

NGC Variable	Legacy Variable	Usage
#12000- #12255	N/A	Discrete outputs
#13000- #13063	N/A	Filtered analog to digital inputs (read only)
#13013	N/A	Coolant level
#14001- #14006	N/A	G110(G154 P1) additional work offsets
#14021- #14026	N/A	G110(G154 P2) additional work offsets
#14041- #14386	N/A	G110(G154 P3- G154 P20) additional work offsets
#14401- #14406	N/A	G110(G154 P21) additional work offsets
#14421- #15966	N/A	G110(G154 P22- G154 P99) additional work offsets
#20000- #29999	N/A	Settings
#30000- #39999	N/A	Parameters
#32014	N/A	Machine Serial Number
#50001- #50200	N/A	Tool Type
#50201- #50400	N/A	Tool material
#50401- #50600	N/A	Tool Offset Point
#50601- #50800	N/A	Estimated RPM
#50801- #51000	N/A	Estimated Feedrate
#51001- #51200	N/A	Offset Pitch
#51201- #51400	N/A	Actually VPS Estimated RPM
#51401- #51600	N/A	Work Material
#51601- #51800	N/A	VPS Feedrate
#51801- #52000	N/A	Approximate length
#52001- #52200	N/A	Approximate diameter
#52201- #52400	N/A	Edge Measure height

NGC Variable	Legacy Variable	Usage
#52401- #52600	N/A	Tool Tolerance
#52601- #52800	N/A	Probe Type

6.13.8 System Variables In-Depth

System variables are associated with specific functions. A detailed description of these functions follows.

#550-#699 #10550- #10699 General and Probe Calibration Data

These general purpose variables are saved on power off. Some of these higher #5xx variables store probe calibration data. Example: #592 sets which side of the table the tool probe is positioned. If these variables are overwritten, you will need to calibrate the probe again.


NOTE:

If the machine does not have a probe installed, you can use these variables as general-purpose variables saved on power off.

#1080-#1097 #11000-#11255 #13000-#13063 1-Bit Discrete Inputs

You can connect designated inputs from external devices with these macros:

Variables	Legacy Variables	Usage
#11000-#11255		256 discrete inputs (read only)
#13000-#13063	#1080-#1087 #1090-#1097	Raw and Filtered analog to digital inputs (read only)

Specific input values can be read from within a program. The format is #11nnn where nnn is the Input Number. Press [DIAGNOSTIC] and select the I/O tab to see the Input and Output numbers for different devices.

Example:

#10000=#11018

This example records the state of #11018, which refers to Input 18 (M-Fin_Input), to variable #10000.

For available User Inputs on the I/O PCB, refer to the Robot Integration Aid reference document in the Haas service website.

- F6.21: Scan the QR code for Robot Integration Aid reference document



#1064-#1268 Maximum Axis Loads

These variables contain the maximum load an axis has achieved since the machine was last powered on, or since that Macro Variable was cleared. The Maximum Axis Load is the greatest load (100.0 = 100%) an axis has seen, not the Axis Load at the time that the control reads the variable.

#1064 = X Axis	#1264 = C axis
#1065 = Y Axis	#1265 = U axis
#1066 = Z Axis	#1266 = V axis
#1067 = A Axis	#1267 = W axis
#1068 = B Axis	#1268 = T axis

#2001-#2800 Tool Offsets

Each tool offset has a length (H) and diameter (D) along with associated wear values.

#2001-#2200	H geometry offsets (1-200) for length.
#2201-#2400	H geometry wear (1-200) for length.

#2401-#2600	D geometry offsets (1-200) for diameter.
#2601-#2800	D geometry wear (1-200) for diameter.

#3000 Programmable Alarm Messages

#3000 Alarms can be programmed. A programmable alarm will act like the built-in alarms. An alarm is generated by setting macro variable #3000 to a number between 1 and 999.

```
#3000= 15 (MESSAGE PLACED INTO ALARM LIST) ;
```

When this is done, *Alarm* flashes at the bottom of the display and the text in the next comment is placed into the alarm list. The alarm number (in this example, 15) is added to 1000 and used as an alarm number. If an alarm is generated in this manner all motion stops and the program must be reset to continue. Programmable alarms are always numbered between 1000 and 1999.

#3001-#3002 Timers

Two timers can be set to a value by assigning a number to the respective variable. A program can then read the variable and determine the time passed since the timer was set. Timers can be used to imitate dwell cycles, determine part-to-part time or wherever time-dependent behavior is desired.

- #3001 Millisecond Timer - The millisecond timer represents the system time after power on in number of milliseconds. The whole number returned after accessing #3001 represents the number of milliseconds.
- #3002 Hour Timer - The hour timer is similar to the millisecond timer except that the number returned after accessing #3002 is in hours. The hour and millisecond timers are independent of each other and can be set separately.

#3003 Single Block Suppression

Variable #3003 overrides the Single Block function in G-code. When #3003 has a value of 1, the control executes each G-code command continuously even though the Single Block function is ON. When #3003 has a value of zero, Single Block operates as normal. You must press **[CYCLE START]** to execute each line of code in single block mode.

```
...
#3003=1 ;
G54 G00 G90 X0 Y0 ;
S2000 M03 ;
G43 H01 Z.1 ;
G81 R.1 Z-0.1 F20. ;
#3003=0 ;
```

```

T02 M06 ;
G43 H02 Z.1 ;
S1800 M03 ;
G83 R.1 Z-1. Q.25 F10. ;
X0. Y0. ;
%
```

#3004 Enables and Disables Feed Hold

Variable #3004 overrides specific control features during operation.

The first bit disables **[FEED HOLD]**. If variable #3004 is set to 1, **[FEED HOLD]** is disabled for the program blocks that follow. Set #3004 to 0 to enable **[FEED HOLD]** again. For example:

```

...
(Approach code - [FEED HOLD] allowed) ;
#3004=1 (Disables [FEED HOLD]) ;
(Non-stoppable code - [FEED HOLD] not allowed) ;
#3004=0 (Enables [FEED HOLD]) ;
(Depart code - [FEED HOLD] allowed) ;
...
...
```

This is a map of variable #3004 bits and the associated overrides.

E = Enabled D = Disabled

#3004	Feed Hold	Feed Rate Override	Exact Stop Check
0	E	E	E
1	D	E	E
2	E	D	E
3	D	D	E
4	E	E	D
5	D	E	D
6	E	D	D
7	D	D	D



NOTE:

When the feed rate override variable is set (#3004 = 2), the control will set the feed rate override to 100% (default). During #3004 = 2 the control will show 100% in red bold text on the display until the variable is reset. Once the feed rate override is reset (#3004 = 0) the feed rate will be restored to the previous value before setting the variable.

#3006 Programmable Stop

You can add stops to the program that act like an M00 - The control stops and waits until you press **[CYCLE START]**, then the program continues with the block after the #3006. In this example, the control displays the comment on the lower-center of the screen.

```
#3006=1 (comment here) ;
```

#3030 Single Block

In Next Generation Control when the system variable #3030 is set to a 1; the control will go into single block mode. There is no need to limit the lookahead using a G103 P1, the Next Generation Control will correctly process this code.



NOTE:

For the Classic Haas Control to process system variable #3030=1 correctly, it is necessary to limit the lookahead to 1 block using a G103 P1 before the #3030=1 code.

#4001-#4021 Last Block (Modal) Group Codes

G-code groups let the machine control process the codes more efficiently. G-codes with similar functions are usually in the same group. For example, G90 and G91 are under group 3. Macro variables #4001 through #4021 store the last or default G code for any of 21 groups.

G-Codes Group number is listed next to its description in the G-Code section.

Example:

G81 Drill Canned Cycle (Group 09)

When a macro program reads the group code, the program can change the behavior of the G-code. If #4003 contains 91, then a macro program could determine that all moves should be incremental rather than absolute. There is no associated variable for group zero; group zero G codes are Non-modal.

#4101-#4126 Last Block (Modal) Address Data

Address codes A-Z (excluding G) are maintained as modal values. The information represented by the last line of code interpreted by the lookahead process is contained in variables #4101 through #4126. The numeric mapping of variable numbers to alphabetic addresses corresponds to the mapping under alphabetic addresses. For example, the value of the previously interpreted D address is found in #4107 and the last interpreted I value is #4104. When aliasing a macro to an M-code, you may not pass variables to the macro using variables #1 - #33. Instead, use the values from #4101 - #4126 in the macro.

#5001-#5006 Last Target Position

The final programmed point for the last motion block can be accessed through variables #5001 - #5006, X, Z, Y, A, B, and C respectively. Values are given in the current work coordinate system and can be used while the machine is in motion.

#5021-#5026 Current Machine Coordinate Position

To get the current machine axis positions, call macro variables #5021-#5026 corresponding to axis X, Y, Z, A, B, and C, respectively.

#5021 X Axis	#5022 Y Axis	#5023 Z Axis
#5024 A Axis	#5025 B Axis	#5026 C Axis



NOTE:

Values CANNOT be read while the machine is in motion.

#5041-#5046 Current Work Coordinate Position

To get the current work coordinate positions, call macro variables #5041-#5046 corresponding to axis X, Y, Z, A, B, and C, respectively.



NOTE:

The values CANNOT be read while the machine is in motion.

#5061-#5069 Current Skip Signal Position

Macro variables #5061-#5069 corresponding to X, Y, Z, A, B, C, U, V and W respectively, give the axis positions where the last skip signal occurred. Values are given in the current work coordinate system and can be used while the machine is in motion.

The value of #5063 (Z) has tool length compensation applied to it.

#5081-#5086 Tool Length Compensation

Macro variables #5081 - #5086 give the current total tool length compensation in axis X, Y, Z, A, B, or C, respectively. This includes tool length offset referenced by the current value set in H (#4008) plus the wear value.

#5201-#5326, #7001-#7386, #14001-#14386 Work Offsets

Macro expressions can read and set all work offsets. This lets you preset coordinates to exact locations, or set coordinates to values based upon the results of skip signal (probed) locations and calculations. When any of the offsets are read, the interpretation look-ahead queue is stopped until that block is executed.

Variables	Legacy Variables	Usage
	#5201- #5206	G52 X, Y, Z, A, B, C OFFSET VALUES
	#5221- #5226	G54 X, Y, Z, A, B, C OFFSET VALUES
	#5241- #5246	G55 X, Y, Z, A, B, C OFFSET VALUES
	#5261- #5266	G56 X, Y, Z, A, B, C OFFSET VALUES
	#5281- #5286	G57 X, Y, Z, A, B, C OFFSET VALUES
	#5301- #5306	G58 X, Y, Z, A, B, C OFFSET VALUES
	#5321- #5326	G59X, Y, Z, A, B, C OFFSET VALUES
#14001-#14006	#7001- #7006	G110 (G154 P1) additional work offsets
#14021-#14026	#7021-#7026	G111 (G154 P2) additional work offsets
#14041-#14046	#7041-#7046	G112 (G154 P3) additional work offsets
#14061-#14066	#7061-#7066	G113 (G154 P4) additional work offsets
#14081-#14086	#7081-#7086	G114 (G154 P5) additional work offsets
#14101-#14106	#7101-#7106	G115 (G154 P6) additional work offsets
#14121-#14126	#7121-#7126	G116 (G154 P7) additional work offsets
#14141-#14146	#7141-#7146	G117 (G154 P8) additional work offsets
#14161-#14166	#7161-#7166	G118 (G154 P9) additional work offsets

Variables	Legacy Variables	Usage
#14181-#14186	#7181-#7186	G119 (G154 P10) additional work offsets
#14201-#14206	#7201-#7206	G120 (G154 P11) additional work offsets
#14221-#14226	#7221-#7226	G121 (G154 P12) additional work offsets
#14241-#14246	#7241-#7246	G122 (G154 P13) additional work offsets
#14261-#14266	#7261-#7266	G123 (G154 P14) additional work offsets
#14281-#14286	#7281-#7286	G124 (G154 P15) additional work offsets
#14301-#14306	#7301-#7306	G125 (G154 P16) additional work offsets
#14321-#14326	#7321-#7326	G126 (G154 P17) additional work offsets
#14341-#14346	#7341-#7346	G127 (G154 P18) additional work offsets
#14361-#14366	#7361-#7366	G128 (G154 P19) additional work offsets
#14381-#14386	#7381-#7386	G129 (G154 P20) additional work offsets

#6198 Next-Generation Control Identifier

The macro variable #6198 has a read-only value of 1000000.

You can test #6198 in a program to detect the control version, and then conditionally run program code for that control version. For example:

```
%  
IF [#6198 EQ 1000000] GOTO5 ;  
(Non-NGC code) ;  
GOTO6 ;  
N5 (NGC code) ;  
N6 M30 ;  
%
```

In this program, if the value stored in #6198 is equal to 1000000, go to Next Generation Control compatible code then end the program. If if the value stored in #6198 is not equal to 1000000, run the non-NGC program and then end the program.

#7501 - #7806, #3028 Pallet Changer Variables

The status of the pallets from the Automatic Pallet Changer is checked with these variables:

#7501-#7506	Pallet priority
#7601-#7606	Pallet status
#7701-#7706	Part program numbers assigned to pallets
#7801-#7806	Pallet usage count
#3028	Number of pallet loaded on receiver

#8500-#8515 Advanced Tool Management

These variables give information on Advanced Tool Management (ATM). Set variable #8500 to the tool group number, then access information for the selected tool group with the read-only macros #8501-#8515.

#8500	Advanced Tool Management (ATM). Group ID
#8501	ATM. Percent of available tool life of all tools in the group.
#8502	ATM. Total available tool usage count in the group.
#8503	ATM. Total available tool hole count in the group.
#8504	ATM. Total available tool feed time (in seconds) in the group.
#8505	ATM. Total available tool total time (in seconds) in the group.
#8510	ATM. Next tool number to be used.
#8511	ATM. Percent of available tool life of the next tool.
#8512	ATM. Available usage count of the next tool.
#8513	ATM. Available hole count of the next tool.

#8514	ATM. Available feed time of the next tool (in seconds).
#8515	ATM. Available total time of the next tool (in seconds).

#8550-#8567 Advanced Tool Management Tooling

These variables give information on tooling. Set variable #8550 to the tool offset number, then access information for the selected tool with the read-only macros #8551-#8567.



NOTE:

Macro variables #1601-#2800 give access to the same data for individual tools as #8550-#8567 give for Tool Group tools.

#8550	Individual tool ID
#8551	Number of Flutes on tool
#8552	Maximum recorded vibration
#8553	Tool length offset
#8554	Tool length wear
#8555	Tool diameter offset
#8556	Tool diameter wear
#8557	Actual diameter
#8558	Programmable coolant position
#8559	Tool feed timer (seconds)
#8560	Total tool timers (seconds)
#8561	Tool life monitor limit
#8562	Tool life monitor counter
#8563	Tool load monitor maximum load sensed so far
#8564	Tool load monitor limit

#12000-#12255 1-Bit Discrete Outputs

The Haas control is capable of controlling up to 256 discrete outputs. However, a number of these outputs are reserved for the Haas control to use.

Variables	Legacy Variables	Usage
#12000-#12255		256 discrete outputs

Specific output values can be read, or written to, from within a program. The format is #12nnn where nnn is the Output Number.

Example:

```
#10000=#12018 ;
```

This example records the state of #12018, which refers to Input 18 (Coolant Pump Motor), to variable #10000.

#20000-#20999 Settings Access with Macro Variables

Access settings through variables #20000 - #20999, starting from setting 1 respectively. Refer to page 439 for the detailed descriptions of the settings that are available in the control.



NOTE:

The #20000 - 20999 range numbers correspond directly to Setting numbers plus 20000.

#50001 - #50200 Tool Type

Use macro variables #50001 - #50200, to read or write the tool type set in the tool offset page.

T6.3: Available Tool Types for Mill

Tool Type	Tool Type #
Drill	1
Tap	2
Shell Mill	3

Tool Type	Tool Type #
End Mill	4
Spot Drill	5
Ball Nose	6
Probe	7
Reserve for Future Use	8-20

6.13.9 Variable Usage

All variables are referenced with a number sign (#) followed by a positive number: #1, #10001, and #10501.

Variables are decimal values that are represented as floating point numbers. If a variable has never been used, it can take on a special `undefined` value. This indicates that it has not been used. A variable can be set to `undefined` with the special variable #0. #0 has the value of undefined or 0.0 depending on its context. Indirect references to variables can be accomplished by enclosing the variable number in brackets: # [<Expression>]

The expression is evaluated and the result becomes the variable accessed. For example:

```
#1=3 ;
#[#1]=3.5 + #1 ;
```

This sets the variable #3 to the value 6.5.

A variable can be used in place of a G-code address where address refers to the letters A-Z.

In the block:

```
N1 G0 G90 X1.0 Y0 ;
```

the variables can be set to the following values:

```
#7=0 ;
#11=90 ;
#1=1.0 ;
#2=0.0 ;
```

and replaced by:

```
N1 G#7 G#11 X#1 Y#2 ;
```

Values in the variables at runtime are used as the address values.

6.13.10 Address Substitution

The usual method of setting control addresses A-Z is the address followed by a number. For example:

```
G01 X2.5 Y3.7 F20.;
```

sets addresses G, X, Y and F to 1, 1.5, 3.7 and 20.0 respectively and thus instructs the control to move linearly, G01, to position X=2.5 Y=3.7 at a feed rate of 20 (in/mm). Macro syntax allows the address values to be replaced with any variable or expression.

The previous statement can be replaced by this code:

```
#1=1 ;
#2=1.5 ;
#3=3.7 ;
#4=20 ;
G#1 X[#1+#2] Y#3 F#4 ;
```

The permissible syntax on addresses A-Z (exclude N or O) is as follows:

<address><variable>	A#101
<address><-><variable>	A-#101
<address>[<expression>]	Z[#5041+3.5]
<address><->[<expression>]	Z-[SIN[#1]]

If the variable value does not agree with the address range, the control generates an alarm. For example, this code causes a range error alarm because the tool diameter numbers range from 0 to 200.

```
#1=250 ;
D#1 ;
```

When a variable or expression is used in place of an address value, the value is rounded to the least significant digit. If #1=.123456, then G01 X#1 would move the machine tool to .1235 on the X Axis. If the control is in the metric mode, the machine would be moved to .123 on the X axis.

When an undefined variable is used to replace an address value, that address reference is ignored. For example, if #1 is undefined, then the block

```
G00 X1.0 Y#1 ;
```

becomes

```
G00 X1.0 ;
```

and no Y movement takes place.

Macro Statements

Macro statements are lines of code that allow the programmer to manipulate the control with features similar to any standard programming language. Included are functions, operators, conditional and arithmetic expressions, assignment statements, and control statements.

Functions and operators are used in expressions to modify variables or values. The operators are essential to expressions while functions make the programmer's job easier.

Functions

Functions are built-in routines that the programmer has available to use. All functions have the form <function_name>[argument] and return floating-point decimal values. The functions provided in the Haas control are as follows:

Function	Argument	Returns	Notes
SIN[]	Degrees	Decimal	Sine
COS[]	Degrees	Decimal	Cosine
TAN[]	Degrees	Decimal	Tangent
ATAN[]	Decimal	Degrees	Arctangent Same as FANUC ATAN[]/[1]

Function	Argument	Returns	Notes
SQRT[]	Decimal	Decimal	Square root
ABS[]	Decimal	Decimal	Absolute value
ROUND[]	Decimal	Decimal	Round off a decimal
FIX[]	Decimal	Integer	Truncate fraction
ACOS[]	Decimal	Degrees	Arc cosine
ASIN[]	Decimal	Degrees	Arcsine
#[]	Integer	Integer	Indirect reference Refer to page 275

Notes on Functions

The function ROUND works differently depending on the context that it is used. When used in arithmetic expressions, any number with a fractional part greater than or equal to .5 is rounded up to the next whole integer; otherwise, the fractional part is truncated from the number.

```
%  
#1=1.714 ;  
#2=ROUND[#1] (#2 is set to 2.0) ;  
#1=3.1416 ;  
#2=ROUND[#1] (#2 is set to 3.0) ;  
%
```

When ROUND is used in an address expression, metric and angle dimensions are rounded to three-place precision. For inch dimensions, four-place precision is the default.

```
%  
#1= 1.00333 ;  
G00 X[ #1 + #1 ] ;  
(Table X Axis moves to 2.0067) ;  
G00 X[ ROUND[ #1 ] + ROUND[ #1 ] ] ;  
(Table X Axis moves to 2.0067) ;  
G00 A[ #1 + #1 ] ;  
(Axis rotates to 2.007) ;  
G00 A[ ROUND[ #1 ] + ROUND[ #1 ] ] ;  
(Axis rotates to 2.007) ;  
D[1.67] (Diameter rounded up to 2) ;
```

%

Fix vs. Round

```
%  
#1=3.54 ;  
#2=ROUND[#1] ;  
#3=FIX[#1].  
%
```

#2 will be set to 4. #3 will be set to 3.

Operators

Operators have (3) categories: Boolean, Arithmetic, and Logical.

Boolean Operators

Boolean operators always evaluate to 1.0 (TRUE) or 0.0 (FALSE). There are six Boolean operators. These operators are not restricted to conditional expressions, but they most often are used in conditional expressions. They are:

EQ - Equal To

NE - Not Equal To

GT - Greater Than

LT - Less Than

GE - Greater Than or Equal To

LE - Less Than or Equal To

Here are four examples of how Boolean and Logical operators can be used:

Example	Explanation
IF [#10001 EQ 0.0] GOTO100 ;	Jump to block 100 if value in variable #10001 equals 0.0.
WHILE [#10101 LT 10] DO1 ;	While variable #10101 is less than 10 repeat loop DO1..END1.

Example	Explanation
#10001=[1.0 LT 5.0] ;	Variable #10001 is set to 1.0 (TRUE).
IF [#10001 AND #10002 EQ #10003] GOTO1 ;	If variable #10001 AND variable #10002 are equal to the value in #10003 then control jumps to block 1.

Arithmetic Operators

Arithmetic operators consist of unary and binary operators. They are:

+	- Unary plus	+1.23
-	- Unary minus	-[COS[30]]
+	- Binary addition	#10001=#10001+5
-	- Binary subtraction	#10001=#10001-1
*	- Multiplication	#10001=#10002*#10003
/	- Division	#10001=#10002/4
MOD	- Remainder	#10001=27 MOD 20 (#10001 contains 7)

Logical Operators

Logical operators are operators that work on binary bit values. Macro variables are floating point numbers. When logical operators are used on macro variables, only the integer portion of the floating point number is used. The logical operators are:

OR - logically OR two values together

XOR - Exclusively OR two values together

AND - Logically AND two values together

Examples:

```
%  
#10001=1.0 ;  
#10002=2.0 ;
```

```
#10003=#10001 OR #10002 ;
%
```

Here the variable #10003 will contain 3.0 after the OR operation.

```
%  
#10001=5.0 ;  
#10002=3.0 ;  
IF [[#10001 GT 3.0] AND [#10002 LT 10]] GOTO1 ;  
%
```

Here control transfers to block 1 because #10001 GT 3.0 evaluates to 1.0 and #10002 LT 10 evaluates to 1.0, thus 1.0 AND 1.0 is 1.0 (TRUE) and the GOTO occurs.



NOTE:

To achieve your desired results, be very careful when you use logical operators.

Expressions

Expressions are defined as any sequence of variables and operators surrounded by the square brackets [and]. There are two uses for expressions: conditional expressions or arithmetic expressions. Conditional expressions return FALSE (0.0) or TRUE (any non zero) values. Arithmetic expressions use arithmetic operators along with functions to determine a value.

Arithmetic Expressions

An arithmetic expression is any expression using variables, operators, or functions. An arithmetic expression returns a value. Arithmetic expressions are usually used in assignment statements, but are not restricted to them.

Examples of Arithmetic expressions:

```
%  
#10001=#10045*#10030 ;  
#10001=#10001+1 ;  
X[#10005+COS[#10001]] ;  
#[#10200+#10013]=0 ;  
%
```

Conditional Expressions

In the Haas control, all expressions set a conditional value. The value is either 0.0 (FALSE) or the value is nonzero (TRUE). The context in which the expression is used determines if the expression is a conditional expression. Conditional expressions are used in the IF and WHILE statements and in the M99 command. Conditional expressions can make use of Boolean operators to help evaluate a TRUE or FALSE condition.

