Post-Processor (PPG) Decoded

VisualCAD/CAM 2025

Published: February 2025

N049 G0
N050 G1Z-0. F29.334
N051 G02Z-0.125I-0.4375J-0.0002 F14.667
N052 G02Z-0.25I-0.4375J-0.0002
N053 G02X-0.4375Y-0.0002I-0.4375J-0.0002
N054 G02X0.4375Y0.0002I0.4375J0.0002
N055 G0Z0.25
N056 (Deep Drill)
N057 (Tool Diameter = 0.25 Leng
N058 G20 T3 M6
N059 G54
N060 S190 M3
N061 G90G0X0.75Y-0.75

MecSoft Corpotation

Table of Contents

Quick Start 3

Resource Guide 6

Introduction 7

1 What is a Post-Processor? 8
2 What is the PPG? 8
3 PPG Variables & Macros 9

Decoding a Sample G-Code File 10

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

The Post-Processor Generator Dialog 24

1 PPG > General 24
2 PPG > Start/End 25
3 PPG > Tool Change 28
4 PPG > Setup 32
5 PPG > Spindle 33
6 PPG > Feed Rate 34
7 PPG > Motion 36
8 PPG > Circle 37
9 PPG > Helical/Spiral 40
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 50
15 PPG > Variables 52

Index 53

Quick Start

PPG Module 2025

Prefer Printed Documentation? Click Here!

What's New | Quick Start Play List

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

How to Access the Quick Start Guide Documents

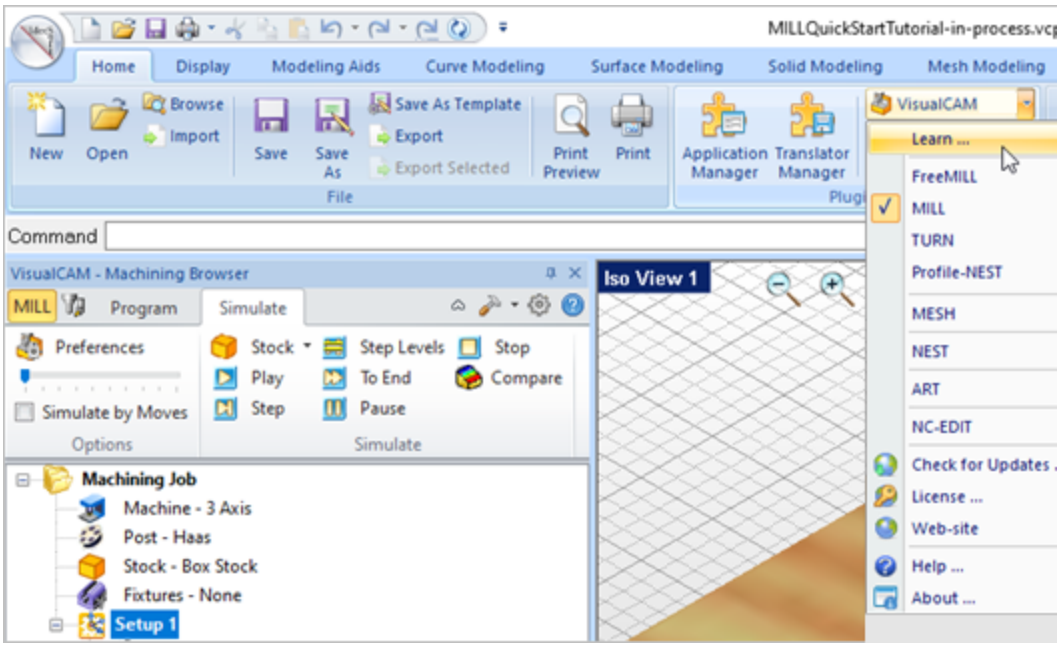
To help you quickly get started in working with each module, select one of the Help buttons located on the VisualCAD/CAM Learning Resources dialog.

You will find:

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

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

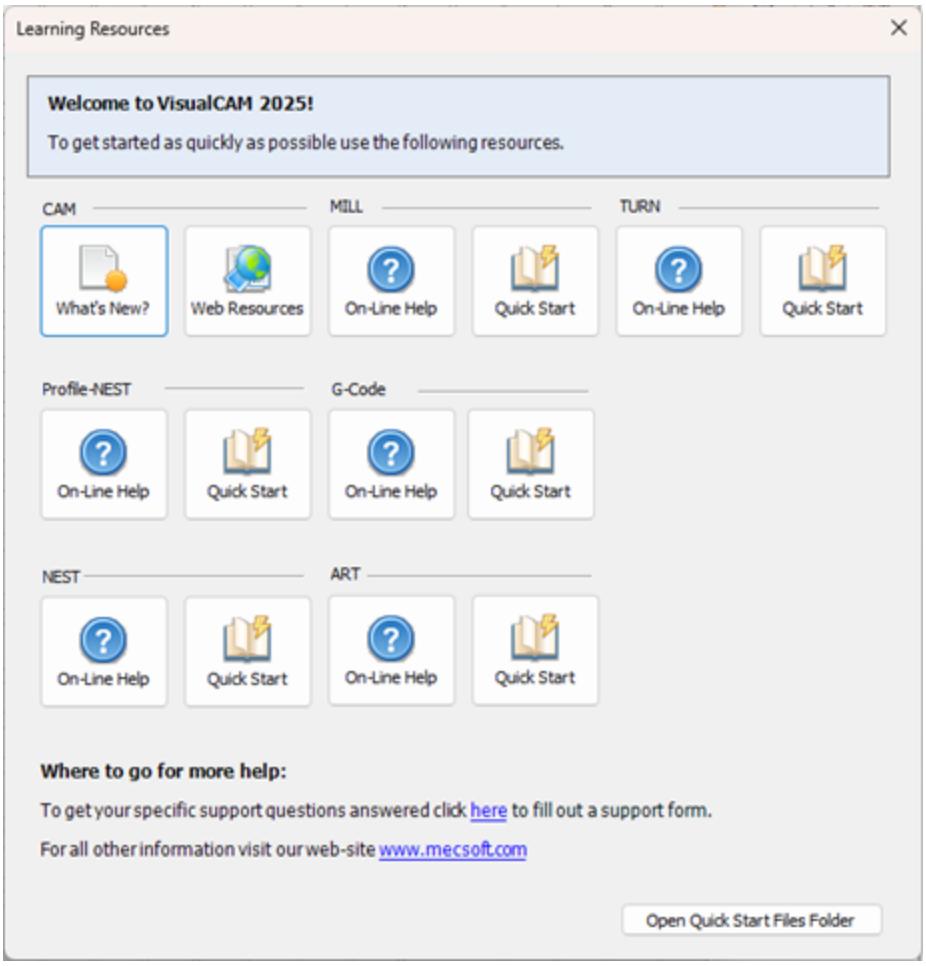
1. From the VisualCAD Home Ribbon Bar, drop down the Main menu and select Learn ...



To access the Learning Resources dilog in VisualCAM

2. Select a document from the Learning Resources dialog to get started using the module of your choice.

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



Learning Resources Dialog

Related Topics

Find More Resources

Resource Guide

Download this PDF Guide for a list of the available VisualCAD/CAM Resources.

2025 VisualCAD/CAM Resource Guide

The 2025 VisualCAD/CAM Resource Guide!

18 Pages

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

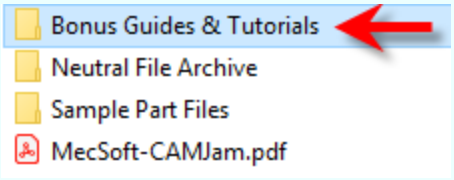
Prefer Printed Documentation? Click Here!

What's New | Quick Start Play List

Introduction

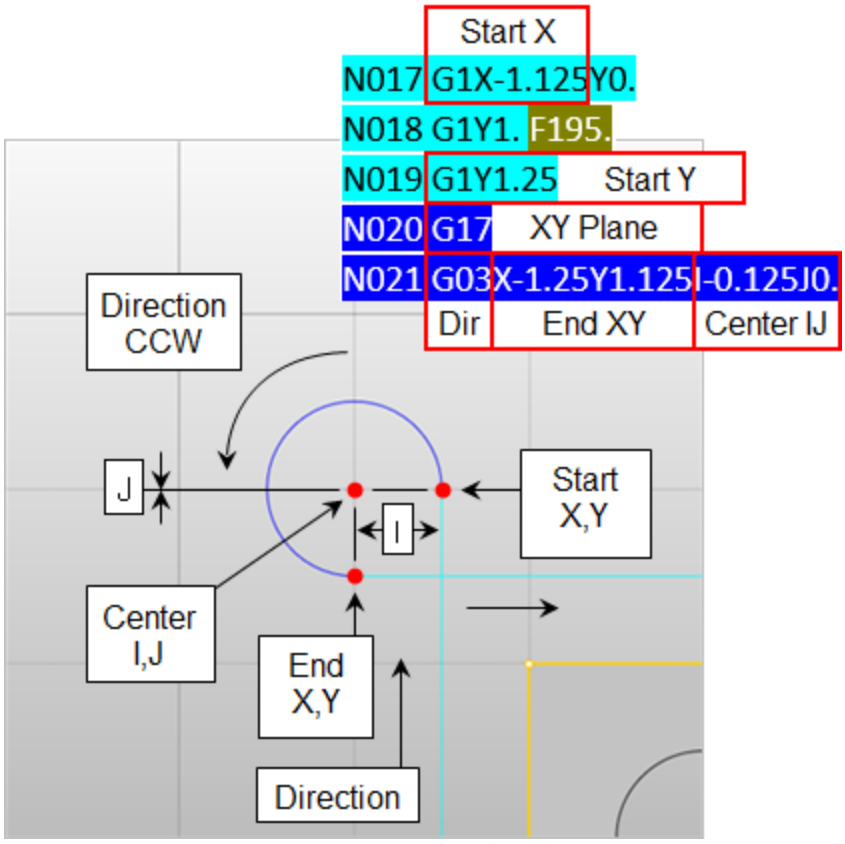
This guide provides the basis for understanding how the Legacy Post-Process Generator (PPG) operates and how you can use it to customize your posted g-code files. It uses a simple part file example with three toolpath operations. Each toolpath is illustrated, examined and the resulting G-Code is color 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.



CAMJam Download Archive

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



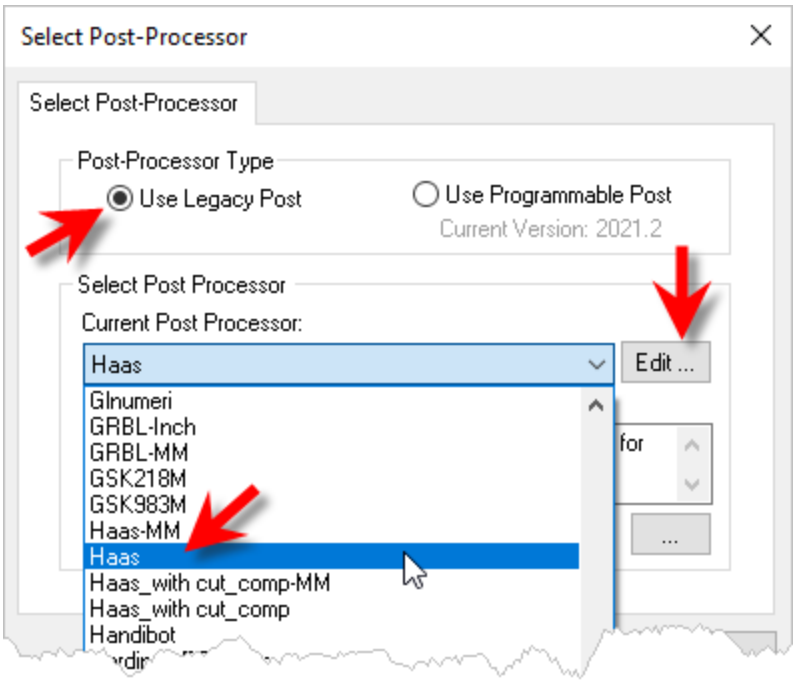
How arc motions are defined for our sample.

