SimplyCam

Customising Post Processor (PST) Files

Version: 1.0
Date: 24-Apr-2008
By: HarryE

Table of Contents

Introduction

SimplyCam is a fully integrated 2D CAD/CAM system that can directly open, create, edit and save drawing files in industry standard DXF format. It is a very flexible application and can also be customised in a number of ways making it a very flexible and low cost start-up for those needing to perform rapid proto-type development using CNC milling systems.

This text describes aspects of Post Processor files as used with SimplyCam to generate the actual G-code needed by CNC controls. It was written for use with SimplyCam v1.51 and is intended to assist users who would like to provide more customisations for various reasons. Specific focus is in and around the TurboCNC system because this is what I use.

Although many of the customisations described here add to the size of the generated G-Code it makes for a wonderful learning and post-editing environment in that it allows the user to identify specific areas of the code for further customisation and/or hand editing. Used in conjunction with the tool path simulation, edits become an absolute breeze to perform.

For example, I like to have line numbers with stepped increments so I can perform manual edits to already developed tool paths. This is easy to do with stepped increments in line numbers. Also, when using SimplyCam to generate a simple part outline, having customised code generation makes it relatively simple to convert the code into a sub-routine based tool path. This allows for the setup of variables and other code inclusions to provide more complex programs.

Location of PST Files

The default location of the Post Processor files is found in the following directory:

C:\Program Files\SimplyCam\pst\

The Post Processor files have a default filename extension of ".pst"

Note there are some 30+ files included with the standard distribution covering many of the more popular CNC controls. Of course more can be added to allow for additional controls or those that are not included.

Like any computer application it cannot be stressed how important file backups are to make for easy recovery. Having said that, before you make any changes, MAKE A BACKUP COPY of any files you intend to modify to make it easy to undo any accidental or erroneous additions. SimplyCam is quite happy to allow you selection of any post processor file added to this directory as long as they end with the ".pst" extension. Note that SimplyCam reads the files in this directory at the time you select the "Create NC Program" button so adding files can be done pretty much on-the-fly!

Sample PST File for TurboCNC

The example below shows the default PST file for TurboCNC 4 in millimetre format. Note the defaults chosen for the sections "[Default]" and "[Block Numbering]" because much of this text centres around these areas and these would be the most common area of interest. More details about this follows but for now notice the values for "File_extension" and "OutputSeq" in these sections respectively.

[Post Comment]
1=Post TurboCnc 4 mm
2=Modal XYZ and Feed
3=Arc defined with R (-R if>180 degree)
4=Tool change (T..M6 and M0)
5=Cycle G81, G83, G84
6=Comment (....)
7=Extension: *.CNC
8=Space between instruction
9=No block number
10=

[Default]
File_extension=NC
DelZero=1
Spaces=1
XYZModal=1
GModal=0
FModal=1
StartComment=(
EndComment=)

[Block Numbering]
OutputSeq=0
Pref=N
SeqStart=1
SeqInc=1
SeqMax=999999

[X Axis]
Pref=X
Format=1.3

[Y Axis]
Pref=Y
Format=1.3

[Z Axis]
Pref=Z
Format=1.3

[Feed]
Pref=F
Format=1.1
Rapid=

[Tool]
Pref=T
Format=1.0
Tofflen=

[Gcode]
Rapid=G00
Linear=G01
Circular_CW=G02
Circular_CCW=G03

[CComp]
None=G40
Left=G41
Right=G42

[Cycle]
Format=1.3
DepthPrefix=Z
Ref_heightPrefix=R
Peck_incrementPrefix=Q
PitchPrefix=F

[Cycle_1]
Name=Drill
Exploded=0
Cycle_def=[n]G81[x][y][depth][ref_height][feedplunge]
Cycle_move=[n][x][y]
Cycle_cancel=[n]G80

[Cycle_2]
Name=PeckDrill
Exploded=0
Cycle_def=[n]G83[x][y][depth][peck_increment][ref_height][feedplunge]
Cycle_move=[n][x][y]
Cycle_cancel=[n]G80

