

# Post-Processor (PPG) Decoded

## *RhinoCAM 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>Quick Start</b>	<b>3</b>
<b>Resource Guide</b>	<b>6</b>
<b>Introduction</b>	<b>7</b>
1 What is a Post-Processor?.....	8
2 What is the PPG?.....	8
3 PPG Variables & Macros.....	9
<b>Decoding a Sample G-Code File</b>	<b>10</b>
1 The Setup & Post Processor.....	10
2 The Sample Toolpaths.....	10
3 The Posted Setup G-Code File.....	12
4 The 2½ Axis Profiling Operation G-Code.....	15
5 The 2½ Axis Hole Profiling Operation G-Code.....	19
6 The Deep Drilling Operation G-Code.....	21
<b>The Post-Processor Generator Dialog</b>	<b>24</b>
1 PPG > General.....	24
2 PPG > Start/End.....	25
3 PPG > Tool Change.....	28
4 PPG > Setup.....	32
5 PPG > Spindle.....	32
6 PPG > Feed Rate.....	34
7 PPG > Motion.....	36
8 PPG > Circle.....	37
9 PPG > Helical/Spiral.....	41
10 PPG > Multi Axis Motion.....	42
11 PPG > Cutter Compensation.....	43
12 PPG > Cut Motion Start/End.....	46
13 PPG > Cycles.....	47
14 PPG > Misc.....	51
15 PPG > Variables.....	52
<b>Index</b>	<b>53</b>

## Quick Start



# PPG Module 2026

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

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

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



### What's New!

[What's New in RhinoCAM 2026](#)



### The Complete Quick Start Video Play List

[Here is a link to the complete 2026 Video Play List](#)



### How to Access the Quick Start Guide Documents

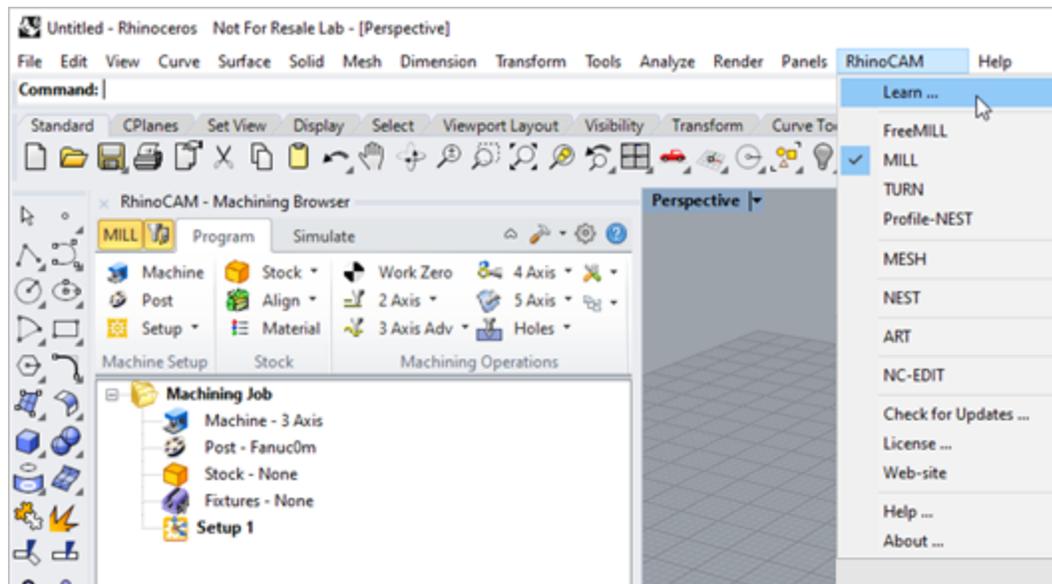
To help you quickly get started in working with each module, select one of the Help buttons located on the [RhinoCAM Learning Resources](#) dialog.

You will find:

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

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

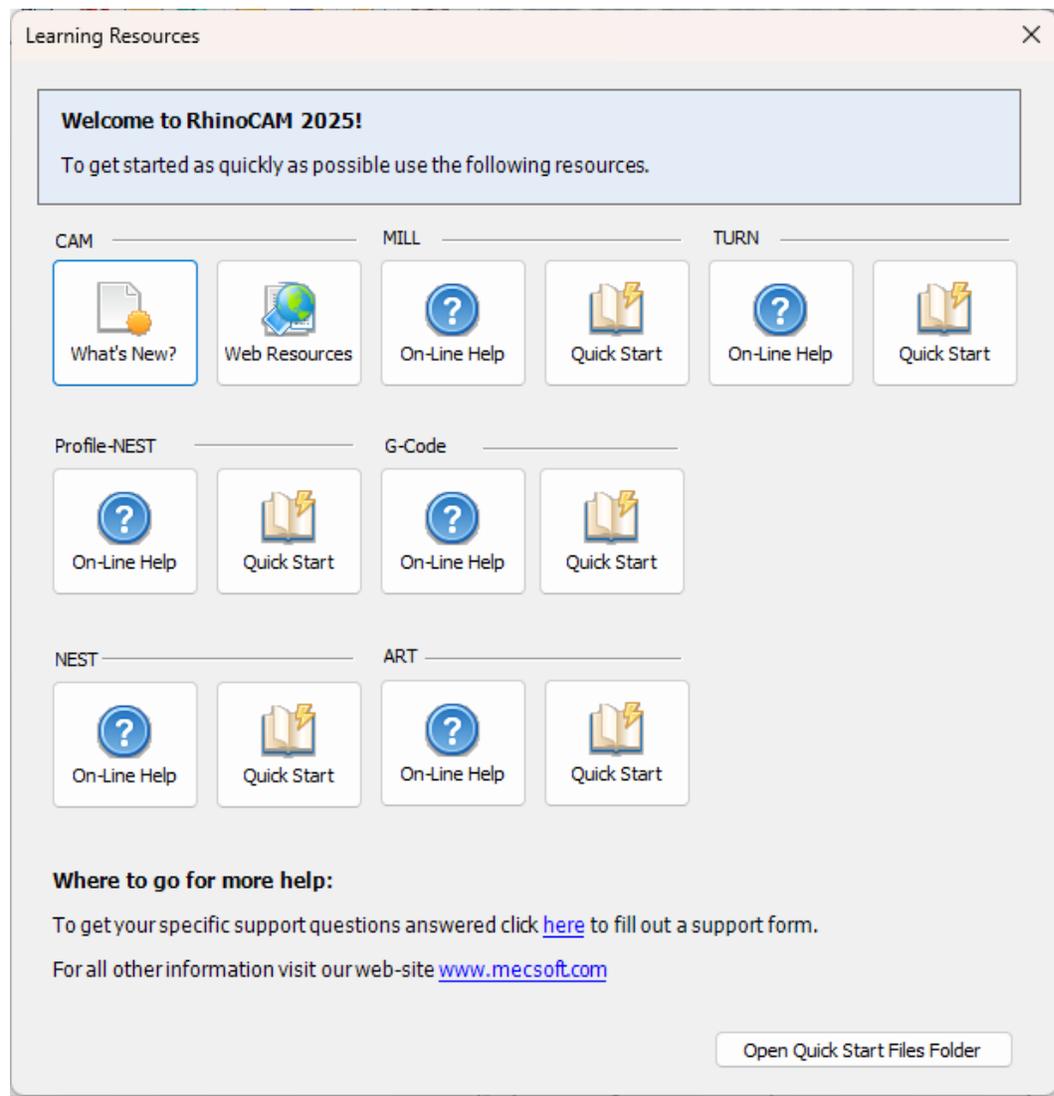
1. From the [Rhino Main Menu](#), drop down the Main menu and select [Learn ...](#)



To access the Learning Resources dialog in RhinoCAM

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

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



Learning Resources Dialog



## Related Topics

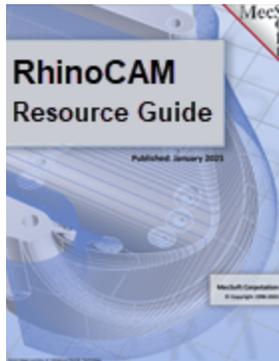
Find More Resources

## Resource Guide

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



### 2025 RhinoCAM Resource Guide



### The 2026 RhinoCAM Resource Guide!

*18 Pages*

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

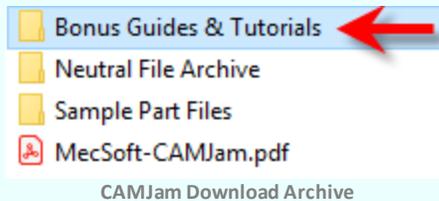
[Prefer Printed Documentation? Check Here!](#)

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

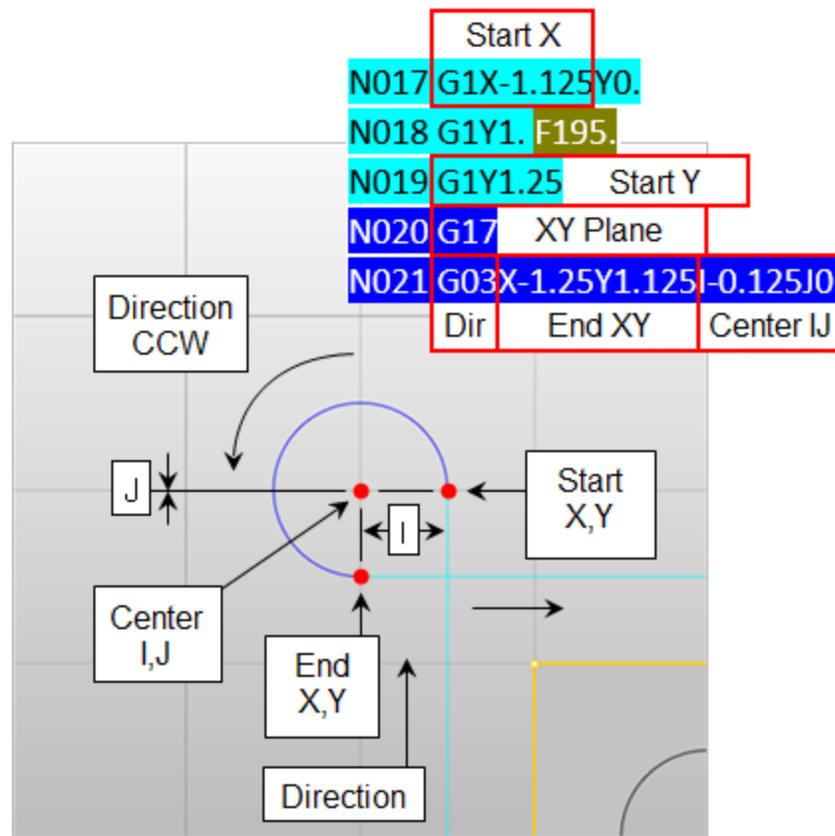
## Introduction

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!