3.1 What is a Post-Processor?

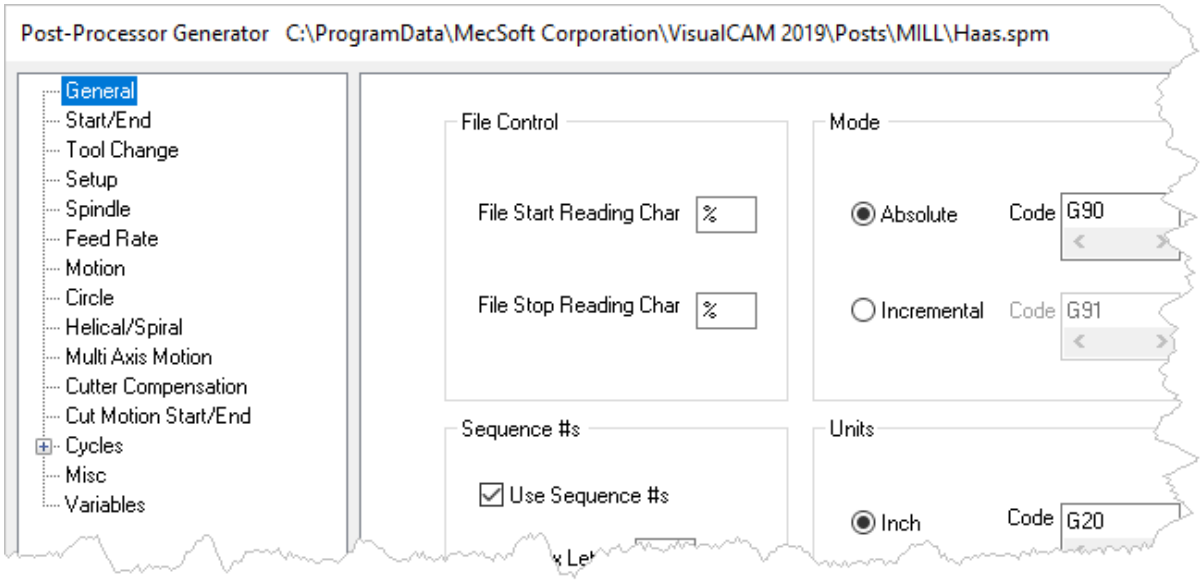
The post-processor converts the toolpath operations you create in MecSoft CAM into a suitable format (called G-Code) that your CNC machine controller can interpret. The G-Code that MecSoft CAM's post-processors create adheres to ISO6983 (the international standard for numerical controls). The post-processor 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

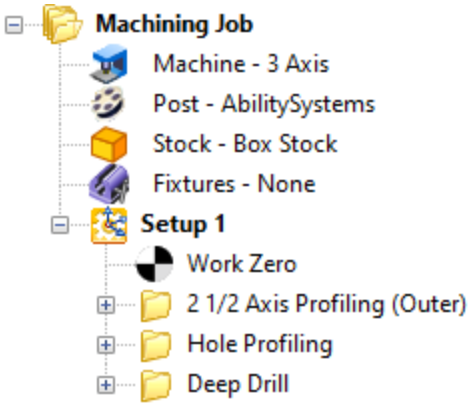
The PPG uses hundreds of variables and macros that allow you to customize each post definition file. The names of the variables are pre-defined. The values for many of the variables are extracted from your CAM cutting tools, setups, toolpath operations, etc. Other variable values are defined within the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

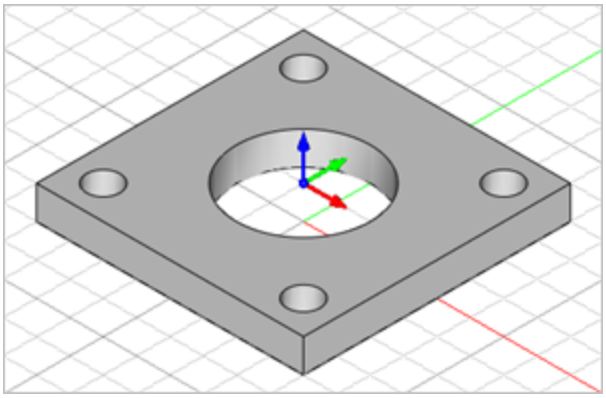
In this guide we will use the following sample part and toolpaths to see how internal variables for CAM definitions such as stock size, tool size, feeds & speeds parameters, toolpath operation types and various other cutting parameters, are used by the post-processor generator and the post definition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



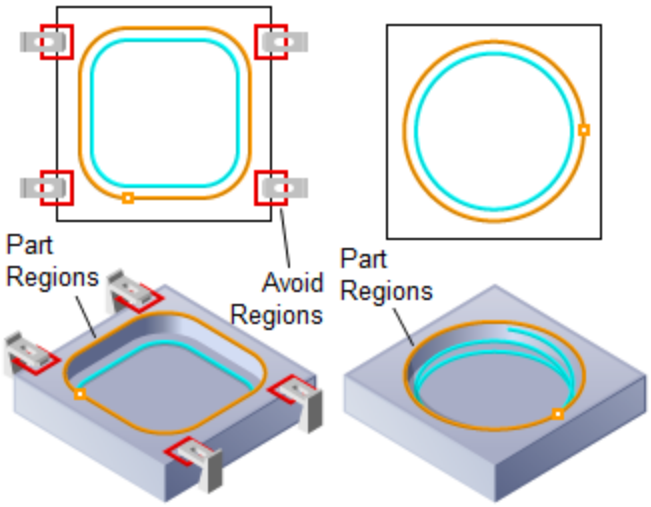
The Sample Machining Job



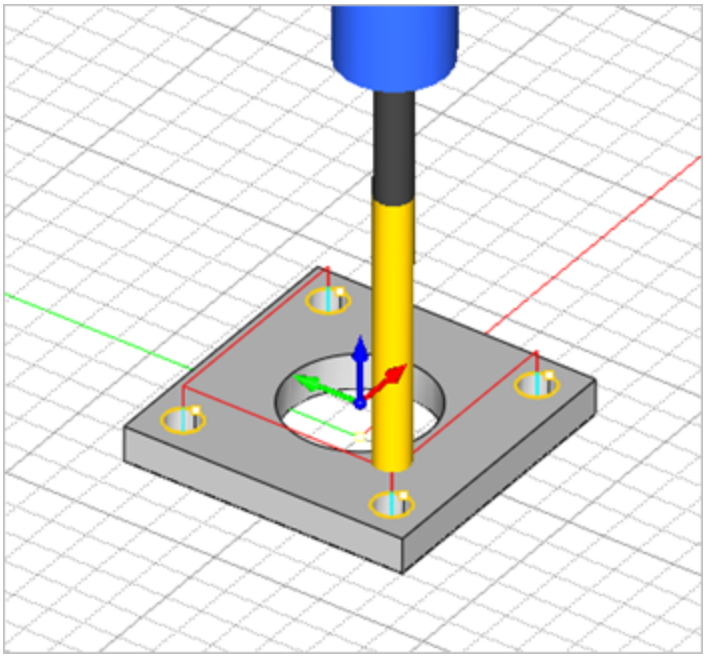
The Sample Part

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

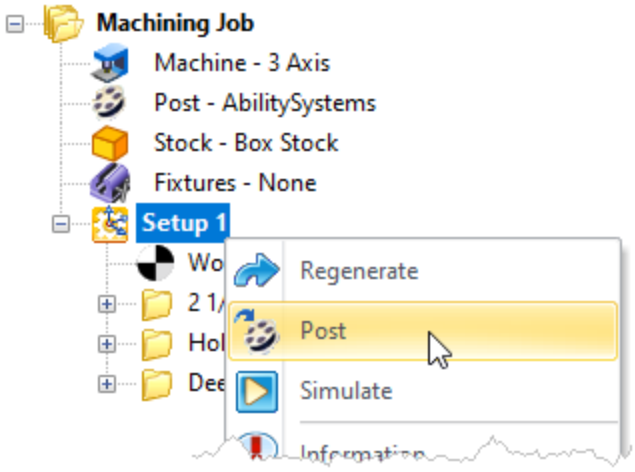
We right-clicked on the Setup and selected Post to output one G-Code file that includes all three machining operations. The G-Code file is shown below. We have color-coded each line of the G-Code file to its corresponding section in the PPG.

PPG Dialog Selections

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 is 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.



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
```

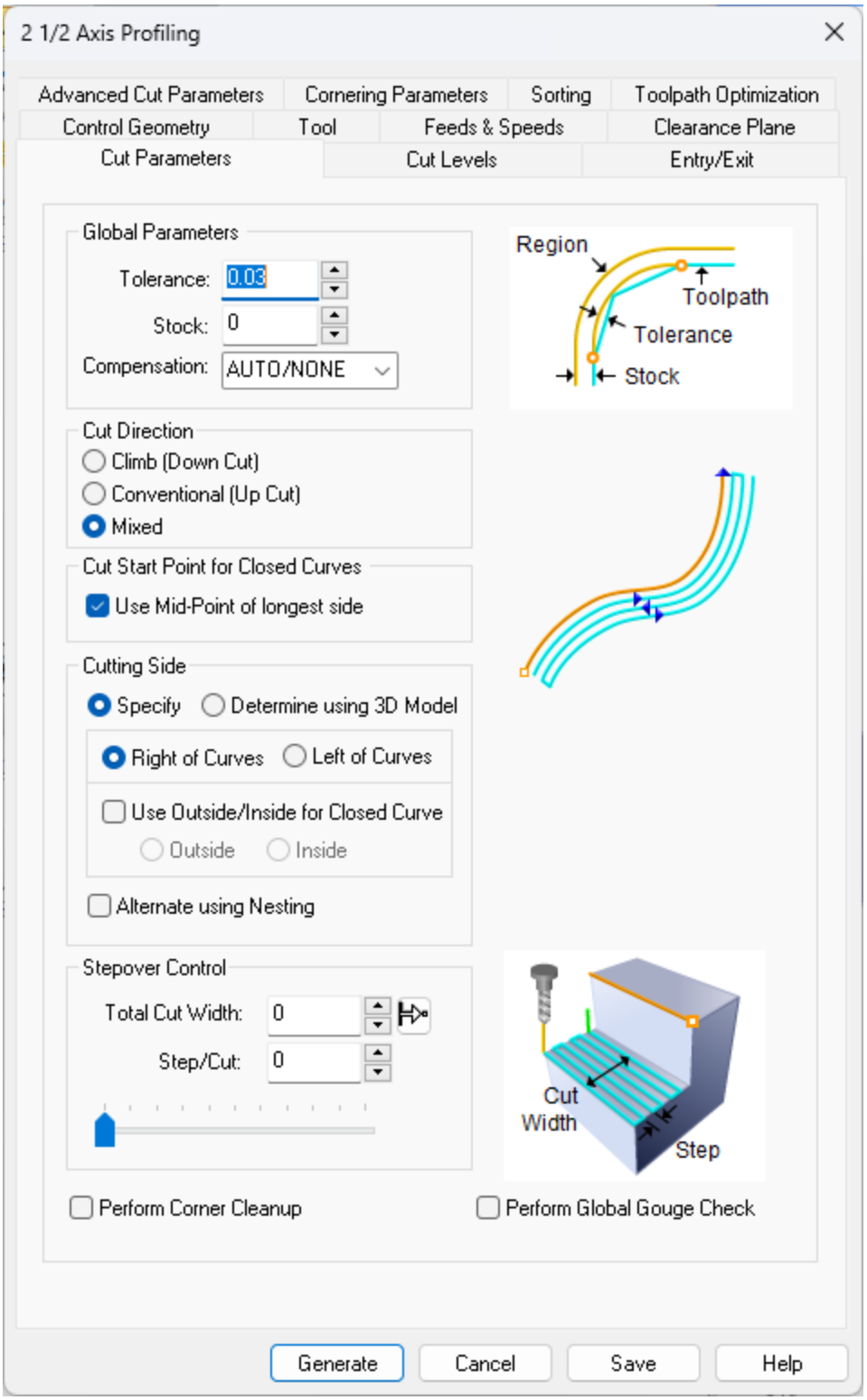
The first half of the complete G-Code file (79 lines) is shown color coded to each PPG section that controls it.

