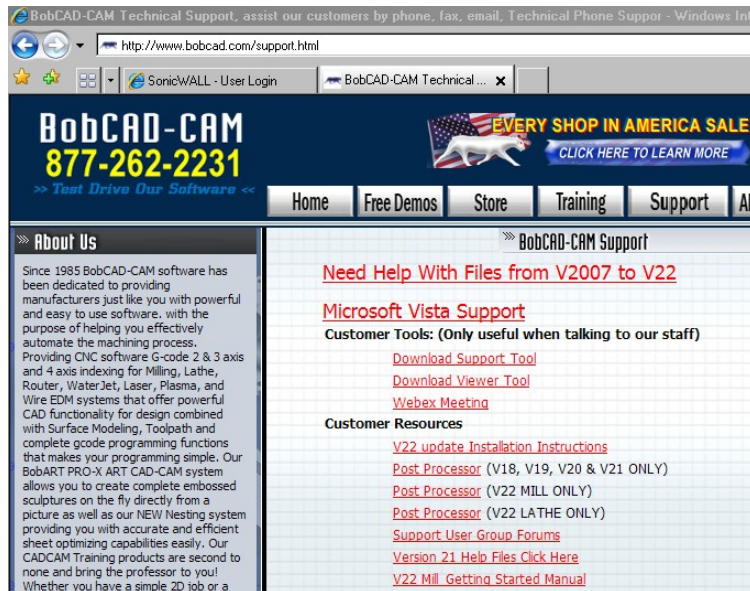




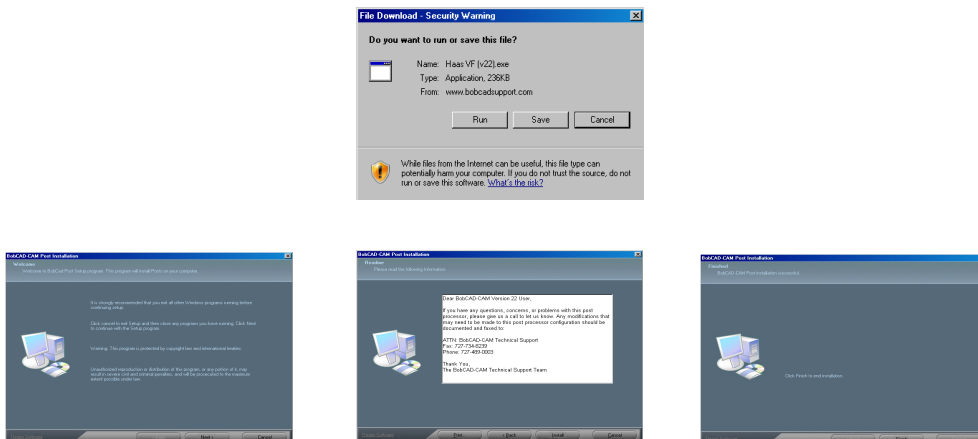
Lathe Post Editor
Installing and Editing Post-Processors

Installing a Post Processor

- Step 1** Go to the www.bobcad.com website.
- Step 2** Then, click on **Support** tab.
- Step 3** Under **Customer Resources**, click on **Post Processors (V22 MILL ONLY)** or **Post Processors (V22 LATHE ONLY)** depending on desired post.



- Step 4** Navigate to the desired **Post Processor** by clicking on the corresponding controller name.
- Step 5** Click on the desired **Post Processor .exe** file, and **Save** the file to your computer. Note that prior to the file extension is a date code to verify revision dates. Example: the **Haas VF (v22 011408).exe** was last modified on January 14 ,2008.
- Step 6** Once **Saved** to your computer, and with the **BobCAD-CAM** software closed; double-click on the file icon and **Run**, this will automatically launch the installation wizard. Click **Next**, **Install** and **Finish**.



Your Post Processor is now installed into the BobCAD-CAM Version 22 software.

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**

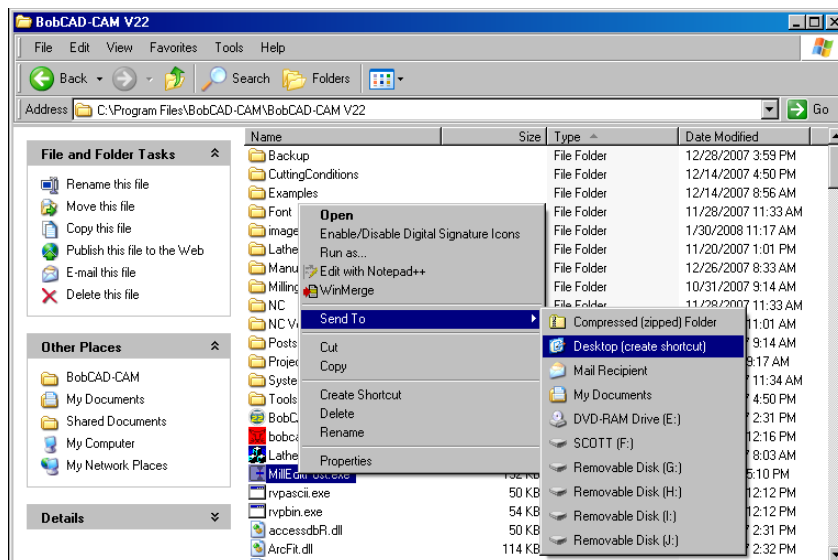
Create a copy of the post you desire to edit, and rename before editing.

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

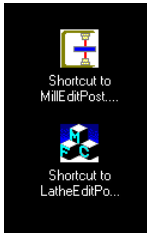
C:\Program Files\BobCAD-CAM\BobCAD-CAM V22\ MillEditPost.exe

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:

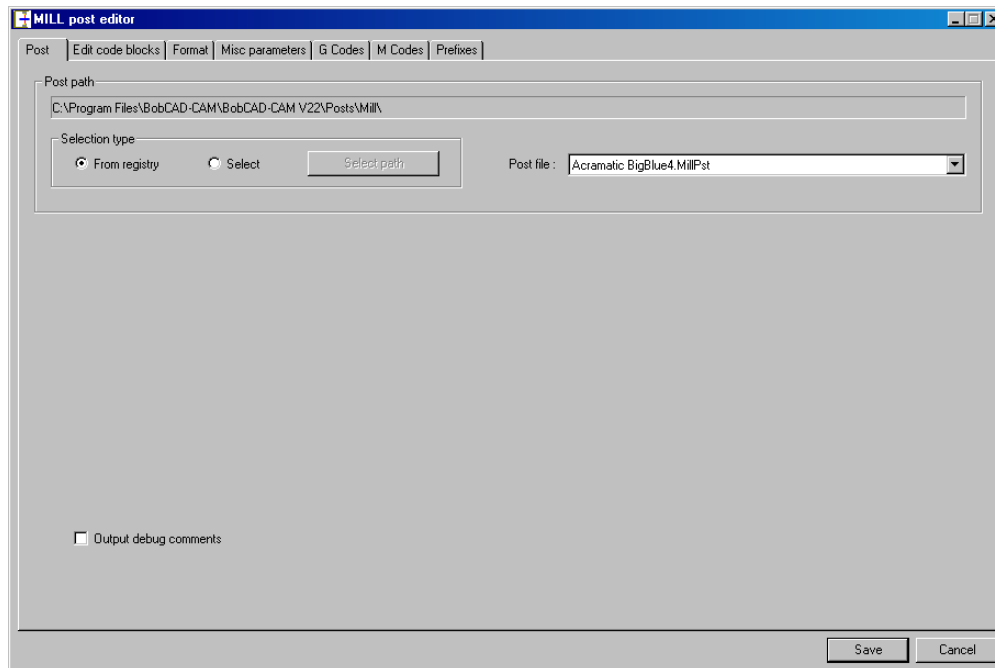


For editing all mill posts with the **.MillPst** extension

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

The following window will appear for the corresponding applications:

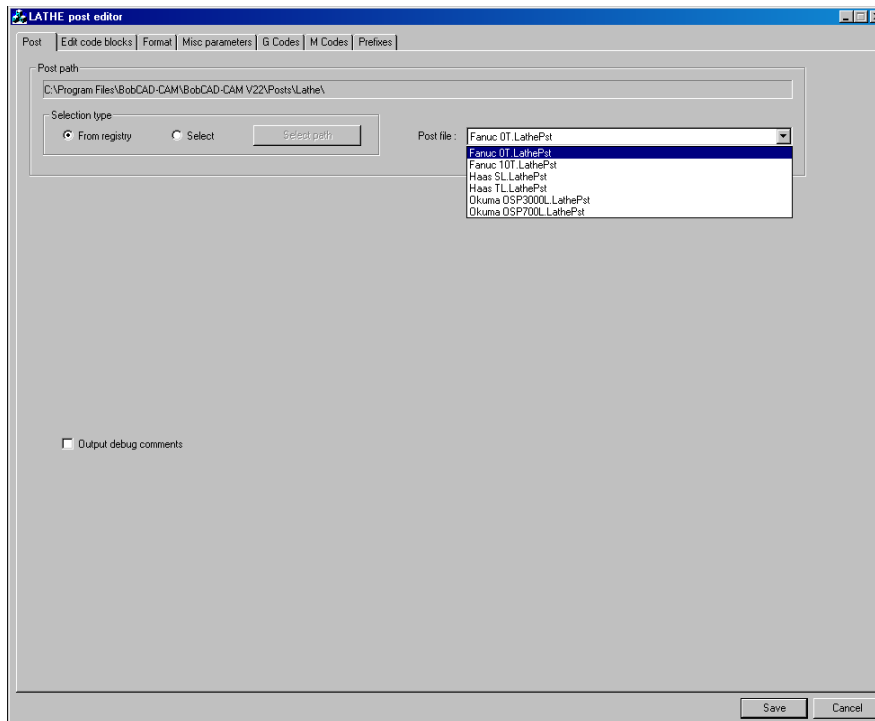
Mill Post Editor



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.

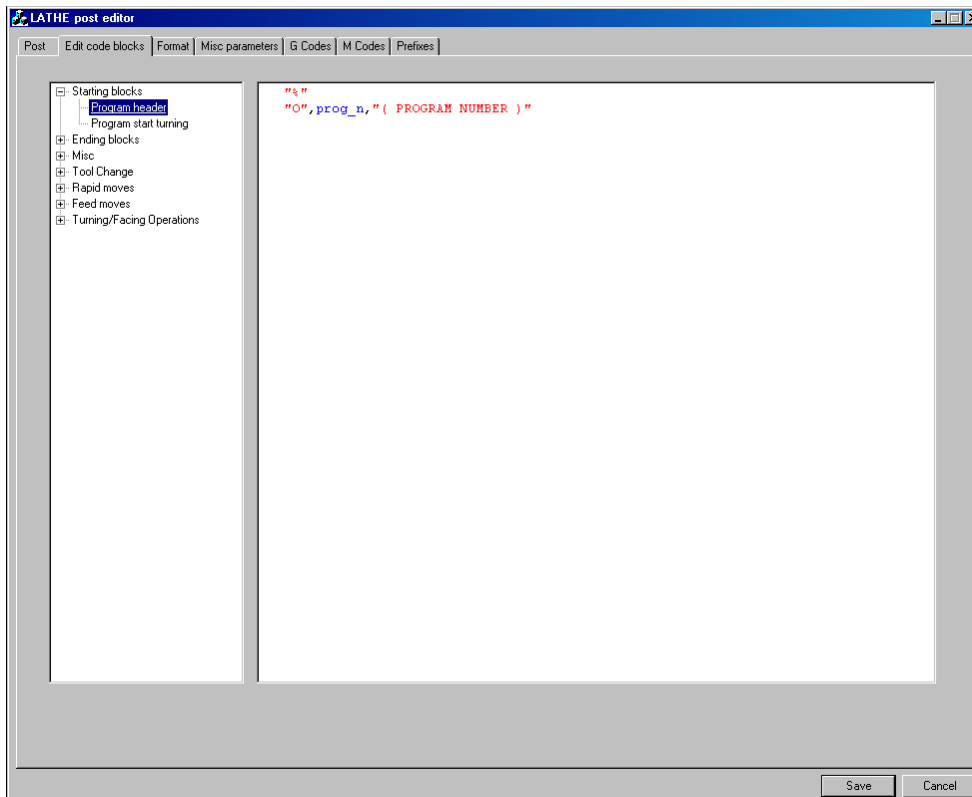
Lathe - Post Tab

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 **Fanuc 0T.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.



