

Post-Process (PPG) Reference Guide

VisualCAM-PPG 2025

Published: February 2025

N049 G0

N050 G1Z-0. F29.334

N051 G02Z-0.125I-0.4375J-0.0002 F14.667

N052 G02Z-0.25I-0.4375J-0.0002

N053 G02X-0.4375Y-0.0002I-0.4375J-0.0002

N054 G02X0.4375Y0.0002I0.4375J0.0002

N055 G0Z0.25

N056 (Deep Drill)

N057 (Tool Diameter = 0.25 Leng

N058 G20 T3 M6

N059 G54

N060 S190 M3

N061 G90G0X0.75Y-0.75

MecSoft Corporation

© Copyright 1998-2025

Table of Contents

Welcome	3
Quick Start	4
Resource Guide	7
Good Things to Know	8
Use Legacy Post	20
1 Dialogs.....	23
Post Processor File Browser	23
PPG Editor	24
Variable List Dialog	26
2 PPG Editor.....	27
General	27
StartEnd	31
Tool Change	32
Setup	35
Spindle	36
Feed Rate	38
Motion	41
Circle	43
Helical/Spiral	45
Multi Axis Motion	47
Cutter Compensation	49
Cut Motion Start/End	50
Cycles	52
Miscellaneous	54
Variables	56
3 Macros.....	74
Macros	74
How to edit Macros	75
Use Programmable Post	76
Find More Resources	77
Good Things to Know	78
Index	90

Welcome



PPG Module 2025

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

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

The [VisualCAD/CAM Post-Processor Generator](#) is used to edit post processor files (SPM Files). These files are used by [VisualCAD/CAM](#) during toolpath post-processing. [VisualCAD/CAM](#) reads in a user specified [SPM](#) file, each file corresponding to a single machine tool controller, and generates the post-processed output using the rules resident in these files. Users have the ability to edit these files to modify these rules, thereby controlling the output that [VisualCAD/CAM](#) generates.

Using the [VisualCAD/CAM Post-Processor Generator](#), these [SPM](#) files can be edited by following these steps:

First choose the required [SPM](#) file to edit from the [Post Processor File Browser](#). After selecting the file, it can be edited using the [Editor](#) dialog. The format of various output blocks, such as motion, feed rates, spindle etc., can be edited by selecting the appropriate tabs in this dialog.

In addition to predefined block definitions, you can add startup codes as well as termination codes specific to the controller and shop practices. These blocks can be user-defined statements that may contain built in variables.



Related Topics

[Good Things to Know](#)

[Post Processor File Browser](#)

[Main Editor](#)

[Variable List Dialog](#)

Quick Start



PPG Module 2025

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

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

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



How to Access the Quick Start Guide Documents

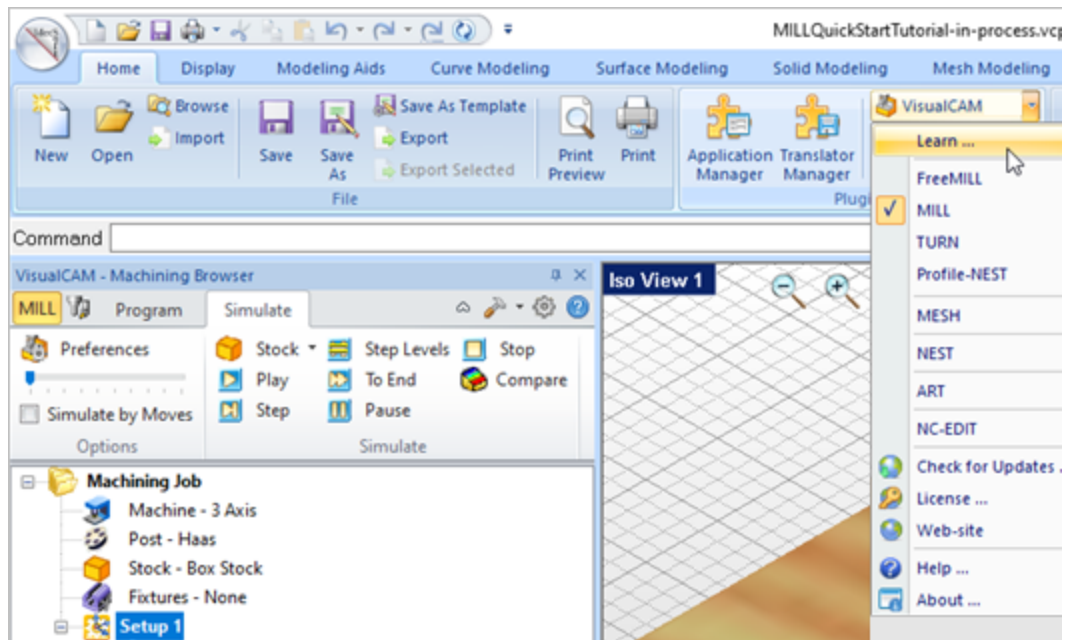
To help you quickly get started in working with each module, select one of the Help buttons located on the [VisualCAD/CAM Learning Resources](#) dialog.

You will find:

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

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

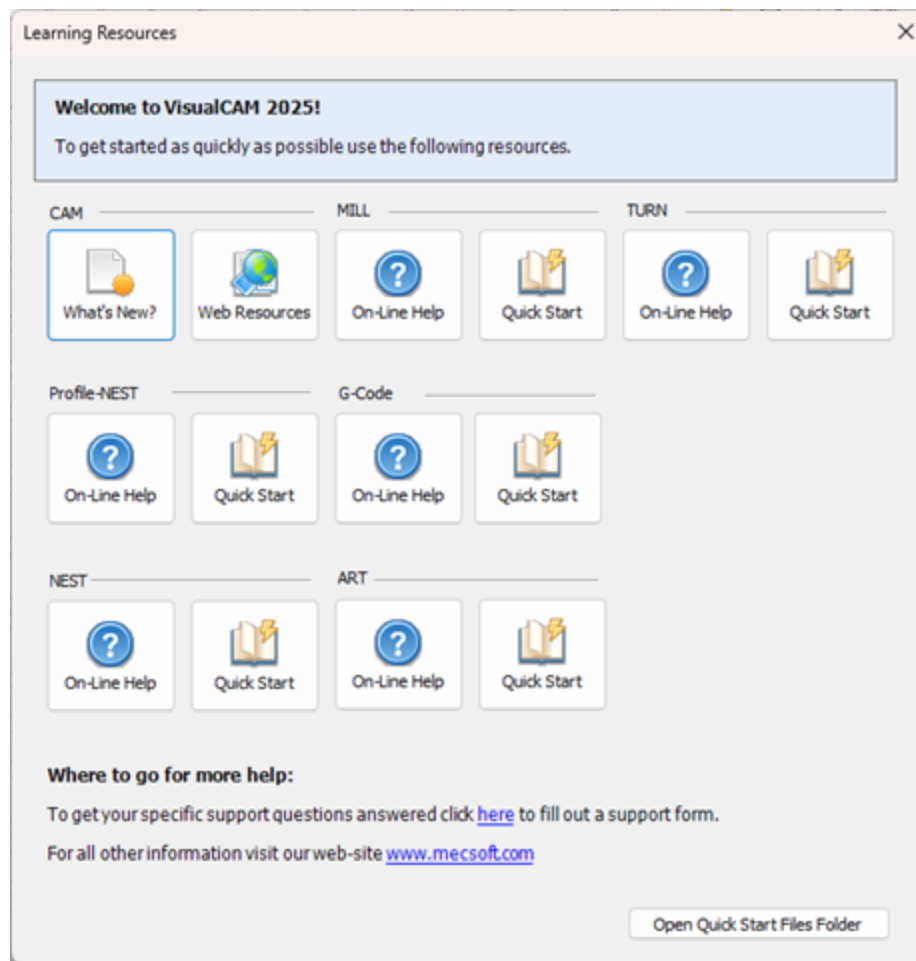
1. From the [VisualCAD Home Ribbon Bar](#), drop down the Main menu and select [Learn ...](#)



To access the Learning Resources dialog in VisualCAM

2. Select a document from the [Learning Resources](#) dialog to get started using the module of your choice.

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



Learning Resources Dialog



Related Topics

[Find More Resources](#)

Resource Guide

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



2025 VisualCAD/CAM Resource Guide



The 2025 VisualCAD/CAM Resource Guide!

18 Pages

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

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

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

Good Things to Know

Here is a list of things you should know when posting G-Code using a customized post created from the [Post-Processor Generator](#) in [VisualCAD/CAM](#).



Algebraic Expressions when Posting

The [Post Process Generator](#) supports [Algebraic Expressions](#) in all input fields.

Here are some guidelines for using expressions:

1. Each expression should be placed in 'E{', 'E}' tags.
2. In expression can be used next operations: -, +, /, *, ^
3. Negative values should be placed in parentheses '()'
4. Expression parts can be placed in parentheses '()'
5. Floating point numbers should be delimited by point symbol, use 0.xx in case of fractional numbers
6. Expressions can contain spaces in any place, spaces will be removed while parsing
7. Numbers in [-0.9; 0.9] can be written as [-.9; .9]

Examples:

- E{ ([SOME_VAR1]/2 + ([SOME_VAR]*(-3.2)))^3 E}
- E{[SOME_VAR1] + .3 E} SOME_TEXT E{ [SOME_VAR1] *(-1) E}



Posting Drill Cycles & Indexed Machining

Drill cycles will be converted to simulated cycles (i.e., using linear motions) if the setup the drill cycles appear in is not aligned with the machine Z axis. This is done only when the machine has a head configuration defined and [Output all coordinates in local Setup Coordinate System](#) is not checked. See [Machine Tool Setup](#) for more information.



Posting Cutter Compensation (G40, G41, G42)

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 [VisualCAD/CAM](#). 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 [VisualCAD/CAM](#).

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

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

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

Cutter Compensation Left

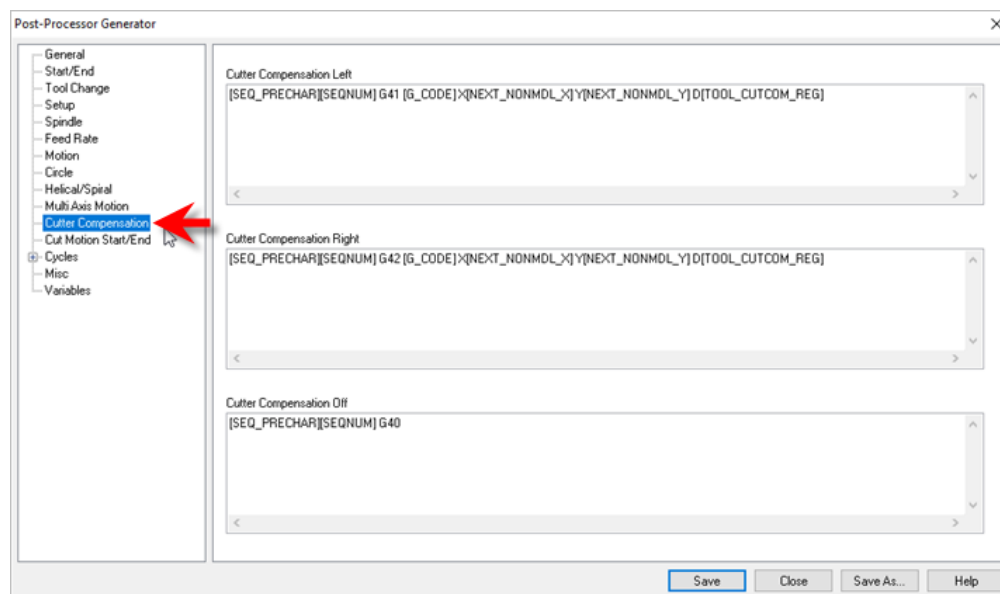
```
[SEQ_PRECHAR][SEQNUM] G41 [G_CODE] X[NEXT_NONMDL_X] Y[NEXT_NONMDL_Y]
D[TOOL_CUTCOM_REG]
```

Cutter Compensation Right

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

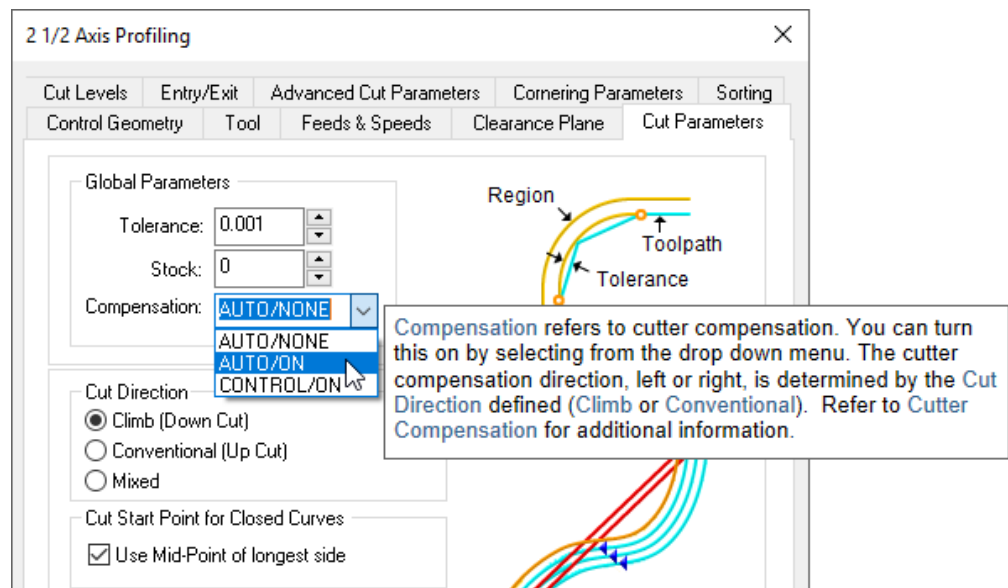
Cutter Compensation Off

```
[SEQ_PRECHAR][SEQNUM] G40
```



A few things to watch out for:

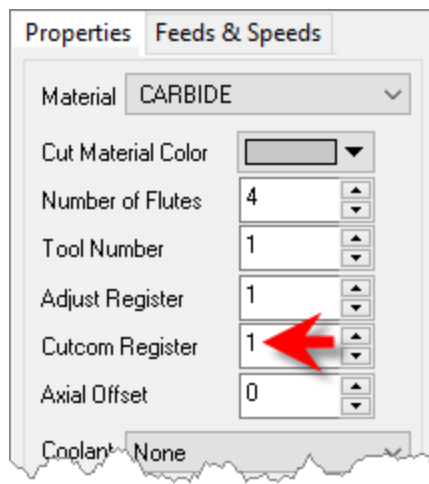
1. [Cutter compensation](#) makes sense only in 2-1/2 axis operations. If you are using roughing (pocketing & facing) the compensation will be turned on only in the final passes.
2. Make sure you are using [Climb](#) or [Conventional](#) cut traversal in any of the methods that you want to turn compensation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



3. Make sure you have a linear motion for the controller to turn on the compensation for. If your first motion is an arc the controller will not be able to turn on the compensation. Thus, in **2-1/2 axis profiling**, make sure there is a linear entry motion for the controller to be able to turn compensation on & linear exit to turn off compensation.

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

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



Posting 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:

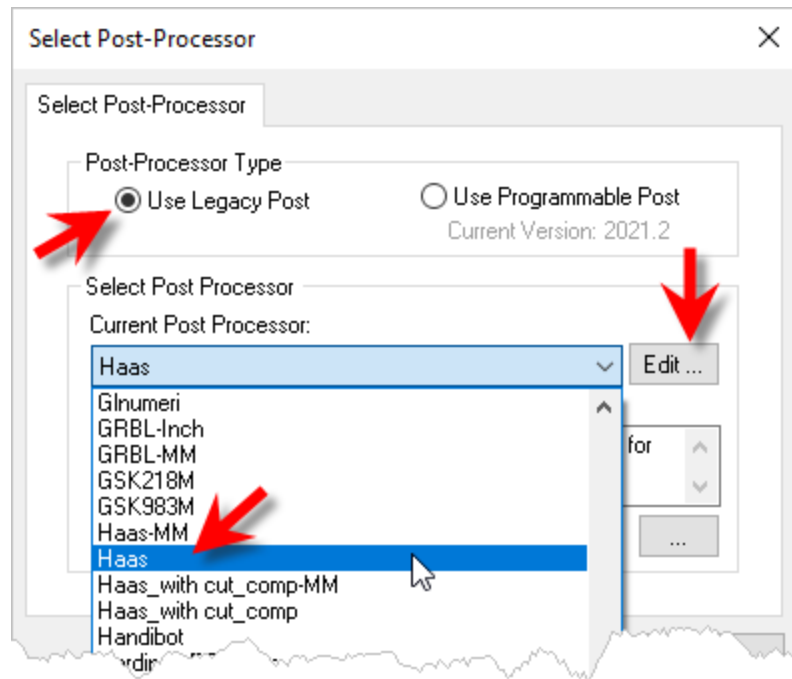


For 2 and 3 Axis Output

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:

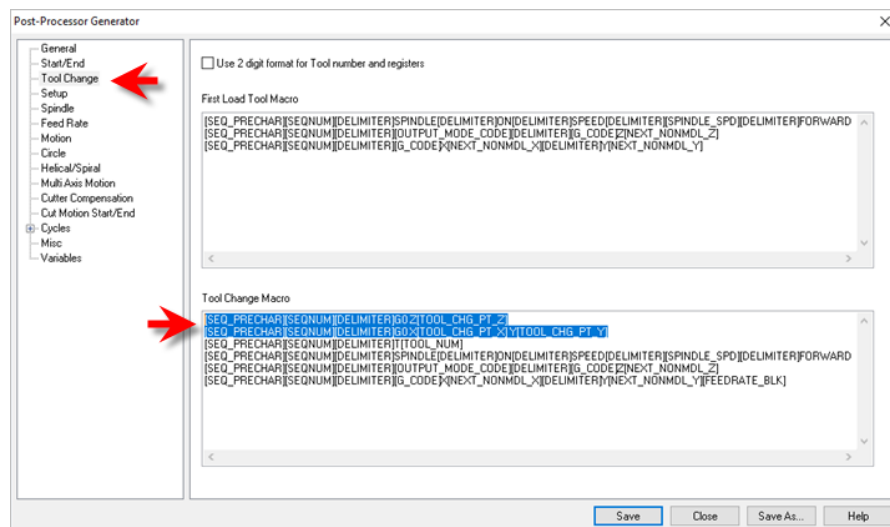
X: -4, Y: 0 Z: 0

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.