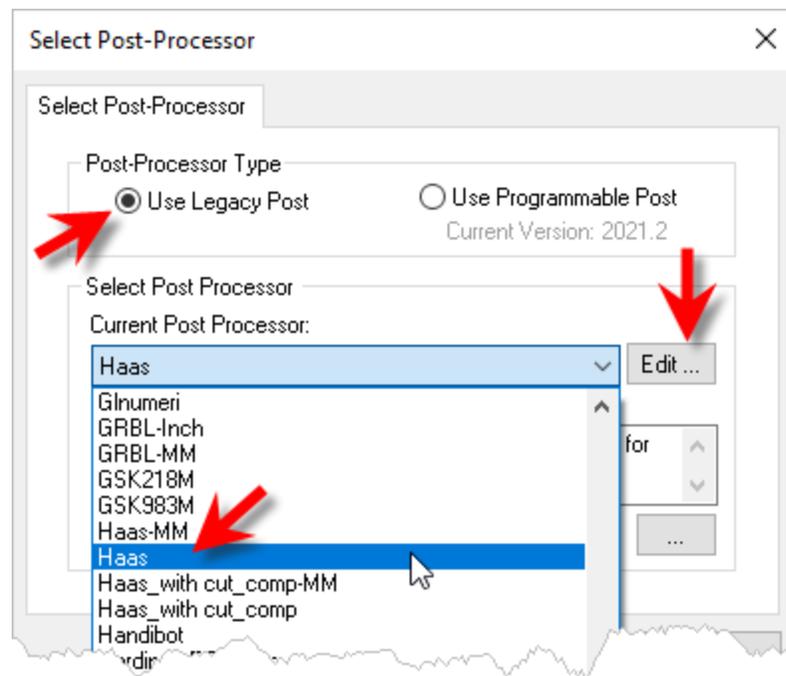
How arc motions are defined for our sample.

### 3.1 What is a Post-Processor?

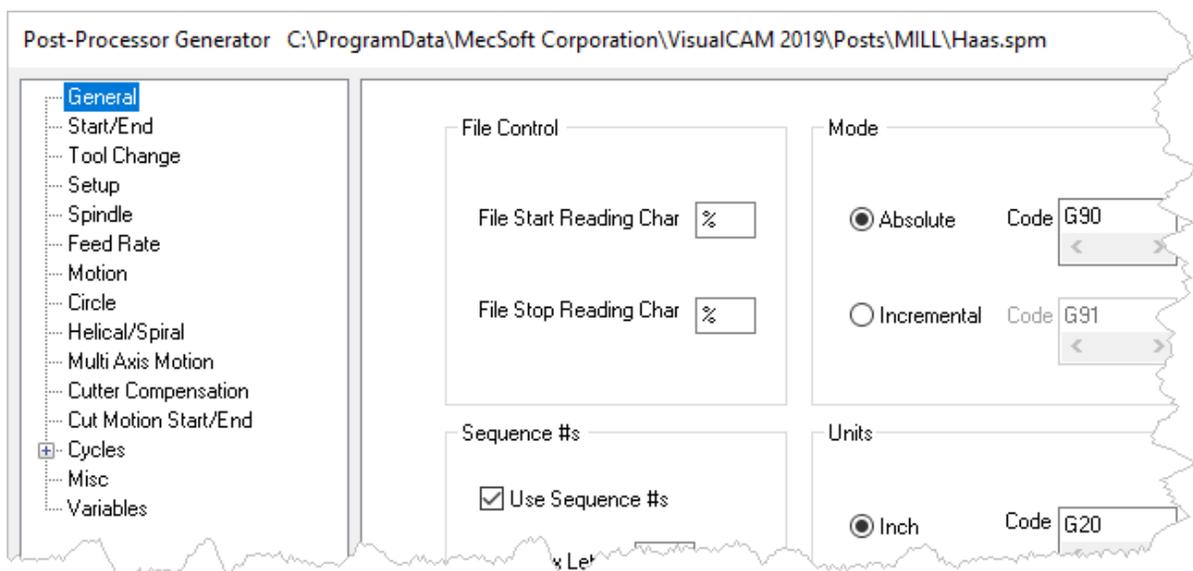
The post-processor converts the toolpath operations you create in MecSoft CAM into a suitable format (called G-Code) that your CNC machine controller can interpret. The G-Code that MecSoft CAM's post-processors create adheres to [ISO6983](#) (the international standard for numerical controls). The post-processor also tailors the G-Code to a specific type of CNC controller such as [Haas](#), [Fanuc](#), [Mach3](#), [WinCNC](#), etc. MecSoft's CAM plugins come installed with over 300 post-processor definition files that are pre-configured for a wide range of CNC machines & controllers available on the market today.

### 3.2 What is the PPG?

The [Post-Processor Generator \(PPG\)](#) allows you to create and customize your post definition files. These files have the file extension of `*.spm`. The names of these post definition files are what you are selecting when you define your post-processor. For example when you select [MILL > Program > Post](#) and select the [Haas](#) post-processor you can also select the [Edit ...](#) button and the [Post-Processor Generator](#) will display with the [Haas.spm](#) post definition file loaded for editing. The default folder where these files are stored is also listed just below the post selection. This is illustrated in the two dialog images below.



Set Post-Processor Options



The Post-Processor Generator (PPG)

### 3.3 PPG Variables & Macros

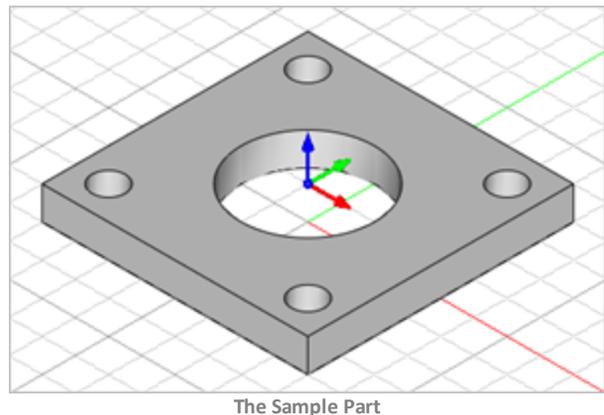
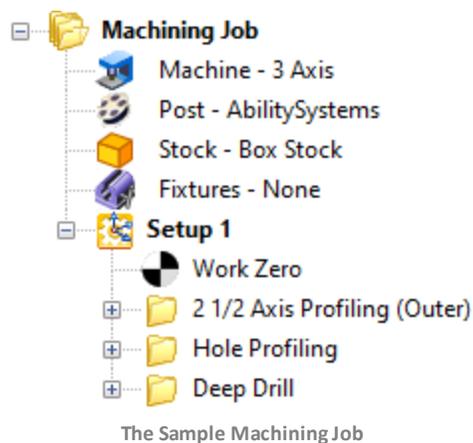
The **PPG** uses hundreds of variables and macros that allow you to customize each post definition file. The names of the variables are pre-defined. The values for many of the variables are extracted from your CAM cutting tools, setups, toolpath operations, etc. Other variable values are defined within 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.

## Decoding a Sample G-Code File

In this guide we will use the following sample part and toolpaths to see how internal variables for CAM definitions such as stock size, tool size, feeds & speeds parameters, toolpath operation types and various other cutting parameters, are used by the post-processor generator and the post definition file to define the format and output of the resulting G-Code file.

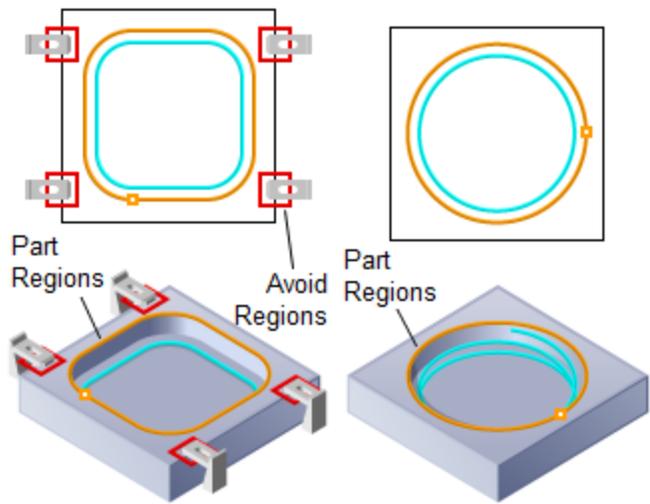
### 4.1 The Setup & Post Processor

The Machine is defined as **3 Axis** (this is used for both **2½ Axis** and **3 Axis** applications). The post-processor definition file selected is a Haas post that we have specifically customized for this article. The Haas machining center has an automatic tool changer. The post definition file is [haas-blog.spm](#). The stock is a 2-1/8" x 2-1/8" x 1/4" 6061 Aluminum plate. All sample files are included in the download archive at the end of this document.



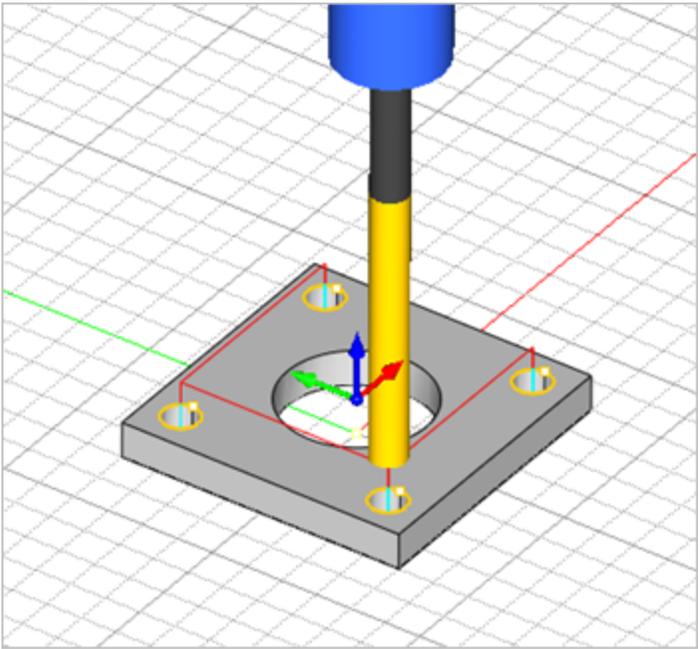
### 4.2 The Sample Toolpaths

The sample toolpaths included in the test part are **2½ Axis Profiling** with Cutter Compensation and External Corners Loop using a ¼" diameter end mill, **2½ Axis Hole Profiling** with a helical path using a 1/8" diameter end mill and a **Deep Drilling** operation using a ¼" drill.



