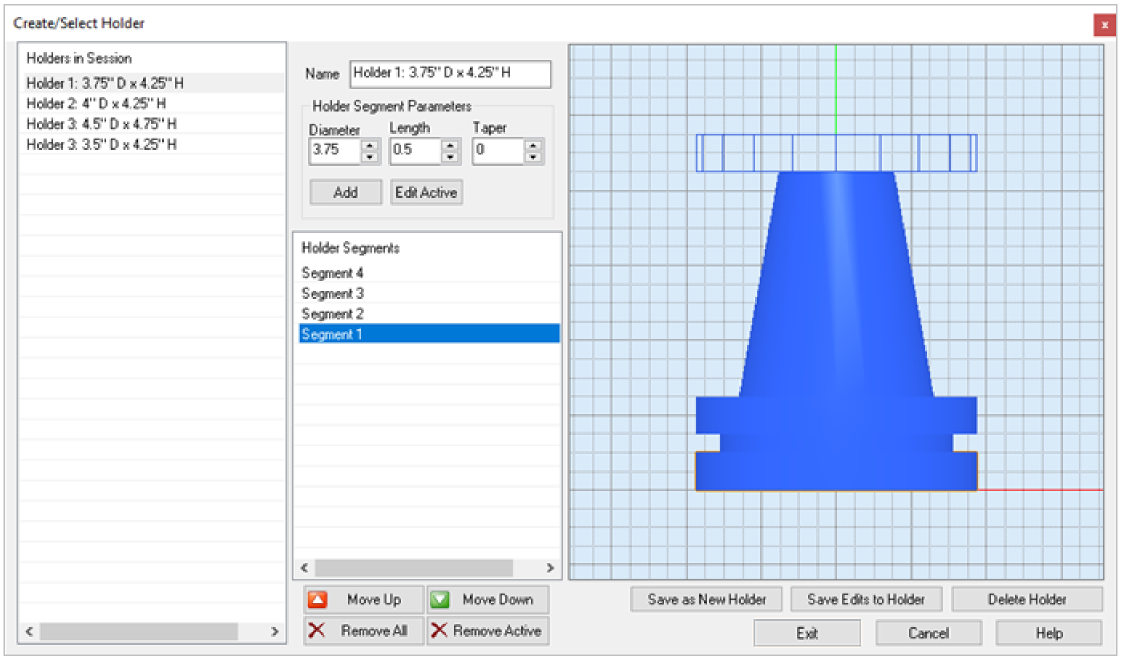
Dialog Box: Create/Select Tools

3. 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

4. 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.

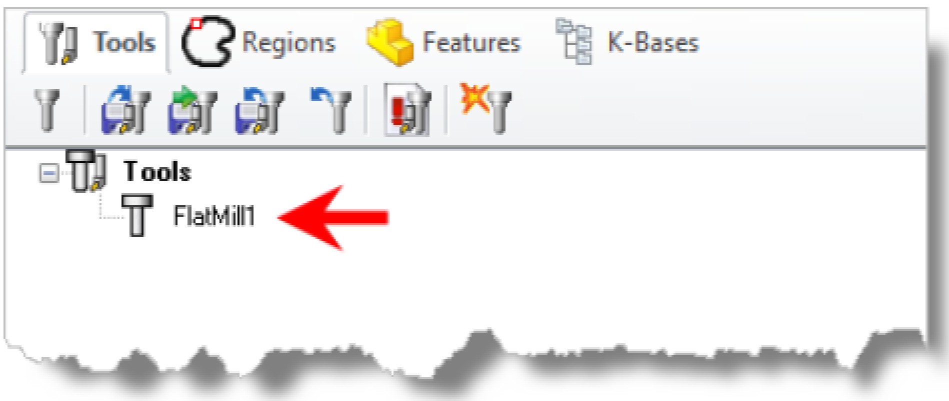
5.3 Create a Tool Library

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:

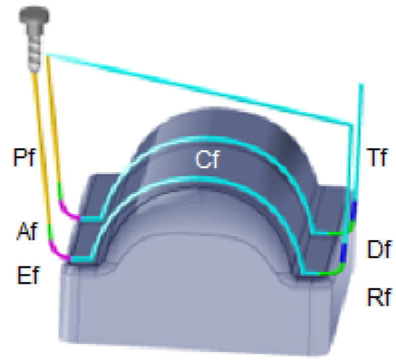
3. From the Save As dialog that displays, change the Save as Type selector to *.vkb. Both formats (*.vkb or *.csv) allow the Feeds & Speeds parameters to be saved along with each tool definition but the *.vkb format is recommended.

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

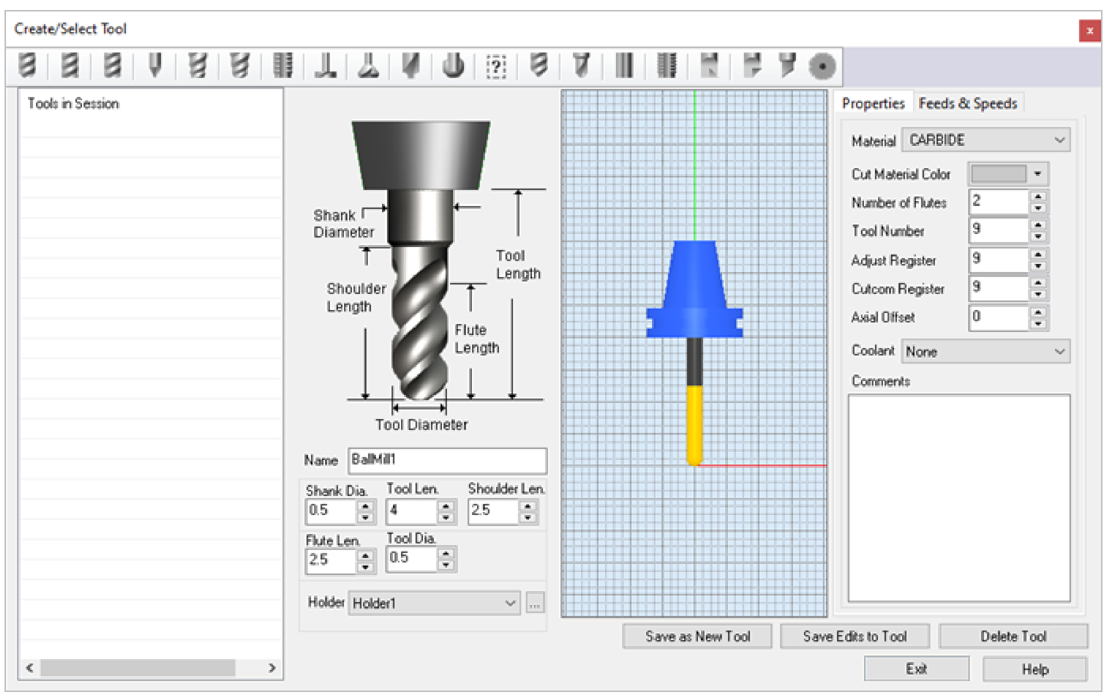
The MILL Module has a built-in Feeds & Speeds Calculator. That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various 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.

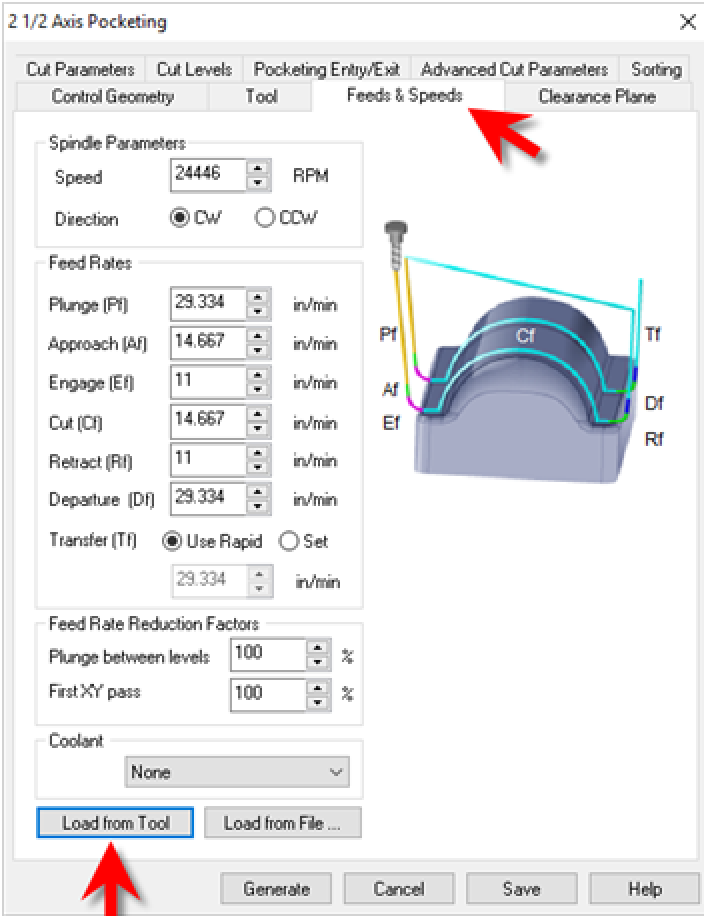


Dialog Box: Create/Select Tools

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

© 2026 MecSoft Corporation

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)	**Surface Speed (SFM) x 12 / (π x Tool Diameter(in))**
Cut Feed (IPM)	**Feed Per Tooth(IPT) x # of Flutes x 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)	**Surface Speed (SMM) * 1000 / (π x Tool Diameter (mm))**
Cut Feed (IPM)	**Feed Per Tooth(MMPT) x # of Flutes x Spindle Speed(RPM)**

© 2026 MecSoft Corporation

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!

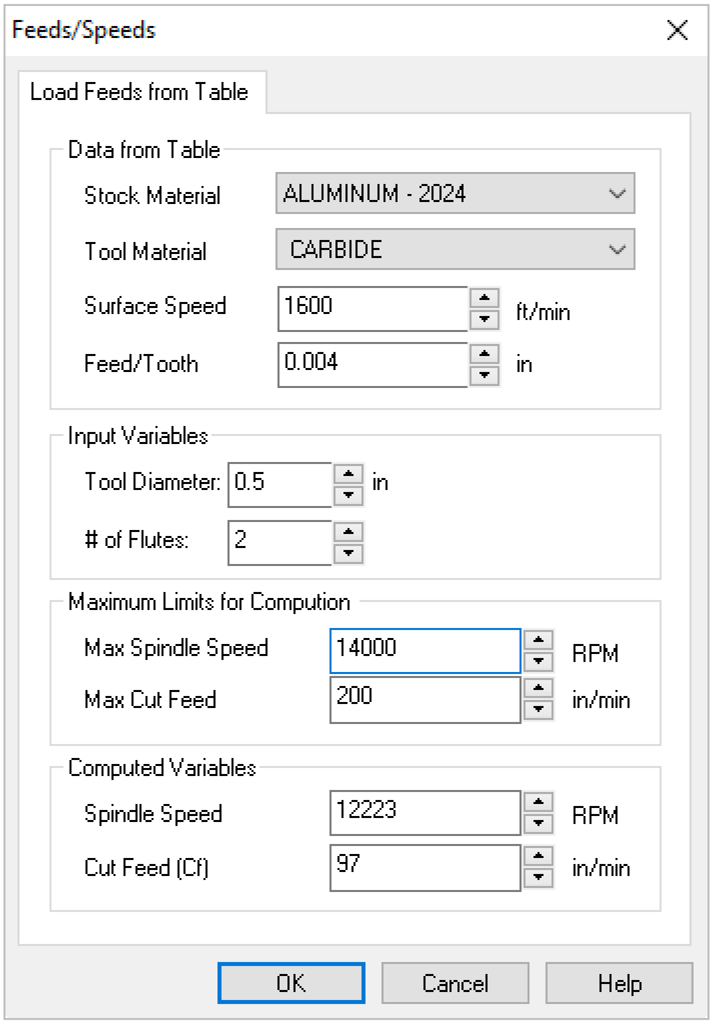
More information about the Load from File button:

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.

© 2026 MecSoft Corporation



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

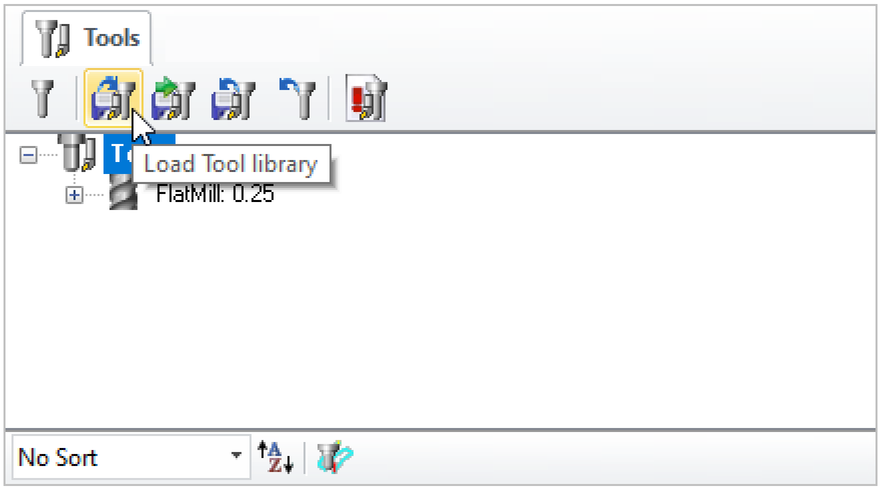
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override 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

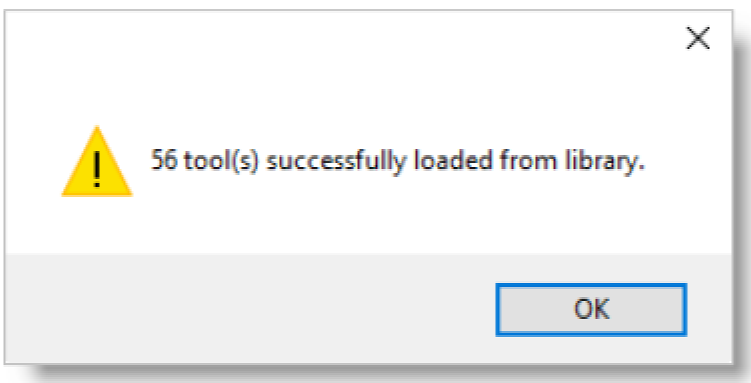
RhinoCAM MILL. comes with two Tool Libraries (INCH and METRIC) each containing over 50 Tools that you can use and modify to create your own customized Tool Library. To see a complete list of the tools in these default libraries, visit the Default Tool Libraries section of this guide.

How to load the default Tool Library:

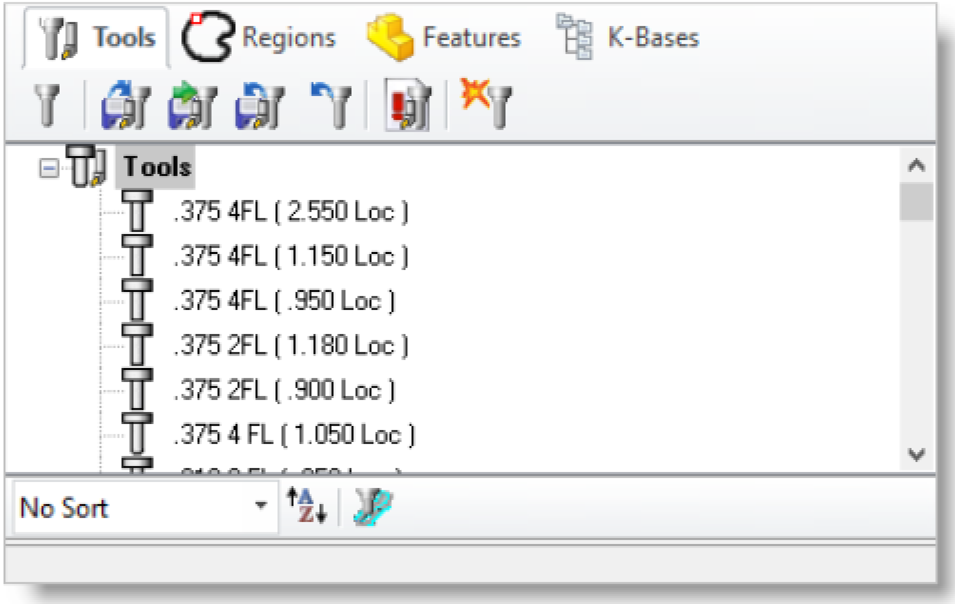
1. Go to the Tools tab and select the Load Tool Library icon.



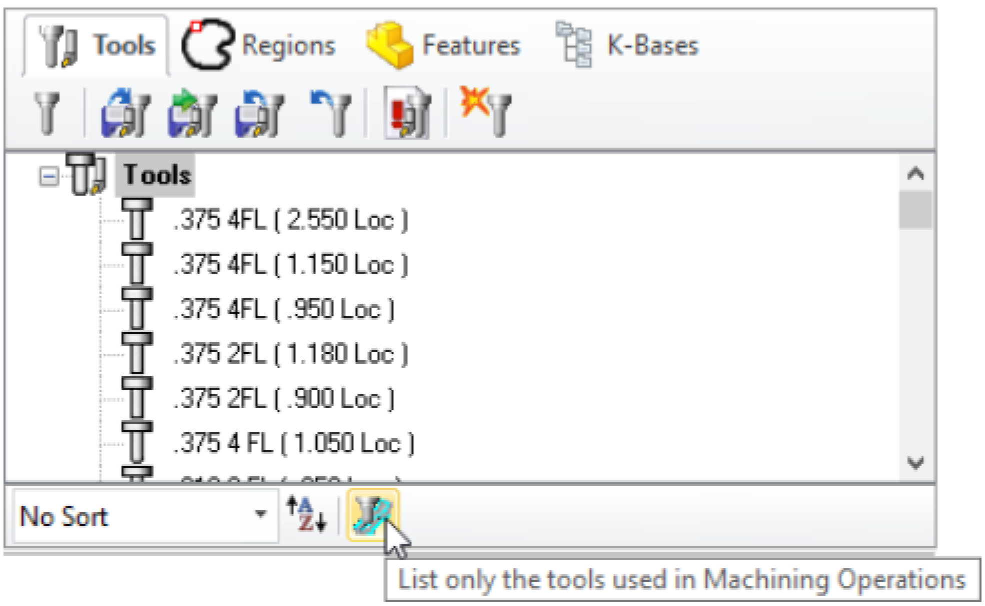
2. Navigate to the /Data folder of the install path in RhinoCAM.
   By default it is located at: *C:\ProgramData\MecSoft Corporation\RhinoCAM 20xx for Rhino x\Data*
3. In the File Open dialog, change the file type to Tool Library Text Files (*.csv).
4. Select the *DefaultEnglishTools.csv* or the *DefaultMetricTools.csv*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



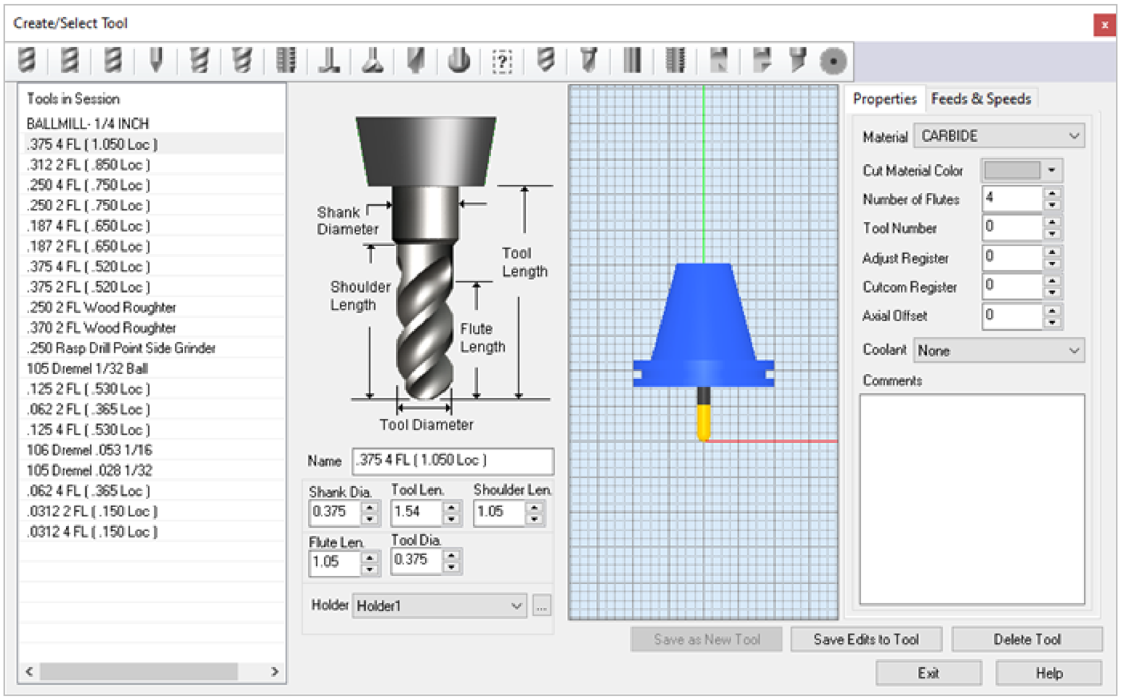
6. The tools in the library will also be listed in the Tools tab.



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!



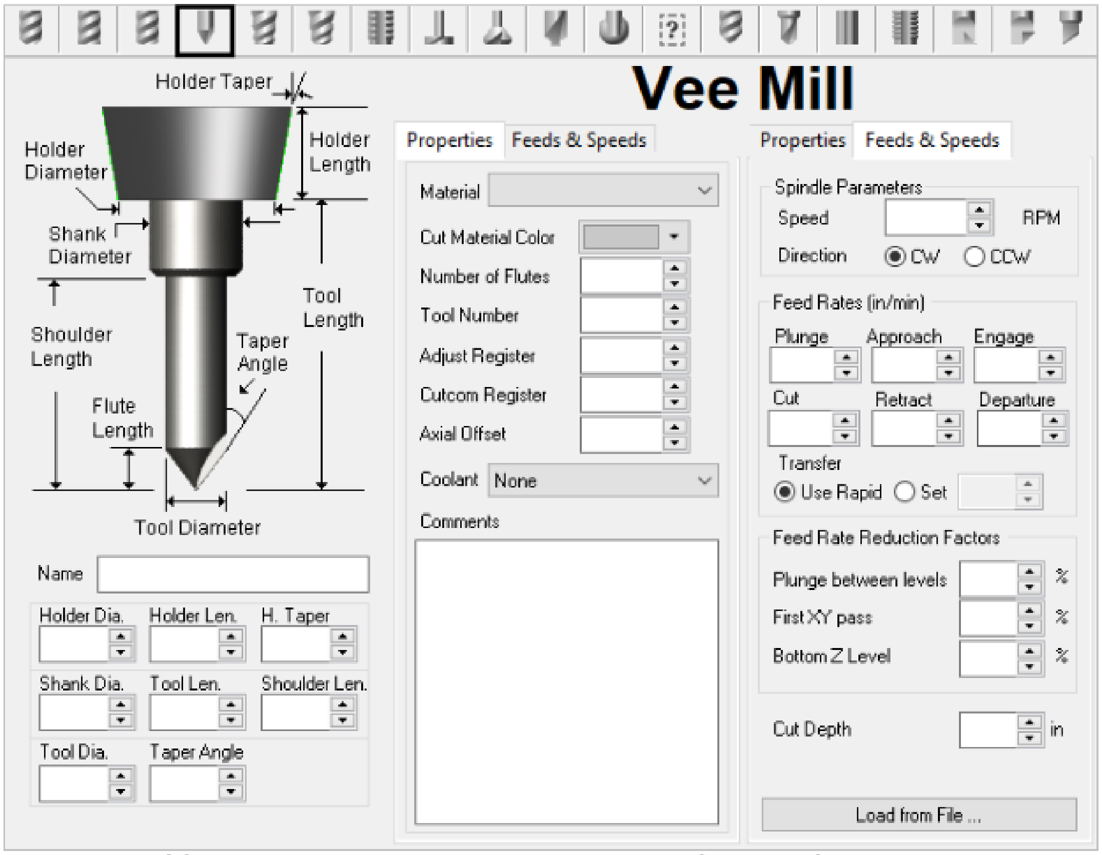
Dialog Box: Create/Select Tools

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:



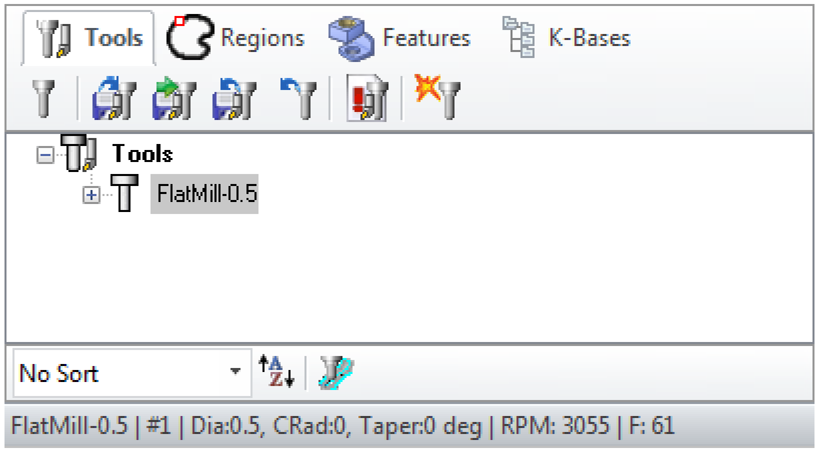
This is a worksheet to help you document tool paramaters. This IS NOT a RhinoCAM menu!

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



The Machining Objects (MObs) Browser, Tools Tab

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

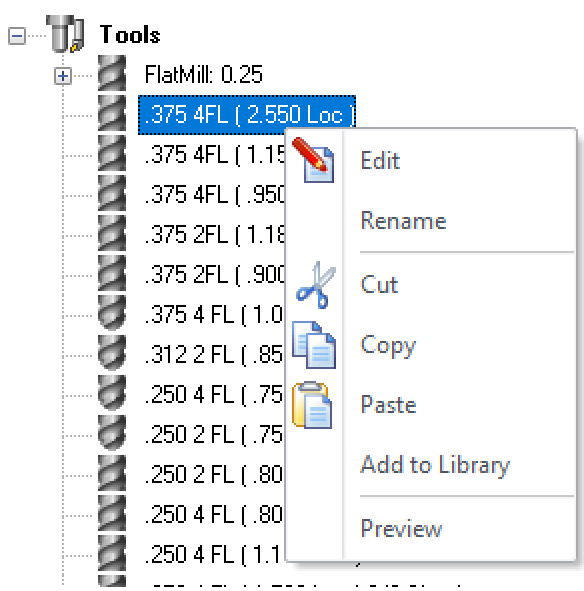
Tools Tab Functions

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:



Right-click Options on a Tool

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

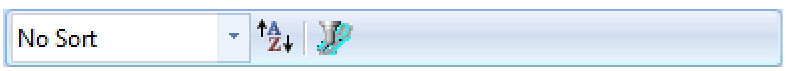
This allows you to Add the selected Tool to an exiting Tool Library *.csv data file.