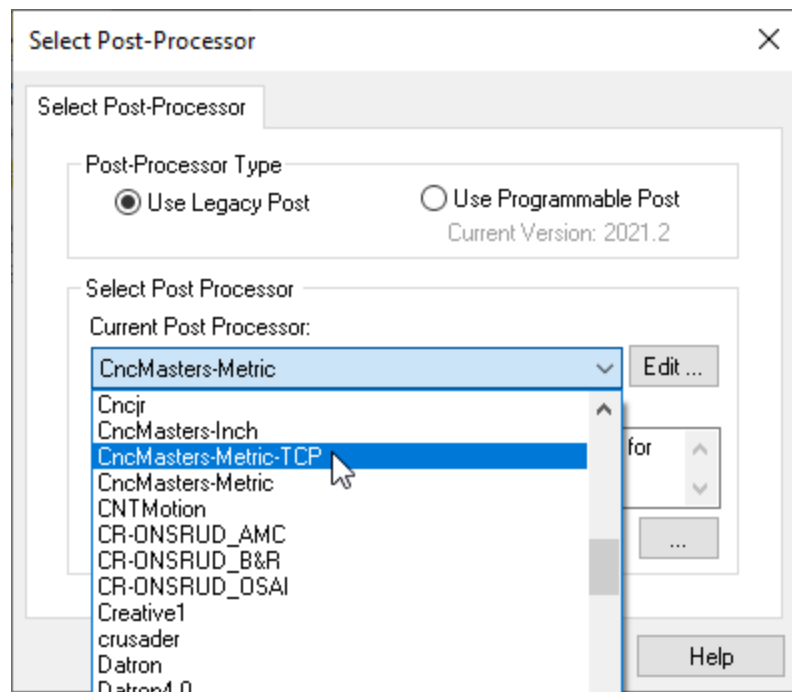
```
[SEQ_PRECHAR][SEQNUM][DELIMITER]G0 Z[T[TOOL_CHG_PT_Z]
[SEQ_PRECHAR][SEQNUM][DELIMITER]G0 X[T[TOOL_CHG_PT_X] Y[T[TOOL_CHG_PT_Y]
[SEQ_PRECHAR][SEQNUM][DELIMITER]T[T[TOOL_NUM]
...
...
```