Sample Part: 2½ Axis Profiling

Sample Part: 2½ Axis Hole Profiling



Sample Part: Deep Drilling

## 4.3 The Posted Setup G-Code File

We right-clicked on the **Setup** and selected **Post** to output one G-Code file that includes all three machining operations. The G-Code file is shown below. We have color-coded each line of the G-Code file to its corresponding section in the **PPG**.

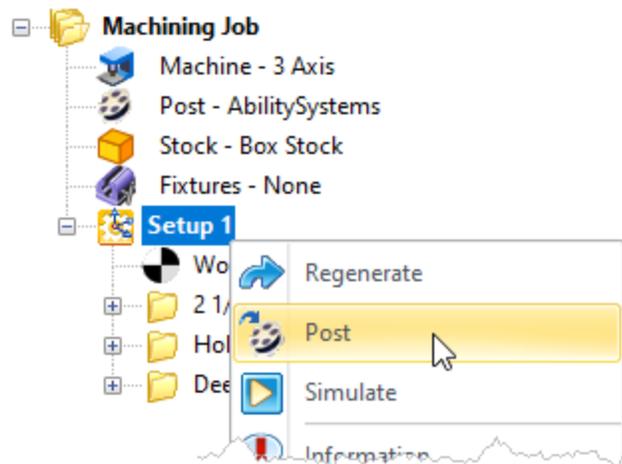
The **PPG** we have selected (*haas-blog.spm*) directs the format and content of code. Using these three toolpaths you see that in the 70 lines of code 12 of the 15 **PPG** sections are working together to define the G-Code file. The **PPG** sections are shown in the image below. Again, we have color-coded each section to help you identify each line in the G-Code file.

Three **PPG** sections are not represented in this G-Code sample. They are listed below along with the reason why. However these sections are discussed in this article.

1. **Setup**: This is used for 4 and 5 Axis setups only. It does not affect our G-Code sample file.
2. **Multi-Axis Motion**: Again, this is used for 4 and 5 Axis setups only. It does not affect our G-Code sample file.
3. **Cut Motion Start/End**: This **PPG** section is not required for our G-Code sample file.



PPG Dialog Selections



Highlight Setup, right-click and select Post

```

%
O1234
N001 G40 G49 G80
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)
N004 (Work Zero)
N005 G54
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
N013 G0
N014 G1Z-0.25 F97.5
N015 G41 G1 X-1.2533 Y-0.3524 D1 F146.25
N016 G1X-1.1678Y-0.1175
N017 G1X-1.125Y0.
N018 G1Y1. F195.
N019 G1Y1.25
N020 G17
N021 G03X-1.25Y1.125I-0.125J0.
N022 G1X-1.
N023 G1X1.
N024 G1X1.25
N025 G03X1.125Y1.25I0.J0.125
N026 G1Y1.
N027 G1Y-1.
N028 G1Y-1.25
N029 G03X1.25Y-1.125I0.125J0.
N030 G1X1.
N031 G1X-1.
N032 G1X-1.25
N033 G03X-1.125Y-1.25I0.J-0.125

```

The first half of the complete G-Code file (79 lines) is shown color coded to each PPG section that controls it.

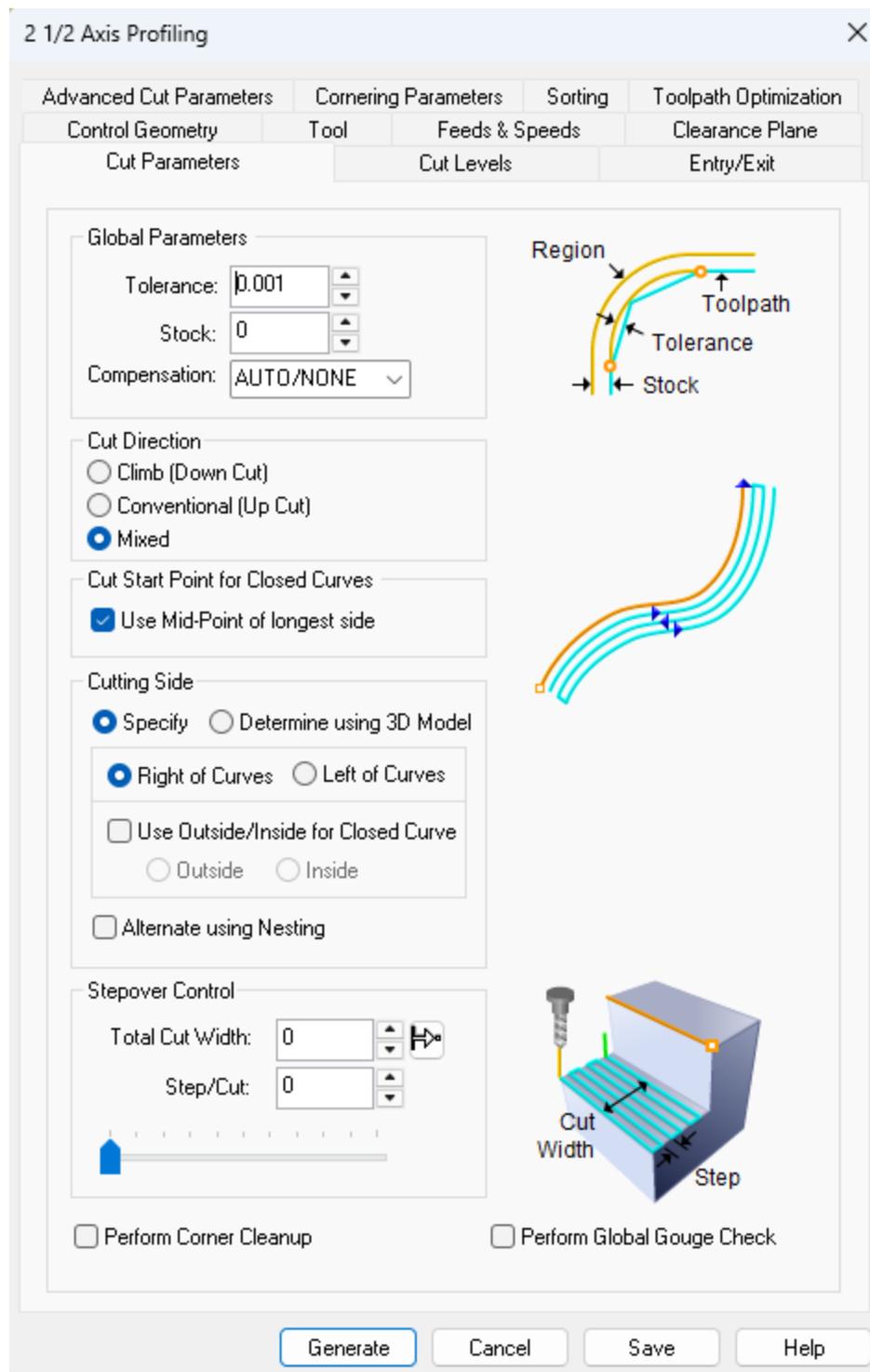
```

N033 G03X-1.125Y-1.25I0.J-0.125
N034 G1Y-1.
N035 G1Y0.
N036 G1Y0.1
N037 G1X-1.2105Y0.3349 F243.75
N038 G40
N039 G1X-1.296Y0.5698 F292.5
N040 G0Z0.25
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
N049 G0
N050 G1Z-0. F97.5
N051 G02Z-0.125I-0.4375J-0.0002 F195.
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 Length = 3.0 )
N058 G20 T3 M6
N059 G54
N060 S190 M3
N061 G90G0X0.75Y-0.75
N062 G43Z0.25H3
N063 M7
N064 G0
N065 G83X0.75Y-0.75Z-0.3222R0.1Q0.1F10.
N066X-0.75Y-0.75
N067X-0.75Y0.75
N068X0.75Y0.75
N069 G80
N070 G0Z0.25
N071 M30
%
```

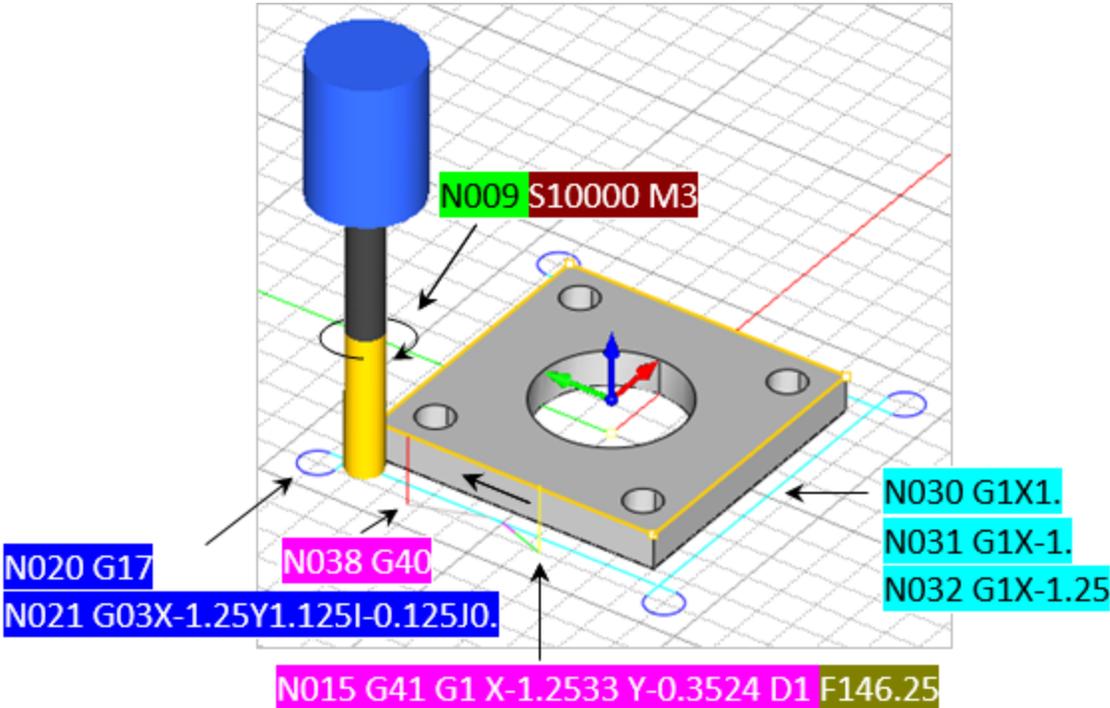
The second half of the complete G-Code file.