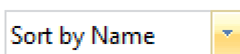
Preview

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.



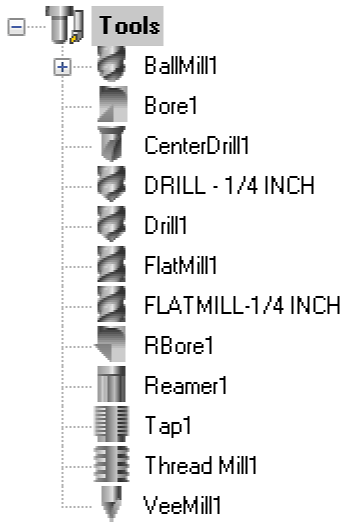


Sorting Selector: This allows you to sort the tool list. You can select No Sort or sort by Name, Number, Type and Diameter.

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!

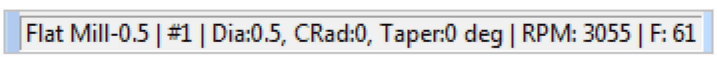


Tools Sorted by Name

Tools Sorted by Used

Tools Status Bar

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

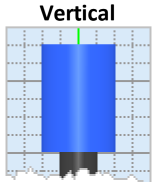


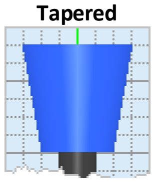
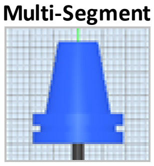
Status Bar, Tools Tab, Machining Objects Browser

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		✓	✓	✓	✓
Lollypop Cutters		✓	✓	✓	✓
User Defined Cutters		✓	✓	✓	✓

© 2026 MecSoft Corporation

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.

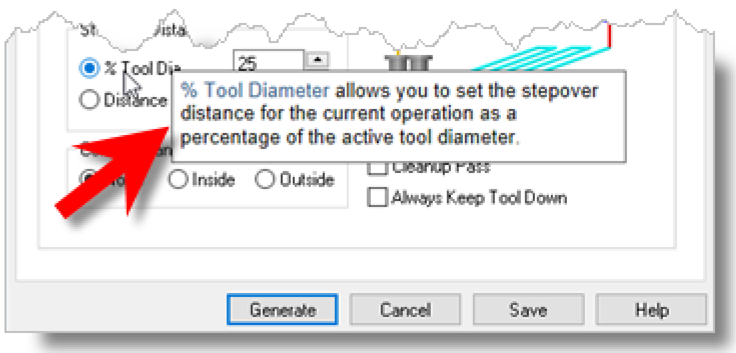
6.1 Where can I find Tool related Preferences?

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:

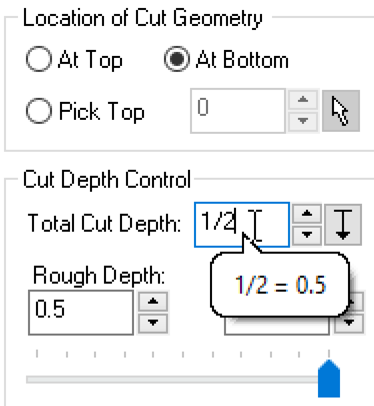
Show context ToolTips

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.



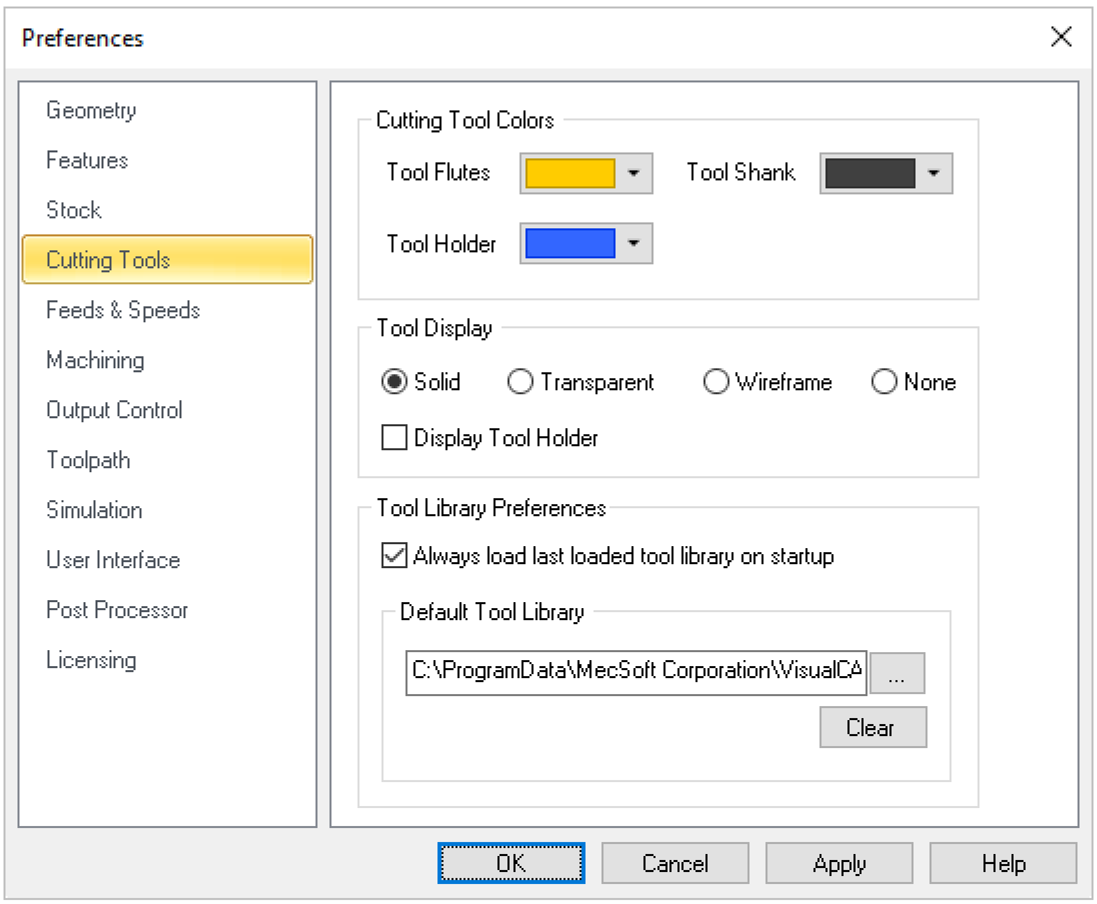
Show Expressions in Tooltip

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.



Show Expressions in ToolTip

3. Select the Cutting Tools item from the left.

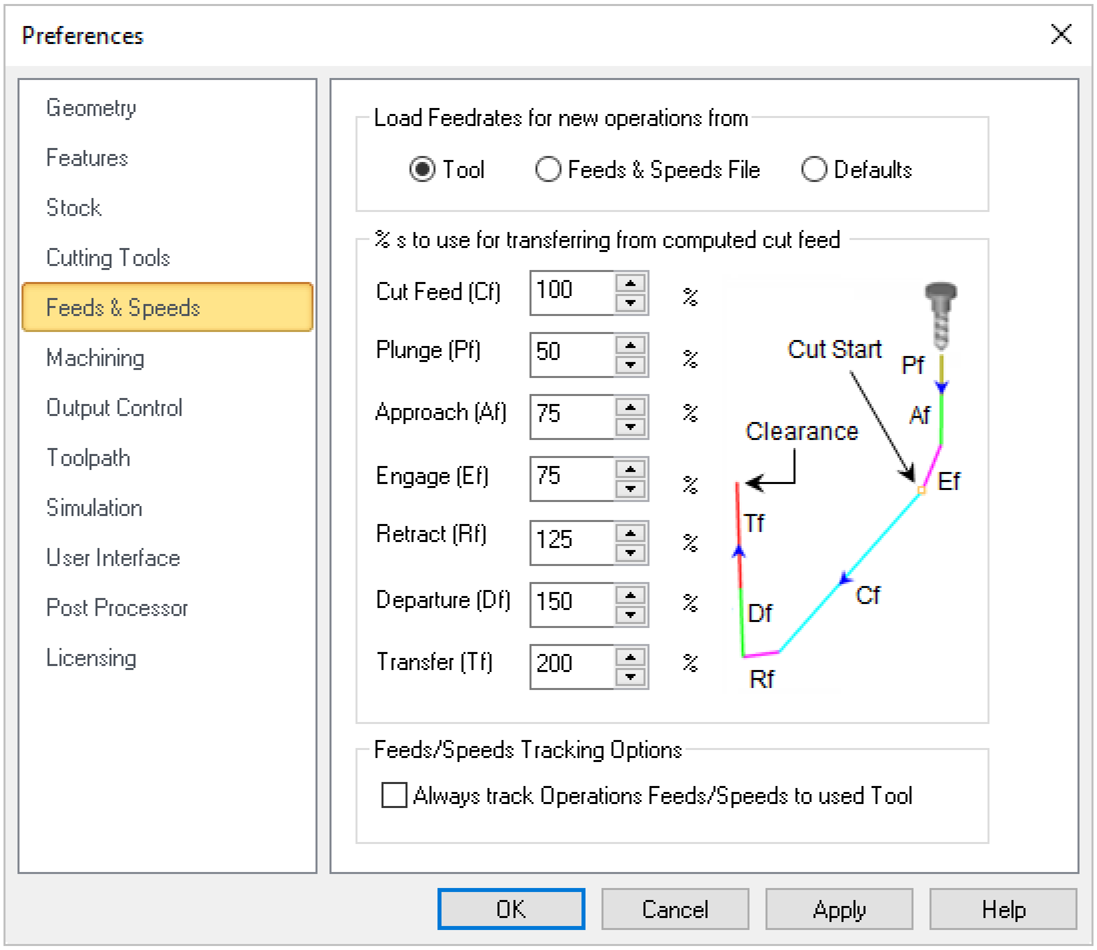


CAM Preferences > Cutting Tools
Note: Menu selections on the left may change depending on module and configuration

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.

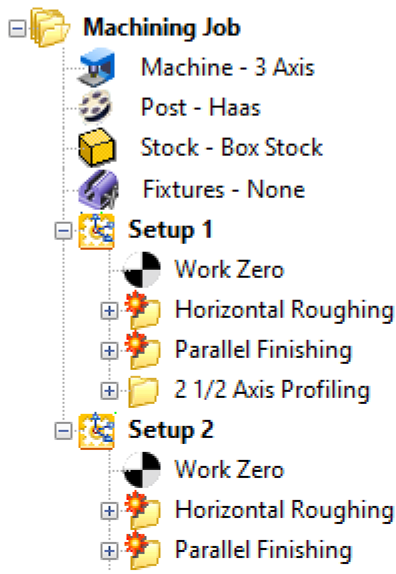
5. You can also set the Colors used when the Tool is displayed on the screen.

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.



CAM Preferences > Feeds & Speeds

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.

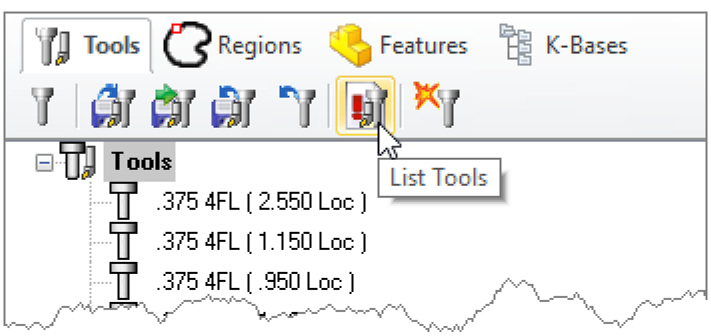


6.2 How can I Print a Tool List?

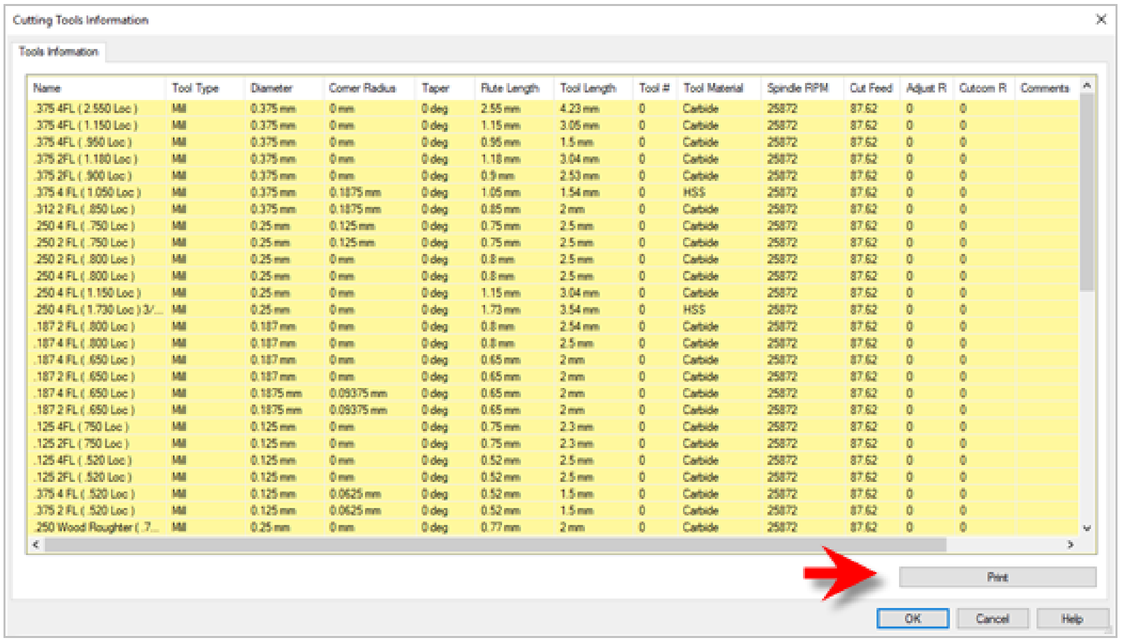
Follow the steps outlined below to print your Tool List:

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.



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)	Mill	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)	Mill	0.375 mm	0 mm	0 deg	1.15 mm	3.05 mm	0	Carbide	25872	87.62	0	0	
.375 4FL (.950 Loc)	Mill	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)	Mill	0.375 mm	0 mm	0 deg	1.18 mm	3.04 mm	0	Carbide	25872	87.62	0	0	
.375 2FL (.900 Loc)	Mill	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)	Mill	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)	Mill	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)	Mill	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)	Mill	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)	Mill	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)	Mill	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)	Mill	0.25 mm	0 mm	0 deg	1.15 mm	3.04 mm	0	Carbide	25872	87.62	0	0	
.250 4 FL (1.730 Loc) 3/...	Mill	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)	Mill	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)	Mill	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)	Mill	0.187 mm	0 mm	0 deg	0.65 mm	2 mm	0	Carbide	25872	87.62	0	0	
.187 2 FL (.650 Loc)	Mill	0.187 mm	0 mm	0 deg	0.65 mm	2 mm	0	Carbide	25872	87.62	0	0	
.187 4 FL (.650 Loc)	Mill	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)	Mill	0.1875 mm	0.09375 mm	0 deg	0.65 mm	2 mm	0	Carbide	25872	87.62	0	0	
.125 4FL (750 Loc)	Mill	0.125 mm	0 mm	0 deg	0.75 mm	2.3 mm	0	Carbide	25872	87.62	0	0	
.125 2FL (750 Loc)	Mill	0.125 mm	0 mm	0 deg	0.75 mm	2.3 mm	0	Carbide	25872	87.62	0	0	
.125 4FL (.520 Loc)	Mill	0.125 mm	0 mm	0 deg	0.52 mm	2.5 mm	0	Carbide	25872	87.62	0	0	
.125 2FL (.520 Loc)	Mill	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)	Mill	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)	Mill	0.125 mm	0.0625 mm	0 deg	0.52 mm	1.5 mm	0	Carbide	25872	87.62	0	0	
.250 Wood Roughter (.7...	Mill	0.25 mm	0 mm	0 deg	0.77 mm	2 mm	0	Carbide	25872	87.62	0	0	

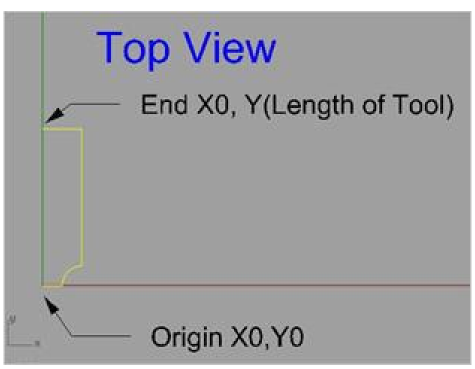
6.3 How can I add a Custom Tool?

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

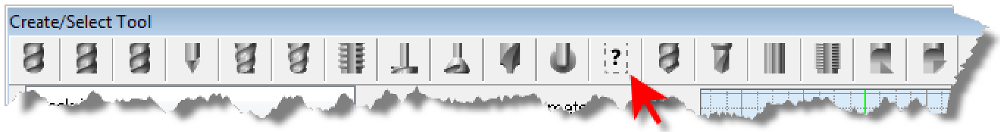


Steps to create a user defined tool

Steps to create a user defined tool:

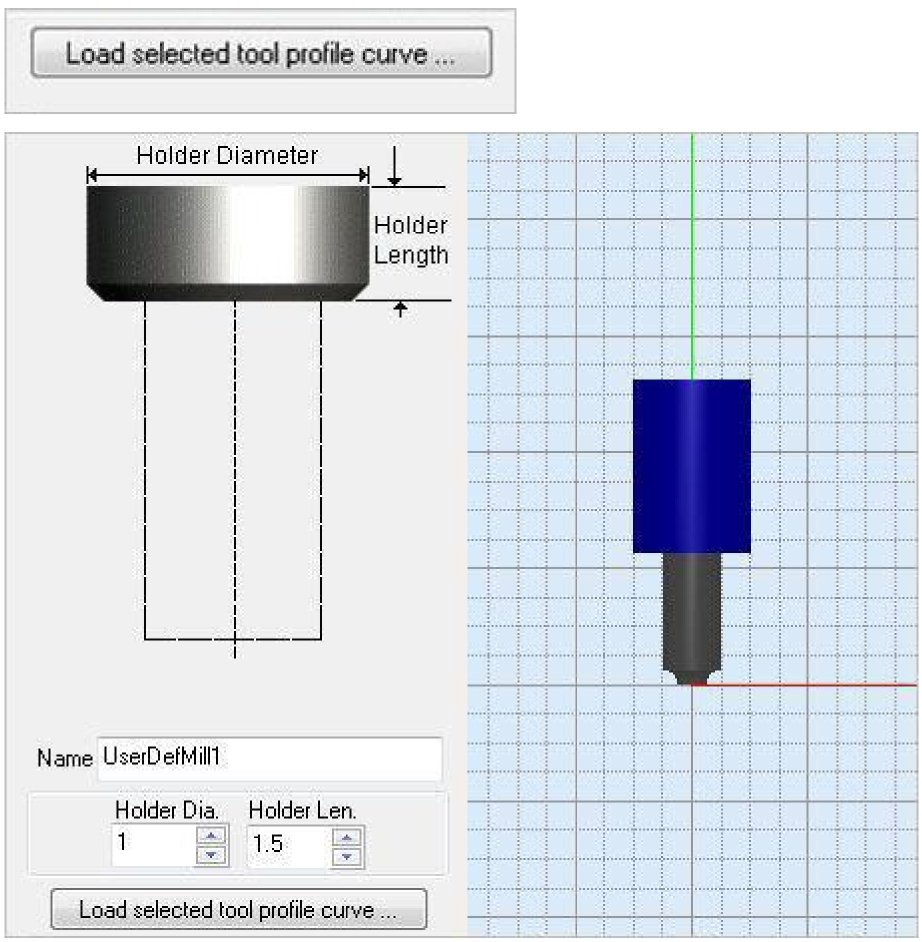
1. Draw half the tool profile from the top view (XY plane of the world coordinate system) as shown in the picture above and then join the curves to form a single curve.
2. Make sure one end of the curve (tool tip) is at origin (0,0) and the other end at X0, Y<value>.

3. From the Tools tab under the Machining Objects Browser, click Create/Select Tool and select User Defined Tool.



click Create/Select Tool and select User Defined Tool

4. Click Load selected tool profile curve.

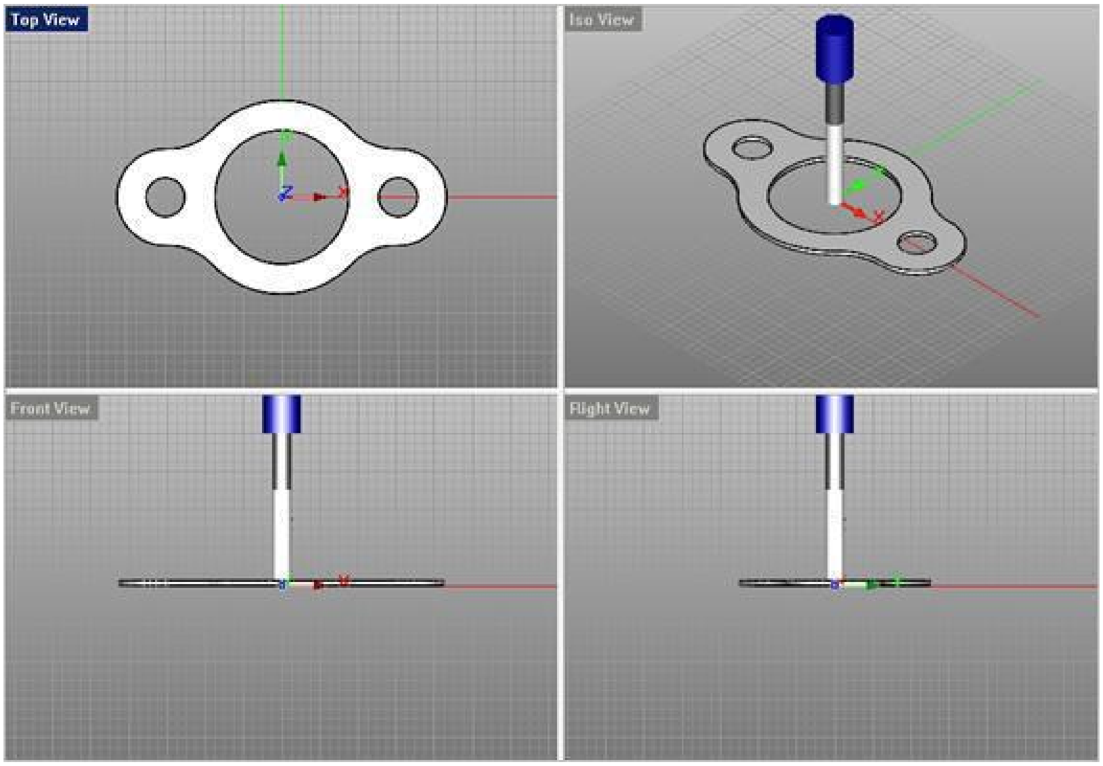


Click Load selected tool profile curve

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.

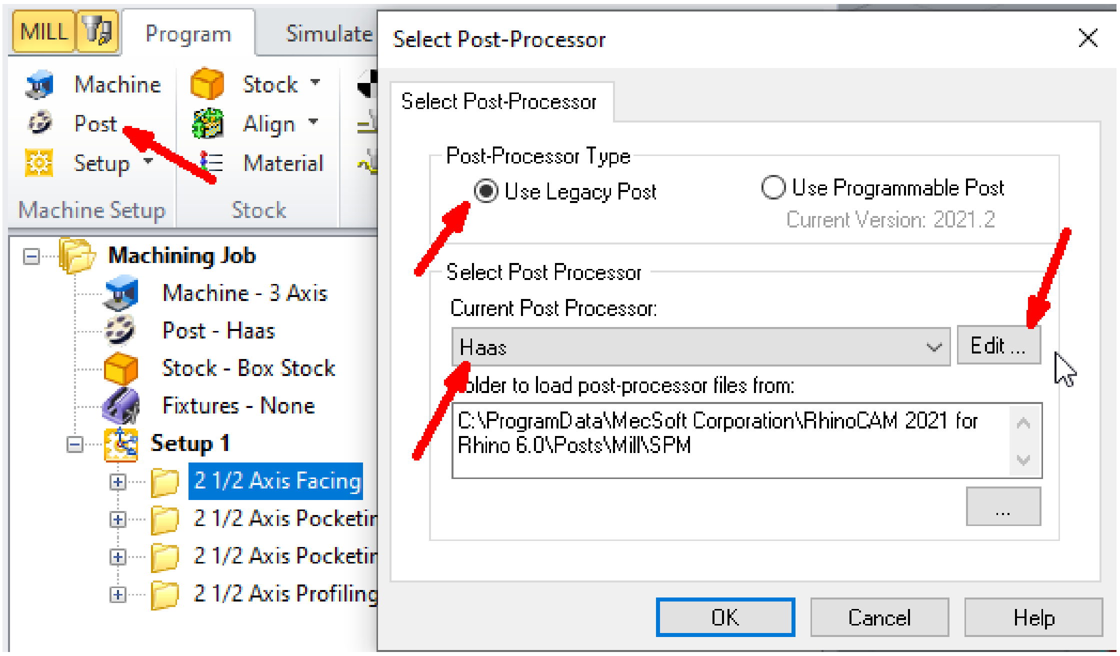


Preview your Tool

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

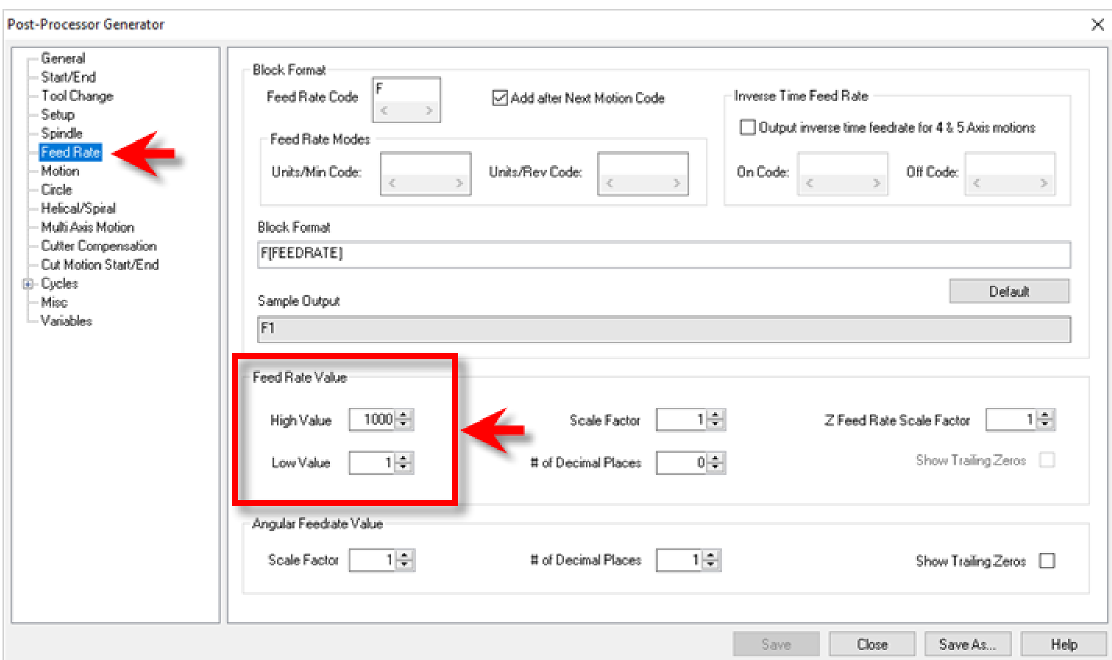
If your Feed Rate values in your posted g-code are too High or too Low, you can adjust the High/Low parameters in the Post Processor following the steps outlined below:

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

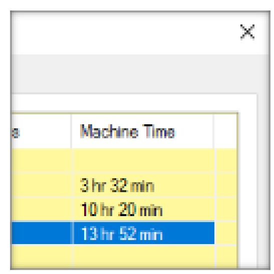
[CYCL_TPI] – Thread Pitch

[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...

