

# Post-Process (PPG) Reference Guide

## *VisualCAM-PPG 2026*

Published: March 2026

**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 1998-2026

# Table of Contents

<b>Welcome</b>	<b>4</b>
<b>Quick Start</b>	<b>5</b>
<b>Resource Guide</b>	<b>8</b>
<b>Good Things to Know</b>	<b>9</b>
<b>Use Legacy Post</b>	<b>21</b>
<b>1 Dialogs</b> .....	<b>24</b>
Post Processor File Browser .....	24
PPG Editor .....	25
Variable List Dialog .....	27
<b>2 PPG Editor</b> .....	<b>28</b>
General .....	28
StartEnd .....	32
Tool Change .....	33
Setup .....	36
Spindle .....	37
Feed Rate .....	39
Motion .....	42
Circle .....	44
Helical/Spiral .....	46
Multi Axis Motion .....	48
Cutter Compensation .....	50
Cut Motion Start/End .....	51
Cycles .....	53
Miscellaneous .....	55
Variables .....	57
Introduction.....	62
PPG > Feed Rate.....	63
PPG > General.....	65
PPG > Helical/Spiral.....	66
PPG > Misc.....	68
PPG > Motion.....	70
PPG > Multi Axis Motion.....	71
PPG > Setup.....	72
PPG > Spindle.....	73
PPG > Start/End.....	75
PPG > Tool Change.....	77
PPG > Variables.....	81
PPG Variables & Macros.....	81
<b>3 Macros</b> .....	<b>81</b>
Macros .....	81
How to edit Macros .....	82

<b>Use Programmable Post</b>	<b>83</b>
<b>Find More Resources</b>	<b>84</b>
<b>Good Things to Know</b>	<b>85</b>
<b>Index</b>	<b>97</b>

## Welcome



# PPG Module 2026

[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 Thinks to Know](#)

[Post Processor File Browser](#)

[Main Editor](#)

[Variable List Dialog](#)

## Quick Start



# PPG Module 2026

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

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

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



### What's New!

[What's New in VisualCAD/CAM 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

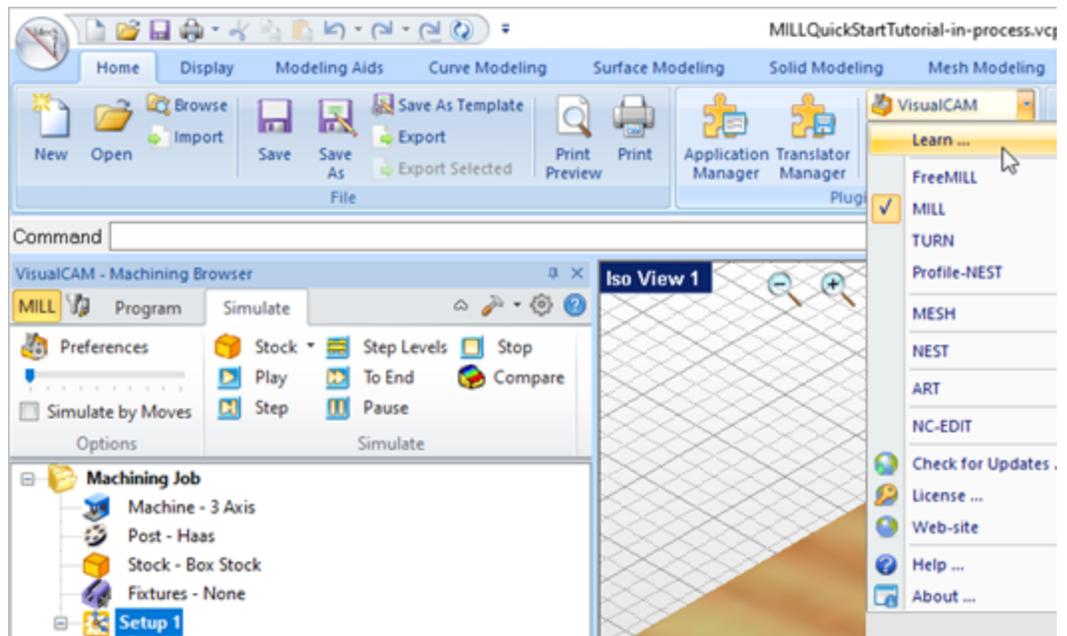
To help you quickly get started in working with each module, select one of the Help buttons located on the [VisualCAD/CAM Learning Resources](#) dialog.

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:

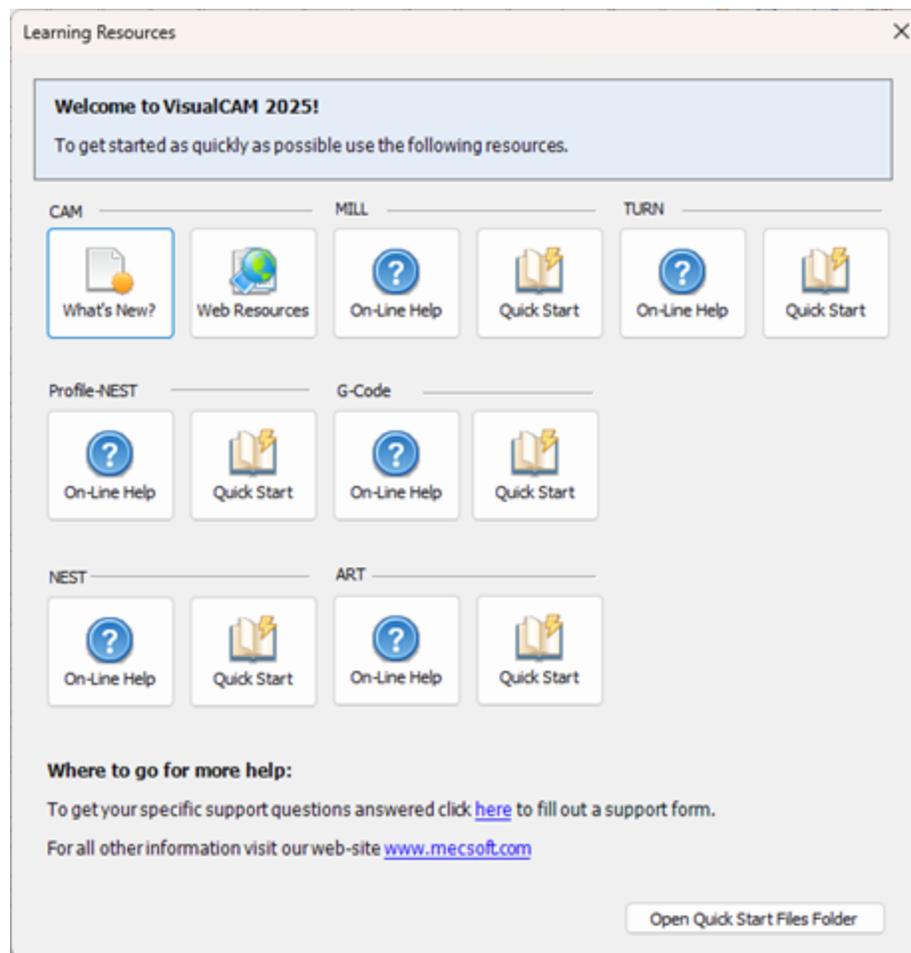
1. From the [VisualCAD Home Ribbon Bar](#), drop down the Main menu and select [Learn ...](#)



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

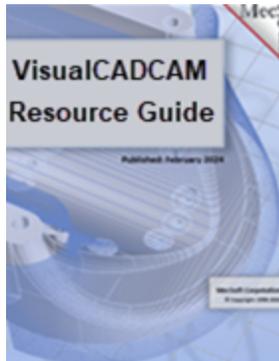
[Find More Resources](#)

## Resource Guide

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



### 2025 VisualCAD/CAM Resource Guide



### The 2026 VisualCAD/CAM Resource Guide!

*18 Pages*

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

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

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

## 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**

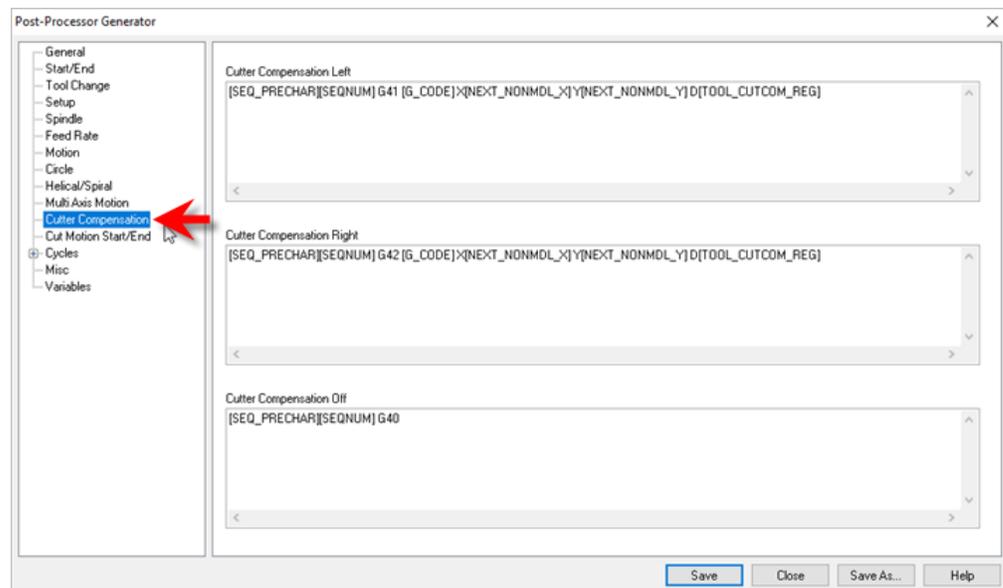
```
[SEQ_PRECHAR][SEQNUM] G41 [G_CODE] X[NEXT_NONMDL_X] Y[NEXT_NONMDL_Y]
D[TOOL_CUTCOM_REG]
```

**Cutter Compensation Right**

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

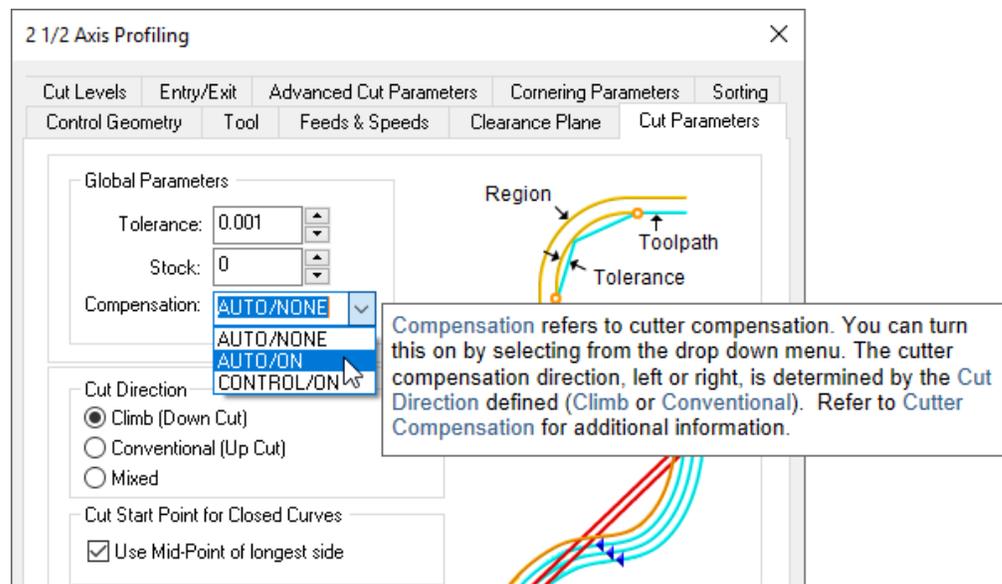
**Cutter Compensation Off**

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



**A few things to watch out for:**

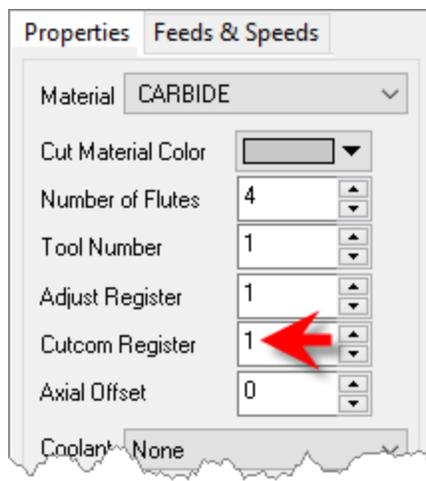
1. [Cutter compensation](#) makes sense only in 2-1/2 axis operations. If you are using roughing (pocketing & facing) the compensation will be turned on only in the final passes.
2. Make sure you are using [Climb](#) or [Conventional](#) cut traversal in any of the methods that you want to turn compensation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



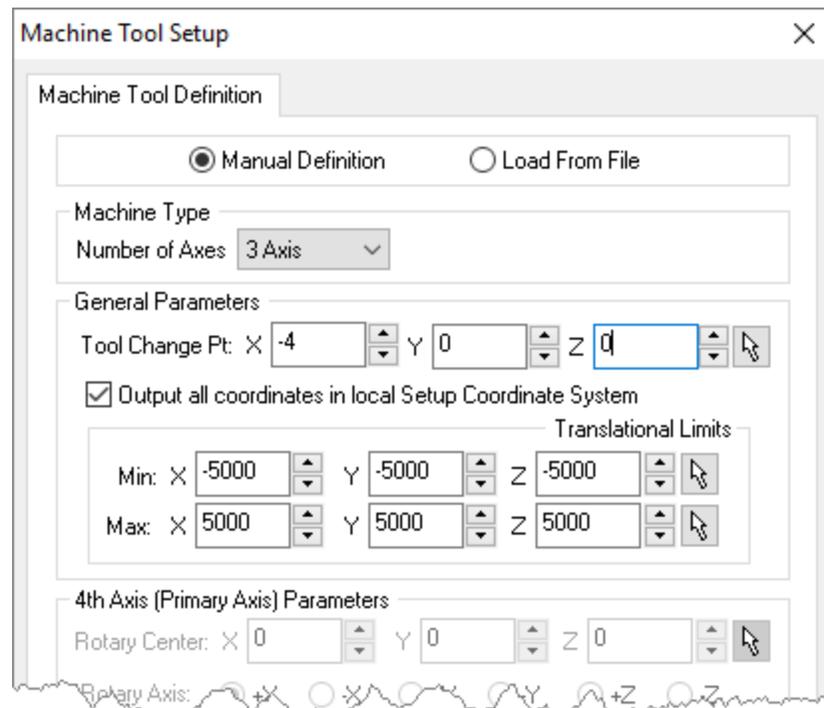
## Posting a Tool Change Point

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:

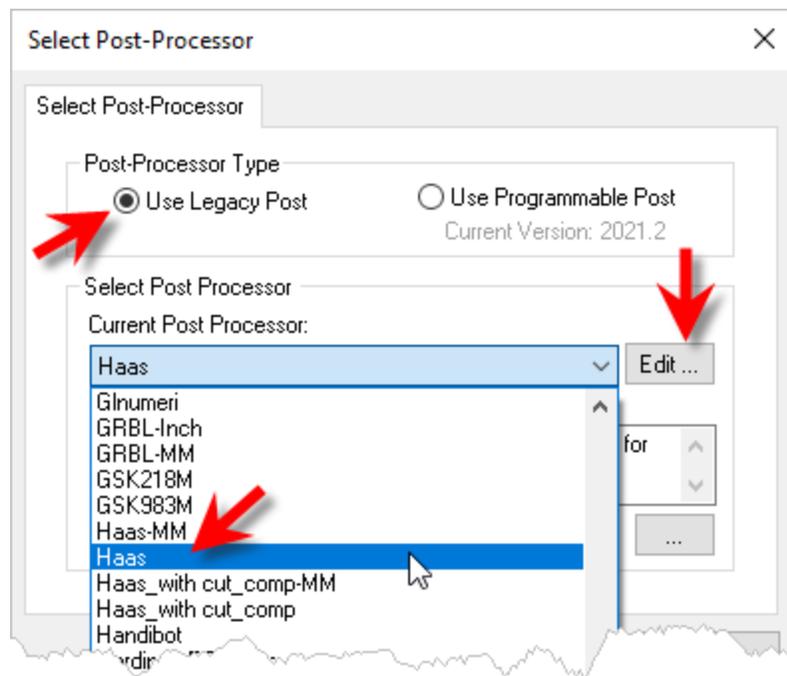
#### For 2 and 3 Axis Output

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**

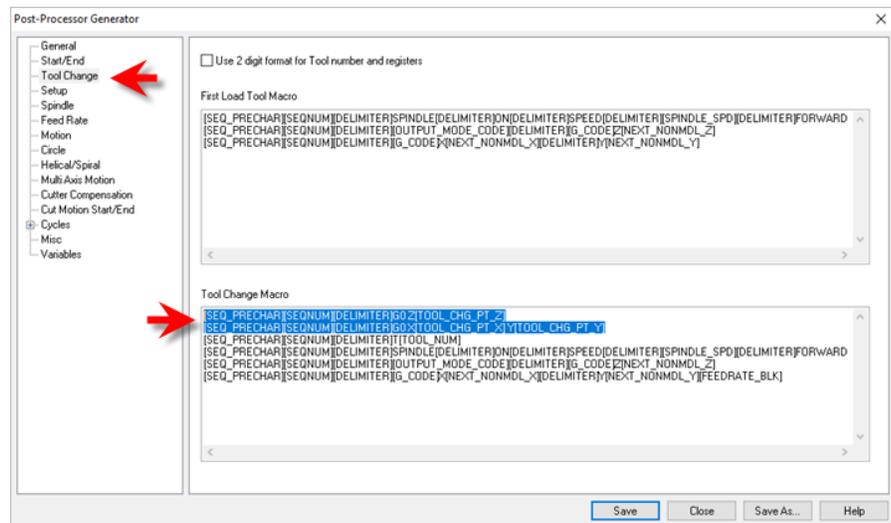


3. Edit your post processor by selecting [Program](#) tab > [Post](#) > [Edit](#).



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.

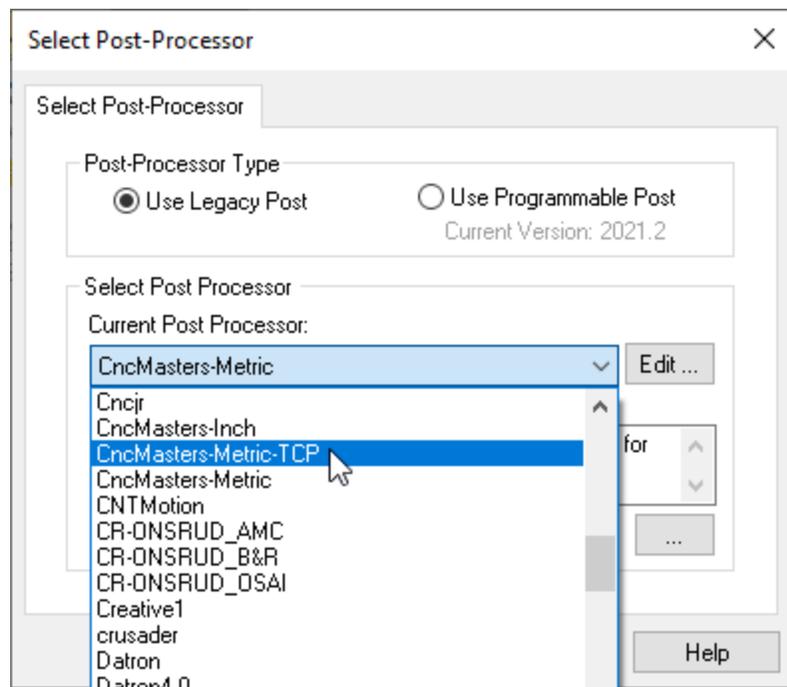
```
[SEQ_PRECHAR][SEQNUM][DELIMITER]GO Z[TOOL_CHG_PT_Z]
[SEQ_PRECHAR][SEQNUM][DELIMITER]GO X[TOOL_CHG_PT_X] Y[TOOL_CHG_PT_Y]
[SEQ_PRECHAR][SEQNUM][DELIMITER]T[TOOL_NUM]
...
...
```



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

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

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



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

```

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

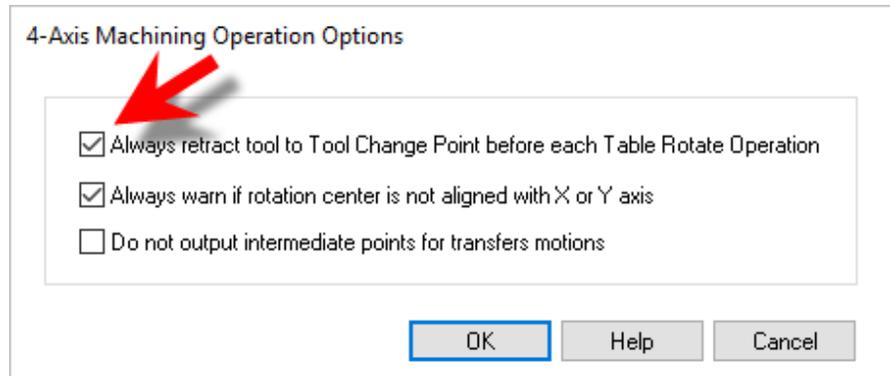
```

15. That's it!

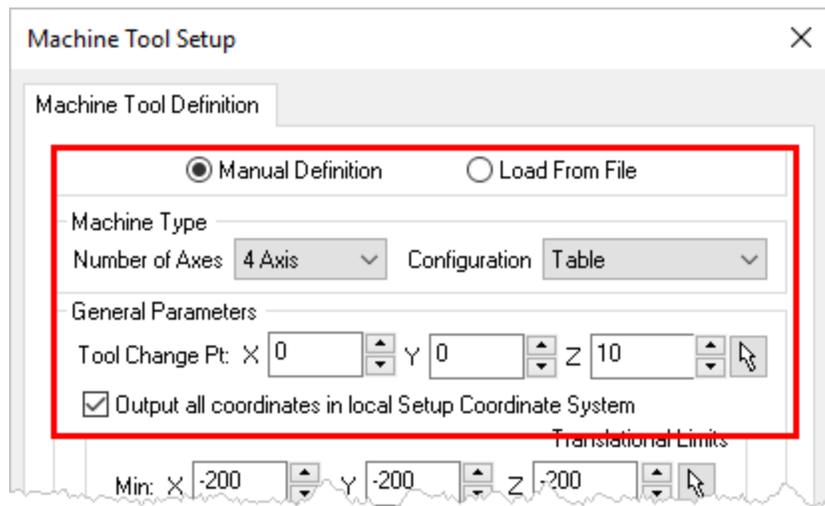
### For 4 Axis Output

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.

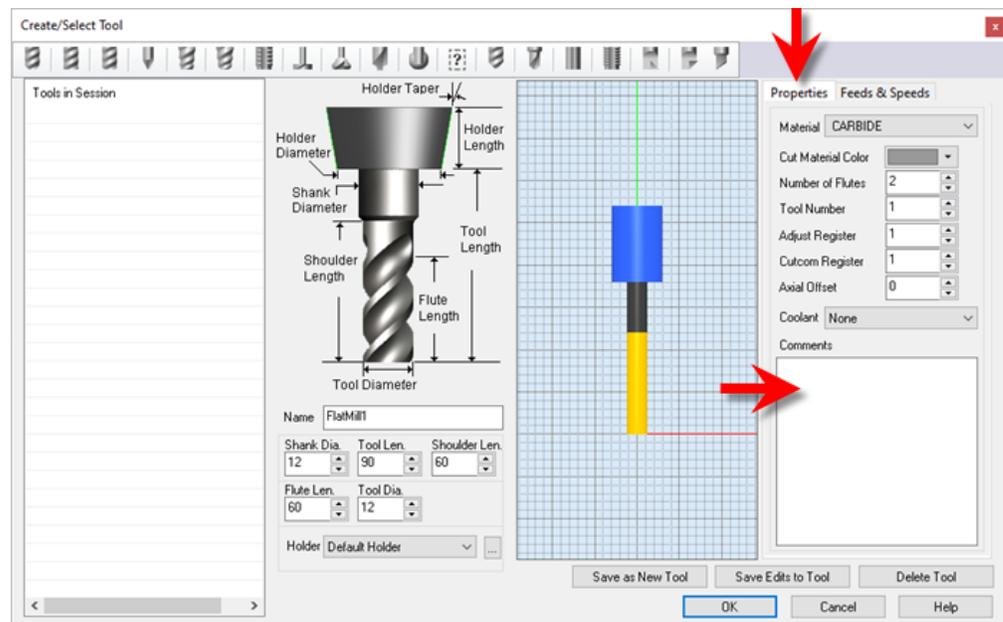
...  
...

## Posting Tool Comments

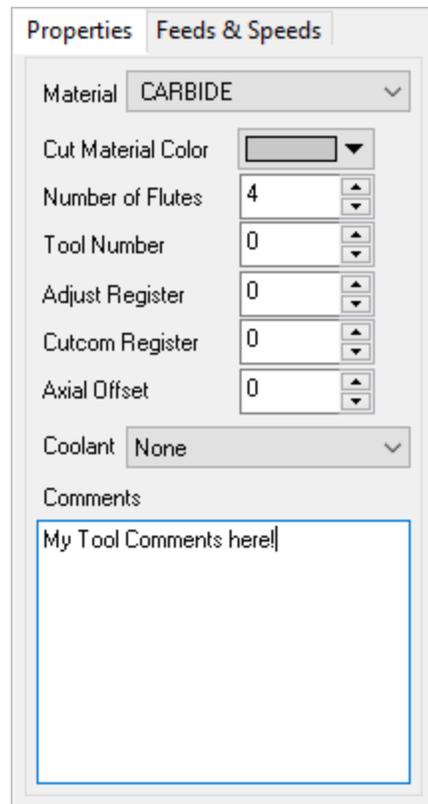
You can add comments associated with a **Tool**. These **Comments** are saved with the **Tool** in your **Tool Library**. They are also posted to your g-code when the tool is used.

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.

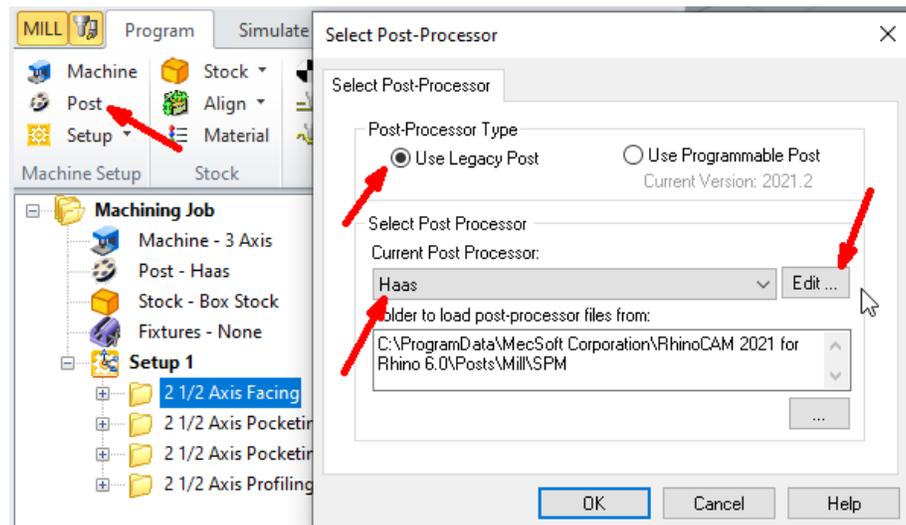


3. Add text to the **Comments** window.



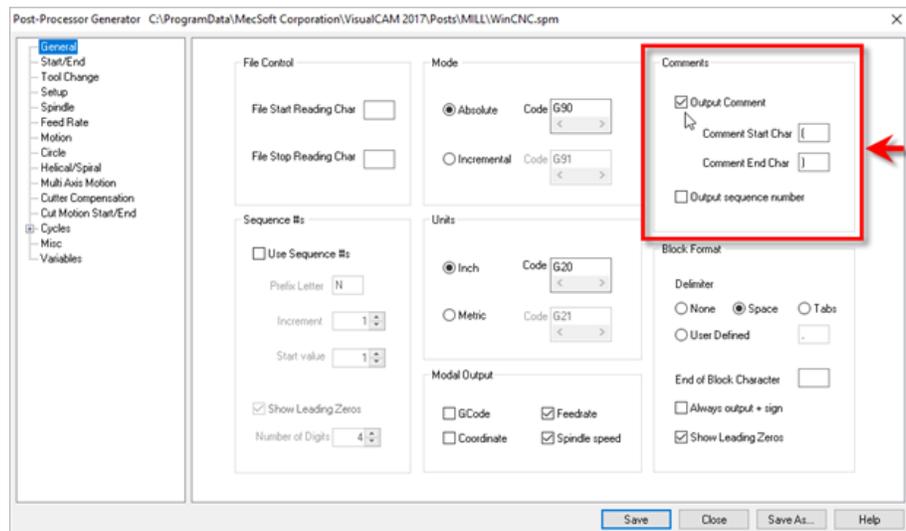
4. Make sure [Comments](#) are enabled in your post.

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 

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](#)

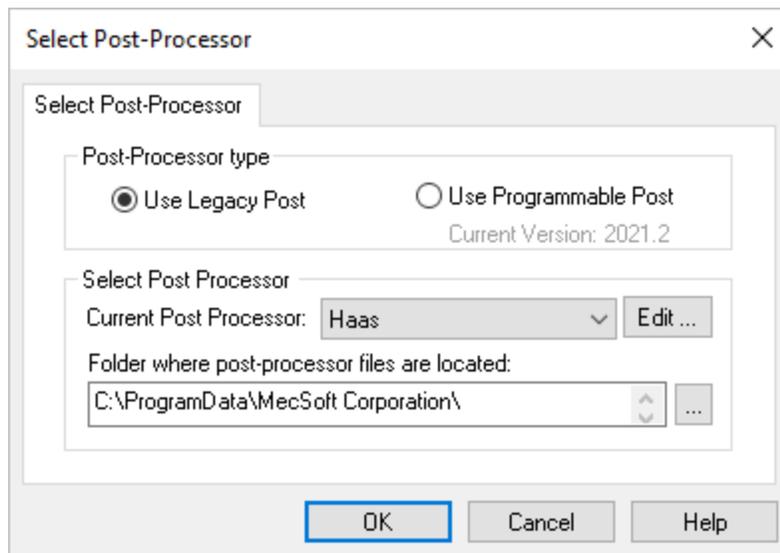
[Main Editor](#)

[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.