```
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
%
```

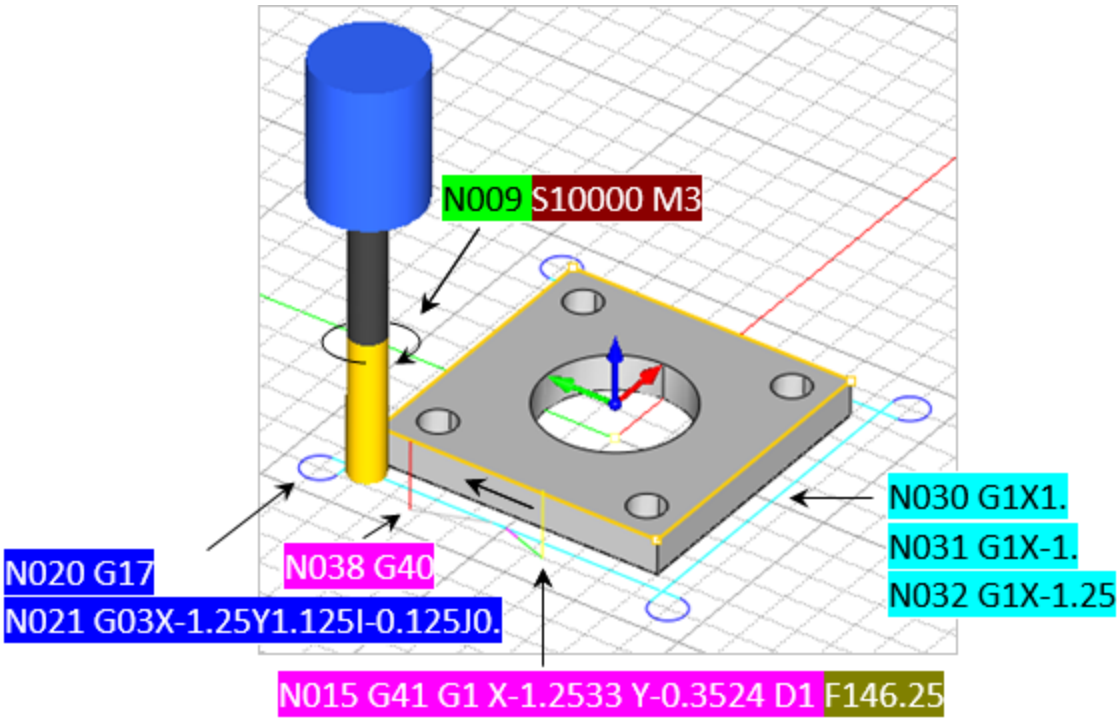
The second half of the complete G-Code file.

4.4 The 2½ Axis Profiling Operation G-Code

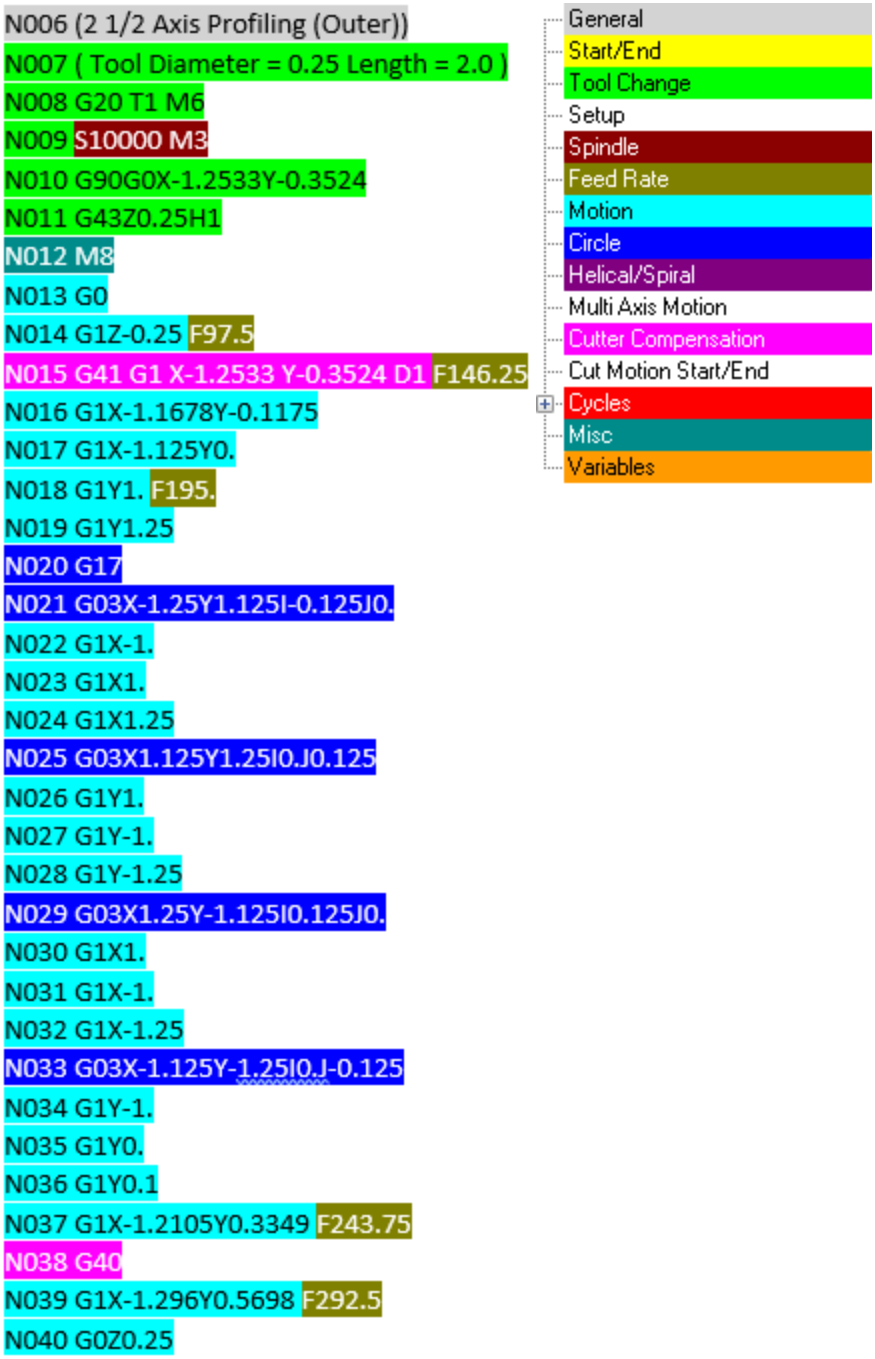
This toolpath strategy has Compensation enabled, a Climb Cut Direction and linear entry and exit motions. It also has External Corner Type set to Loop from the Cornering Parameters tab. With the Fit Arcs option enabled from the Advanced Cut Parameters tab, the four corner loops are automatically converted 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



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:

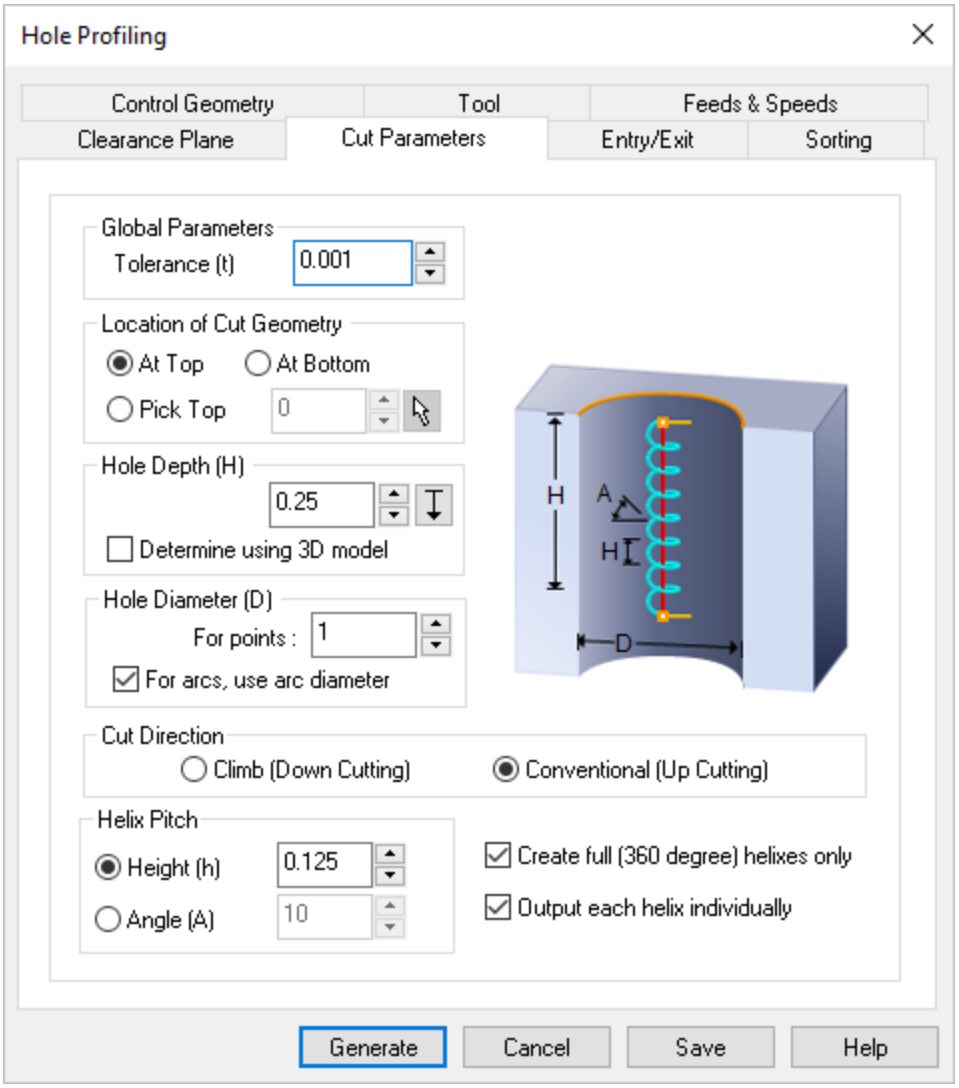
- General
- Tool Change
- Misc
- Motion

Cutter Compensation

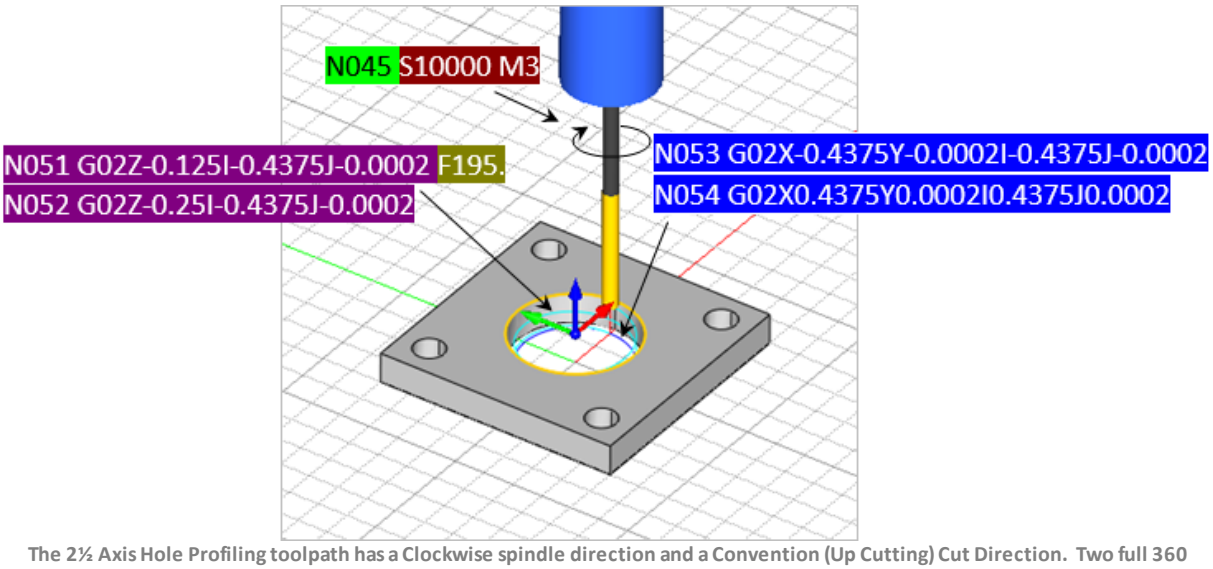
Cycles

4.5 The 2½ Axis Hole Profiling Operation G-Code

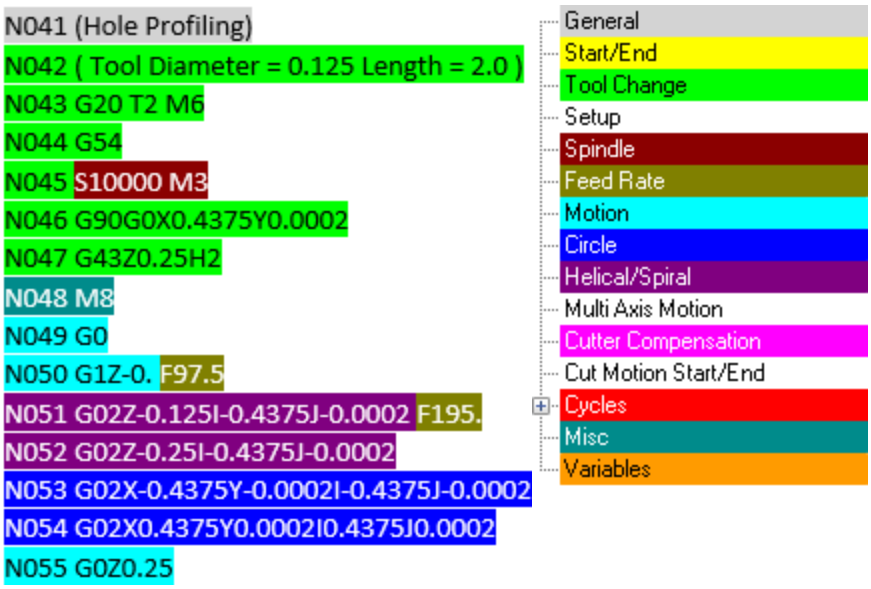
This toolpath strategy is defined by a helical motion. Because Hole Depth is set to 0.25 and the Helix Pitch Height is set to 0.125, a total of two helical motions are required. The options to Create full (360 degree) helixes only and Output each helix individually are enabled. The Cut Direction is set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



The 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.



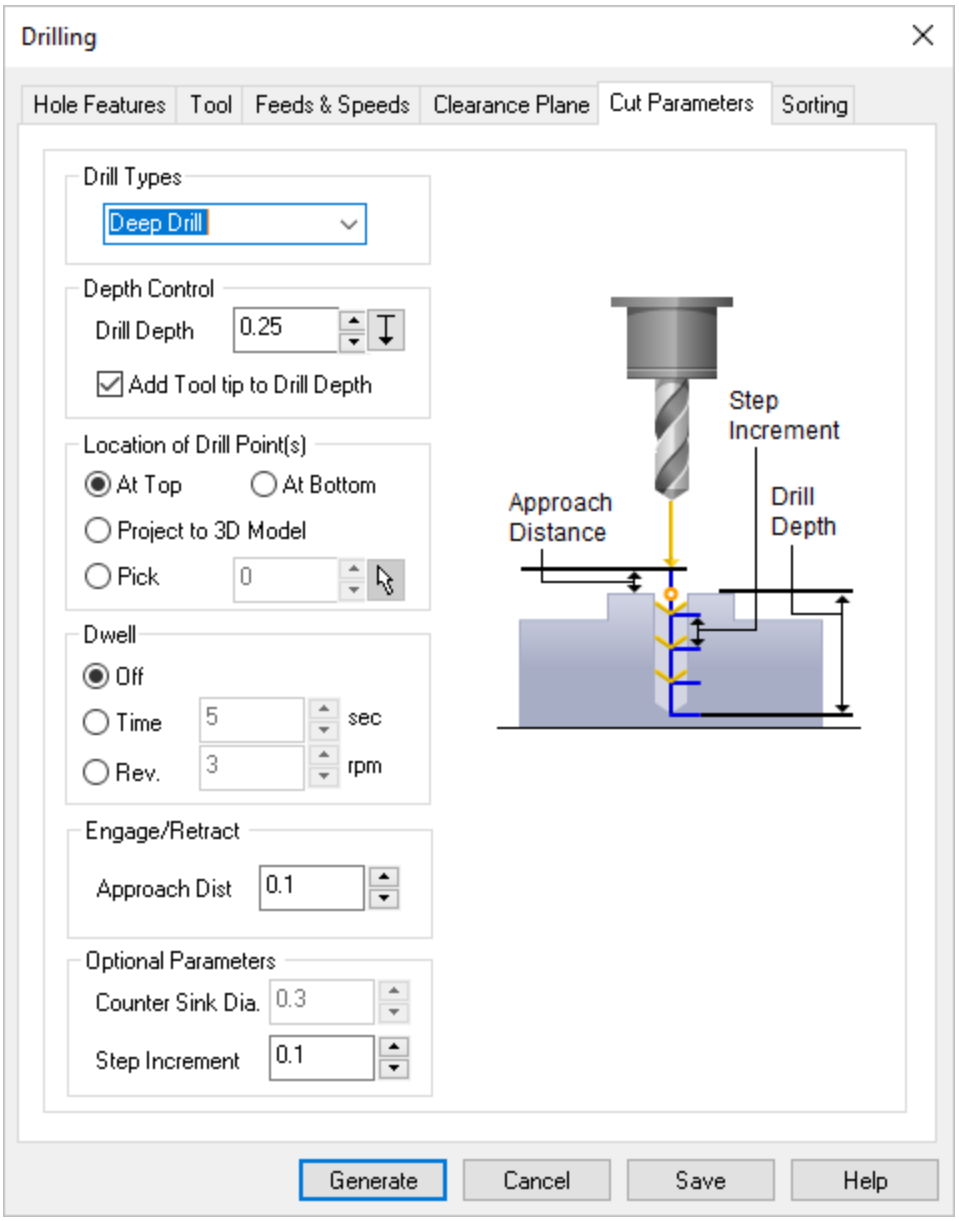
The portion of the G-Code sample file created for the 2½ Axis Hole Profiling operation is shown.

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

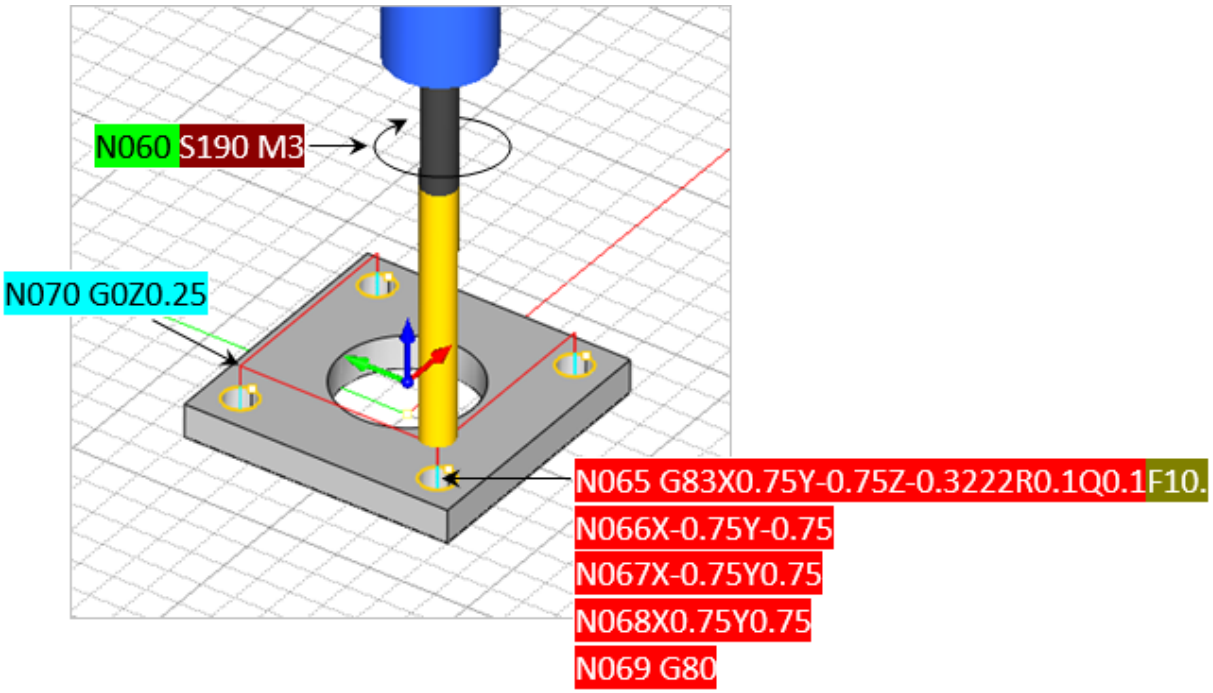
- General
- Tool Change
- Misc
- Motion
- Helical/Spiral
- Circle
- Cycles

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)
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 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

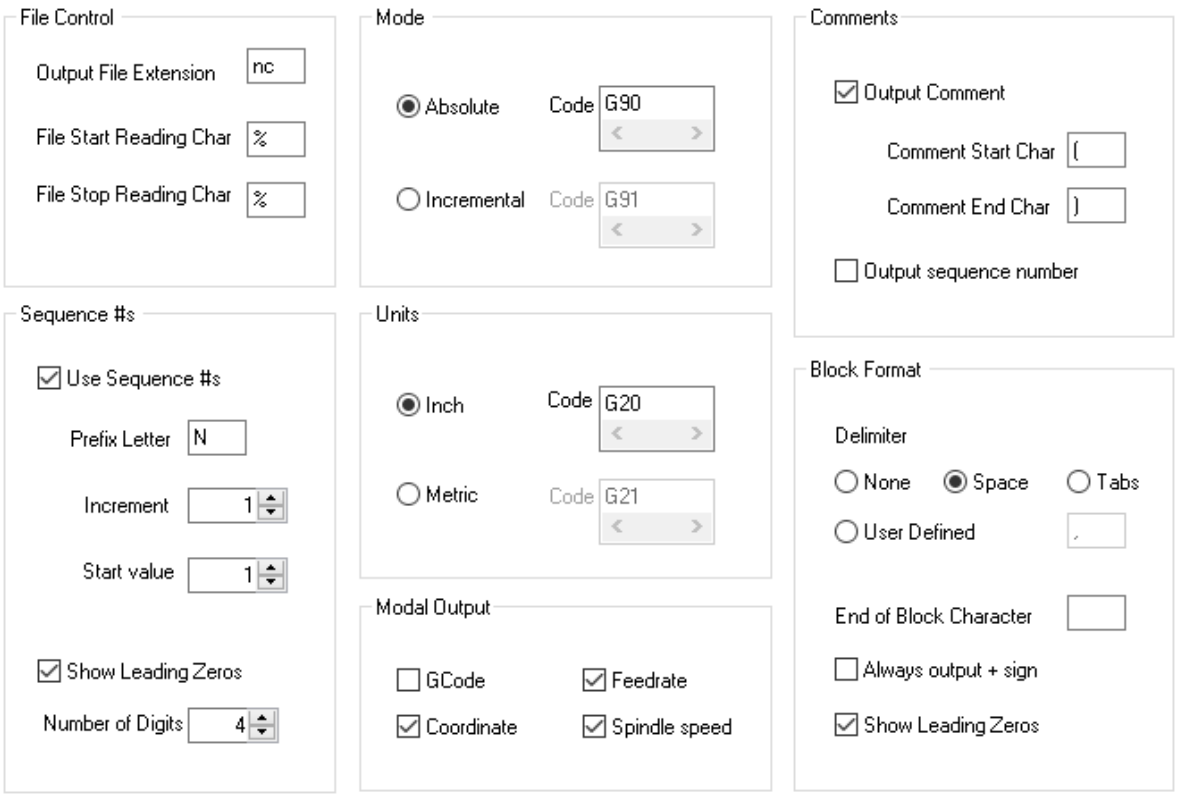