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
...
...

```

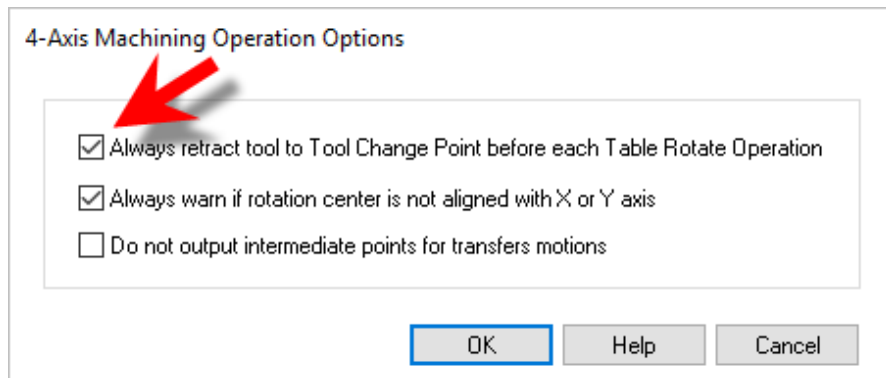
15. That's it!



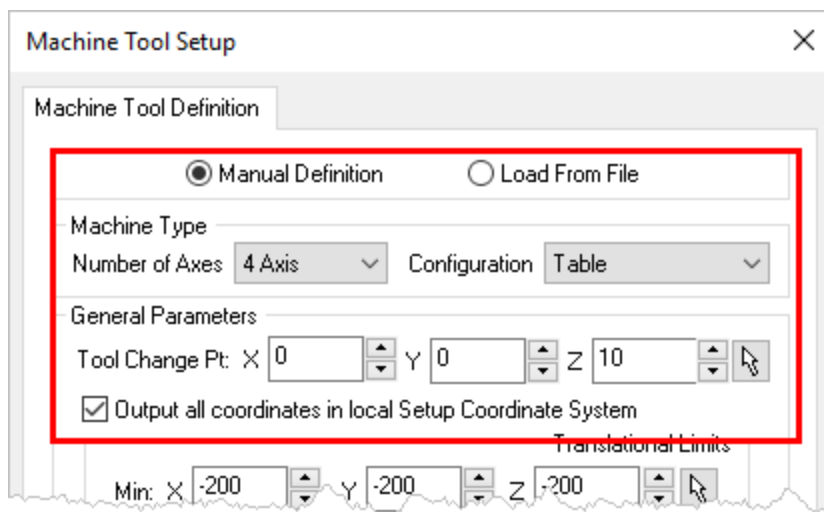
For 4 Axis Output

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.

...
...

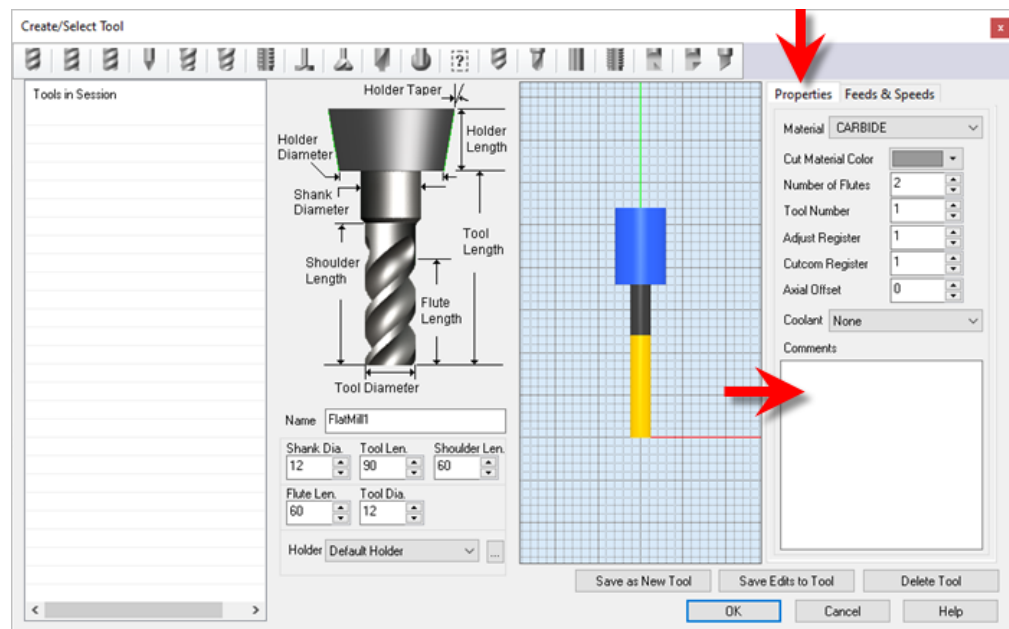


Posting Tool Comments

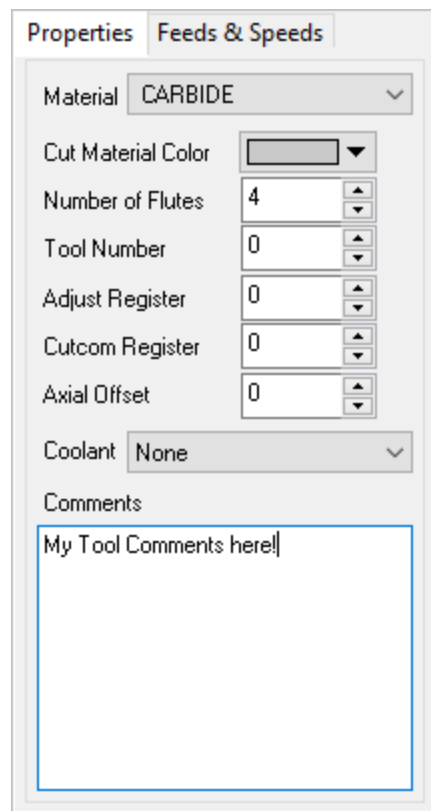
You can add comments associated with a **Tool**. These **Comments** are saved with the **Tool** in your **Tool Library**. They are also posted to your g-code when the tool is used.

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.

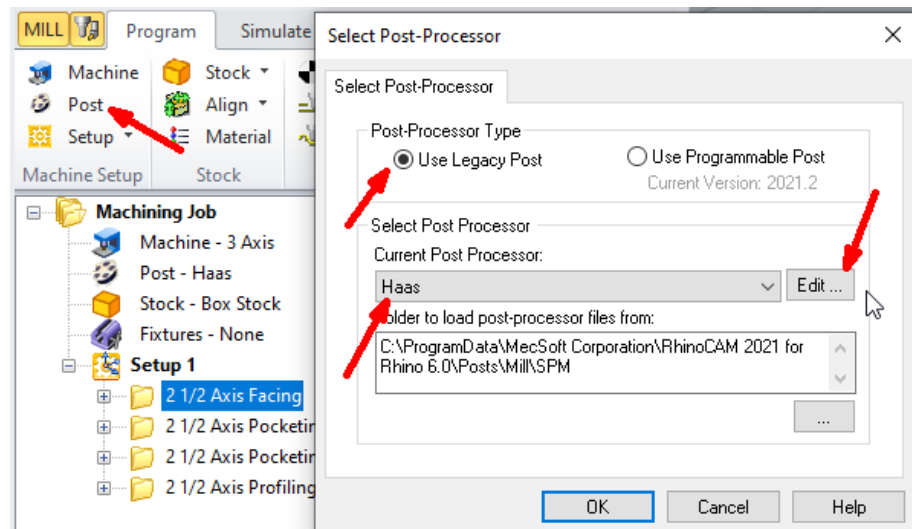


3. Add text to the **Comments** window.



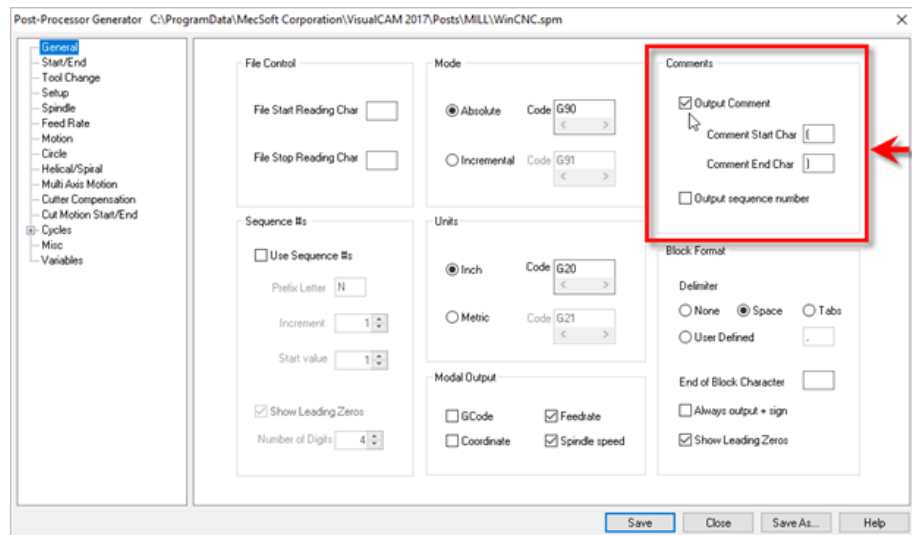
4. Make sure [Comments](#) are enabled in your post.

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.

Properties **Feeds & Speeds**

Material: CARBIDE

Cut Material Color: [Grey Swatch]

Number of Flutes: 4

Tool Number: 0

Adjust Register: 0

Cutcom Register: 0

Axial Offset: 0

Coolant: None

Comments: \$My g-code Here!

...

...

G1 X0.4655 Y-0.7171 Z0.7480

G1 X0.5301 Y-0.7171 Z0.7480

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

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

G0 Z0.9843

G0 X0.7801 Y-0.2171

(2 1/2 Axis Profiling)

My g-code Here!

S18000

G0 Z0.9843

G0 X0.5301 Y-0.7097

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

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

G1 X0.5873 Y-0.6345 Z0.7480

...

...



Related Topics

[Post Processor File Browser](#)

[Main Editor](#)

[Variable List Dialog](#)

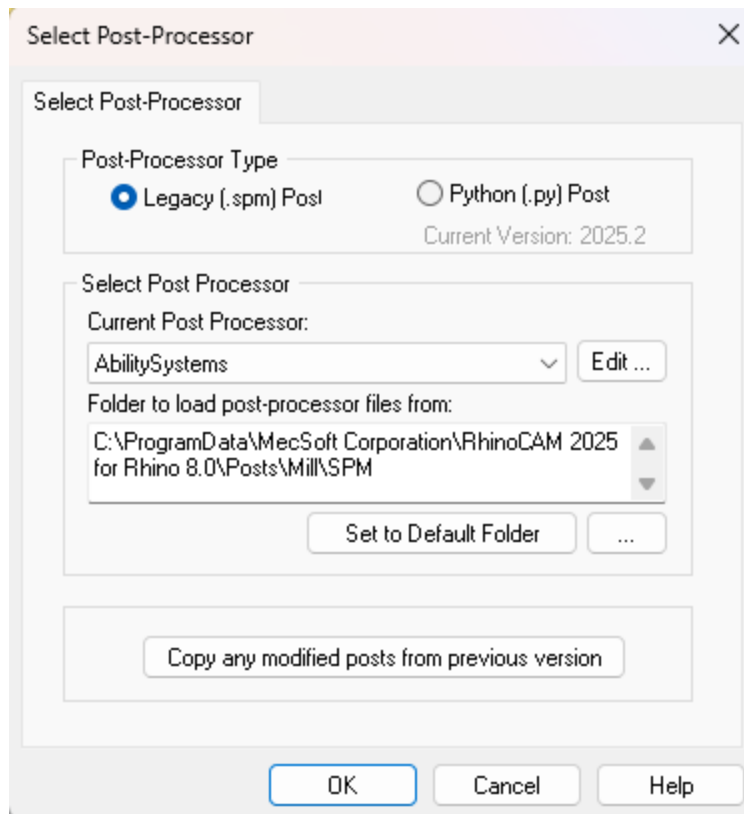
Use Legacy Post



Select this option from the Post-Processor Options dialog to use your *"Legacy Post"* when posting toolpath operations. *"Legacy Post"* refers to your post definition (*.spm) file you used prior to version 2021. This option also lists the over 300 existing post-processors. See [Current Post Processor](#) below.



Dialog Box: Set Post-Processor Options



Dialog Box: Set Post-Processor Options



Post Processor Type

Use these options to define the post type to use when posting toolpath operations. Each option is documented further in the [Post-processor Generator On-Line Help](#).

Legacy (.spm) Post

Select this option to use your *"Legacy Post"* when posting toolpath operations. *"Legacy Post"* refers to your post definition (*.spm) file you used prior to this current release. This option also lists the over 300 existing post-processors. See [Current Post Processor](#) below. Refer to the [PPG Online Help](#), [Reference Guide](#) and [PPG Decoded Guide](#) for help understanding the [Legacy Post Definition](#) files.

Python (.py) Post Post

Select this option to use your "*Programmable Post*" (PPPG) when posting toolpath operations. "*Programmable Post*" refers to your programmable post created with our version 2021 and newer CAM plugins. This option will list only one post processor selection called [PostModifier](#). This post definition is defined using the [Programmable Post API](#). The current version of the PPPG is shown in the dialog. Refer to the [PPG Online Help, Reference Guide](#) and [PPG Decoded Guide](#) for help understanding the [Programmable Post](#) files.



Select Post Processor

Current Post Processor

You can change the current post processor by selecting a post from the list of available post processors under [Current Post Processor](#). If [Use Legacy Post](#) is selected, the [Current Post Processor](#) list will contain MecSoft's standard list of over 300 post processors. If [Use Programmable Post](#) is selected, this will list only one post processor selection called [PostModifier](#). This post definition is defined using the [Programmable Post API](#). Refer to the [Programmable Post API](#) documentation for more about the API.



Current Post Processor

You can change the default post processor by selecting a post from the list of available post processors under [Current Post Processor](#). If [Use Legacy Post](#) is selected, the [Current Post Processor](#) list will contain MecSoft's standard list of over 300 post processors. If [Use Programmable Post](#) is selected, this will list only one post processor selection called [PostModifier](#). This post definition is defined using the [Programmable Post API](#). Refer to the [Programmable Post API](#) documentation for more about the API.



Folder to load post-processor files from

The [MILL](#) module uses macro files with a *.spm* file extension to handle post-processing to different controllers. These files are typically located in the "[Posts](#)" directory under the installation folder(*C:\ProgramData\MecSoft Corporation\Posts\Mill*).

The [MILL](#) module by default looks in this directory to build the list of available post-processors shown under the [Current Post](#).

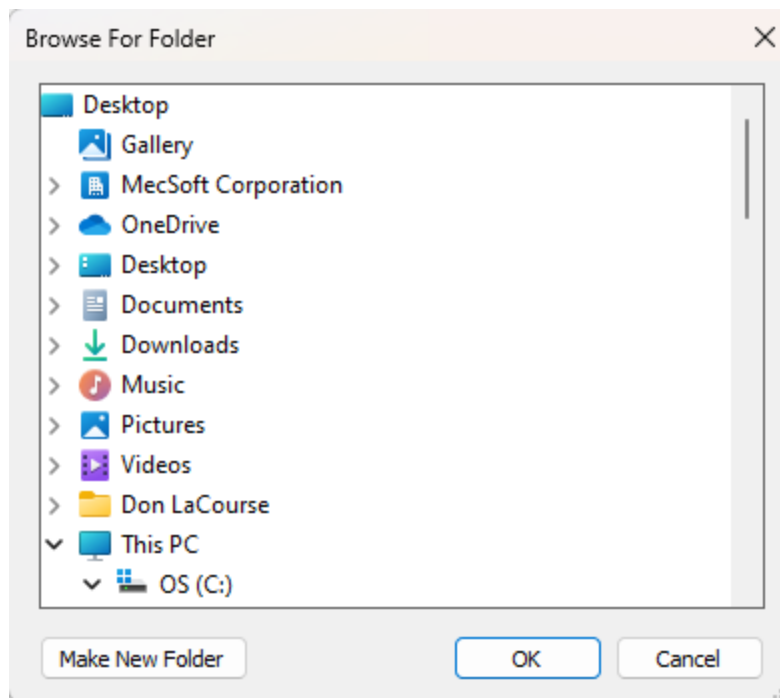
Set to Default Folder

Select this button to change the posts location folder to the system default location. This folder location will be located within the current plug-in installation path.

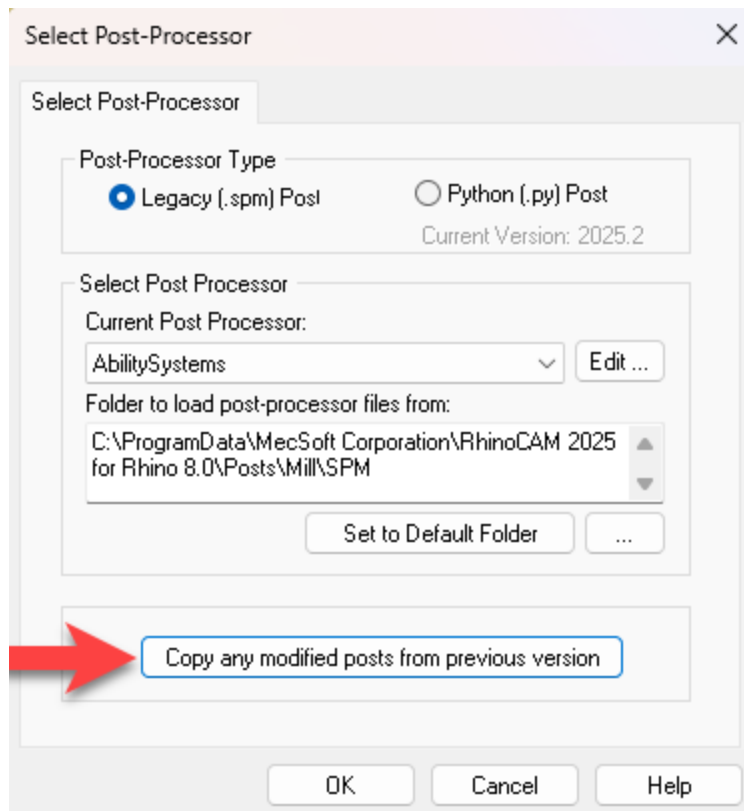
Browse for folder



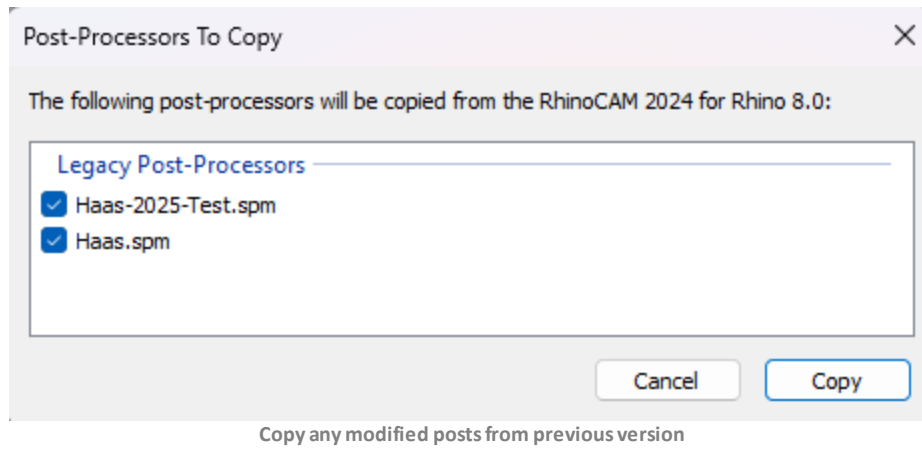
To set the posts location folder to a folder of your choice, select the ... button and then select a folder using the [Browse for Folder](#) dialog shown below.



Copy any modified posts from previous version



This button will be active when you have post definition files from the previous [RhinoCAM](#) or [VisualCAM](#) version residing in the "Folder to load post processor files from".



To use this feature:

1. First, you must have posts you your "[Folder to load post-processor files from](#)" that are new or have been revised after the previous [RhinoCAM](#) or [VisualCAM](#) version was installed.
2. Select the [Copy any modified posts from previous version](#) button.
3. The [Post-Processors To Copy](#) dialog will display listing the post that will be copied.
4. Then pick **Copy** to accept and close the dialog.
5. When you display the [Select Post-processor](#) dialog again, your previous posts will be listed for selection from the [Current post-processor](#) list.

NOTE: It is advisable that you backup all of your post-processor definition files and keep them in a safe location outside of the plug-in installation path so that you can access them manually if needed.

NOTE: This feature only looks for revised posts from the immediate previous version (i.e., one version back) and not multiple previous versions. Also, the revised posts must be newer than when the previous version was installed.

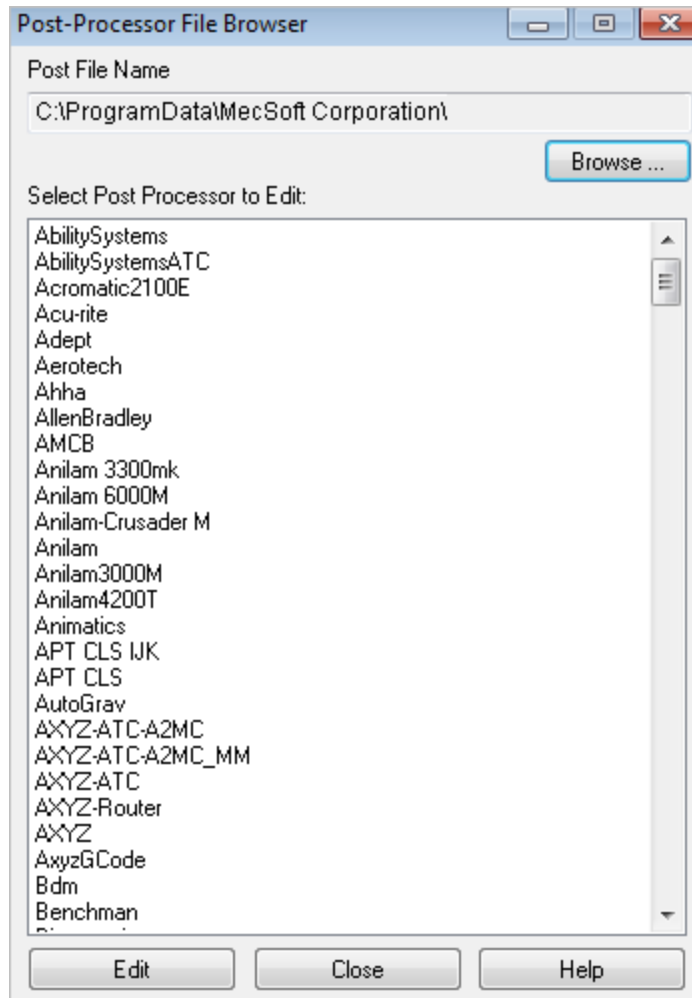
5.1 Dialogs

5.1.1 Post Processor File Browser

The dialog shown below is used to select the post processor file to be edited. (SPM File). The name and location of the post processor file can be either entered in the edit box provided, or can be selected using the browse button. You can double click on the required [SPM File](#) to invoke the [Editor](#).



Dialog Box: Post Processor File Browser



Dialog Box: Post Processor File Browser



Related Topics

[Introduction](#)

[Post Processor Editor](#)

[Variable List Dialog](#)

5.1.2 PPG Editor

The [VisualCAD/CAM Post-Processor Generator \(PPG\) Editor](#) is shown below. This [Editor](#) is divided into sections (listed on the left) allowing you to define each block type. Refer to the list of sections below.



Dialog Box: Post Processor Generator: Editor

Post-Processor Generator

General

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

File Control

Output File Extension: .nc

File Start Reading Char: %

File Stop Reading Char: %

Mode

☒ Absolute Code: G90

☐ Incremental Code: G91

Comments

☒ Output Comment

Comment Start Char: {

Comment End Char: }

☐ Output sequence number

Sequence #s

☐ Use Sequence #s

Prefix Letter: N

Increment: 10

Start value: 0

☐ Show Leading Zeros

Number of Digits: 4

Units

☐ Inch Code: G20

☒ Metric Code: G21

Block Format

Delimiter

☐ None ☒ Space ☐ Tabs

☐ User Defined

End of Block Character: }

☐ Always output + sign

☒ Show Leading Zeros

Modal Output

☐ GCode ☒ Feedrate

☒ Coordinate ☒ Spindle speed

Save Close Save As... Help

Dialog Box: Post Processor Generator: Editor



PPG Editor Sections

- [General](#)
This folder helps you set up file information, G-Code format, mode and the units of operation.
- [Start End](#)
Start and End code specifier.
- [Tool Change](#)
Load and Tool Change Macro specifier.
- [Setup](#)
Setup change and Rotate Table Setup specifier.
- [Spindle](#)
Spindle code specifier.
- [Feed Rate](#)
Feed Rate specifier.
- [Motion](#)
Motion block specifier
- [Circle](#)
Circle block specifier.
- [Helical/Spiral](#)
Helical and Spiral motion block specifier.

- [Multi Axis Motion](#)
Multi Axis Motion specifier.
- [Cutter Compensation](#)
Motion blocks for [Cutter Compensation Right](#), [Left](#) and [Off](#).
- [Cut Motion Start/End](#)
Cut Motion Start and End macro blocks.
- [Cycles](#)
Cycle G-Code and format specifier.
- [Miscellaneous](#)
Coolant and Compensation code specifier
- [Variables](#)
Lists [variables](#) and their values used in post-processing



Related Topics

[Introduction](#)

[Post Processor File Browser](#)

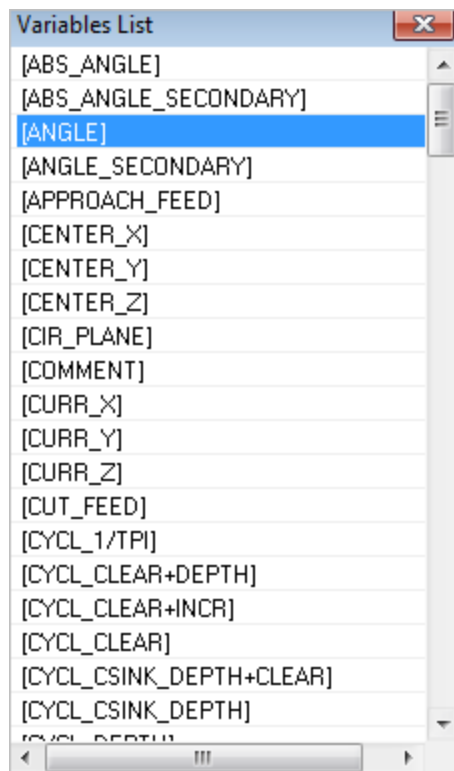
[Variable List Dialog](#)

5.1.3 Variable List Dialog

The [Variable List](#) dialog shown below can be invoked by pressing the right mouse button from within edit boxes that are used in setting up startup and termination code for a post-processor. This dialog can be used to add [variables](#) to the active edit box for [macros](#).



Dialog Box: Variable List



Dialog Box: Variable List



Related Topics

[Post Processor File Browser](#)

[Post Processor Editor](#)

[Variables List](#)

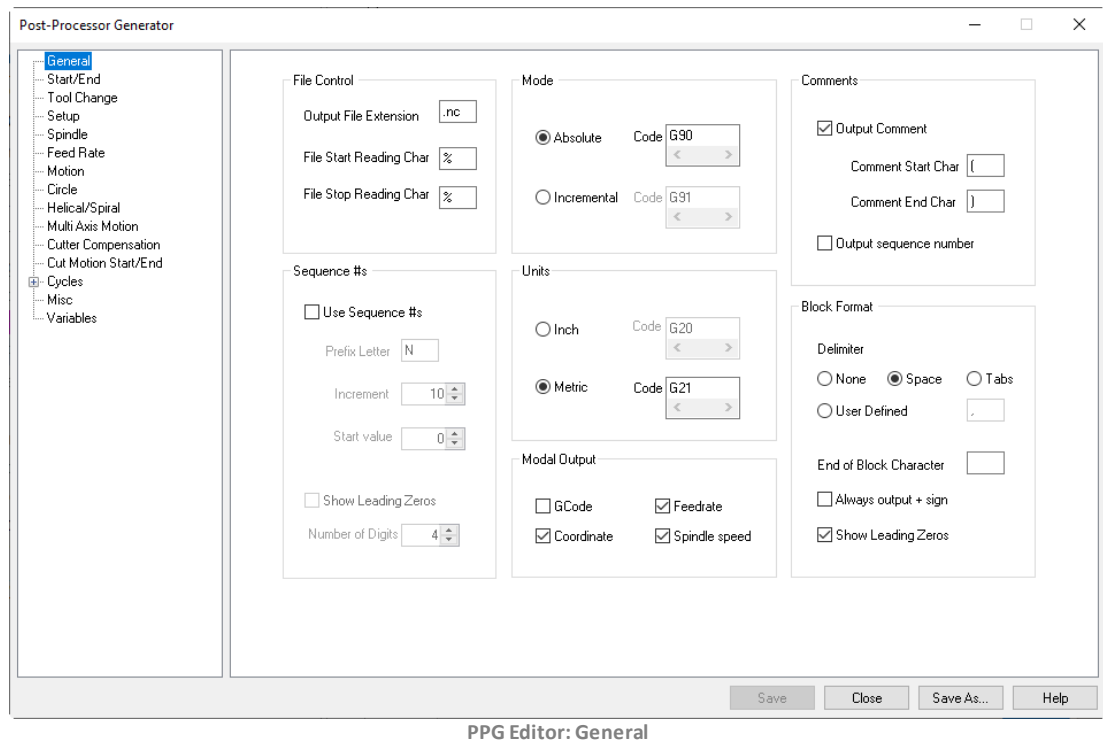
5.2 PPG Editor

5.2.1 General

The [General](#) section allows you to define the general behavior of the post-processor output. Each editable parameter is described below.



PPG Editor: General



File Control

This section allows you to set the default extension of the output file.

- **Output File Extension**
This allows you to set the posted file extension here in the PPG. Once set, go to the Post Preferences dialog ([CAM Preferences > Post](#)) and check the box "File Extension from Post Processor". The file extension will also be shown in that dialog.
- **File Start Reading Char**
The user can also set up an optional [File Start](#) character. This file start character will be the first character written to the output file.
- **File Stop Reading Char**
Similarly the file end character will be the last character written to the output file. Most standard controllers look for a percent sign (%) as this last character.



Mode

Coordinate values can be set to be either [Absolute](#) or [Incremental](#). In the [Absolute](#) mode, coordinate values (X, Y, Z, I, J, K) are output as absolute values. In the [Incremental](#) mode, coordinate values are output as incremental values.



Comments

This section defines the general format of commentary blocks.

- [Output Comment](#): Set parameter to output comments
- [Comment Start Char](#): Comment start character
- [Comment End Char](#): Comment end character
- [Output Sequence Number](#): Toggle sequence number for comments

If you want the comments to be output just as it is without the [Start Character](#) and the [End Character](#), then in [VisualCAD/CAM](#), when you are inserting a comment, type in a \$ sign in front of it, like:

\$ Comment



Sequence #s

This section allows you to control the format of sequence numbers to the output file. This option is available for all G-code lines except [Tool change Macro](#) and [Start and End Code](#).

- [Use Sequence #s](#)
Sequence number output can be turned on or off.
- [Prefix Letter](#)
When on, a prefix letter can optionally be added at the beginning of every sequence number.
- [Increment](#)
Sequence numbers can also be output in increments rather than sequentially. This increment value can be specified here.
- [Start Value](#)
This specifies the starting value for the sequence numbers.
- [Show Leading Zeros](#)
In addition the number of digits output as well as presence/absence of leading zeros in the sequence numbers can be controlled.



Units

This tells the post-processor the units of the output file by outputting a units code that can be defined here. Output units can either be in the English system (inches) or in the Metric system (mm).

- [Inch](#)
Sets the units to Inches and post the code [G20](#) by default. This code can be changed if desired.
- [Metric](#)
Sets the units to Millimeters and post the code [G21](#) by default. This code can be changed if desired.



Block Format

This section defines the general format of all output blocks. Each of the options is described below.

- Delimiter:**
 This is the delimiter used between G-codes.
 Example where delimiter is set to **[D]**: G01[D]X1.0[D]Y2.0[D]Z3.0[D]
 S3000M03[D]F20

None: No delimiter is posted in the output.
Space: A space character is output as the delimiter
Tab: A tab character is output as the delimiter
User Defined: Select this option and then enter the delimiter character to use.
- End of Block Character:** End of block character to output
- Always output +sign:** Outputs a '+' (no quote marks included) for positive values.
- Show Leading Zeros:** Check this box to include leading zeros in the posted code. Example (G01 X+1.0 Y+1.0 Z+1.0)



Modal Output

The **Post processor generator** allows the following parameters to be set as modal or non-modal. The modal output setting will output the value of a variable only if it is different from the value that was last output.

- Gcode:** G-Code modal option sample
- Coordinate:** Coordinates modal option sample
- Feedrate:** Feed Rate modal option sample
- Spindle speed:** Spindle Speed modal option sample

An example of non-modal data is shown below. The repeated values are shown in colored text.