[Cycle_3]
Name=Tap
Exploded=0
Cycle_def=[n]G84[x][y][depth][pitch][ref_height]
Cycle_move=[n][x][y]
Cycle_cancel=[n]G80

[ArcDef]
;0 = IJ, 1 = R no sign, 2 = R signed neg. over 180
ArcOutput=2
;Arc center (if ArcOutput=0) 1=Abs, 2=Inc(Ct-Start), 3=Inc(Start-Ct), 4=Unsigned inc.
ArcCenter=1
;Break arcs, 0 = no, 1 = quadrants, 2 = 180deg. max arcs
BreakArc=0
IPref=I
JPref=J
RPref=R
Format=1.3

[Start_of_file]
1=[progname]
2=G90
3=G71
4=M06[tool] [tool_info]
5=M00(CHANGE TOOL)
6=M03
7=M08
8=G00[xrapid][yrapid]
9=G00[zrapid]

[Tool_change]
1=M05
2=M09
3=M06[tool] [tool_info]
4=M00(CHANGE TOOL)
5=M03
6=M08
7=G00[xrapid][yrapid]
8=G00[zrapid]
9=
10=

[End_of_file]
1=G00 X0. Y0.
2=M05
3=M09
4=M02
5=
6=
7=
8=
9=
10=

Structure of Post Processor (PST) Files

The structure of SimplyCam PST files is a modular design similar to the design and layout of many computer application and operating system initialisation files. The files are organised as Sections which provide major and minor function, control and other information for how SimplyCam behaves when it generates G-code.

Sections

Sections begin with predefined keywords contained within square brackets "[keyword]" and end with a blank line. Additional sections with their name follow after a blank line if applicable. Within each section is a list of keywords and possible values following the keyword after an equal "=" sign. Note that section names and keywords are case sensitive meaning that OutputSeq is not the same as OUTPUTSEQ which is not the same as outputseq. This is important to note because it is very easy to overlook this when editing PST files and wonder why your changes don't work..

Keywords

Keywords describe a particular action or are used to provide a substituted value during G-code generation. Keywords are either flags which denote some Boolean (True or False) function or represent some variable which is used to provide a substituted constant or variable value or string. Flags can be thought of as switches whereby their value represents an On or Off indicator for SimplyCam. These are represented as 0 (zero) is off and 1 (one) is on. When used as a variable, the value assigned to a keyword is substituted in the generated G-code. More on this later.

Keyword Flags

As mentioned, flags tell SimplyCam what to do or how to treat other aspects of code generation. For example, the flag "OutputSeq" is used to tell SimplyCam whether or not it should prefix code blocks with line numbers in the form specified by other keywords contained within the "[Block Numbering]" section. Having said that, if you now examine this section, most of the keywords defined in this section should make a lot of sense.

For example:

The keyword "Pref" is the prefix given before the actual line number. It is case-sensitive so you can choose either upper, lower or even mixed case depending on if your CNC controls allow this. My preference is lowercase because it's easier to read on screen.

The "SeqStart" keyword defines the first line number to use and obviously the "SeqInc" value tells SimplyCam what line number stepping to use. A value here of 10 is good because it allows you a fair amount of flexibility to add additional lines if you perform manual additions to your code.

Keyword Values or Parameters

Values assigned to keywords are substituted into the produced output file at G-code generation time. Comments may be specified in the PST file either surrounded by parenthesis or preceded with the semi-colon ";" character. This also holds for many popular CNC control software including TurboCNC and Mach3. Other CNC control software may only accept comments surrounded by parentheses as per the XYZ G-code standard.

Examples of PST Sections

[Default]
File_extension=NC
DelZero=1
Spaces=1
XYZModal=1
GModal=0
FModal=1
StartComment=(
EndComment=)

[Block Numbering]
OutputSeq=0
Pref=N
SeqStart=1
SeqInc=1
SeqMax=999999

Keyword Scope and Meanings

Keywords are specific to the section in which they are contained. For this reason, you may see the same keyword with a different value contained in more than one section. This concept is referred to as the "scope" of a keyword whereby its value is only applicable to the section in which it is contained.