The M99 conditional construct is unique to the Haas control. Without macros, M99 in the Haas control has the ability to branch unconditionally to any line in the current subprogram by placing a P code on the same line. For example:

```
N50 M99 P10 ;
```

branches to line N10. It does not return control to the calling subprogram. With macros enabled, M99 can be used with a conditional expression to branch conditionally. To branch when variable #10000 is less than 10 we could code the above line as follows:

```
N50 [#10000 LT 10] M99 P10 ;
```

In this case, the branch occurs only when #10000 is less than 10, otherwise processing continues with the next program line in sequence. In the above, the conditional M99 can be replaced with

```
N50 IF [#10000 LT 10] GOTO10 ;
```

Assignment Statements

Assignment statements let you modify variables. The format of the assignment statement is:

```
<expression>=<expression>
```

The expression on the left of the equal sign must always refer to a macro variable, whether directly or indirectly. This macro initializes a sequence of variables to any value. This example uses both direct and indirect assignments.

```
%  
O50001 (INITIALIZE A SEQUENCE OF VARIABLES) ;  
N1 IF [#2 NE #0] GOTO2 (B=base variable) ;  
#3000=1 (Base variable not given) ;
```

```

N2 IF [#19 NE #0] GOTO3 (S=size of array) ;
#3000=2 (Size of array not given) ;
N3 WHILE [#19 GT 0] DO1 ;
#19=#19-1 (Decrement count) ;
#[#2+#19]=#22 (V=value to set array to) ;
END1 ;
M99 ;
%
```

You could use the above macro to initialize three sets of variables as follows:

```

%
G65 P300 B101. S20 (INIT 101..120 TO #0) ;
G65 P300 B501. S5 V1. (INIT 501..505 TO 1.0) ;
G65 P300 B550. S5 V0 (INIT 550..554 TO 0.0) ;
%
```

The decimal point in B101., etc. would be required.

Control Statements

Control statements allow the programmer to branch, both conditionally and unconditionally. They also provide the ability to iterate a section of code based on a condition.

Unconditional Branch (GOTOnnn and M99 Pnnnn)

In the Haas control, there are two methods of branching unconditionally. An unconditional branch will always branch to a specified block. M99 P15 will branch unconditionally to block number 15. The M99 can be used whether or not macros is installed and is the traditional method for branching unconditionally in the Haas control. GOTO15 does the same as M99 P15. In the Haas control, a GOTO command can be used on the same line as other G-codes. The GOTO is executed after any other commands like M codes.

Computed Branch (GOTO#n and GOTO [expression])

Computed branching allows the program to transfer control to another line of code in the same subprogram. The control can compute the block while the program runs, using the GOTO [expression] form, or it can pass the block in through a local variable, as in the GOTO#n form.

The GOTO rounds the variable or expression result that is associated with the Computed branch. For instance, if variable #1 contains 4.49 and the program contains a GOTO#1 command, the control attempts to transfer to a block that contains N4. If #1 contains 4.5, then the control transfers to a block that contains N5.

Example: You could develop this code skeleton into a program that adds serial numbers to parts:

```
%  
O50002 (COMPUTED BRANCHING) ;  
(D=Decimal digit to engrave) ;  
;  
IF [[#7 NE #0] AND [#7 GE 0] AND [#7 LE 9]] GOTO99 ;  
#3000=1 (Invalid digit) ;  
;  
N99;  
#7=FIX[#7] (Truncate any fractional part) ;  
;  
GOTO#7 (Now engrave the digit) ;  
;  
N0 (Do digit zero) ;  
M99 ;  
;  
N1 (Do digit one) ;  
;  
M99 ;  
%
```

With the above subprogram, you would use this call to engrave the fifth digit:

```
G65 P9200 D5 ;
```

Computed GOTOS using expression could be used to branch processing based on the results of reading hardware inputs. For example:

```
%  
GOTO [[#1030*2]+#1031] ;  
N0(1030=0, 1031=0) ;  
...M99 ;  
N1(1030=0, 1031=1) ;  
...M99 ;  
N2(1030=1, 1031=0) ;  
...M99 ;  
N3(1030=1, 1031=1) ;  
...M99 ;  
%
```

#1030 and #1031.

Conditional Branch (IF and M99 Pnnnn)

Conditional branching allows the program to transfer control to another section of code within the same subprogram. Conditional branching can only be used when macros are enabled. The Haas control allows two similar methods for accomplishing conditional branching:

```
IF [<conditional expression>] GOTON
```

As discussed, <conditional expression> is any expression that uses any of the six Boolean operators EQ, NE, GT, LT, GE, or LE. The brackets surrounding the expression are mandatory. In the Haas control, it is not necessary to include these operators. For example:

```
IF [#1 NE 0.0] GOTO5 ;
```

could also be:

```
IF [#1] GOTO5 ;
```

In this statement, if the variable #1 contains anything but 0.0, or the undefined value #0, then branching to block 5 occurs; otherwise, the next block is executed.

In the Haas control, a <conditional expression> is also used with the M99 Pnnnn format. For example:

```
G00 X0 Y0 [#1EQ#2] M99 P5;
```

Here, the conditional is for the M99 portion of the statement only. The machine tool is instructed to go to X0, Y0 whether or not the expression evaluates to True or False. Only the branch, M99, is executed based on the value of the expression. It is recommended that the IF GOTO version be used if portability is desired.

Conditional Execution (IF THEN)

Execution of control statements can also be achieved by using the IF THEN construct. The format is:

```
IF [<conditional expression>] THEN <statement> ;
```



NOTE:

To preserve compatibility with FANUC syntax `THEN` may not be used with `GOTO`.

This format is traditionally used for conditional assignment statements such as:

```
IF [#590 GT 100] THEN #590=0.0 ;
```

Variable #590 is set to zero when the value of #590 exceeds 100.0. In the Haas control, if a conditional evaluates to FALSE (0.0), then the remainder of the IF block is ignored. This means that control statements can also be conditioned so that we could write something like:

```
IF [#1 NE #0] THEN G01 X#24 Y#26 F#9 ;
```

This executes a linear motion only if variable #1 has been assigned a value. Another example is:

```
IF [#1 GE 180] THEN #101=0.0 M99 ;
```

This says that if variable #1 (address A) is greater than or equal to 180, then set variable #101 to zero and return from the subprogram.

Here is an example of an IF statement that branches if a variable has been initialized to contain any value. Otherwise, processing continues and an alarm is generated. Remember, when an alarm is generated, program execution is halted.

```
%  
N1 IF [#9NE#0] GOTO3 (TEST FOR VALUE IN F) ;  
N2 #3000=11 (NO FEED RATE) ;  
N3 (CONTINUE) ;  
%
```

Iteration/Looping (WHILE DO END)

Essential to all programming languages is the ability to execute a sequence of statements a given number of times or to loop through a sequence of statements until a condition is met. Traditional G coding allows this with the use of the L address. A subprogram can be executed any number of times by using the L address.

```
M98 P2000 L5 ;
```

This is limited since you cannot terminate execution of the subprogram on condition. Macros allow flexibility with the WHILE-DO-END construct. For example:

```
%  
WHILE [<conditional expression>] DOn ;  
<statements> ;  
ENDn ;  
%
```

This executes the statements between `DOn` and `ENDn` as long as the conditional expression evaluates to True. The brackets in the expression are necessary. If the expression evaluates to False, then the block after `ENDn` is executed next. `WHILE` can be abbreviated to `WH`. The `DOn-ENDn` portion of the statement is a matched pair. The value of `n` is 1-3. This means that there can be no more than three nested loops per subprogram. A nest is a loop within a loop.

Although nesting of `WHILE` statements can only be up to three levels, there really is no limit since each subprogram can have up to three levels of nesting. If there is a need to nest to a level greater than 3, then the segment containing the three lowest levels of nesting can be made into a subprogram thus overcoming the limitation.

If two separate `WHILE` loops are in a subprogram, they can use the same nesting index. For example:

```
%  
#3001=0 (WAIT 500 MILLISECONDS) ;  
WH [#3001 LT 500] DO1 ;  
END1 ;  
<Other statements>  

```

You can use `GOTO` to jump out of a region encompassed by a `DO-END`, but you cannot use a `GOTO` to jump into it. Jumping around inside a `DO-END` region using a `GOTO` is allowed.

An infinite loop can be executed by eliminating the `WHILE` and expression. Thus,

```
%  
DO1 ;  
<statements>
```

```
END1 ;  
%
```

executes until the `RESET` key is pressed.



CAUTION: *The following code can be confusing:*

```
%  
WH [#1] D01 ;  
END1 ;  
%
```

In this example, an alarm results indicating no `Then` was found; `Then` refers to the `D01`. Change `D01` (zero) to `DO1` (letter O).

6.13.11 Communication With External Devices - DPRNT[]

Macros allow additional capabilities to communicate with peripheral devices. With user provided devices you can digitize parts, provide runtime inspection reports, or synchronize controls.

Formatted Output

The `DPRNT` statement lets programs send formatted text to the serial port. `DPRNT` can print any text and any variable to the serial port. The form of the `DPRNT` statement is as follows:

```
DPRNT [<text> <#nnnn[wf]>... ] ;
```

`DPRNT` must be the only command in the block. In the previous example, `<text>` is any character from A to Z or the letters (+,-,/*, and the space). When an asterisk is output, it is converted to a space. The `<#nnnn[wf]>` is a variable followed by a format. The variable number can be any macro variable. The format `[wf]` is required and consists of two digits within square brackets. Remember that macro variables are real numbers with a whole part and a fractional part. The first digit in the format designates the total places reserved in the output for the whole part. The second digit designates the total places reserved for the fractional part. The control can use any number from 0-9 for both whole and fractional parts.

A decimal point is printed out between the whole part and the fractional part. The fractional part is rounded to the least significant place. When zero places are reserved for the fractional part, then no decimal point is printed out. Trailing zeros are printed if there is a fractional part. At least one place is reserved for the whole part, even when a zero is used. If the value of the whole part has fewer digits than have been reserved, then leading spaces are output. If the value of the whole part has more digits than has been reserved, then the field is expanded so that these numbers are printed.

The control sends a carriage return after every DPRNT block.

DPRNT[] Example:

Code	Output
#1= 1.5436 ;	
DPRNT [X#1[44]*Z#1[03]*T#1[40]] ;	X1.5436 Z 1.544 T 1
DPRNT [***MEASURED*INSIDE*DIAMETER** *] ;	MEASURED INSIDE DIAMETER
DPRNT [] ;	(no text, only a carriage return)
#1=123.456789 ;	
DPRNT [X-#1[35]] ;	X-123.45679 ;

DPRNT[] Settings

Setting 261 determines the destination for DPRNT statements. You can choose to output them to a file, or to a TCP port. Settings 262 and 263 specify the destination for DPRNT output. Refer to the Settings section of this manual for more information.

Execution

DPRNT statements are executed at look-ahead time. This means that you must be careful about where the DPRNT statements appear in the program, particularly if the intent is to print out.

G103 is useful for limiting lookahead. If you wanted to limit look-ahead interpretation to one block, you would include this command at the start of your program: This causes the control to look ahead (2) blocks.

```
G103 P1 ;
```

To cancel the lookahead limit, change the command to G103 P0. G103 cannot be used when cutter compensation is active.

Editing

Improperly structured or improperly placed macro statements will generate an alarm. Be careful when editing expressions; brackets must be balanced.

The DPRNT [] function can be edited much like a comment. It can be deleted, moved as a whole item, or individual items within the bracket can be edited. Variable references and format expressions must be altered as a whole entity. If you wanted to change [24] to [44], place the cursor so that [24] is highlighted, enter [44] and press [**ENTER**]. Remember, you can use the jog handle to maneuver through long DPRNT [] expressions.

Addresses with expressions can be somewhat confusing. In this case, the alphabetic address stands alone. For instance, this block contains an address expression in X:

```
G01 G90 X [COS [90]] Y3.0 (CORRECT) ;
```

Here, the X and brackets stand-alone and are individually editable items. It is possible, through editing, to delete the entire expression and replace it with a floating-point constant.

```
G01 G90 X 0 Y3.0 (WRONG) ;
```

The above block will result in an alarm at runtime. The correct form looks as follows:

```
G01 G90 X0 Y3.0 (CORRECT) ;
```

**NOTE:**

There is no space between the X and the Zero (0). REMEMBER when you see an alpha character standing alone it is an address expression.

6.13.12 G65 Macro Subprogram Call Option (Group 00)

G65 is the command that calls a subprogram with the ability to pass arguments to it. The format follows:

```
G65 Pnnnnn [Lnnnn] [arguments] ;
```

Arguments italicized in square brackets are optional. See the Programming section for more details on macro arguments.

The G65 command requires a **P** address corresponding to a program number currently located in the control's drive or path to a program. When the **L** address is used the macro call is repeated the specified number of times.

When a subprogram is called, the control looks for the subprogram on the active drive or the path to the program. If the subprogram cannot be located on the active drive, the control looks in the drive designated by Setting 251. Refer to the Setting Up Search Locations section for more information on subprogram searching. An alarm occurs if the control does not find the subprogram.

In Example 1, subprogram 1000 is called once without conditions passed to the subprogram. G65 calls are similar to, but not the same as, M98 calls. G65 calls can be nested up to 9 times, which means, program 1 can call program 2, program 2 can call program 3 and program 3 can call program 4.

Example 1:

```
%  
G65 P1000 (Call subprogram O01000 as a macro) ;  
M30 (Program stop) ;  
O01000 (Macro Subprogram) ;  
...  
M99 (Return from Macro Subprogram) ;  
%
```

In Example 2, the program LightHousing.nc is called using the path that it is in.

Example 2:

```
%
```

```
G65 P15 A1. B1.;  
G65 (/Memory/LightHousing.nc) A1. B1.;
```



NOTE:

Paths are case sensitive.

In Example 3, subprogram 9010 is designed to drill a sequence of holes along a line whose slope is determined by the X and Y arguments that are passed to it in the G65 command line. The Z drill depth is passed as Z, the feed rate is passed as F, and the number of holes to be drilled is passed as T. The line of holes is drilled starting from the current tool position when the macro subprogram is called.

Example 3:



NOTE:

The subprogram program O09010 should reside on the active drive or on a drive designated by Setting 252.

```
%  
G00 G90 X1.0 Y1.0 Z.05 S1000 M03 (Position tool) ;  
G65 P9010 X.5 Y.25 Z.05 F10. T10 (Call O09010) ;  
M30 ;  
O09010 (Diagonal hole pattern) ;  
F#9 (F=Feedrate) ;  
WHILE [#20 GT 0] D01 (Repeat T times) ;  
G91 G81 Z#26 (Drill To Z depth) ;  
#20=#20-1 (Decrement counter) ;  
IF [#20 EQ 0] GOTO5 (All holes drilled) ;  
G00 X#24 Y#25 (Move along slope) ;  
N5 END1 ;  
M99 (Return to caller) ;  
%
```

6.13.13 Aliasing

Aliased codes are user defined G and M-codes that reference a macro program. There are 10 G alias codes and 10 M alias codes available to users. Program numbers 9010 through 9019 are reserved for G-code aliasing and 9000 through 9009 are reserved for M-code aliasing.

Aliasing is a means of assigning a G-code or M-code to a G65 P##### sequence. For instance, in the previous Example 2, it would be easier to write:

```
G06 X.5 Y.25 Z.05 F10. T10 ;
```

When aliasing, variables can be passed with a G-code; variables cannot be passed with an M-code.

Here, an unused G-code has been substituted, G06 for G65 P9010. In order for the previous block to work, the value associated with subprogram 9010 must be set to 06. Refer to the Setting Aliases section for how to setup aliases.

**NOTE:**

G00, G65, G66, and G67 cannot be aliased. All other codes between 1 and 255 can be used for aliasing.

If a macro call subprogram is set to a G-code and the subprogram is not in memory, then an alarm is given. Refer to the G65 Macro Subprogram Call section on page 291 on how to locate the subprogram. An alarm occurs if the sub-program is not found.

Setting Aliases

The G-code or M-code alias setup is done on the Alias Codes window. To setup an alias:

1. Press **[SETTING]** and navigate to the **Alias Codes** tab.
2. Press **[EMERGENCY STOP]** on the control.
3. Using the cursor keys select the M or G Macro Call to be used.
4. Enter the number of the G-code or M-code you want to alias. For example, if you want to alias G06 type 06.
5. Press **[ENTER]**.
6. Repeat steps 3 - 5 for other aliased G or M-codes.
7. Release the **[EMERGENCY STOP]** on the control.

Setting an alias value to 0 disables aliasing for the associated subprogram.

F6.22: Alias Codes Window

Settings And Graphics					
Graphics	Settings	Network	Notifications	Rotary	Alias Codes
M-Codes & G-Codes Program Aliases					Value
M MACRO CALL 09000					0
M MACRO CALL 09001					0
M MACRO CALL 09002					0
M MACRO CALL 09003					0
M MACRO CALL 09004					0
M MACRO CALL 09005					0
M MACRO CALL 09006					0
M MACRO CALL 09007					0
M MACRO CALL 09008					0
M MACRO CALL 09009					0
G MACRO CALL 09010					0
G MACRO CALL 09011					0
G MACRO CALL 09012					0
G MACRO CALL 09013					0
G MACRO CALL 09014					0
G MACRO CALL 09015					0
G MACRO CALL 09016					0
G MACRO CALL 09017					0
G MACRO CALL 09018					0
G MACRO CALL 09019					0

6.13.14 More Information Online

For updated information, including tips, tricks, maintenance procedures, and more, visit the Haas Service page at www.HaasCNC.com.

For the most current Operator's and Service Manuals scan the code below with your mobile device:



6.14 Pallet Pool M-Codes

The following are the M-codes used by the pallet pool.

6.14.1 M46 Qn Pmm Jump to Line

Jump to line mm in the current program if pallet n is loaded, otherwise go to the next block.

6.14.2 M48 Validate That The Current Program is Appropriate for Loaded Pallet

Checks in the Pallet Schedule Table that the current program is assigned to the loaded pallet. If the current program is not in the list or the loaded pallet is incorrect for the program, an alarm is generated. **M48** can be in a program listed in the PST, but never in a subroutine of the PST program. An alarm will occur if **M48** is incorrectly nested.

6.14.3 M50 Pallet Change Sequence

***P** - Pallet number

*indicates optional

This M-code is used to call a pallet change sequence. An **M50** with a **P** command will call a specific pallet. **M50 P3** will change to pallet 3, commonly used with Pallet Pool machines. Refer to the Pallet Changer section of the manual.

6.14.4 M199 Pallet / Part Load or Program End

M199 takes the place of an **M30** or **M99** at the end of a program. When running in Memory or MDI mode, pressing **Cycle Start** to run the program, the **M199** will behave the same as an **M30**. It will stop and rewind the program back to the beginning. While running in Pallet Change mode, pressing **INSERT** while on the Pallet Schedule Table to run a program, the **M199** behaves the same as an **M50 + M99**. It will end the program, get the next scheduled pallet and associated program, then continue to run until all scheduled pallets are completed.

Chapter 7: G-codes

7.1 Introduction

This chapter gives detailed descriptions of the G-codes that you use to program your machine.

For the most current G-code information scan the code below with your mobile device.

F7.1: Mill - G-Codes Guide



7.1.1 List of G-codes



CAUTION:

The sample programs in this manual have been tested for accuracy, but they are for illustrative purposes only. The programs do not define tools, offsets, or materials. They do not describe workholding or other fixturing. If you choose to run a sample program on your machine, do so in Graphics mode. Always follow safe machining practices when you run an unfamiliar program.



NOTE:

The sample programs in this manual represent a very conservative programming style. The samples are intended to demonstrate safe and reliable programs, and they are not necessarily the fastest or most efficient way to operate a machine. The sample programs use G-codes that you might choose not to use in more efficient programs.

Code	Description	Group	Page
G00	Rapid Motion Positioning	01	308
G01	Linear Interpolation Motion	01	309
G02	Circular Interpolation Motion CW	01	311
G03	Circular Interpolation Motion CCW	01	311
G04	Dwell	00	319
G09	Exact Stop	00	320
G10	Set Offsets	00	320
G12	Circular Pocket Milling CW	00	321
G13	Circular Pocket Milling CCW	00	321
G17	XY Plane Selection	02	324
G18	XZ Plane Selection	02	324
G19	YZ Plane Selection	02	324
G20	Select Inches	06	324
G21	Select Metric	06	324
G28	Return To Machine Zero Point	00	324
G29	Return From Reference Point	00	325
G31	Feed Until Skip	00	325
G35	Automatic Tool Diameter Measurement	00	327
G36	Automatic Work Offset Measurement	00	329
G37	Automatic Tool Offset Measurement	00	331
G40	Cutter Compensation Cancel	07	332
G41	2D Cutter Compensation Left	07	332

Code	Description	Group	Page
G42	2D Cutter Compensation Right	07	332
G43	Tool Length Compensation + (Add)	08	333
G44	Tool Length Compensation - (Subtract)	08	333
G47	Text Engraving	00	333
G49	G43/G44/G143 Cancel	08	339
G50	Cancel Scaling	11	339
G51	Scaling	11	339
G52	Set Work Coordinate System	00 or 12	344
G53	Non-Modal Machine Coordinate Selection	00	344
G54	Select Work Coordinate System #1	12	344
G55	Select Work Coordinate System #2	12	344
G56	Select Work Coordinate System #3	12	344
G57	Select Work Coordinate System #4	12	344
G58	Select Work Coordinate System #5	12	344
G59	Select Work Coordinate System #6	12	344
G60	Uni-Directional Positioning	00	345
G61	Exact Stop Mode	15	345
G64	G61 Cancel	15	345
G65	Macro Subprogram Call Option	00	345
G68	Rotation	16	345
G69	Cancel G68 Rotation	16	349
G70	Bolt Hole Circle	00	349
G71	Bolt Hole Arc	00	350

Code	Description	Group	Page
G72	Bolt Holes Along an Angle	00	350
G73	High-Speed Peck Drilling Canned Cycle	09	351
G74	Reverse Tap Canned Cycle	09	352
G76	Fine Boring Canned Cycle	09	353
G77	Back Bore Canned Cycle	09	354
G80	Canned Cycle Cancel	09	357
G81	Drill Canned Cycle	09	357
G82	Spot Drill Canned Cycle	09	358
G83	Normal Peck Drilling Canned Cycle	09	360
G84	Tapping Canned Cycle	09	362
G85	Boring Canned Cycle	09	364
G86	Bore and Stop Canned Cycle	09	364
G89	Bore In, Dwell, Bore Out Canned Cycle	09	365
G90	Absolute Position Command	03	366
G91	Incremental Position Command	03	366
G92	Set Work Coordinate Systems Shift Value	00	366
G93	Inverse Time Feed Mode	05	367
G94	Feed Per Minute Mode	05	367
G95	Feed per Revolution	05	367
G98	Canned Cycle Initial Point Return	10	364
G99	Canned Cycle R Plane Return	10	369
G100	Cancel Mirror Image	00	370
G101	Enable Mirror Image	00	370

Code	Description	Group	Page
G103	Limit Block Buffering	00	371
G107	Cylindrical Mapping	00	372
G110	#7 Coordinate System	12	372
G111	#8 Coordinate System	12	372
G112	#9 Coordinate System	12	372
G113	#10 Coordinate System	12	372
G114	#11 Coordinate System	12	372
G115	#12 Coordinate System	12	372
G116	#13 Coordinate System	12	372
G117	#14 Coordinate System	12	372
G118	#15 Coordinate System	12	372
G119	#16 Coordinate System	12	372
G120	#17 Coordinate System	12	372
G121	#18 Coordinate System	12	372
G122	#19 Coordinate System	12	372
G123	#20 Coordinate System	12	372
G124	#21 Coordinate System	12	372
G125	#22 Coordinate System	12	372
G126	#23 Coordinate System	12	372
G127	#24 Coordinate System	12	372
G128	#25 Coordinate System	12	372
G129	#26 Coordinate System	12	372
G136	Automatic Work Offset Center Measurement	00	373

Code	Description	Group	Page
G141	3D+ Cutter Compensation	07	375
G143	5-Axis Tool Length Compensation +	08	378
G150	General Purpose Pocket Milling	00	379
G154	Select Work Coordinates P1-P99	12	387
G156	Broaching Canned Cycle	09	389
G167	Modify Setting	00	392
G174	CCW Non-Vertical Rigid Tap	00	393
G184	CW Non-Vertical Rigid Tap	00	393
G187	Setting the Smoothness Level	00	394
G234	Tool Center Point Control (TCPC)	08	394
G253	G253 Orient Spindle Normal To Feature Coordinate System	00	400
G254	Dynamic Work Offset (DWO)	23	400
G255	Cancel Dynamic Work Offset (DWO)	23	404
G266	Visible Axes Linear Rapid % Motion	00	405
G268 /	Enable Feature Coordinate System	02	405
G269	Disable Feature Coordinate System	02	405

About G-codes

G-codes tell the machine tool what type of action to do, such as:

- Rapid moves
- Move in a straight line or arc
- Set tool information
- Use letter addressing
- Define axis and beginning and ending positions
- Pre-set series of moves that bore a hole, cut a specific dimension, or a contour (canned cycles)

G-code commands are either modal or non-modal. A modal G-code stays in effect until the end of the program or until you command another G-code from the same group. A non-modal G-code affects only the line it is in; it does not affect the next program line. Group 00 codes are non-modal; the other groups are modal.