Lathe - 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**. In a future Build it will define the basic stock geometry and tool definitions for the **Backplot** feature within the **Predator CNC Editor – Level2** software. This information will be located prior to the hard-coded “%”, indicating the beginning of the **NC** program file, it will be disregarded by the controller. The current code blocks are written as follows:

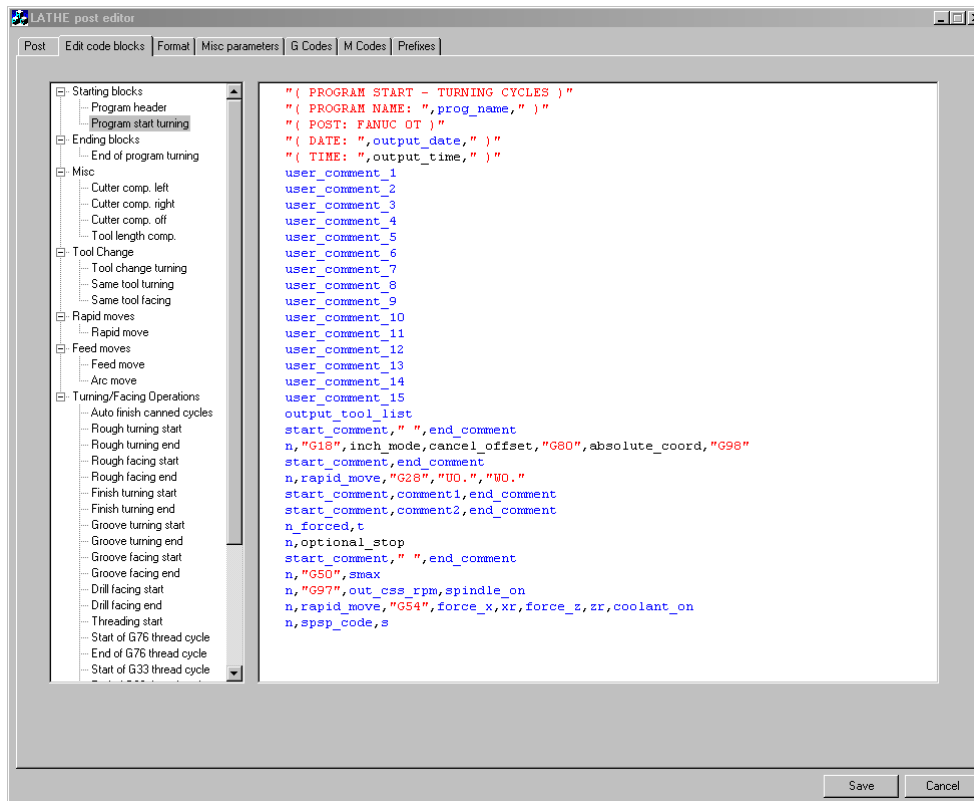
```
"%"
"O",prog_n," ( PROGRAM NUMBER )"
default_add_spaces
```

This command outputs nothing, only adds spaces to the program

The corresponding output will be posted as follows:

```
()          NC file start
%          Percent sign indicating start of NC file
O00001 ( PROGRAM NUMBER )  Program number as defined in the BobCAD software, no line number
```

Lathe - Edit code blocks tab – Starting blocks folder – Program start turning



The above example is information contained in the **Program start turning**. 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 START - TURNING CYCLES )"
"( PROGRAM NAME: ",prog_name," )"
"( POST: FANUC OT )"
"( 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
output_tool_list
start_comment," ",end_comment
n,"G18",inch_mode,cancel_offset,"G80",absolute_coord,"G98"
start_comment,end_comment
n,rapid_move,"G28","U0.", "W0."
start_comment,comment1,end_comment
start_comment,comment2,end_comment
n_forced,t
n,optional_stop
start_comment," ",end_comment
n,"G50",smax
n,"G97",out_css_rpm,spindle_on
n,rapid_move,"G54",force_x,xr,force_z,zr,coolant_on
n,spsp_code,s

```

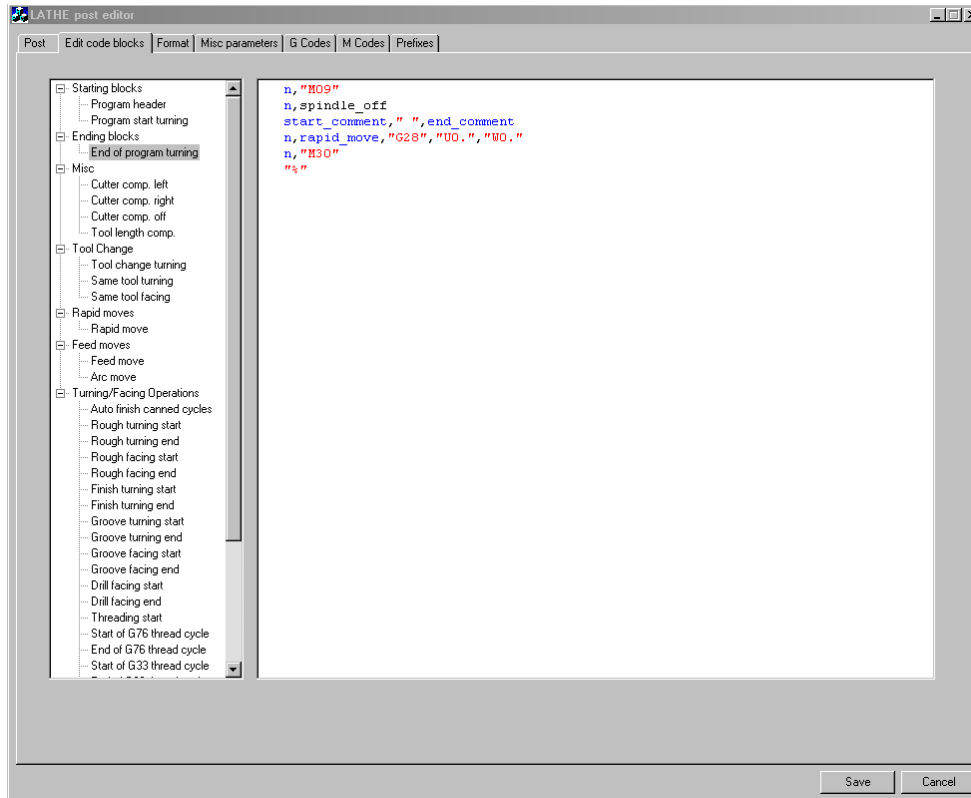
The user comment will only display if selected to output and the field contains text

The corresponding output will be posted as follows:

| | |
|---------------------------------------|--|
| (PROGRAM START - TURNING CYCLES) | Program start |
| (PROGRAM NAME: POST TEST.NC) | Program name is output |
| (POST: FANUC 0T) | Post name is output |
| (DATE: FRI. 02/01/2008) | Date output |
| (TIME: 03:29PM) | Time output |
| () | Start of safety line |
| N01 G18 G20 G40 G80 G90 G98 | Safety line g-code Variables and hard-code |
| () | Start of first tool change |
| N02 G00 G28 U0. W0. | Automatic return to reference point, location hard-coded |
| (JOB 0 ROUGH CYCLE) | System comment as generate by Feature in CAM Tree |
| (TOOL #1 80 DEG. 1/64 ROUGH TURNING) | Feature name comment, as define by selected tool in Feature |
| N03 T0101 | First tool change with offset |
| N04 M01 | Optional stop to inspect insert |
| () | Start of speeds and move to machining location |
| N05 G50 S3000 | Set maximum spindle speed |
| N06 G97 S454 M03 | RPM mode, speed and spindle on |
| N07 G54 X2. Z.1312 M08 | Work coordinate, move to location and coolant on |
| N08 G96 S500 | Change to CSS mode and speed |

Lathe - Edit code blocks tab – Ending blocks folder – End of program turning

The ***End of program turning*** 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 turning**. 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:

```
n, "M09"
n, spindle_off
start_comment, " ", end_comment
n, rapid_move, "G28", "U0.", "W0."
n, "M30"
"%"
```

The corresponding output will be posted as follows:

| | |
|------------------|--|
| N368 M09 | Turn coolant off |
| N369 M05 | Turn spindle off |
| () | Start of program end |
| N370 G28 U0. W0. | Automatic return to reference point, location hard-coded |
| N371 M30 | End of program |
| % | End of NC file |

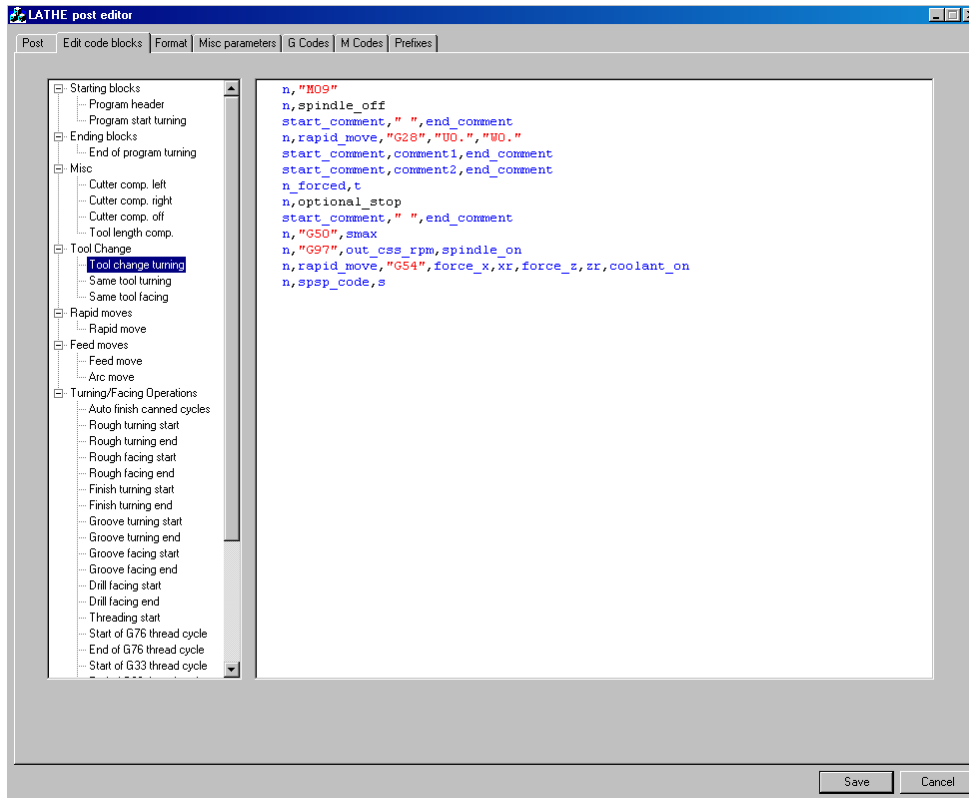
Lathe - Edit code blocks tab – Misc. folder

The **Misc.** folder contains the following fields for various code block definition, for calling within the post in other fields:

| | |
|---------------------------|--------------|
| Cutter comp. left | "G41" |
| Cutter comp. right | "G42" |
| Cutter comp. off | "G40" |
| Tool length comp. | "G43" |

Lathe - Edit code blocks tab – Tool Change folder – Tool change turning

The **Tool Change** folder contains the **Tool change turning**, the **Same tool turning** and the **Same tool facing** fields. These define how a tool change is to be executed in the NC file.



The above example is a code block definition contained in the **Tool change turning** 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 **Program start turning**. The code blocks are written as follows:

```
n,"M09"
n,spindle_off
start_comment," ",end_comment
n,rapid_move,"G28","U0. ","W0."
start_comment,comment1,end_comment
start_comment,comment2,end_comment
n_forced,t
n,optional_stop
start_comment," ",end_comment
n,"G50",smax
n,"G97",out_css_rpm,spindle_on
n,rapid_move,"G54",force_x,xr,force_z,zr,coolant_on
n,spsp_code,s
```

The corresponding output will be posted as follows:

| | |
|----------------------------|--|
| N209 M09 | Coolant off, defined in M Codes tab |
| N210 M05 | Spindle off, defined in M Codes tab |
| () | Start of tool change |
| N211 G00 G28 U0. W0. | Automatic return to reference point |
| (JOB 0 GROOVE CYCLE) | System comment as generate by Feature in CAM Tree |
| (TOOL #19 1/8 FACE ANVIL) | Feature name comment, as define by selected tool in Feature |
| N212 T1919 | Tool change with offset |
| N213 M01 | Optional stop to inspect insert |

| | |
|-------------------------|--|
| () | Start of speeds and move to machining location |
| N214 G50 S3000 | Set maximum spindle speed |
| N215 G97 S2750 M03 | RPM mode, speed and spindle on |
| N216 G54 X.3125 Z0. M08 | Work coordinate, move to location and coolant on |
| N217 G96 S450 | Change to CSS mode and speed |

Lathe - Edit code blocks tab – Tool Change folder – Same tool turning / grooving

Since the lathe philosophy and theory are different than mill, as far as the NC program layout, each cut is treated much like a program unto itself. As such, the **Same tool turning** and the **Same tool facing** fields are the same as the **Tool change turning**. This may NOT be the case for all machine tool controllers (post processors), but this applies to the current example. No fields are shown.

Lathe - 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:

Rapid move z `n,rapid_move,xr,zr`

Lathe - Edit code blocks tab – Feed moves folder

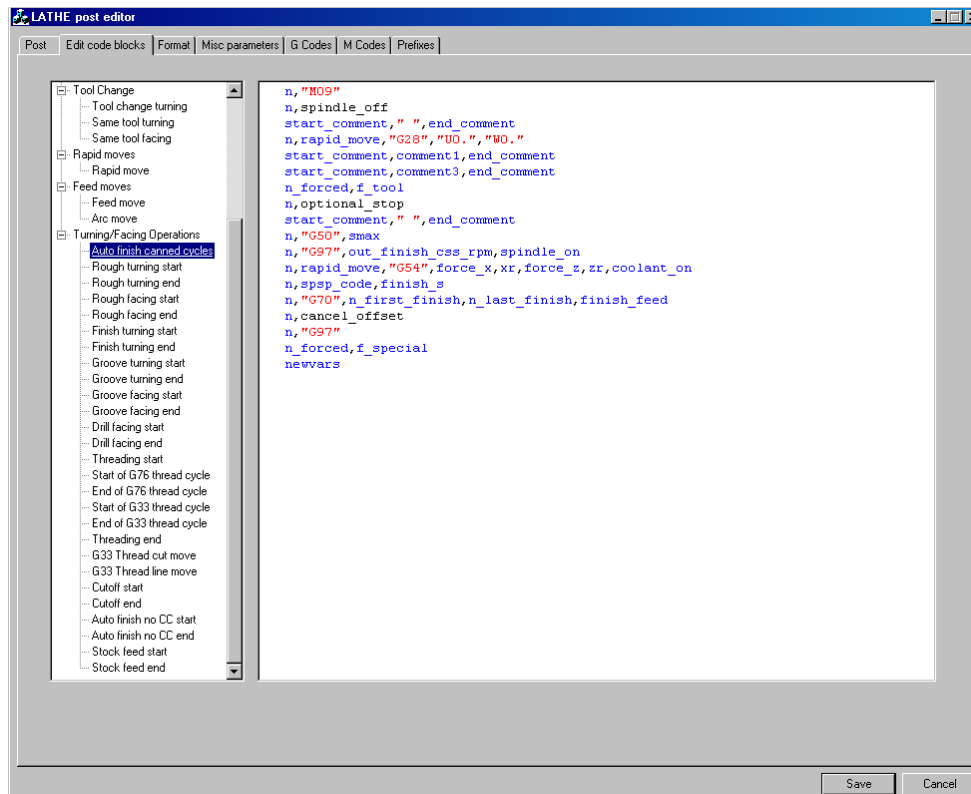
The **Feed moves** folder contains all code block definitions for rapid moves within the **NC** file. It contains the following fields:

Feed Move `n,cc,feed_move,x_f,z_f,f`
 Arc move `n,g_arc_move,x_f,z_f,arc_center,f`

Lathe - Edit code blocks tab – Turning/Facing Operations

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 **Turning/Facing Operations** folder contains all the fields for defining the code blocks for each canned cycle start and the independent termination of each cycle. It contains the following fields:

Turning/Facing Operations - Auto finish canned cycles (G70)

```
n, "M09"
n, spindle_off
start_comment, " ", end_comment
n, rapid_move, "G28", "U0.", "W0."
start_comment, comment1, end_comment
start_comment, comment3, end_comment
n_forced, f_tool
n, optional_stop
start_comment, " ", end_comment
n, "G50", smax
n, "G97", out_finish_css_rpm, spindle_on
n, rapid_move, "G54", force_x, xr, force_z, zr, coolant_on
n, spsp_code, finish_s
n, "G70", n_first_finish, n_last_finish, finish_feed
n, cancel_offset
n, "G97"
n_forced, f_special
newvars
```

The corresponding output will be posted as follows (when selected in the Feature):

| | |
|--|--|
| N24 M09 | Coolant off, defined in M Codes tab |
| N25 M05 | Spindle off, defined in M Codes tab |
| () | Start of tool change |
| N26 G00 G28 U0. W0. | Automatic return to reference point |
| (JOB 0 ROUGH CYCLE) | System comment as generate by Feature in CAM Tree |
| (TOOL #2 80 DEG. 1/64 FINISH TURNING) | Feature name comment, as define by BobCAD and selected tool in Feature |
| N27 T0202 | Tool change with offset |
| N28 M01 | Optional stop to inspect insert |
| () | Start of speeds and move to machining location |
| N29 G50 S3000 | Set maximum spindle speed |
| N30 G97 S429 M03 | RPM mode, speed and spindle on |
| N31 G54 X-.0156 Z0. M08 | Work coordinate, move to location and coolant on |
| N32 G96 S450 | Change to CSS mode and speed |
| N33 G70 P11 Q20 F.015 | Executes G70 Finish Cycle for line 11 through 20 in previous cycle |
| N34 G40 | Cancel offset |
| N35 G97 | Cancel CSS mode, return to RPM |
| N36 T0200 | Cancel tool offset |

Turning/Facing Operations - Rough turning start (G71)

```
n, "G71", roughing_depth_of_cut_x, rough_retract_amount
n, "G71", n_first_rough, n_last_rough, rough_x_allowance, rough_z_allowance, rough_feed
```

The corresponding output will be posted as follows (when selected in the Feature):

| | |
|---|--|
| N09 G71 U.1 R.1 | G71 Rough turning cycle – Two line format |
| N10 G71 P11 Q20 U.05 W.05 F15. | Executes G71 Rough turning cycle for |
| N11 G00 X-.0156 | line 11 through 20 with established parameters |
| N12 G42 G01 X.7834 | |
| N13 G03 X.9153 Z-.0919 I-.1406 K0. | |
| N14 G01 X1. Z-.3209 | |
| N15 Z-.7656 | |
| N16 G02 X1.1977 Z-.9971 I0. K.2344 | |
| N17 G01 X1.256 Z-1.0063 | |
| N18 G03 X1.6365 Z-1.285 I-.5093 K-.0806 | |
| N19 G01 X2. Z-2.0119 | |
| N20 Z-2.7656 | |

Turning/Facing Operations - Rough turning end

```
n, cancel_offset  
n, "G97"  
n_forced, special
```

The corresponding output will be posted as follows (for all):

```
N21 G40          Cancel offset  
N22 G97          Cancel CSS mode, return to RPM  
N23 T0100        Cancel tool offset
```

Turning/Facing Operations - Rough facing start (G72)

```
n, "G72", roughing_depth_of_cut_z, rough_retract_amount  
n, "G72", n_first_rough, n_last_rough, rough_x_allowance, rough_z_allowance, rough_feed
```

The corresponding output will be posted as follows (when selected in the Feature):

```
N09 G72 W.1 R.1          G72 Rough facing cycle – Two line format  
N10 G72 P11 Q20 U.05 W.05 F15. Executes G72 Rough facing cycle for  
N11 G00 X-.0156          line 11 through 20 with established parameters  
N12 G42 G01 X.7834  
N13 G03 X.9153 Z-.0919 I-.1406 K0.  
N14 G01 X1. Z-.3209  
N15 Z-.7656  
N16 G02 X1.1977 Z-.9971 I0. K.2344  
N17 G01 X1.256 Z-1.0063  
N18 G03 X1.6365 Z-1.285 I-.5093 K-.0806  
N19 G01 X2. Z-2.0119  
N20 Z-2.7656
```

Turning/Facing Operations - Rough facing end

```
n, cancel_offset  
n, "G97"  
n_forced, special
```

The corresponding output will be posted as follows (for all):

```
N21 G40          Cancel offset  
N22 G97          Cancel CSS mode, return to RPM  
N23 T0100        Cancel tool offset
```

Turning/Facing Operations - Finish turning start (G73)

```
n, "G73", finish_x_spring_pass_stock_radius, finish_z_spring_pass_stock_radius  
          , finish_number_of_cuts_two_line  
n, "G73", n_first_rough, n_last_rough, finish_x_allowance, finish_z_allowance  
          , rough_feed
```

The corresponding output will be posted as follows (when selected in the Feature):

```
N33 G73 U.04 W.04 R5          G73 Finish turning cycle – Two line format  
N34 G73 P35 Q45 U0. W0. F5. Executes G73 Rough turning cycle for  
N35 G00 X-.0156          line 35 through 45 with established parameters  
N36 G42 G01 X.7834  
N37 G03 X.9153 Z-.0919 I-.1406 K0.  
N38 G01 X1. Z-.3209  
N39 Z-.7656  
N40 G02 X1.1977 Z-.9971 I0. K.2344  
N41 G01 X1.256 Z-1.0063  
N42 G03 X1.6365 Z-1.285 I-.5093 K-.0806  
N43 G01 X2. Z-2.0119  
N44 Z-2.7656  
N45 G00 Z0.
```

Turning/Facing Operations - Finish turning end

```
n,cancel_offset  
n,"G97"  
n_forced,special
```

The corresponding output will be posted as follows (for all):

```
N46 G40          Cancel offset  
N47 G97          Cancel CSS mode, return to RPM  
N48 T1000        Cancel tool offset
```

Turning/Facing Operations - Groove turning start (G75)

```
n,"G75",groove_retract_amount  
n,"G75",groove_x_bottom,groove_z_bottom,groove_peck_increment,groove_depth_of_cut  
      ,groove_rvalue,rough_feed
```

The corresponding output will be posted as follows (when selected in the Feature):

```
N72 G75 R.1      G75 Groove turning cycle – Two line format  
N73 G75 X5. Z-2.5838 I0. K.05 R0. F12.  Executes G75 Groove turning cycle
```

Turning/Facing Operations - Groove turning end

```
n,cancel_offset  
n,"G97"  
n_forced,special
```

The corresponding output will be posted as follows (for all):

```
N74 G40          Cancel offset  
N75 G97          Cancel CSS mode, return to RPM  
N76 T2200        Cancel tool offset
```

Turning/Facing Operations - Groove facing start (G74)

```
n,"G74",groove_retract_amount  
n,"G74",groove_x_bottom,groove_z_bottom,groove_peck_increment,groove_depth_of_cut  
      ,groove_rvalue,rough_feed
```

The corresponding output will be posted as follows (when selected in the Feature):

```
N58 G74 R.1      G74 Groove facing cycle – Two line format  
N59 G74 X.5625 Z5. I.025 K.025 R0. F10.  Executes G74 Groove facing cycle
```

Turning/Facing Operations - Groove facing end

```
n,cancel_offset  
n,"G97"  
n_forced,special
```

The corresponding output will be posted as follows (for all):

```
N60 G40          Cancel offset  
N61 G97          Cancel CSS mode, return to RPM  
N62 T1900        Cancel tool offset
```

Turning/Facing Operations - Drill facing start (G74)

```
n, "G74", groove_retract_amount  
n, "G74", groove_x_bottom, groove_z_bottom, groove_peck_increment, groove_depth_of_cut  
    , groove_rvalue, rough_feed
```

The corresponding output will be posted as follows (when selected in the Feature):

```
N103 G74 R.1  
N104 G74 X0. Z-1. I0. K.25 R0. F12.           G74 Drill facing cycle – Two line format  
                                           Executes G74 Drill facing cycle
```

Turning/Facing Operations - Drill facing end

```
n, "G97"  
n_forced, special
```

The corresponding output will be posted as follows (for all):

```
N105 G97  
N106 T5200                                Cancel CSS mode, return to RPM  
                                           Cancel tool offset
```

Turning/Facing Operations - Threading start

In this post processor case the field is blank.

Turning/Facing Operations - Start of G76 thread cycle

```
n, thread_angle_out_start  
n, "G76", thread_angle_in, thread_last_cut, thread_machine_allowance  
n, "G76", thread_x2, thread_z2, thread_height, thread_first_cut, thread_lead  
n, thread_angle_out_end
```

The corresponding output will be posted as follows (when selected in the Feature):

