



Mill Post Editor
Installing and Editing Post-Processors

Editing a Post Processor

NOTE: Included with your **BobCAD-CAM** software is free post processor modifications and creation. Please contact Technical Support at (727) 489-0003 for information regarding a post processor for your machine controller. This section is intended for those who wish to modify or create a post processors. This section assumes that the end-user is already familiar with **Windows** operating system software.

IMPORTANT:

Be certain to create a copy of your post processor and rename the file, to avoid overwriting, as saved changes cannot be easily undone.

Step 1 The post processor files are in the following location:

C:\Program Files\BobCAD-CAM\BobCAD-CAM V22\Posts\Mill, for the mill post processors with the .MillPst file extension. Example: **Haas VF.MillPst**

C:\Program Files\BobCAD-CAM\BobCAD-CAM V22\Posts\Lathe, for the lathe post processors with the .LathePst file extension. Example: **Fanuc0T.LathePst**

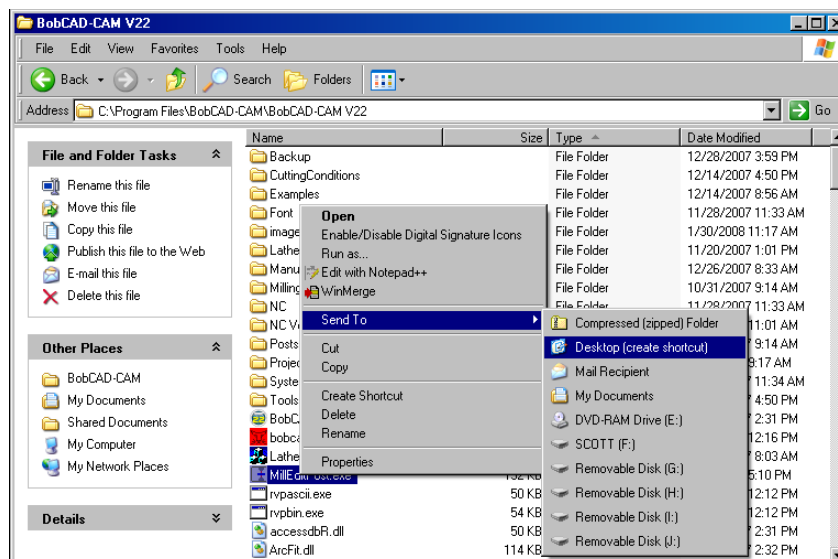
Create a copy of the post you desire to edit, and rename before editing.

Step 2 To edit the post processor you will first need to open the **Mill Post Editor** (or **Lathe Post Editor**) this is an executable file located in the **BobCAD-CAM V22** directory:

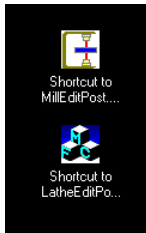
C:\Program Files\BobCAD-CAM\BobCAD-CAM V22\ MillEditPost.exe

C:\Program Files\BobCAD-CAM\BobCAD-CAM V22\ LatheEditPost.exe

Right-click on each file icon and send a shortcut to the desktop.



Step 3 Double-click on the short-cut icon to launch the application:



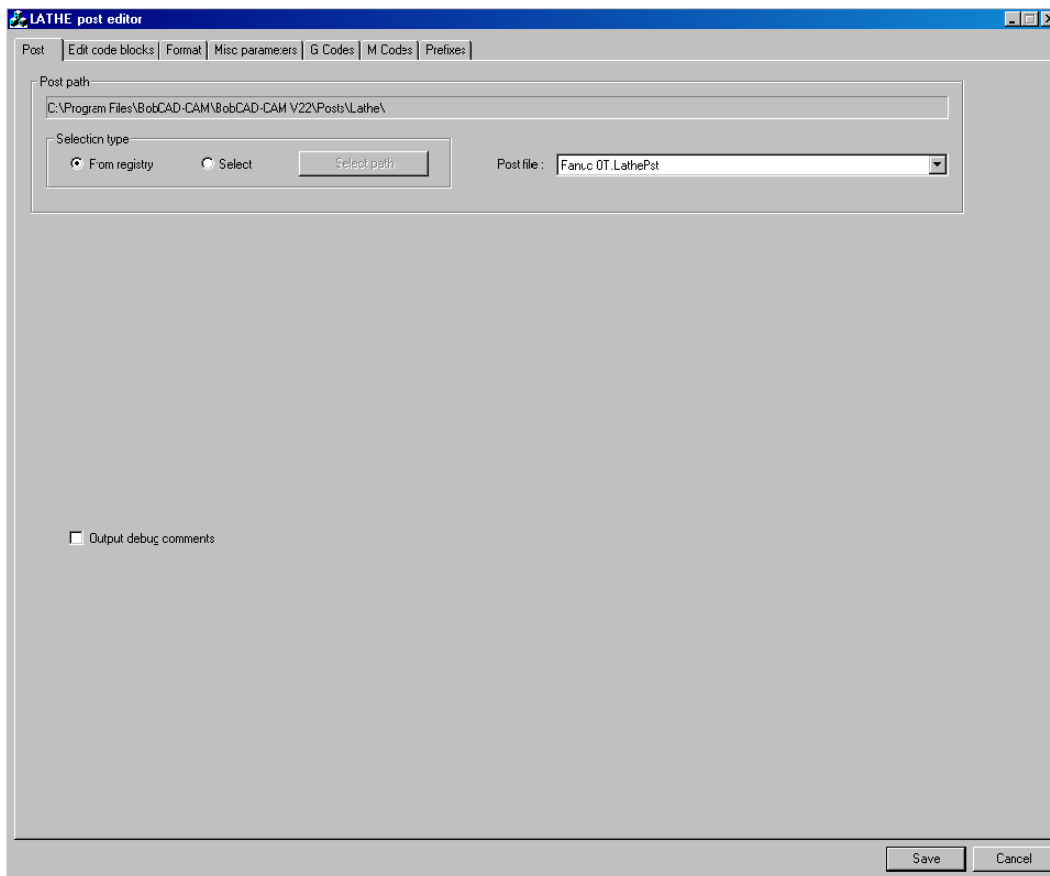
For editing all mill posts with the **.MillPst** extension

For editing all lathe posts with the **.LathePst** extension

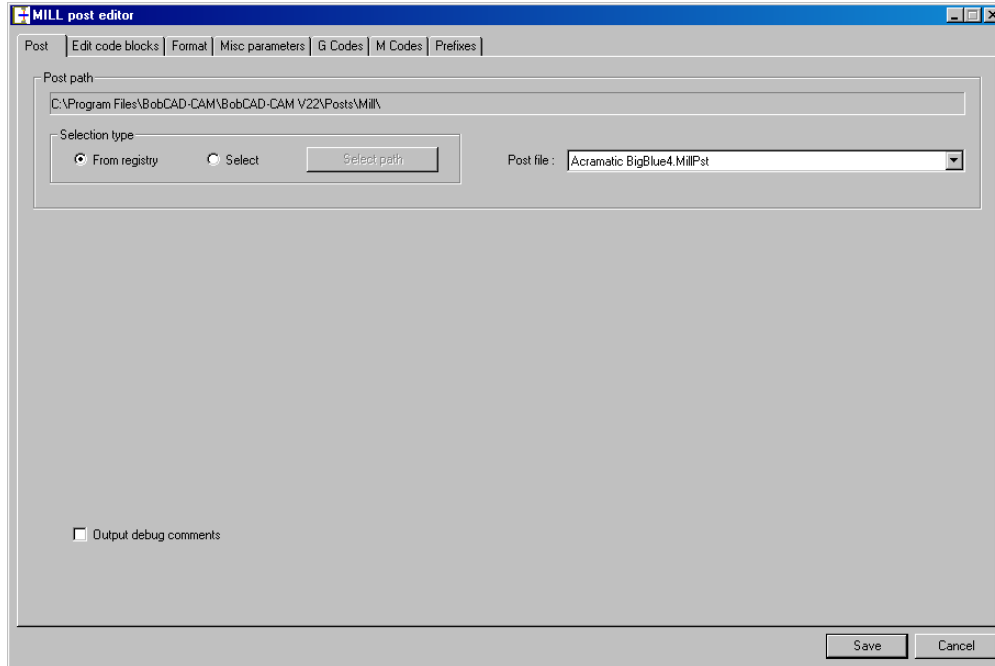
Note: Most post editing is only slight; requiring only relocating existing variables, changing and or adding hard output text. All mill and lathe **Variables** and **Commands**, with a brief description, are listed in the end of this document.

The following window will appear for the corresponding applications:

Lathe Post Editor

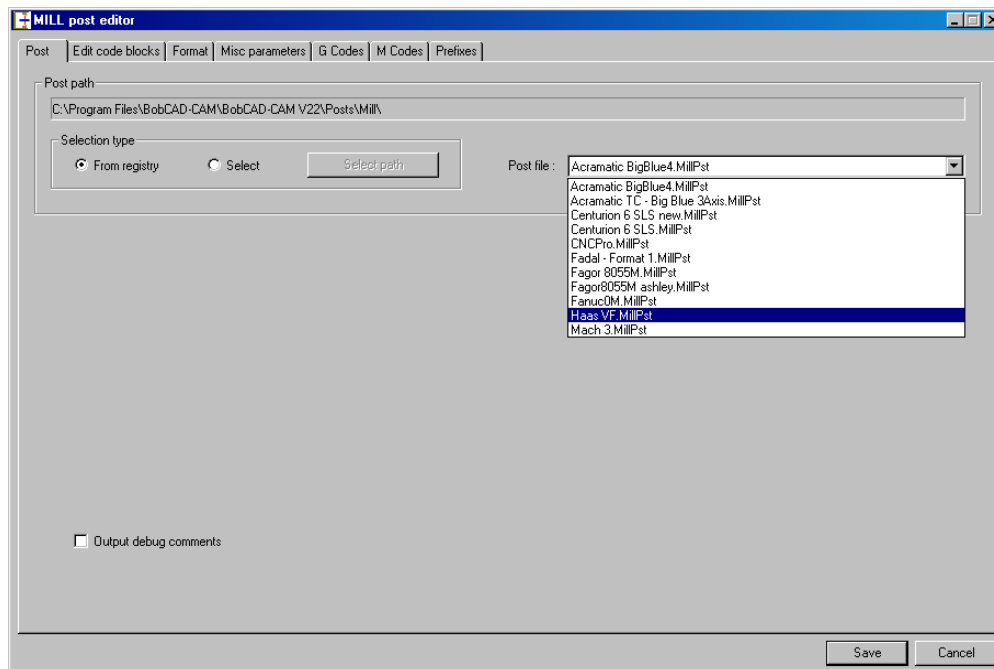


Mill Post Editor



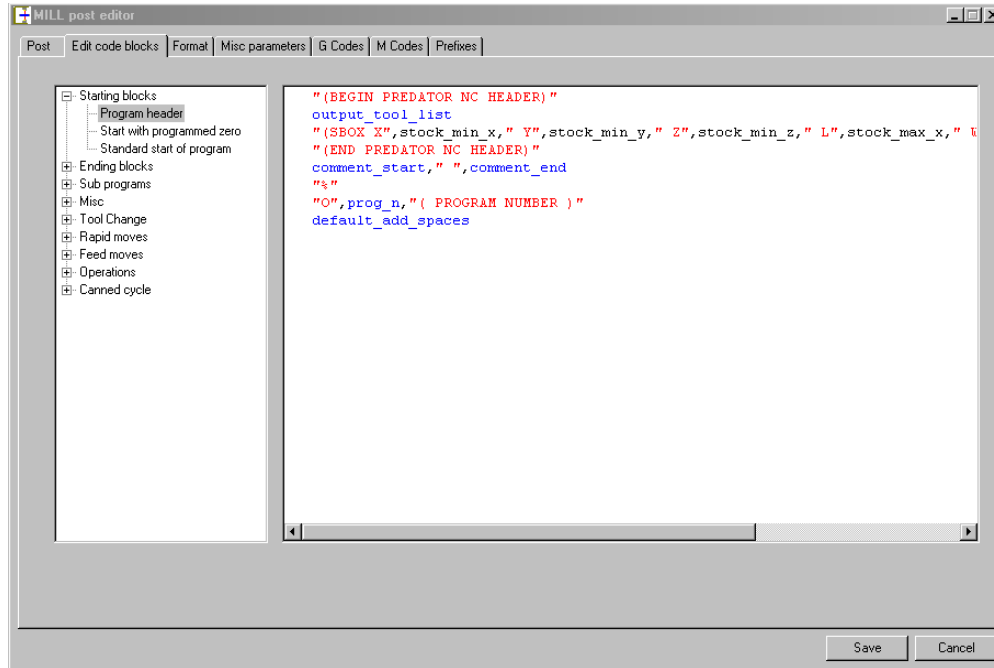
Mill - Post Tab

The following information is true for both post editors. On the **Post** tab, you may select the desired post to be modified. Selection is determined from the pull down menu. You will note in the lower left hand corner of the window is the **Output debug comments** check box. This is a handy feature for determining from which code block the resultant **NC** code is generated. It will be discussed in detail, further in this document. The following examples will be based upon the **Haas VF.MillPst** post processor, it has been selected for their complete use of **standardized G and M codes**. Post can vary considerably, as such, the following examples are purely to be used as a guide. They are not intend to, nor do they match all machine tool controllers.



Mill - Edit code blocks tab – Starting blocks folder - Program header

After selecting the desired post to be modified, click on the **Edit code blocks** tab. In these fields is where you may edit the various information to generate the desired machine tool **NC** output as generated by the **BobCAD-CAM V22** software. The blocks are programmed using **Variables** and **Commands**, displayed in **BLUE**. Characters in quotes will be hard-coded, and displayed in **RED**. You may encounter a variable in **BLACK**; this mean that the variable name is misspelled or the variable is not defined in the **PostingVariables.txt** file. Even if a **Variable** or **Command** is displayed in **BLACK** and not defined in the **PostingVariables.txt**, it still may function.



The above example is information contained in the **Program header** under **Starting blocks**, it is regarding the set up for the **Predator NC Header**. It is used to define the basic stock geometry and tool definitions for the **Backplot** feature within the **Predator CNC Editor – Level2** software. Since this information is located prior to the hard-coded “%”, indicating the beginning of the **NC** program file, it will be disregarded by the controller. The code blocks are written as follows:

```
(BEGIN PREDATOR NC HEADER)
output_tool_list
(SBOX X,stock_min_x,\" Y\",stock_min_y,\" Z\",stock_min_z,
\" L\",stock_max_x,\" W\",stock_max_y,\" H\",stock_max_z,)\",
( END PREDATOR NC HEADER)
comment_start,\" \",comment_end
\"%\"
\"O\",prog_n,\" ( PROGRAM NUMBER )\"
default_add_spaces
```

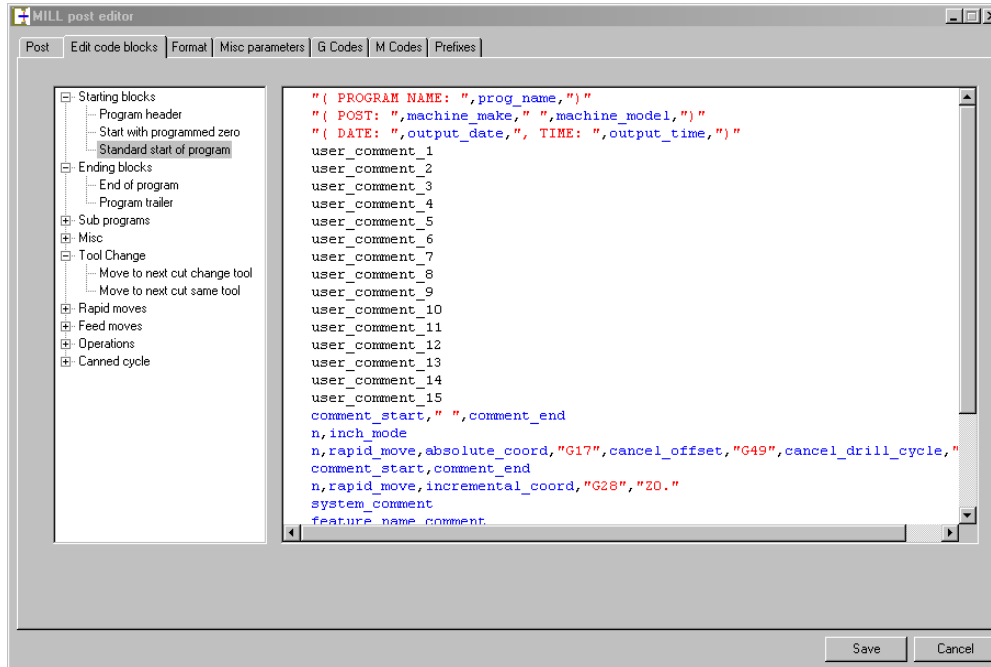
This command outputs nothing, only adds spaces to the program

The corresponding output will be as follows:

(BEGIN PREDATOR NC HEADER)	Predator Header start
(MTOOL T1 S1 D.5 H5.)	Tool 1 defined
(MTOOL T2 S1 D.625 H5.)	Tool 2 defined
(MTOOL T3 S1 D.125 H5.)	Tool 3 defined
(MTOOL T4 S1 D.201 H5.)	Tool 4 defined
(MTOOL T5 S1 D.35 H5.)	Tool 5 defined
(MTOOL T6 S1 D.25 H5.)	Tool 6 defined
(SBOX X-2.25 Y-2.25 Z-1. L4.5 W4.75 H1.)	Stock geometry defined
(END PREDATOR NC HEADER)	Predator Header end
()	NC file start
%	Percent sign indicating start of NC file
O12345 (PROGRAM NUMBER)	Program number as defined in the BobCAD software, no line number

Mill - Edit code blocks tab – Starting blocks folder – Standard start of program

The **Start with programmed zero** and **Standard start of program** under **Starting blocks**, should contain the same code block layout. Currently the software is only referencing the **Standard start of program**, the previous field is for future development.



The above example is information contained in the **Standard start of program**. It is used to output information regarding the program creation, any user comments, initial safety line, first tool change and location of first machining operation. The code blocks are written as follows:

```
"( PROGRAM NAME: ",prog_name,")"
"( POST: ",machine_make," ",machine_model,")"
"( DATE: ",output_date," ", TIME: ",output_time,")"
user_comment_1
user_comment_2
user_comment_3
user_comment_4
user_comment_5
user_comment_6
user_comment_7
user_comment_8
user_comment_9
user_comment_10
user_comment_11
user_comment_12
user_comment_13
user_comment_14
user_comment_15
comment_start," ",comment_end
n,inch mode
n,rapid_move,absolute_coord,"G17","G40","G49",cancel_drill_cycle,"G99"
comment_start,comment_end
n,rapid_move,incremental_coord,"G28","Z0."
system_comment
feature_name_comment
n,t,"M06"
n,s,spindle_on
n,rapid_move,absolute_coord,work_coord,force_x,xr,force_y,yr
n,rapid_move,length_offset
n,coolant_on
comment_start," ",comment_end
```

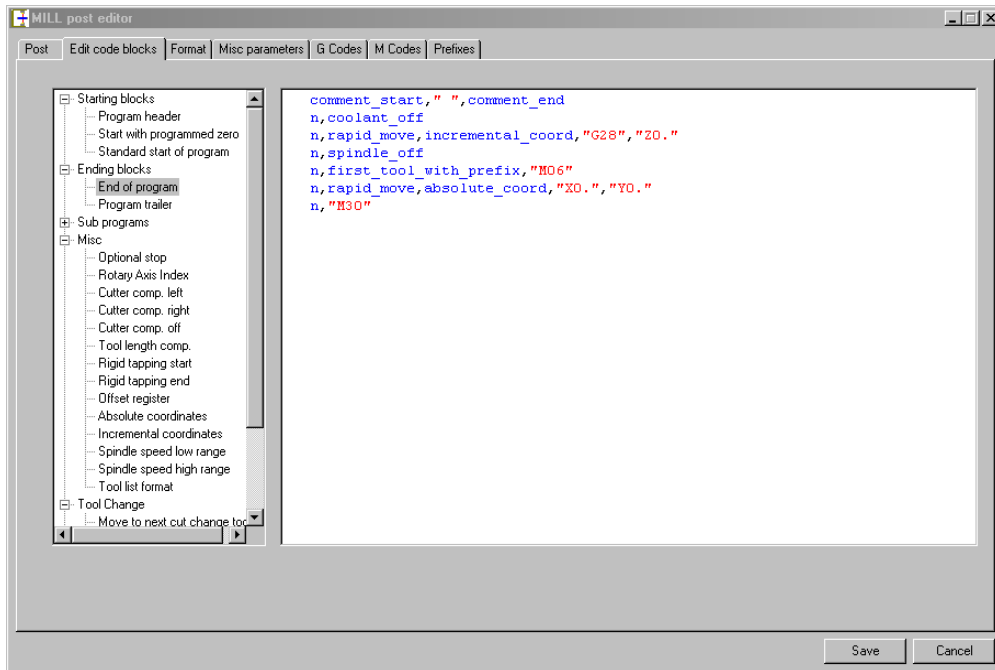
Comments are only displayed if they contain characters.
And have been selected to output, in the BobCAD software.

The corresponding output will be as follows:

(PROGRAM NAME: POST TEST TEMPLATE.NC)	Program name is output
(POST: HAAS VF)	Post name is output, as define in post file
(DATE: THU. 01/31/2008 , TIME: 12:25PM)	Date and Time is output
(INFORMATION IN COMMENT 1 FIELD)	Any comments are output
(THROUGH)	Any comments are output
(INFORMATION IN COMMENT 15 FIELD)	Any comments are output
()	Start of safety line
N10 G20	Inch mode is established
N20 G00 G90 G17 G40 G49 G80 G99	Safety line g-code Variables and hard-code
()	Start of first tool change
N30 G91 G28 Z0.	Automatic return to reference point, location hard-coded
(JOB 1 POCKET)	System comment as generate by Feature in CAM Tree
(TOOL #1 0.5000 ENDMILL ROUGH)	Feature name comment, as define by BobCAD and selected tool in Feature
N40 T1 M06	First tool change
N50 S10000 M03	Spindle speed, as define by Feature , spindle on, defined in M Codes tab
N60 G90 G54 X0. Y0.	Rapid to first cut location, and establish work coordinates
N70 G43 H1 Z.5	Length offset as define in Misc. and Tool length comp. field in post editor
N80 M08	Coolant is turned on, defined in M Codes tab
()	End of first tool change

Mill - Edit code blocks tab – Ending blocks folder – End of program

The **End of program** under **Ending blocks**, should contain information for turning off operations and returning to a home or safe position.