The Post-Processor Generator Dialog

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.

This is the color code for the General section.



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)

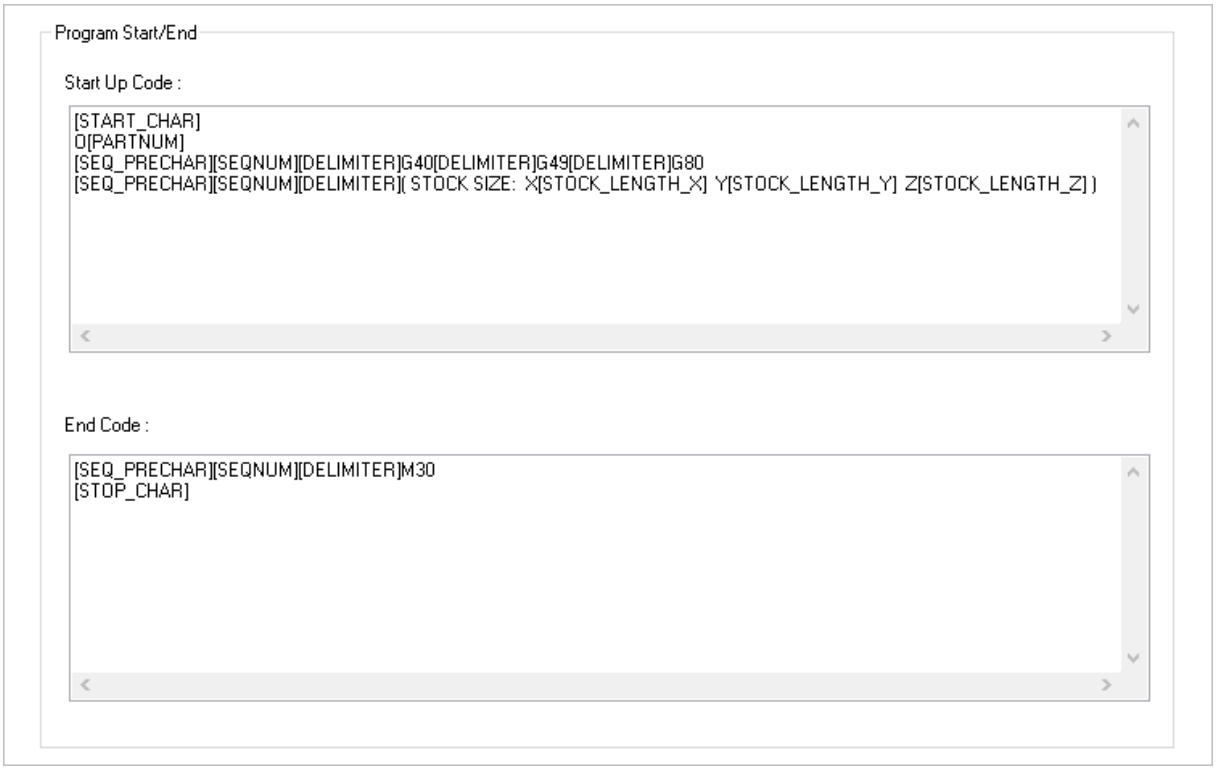
5.2 PPG > Start/End

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 ( ).



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

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])	
Variable	**Derived From**
[START_CHAR]	File Start Reading Character from the General section of the PPG.
[DELIMITER]	Delimiter type from the General 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 General section of the PPG.
[SEQNUM]	Use Sequence #s from the General 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	
% O1234 N001 G40 G49 G80 N002 (STOCK SIZE: X2.125 Y2.125 Z0.25) (BEGIN TOOL LIST)	

End Code	
[SEQ_PRECHAR][SEQNUM][DELIMITER]M30 [STOP_CHAR]	
Variable	**Derived From**
[DELIMITER]	Delimiter type from the General section of the PPG.

[SEQ_PRECHAR]	Prefix Letter from the General section of the PPG.
[SEQNUM]	Use Sequence #s from the General section of the PPG.
[STOP_CHAR]	File Stop Reading Character from the General section of the PPG.
ISO Codes	Function
M30	End of Program
Resulting G-Code Sample	
N069 G80 N070 G0Z0.25 N071 M30 %	

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.

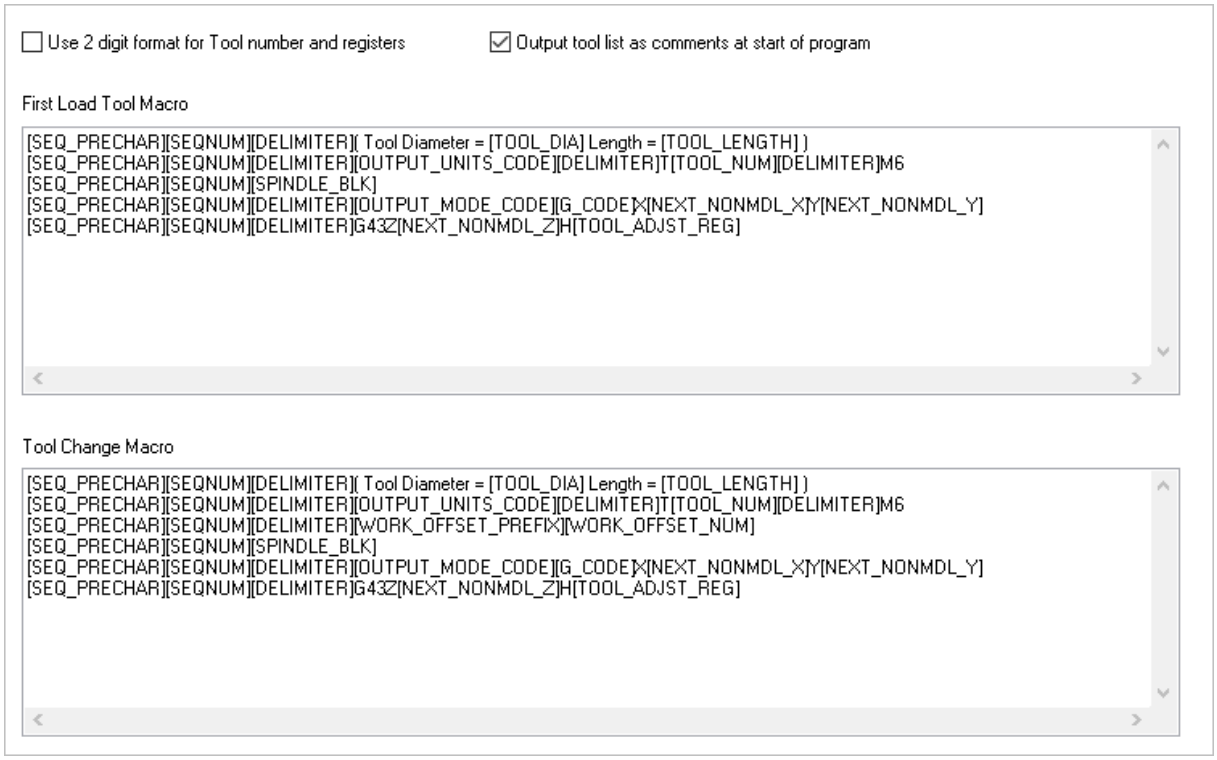
This section is ignored if your CNC machine does not have an automatic tool changer. The reason these are called Macros is because they use nested variables. For example [SPINDLE_BLK] is the result of the Spindle Block Format code defined in the Spindle section of the PPG.

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.



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

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]
```