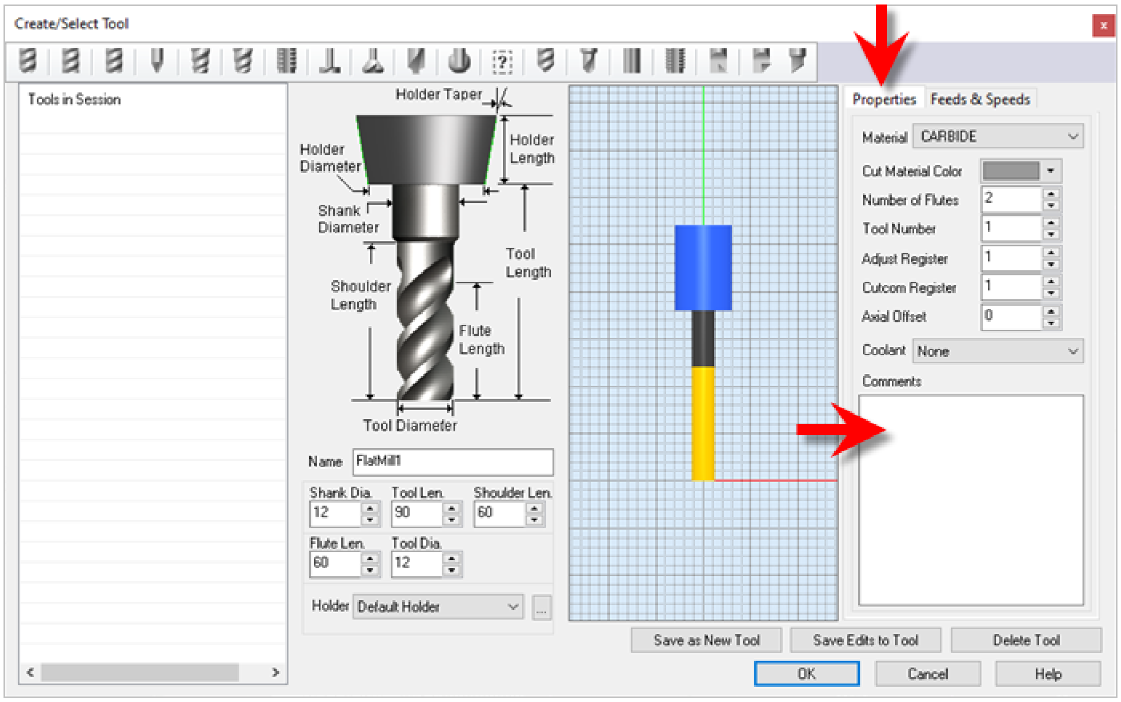
More Advanced Topics

7.1 How can I add Tool Comments?

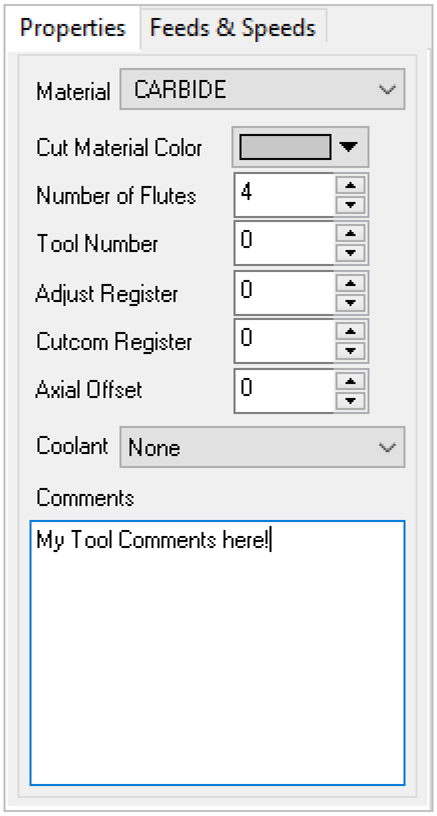
You can add comments associated with a Tool. These Comments are saved with the Tool in your Tool Library. They are also posted to your g-code when the tool is used.

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.

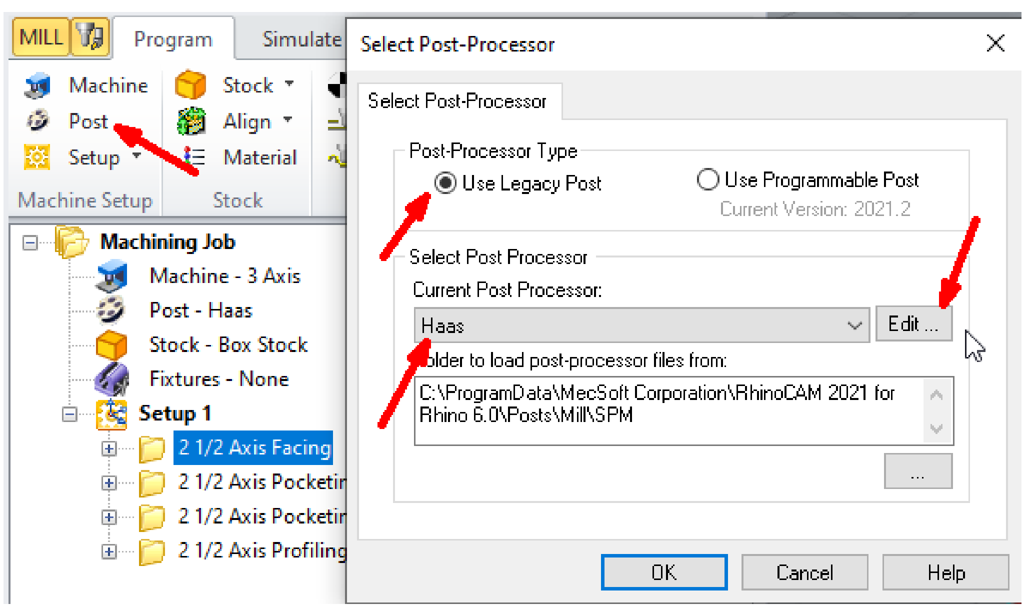


3. Add text to the Comments window.



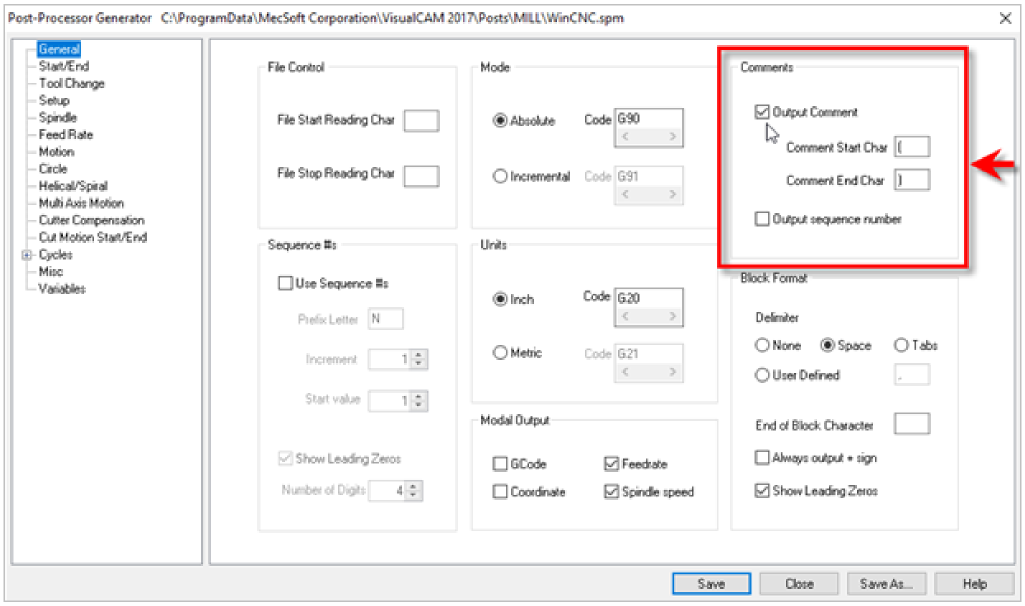
4. Make sure Comments are enabled in your post.

   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.

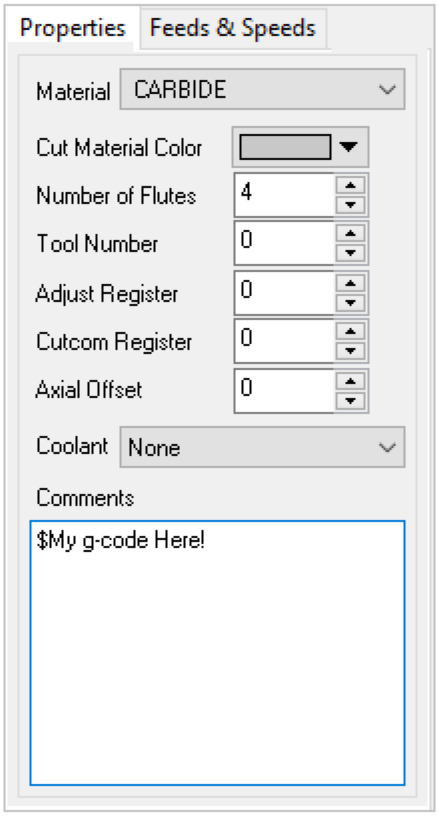


D. Then pick Save or Save As.

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.




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

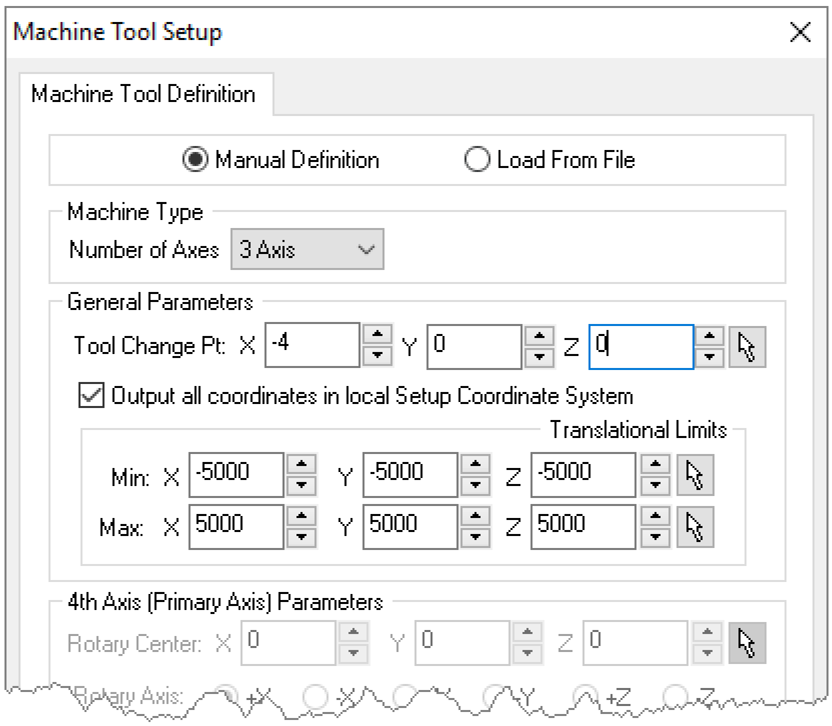
7.2 How can add a Tool Change Point?

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:

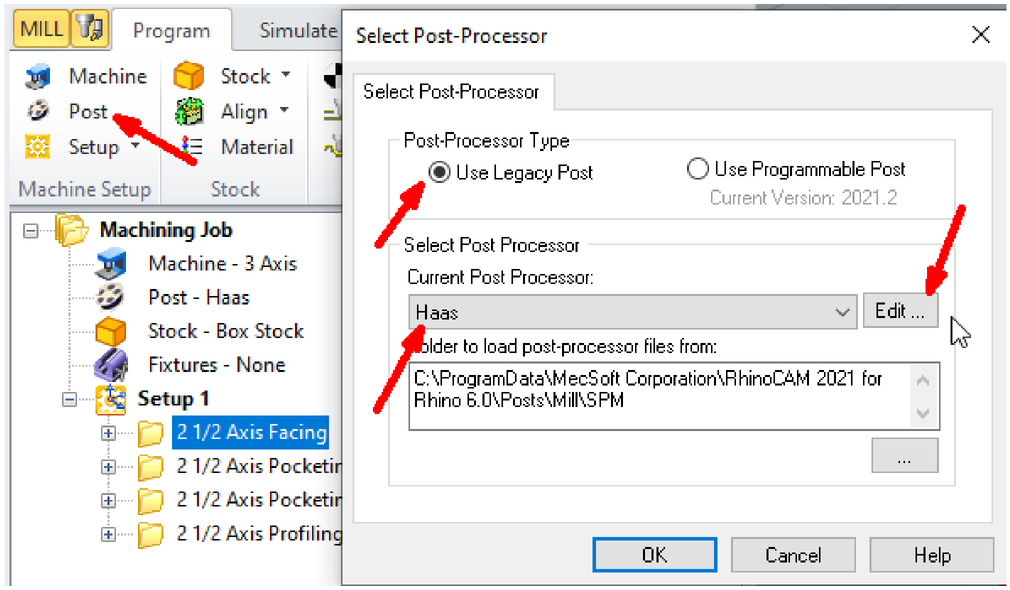
For 2 and 3 Axis Output

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



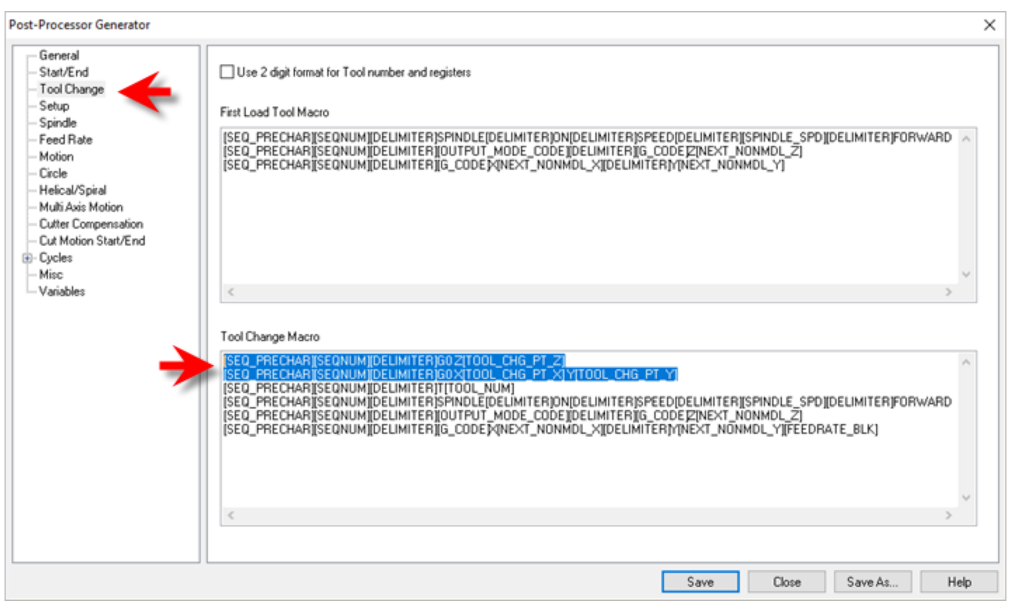
3. Edit your post processor by selecting Program tab > Post > Edit.



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.

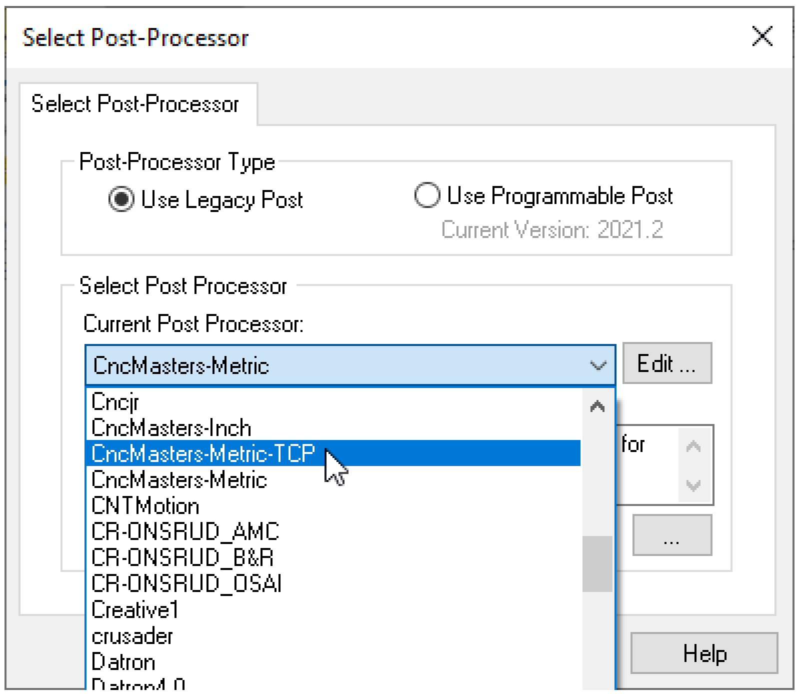
```
[SEQ_PRECHAR][SEQNUM][DELIMITER]G0 Z[TOOL_CHG_PT_Z]
[SEQ_PRECHAR][SEQNUM][DELIMITER]G0 X[TOOL_CHG_PT_X] Y[TOOL_CHG_PT_Y]
[SEQ_PRECHAR][SEQNUM][DELIMITER]T[TOOL_NUM]
...
...
```



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

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



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

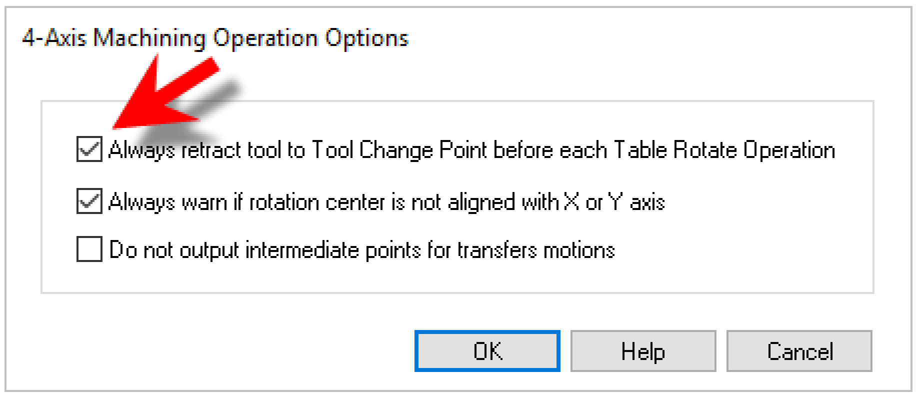
...

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

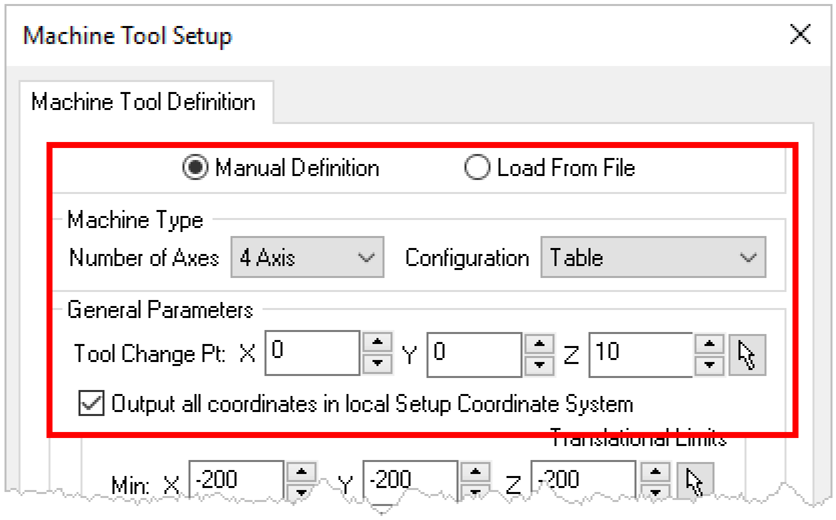
15. That's it!

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!

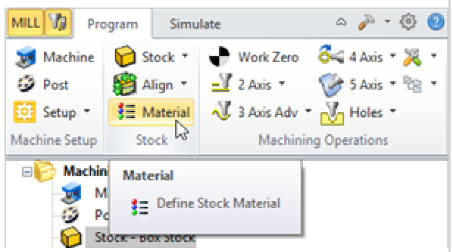
However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library?

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.

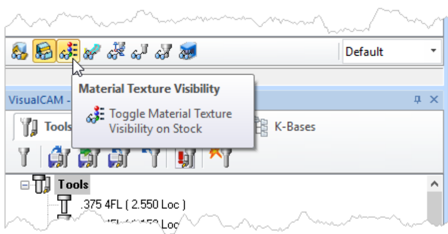
How Materials are used

© 2026 MecSoft Corporation

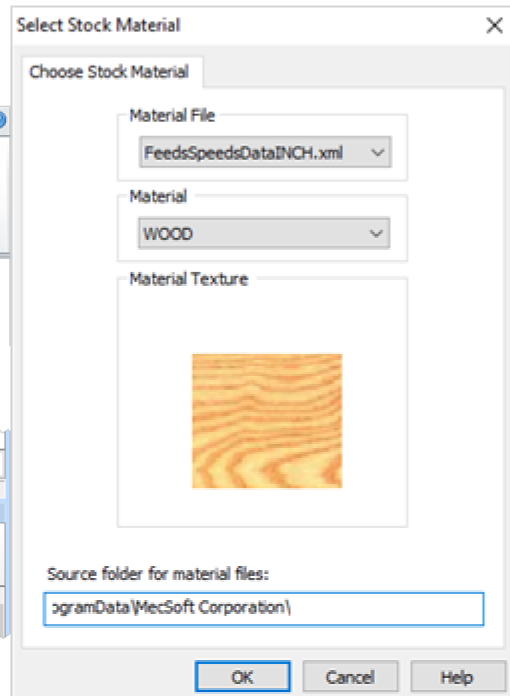
First let's review how material definitions are used in RhinoCAM. From the Stock pane of the Program tab in the Machining Browser you will find the Material icon. Selecting it will display the Select Stock Material dialog. Here you can select a stock material from the Material File. If there is an associated 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.