The above example is information contained in the **End of program**. It is used to output variables regarding the program end of file, turning off operations and returning to a safe location. The code blocks are written as follows:

```
comment_start, " ", comment_end
n, coolant_off
n, rapid_move, incremental_coord, "G28", "Z0."
n, spindle_off
n, first_tool_with_prefix, "M06"
n, rapid_move, absolute_coord, "X0.", "Y0."
n, "M30"
```

The corresponding output will be as follows:

()	Start of program end
N1620 M09	Coolant off, defined in M Codes tab
N1630 G00 G91 G28 Z0.	Automatic return to reference point
N1640 M05	Spindle off, defined in M Codes tab
N1650 T1 M06	Change to first tool
N1660 G90 X0. Y0.	Rapid to safe location
N1670 M30	End of program

The above example is a code block definition contained in the **Move to next cut change tool** field. It is used to output variables regarding turning off operation, moving to the tool change position, executing the change and moving to the next machining operation. You will note that the code blocks are similar, if not the same as, the first tool change defined in the **Standard start of program**. The code blocks are written as follows:

```
comment_start," ",comment_end
n,coolant_off
n,spindle_off
n,"G49"
n,rapid_move,incremental_coord,"G28","Z0."
system_comment
feature_name_comment
n,t,"M06"
n,s,spindle_on
n,rapid_move,absolute_coord,work_coord,force_x,xr,force_y,yr
n,rapid_move,length_offset
n,coolant_on
comment_start," ",comment_end
```

The corresponding output will be as follows:

()	Start of tool change
N790 M09	Coolant off, defined in M Codes tab
N800 M05	Spindle off, defined in M Codes tab
N810 G49	Length offset cancel, hard-coded
N820 G91 G28 Z0.	Automatic return to reference point
(JOB 1 POCKET)	System comment as generate by Feature in CAM Tree
(TOOL #2 0.6250 ENDMILL FINISH)	Feature name comment, as define by BobCAD and selected tool in Feature
N830 T2 M06	Change to next tool
N840 S403 M03	Spindle speed, as define by Feature , spindle on, defined in Mcodes tab
N850 G90 G54 X0. Y.4375	Rapid to first cut location, and establish work coordinates
N860 G43 H2 Z.5	Length offset as define in Misc. and Tool length comp. field in post editor
N870 M08	Coolant is turned on, defined in M Codes tab
()	End of next tool change

Mill - Edit code blocks tab – Tool Change folder – Move to next cut same tool

The **Move to next cut same tool** field is used mainly to set the spindle speed (if different from previous operation), comment the operation and move to the next cut. The code blocks are written as follows:

```
comment_start," ",comment_end
system_comment
feature_name_comment
n,s
n,rapid_move,absolute_coord,work_coord,force_x,xr,force_y,yr
comment_start," ",comment_end
```

The corresponding output will be as follows:

()	Start of next cut with same tool
(JOB 2 CONTOUR)	System comment as generate by Feature in CAM Tree
(TOOL #1 0.5000 ENDMILL ROUGH)	Feature name comment, as define by BobCAD and selected tool in Feature
N640 S374	Spindle speed, as define by Feature , spindle is already on, No MO3
N650 G90 G54 X0. Y2.25	Rapid to first cut location, and establish work coordinates
()	End of next cut with same tool

Mill - Edit code blocks tab – Rapid moves folder

The **Rapid moves** folder contains all code block definitions for rapid moves within the **NC** file. It contains the following fields:

First rapid move z	n,rapid_move,zr
Position move z	n,rapid_move,zr
Rapid move xy	n,rapid_move,xr,yr
Rapid move z	n,rapid_move,zr
Rapid move xyz	n,rapid_move,xr,yr,zr

Mill - Edit code blocks tab – Feed moves folder

The **Feed moves** folder contains all code block definitions for rapid moves within the **NC** file. It contains the following fields:

Feed move xy leadin	<code>n,cc,feed_move,x_f,y_f,feed_rate</code>
Feed move xy	<code>n,feed_move,x_f,y_f,feed_rate</code>
Feed move xy leadout	<code>n,cc,feed_move,x_f,y_f,feed_rate</code>
Feed move z	<code>n,feed_move,z_f,feed_rate</code>
Feed move xyz	<code>n,feed_move,x_f,y_f,z_f,feed_rate</code>
Arc move	<code>n,g_arc_move,x_f,y_f,arc_center,feed_rate</code>

Mill - Edit code blocks tab – Operations folder

The **Operations** folder defines the code block for multiple operations within a single **BobCAD** (.bbcd) drawing file and the resultant generated g-code NC file. It contains the following fields:

Start of operation	<code>" (START OF OPERATION) "</code>
End of operation	<code>" (END OF OPERATION) "</code>

Mill - Edit code blocks tab – Canned Cycles folder

IMPORTANT:

These code blocks are independent for each controller; as a result they are the most difficult to properly program. However, they will greatly reduce the length of the g-code **NC** program file and increase the machine tool efficiency. Care must be taken when verifying the generated output to ensure correct functionality.

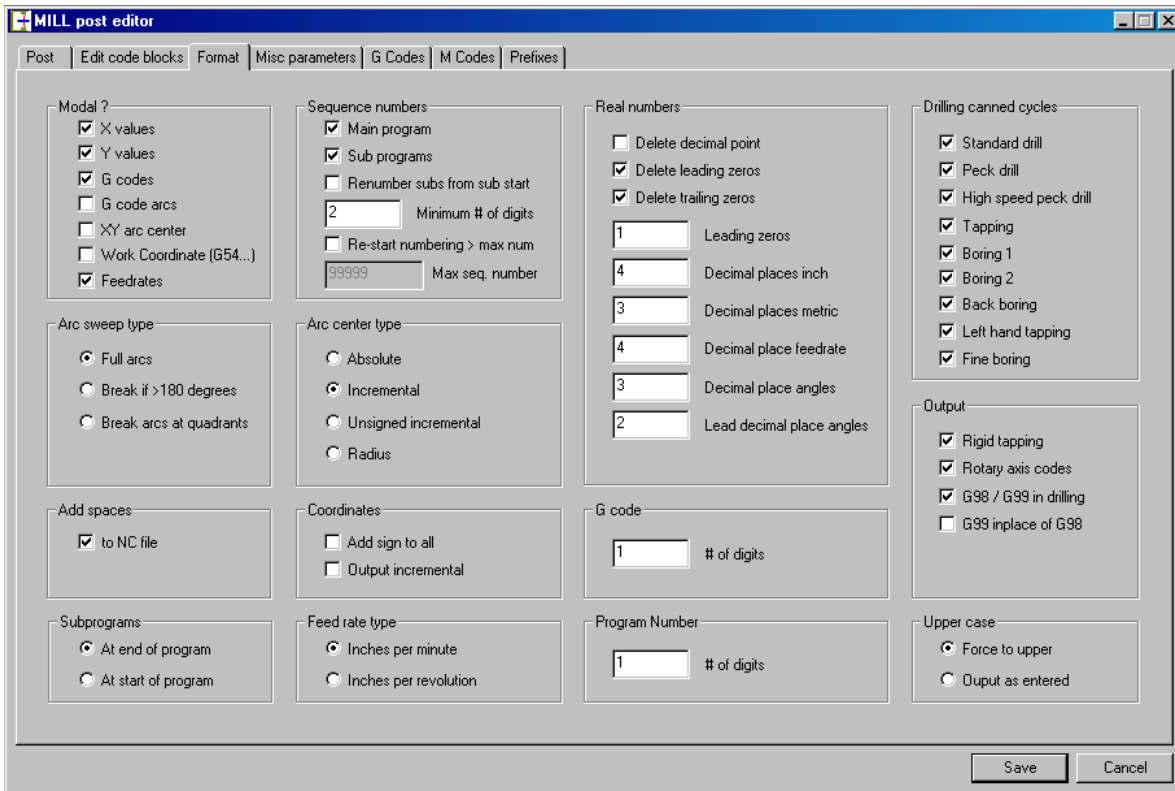
The **Canned cycle** folder contains all the fields for defining the code blocks for each canned cycle, the point format for each cycle (with subprogram and without) and the termination of the cycles. It contains the following fields:

Standard drill	<code>n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,canned_feed_rate</code>
Standard drill with dwell	<code>n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,dwell,canned_feed_rate</code>
Peck drill	<code>n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,peck_drill_increment,dwell,canned_feed_rate</code>
High speed peck drill	<code>n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,peck_drill_increment,dwell,canned_feed_rate</code>
Tap	<code>rigid_tapping_start</code>
	<code>n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,dwell,canned_feed_rate</code>
Left handed tap	<code>rigid_tapping_start</code>
	<code>n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,dwell,canned_feed_rate</code>
Fine boring	<code>n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,dwell,canned_feed_rate</code>
Boring 1	<code>n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,canned_feed_rate</code>
Boring 2	<code>n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,canned_feed_rate</code>
Boring 1 with dwell	<code>n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,dwell,canned_feed_rate</code>
Boring 2 with dwell	<code>n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,dwell,canned_feed_rate</code>
Back boring	<code>n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,dwell,canned_feed_rate</code>
Canned cycle cancel	<code>rigid_tapping_end</code> <code>n,cancel_drill_cycle</code>
Canned cycle point format	<code>n,x_f,y_f,z_f,canned_feed_rate</code>
Point format Standard drill	<code>n,x_f,y_f,z_f,canned_feed_rate</code>
Point format Standard drill with dwell	<code>n,x_f,y_f,z_f,canned_feed_rate</code>
Point format Peck drill	<code>n,x_f,y_f,z_f,canned_feed_rate</code>
Point format High speed peck drill	<code>n,x_f,y_f,z_f,canned_feed_rate</code>
Point format Tap	<code>n,x_f,y_f,z_f,canned_feed_rate</code>
Point format Left handed tap	<code>n,x_f,y_f,z_f,canned_feed_rate</code>
Point format Fine boring	<code>n,x_f,y_f,z_f,canned_feed_rate</code>
Point format Boring 1	<code>n,x_f,y_f,z_f,canned_feed_rate</code>
Point format Boring 2	<code>n,x_f,y_f,z_f,canned_feed_rate</code>
Point format Boring 1 with dwell	<code>n,x_f,y_f,z_f,canned_feed_rate</code>
Point format Boring 2 with dwell	<code>n,x_f,y_f,z_f,canned_feed_rate</code>
Point format Back boring	<code>n,x_f,y_f,z_f,canned_feed_rate</code>

The following tabs in the **MILL Post Editor** contain small fields and check boxes for editing and tuning the **NC** program file to the required output. They are self explanatory, as such, we will not show an output for each field change. The best method for understanding the functionality is; edit a single field, save the post file and re-post **NC** code to view results.