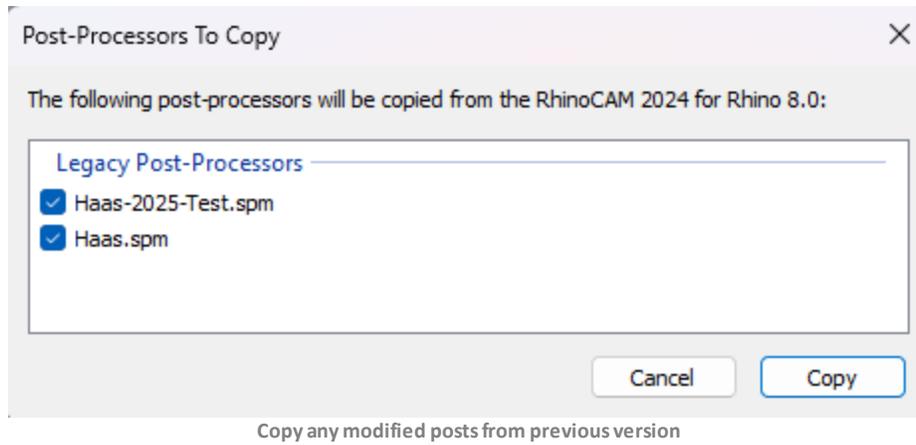
### Browse for folder



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.



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.

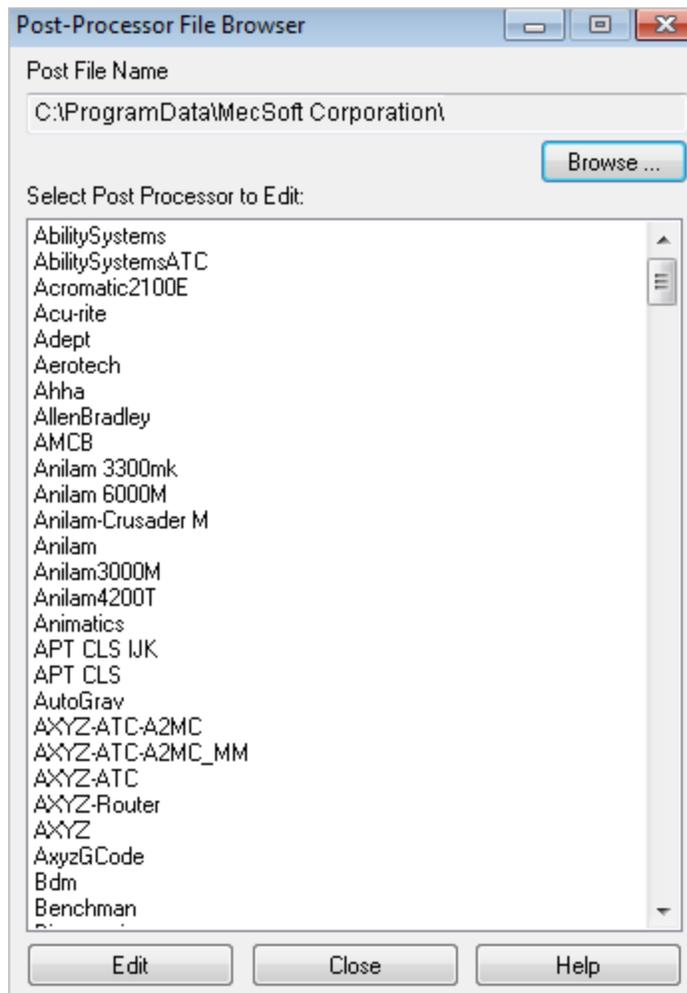
 **NOTE:** It is advisable that you backup all of your post-processor definition files and keep them in a safe location outside of the plug-in installation path so that you can access them manually if needed.

 **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**

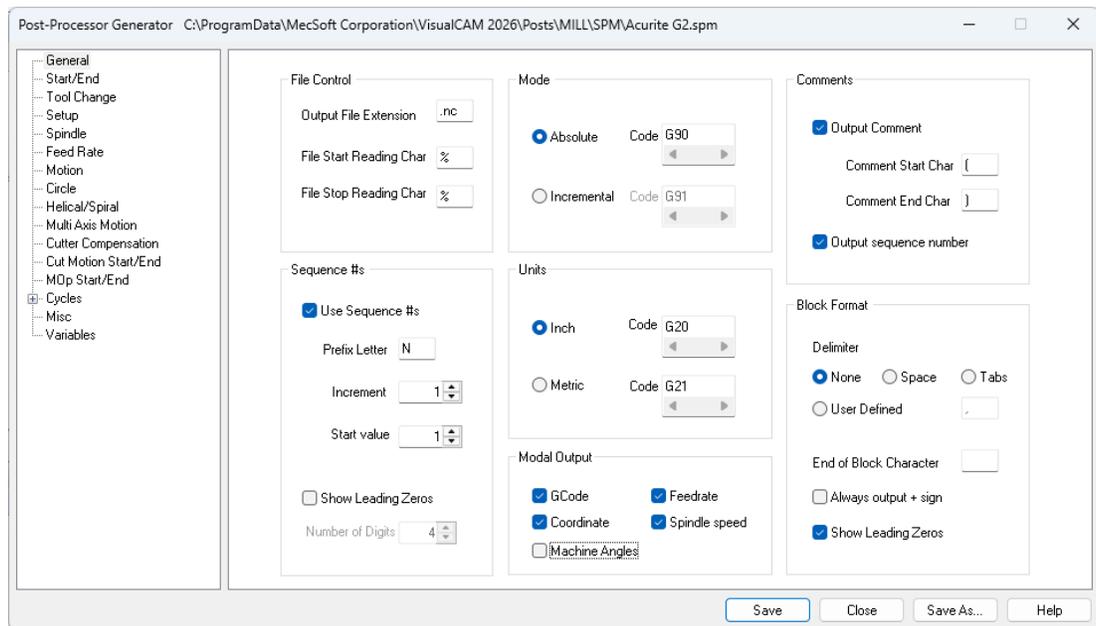
Dialog Box: Post Processor File Browser

 **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**



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](#)

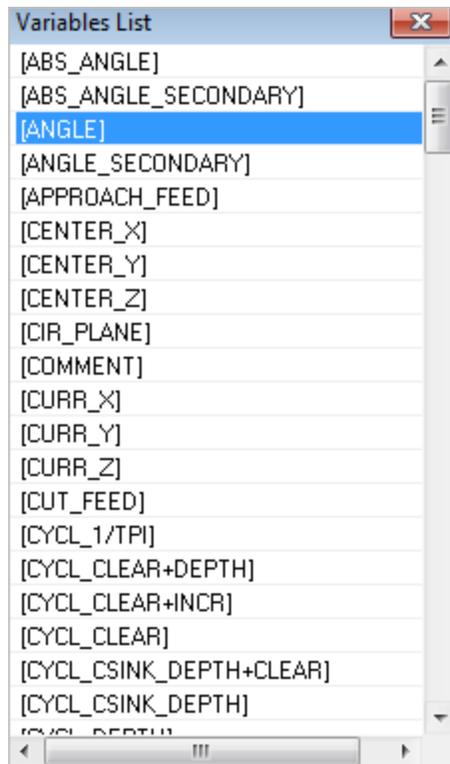
[Variable List Dialog](#)

### 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](#)

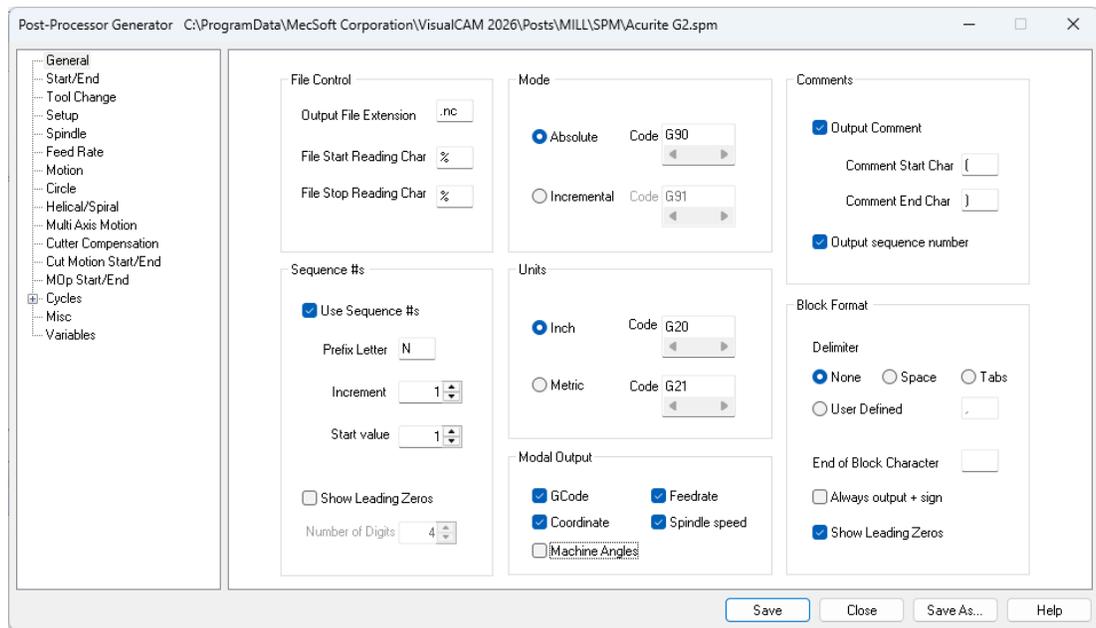
## 5.2 PPG Editor

### 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



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](#)

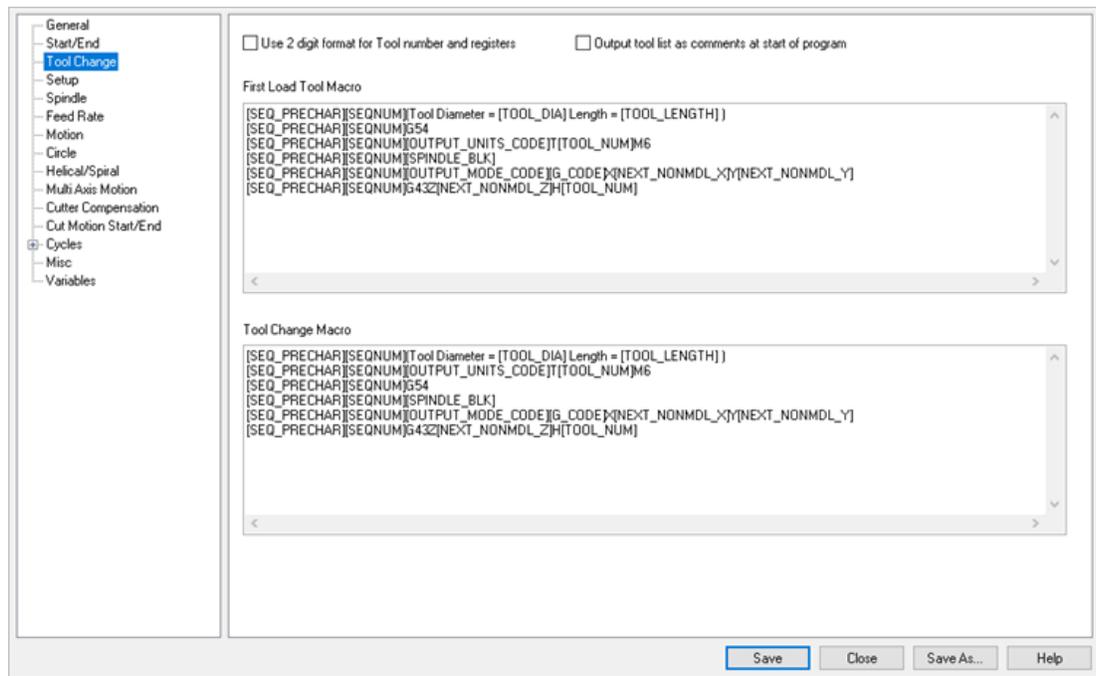
[Variables](#)

## 5.2.3 Tool Change

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



### PPG Editor: Tool Change



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 Marco

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](#)

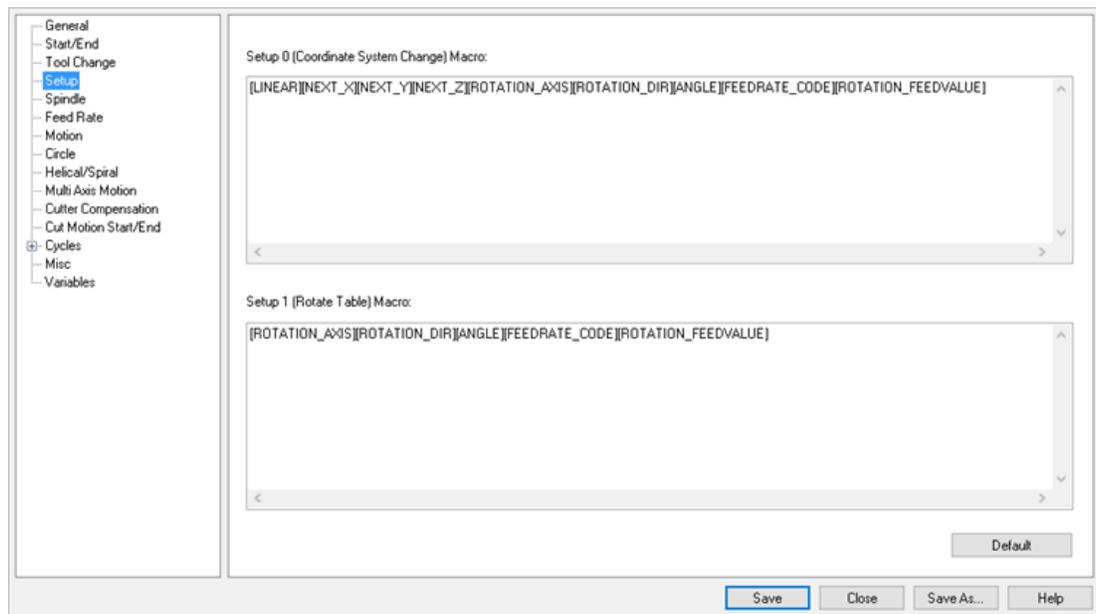
[Miscellaneous](#)

[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

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
  - Unites/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](#)