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



The Material icons in the Machining Browser / MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST



The Select Stock Material dialog

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 - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00, 0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00, 0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
...
...
```

Here is what the file looks like in tabulated form:

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

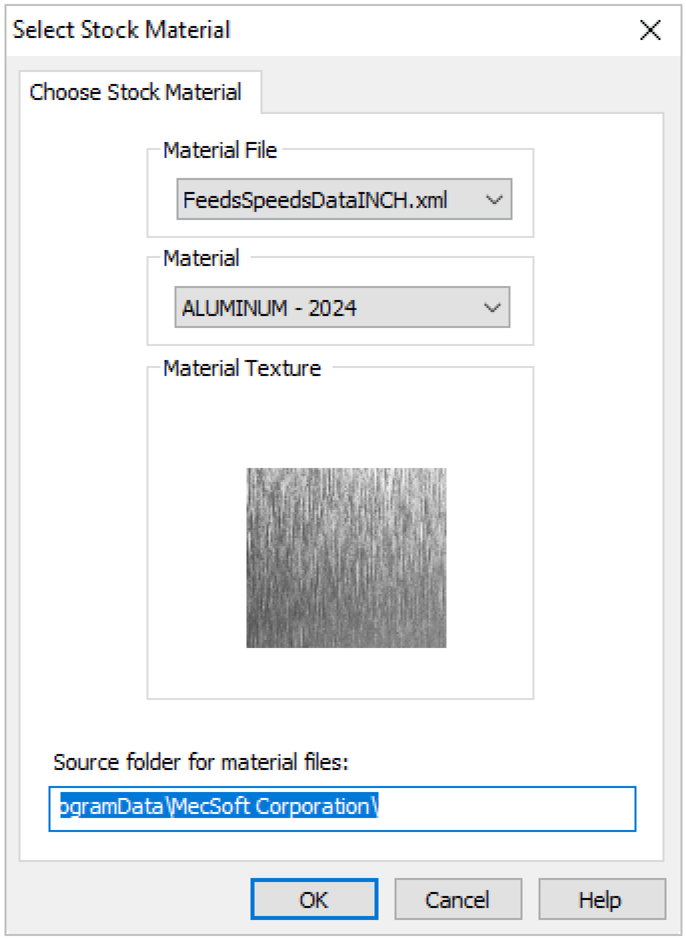
4. You will need to know the Surface Speed and Feed/Tooth specifications for each instance of the Material you are adding to the XML file. Here is the information format:

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

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

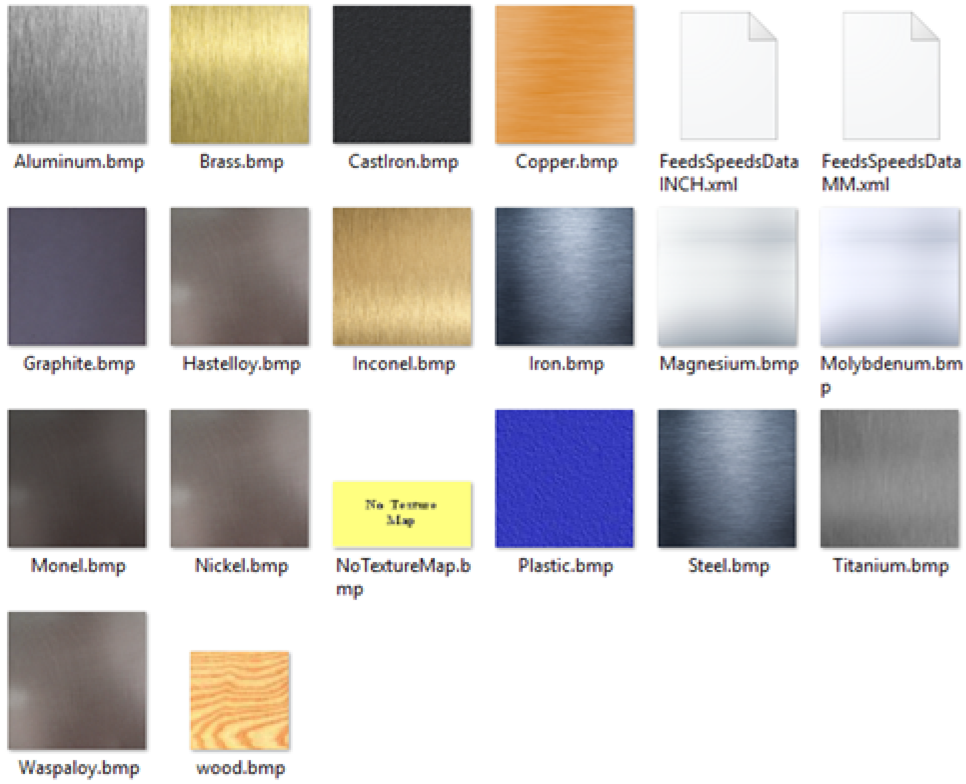
   <Version>1.0</Version>
   <Units>Imperial</Units>
   <FeedsSpeeds>
   <Material>
   ...
   ...

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



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

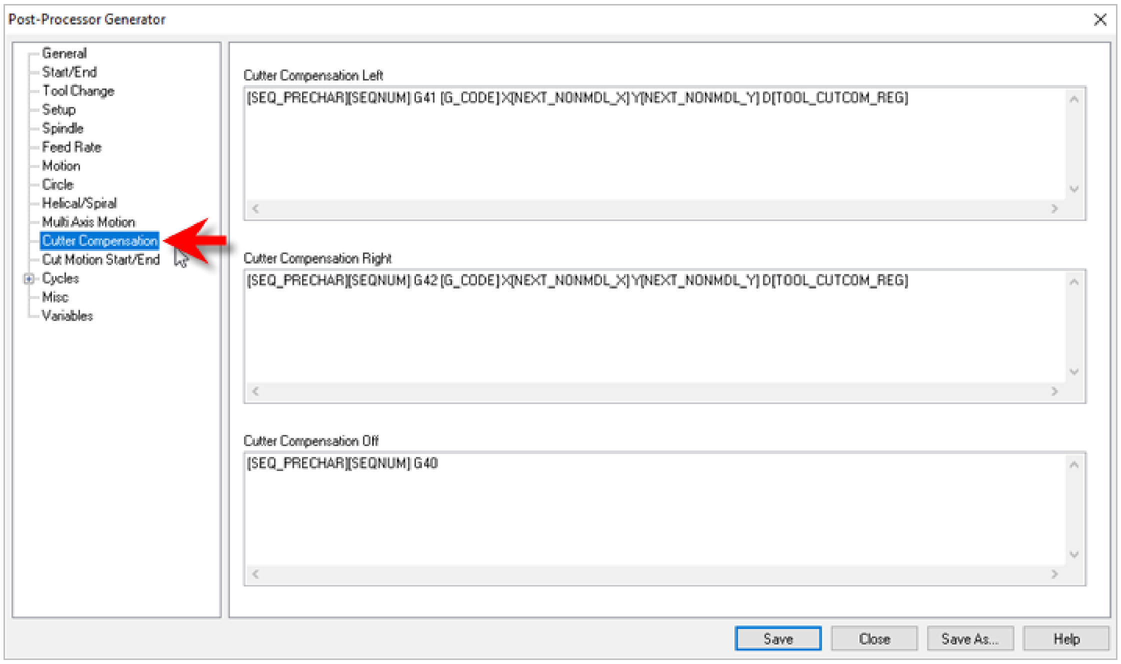
   [SEQ_PRECHAR][SEQNUM] G41 [G_CODE] X[NEXT_NONMDL_X] Y[NEXT_NONMDL_Y] D[TOOL_CUTCOM_REG]

   Cutter Compensation Right

   [SEQ_PRECHAR][SEQNUM] G42 [G_CODE] X[NEXT_NONMDL_X] Y[NEXT_NONMDL_Y] D[TOOL_CUTCOM_REG]