```
S1000M03
G00 X1.0 Y2.0 Z0.0 F10
S2000M03
G01 X1.0 Y2.0 Z3.0 F20
G01 X1.0 Y3.0 Z3.0 F20
S2000M03
```



Related Topics

Other **PPG Editor** Sections:

[Start/End](#)

[Tool Change](#)[Setup](#)[Spindle](#)[Feed Rate](#)[Motion](#)[Circle](#)[Helical/Spiral Cycles](#)[Multi Axis Motion](#)[Cutter Compensation](#)[Cut Motion Start/End](#)[Cycles](#)[Miscellaneous](#)[Variables](#)

5.2.2 StartEnd

This tab is used to set the [Start](#) and [End](#) G-Codes.



PPG Editor: Start/End

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

Program Start/End

Start Up Code :

```
[START_CHAR]  
O[PARTNUM]  
[SEQ_PRECHAR][SEQNUM]G40G49G80
```

End Code :

```
[SEQ_PRECHAR][SEQNUM]M30  
[STOP_CHAR]
```

Save Close Save As... Help

PPG Editor: Start/End



Startup Code

First macro output in the generated NC file.

Example:

```
[START_CHAR]  
O[PARTNUM]  
[SEQ_PRECHAR][SEQNUM]G40G49G80
```



End Code

Last macro output in the generated NC file.

Example:

```
[SEQ_PRECHAR][SEQNUM]M30  
[STOP_CHAR]
```



Related Topics

[How to edit macros](#)

Other PPG Editor Sections:

[General](#)

[Tool Change](#)

[Setup](#)

[Spindle](#)

[Feed Rate](#)

[Motion](#)

[Circle](#)

[Helical/Spiral Cycles](#)

[Multi Axis Motion](#)

[Cutter Compensation](#)

[Cut Motion Start/End](#)

[Cycles](#)

[Miscellaneous](#)

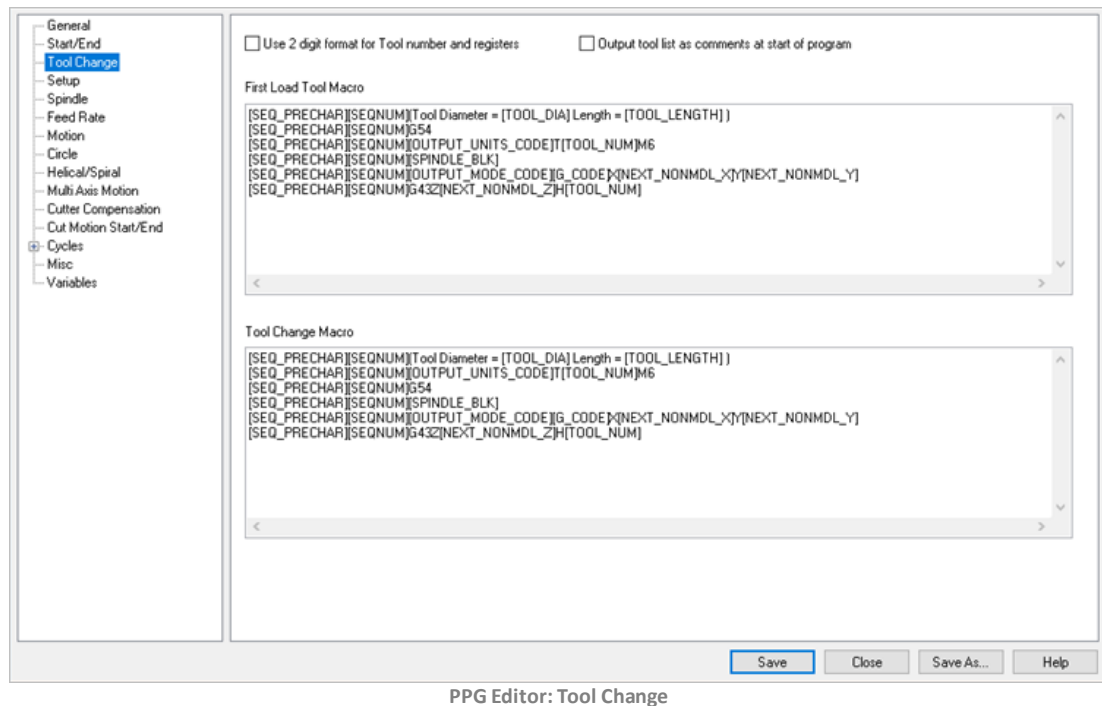
[Variables](#)

5.2.3 Tool Change

This tab is used to set the [Load Tool](#) and [Tool Change](#) macros.



PPG Editor: Tool Change



Use 2 digit format for Tool number and registers

Check this box to use 2-digit format for **Tool Numbers** and **Tool Registers** (i.e., T01)



Output tool list as comments at start of program

Check this box to include the tool list as a comment at the start of the program.

Example:

```
%
Ob
N1G40G49G80
( BEGIN TOOL LIST )
( TOOL 1 - FLATMILL- 1/2 INCH - DESC: 0.5000 DIA, 2 FLUTE, CARBIDE MAT )
( TOOL 2 - BALLMILL- 1/4 INCH - DESC: 0.2500 DIA, 2 FLUTE, CARBIDE MAT )
( ENDOF TOOL LIST )
(Setup 1)
(Horizontal Roughing)
N2(Tool Diameter = 0.5 Length = 4.0 )
N3G54
...
...
...
```



First Load Tool Macro

Macro for the first load tool command.

Example:

```
[SEQ_PRECHAR][SEQNUM](Tool Diameter = [TOOL_DIA] Length = [TOOL_LENGTH] )  
[SEQ_PRECHAR][SEQNUM]G54  
[SEQ_PRECHAR][SEQNUM][OUTPUT_UNITS_CODE]T[TOOL_NUM]M6  
[SEQ_PRECHAR][SEQNUM][SPINDLE_BLK]  
[SEQ_PRECHAR][SEQNUM][OUTPUT_MODE_CODE][G_CODE]X[NEXT_NONMDL_X]  
Y[NEXT_NONMDL_Y]  
[SEQ_PRECHAR][SEQNUM]G43Z[NEXT_NONMDL_Z]H[TOOL_NUM]
```



Tool Change Macro

Macro for tool change command. (Not including the first load tool.)

Example:

```
[SEQ_PRECHAR][SEQNUM](Tool Diameter = [TOOL_DIA] Length = [TOOL_LENGTH] )  
[SEQ_PRECHAR][SEQNUM][OUTPUT_UNITS_CODE]T[TOOL_NUM]M6  
[SEQ_PRECHAR][SEQNUM]G54  
[SEQ_PRECHAR][SEQNUM][SPINDLE_BLK]  
[SEQ_PRECHAR][SEQNUM][OUTPUT_MODE_CODE][G_CODE]X[NEXT_NONMDL_X]  
Y[NEXT_NONMDL_Y]  
[SEQ_PRECHAR][SEQNUM]G43Z[NEXT_NONMDL_Z]H[TOOL_NUM]
```



Related Topics

[How to edit macros](#)

Other PPG Editor Sections:

[General](#)

[Start/End](#)

[Setup](#)

[Spindle](#)

[Feed Rate](#)

[Motion](#)

[Circle](#)

[Helical/Spiral Cycles](#)

[Multi Axis Motion](#)

[Cutter Compensation](#)

[Cut Motion Start/End](#)

[Cycles](#)

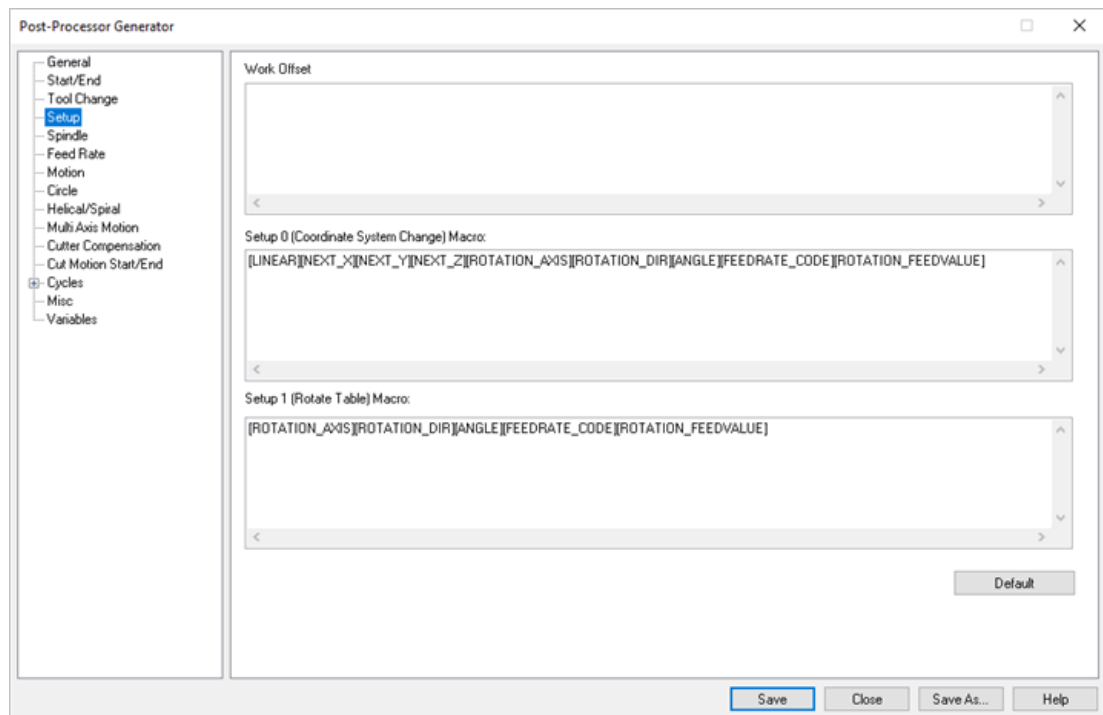
[Miscellaneous](#)

[Variables](#)

5.2.4 Setup



PPG Editor: Setup



PPG Editor: Setup



Work Offset

You can define macros here that will control the format of each [Work Offset](#) in the posted g-code file.



Setup 0 (Coordinate System Change) Macro

Macro to execute before every [Setup](#) change.

Example:

```
[LINEAR][NEXT_X][NEXT_Y][NEXT_Z][ROTATION_AXIS][ROTATION_DIR][ANGLE]
[FEEDRATE_CODE][ROTATION_FEEDVALUE]
```



Setup 1 (Rotate Table) Macro

Macro to execute before every [Rotate Table Setup](#) change.

Example:

```
[ROTATION_AXIS][ROTATION_DIR][ANGLE][FEEDRATE_CODE][ROTATION_FEEDVALUE]
```



Related Topics

Other PPG Editor Sections:

[General](#)

[Start/End](#)

[Tool Change](#)

[Spindle](#)

[Feed Rate](#)

[Motion](#)

[Circle](#)

[Helical/Spiral Cycles](#)

[Multi Axis Motion](#)

[Cutter Compensation](#)

[Cut Motion Start/End](#)

[Cycles](#)

[Miscellaneous](#)

[Variables](#)

5.2.5 Spindle

This tab is used to set parameters for controlling the spindle.



PPG Editor: Spindle

PPG Editor: Spindle



Block Format

Sets the spindle block format.

- **Spindle Code:** Register used for the feed rate value.
- **Spindle Direction:** Specify the direction code for the spindle
 - Clockwise Rotation Code:** Clockwise spindle code
 - C-Clockwise Rotation Code:** Counter clockwise spindle code
 - Spindle Off Code:** Spindle off code
- **Spindle Mode:**
 - Constant Surface Speed:**
 - Constant Rotation Speed:**
- **Spindle Block Format:** Defines the block format for the spindle. Example: `S[SPINDLE_SPD][SPINDLE_ARC]`
- **Default:** Reset all values in this section to their system defaults.
- **Sample output:** This field displays sample output of the spindle block. It is a non-editable field.



Spindle RPM

- **High Value:** Maximum spindle value. The spindle RPM is capped to this high value.

- **Low Value:** Minimum spindle value. The spindle RPM is capped to this low value.
- **Scale Factor:** Scale factor of Spindle value.
- **# of Decimal Places:** Number of digits output after the decimal point
- **Show Trailing Zeros:** Show trailing zeros when **# of Decimal Places** is set to a value greater than zero.



Related Topics

Other PPG Editor Sections:

[General](#)

[Start/End](#)

[Tool Change](#)

[Setup](#)

[Feed Rate](#)

[Motion](#)

[Circle](#)

[Helical/Spiral Cycles](#)

[Multi Axis Motion](#)

[Cutter Compensation](#)

[Cut Motion Start/End](#)

[Cycles](#)

[Miscellaneous](#)

[Variables](#)

5.2.6 Feed Rate

This tab sets the feeds and speeds parameters



PPG Editor: Feed Rate

The screenshot shows the 'PPG Editor: Feed Rate' dialog box. On the left is a sidebar with a tree view containing the following items: General, Start/End, Tool Change, Setup, Spindle, **Feed Rate** (highlighted), Motion, Circle, Helical/Spiral, Multi Axis Motion, Cutter Compensation, Cut Motion Start/End, Cycles, Misc, and Variables. The main panel is divided into several sections:

- Block Format:** Includes a 'Feed Rate Code' dropdown set to 'F', a checked 'Add after Next Motion Code' checkbox, and an 'Inverse Time Feed Rate' section with an unchecked checkbox 'Output inverse time feedrate for 4 & 5 Axis motions' and 'On Code' and 'Off Code' dropdowns.
- Feed Rate Modes:** Includes 'Units/Min Code' and 'Units/Rev Code' dropdowns.
- Block Format:** A text field containing '[FEEDRATE]'.
- Sample Output:** A text field containing 'F0.' with a 'Default' button to its right.
- Feed Rate Value:** Includes 'High Value' (300), 'Low Value' (0), 'Scale Factor' (1), '# of Decimal Places' (3), 'Z Feed Rate Scale Factor' (1), and a 'Show Trailing Zeros' checkbox.
- Angular Feedrate Value:** Includes 'Scale Factor' (1), '# of Decimal Places' (1), and a 'Show Trailing Zeros' checkbox.

At the bottom right of the dialog are four buttons: 'Save', 'Close', 'Save As...', and 'Help'.

PPG Editor: Feed Rate



Block Format

Characters for the feed rate block:

- **Feed Rate Code:** Register used for the feed rate value
- **Add after Next Motion Code:**
- **Feed Rate Modes**
 - Units/Min Code:** Typically a G94
 - Units/Rev Code:** Typically a G95
- **Inverse Time Feed Rate**

When **Output inverse time feedrate for 4 & 5 Axis motions** is selected the **On Code** is output at the start of the operation and **Off Code** at the end.

You turn on the **Inverse Time Feedrate** using a checkbox in the **Post/Feedrate** tab. You can set up the code for **On** and **Off** in the same tab. Once this is set, then all 4 and 5 axis motions will be processed with this feedrate being output.

The way the feedrate is computed is as follows:

First the distance traveled by the tool is computed for each move = **dist**

Then the time taken for traversing the move is computed thus:
 $\text{timeForTravel} = \text{dist} / \text{cutFeedRate};$

Then the inverse time feedrate is = $1.0/\text{timeForTravel};$

And it is output for each move if different from previous move.

[On Code](#): G93

[Off Code](#): G94

- [Block Format](#): Format for the feed rate block
- [Default](#): Reset all values in this section to their system defaults.
- [Sample Output](#): Sample output displays sample code of the feed rate. It is a non-editable field.



Feed Rate values

Parameters for the adjustment of feed rate value:

- [High Value](#): Maximum Feed rate value.
- [Low Value](#): Minimum Feed rate value.
- [Scale Factor](#): Scale factor of Feed rate value.
- [# of Decimal Places](#): Number of digits output after the decimal point.
- [Z Feed Rate Scale Factor](#): Feed rate scale factor for Z Feed Rate (use [ZFEDRATE] variable to get the Z feed rate)
- [Show Trailing Zeros](#): Show trailing zeros when [# of Decimal Places](#) is set to a value greater than zero.



Angular Rate values

- [Scale Factor](#): Enter the scale factor for angular feed rate values.
- [# of Decimal Places](#): Number of digits output after the decimal point
- [Show Trailing Zeros](#): Show trailing zeros when [# of Decimal Places](#) is set to a value greater than zero.



Related Topics

Other [PPG Editor](#) Sections:

[General](#)

[Start/End](#)

[Tool Change](#)

[Setup](#)

[Spindle](#)

[Motion](#)

[Circle](#)

[Helical/Spiral Cycles](#)[Multi Axis Motion](#)[Cutter Compensation](#)[Cut Motion Start/End](#)[Cycles](#)[Miscellaneous](#)[Variables](#)

5.2.7 Motion

This tab is used to define the linear motion outputs of a post-processor. You can use multiple lines to define the [Block Format](#) in both the [Linear Motion Block](#) and the [Rapid Motion Block](#).



PPG Editor: Motion

PPG Editor: Motion



Linear Motion Block

Used to define the output format for the cut motions (e.g.: G1). The sample output can be seen in the non-editable Sample Output field. Multiple lines of code are supported.

- [G Code](#): Typically a G1

- **Block Format:** Example: `[G_CODE][NEXT_X][NEXT_Y][NEXT_Z]`
- **Default:** Reset all values in this section to their system defaults.
- **Sample Output:** This displays how the posted output will look.
- **Output cut motion start macro before plunge motion:** Checking this box will output the cut motion start macro defined under [Cut Motion Start/End](#) section before the plunge motion in the posted code.



Rapid Motion Block Format

Used to define the output format for the rapid motions (e.g.: G0). The sample output can be seen in the non-editable Sample Output field. Multiple lines of code are supported.

- **G Code:** Typically G0
- **Block Format:** Example:
`[G_CODE][NEXT_Z]`
`[NEXT_X][NEXT_Y]`
- **Default:** Reset all values in this section to their system defaults.
- **Sample Output:** This displays how the posted output will look.



Motion Coordinates

- **Scale Factor for X, Y, Z:** Scale factor of the coordinate values. (Includes circles and cycles)
- **Show Trailing Zeros:** Show the trailing zeros (e.g. 5.4 is output as 5.4000)
- **# of Decimal Places:** Number of digits output after the decimal point



Related Topics

Other PPG Editor Sections:

[General](#)

[Start/End](#)

[Tool Change](#)

[Setup](#)

[Spindle](#)

[Feed Rate](#)

[Circle](#)

[Helical/Spiral Cycles](#)

[Multi Axis Motion](#)

[Cutter Compensation](#)

[Cut Motion Start/End](#)

[Cycles](#)

[Miscellaneous](#)

[Variables](#)

5.2.8 Circle

This tab is used to define the circle block output.



PPG Editor: Circle

G Code Clockwise Arc Code G02 ← → C-clockwise Arc Code G03 ← →		Plane Code XY G17 ← → ZX G18 ← → YZ G19 ← →	
Output Format ☐ I, J, K, and Radius ☒ I, J, K only ☐ Radius only ☐ Output values only when different ☐ Use -R for CW Arcs		Arc Center (I, J, K) ☐ Absolute ☐ Vector from Center to Start ☒ Vector from Start to Center ☐ Unsigned Vector from Start to Center	
Block Format Select Plane Plane XY Plane ZX [CIR_PLANE] [G_CODE][NEXT_X][NEXT_Y][NEXT_I][NEXT_J] Sample Output {G17/G18/G19} {G02/G03}X0.0Y0.0I0J0			
☐ Limit Arcs to Angle 180		Default	

PPG Editor: Circle



G Code

Arc Direction Code:

- **Clockwise Arc Code:** Typically G02
- **C-clockwise Arc Code:** Typically G03



Plane Code

The G Code for the principal planes in which the Arc motion is output.

- [XY](#): Typically G17
- [ZX](#): Typically G18
- [YZ](#): Typically G19



Output Format

These options help define the output format for the circle command.

- [I,J,K and Radius](#): Output [I,J,K](#) and [Radius](#)
- [I,J,K only](#): Output only [I,J,K](#).
- [Radius only](#): Output only [Radius](#).
- [Output values only when different](#): Output [I,J,K,R](#) values only when different from the previous values.
- [Use -R for CW Arcs](#): Prefixes [-R](#) before clockwise arc motions.



Arc Center (I,J,K)

Defines the calculation of the arc center coordinates.

- [Absolute](#): is the absolute center
- [Vector from Center to Start](#): is (Center - Start)
- [Vector from Start to Center](#): is (Start - Center)
- [Unsigned vector](#): is the unsigned distance between center and start.



Block Format

Used to specify the block format for three different planes. Use the default button to see the default values and edit them if needed.



Sample Output

This field displays sample output of the arc motion block. This field is non-editable. It indicates the parameters selected from the available options.



Limit Arcs to Angle

This is used to limit arcs to a certain angle. This is helpful for certain types of controllers which cannot output arcs greater than a certain angle



Default

Reset all values in this section to their system defaults.



Related Topics

Other [PPG Editor](#) Sections:

[General](#)

[Start/End](#)

[Tool Change](#)

[Setup](#)

[Spindle](#)

[Feed Rate](#)

[Motion](#)

[Helical/Spiral Cycles](#)

[Multi Axis Motion](#)

[Cutter Compensation](#)

[Cut Motion Start/End](#)

[Cycles](#)

[Miscellaneous](#)

[Variables](#)

5.2.9 Helical/Spiral

This tab is used to define the Helix and Spiral block output. For both these cycles, the [Arc Center](#) and the [Plane Code](#) are the same as those defined in the [Circle Section](#). Please define those first before defining these cycles.



PPG Editor: Helical/Spiral Cycles

PPG Editor: Helical/Spiral Cycles



Select Interpolation Type

Used to select the Interpolation type to define the parameters for the [Helical Cycles](#) or the [Spiral Cycles](#).



G Code

Used to define the [Clockwise](#) and the [Counter Clockwise Codes](#) for the [Helical](#) or [Spiral Cycles](#)



Block Format

Used to specify the block format for three different planes. Use the default button to see the default values and edit them if needed.



Default

Reset all values in this section to their system defaults.



Sample Output

This field displays sample output of the [Helical/Spiral](#) motion block. This field is non-editable. It indicates the parameters selected from the available options.



Related Topics

Other [PPG Editor](#) Sections:

[General](#)

[Start/End](#)

[Tool Change](#)[Setup](#)[Spindle](#)[Feed Rate](#)[Motion](#)[Circle](#)[Multi Axis Motion](#)[Cutter Compensation](#)[Cut Motion Start/End](#)[Cycles](#)[Miscellaneous](#)[Variables](#)

5.2.10 Multi Axis Motion

This tab is used to set 4th Axis Motion Section parameters.



PPG Editor: Multi Axis Motion

Block Format

Rotation Axis Code

Primary Axis: Secondary Axis: Tertiary Axis:

Rotation Direction Code

Clockwise Rotation: Counter Clockwise Rotation: ☒ Ignore in continuous rotation toolpaths

Angle Value

Scale Factor: # Decimal Places: Trailing Zeros: ☐

Motion Block

Block Format

[LINEAR][NEXT_X][NEXT_Y][NEXT_Z][ROTATION_AXIS][ROTATION_DIR][ANGLE][FEEDRATE_CODE][ROTATION_FEEDVALUE]

Sample Output

G1X0.0Y0.0Z0.0A-0F0

Rapid Block

Block Format

[RAPID][NEXT_X][NEXT_Y][NEXT_Z][ROTATION_AXIS][ROTATION_DIR][ANGLE][FEEDRATE_CODE][ROTATION_FEEDVALUE]

Sample Output

G0X0.0Y0.0Z0.0A-0F0

PPG Editor: Multi Axis Motion



Block Format



Rotation Axis Code

Characters for Rotation Axis Code.

- **Primary Axis:** Sets the primary axis code. Typically A
- **Secondary Axis:** Sets the secondary axis code. Typically B
- **Tertiary Axis:** Sets the Tertiary axis code. Typically C



Rotation Direction Code (Only for Rotate Table)

Characters for Rotation Direction Code

- **Clockwise Rotation:** Clockwise rotation code
- **Counter Clockwise Rotation:** Counter Clockwise rotation code
- **Ignore in continuous rotation toolpaths:** Check this box to ignore rotation direction in 4 Axis continuous rotation toolpaths.



Angle Values

Angle Value for 4th Axis Motion

- **Scale Factor:** Scale Factor for Angle Value (the angle is in radians, to convert to degrees use a scale factor of 57.295779513082)
- **# Decimal Places:** No of Decimal Places
- **Trailing Zeros:** Number of Trailing Zeros after Decimal places



Motion Block

Motion Code for 4th Axis Motion

- **Block Format:** Helps to define the output format for the 4th Axis motion code.
- **Sample Output:** Sample output displays sample code of the 4th Axis motion. It is a non-editable field.
- **Default:** Reset all values in this section to their system defaults.



Rapid Block

Rapid Code for 4th Axis Motion

- **Block Format:** Helps to define the output format for the 4th Axis rapid code
- **Sample Output:** Sample output displays sample code of the 4th Axis rapids. It is a non-editable field.
- **Default:** Reset all values in this section to their system defaults.



Related Topics

Other PPG Editor Sections:

[General](#)

[Start/End](#)

[Tool Change](#)

[Setup](#)

[Spindle](#)

[Feed Rate](#)

[Motion](#)

[Circle](#)

[Helical/Spiral Cycles](#)

[Cutter Compensation](#)

[Cut Motion Start/End](#)

[Cycles](#)

[Miscellaneous](#)

[Variables](#)

5.2.11 Cutter Compensation



PPG Editor: Cutter Compensation

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

Cutter Compensation Left
[SEQ_PRECHAR][SEQNUM]G41[G_CODE]>[NEXT_NONMDL_X]'Y[NEXT_NONMDL_Y]D[TOOL_NUM]

Cutter Compensation Right
[SEQ_PRECHAR][SEQNUM]G42[G_CODE]>[NEXT_NONMDL_X]'Y[NEXT_NONMDL_Y]D[TOOL_NUM]

Cutter Compensation Off
[SEQ_PRECHAR][SEQNUM]G40[G_CODE]>[NEXT_NONMDL_X]'Y[NEXT_NONMDL_Y]

Save Close Save As... Help

PPG Editor: Cutter Compensation



Cutter Compensation Left

Macro used to define the output when the cutter compensation left is detected in the output



Cutter Compensation Right

Macro used to define the output when the cutter compensation right is detected in the output



Cutter Compensation Off

Macro used to define the output when the cutter compensation cancel (off) is detected in the output



Related Topics

Other [PPG Editor](#) Sections:

[General](#)

[Start/End](#)

[Tool Change](#)

[Setup](#)

[Spindle](#)

[Feed Rate](#)

[Motion](#)

[Circle](#)

[Helical/Spiral Cycles](#)

[Multi Axis Motion](#)

[Cut Motion Start/End](#)

[Cycles](#)

[Miscellaneous](#)

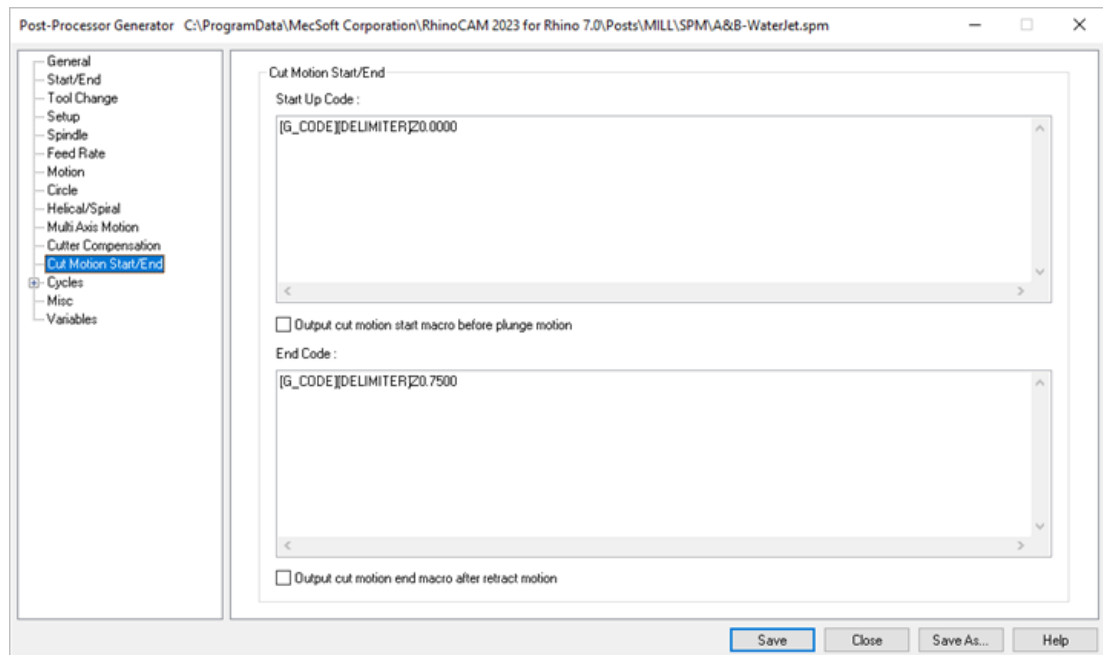
[Variables](#)

5.2.12 Cut Motion Start/End

The Cut Motion Start/End dialog allows you to define [Start](#) and [End](#) cut motion codes.



PPG Editor: Cut Motion Start/End



PPG Editor: Cut Motion Start/End

Output cut motion start macro before plunge motion

You can have the [Cut Motion Start Up Code](#) posted before plunge motions by checking the box.

Output cut motion end macro after retract motion

You can have the [Cut Motion End Macro](#) posted after the retract motions by checking this box.



Related Topics

Other PPG Editor Sections:

[General](#)

[Start/End](#)

[Tool Change](#)

[Setup](#)

[Spindle](#)

[Feed Rate](#)

[Motion](#)

[Circle](#)

[Helical/Spiral Cycles](#)

[Multi Axis Motion](#)

[Cutter Compensation](#)

[Cycles](#)

[Miscellaneous](#)

[Variables](#)

5.2.13 Cycles

This tab is to set the [cycle](#) parameters in the [VisualCAD/CAM Post-Processor](#).



PPG Editor: Cycles

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

Cycle G-Code: G80

Cycle Code:
[SEQ_PRECHAR][SEQNUM][G_CODE]

Optimize Cycle Options
☐ Optimize Cycle Output

Cycle Coordinate Block:

Dwell Options
Scale Factor 1 # of Decimal Places 0

Thread First Depth Options
Scale Factor 10000 # of Decimal Places 0

Save Close Save As... Help

PPG Editor: Cycles



Cycle G-Code

Macros for the cycle commands. It represents the selected G-code value that defines each specific cycle. This value is displayed in the edit box below all the available options. It can be changed if required.



Cycle Code

The following Cycles are supported:

- Cycle Off (G80)
- Standard Drill (G81)
- Standard Drill with Dwell On (G82)
- Deep (G83)

- Break Chip (G87)
- Counter Sink (G82)
- Tap (Clockwise) (G84)
- Tap (C-Clockwise) (G84)
- Peck Tapping (Clockwise) (G84)
- Peck Tapping (C-Clockwise) (G84)
- Rigid Tap (Clockwise) (G84)
- Rigid Tap (C-Clockwise) (G74)
- Bore (Drag) Dwell Off (G85)
- Bore (Drag) Dwell On (G89)
- Bore (No Drag) Dwell Off Orient On (G76)
- Bore (No Drag) Dwell On Orient On (G76)
- Bore (No Drag) Dwell Off Orient Off (G86)
- Bore (No Drag) Dwell On Orient Off (G86)
- Bore (Reverse) Dwell Off (G87)
- Bore (Manual) Dwell On (G88)
- Bore (Reverse) Dwell Off (G77)
- Bore (Reverse) Dwell On (G77)
- User Defined Drill Cycle 1
- User Defined Drill Cycle 2
- User Defined Drill Cycle 3
- User Defined Drill Cycle 4
- User Defined Tap Cycle 1
- User Defined Tap Cycle 2
- User Defined Tap Cycle 3
- User Defined Tap Cycle 4
- User Defined Bore Cycle 1
- User Defined Bore Cycle 2
- User Defined Reverse Bore Cycle 1
- User Defined Reverse Bore Cycle 2
- Turn Thread Cycle Automatic
- Turn Thread Cycle Box Cycle
- Turn Thread Cycle Single Block
- Machine Control Cycle 1
- Machine Control Cycle 2
- Machine Control Cycle 3
- Machine Control Cycle 4



Optimize Cycle Options

Optimize Cycle output will define the cycle format only once and will output the X,Y values for all the other holes. This will result in significant reduction in the file size for output.

- **Optimize Cycle Output:** Enables the optimized cycle options.

- [Cycle Coordinate Block](#): Specific cycle block variables to use.



Dwell Options

- [Scale Factor for Dwell](#): Sets the scale factor for dwell output.
- [# of Decimal Places](#): Sets the number of decimal places for output.



Thread First Depth Options

- [Scale Factor](#): Sets the scale factor for the first thread depth.
- [# of Decimal Places](#): Sets the number of decimal places for output.



Related Topics

[How to edit macros](#)

Other PPG Editor Sections:

[General](#)

[Start/End](#)

[Tool Change](#)

[Setup](#)

[Spindle](#)

[Feed Rate](#)

[Motion](#)

[Circle](#)

[Helical/Spiral Cycles](#)

[Multi Axis Motion](#)

[Cutter Compensation](#)

[Cut Motion Start/End](#)

[Miscellaneous](#)

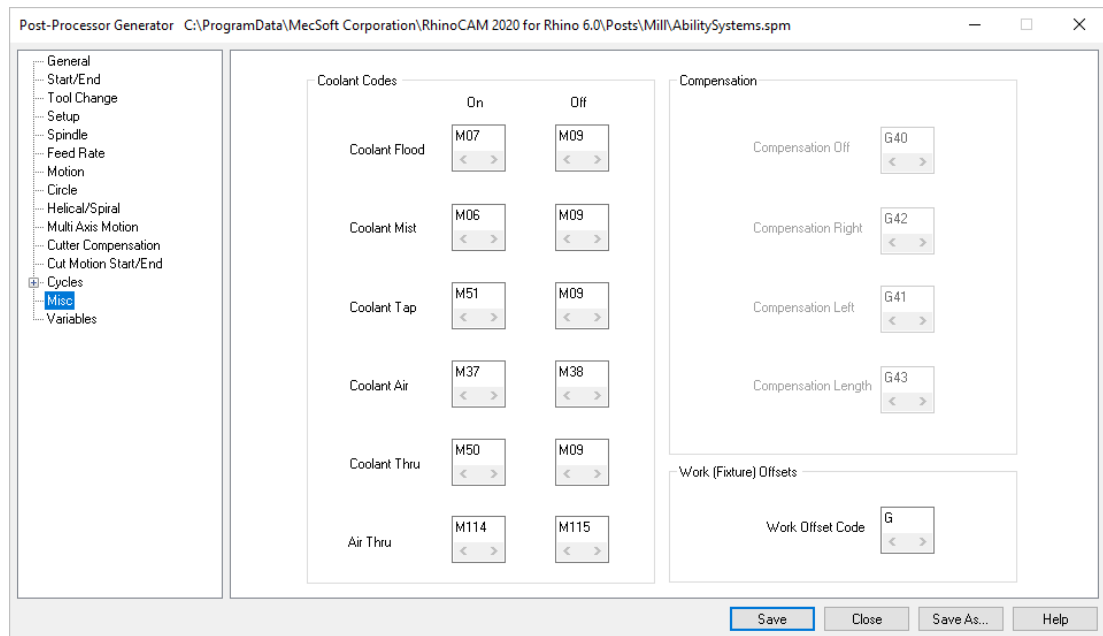
[Variables](#)

5.2.14 Miscellaneous

This tabbed dialog is used to set miscellaneous parameters. The various miscellaneous parameters are shown in the dialog.



PPG Editor: Miscellaneous



PPG Editor: Miscellaneous



Coolant Codes

- **Coolant Flood:** Coolant Flood On/Off code
- **Coolant Mist:** Coolant Mist On/Off code
- **Coolant Tap:** Coolant Tap On/Off code
- **Coolant Air:** Coolant Air On/Off code
- **Coolant Thru:** Coolant Thru On/Off code
- **Air Through:** Coolant Air Thru On/Off code