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	

```
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
```

Tool Change Macro

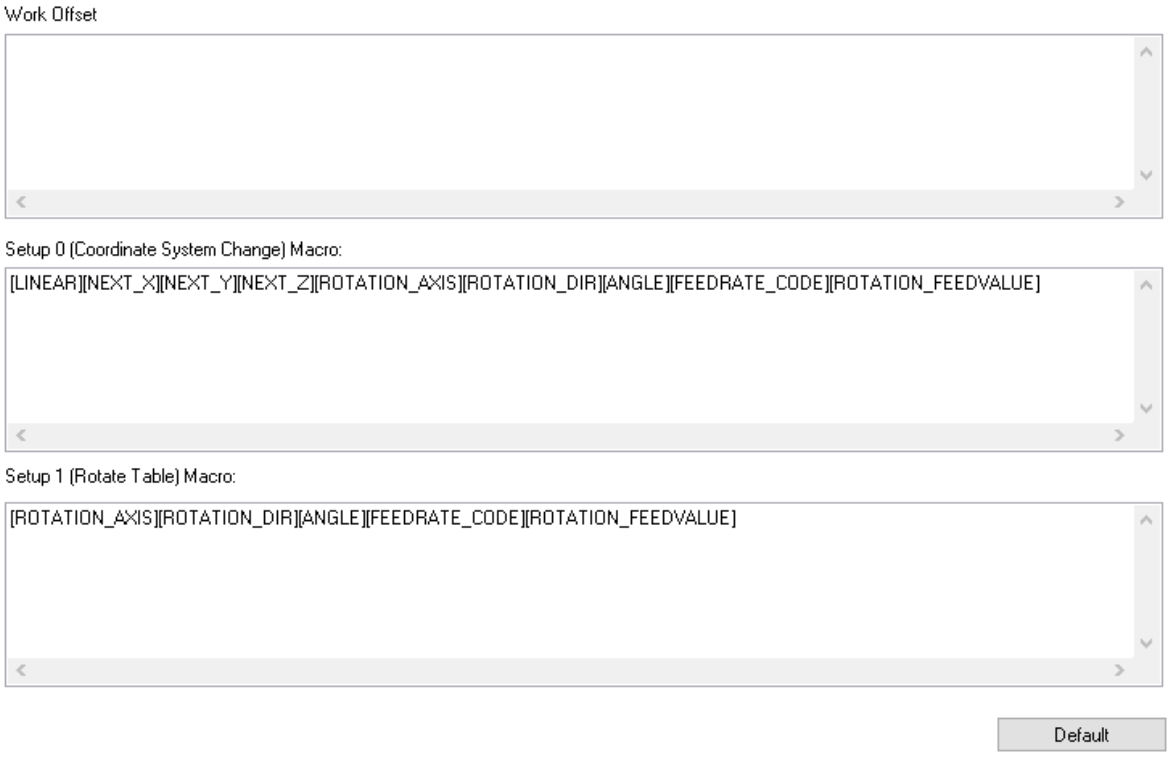
[SEQ_PRECHAR][SEQNUM][DELIMITER](Tool Diameter = [TOOL_DIA] Length = [TOOL_LENGTH])
[SEQ_PRECHAR][SEQNUM][DELIMITER][OUTPUT_UNITS_CODE][DELIMITER]T[TOOL_NUM][DELIMITER]M6
[SEQ_PRECHAR][SEQNUM][DELIMITER][WORK_OFFSET_PREFIX][WORK_OFFSET_NUM]
[SEQ_PRECHAR][SEQNUM][SPINDLE_BLK]
[SEQ_PRECHAR][SEQNUM][DELIMITER][OUTPUT_MODE_CODE][G_CODE]X[NEXT_NONMDL_X]Y[NEXT_NONMDL_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

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	

```
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
```

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.

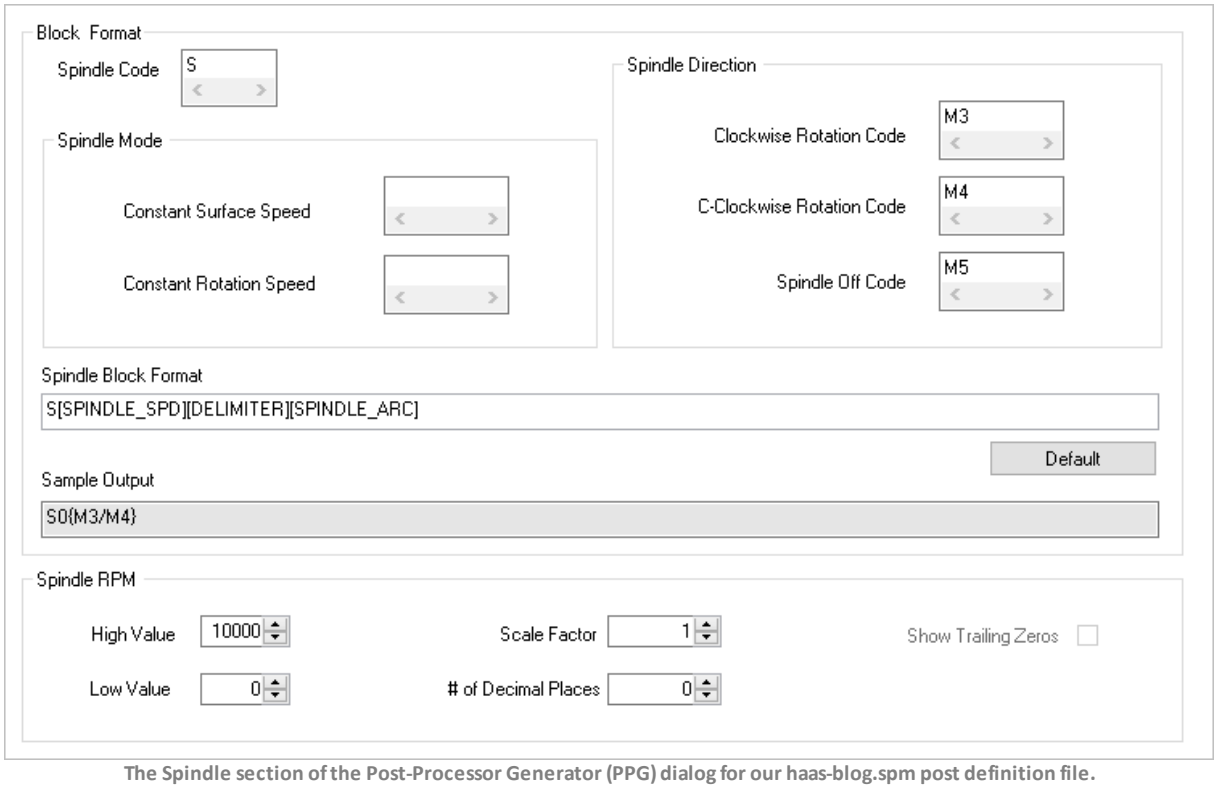


The Setup section of the Post-Processor Generator (PPG) dialog for our haas-blog.spm post definition file.

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.

This is the color code for the Spindle section.



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

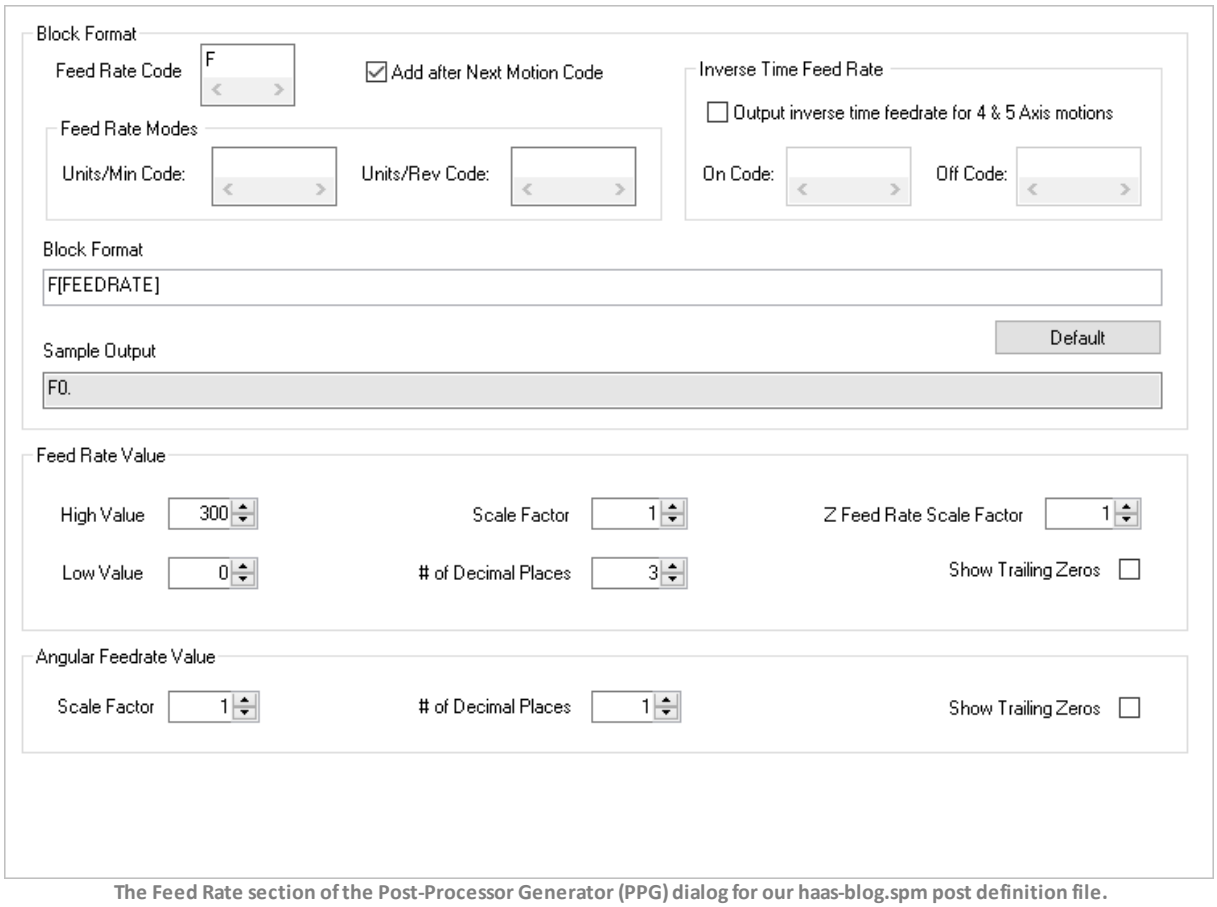
Spindle Block Format	
S[SPINDLE_SPD][DELIMITER][SPINDLE_ARC]	
Variable	**Derived From**
[SPINDLE_SPD]	MecSoft CAM > Tools > Create/Select Tool dialog > Feeds & Speeds or Mop > Feeds & Speeds > Spindle Speed

[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	
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	

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.



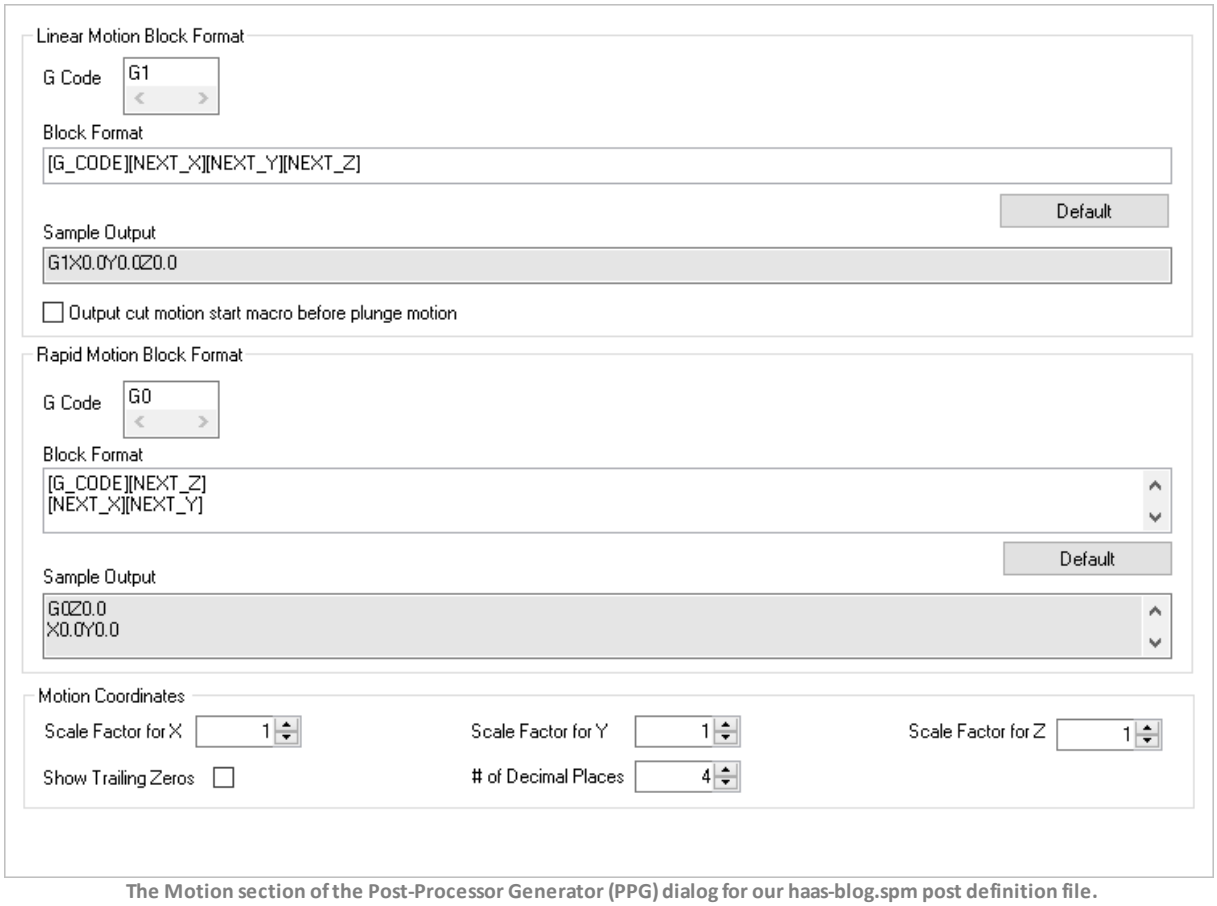
The Feed Rate section of the Post-Processor Generator (PPG) dialog for our haas-blog.spm post definition file.