Section Descriptions

The default section names contained within the TurboCNC PST file are described below:

Section Name	Description	Purpose	Function or Value
[Post Comment]	This section contains the text lines appearing at the beginning of the first screen of the G-code generation process. Basically, it is used to provide information about G-code characteristics for a given Post Processor.	This section does not use keyword names, rather it uses a sequence of numbers which denote the order in which the text after the equal (=) sign is displayed. There can be up to 10 lines in this section.	Informational only.

| [Default] | Describes the characteristics used by SimplyCam when G-code is generated. For example specifies whether G or M codes are modal in scope. Allows the user to define the default file extension for G-code files generated. | **Keyword Name Default Value Comments**

File_extension
NC
Provides the default extension name used to generate G-code files.

DelZero
1
Flag specifies if trailing zeros are omitted from co-ordinate values. Having these suppressed creates smaller output files although column alignment can be more difficult to read when performing manual edits to the files.

Spaces
1
Flag specifies whether spaces are produced between G-code words on each block. Omitting them reduces output file size but makes code harder to read.

XYZModal
1
Flag indicates whether the XYZ movement codes are modal and if next XY or Z coordinate should produce output if its current value is the same as its | | |

| [Block Numbering] | Specifies whether code blocks should contain line numbering and if so, how the line numbers are generated. | **Keyword Name Default Value Comments**

OutputSeq
Depends on PST used
Flag denoting if block numbers should be generated. 1=Yes

Pref
N
Prefix character pre-pended to block/line numbers. Preserves case chosen

SeqStart
1
Defines the first block/line number to use if numbers are generated. If you plan to edit the top of the file with comments or other setup information choose a higher starting value to allow you to use numbers lower than this one.

SeqInc
1
Defines the incremental value between consecutive block/line numbers. If you plan to edit code manually after generation a good value to use is 10

SeqMax
999999
Specifies the upper | | |

[X Axis]		**Keyword Name Default Value Comments** *Pref* X Prefix used for X axis values *Format* 1.3 Format of value produced in the output file. 1.3 denotes 1 integer and 3 decimal places		
[Y Axis]		Same description as for [X Axis] but applies to the Y axis		
[Z Axis]		Same description as for [X Axis] but applies to the Z axis		

[Feed]		**Keyword Name Default Value Comments** *Pref* F Prefix used for feed rate values *Format* 1.1 Format of output value. 1.1 denotes 1 integer and 1 decimal place. *Rapid* NULL Used by particular CNC controls eg: Heiden		

[Tool]		Keyword Name Default Value Comments		
		Pref T Prefix used for tool references *Format* 1.0 Format of output value. Since tools have no decimal part, the zero is used to suppress any decimal output. *Tofflen* NULL Specifies a tool offset length when using Tool Length compensation.		

[Gcode]		**Keyword Name** **Default Value** **Comments** *Rapid* G00 G-code used to denote a rapid travel command. *Linear* G01 G-code used to denote a linear interpolation movement. *Circular_CW* G02 G-code used to denote a clockwise movement *Circular_CCW* G03 G-code used to denote a counter-clockwise movement		

| [CComp] | | Keyword Name Default Value Comments

None
G40
G-code used to cancel cutter radius compensation

Left
G41
G-code used to enable cutter radius compensation to the left of the cut

Right
G42
G-code used to enable cutter compensation to the right of the cut | | |
|---|---|---|---|---|
| | | | | |