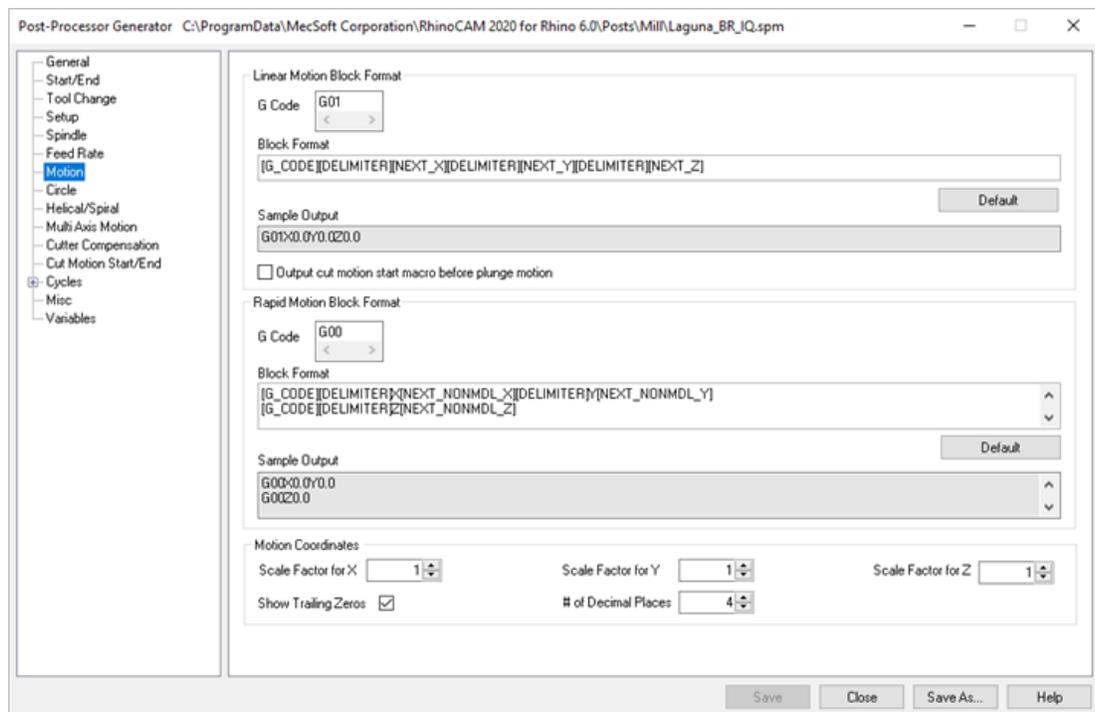
[Miscellaneous](#)

[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



Post-Processor Generator C:\ProgramData\MecSoft Corporation\RhinoCAM 2020 for Rhino 6.0\Posts\Mill\Laguna\_BR\_IQ.spm

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

Linear Motion Block Format

G Code

Block Format  
[G\_CODE][DELIMITER][NEXT\_X][DELIMITER][NEXT\_Y][DELIMITER][NEXT\_Z]

Sample Output  
G01:X.0Y0.0Z0.0

Output cut motion start macro before plunge motion

Rapid Motion Block Format

G Code

Block Format  
[G\_CODE][DELIMITER][NEXT\_NONMDL\_X][DELIMITER][NEXT\_NONMDL\_Y]  
[G\_CODE][DELIMITER][NEXT\_NONMDL\_Z]

Sample Output  
G00:X.0Y0.0  
G00Z0.0

Motion Coordinates

Scale Factor for X  Scale Factor for Y  Scale Factor for Z

Show Trailing Zeros  # of Decimal Places

Save Close Save As... Help

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**

<b>G Code</b> Clockwise Arc Code <input type="text" value="G02"/> C-clockwise Arc Code <input type="text" value="G03"/>		<b>Plane Code</b> XY <input type="text" value="G17"/> ZX <input type="text" value="G18"/> YZ <input type="text" value="G19"/>	
<b>Output Format</b> <input type="radio"/> I, J, K, and Radius <input checked="" type="radio"/> I, J, K only <input type="radio"/> Radius only <input type="checkbox"/> Output values only when different <input type="checkbox"/> Use -R for CW Arcs		<b>Arc Center (I, J, K)</b> <input type="radio"/> Absolute <input type="radio"/> Vector from Center to Start <input checked="" type="radio"/> Vector from Start to Center <input type="radio"/> Unsigned Vector from Start to Center	
<b>Block Format</b> 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			
<input type="checkbox"/> Limit Arcs to Angle <input type="text" value="180"/>		<input type="button" value="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](#)

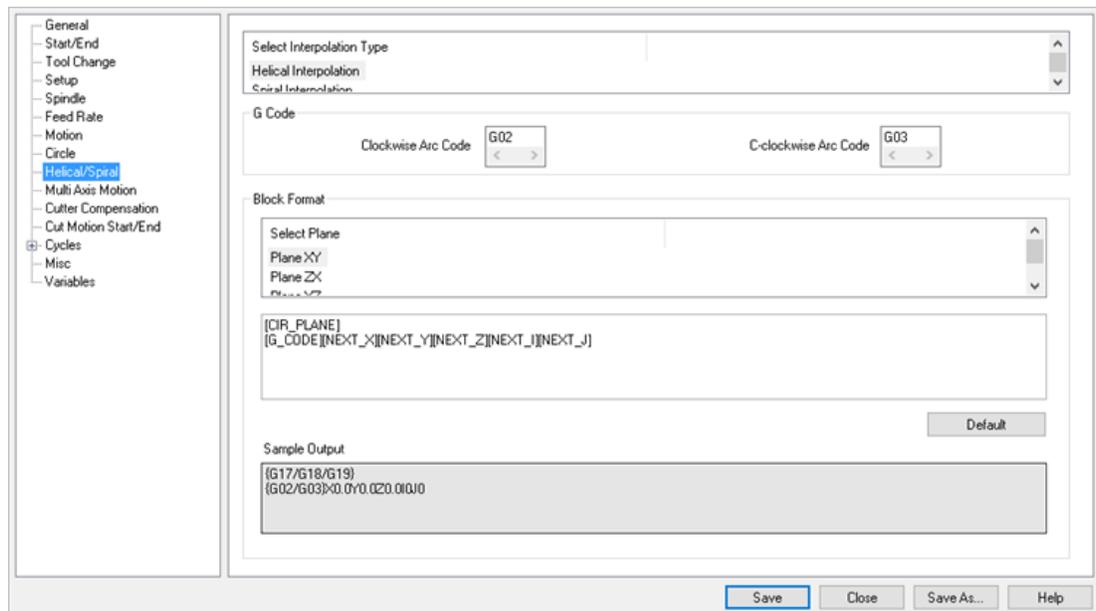
[Variables](#)

## 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](#)

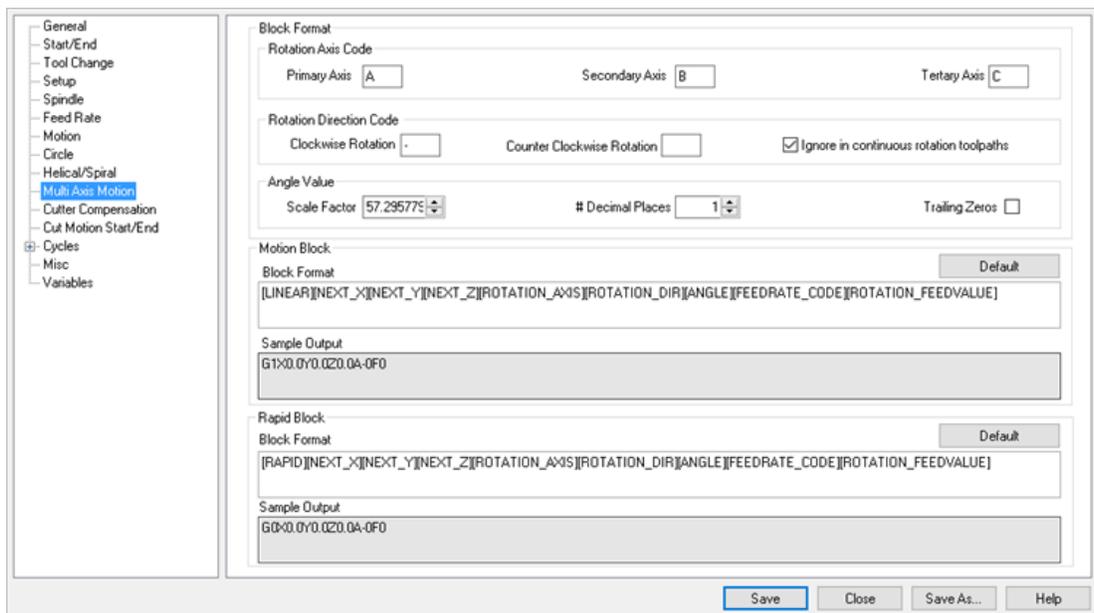
[Start/End](#)

[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](#)

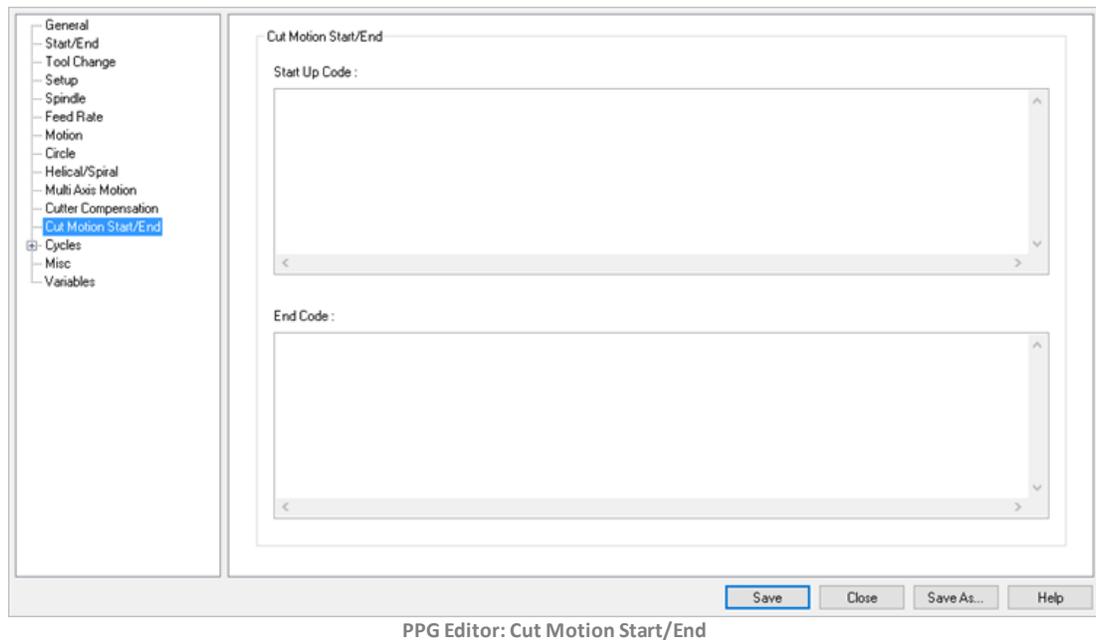
[Miscellaneous](#)

[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**



### 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](#)

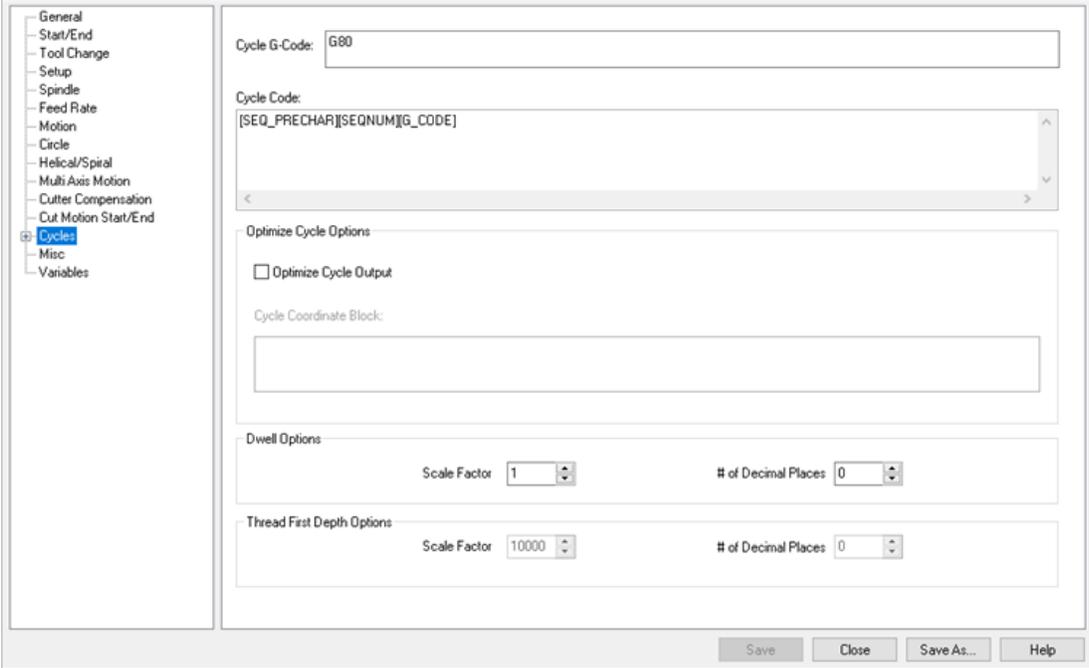
[Miscellaneous](#)

[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](#)

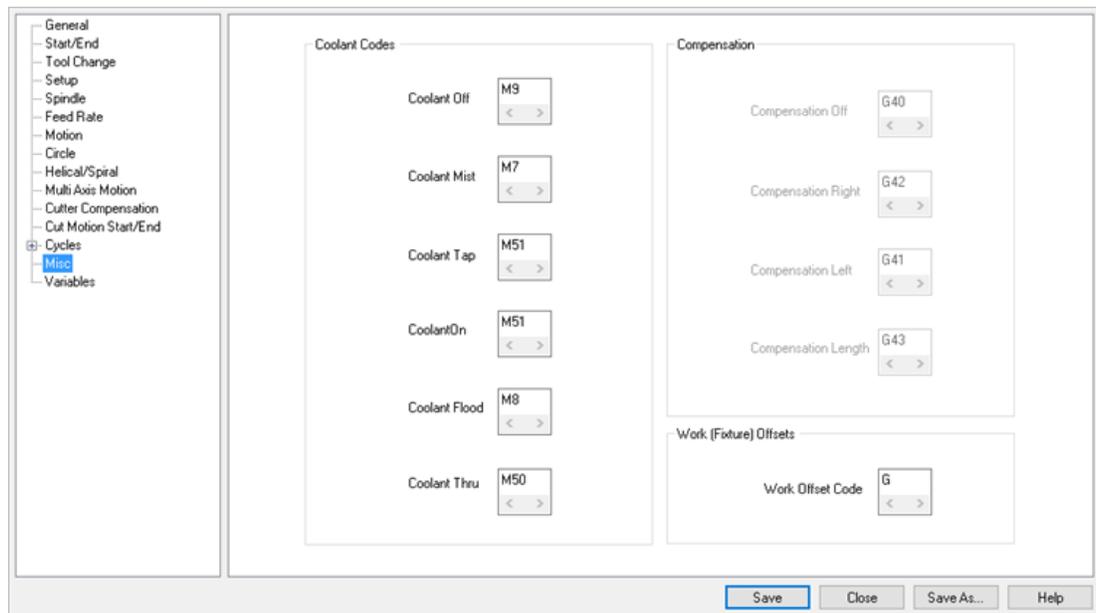
[Miscellaneous](#)