For a description of basic programming, refer to the basic programming section of the Programming chapter, starting on page **186**.

**NOTE:**

The Visual Programming System (VPS) is an optional programming mode that lets you program part features without manually writing G-code.

**NOTE:**

A program block can contain more than one G-code, but you cannot put two G-codes from the same group in the same program block.

Canned Cycles

Canned cycles are G-codes that do repetitive operations such as drilling, tapping, and boring. You define a canned cycle with alphabetic address codes. While the canned cycle is active, the machine does the defined operation every time you command a new position, unless you specify not to.

Using Canned Cycles

You can program canned cycle X and Y positions in either absolute (G90) or incremental (G91).

Example:

```
%  
G81 G99 Z-0.5 R0.1 F6.5 (This drills one hole);  
(at the present location);  
G91 X-0.5625 L9 (This drills 9 more holes 0.5625);  
(equally spaced in the X-negative direction);  
%
```

There are (3) possible ways for a canned cycle to behave in the block in which you command it:

- If you command an X/Y position in the same block as the canned cycle G-code, the canned cycle executes. If Setting 28 is **OFF**, the canned cycle executes in the same block only if you command an X/Y position in that block.

- If Setting 28 is **ON**, and you command a canned cycle G-code with or without an X/Y position in the same block, the canned cycle executes in that block—either at the position where you commanded the canned cycle, or at the new X/Y position.
- If you include a loop count of zero (**L0**) in the same block as the canned cycle G-code, the canned cycle does not execute in that block. The canned cycle does not execute regardless of Setting 28 and whether or not the block also contains an X/Y position.

**NOTE:**

Unless otherwise noted, the program examples given here assume that Setting 28 is ON.

When a canned cycle is active, it repeats at every new X/Y position in the program. In the example above, with each incremental move of -0.5625 in the X axis, the canned cycle (G81) drills a 0.5" deep hole. The **L** address code in the incremental position command (G91) repeats this operation (9) times.

Canned cycles operate differently depending on whether incremental (G91) or absolute (G90) positioning is active. Incremental motion in a canned cycle is often useful, because it lets you use a loop (**L**) count to repeat the operation with an incremental X or Y move between cycles.

Example:

```
%  
X1.25 Y-0.75 (center location of bolt hole pattern) ;  
G81 G99 Z-0.5 R0.1 F6.5 L0;  
(L0 on the G81 line will not drill a hole) ;  
G70 I0.75 J10. L6 (6-hole bolt hole circle) ;  
%
```

The R plane value and the Z depth value are important canned cycle address codes. If you specify these addresses in a block with XY commands, the control does the XY move, and it does all of the subsequent canned cycles with the new R or Z value.

The X and Y positioning in a canned cycle is done with rapid moves.

G98 and G99 change the way the canned cycles operate. When G98 is active, the Z-Axis will return to the initial start plane at the completion of each hole in the canned cycle. This allows for positioning up and around areas of the part and/or clamps and fixtures.

When G99 is active, the Z-Axis returns to the R (rapid) plane after each hole in the canned cycle for clearance to the next XY location. Changes to the G98/G99 selection can also be made after the canned cycle is commanded, which will affect all later canned cycles.

A *P* address is an optional command for some canned cycles. This is a programmed pause at the bottom of the hole to help break chips, provide a smoother finish, and relieve any tool pressure to hold closer tolerance.

**NOTE:**

*A *P* address used for one canned cycle is used in others unless canceled (G00, G01, G80 or the [RESET] button).*

You must define an *S* (spindle speed) command in or before the canned cycle G-code block.

Tapping in a canned cycle needs a feed rate calculated. The feed formula is:

Spindle speed divided by threads per inch of the tap = feedrate in inches per minute

The metric version of the feed formula is:

RPM times metric pitch = feedrate in mm per minute

Canned cycles also benefit from the use of Setting 57. If this setting is **ON**, the machine stops after the X/Y rapids before it moves the Z Axis. This is useful to avoid nicking the part when the tool exits the hole, especially if the R plane is close to the part surface.

**NOTE:**

*The *Z*, *R*, and *F* addresses are required data for all canned cycles.*

Canceling a Canned Cycle

G80 cancels all canned cycles. G00 or G01 code also cancel a canned cycle. A canned cycle stays active until G80, G00, or G01 cancels it.

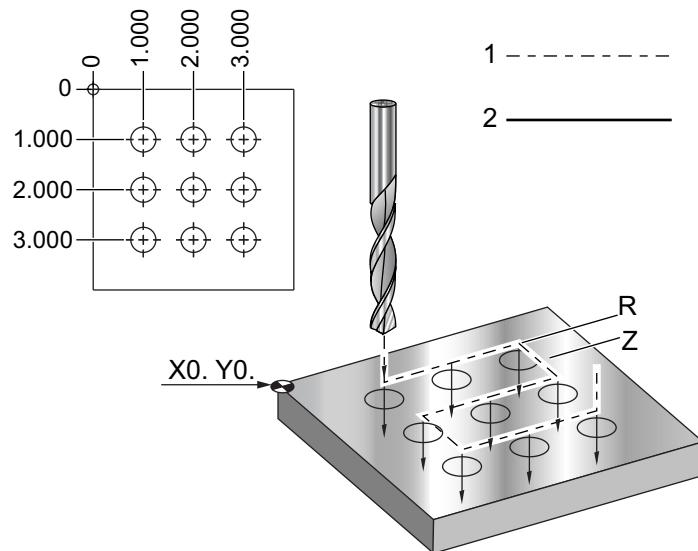
Looping Canned Cycles

This is an example of a program that uses an incrementally looped drilling canned cycle.

**NOTE:**

The sequence of drilling used here is designed to save time and to follow the shortest path from hole to hole.

F7.2: G81 Drilling Canned Cycle: [R] R Plane, [Z] Z Plane, [1] Rapid, [2] Feed.



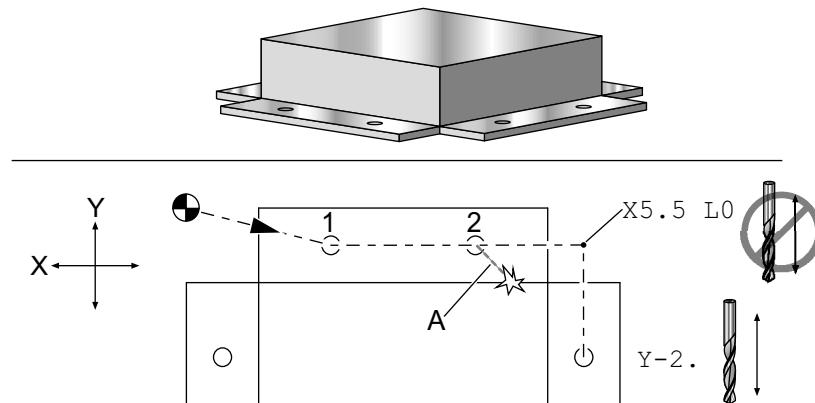
```
%  
O60810 (Drilling grid plate 3x3 holes) ;  
(G54 X0 Y0 is at the top-left of part) ;  
(Z0 is at the top of the part) ;  
(T1 is a drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X1.0 Y-1.0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G81 Z-1.5 F15. R.1 (Begin G81 & drill 1st hole) ;  
G91 X1.0 L2 (Drill 1st row of holes) ;  
G90 Y-2.0 (1st hole of 2nd row) ;  
G91 X-1.0 L2 (2nd row of holes) ;  
G90 Y-3.0 (1st hole of 3rd row) ;  
G91 X1.0 L2 (3rd row of holes) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

X/Y Plane Obstacle Avoidance in a Canned Cycle

If you put an `L0` on a canned cycle line, you can make an X, Y move without the Z-Axis canned operation. This is a good way to avoid obstacles in the X/Y plane.

Consider a 6" square aluminum block, with a 1" by 1" deep flange on each side. The print calls for two holes centered on each side of the flange. You use a `G81` canned cycle to make the holes. If you simply command the hole positions in the drill canned cycle, the control takes the shortest path to the next hole position, which puts the tool through the corner of the workpiece. To prevent this, command a position past the corner, so the move to the next hole position does not go through the corner. The drill canned cycle is active, but you do not want a drill cycle at that position, so use `L0` in this block.

- F7.3:** Canned Cycle Obstacle Avoidance. The program drills holes [1] and [2], then moves to X5.5. Because of the `L0` address in this block, there is no drill cycle in this position. Line [A] shows the path that the canned cycle would follow without the obstacle avoidance line. The next move is in the Y Axis only to the position of the third hole, where the machine does another drill cycle.



```
%  
O60811 (X Y OBSTACLE AVOIDANCE) ;  
(G54 X0 Y0 is at the top-left of part) ;  
(Z0 is at the top of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y-0.5(Rapid to first position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 M08 (Activate tool offset 1) ;  
(Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G81 Z-2. R-0.9 F15. (Begin G81 & Drill 1st hole) ;  
X4. (Drill 2nd hole) ;  
X5.5 L0 (Corner avoidance) ;  
Y-2. (3rd hole) ;
```

```
Y-4. (4th hole) ;
Y-5.5 L0 (Corner avoidance) ;
X4. (5th hole) ;
X2. (6th hole) ;
X0.5 L0 (Corner avoidance) ;
Y-4. (7th hole) ;
Y-2. (8th hole) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

G00 Rapid Motion Positioning (Group 01)

***X** - Optional X-Axis motion command

***Y** - Optional Y-Axis motion command

***Z** - Optional Z-Axis motion command

***A** - Optional A-Axis motion command

***B** - Optional B-Axis motion command

***C** - Optional C-axis motion command

* **E** - Optional code to specify the rapid rate of the block as a percent.

*indicates optional

G00 is used to move the machine axes at the maximum speed. It is primarily used to quickly position the machine to a given point before each feed (cutting) command. This G code is modal, so a block with G00 causes all following blocks to be rapid motion until another Group 01 code is specified.

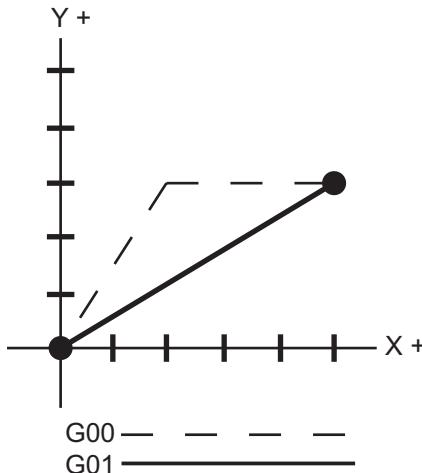
A rapid move also cancels an active canned cycle, just like G80 does.



NOTE:

Generally, rapid motion will not be in a single straight line. Each axis specified moves at its maximum speed, but all axes will not necessarily complete their motions at the same time. The machine waits until all motions are complete before starting the next command.

F7.4: G00 Multi-linear Rapid Motion



Setting 57 (Exact Stop Canned X-Y) can change how closely the machine waits for a precise stop before and after a rapid move.

G01 Linear Interpolation Motion (Group 01)

F - Feedrate

- * **X** - X-Axis motion command
- * **Y** - Y-Axis motion command
- * **Z** - Z-Axis motion command
- * **A** - A-Axis motion command
- * **B** - B-Axis motion command
- * **C** - C-axis motion command
- * **,R** - Radius of the arc
- * **,C** - Chamfer distance

*indicates optional

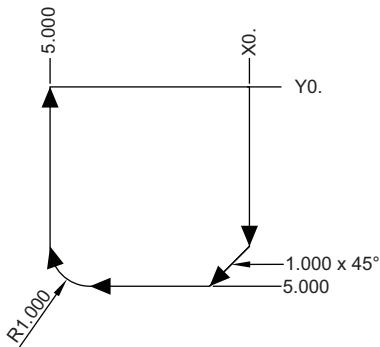
G01 moves the axes at a commanded feed rate. It is primarily used to cut the workpiece. A G01 feed can be a single axis move or a combination of the axes. The rate of axes movement is controlled by feedrate (F) value. This F value can be in units (inch or metric) per minute (G94) or per spindle revolution (G95), or time to complete the motion (G93). The feedrate value (F) can be on the current program line, or a previous line. The control will always use the most recent F value until another F value is commanded. If in G93, an F value is used on each line. Refer also to G93.

G01 is a modal command, which means that it will stay in effect until canceled by a rapid command such as G00 or a circular motion command like G02 or G03.

Once a G01 is started all programmed axes move and reach the destination at the same time. If an axis is not capable of the programmed feedrate the control will not proceed with the G01 command and an alarm (max feedrate exceeded) will be generated.

Corner Rounding and Chamfering Example

F7.5: Corner Rounding and Chamfering Example #1



```
%  
O60011 (G01 CORNER ROUNDING & CHAMFER) ;  
(G54 X0 Y0 is at the top-right of part) ;  
(Z0 is on top of the part) ;  
(T1 is an end mill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.5 F20. (Feed to cutting depth) ;  
Y-5. ,C1. (Chamfer) ;  
X-5. ,R1. (Corner-round) ;  
Y0 (Feed to Y0.) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

A chamfer block or a corner-rounding block can be automatically inserted between two linear interpolation blocks by specifying ,C (chamfering) or ,R (corner rounding). There must be a terminating linear interpolation block after the beginning block (a G04 pause may intervene).

These two linear interpolation blocks specify a corner of intersection. If the beginning block specifies a , C, the value after the , C is the distance from the intersection to where the chamfer begins, and also the distance from the intersection to where the chamfer ends. If the beginning block specifies an , R, the value after the , R is the radius of a circle tangent to the corner at two points: the beginning of the corner-rounding arc and the endpoint of that arc. There can be consecutive blocks with chamfering or corner rounding specified. There must be movement on the two axes specified by the selected plane, whether the active plane is XY (G17), XZ (G18) or YZ (G19).

G02 CW / G03 CCW Circular Interpolation Motion (Group 01)

F - Feedrate

- ***I** - Distance along X Axis to center of circle
- ***J** - Distance along Y Axis to center of circle
- ***K** - Distance along Z Axis to center of circle
- ***R** - Radius of circle
- ***X** - X-Axis motion command
- ***Y** - Y-Axis motion command
- ***Z** - Z-Axis motion command
- ***A** - A-Axis motion command

*indicates optional



NOTE:

I, J and K is the preferred method to program a radius. R is suitable for general radii.

These G codes are used to specify circular motion. Two axes are necessary to complete circular motion and the correct plane, G17-G19, must be used. There are two methods of commanding a G02 or G03, the first is using the I, J, K addresses and the second is using the R address.

Using I, J, K addresses

I, J and K address are used to locate the arc center in relation to the start point. In other words, the I, J, K addresses are the distances from the starting point to the center of the circle. Only the I, J, or K specific to the selected plane are allowed (G17 uses IJ, G18 uses IK and G19 uses JK). The x, y, and z commands specify the end point of the arc. If the x, y, and z location for the selected plane is not specified, the endpoint of the arc is the same as the starting point for that axis.

To cut a full circle the I, J, K addresses must be used; using an R address will not work. To cut a full circle, do not specify an ending point (x, y, and z); program I, J, or K to define the center of the circle. For example:

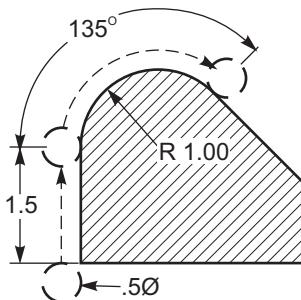
```
G02 I3.0 J4.0 (Assumes G17; XY plane) ;
```

Using the R address

The R-value defines the distance from the starting point to the center of the circle. Use a positive R-value for radii of 180° or less, and a negative R-value for radii more than 180°.

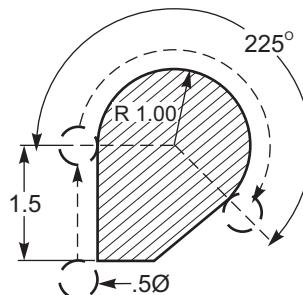
Programming Example

F7.6: Positive R Address Programming Example



```
%  
O60021 (G02 POSITIVE R ADDRESS) ;  
(G54 X0 Y0 is at the bottom-left of part) ;  
(Z0 is on top of the part) ;  
(T1 is a .5 in dia endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X-0.25 Y-0.25 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.5 F20. (Feed to cutting depth) ;  
G01 Y1.5 F12. (Feed to Y1.5) ;  
G02 X1.884 Y2.384 R1.25 (CW circular motion) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

F7.7: Negative R Address Programming Example



```
%  
O60022 (G02 NEGATIVE R ADDRESS) ;  
(G54 X0 Y0 is at the bottom-left of part) ;  
(Z0 is on top of the part) ;  
(T1 is a .5 in dia endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X-0.25 Y-0.25 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.5 F20. (Feed to cutting depth) ;  
G01 Y1.5 F12. (Feed to Y1.5) ;  
G02 X1.884 Y0.616 R-1.25 (CW circular motion) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

Thread Milling

Thread milling uses a standard G02 or G03 move to create the circular move in X-Y, then adds a Z move on the same block to create the thread pitch. This generates one turn of the thread; the multiple teeth of the cutter generate the rest. Typical block of code:

```
N100 G02 I-1.0 Z-.05 F5. (generates 1-inch radius for 20-pitch  
thread) ;
```

Thread milling notes:

Internal holes smaller than 3/8 inch may not be possible or practical. Always climb cut the cutter.

Use a G03 to cut I.D. threads or a G02 to cut O.D. threads. An I.D. right hand thread will move up in the Z-Axis by the amount of one thread pitch. An O.D. right hand thread will move down in the Z-Axis by the amount of one thread pitch. PITCH = 1/Threads per inch (Example - 1.0 divided by 8 TPI = .125)

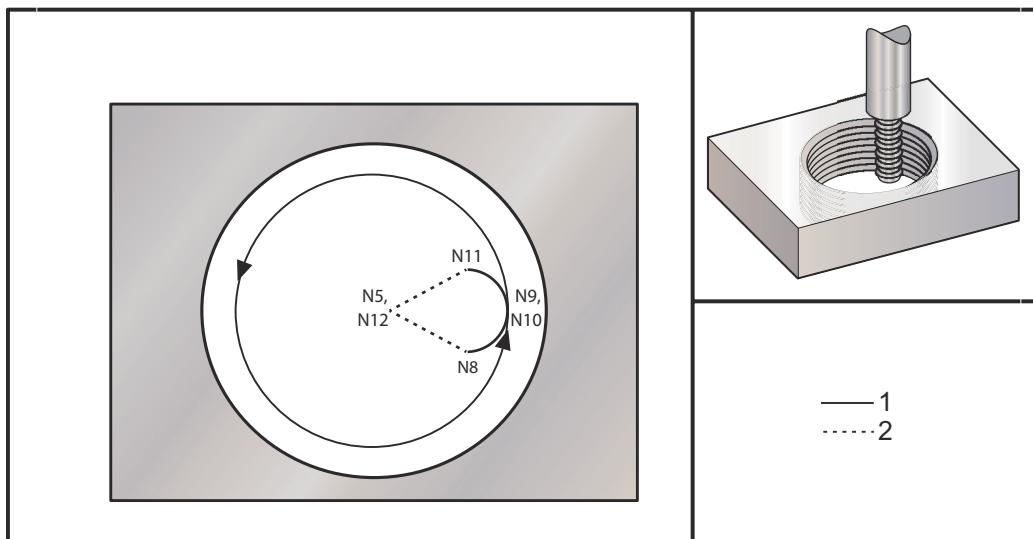
This program I.D. thread mills a 1.5 diameter x 8 TPI hole with a 0.750" diameter x 1.0" thread hob.

1. To start, take the hole diameter (1.500). Subtract the cutter diameter .750 and then divide by 2. $(1.500 - .75) / 2 = .375$
The result (.375) is the distance the cutter starts from the I.D. of the part.
2. After the initial positioning, the next step of the program is to turn on cutter compensation and move to the I.D. of the circle.
3. The next step is to program a complete circle (G02 or G03) with a Z-Axis command of the amount of one full pitch of the thread (this is called Helical Interpolation).
4. The last step is to move away from the I.D. of the circle and turn off cutter compensation.

You cannot turn cutter compensation off or on during an arc movement. You must program a linear move, either in the X or Y Axis, to move the tool to and from the diameter to cut. This move will be the maximum compensation amount that you can adjust.

Thread Milling Example

- F7.8:** Thread Milling Example, 1.5 Diameter X 8 TPI: [1]Tool Path, [2] Turn on and off cutter compensation.



NOTE:

Many thread mill manufacturers offer free online software to help you create your threading programs.

```
%  
O60023 (G03 THREAD MILL 1.5-8 UNC) ;  
(G54 X0 Y0 is at the center of the bore) ;  
(Z0 is on top of the part) ;  
(T1 is a .5 in dia thread mill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.5156 F50. (Feed to starting depth) ;  
(Z-0.5 minus 1/8th of the pitch = Z-0.5156) ;  
G41 X0.25 Y-0.25 F10. D01 (cutter comp on) ;  
G03 X0.5 Y0 I0 J0.25 Z-0.5 (Arc into thread) ;  
(Ramps up by 1/8th of the pitch) ;  
I-0.5 J0 Z-0.375 F20. (Cuts full thread) ;  
(Z moving up by the pitch value to Z-0.375) ;
```

```
X0.25 Y0.25 I-0.25 J0 Z-0.3594 (Arc out of thread) ;  
(Ramp up by 1/8th of the pitch) ;  
G40 G01 X0 Y1 (cutter comp off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

N5 = XY at the center of the hole

N7 = Thread depth, minus 1/8 pitch

N8 = Enable Cutter Compensation

N9 = Arcs into thread, ramps up by 1/8 pitch

N10 = Cuts full thread, Z moving up by the pitch value

N11 = Arcs out of thread, ramps up 1/8 pitch

N12 = Cancel Cutter Compensation

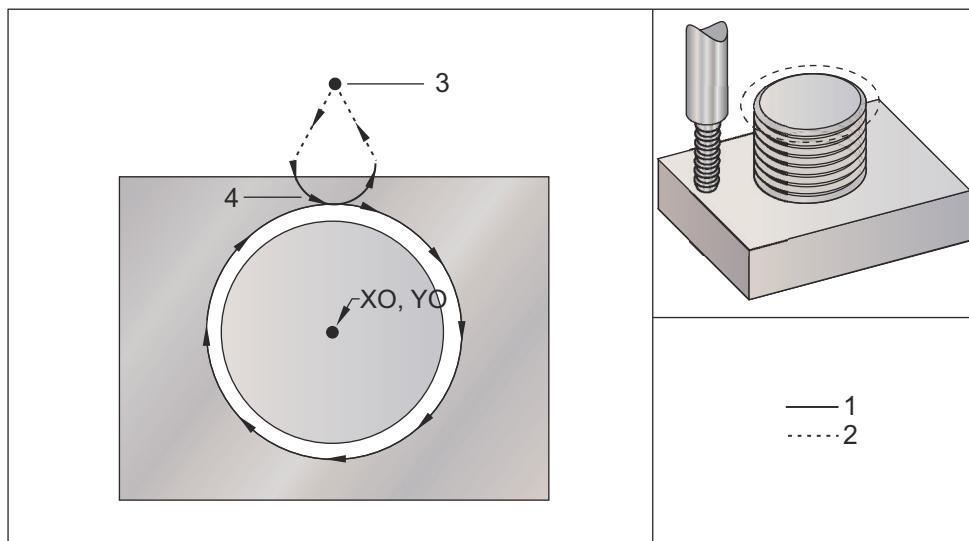


NOTE:

Maximum cutter compensation adjustability is 0.175.