[Cycle]		**Keyword Name Default Value Comments** *Format* 1.3 Format of value produced in the output file. 1.3 denotes 1 integer and 3 decimal places *DepthPrefix* Z Character used to denote Z axis references *Ref_heightPrefix* R Character used to denote reference height when using canned cycled codes *Peck_incrementPrefix x* Q Character used to denote peck cycle increment values *PitchPrefix* F Character used to denote pitch when doing thread cutting		

| [Cycle_1] | | Keyword Name Default Value Comments

Name
Drill
Name that appears in SimplyCam dialog

Exploded
0
0=Canned cycle, 1=Linear move

Cycle_def
[n] G81 [x] [y] [depth] [ref_height]
First cycle definition

Cycle_move
[n] [x] [y]
Subsequent points for this cycle

Cycle_cancel
[n] G80
G-code to cancel this cycle | | |
| | | | | |

[Cycle_2]		Keyword Name Default Value Comments		
		Name PeckDrill As for Cycle_1		
		Exploded 0		
		Cycle_def [n] G83 [x] [y] [depth] [peck_increment] [ref_height]		
		Cycle_move [n] [x] [y]		
		Cycle_cancel [n] G80		

[Cycle_3]		Keyword Name Default Value Comments		
		Name Tap As for Cycle_1		
		Exploded 0		
		Cycle_def [n] G84 [x] [y] [depth] [pitch] [ref_height]		
		Cycle_move [n] [x] [y]		
		Cycle_cancel [n] G80		

[ArcDef]		Keyword Name Default Value Comments		
		;0 = IJ, 1 = R no sign, 2 = R signed neg. over 180		
		ArcOutput 2 0=IJ, 1=R no sign, 2=R signed negative over 180 degrees		
		;Arc center (if ArcOutput=0) 1=Abs, 2=Inc(Ct-Start), 3=Inc(Start-Ct), 4=Unsigned inc.		
		ArcCenter 1 If ArcOutput=0 1=Abs, 2=Inc(Ct-Start), 3=Inc(Start-Ct, 4=Unsigned inc.		
		;Break arcs, 0 = no, 1 = quadrants, 2 = 180deg. max arcs		
		BreakArc 0 0=No, 1=Quadrants, 2=180 degree max arcs		
		IPref I Prefix for X center		

| [Start_of_file] | This section does not use keyword names, rather it uses a sequence of numbers which denote the order in which the text after the equal (=) sign is displayed.

The "Default Value" items specified here appear at the top of the generated G-code file.

You can place Initialisation code and comments here that you want to appear at the top of the file. | **Keyword Name**
Default Value
Comments

1
[progname]
The program name

2
G90
Fixed string

3
G70
Fixed string

4
M06[tool]
[tool_info]
Fixed string + variables

5
M00(CHANGE TOOL)
Fixed string

6
M03
Fixed string

7
M08
Fixed string

8
G00[xrapid]
[yrapid]

9
G00[zrapid]
Fixed string + variables | | |
| [Start_of_file] | | **Keyword Name**
Default Value
Comments | | |

[Tool_change]	This section does not use keyword names, rather it uses a sequence of numbers which denote the order in which the text after the equal (=) sign is displayed. The "Default Value" items here specify the sequence of code used to perform a tool change.	**Keyword name** **Default Value** **Comments** *1* M05 Fixed string *2* M09 *3* M06[tool] [tool_info] Fixed string + variables *4* M00(CHANGE TOOL) *5* M03 *6* M08 *7* G00[xrapid] [yrapid] *8* G00[zrapid] Fixed string + variables		

[End_of_file]	This section does not use keyword names, rather it uses a sequence of numbers which denote the order in which the text after the equal (=) sign is displayed. The "Default Value" items here appear at the end of the file and are used to clean-up any machine settings, turn off the spindle and/or coolant.	**Keyword Name** **Default Value** **Comments** *1* G00 X0. Y0. Fixed string *2* M05 *3* M09 *4* M02 *5* See Note		
		In this example, we rapid the X and Y positions back to zero, stop the spindle and turn off the coolant flow. Finally we specify the end of the program.		

		Up to a maximum of 20 lines is allowed here. Note: We could put an M30 here (End of Program, reset to Start) if we wanted to re-set our machine control. As far as I can tell SimplyCam ignores this and does not undo the trace provided in the tool path simulator.	