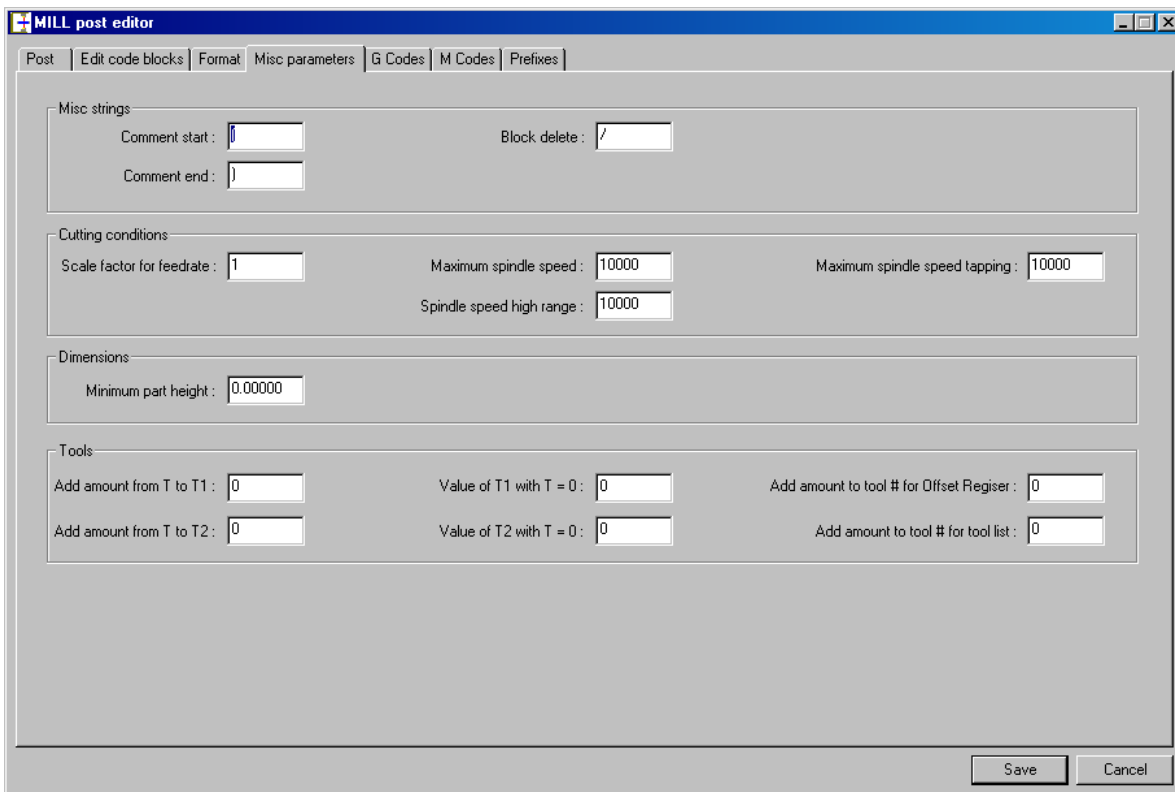
Mill – Format tab

The **Format** tab contains fields to define or select which code is **Modal** output (or not), how **Sequence numbers** output, leading and trailing output for **Real numbers**, **Drilling canned cycles** for controller, amount of **Arc sweep type**, **Arc center type**, **Add spaces** to entire file, add sign and change **Coordinates** type, number of **G code** digits, **Subprograms** location, **Feed rate type**, number of digits for **Program Numbers**, canned cycle **Output**, and changing to **Uppercase** in the **NC** file.



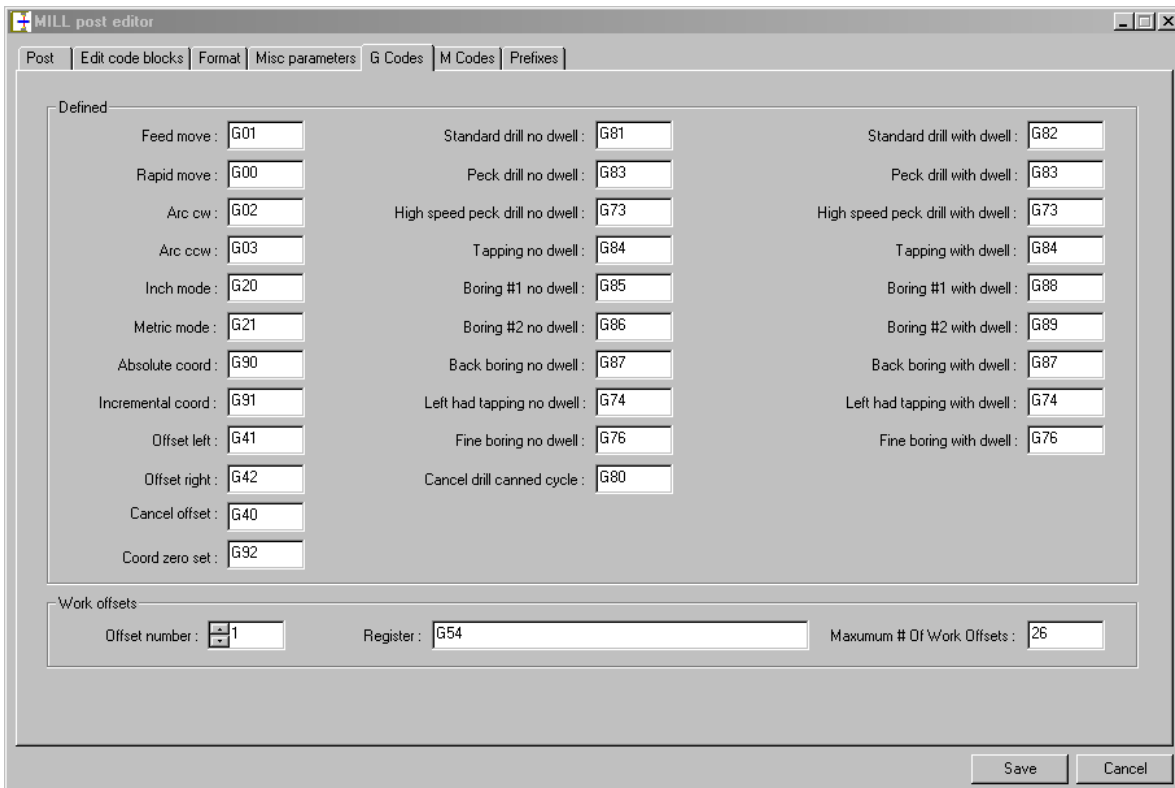
Mill – Misc. parameters tab

The **Misc. parameters** tab contains fields to define or select the **Misc strings** for comments and block deletes, **Cutting conditions** to set maximum spindle speeds and gear ranges, **Dimensions** for establishing **Minimum part height** and **Tools** for defining tool number and offset output.



Mill – G Codes tab

The **G Codes** tab contains fields to enter the format of the **Defined** variables to be output in the **NC** file and the **Work Offsets** number, g code definition and number of offsets.



Mill – M Codes tab

The **M Codes** tab contains fields to enter the format of the **Defined** variables to be output in the NC file.



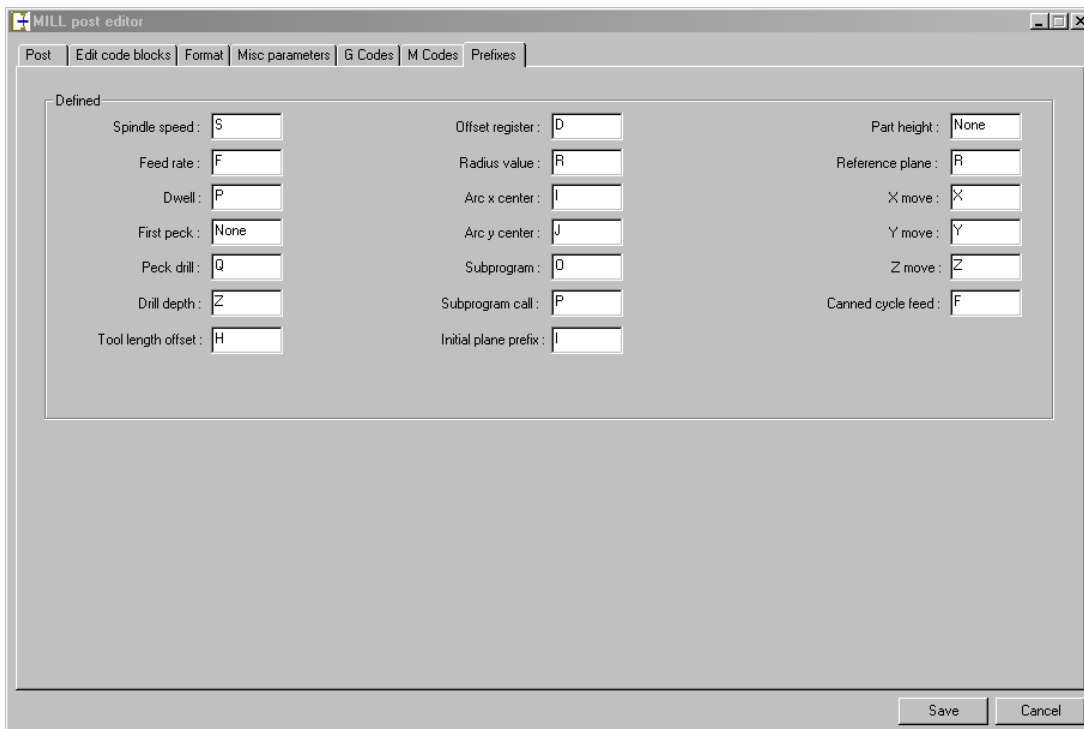
The screenshot shows the 'MILL post editor' window with the 'M Codes' tab selected. The 'Defined' section contains the following fields:

End of file:	M02	Spindle forward:	M03
Stop:	M00	Spindle Reverse:	M04
Optional stop:	M01	Spindle stop:	M05
Subprogram call:	M98	Coolant on:	M08
Subprogram return:	M99	Coolant off:	M09

Buttons for 'Save' and 'Cancel' are located at the bottom right of the window.

Mill – Prefixes tab

The **Prefixes** tab contains fields to enter the format of the **Defined** variables to be output in the NC file.



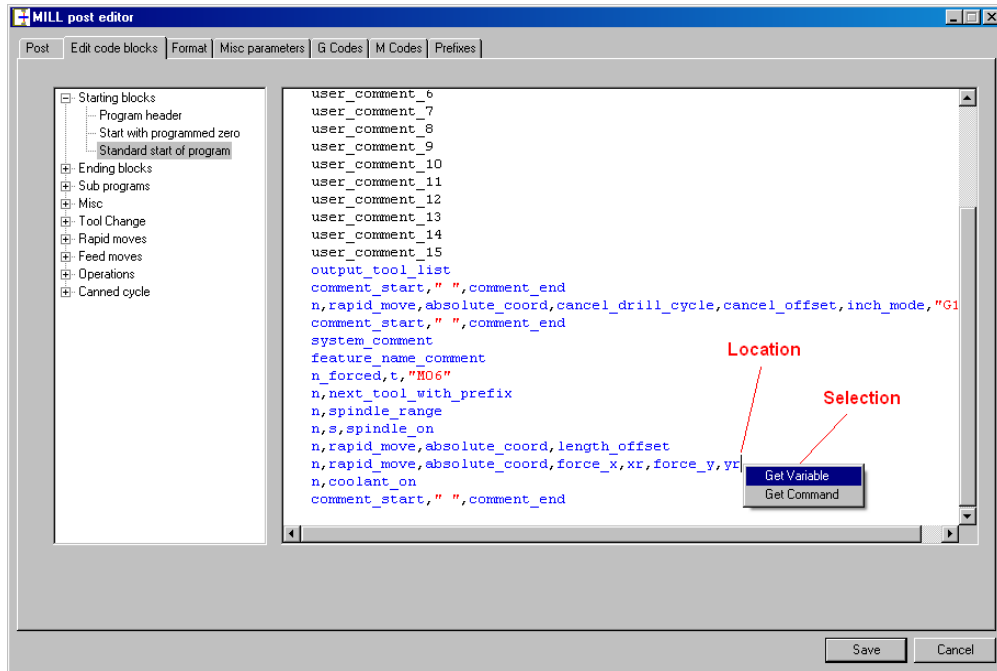
The screenshot shows the 'MILL post editor' window with the 'Prefixes' tab selected. The 'Defined' section contains the following fields:

Spindle speed:	S	Offset register:	D	Part height:	None
Feed rate:	F	Radius value:	R	Reference plane:	R
Dwell:	P	Arc x center:	I	X move:	X
First peck:	None	Arc y center:	J	Y move:	Y
Peck drill:	Q	Subprogram:	D	Z move:	Z
Drill depth:	Z	Subprogram call:	P	Canned cycle feed:	F
Tool length offset:	H	Initial plane prefix:	I		

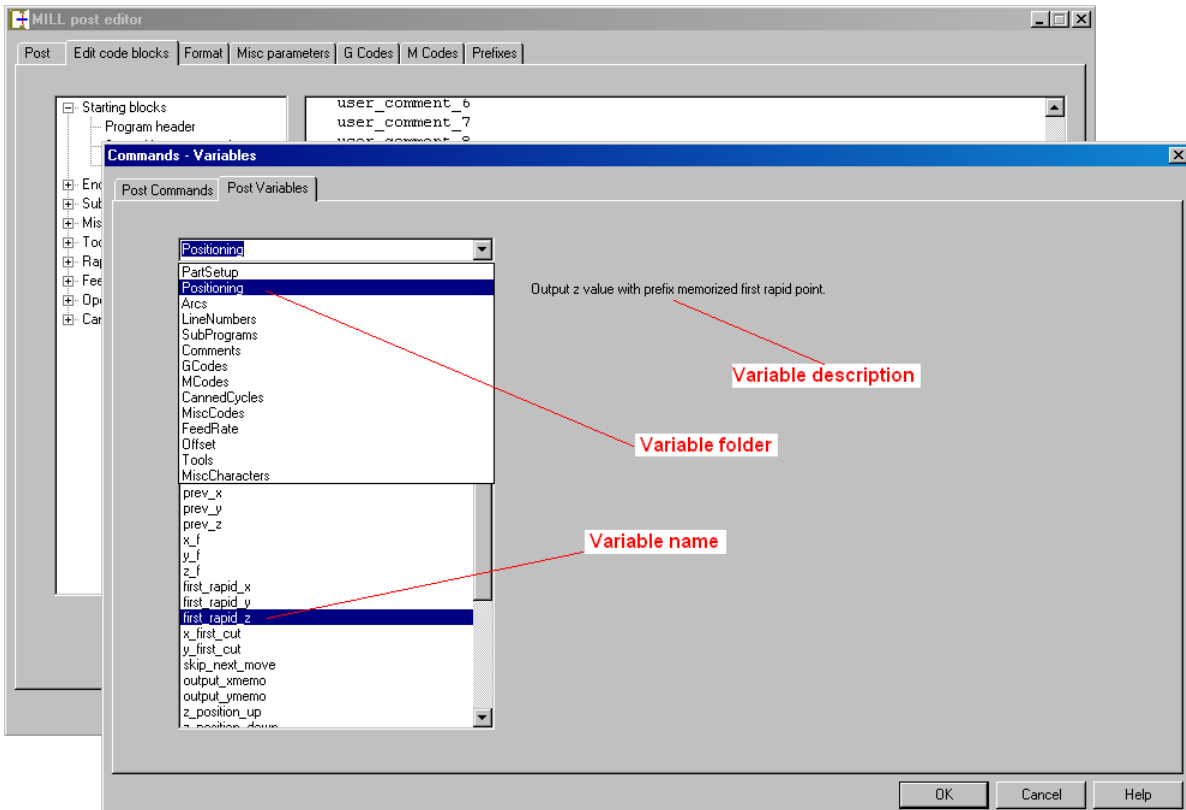
Buttons for 'Save' and 'Cancel' are located at the bottom right of the window.

Mill – Notes on Editing the Code blocks field – Getting Variables/Commands

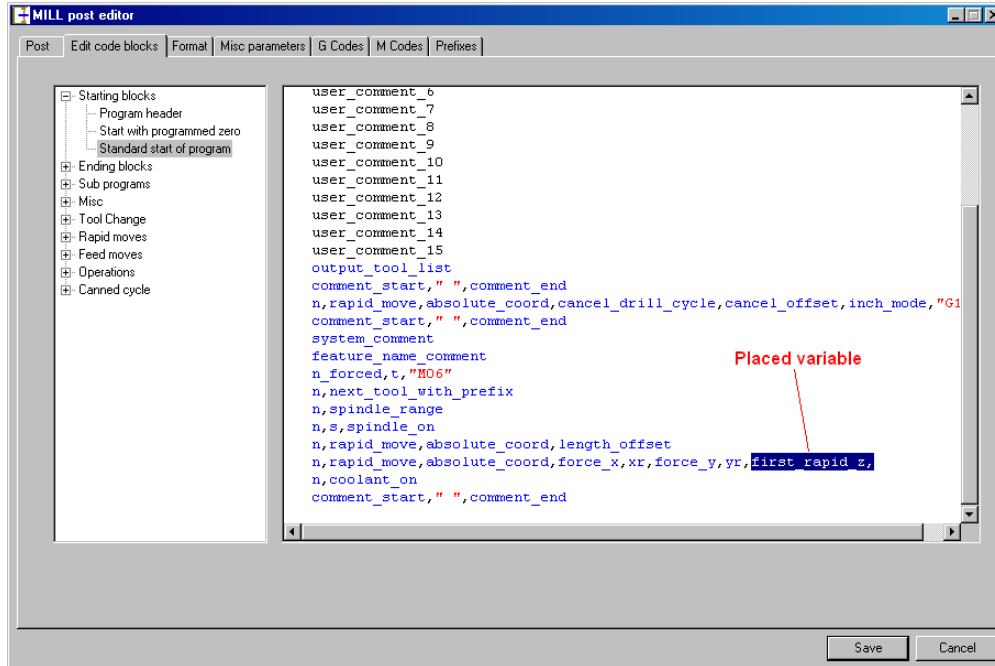
All **Commands** and **Variables** can be pulled-up (displayed) and selected when editing a field in the **Edit code blocks** tab. Simply place your cursor at the location to enter the desired variables and or commands. Right-click, and select the desired folder, **Get Variable** or **Get Command**.



A new window called, **Commands – Variables** will display. You may change selections by clicking on the corresponding **Post Commands** or **Post Variables** tab. Select the desired folder, by click on the on the pull-down menu and then double-clicking on the variable name. You may also high-light the variable name and click **OK**.

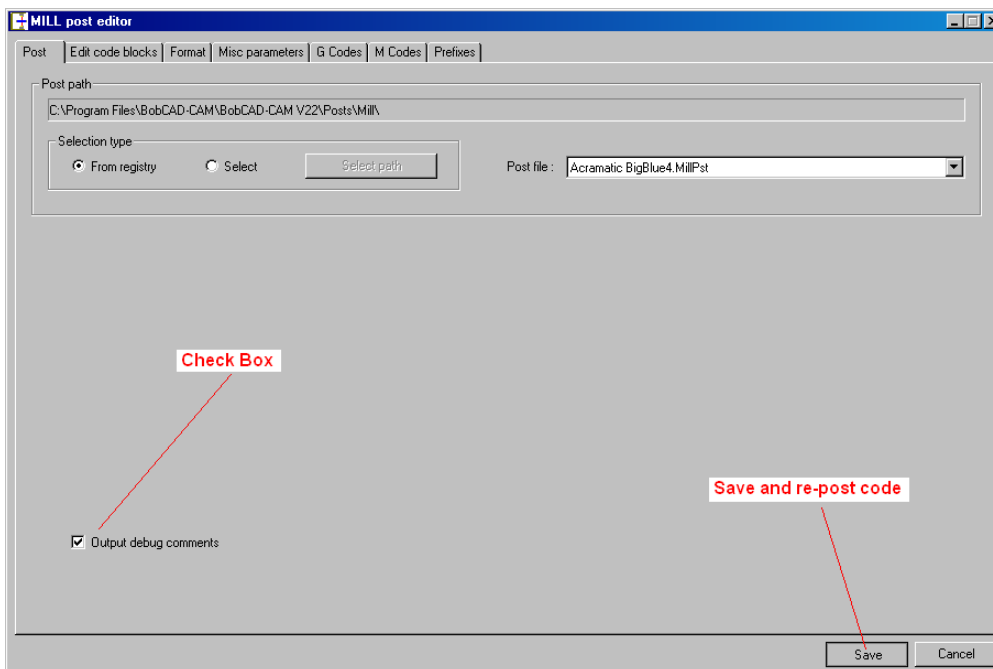


With either method the high-lighted variable will be placed in the location we determine in the **Edit code blocks** field. As you will see, it is quite easy to build the **Edit code blocks** fields to generate the **NC** file for your machine tool with your edited post processor.



Mill – Notes on Editing the Code blocks field – Outputting debug comments

On the **Post** tab in the lower left-hand corner is the **Output debug comments** check box. If you check the box, and click **Save**, the code will post with comments. These comments will indicate where the **Edit code block field** and the resultant output was derived.



The example below this a program using **Output debug comments** (high-lighted in yellow).

```
***** 28 - Rapid moves//Position move Z *****
(BEGIN PREDATOR NC HEADER)
(MTOOL T1 S1 D.5 H5.)
(SBOX X-2.25 Y-2.25 Z-1. L4.5 W4.75 H1.)
(END PREDATOR NC HEADER)
()
%
O100 ( PROGRAM NUMBER )
***** 2 - Starting blocks//Standard start of program *****
( PROGRAM NAME: POST TEST TEMPLATE.NC)
( POST: HAAS VF)
( DATE: FRI. 02/01/2008 , TIME: 11:12AM)
()
N10 G20
N20 G00 G90 G17 G40 G49 G80 G99
()
N30 G91 G28 Z0.
(JOB 2 CONTOUR)
(TOOL #1 0.5000 ENDMILL ROUGH)
N40 T1 M06
N50 S385 M03
N60 G90 G54 X0. Y2.25
N70 G43 H1 Z.25
N80 M08
()
***** 40 - Operations//Start of operation *****
( START OF OPERATION )
***** 50 - Rapid moves//Rapid move Z *****
N90 Z.1
***** 51 - Feed moves//Feed move Z *****
N100 G01 Z-1. F7.
***** 56 - Feed moves//Feed move xy leadin *****
N110 G41 D1 X-.25 F15.
***** 64 - Feed moves//Arc move *****
N120 G03 X0. Y2. I.25 J0.
***** 53 - Feed moves//Feed move XY *****
N130 G01 X2.
***** 53 - Feed moves//Feed move XY *****
N140 Y-2.
***** 53 - Feed moves//Feed move XY *****
N150 X-2.
***** 53 - Feed moves//Feed move XY *****
N160 Y2.
***** 53 - Feed moves//Feed move XY *****
N170 X0.
***** 53 - Feed moves//Feed move XY *****
N180 X.1
***** 64 - Feed moves//Arc move *****
N190 G03 X.35 Y2.25 I0. J.25
***** 57 - Feed moves//Feed move xy leadout *****
N200 G40 G01 X.1
***** 50 - Rapid moves//Rapid move Z *****
N210 G00 Z.25
***** 5 - Ending blockes//End of program *****
()
N220 M09
N230 G91 G28 Z0.
N240 M05
N250 T1 M06
N260 G90 X0. Y0.
N270 M30

***** 24 - Ending blocks//Program trailer *****
%
```

You will note that each debug comment has a number prefix, this corresponds to the function number in the **.MillPst** file. Remember to turn off **Output debug comments**, **Save** the changes and re-post code; prior to saving or sending the NC file to your machine tool.