Outside Diameter (O.D.) Thread Milling

- F7.9:** O.D. Thread Milling Example, 2.0 diameter post x 16 TPI: [1] Tool Path [2] Rapid Positioning, Turn on and off cutter compensation, [3] Start Position, [4] Arc with Z.



```
%  
O60024 (G02 G03 THREAD MILL 2.0-16 UNC) ;  
(G54 X0 Y0 is at the center of the post) ;  
(Z0 is on top of the opost) ;  
(T1 is a .5 in dia thread mill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y2.4 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G00 Z-1. (Rapids to Z-1.) ;  
G01 G41 D01 X-0.5 Y1.4 F20. (Linear move) ;  
(Cutter comp on) ;  
G03 X0 Y0.962 R0.5 F25. (Arc into thread) ;  
G02 J-0.962 Z-1.0625 (Cut threads while lowering Z) ;  
G03 X0.5 Y1.4 R0.5 (Arc out of thread) ;  
G01 G40 X0 Y2.4 F20. (Linear move) ;  
(Cutter comp off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;
```

%

**NOTE:**

A cutter compensation move can consist of any X or Y move from any position as long as the move is greater than the amount being compensated.

Single-Point Thread Milling

This program is for a 1.0" diameter hole with a cutter diameter of 0.500" and a thread pitch of 0.125 (8TPI). This program positions itself in Absolute G90 and then switches to G91 Incremental mode on line N7.

The use of an Lxx value on line N10 allows us to repeat the thread milling arc multiple times, with a Single-Point Thread Mill.

```
%  
O60025 (G03 SNGL PNT THREAD MILL 1.5-8 UNC) ;  
(G54 X0 Y0 is at the center of the bore) ;  
(Z0 is on top of the part) ;  
(T1 is a .5 in dia thread mill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G91 G01 Z-0.5156 F50. (Feed to starting depth) ;  
(Z-0.5 minus 1/8th of the pitch = Z-0.5156) ;  
G41 X0.25 Y-0.25 F20. D01 (Cutter comp on) ;  
G03 X0.25 Y0.25 I0 J0.25 Z0.0156 (Arc into thread) ;  
(Ramps up by 1/8th of the pitch) ;  
I-0.5 J0 Z0.125 L5 (Thread cut, repeat 5 times) ;  
X-0.25 Y0.25 I-0.25 J0 Z0.0156 (Arc out of thread) ;  
(Ramps up by 1/8th of the pitch) ;  
G40 G01 X-0.25 Y-0.25 (Cutter comp off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

Specific line description:

N5 = XY at the center of the hole

N7 = Thread depth, minus 1/8 pitch. Switches to G91

N8 = Enable Cutter Compensation

N9 = Arcs into thread, ramps up by 1/8 pitch

N10 = Cuts full thread, Z moving up by the pitch value

N11 = Arcs out of thread, ramps up 1/8 pitch

N12 = Cancel Cutter Compensation

N13 = Switches back to G90 Absolute positioning

Helical Motion

Helical (spiral) motion is possible with G02 or G03 by programming the linear axis that is not in the selected plane. This third axis will be moved along the specified axis in a linear manner, while the other two axes will be moved in the circular motion. The speed of each axis will be controlled so that the helical rate matches the programmed feedrate.

G04 Dwell (Group 00)

P - The dwell time in seconds or milliseconds



NOTE:

The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.

G04 specifies a delay or dwell in the program. The block with G04 delay for the time specified by the P address code. For example:

G04 P10.0. ;

Delays the program for 10 seconds.



NOTE:

G04 P10. is a dwell of 10 seconds; G04 P10 is a dwell of 10 milliseconds. Make sure you use decimal points correctly so that you specify the correct dwell time.

G09 Exact Stop (Group 00)

G09 code is used to specify a controlled axes stop. It affects only the block in which it is commanded. It is non-modal and does not affect the blocks that come after the block where it is commanded. Machine moves decelerate to the programmed point before the control processes the next command.

G10 Set Offsets (Group 00)

G10 lets you set offsets within the program. G10 replaces manual offset entry (i.e. Tool length and diameter, and work coordinate offsets).

L – Selects offset category.

L2 Work coordinate origin for G52 and G54-G59

L10 Length offset amount (for H code)

L11 or **L1** Tool wear offset amount (for H code)

L12 Diameter offset amount (for D code)

L13 Diameter wear offset amount (for D code)

L20 Auxiliary work coordinate origin for G110-G129

P – Selects a specific offset.

P1-P200 Used to reference D or H code offsets (L10-L13)

P0 G52 references work coordinate (L2)

P1-P6 G54-G59 references work coordinates (L2)

P1-P20 G110-G129 references auxiliary coordinates (L20)

P1-P99 G154 reference auxiliary coordinate (L20)

***R** Offset value or increment for length and diameter.

***X** X-Axis zero location.

***Y** Y-Axis zero location.

***Z** Z-Axis zero location.

***A** A-Axis zero location.

***B** B-Axis zero location.

***C** C-Axis zero location.

*indicates optional

```
%  
O60100 (G10 SET OFFSETS) ;  
G10 L2 P1 G91 X6.0 ;  
    (Move coordinate G54 6.0 to the right) ;  
;  
G10 L20 P2 G90 X10. Y8. ;  
    (Set work coordinate G111 to X10.0 Y8.0) ;  
;  
G10 L10 G90 P5 R2.5 ;  
    (Set offset for Tool #5 to 2.5) ;  
;  
G10 L12 G90 P5 R.375 ;  
    (Set diameter for Tool #5 to .375") ;
```

```

;
G10 L20 P50 G90 X10. Y20. ;
(Set work coordinate G154 P50 to X10. Y20.) ;
%
```

G12 Circular Pocket Milling CW / G13 Circular Pocket Milling CCW (Group 00)

These G-codes mill circular shapes. They are different only in that G12 uses a clockwise direction and G13 uses a counterclockwise direction. Both G-codes use the default XY circular plane (G17) and imply the use of G42 (cutter compensation) for G12 and G41 for G13. G12 and G13 are non-modal.

***D** - Tool radius or diameter selection**

F - Feedrate

I - Radius of first circle (or finish if no K). I value must be greater than Tool Radius, but less than K value.

***K** - Radius of finished circle (if specified)

***L** - Loop count for repeating deeper cuts

***Q** - Radius increment, or stepover (must be used with K)

Z - Depth of cut or increment

*indicates optional

**To get the programmed circle diameter, the control uses the selected D code tool size. To program tool centerline select D0.



NOTE:

Specify D00 if you do not want to use cutter compensation. If you do not specify a D value in the G12/G13 block, the control uses the last commanded D value, even if it was previously canceled with a G40.

Rapid-position the tool to the center of the circle. To remove all the material inside the circle, use I and Q values less than the tool diameter and a K value equal to the circle radius. To cut a circle radius only, use an I value set to the radius and no K or Q value.

```

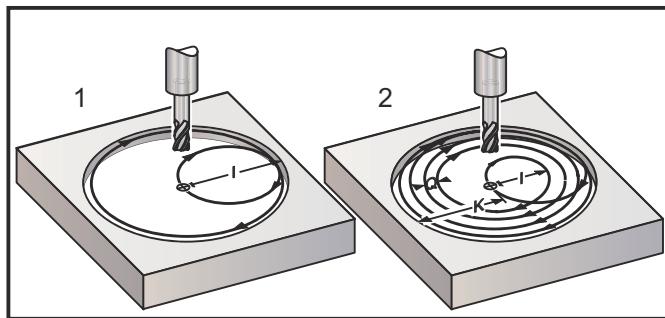
%
O60121(SAMPLE G12 AND G13) ;
(G54 X0 Y0 is center of first pocket) ;
(Z0 is on top of the part) ;
(T1 is a .25 in. dia endmill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X0 Y0 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
```

```

G43 H01 Z0.1 (Tool offset 1 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G12 I0.75 F10. Z-1.2 D01 (Finish pocket CW) ;
G00 Z0.1 (Retract) ;
X5. (Move to center of next pocket) ;
G12 I0.3 K1.5 Q1. F10. Z-1.2 D01 ;
(Rough & finish CW) ;
G00 Z0.1 (Retract) ;
X10. (Move to center of next pocket) ;
G13 I1.5 F10. Z-1.2 D01 (Finish CCW) ;
G00 Z0.1 (Retract) ;
X15. (Move to center of the last pocket) ;
G13 I0.3 K1.5 Q0.3 F10. Z-1.2 D01 ;
(Rough & finish CCW) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%

```

F7.10: Circular Pocket Milling, G12 Clockwise shown: [1] I only, [2] I, K and Q only.



These G codes assume cutter compensation, so you do not need to program G41 or G42 in the program block. However, you must include a D offset number, for cutter radius or diameter, to adjust the circle diameter.

These program examples show the G12 and G13 format, and the different ways that you can write these programs.

Single Pass: Use I only.

Applications: One-pass counter boring; rough and finish pocketing of smaller holes, ID cutting of O-ring grooves.

Multiple Pass: Use I , K , and Q .

Applications: Multiple-pass counter boring; rough and finish pocketing of large holes with cutter overlap.

Multiple Z-Depth Pass: Using **I** only, or **I**, **K**, and **Q** (**G91** and **L** may also be used).

Applications: Deep rough and finish pocketing.

The previous figures show the tool path during the pocket milling G-codes.

Example G13 multiple-pass using **I**, **K**, **Q**, **L**, and **G91**:

This program uses **G91** and an **L** count of 4, so this cycle will execute a total of four times. The Z depth increment is 0.500. This is multiplied by the **L** count, making the total depth of this hole 2.000.

The **G91** and **L** count can also be used in a **G13 I** only line.

```
%  
O60131 (G13 G91 CCW EXAMPLE) ;  
(G54 X0 Y0 is center of 1st pocket) ;  
(Z0 is on top of the part) ;  
(T1 is a 0.5 in. dia endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G13 G91 Z-.5 I.400 K2.0 Q.400 L4 D01 F20. ;  
(Rough & finish CCW) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G90 Z0.1 M09 (Rapid retract, coolant off) ;  
G53 G49 Z0 M05 (Z home, spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

G17 XY / G18 XZ / G19 YZ Plane Selection (Group 02)

The face of the workpiece to have a circular milling operation (G02, G03, G12, G13) done to it must have two of the three main axes (X, Y and Z) selected. One of three G codes is used to select the plane, G17 for XY, G18 for XZ, and G19 for YZ. Each is modal and applies to all subsequent circular motions. The default plane selection is G17, which means that a circular motion in the XY plane can be programmed without selecting G17. Plane selection also applies to G12 and G13, circular pocket milling, (always in the XY plane).

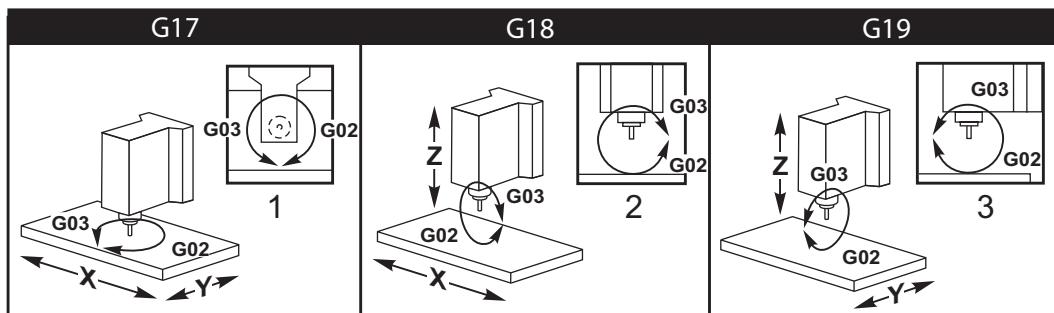
If cutter radius compensation is selected (G41 or G42), only use the XY plane (G17) for circular motion.

G17 Defined - Circular motion with the operator looking down on the XY table from above. This defines the motion of the tool relative to the table.

G18 Defined - Circular motion is defined as the motion for the operator looking from the rear of the machine toward the front control panel.

G19 Defined - Circular motion is defined as the motion for the operator looking across the table from the side of the machine where the control panel is mounted.

F7.11: G17, G18, and G19 Circular Motion Diagrams: [1] Top view, [2] Front view, [3] Right view.



G20 Select Inches / G21 Select Metric (Group 06)

Use G20 (inch) and G21 (mm) codes are to make sure that the inch/metric selection is set correctly for the program. Use Setting 9 to select between inch and metric programming. G20 in a program causes an alarm if Setting 9 is not set to inch.

G28 Return to Machine Zero Point (Group 00)

The G28 code returns all axes (X, Y, Z, A and B) simultaneously to the machine zero position when no axis is specified on the G28 line.

Alternatively, when one or more axes locations are specified on the G28 line, G28 will move to the specified locations and then to machine zero. This is called the G29 reference point; it is saved automatically for optional use in G29.

Setting 108 affects the way that rotary axes return when you command a G28. Refer to page 463 for more information.

```
%  
G28 G90 X0 Y0 Z0 (moves to X0 Y0 Z0) ;  
G28 G90 X1. Y1. Z1. (moves to X1. Y1. Z1.) ;  
G28 G91 X0 Y0 Z0 (moves directly to machine zero) ;  
G28 G91 X-1. Y-1. Z-1 (moves incrementally -1.) ;  
%
```

G29 Return From Reference Point (Group 00)

G29 moves the axes to a specific position. The axes selected in this block are moved to the G29 reference point saved in G28, and then moved to the location specified in the G29 command.

G31 Feed Until Skip (Group 00)

(This G-code is optional and requires a probe)

This G-code is used to record a probed location to a macro variable.

F - Feedrate

- ***X** - X-Axis absolute motion command
- ***Y** - Y-Axis absolute motion command
- ***Z** - Z-Axis absolute motion command
- ***A** - A-Axis absolute motion command
- ***B** - B-Axis absolute motion command
- ***C** - C-axis absolute motion command (UMC)

*indicates optional

This G-code moves the programmed axes while looking for a signal from the probe (skip signal). The specified move is started and continues until the position is reached or the probe receives a skip signal. If the probe receives a skip signal during the G31 move, axis motion stops, the control beeps and records the skip signal position to macro variables. The program will then execute the next line of code. If the probe does not receive a skip signal during the G31 move, the control will not beep and the skip signal position will be recorded at the end of the programmed move. The program will continue. This G-code requires at least one Axis specified and a feedrate. If the command contains neither, an alarm is generated.

Macro variables #5061 through #5066 are designated to store skip signal positions for each axis. For more information about these skip signal variables see the macro section of this manual.

Notes:

This code is non-modal and only applies to the block of code in which G31 is specified.

Do not use Cutter Compensation (G41, G42) with a G31.

The G31 line must have a Feed command. To avoid damaging the probe, use a feed rate below F100. (inch) or F2500. (metric).

Turn on the probe before using G31.

If your mill has the standard Renishaw probing system, use the following commands to turn on the probe.

Use the following code to turn on the spindle probe.

```
M59 P1134 ;
```

Use the following code to turn on the tool-setting probe.

```
%  
M59 P1133 ;  
G04 P1.0 ;  
M59 P1134 ;  
%
```

Use the following code to turn off either probe.

```
M69 P1134 ;
```

Also see M75, M78 and M79 ;

Sample program:

This sample program measures the top surface of a part with the spindle probe traveling in the Z negative direction. To use this program, the G54 part location must be set at, or close to the surface to be measured.

```
%  
O60311 (G31 SPINDLE PROBE) ;  
(G54 X0. Y0. is at the center of the part) ;  
(Z0. is at, or close to the surface) ;  
(T1 is a Spindle probe) ;  
(PREPARATION) ;  
T1 M06 (Select Tool 1) ;  
G00 G90 G54 X0 Y0 (Rapid to X0. Y0.) ;  
M59 P1134 (Spindle probe on) ;  
G43 H1 Z1. (Activate tool offset 1) ;  
(PROBING) ;  
G31 Z-0.25 F50. (Measure top surface) ;
```

```

Z1. (Retract to Z1.) ;
M69 P1134 (Spindle probe off) ;
(COMPLETION) ;
G00 G53 Z0. (Rapid retract to Z home) ;
M30 (End program) ;
%
```

G35 Automatic Tool Diameter Measurement (Group 00)

(This G-code is optional and requires a probe)

This G-code is used to set a tool diameter offset.

F - Feedrate

***D** - Tool diameter offset number

***X** - X-Axis command

***Y** - Y-Axis command

*indicates optional

Automatic Tool Diameter Offset Measurement function (G35) is used to set the tool diameter (or radius) using two touches of the probe; one on each side of the tool. The first point is set with a G31 block using an M75, and the second point is set with the G35 block. The distance between these two points is set into the selected (non-zero) Dnnn offset.

Setting 63 Tool Probe Width is used to reduce the measurement of the tool by the width of the tool probe. See the settings section of this manual for more information about Setting 63.

This G-code moves the axes to the programmed position. The specified move is started and continues until the position is reached or the probe sends a signal (skip signal).

NOTES:

This code is non-modal and only applies to the block of code in which G35 is specified.

Do not use Cutter Compensation (G41, G42) with a G35.

To avoid damaging the probe, use a feed rate below F100. (inch) or F2500. (metric).

Turn on the tool-setting probe before using G35.

If your mill has the standard Renishaw probing system, use the following commands to turn on the tool-setting probe.

```

%
M59 P1133 ;
G04 P1.0 ;
M59 P1134 ;
%
```

Use the following commands to turn off the tool-setting probe.

```
M69 P1134 ;
```

Turn on the spindle in reverse (M04), for a right handed cutter.

Also see M75, M78, and M79.

Also see G31.

Sample program:

This sample program measures the diameter of a tool and records the measured value to the tool offset page. To use this program, the G59 Work Offset location must be set to the tool-setting probe location.

```
%  
O60351 (G35 MEASURE AND RECORD TOOL DIA OFFSET) ;  
(G59 X0 Y0 is the tool setting probe location) ;  
(Z0 is at the surface of tool-setting probe) ;  
(T1 is a spindle probe) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G59 X0 Y-1. (Rapid tool next to probe) ;  
M59 P1133 (Select tool-setting probe) ;  
G04 P1. (Dwell for 1 second) ;  
M59 P1134 (Probe on) ;  
G43 H01 Z1. (Activate tool offset 1) ;  
S200 M04 (Spindle on CCW) ;  
(BEGIN PROBING BLOCKS) ;  
G01 Z-0.25 F50. (Feed tool below surface of probe) ;  
G31 Y-0.25 F10. M75 (Set reference point) ;  
G01 Y-1. F25. (Feed away from the probe) ;  
Z0.5 (Retract above the probe) ;  
Y1. (Move over the probe in Y-axis) ;  
Z-0.25 (Move tool below surface of the probe) ;  
G35 Y0.205 D01 F10. ;  
(Measure & record tool diameter) ;  
(Records to tool offset 1);  
G01 Y1. F25. (Feed away from the probe) ;  
Z1. (Retract above the probe) ;  
M69 P1134 (Probe off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G53 Z0. (Rapid retract to Z home) ;  
M30 (End program) ;
```

%

G36 Automatic Work Offset Measurement (Group 00)

(This G-code is optional and requires a probe)

This G-code is used to set work offsets with a probe.

F - Feedrate

***I**- Offset distance along X-Axis

***J** - Offset distance along Y-Axis

***K** - Offset distance along Z-Axis

***X** - X-Axis motion command

***Y** - Y-Axis motion command

***Z** - Z-Axis motion command

*indicates optional

Automatic Work Offset Measurement (G36) is used to command a probe to set work coordinate offsets. A G36 will feed the axes of the machine in an effort to probe the work piece with a spindle mounted probe. The axis (axes) will move until a signal from the probe is received or the end of the programmed move is reached. Tool compensation (G41, G42, G43, or G44) must not be active when this function is performed. The point where the skip signal is received becomes the zero position for the currently active work coordinate system of each axis programmed. This G-code requires at least one Axis specified, if neither are found, an alarm is generated.

If an I, J, or K is specified, the appropriate axis work offset is shifted by the amount in the I, J, or K command. This allows the work offset to be shifted away from where the probe actually contacts the part.

NOTES:

This code is non-modal and only applies to the block of code in which G36 is specified.

The points probed are offset by the values in Settings 59 through 62. See the settings section of this manual for more information.

Do not use Cutter Compensation (G41, G42) with a G36.

Do not use tool length Compensation (G43, G44) with G36

To avoid damaging the probe, use a feed rate below F100. (inch) or F2500. (metric).

Turn on the spindle probe before using G36.

If your mill has the standard Renishaw probing system, use the following commands to turn on the spindle probe.

Use the following commands to turn off the spindle probe.

```
M69 P1134 ;
```

Also see M78, and M79.

```
%  
O60361 (G36 AUTO WORK OFFSET MEASUREMENT) ;  
(G54 X0 Y0 is at the top-center of the part) ;  
(Z0 is at the surface of part) ;  
(T1 is a Spindle probe) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 20) ;  
G00 G90 G54 X0 Y1. (Rapid to 1st position) ;  
(BEGIN PROBING BLOCKS) ;  
M59 P1134 (Spindle probe on) ;  
Z-.5 (Move the probe below surface of part) ;  
G01 G91 Y-0.5 F50. (Feed towards the part) ;  
G36 Y-0.7 F10. (Measure and record Y offset) ;  
G91 Y0.25 F50. (Move incrementally away from part) ;  
G00 Z1. (Rapid retract above part) ;  
M69 P1134 (Spindle probe off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G90 G53 Z0. (Rapid retract to Z home) ;  
M30 (End program) ;  
%
```

G37 Automatic Tool Offset Measurement (Group 00)

(This G-code is optional and requires a probe)

This G-code is used to set tool length offsets.

F - Feedrate

H - Tool offset number

Z - Required Z-Axis offset

Automatic Tool Length Offset Measurement (G37) is used to command a probe to set tool length offsets. A G37 will feed the Z-Axis in an effort to probe a tool with a tool-setting probe. The Z-Axis will move until a signal from the probe is received or the travel limit is reached. A non-zero H code and either G43 or G44 must be active. When the signal from the probe is received (skip signal) the Z position is used to set the specified tool offset (H_{nnnn}). The resulting tool offset is the distance between the current work coordinate zero point and the point where the probe is touched. If a non-zero Z value is on the G37 line of code the resulting tool offset will be shifted by the non-zero amount. Specify Z0 for no offset shift.

The work coordinate system (G54, G55, etc.) and the tool length offsets

(H01-H200) may be selected in this block or the previous block.

NOTES:

This code is non-modal and only applies to the block of code in which G37 is specified.

A non-zero H code and either G43 or G44 must be active.

To avoid damaging the probe, use a feed rate below F100. (inch) or F2500. (metric).

Turn on the tool-setting probe before using G37.

If your mill has the standard Renishaw probing system, use the following commands to turn on the tool-setting probe.

```
%  
M59 P1133 ;  
G04 P1. ;  
M59 P1134 ;  
%
```

Use the following command to turn off the tool-setting probe.

```
M69 P1134 ;
```

Also see M78 and M79.

Sample program:

This sample program measures the length of a tool and records the measured value on the tool offset page. To use this program, the G59 work offset location must be set to the tool-setting probe location.

```
%  
O60371 (G37 AUTO TOOL OFFSET MEASUREMENT) ;  
(G59 X0 Y0 is center of tool-setting probe) ;  
(Z0 is at the surface of tool-setting probe) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G59 X0 Y0 (Rapid to center of the probe) ;  
G00 G43 H01 Z5. (Activate tool offset 1) ;  
(BEGIN PROBING BLOCKS) ;  
M59 P1133 (Select tool-setting probe) ;  
G04 P1. (Dwell for 1 second) ;  
M59 P1134 (Probe on) ;  
G37 H01 Z0 F30. (Measure & record tool offset) ;  
M69 P1134 (Probe off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G53 Z0. (Rapid retract to Z home) ;  
M30 (End program) ;  
%
```

G40 Cutter Comp Cancel (Group 07)

G40 cancels G41 or G42 cutter compensation.

G41 2D Cutter Compensation Left / G42 2D Cutter Comp. Right (Group 07)

G41 will select cutter compensation left; that is, the tool is moved to the left of the programmed path to compensate for the size of the tool. A D address must be programmed to select the correct tool radius or diameter offset. If the value in the selected offset is negative, cutter compensation will operate as though G42 (Cutter Comp Right.) was specified.

The right or left side of the programmed path is determined by looking at the tool as it moves away. If the tool needs to be on the left of the programmed path as it moves away, use G41. If it needs to be on the right of the programmed path as it moves away, use G42. For more information, refer to the Cutter Compensation section.

G43 Tool Length Compensation + (Add) / G44 Tool Length Comp - (Subtract) (Group 08)

A G43 code selects tool length compensation in the positive direction; the tool length in the offsets page is added to the commanded axis position. A G44 code selects tool length compensation in the negative direction; the tool length in the offsets page is subtracted from the commanded axis position. A non-zero H address must be entered to select the correct entry from the offsets page.