## 4.4 The 2½ Axis Profiling Operation G-Code

This toolpath strategy has [Compensation](#) enabled, a [Climb Cut Direction](#) and linear entry and exit motions. It also has [External Corner Type](#) set to [Loop](#) from the [Cornering Parameters](#) tab. With the [Fit Arcs](#) option enabled from the [Advanced Cut Parameters](#) tab, the four corner loops are automatically converted to arc motions, shown in dark blue in the illustration below.



The 2½ Axis Profiling Cut Parameters



The 2½ Axis Profiling Operation and G-Code

N006 (2 1/2 Axis Profiling (Outer))	General
N007 ( Tool Diameter = 0.25 Length = 2.0 )	Start/End
N008 G20 T1 M6	Tool Change
N009 S10000 M3	Setup
N010 G90G0X-1.2533Y-0.3524	Spindle
N011 G43Z0.25H1	Feed Rate
N012 M8	Motion
N013 G0	Circle
N014 G1Z-0.25 F97.5	Helical/Spiral
N015 G41 G1 X-1.2533 Y-0.3524 D1 F146.25	Multi Axis Motion
N016 G1X-1.1678Y-0.1175	Cutter Compensation
N017 G1X-1.125Y0.	Cut Motion Start/End
N018 G1Y1. F195.	Cycles
N019 G1Y1.25	Misc
N020 G17	Variables
N021 G03X-1.25Y1.125I-0.125J0.	
N022 G1X-1.	
N023 G1X1.	
N024 G1X1.25	
N025 G03X1.125Y1.25I0.J0.125	
N026 G1Y1.	
N027 G1Y-1.	
N028 G1Y-1.25	
N029 G03X1.25Y-1.125I0.125J0.	
N030 G1X1.	
N031 G1X-1.	
N032 G1X-1.25	
N033 G03X-1.125Y-1.25I0.J-0.125	
N034 G1Y-1.	
N035 G1Y0.	
N036 G1Y0.1	
N037 G1X-1.2105Y0.3349 F243.75	
N038 G40	
N039 G1X-1.296Y0.5698 F292.5	
N040 G0Z0.25	

The portion of the G-Code sample file created for the 2½ Axis Profiling operation is shown.

The following PPG sections affect how the G-Code for this operation is defined:

[General](#)

[Tool Change](#)

[Misc](#)

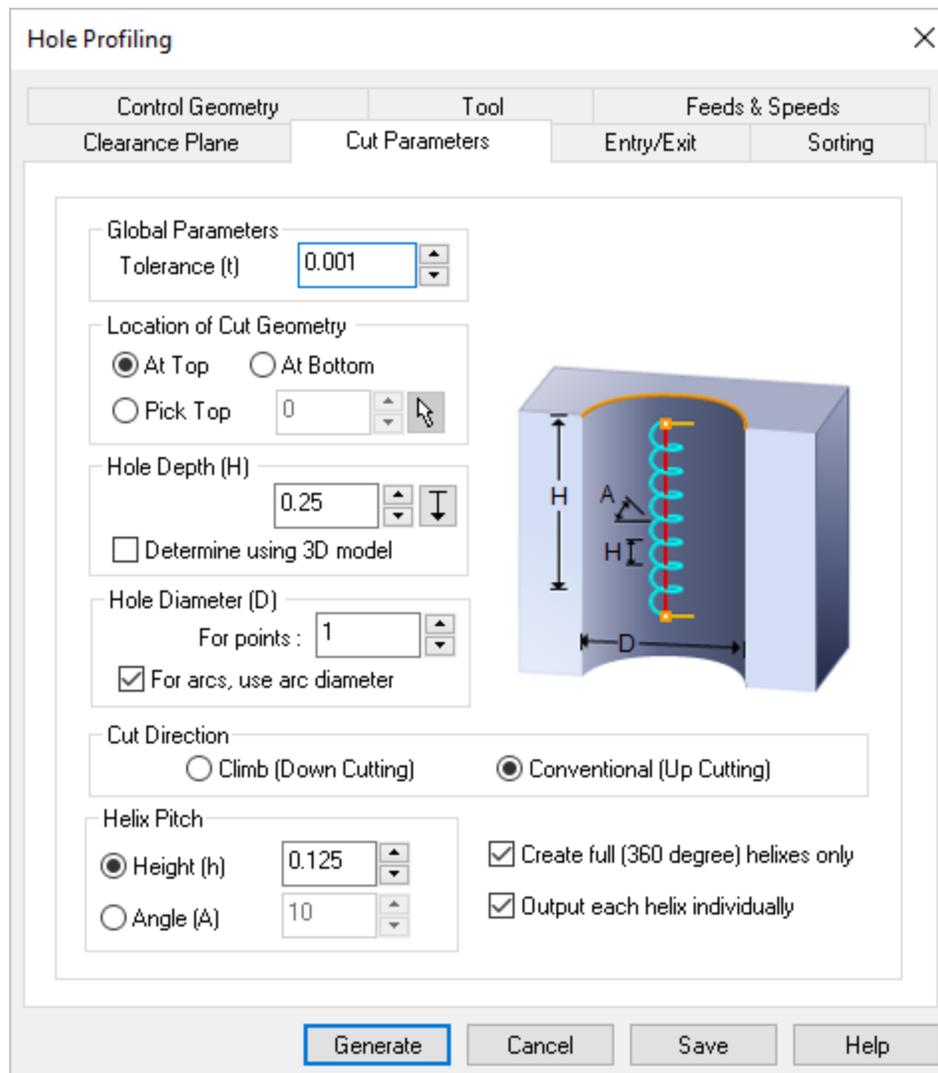
[Motion](#)

[Cutter Compensation](#)

[Cycles](#)

## 4.5 The 2½ Axis Hole Profiling Operation G-Code

This toolpath strategy is defined by a helical motion. Because [Hole Depth](#) is set to 0.25 and the [Helix Pitch Height](#) is set to 0.125, a total of two helical motions are required. The options to [Create full \(360 degree\) helixes only](#) and [Output each helix individually](#) are enabled. The [Cut Direction](#) is set to [Conventional \(Up Cutting\)](#).



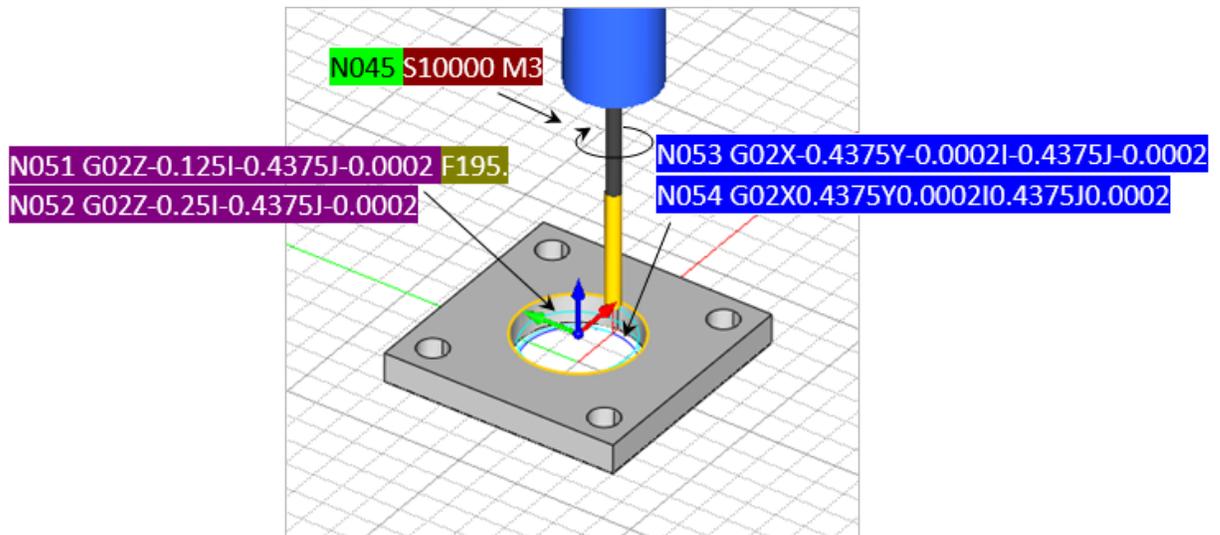
The screenshot shows the 'Hole Profiling' dialog box with the following settings:

- Control Geometry:** Clearance Plane
- Tool:** Cut Parameters
- Feeds & Speeds:** Entry/Exit, Sorting
- Global Parameters:** Tolerance (t) = 0.001
- Location of Cut Geometry:** At Top (selected), At Bottom, Pick Top = 0
- Hole Depth (H):** 0.25, Determine using 3D model (unchecked)
- Hole Diameter (D):** For points: 1, For arcs, use arc diameter (checked)
- Cut Direction:** Conventional (Up Cutting) (selected), Climb (Down Cutting)
- Helix Pitch:** Height (h) = 0.125 (selected), Angle (A) = 10
- Options:** Create full (360 degree) helixes only (checked), Output each helix individually (checked)

A 3D diagram of a hole in a block is shown, illustrating the helical motion. The hole depth is labeled 'H', the hole diameter is 'D', the helix pitch height is 'h', and the helix angle is 'A'.

Buttons at the bottom: Generate, Cancel, Save, Help

The 2½ Axis Hole Profiling Operation Cut Parameters



The 2½ Axis Hole Profiling toolpath has a Clockwise spindle direction and a Convention (Up Cutting) Cut Direction. Two full 360 degree helical motions are created. At the bottom of the hole profile two 180 degree arc motions are defined.

N041 (Hole Profiling)	General
N042 ( Tool Diameter = 0.125 Length = 2.0 )	Start/End
N043 G20 T2 M6	Tool Change
N044 G54	Setup
N045 S10000 M3	Spindle
N046 G90G0X0.4375Y0.0002	Feed Rate
N047 G43Z0.25H2	Motion
N048 M8	Circle
N049 G0	Helical/Spiral
N050 G1Z-0. F97.5	Multi Axis Motion
N051 G02Z-0.125I-0.4375J-0.0002 F195.	Cutter Compensation
N052 G02Z-0.25I-0.4375J-0.0002	Cut Motion Start/End
N053 G02X-0.4375Y-0.0002I-0.4375J-0.0002	Cycles
N054 G02X0.4375Y0.0002I0.4375J0.0002	Misc
N055 G0Z0.25	Variables

The portion of the G-Code sample file created for the 2½ Axis Hole Profiling operation is shown.