[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](#)

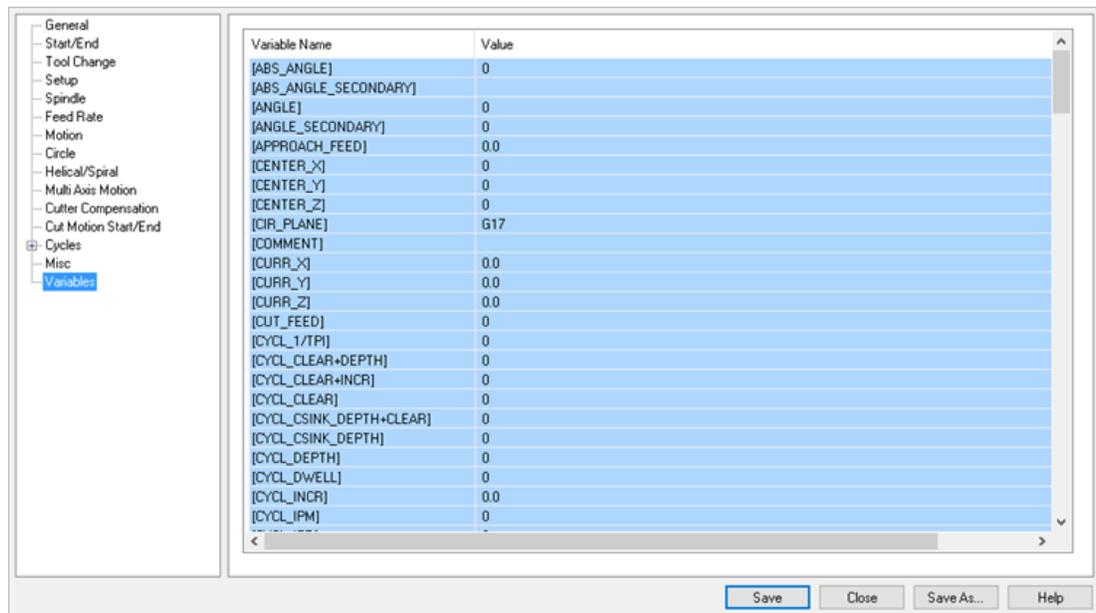
[Variables](#)

### 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]	<a href="#">Output comments.</a>
[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)



## 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]	<a href="#">Output comments.</a>
[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)



## 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](#)

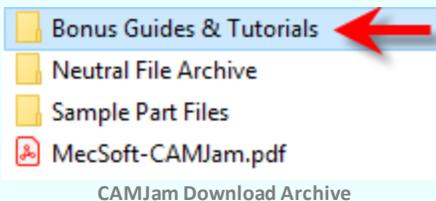
[Cycles](#)

[Miscellaneous](#)

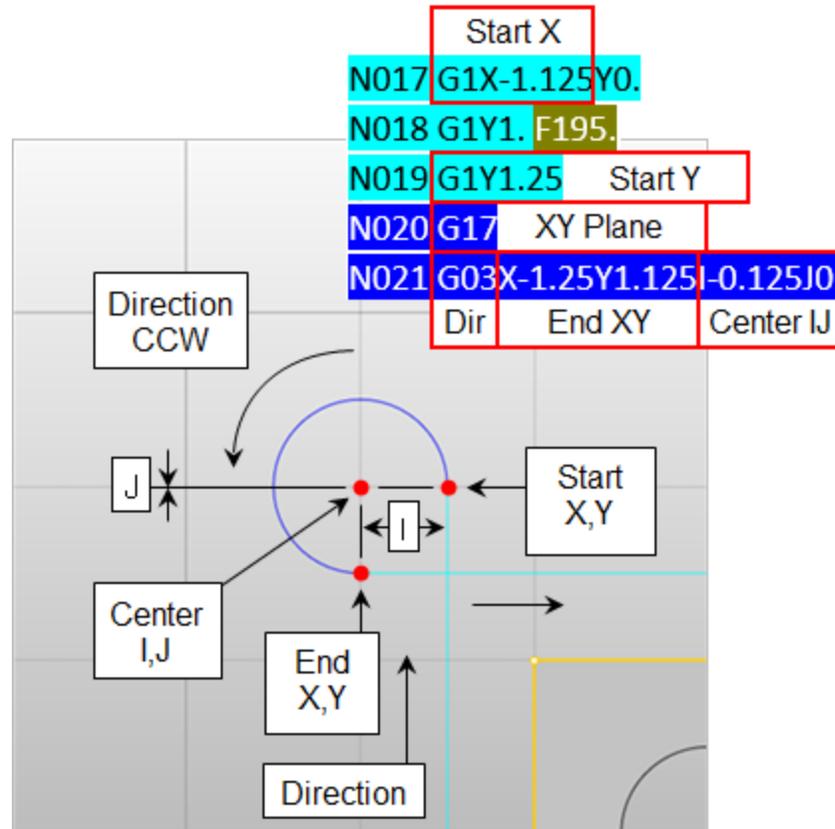
### 5.2.15.1 Introduction

This guide provides the basis for understanding how the [Legacy Post-Process Generator \(PPG\)](#) operates and how you can use it to customize your posted g-code files. It uses a simple part file example with three toolpath operations. Each toolpath is illustrated, examined and the resulting G-Code is color coded to the portion of the [PPG](#) that controls it. Each section of the [PPG](#) is then explained and examined for the part file example and its resulting G-Code.

 **NOTE:** This guide includes sample CAM files, toolpath operations, a customized Haas post definition file and the sample G-Code file. These source files are included in your CAMJam download archive.



**Remember:** The PPG is your friend! It can save you and your CNC operator enormous amounts of time during production!



How arc motions are defined for our sample.

### 5.2.15.2 PPG > Feed Rate

The **Feed Rate** section of the **PPG** defines the block format for Feed Rate. The **Feed Rate Block** is inserted only when the feed rate changes if **Feedrate** is checked under **Modal Output** in the **General** section of the **PPG**. You can also set the **High** and **Low** values for **Feedrate** here as well as other parameters related to feed rate.

**Feed  
Rate**

This is the color code for the **Feed Rate** section.

Block Format

Feed Rate Code:   Add after Next Motion Code

Feed Rate Modes

Units/Min Code:  Units/Rev Code:

Inverse Time Feed Rate

Output inverse time feedrate for 4 & 5 Axis motions

On Code:  Off Code:

Block Format

Sample Output Default

Feed Rate Value

High Value:  Scale Factor:  Z Feed Rate Scale Factor:

Low Value:  # of Decimal Places:  Show Trailing Zeros:

Angular Feedrate Value

Scale Factor:  # of Decimal Places:  Show Trailing Zeros:

The Feed Rate section of the Post-Processor Generator (PPG) dialog for our haas-blog.spm post definition file.

Feed Rate Block Format	
F[FEEDRATE]	
Variable	Derived From
[FEEDRATE]	MecSoft CAM > Tools > Create/Select Tool dialog > Feeds & Speeds or Mop > Feeds & Speeds
Resulting G-Code Sample	
N049 G0	
N050 G1Z-0. F97.5	
N051 G0Z-0.125I-0.4375J-0.0002 F195.	
N052 G0Z-0.25I-0.4375J-0.0002	

### 5.2.15.3 PPG > General

The **General** section of the **PPG** includes options that affect the entire G-Code file. **Modes** are set, **Units** are defined, **Comments** are enabled, **Sequence #s** are enabled and defined, along with other formatting options. The **Modal Output** check boxes determine if values are repeated on every line or only when the value changes.

## General

This is the color code for the **General** section.

<b>File Control</b> Output File Extension <input type="text" value="nc"/> File Start Reading Char <input data-bbox="526 856 581 886" type="text" value="%"/> File Stop Reading Char <input data-bbox="526 909 581 938" type="text" value="%"/>	<b>Mode</b> <input checked="" type="radio"/> Absolute Code <input data-bbox="878 814 992 886" type="text" value="G90"/> <input type="radio"/> Incremental Code <input data-bbox="878 909 992 980" type="text" value="G91"/>	<b>Comments</b> <input checked="" type="checkbox"/> Output Comment Comment Start Char <input data-bbox="1344 867 1399 896" type="text" value="("/> Comment End Char <input data-bbox="1344 919 1399 949" type="text" value=")"/> <input type="checkbox"/> Output sequence number
<b>Sequence #s</b> <input checked="" type="checkbox"/> Use Sequence #s Prefix Letter <input data-bbox="467 1150 522 1180" type="text" value="N"/> Increment <input data-bbox="467 1213 565 1243" type="text" value="1"/> Start value <input data-bbox="467 1276 565 1306" type="text" value="1"/> <input checked="" type="checkbox"/> Show Leading Zeros Number of Digits <input data-bbox="467 1434 565 1463" type="text" value="4"/>	<b>Units</b> <input checked="" type="radio"/> Inch Code <input data-bbox="878 1119 992 1190" type="text" value="G20"/> <input type="radio"/> Metric Code <input data-bbox="878 1213 992 1285" type="text" value="G21"/>	<b>Block Format</b> Delimiter <input type="radio"/> None <input checked="" type="radio"/> Space <input type="radio"/> Tabs <input type="radio"/> User Defined <input data-bbox="1344 1245 1399 1274" type="text" value=","/> End of Block Character <input data-bbox="1344 1329 1399 1358" type="text"/> <input type="checkbox"/> Always output + sign <input checked="" type="checkbox"/> Show Leading Zeros
<b>Modal Output</b> <input type="checkbox"/> GCode <input checked="" type="checkbox"/> Feedrate <input checked="" type="checkbox"/> Coordinate <input checked="" type="checkbox"/> Spindle speed		

The General section of the Post-Processor Generator (PPG) dialog for our haas-blog.spm post definition file.

Resulting G-Code Sample
( ENDOF TOOL LIST )
N003 (Setup 1)
N004 (Work Zero)
N005 G54
N006 (2 1/2 Axis Profiling (Outer))
N007 ( Tool Diameter = 0.25 Length = 2.0 )

#### 5.2.15.4 PPG > Helical/Spiral

The [Helical/Spiral](#) section of the [PPG](#) defines how all helical and spiral motions are defined. The only difference between an arc and a helix is that a helix has a Z value (i.e., pitch). In fact, a helix is defined using the Arc G-Code with the added Z value. We strongly recommended that you first read the [PPG > Circle](#) section above to understand how arcs are defined before continuing.



This is the color code for the [Helical/Spiral](#) section.

Select Interpolation Type  
Helical Interpolation  
Spiral Interpolation

G Code  
Clockwise Arc Code G02  
C-clockwise Arc Code G03

Block Format  
Select Plane  
Plane XY  
Plane ZX  
Plane YZ

[CIR\_PLANE]  
[G\_CODE][NEXT\_X][NEXT\_Y][NEXT\_Z][NEXT\_I][NEXT\_J]

Default

Sample Output  
{G17/G18/G19}  
{G02/G03}X0.0Y0.0Z0.0I0J0

The Helical/Spiral section of the Post-Processor Generator (PPG) dialog for our haas-blog.spm post definition file.

Helical/Spiral Block Format	
[CIR_PLANE] [G_CODE][NEXT_X][NEXT_Y][NEXT_Z][NEXT_I][NEXT_J]	
Variable	Derived From
[CIR_PLANE]	The Plane Code defined in this section of the PPG (G17=XY Plane, G18=ZX Plane and G19=YZ Plane)
[G_CODE]	The G Code defined in this section of the PPG (G02 for a clockwise arc and G03 for a counter clockwise arc)
[NEXT_X]	The X coordinate value for the helix end point
[NEXT_Y]	The Y coordinate value for the helix end point
[NEXT_Z]	The Z coordinate value for the helix end point
[NEXT_I]	The X coordinate value for the helix center point
[NEXT_J]	The Y coordinate value for the helix center point
[NEXT_K]	The Z coordinate value for the arc center point
Resulting G-Code Sample	
N046 G90G0X0.4375Y0.0002	← X, Y Start of Helix
N047 G43Z0.25H2	
N048 M8	
N049 G0	
N050 G1Z-0. F97.5	← Z Start of Helix
N051 G02Z-0.125I-0.4375J-0.0002 F195.	← First Full Helix
N052 G02Z-0.25I-0.4375J-0.0002	← Second Full Helix
N053 G02X-0.4375Y-0.0002I-0.4375J-0.0002	← First Arc Clean up
N054 G02X0.4375Y0.0002I0.4375J0.0002	← Second Arc Clean up
Notes:	
<ol style="list-style-type: none"> <li>1. Notice that the X, Y start of the helix is one line N046 and the Z start of the helix is on line N050.</li> <li>2. Lines N051 and N052 are the two helical motions. Notice that each includes a Z coordinate value.</li> <li>3. Line N051 create the first full helix starting a Z0 and ending at Z0.125.</li> <li>4. Line N052 create the second full helix starting a Z0.125 and ending at Z0.25.</li> <li>5. Lines N053 and N054 are the two arc motions located at the base of the hole. Together they form a complete circle which serves as a clean up pass.</li> </ol>	

### 5.2.15.5 PPG > Misc

The [Misc](#) section of the PPG is primarily used for [Coolant Codes](#) but also contains the [Work Offset Code](#).

## Misc

This is the color code for the Misc section.

Coolant Codes		Compensation	
Coolant Off	M9	Compensation Off	G40
Coolant Mist	M7	Compensation Right	G42
Coolant Tap	M51	Compensation Left	G41
Coolant On	M51	Compensation Length	G43
Coolant Flood	M8	Work (Fixture) Offsets	
Coolant Thru	M50	Work Offset Code	G

The Misc section of the Post-Processor Generator (PPG) dialog for our haas-blog.spm post definition file.

### Resulting G-Code Sample

```

N006 (2 1/2 Axis Profiling (Outer))
N007 ( Tool Diameter = 0.25 Length = 2.0 )
N008 G20 T1 M6
N009 S10000 M3
N010 G90G0X-1.2533Y-0.3524
N011 G43Z0.25H1
N012 M8 ←————— Coolant = Flood for Tool #1
N013 G0
  
```

#### Notes:

1. To enable Coolant codes, make sure Coolant is set on the Feeds & Speeds tab of the Mop (MecSoft CAM > Holes > Drill > Feeds & Speeds > Coolant).
2. Even if you have Coolant set for the Tool (MecSoft CAM > Create/Select Tool > Properties > Coolant), it could be overwritten in the Feeds & Speeds tab of the Mop so see note #1 also.

### 5.2.15.6 PPG > Motion

The **Motion** section of the **PPG** defines the **Linear Motion** and **Rapid Motion Block Format**. Anytime a linear or rapid motion is defined it is output in this format. This section also allows you to control other aspects of motion coordinates including X, Y and Z Scale Factors, # of Decimal Places and more.

## Motion

This is the color code for the **Motion** section.

**Linear Motion Block Format**

G Code

Block Format

Sample Output  Default

Output cut motion start macro before plunge motion

**Rapid Motion Block Format**

G Code

Block Format

Sample Output  Default

**Motion Coordinates**

Scale Factor for X       Scale Factor for Y       Scale Factor for Z

Show Trailing Zeros       # of Decimal Places

The Motion section of the Post-Processor Generator (PPG) dialog for our haas-blog.spm post definition file.