```
N86 G01 X1.6382 F.0714                     Start of thread  
N87 M23                                     Chamfer on, selected in Feature, from M Codes tab  
N88 G76 P010060 Q.03 R.001                 G76 Complex thread cycle – Two line format  
N89 G76 X1.5763 Z-2. K2999 D0500 F.071429 Executes Complex thread cycle with established parameters  
N90 M24                                     Chamfer off selected in Feature, from M Codes tab  
N91 G00 X5.                                Rapid to location X  
N92 Z5.                                    Rapid to location Z
```

Turning/Facing Operations - End of G76 thread cycle

In this post processor case the field is blank.

Turning/Facing Operations - Start of G33 thread cycle

In this post processor case the field is blank.

Turning/Facing Operations - End of G33 thread cycle

In this post processor case the field is blank.

Turning/Facing Operations - Threading end

```
n, cancel_offset  
n, "G97"  
n_forced, special
```

The corresponding output will be posted as follows (for all):

```
N93 G40          Cancel offset  
N94 G97          Cancel CSS mode, return to RPM  
N95 T0400        Cancel tool offset
```

Turning/Facing Operations - G33 Thread cut move

```
n, thread_g33_gcode, thread_g33_xmove, thread_g33_zmove, thread_g33_feedrate
```

The corresponding output will be posted as follows (when selected in the Feature):

```
N88G33 X1.9579 Z-2. F.0714      Start of G33 Simple thread cut move  
N89 X2.0619  
N90 X1.6522 Z-1.1807  
N91 X1.5285
```

Turning/Facing Operations - G33 Thread line move

```
n, thread_g33_gcode, thread_g33_xmove, thread_g33_zmove, thread_g33_feedrate
```

The corresponding output will be posted as follows (when selected in the Feature):

```
N92G33 X1.9381 Z-2. F.0714      Start of G33 Simple thread line move  
N93 X2.0619  
N94 G00 X5.  
N95 Z5.
```

Turning/Facing Operations - Cutoff start

In this post processor case the field is blank. The corresponding output will be posted as follows (for all, depending on Feature parameters selection):

```
N136 G42 G01 X1.9 F15.          Feed into cutoff location  
N137 G00 X2.1                   Retract to X clearance  
N138 Z-3.025                    Rapid to Z location  
N139 G01 X2.                     Feed to X location  
N140 G03 X1.9 Z-3.125 I0. K-.1  Fillet Corner break selected in Feature  
N141 G01 X-.01                  Feed past center  
N142 G00 X5.                     Rapid to X location  
N143 Z5.                         Rapid to Z location
```

Turning/Facing Operations - Cutoff end

```
n, cancel_offset  
n, "G97"  
n_forced, special  
n, "M09"  
n, spindle_off
```

The corresponding output will be posted as follows (for all):

| | |
|------------|---|
| N144 G40 | Cancel offset |
| N145 G97 | Cancel CSS mode, return to RPM |
| N146 T0800 | Cancel tool offset |
| N147 M09 | Spindle off , as defined in M Codes tab |
| N148 M05 | Spindle off , as defined in M Codes tab |

Turning/Facing Operations - Auto finish no CC start

In this post processor case the field is blank.

Turning/Facing Operations - Auto finish no CC end

```
n,cancel_offset  
n,"G97"  
n,special
```

The corresponding output will be posted as follows (for all):

| | |
|------------|--|
| N154 G40 | Cancel offset |
| N155 G97 | Cancel CSS mode, return to RPM |
| N156 T0100 | Cancel tool offset |

Turning/Facing Operations - Stock feed start