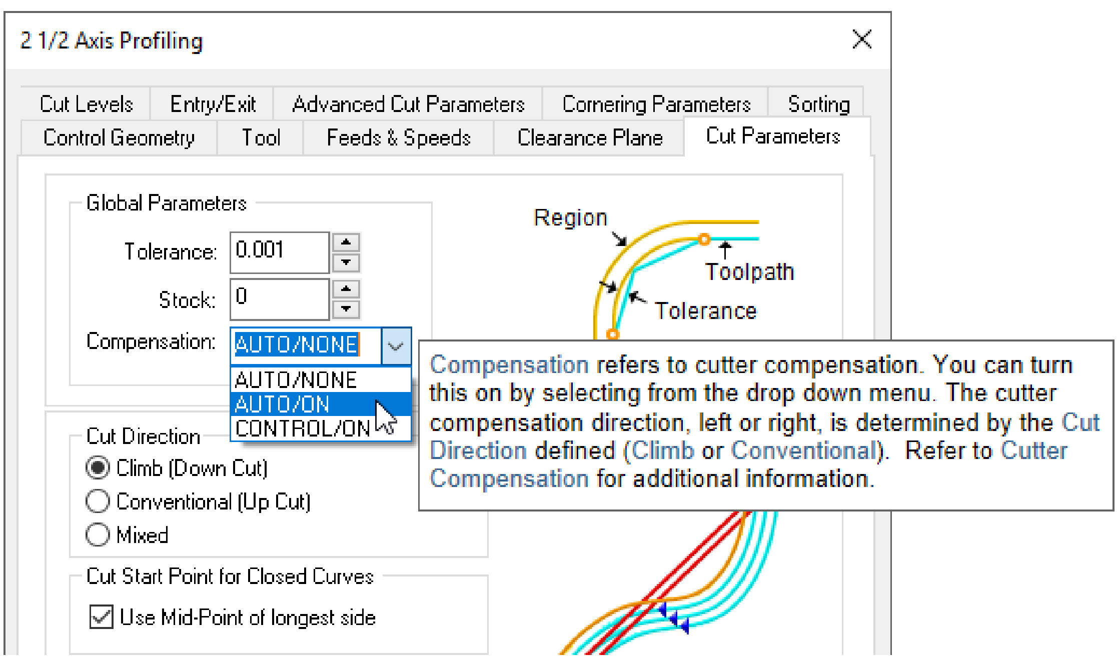
   Cutter Compensation Off

   [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.



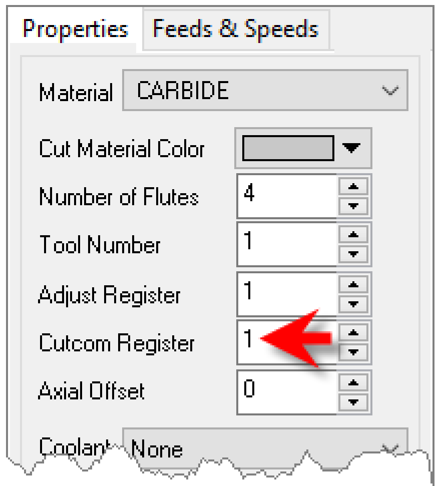
3. Make sure you have a linear motion for the controller to turn on the compensation for. If your first motion is an arc the controller will not be able to turn on the compensation.

© 2026 MecSoft Corporation

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.

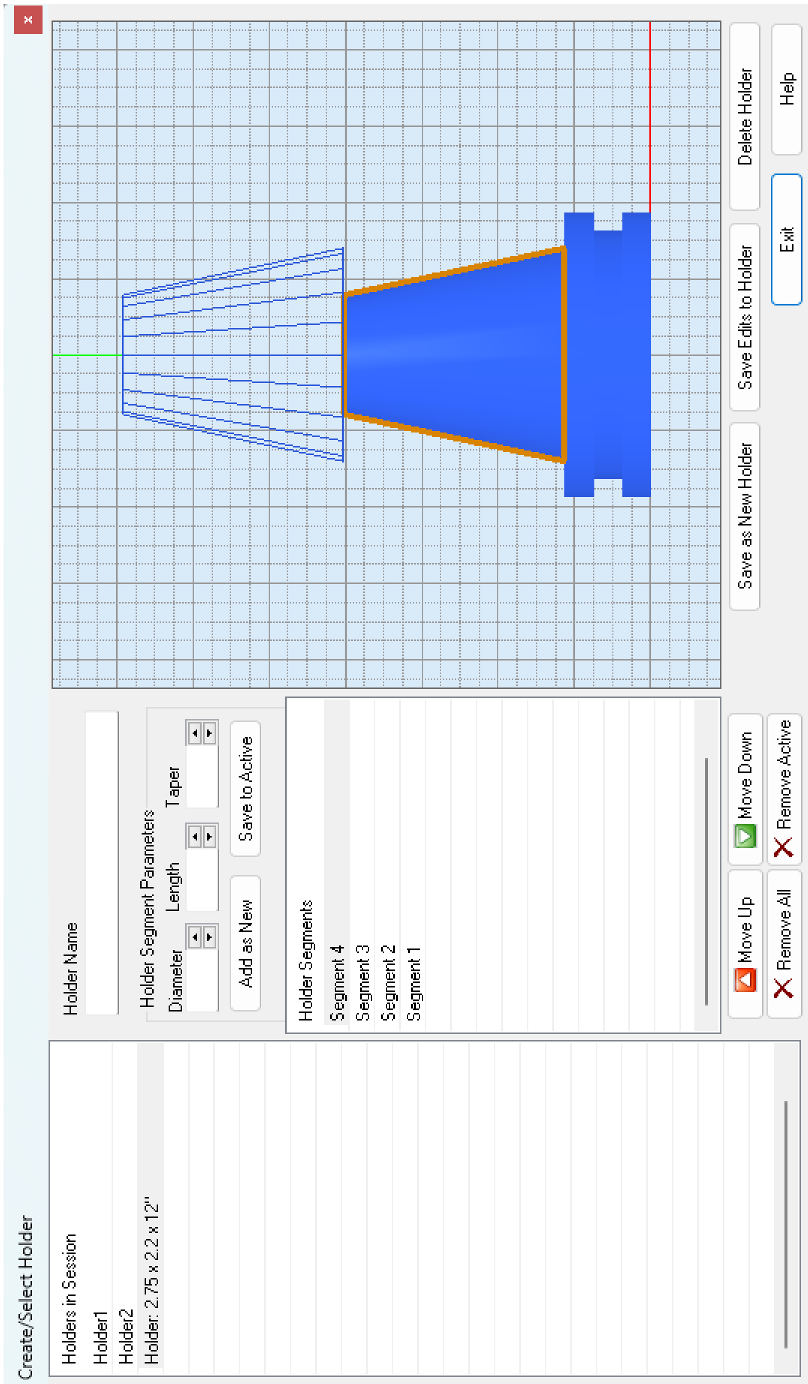
If you are looking to compensate for the full tool diameter, set Stock = -0.125 under the cut parameters tab. (0.125 being the radius of the tool). This would generate the toolpath ON the curve. This would invalidate the simulation as the tool tip stays on the drive geometry.

Note: The Cutcom Register is set under the Create/Select Tool definition dialog.



Reference

8.1 MILL Holder Worksheet



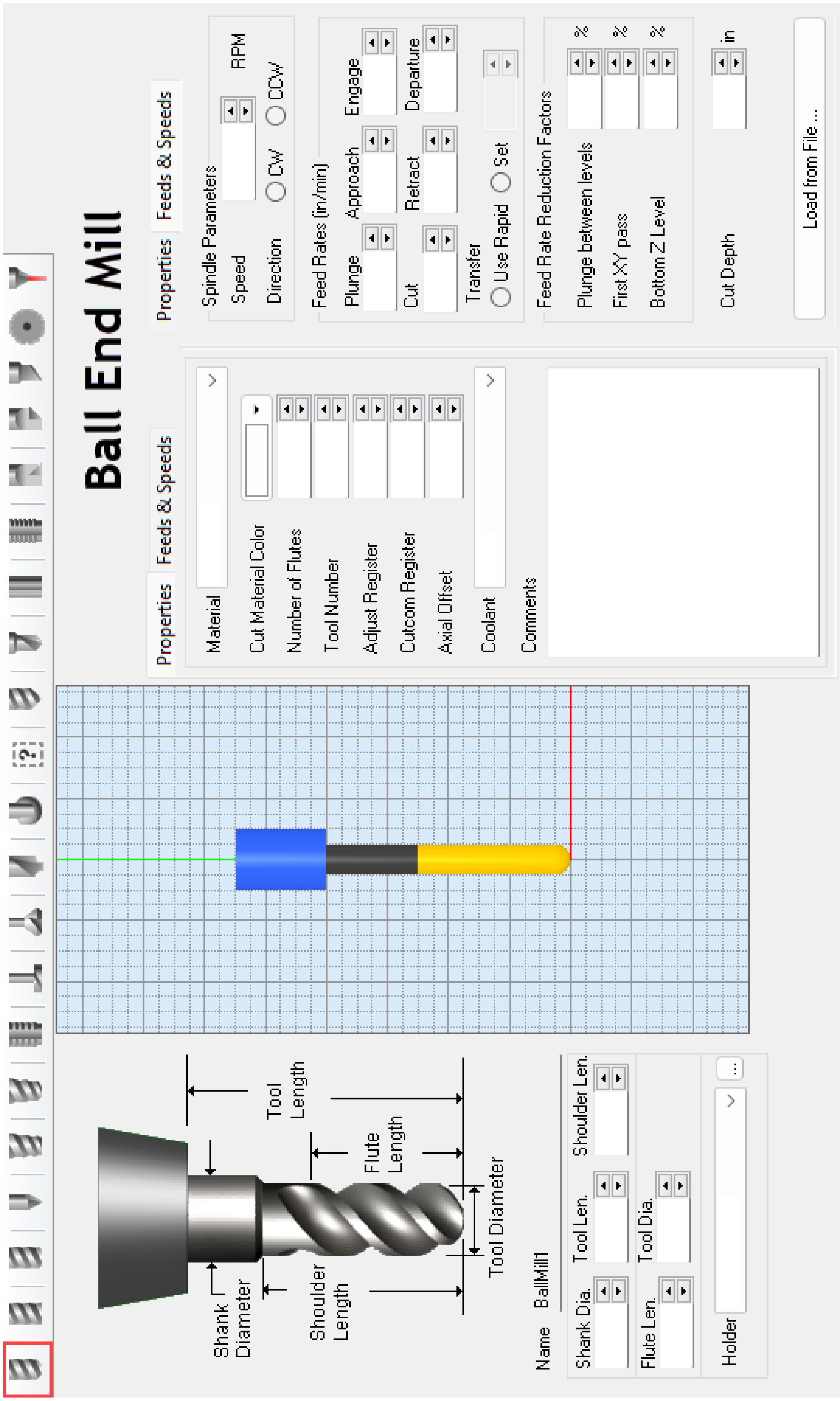
© 2026 MecSoft Corporation

8.2 Mill Tool Worksheets

The following Mill Tool Worksheets are available:

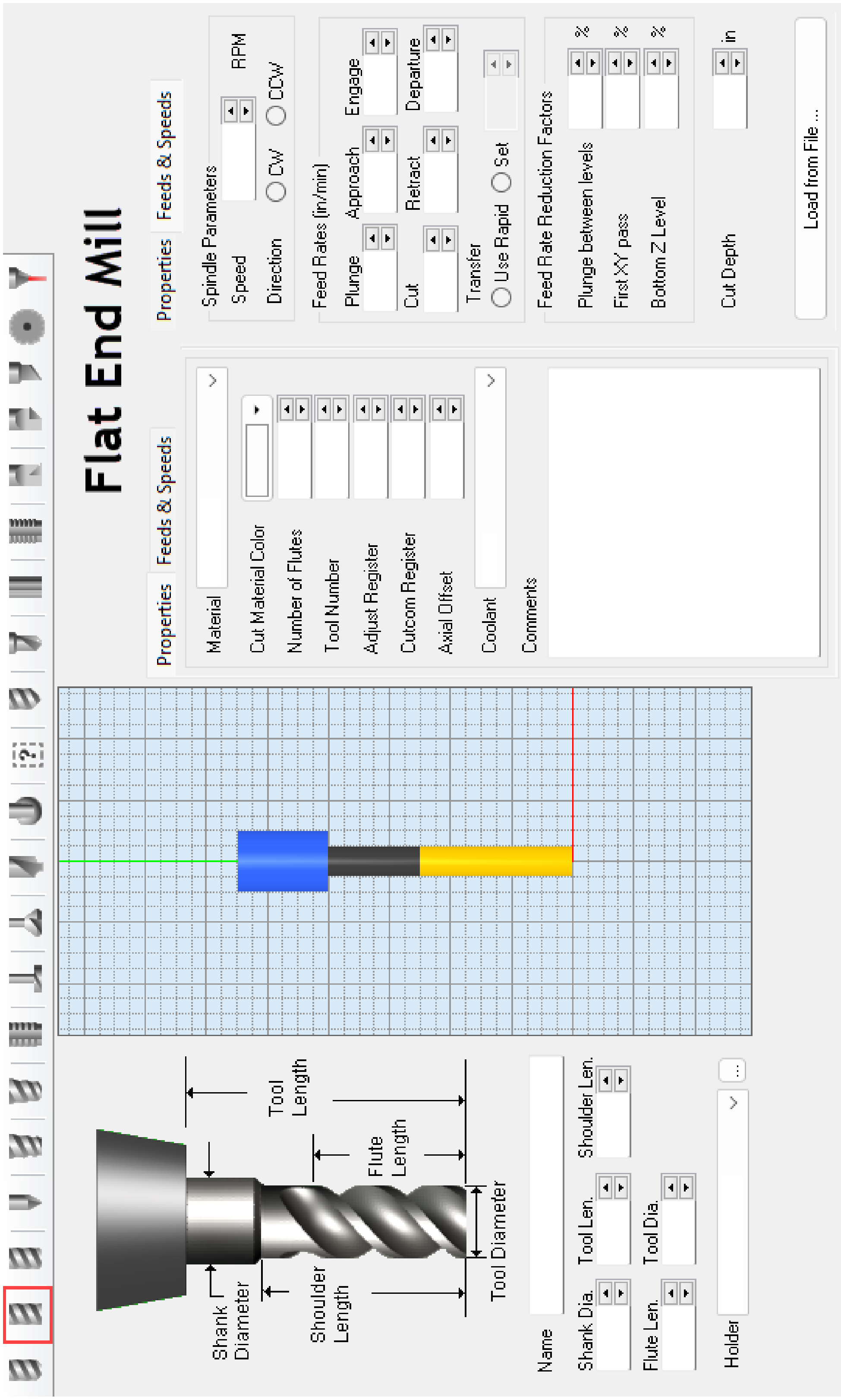
1. Ball Mill Worksheet
2. Flat Mill Worksheet
3. Corner Radius Mill Worksheet
4. Vee Mill Worksheet
5. Chamfer Mill Worksheet
6. Taper Mill Worksheet
7. Thread Mill Worksheet
8. Dovetail Mill Worksheet
9. Fillet Mill Worksheet
10. Lollipop Mill Worksheet
11. User Defined Mill Worksheet