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 G02Z-0.125I-0.4375J-0.0002 F195. N052 G02Z-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.

This is the color code for the Motion section.



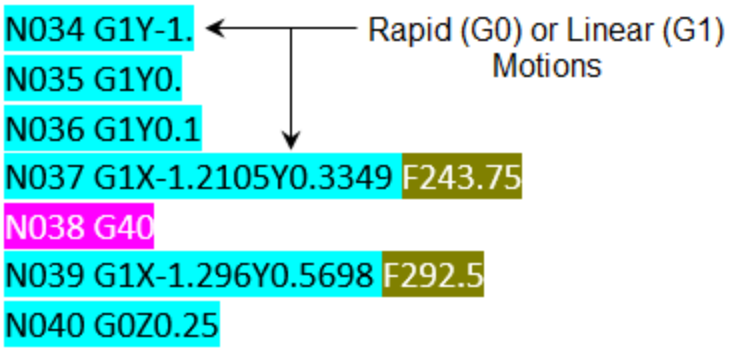
The Motion section of the Post-Processor Generator (PPG) dialog for our haas-blog.spm post definition file.

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



5.8 PPG > Circle

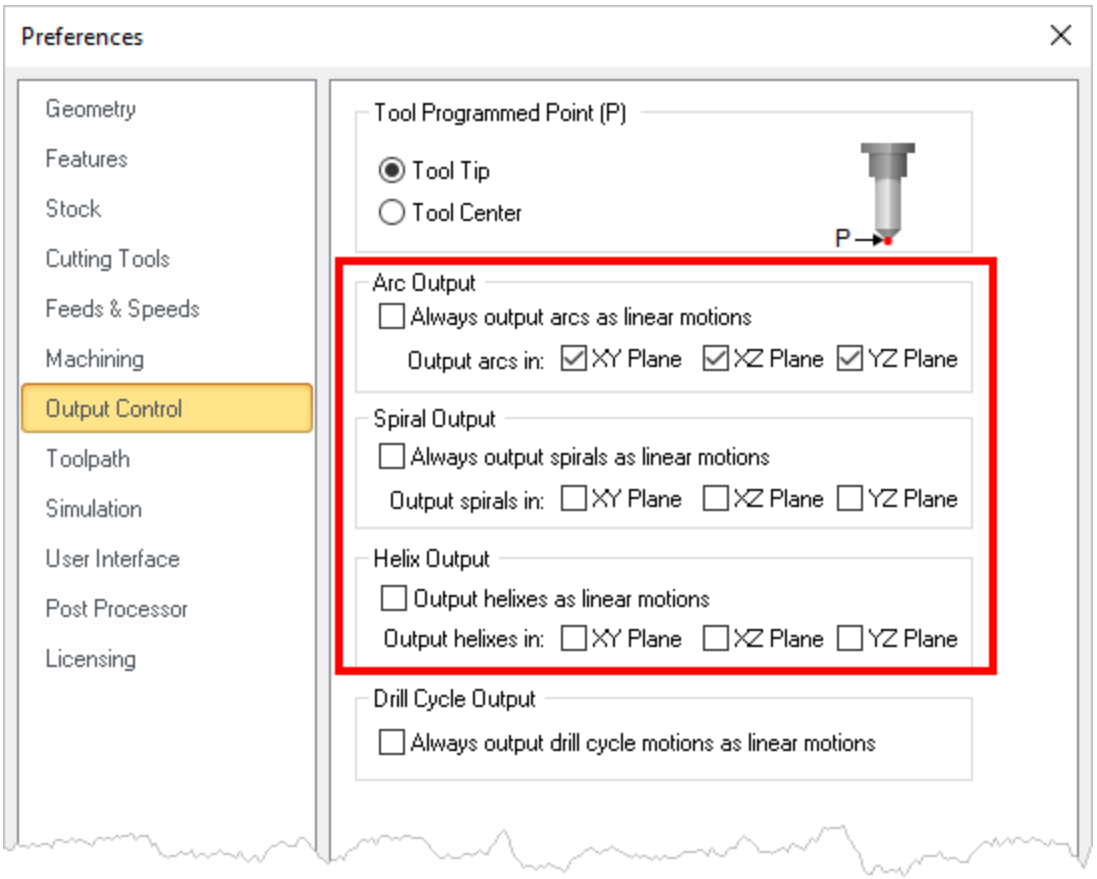
The Circle section of the PPG defines all arc motions. At the top you see the G-Code and Plane Code sections. This is followed by the Output Format and Arc Center (I, J, K) sections. The options in these two sections define how the arc code is formatted and specifically how the arc center is calculated.

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.

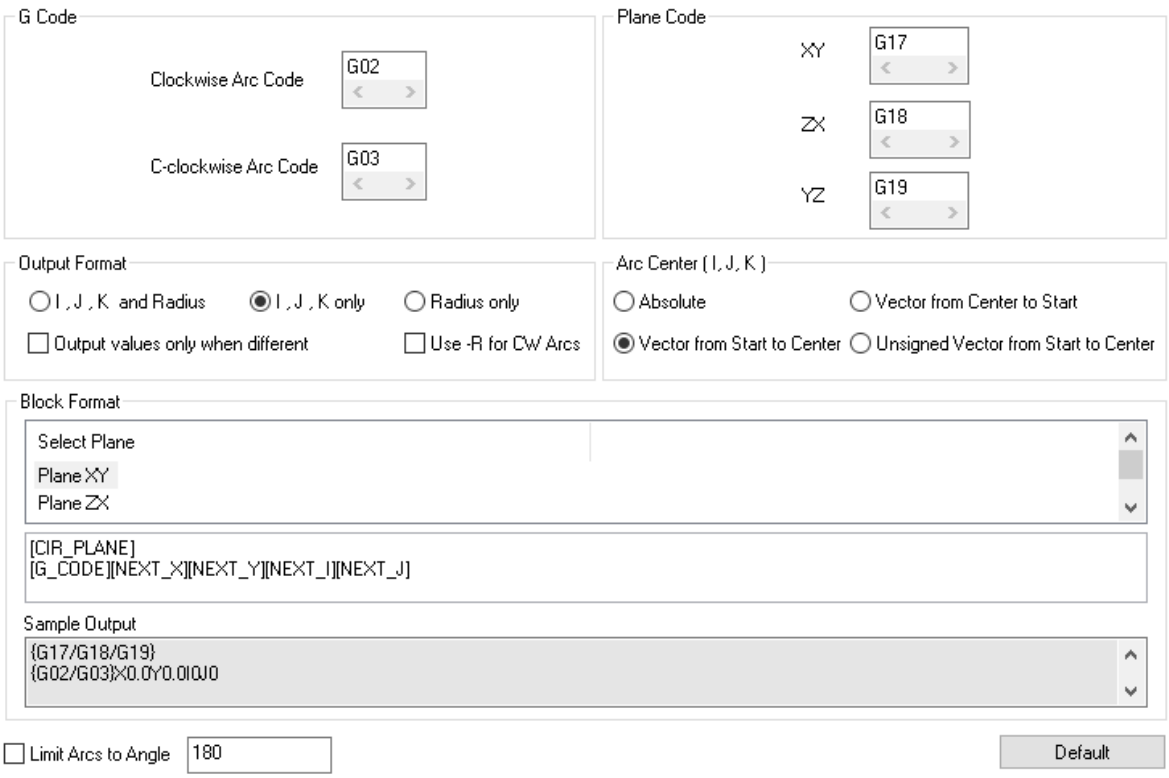


MILL > CAM Preferences > Machining dialog

- 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

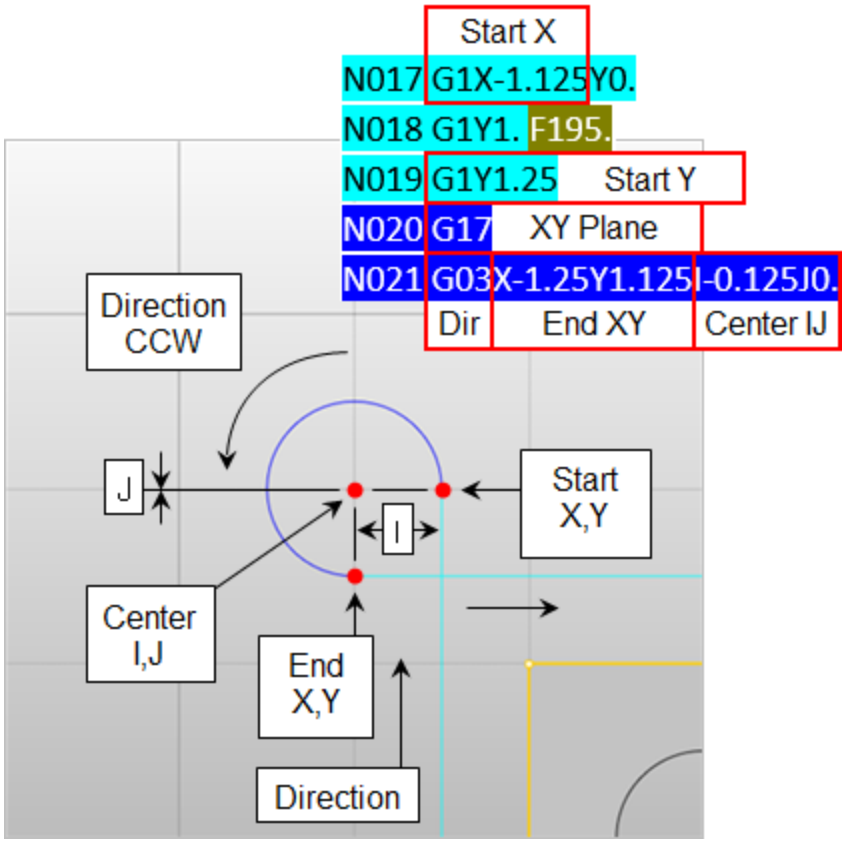
Perform Arc Fitting on the Advanced Cut Parameters tab for for 2½ Axis Profiling



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 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 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



How arc motions are defined for our sample.

Notes:

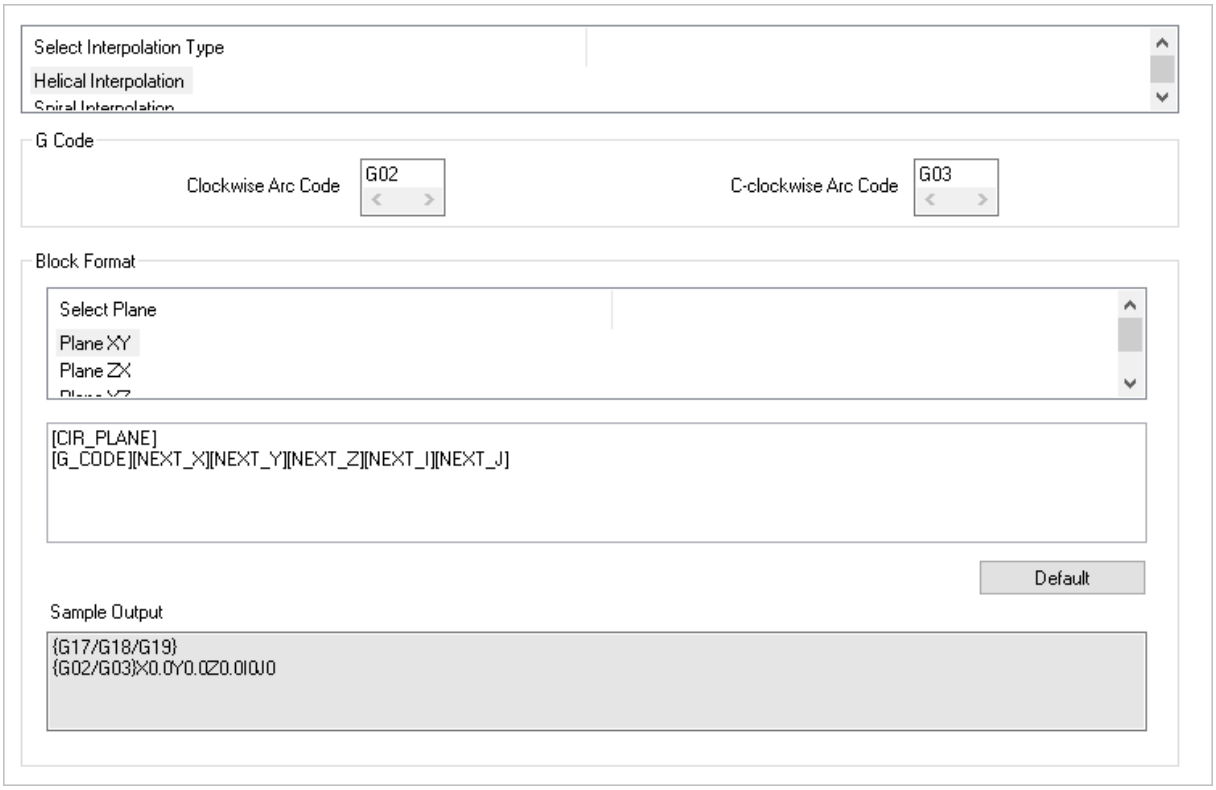
1. I, J and K represent the arc center as an offset distance from the arc start. This is because in the PPG, Arc Center (I, J, K) section we have select Vector from Center to Start. If you want the arc center posted in absolute coordinates, select the Absolute option.
2. Because the plane of the arc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

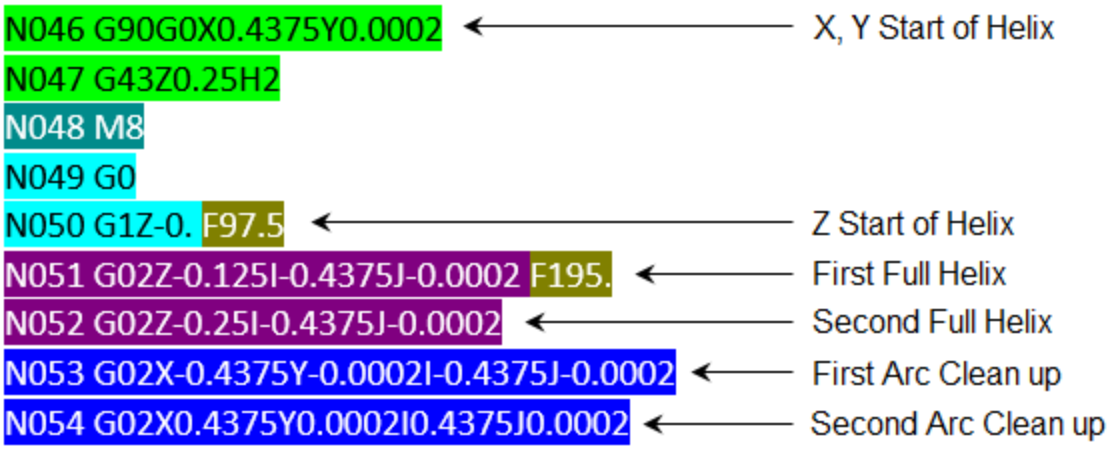


The Helical/Spiral section of the Post-Processor Generator (PPG) dialog for our haas-blog.spm post definition file.