```
start_comment," ",end_comment  
"( STOCK FEED )"  
start_comment," ",end_comment  
rapid_to_pickup_z  
rapid_to_position_x  
n,"M11"  
n,"G04","X.5"  
rapid_to_stock_feed_z  
n,"G04","X.5"  
n,"M10"  
n,"G04","X.5"  
rapid_to_position_z
```

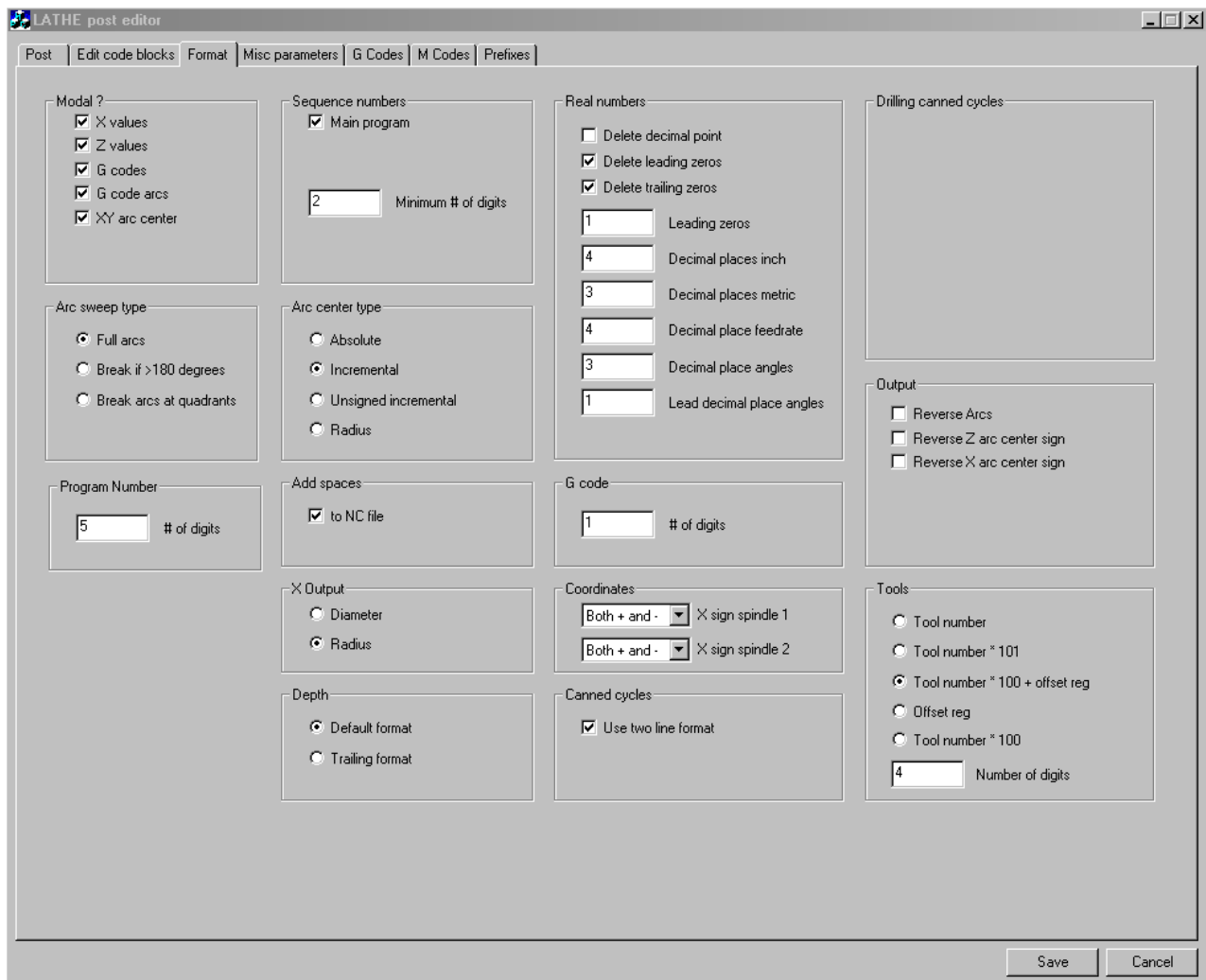
The corresponding output will be posted as follows (for all):

| | |
|----------------|--|
| () | |
| (STOCK FEED) | Stock feed start |
| () | |
| N149 Z0. | Rapid to Z pick up |
| N150 X2. | Rapid to X position |
| N151 M11 | Clamp off |
| N152 G04 X.5 | Dwell for 0.5 seconds |
| N153 Z-3.25 | Rapid to stock feed Z location |
| N154 G04 X.5 | Dwell for 0.5 seconds |
| N155 M10 | Clamp on |
| N156 G04 X.5 | Dwell for 0.5 seconds |
| N157 Z3. | Rapid to Z position |

The following tabs in the **Lathe 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.

Lathe – 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 (blank), amount of **Arc sweep type**, **Arc center type**, **# of digits for Program Numbers**, **Add spaces to NC file**, **# of digits for G code**, **Output** for reversing signs, **X Output** for **Radius** or **Diameter** mode, **Coordinates** sign, **Depth** format, **Canned cycles** type and **Tool** numbering and offset definitions.



Lathe – 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, **Tools** for defining tool number and offset output and **Two line canned cycles** retract and allowances.

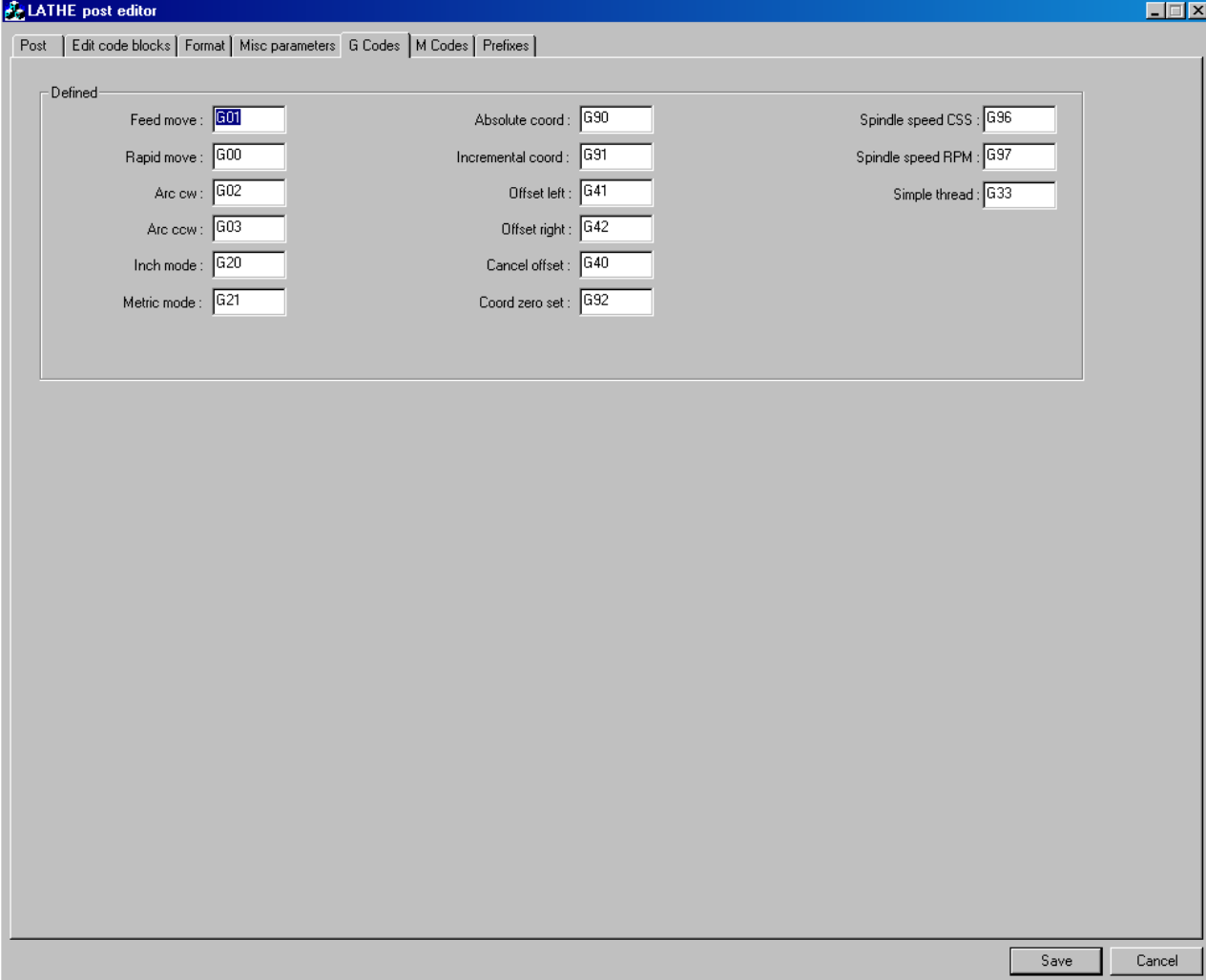
The screenshot shows the 'LATHE post editor' window with the 'Misc parameters' tab selected. The window has a menu bar with 'Post', 'Edit code blocks', 'Format', 'Misc parameters', 'G Codes', 'M Codes', and 'Prefixes'. The main area is divided into four sections, each with a title and several input fields:

- Misc strings:** Contains 'Comment start:' with a text box containing '(', 'Comment end:' with a text box containing ')', and 'Block delete:' with a text box containing '/'. There is also an empty text box to the right of the 'Block delete:' label.
- Cutting conditions:** Contains 'Scale factor for feedrate:' with a text box containing '1', 'Medium range spindle speed:' with a text box containing '360', and 'High range spindle speed:' with a text box containing '780'.
- Tools:** Contains 'Add amount from T to T1:' with a text box containing '0', 'Value of T1 with T = 0:' with a text box containing '0', 'Add amount from T to T2:' with a text box containing '0', and 'Value of T2 with T = 0:' with a text box containing '0'.
- Two line canned cycles:** Contains 'Rough retract:' with a text box containing '0.10000', 'Groove retract:' with a text box containing '0.10000', and 'Thread machine allowance:' with a text box containing '0.00100'.

At the bottom right of the window, there are 'Save' and 'Cancel' buttons.

Lathe – 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.



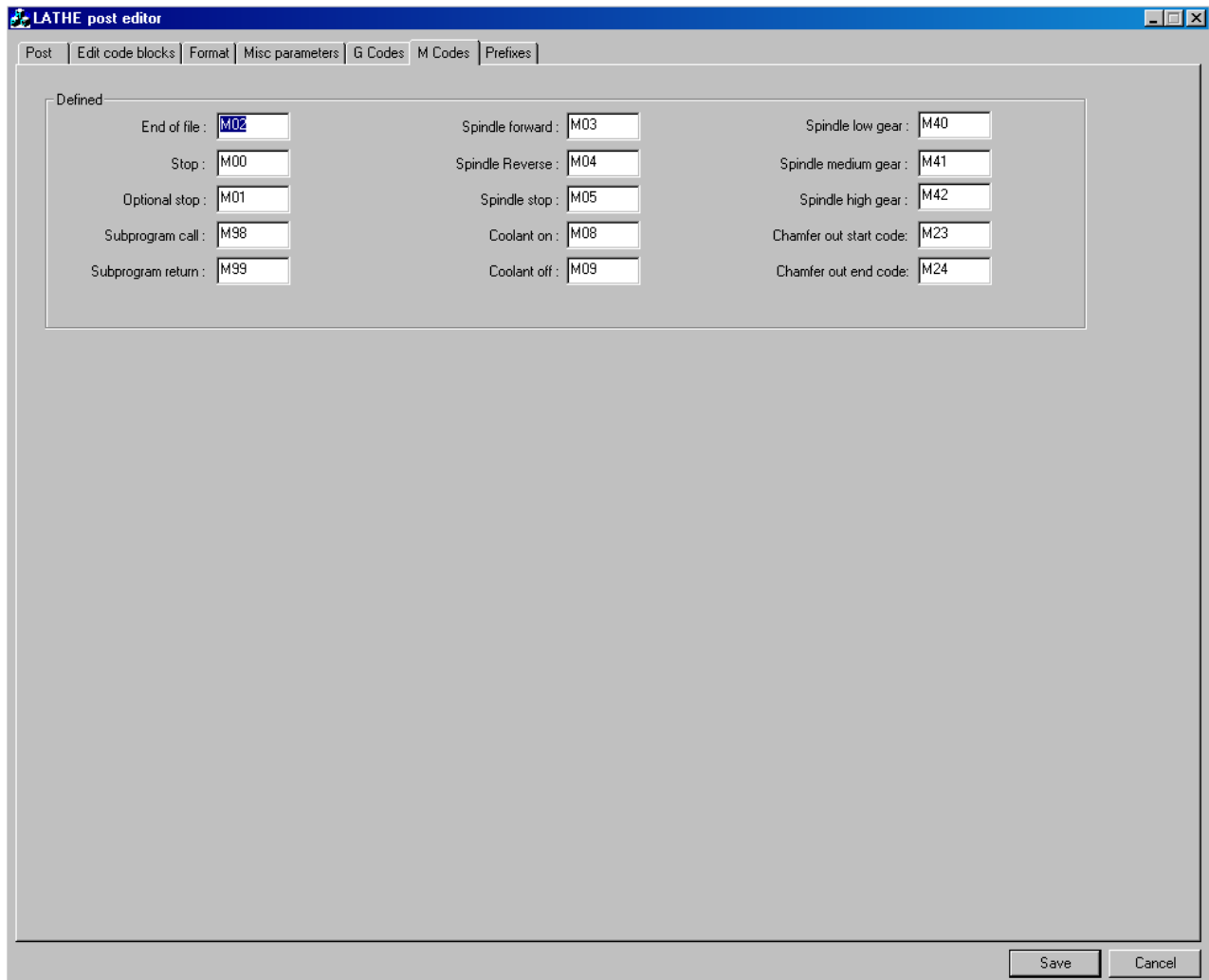
The screenshot shows the 'LATHE post editor' window with the 'G Codes' tab selected. The 'Defined' section contains the following fields:

| Variable Name | Value |
|-------------------|-------|
| Feed move | G01 |
| Rapid move | G00 |
| Arc cw | G02 |
| Arc ccw | G03 |
| Inch mode | G20 |
| Metric mode | G21 |
| Absolute coord | G90 |
| Incremental coord | G91 |
| Offset left | G41 |
| Offset right | G42 |
| Cancel offset | G40 |
| Coord zero set | G92 |
| Spindle speed CSS | G96 |
| Spindle speed RPM | G97 |
| Simple thread | G33 |

Buttons: Save, Cancel

Lathe – 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 'LATHE post editor' window with the 'M Codes' tab selected. The 'Defined' section contains the following variables and their values:

| Variable | Value |
|------------------------|-------|
| End of file | M02 |
| Stop | M00 |
| Optional stop | M01 |
| Subprogram call | M98 |
| Subprogram return | M99 |
| Spindle forward | M03 |
| Spindle Reverse | M04 |
| Spindle stop | M05 |
| Coolant on | M08 |
| Coolant off | M09 |
| Spindle low gear | M40 |
| Spindle medium gear | M41 |
| Spindle high gear | M42 |
| Chamfer out start code | M23 |
| Chamfer out end code | M24 |

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

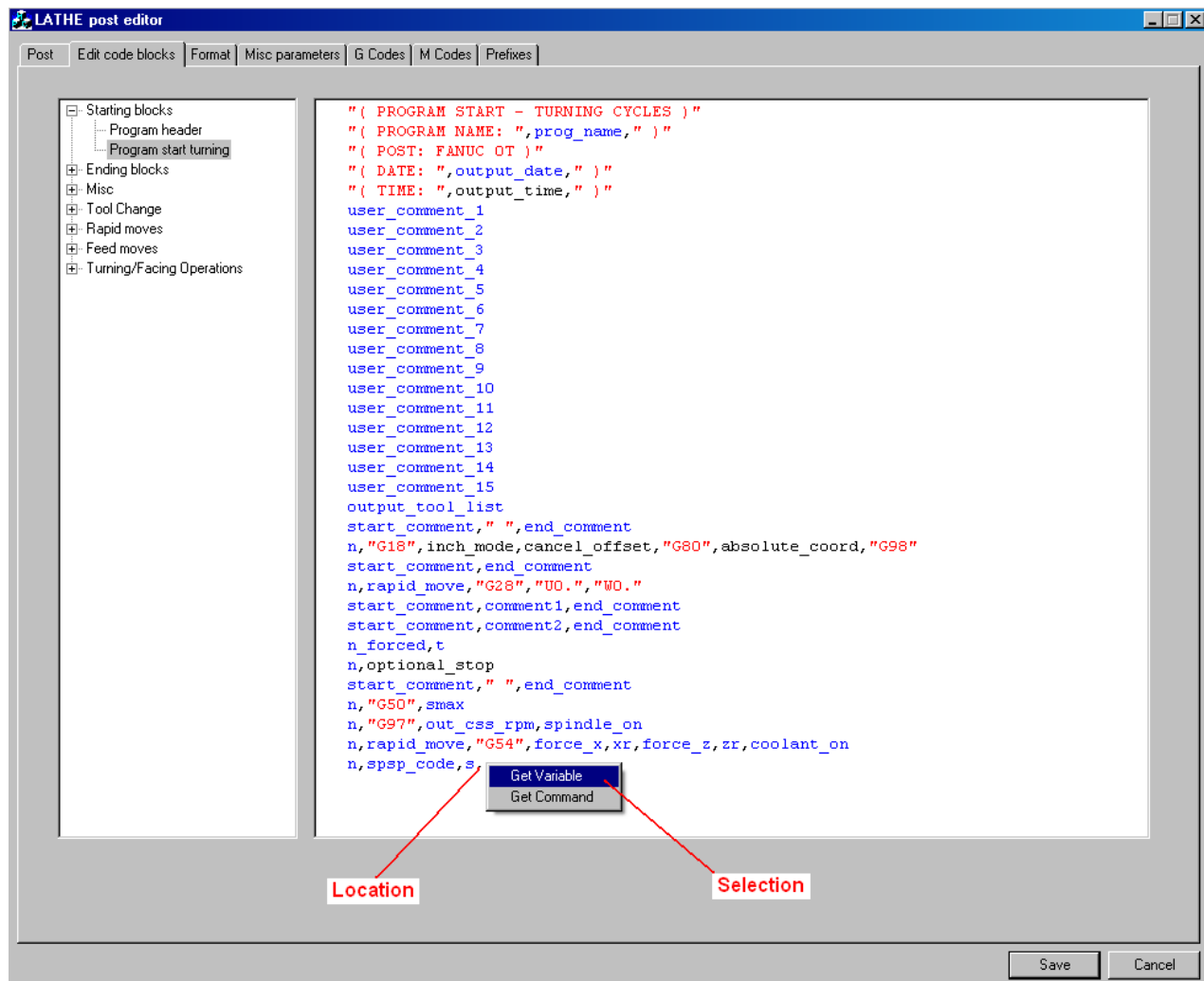
Lathe – Prefixes tab

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

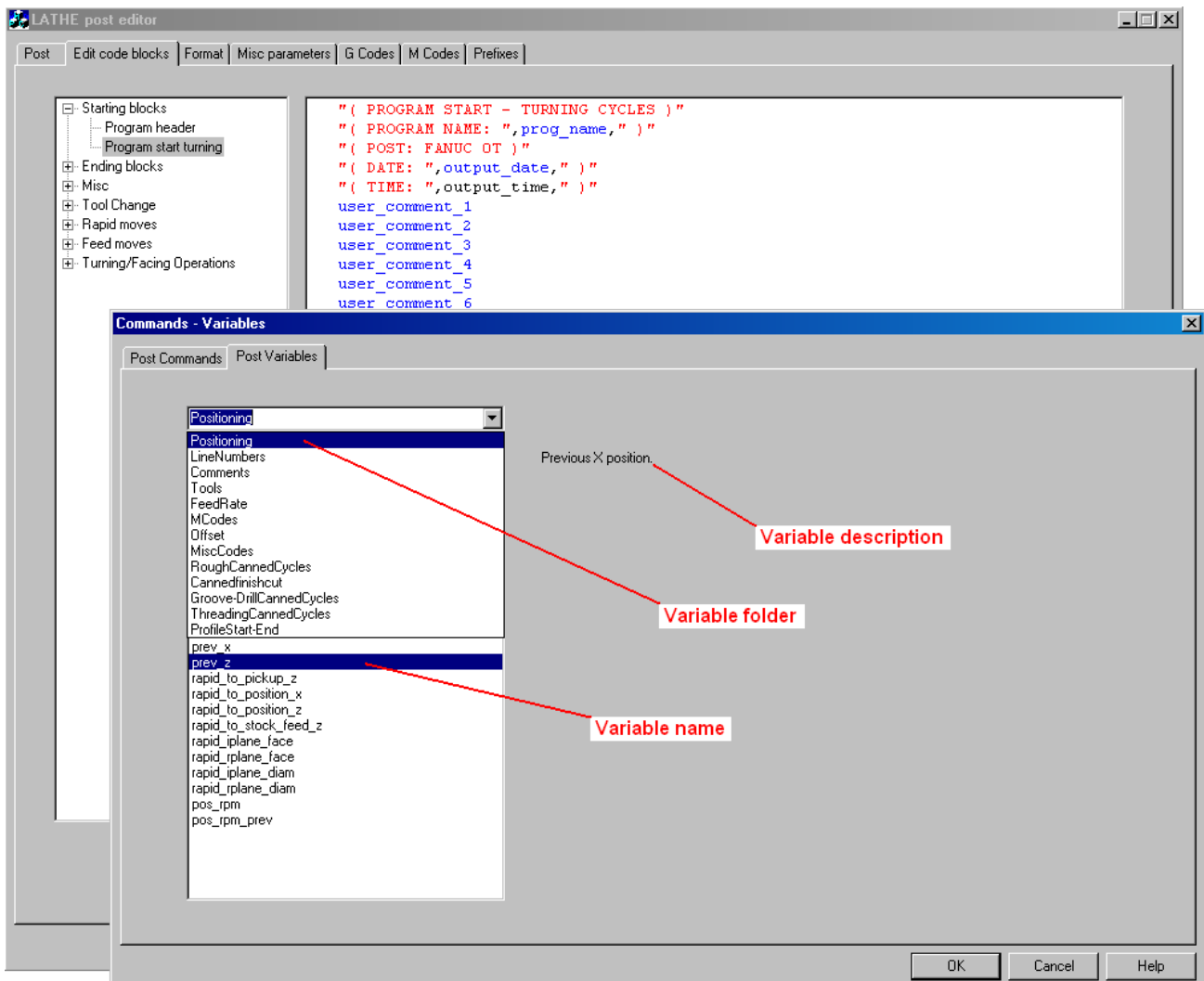
| Defined | | |
|-------------------|------------------------------|---|
| Spindle speed : S | Canned cycle start line #: P | Groove cycle peck: I |
| Feed rate IPM : F | Canned cycle end line #: Q | Rough cycle depth: D |
| Feed rate IPR: F | Thread first cut amount: D | Rough cycle stock allowance X: U |
| Thread lead: F | Thread Height: K | Rough cycle stock allowance Z: W |
| Tool number: T | Thread Angle: P | Rough cycle retract: R |
| Radius value: R | Thread last cut amount: Q | Finish cycle stock allowance X: U |
| Arc Z center: I | Thread machine: R | Finish cycle stock allowance Z: W |
| Arc X center: K | Groove cycle retract: R | Finish cycle stock for spring passes X: U |
| X move: X | Groove cycle X bottom: X | Finish cycle stock for spring passes Z: W |
| Y move: Y | Groove cycle X bottom: Z | Rough cycle X depth: U |
| Z move: Z | Groove cycle depth: K | Rough cycle Z depth: W |

Lathe – 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**.



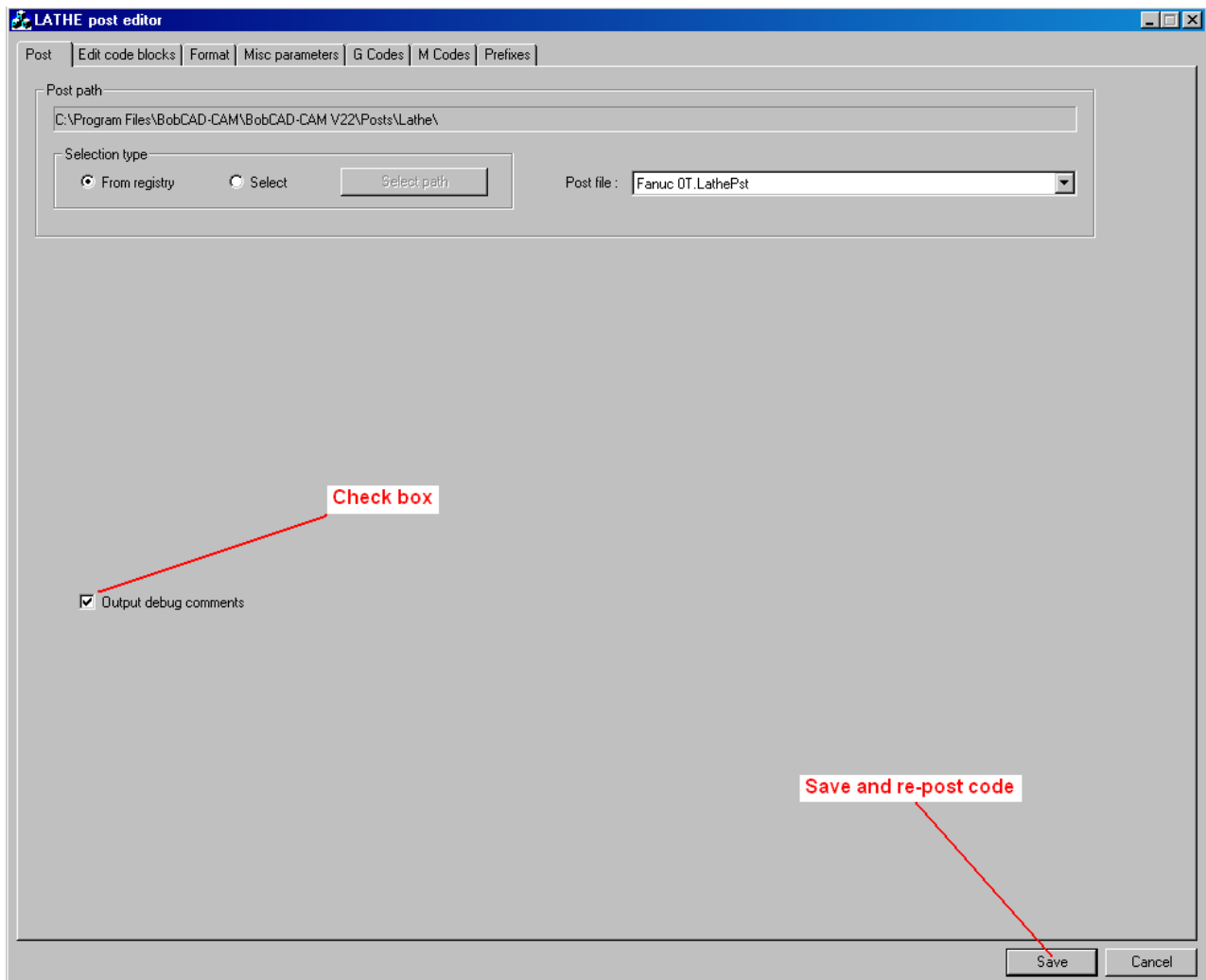
A new window called, **Commands – Variables** will display. You may change selections by clicking on the corresponding **Post Commands** or **Post Variables** tab. Select the desired folder, by click on the on the pull-down menu and then double-clicking on the variable name. You may also high-light the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

Lathe – 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).