Linear Motion Block Format	
[G_CODE][NEXT_X][NEXT_Y][NEXT_Z]	
Rapid Motion Block Format	
[G_CODE][NEXT_Z] [NEXT_X][NEXT_Y]	
Variable	Derived From
[G_CODE]	Will be G1 for a Linear motion or G0 for a Rapid motion
[NEXT_Z]	The next Z coordinate value
[NEXT_X]	The next X coordinate value
[NEXT_Y]	The next Y coordinate value
Resulting G-Code Sample	
<p>N034 G1Y-1. ← Rapid (G0) or Linear (G1) Motions</p> <p>N035 G1Y0.</p> <p>N036 G1Y0.1</p> <p>N037 G1X-1.2105Y0.3349 F243.75</p> <p>N038 G40</p> <p>N039 G1X-1.296Y0.5698 F292.5</p> <p>N040 G0Z0.25</p>	

### 5.2.15.7 PPG > Multi Axis Motion

The [Multi Axis Motion](#) section of the PPG is only used if your [Machine](#) definition is set to [4 Axis](#) or [5 Axis](#) (MecSoft CAM > [Machining Job](#) > [Machine](#)). It is used to define the [Rotation Axis Code](#), [Rotation Direction Code](#), [Angle Value](#), and the [Motion Block](#). This will be covered in a future blog post.

Block Format		
Rotation Axis Code		
Primary Axis	<input type="text" value="A"/>	Secondary Axis <input type="text" value="B"/>
		Tertiary Axis <input type="text" value="C"/>
Rotation Direction Code		
Clockwise Rotation	<input type="text" value="-"/>	Counter Clockwise Rotation <input type="text"/>
		<input checked="" type="checkbox"/> Ignore in continuous rotation toolpaths
Angle Value		
Scale Factor	<input type="text" value="57.295779"/>	# Decimal Places <input type="text" value="1"/>
		Trailing Zeros <input type="checkbox"/>
Motion Block		
Block Format		<input type="button" value="Default"/>
<input type="text" value="[LINEAR][NEXT_X][NEXT_Y][NEXT_Z][ROTATION_AXIS][ROTATION_DIR][ANGLE][FEEDRATE_CODE][ROTATION_FEEDVALUE]"/>		
Sample Output		
<input type="text" value="G1X0.0Y0.0Z0.0A-0F0"/>		
Rapid Block		
Block Format		<input type="button" value="Default"/>
<input type="text" value="[RAPID][NEXT_X][NEXT_Y][NEXT_Z][ROTATION_AXIS][ROTATION_DIR][ANGLE][FEEDRATE_CODE][ROTATION_FEEDVALUE]"/>		
Sample Output		
<input type="text" value="G0X0.0Y0.0Z0.0A-0F0"/>		

The Multi Axis Motion section of the Post-Processor Generator (PPG) dialog for our haas-blog.spm post definition file.

### 5.2.15.8 PPG > Setup

The [Setup](#) section of the [PPG](#) is only used if your [Machine](#) definition is set to [4 Axis](#) or [5 Axis](#) (MecSoft CAM > [Machining Job](#) > [Machine](#)). It is used to output the rotation axis and angles for the setup. This will be covered in a future guide.

Work Offset

Setup 0 (Coordinate System Change) Macro:

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

Setup 1 (Rotate Table) Macro:

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

Default

The Setup section of the Post-Processor Generator (PPG) dialog for our haas-blog.spm post definition file.

### 5.2.15.9 PPG > Spindle

The [Spindle](#) section of the [PPG](#) defines the [Spindle Block](#) variable `[SPINDLE_BLK]`. It is also used to set the [High](#) and [Low](#) values for the [Spindle RPM](#). These values will not be exceeded regardless of the tool or toolpath parameters. This section also defines the [Spindle Direction](#) and other spindle related values.



This is the color code for the [Spindle](#) section.

Block Format

Spindle Code

Spindle Mode

Constant Surface Speed

Constant Rotation Speed

Spindle Direction

Clockwise Rotation Code

C-Clockwise Rotation Code

Spindle Off Code

Spindle Block Format

Sample Output

Spindle RPM

High Value  Scale Factor  Show Trailing Zeros

Low Value  # of Decimal Places

The Spindle section of the Post-Processor Generator (PPG) dialog for our haas-blog.spm post definition file.

Spindle Block Format	
<input style="background-color: #f0f0f0;" type="text" value="S{SPINDLE_SPD}{DELIMITER}{SPINDLE_ARC}"/>	
Variable	Derived From
[SPINDLE_SPD]	MecSoft CAM > Tools > Create/Select Tool dialog > Feeds & Speeds or Mop > Feeds & Speeds > Spindle Speed
[SPINDLE_ARC]	MecSoft CAM > Tools > Create/Select Tool dialog > Feeds & Speeds or Mop > Feeds & Speeds > Direction
ISO Codes	Function
M3	Spindle On, Clockwise rotation
M4	Spindle On, Counter Clockwise rotation
Resulting G-Code Sample	
<input style="background-color: #f0f0f0;" type="text" value="N006 (2 1/2 Axis Profiling (Outer))"/>	
<input style="background-color: #90ee90;" type="text" value="N007 ( Tool Diameter = 0.25 Length = 2.0 )"/>	
<input style="background-color: #90ee90;" type="text" value="N008 G20 T1 M6"/>	
<input style="background-color: #90ee90;" type="text" value="N009 S10000 M3"/>	
<input style="background-color: #90ee90;" type="text" value="N010 G90G0X-1.2533Y-0.3524"/>	
<input style="background-color: #90ee90;" type="text" value="N011 G43Z0.25H1"/>	
<input style="background-color: #4682b4;" type="text" value="N012 M8"/>	

### 5.2.15.10 PPG > Start/End

The [Start/End](#) section of the [PPG](#) allows you to define how the start and end sections of the G-Code file are formatted and what codes are included in these sections. The [Start Up](#) section typically includes codes to make sure various machine functions are turned off. We have customized it to add the stock size.



This is the color code for the [Start/End](#) section.

The tables below lists [Start Up Code](#) and the [End Code](#) along with the variables used and how those variables were defined. Note that variables are written within brackets [ ] and comments are written within parentheses ( ).

Program Start/End

Start Up Code :

```
[START_CHAR]
Q(PARTNUM)
[SEQ_PRECHAR][SEQNUM][DELIMITER]G40[DELIMITER]G49[DELIMITER]G80
[SEQ_PRECHAR][SEQNUM][DELIMITER]( STOCK SIZE: X{STOCK_LENGTH_X} Y{STOCK_LENGTH_Y} Z{STOCK_LENGTH_Z} )
```

End Code :

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

The Start/End section of the Post-Processor Generator (PPG) dialog for our `haas-blog.spm` post definition file.

Start Up Code	
<pre>[START_CHAR] O(PARTNUM) [SEQ_PRECHAR][SEQNUM][DELIMITER]G40[DELIMITER]G49[DELIMITER]G80 [SEQ_PRECHAR][SEQNUM][DELIMITER]( STOCK SIZE: X[STOCK_LENGTH_X] Y[STOCK_LENGTH_Y] Z[STOCK_LENGTH_Z])</pre>	
Variable	Derived From
[START_CHAR]	File Start Reading Character from the <a href="#">General</a> section of the PPG.
[DELIMITER]	Delimiter type from the <a href="#">General</a> section of the PPG. This is set to insert a space between values just to make it easier to read the posted output file.
[PARTNUM]	Right-click > Properties on the first Machining Operation (Mop) under the first Setup in the Machining Job.
[SEQ_PRECHAR]	Prefix Letter from the <a href="#">General</a> section of the PPG.
[SEQNUM]	Use Sequence #s from the <a href="#">General</a> section of the PPG.
[STOCK_LENGTH_X]	MecSoft CAM > MILL > Machining Job > Stock > Length (L)
[STOCK_LENGTH_Y]	MecSoft CAM > MILL > Machining Job > Stock > Width (W)
[STOCK_LENGTH_Z]	MecSoft CAM > MILL > Machining Job > Stock > Height (H)
ISO Codes	Function
G40	Cutter Compensation Off
G49	Tool Length Compensation Off
G80	Canned Cycle Off
Resulting G-Code Sample	
<pre>% O1234 N001 G40 G49 G80 N002 ( STOCK SIZE: X2.125 Y2.125 Z0.25 ) ( BEGIN TOOL LIST )</pre>	

End Code	
[SEQ_PRECHAR][SEQNUM][DELIMITER]M30 [STOP_CHAR]	
Variable	Derived From
[DELIMITER]	Delimiter type from the <a href="#">General</a> section of the PPG.
[SEQ_PRECHAR]	Prefix Letter from the <a href="#">General</a> section of the PPG.
[SEQNUM]	Use Sequence #s from the <a href="#">General</a> section of the PPG.
[STOP_CHAR]	File Stop Reading Character from the <a href="#">General</a> section of the PPG.
ISO Codes	Function
M30	End of Program
Resulting G-Code Sample	
<pre> N069 G80 N070 G0Z0.25 N071 M30 %</pre>	

### 5.2.15.11 PPG > Tool Change

The [Tool Change](#) section of the PPG allows you to define the [First Tool Load Macro](#) and the [Tool Change Macro](#) for subsequent tool changes. These macros define all of the information your CNC controller needs to load and change cutting tools and move the tool into position.



This is the color code for the [Tool Change](#) section.

This section is ignored if your CNC machine does not have an automatic tool changer. The reason these are called Macros is because they use nested variables. For example [\[SPINDLE\\_BLK\]](#) is the result of the [Spindle Block Format](#) code defined in the [Spindle](#) section of the PPG.

First you will notice that [Output tool list as comments at start of program](#) is checked. Because [Comments](#) are enabled in the [General](#) section of the PPG, the following comments are listed at the top of the G-Code file:

```

N002 ( STOCK SIZE: X2.125 Y2.125 Z0.25 )
( BEGIN TOOL LIST )
( TOOL 1 - FlatMill:0.25 - DESC: 0.2500 DIA, 2 FLUTE, CARBIDE MAT )
( TOOL 2 - FlatMill:0.125 - DESC: 0.1250 DIA, 2 FLUTE, CARBIDE MAT )
( TOOL 3 - Drill: 1/4" - DESC: 0.2500 DIA, 2 FLUTE, CARBIDE MAT )
( ENDOF TOOL LIST )
N003 (Setup 1)

```

Output tool list as comments

Now onto the macros.

Use 2 digit format for Tool number and registers
  Output tool list as comments at start of program

First Load Tool Macro

```

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

```

Tool Change Macro

```

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

```

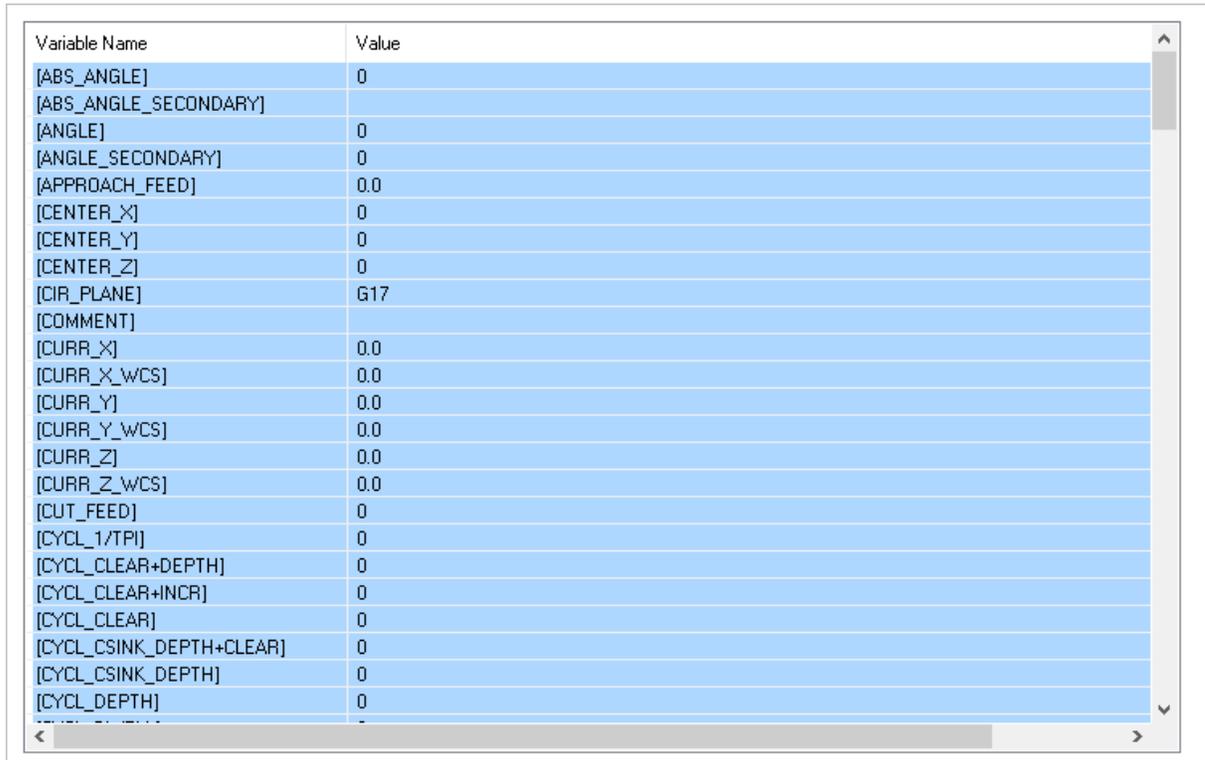
The Tool Change section of the Post-Processor Generator (PPG) dialog for our haas-blog.spm post definition file.