[MCode]		**Keyword Name** **Default Value** **Comments**		
		SpindleCW M3		
		SpindleCCW M4		
		SpindleOff M5		
		CoolantOn M8		
		CoolantOff M9		
		SpindleCWCoolOn M12 Spindle On Clockwise and Coolant On		
		SpindleCCWCoolOn M14 Spindle On Counter-Clockwise and Coolant On		
[MCode]				

Sample Customised Version of TurboCNC v4.x file with Comments

Here is a customised PST file for use with TurboCNC4 in metric. Only the customised sections are shown. The section following show the G-code file generated using the customisations as shown here. Notice the block/line numbers and comments scattered throughout the file. My preference is to use the file extension CNC rather than the default NC as used by TurboCNC.

Also note I chose to add additional comments to some of the canned cycle sequences to make it easier to determine what the code is doing.

[Post Comment]
1=Post TurboCnc4 (Dimensions in mm)
2=Modal XYZ and Feed
3=Arc defined with R (-R if>180 degree)
4=Tool change (T..M6 and M0)
5=Cycle G81, G83, G84
6=Comment (....)
7=Extension: *.CNC
8=Space between instruction
9=Block number
10=By: HarryE

[Default]
File_extension=CNC
DelZero=0
Spaces=1
XYZModal=1
GModal=0
FModal=1
StartComment=(
EndComment=)

[Block Numbering]
OutputSeq=1
Pref=n
SeqStart=1000
SeqInc=10
SeqMax=999999

[Cycle_1]
Name=Drill
Exploded=0
Cycle_def=[n] G81 [x] [y] [depth] [ref_height] (Drilling Cycle)
Cycle_move=[n] [x] [y]
Cycle_cancel=[n] G80 (Canned Cycle CANCEL)

[Cycle_2]
Name=PeckDrill
Exploded=0
Cycle_def=[n] G83 [x] [y] [depth] [peck_increment] [ref_height] (Deep Hole Drilling Cycle)
Cycle_move=[n] [x] [y]
Cycle_cancel=[n] G80 (Canned Cycle CANCEL)

[Cycle_3]
Name=Tap
Exploded=0
Cycle_def=[n] G84 [x] [y] [depth] [pitch] [ref_height] (Tapping Cycle)
Cycle_move=[n] [x] [y]
Cycle_cancel=[n] G80 (Canned Cycle CANCEL)

[Start_of_file]
1=n0010 [progname]
2=n0020 (By: HarryE)
3=()
4=n0040 g17 (Select Plane XY=17 XZ=18 YZ=19)
5=n0050 g23 (Stored Stroke Limit OFF 22=ON)
6=n0060 g40 (Cutter Compensation CANCEL 40=Off 41=Left 42=Right)
7=n0070 g49 (Tool Length Compensation CANCEL
8=n0080 g54 (Workpiece Co-Ordinate System Default=0)
9=()
10=n0100 G90 (Positioning System 90=Absolute 91=Relative)
11=n0110 G71 (Measurement Unit 21 and 71=Metric ie:mm 20 and 70=Inch)
12=n0120 M06 [tool] [tool_info]
13=n0130 M00 (Program Stop to Change Cutting Tool)
14=n0140 M03 (Spindle ON 03=CW 04=CCW 05=OFF)
15=n0150 M08 (Coolant 08=On 09=Off)
16=n0160 G01 [xrapid][yrapid] (Move to initial XY using interpolated move)
17=n0170 G00 [zrapid] (Rapid Z to Feed Plane)
18=()
19=(Start Program)
20=n0200 ()

[Tool_change]
1=M05
2=M09
3=M06 [tool] [tool_info]
4=M00 (Change Cutting Tool)
5=M03
6=M08
7=G00 [xrapid][yrapid]
8=G00 [zrapid]

[End_of_file]
1=()
2=(End Program)

3=[n] g04 P001 (Dwell for 001 time periods)
4=[n] G00 Z0.000 X0.000 Y0.000 (Move Axes to Start Positions)
5=[n] M05 (Spindle 05=OFF 04=CCW 03=CW)
6=[n] M09 (Coolant 09=OFF 08=ON)
7=[n] M02 (Program End)

Sample Code for a Part with a Simple Tool Path

Here is a simple tool path generated using the above customisations. It is a simple bearing block with the following dimensions in millimetres:

To see what it looks like, past the following code into SimplyCam's Edit or Simulate window and run the simulation. You can then verify the X, Y and Z dimensions by clicking on the "Nc Limits" button.