Compensation

- **Compensation Off:** (This is not editable - reserved for future releases)
- **Compensation Left:** (This is not editable - reserved for future releases)
- **Compensation Right:** (This is not editable - reserved for future releases)
- **Compensation Length:** (This is not editable - reserved for future releases)



Work (Fixture) Offsets

- **Work (Fixture) Offsets:** Sets the work offset prefix code. Typically G. You can enable the output of the work offset code by selecting **Output Work Offset** from the **Work Zero** dialog and entering the offset code. For example, enabling and entering **54** in the **Work Zero** dialog activates this portion of the post and prefixes the code with this value. Example: **G54**.



Related Topics

Other PPG Editor Sections:

[General](#)

[Start/End](#)

[Tool Change](#)

[Setup](#)

[Spindle](#)

[Feed Rate](#)

[Motion](#)

[Circle](#)

[Helical/Spiral Cycles](#)

[Multi Axis Motion](#)

[Cutter Compensation](#)

[Cut Motion Start/End](#)

[Cycles](#)

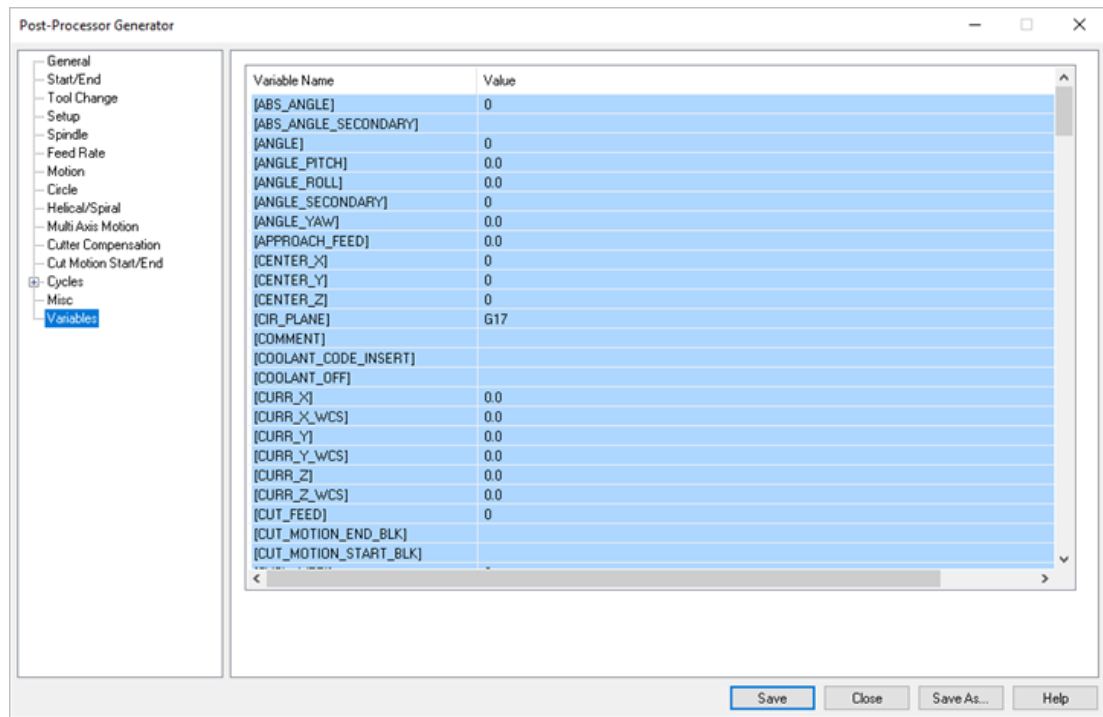
[Variables](#)

5.2.15 Variables

This tab lists all the [variables](#) used in the macros in the [VisualCAD/CAM Post Processor](#).



PPG Editor: Variables



PPG Editor: Variables



Variables

The variables that can be used are listed in the table below.

The variables that start with "CYCL_" are used only for the cycles commands.

Variable	Comments
[ABS_ANGLE]	
[ABS_ANGLE_SECONDARY]	
[ANGLE]	
[ANGLE_PITCH]	
[ANGLE_ROLL]	
[ANGLE_SECONDARY]	
[ANGLE_YAW]	
[APPROACH_FEED]	
[CENTER_X]	
[CENTER_Y]	

[CENTER_Z]	
[CIR_PLANE]	
[COMMENT]	Output comments.
[COOLANT_CODE_INSERT]	
[COOLANT_OFF]	
[CURR_X]	The X coordinate of current point.
[CURR_X_WCS]	The X coordinate of current point in World Coordinates
[CURR_Y]	The Y coordinate of current point.
[CURR_Y_WCS]	The Y coordinate of current point in World Coordinates
[CURR_Z]	The Z coordinate of current point.
[CURR_Z_WCS]	The Z coordinate of current point in World Coordinates
[CUT_FEED]	
[CUT_MOTION_END_BLK]	
[CUT_MOTION_START_BLK]	
[CYCL_1/TPI]	1/TPI (only for TAP cycle)
[CYCL_CLEAR+DEPTH]	Drill Depth + Clear. (only for cycles except C-SINK)
[CYCL_CLEAR+INCR]	
[CYCL_CLEAR]	Clearance. (only for cycles)
[CYCL_CSINK_DEPTH+CLEAR]	Drill Depth + Clear. (only for C-SINK cycle)
[CYCL_CSINK_DEPTH]	Drill Depth. (only for C-Sink cycle)
[CYCL_DEPTH]	Drill Depth. (only for cycles except C-SINK)
[CYCL_DWELL]	Dwell. (only for cycles)
[CYCL_INCR]	Step Increment. (only for cycles)
[CYCL_IPM]	IPM. (only for cycles except TAP)
[CYCL_IPR]	IPR. (only for TAP cycle)
[CYCL_NEG_CLEAR+DEPTH]	-(Drill Depth + Clear). (only for cycles except C-SINK)
[CYCLE_NUM_STEPS]	

[CYCL_ORIENT]	Orient. (only for cycles)
[CYCL_SCALED_DWELL]	Dwell * Scale Factor (only for cycles)
[CYCL_TPI]	
[CYCLE_Z]	This variable stores the Z value of drill points.
[CYCL_Z+CLEAR]	Next Z + Clearance. (only for cycles)
[CYCL_Z+DEPTH+CLEAR]	Next Z + Depth + Clearance. (only for cycles)
[CYCL_Z+DEPTH]	Next Z + Depth. (only for cycles)
[CYCL_Z-DEPTH]	Next Z - Depth. (only for cycles)
[DELIMITER]	Delimiter definition.
[DEPART_FEED]	Outputs the Departure feedrate value for the currently operation.
[ENGAGE_FEED]	Outputs the Engage feedrate value for the currently operation.
[EOB]	The end of block character.
[EXTRUSION]	
[EXTRUSION_BLK]	
[EXTRUSION_CODE]	
[FEEDRATE]	FeedRate Value.
[FEEDRATE_BLK]	FeedRate Block.
[FEEDRATE_CODE]	Outputs the Feedrate code
[FEEDRATE_UNITS_CODE]	Outputs the Feedrate Units code
[FIRST_TOOL_NUM]	
[G_CODE]	The next G-Code. This is translated to linear, rapid, arc or cycle G-Code.
[HELIX_ANGLE]	
[HELIX_CCW_ARC]	
[HELIX_CW_ARC]	
[HELIX_LEAD]	
[HELIX_NUM_REV]	

[HELIX_RADIUS]	
[HELIX_TOTAL_DEPTH]	
[INPUTFILE_NAME]	
[INPUTFILE_NAME_LONG]	
[INV_TIME_FEEDRATE_OFF]	
[INV_TIME_FEEDRATE_ON]	
[LINEAR]	The linear motion code .
[MOP_NAME]	
[NEXT_ABS_X_WCS]	The next absolute X coordinate point in World Coordinates.
[NEXT_ABS_Y_WCS]	The next absolute Y coordinate point in World Coordinates.
[NEXT_ABS_Z_WCS]	The next absolute Z coordinate point in World Coordinates.
[NEXT_I]	
[NEXT_J]	
[NEXT_K]	
[NEXT_NONMDL_E]	
[NEXT_NONMDL_I]	
[NEXT_NONMDL_J]	
[NEXT_NONMDL_K]	
[NEXT_NONMDL_L]	
[NEXT_NONMDL_R]	
[NEXT_NONMDL_X]	The next non-modal X coordinate point in local Machine Coordinates.
[NEXT_NONMDL_X_WCS]	The next non-modal X coordinate point in World Coordinates.
[NEXT_NONMDL_Y]	The next non-modal Y coordinate point in local Machine Coordinates.
[NEXT_NONMDL_Y_WCS]	The next non-modal Y coordinate point in World Coordinates.

[NEXT_NONMDL_Z]	The next non-modal Z coordinate point in local Machine Coordinates.
[NEXT_NONMDL_Z_WCS]	The next non-modal Z coordinate point in World Coordinates.
[NEXT_R]	
[NEXT_TOOL_NAME]	
[NEXT_TOOL_NUM]	
[NEXT_X]	The next X coordinate point in Machine Coordinates.
[NEXT_X_WCS]	The next X coordinate point in World Coordinates.
[NEXT_Y]	The next Y coordinate point in Machine Coordinates.
[NEXT_Y_WCS]	The next Y coordinate point in World Coordinates.
[NEXT_Z]	The next Z coordinate point in Machine Coordinates.
[NEXT_Z_WCS]	The next Z coordinate point in World Coordinates.
[OUTPUTE_MODE_CODE]	
[OUTPUT_UNITS_CODE]	English or Metric outputs code.
[OUTPUTFILE_NAME]	
[OUTPUT_FILENAME_LONG]	
[PARTNAME]	
[PARTNUM]	
[PLUNGE_FEED]	
[POST_NAME]	
[POST_NAME_LONG]	
[PREV_TOOL_ADJST_REG]	
[PREV_TOOL_CUTCOM_REG]	
[PREV_TOOL_NUM]	
[PREV_TOOL_NUM_FLUTES]	

[PREV_TOOL_ZOFFSET]	
[PROGRAM_END_BLK]	
[PROGRAM_START_BLK]	
[RAPID]	The rapid motion code .
[RAPID_FEED]	
[RETRACT_FEED]	
[ROTATION_AXIS]	
[ROTATION_AXIS_SECONDARY]	
[ROTATION_DIR]	
[ROTATION_DIR_SECONDARY]	
[ROTATION_FEEDVALUE]	
[ROTATION_MODE]	
[SEQ_PRECHAR]	Letter that is prefixed before the sequence number
[SEQNUM]	The actual sequence number.
[SETUP_NAME]	
[SETUP_X]	
[SETUP_XAXIS_X]	
[SETUP_XAXIS_Y]	
[SETUP_XAXIS_Z]	
[SETUP_Y]	
[SETUP_YAXIS_X]	
[SETUP_YAXIS_Y]	
[SETUP_YAXIS_Z]	
[SETUP_Z]	
[SETUP_ZAXIS_X]	
[SETUP_ZAXIS_Y]	

[SETUP_ZAXIS_Z]	
[SPINDLE_ARC]	Spindle direction.
[SPINDLE_BLK]	Spindle Block.
[SPINDLE_CODE]	
[SPINDLE_SPD]	Spindle speed.
[SPINDLE_SPD_MAX]	
[SPINDLE_SPD_TYPE]	
[SPIRAL_ANGLE]	
[SPIRAL_CCW_ARC]	
[SPIRAL_CW_ARC]	
[SPIRAL_END_RADIUS]	
[SPIRAL_LEAD]	
[SPIRAL_NUM_REV]	
[SPIRAL_START_RADIUS]	
[SPIRAL_TOTAL_LENGTH]	
[START_CHAR]	The program start character.
[START_POSITION_X]	
[START_POSITION_Y]	
[START_POSITION_Z]	
[START_X]	The X coordinate of start point.
[START_X_WCS]	The X coordinate of start point in World Coordinates.
[START_Y]	The Y coordinate of start point.
[START_Y_WCS]	The Y coordinate of start point in World Coordinates.
[START_Z]	The Z coordinate of start point.
[START_Z_WCS]	The Z coordinate of start point in World Coordinates.
[STEP_NEXT_X]	

[STEP_NEXT_Y]	
[STEP_NEXT_Z]	
[STEP_START_DEPTH]	
[STOCK_LENGTH_X]	UNDEFINED is output if there is no stock defined when posting occurs.
[STOCK_LENGTH_Y]	
[STOCK_LENGTH_Z]	
[STOCK_MAX_X]	
[STOCK_MAX_Y]	
[STOCK_MAX_Z]	
[STOCK_MIN_X]	
[STOCK_MIN_Y]	
[STOCK_MIN_Z]	
[STOP_CHAR]	The program end character.
[TEMPERATURE]	
[TEMPERATURE_BED_SET_CODE]	
[TEMPERATURE_BED_WAIT_CODE]	
[TEMPERATURE_EXTRUDER_SET_CODE]	
[TEMPERATURE_EXTRUDER_WAIT_CODE]	
[TEMPERATURE_SET_BLK]	
[TEMPERATURE_SET_CODE]	
[TEMPERATURE_WAIT_BLK]	
[TEMPERATURE_BED_WAIT_CODE]	
[THREAD_ANGLE]	
[THREAD_DEPTH]	
[THREAD_DIR]	
[THREAD_FINISH_NUMCUTS]	
[THREAD_FINISH_STOCK]	
[THREAD_FINISH_Z]	
[THREAD_FIRST_DEPTH]	
[THREAD_INFEED_TYPE]	

[THREAD_LENGTH]	
[THREAD_MAJOR_DIR]	
[THREAD_MIN_DEPTH]	
[THREAD_MINOR_DIA]	
[THREAD_PITCH]	
[THREAD_PULL_OUT_DIST]	The pull out value is specified under G76 parameters in threading mop in turning.
[THREAD_TAPER]	
[TIME_STAMP]	
[TOOL_ADJ_REG]	Tool Adjust Register number.
[TOOL_CHG_PT_X]	
[TOOL_CHG_PT_Y]	
[TOOL_CHG_PT_Z]	
[TOOL_CUTCOM_REG]	
[TOOL_DIA]	Tool Diameter.
[TOOL_LENGTH]	Tool length.
[TOOL_NAME]	
[TOOL_NUM]	Tool number.
[TOOL_NUM_FLUTES]	
[TOOL_RAD]	Tool Radius.
[TOOL_ZOFFSET]	
[TOOLPATH_MAX_X]	
[TOOLPATH_MAX_Y]	
[TOOLPATH_MAX_Z]	
[TOOLPATH_MIN_X]	
[TOOLPATH_MIN_Y]	
[TOOLPATH_MIN_Z]	

[VMPFILE_NAME]	
[WMPFILE_NAME_LONG]	
[WORK_OFFSET_CODE]	
[WORK_OFFSET_NUM]	
[WORK_OFFSET_PREFIX]	
[ZFEEDRATE]	
INV_TIME_FEEDRATE_FLAG	



Variables

The variables that can be used are listed in the table below.

The variables that start with “CYCL_” are used only for the cycles commands.

Variable	Comments
[ABS_ANGLE]	
[ABS_ANGLE_SECONDARY]	
[ANGLE]	
[ANGLE_PITCH]	
[ANGLE_ROLL]	
[ANGLE_SECONDARY]	
[ANGLE_YAW]	
[APPROACH_FEED]	
[CENTER_X]	
[CENTER_Y]	
[CENTER_Z]	
[CIR_PLANE]	
[COMMENT]	Output comments.
[COOLANT_CODE_INSERT]	
[COOLANT_OFF]	
[CURR_X]	The X coordinate of current point.

[CURR_X_WCS]	The X coordinate of current point in World Coordinates
[CURR_Y]	The Y coordinate of current point.
[CURR_Y_WCS]	The Y coordinate of current point in World Coordinates
[CURR_Z]	The Z coordinate of current point.
[CURR_Z_WCS]	The Z coordinate of current point in World Coordinates
[CUT_FEED]	
[CUT_MOTION_END_BLK]	
[CUT_MOTION_START_BLK]	
[CYCL_1/TPI]	1/TPI (only for TAP cycle)
[CYCL_CLEAR+DEPTH]	Drill Depth + Clear. (only for cycles except C-SINK)
[CYCL_CLEAR+INCR]	
[CYCL_CLEAR]	Clearance. (only for cycles)
[CYCL_CSINK_DEPTH+CLEAR]	Drill Depth + Clear. (only for C-SINK cycle)
[CYCL_CSINK_DEPTH]	Drill Depth. (only for C-Sink cycle)
[CYCL_DEPTH]	Drill Depth. (only for cycles except C-SINK)
[CYCL_DWELL]	Dwell. (only for cycles)
[CYCL_INCR]	Step Increment. (only for cycles)
[CYCL_IPM]	IPM. (only for cycles except TAP)
[CYCL_IPR]	IPR. (only for TAP cycle)
[CYCL_NEG_CLEAR+DEPTH]	-(Drill Depth + Clear). (only for cycles except C-SINK)
[CYCLE_NUM_STEPS]	
[CYCL_ORIENT]	Orient. (only for cycles)
[CYCL_SCALED_DWELL]	Dwell * Scale Factor (only for cycles)
[CYCL_TPI]	
[CYCLE_Z]	This variable stores the Z value of drill points.

[CYCL_Z+CLEAR]	Next Z + Clearance. (only for cycles)
[CYCL_Z+DEPTH+CLEAR]	Next Z + Depth + Clearance. (only for cycles)
[CYCL_Z+DEPTH]	Next Z + Depth. (only for cycles)
[CYCL_Z-DEPTH]	Next Z - Depth. (only for cycles)
[DELIMITER]	Delimiter definition.
[DEPART_FEED]	Outputs the Departure feedrate value for the currently operation.
[ENGAGE_FEED]	Outputs the Engage feedrate value for the currently operation.
[EOB]	The end of block character.
[EXTRUSION]	
[EXTRUSION_BLK]	
[EXTRUSION_CODE]	
[FEEDRATE]	FeedRate Value.
[FEEDRATE_BLK]	FeedRate Block.
[FEEDRATE_CODE]	Outputs the Feedrate code
[FEEDRATE_UNITS_CODE]	Outputs the Feedrate Units code
[FIRST_TOOL_NUM]	
[G_CODE]	The next G-Code. This is translated to linear, rapid, arc or cycle G-Code.
[HELIX_ANGLE]	
[HELIX_CCW_ARC]	
[HELIX_CW_ARC]	
[HELIX_LEAD]	
[HELIX_NUM_REV]	
[HELIX_RADIUS]	
[HELIX_TOTAL_DEPTH]	
[INPUTFILE_NAME]	
[INPUTFILE_NAME_LONG]	
[INV_TIME_FEEDRATE_OFF]	

[INV_TIME_FEEDRATE_ON]	
[LINEAR]	The linear motion code .
[MOP_NAME]	
[NEXT_ABS_X_WCS]	The next absolute X coordinate point in World Coordinates.
[NEXT_ABS_Y_WCS]	The next absolute Y coordinate point in World Coordinates.
[NEXT_ABS_Z_WCS]	The next absolute Z coordinate point in World Coordinates.
[NEXT_I]	
[NEXT_J]	
[NEXT_K]	
[NEXT_NONMDL_E]	
[NEXT_NONMDL_I]	
[NEXT_NONMDL_J]	
[NEXT_NONMDL_K]	
[NEXT_NONMDL_L]	
[NEXT_NONMDL_R]	
[NEXT_NONMDL_X]	The next non-modal X coordinate point in local Machine Coordinates.
[NEXT_NONMDL_X_WCS]	The next non-modal X coordinate point in World Coordinates.
[NEXT_NONMDL_Y]	The next non-modal Y coordinate point in local Machine Coordinates.
[NEXT_NONMDL_Y_WCS]	The next non-modal Y coordinate point in World Coordinates.
[NEXT_NONMDL_Z]	The next non-modal Z coordinate point in local Machine Coordinates.
[NEXT_NONMDL_Z_WCS]	The next non-modal Z coordinate point in World Coordinates.
[NEXT_R]	
[NEXT_TOOL_NAME]	
[NEXT_TOOL_NUM]	

[NEXT_X]	The next X coordinate point in Machine Coordinates.
[NEXT_X_WCS]	The next X coordinate point in World Coordinates.
[NEXT_Y]	The next Y coordinate point in Machine Coordinates.
[NEXT_Y_WCS]	The next Y coordinate point in World Coordinates.
[NEXT_Z]	The next Z coordinate point in Machine Coordinates.
[NEXT_Z_WCS]	The next Z coordinate point in World Coordinates.
[OUTPUT_MODE_CODE]	
[OUTPUT_UNITS_CODE]	English or Metric outputs code.
[OUTPUTFILE_NAME]	
[OUTPUT_FILENAME_LONG]	
[PARTNAME]	
[PARTNUM]	
[PLUNGE_FEED]	
[POST_NAME]	
[POST_NAME_LONG]	
[PREV_TOOL_ADJST_REG]	
[PREV_TOOL_CUTCOM_REG]	
[PREV_TOOL_NUM]	
[PREV_TOOL_NUM_FLUTES]	
[PREV_TOOL_ZOFFSET]	
[RAPID]	The rapid motion code .
[RAPID_FEED]	
[RETRACT_FEED]	
[ROTATION_AXIS]	
[ROTATION_AXIS_SECONDARY]	

[ROTATION_DIR]	
[ROTATION_DIR_SECONDARY]	
[ROTATION_FEEDVALUE]	
[ROTATION_MODE]	
[SEQ_PRECHAR]	Letter that is prefixed before the sequence number
[RT_NXT_X]	The next X coordinate. (Modal)
[RT_NXT_Y]	The next Y coordinate. (Modal)
[RT_NXT_Z]	The next Z coordinate. (Modal)
[RT_NXT_NONMDL_X]	The next X coordinate. (NonModal)
[RT_NXT_NONMDL_Y]	The next Y coordinate. (NonModal)
[RT_NXT_NONMDL_Z]	The next Z coordinate. (NonModal)
[SEQNUM]	The actual sequence number.
[SPINDLE_ARC]	Spindle direction.
[SPINDLE_BLK]	Spindle Block.
[SPINDLE_CODE]	
[SPINDLE_SPD]	Spindle speed.
[SPINDLE_SPD_MAX]	
[SPINDLE_SPD_TYPE]	
[SPIRAL_ANGLE]	
[SPIRAL_CCW_ARC]	
[SPIRAL_CW_ARC]	
[SPIRAL_END_RADIUS]	
[SPIRAL_LEAD]	
[SPIRAL_NUM_REV]	
[SPIRAL_START_RADIUS]	
[SPIRAL_TOTAL_LENGTH]	
[START_CHAR]	The program start character.
[START_POSITION_X]	

[START_POSITION_Y]	
[START_POSITION_Z]	
[START_X]	The X coordinate of start point.
[START_X_WCS]	The X coordinate of start point in World Coordinates.
[START_Y]	The Y coordinate of start point.
[START_Y_WCS]	The Y coordinate of start point in World Coordinates.
[START_Z]	The Z coordinate of start point.
[START_Z_WCS]	The Z coordinate of start point in World Coordinates.
[STEP_NEXT_X]	
[STEP_NEXT_Y]	
[STEP_NEXT_Z]	
[STEP_START_DEPTH]	
[STOCK_LENGTH_X]	UNDEFINED is output if there is no stock defined when posting occurs.
[STOCK_LENGTH_Y]	
[STOCK_LENGTH_Z]	
[STOCK_MAX_X]	
[STOCK_MAX_Y]	
[STOCK_MAX_Z]	
[STOCK_MIN_X]	
[STOCK_MIN_Y]	
[STOCK_MIN_Z]	
[STOP_CHAR]	The program end character.
[TEMPERATURE]	
[TEMPERATURE_BED_SET_CODE]	
[TEMPERATURE_BED_WAIT_CODE]	
[TEMPERATURE_EXTRUDER_SET_CODE]	
[TEMPERATURE_EXTRUDER_WAIT_CODE]	
[TEMPERATURE_SET_BLK]	
[TEMPERATURE_SET_CODE]	
[TEMPERATURE_WAIT_BLK]	

[TEMPERATURE_BED_WAIT_CODE]	
[THREAD_ANGLE]	
[THREAD_DEPTH]	
[THREAD_DIR]	
[THREAD_FINISH_NUMCUTS]	
[THREAD_FINISH_STOCK]	
[THREAD_FINISH_Z]	
[THREAD_FIRST_DEPTH]	
[THREAD_INFEED_TYPE]	
[THREAD_LENGTH]	
[THREAD_MAJOR_DIR]	
[THREAD_MIN_DEPTH]	
[THREAD_MINOR_DIA]	
[THREAD_PITCH]	
[THREAD_PULL_OUT_DIST]	The pull out value is specified under G76 parameters in threading mop in turning.
[THREAD_TAPER]	
[TIME_STAMP]	
[TOOL_ADJ_REG]	Tool Adjust Register number.
[TOOL_CHG_PT_X]	
[TOOL_CHG_PT_Y]	
[TOOL_CHG_PT_Z]	
[TOOL_CUTCOM_REG]	
[TOOL_DIA]	Tool Diameter.
[TOOL_LENGTH]	Tool length.
[TOOL_NAME]	
[TOOL_NUM]	Tool number.

[TOOL_NUM_FLUTES]	
[TOOL_RAD]	Tool Radius.
[TOOL_ZOFFSET]	
[VMPFILE_NAME]	
[WMPFILE_NAME_LONG]	
[WORK_OFFSET_NUM]	
[WORK_OFFSET_PREFIX]	
[ZFEEDRATE]	
INV_TIME_FEEDRATE_FLAG	



Related Topics

Other PPG Editor Sections:

[General](#)

[Start/End](#)

[Tool Change](#)

[Setup](#)

[Spindle](#)

[Feed Rate](#)

[Motion](#)

[Circle](#)

[Helical/Spiral Cycles](#)

[Multi Axis Motion](#)

[Cutter Compensation](#)

[Cut Motion Start/End](#)

[Cycles](#)

[Miscellaneous](#)

5.3 Macros

5.3.1 Macros

The following is a list of the available macros in the VisualCAD/CAM Post-Processor generator.

[StartEnd Tab](#)

These macros are output **before** and **after** a tool path and therefore Runtime variables cannot be used.

Program Start up Code Program End Code

[Tool Change Tab](#)

These macros are output only when a tool is **loaded** or **changed**.

First Load Tool Macro Tool Change Macro

[Cycles Tab](#)

These macros are output only when the motion command is a **cycle** command.

Drill Code	Deep Code	Break Chip Code
Counter Sink Code	Tap Code	Bore [Drag] Code
Bore [No Drag] Code	Bore [Manual] Code	Bore [Reverse] Code

5.3.2 How to edit Macros

This section describes the procedure for editing [macros](#).

You can directly type the macro. With the exception of '[' and ']' characters as variables.

[Usage of Variable List Dialog](#)

You can also add variables directly in the following manner.

- | | Operation | Dialog to operate |
|---|--|--------------------------------------|
| 1 | Click the right mouse button on the required edit box | Main Editor |
| 2 | Variable List Dialog is displayed. | Variable List Dialog |
| 3 | Select the position. | Main Editor |
| 4 | Select the variable to add. (This procedure is for adding)
Double click the left mouse button on the variable list and the highlighted variable is added. | Variable List Dialog |
| 5 | Use the [Add] or [Undo] or [Undo All] buttons to perform the standard add and undo operations. | Variable List Dialog |

Note : Multiple operations would require you to repeat steps 3 to 5.

Use Programmable Post

The Programmable Post Processor allows you to use Python script files for G-Code generation during toolpath processing by each MecSoft CAM plugin. You can create new post scripts or modify existing ones. Using the Python language for post scripts allows you to more efficiently generate the needed G-Code output allowing the use of programming elements, such as condition processing, loops, logical operations, and more.

You can find documentation and sample files on our programmable Post here:

<C:\Program Files\Rhino x\Plug-ins\RhinoCAM 20xx for Rx\SDK>

Find More Resources

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



2025 VisualCAD/CAM Resource Guide



The 2025 VisualCAD/CAM Resource Guide!

18 Pages

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

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

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

Good Things to Know

Here is a list of things you should know when posting G-Code using a customized post created from the [Post-Processor Generator](#) in [VisualCAD/CAM](#).



Algebraic Expressions when Posting

The [Post Process Generator](#) supports [Algebraic Expressions](#) in all input fields.

Here are some guidelines for using expressions:

1. Each expression should be placed in 'E{', 'E}' tags.
2. In expression can be used next operations: -, +, /, *, ^
3. Negative values should be placed in parentheses '()'
4. Expression parts can be placed in parentheses '()'
5. Floating point numbers should be delimited by point symbol, use 0.xx in case of fractional numbers
6. Expressions can contain spaces in any place, spaces will be removed while parsing
7. Numbers in [-0.9; 0.9] can be written as [-.9; .9]

Examples:

- E{ ([SOME_VAR1]/2 + ([SOME_VAR]*(-3.2)))^3 E}
- E{[SOME_VAR1] + .3 E} SOME_TEXT E{ [SOME_VAR1] *(-1) E}



Posting Drill Cycles & Indexed Machining

Drill cycles will be converted to simulated cycles (i.e., using linear motions) if the setup the drill cycles appear in is not aligned with the machine Z axis. This is done only when the machine has a head configuration defined and [Output all coordinates in local Setup Coordinate System](#) is not checked. See [Machine Tool Setup](#) for more information.



Posting Cutter Compensation (G40, G41, G42)

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 [VisualCAD/CAM](#). 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 [VisualCAD/CAM](#).

[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

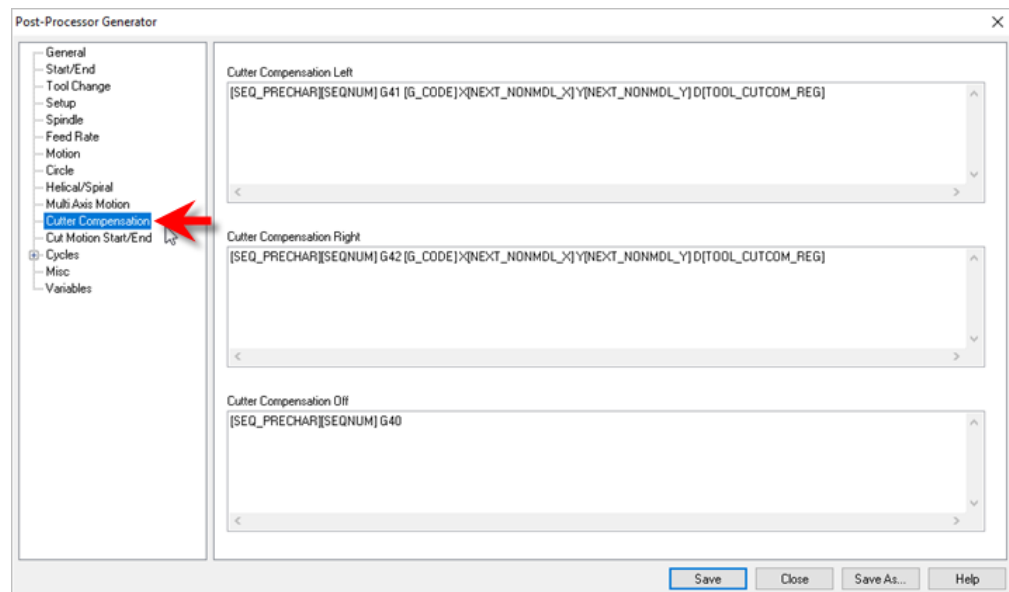
```
[SEQ_PRECHAR][SEQNUM] G41 [G_CODE] X[NEXT_NONMDL_X] Y[NEXT_NONMDL_Y]  
D[TOOL_CUTCOM_REG]
```

Cutter Compensation Right

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

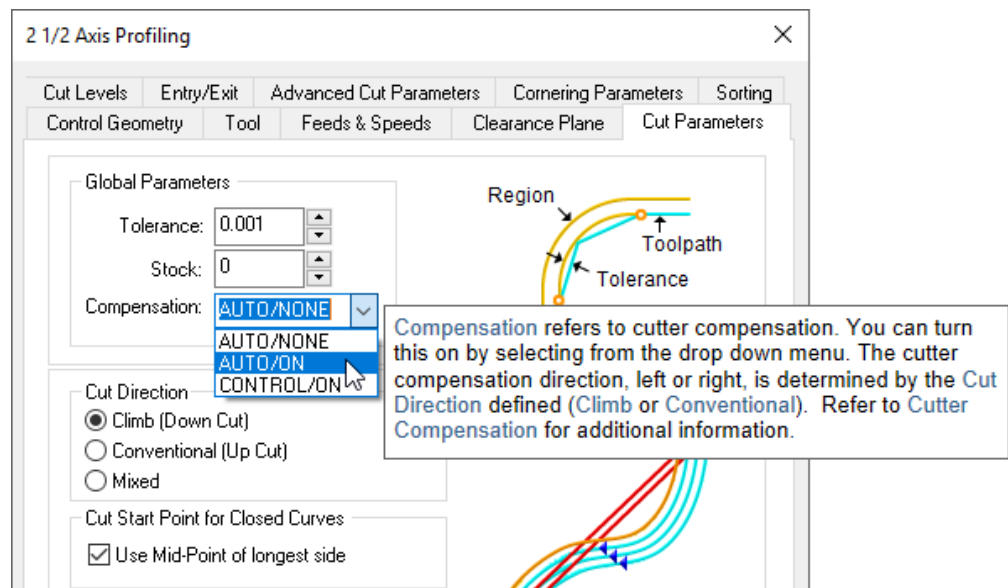
Cutter Compensation Off