```
*****0 - Starting blocks//Program header *****
%
O00001 ( PROGRAM NUMBER )
*****2 - Starting blocks//Program start turning *****
( PROGRAM START - TURNING CYCLES )
( PROGRAM NAME: POST TEST.NC )
( POST: FANUC 0T )
( DATE: MON. 02/04/2008 )
( TIME: 03:45PM )
( )
N01 G18 G20 G40 G80 G90 G98
( )
N02 G00 G28 U0. W0.
(JOB 0 ROUGH CYCLE )
(TOOL #1 80 DEG. 1/64 ROUGH TURNING )
N03 T0101
N04 M01
( )
N05 G50 S3000
N06 G97 S477 M03
N07 G54 X2. Z0. M08
N08 G96 S500
*****20 - Rapid moves//Rapid move *****
*****62 - Turning//Facing Operations/Rough turning start *****
N09 G71 U.1 R.1
N10 G71 P11 Q20 U.05 W.05 F15.
*****20 - Rapid moves//Rapid move *****
N11 G00 X-.0156
*****21 - Feed moves//Feed move *****
N12 G42 G01 X.7834
*****25 - Feed moves//Arc move *****
N13 G03 X.9153 Z-.0919 I-.1406 K0.
*****21 - Feed moves//Feed move *****
N14 G01 X1. Z-.3209
*****21 - Feed moves//Feed move *****
N15 Z-.7656
*****25 - Feed moves//Arc move *****
N16 G02 X1.1977 Z-.9971 I0. K.2344
*****21 - Feed moves//Feed move *****
N17 G01 X1.256 Z-1.0063
*****25 - Feed moves//Arc move *****
N18 G03 X1.6365 Z-1.285 I-.5093 K-.0806
*****21 - Feed moves//Feed move *****
N19 G01 X2. Z-2.0119
*****21 - Feed moves//Feed move *****
N20 Z-2.7656
*****63 - Turning//Facing Operations/Rough turning end *****
N21 G40
N22 G97
N23 T0100
*****60 - Turning//Facing Operations/Auto finish canned cycles *****
N24 M09
N25 M05
( )
N26 G00 G28 U0. W0.
(JOB 0 ROUGH CYCLE )
(TOOL #2 80 DEG. 1/64 FINISH TURNING )
N27 T0201
N28 M01
( )
N29 G50 S3000
N30 G97 S429 M03
N31 G54 X-.0156 Z0. M08
N32 G96 S450
```

```

N33 G70 P11 Q20 F.015
N34 G40
N35 G97
N36 T0200
*****5 - Ending blocks//End of program turning *****
N37 M09
N38 M05
( )
N39 G00 G28 U0. W0.
N40 M30
%
```

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.

```

%                                "% "
O00001 ( PROGRAM NUMBER )       "O",prog_n,"( PROGRAM NUMBER )"
( PROGRAM START - TURNING CYCLES ) "( PROGRAM START - TURNING CYCLES )"
( PROGRAM NAME: POST TEST.NC )  "( PROGRAM NAME: ",prog_name," )"
( POST: FANUC 0T )              "( POST: FANUC 0T )"
( DATE: MON. 02/04/2008 )       "( DATE: ",output_date," )"
( TIME: 03:49PM )               "( TIME: ",output_time," )"
( )                               start_comment," ",end_comment
N01 G18 G20 G40 G80 G90 G98     n,"G18",inch_mode,cancel_offset,"G80",absolute_coord
                                ,"G98"
( )                               start_comment," ",end_comment
N02 G00 G28 U0. W0.             n,rapid_move,"G28","U0.", "W0."
(JOB 0 ROUGH CYCLE )           start_comment,comment1,end_comment
(TOOL #1 80 DEG. 1/64 ROUGH TURNING ) start_comment,comment2,end_comment
N03 T0101                       n_forced,t
N04 M01                          n,optional_stop
( )                               start_comment," ",end_comment
N05 G50 S3000                   n,"G50",smax
N06 G97 S477 M03                 n,"G97",out_css_rpm,spindle_on
N07 G54 X2. Z0. M08              n,rapid_move,"G54",force_x,xr,force_z,zr,coolant_on
N08 G96 S500                     n,spsp_code,s
N09 G71 U.1 R.1                 n,"G71",roughing_depth_of_cut_x,rough_retract_amount
N10 G71 P11 Q20 U.05 W.05 F15.   n,"G71",n_first_rough,n_last_rough,rough_x_allowance
                                ,rough_z_allowance,rough_feed
N11 G00 X-.0156                 n,rapid_move,xr,zr
N12 G42 G01 X.7834              n,cc,feed_move,x_f,z_f,f
N13 G03 X.9153 Z-.0919 I-.1406 K0. n,g_arc_move,x_f,z_f,arc_center,f
N14 G01 X1. Z-.3209             n,cc,feed_move,x_f,z_f,f
N15 Z-.7656                     n,cc,feed_move,x_f,z_f,f
N16 G02 X1.1977 Z-.9971 I0. K.2344 n,g_arc_move,x_f,z_f,arc_center,f
N17 G01 X1.256 Z-1.0063         n,cc,feed_move,x_f,z_f,f
N18 G03 X1.6365 Z-1.285 I-.5093 K-.0806 n,g_arc_move,x_f,z_f,arc_center,f
N19 G01 X2. Z-2.0119           n,cc,feed_move,x_f,z_f,f
N20 Z-2.7656                   n,cc,feed_move,x_f,z_f,f
N21 G40                         n,cancel_offset
N22 G97                         n,"G97"
N23 T0100                       n_forced,special
N24 M09                         n,"M09"
N25 M05                         n,spindle_off
( )                               start_comment," ",end_comment
N26 G00 G28 U0. W0.             n,rapid_move,"G28","U0.", "W0."
(JOB 0 ROUGH CYCLE )           start_comment,comment1,end_comment
(TOOL #2 80 DEG. 1/64 FINISH TURNING ) start_comment,comment3,end_comment
N27 T0201                       n_forced,f_tool
N28 M01                         n,optional_stop
( )                               start_comment," ",end_comment
```

```

N29 G50 S3000          n,"G50",smax
N30 G97 S429 M03      n,"G97",out_finish_css_rpm,spindle_on
N31 G54 X-.0156 Z0. M08 n,rapid_move,"G54",force_x,xr,force_z,zr,coolant_on
N32 G96 S450          n,spsp_code,finish_s
N33 G70 P11 Q20 F.015 n,"G70",n_first_finish,n_last_finish,finish_feed
N34 G40               n,cancel_offset
N35 G97               n,"G97"
N36 T0200             n_forced,f_special
N37 M09               n,"M09"
N38 M05               n,spindle_off
( )                   start_comment," ",end_comment
N39 G00 G28 U0. W0.  n,rapid_move,"G28","U0.", "W0."
N40 M30               n,"M30"
%                     "%"
```

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

The **.LathePst** 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 **Lathe 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 **Fanuc 0T.LathePst** file, opened in the **Predator CNC Editor**.