X=100, Y=50, Z=-1

```
n0010 (BEARINGBLOCK_001.CNC)
n0020 (By: Harry Eleftheriou)
()
n0040 g17 (Select Plane XY=17 XZ=18 YZ=19)
n0050 g23 (Stored Stroke Limit OFF 22=ON)
n0060 g40 (Cutter Compensation CANCEL 40=Off 41=Left 42=Right)
n0070 g49 (Tool Length Compensation CANCEL
n0080 g54 (Workpiece Co-Ordinate System Default=0)
()
n0100 G90 (Positioning System 90=Absolute 91=Relative)
n0110 G71 (Measurement Unit 21 and 71=Metric ie:mm 20 and 70=Inch)
n0120 M06  T1 (TLDIA=3)
n0130 M00 (Program Stop to Change Cutting Tool)
n0140 M03 (Spindle ON 03=CW 04=CCW 05=OFF)
n0150 M08 (Coolant 08=On 09=Off)
n0160 G01  X62.500 Y30.000 (Move to initial XY using interpolated move)
n0170 G00  Z2.000 (Rapid Z to Feed Plane)
()
(Start Program)
n0200 ()
n1000 G01 Z-1.000 F100
n1010 G03 X37.500 R12.500 F200
n1020 G03 X62.500 R12.500
n1030 G00 Z2.000
n1040 G00 X25.000 Y10.000
n1050 G01 Z-1.000 F100
n1060 G03 X15.000 R5.000 F200
n1070 G03 X25.000 R5.000
n1080 G00 Z2.000
n1090 G00 X85.000
n1100 G01 Z-1.000 F100
n1110 G03 X75.000 R5.000 F200
n1120 G03 X85.000 R5.000
n1130 G00 Z2.000
n1140 G00 X64.478 Y17.272
n1150 G01 Z-1.000 F100
n1160 G03 X60.978 R1.750 F200
n1170 G03 X64.478 R1.750
n1180 G00 Z2.000
n1190 G00 Y42.728
n1200 G01 Z-1.000 F100
n1210 G03 X60.978 R1.750 F200
n1220 G03 X64.478 R1.750
n1230 G00 Z2.000
n1240 G00 X39.022
n1250 G01 Z-1.000 F100
n1260 G03 X35.522 R1.750 F200
n1270 G03 X39.022 R1.750
n1280 G00 Z2.000
n1290 G00 Y17.272
n1300 G01 Z-1.000 F100
n1310 G03 X35.522 R1.750 F200
n1320 G03 X39.022 R1.750
n1330 G00 Z2.000
```

```
n1340 G00 X40.000 Y50.000
n1350 G01 Z-1.000 F100
n1360 G01 X0.000 Y30.000 F200
n1370 G01 Y0.000
n1380 G01 X100.000
n1390 G01 Y30.000
n1400 G01 X60.000 Y50.000
n1410 G01 X40.000
n1420 G00 Z2.000
()
(End Program)
n1430 g04 P001 (Dwell for 001 time periods)
n1440 G00 Z0.000 X0.000 Y0.000 (Move Axes to Start Positions)
n1450 M05 (Spindle 05=OFF 04=CCW 03=CW)
n1460 M09 (Coolant 09=OFF 08=ON)
n1470 M02 (Program End)
```

Additional Sections, Keywords and Variables

This section lists additional sections, keywords and inbuilt variables for completeness. Many are not defined in the standard PST files so they are included here for completeness.