```
[SEQ_PRECHAR][SEQNUM] G40
```



A few things to watch out for:

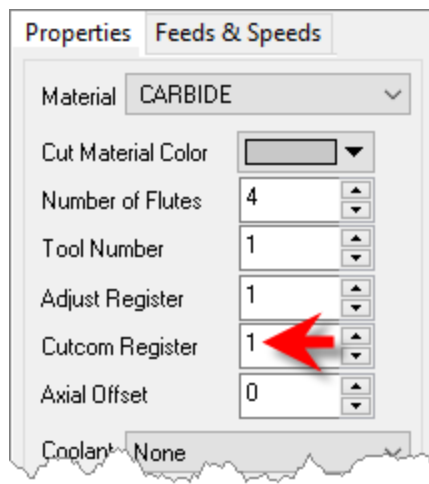
1. [Cutter compensation](#) makes sense only in 2-1/2 axis operations. If you are using roughing (pocketing & facing) the compensation will be turned on only in the final passes.
2. Make sure you are using [Climb](#) or [Conventional](#) cut traversal in any of the methods that you want to turn compensation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



- Make sure you have a linear motion for the controller to turn on the compensation for. If your first motion is an arc the controller will not be able to turn on the compensation. Thus, in **2-1/2 axis profiling**, make sure there is a linear entry motion for the controller to be able to turn compensation on & linear exit to turn off compensation.

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

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



Posting 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:

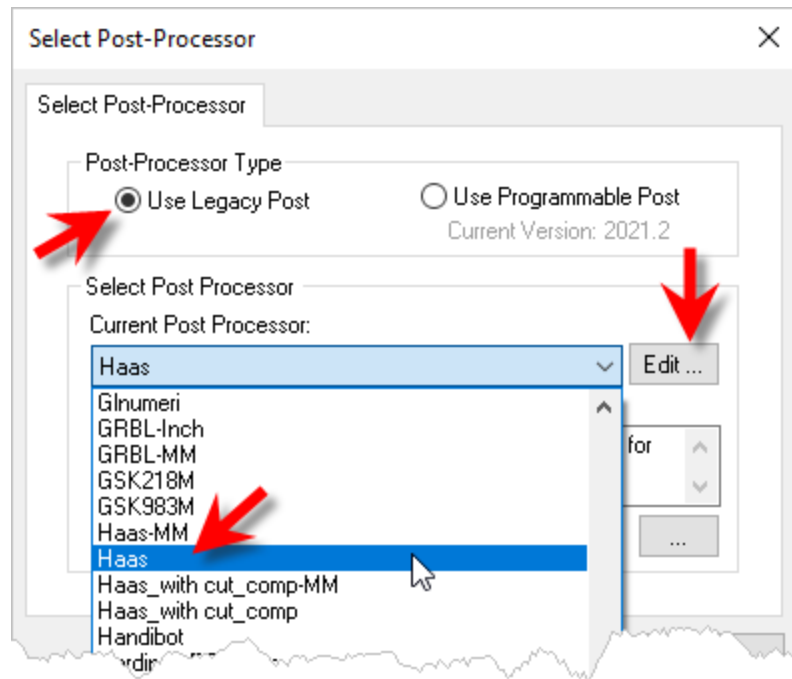


For 2 and 3 Axis Output

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:

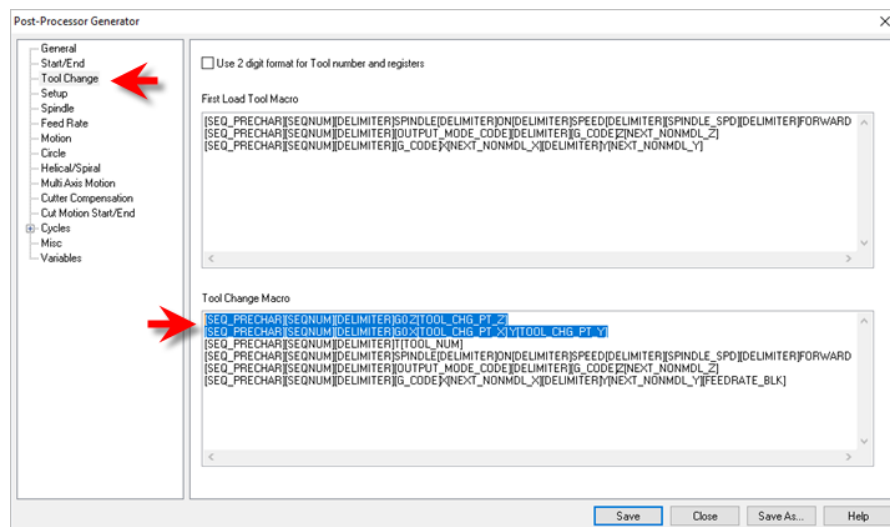
X: -4, Y: 0 Z: 0

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.

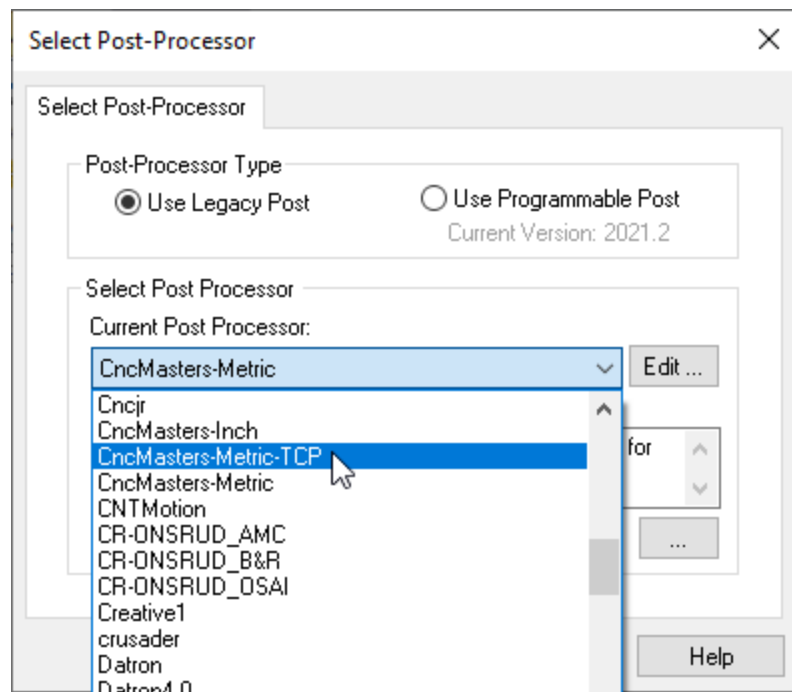
```
[SEQ_PRECHAR][SEQNUM][DELIMITER]G0 Z[TOOL_CHG_PT_Z]
[SEQ_PRECHAR][SEQNUM][DELIMITER]G0 X[TOOL_CHG_PT_X] Y[TOOL_CHG_PT_Y]
[SEQ_PRECHAR][SEQNUM][DELIMITER]T[TOOL_NUM]
...
...
```



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
...
...