12. Drag Knife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

8.2.1 Ball Mill Worksheet

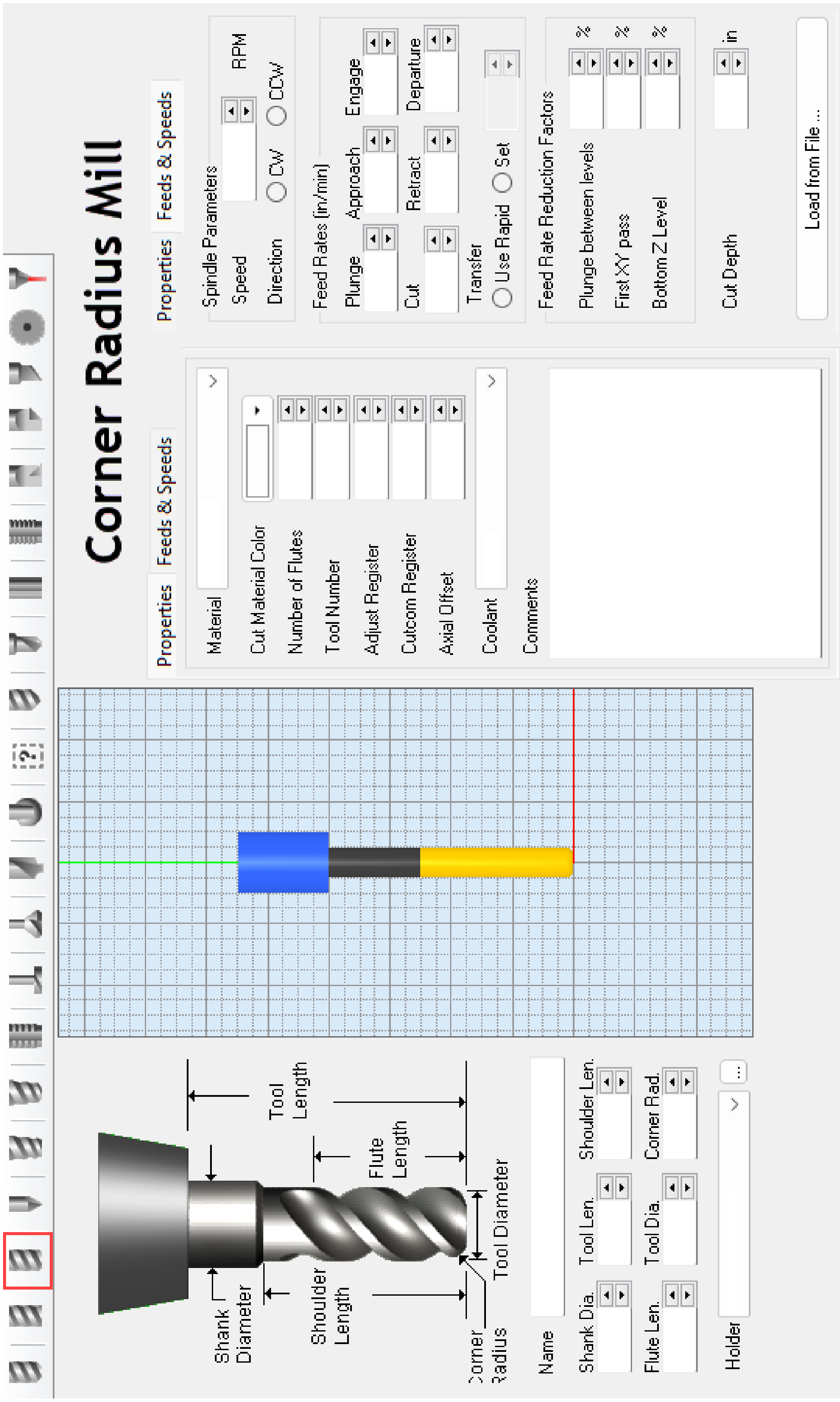


© 2026 MecSoft Corporation

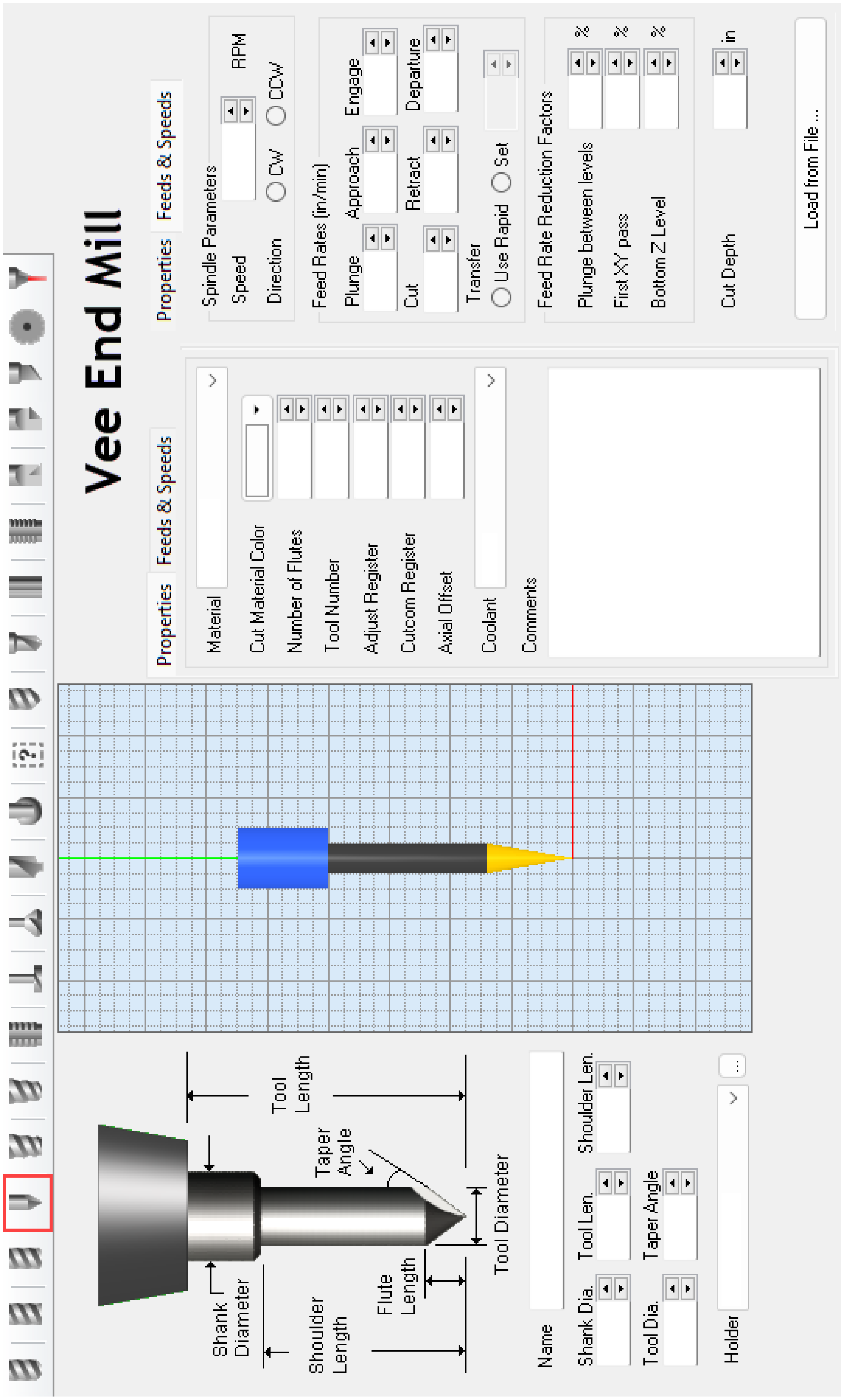
8.2.2 Flat Mill Worksheet



8.2.3 Corner Radius Mill Worksheet

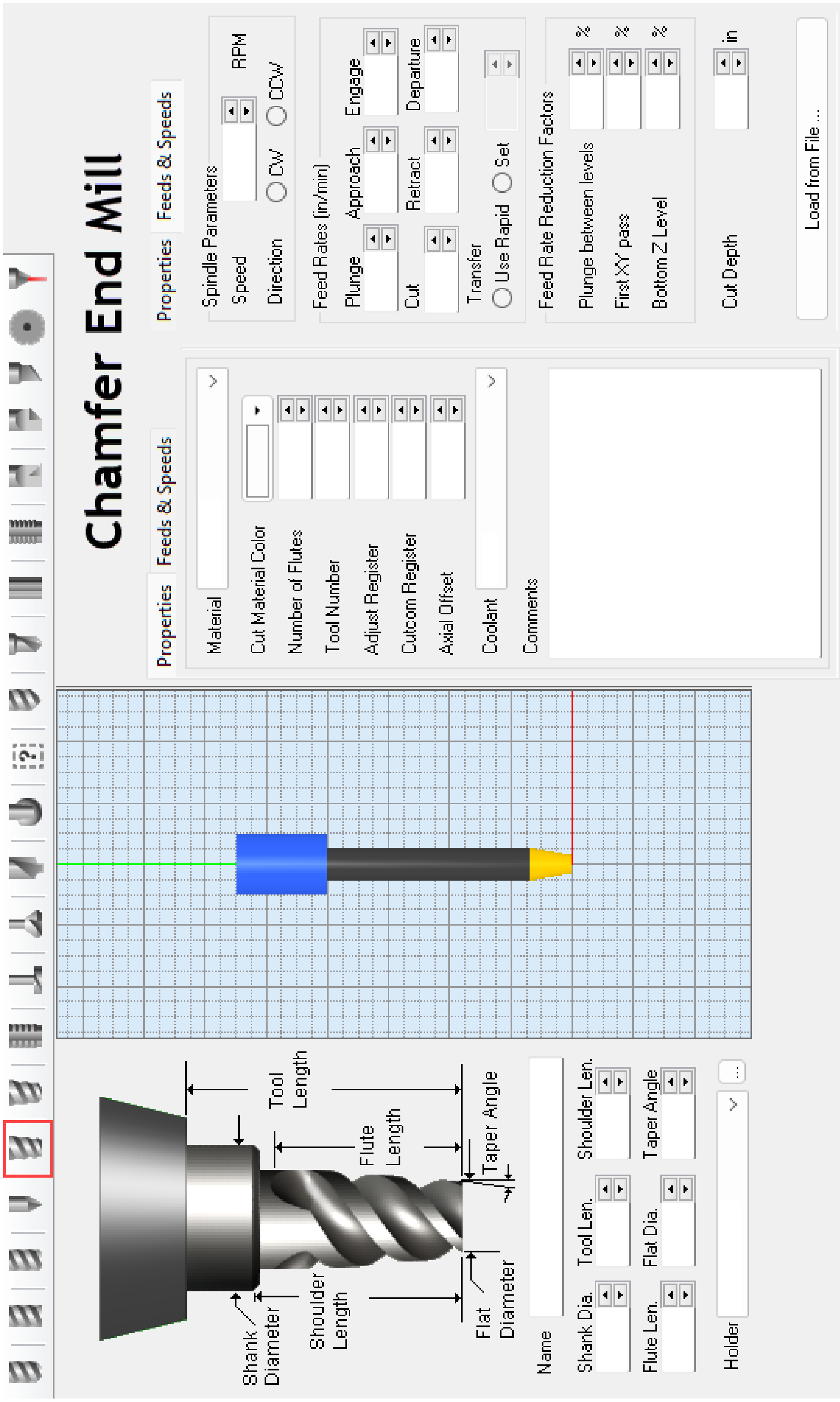


8.2.4 Vee Mill Worksheet

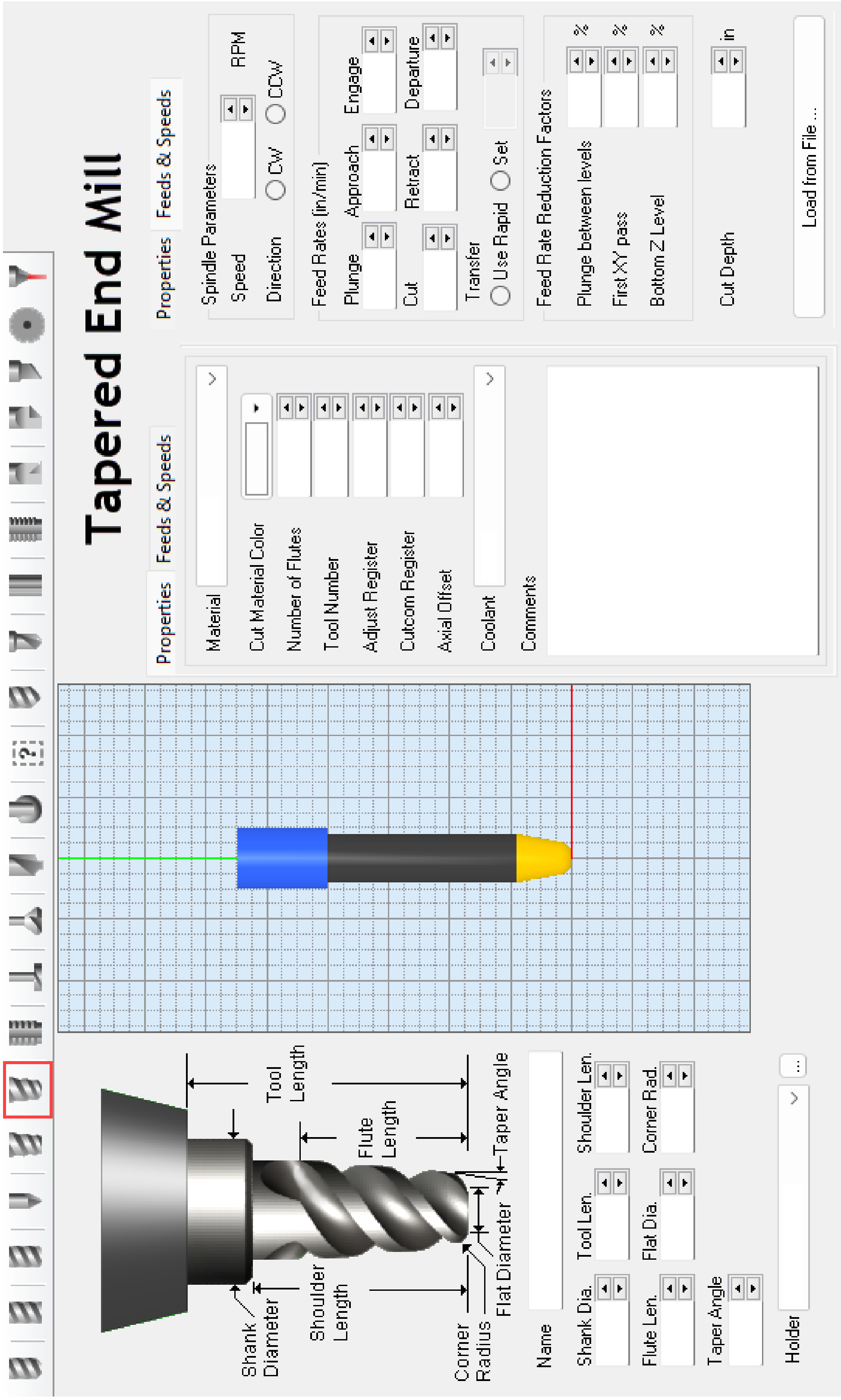


© 2026 MecSoft Corporation

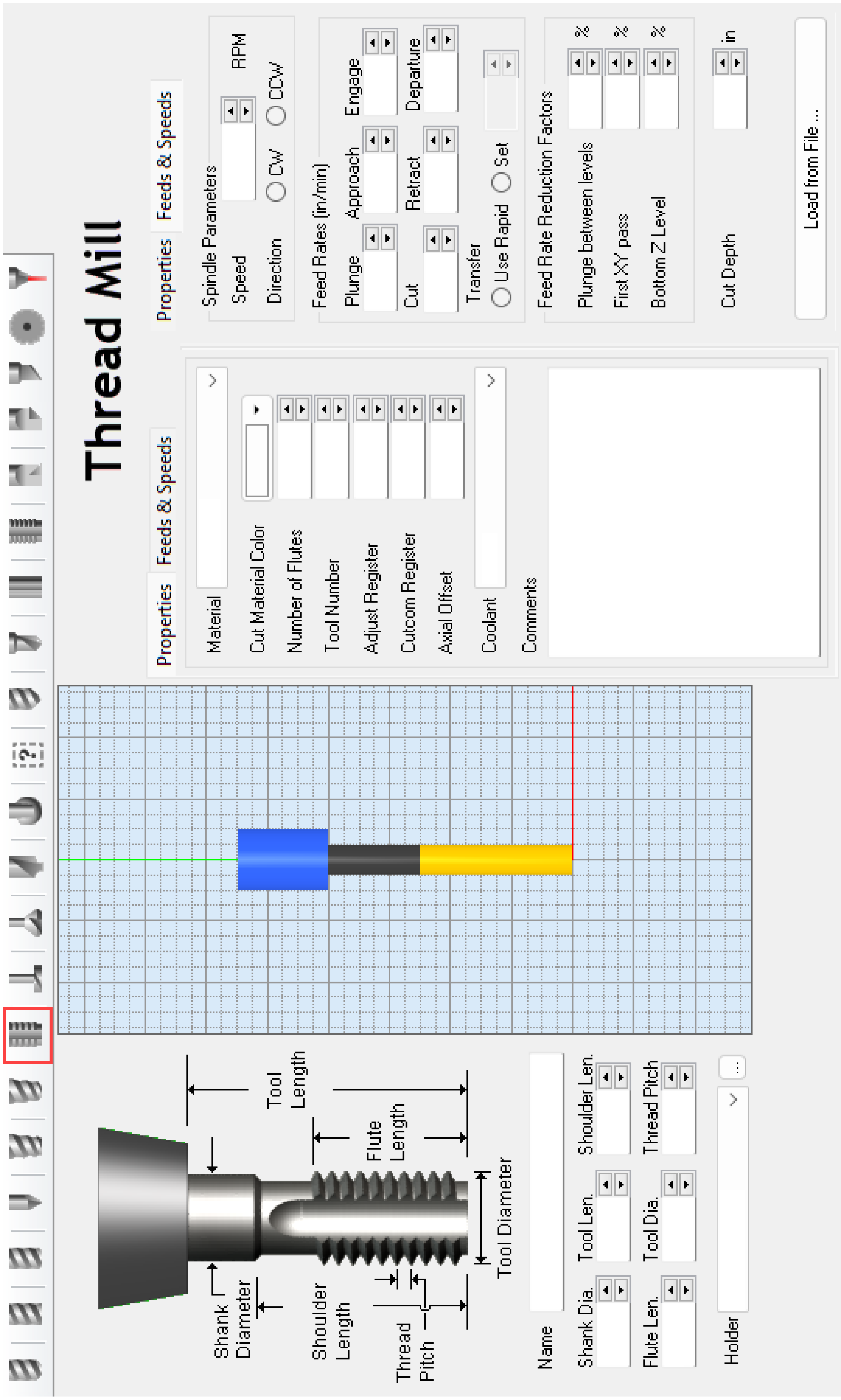
8.2.5 Chamfer Mill Worksheet



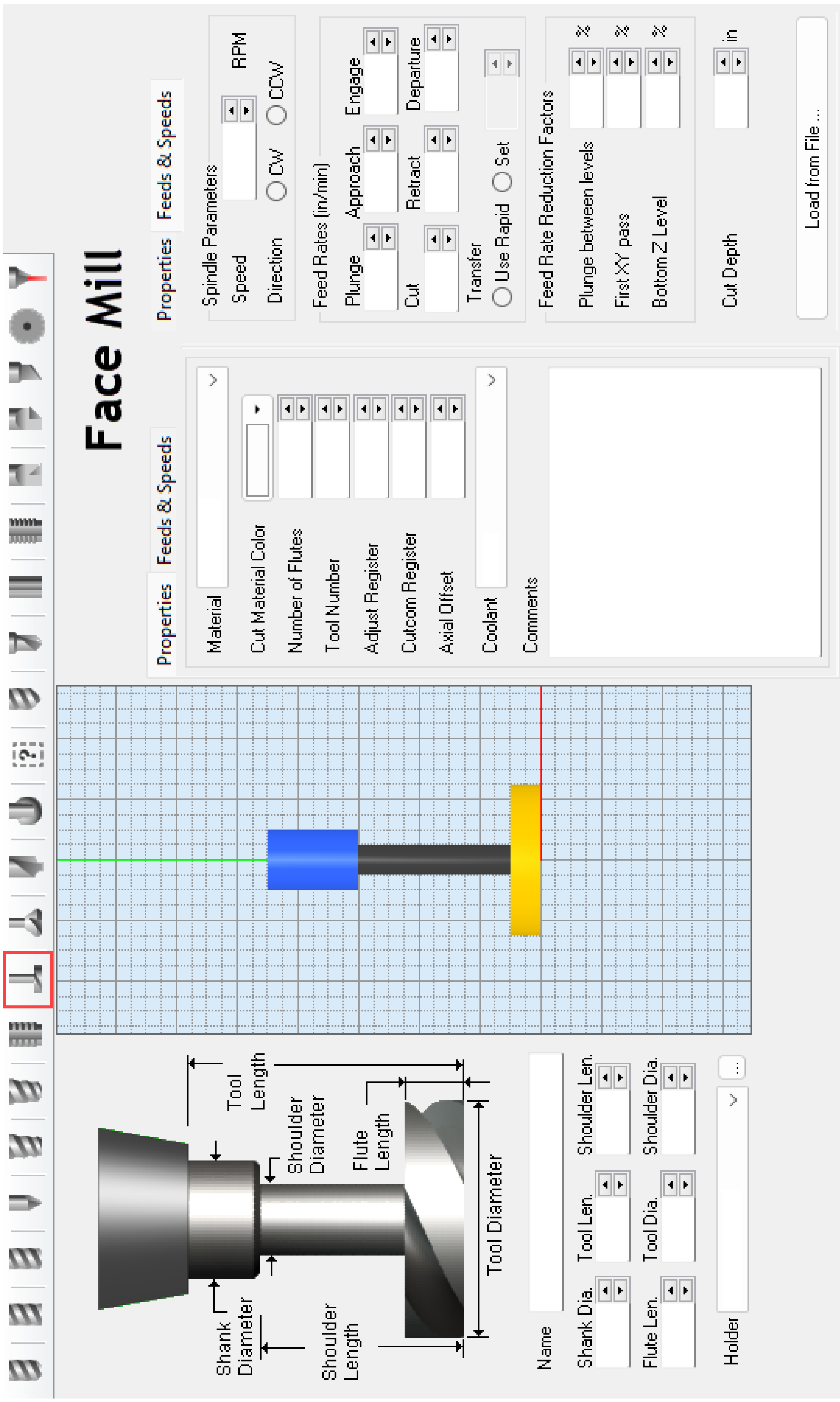
8.2.6 Taper Mill Worksheet



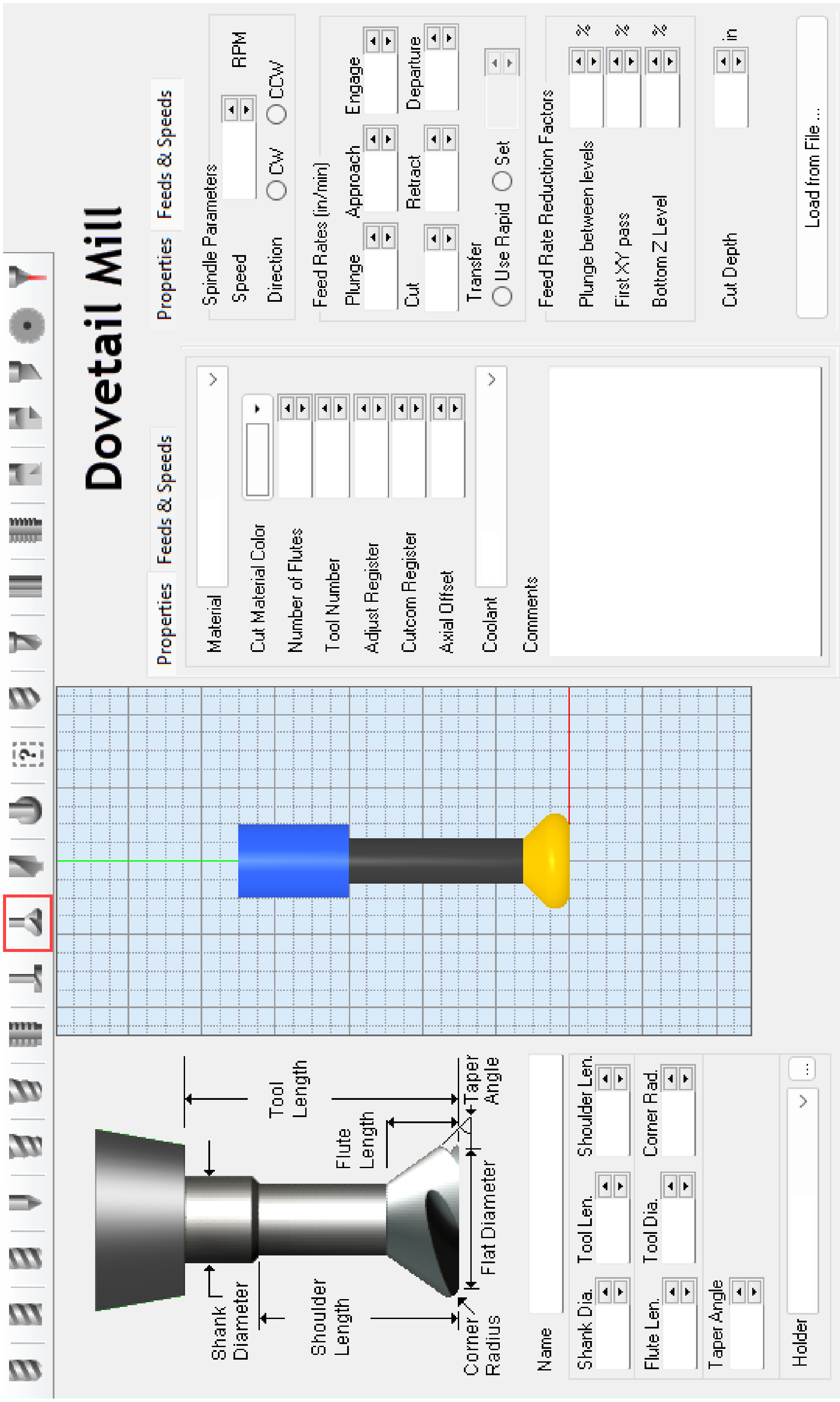
8.2.7 Thread Mill Worksheet



8.2.8 Face Mill Worksheet

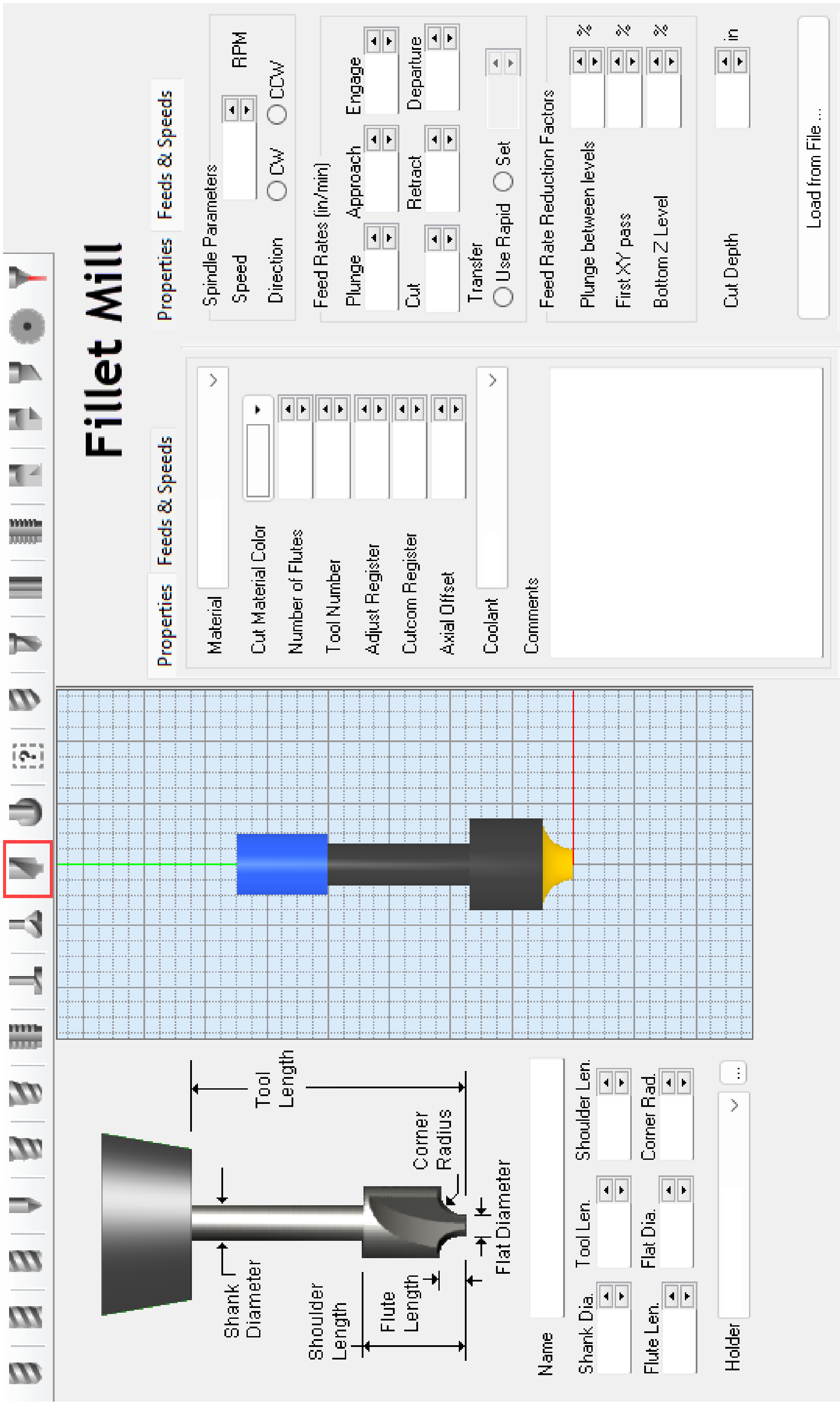


8.2.9 Dovetail Mill Worksheet

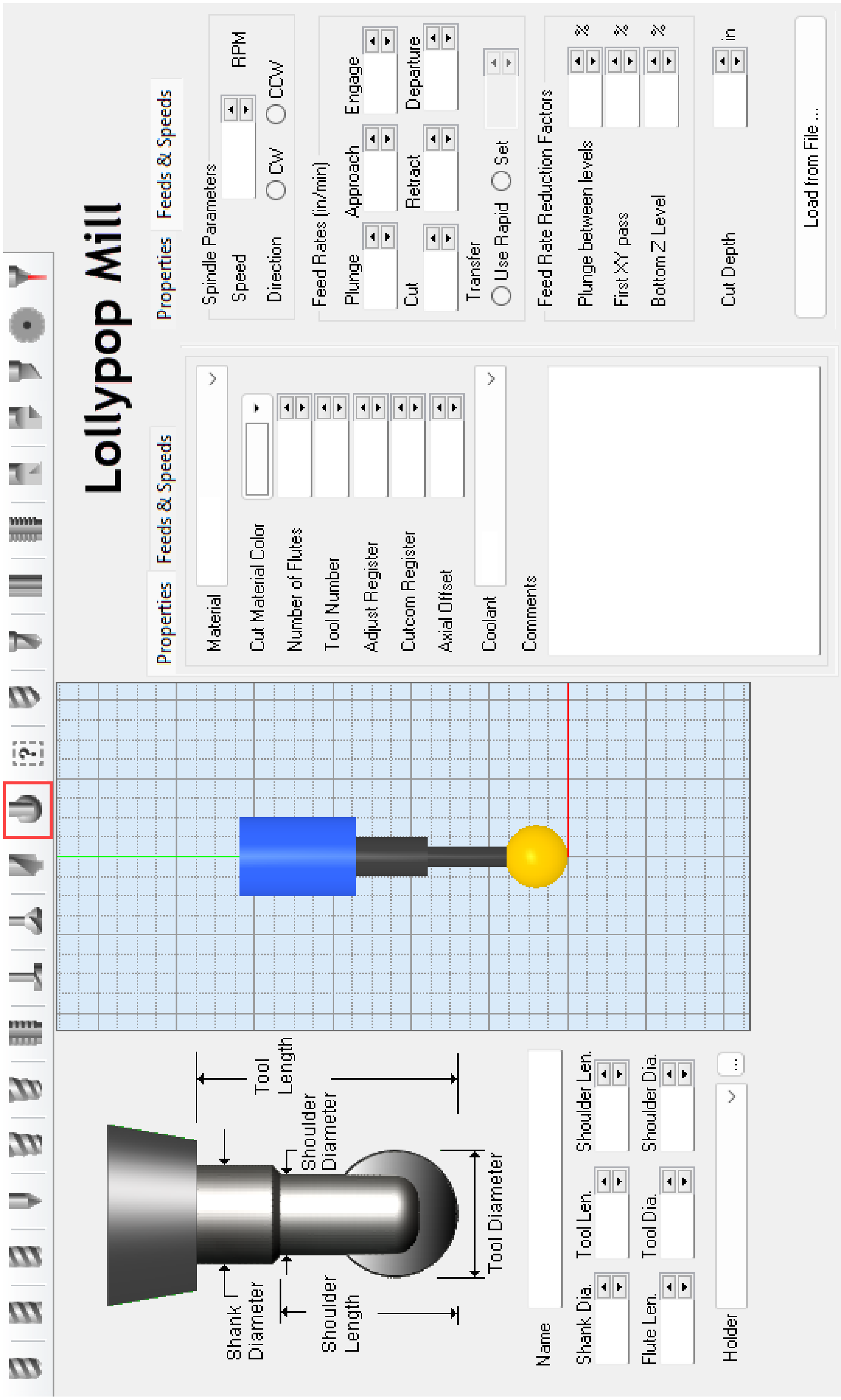


© 2026 MecSoft Corporation

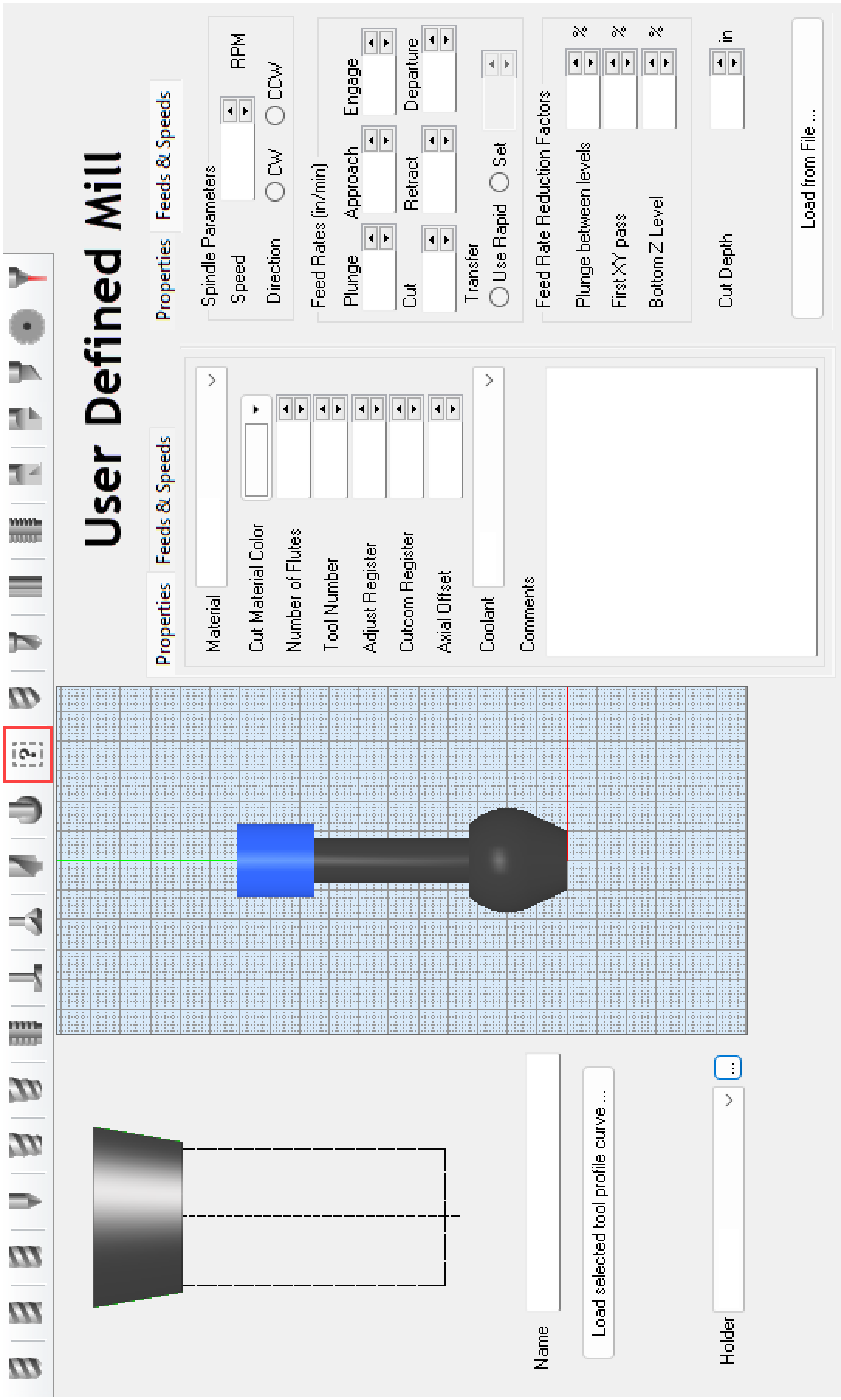
8.2.10 Fillet Mill Worksheet



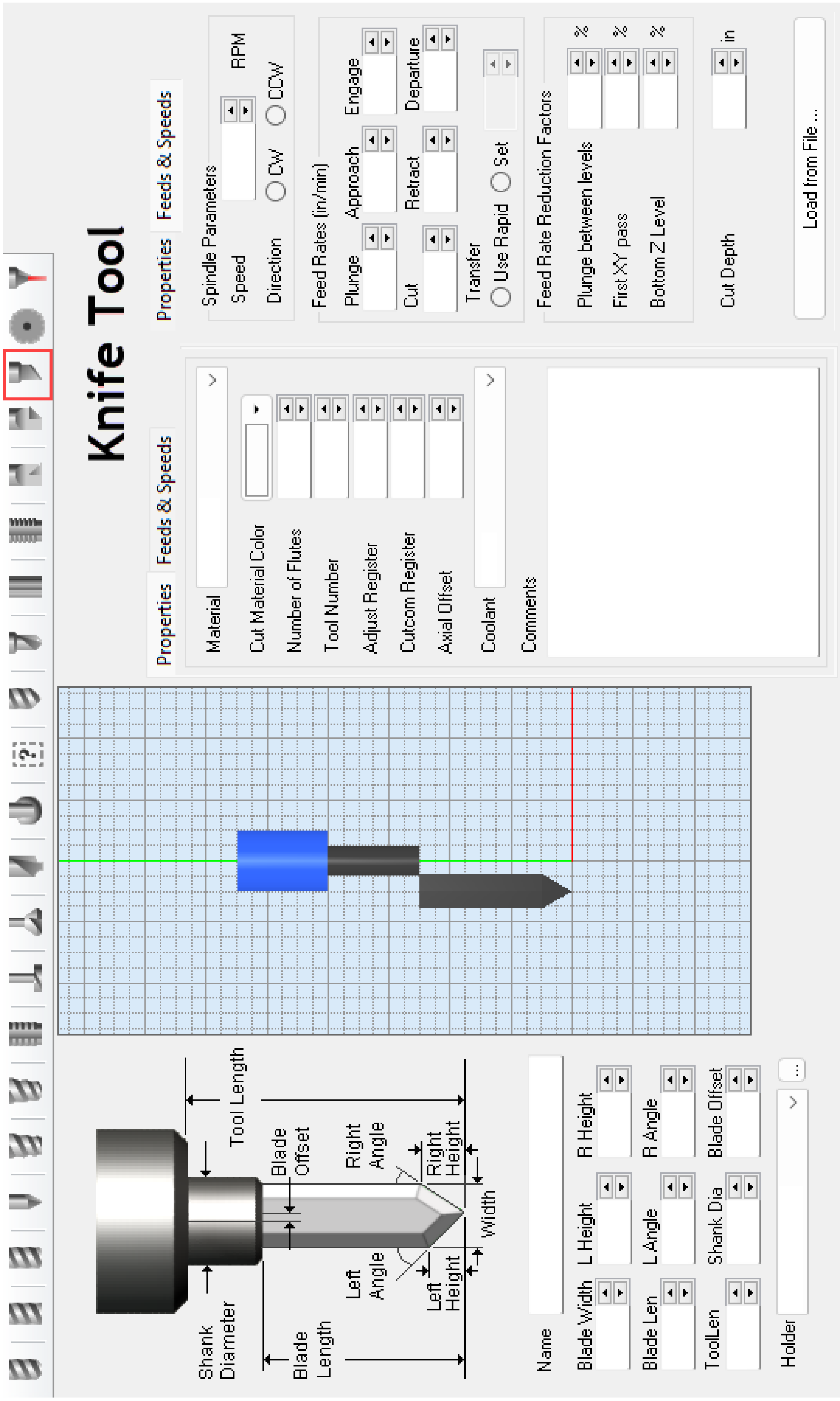
8.2.11 Lollipop Mill Worksheet



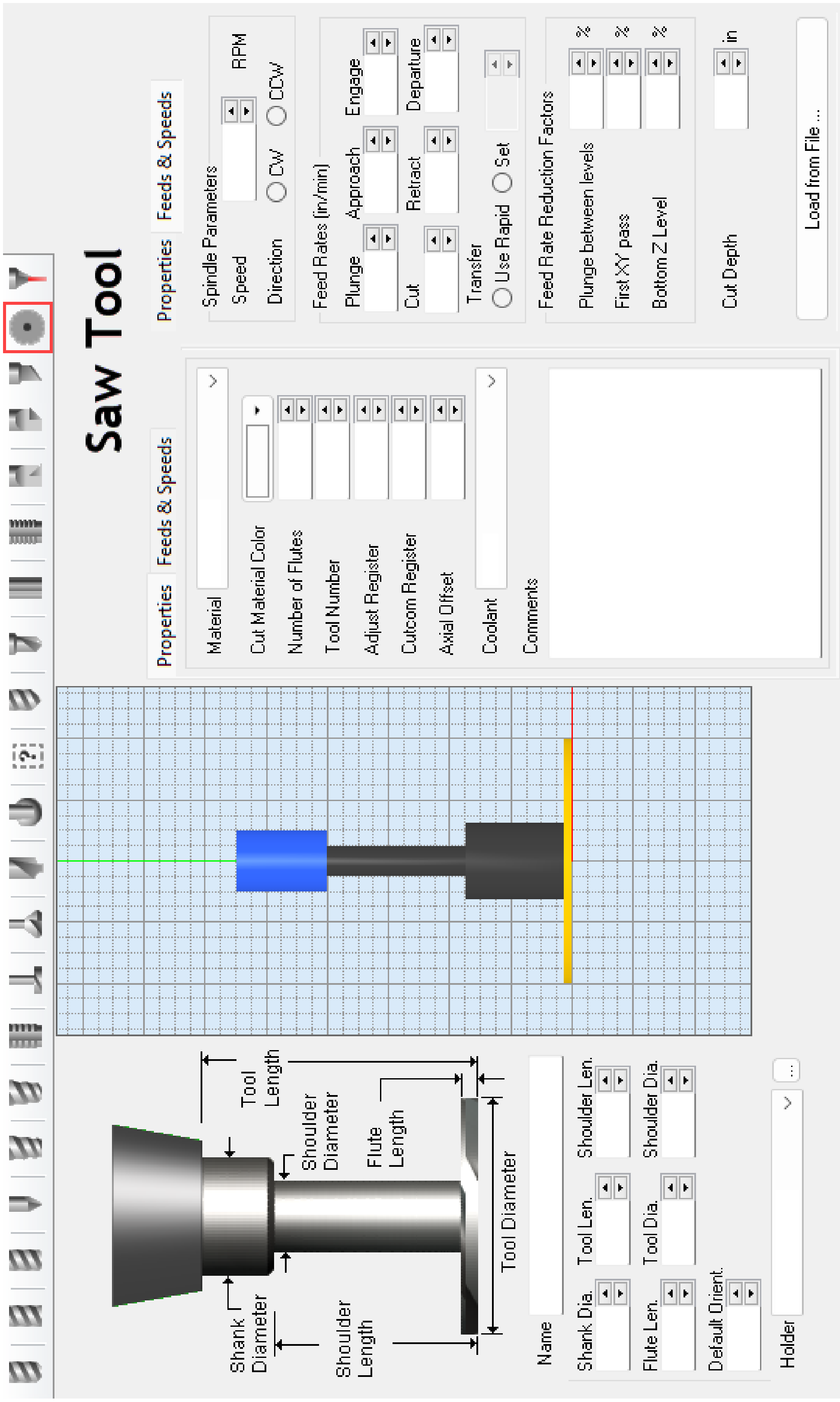
8.2.12 User Defined Mill Worksheet



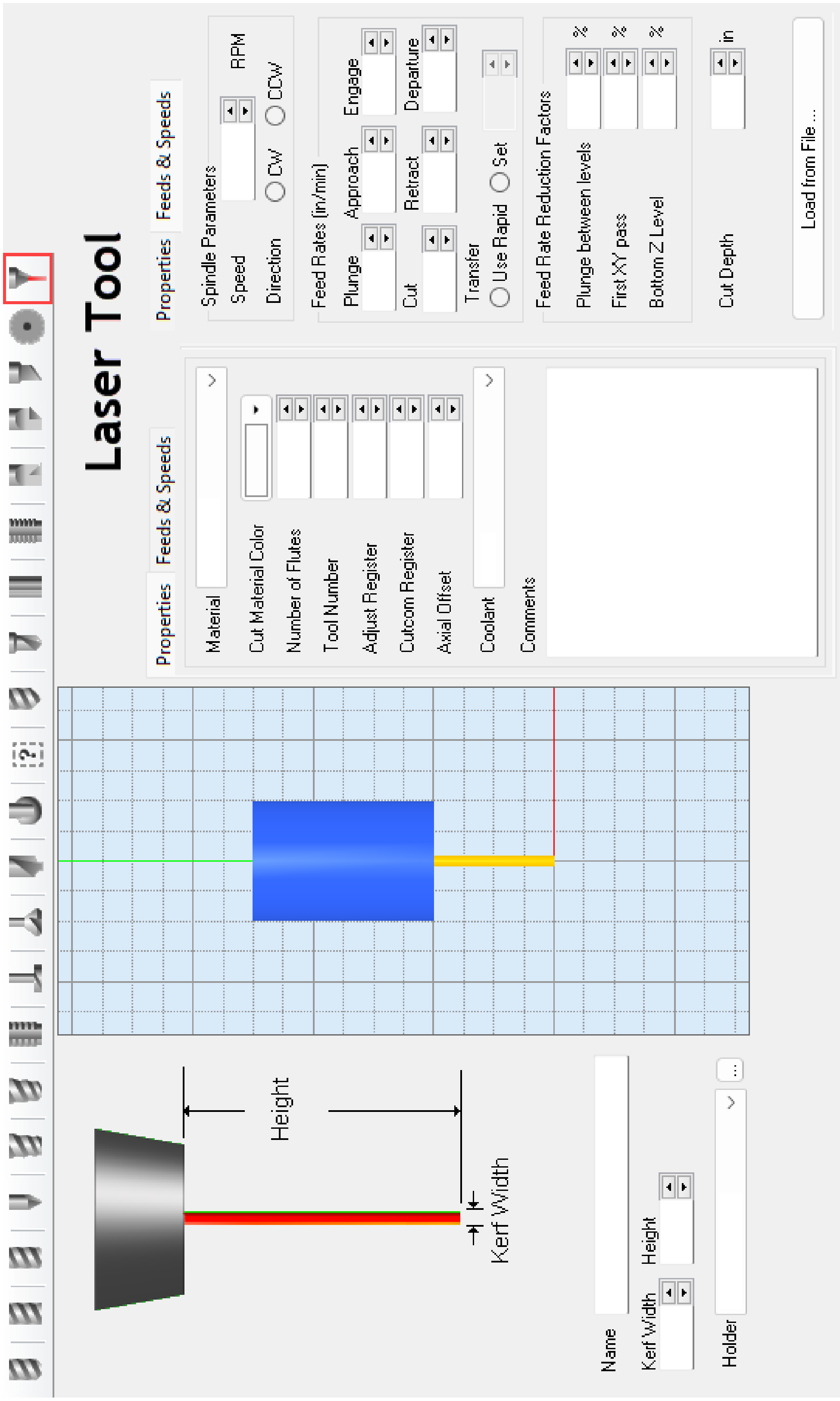
8.2.13 Drag Knife Worksheet



8.2.14 Saw Tool Worksheet



8.2.15 Laser Tool Worksheet



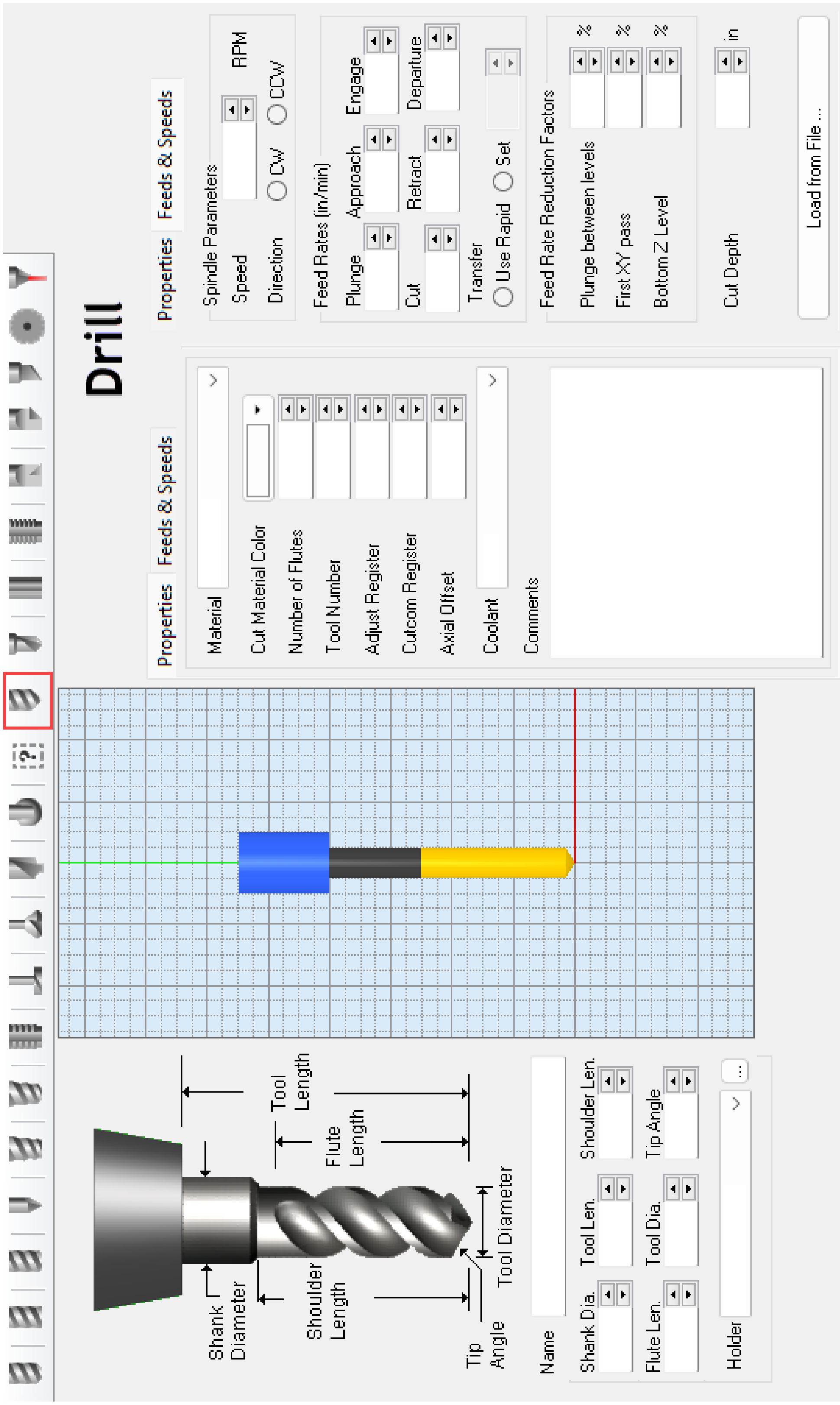
8.3 Drill Tool Worksheets

The following Drill Tool Worksheets are available:

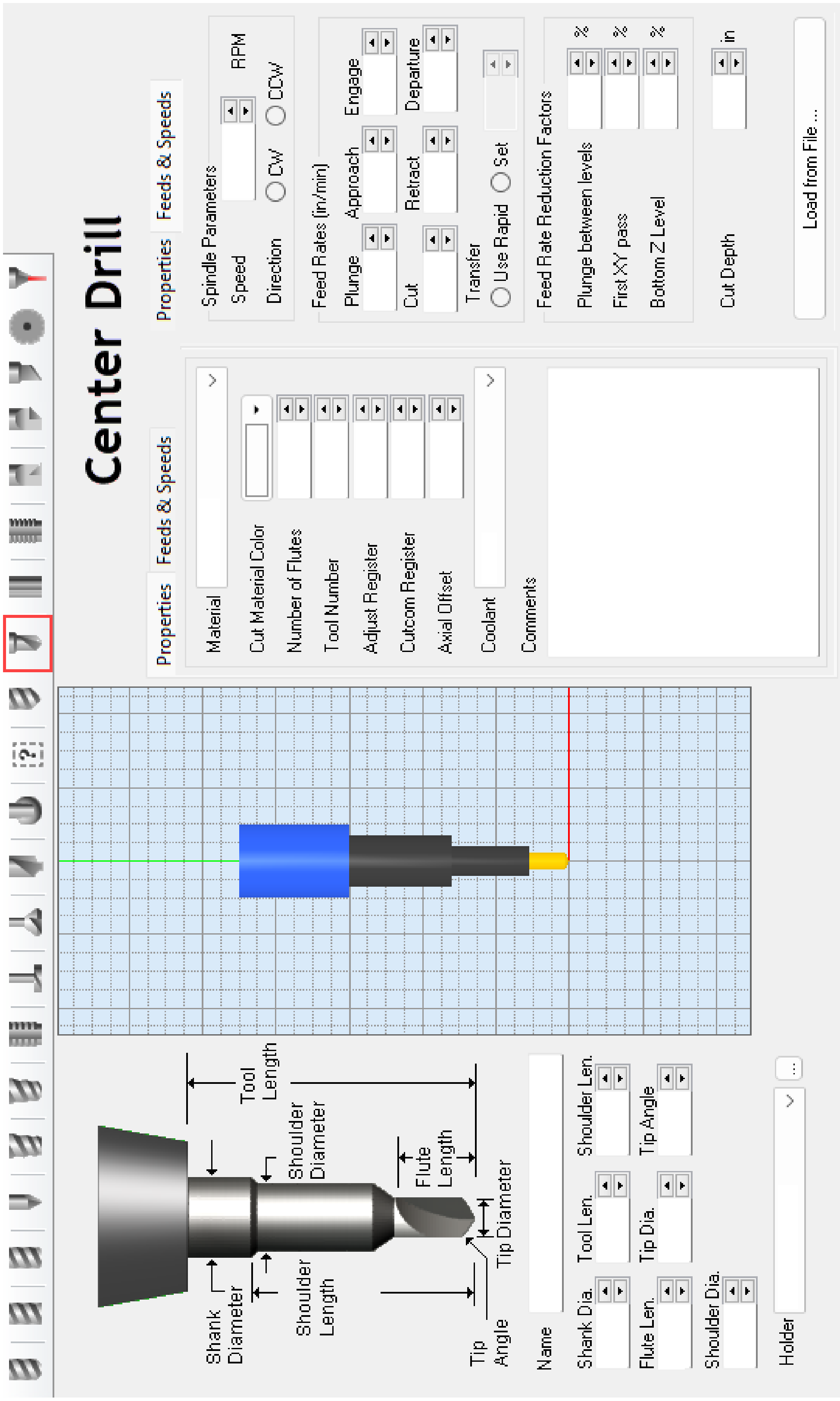
1. Drill Worksheet