The following PPG sections affect how the G-Code for this operation is defined:

[General](#)

[Tool Change](#)

[Misc](#)

[Motion](#)

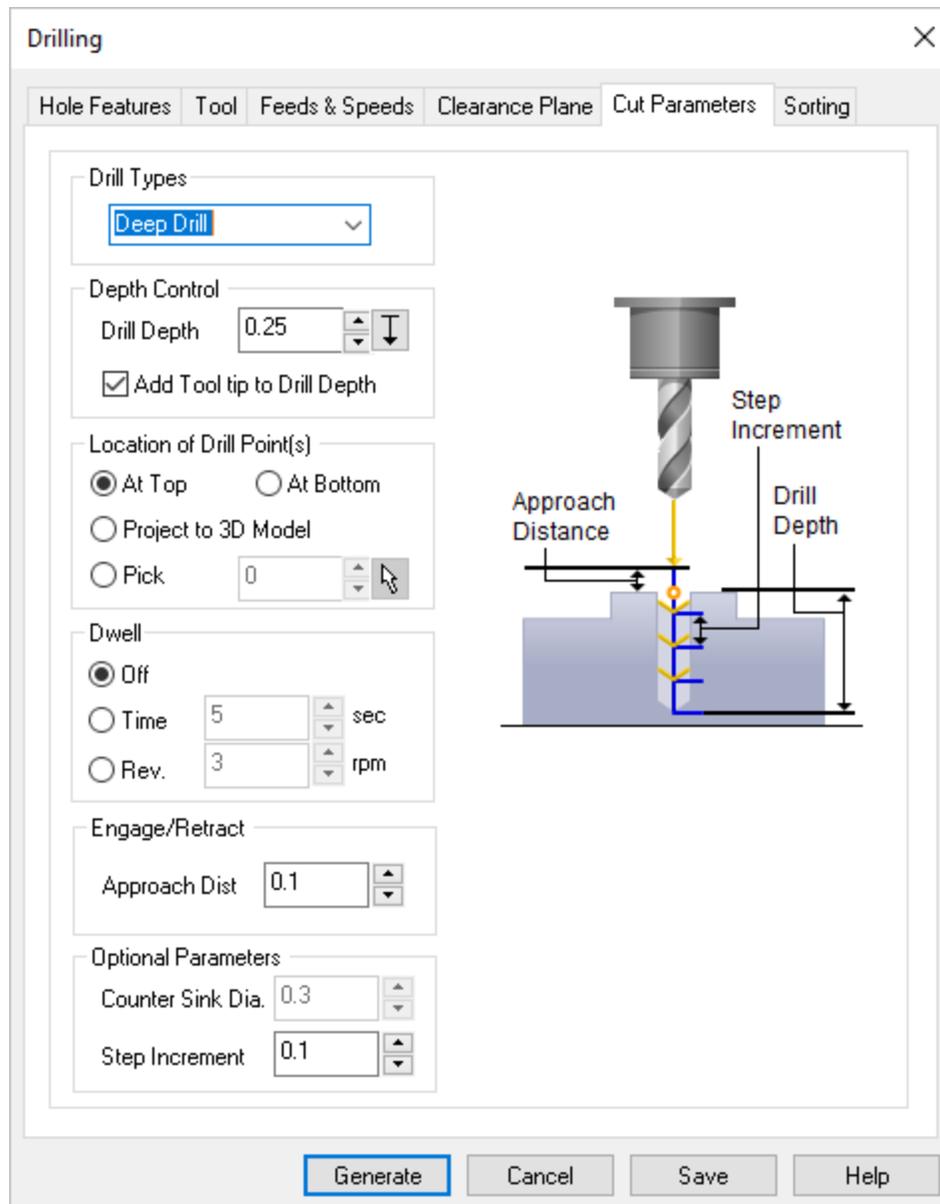
[Helical/Spiral](#)

[Circle](#)

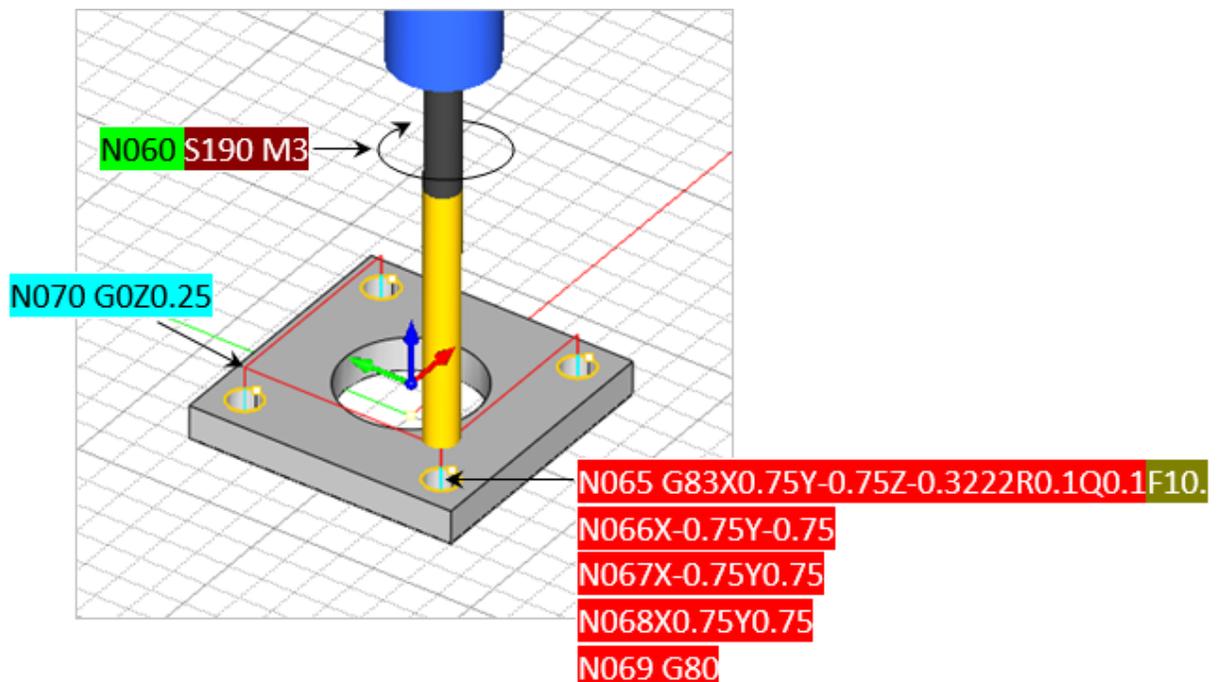
[Cycles](#)

## 4.6 The Deep Drilling Operation G-Code

This toolpath strategy is defined by a **Drill** cycle. The **Hole Depth** is set to 0.25. The **Drill Type** is set to **Deep Drill**. This enables the **Step Increment** option which is set to 0.1 creating three drill increments for each of the four holes (i.e., the drill will peck down three times to complete one hole). We have also enabled **Add Tool tip to Drill Depth**. This ensures a thru hole by taking into account the tip angle of the tool when calculating the drill depth. **Minimum Distance Sort** is also selected from the **Sorting** tab determining the fastest drill sequence.



The Cut Parameters tab for the Drilling operation.



The Drill toolpath is set to Deep Drill (G83) and has an Approach Distance of 0.1 and a Step Increment of 0.1. Add Tool tip to Drill Depth makes the total drill depth 0.32 (shown on the G83 line) when the stock is only 0.25.

N056 (Deep Drill)	General
N057 ( Tool Diameter = 0.25 Length = 3.0 )	Start/End
N058 G20 T3 M6	Tool Change
N059 G54	Setup
N060 S190 M3	Spindle
N061 G90G0X0.75Y-0.75	Feed Rate
N062 G43Z0.25H3	Motion
N063 M7	Circle
N064 G0	Helical/Spiral
N065 G83X0.75Y-0.75Z-0.3222R0.1Q0.1F10.	Multi Axis Motion
N066X-0.75Y-0.75	Cutter Compensation
N067X-0.75Y0.75	Cut Motion Start/End
N068X0.75Y0.75	Cycles
N069 G80	Misc
N070 G0Z0.25	Variables
N071 M30	
%	

The portion of the G-Code sample file created for the Drilling operation is shown.

The following PPG sections affect how the G-Code for this operation is defined:

[General](#)

[Tool Change](#)

[Misc](#)

[Motion](#)

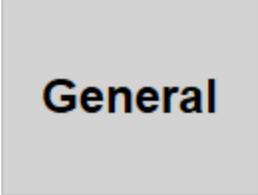
[Cycles](#)

## The Post-Processor Generator Dialog

The topics below show you how our sample post (*haas-blog.spm*) is defined within the PPG. We began with the standard *haas.spm* post definition file and added a few changes and customizations. ALL of the customizations were performed using the PPG dialog. The sample G-Code file was not edited or formatted manually except for adding our color-codes. We added the colors to make it easier for you to follow this article and better understand how the different PPG sections work together to produce the G-Code.