```

"Customization file for Fanuc 0T Two Line format Lathe - SDP"

Post variables listed in PostVariables.txt

****Version number MONTH DAY YEAR****
1000. Version Information = Version Month Day Year? "1.2 01 21 2008"

0. File Header.
"% "
"O",prog_n,"( PROGRAM NUMBER )"
default_add_spaces

1. Start of program live tooling cycle.
"( PROGRAM START - LIVE TOOLING CYCLES )"
"( PROGRAM NAME: ",prog_name," )"
"( POST: FANUC 0T )"
"( 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
output_tool_list
```

```

n,absolute_coord,"G80",cancel_offset,inch_mode
n_forced,rapid_move,"G28","U0.", "W0."
start_comment,comment1,end_comment
start_comment,comment2,end_comment
n,"M45"
n,"G28","H0"
n,rapid_move,"G54","G97","S100",t,live_spindle_on

```

3. Tool change for turning cycles.

```

n,"M09"
n,spindle_off
start_comment," ",end_comment
n,rapid_move,"G28","U0.", "W0."
start_comment,comment1,end_comment
start_comment,comment2,end_comment
n_forced,t
n,optional_stop
start_comment," ",end_comment
n,"G50",smax
n,"G97",out_css_rpm,spindle_on
n,rapid_move,"G54",force_x,xr,force_z,zr,coolant_on
n,spsp_code,s

```

4. Next operation same tool turning.

```

n,"M09"
n,spindle_off
start_comment," ",end_comment
n,rapid_move,"G28","U0.", "W0."
start_comment,comment1,end_comment
start_comment,comment2,end_comment
n_forced,t
n,optional_stop
start_comment," ",end_comment
n,"G50",smax
n,"G97",out_css_rpm,spindle_on
n,rapid_move,"G54",force_x,xr,force_z,zr,coolant_on
n,spsp_code,s

```

5. End of turning cycles.

```

n,"M09"
n,spindle_off
start_comment," ",end_comment
n,rapid_move,"G28","U0.", "W0."
n,"M30"
"%"

```

6. Next operation same tool facing.

```

n,"M09"
n,spindle_off
start_comment," ",end_comment
n,rapid_move,"G28","U0.", "W0."
start_comment,comment1,end_comment
start_comment,comment2,end_comment
n_forced,t
n,optional_stop
start_comment," ",end_comment
n,"G50",smax
n,"G97",out_css_rpm,spindle_on
n,rapid_move,"G54",force_x,xr,force_z,zr,coolant_on
n,spsp_code,s

```

8. Set debug.

```

debug_off

```

10. Cancel cutter compensation.

```

"G40"

```

11. Cutter compensation left.

```

"G41"

```

12. Cutter compensation right.