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

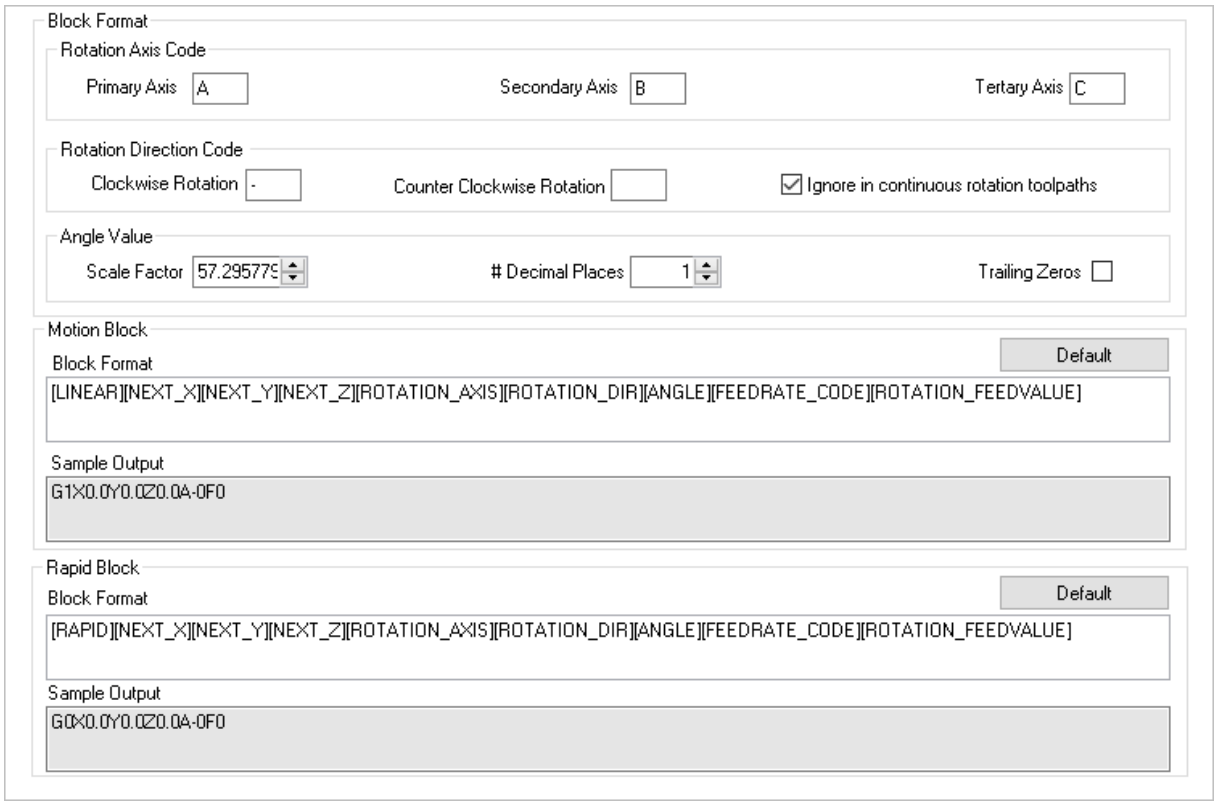


Notes:

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.
2. Lines N051 and N052 are the two helical motions. Notice that each includes a Z coordinate value.
3. Line N051 create the first full helix starting a Z0 and ending at Z0.125.
4. Line N052 create the second full helix starting a Z0.125 and ending at Z0.25.
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.

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.



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

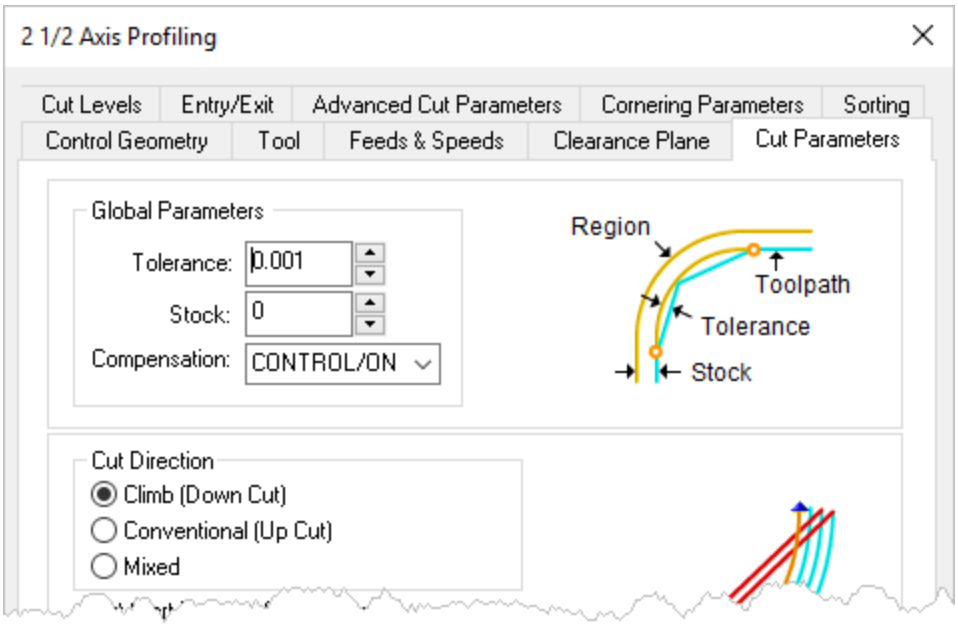
5.11 PPG > Cutter Compensation

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.

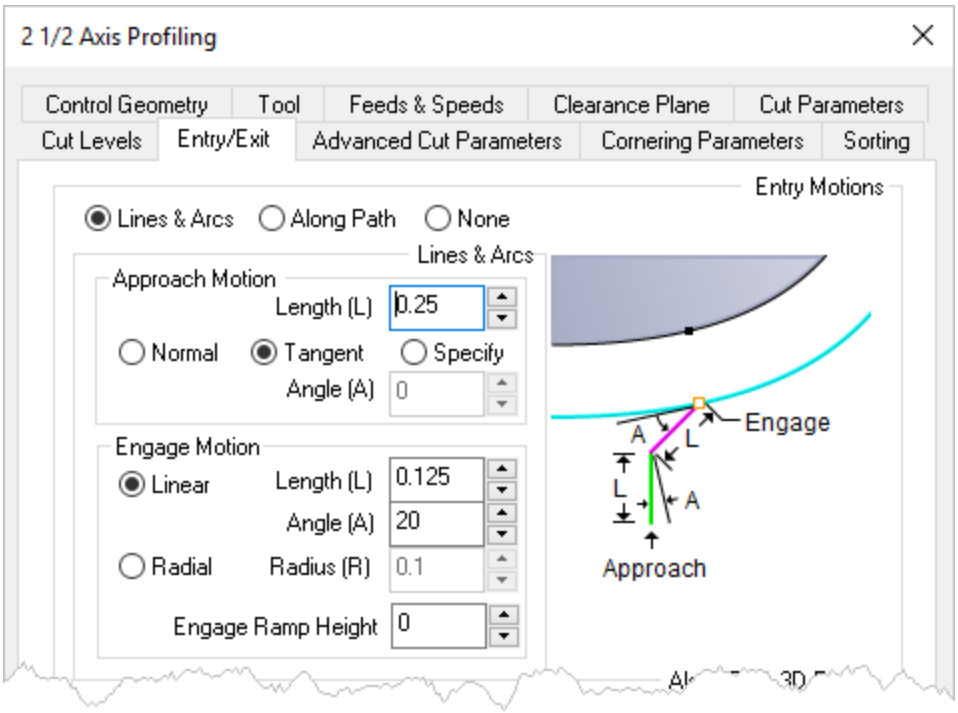
What is needed to output Cutter Compensation:

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.



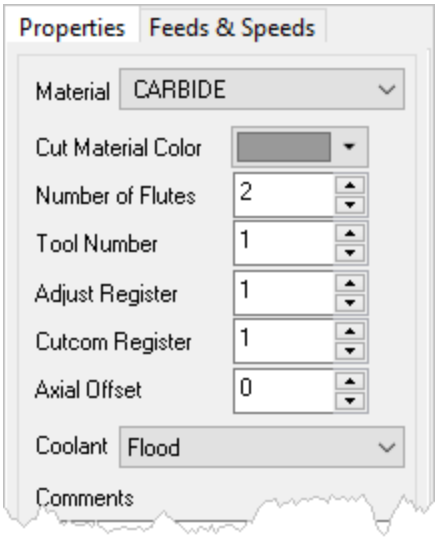
Compensation is set to CONTROL/ON and Cut Direction is set to Climb

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 2½ 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 tab of the Create/Select Tool dialog