Additional Sections

Section Name	Description	Purpose

[BeforeFirstFeedMove]	Optional code added before each first feed move	**Keyword Name Default Value Comments**		
		1 [n] M4 Example: Laser Power On		
		2 [n] M18 Laser Beam On		
		3		
		...		
		20 Up to 20 lines		
[AfterFirstFeedMove]	Same as above but after first feed move	Same format as above.		
[BeforeLastFeedMove]	Optional code added before last feed move	Same format as above.		

[AfterLastFeedMove]	As above but after last feed move	**Keyword Name Default Value Comments** *1* [n] M7 Laser Beam Off *2* [n] M19 Laser Power Off *3* ... *20* Up to 20 lines		
[BeforeFirstZMove]	The following don't yet exist	I suggested these as an enhancement to allow easy identification of code blocks in cycle sequences.		
[AfterFirstZMove]				
[BeforeLastZMove]				
[AfterLastZMove]				

List of Variables

Here is a list of the pre-defined variables. These are global definitions and may be used in any section AFAIK.

All variable names are all lowercase and are enclosed between square brackets ([]).

Variable Name,Description

Variable Name	Description
[progname]	Filename of CNC file open in SimplyCam
;	
[n]	Output the block/line number
;	
[x]	Output X value
[y]	Output Y value
[z]	Output Z value
[a]	Output 4th axis value (N/A)
;	
[xrapid]	Output first X rapid move
[yrapid]	Output first Y rapid move
[zrapid]	Output first Z rapid move
;	
[f_xrapid]	As above but forced move
[f_yrapid]	
[f_zrapid]	
;	
[tool]	Output number of tool
[tool_info]	Output tool info (TLDIA for Simulation)
[offlen]	
;	
[speed]	Spindle speed
[spindle_on]	Code for Spindle On
[spindle_off]	Code for Spindle Off
[spindle_and_coolant_on]	Code for Spindle and Coolant On
;	
[feed]	Feedrate in XY move
[feedplunge]	Feedrate of Z move
;	
[coolant_on]	Code for Coolant On
[coolant_off]	Code for Coolant Off
;	
[depth]	Output depth of cycle
[ref_height]	Output rapid plane of cycle
[peck_increment]	Output increment in G83 cycle
[pitch]	Output pitch in thread cycle

Samples

Peck Drill Cycle

```
n0010 (T.CNC)
N0015 (This line added manually to show why the line increment value was used)
n0020 (By: SimplyCam and HarryE)
()
n0040 g17 (Select Plane XY=17 XZ=18 YZ=19)
n0050 g23 (Stored Stroke Limit OFF 22=ON)
n0060 g40 (Cutter Compensation CANCEL 40=Off 41=Left 42=Right)
n0070 g49 (Tool Length Compensation CANCEL
n0080 g54 (Workpiece Co-Ordinate System Default=0)
()
n0100 G90 (Positioning System 90=Absolute 91=Relative)
n0110 G71 (Measurement Unit 21 and 71=Metric ie:mm 20 and 70=Inch)
()
n0130 M06  T5 (TLDIA=1)
n0140 M00 (Program Stop to Change Cutting Tool)
n0150 M03 (Spindle ON 03=CW 04=CCW 05=OFF)
n0160 M08 (Coolant 08=On 09=Off)
()
(Position XY to start co-ordinates and Z to Feed Plane)
n0900 G01  X110.000 Y60.000 (Move to initial XY using interpolated move)
n0910 G00  Z2.000 (Rapid Z to Feed Plane)
n1000 G83  X110.000  Y60.000  Z-10.000  Q5.000  R2.000 (Deep Hole Drilling Cycle)
n1010  X95.355  Y95.355
n1020  X60.000  Y110.000
n1030  X24.645  Y95.355
n1040  X10.000  Y60.000
n1050  X24.645  Y24.645
n1060  X60.000  Y10.000
n1070  X95.355  Y24.645
n1080  X60.000  Y60.000
n1090 G80 (Canned Cycle CANCEL)
()
(End Program)
n1100 g04 P001 (Dwell for 001 time periods)
n1110 G00 Z0.000 X0.000 Y0.000 (Move Axes to Start Positions)
n1120 M05 (Spindle 05=OFF 04=CCW 03=CW)
n1130 M09 (Coolant 09=OFF 08=ON)
n1140 M02 (Program End)
```