2. Center Drill Mill Worksheet
3. Reamer Worksheet
4. Tap Worksheet
5. Bore Worksheet
6. Reverse Bore Worksheet

© 2026 MecSoft Corporation

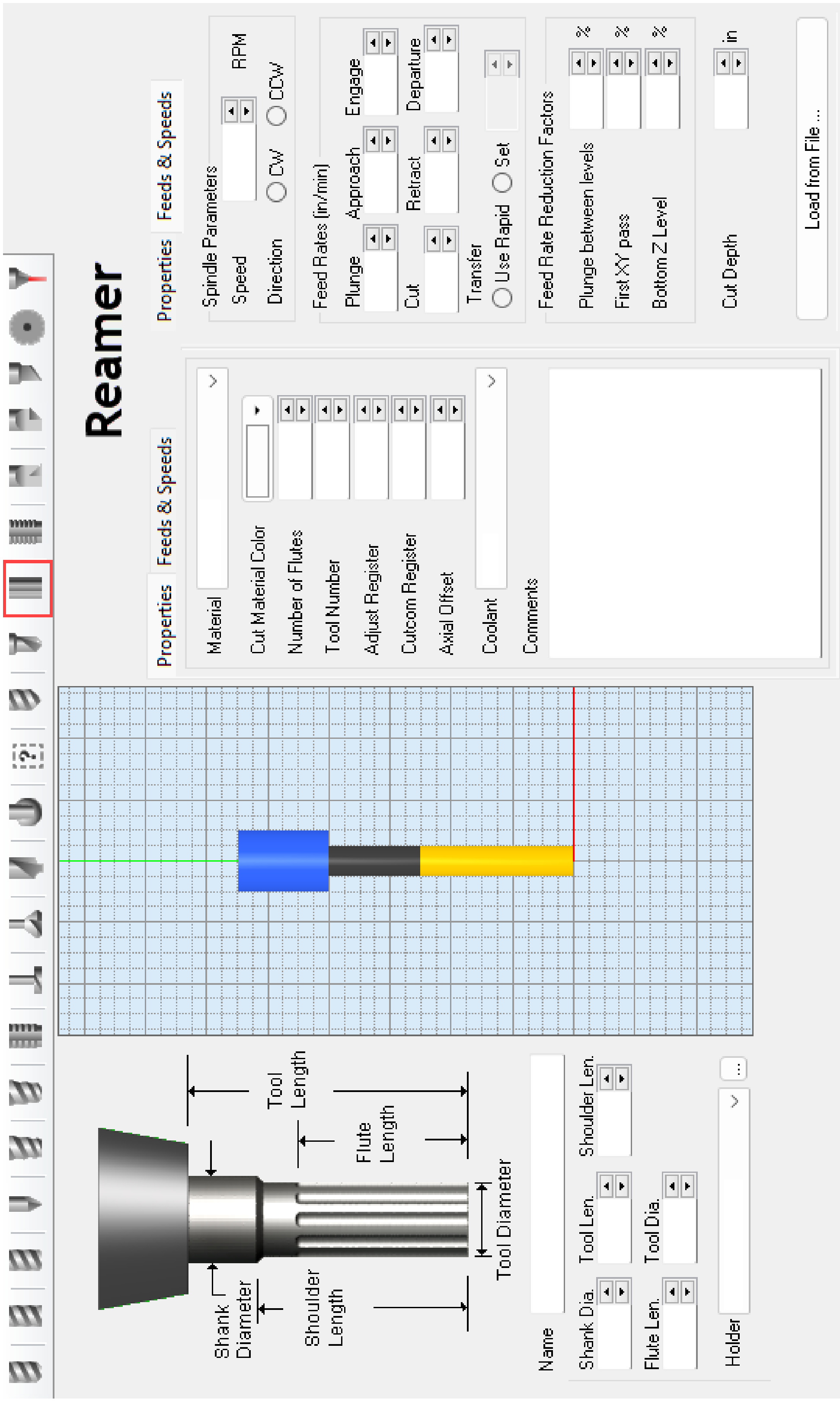
8.3.1 Drill Worksheet



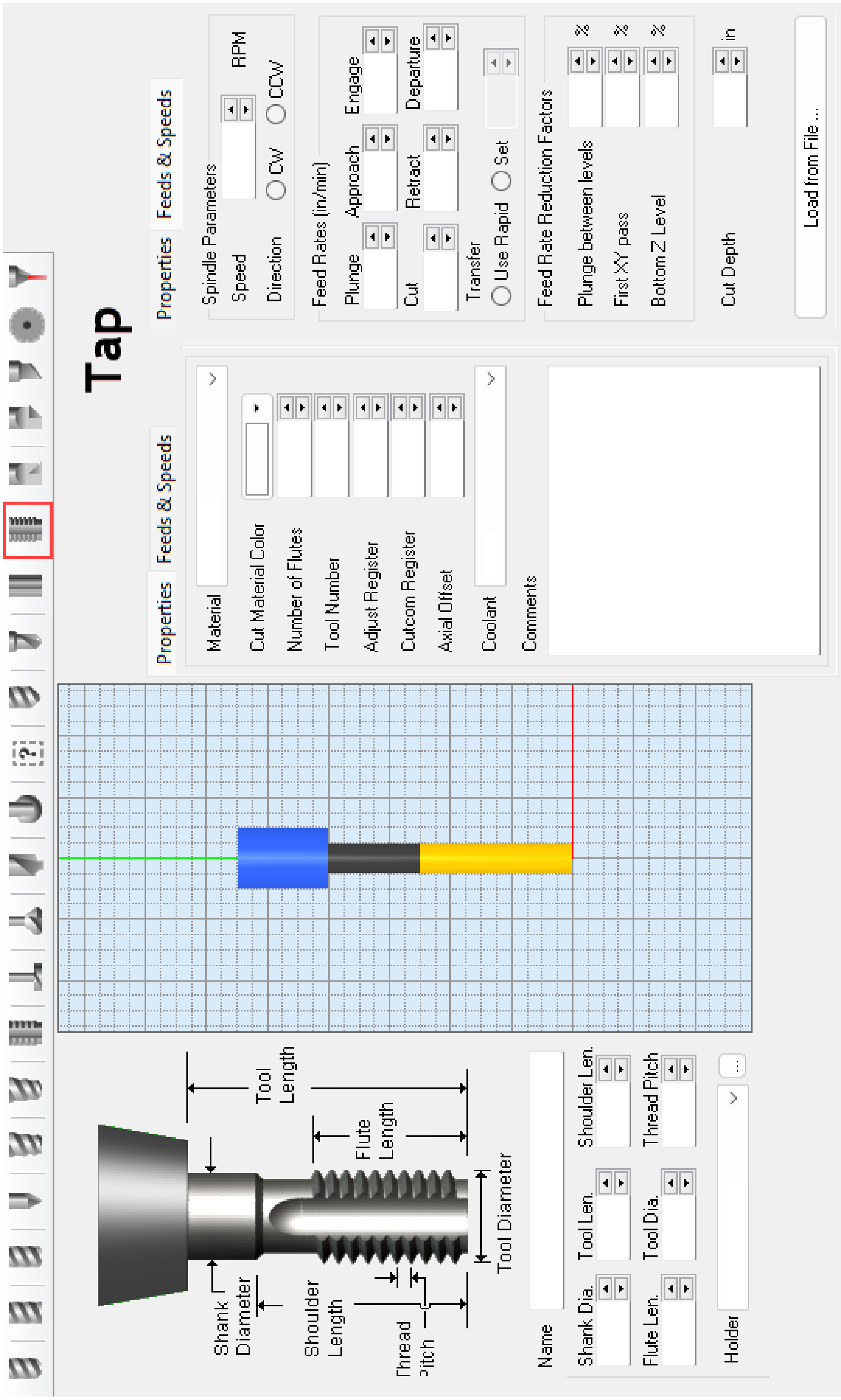
8.3.2 Center Drill Worksheet



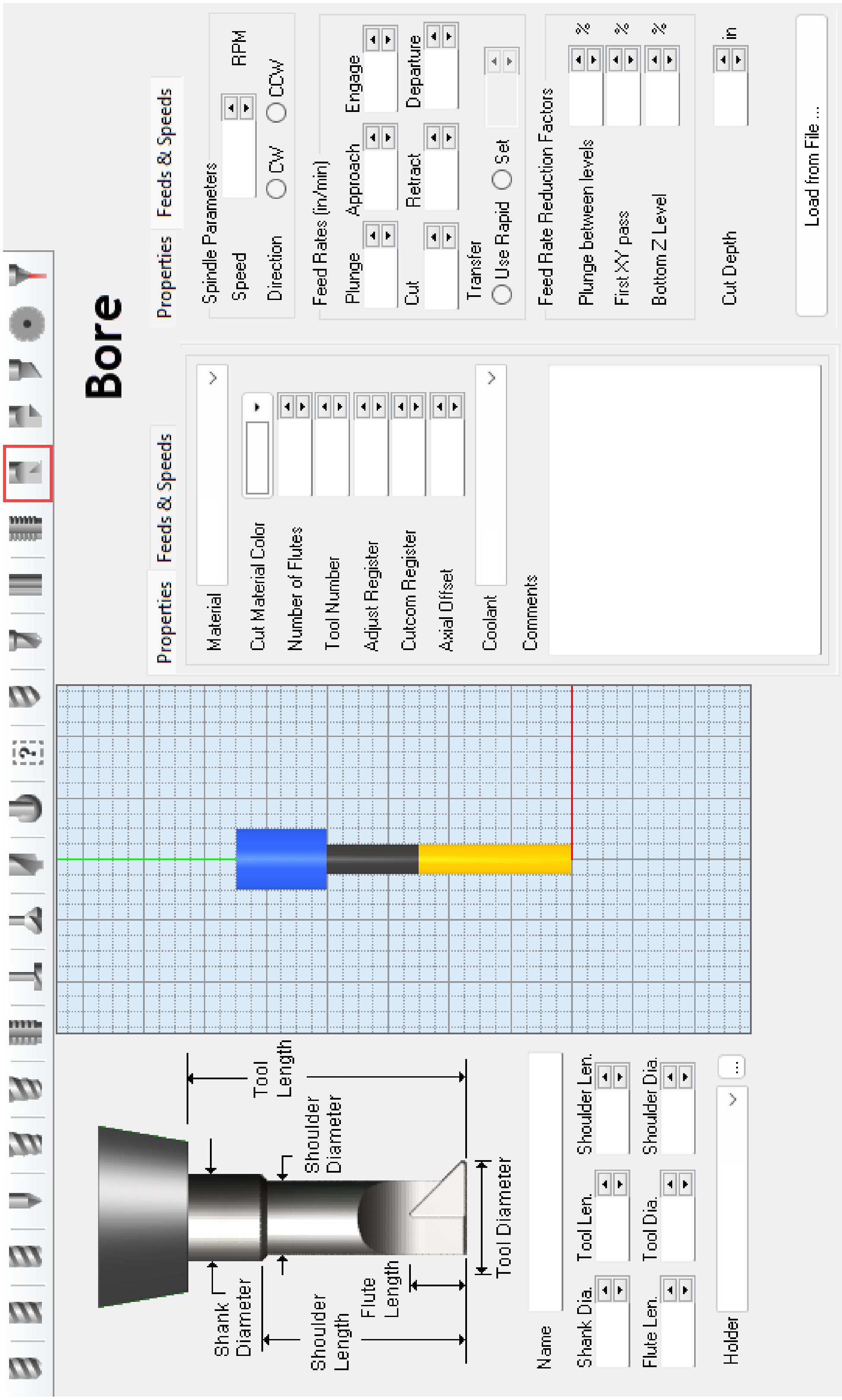
8.3.3 Reamer Worksheet



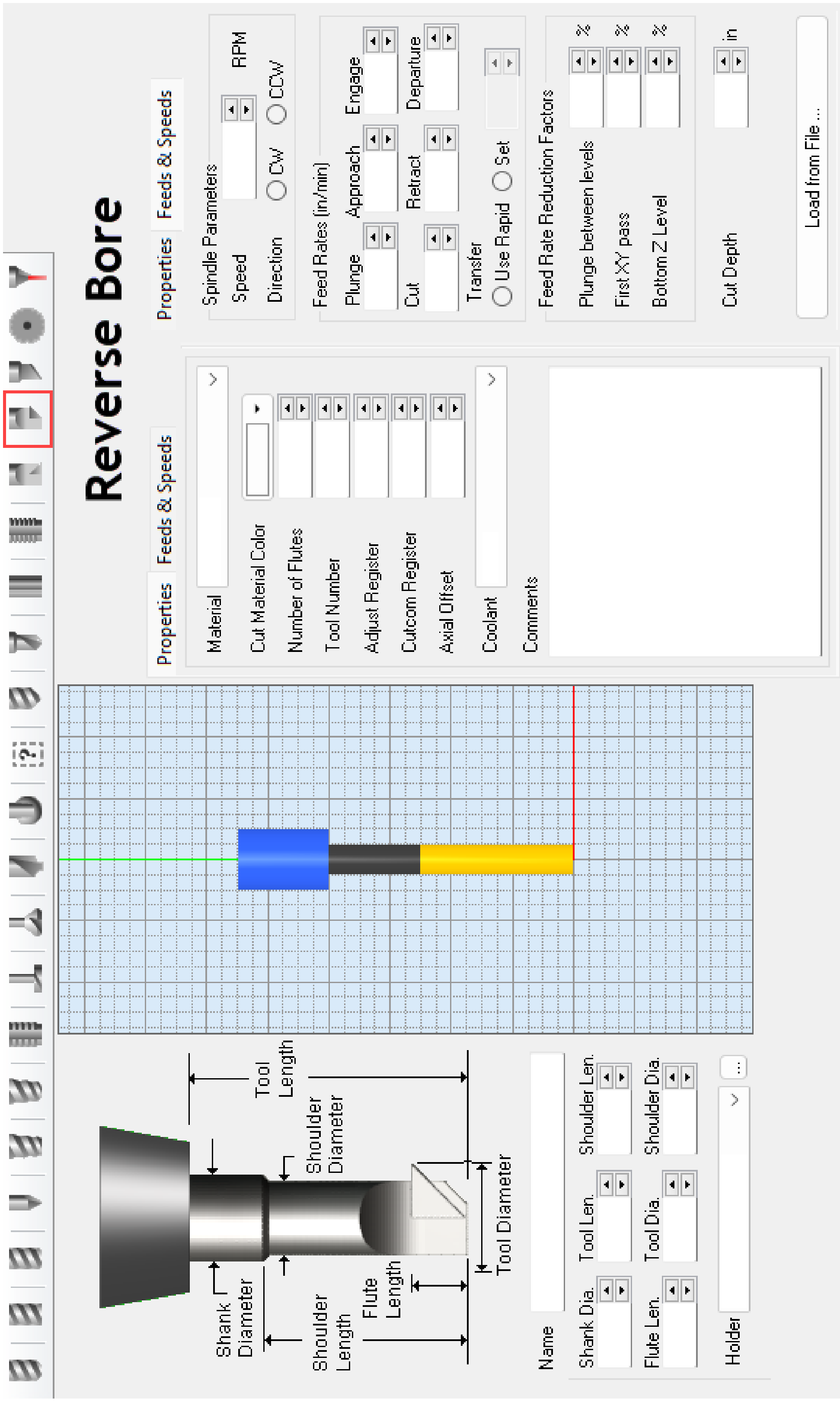
8.3.4 Tap Worksheet



8.3.5 Bore Worksheet



8.3.6 Reverse Bore Worksheet

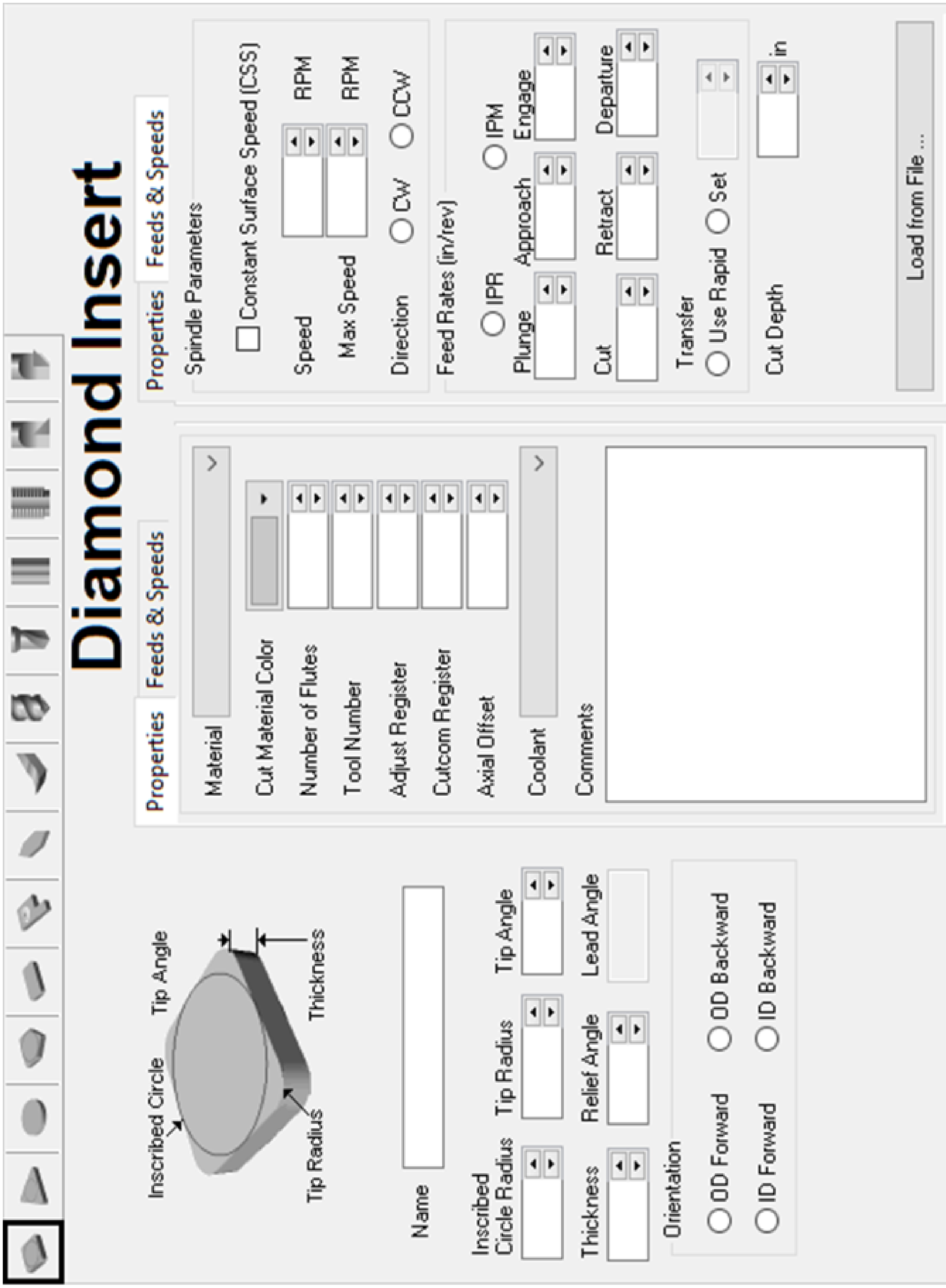


8.4 Turn Tool Worksheets

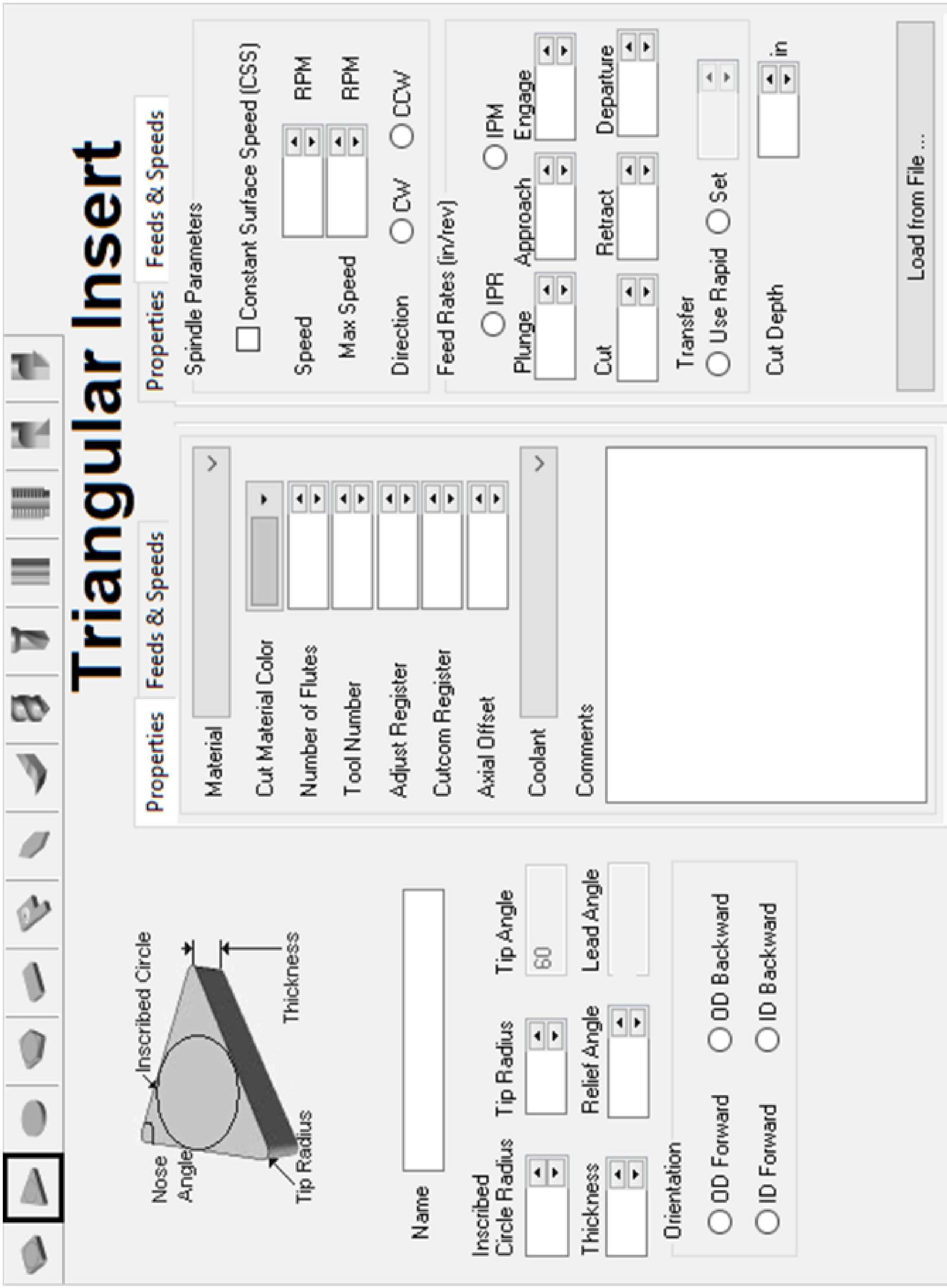
The following TURN Tool Worksheets are available:

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

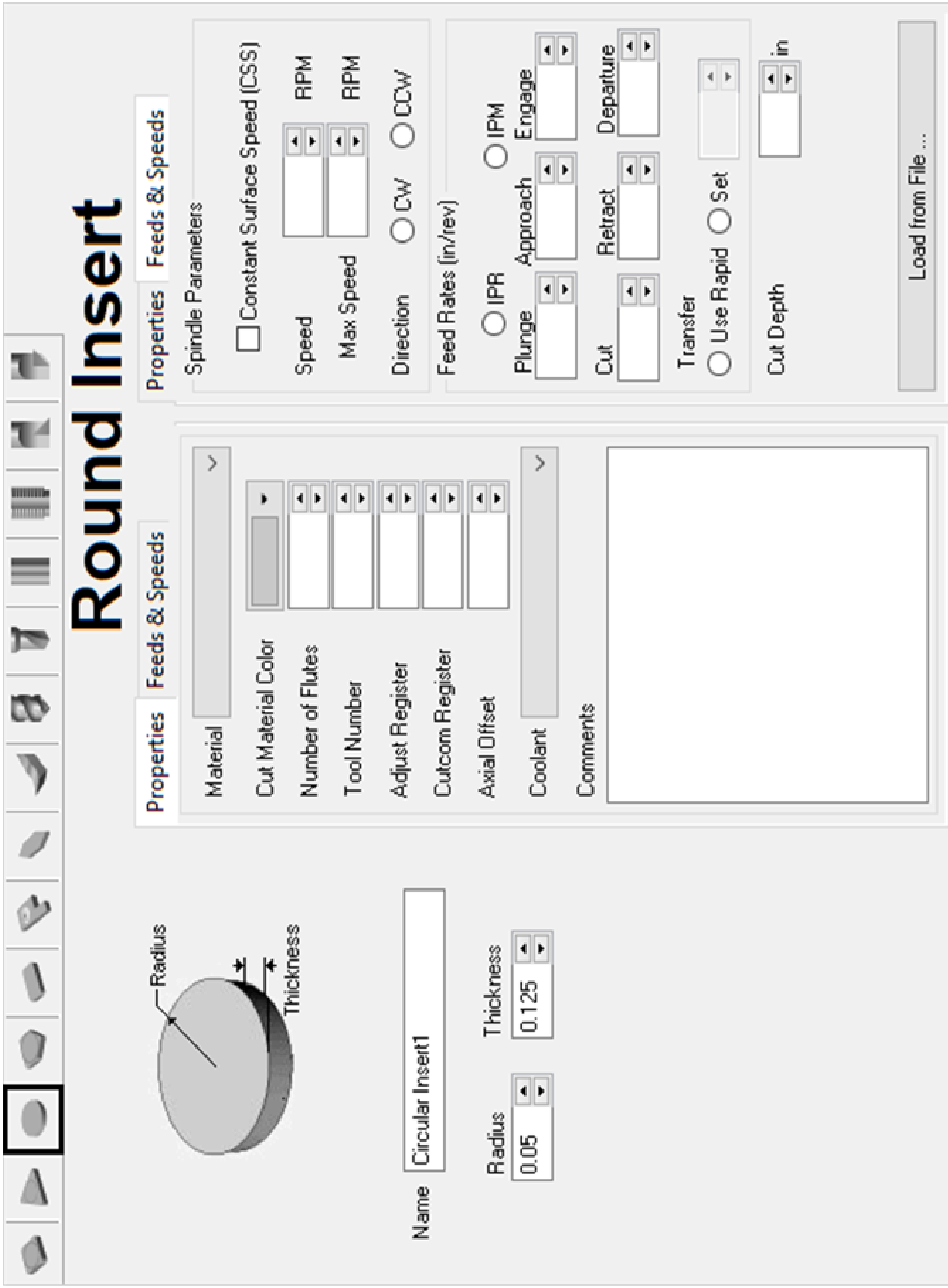
8.4.1 Diamond Insert Worksheet



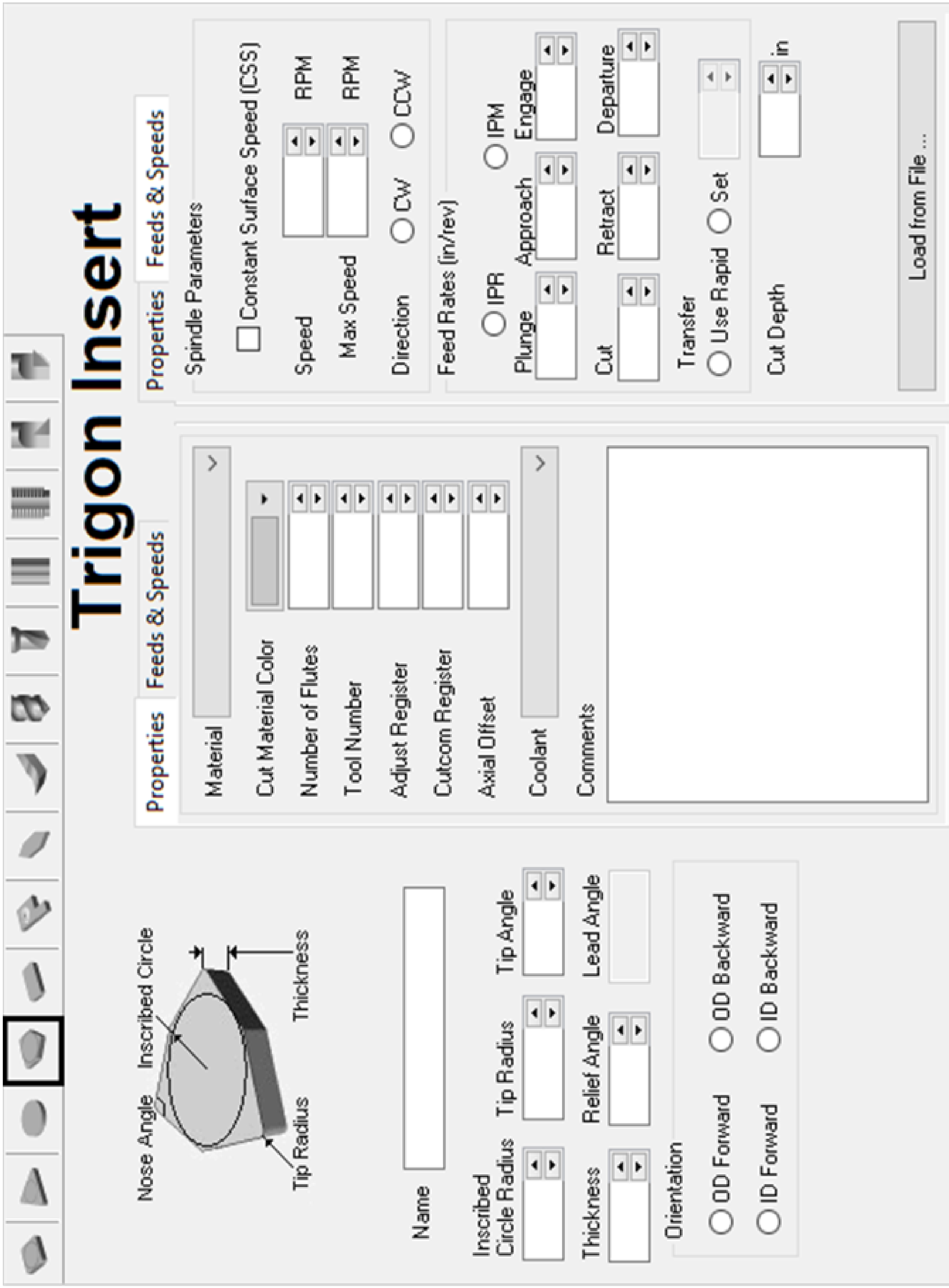
8.4.2 Triangular Insert Worksheet



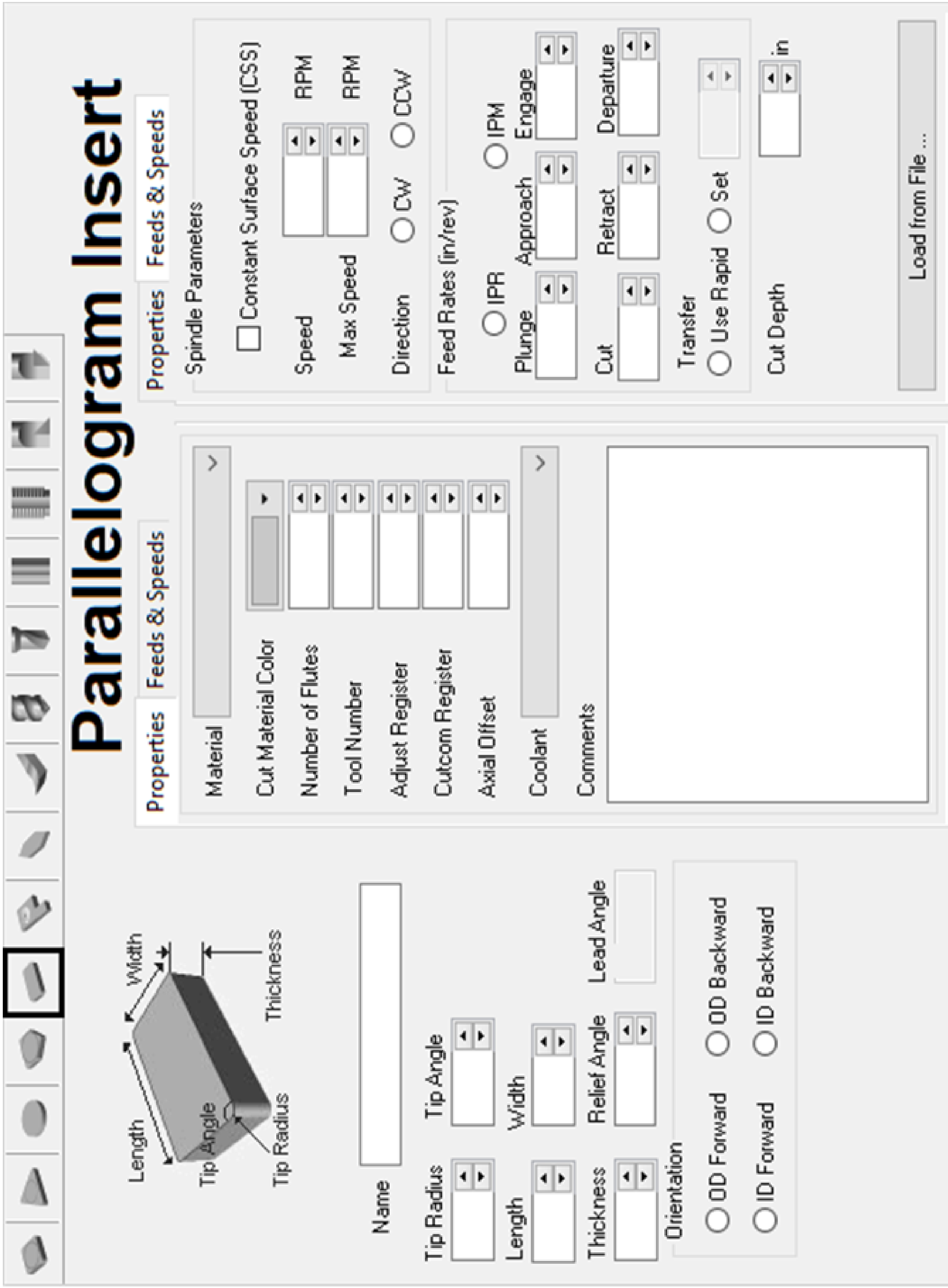
8.4.3 Round Insert Worksheet



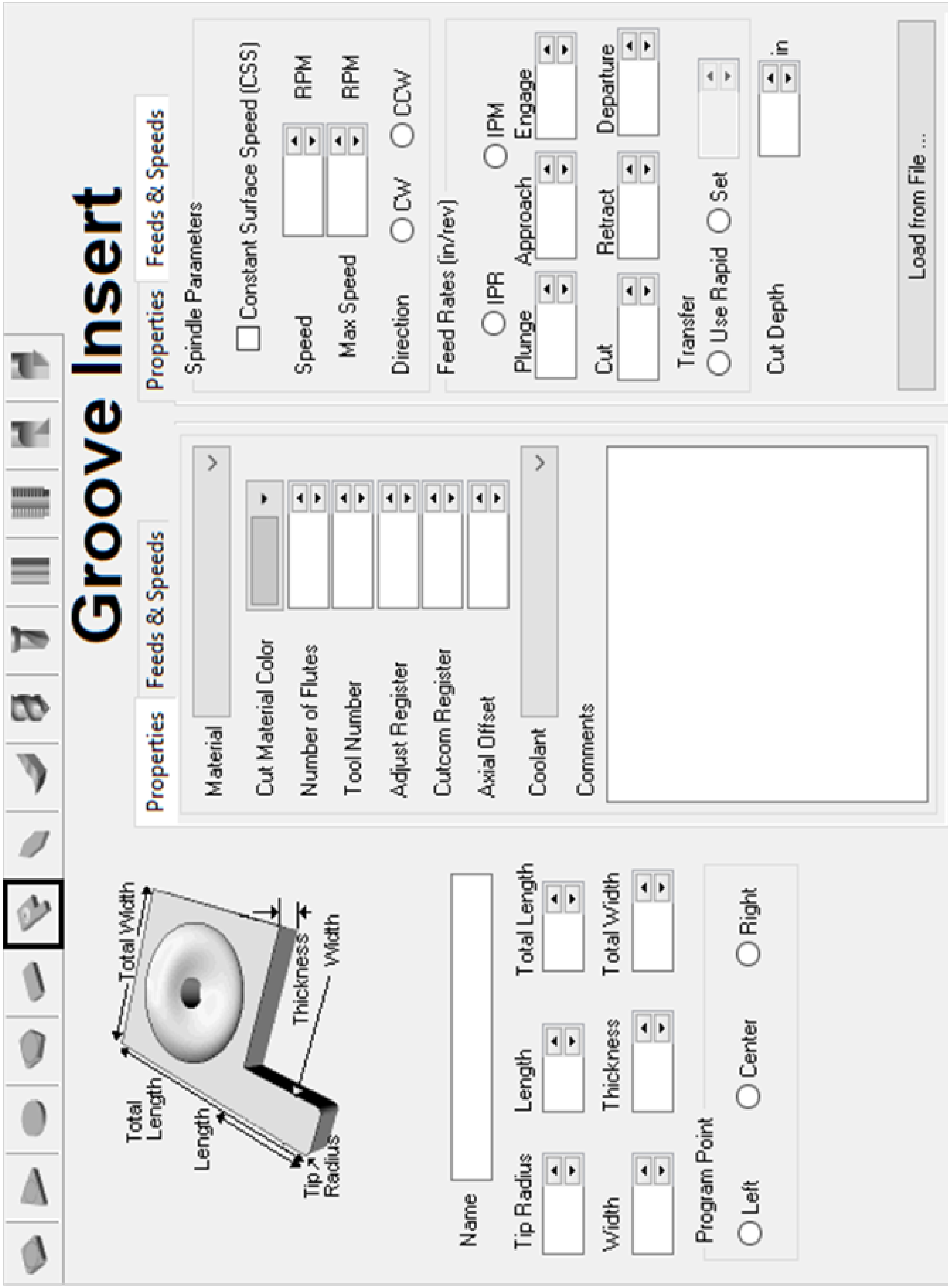
8.4.4 Trigon Insert Worksheet



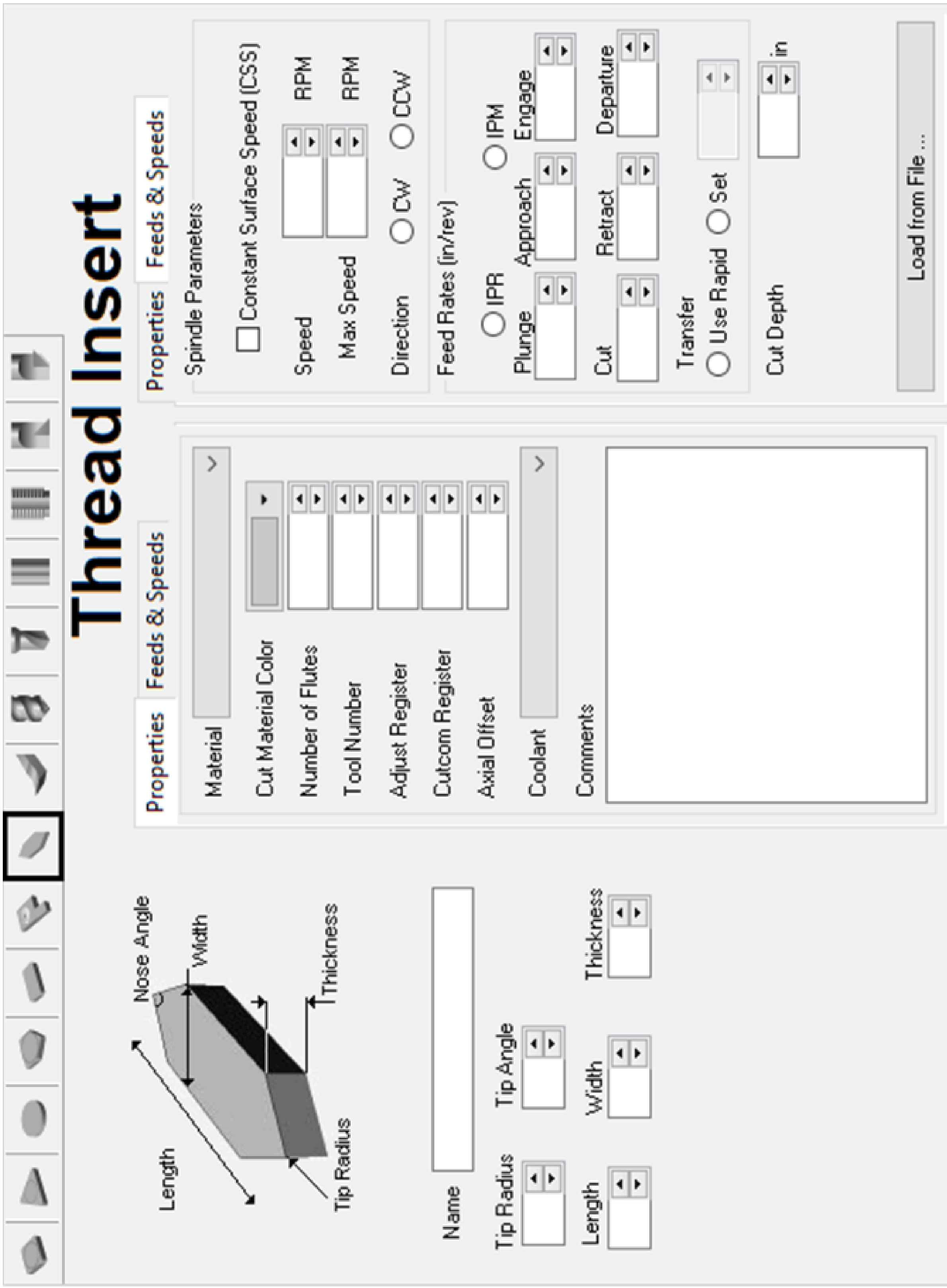
8.4.5 Parallelogram Insert Worksheet



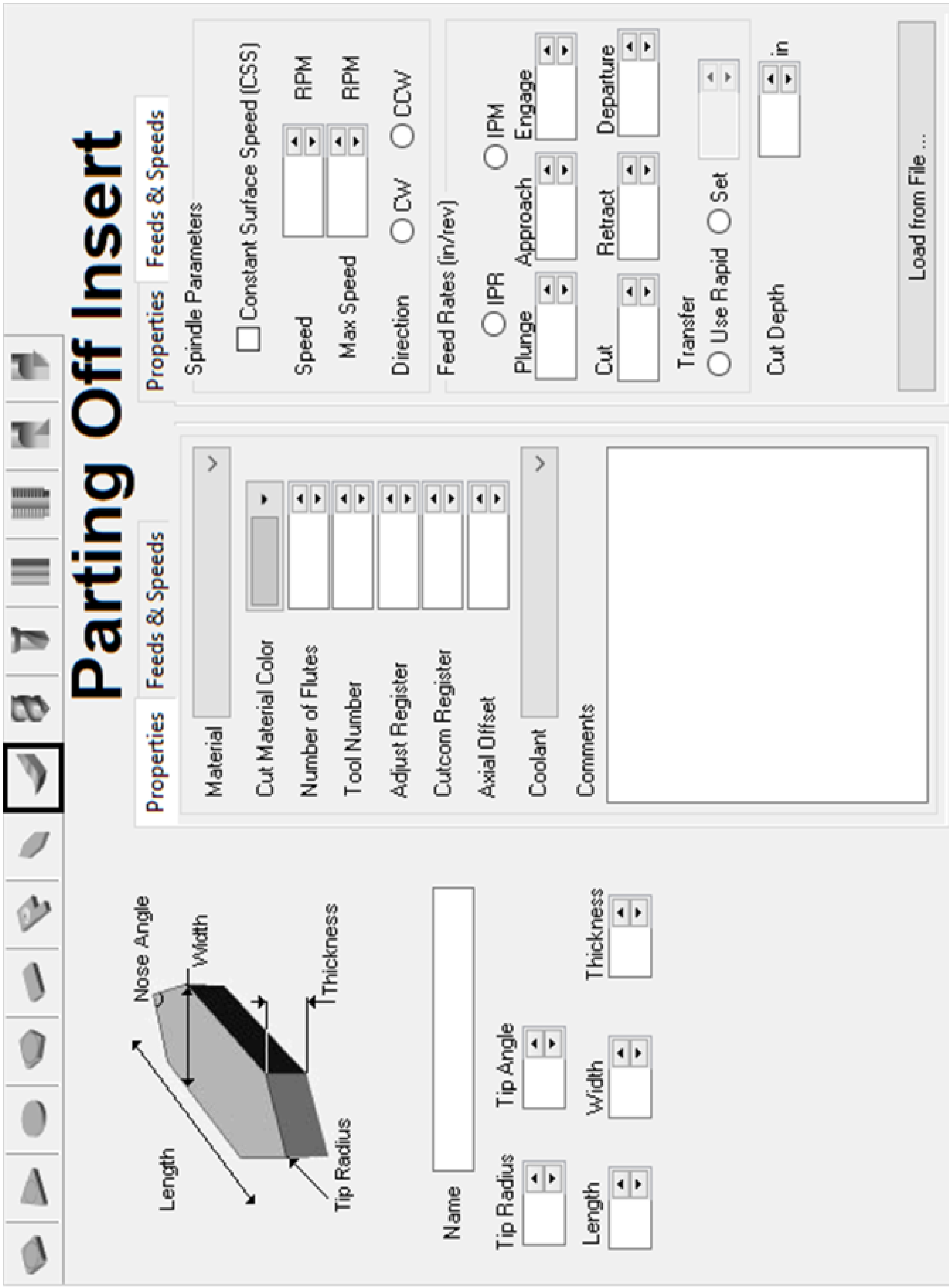
8.4.6 Groove Insert Worksheet



8.4.7 Thread Insert Worksheet



8.4.8 Part Off Insert Worksheet



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
	[illegible]	[illegible]	[illegible]	[illegible]

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