NOTE:

Starting in NGC Software Version 100.21.000.1100, the tool length offset behavior has been modified on Haas machines in the following ways: By default, tool length offsets will now always be applied, unless a G49/H00 (Mill) or Txx00 offset (Lathe) is explicitly specified. On mills, when a tool change occurs, the tool length offset will automatically update to match the new tool. The current tool length offset and mill group 8 code will now persist through power cycles.

G47 Text Engraving (Group 00)

G47 lets you engrave a line of text, or sequential serial numbers, with a single G-code. To use G47, Settings 29 (G91 Non-Modal) and 73 (G68 Incremental Angle) must be OFF.


NOTE:

Engraving along an arc is not supported.

***D** - Controls the smoothness level, D1(rough), D2(medium), or D3(finish). If **D** is not used then default is D3.

***E** - Plunge feed rate (units/min)

F - Engraving feed rate (units/min)

***I** - Angle of rotation (-360. to +360.); default is 0

***K** - Sets the max corner rounding value. If **K** is not used then default is K0.002.

***J** - Height of text in in/mm (minimum = 0.001 inch); default is 1.0 inch (1.0 mm)

P - 0 for literal text engraving

- 1 for sequential serial number engraving

- 32-126 for ASCII characters

***R** - Return plane

***X** - X start of engraving

***Y** - Y start of engraving

***Z** - Depth of cut

*indicates optional

Literal Text Engraving

This method is used to engrave text on a part. The text should be in the form of a comment on the same line as the G47 command. For example, G47 P0 (TEXT TO ENGRAVE), will engrave *TEXT TO ENGRAVE* on the part.

**NOTE:**

Corner rounding can cause engraved characters to appear rounded and make them harder to read. To improve the sharpness and readability of engraved text, consider lowering the corner-rounding values with a G187 E.xxx value before the G47 command. Suggested starting E values are E0.002 (inch) or E0.05 (metric). Command a G187 alone after the engraving cycle to restore the default corner-rounding level. Refer to the example below:

```
G187 E.002 (PREFACE ENGRAVING WITH A G187 E.xxx);  
G47 P0 X.15 Y0. I0. J.15 R.1 Z-.004 F80. E40. (Engraving  
Text);  
G00 G80 Z0.1;  
G187 (RESTORE NORMAL CORNER ROUNDING FOR SMOOTHNESS);
```

The characters available for engraving are:

A-Z, a-z 0-9, and ` ~ ! @ # \$ % ^ & * - _ = + [] { } \ | ; : ' " , . / < > ?

Not all of these characters can be entered from the control. When programming from the mill keypad, or engraving parenthesis (), refer to the following Engraving Special Characters section.

This example creates the figure shown.

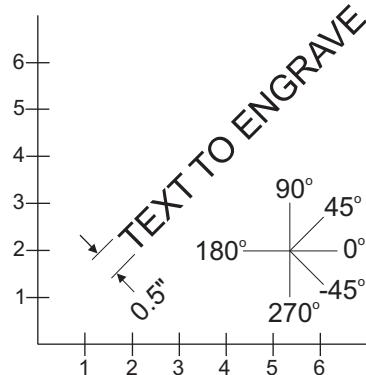
```
%  
O60471 (G47 TEXT ENGRAVING) ;  
(G54 X0 Y0 is at the bottom-left of part) ;  
(Z0 is on top of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y2. (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G47 P0 (TEXT TO ENGRAVE) X2. Y2. I45. J0.5 R0.05 Z-0.005 F15.  
E10. ;  
(Starts at X2. Y2., engraves text at 45 deg) ;  
(BEGIN COMPLETION BLOCKS) ;
```

```

G00 G80 Z0.1 (Cancel canned cycle) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%

```

F7.12: Engraving Program Example



In this example, G47 P0 selects literal string engraving. X2.0 Y2.0 sets the starting point for the text at the bottom left corner of first letter. I45. places the text at a positive 45° angle. J.5 sets the text height to 0.5 units-in/mm. R.05 retracts cutter to 0.05 units above part after engraving. Z-0.005 sets an engraving depth of -0.005 units. F15.0 sets an engraving, XY move, feedrate of 15 units per minute. E10.0 sets a plunge, -Z move, feedrate of 10 units per minute.

Special Characters

Engraving Special Characters involves using G47 with specific P values (G47 P32-126).

P- values to engrave specific characters

T7.1: G47 P Values for Special Characters

32		space	59	;	semicolon
33	!	exclamation mark	60	<	less than
34	"	double quotation mark	61	=	equals
35	#	number sign	62	>	greater than
36	\$	dollar sign	63	?	question mark
37	%	percent sign	64	@	at sign

38	&	ampersand	65-90	A-Z	capitol letters
39	'	closed single quote	91	[open square bracket
40	(open parenthesis	92	\	backslash
41)	close parenthesis	93]	closed square bracket
42	*	asterisk	94	^	caret
43	+	plus sign	95	_	underscore
44	,	comma	96	'	open single quote
45	-	minus sign	97-122	a-z	lowercase letters
46	.	period	123	{	open curly bracket
47	/	slash	124		vertical bar
48-57	0-9	numbers	125	}	closed curly bracket
58	:	colon	126	~	tilde

Example:

To engrave \$2.00, you need (2) blocks of code. The first block uses a P36 to engrave the dollar sign (\$), and the second block uses P0 (2.00).



NOTE:

Shift the X/Y start location between the first and second line of code to make a space between the dollar sign and the 2.

This is the only method to engrave parenthesis () .

Engraving Sequential Serial Numbers

This method is used to engrave numbers on a series of parts with the number being increased by one each time. The # symbol is used to set the number of digits in the serial number. For example, G47 P1 (####), limits the number to four digits while (##) would limit the serial number to two digits.

This program engraves a four digit serial number.

```
%  
O00037 (SERIAL NUMBER ENGRAVING) ;  
T1 M06 ;
```

```
G00 G90 G98 G54 X0. Y0. ;  
S7500 M03 ;  
G43 H01 Z0.1 ;  
G47 P1 (###) X2. Y2. I0. J0.5 R0.05 Z-0.005 F15. E10. ;  
G00 G80 Z0.1 ;  
M05 ;  
G28 G91 Z0 ;  
M30 ;  
%
```

Initial Serial Number

There are two ways to set the initial serial number to be engraved. The first requires replacing the # symbols within the parenthesis with the first number to be engraved. With this method, nothing is engraved when the G47 line is executed (it is only setting the initial serial number). Execute this once and then change the value within the parenthesis back to # symbols to engrave normally.

The following example will set the initial serial number to be engraved to 0001. Run this code once and then change (0001) to (####).

```
G47 P1 (0001) ;
```

The second method for setting the initial serial number to be engraved is to change the Macro Variable where this value is stored (Macro Variable 599). The Macros option does not need to be enabled.

Press **[CURRENT COMMANDS]** then press **[PAGE UP]** or **[PAGE DOWN]** as needed to display the **MACRO VARIABLES** page. From that screen, enter 599 and press Down cursor.

Once 599 is highlighted on the screen, type in the initial serial number to engrave, **[1]** for example, then press **[ENTER]**.

The same serial number can be engraved multiple times on the same part with the use of a macro statement. The macros option is required. A macro statement as shown below could be inserted between two G47 engraving cycles to keep the serial number from incrementing to the next number. For more details, see the Macros section of this manual.

Macro Statement: #599=[#599-1]

Engraving Around the Outside of a Rotary Part (G47, G107)

You can combine a G47 Engraving cycle with a G107 Cylindrical Mapping cycle to engrave text (or a serial number) along the outside diameter of a rotary part.

This code engravings a four digit serial number along the outer diameter of a rotary part.

```
%  
O01832 (CHANNEL ON 1.5 ROTARY PART);  
(MOUNT ROTARY ON RIGHT SIDE OF TABLE);  
(X ZERO IS FACE OF STOCK);  
(Y ZERO IS ROTARY CL) (TOUCH OFF TOOLS ON TOP OF PART);  
(STOCK IS 1.5 DIA);  
(T11 = ENGRAVING TOOL);  
(WRAP ENGRAVING AROUND CYLINDER, G107 G47);  
T11 M06;  
M11;  
M03 S12000;  
G57 G90 G00 G17 G40 G80;  
X0.323 Y0. A0. (START POINT OF ENGRAVE);  
G43 H11 Z0.1;  
/ G107 A0. Y0. R0.75;  
G187 P3 E0.002;  
G47 P0 (ROTARY) X0.323 Y0.177 I45. J0.15 R0.05 Z-0.004 F30.  
E10.;  
G00 Z0.1;  
G187;  
G107;  
T11 M06;  
M11;  
M03 S12000;  
G57 G90 G00 G17 G40 G80;  
X0.323 Y0. A0. (START POINT OF ENGRAVE);  
G43 H11 Z0.1;  
/ G107 A0. Y0. R0.75;  
G187 P3 E0.002;  
G47 P1 (S/N #####) X0.79 Y-0.28 I45. J0.15 R0.05 Z-0.004 F30.  
E10.;  
G00 Z2. M09;  
G107;  
G90 G00 A70.;  
G53 G00 G90 Y0;  
G187;  
M30;  
%
```

For more details on this cycle, refer to the G107 section.

G49 Tool Length Compensation Cancel (Group 08)

This G code cancels tool length compensation.



NOTE:

An *H0*, *M30*, and [**RESET**] will also cancel tool length compensation.

G50 Cancel Scaling (Group 11)

G50 cancels the optional scaling feature. Any axis scaled by a previous G51 command is no longer in effect.

G51 Scaling (Group 11)



NOTE:

You must purchase the Rotation and Scaling option to use this G-code. A 200-hour option tryout is also available; refer to page 222 for instructions.

***X** - Center of scaling for the X Axis

***Y** - Center of scaling for the Y Axis

***Z** - Center of scaling for the Z Axis

***P** - Scaling factor for all axes; three-place decimal from 0.001 to 999.999

*indicates optional

G51 [X...] [Y...] [Z...] [P...] ;

The control always uses a scaling center to determine the scaled position. If you do not specify a scaling center in the G51 command block, then the control uses the last commanded position as the scaling center.

With a scaling (G51) command, the control multiplies by a scaling factor (*P*) all X, Y, Z, A, B, and C end points for rapids, linear feeds, and circular feeds. G51 also scales I, J, K, and R for G02 and G03. The control offsets all of these positions relative to a scaling center.

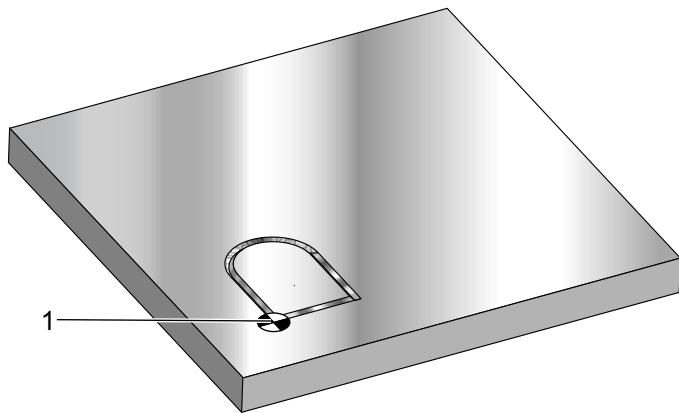
There are (3) ways to specify the scaling factor:

- A *P* address code in the G51 block applies the specified scaling factor to all axes.
- Setting 71 applies its value as a scaling factor to all axes if it has a nonzero value and you do not use a *P* address code.
- Settings 188, 189, and 190 apply their values as scaling factors to the X, Y, and Z axes independently if you do not specify a *P* value and Setting 71 has a value of zero. These settings must have equal values to use them with G02 or G03 commands.

G51 affects all appropriate positioning values in the blocks after the G51 command.

These example programs show how different scaling centers affect the scaling command.

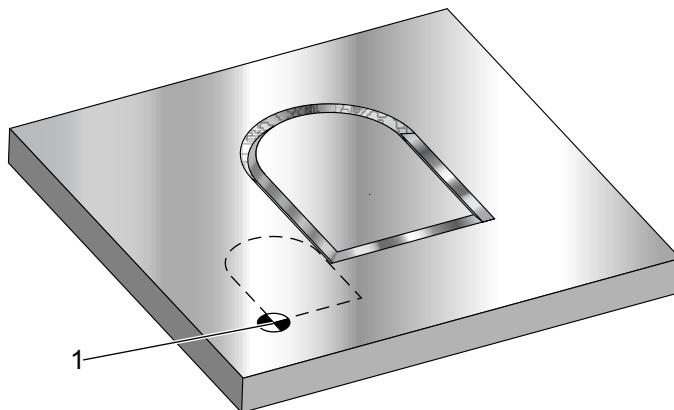
F7.13: G51 No Scaling Gothic Window: [1] Work coordinate origin.



```
%  
O60511 (G51 SCALING SUBPROGRAM) ;  
(G54 X0 Y0 is at the bottom left of window) ;  
(Z0 is on top of the part) ;  
(Run with a main program) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 X2. ;  
Y2. ;  
G03 X1. R0.5 ;  
G01 Y1. ;  
M99 ;  
%
```

The first example illustrates how the control uses the current work coordinate location as a scaling center. Here, it is X0 Y0 Z0.

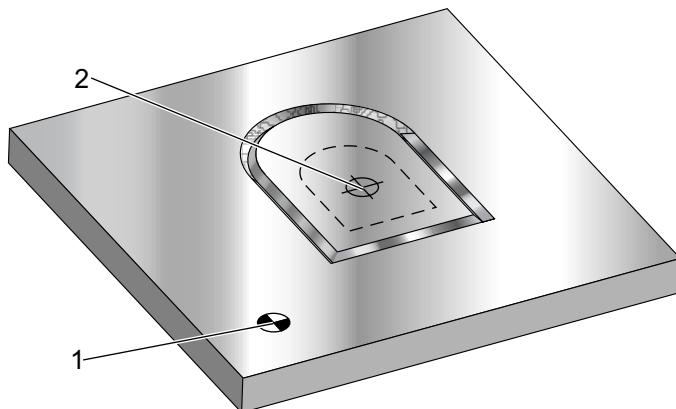
- F7.14:** G51 Scaling Current Work Coordinates: The Origin [1] is the work origin and the center of scaling.



```
%  
o60512 (G51 SCALING FROM ORIGIN) ;  
(G54 X0 Y0 is at the bottom left of part) ;  
(Z0 is on top of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 M08 (Activate tool offset 1) ;  
(Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.1 F25. (Feed to cutting depth) ;  
M98 P60511 (Cuts shape without scaling) ;  
G00 Z0.1 (Rapid Retract) ;  
G00 X2. Y2. (Rapid to new scale position) ;  
G01 Z-.1 F25. (Feed to cutting depth) ;  
G51 X0 Y0 P2. (2x scale from origin) ;  
M98 P60511 (run subprogram) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09(Rapid retract, Coolant off) ;  
G50 (CANCELS G51);  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

The next example specifies the center of the window as the scaling center.

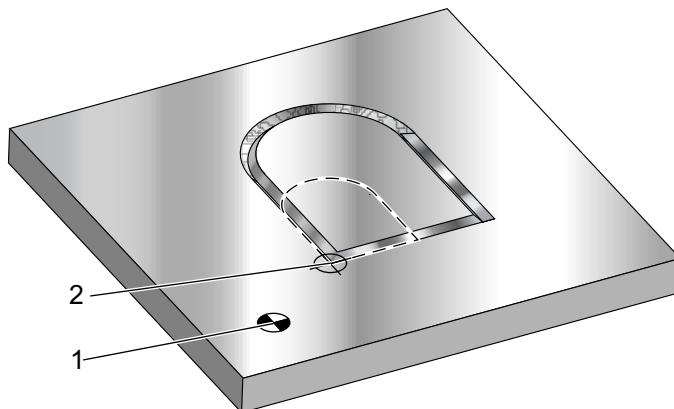
F7.15: G51 Scaling Center of Window: [1] Work coordinate origin, [2] Center of scaling.



```
%  
o60513 (G51 SCALING FROM CENTER OF WINDOW) ;  
(G54 X0 Y0 is at the bottom left of part) ;  
(Z0 is on top of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 M08 (Activate tool offset 1) ;  
(Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.1 F25. (Feed to cutting depth) ;  
M98 P60511 (Cuts shape without scaling) ;  
G00 Z0.1 (Rapid Retract) ;  
G00 X0.5 Y0.5 (Rapid to new scale position) ;  
G01 Z-.1 F25. (Feed to cutting depth) ;  
G51 X1.5 Y1.5 P2. (2x scale from center of window) ;  
M98 P60511 (run subprogram) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09(Rapid retract, Coolant off) ;  
G50 (CANCEL G51);  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

The last example illustrates how scaling can be placed at the edge of tool paths as if the part was being set against locating pins.

F7.16: G51 Scaling Edge of Tool Path: [1] Work coordinate origin, [2] Center of scaling.



```
%  
O60514 (G51 SCALING FROM EDGE OF TOOLPATH) ;  
(G54 X0 Y0 is at the bottom left of part) ;  
(Z0 is on top of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 M08 (Activate tool offset 1) ;  
(Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.1 F25. (Feed to cutting depth) ;  
M98 P60511 (Cuts shape without scaling) ;  
G00 Z0.1 (Rapid Retract) ;  
G00 X1. Y1. (Rapid to new scale position) ;  
G01 Z-.1 F25. (Feed to cutting depth) ;  
G51 X1. Y1. P2. (2x scale from edge of toolpath) ;  
M98 P60511 (run subprogram) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G50 (CANCEL G51) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

Tool offsets and cutter compensation values are not affected by scaling.

For canned cycles, G51 scales the initial point, depth, and return plane relative to the center of scaling.

To retain the functionality of canned cycles, G51 does not scale these:

- In G73 and G83:
 - Peck depth (Q)
 - Depth of first peck (I)
 - Amount to reduce peck depth per pass (J)
 - Minimum peck depth (K)
- In G76 and G77:
 - The shift value (Q)

The control rounds the final results of scaling to the lowest fractional value of the variable being scaled.

G52 Set Work Coordinate System (Group 00 or 12)

G52 works differently depending on the value of Setting 33. Setting 33 selects the Fanuc or Haas style of coordinates.

If **FANUC** is selected, G52 is a group 00 G-code. This is a global work coordinate shift. The values entered into the G52 line of the work offset page are added to all work offsets. All of the G52 values in the work offset page will be set to zero (0) when powered on, reset is pressed, changing modes, at the end of the program, by an M30, G92 or a G52 X0 Y0 Z0 A0 B0. When using a G92 (Set Work Coordinate Systems Shift Value), in Fanuc format, the current position in the current work coordinate system is shifted by the values of G92 (X, Y, Z, A, and B). The values of the G92 work offset are the difference between the current work offset and the shifted amount commanded by G92.

If **HAAS** is selected, G52 is a group 00 G-code. This is a global work coordinate shift. The values entered into the G52 line of the work offset page are added to all work offsets. All of the G52 values will be set to zero (0) by a G92. When using a G92 (Set Work Coordinate Systems Shift Value), in Haas format, the current position in the current work coordinate system is shifted by the values of G92 (X, Y, Z, A, and B). The values of the G92 work offset are the difference between the current work offset and the shifted amount commanded by G92 (Set Work Coordinate Systems Shift Value).

G53 Non-Modal Machine Coordinate Selection (Group 00)

This code temporarily cancels work coordinate offsets and uses the machine coordinate system. This code will also ignore tool offsets. In the machine coordinate system, the zero point for each axis is the position where the machine goes when a Zero Return is performed. G53 will revert to this system for the block in which it is commanded.

G54-G59 Select Work Coordinate System #1 - #6 (Group 12)

These codes select one of more than six user coordinate systems. All future references to axes positions will be interpreted using the new (G54 G59) coordinate system. See also **387** for additional work offsets.

G60 Uni-Directional Positioning (Group 00)

This G code is used to provide positioning only from the positive direction. It is provided only for compatibility with older systems. It is non-modal, so does not affect the blocks that follow it. Also refer to Setting 35.

G61 Exact Stop Mode (Group 15)

The G61 code is used to specify an exact stop. It is modal; therefore, it affects the blocks that follow it. The machine axes will come to an exact stop at the end of each commanded move.

G64 Cancels Exact Stop Mode (Group 15)

G64 code cancels exact stop (G61).

G65 Macro Subprogram Call Option (Group 00)

G65 is described in the Macros programming section.

G68 Rotation (Group 16)


NOTE:

You must purchase the Rotation and Scaling option to use this G-code. A 200-hour option tryout is also available; refer to page 222 for instructions.

***G17, G18, G19** - Plane of rotation, default is current

***X/Y, X/Z, Y/Z** - Center of rotation coordinates on the selected plane**

***R** - Angle of rotation, in degrees. Three-place decimal, -360.000 to 360.000.

*indicates optional

**The axis designation you use for these address codes corresponds to the axes of the current plane. For example, in the G17 (XY plane), you would use X and Y to specify the center of rotation.

When you command a G68, the control rotates all X, Y, Z, I, J, and K values about a center of rotation to a specified angle (R).,

You can designate a plane with G17, G18, or G19 before G68 to establish the axis plane to rotate. For example:

```
G17 G68 Xnnn Ynnn Rnnn ;
```

If you do not designate a plane in the G68 block, the control uses the currently active plane.

The control always uses a center of rotation to determine the positional values after rotation. If you do not specify a center of rotation, the control uses the current location.

G68 affects all appropriate positional values in the blocks after the G68 command. Values in the line that contains the G68 command are not rotated. Only the values in the plane of rotation are rotated; therefore, if G17 is the current plane of rotation, the command affects only the X and Y values.

A positive number (angle) in the R address rotates the feature counterclockwise.

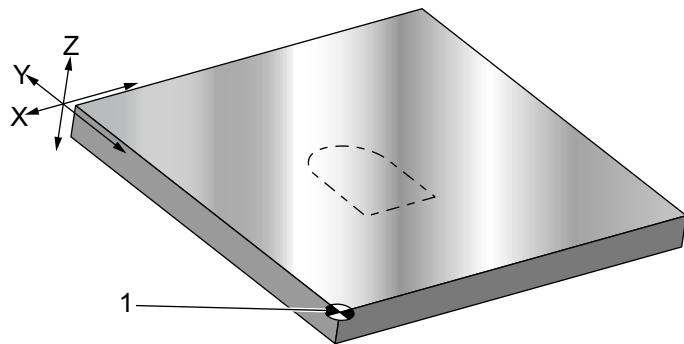
If you do not specify the angle of rotation (R), then the control uses the value in Setting 72.

In G91 mode (incremental) with Setting 73 ON, the rotation angle changes by the value in R. In other words, each G68 command changes the rotation angle by the value specified in R.

The rotational angle is set to zero at the beginning of the program, or you can set it to a specific angle with G68 in G90 mode.

These examples illustrate rotation with G68. The first program defines a Gothic window shape to cut. The rest of the programs use this program as a subprogram.

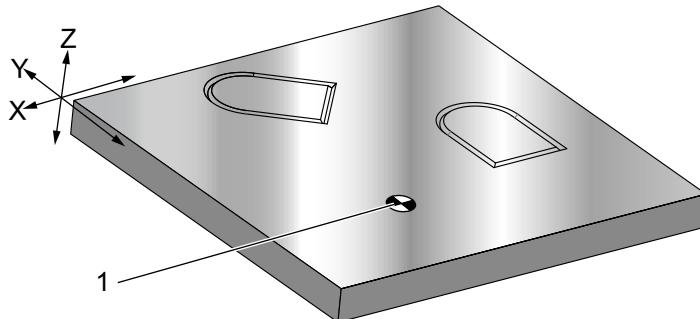
F7.17: G68 Start Gothic Window, No rotation: [1] Work coordinate origin.



```
%  
O60681 (GOTHIC WINDOW SUBPROGRAM) ;  
F50. S15000 (SET FEED AND SPINDLE SPEED) ;  
G00 X1. Y1. Z0.1 (RAPID TO LOWER-LEFT WINDOW CORNER) ;  
Z-.005;  
G01 X2. (BOTTOM OF WINDOW) ;  
Y2. (RIGHT SIDE OF WINDOW);  
G03 X1. R0.5 (TOP OF WINDOW) ;  
G01 Y1. (FINISH WINDOW) ;  
Z0.1 (RETRACT Z BEFORE GOING TO THE NEXT WINDOW) ;  
M99;  
%
```

The first example illustrates how the control uses the current work coordinate location as a rotation center ($X_0 Y_0 Z_0$).

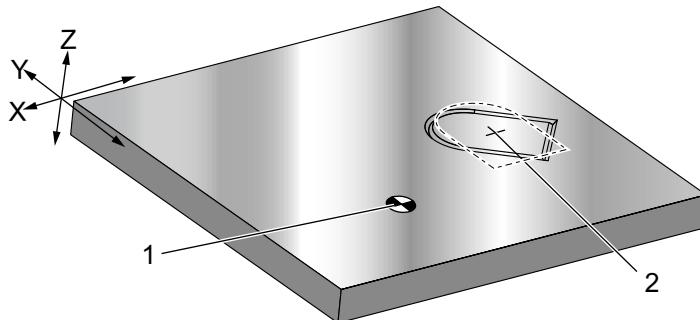
- F7.18:** G68 Rotation Current Work Coordinate: [1] Work coordinate origin and center of rotation.