```

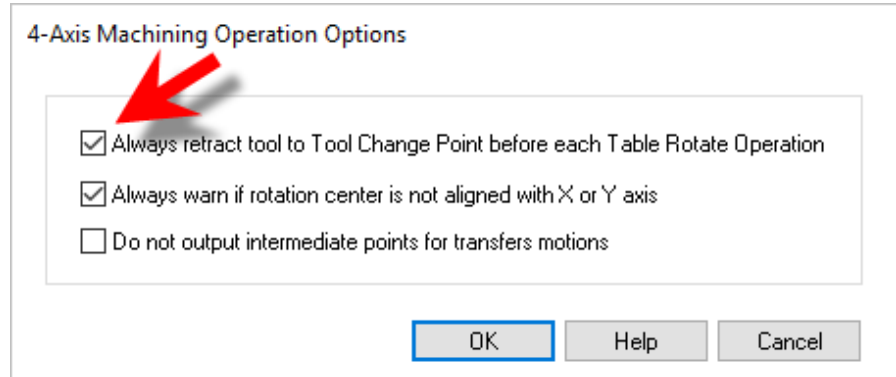
15. That's it!



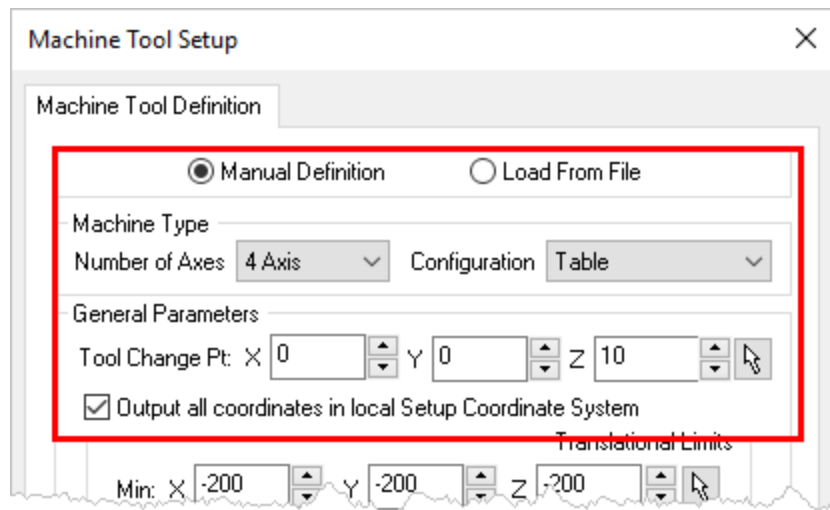
For 4 Axis Output

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.

...
...

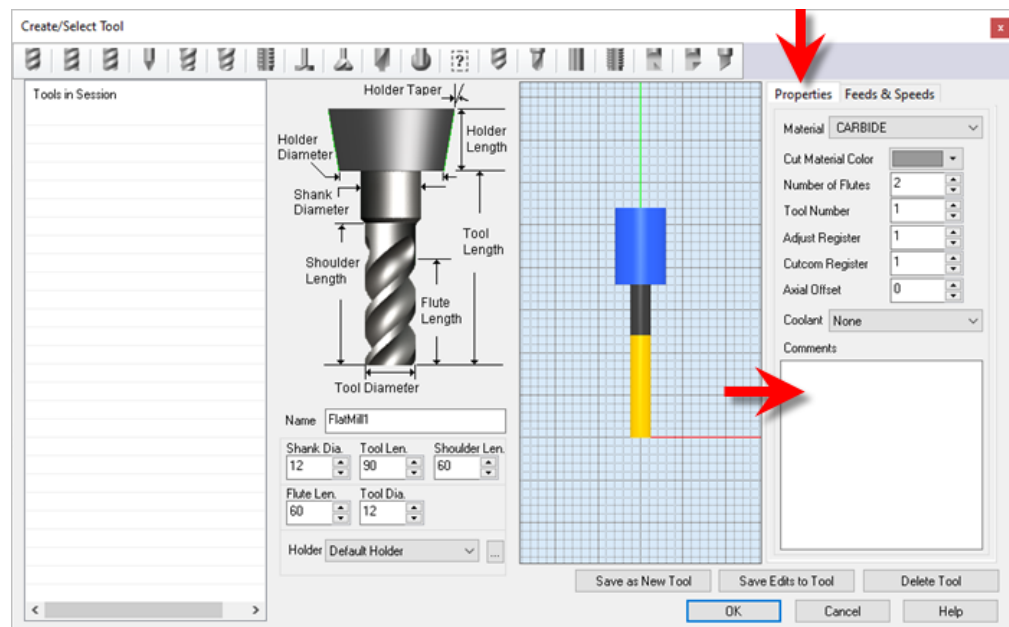


Posting Tool Comments

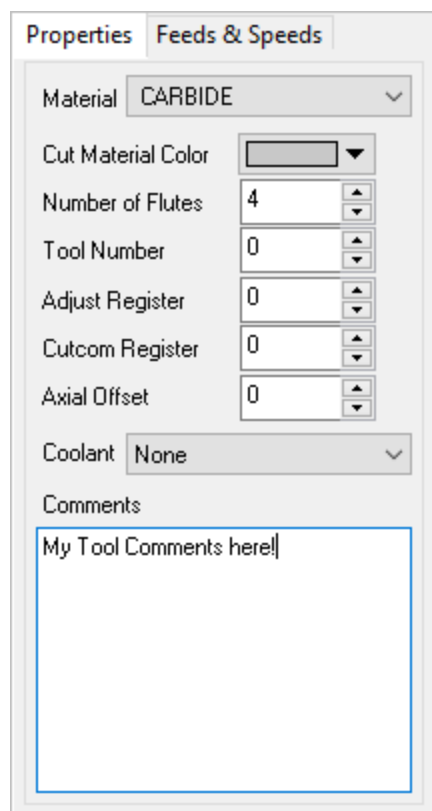
You can add comments associated with a **Tool**. These **Comments** are saved with the **Tool** in your **Tool Library**. They are also posted to your g-code when the tool is used.

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.

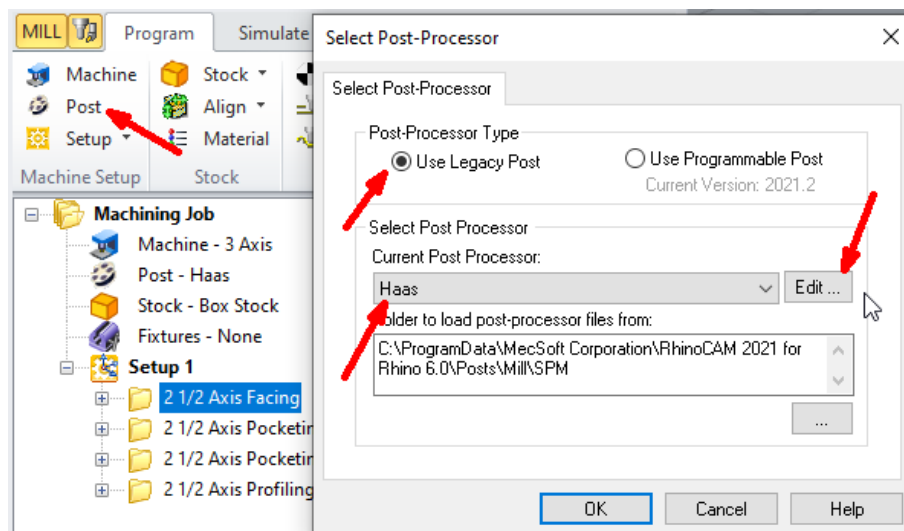


3. Add text to the **Comments** window.



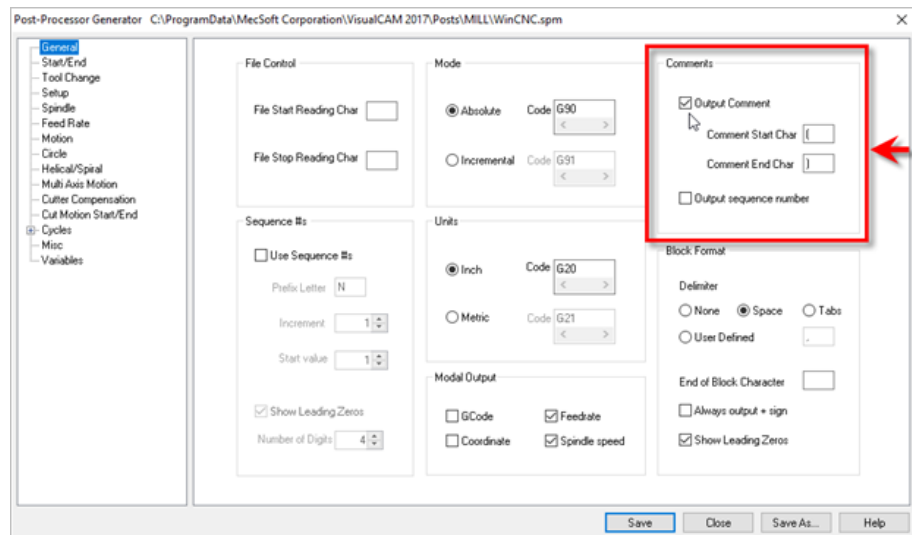
4. Make sure [Comments](#) are enabled in your post.

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.

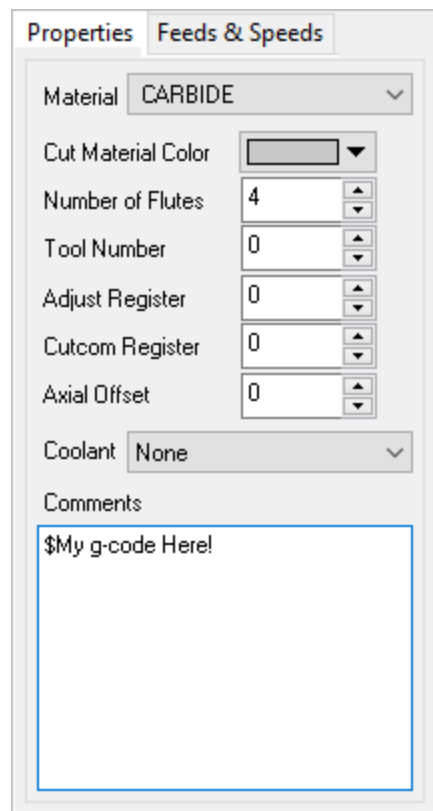


D. Then pick **Save** or **Save As**.

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

...

...



Related Topics

[Post Processor File Browser](#)

[Main Editor](#)

[Variable List Dialog](#)

Index

- A -

Algebraic Expressions 8, 78
 arc center [i,j,k] 43
 arc motion block 43

- B -

block format 27

- C -

Circle 43
 comments (o/p, start, end, seq) 27
 comments on output 27
 compensation 54
 coolant code 54
 Cutter Compensation 8, 78
 Cycles 52

- D -

delimiter 27
 Drill Cycles & Indexed Machining 8, 78

- F -

Feed Rate 38
 feed rate value 38
 feedrate block 38
 file control 27
 file select dialog 23
 format 43

- H -

How to edit 75

- I -

Introduction 3

- L -

Legacy Post 20
 linear motion block 41

- M -

Macros 74
 main editor 24
 Miscellaneous 54
 modal output 27
 mode 27
 More Help 77
 Motion 41
 motion coordinates 41
 motion order 41

- P -

plane 43
 Post
 Use Legacy 20
 Programmable Post
 About 76

- S -

sample output (feedrate) 38
 sample output (motion) 41
 sample output (spindle) 36
 sequence number 27
 Spindle 36
 spindle block 36
 spindle direction 36
 spindle RPM 36
 StartEnd 31

- T -

Tool Change Point 8, 78
 Tool Comments 8, 78
 ToolAxis 47
 ToolChange 32

- U -

units 27

- V -

variable list dialog 26

Variables 56