```
<Version>1.0</Version>
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>ALUMINUM - 2026</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>
```


© 2026 MecSoft Corporation

```
  <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 - 2026</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)	Mill	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)	Mill	0.375 in	0 in	0 deg	1.15 in	3.05 in	0	Carbide	24446	14.67	0	0
.375 4FL (.950 Loc)	Mill	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)	Mill	0.375 in	0 in	0 deg	1.18 in	3.04 in	0	Carbide	24446	14.67	0	0
.375 2FL (.900 Loc)	Mill	0.375 in	0 in	0 deg	0.9 in	2.53 in	0	Carbide	24446	14.67	0	0
1/8-Bore Faker	Mill	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...	Mill	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)	Mill	0.0625 in	0 in	0 deg	0.25 in	1.5 in	0	Carbide	24446	14.67	0	0
.750 Wood Roughter	Mill	0.75 in	0 in	0 deg	0.75 in	2 in	0	HSS	24446	14.67	0	0
.250 2 FL (.800 Loc)	Mill	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)	Mill	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)	Mill	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/...	Mill	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)	Mill	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)	Mill	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)	Mill	0.187 in	0 in	0 deg	0.65 in	2 in	0	Carbide	24446	14.67	0	0
.187 2 FL (.650 Loc)	Mill	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...	Mill	0.125 in	0 in	0 deg	0.74 in	2.15 in	0	HSS	24446	14.67	0	0
.373 Wood Roughter (.7...	Mill	0.373 in	0 in	0 deg	0.75 in	2 in	0	HSS	24446	14.67	0	0
.125 4FL (750 Loc)	Mill	0.125 in	0 in	0 deg	0.75 in	2.3 in	0	Carbide	24446	14.67	0	0
.125 2FL (750 Loc)	Mill	0.125 in	0 in	0 deg	0.75 in	2.3 in	0	Carbide	24446	14.67	0	0
.125 4FL (.520 Loc)	Mill	0.125 in	0 in	0 deg	0.52 in	2.5 in	0	Carbide	24446	14.67	0	0
.125 2FL (.520 Loc)	Mill	0.125 in	0 in	0 deg	0.52 in	2.5 in	0	Carbide	24446	14.67	0	0
.250 Wood Roughter (1.0...	Mill	0.25 in	0 in	0 deg	1.04 in	2.25 in	0	HSS	24446	14.67	0	0
.312 Wood Roughter (.7...	Mill	0.312 in	0 in	0 deg	0.76 in	2 in	0	HSS	24446	14.67	0	0
.250 Wood Roughter (.7...	Mill	0.25 in	0 in	0 deg	0.77 in	2 in	0	Carbide	24446	14.67	0	0
.375 2 FL (.520 Loc)	Mill	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)	Mill	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)	Mill	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)	Mill	0.1875 in	0.09375 in	0 deg	0.65 in	2 in	0	Carbide	24446	14.67	0	0
.250 2 FL Wood Roughter	Mill	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 Roughter	Mill	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 ...	Mill	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	Cutcom R
.250 4 FL (.750 Loc)	Mill	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	Mill	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)	Mill	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)	Mill	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)	Mill	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)	Mill	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	Mill	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	Mill	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)	Mill	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)	Mill	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)	Mill	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)	Mill	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...	Mill	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/...	Mill	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/...	Mill	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/...	Mill	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/...	Mill	0.028 in	0.014 in	10 deg	0.028 in	1.75 in	0	HSS	24446	14.67	0	0
.030 Engraver	Mill	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

© 2026 MecSoft Corporation

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	Mill	4 mm	0 mm	0 deg	10 mm	14 mm	0	HSS	25872	87.62	0	0
30mm End Mill	Mill	30 mm	0 mm	0 deg	75 mm	105 mm	0	HSS	25872	87.62	0	0
25mm End Mill	Mill	25 mm	0 mm	0 deg	62.5 mm	87.5 mm	0	HSS	25872	87.62	0	0
20mm End Mill	Mill	20 mm	0 mm	0 deg	50 mm	70 mm	0	HSS	25872	87.62	0	0
16mm End Mill	Mill	16 mm	0 mm	0 deg	40 mm	56 mm	0	HSS	25872	87.62	0	0
12mm End Mill	Mill	12 mm	0 mm	0 deg	30 mm	42 mm	0	HSS	25872	87.62	0	0
10mm End Mill	Mill	10 mm	0 mm	0 deg	25 mm	35 mm	0	HSS	25872	87.62	0	0
8mm End Mill	Mill	8 mm	0 mm	0 deg	20 mm	28 mm	0	HSS	25872	87.62	0	0
6mm End Mill	Mill	6 mm	0 mm	0 deg	15 mm	21 mm	0	HSS	25872	87.62	0	0
5mm End Mill	Mill	5 mm	0 mm	0 deg	12.5 mm	17.5 mm	0	HSS	25872	87.62	0	0
3mm End Mill	Mill	3 mm	0 mm	0 deg	7.5 mm	10.5 mm	0	HSS	25872	87.62	0	0
2mm End Mill	Mill	2 mm	0 mm	0 deg	5 mm	7 mm	0	HSS	25872	87.62	0	0
1.5mm End Mill	Mill	1.5 mm	0 mm	0 deg	3 mm	5 mm	0	HSS	25872	87.62	0	0
1mm End Mill	Mill	1 mm	0 mm	0 deg	2.5 mm	3.5 mm	0	HSS	25872	87.62	0	0
0.8mm End Mill	Mill	0.8 mm	0 mm	0 deg	1.5 mm	3 mm	0	HSS	25872	87.62	0	0
0.6mm End Mill	Mill	0.6 mm	0 mm	0 deg	1.5 mm	3 mm	0	HSS	25872	87.62	0	0
0.5mm End Mill	Mill	0.5 mm	0 mm	0 deg	1.5 mm	3 mm	0	HSS	25872	87.62	0	0
50mm End Mill	Mill	50 mm	0 mm	0 deg	100 mm	130 mm	0	HSS	25872	87.62	0	0
25mm Ballnose	Mill	25 mm	12.5 mm	0 deg	62.5 mm	100 mm	0	HSS	25872	87.62	0	0
20mm Ballnose	Mill	20 mm	10 mm	0 deg	50 mm	80 mm	0	HSS	25872	87.62	0	0
18mm Ballnose	Mill	18 mm	9 mm	0 deg	45 mm	72 mm	0	HSS	25872	87.62	0	0
16mm Ballnose	Mill	16 mm	8 mm	0 deg	40 mm	64 mm	0	HSS	25872	87.62	0	0
12mm Ballnose	Mill	12 mm	6 mm	0 deg	30 mm	48 mm	0	HSS	25872	87.62	0	0
10mm Ballnose	Mill	10 mm	5 mm	0 deg	25 mm	40 mm	0	HSS	25872	87.62	0	0
8mm Ballnose	Mill	8 mm	4 mm	0 deg	20 mm	32 mm	0	HSS	25872	87.62	0	0
6mm Ballnose	Mill	6 mm	3 mm	0 deg	15 mm	24 mm	0	HSS	25872	87.62	0	0
0.5mm Ballnose	Mill	0.5 mm	0.25 mm	0 deg	1.5 mm	5 mm	0	HSS	25872	87.62	0	0
5mm Ballnose	Mill	5 mm	2.5 mm	0 deg	12.5 mm	20 mm	0	HSS	25872	87.62	0	0
4mm Ballnose	Mill	4 mm	2 mm	0 deg	10 mm	16 mm	0	HSS	25872	87.62	0	0
3mm Ballnose	Mill	3 mm	1.5 mm	0 deg	7.5 mm	12 mm	0	HSS	25872	87.62	0	0
2mm Ballnose	Mill	2 mm	1 mm	0 deg	5 mm	8 mm	0	HSS	25872	87.62	0	0

© 2026 MecSoft Corporation

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	Mill	1.5 mm	0.75 mm	0 deg	3 mm	5 mm	0	HSS	25872	87.62	0	0
1.2mm Ballnose	Mill	1.2 mm	0.6 mm	0 deg	4 mm	10 mm	0	HSS	25872	87.62	0	0
1mm Ballnose	Mill	1 mm	0.5 mm	0 deg	2.5 mm	5 mm	0	HSS	25872	87.62	0	0
0.8mm Ballnose	Mill	0.8 mm	0.4 mm	0 deg	2 mm	5 mm	0	HSS	25872	87.62	0	0
0.6mm Ballnose	Mill	0.6 mm	0.3 mm	0 deg	1.5 mm	5 mm	0	HSS	25872	87.62	0	0
30mm Ballnose	Mill	30 mm	15 mm	0 deg	75 mm	120 mm	0	HSS	25872	87.62	0	0
6mm R 0.5	Mill	6 mm	0.5 mm	0 deg	12.5 mm	17.5 mm	0	HSS	25872	87.62	0	0
6mm R 1.0	Mill	6 mm	1 mm	0 deg	10 mm	20 mm	0	HSS	25872	87.62	0	0
8mm R 1.0	Mill	8 mm	1 mm	0 deg	7 mm	25 mm	0	HSS	25872	87.62	0	0
8mm R 1.5	Mill	8 mm	1.5 mm	0 deg	7 mm	25 mm	0	HSS	25872	87.62	0	0
8mm R 2.0	Mill	8 mm	2 mm	0 deg	7 mm	25 mm	0	HSS	25872	87.62	0	0
10mm R 1.5	Mill	10 mm	1.5 mm	0 deg	25 mm	35 mm	0	HSS	25872	87.62	0	0
10mm R 2.0	Mill	10 mm	2 mm	0 deg	25 mm	35 mm	0	HSS	25872	87.62	0	0
12mm R 2.0	Mill	12 mm	2 mm	0 deg	25 mm	35 mm	0	HSS	25872	87.62	0	0
16mm R 2.0	Mill	16 mm	2 mm	0 deg	16 mm	50 mm	0	HSS	25872	87.62	0	0
50mm Surfacer	Mill	58 mm	4 mm	0 deg	8 mm	40 mm	0	HSS	25872	87.62	0	0
63mm Surfacer	Mill	71 mm	4 mm	0 deg	8 mm	50 mm	0	HSS	25872	87.62	0	0
80mm Surfacer	Mill	88 mm	4 mm	0 deg	8 mm	50 mm	0	HSS	25872	87.62	0	0
63 Alesa Surfacer	Mill	75 mm	6 mm	0 deg	6 mm	100 mm	0	HSS	25872	87.62	0	0
17.5º Eng R 0.1	Mill	0.2 mm	0.1 mm	17.5 deg	4.5 mm	10 mm	0	HSS	25872	87.62	0	0
22º Eng R 0.1	Mill	0.2 mm	0.1 mm	22 deg	5.1 mm	10 mm	0	HSS	25872	87.62	0	0
17.5º Eng R 0.05	Mill	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

© 2026 MecSoft Corporation

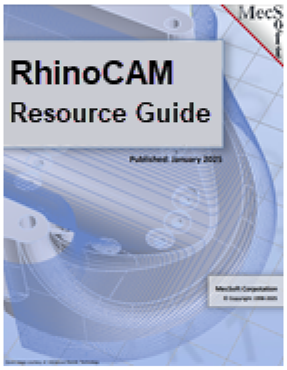
Name	Tool Type	Diameter	Corner Radius	Taper	Flute Length	Tool Length	Tool #	Tool Material	Spindle RPM	Cut Feed	Adjust R	Cutcom R
28.0mm HSS Drill	Drill	28 mm	0 mm	0 deg	70 mm	364 mm	0	HSS	15523	63.08	0	0
30.0mm HSS Drill	Drill	30 mm	0 mm	0 deg	75 mm	390 mm	0	HSS	15523	63.08	0	0
12 mm 90º CDrill	Drill	12 mm	0 mm	0 deg	20 mm	30 mm	0	HSS	15523	63.08	0	0
10 mm 90º CDrill	Drill	10 mm	0 mm	0 deg	20 mm	30 mm	0	HSS	15523	63.08	0	0
8mm 120º CDrill	Drill	8 mm	0 mm	0 deg	10 mm	30 mm	0	HSS	15523	63.08	0	0
8mm 90º CDrill	Drill	8 mm	0 mm	0 deg	15 mm	25 mm	0	HSS	15523	63.08	0	0
6mm 90º CDrill	Drill	6 mm	0 mm	0 deg	10 mm	20 mm	0	HSS	15523	63.08	0	0
5mm 90º CDrill	Drill	5 mm	0 mm	0 deg	10 mm	62 mm	0	HSS	15523	63.08	0	0
4mm 90º CDrill	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.

2026 RhinoCAM Resource Guide



The 2026 RhinoCAM Resource Guide!

18 Pages

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

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

Index

© 2026 MecSoft Corporation

- U -