The example below this the same program with **Output debug comments** turned off and the corresponding code blocks listed to the right of each line.

```

(BEGIN PREDATOR NC HEADER)
(MTOOL T1 S1 D.5 H5.)
(SBOX X-2.25 Y-2.25 Z-1. L4.5 W4.75 H1.)

(END PREDATOR NC HEADER)
()
%
O100 ( PROGRAM NUMBER )
( PROGRAM NAME: POST TEST TEMPLATE.NC)
( POST: HAAS VF)
( DATE: FRI. 02/01/2008 , TIME: 11:21AM)
()
N10 G20
N20 G00 G90 G17 G40 G49 G80 G99

()
N30 G91 G28 Z0.
(JOB 2 CONTOUR)
(TOOL #1 0.5000 ENDMILL ROUGH)
N40 T1 M06
N50 S385 M03
N60 G90 G54 X0. Y2.25

N70 G43 H1 Z.25
N80 M08
()
( START OF OPERATION )
N90 Z.1
N100 G01 Z-1. F7.
N110 G41 D1 X-.25 F15.
N120 G03 X0. Y2. I.25 J0.
N130 G01 X2.
N140 Y-2.
N150 X-2.
N160 Y2.
N170 X0.
N180 X.1
N190 G03 X.35 Y2.25 I0. J.25
N200 G40 G01 X.1
N210 G00 Z.25
()
N220 M09
N230 G91 G28 Z0.
N240 M05
N250 T1 M06
N260 G90 X0. Y0.
N270 M30

%
" (BEGIN PREDATOR NC HEADER) "
output_tool_list
"(SBOX X",stock_min_x," Y",stock_min_y," Z",stock_min_z,
" L",stock_max_x," W",stock_max_y," H",stock_max_z,")"
"(END PREDATOR NC HEADER)"
comment_start," ",comment_end
"% "
"O",prog_n,"( PROGRAM NUMBER )"
"( PROGRAM NAME: ",prog_name,")"
"( POST: ",machine_make," ",machine_model,")"
"( DATE: ",output_date," , TIME: ",output_time,")"
comment_start," ",comment_end
n,inch_mode
n,rapid_move,absolute_coord,"G17",cancel_offset,"G49",
cancel_drill_cycle,"G99"
comment_start," ",comment_end
n,rapid_move,incremental_coord,"G28","Z0."
system_comment
feature_name_comment
n,t,"M06"
n,s,spindle_on
n,rapid_move,absolute_coord,work_coord,force_x,xr,
force_y,yr
n,rapid_move,length_offset
n,coolant_on
comment_start," ",comment_end
"( START OF OPERATION )"
n,rapid_move,zr
n,feed_move,z_f,feed_rate
n,cc,feed_move,x_f,y_f,feed_rate
n,g_arc_move,x_f,y_f,arc_center,feed_rate
n,feed_move,x_f,y_f,feed_rate
n,feed_move,x_f,y_f,feed_rate
n,feed_move,x_f,y_f,feed_rate
n,feed_move,x_f,y_f,feed_rate
n,feed_move,x_f,y_f,feed_rate
n,feed_move,x_f,y_f,feed_rate
n,g_arc_move,x_f,y_f,arc_center,feed_rate
n,cc,feed_move,x_f,y_f,feed_rate
n,rapid_move,zr
comment_start," ",comment_end
n,coolant_off
n,rapid_move,incremental_coord,"G28","Z0."
n,spindle_off
n,first_tool_with_prefix,"M06"
n,rapid_move,absolute_coord,"X0. ","Y0."
n,"M30"

"% "

```

Mill – Notes on Editing the Code blocks field – Post text file

The **.MillPst** are actually **ASCII** text files, as such they may be opened with a text editor. **NotePad**, **NotePad++** or the **Predator CNC Editor** may be used to manually edit the post processor file without going through the **Mill Post Editor** application.

IMPORTANT:

Extreme care should be taken when editing with a text editor application, since you cannot verify the variable spelling or post format and the text color will not be indicated.

The file below is the **Haas VF.MillPst** file, opened in the **Predator CNC Editor**.

Customization file for HAAS VF

Post variables listed in MillPostVariables.pst

****Version number MONTH DAY YEAR****

1000. Version Information = Version Month Day Year? "2.0 01 14 2008"

0. File header

```
"(BEGIN PREDATOR NC HEADER)"
output_tool_list
"(SBOX X",stock_min_x," Y",stock_min_y," Z",stock_min_z," L",stock_max_x," W",stock_max_y, " H",stock_max_z,)"
"(END PREDATOR NC HEADER)"
comment_start," ",comment_end
"% "
"O",prog_n,"( PROGRAM NUMBER )"
default_add_spaces
```

1. Start of file programmed zero

```
"( PROGRAM NAME: ",prog_name,)"
"( POST: ",machine_make," ",machine_model,)"
"( DATE: ",output_date,", TIME: ",output_time,)"
user_comment_1
user_comment_2
user_comment_3
user_comment_4
user_comment_5
user_comment_6
user_comment_7
user_comment_8
user_comment_9
user_comment_10
user_comment_11
user_comment_12
user_comment_13
user_comment_14
user_comment_15
comment_start," ",comment_end
n,inch_mode
n,rapid_move,absolute_coord,"G17",cancel_offset,"G49",cancel_drill_cycle,"G99"
comment_start,comment_end
n,rapid_move,incremental_coord,"G28","Z0."
system_comment
feature_name_comment
n,t,"M06"
n,s,spindle_on
n,rapid_move,absolute_coord,work_coord,force_x,xr,force_y,yr
n,rapid_move,length_offset
n,coolant_on
comment_start," ",comment_end
```

2. Start of file Standard

```
"( PROGRAM NAME: ",prog_name,)"
"( POST: ",machine_make," ",machine_model,)"
```

```

"( DATE: ",output_date,", TIME: ",output_time,")"
user_comment_1
user_comment_2
user_comment_3
user_comment_4
user_comment_5
user_comment_6
user_comment_7
user_comment_8
user_comment_9
user_comment_10
user_comment_11
user_comment_12
user_comment_13
user_comment_14
user_comment_15
comment_start," ",comment_end
n,inch_mode
n,rapid_move,absolute_coord,"G17",cancel_offset,"G49",cancel_drill_cycle,"G99"
comment_start," ",comment_end
n,rapid_move,incremental_coord,"G28","Z0."
system_comment
feature_name_comment
n,t,"M06"
n,s,spindle_on
n,rapid_move,absolute_coord,work_coord,force_x,xr,force_y,yr
n,rapid_move,length_offset
n,coolant_on
comment_start," ",comment_end

3. Tool change
comment_start," ",comment_end
n,coolant_off
n,spindle_off
n,"G49"
n,rapid_move,incremental_coord,"G28","Z0."
system_comment
feature_name_comment
n,t,"M06"
n,s,spindle_on
n,rapid_move,absolute_coord,work_coord,force_x,xr,force_y,yr
n,rapid_move,length_offset
n,coolant_on
comment_start," ",comment_end

4. Null tool change
comment_start," ",comment_end
system_comment
feature_name_comment
n,s
n,rapid_move,absolute_coord,work_coord,force_x,xr,force_y,yr
comment_start," ",comment_end

5. End of file for non-zero tool
comment_start," ",comment_end
n,coolant_off
n,rapid_move,incremental_coord,"G28","Z0."
n,spindle_off
n,first_tool_with_prefix,"M06"
n,rapid_move,absolute_coord,"X0.", "Y0."
n,"M30"

6. Optional Stop
n,optional_stop

7. Sub program call
n,sub_call,sub_num,"( SUBPROGRAM CALL )"

8. Sub definition
" "
sub_num_with_prefix,sub_comment

9. Sub program return

```

n,sub_return,"(SUBPROGRAM RETURN)"

10. Rotary axis index string

n,"B",rotary_angle

11. Cancel cutter compensation

"G40"

12. Cutter compensation left

"G41",d_offset

13. Cutter compensation right

"G42",d_offset

14. Tool length compensation

" G43",h,force_z,zr

20. Spindle speed low range

"M40","(LOW RANGE SPINDLE)"

21. Spindle speed high range

"M41","(HIGH RANGE SPINDLE)"

22. Rigid tapping start.

n,"M29",s,"(RIGID TAP START)"

23. Rigid tapping end.

n,"M28","(RIGID TAP END)"

24. File trailer.

"%"

25. Format for offset when using offset registers.

26. Set debug.

debug_off

27. First Rapid Move.

n,rapid_move,zr

28. Rapid Move.

n,rapid_move,zr

40. Start of 2axis contour.

"(START OF OPERATION)"

50. Line first rapid move Z.

n,rapid_move,zr

51. Line feed move Z.

n,feed_move,z_f,feed_rate

52. Line rapid move XY.

n,rapid_move,xr,yr

53. Line feed move XY.

n,feed_move,x_f,y_f,feed_rate

54. Line rapid move XYZ.

n,rapid_move,xr,yr,zr

55. Line feed move XYZ.

n,feed_move,x_f,y_f,z_f,feed_rate

56. Line feed move XY on Leadin.

n,cc,feed_move,x_f,y_f,feed_rate

57. Line feed move XY on Leadout.

n,cc,feed_move,x_f,y_f,feed_rate

64. Arc move.

n,g_arc_move,x_f,y_f,arc_center,feed_rate

71. End of 2axis cutting.
"(END OF OPERATION)"
73. High speed peck drill canned cycle.
n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,peck_drill_increment,canned_feed_rate
74. Left handed tapping canned cycle.
rigid_tapping_start
n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,canned_feed_rate
76. Fine boring canned cycle.
n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,dwell,canned_feed_rate
80. Drill canned cycle cancel.
rigid_tapping_end
n,cancel_drill_cycle
81. Standard drill canned cycle.
n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,canned_feed_rate
82. Standard drill canned cycle with dwell.
n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,dwell,canned_feed_rate
83. Peck drill canned cycle.
n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,peck_drill_increment,dwell,canned_feed_rate
84. Tapping canned cycle.
rigid_tapping_start
n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,dwell,canned_feed_rate
85. Boring cycle #1 canned cycle.
n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,canned_feed_rate
86. Boring cycle #2 canned cycle.
n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,canned_feed_rate
87. Back boring cycle canned cycle.
n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,dwell,canned_feed_rate
88. Boring cycle #1 canned cycle with dwell.
n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,dwell,canned_feed_rate
89. Boring cycle #2 canned cycle with dwell.
n,g_canned_cycle,g98_g99,x_f,y_f,drill_depth,reference_plane,dwell,canned_feed_rate
90. Canned cycle drill point format WITH SUBPROGRAMS.
n,x_f,y_f,z_f,canned_feed_rate
113. Canned cycle drill point format for high speed peck drill canned cycle NO SUBPROGRAMS.
n,x_f,y_f,z_f,canned_feed_rate
114. Canned cycle drill point format for left handed tapping canned cycle NO SUBPROGRAMS.
n,x_f,y_f,z_f,canned_feed_rate
116. Canned cycle drill point format for fine boring canned cycle NO SUBPROGRAMS.
n,x_f,y_f,z_f,canned_feed_rate
91. Canned cycle drill point format for standard drill canned cycle NO SUBPROGRAMS.
n,x_f,y_f,z_f,canned_feed_rate
92. Canned cycle drill point format for standard drill canned cycle with dwell NO SUBPROGRAMS.
n,x_f,y_f,z_f,canned_feed_rate
93. Canned cycle drill point format for peck drill canned cycle NO SUBPROGRAMS.
n,x_f,y_f,z_f,canned_feed_rate
94. Canned cycle drill point format for tapping canned cycle NO SUBPROGRAMS.
n,x_f,y_f,z_f,canned_feed_rate
95. Canned cycle drill point format for boring cycle #1 canned cycle NO SUBPROGRAMS.
n,x_f,y_f,z_f,canned_feed_rate
96. Canned cycle drill point format for boring cycle #2 canned cycle NO SUBPROGRAMS.