### 5.1 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 type="text" value="‰"/> File Stop Reading Char <input type="text" value="‰"/>	<b>Mode</b> <input checked="" type="radio"/> Absolute Code <input type="text" value="G90"/> <input type="radio"/> Incremental Code <input type="text" value="G91"/>	<b>Comments</b> <input checked="" type="checkbox"/> Output Comment Comment Start Char <input type="text" value="("/> Comment End Char <input type="text" value=")"/> <input type="checkbox"/> Output sequence number
<b>Sequence #s</b> <input checked="" type="checkbox"/> Use Sequence #s Prefix Letter <input type="text" value="N"/> Increment <input type="text" value="1"/> Start value <input type="text" value="1"/> <input checked="" type="checkbox"/> Show Leading Zeros Number of Digits <input type="text" value="4"/>	<b>Units</b> <input checked="" type="radio"/> Inch Code <input type="text" value="G20"/> <input type="radio"/> Metric Code <input 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 type="text" value=","/> End of Block Character <input type="text" value=""/> <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
<b>( ENDOF TOOL LIST )</b>
N003 (Setup 1)
N004 (Work Zero)
N005 G54
N006 (2 1/2 Axis Profiling (Outer))
<b>N007 ( Tool Diameter = 0.25 Length = 2.0 )</b>

## 5.2 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.



**Start  
End**

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]
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])
```

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>	

### 5.3 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][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] 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_ADJUST_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_ADJUST_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>	

## 5.4 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]
```

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

## 5.5 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.



Spindle

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

The Spindle section of the Post-Processor Generator (PPG) dialog is shown below. It includes the following fields and controls:

- Block Format:** Spindle Code: S
- Spindle Mode:** Constant Surface Speed, Constant Rotation Speed
- Spindle Direction:** Clockwise Rotation Code: M3, C-Clockwise Rotation Code: M4, Spindle Off Code: M5
- Spindle Block Format:** S[SPINDLE\_SPD][DELIMITER][SPINDLE\_ARC]
- Sample Output:** S0(M3/M4)
- Spindle RPM:** High Value: 10000, Low Value: 0, Scale Factor: 1, # of Decimal Places: 0, Show Trailing Zeros:

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

Spindle Block Format	
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	
<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>	

## 5.6 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.



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	

## 5.7 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

Output cut motion start macro before plunge motion

**Rapid Motion Block Format**

G Code

Block Format

Sample Output

**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	
<pre> N034 G1Y-1. ← Rapid (G0) or Linear (G1) N035 G1Y0.   Motions N036 G1Y0.1 N037 G1X-1.2105Y0.3349 F243.75 N038 G40 N039 G1X-1.296Y0.5698 F292.5 N040 G0Z0.25 </pre>	

## 5.8 PPG > Circle

The [Circle](#) section of the [PPG](#) defines all arc motions. At the top you see the [G-Code](#) and [Plane Code](#) sections. This is followed by the [Output Format](#) and [Arc Center \(I, J, K\)](#) sections. The options in these two sections define how the arc code is formatted and specifically how the arc center is calculated.



Circle

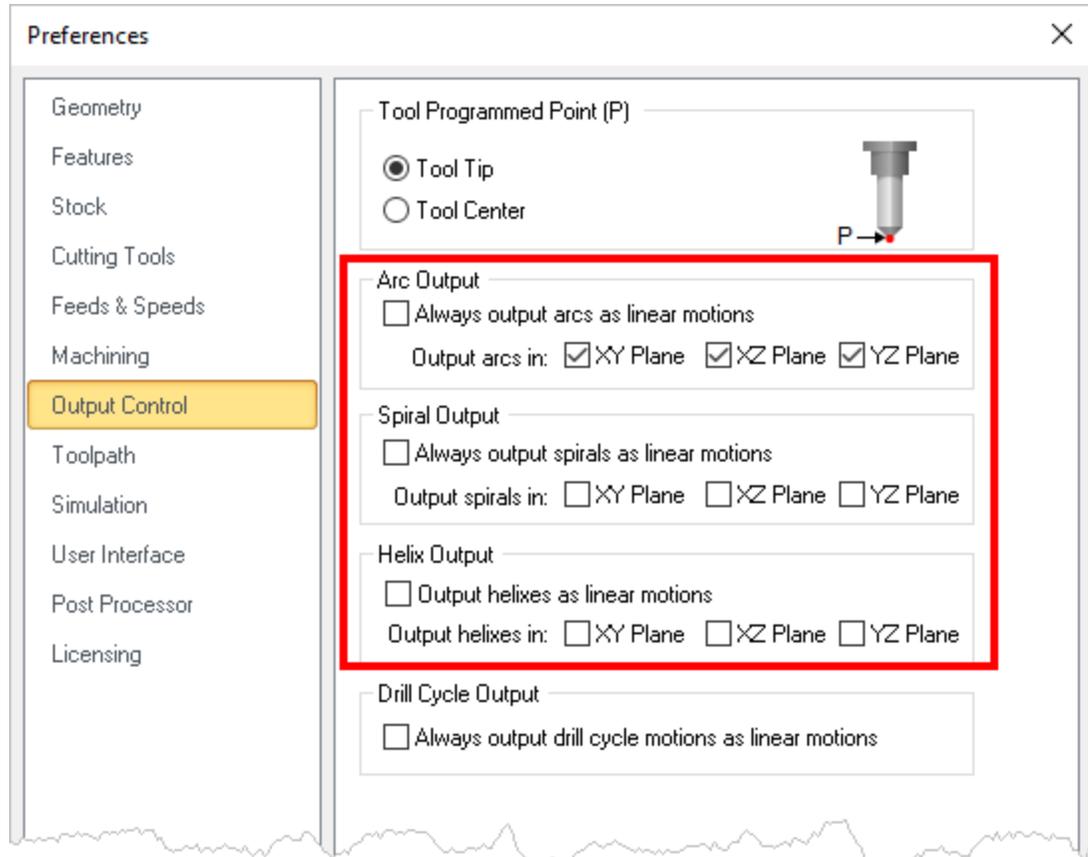
This is the color code for the [Circle](#) section.

Be sure to read the resulting G-Code sample illustration below to understand how the arc motions are defined for our sample part.

What is needed to output Arc Motions:

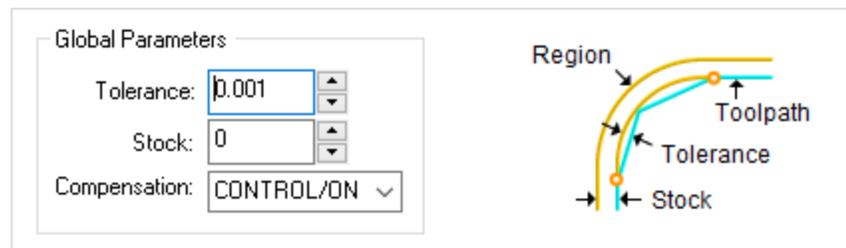
- Your CNC controller must be able to read G02 and G03 arc motions.

- In MecSoft CAM go to **MILL > Preferences > Machining** and uncheck **Output arcs as linear segments** and also uncheck **Output helix motions as linear segments**. If these boxes are checked, arcs are converted to linear motions.



MILL &gt; CAM Preferences &gt; Machining dialog

- In the **Advanced Cut Parameters** tab for the **Mop**, check the box for **Perform Arc Fitting** and enter a value for **Fitting tolerance (t)**. As a rule, this value should be 2 times the value you entered for **Tolerance** on the **Cut Parameters** tab. If you need a tighter tolerance, adjust these two values together.

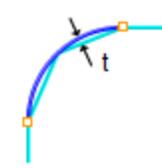


Tolerance value on the Cut Parameters tab for 2 ½ Axis Profiling

**Cut Arc Fitting**

Perform Arc Fitting

Fitting Tolerance (t)



Perform Arc Fitting on the Advanced Cut Parameters tab for for 2½ Axis Profiling

**G Code**

Clockwise Arc Code

C-clockwise Arc Code

**Plane Code**

XY

ZX

YZ

**Output Format**

I, J, K and Radius     I, J, K only     Radius only

Output values only when different     Use -R for CW Arcs

**Arc Center (I, J, K)**

Absolute     Vector from Center to Start

Vector from Start to Center     Unsigned Vector from Start to Center

**Block Format**

Select Plane

Plane XY

Plane ZX

[CIR\_PLANE]

[G\_CODE][NEXT\_X][NEXT\_Y][NEXT\_I][NEXT\_J]

Sample Output

{G17/G18/G19}

{G02/G03}X0.0Y0.0IQJ0

Limit Arcs to Angle

Default

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

Circle Block Format	
[CIR_PLANE] [G_CODE][NEXT_X][NEXT_Y][NEXT_I][NEXT_J]	
Variable	Derived From
[CIR_PLANE]	The <a href="#">Plane Code</a> defined in this section of the PPG (G17=XY Plane, G18=ZX Plane and G19=YZ Plane)
[G_CODE]	The <a href="#">G Code</a> 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 arc end point
[NEXT_Y]	The Y coordinate value for the arc end point
[NEXT_Z]	The Z coordinate value for the arc end point
[NEXT_I]	The X coordinate value for the arc center point
[NEXT_J]	The Y coordinate value for the arc center point
[NEXT_K]	The Z coordinate value for the arc center point
Resulting G-Code Sample	
<p>The diagram illustrates the definition of an arc motion. It shows a coordinate system with a grid. A blue arc is drawn counter-clockwise. Callouts include: 'Start X' pointing to the beginning of the arc; 'Start Y' pointing to the vertical coordinate of the start; 'Center I,J' pointing to the center of the arc; 'End X,Y' pointing to the end of the arc; and 'Direction CCW' indicating the arc's direction. A table below the diagram maps G-code parameters to these elements: N017 G1X-1.125Y0. (Start X), N018 G1Y1. F195. (Start Y), N019 G1Y1.25 (Start Y), N020 G17 XY Plane (XY Plane), and N021 G03X-1.25Y1.125I-0.125J0. (Dir, End XY, Center IJ).</p>	
<p>N017 G1X-1.125Y0.</p> <p>N018 G1Y1. F195.</p> <p>N019 G1Y1.25 Start Y</p> <p>N020 G17 XY Plane</p> <p>N021 G03X-1.25Y1.125I-0.125J0.</p> <p>Dir End XY Center IJ</p>	
<p>How arc motions are defined for our sample.</p>	

**Notes:**

1. I, J and K represent the arc center as an offset distance from the arc start. This is because in the PPG, [Arc Center \( I, J, K \)](#) section we have select [Vector from Center to Start](#). If you want the arc center posted in absolute coordinates, select the [Absolute](#) option.

2. Because the plane of the arc is XY, no Z or K values are shown for the arc start and end.
3. The arc start is located on line N017 because it is the end point of the last linear motion.
4. The red boxes are our added annotations and are not part of the G-Code sample.