```
%  
O60682 (ROTATE ABOUT WORK COORDINATE) ;  
G59 (OFFSET) ;  
G00 G90 X0 Y0 Z0.1 (WORK COORDINATE ORIGIN) ;  
M98 P60681 (CALL SUBPROGRAM) ;  
G90 G00 X0 Y0 (LAST COMMANDED POSITION) ;  
G68 R60. (ROTATE 60 DEGREES) ;  
M98 P60681 (CALL SUBPROGRAM) ;  
G69 G90 X0 Y0 (CANCEL G68) ;  
M30;  
%
```

The next example specifies the center of the window as the rotation center.

- F7.19:** G68 Rotation Center of Window: [1] Work coordinate origin, [2] Center of rotation.

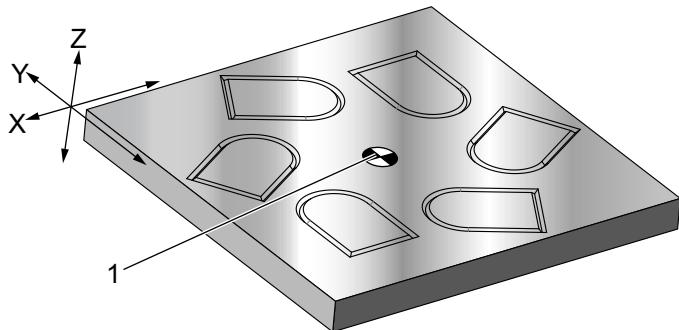


```
%  
O60683 (ROTATE ABOUT CENTER OF WINDOW) ;  
G59 (OFFSET) ;  
G00 G90 X0 Y0 Z0.1 (WORK COORDINATE ORIGIN) ;  
G68 X1.5 Y1.5 R60. ;
```

```
(ROTATE SHAPE 60 DEGREES ABOUT CENTER) ;  
M98 P60681 (CALL SUBPROGRAM) ;  
G69 G90 G00 X0 Y0 (CANCEL G68, LAST COMMANDED POSITION) ;  
M30 ;  
%
```

This next example shows how the G91 mode can be used to rotate patterns about a center. This is often useful for making parts that are symmetric about a given point.

F7.20: G68 Rotate Patterns About Center: [1] Work coordinate origin and center of rotation.



```
%  
O60684 (ROTATE PATTERN ABOUT CENTER) ;  
G59 (OFFSET) ;  
G00 G90 X0 Y0 Z0.1 (WORK COORDINATE ORIGIN) ;  
M97 P1000 L6 (CALL LOCAL SUBPROGRAM, LOOP 6 TIMES) ;  
M30 (END AFTER SUBPROGRAM LOOP) ;  
N1000 (BEGIN LOCAL SUBPROGRAM) ;  
G91 G68 R60. (ROTATE 60 DEGREES) ;  
G90 M98 P60681 (CALL WINDOW SUBPROGRAM) ;  
G90 G00 X0 Y0 (LAST COMMANDED POSITION) ;  
M99;  
%
```

Do not change the plane of rotation while G68 is in effect.

Rotation with Scaling:

If you use scaling and rotation at the same time, you should turn on scaling before rotation, and use separate blocks. Use this template:

```
%  
G51 ... (SCALING) ;  
... ;  
G68 ... (ROTATION) ;
```

```

... program ;
G69 ... (ROTATION OFF) ;
...
G50 ... (SCALING OFF) ;
%
```

Rotation with Cutter Compensation:

Turn on cutter compensation after the rotation command. Turn off cutter compensation before you turn off rotation.

G69 Cancel Rotation (Group 16)

(This G-code is optional and requires Rotation and Scaling.)

G69 cancels rotation mode.

G70 Bolt Hole Circle (Group 00)

I - Radius

*J - Starting angle (0 to 360.0 degrees CCW from horizontal; or 3 o'clock position)

L - Number of holes evenly spaced around the circle

*indicates optional

This non-modal G code must be used with one of the canned cycles G73, G74, G76, G77, or G81-G89. A canned cycle must be active so that at each position, a drill or tap function is performed. See also G-code Canned Cycles section.

```

%
O60701 (G70 BOLT HOLE CIRCLE) ;
(G54 X0 Y0 is center of the circle ) ;
(Z0 is on the top of the part) ;
(T1 is a drill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X0 Y0 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Activate tool offset 1) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G81 G98 Z-1. R0.1 F15. L0 (Begin G81) ;
(L0 skip drilling X0 Y0 position) ;
G70 I5. J15. L12 (Begin G70) ;
(Drills 12 holes on a 10.0 in. diameter circle) ;
G80 (Canned Cycles off) ;
(BEGIN COMPLETION BLOCKS) ;
```

```

G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home and Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%

```

G71 Bolt Hole Arc (Group 00)

I - Radius

***J** - Starting angle (degrees CCW from horizontal)

K - Angular spacing of holes (+ or -)

L - Number of holes

*indicates optional

This non-modal G code is similar to G70 except that it is not limited to a complete circle. G71 belongs to Group 00 and thus is non-modal. A canned cycle must be active so that at each position, a drill or tap function is performed.

G72 Bolt Holes Along an Angle (Group 00)

I - Distance between holes

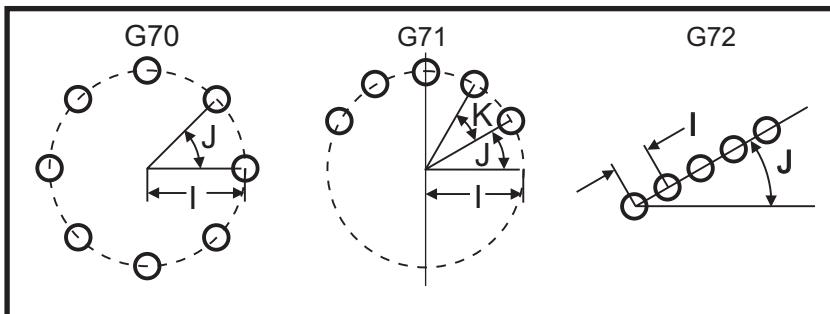
***J** - Angle of line (degrees CCW from horizontal)

L - Number of holes

*indicates optional

This non-modal G code drills L number of holes in a straight line at the specified angle. It operates similarly to G70. For a G72 to work correctly, a canned cycle must be active so that at each position, a drill or tap function is performed.

F7.21: G70, G71, and G72 Bolt Holes: [I] Radius of bolt circle (G70, G71), or distance between holes (G72), [J] Starting angle from the 3 o'clock position, [K] Angular spacing between holes, [L] Number of holes.



G73 High-Speed Peck Drilling Canned Cycle (Group 09)

F - Feedrate

***I** - First peck depth

***J** - Amount to reduce pecking depth for pass

***K** - Minimum peck depth (The control calculates the number of pecks)

***L** - Number of loops (Number of holes to drill) if G91 (Incremental Mode) is used

***P** - Pause at the bottom of the hole (in seconds)

***Q** - Peck Depth (always incremental)

***R** - Position of the R plane (Distance above part surface)

***X** - X-Axis location of hole

***Y** - Y-Axis location of hole

Z - Position of the Z-Axis at the bottom of hole

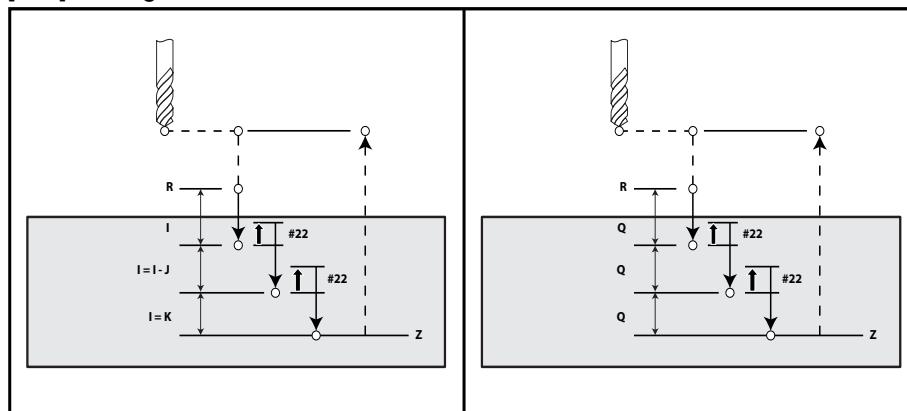
* indicates optional



NOTE:

The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.

F7.22: G73 Peck Drilling. Left: Using I, J, and K Addresses. Right: Using Only the Q Address. [#22] Setting 22.



I, J, K, and Q are always positive numbers.

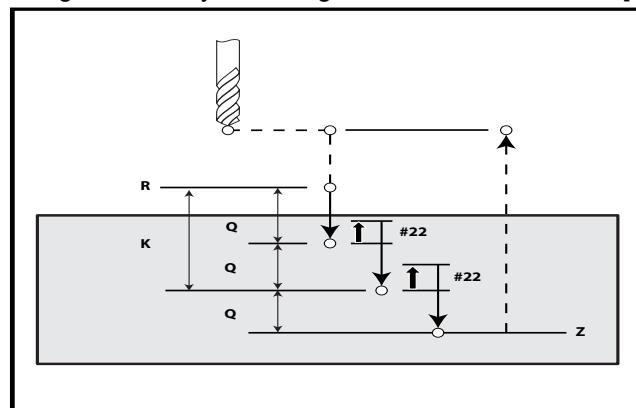
There are three methods to program a G73: using the I, J, K addresses, using the K and Q addresses, and using only the Q address.

If I, J, and K are specified, The first pass will cut in by the value I, each succeeding cut will be reduced by the value of J, and the minimum cutting depth is K. If P is specified, the tool will pause at the bottom of the hole for that amount of time.

If K and Q are both specified, a different operating mode is selected for this canned cycle. In this mode, the tool is returned to the R plane after the number of passes totals up to the K amount.

If only Q is specified, a different operating mode is selected for this canned cycle. In this mode, the tool is returned to the R plane after all pecks are completed, and all pecks will be equal to the Q value.

F7.23: G73 Peck Drilling Canned Cycles using the K and Q Addresses: [#22] Setting 22.



G74 Reverse Tap Canned Cycle (Group 09)

F - Feedrate. Use the formula described in the canned cycle introduction to calculate feedrate and spindle speed.

* **J** - Retract Multiple (How fast to retract - see Setting 130)

* **L** - Number of loops (How many holes to tap) if G91 (Incremental Mode) is used

* **R** - Position of the R plane (position above the part) where tapping starts

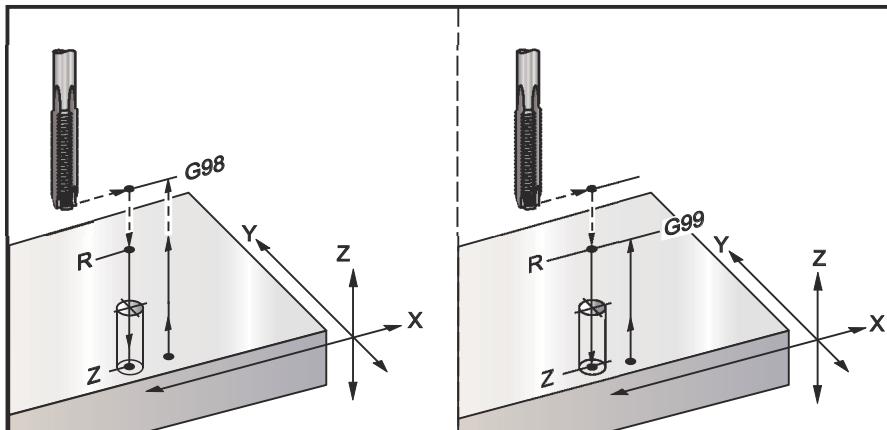
* **X** - X-Axis location of hole

* **Y** - Y-Axis location of hole

Z - Position of the Z-Axis at the bottom of hole

*indicates optional

F7.24: G74 Tapping Canned Cycle

**G76 Fine Boring Canned Cycle (Group 09)****F** - Feedrate***I** - Shift value along the X-Axis before retracting, if **Q** is not specified***J** - Shift value along the Y-Axis before retracting, if **Q** is not specified***L** - Number of holes to bore if **G91** (Incremental Mode) is used***P** - The dwell time at the bottom of the hole***Q** - The shift value, always incremental***R** - Position of the R plane (position above the part)***X** - X-Axis location of hole***Y** - Y-Axis location of hole**Z** - Position of the Z-Axis at the bottom of hole

* indicates optional

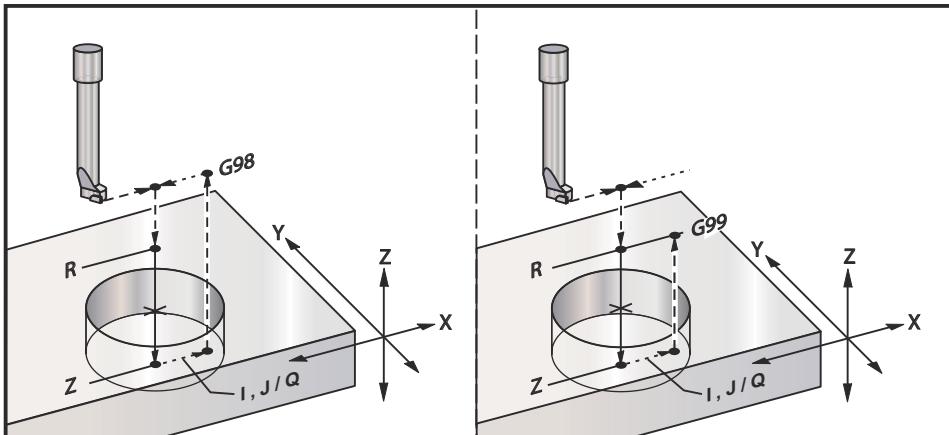
**NOTE:**

The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.

**CAUTION:**

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a “no-spin” cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

F7.25: G76 Fine Boring Canned Cycles



In addition to boring the hole, this cycle will shift the X and/or Y Axis prior to retracting in order to clear the tool while exiting the part. If *Q* is used Setting 27 determines the shift direction. If *Q* is not specified, the optional *I* and *J* values are used to determine the shift direction and distance.

G77 Back Bore Canned Cycle (Group 09)

F - Feedrate

***I** - Shift value along the X Axis before retracting, if *Q* is not specified

***J** - Shift value along the Y Axis before retracting, if *Q* is not specified

***L** - Number of holes to bore if G91 (Incremental Mode) is used

***Q** - The shift value, always incremental

***R** - Position of the R plane

***X** - X-Axis location of hole

***Y** - Y-Axis location of hole

Z - Z-Axis position to cut to

* indicates optional

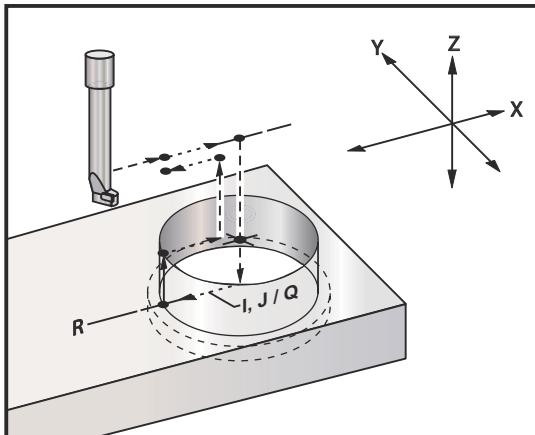


CAUTION:

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a "no-spin" cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

In addition to boring the hole, this cycle shifts the X and Y Axis before and after the cut, to clear the tool while it enters and exits the workpiece (refer to G76 for an example of a shift move). Setting 27 defines the shift direction. If you do not specify a Q value, the control uses the optional I and J values to determine the shift direction and distance.

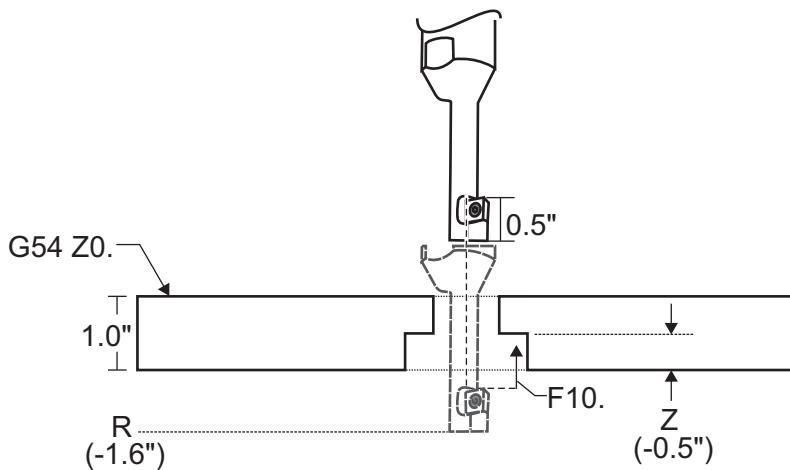
F7.26: G77 Back Boring Canned Cycle Example



Program Example

```
%  
O60077 (G77 CYCLE-WORKPIECE IS 1.0" THICK) ;  
T5 M06 (BACK COUNTERBORE TOOL) ;  
G90 G54 G00 X0 Y0 (INITIAL POSITION) ;  
S1200 M03 (SPINDLE START) ;  
G43 H05 Z.1 (TOOL LENGTH COMPENSATION) ;  
G77 Z-1. R-1.6 Q0.1 F10. (1ST HOLE) ;  
X-2. (2ND HOLE) ;  
G80 G00 Z.1 M09 (CANCEL CANNED CYCLE) ;  
G28 G91 Z0. M05 ;  
M30 ;  
%
```

F7.27: G77 Approximate Toolpath Example. This example shows the entrance motion only. Dimensions are not to scale.



NOTE:

For this example, the “top” of the workpiece is the surface defined as Z0. in the current work offset. The “bottom” of the workpiece is the opposite surface.

In this example, when the tool reaches the R depth, it then moves 0.1" in X (the Q value and Setting 27 define this movement; in this example, Setting 27 is $x+$). The tool then feeds to the Z value at the given feedrate. When the cut is finished, the tool shifts back toward the center of the hole and retracts out of it. The cycle repeats at the next commanded position until the G80 command.



NOTE:

The R value is negative, and it must go past the bottom of the part for clearance.



NOTE:

The Z value is commanded from the active Z work offset.



NOTE:

You do not need to command an initial point return (G98) after a G77 cycle; the control assumes this automatically.

G80 Canned Cycle Cancel (Group 09)

G80 cancels all active canned cycles.



NOTE:

G00 or G01 also cancel canned cycles.

G81 Drill Canned Cycle (Group 09)

***E** - Chip-clean RPM (Spindle reverses to remove chips after each cycle)

F - Feedrate

***L** - Number of holes to drill if G91 (Incremental Mode) is used

***R** - Position of the R plane (position above the part)

***X** - X-Axis motion command

***Y** - Y-Axis motion command

Z - Position of the Z Axis at the bottom of hole

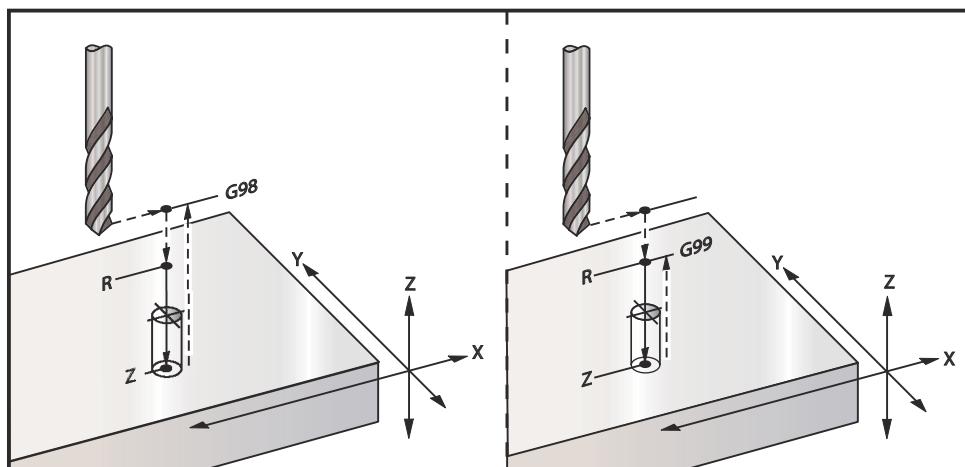
* indicates optional



CAUTION:

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a “no-spin” cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

F7.28: G81 Drill Canned Cycle



This is a program to drill through an aluminum plate:

```
%  
O60811 (G81 DRILLING CANNED CYCLE) ;  
(G54 X0 Y0 is at the top-left of part) ;  
(Z0 is on top of the part) ;  
(T1 is a .5 in drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y-2. (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G81 Z-0.720 R0.1 F15. (Begin G81) ;  
(Drill 1st hole at current X Y location) ;  
X2. Y-4. (2nd hole) ;  
X4. Y-4. (3rd hole) ;  
X4. Y-2. (4th hole) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G90 Z1. M09 (Rapid retract, coolant off) ;  
G53 G49 Z0 M05 (Z home, spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

G82 Spot Drill Canned Cycle (Group 09)

***E** - Chip-clean RPM (Spindle reverses to remove chips after each cycle)

F - Feedrate

***L** - Number of holes if G91 (Incremental Mode) is used.

***P** - The dwell time at the bottom of the hole

***R** - Position of the R plane (position above the part)

***X** - X-Axis location of hole

***Y** - Y-Axis location of hole

Z - Position of bottom of hole

* indicates optional



NOTE:

The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.