- n,x_f,y_f,z_f,canned_feed_rate
97. Canned cycle drill point format for back boring cycle canned cycle NO SUBPROGRAMS.
n,x_f,y_f,z_f,canned_feed_rate
98. Canned cycle drill point format for boring cycle #1 canned cycle with dwell NO SUBPROGRAMS.
n,x_f,y_f,z_f,canned_feed_rate
99. Canned cycle drill point format for boring cycle #2 canned cycle with dwell NO SUBPROGRAMS.
n,x_f,y_f,z_f,canned_feed_rate
100. Absolute coordinate output.
absolute_coord
101. Incremental coordinate output.
incremental_coord
110. Tool list format.
"(MTOOL T",list_tool_number,"S1 D",tool_diameter," H",tool_length,")"
200. Is X modal? y
201. Is Y modal? y
202. Is Z modal? y
203. Are the g codes modal? y
204. Are the g codes (G02 and G03) modal in arc milling? n
205. Are the xy (or yz or xz) coordinates modal in arc milling? n
206. Are work coordinates modal ? n
207. Output sequence numbers in subprograms ? y
208. Output sequence numbers? y
209. Number of places for sequence numbers? 2
210. Delete the decimal point? n
211. Delete leading zeros? y
212. Delete trailing zeros? y
213. English or Metric format (E/M)? E
214. Places before decimal point for reals (X, Y, Z)? 1
215. Number of places for G and M codes? 1
216. Places after decimal for feedrate ? 4
217. Scale factor for feedrate ? 1
219. Add spaces to the program? y
221. Break arcs into quadrants? n
222. Arc center a=absolute, b=incremental, d=unsigned inc., e=radius? b
223. Break arcs into two pieces if greater than 180 degrees? n
227. Output G40 after, rather than with, the last linear or arc move? y
230. Use Standard drilling canned cycle? y
231. Use peck drill canned cycle? y
232. Use High speed peck drill canned cycle? y
233. Use tapping canned cycle? y
234. Use boring cycle #1 canned cycle? y
235. Use boring cycle #2 canned cycle? y
236. Use back boring cycle canned cycle? y
237. Use left hand tap cycle canned cycle? y
238. Use fine boring cycle canned cycle? y
240. Amount to add to t to obtain t1? 0
241. Amount to add to t to obtain t2? 0
242. Value of t1 at t = 0? 0
243. Value of t2 at t = 0? 0
258. Maximum number of work offsets? 26
262. Add sign to all coordinate values? n
267. Amount to add to tool # for tool register value? 0
272. Rigid tapping? y
273. Output programmable rotary axis codes? y
313. Z clearance for auto Z programming for XY move? 0.20000
314. Z clearance for auto Z programming for cutting? 0.00000
315. Minimum part height? 0.00000
414. Number of decimal places for metric numbers ? 3
415. Number of decimal places for english numbers? 4
425. Number of decimal for angles? 3
426. Number of leading decimal places for angles? 2
427. Tapping feed rate (1=ipm 2=ipr)? 2

428. Feed rates other than tapping (1=ipm 2=ipr)? 1
 429. Maximum spindle speed for tapping? 10000
 430. Maximum spindle speed? 10000
 431. Spindle speed for high range? 10000
 432. Add amount for tool number in tool list? 0
 433. Maximum sequence number allowed (used when #534 is y)? 99999

 511. Use incremental coordinates? n
 512. Use block delete for stop codes (M00)? n
 513. Output F feedrate values? n
 515. Output G99 instead of G98 in drilling? n
 516. Output G98/G99 in drilling? y
 518. Output M00 codes? n
 526. Start renumbering from start number for subprograms? n
 527. Create subdirectory for nc file? n
 531. Replace spaces in comment with commas ? n
 533. Output subprograms at the beginning of the program? n
 534. Re-Start sequence numbering once max sequence number is reached? n
 535. Are Feed Rates modal? y
 536. Force all codes to upper case ? y

 605. Spindle speed prefix? "S"
 606. Feedrate prefix? "F"
 607. Dwell prefix? "P"
 610. Miscellaneous end of file string? ""
 613. Pattern contour subprogram start code? ""
 614. Inch mode machining? "G20"
 615. Metric mode machining? "G21"
 620. Absolute coordinates? "G90"
 621. Incremental coordinates? "G91"
 622. Coordinate zero set? "G92"
 625. End of file? "M02"
 626. Stop? "M00"
 627. Optional Stop? "M01"
 628. Subprogram call? "M98"
 629. Subprogram return? "M99"
 630. Comment Start? "("
 631. Comment End? ")"
 639. Cancel wire offset? "G40"
 641. Prefix for radius values? "R"
 642. Prefix for arc X center? "I"
 643. Prefix for arc Y center? "J"
 645. Subprogram prefix? "O"
 646. Machine maker? "HAAS"
 647. Machine model? "VF"
 648. Part Height prefix? "None"
 649. Reference Plane prefix? "R"
 650. Wire comp left? "G41"
 651. Wire comp right? "G42"
 656. Block delete? ""
 658. Subprogram call subnumber prefix? "P"
 659. Add these characters to the end of each line? ""
 670. Spindle forward string? "M03"
 671. Spindle reverse string? "M04"
 672. Spindle off string? "M05"
 673. Coolant on string? "M08"
 674. Coolant off String? "M09"
 675. First peck prefix? "None"
 676. Peck drill prefix? "Q"
 677. Drill depth prefix? "Z"
 678. Cutter offset prefix? "D"
 679. Cancel drilling canned cycle? "G80"
 680. GCode for Rapid Move? "G00"
 681. GCode for Feed Move? "G01"
 682. GCode for Arc CW? "G02"
 683. GCode for Arc CCW? "G03"
 684. Prefix for X Move? "X"
 685. Prefix for Y Move? "Y"
 686. Prefix for Z Move? "Z"
 687. Prefix for drill canned cycle feed rate? "F"
 688. Prefix for tool length offset? "H"
 689. Prefix for initial plane? "I"

800. Standard drilling cycle no dwell #1? "G81"
801. Peck drill cycle no dwell #2? "G83"
802. High speed peck drill cycle no dwell #3? "G73"
803. Tapping cycle no dwell #4? "G84"
804. Boring cycle #1 no dwell #5? "G85"
805. Boring cycle #2 no dwell #6? "G86"
806. Back boring cycle no dwell #7? "G87"
807. Left hand tapping cycle no dwell #8? "G74"
808. Fine boring cycle no dwell #9? "G76"
809. Hole making cycle no dwell #10?""
810. Hole making cycle no dwell #11?""
811. Hole making cycle no dwell #12?""

820. Standard drilling cycle with dwell #1? "G82"
821. Peck drill cycle with dwell #2? "G83"
822. High speed peck drill cycle with dwell #3? "G73"
823. Tapping cycle with dwell #4? "G84"
824. Boring cycle #1 with dwell #5? "G88"
825. Boring cycle #2 with dwell #6? "G89"
826. Back boring cycle with dwell #7? "G87"
827. Left hand tapping cycle with dwell #8? "G74"
828. Fine boring cycle with dwell #9? "G76"
829. Hole making cycle with dwell #10?""
830. Hole making cycle with dwell #11?""
831. Hole making cycle with dwell #12?""

900. Work shift #1? "G54"
901. Work shift #2? "G55"
902. Work shift #3? "G56"
903. Work shift #4? "G57"
904. Work shift #5? "G58"
905. Work shift #6? "G59"
906. Work shift #7? "G110"
907. Work shift #8? "G111"
908. Work shift #9? "G112"
909. Work shift #10? "G113"
910. Work shift #11? "G114"
911. Work shift #12? "G115"
912. Work shift #13? "G116"
913. Work shift #14? "G117"
914. Work shift #15? "G118"
915. Work shift #16? "G119"
916. Work shift #17? "G120"
917. Work shift #18? "G121"
918. Work shift #19? "G122"
919. Work shift #20? "G123"
920. Work shift #21? "G124"
921. Work shift #22? "G125"
922. Work shift #23? "G126"
923. Work shift #24? "G127"
924. Work shift #25? "G128"
925. Work shift #26? "G129"
926. Work shift #27? ""
927. Work shift #28? ""
928. Work shift #29? ""
929. Work shift #30? ""
930. Work shift #31? ""
931. Work shift #32? ""
932. Work shift #33? ""

... to ...

993. Work shift #93? ""
994. Work shift #94? ""
995. Work shift #95? ""
996. Work shift #96? ""
997. Work shift #97? ""
998. Work shift #98? ""
999. Work shift #99? ""

Customization Variables and Commands for Mill Post Processing

Commands

<code>oldvars</code>	= use old position variables and misc. variables.
<code>newvars</code>	= use new position position and misc. variables.
<code>force_x</code>	= force the x value to be output next time even if modal.
<code>force_y</code>	= force the y value to be output next time even if modal.
<code>force_z</code>	= force the z value to be output next time even if modal.
<code>force_drill_z</code>	= force the z value for drilling to be output next time even if modal.
<code>memo_xmove</code>	= Memorize the current x movement for use later.
<code>memo_ymove</code>	= Memorize the current y movement for use later.
<code>memorize_line_number</code>	= Memorize sequence number for use later.
<code>memorized_line_number</code>	= Use memorize sequence number.
<code>debug_on</code>	= Turns on automatic debug comments in post output.
<code>debug_off</code>	= Turns off automatic debug comments in post output.
<code>outputxmemo</code>	= Output X value from memo_xmove
<code>outputymemo</code>	= Output Y value from memo_ymove
<code>start_add_block_delete</code>	= Add block delete to all lines until stop_add_block_delete is used
<code>stop_add_block_delete</code>	= Stop adding block delete that started from using start_add_block_delete
<code>force_no_add_spaces</code>	= Force no spaces even when default is set to add spaces.
<code>default_add_spaces</code>	= Set add spaces back to condition before force_no_add_spaces was called.
<code>output_tool_list</code>	= Output tool list.