```

"G42"

```

13. Tool length compensation.
"G43"

20. Rapid move.
n,rapid_move,xr,zr

21. Feed move.
n,cc,feed_move,x_f,z_f,f

25. Arc move.
n,g_arc_move,x_f,z_f,arc_center,f

60. Auto finish cycle for G71,G72,G73 for turning.
n,"M09"
n,spindle_off
start_comment,"",end_comment
n,rapid_move,"G28","U0.,"W0."
start_comment,comment1,end_comment
start_comment,comment3,end_comment
n_forced,f_tool
n,optional_stop
start_comment,"",end_comment
n,"G50",smax
n,"G97",out_finish_css_rpm,spindle_on
n,rapid_move,"G54",force_x,xr,force_z,zr,coolant_on
n,spsp_code,finish_s
n,"G70",n_first_finish,n_last_finish,finish_feed
n,cancel_offset
n,"G97"
n_forced,f_special
newvars

62. Start of rough (G71) turning cycle.
n,"G71",roughing_depth_of_cut_x,rough_retract_amount
n,"G71",n_first_rough,n_last_rough,rough_x_allowance,rough_z_allowance,rough_feed

63. End of rough (G71) turning cycle.
n,cancel_offset
n,"G97"
n_forced,special

66. Start of rough (G72) facing cycle.
n,"G72",roughing_depth_of_cut_z,rough_retract_amount
n,"G72",n_first_rough,n_last_rough,rough_x_allowance,rough_z_allowance,rough_feed

67. End of rough (G72) facing cycle
n,cancel_offset
n,"G97"
n_forced,special

70. Start of finish (G73) turning cycle
n,"G73",finish_x_spring_pass_stock_radius,finish_z_spring_pass_stock_radius,finish_number_of_cuts_two_line
n,"G73",n_first_rough,n_last_rough,finish_x_allowance,finish_z_allowance,rough_feed

71. End of finish G73) turning cycle
n,cancel_offset
n,"G97"
n_forced,special

74. Start of groove (G75) turning cycle
n,"G75",groove_retract_amount
n,"G75",groove_x_bottom,groove_z_bottom,groove_peck_increment,groove_depth_of_cut,groove_rvalue,rough_feed

75. End of groove (G75) turning cycle
n,cancel_offset
n,"G97"
n_forced,special

78. Start of groove (G74) facing cycle
n,"G74",groove_retract_amount
n,"G74",groove_x_bottom,groove_z_bottom,groove_peck_increment,groove_depth_of_cut,groove_rvalue,rough_feed

79. End of groove (G74) facing cycle

n,cancel_offset
 n,"G97"
 n_forced,special

82. Start of drill (G74) cycle
 n,"G74",groove_retract_amount
 n,"G74",groove_x_bottom,groove_z_bottom,groove_peck_increment,groove_depth_of_cut,groove_rvalue,rough_feed

83. End of drill (G74) cycle
 n,"G97"
 n_forced,special

86. Start of threading cycles.

87. Start of thread (G76) cycle
 n,thread_angle_out_start
 n,"G76",thread_angle_in,thread_last_cut,thread_machine_allowance
 n,"G76",thread_x2,thread_z2,thread_height,thread_first_cut,thread_lead
 n,thread_angle_out_end

88. End of thread (G76) cycle

89. Start of thread (G33) cycle

90. End of thread (G33) cycle

91. End of threading cycles.
 n,cancel_offset
 n,"G97"
 n_forced,special

92. G33 line move.
 n,thread_g33_gcode,thread_g33_xmove,thread_g33_zmove,thread_g33_feedrate

93. G33 thread cut move.
 n,thread_g33_gcode,thread_g33_xmove,thread_g33_zmove,thread_g33_feedrate

100. Start of cutoff cycle

101. End of cutoff cycle
 n,cancel_offset
 n,"G97"
 n_forced,special

104. Start of auto finish no canned cycle.

105. End of auto finish no canned cycle.
 n,cancel_offset
 n,"G97"
 n,special

108. Start of auto stock feed cycle.
 start_comment," ",end_comment
 "(STOCK FEED)"
 start_comment," ",end_comment
 rapid_to_pickup_z
 rapid_to_position_x
 n,"M11"
 n,"G04","X.5"
 rapid_to_stock_feed_z
 n,"G04","X.5"
 n,"M10"
 n,"G04","X.5"
 rapid_to_position_z

109. End of auto stock feed cycle.
 n,optional_stop

199. Not Used

220. Is X modal? y
 222. Is Z modal? y
 223. Are the g codes modal? y

224. Are the xy (or yz or xz) coordinates modal in arc milling? y
 225. Are the g codes (G02 and G03) modal in arc milling? y
 228. Output sequence numbers? y
 230. Delete the decimal point? n
 231. Delete leading zeros? y
 232. Delete trailing zeros? y
 234. Places before decimal point for reals? 1
 235. Number of places for G and M codes? 1
 236. Places after decimal for feedrate ? 4
 237. Scale factor for feedrate ? 1
 239. Add spaces to the program? y
 241. Break arcs into quadrants? n
 242. Arc center a=absolute, b=incremental, d=unsigned inc., e=radius? b
 243. Break arcs into two pieces if greater than 180 degrees? n
 247. Sign of X Spindle 1 (p = pos, n = neg, e = either, r = reverse)? e
 248. Sign of X Spindle 2 (p = pos, n = neg, e = either, r = reverse)? e
 249. Output X as a diameter or radius (d/r)? r
 250. Use canned cycle for drilling? n
 251. Use canned cycle for peck drilling? n
 252. Use canned cycle for chip break? n
 253. Use canned cycle for tap? n
 254. Use canned cycle for bore #1? n
 255. Use canned cycle for bore #2? n
 258. Use two line canned cycle format? y
 259. Depth format (d = default, t = trailing)? d
 260. Amount to add to t to obtain t1? 0
 261. Amount to add to t to obtain t2? 0
 262. Value of t1 at t = 0? 0
 263. Value of t2 at t = 0? 0
 265. Places before decimal point for angles? 1
 300. Number of places for sequence numbers? 2
 302. Number of places for program number? 5

411. R value (Retract amount) for first line of 2 line rough cycle? 0.1
 412. R value (Retract amount) for first line of 2 line groove cycle? 0.1
 413. R value (Thread machine allowance) 2 line G76 cycle? 0.001

512. Tool number format (1=n, 2=n*101, 3=100*n+toffset, 4=toffset, 5=n*100)? 3
 513. Number of digits for tool number? 4
 517. Medium gear spindle speed? 360
 518. High gear spindle speed? 780
 540. Number of decimal places real numbers metric ? 3
 541. Number of decimal places real numbers inch ? 4
 542. Number of decimal places angles ? 3

612. Reverse G2 and G3? n
 613. Reverse the sign of I? n
 614. Reverse the sign of K? n

705. Symbol for lead? "F"
 706. Symbol for IPR feedrate? "F"
 707. Symbol for IPM feedrate? "F"
 710. Begin comment character? "("
 711. End comment character? ")"
 712. Threading chamfer out start code? "M23"
 713. Threading chamfer out end code? "M24"
 714. Inch mode machining? "G20"
 715. Metric mode machining? "G21"
 720. Absolute coordinates? "G90"
 721. Incremental coordinates? "G91"
 722. Coordinate zero set? "G92"
 725. End of file? "M02"
 726. Stop? "M00"
 727. Optional Stop? "M01"
 728. Subprogram call? "M98"
 729. Subprogram return? "M99"
 739. Cancel wire offset? "G40"
 741. Prefix for radius values? "R"
 742. Prefix for arc Z center? "I"
 743. Prefix for arc X center? "K"
 750. Wire comp left? "G41"
 751. Wire comp right? "G42"
 756. Block delete? "/"

773. Coolant on String ? "M08"
 774. Coolant off String ? "M09"
 780. Symbol for Spindle Speed ? "S"
 781. Symbol for Tool Number Prefix ? "T"
 782. Spindle forward String ? "M03"
 783. Spindle reverse String ? "M04"
 784. Spindle off String ? "M05"
 785. Symbol for CSS spindle speed? "G96"
 786. Symbol for RPM spindle speed? "G97"
 787. Code for low spindle gear? "M40"
 788. Code for medium spindle gear? "M41"
 789. Code for high spindle gear? "M42"
 790. Prefix for canned cycle start line Number? "P"
 791. Prefix for canned cycle end line Number? "Q"
 792. Prefix for thread G76 first cut amount? "D"
 793. Prefix for thread height? "K"
 794. Prefix for thread angle in? "P"
 795. Prefix for thread last cut amount? "Q"
 796. Prefix for thread machine allowance? "R"
 797. GCode for Rapid Move? "G00"
 798. GCode for Feed Move? "G01"
 799. GCode for Arc CW? "G02"
 800. GCode for Arc CCW? "G03"
 801. Prefix for X Move? "X"
 802. Prefix for Y Move? "Y"
 803. Prefix for Z Move? "Z"
 804. Prefix for Rough Cycle depth of cut? "D"
 805. Prefix for Rough Cycle stock allowance X? "U"
 806. Prefix for Rough Cycle stock allowance Z? "W"
 807. Prefix for Rough Cycle retract amount? "R"
 808. Prefix for Finish Cycle stock allowance X? "U"
 809. Prefix for Finish Cycle stock allowance Z? "W"
 810. Prefix for Finish Cycle X stock amount for spring passes? "U"
 811. Prefix for Finish Cycle Z stock amount for spring passes? "W"
 812. Prefix for Groove Cycle retract amount? "R"
 813. Prefix for Groove Cycle X bottom? "X"
 814. Prefix for Groove Cycle Z bottom? "Z"
 815. Prefix for Groove Cycle depth of cut? "K"
 816. Prefix for Groove Cycle peck increment? "I"
 822. GCode for simple threading cycle? "G33"
 823. Prefix for X depth for roughing cycle? "U."
 824. Prefix for Z depth for roughing cycle? "W."
 825. Prefix for groove dwell? "D"
 826. Prefix for groove peck clearance? "R"
 827. Prefix for groove peck retract? "R"

 860. Standard Drill cycle live tool face? "G81"
 861. Peck drill cycle live tool face? "G83"
 862. High speed peck cycle live tool face? "G83"
 863. Tap cycle live tool face? "G84"
 864. Boring cycle 1 live tool face? "G85"
 865. Boring cycle 2 live tool face? "G86"
 866. Boring cycle 3 live tool face? "G89"

 880. Standard Drill cycle live tool diameter? "G81"
 881. Peck drill cycle live tool diameter? "G83"
 882. High speed peck cycle live tool diameter? "G83"
 883. Tap cycle live tool diameter? "G84"
 884. Boring cycle 1 live tool diameter? "G85"
 885. Boring cycle 2 live tool diameter? "G86"
 886. Boring cycle 3 live tool diameter? "G89"

 890. Canned cycle cancel? "G80"

Customization Variables and Commands for Lathe 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 – Positioning

| | |
|------------------------------------|---|
| <code>rapid_move</code> | = Output rapid move (G00). |
| <code>feed_move</code> | = Output feed move (G01). |
| <code>arc_move</code> | = Output arc feed move (G02,G03). |
| <code>cr</code> | = C rapid with live tool, diameter and face machining. |
| <code>xr</code> | = X rapid with live tool, diameter and face machining. |
| <code>zr</code> | = Z rapid with live tool, diameter and face machining. |
| <code>crl</code> | = C rapid with live tool, diameter and face machining. |
| <code>xrl</code> | = X rapid with live tool, diameter and face machining. |
| <code>zrl</code> | = Z rapid with live tool, diameter and face machining. |
| <code>xh</code> | = X home position. |
| <code>zh</code> | = Z home position. |
| <code>xhl</code> | = X home position with live tooling. |
| <code>zhl</code> | = Z home position with live tooling. |
| <code>home</code> | = X, Z tool home position. |
| <code>prev_x</code> | = Previous X position. |
| <code>prev_z</code> | = Previous X position. |
| <code>rapid_to_pickup_z</code> | = |
| <code>rapid_to_position_x</code> | = |
| <code>rapid_to_position_z</code> | = |
| <code>rapid_to_stock_feed_z</code> | = |
| <code>rapid_iplane_face</code> | = Rapid to iplane in face live tool (G0 Z.1) |
| <code>rapid_rplane_face</code> | = Rapid to rplane in face live tool (G0 Z.1) |
| <code>rapid_iplane_diam</code> | = |
| <code>rapid_rplane_diam</code> | = |
| <code>pos_rpm</code> | = Position move in RPM mode. |
| <code>pos_rpm_prev</code> | = Position move with previous RPM . |

Variables – Line Numbers

| | |
|-----------------------------|--|
| <code>n</code> | = Sequence number. |
| <code>n_forced</code> | = Output sequence number even if Q. 228 is set to NO. |
| <code>n_first</code> | = P word for the first line number of turning canned cycle. |
| <code>n_last</code> | = Q word for the last line number of turning canned cycle. |
| <code>n_first_finish</code> | = P word for the first line number of turning canned cycle output to finish pass. |
| <code>n_last_finish</code> | = Q word for the last line number of turning canned cycle output to finish pass. |

Variables – Comments

| | |
|------------------------------|---|
| <code>prog_n</code> | = Program number. |
| <code>prog_name</code> | = Program name. |
| <code>output_date</code> | = Output current date. |
| <code>comment1</code> | = Job comment. |
| <code>comment2</code> | = Tool comment. |
| <code>comment3</code> | = Finish tool comment. |
| <code>start_comment</code> | = Output start of comment string. |
| <code>end_comment</code> | = Output end of comment string. |
| <code>user_comment_1</code> | = Output user comment 1 in Turning tools, Current settings, Posting |
| <code>user_comment_2</code> | = Output user comment 2 in Turning tools, Current settings, Posting |
| <code>user_comment_3</code> | = Output user comment 3 in Turning tools, Current settings, Posting |
| <code>user_comment_4</code> | = Output user comment 4 in Turning tools, Current settings, Posting |
| <code>user_comment_5</code> | = Output user comment 5 in Turning tools, Current settings, Posting |
| <code>user_comment_6</code> | = Output user comment 6 in Turning tools, Current settings, Posting |
| <code>user_comment_7</code> | = Output user comment 7 in Turning tools, Current settings, Posting |
| <code>user_comment_8</code> | = Output user comment 8 in Turning tools, Current settings, Posting |
| <code>user_comment_9</code> | = Output user comment 9 in Turning tools, Current settings, Posting |
| <code>user_comment_10</code> | = Output user comment 10 in Turning tools, Current settings, Posting |
| <code>user_comment_11</code> | = Output user comment 11 in Turning tools, Current settings, Posting |
| <code>user_comment_12</code> | = Output user comment 12 in Turning tools, Current settings, Posting |
| <code>user_comment_13</code> | = Output user comment 13 in Turning tools, Current settings, Posting |
| <code>user_comment_14</code> | = Output user comment 14 in Turning tools, Current settings, Posting |
| <code>user_comment_15</code> | = Output user comment 15 in Turning tools, Current settings, Posting |

Variables – Tools

| | |
|--------------------------|---|
| <code>t</code> | = Output first tool number without prefix. |
| <code>f_tool</code> | = Canned cycle auto finish tool number. |
| <code>t1</code> | = Tool number + #260 (without "T" prefix) |
| <code>t2</code> | = Tool number + #261 (without "T" prefix) |
| <code>special</code> | = Tool number with offset canceled (ex. T0100) |
| <code>f_special</code> | = Finish tool number with offset canceled (ex. T0100) |
| <code>n_tool</code> | = N word with tool number. |
| <code>n_ftool</code> | = N word with finish tool number. |
| <code>first_tool</code> | = First tool number without 'T' code. |
| <code>next_tool</code> | = Next tool number without 'T' code. |
| <code>p_tool_num</code> | = P word with tool number for G10 line. |
| <code>p_ftool_num</code> | = P word with finish tool for G10 line. |
| <code>p_special</code> | = P word G10 line tool number with offset canceled. |

Variables – Feed Rates

| | |
|--------------------------|---------------------------------------|
| <code>f</code> | = Feed rate with prefix. |
| <code>finish_feed</code> | = F word with finish cycle feed rate. |
| <code>rough_feed</code> | = F word with rough cycle fee rate |

Variables – M Codes

| | |
|------------------------------|---|
| <code>coolant_on</code> | = M08 if coolant used, null if coolant not used. |
| <code>coolant_off</code> | = M09 if coolant used, null if coolant not used. |
| <code>live_spindle_on</code> | = M code for live tool on. |
| <code>spindle_on</code> | = M03,M04 code for spindle on. |
| <code>spindle_off</code> | = M05 code for spindle off |
| <code>stop</code> | = M00 for program stop |
| <code>optional_stop</code> | = M01 for program optional stop |

Variables – Offset

| | |
|-----------------|--|
| <code>cc</code> | = Cutter compensation code (G41,G42). |
| <code>h</code> | = Output length offset register with prefix. |

Variables – Misc. Codes

| | |
|---------------------------------|---|
| <code>s</code> | = Output spindle speed with prefix " S1000 ". |
| <code>finish_s</code> | = Output finish spindle speed with prefix " S1000 ". |
| <code>smax</code> | = Maximum spindle speed. |
| <code>sp_on_css</code> | = Spindle on using constant surface speed. |
| <code>sp_on_rpm</code> | = Spindle on using RPM. |
| <code>spsp_code</code> | = Rough spindle speed code (G96, G97). |
| <code>spindle_gear</code> | = Output spindle gear. |
| <code>g112</code> | = Activate G112 live tooling mode. |
| <code>g113</code> | = Cancel G112 live tooling mode. |
| <code>mill_radius_string</code> | = R word with live tool radius. |
| <code>out_css_rpm</code> | = Output rough spindle speed in CSS/RPM mode |
| <code>out_finish_css_rpm</code> | = Output finish spindle speed in CSS/RPM mode |
| <code>finish_spsp_code</code> | = Finish spindle speed code (G96, G97). |

Variables – Rough Canned Cycles

| | |
|--|--|
| <code>n_first_rough</code> | = word for the first line number of Roughing canned cycle. |
| <code>n_last_rough</code> | = word for the last line number of Roughing canned cycle. |
| <code>rough_x_allowance</code> | = Rough allowance X axis |
| <code>rough_z_allowance</code> | = Rough allowance Z axis |
| <code>rough_depth_of_cut</code> | = Rough depth of cut |
| <code>rough_depth_of_cut_two_line</code> | = Rough depth of cut for two line canned cycles |
| <code>rough_depth_of_cut_with_decimal</code> | = Rough depth of cut with decimal place |
| <code>rough_depth_of_cut_two_line_decimal</code> | = Rough depth of cut with decimal place for two line canned cycles |
| <code>rough_retract_amount</code> | = Rough retract amount |
| <code>roughing_depth_of_cut_x</code> | = Rough depth of cut along X axis |
| <code>roughing_depth_of_cut_z</code> | = Rough depth of cut along Z axis |
| <code>roughing_depth_of_cut_x_decimal</code> | = Rough depth of cut with decimal along X axis |
| <code>roughing_depth_of_cut_z_decimal</code> | = Rough depth of cut with decimal along Z axis |

Variables – Canned Finish Cut

| | |
|--|--|
| <code>finish_x_spring_pass_stock_diameter</code> | = Spring pass allowance X axis for diameter mode |
| <code>finish_z_spring_pass_stock_diameter</code> | = Spring pass allowance Z axis for diameter mode |
| <code>finish_x_allowance</code> | = Finish allowance X axis |
| <code>finish_z_allowance</code> | = Finish allowance Z axis |
| <code>finish_number_of_cuts</code> | = Number of passes for finish cut |
| <code>finish_x_spring_pass_stock_radius</code> | = Spring pass allowance X axis for radius mode |
| <code>finish_z_spring_pass_stock_radius</code> | = Spring pass allowance Z axis for radius mode |
| <code>finish_number_of_cuts_two_line</code> | = Number of passes for finish cut for two line canned cycles |

Variables – Groove-Drill Canned Cycles

| | |
|------------------------------------|--|
| <code>groove_x_bottom</code> | = Groove X location for final cut |
| <code>groove_z_bottom</code> | = Groove Z location for final cut |
| <code>groove_peck_increment</code> | = Groove peck increment |
| <code>groove_depth_of_cut</code> | = Groove depth of cut |
| <code>groove_retract_amount</code> | = Groove retract amount |
| <code>groove_rvalue</code> | = Groove r value |
| <code>drill_dwell</code> | = Dwell |
| <code>first_peck</code> | = First peck increment |
| <code>peck_increment</code> | = Next or peck increment |
| <code>peck_clearance</code> | = Peck clearance retract |
| <code>peck_retract</code> | = Peck retract |
| <code>last_var_name</code> | = Output last variable name |

Variables – Threading Canned Cycles

| | |
|-------------------------------------|---------------------------------|
| <code>thread_x2</code> | = Thread location X axis |
| <code>thread_z2</code> | = Thread location Z axis |
| <code>thread_angle_out_start</code> | = Thread angle out start |
| <code>thread_angle_out_end</code> | = Thread angle out end |

Variables – Threading Canned Cycles - Cont'd

| | |
|---------------------------------------|---|
| <code>thread_height</code> | = Thread height |
| <code>thread_first_cut</code> | = Thread first cut amount |
| <code>thread_lead</code> | = Thread lead |
| <code>thread_angle_in</code> | = Thread angle in |
| <code>thread_last_cut</code> | = Amount of thread last cut |
| <code>thread_machine_allowance</code> | = Thread machine allowance, defined in post file |
| <code>thread_g33_gcode</code> | = Simple thread g code, defined in G Codes tab |
| <code>thread_g33_xmove</code> | = Simple thread X move |
| <code>thread_g33_zmove</code> | = Simple thread Z move |
| <code>thread_g33_feedrate</code> | = Simple thread feed rate |

Variables – Profile Start-End

| | |
|------------------------------|----------------------------------|
| <code>profile_start_x</code> | = Start of profile X axis |
| <code>profile_start_z</code> | = Start of profile Z axis |
| <code>profile_end_x</code> | = End of profile X axis |
| <code>profile_end_z</code> | = End of profile Z axis |