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

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

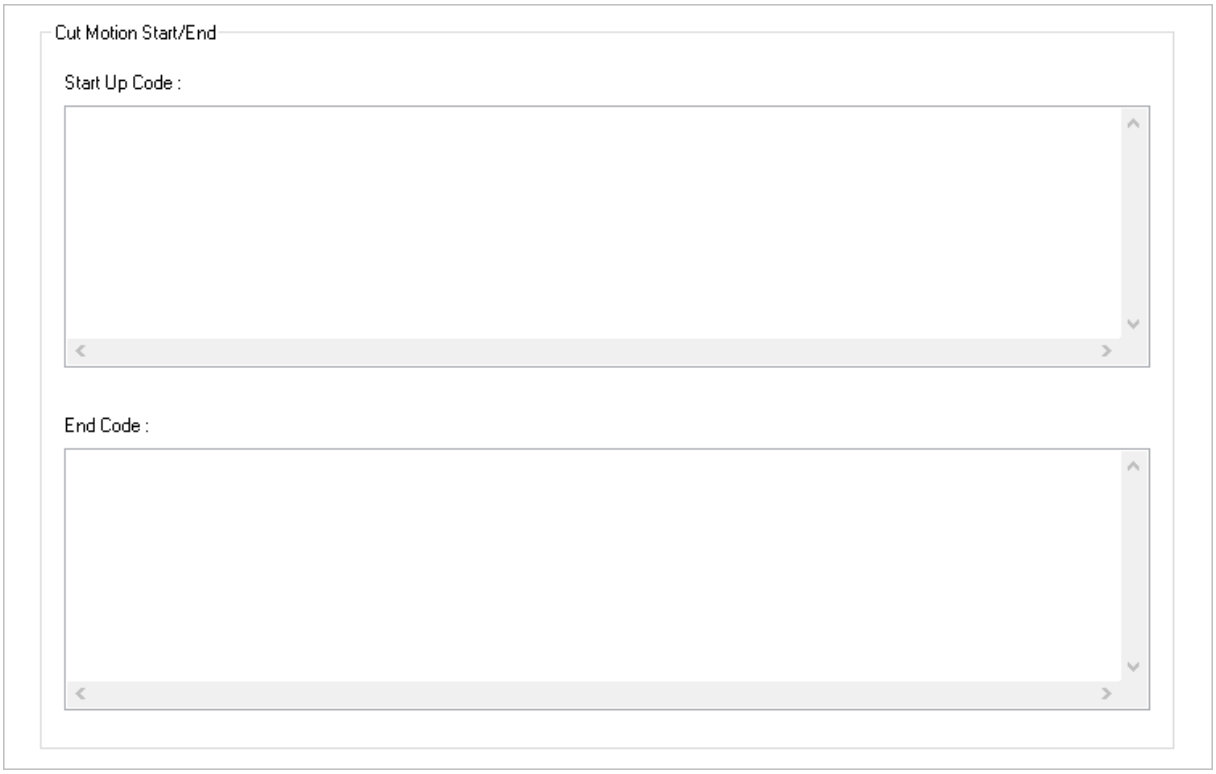
Resulting G-Code Sample

```
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.

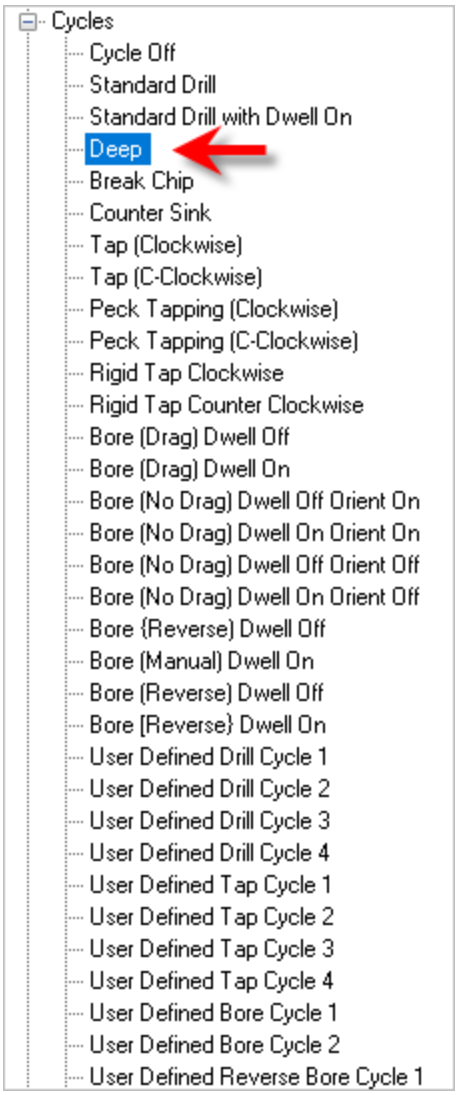
5.13 PPG > Cycles

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.

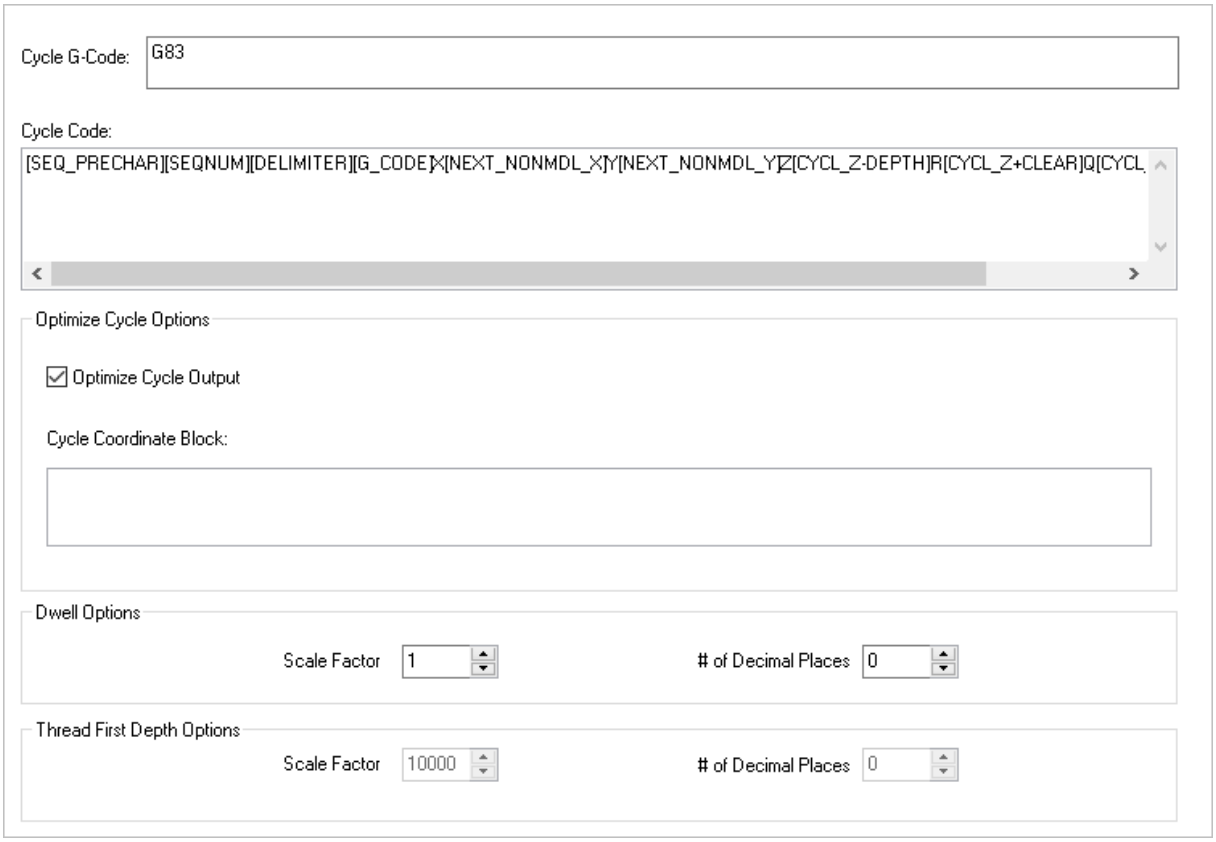
This is the color code for the Cycles section.

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).



Canned Cycles Supported



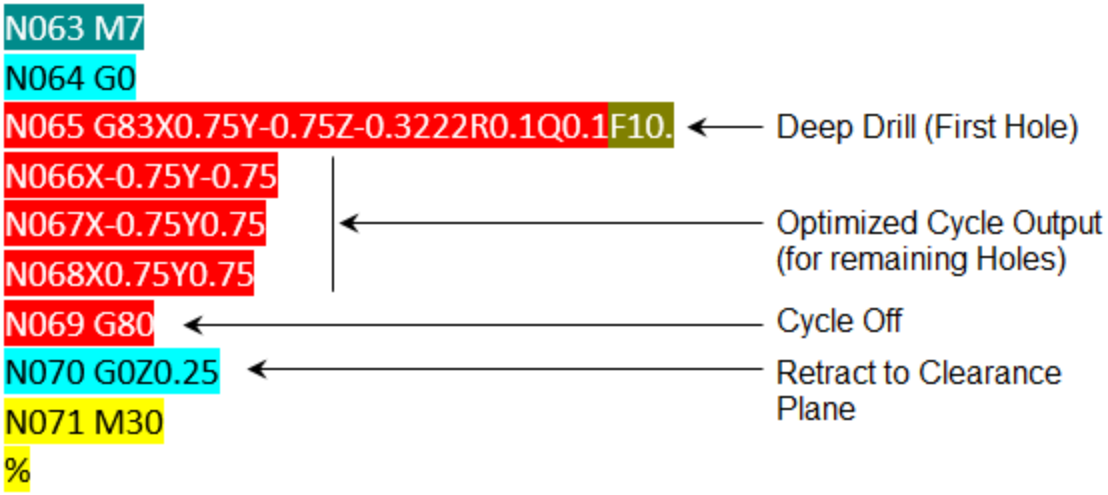
The Cycles > Deep Drill section of the Post-Processor Generator (PPG) dialog for our haas-blog.spm post definition file.

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]X[NEXT_NONMDL_X]Y[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 Deep Drill G83 cycle.
[CYCL_Z-DEPTH]	MecSoft CAM > Drill > Cut Parameters > Drill Depth.
[CYCLE_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

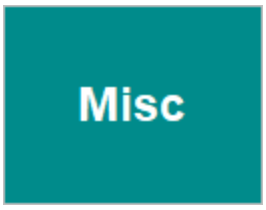


Notes:

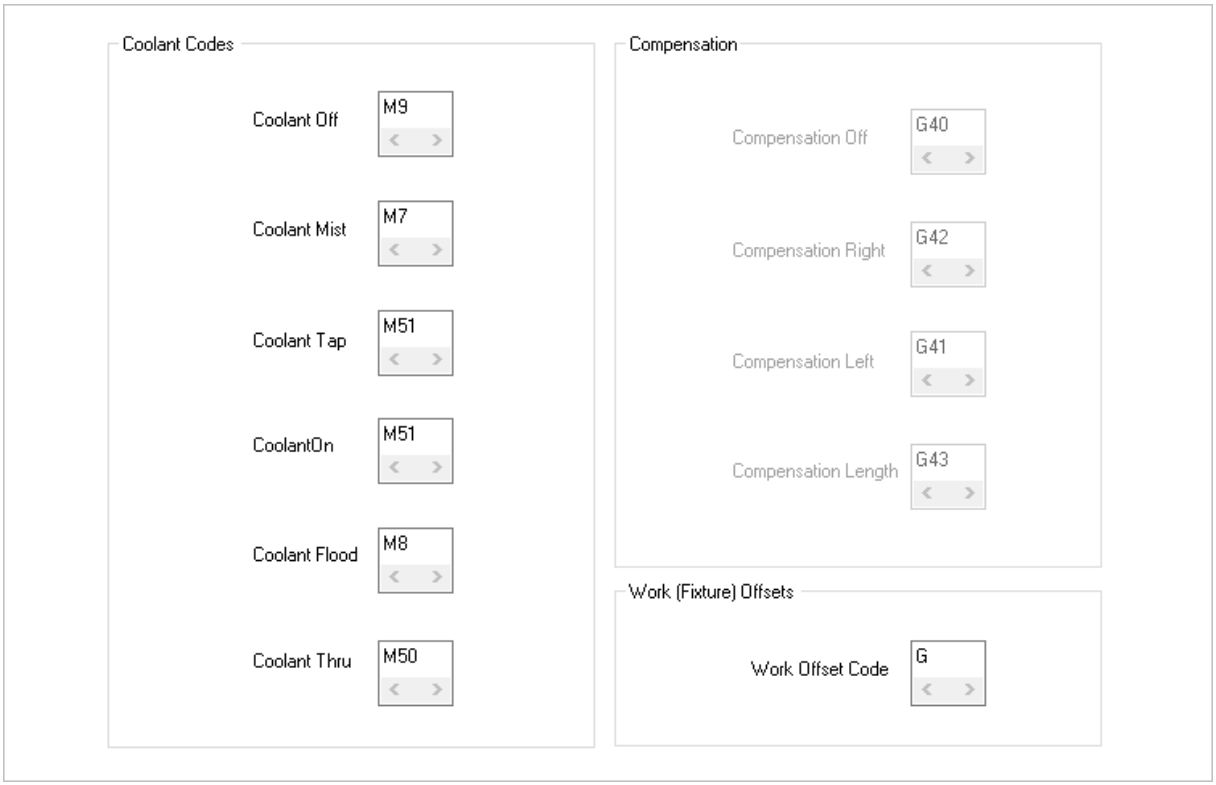
1. Line N065 is the first deep drill cycle (G83). On this line Z is the Drill Depth of the hole, R is the Z location of the Approach Distance value and Q is the Step Increment value.
2. Lines N066, N067 and N068 are the three remaining holes in the set. With Optimized Cycle Output checked, only the change in X and Y are posted.
3. 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.



This is the color code for the Misc section.



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

Resulting G-Code Sample

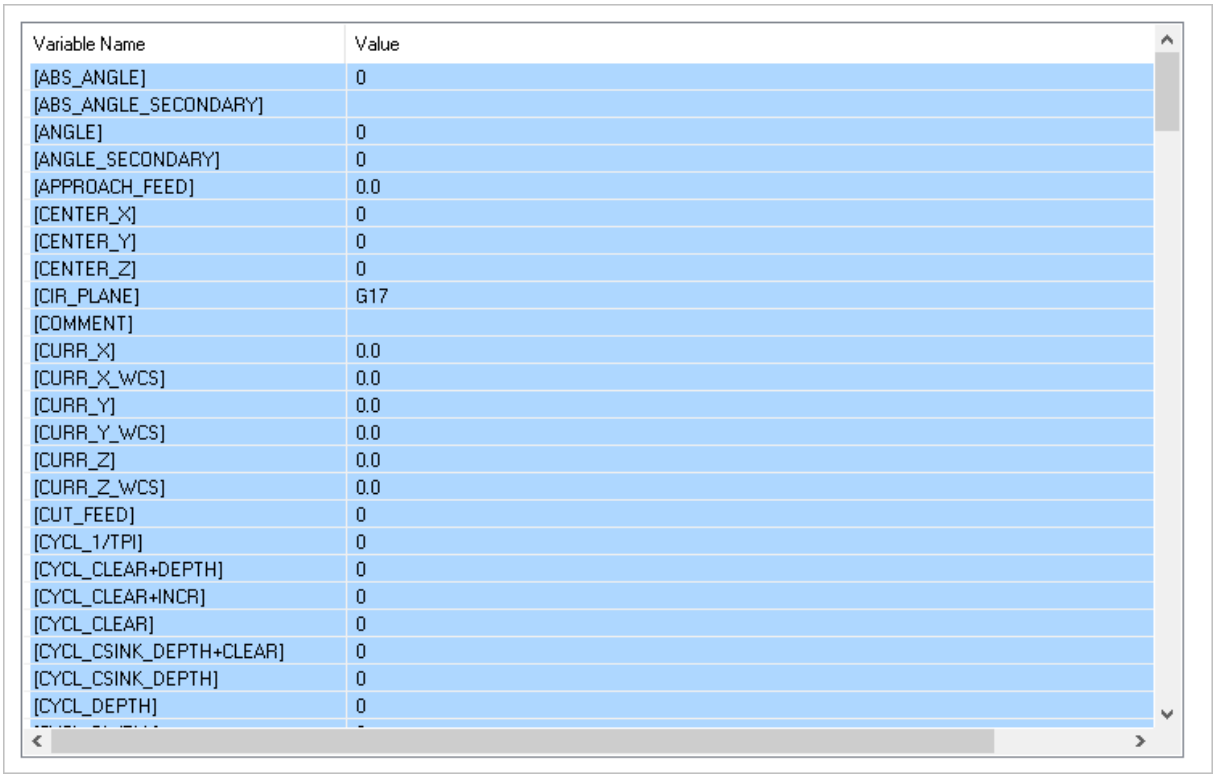
```
N006 (2 1/2 Axis Profiling (Outer))
N007 ( Tool Diameter = 0.25 Length = 2.0 )
N008 G20 T1 M6
N009 S10000 M3
N010 G90G0X-1.2533Y-0.3524
N011 G43Z0.25H1
N012 M8 <---------- Coolant = Flood for Tool #1
N013 G0
```

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 33
  Sample G-Code Control 34

- G -

General
  G-Code Control 24, 32, 33

- H -

Helical 43
Helical Motions
  Sample G-Code 40
  Understanding 40

- 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 40
  Misc 50
  Motion 36
  Multi Axis Motion 42
  Sample G-Code 24, 25, 28, 33, 34, 36, 37, 40, 43, 47, 50
  Setup 32
  Spindle 33
  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 50
Helical 40
Tool Change 28
Sample Toolpath 10
Set Post 10
Setup 10
Spindle
Example G-Code 33
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