Variables - Part Setup

<code>absolute_coord</code>	= Output code to set coordinates to absolute mode (G90). Code defined in " G Codes/Absolute Coord ".
<code>absolute_coord_no_code</code>	= Set output to absolute coordinates with no output.
<code>incremental_coord</code>	= Output code to set coordinates to incremental mode (G91). Code defined in " G Codes/Incremental Coord ".
<code>incremental_coord_no_code</code>	= Set output to incremental coordinates with no output.
<code>work_coord</code>	= Output work coordinates when it changing work pieces. (G54 etc) Code defined in " G Codes/Work offsets ".
<code>pattern_work_coord</code>	= Output work coordinates for pattern contours. (G54 etc) Code defined in " G Codes/Work offsets ".
<code>zero_set_xp_yp</code>	= Output code to set current work coordinate to programmed zero (G92X__Y__). Code defined in " G Codes/Coord zero set ".
<code>zero_set_xr_yr</code>	= Output code to set current work coordinate to current cycle start point (G92X__Y__). Code defined in " G Codes/Coord zero set ".
<code>zero_set</code>	= Output set zero command with out XY values (G92). Code defined in " G Codes/Coord zero set ".
<code>metric_mode</code>	= Set output mode to metric.
<code>inch_mode</code>	= Set output mode to inch.
<code>ref_plane</code>	= Output reference plane value with prefix (Charmille). Prefix defined in " Prefixes/Reference plane ".
<code>part_height</code>	= Output workpiece height with prefix. Prefix defined in " Prefixes/Part height ".
<code>part_length</code>	= Output workpiece length with prefix.
<code>part_width</code>	= Output workpiece with with prefix.
<code>h</code>	= Output length offset register with prefix.

Variables – Positioning

<code>z_drill</code>	= Output drilling z value with prefix.
<code>z_clearance</code>	= Output z clearance z value with prefix.
<code>xr</code>	= Output rapid position x with prefix (X1.25).
<code>yr</code>	= Output rapid position y with prefix (Y1.25).
<code>zr</code>	= Output rapid position z with prefix
<code>xr_no_output</code>	= X rapid value to previous move without outputting code.
<code>yr_no_output</code>	= Y rapid value to previous move without outputting code.
<code>zr_no_output</code>	= Z rapid value to previous move without outputting code.
<code>xh</code>	= Output home position x with prefix (X1.25).
<code>yh</code>	= Output home position y with prefix (Y1.25).
<code>zh</code>	= Output home position z with prefix (Z1.25).
<code>xp</code>	= Output programmed zero x (X__).
<code>yp</code>	= Output programmed zero y (Y__).
<code>prev_x</code>	= Output previous x (X__).

Variables – Positioning cont'd

<code>prev_y</code>	= Output previous y (Y ___).
<code>prev_z</code>	= Output previous z (Z ___).
<code>x_f</code>	= Output code for X feed value with prefix.
<code>y_f</code>	= Output code for Y feed value with prefix.
<code>z_f</code>	= Output code for Z feed value with prefix.
<code>first_rapid_x</code>	= Output x value with prefix memorized first rapid point.
<code>first_rapid_y</code>	= Output y value with prefix memorized first rapid point.
<code>first_rapid_z</code>	= Output z value with prefix memorized first rapid point.
<code>x_first_cut</code>	= Output x value with prefix for entrance cut when special cutting conditions are used to enter contour.
<code>y_first_cut</code>	= Output y value with prefix for entrance cut when special cutting conditions are used to enter contour.
<code>skip_next_move</code>	= Skip next movement in order to output later. Used for line swapping in Agie.
<code>output_xmemo</code>	= Output the x value that was memorized with memo_xmove .
<code>output_ymemo</code>	= Output the y value that was memorized with memo_ymove .
<code>z_position_up</code>	= Programmable Z position up.
<code>z_position_down</code>	= Programmable Z position down.
<code>rotary_angle</code>	= Output rotary angle position.
<code>output_rotary_angle</code>	= Output block that defines the output for setting the rotary angle.
<code>out_rotary_clearance</code>	= Output rotary clearance move with prefix.
<code>rotary_clearance</code>	= Output rotary clearance value without prefix.
<code>zero_position_x</code>	= Output part zero position x without prefix.
<code>zero_position_y</code>	= Output part zero position y without prefix.
<code>zero_position_z</code>	= Output part zero position z without prefix.
<code>return_plane_distance</code>	= Output return plane distance without prefix.
<code>security_plane_distance</code>	= Output security plane distance without prefix.

Variables – Arcs

<code>xcenter</code>	= Output x center of arc with prefix (" Format/Arc center type "). Prefix define " Prefixes/Arc x center ".
<code>ycenter</code>	= Output y center of arc with prefix (" Format/Arc center type "). Prefix define " Prefixes/Arc y center ".
<code>rcenter</code>	= Output Radius value of arc with prefix (" Format/Arc center type "). Prefix define " Prefixes/Radius value ".
<code>arc_center</code>	= Output x, and y center of arc with prefixes (" Format/Arc center type "). Prefix define " Prefixes/Arc x center :" & " Prefixes/Arc y center ".

Variables – Line Numbers

<code>n</code>	= Output next sequence number with prefix.
<code>n_forced</code>	= Force output of next sequence number with prefix (N). Sequence number for set in " Format/sequence numbers ".
<code>seq_only</code>	= Output sequence number only without prefix.

Variables – Sub Programs

<code>sub_call</code>	= Output subprogram call. Prefix defined in " Prefixes/Arc x center ".
<code>sub_return</code>	= Output subprogram return. Prefix defined in " Prefixes/Arc y center ".
<code>sub_num</code>	= Output subprogram number without prefix.
<code>sub_num_no_prefix</code>	= Output subprogram number without prefix.
<code>sub_num_with_prefix</code>	= Output subprogram number with prefix. Prefix defined in " Prefixes/Subprogram ".
<code>pattern_contour_sub_start</code>	= Output pattern contour code for subprogram start.

Variables – Comments

<code>comment_start</code>	= Output comment start. Code defined in " Misc parameters/Comment start ".
<code>comment_end</code>	= Output comment end. Code defined in " Misc parameters/Comment end ".
<code>system_comment</code>	= Output the automatic comment " FEATURE 1 CNT2X ROUGH CUT FORWARD ".
<code>feature_name_comment</code>	= Output Feature Name comment " 2 AXIS CONTOUR1-RECTANGULAR DIE3 ".
<code>pass_name_comment</code>	= Output pass name comment " ROUGH1 ".
<code>feature_pass_names_comment</code>	= Output Feature and pass names " 2 AXIS CONTOUR1-RECTANGULAR DIE3 ROUGH1 ".
<code>sub_comment</code>	= Output subprogram description comment.
<code>output_date</code>	= Output the date in the form " Tue. 03/05/2002 ".
<code>output_time</code>	= Output the current time in the form " 11:35AM ".

Variables – Comments cont'd

<code>prog_n</code>	= Output program number without prefix (O12345). Hard code prefix (" O ", <code>prog_n</code>)
<code>prog_name</code>	= Output program name (file name of saved program on disk)
<code>machine_make</code>	= Output machine manufacturers name as string.
<code>machine_model</code>	= Output machine model name as string.
<code>pass_name</code>	= Output pass name. (Used in comment)
<code>feature_name</code>	= Output feature name. (Used in comment)
<code>workpiece_name</code>	= Output workpiece name. (Used in comment)

Variables – G Codes

<code>feed_move</code>	= Set movement to feed and output feed move command (G01). Value defined in " G Codes/Feed move ".
<code>rapid_move</code>	= Set movement to rapid and output rapid move command (G00). Value defined in " G Codes/Rapid move ".
<code>cc</code>	= Output code for Cutter compensation. Codes defined in " G Codes ".
<code>measurement</code>	= Output measurement mode (inch/metric). Inch defined in " G Codes/Inch mode :" & " G Codes/Metric mode ".
<code>g_arc_move</code>	= Output G code for arc moves.
<code>rigid_tapping_start</code>	= Output G Code for rigid tapping start.
<code>rigid_tapping_end</code>	= Output G Code for rigid tapping end.

Variables – M Codes

<code>end_of_file</code>	= Output end of file code (M02). Code define in " M Codes/End of file ".
<code>stop</code>	= Output stop code (M00). Code define in " M Codes/Stop ".
<code>optional_stop</code>	= Output code for Optional Stop (M01). Code define in " M Codes/Optional stop ".
<code>coolant_on</code>	= Output code for turning on coolant. (M07, M08)
<code>coolant_off</code>	= Output code for turning off coolant. (M09)
<code>spindle_on</code>	= Output code for turning on spindle. (M03, M04, M13, M14)
<code>spindle_off</code>	= Output code for turning off spindle. (M05, M15)
<code>mcode_rotary_index</code>	= Output code for 4th axis rotary index.

Variables – Canned Cycles

<code>g_canned_cycle</code>	= Output code for canned cycle. (G73,G74,G76,G81, G82, G83, etc.)
<code>cancel_drill_cycle</code>	= Output code to cancel canned cycle (G80)
<code>g98_g99</code>	= Output G98, G99 codes.
<code>drill_depth</code>	= Output drill depth.
<code>abs_drill_depth</code>	= Output absolute value of drill depth.
<code>reference_plane</code>	= Output reference plane with prefix.
<code>initial_plane</code>	= Output initial plane with prefix.
<code>canned_feed_rate</code>	= Output canned cycle feed rate.
<code>peck_drill_increment</code>	= Output peck drill increment.

Variables – Misc Codes

<code>s</code>	= Output spindle speed with prefix " S1000 ".
<code>sp_speed</code>	= Output spindle speed no prefix.
<code>smax</code>	= Output maximum spindle speed with prefix.
<code>spindle_range</code>	= Output block that defines setting the spindle speed range (High/Low).
<code>length_offset</code>	= Output block that defines the all codes for length offset.
<code>null</code>	= Output ascii character 0 or NULL .
<code>list_tool_number</code>	= Output tool number with added value for tools in the tool list.
<code>tool_diameter</code>	= Output tool diameter without prefix.
<code>tool_radius</code>	= Output tool radius without prefix.
<code>tool_name</code>	= Output name of current tool.
<code>tool_length</code>	= Output tool length without prefix
<code>heiden_ref_height</code>	= Output reference plane for Heidenhein with prefix.
<code>heiden_depth</code>	= Output depth for Heidenhein with prefix.
<code>heiden_peck_incr</code>	= Output peck increment for Heidenhein with prefix.
<code>heiden_dwell</code>	= Output dwell for Heidenhein with prefix.
<code>heiden_pluge_rate</code>	= Output plunge rate for Heidenhein with prefix.

Variables – Feed Rate

`feed_rate` = Output code for feed rate.

Variables – Offset

`cc` = Output code for Cutter compensation. Codes defined in "**G Codes**".
`cancel_offset` = Output cancel wire offset in 2Axis. Code defined in "**G Codes/Cancel offset**".
`d_offset` = Output diameter compensation (offset) register with add amount and prefix "**D1**".
`t_offset` = Output tool offset register number without prefix.
`l_offset` = Output tool length offset register number without prefix
`offset_value` = Output cutter offset value without prefix.

Variables – Tools

`t` = Output current tool number with prefix.
`first_tool` = Output first tool number without prefix.
`first_tool_with_prefix` = Output first tool number with prefix.
`next_tool` = Output next tool without prefix.
`next_tool_with_prefix` = Output next tool with prefix.
`tool_with_offset` = Output tool number and offset register number together with tool prefix (**T0101**).
`next_tool_with_offset` = Output next tool number and next offset register number together with tool prefix (**T0202**).

Variables – Misc Characters

`cr_lf` = Output carriage return line feed characters.
`single_quote` = Output single quote character.
`quote` = Output double quote character.
`n_spaces` = Output spaces for the number of characters in the previous sequence numbers