## 5.9 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.

### Helical Spiral

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>	

## 5.10 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       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

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

## 5.11 PPG > Cutter Compensation

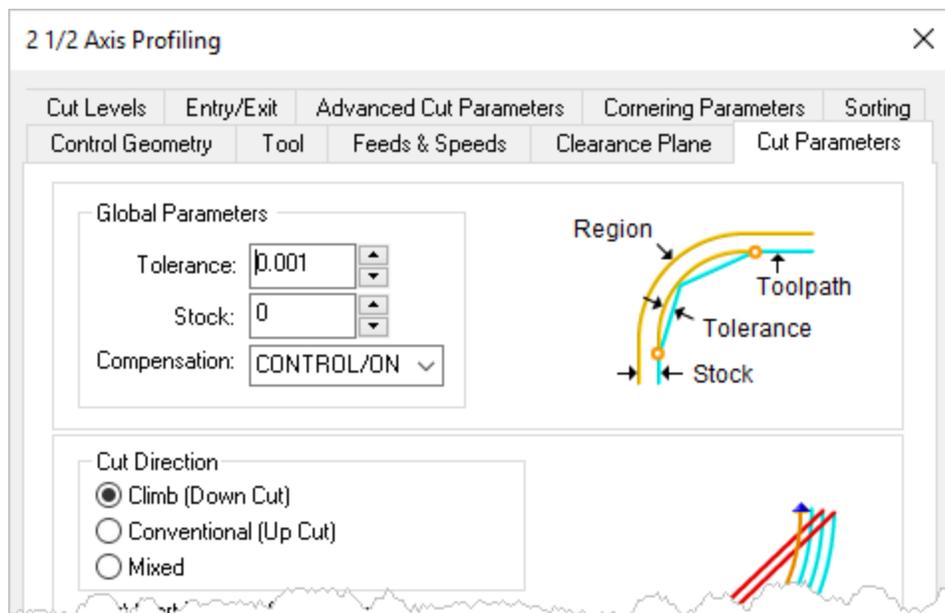
The [Cutter Compensation](#) section of the [PPG](#) is used to provide a way to adjust the toolpath at the machine to compensate for tool size, tool wear, and tool deflection. The dialog provides the block format for [Cutter Compensation Left \(G41\)](#), [Cutter Compensation Right \(G42\)](#) and [Cutter Compensation Off \(G40\)](#). G41 is used for a climb (down cut) direction with a clockwise (right-handed) spindle direction. G42 is used for a conventional (up cut) direction.



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

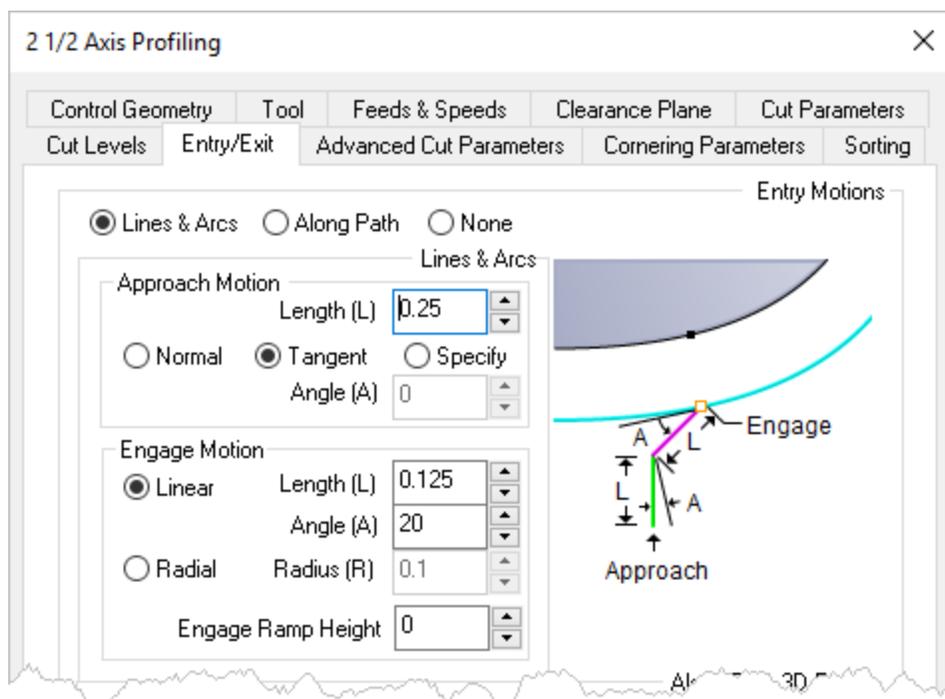
What is needed to output Cutter Compensation:

1. In the [Cut Parameters](#) tab under [Global Parameters](#) set [Compensation](#) to either [Auto/On](#) or [Control/On](#). In our example, we have it set to [Control/On](#).



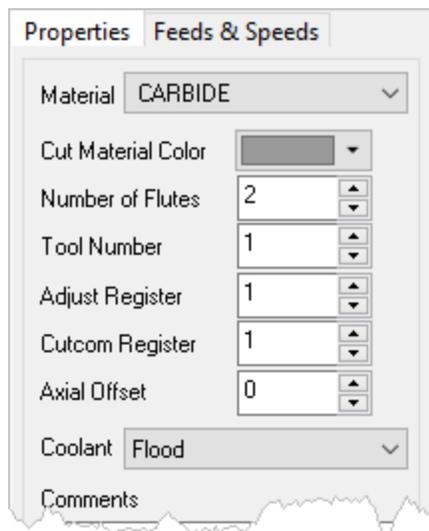
Compensation is set to CONTROL/ON and Cut Direction is set to Climb

2. In the **Cut Parameters** tab set **Cut Direction** to either **Climb** or **Conventional**. Do not set it to **Mixed** as this will disable **Cutter Compensation**. In our example we have it set to **Climb**.
3. Your first cut motion must be linear. In our example, on the **Entry/Exit** tab of the **2 1/2 Axis Profiling Mop**, **Entry Motions** is set to **Lines & Arcs** and the **Engage Motion** is set to **Linear**. Similarly, the **Exit Motions** is set to **Lines & Arcs** and the **Retract Motion** is set to **Linear**.



Entry/Exit is set to Lines & Arcs with Engage Motion set to Linear

4. In your tool definition make sure the **Cutcom Register** is set to the same value as the **Tool Number**. **Tool Number**, **Adjust Register** and **Cutcom Register** should all be the same value.