**CAUTION:**

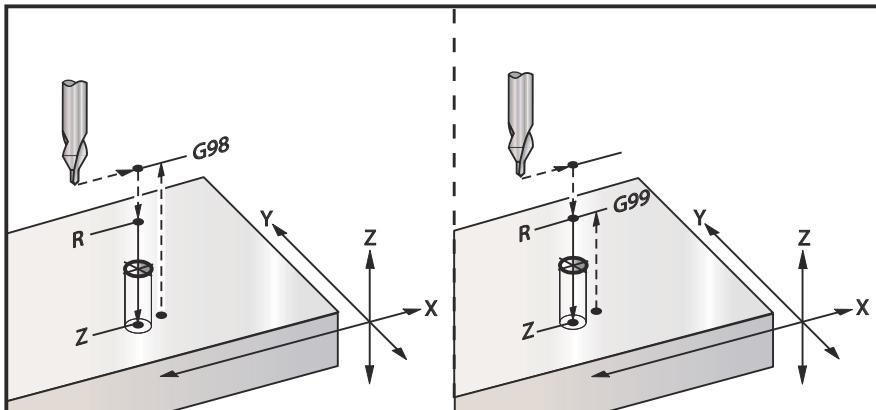
Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a “no-spin” cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

**NOTE:**

G82 is similar to G81 except that there is the option to program a dwell (P).

```
%  
O60821 (G82 SPOT DRILLING CANNED CYCLE) ;  
(G54 X0 Y0 is at the top-left of part) ;  
(Z0 is on top of the part) ;  
(T1 is a 0.5 in 90 degree spot drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y-2. (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G82 Z-0.720 P0.3 R0.1 F15. (Begin G82) ;  
(Drill 1st hole at current X Y location) ;  
X2. Y-4. (2nd hole) ;  
X4. Y-4. (3rd hole) ;  
X4. Y-2. (4th hole) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z1. M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

F7.29: G82 Spot Drilling Example

**G83 Normal Peck Drilling Canned Cycle (Group 09)**

***E** - Chip-clean RPM (Spindle reverses to remove chips after each cycle)

F - Feedrate

***I** - Size of first peck depth

***J** - Amount to reduce peck depth each pass

***K** - Minimum depth of peck

***L** - Number of holes if G91 (Incremental Mode) is used, also G81 through G89.

***P** - Pause at end of last peck, in seconds (Dwell)

***Q** - Peck depth, always incremental

***R** - Position of the R plane (position above the part)

***X** - X-Axis location of hole

***Y** - Y-Axis location of hole

Z - Position of the Z-Axis at the bottom of hole

* indicates optional

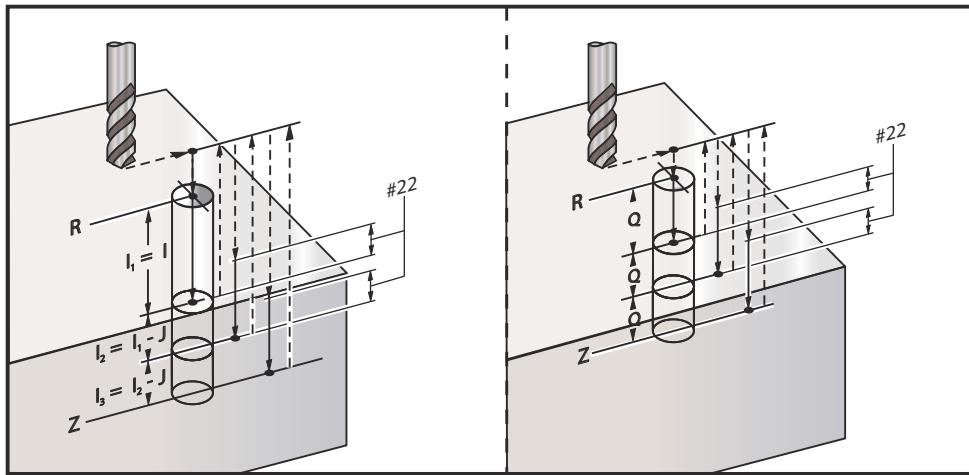
If I, J, and K are specified, the first pass will cut in by the amount of I, each succeeding cut will be reduced by amount J, and the minimum cutting depth is K. Do not use a Q value when programming with I, J, and K.

If P is specified, the tool will pause at the bottom of the hole for that amount of time. The following example will peck several times and dwell for 1.5 seconds:

```
G83 Z-0.62 F15. R0.1 Q0.175 P1.5 ;
```

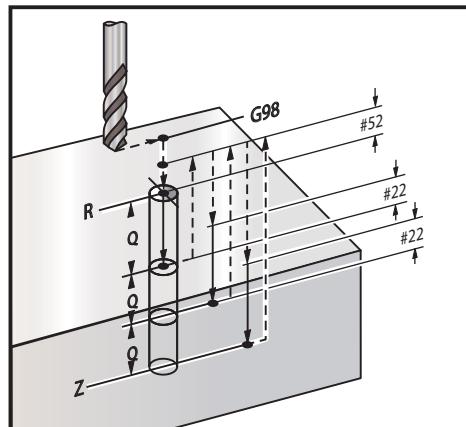
The same dwell time will apply to all subsequent blocks that do not specify a dwell time.

F7.30: G83 Peck Drilling with I, J, K and Normal Peck Drilling: [#22] Setting 22.



Setting 52 changes the way G83 works when it returns to the R plane. Usually the R plane is set well above the cut to ensure that the peck motion allows the chips to get out of the hole. This wastes time as the drill starts by drilling empty space. If Setting 52 is set to the distance required to clear chips, you can set the R plane much closer to the part. When the chip-clearing move to R occurs, Setting 52 determines the Z-Axis distance above R.

F7.31: G83 peck Drilling Canned Cycle with Setting 52 [#52]



```
%  
O60831 (G83 PECK DRILLING CANNED CYCLE) ;  
(G54 X0 Y0 is at the top-left of part) ;  
(Z0 is on top of the part) ;  
(T1 is a 0.3125 in. stub drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;
```

```
G00 G54 X2. Y-2. (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Activate tool offset 1) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G83 Z-0.720 Q0.175 R0.1 F15. (Begin G83) ;
(Drill 1st hole at current X Y location) ;
X2. Y-4. (2nd hole) ;
X4. Y-4. (3rd hole) ;
X4. Y-2. (4th hole) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

G84 Tapping Canned Cycle (Group 09)

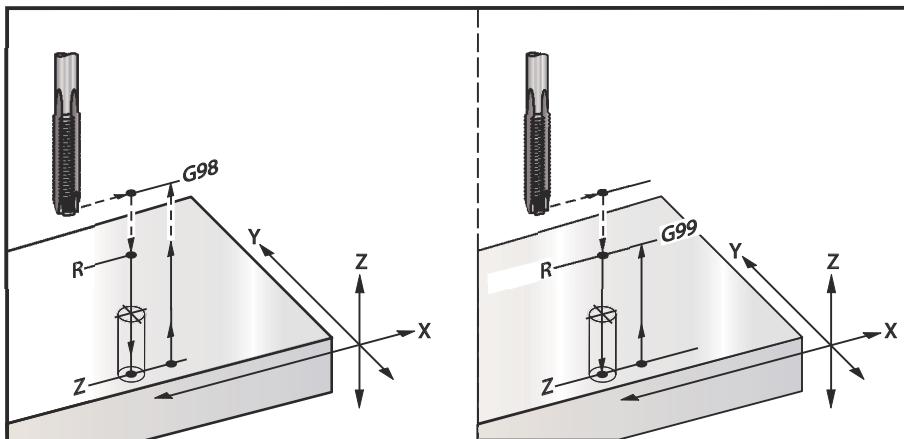
- * **E** - Chip-clean RPM (Spindle reverses to remove chips after each cycle)
 - F** - Feedrate
 - * **J** - Retract Multiple (Example: **J2** retracts twice as fast as the cutting speed, also refer to Setting 130)
 - * **L** - Number of holes if **G91** (Incremental Mode) is used
 - * **R** - Position of the R plane (Position above the part)
 - * **X** - X-Axis location of hole
 - * **Y** - Y-Axis location of hole
 - Z** - Position of the Z Axis at the bottom of hole
 - * **S** - Spindle speed
- * indicates optional



NOTE:

*You do not need to command a spindle start (M03 / M04) before G84.
The canned cycle starts and stops the spindle as needed.*

F7.32: G84 Tapping Canned Cycle



```
%  
O60841 (G84 TAPPING CANNED CYCLE) ;  
(G54 X0 Y0 is at the top-left of part) ;  
(Z0 is on top of the part) ;  
(T1 is a 3/8-16 tap) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y-2. (Rapid to 1st position) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G84 Z-0.600 R0.1 F56.25 S900 (Begin G84) ;  
(900 rpm divided by 16 tpi = 56.25 ipm) ;  
(Drill 1st hole at current X Y location) ;  
X2. Y-4. (2nd hole) ;  
X4. Y-4. (3rd hole) ;  
X4. Y-2. (4th hole) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z1. M09 (Canned cycle off, rapid retract) ;  
(Coolant off) ;  
G53 G49 Z0 (Z home) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

G85 Bore In, Bore Out Canned Cycle (Group 09)

F - Feedrate

***L** - Number of holes if G91 (Incremental Mode) is used

***R** - Position of the R plane (position above the part)

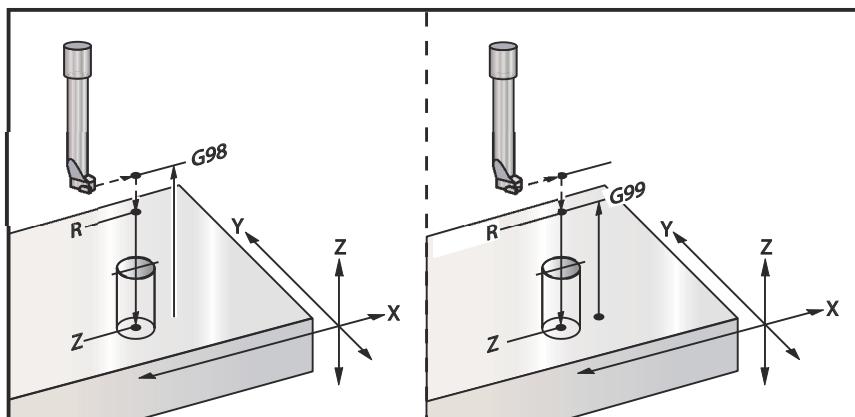
***X** - X-Axis location of holes

***Y** - Y-Axis location of holes

Z - Position of the Z Axis at the bottom of hole

* indicates optional

F7.33: G85 Boring Canned Cycle



G86 Bore and Stop Canned Cycle (Group 09)

F - Feedrate

***L** - Number of holes if G91 (Incremental Mode) is used

***R** - Position of the R plane (position above the part)

***X** - X-Axis location of hole

***Y** - Y-Axis location of hole

Z - Position of the Z Axis at the bottom of hole

* indicates optional

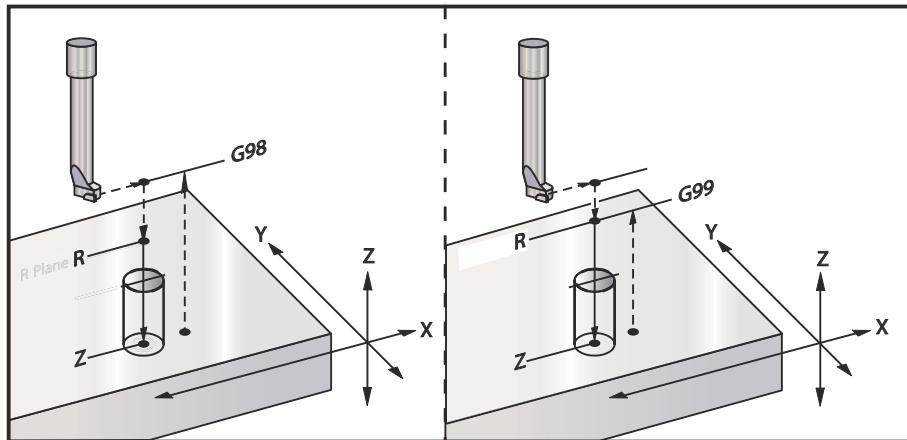


CAUTION:

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a "no-spin" cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

This G code will stop the spindle once the tool reaches the bottom of the hole. The tool is retracted once the spindle has stopped.

F7.34: G86 Bore and Stop Canned Cycles



G89 Bore In, Dwell, Bore Out Canned Cycle (Group 09)

F - Feedrate

L - Number of holes if G91 (Incremental Mode) is used

P - The dwell time at the bottom of the hole

***R** - Position of the R plane (position above the part)

X - X-Axis location of holes

Y - Y-Axis location of holes

Z - Position of the Z Axis at the bottom of hole

* indicates optional



NOTE:

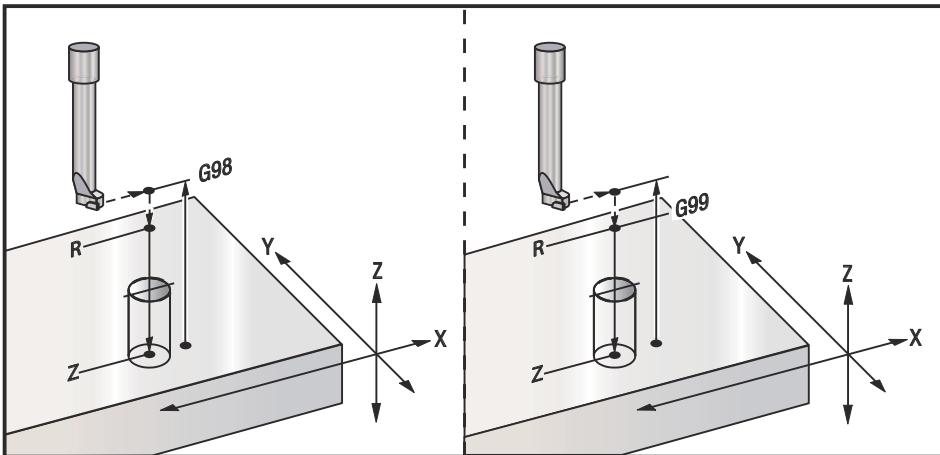
The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.



CAUTION:

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a "no-spin" cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

F7.35: G89 Bore and Dwell and Canned Cycle



G90 Absolute / G91 Incremental Position Commands (Group 03)

These G codes change the way the axis commands are interpreted. Axes commands following a G90 will move the axes to the machine coordinate. Axes commands following a G91 will move the axis that distance from the current point. G91 is not compatible with G143 (5-Axis Tool Length Compensation).

The Basic Programming section of this manual, beginning on page 190, includes a discussion of absolute versus incremental programming.

G92 Set Work Coordinate Systems Shift Value (Group 00)

This G-code does not move any of the axes; it only changes the values stored as user work offsets. G92 works differently depending on Setting 33, which selects a FANUC or HAAS coordinate system.

FANUC or HAAS

If Setting 33 is set to **FANUC** or **HAAS**, a G92 command shifts all work coordinate systems (G54-G59, G110-G129) so that the commanded position becomes the current position in the active work system. G92 is non-modal.

A G92 command cancels any G52 in effect for the commanded axes. Example: G92 X1.4 cancels the G52 for the X-Axis. The other axes are not affected.

The G92 shift value is displayed at the bottom of the Work Offsets page and may be cleared there if necessary. It is also cleared automatically after power-up, and any time [**ZERO RETURN**] and [**ALL**] or [**ZERO RETURN**] and [**SINGLE**] are used.

G92 Clear Shift Value From Within a Program

G92 shifts may be canceled by programming another G92 shift to change the current work offset back to the original value.

```
%  
O60921 (G92 SHIFT WORK OFFSETS) ;  
(G54 X0 Y0 Z0 is at the center of mill travel) ;  
G00 G90 G54 X0 Y0 (Rapid to G54 origin) ;  
G92 X2. Y2. (Shifts current G54) ;  
G00 G90 G54 X0 Y0 (Rapid to G54 origin) ;  
G92 X-2. Y-2. (Shifts current G54 back to original) ;  
G00 G90 G54 X0 Y0 (Rapid to G54 origin) ;  
M30 (End program) ;  
%
```

G93 Inverse Time Feed Mode (Group 05)

F - Feed Rate (strokes per minute)

This G code specifies that all F (feedrate) values are interpreted as strokes per minute. In other words the time (in seconds) to complete the programmed motion using G93 is, 60 (seconds) divided by the F value.

G93 is generally used in 4 and 5-axis work when the program is generated using a CAM system. G93 is a way of translating the linear (inches/min) feedrate into a value that takes rotary motion into account. When G93 is used, the F value will tell you how many times per minute the stroke (tool move) can be repeated.

When G93 is used, feedrate (F) is mandatory for all interpolated motion blocks. Therefore each non-rapid motion block must have its own feedrate (F) specification.



NOTE:

Pressing [RESET] will set the machine to G94 (Feed per Minute) mode. Settings 34 and 79 (4th & 5th axis diameter) are not necessary when using G93.

G94 Feed Per Minute Mode (Group 05)

This code deactivates G93 (Inverse Time Feed Mode) and returns the control to Feed Per Minute mode.

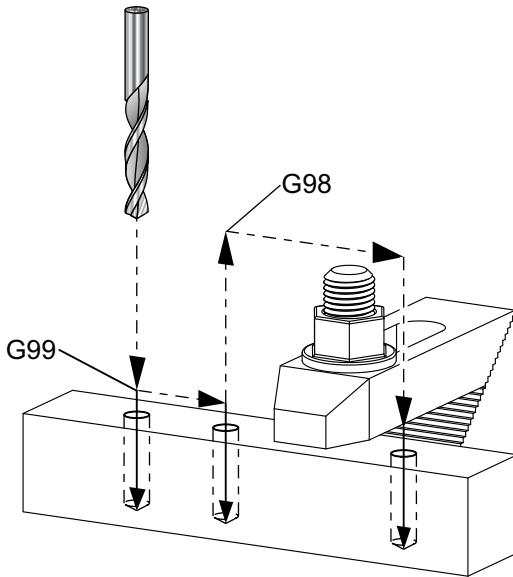
G95 Feed per Revolution (Group 05)

When G95 is active, a spindle revolution will result in a travel distance specified by the Feed value. If Setting 9 is set to INCH, then the feed value F will be taken as inches/rev (set to MM, then the feed will be taken as mm/rev). Feed Override and Spindle Override will affect the behavior of the machine while G95 is active. When a Spindle Override is selected, any change in the spindle speed will result in a corresponding change in feed in order to keep the chip load uniform. However, if a Feed Override is selected, then any change in the Feed Override will only affect the feed rate and not the spindle.

G98 Canned Cycle Initial Point Return (Group 10)

Using G98, the Z-Axis returns to its initial starting point (the Z position in the block before the canned cycle) between each X/Y position. This lets you program up and around areas of the part, clamps, and fixtures.

- F7.36:** G98 Initial Point Return. After the second hole, the Z Axis returns to the starting position [G98] to move over the toe clamp to the next hole position.



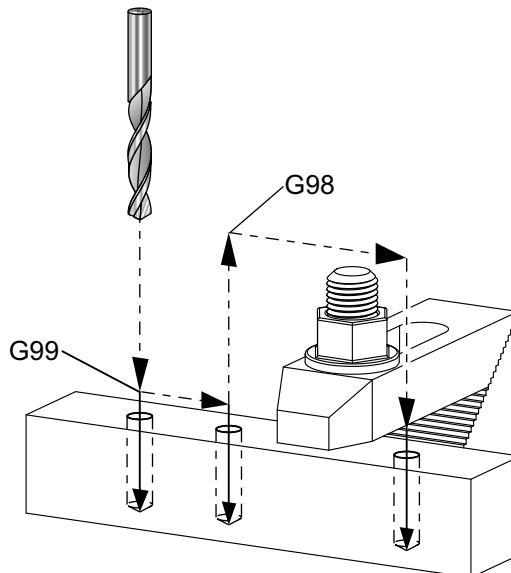
```
%  
O69899 (G98/G99 INITIAL POINT & R PLANE RETURN) ;  
(G54 X0 Y0 is top right corner of part) ;  
(Z0 is on top of the part) ;  
(T1 is a drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G17 G40 G49 G54 (Safe startup) ;  
G00 G54 X1. Y-0.5 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z2. (Tool offset 1 on) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G81 G99 X1. Z-0.5 F10. R0.1 (Begin G81 using G99) ;  
G98 X2. (2nd hole and then clear clamp with G98) ;  
X4. (Drill 3rd hole) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z2. M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;
```

```
M30 (End program) ;
%
```

G99 Canned Cycle R Plane Return (Group 10)

Using G99, the Z-Axis will stay at the R plane between each X and/or Y location. When obstructions are not in the path of the tool G99 saves machining time.

- F7.37:** G99R Plane Return. After the first hole, the Z Axis returns to the R plane position [G99] and moves to the second hole position. This is a safe move in this case because there are no obstacles.



```
%  
O69899 (G98/G99 INITIAL POINT & R PLANE RETURN) ;  
(G54 X0 Y0 is top right corner of part) ;  
(Z0 is on top of the part) ;  
(T1 is a drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G17 G40 G49 G54 (Safe startup) ;  
G00 G54 X1. Y-0.5 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z2. (Tool offset 1 on) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G81 G99 X1. Z-0.5 F10. R0.1 (Begin G81 using G99) ;  
G98 X2. (2nd hole and then clear clamp with G98) ;  
X4. (Drill 3rd hole) ;
```

```
(BEGIN COMPLETION BLOCKS) ;
G00 Z2. M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

G100 Disable / G101 Enable Mirror Image (Group 00)

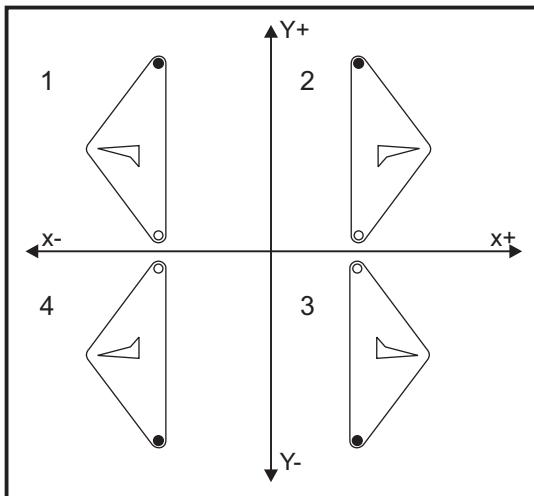
- ***X** - X-Axis command
- ***Y** - Y-Axis command
- ***Z** - Z-Axis command
- ***A** - A-Axis command
- ***B** - B-Axis command
- ***C** - C-Axis command

* indicates optional

Programmable mirror imaging is used to turn on or off any of the axes. When one is **ON**, axis motion may be mirrored (or reversed) around the work zero point. These G codes should be used in a command block without any other G codes. They do not cause any Axis motion. The bottom of the screen indicates when an axis is mirrored. Also see Settings 45, 46, 47, 48, 80, and 250 for mirror imaging.

The format for turning Mirror Image on and off is:

```
G101 X0. (turns on mirror imaging for the X-Axis) ;
G100 X0. (turns off mirror imaging for the X-Axis) ;
```

F7.38: X-Y Mirror Image**G103 Limit Block Look-Ahead (Group 00)**

G103 specifies the maximum number of blocks the control looks ahead (Range 0-15), for example:

```
G103 [P..] ;
```

During machine motions, the control prepares future blocks (lines of code) ahead of time. This is commonly called “Block Look-ahead.” While the control executes the current block, it has already interpreted and prepared the next block for continuous motion.

A program command of G103 P0, or simply G103, disables block limiting. A program command of G103 Pn limits look-ahead to n blocks.

G103 is useful for debugging macro programs. The control interprets Macro expressions during look-ahead time. If you insert a G103 P1 into the program, the control interprets macro expressions (1) block ahead of the currently executing block.

It is best to add several empty lines after a G103 P1 is called. This ensures that no lines of code after the G103 P1 are interpreted until they are reached.

G103 affects cutter compensation and High Speed Machining.

**NOTE:**

The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.

G107 Cylindrical Mapping (Group 00)

***X** - X-Axis command
***Y** - Y-Axis command
***Z** - Z-Axis command
***A** - A-Axis command
***B** - B-Axis command
C - C-Axis command
***Q** - Diameter of the cylindrical surface
***R** - Radius of the rotary axis

* indicates optional

This G code translates all programmed motion occurring in a specified linear axis into the equivalent motion along the surface of a cylinder (as attached to a rotary axis) as shown in the following figure. It is a Group 0 G code, but its default operation is subject to Setting 56 (M30 Restores Default G). The G107 command is used to either activate or deactivate cylindrical mapping.

- Any linear-axis program can be cylindrically mapped to any rotary axis (one at a time).
- An existing linear-axis G-code program can be cylindrically mapped by inserting a G107 command at the beginning of the program.
- The radius (or diameter) of the cylindrical surface can be redefined, allowing cylindrical mapping to occur along surfaces of different diameters without having to change the program.
- The radius (or diameter) of the cylindrical surface can either be synchronized with or be independent of the rotary axis diameter(s) specified in the Settings 34 and 79.
- G107 can also be used to set the default diameter of a cylindrical surface, independently of any cylindrical mapping that may be in effect.

G110-G129 Coordinate System #7-26 (Group 12)

These codes select one of the additional work coordinate systems. All subsequent references to axis positions will be interpreted in the new coordinate system. Operation of G110 to G129 is the same as G54 to G59.

G136 Automatic Work Offset Center Measurement (Group 00)

This G-code is optional and requires a probe. Use it to set work offsets to the center of a work piece with a work probe.

F - Feedrate

- ***I** - Optional offset distance along X-Axis
- ***J** - Optional offset distance along Y-Axis
- ***K** - Optional offset distance along Z-Axis
- ***X** - Optional X-Axis motion command
- ***Y** - Optional Y-Axis motion command
- ***Z** - Optional Z-Axis motion command

* indicates optional

Automatic Work Offset Center Measurement (G136) is used to command a spindle probe to set work offsets. A G136 will feed the axes of the machine in an effort to probe the work piece with a spindle mounted probe. The axis (axes) will move until a signal (skip signal) from the probe is received or the end of the programmed move is reached. Tool compensation (G41, G42, G43, or G44) must not be active when this function is performed. The currently active work coordinate system is set for each axis programmed. Use a G31 cycle with an M75 to set the first point. A G136 will set the work coordinates to a point at the center of a line between the probed point and the point set with an M75. This allows the center of the part to be found using two separate probed points.

If an I, J, or K is specified, the appropriate axis work offset is shifted by the amount in the I, J, or K command. This allows the work offset to be shifted away from the measured center of the two probed points.

Notes:

This code is non-modal and only applies to the block of code in which G136 is specified.

The points probed are offset by the values in Settings 59 through 62. See the Settings section of this manual for more information.

Do not use Cutter Compensation (G41, G42) with a G136.