Same Drawing using a Bolt Hole with Contour Setting

This illustrates more use of the different customisations in the PST file.

```
n0010 (T.CNC)
n0020 (By: SimplyCan and HarryE)
()
n0040 g17 (Select Plane XY=17 XZ=18 YZ=19)
n0050 g23 (Stored Stroke Limit OFF 22=ON)
n0060 g40 (Cutter Compensation CANCEL 40=Off 41=Left 42=Right)
n0070 g49 (Tool Length Compensation CANCEL
n0080 g54 (Workpiece Co-Ordinate System Default=0)
()
n0100 G90 (Positioning System 90=Absolute 91=Relative)
n0110 G71 (Measurement Unit 21 and 71=Metric ie:mm 20 and 70=Inch)
()
n0130 M06  T5 (TLDIA=1)
n0140 M00 (Program Stop to Change Cutting Tool)
n0150 M03 (Spindle ON 03=CW 04=CCW 05=OFF)
n0160 M08 (Coolant 08=On 09=Off)
()
(Position XY to start co-ordinates and Z to Feed Plane)
n0900 G01  X27.145 Y24.645 (Move to initial XY using interpolated move)
n0910 G00  Z2.000 (Rapid Z to Feed Plane)
n1000 G01 Z-1.000 F75
()
n1010 G03 X22.145 R2.500 F125
n1020 G03 X27.145 R2.500
(AfterLastFeedMove)
n1030 G00 Z2.000
n1040 G00 X62.500 Y10.000
n1050 G01 Z-1.000 F75
()
n1060 G03 X57.500 R2.500 F125
n1070 G03 X62.500 R2.500
(AfterLastFeedMove)
n1080 G00 Z2.000
n1090 G00 X97.855 Y24.645
n1100 G01 Z-1.000 F75
()
n1110 G03 X92.855 R2.500 F125
n1120 G03 X97.855 R2.500
(AfterLastFeedMove)
n1130 G00 Z2.000
n1140 G00 X112.500 Y60.000
n1150 G01 Z-1.000 F75
()
n1160 G03 X107.500 R2.500 F125
n1170 G03 X112.500 R2.500
(AfterLastFeedMove)
n1180 G00 Z2.000
n1190 G00 X97.855 Y95.355
n1200 G01 Z-1.000 F75
()
n1210 G03 X92.855 R2.500 F125
n1220 G03 X97.855 R2.500
(AfterLastFeedMove)
n1230 G00 Z2.000
n1240 G00 X62.500 Y110.000
n1250 G01 Z-1.000 F75
()
n1260 G03 X57.500 R2.500 F125
n1270 G03 X62.500 R2.500
(AfterLastFeedMove)
n1280 G00 Z2.000
n1290 G00 X27.145 Y95.355
n1300 G01 Z-1.000 F75
()
n1310 G03 X22.145 R2.500 F125
n1320 G03 X27.145 R2.500
(AfterLastFeedMove)
```

```
n1330 G00 Z2.000
n1340 G00 X12.500 Y60.000
n1350 G01 Z-1.000 F75
()
n1360 G03 X7.500 R2.500 F125
n1370 G03 X12.500 R2.500
(AfterLastFeedMove)
n1380 G00 Z2.000
n1390 G00 X55.000
n1400 G01 Z-1.000 F75
()
n1410 G03 X65.000 R5.000 F125
n1420 G03 X55.000 R5.000
(AfterLastFeedMove)
n1430 G00 Z2.000
()
(End Program)
n1440 g04 P001 (Dwell for 001 time periods)
n1450 G00 Z0.000 X0.000 Y0.000 (Move Axes to Start Positions)
n1460 M05 (Spindle 05=OFF 04=CCW 03=CW)
n1470 M09 (Coolant 09=OFF 08=ON)
n1480 M02 (Program End)
```