Properties Feeds & Speeds

Material CARBIDE

Cut Material Color

Number of Flutes 2

Tool Number 1

Adjust Register 1

Cutcom Register 1

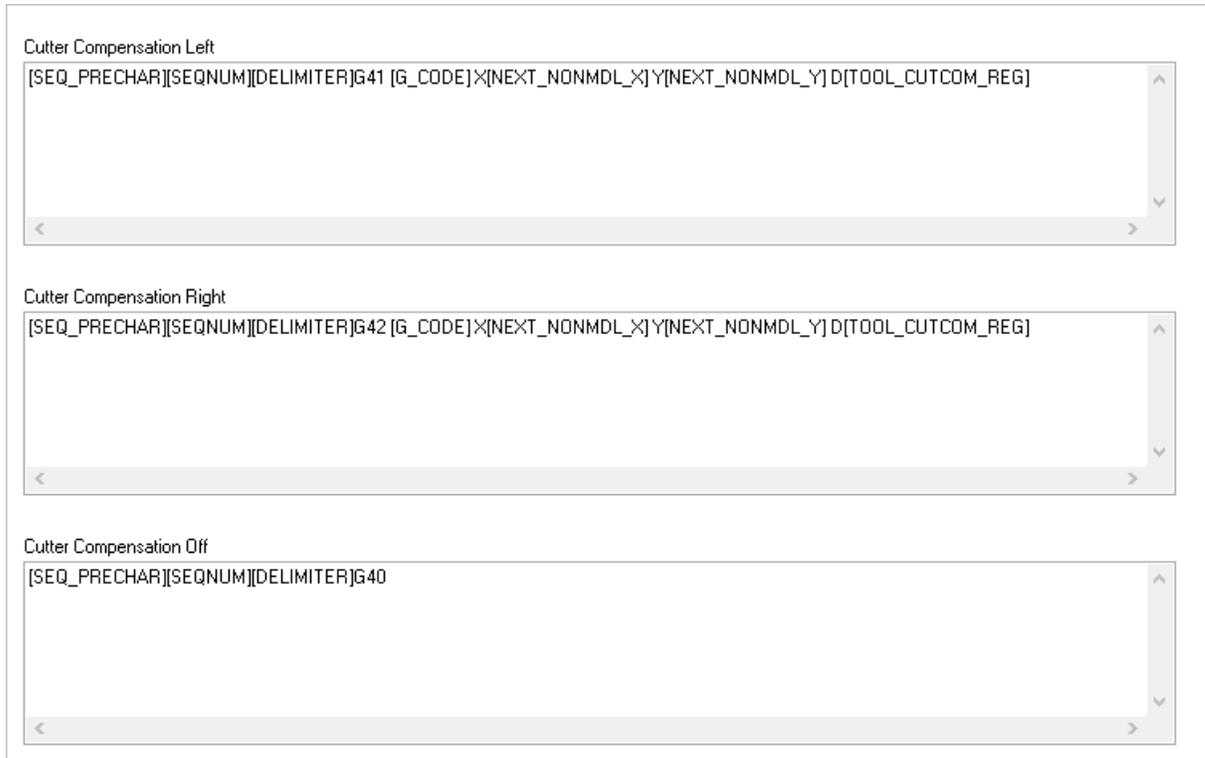
Axial Offset 0

Coolant Flood

Comments

Properties tab of the Create/Select Tool dialog

5. Make sure to specify the cutter compensation value and the compensation register in your controller.



Cutter Compensation Left

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

Cutter Compensation Right

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

Cutter Compensation Off

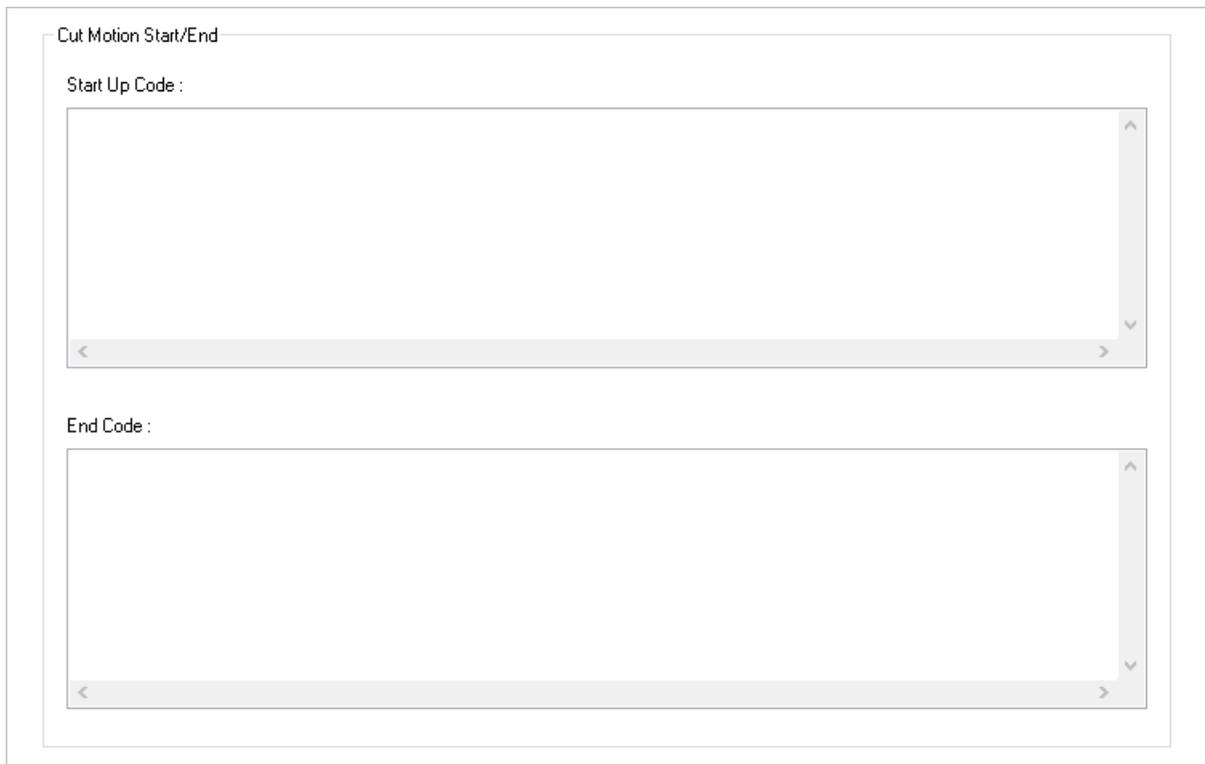
```
[SEQ_PRECHAR][SEQNUM][DELIMITER]G40
```

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

Cutter Compensation Block Format	
[SEQ_PRECHAR][SEQNUM][DELIMITER]G41 [G_CODE]X[NEXT_NONMDL_X]Y[NEXT_NONMDL_Y]D[TOOL_CUTCOM_REG]	
[SEQ_PRECHAR][SEQNUM][DELIMITER]G42 [G_CODE]X[NEXT_NONMDL_X]Y[NEXT_NONMDL_Y]D[TOOL_CUTCOM_REG]	
[SEQ_PRECHAR][SEQNUM][DELIMITER]G40	
Variable	Derived From
[G_CODE]	Will be a G1 linear motion
[TOOL_CUTCOM_REG]	MecSoft CAM > Tools > Create/Select Tool > Properties > Cutcom Register. Should be the same as the Tool Number.
ISO Codes	Function
G41	Cutter Compensation Left
G42	Cutter Compensation Right
G40	Cutter Compensation Off
Resulting G-Code Sample	
N012 M8	
N013 G0	
N014 G1Z-0.25 F97.5	
N015 G41 G1 X-1.2533 Y-0.3524 D1 F146.25 ← Cutter Compensation Left	
N016 G1X-1.1678Y-0.1175	
N037 G1X-1.2105Y0.3349 F243.75	
N038 G40 ← Cutter Compensation Off	
N039 G1X-1.296Y0.5698 F292.5	

## 5.12 PPG > Cut Motion Start/End

The [Cut Motion Start/End](#) section of the PPG is used for inserting additional code at the start or the end of every cut motion. For example, in the Motion section you can check the box to [Output cut motion start macro before plunge motion](#). If you add codes to the [Start Up Code](#) section below, this option will include it prior to a Z plunge motion.



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

## 5.13 PPG > Cycles

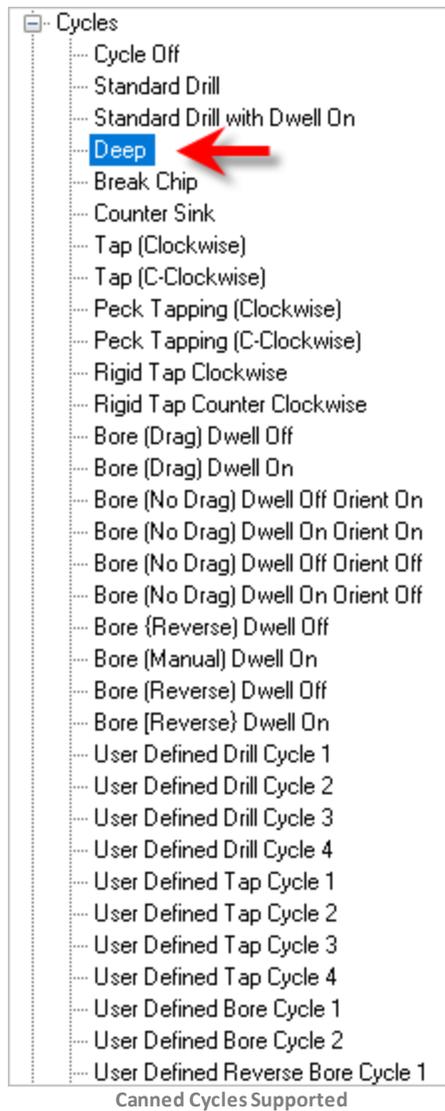
The [Cycles](#) section of the [PPG](#) is used for all canned cycles. These are cycles that your CNC controller is programmed to understand. For example all [Drill](#), [Tap](#) and [Bore Mops](#) are posted out as canned cycles. Other cycles include [User Defined Cycles](#), [Turn Thread Cycles](#) and [Machine Control Cycles](#).



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

All of these are defined in this [Cycles](#) section of the [PPG](#). We have shown the [Deep Drill](#) cycle dialog below since our part example has a [Drill](#) operation that is set to [Deep Drill](#).

On the left of the [PPG](#) dialog you would expand the [Cycles](#) selection to see all of the supported canned cycles. We are using the [Deep](#) cycle (G83).



Cycle G-Code:

Cycle Code:

Optimize Cycle Options

Optimize Cycle Output

Cycle Coordinate Block:

Dwell Options

Scale Factor  # of Decimal Places

Thread First Depth Options

Scale Factor  # of Decimal Places

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

**Note:** The [Cycle Code](#) shown below is actually one line! We separated it into two lines for presentation purposes. Your G83 cycle code should be all on one line as shown in our part sample below.

Cycle Code	
[SEQ_PRECHAR][SEQNUM][DELIMITER][G_CODE][NEXT_NONMDL_X][NEXT_NONMDL_Y] Z[CYCL_Z-DEPTH]R[CYCL_Z+CLEAR]Q[CYCL_INCR]F[CYCL_IPM]	
Variable	Derived From
[G_CODE]	This will be the canned cycle code. Our part sample requires the <a href="#">Deep Drill G83</a> cycle.
[CYCL_Z-DEPTH]	MecSoft CAM > Drill > Cut Parameters > Drill Depth.
[CYCL_Z+CLEAR]	MecSoft CAM > Drill > Cut Parameters > Approach Distance.
[CYCLE_INC]	MecSoft CAM > Drill > Cut Parameters > Step Increment.
[CYCLE_IPM]	MecSoft CAM > Drill > Feeds & Speeds > Cut (Cf)
ISO Codes	Function
G83	Deep Drill Cycle
R	Position of the R (retract) plane
Q	Depth of cut for each cutting feed (depth of each peck)
Resulting G-Code Sample	
<pre> N063 M7 N064 G0 N065 G83X0.75Y-0.75Z-0.3222R0.1Q0.1F10. ← Deep Drill (First Hole) N066X-0.75Y-0.75 N067X-0.75Y0.75 ← Optimized Cycle Output N068X0.75Y0.75 (for remaining Holes) N069 G80 ← Cycle Off N070 G0Z0.25 ← Retract to Clearance N071 M30 Plane %</pre>	

## Notes:

- Line N065 is the first deep drill cycle (G83). On this line Z is the Drill Depth of the hole, R is the Z location of the Approach Distance value and Q is the Step Increment value.
- Lines N066, N067 and N068 are the three remaining holes in the set. With Optimized Cycle Output checked, only the change in X and Y are posted.
- Line N069 cancels the canned cycle (G80).

## 5.14 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.

Category	Parameter	Value
Coolant Codes	Coolant Off	M9
	Coolant Mist	M7
	Coolant Tap	M51
	Coolant On	M51
	Coolant Flood	M8
	Coolant Thru	M50
Compensation	Compensation Off	G40
	Compensation Right	G42
	Compensation Left	G41
	Compensation Length	G43
Work (Fixture) Offsets	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.

## 5.15 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.

# Index

## - 2 -

- 2 Axis Hole Profiling 10
  - Cut Parameters 19
  - Posted G-Code 19
- 2 Axis Profiling 10
  - Cut Parameters 15
  - Posted G-Code 15

## - A -

- About this Guide 7
- Arc/Circle
  - Sample G-Code 37
  - Similar to Helix 37
  - Variables 37

## - C -

- Cutter Compension
  - Sample G-Code 43
  - Variables 43
  - What to do first 43
  - What to Know 43
- Cycles
  - Sample Deep Drill Cycle 47

## - D -

- Drilling 10
  - Cut Parameters 21
  - Posted G-Code 21

## - F -

- Feed Rate 32
  - Sample G-Code Control 34

## - G -

- General
  - G-Code Control 24, 32

## - H -

- Helical 43
- Helical Motions
  - Sample G-Code 41
  - Understanding 41

## - M -

- Macros 9
- Motion
  - Example G-Code 36
- Multi Axis Motion 42

## - P -

- Post-Processor 8
- Post-Processor Generator 8
  - Circle 37
  - Cut Motion Start/End 46
  - Cutter Compensation 43
  - Cycles 47
  - Feed Rate 34
  - General 24
  - Helical/Spiral 41
  - Misc 51
  - Motion 36
  - Multi Axis Motion 42
  - Sample G-Code 24, 25, 28, 32, 34, 36, 37, 41, 43, 47, 51
  - Setup 32
  - Spindle 32
  - Start/End 25
  - Tool Change 28
  - Variables 52
- Post-Processor Generator Dialog 24
- Print Tool List 28

## - S -

- Sample G-Code
  - 2 Axis Hole Profiling 19
  - 2 Axis Profiling 15
  - Arc/Circles 37
  - Cutter Compensation 43
  - Drill Cycles 47

## Sample G-Code

Drilling 21

File 12

Flood/Mist 51

Helical 41

Tool Change 28

Sample Toolpath 10

Set Post 10

Setup 10

## Spindle

Example G-Code 32

Start/End G-Code Control 25

**- T -**

## Tool Change

Example G-Code 28

First Tool Load Macro 28

Print Tool List 28

Tool Change Macro 28

Variables Used 28

**- V -**

Variables 9