First Load Tool Macro	
<pre>[SEQ_PRECHAR][SEQNUM][DELIMITER] Tool Diameter = [TOOL_DIA] Length = [TOOL_LENGTH] ) [SEQ_PRECHAR][SEQNUM][DELIMITER][OUTPUT_UNITS_CODE][DELIMITER]T[TOOL_NUM][DELIMITER]M6 [SEQ_PRECHAR][SEQNUM][DELIMITER][SPINDLE_BLK] [SEQ_PRECHAR][SEQNUM][DELIMITER][OUTPUT_MODE_CODE][G_CODE]X[NEXT_NONMDL_X]Y[NEXT_NONMDL_Y] [SEQ_PRECHAR][SEQNUM][DELIMITER]G43Z[NEXT_NONMDL_Z]H[TOOL_ADJST_REG]</pre>	
Variable	Derived From
[TOOL_DIA]	MecSoft CAM > Tools > Create/Select Tool > Tool Dia.
[TOOL_LENGTH]	MecSoft CAM > Tools > Create/Select Tool > Tool Length.
[OUTPUT_UNITS_CODE]	Units Code from the General section of the PPG.
[TOOL_NUM]	MecSoft CAM > Tools > Create/Select Tool > Tool Number.
[SPINDLE_BLK]	Spindle Block from the Spindle section of the PPG.
[OUTPUT_MODE_CODE]	Mode Code from the Spindle section of the PPG.
[G_CODE]	This will be either G0 or G1 depending on the Transfer feed rate value on the Feeds & Speeds tab of the first Mop. G0=Rapid, G1=Feed Rate
[NEXT_NONMDL_X]	The next X coordinate if different than the last (non modal). Because Coordinate is checked under Model Output in the General section, coordinates are only displayed if it is different than the last coordinate value.
[NEXT_NONMDL_Y]	The next Y coordinate if different than the last (non modal).
[NEXT_NONMDL_Z]	The next Z coordinate if different than the last (non modal).
[TOOL_ADJST_REG]	MecSoft CAM > Tools > Create/Select Tool dialog > Adjust Register. Should be the same as Tool Number.
ISO Codes	Function
M6	Tool Change
G43	Apply Tool Length Compensation
Resulting G-Code Sample	
<pre>N006 (2 1/2 Axis Profiling (Outer)) N007 ( Tool Diameter = 0.25 Length = 2.0 ) N008 G20 T1 M6 N009 S10000 M3 N010 G90G0X-1.2533Y-0.3524 N011 G43Z0.25H1 N012 M8</pre>	

Tool Change Macro	
<pre>(SEQ_PRECHAR)[SEQNUM][DELIMITER] T Tool Diameter = [TOOL_DIA] Length = [TOOL_LENGTH] ) (SEQ_PRECHAR)[SEQNUM][DELIMITER][OUTPUT_UNITS_CODE][DELIMITER]T[TOOL_NUM][DELIMITER]M6 (SEQ_PRECHAR)[SEQNUM][DELIMITER][WORK_OFFSET_PREFIX][WORK_OFFSET_NUM] (SEQ_PRECHAR)[SEQNUM][DELIMITER][SPINDLE_BLK] (SEQ_PRECHAR)[SEQNUM][DELIMITER][OUTPUT_MODE_CODE][G_CODE]X[NEXT_NONMDL_X]Y[NEXT_NONMDL_Y] (SEQ_PRECHAR)[SEQNUM][DELIMITER]G43Z[NEXT_NONMDL_Z]H[TOOL_ADJST_REG]</pre>	
Variable	Derived From
[TOOL_DIA]	MecSoft CAM > Tools > Create/Select Tool > Tool Dia.
[TOOL_LENGTH]	MecSoft CAM > Tools > Create/Select Tool > Tool Length.
[OUTPUT_UNITS_CODE]	Units Code from the General section of the PPG.
[TOOL_NUM]	MecSoft CAM > Tools > Create/Select Tool > Tool Number.
[WORK_OFFSET_PREFIX]	Work Offset Code from the Misc section of the PPG.
[WORK_OFFSET_NUM]	MecSoft CAM > Machining Job > Work Zero > Work Offset Register Number.
[SPINDLE_BLK]	Spindle Block from the Spindle section of the PPG.
[OUTPUT_MODE_CODE]	Mode Code from the Spindle section of the PPG.
[G_CODE]	This will be either G0 or G1 depending on the Transfer feed rate value on the Feeds & Speeds tab of the first Mop. G0=Rapid, G1=Feed Rate
[NEXT_NONMDL_X]	The next X coordinate if different than the last (non modal). Because Coordinate is checked under Model Output in the General section, coordinates are only displayed if it is different than the last coordinate value.
[NEXT_NONMDL_Y]	The next Y coordinate if different than the last (non modal).
[NEXT_NONMDL_Z]	The next Z coordinate if different than the last (non modal).
[TOOL_ADJST_REG]	MecSoft CAM > Tools > Create/Select Tool dialog > Adjust Register. Should be the same as Tool Number.
ISO Codes	Function
M6	Tool Change
G43	Apply Tool Length Compensation
Resulting G-Code Sample	
<pre>N041 (Hole Profiling) N042 ( Tool Diameter = 0.125 Length = 2.0 ) N043 G20 T2 M6 N044 G54 N045 S10000 M3 N046 G90G0X0.4375Y0.0002 N047 G43Z0.25H2 N048 M8</pre>	

### 5.2.15.12 PPG > Variables

The [Variables](#) section of the [PPG](#) lists ALL of the available variables in MecSoft CAM along with the format of the expected values. Variables are used throughout the [PPG](#) and affect the format and output of the entire sample G-Code file. We have color-coded this section but instead have listed each variable used in each [PPG](#) section that affects the sample G-Code file.



Variable Name	Value
[ABS_ANGLE]	0
[ABS_ANGLE_SECONDARY]	
[ANGLE]	0
[ANGLE_SECONDARY]	0
[APPROACH_FEED]	0.0
[CENTER_X]	0
[CENTER_Y]	0
[CENTER_Z]	0
[CIR_PLANE]	G17
[COMMENT]	
[CURR_X]	0.0
[CURR_X_WCS]	0.0
[CURR_Y]	0.0
[CURR_Y_WCS]	0.0
[CURR_Z]	0.0
[CURR_Z_WCS]	0.0
[CUT_FEED]	0
[CYCL_1/TPI]	0
[CYCL_CLEAR+DEPTH]	0
[CYCL_CLEAR+INCR]	0
[CYCL_CLEAR]	0
[CYCL_CSINK_DEPTH+CLEAR]	0
[CYCL_CSINK_DEPTH]	0
[CYCL_DEPTH]	0

The Variables section of the Post-Processor Generator (PPG) dialog for our haas-blog.spm post definition file.

### 5.2.15.13 PPG Variables & Macros

The [PPG](#) uses hundreds of variables and macros that allow you to customize each post definition file. The names of the variables are pre-defined. The values for many of the variables are extracted from your CAM cutting tools, setups, toolpath operations, etc. Other variable values are defined within the [PPG](#) itself specific to the post definition file. Load the [PPG](#) and select [Variables](#) to see the complete name and value of each available variable. You can also use algebraic expressions within the [PPG](#). Just pick the [Help](#) button from the [PPG](#) dialog to learn more.

## 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.

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

**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:

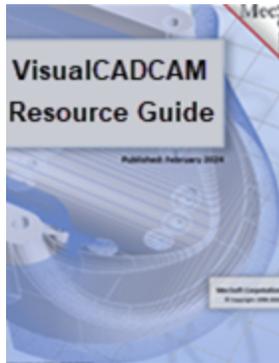
<C:\Program Files\Rhino x\Plug-ins\RhinoCAM 20xx for Rx\SDK>

## Find More Resources

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



### 2025 VisualCAD/CAM Resource Guide



## The 2026 VisualCAD/CAM Resource Guide!

*18 Pages*

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

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

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

## 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**

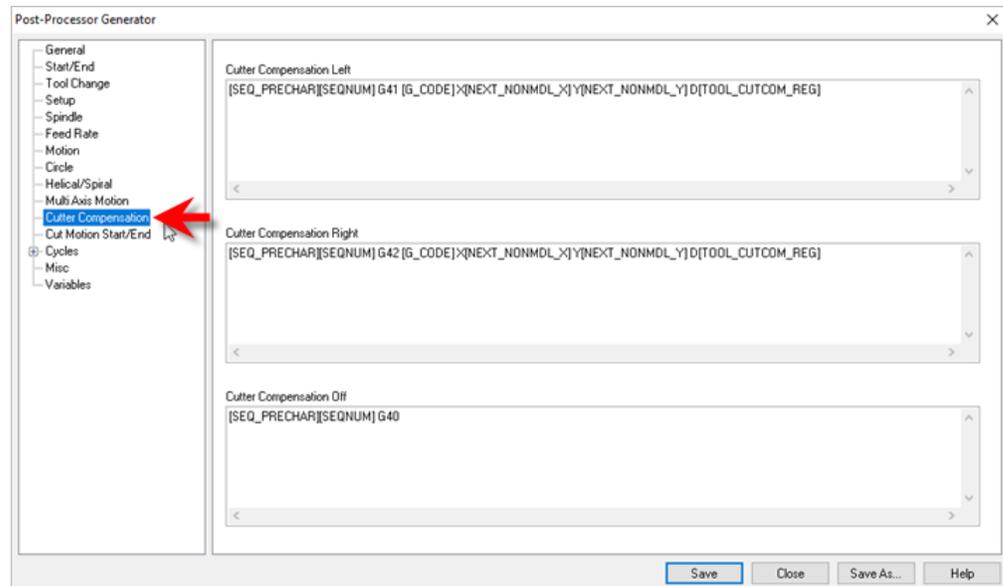
```
[SEQ_PRECHAR][SEQNUM] G41 [G_CODE] X[NEXT_NONMDL_X] Y[NEXT_NONMDL_Y]
D[TOOL_CUTCOM_REG]
```

**Cutter Compensation Right**

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

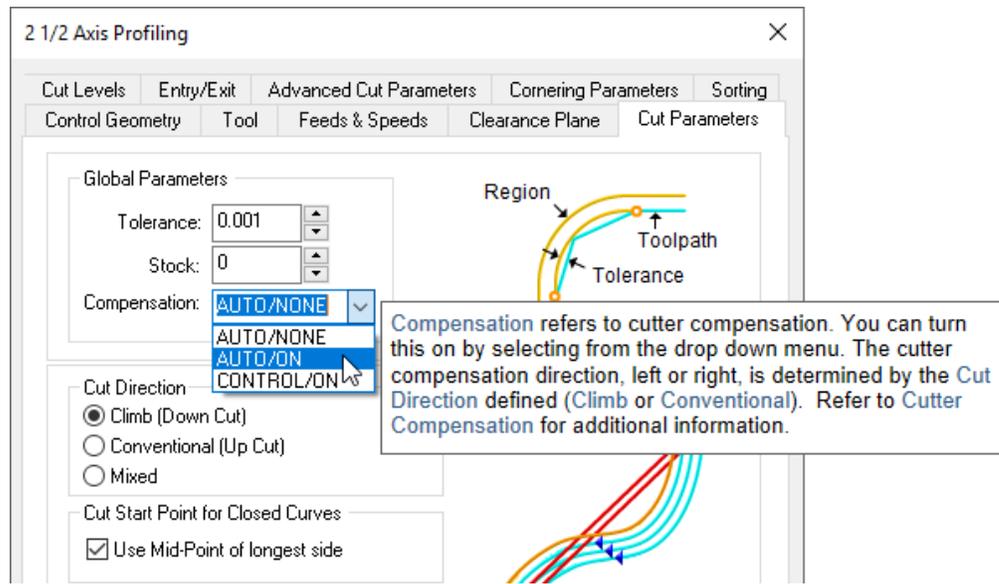
**Cutter Compensation Off**

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



**A few things to watch out for:**

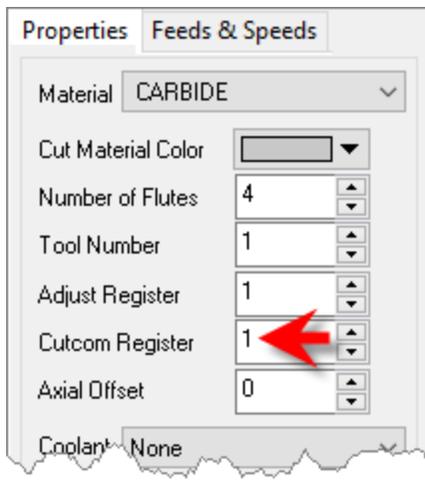
1. **Cutter compensation** makes sense only in 2-1/2 axis operations. If you are using roughing (pocketing & facing) the compensation will be turned on only in the final passes.
2. Make sure you are using **Climb** or **Conventional** cut traversal in any of the methods that you want to turn compensation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



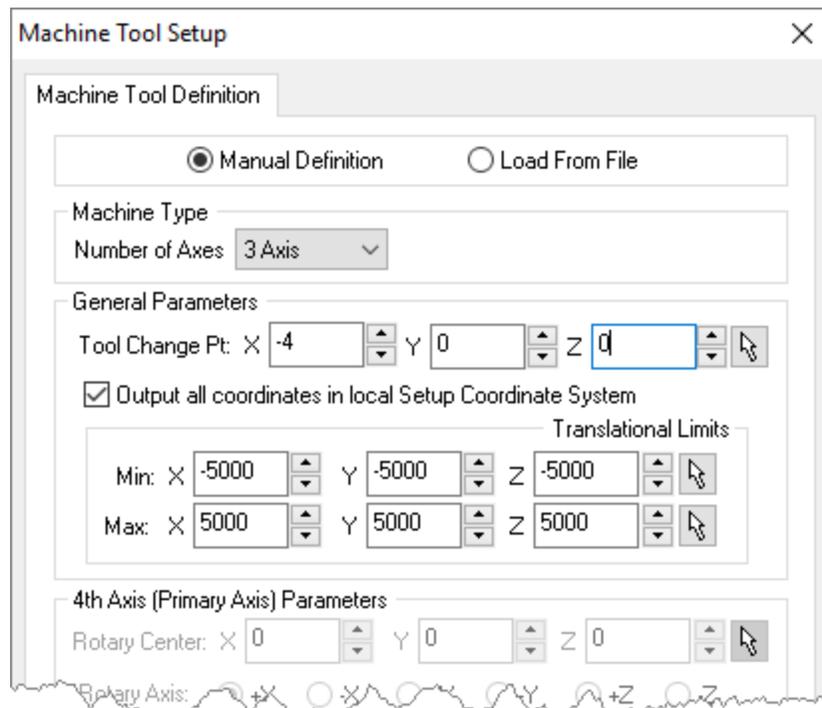
 Posting a Tool Change Point

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:

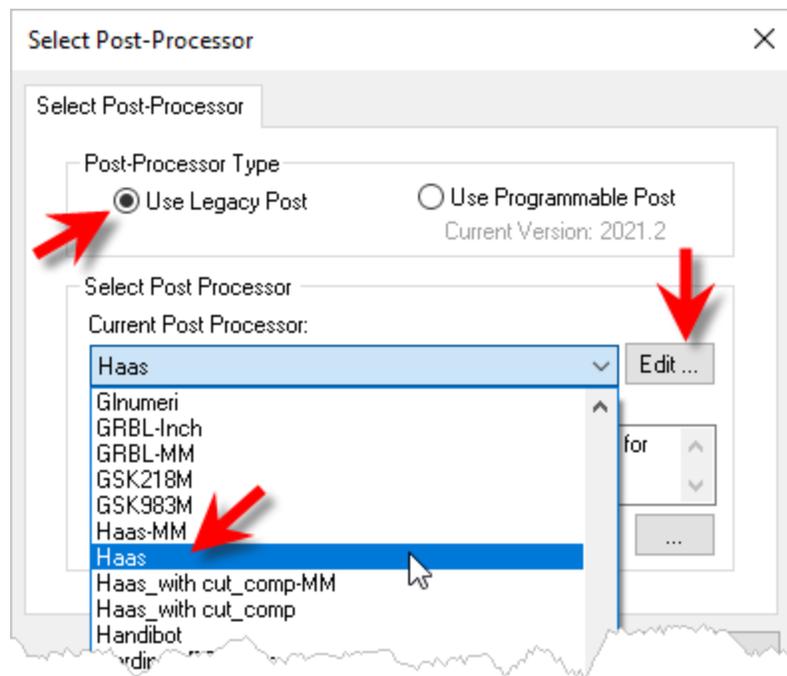
#### For 2 and 3 Axis Output

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**

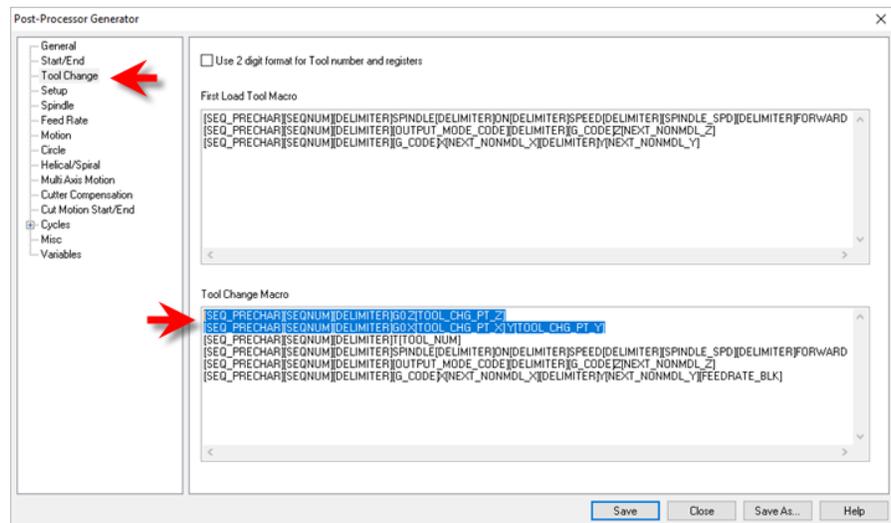


3. Edit your post processor by selecting [Program](#) tab > [Post](#) > [Edit](#).



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.

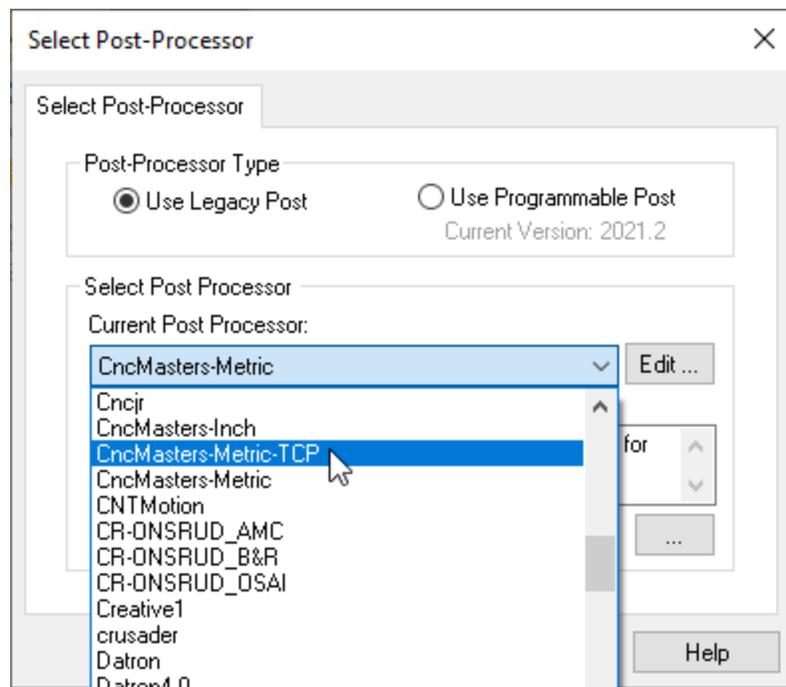
```
[SEQ_PRECHAR][SEQNUM][DELIMITER]GO Z[TOOL_CHG_PT_Z]
[SEQ_PRECHAR][SEQNUM][DELIMITER]GO X[TOOL_CHG_PT_X] Y[TOOL_CHG_PT_Y]
[SEQ_PRECHAR][SEQNUM][DELIMITER]T[TOOL_NUM]
...
...
```



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

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

- From the [Post Process Generator](#) dialog, pick [Save As](#).
- Enter a unique name for your post file ([\\*.spm](#)) for testing and pick [Save](#).
- 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
...
...

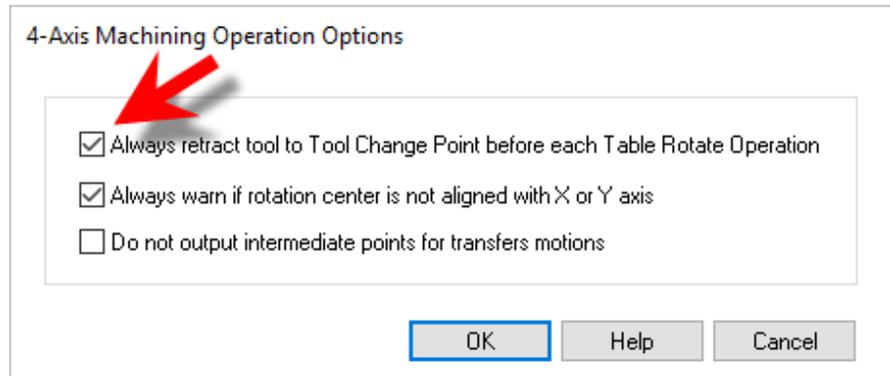
```

15. That's it!

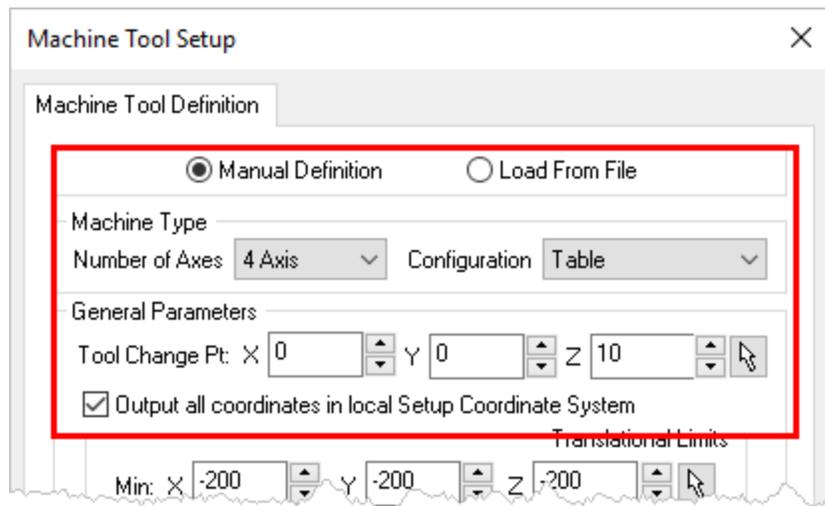
### For 4 Axis Output

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.

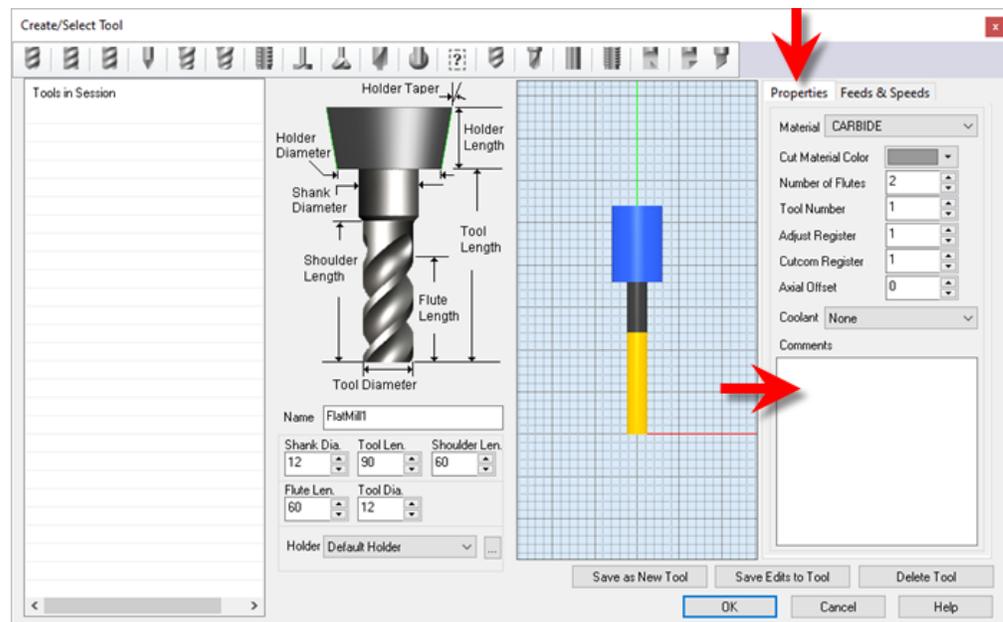
...  
...

## Posting Tool Comments

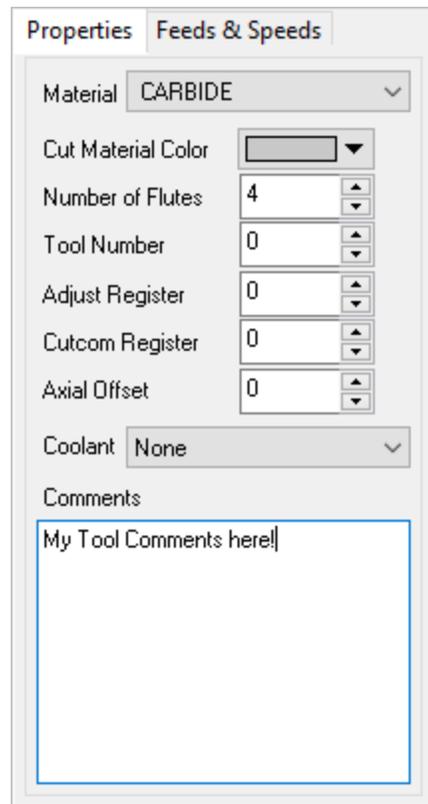
You can add comments associated with a **Tool**. These **Comments** are saved with the **Tool** in your **Tool Library**. They are also posted to your g-code when the tool is used.

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.

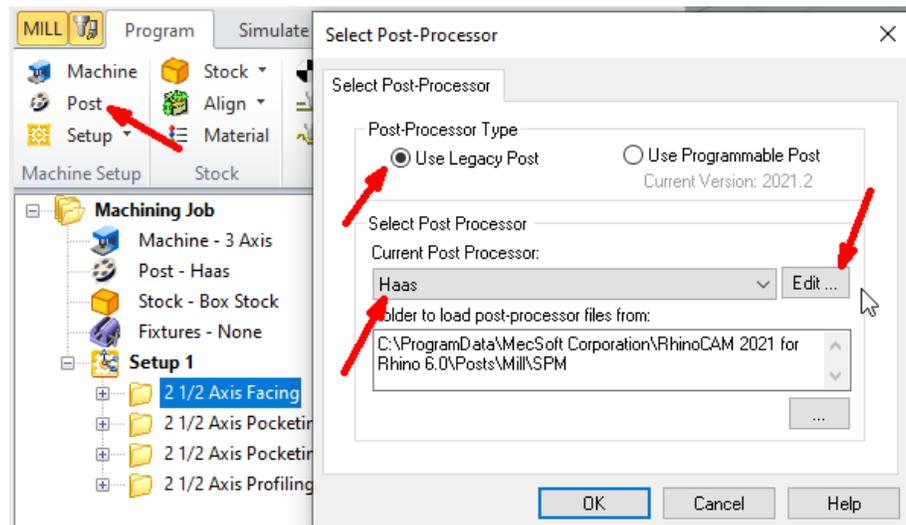


3. Add text to the **Comments** window.



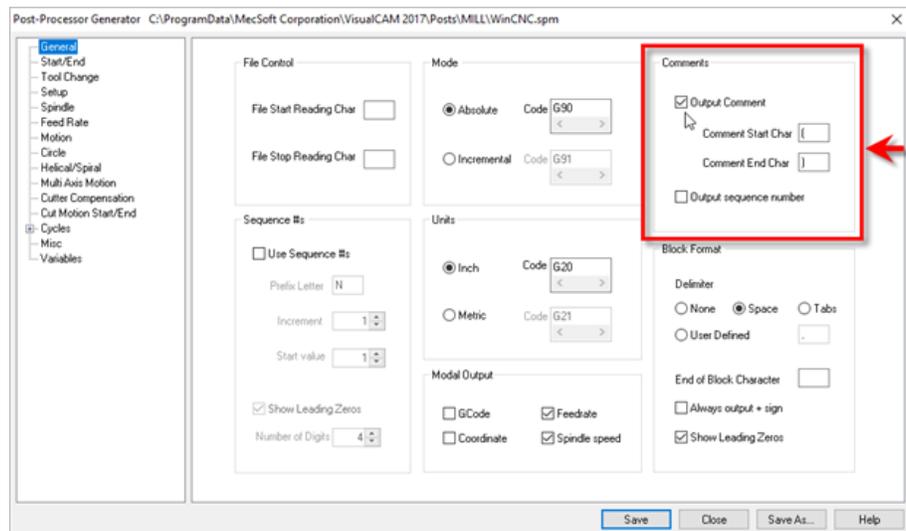
4. Make sure **Comments** are enabled in your post.

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

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](#)

# Index

## - A -

About this Guide 62  
 Algebraic Expressions 9, 85  
 arc center [i,j,k] 44  
 arc motion block 44

## - B -

block format 28

## - C -

Circle 44  
 comments (o/p, start , end , seq ) 28  
 comments on output 28  
 compensation 55  
 coolant code 55  
 Cutter Compensation 9, 85  
 Cycles 53

## - D -

delimiter 28  
 Drill Cycles & Indexed Machining 9, 85

## - F -

Feed Rate 39, 73  
     Sample G-Code Control 63  
 feed rate value 39  
 feedrate block 39  
 file control 28  
 file select dialog 24  
 format 44

## - G -

General  
     G-Code Control 65, 72, 73

## - H -

Helical Motions  
     Sample G-Code 66  
     Understanding 66  
 How to edit 82

## - I -

Introduction 4

## - L -

Legacy Post 21  
 linear motion block 42

## - M -

Macros 81  
 main editor 25  
 Miscellaneous 55  
 modal output 28  
 mode 28  
 More Help 84  
 Motion 42  
     Example G-Code 70  
 motion coordinates 42  
 motion order 42  
 Multi Axis Motion 71

## - P -

plane 44  
 Post  
     Use Legacy 21  
 Post-Processor Generator  
     Feed Rate 63  
     General 65  
     Helical/Spiral 66  
     Misc 68  
     Motion 70  
     Multi Axis Motion 71  
     Sample G-Code 63, 65, 66, 68, 70, 73, 75, 77  
     Setup 72  
     Spindle 73

## Post-Processor Generator

Start/End 75

Tool Change 77

Variables 81

Print Tool List 77

## Programmable Post

About 83

**- S -**

## Sample G-Code

Flood/Mist 68

Helical 66

Tool Change 77

sample output ( feedrate ) 39

sample output ( motion ) 42

sample output ( spindle ) 37

sequence number 28

Spindle 37

Example G-Code 73

spindle block 37

spindle direction 37

spindle RPM 37

Start/End G-Code Control 75

StartEnd 32

**- T -**

## Tool Change

Example G-Code 77

First Tool Load Macro 77

Print Tool List 77

Tool Change Macro 77

Variables Used 77

Tool Change Point 9, 85

Tool Comments 9, 85

ToolAxis 48

ToolChange 33

**- U -**

units 28

**- V -**

variable list dialog 27

Variables 57, 81