Do not use tool length Compensation (G43, G44) with G136

To avoid damaging the probe, use a feed rate below F100. (inch) or F2500. (metric).

Turn on the spindle probe before using G136.

If your mill has the standard Renishaw probing system, use the following commands to turn on the spindle probe:

```
M59 P1134 ;
```

Use the following commands to turn off the spindle probe:

```
M69 P1134 ;
```

Also see M75, M78, and M79.

Also see G31.

This sample program measures the center of a part in the Y Axis and records the measured value to the G58 Y Axis work offset. To use this program, the G58 work offset location must be set at or close to the center of the part to be measured.

```
%  
O61361 (G136 AUTO WORK OFFSET - CENTER OF PART) ;  
(G58 X0 Y0 is at the center of part) ;  
(Z0 is on top of the part) ;  
(T1 is a spindle probe) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G58 X0. Y1. (Rapid to 1st position) ;  
(BEGIN PROBING BLOCKS) ;  
M59 P1134 (Spindle probe on) ;  
Z-10. (Rapid spindle down to position) ;  
G91 G01 Z-1. F20. (Incremental feed by Z-1.) ;  
G31 Y-1. F10. M75 (Measure & record Y reference) ;  
G01 Y0.25 F20. (Feed away from surface) ;  
G00 Z2. (Rapid retract) ;  
Y-2. (Move to opposite side of part) ;  
G01 Z-2. F20. (Feed by Z-2.) ;  
G136 Y1. F10. ;  
(Measure and record center in the Y axis) ;  
G01 Y-0.25 (Feed away from surface) ;  
G00 Z1. (Rapid retract) ;  
M69 P1134 (Spindle probe off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G90 G53 Z0. (Rapid retract to Z home) ;  
M30 (End program) ;  
%
```

G141 3D+ Cutter Compensation (Group 07)

X - X-Axis command

Y - Y-Axis command

Z - Z-Axis command

***A** - A-Axis command (optional)

***B** - B-Axis command (optional)

***D** - Cutter Size Selection (modal)

I - X-Axis cutter compensation direction from program path

J - Y-Axis cutter compensation direction from program path

K - Z-Axis cutter compensation direction from program path

F - Feedrate

* indicates optional

This feature performs three-dimensional cutter compensation.

The form is:

```
G141 Xnnn Ynnn Znnn Innn Jnnn Knnn Fnnn Dnnn
```

Subsequent lines can be:

```
G01 Xnnn Ynnn Znnn Innn Jnnn Knnn Fnnn ;
```

Or

```
G00 Xnnn Ynnn Znnn Innn Jnnn Knnn ;
```

Some CAM systems are able to output the **X**, **Y**, and **Z** with values for **I**, **J**, **K**. The **I**, **J**, and **K** values tell the control the direction in which to apply the compensation at the machine. Similar to other uses of **I**, **J**, and **K**, these are incremental distances from the **X**, **Y**, and **Z** point called.

The **I**, **J**, and **K** specify the normal direction, relative to the center of the tool, to the contact point of the tool in the CAM system. The **I**, **J**, and **K** vectors are required by the control to be able to shift the toolpath in the correct direction. The value of the compensation can be in a positive or negative direction.

The offset amount entered in radius or diameter (Setting 40) for the tool will compensate the path by this amount, even if the tool motions are 2 or 3 axes. Only **G00** and **G01** can use **G141**. A **Dnn** will have to be programmed; the **D**-code selects which tool wear diameter offset to use. A feedrate must be programmed on each line if in **G93** Inverse Time Feed mode.

With a unit vector, the length of the vector line must always equal 1. In the same way that a unit circle in mathematics is a circle with a radius of 1, a unit vector is a line that indicates a direction with a length of 1. Remember, the vector line does not tell the control how far to move the tool when a wear value is entered, just the direction in which to go.

Only the endpoint of the commanded block is compensated in the direction of I, J, and K. For this reason, this compensation is recommended only for surface toolpaths having a tight tolerance (small motion between blocks of code). G141 compensation does not prohibit the toolpath from crossing over itself when excessive cutter compensation is entered. The tool will be offset, in the direction of the vector line, by the combined values of the tool offset geometry plus the tool offset wear. If compensation values are in diameter mode (Setting 40), the move will be half the amount entered in these fields.

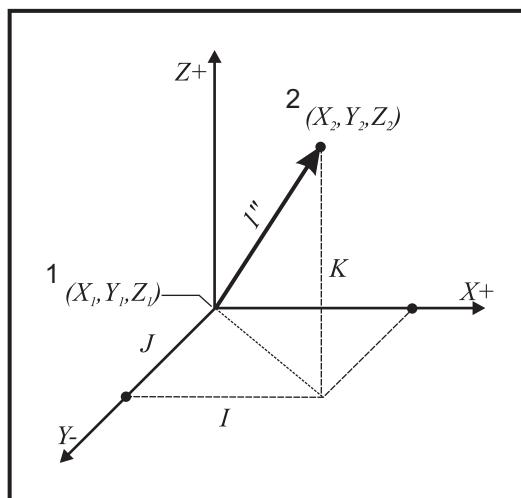
For best results, program from the tool center using a ball nose endmill.

```
%  
O61411 (G141 3D CUTTER COMPENSATION) ;  
(G54 X0 Y0 is at the bottom-left) ;  
(Z0 is on top of the part) ;  
(T1 is a ball nose endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 Z0 A0 B0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G141 D01 X0. Y0. Z0. ;  
(Rapid to position with 3D+ cutter comp) ;  
G01 G93 X.01 Y.01 Z.01 I.1 J.2 K.9747 F300. ;  
(Inverse time feed on, 1st linear motion) ;  
N1 X.02 Y.03 Z.04 I.15 J.25 K.9566 F300. (2nd motion) ;  
X.02 Y.055 Z.064 I.2 J.3 K.9327 F300. (3rd motion) ;  
X2.345 Y.1234 Z-1.234 I.25 J.35 K.9028 F200. ;  
(Last motion) ;  
(BEGIN COMPLETION BLOCKS) ;  
G94 F50. (Inverse time feed off) ;  
G00 G90 G40 Z0.1 M09 (Cutter comp off) ;  
(Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

In the above example, we can see where the I, J, and K were derived by plugging the points into the following formula:

$AB = [(x_2-x_1)^2 + (y_2-y_1)^2 + (z_2-z_1)^2]$, a 3D version of the distance formula. Looking at line N1, we use 0.15 for x_2 , 0.25 for y_2 , and 0.9566 for Z_2 . Because I, J, and K are incremental, we will use 0 for x_1 , y_1 , and z_1 .

- F7.39:** Unit Vector Example: The commanded line endpoint [1] is compensated in the direction of the vector line [2](I,J,K), by the amount of the Tool Offset Wear.



$$\begin{aligned} \text{AB} &= [(.15)^2 + (.25)^2 + (.9566)^2] \\ \text{AB} &= [.0225 + .0625 + .9150] \\ \text{AB} &= 1 \end{aligned}$$

A simplified example is listed below:

```
%  
O61412 (G141 SIMPLE 3D CUTTER COMPENSATION) ;  
(G54 X0 Y0 is at the bottom-left) ;  
(Z0 is on top of the part) ;  
(T1 is a ball nose endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;
```

```
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G141 D01 X0. Y0. Z0. ;
(Rapid to position with 3D+ cutter compensation) ;
N1 G01 G93 X5. Y0. I0. J-1. K0. F300. ;
(Inverse time feed on & linear motion) ;
(BEGIN COMPLETION BLOCKS) ;
G94 F50. (Inverse time feed off) ;
G00 G90 G40 Z0.1 M09 (Cutter compensation off) ;
(Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

In this case, the wear value (DIA) for T01 is set to -.02. Line N1 moves the tool from (X0., Y0., Z0.) to (X5., Y0., Z0.). The J value tells the control to compensate the endpoint of the programmed line only in the Y Axis.

Line N1 could have been written using only the J-1. (not using I0. or K0.), but a Y value must be entered if a compensation is to be made in this axis (J value used).

G143 5-Axis Tool Length Compensation + (Group 08)

(This G-code is optional; it only applies to machines on which all rotary motion is movement of the cutting tool, such as VR-series mills)

This G code allows the user to correct for variations in the length of cutting tools without the need for a CAD/CAM processor. An H code is required to select the tool length from the existing length compensation tables. A G49 or H00 command will cancel 5-axis compensation. For G143 to work correctly there must be two rotary axes, A and B. G90, absolute positioning mode must be active (G91 cannot be used). Work position 0,0 for the A and B axes must be so the tool is parallel with Z-Axis motion.

The intention behind G143 is to compensate for the difference in tool length between the originally posted tool and a substitute tool. Using G143 allows the program to run without having to repost a new tool length.

G143 tool length compensation works only with rapid (G00) and linear feed (G01) motions; no other feed functions (G02 or G03) or canned cycles (drilling, tapping, etc.) can be used. For a positive tool length, the Z-Axis would move upward (in the + direction). If one of X, Y or Z is not programmed, there will be no motion of that axis, even if the motion of A or B produces a new tool length vector. Thus a typical program would use all 5 axes on one block of data. G143 may effect commanded motion of all axes in order to compensate for the A and B axes.

Inverse feed mode (G93) is recommended, when using G143.

```

%
O61431 (G143 5-AXIS TOOL LENGTH) ;
(G54 X0 Y0 is at the top-right) ;
(Z0 is on top of the part) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X0 Y0 Z0 A0 B0 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G143 H01 X0. Y0. Z0. A-20. B-20. ;
(Rapid to position w/ 5 Axis tool length comp) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 G93 X.01 Y.01 Z.01 A-19.9 B-19.9 F300. ;
(Inverse time feed on , 1st linear motion) ;
X0.02 Y0.03 Z0.04 A-19.7 B-19.7 F300. ( 2nd motion) ;
X0.02 Y0.055 Z0.064 A-19.5 B-19.6 F300. (3rd motion) ;
X2.345 Y.1234 Z-1.234 A-4.127 B-12.32 F200. ;
(Last motion) ;
(BEGIN COMPLETION BLOCKS) ;
G94 F50. (Inverse time feed off) ;
G00 G90 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Tool length comp off) ;
(Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%

```

G150 General Purpose Pocket Milling (Group 00)

- D** - Tool radius/diameter offset selection
- F** - Feedrate
- I** - X-Axis cut increment (positive value)
- J** - Y-Axis cut increment (positive value)
- K** - Finishing pass amount (positive value)
- P** - Subprogram number that defines pocket geometry
- Q** - Incremental Z-Axis cut depth per pass (positive value)
- *R** - Position of the rapid R-plane location
- *S** - Spindle speed
- X** - X start position
- Y** - Y start position
- Z** - Final depth of pocket

* indicates optional

The G150 starts by positioning the cutter to a start point inside the pocket, followed by the outline, and completes with a finish cut. The end mill will plunge in the Z-Axis. A subprogram P### is called, which defines the pocket geometry of a closed area using G01, G02, and G03 motions in the X and Y axes on the pocket. The G150 command will search for an internal subprogram with a N-number specified by the P-code. If that is not found the control will search for an external subprogram. If neither are found, alarm 314 Subprogram Not In Memory will be generated.



NOTE:

When defining the G150 pocket geometry in the subprogram, do not move back to the starting hole after the pocket shape is closed.



NOTE:

The pocket geometry subprogram cannot use macro variables.

An I or J value defines the roughing pass amount the cutter moves over for each cut increment. If I is used, the pocket is roughed out from a series of increment cuts in the X-Axis. If J is used, the increment cuts are in the Y-Axis.

The K command defines a finish pass amount on the pocket. If a K value is specified, a finish pass is performed by K amount, around the inside of pocket geometry for the last pass and is done at the final Z depth. There is no finishing pass command for the Z depth.

The R value needs to be specified, even if it is zero (R0), or the last R value that was specified will be used.

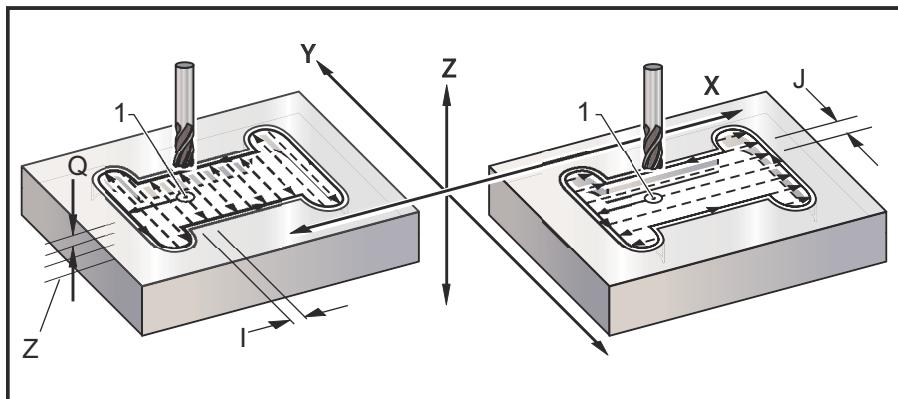
Multiple passes in the pocket area are done, starting from the R plane, with each Q (Z-Axis depth) pass to the final depth. The G150 command will first make a pass around pocket geometry, leaving stock with K, then doing passes of I or J roughing out inside of pocket after feeding down by the value in Q until the Z depth is reached.

The Q command must be in the G150 line, even if only one pass to the Z depth is desired. The Q command starts from the R plane.

Notes: The subprogram (P) must not consist of more than 40 pocket geometry moves.

It may be necessary to drill a starting point, for the G150 cutter, to the final depth (Z). Then position the end mill to the start location in the XY axes within the pocket for the G150 command.

F7.40: G150 General Pocket Milling: [1] Start Point, [Z] Final depth.



```

%
O61501 (G150 GENERAL POCKET MILLING) ;
(G54 X0 Y0 is at the bottom-left) ;
(Z0 is on top of the part) ;
(T1 is a .5" endmill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X3.25 Y4.5 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z1.0 (Activate tool offset 1) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G150 X3.25 Y4.5 Z-1.5 G41 J0.35 K.01 Q0.25 R.1 P61502 D01 F15.
;
(Pocket mill sequence, call pocket subprogram) ;
(Cutter comp on) ;
(0.01" finish pass K on sides) ;
G40 X3.25 Y4.5 (Cutter comp off) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
%
O61502 (G150 GENERAL POCKET MILL SUBPROGRAM) ;
(Subprogram for pocket in O61501) ;
(Must have a feedrate in G150) ;
G01 Y7. (First linear move onto pocket geometry) ;
X1.5 (Linear move) ;
G03 Y5.25 R0.875 (CCW arc) ;

```

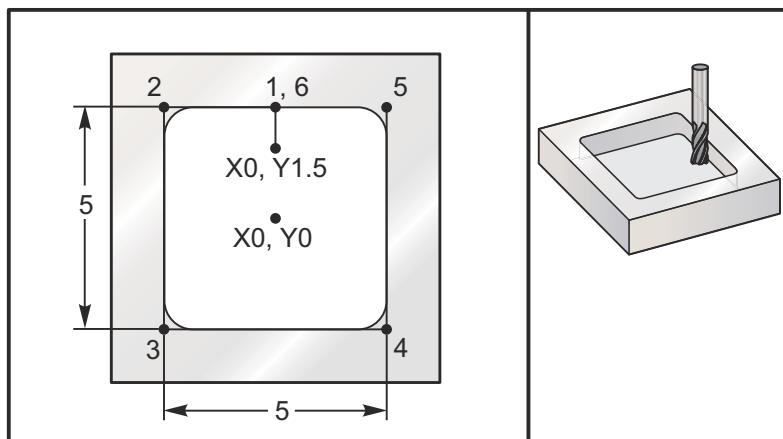
```

G01 Y2.25 (Linear move) ;
G03 Y0.5 R0.875 (CCW arc) ;
G01 X5. (Linear move) ;
G03 Y2.25 R0.875 (CCW arc) ;
G01 Y5.25 (Linear move) ;
G03 Y7. R0.875 (CCW arc) ;
G01 X3.25 (Close pocket geometry) ;
M99 (Exit to Main Program) ;
%

```

Square Pocket

F7.41: G150 General Purpose Pocket Milling: 0.500 diameter endmill.

**5.0 x 5.0 x 0.500 DP. Square Pocket****Main Program**

```

%
O61503 (G150 SQUARE POCKET MILLING) ;
(G54 X0 Y0 is at the center of the part) ;
(Z0 is on top of the part) ;
(T1 is a .5" endmill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X0 Y1.5 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z1.0 (Activate tool offset 1) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z0.1 F10. (Feed right above the surface) ;

```

```

G150 P61504 Z-0.5 Q0.25 R0.01 J0.3 K0.01 G41 D01 F10. ;
(Pocket Mill sequence, call pocket subprogram) ;
(Cutter comp on) ;
(0.01" finish pass K on sides) ;
G40 G01 X0. Y1.5 (Cutter comp off) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract,Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%

```

Subprogram

```

%
O61505 (G150 INCREMENTAL SQUARE POCKET MILLING SUBPROGRAM) ;
(Subprogram for pocket in O61503) ;
(Must have a feedrate in G150) ;
G91 G01 Y0.5 (Linear move to position 1) ;
X-2.5 (Linear move to position 2) ;
Y-5. (Linear move to position 3) ;
X5. (Linear move to position 4) ;
Y5. (Linear move to position 5) ;
X-2.5 (Linear move to position 6, Close Pocket Loop) ;
G90 (Turn off incremental mode, Turn on absolute) ;
M99 (Exit to Main Program) ;
%

```

Absolute and Incremental examples of a subprogram called up by the P#### command in the G150 line:

Absolute Subprogram

```

%
O61504 (G150 ABSOLUTE SQUARE POCKET MILLING SUBPROGRAM) ;
(Subprogram for pocket in O61503) ;
(Must have a feedrate in G150) ;
G90 G01 Y2.5 (Linear move to position 1) ;
X-2.5 (Linear move to position 2) ;
Y-2.5 (Linear move to position 3) ;
X2.5 (Linear move to position 4) ;
Y2.5 (Linear move to position 5) ;
X0. (Linear move to position 6, Close Pocket Loop) ;
M99 (Exit to Main Program) ;
%

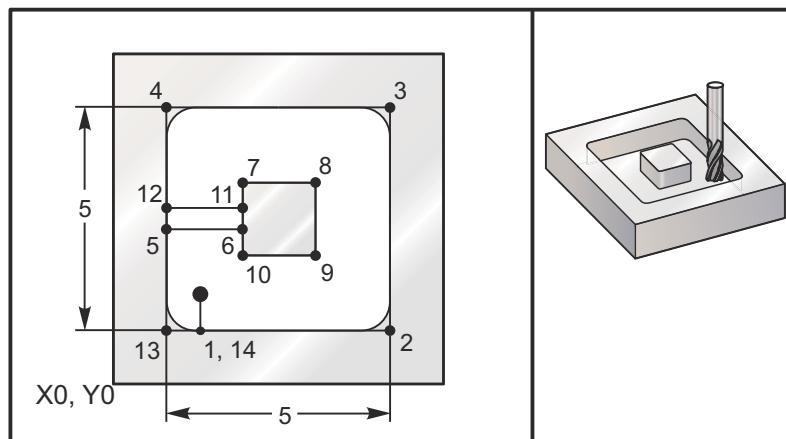
```

Incremental Subprogram

```
%  
O61505 (G150 INCREMENTAL SQUARE POCKET MILLING SUBPROGRAM) ;  
(Subprogram for pocket in O61503) ;  
(Must have a feedrate in G150) ;  
G91 G01 Y0.5 (Linear move to position 1) ;  
X-2.5 (Linear move to position 2) ;  
Y-5. (Linear move to position 3) ;  
X5. (Linear move to position 4) ;  
Y5. (Linear move to position 5) ;  
X-2.5 (Linear move to position 6, Close Pocket Loop) ;  
G90 (Turn off incremental mode, Turn on absolute) ;  
M99 (Exit to Main Program) ;  
%
```

Square Island

F7.42: G150 Pocket Milling Square Island: 0.500 diameter endmill.



5.0 x 5.0 x 0.500 DP. Square Pocket with Square Island

Main Program

```
%  
O61506 (G150 SQUARE ISLAND POCKET MILLING) ;  
(G54 X0 Y0 is at the bottom-left) ;  
(Z0 is on top of the part) ;  
(T1 is a .5" endmill) ;  
(BEGIN PREPARATION BLOCKS) ;
```

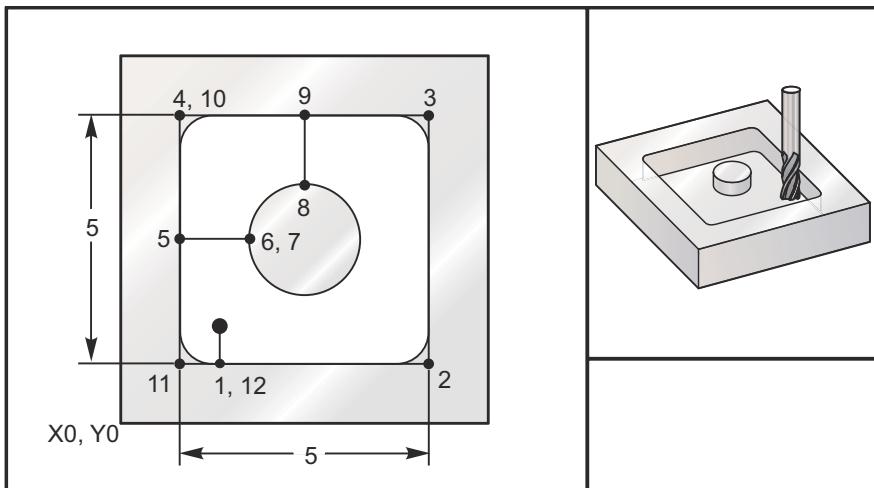
```
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X2. Y2. (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z1.0(Activate tool offset 1) ;
M08(Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z0.01 F30. (Feed right above the surface) ;
G150 P61507 X2. Y2. Z-0.5 Q0.5 R0.01 I0.3 K0.01 G41 D01 F10. ;
(Pocket mill sequence, call pocket subprogram) ;
(Cutter comp off) ;
(0.01" finish pass K on sides) ;
G40 G01 X2.Y2. (Cutter comp off) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

Subprogram

```
%  
O61507 (G150 SQUARE ISLAND POCKET MILLING SUBPROGRAM) ;
(Subprogram for pocket in O61503) ;
(Must have a feedrate in G150) ;
G01 Y1. (Linear move to position 1) ;
X6. (Linear move to position 2) ;
Y6. (Linear move to position 3) ;
X1. (Linear move to position 4) ;
Y3.2 (Linear move to position 5) ;
X2.75 (Linear move to position 6) ;
Y4.25 (Linear move to position 7) ;
X4.25 (Linear move to position 8) ;
Y2.75 (Linear move to position 9) ;
X2.75 (Linear move to position 10) ;
Y3.8 (Linear move to position 11) ;
X1. (Linear move to position 12) ;
Y1. (Linear move to position 13) ;
X2. (Linear move to position 14, Close Pocket Loop) ;
M99 (Exit to Main Program) ;
%
```

Round Island

F7.43: G150 Pocket Milling Round Island: 0.500 diameter endmill.



5.0 x 5.0 x 0.500 DP. Square Pocket with Round Island

Main Program

```
%  
O61508 (G150 SQ POCKET W/ ROUND ISLAND MILLING) ;  
(G54 X0 Y0 is at the bottom-left) ;  
(Z0 is on top of the part) ;  
(T1 is a .5" endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y2. (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z1.0 M08 (Activate tool offset 1) ;  
(Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z0.01 F30. (Feed right above the surface) ;  
G150 P61509 X2. Y2. Z-0.5 Q0.5 R0.01 J0.3 K0.01 G41 D01 F10. ;  
(Pocket mill sequence, call pocket subprogram) ;  
(Cutter comp on) ;  
(0.01" finish pass K on sides) ;  
G40 G01 X2.Y2. (Cutter comp off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;
```

%

Subprogram

```
%  
O61509 (G150 SQ POCKET W/ ROUND ISLAND MILLING SUBPROGRAM) ;  
(Subprogram for pocket in O61503) ;  
(Must have a feedrate in G150) ;  
G01 Y1. (Linear move to position 1) ;  
X6. (Linear move to position 2) ;  
Y6. (Linear move to position 3) ;  
X1. (Linear move to position 4) ;  
Y3.5 (Linear move to position 5) ;  
X2.5 (Linear move to position 6) ;  
G02 I1. (CW circle along X axis at position 7) ;  
G02 X3.5 Y4.5 R1. (CW arc to position 8) ;  
G01 Y6. (Linear move to position 9) ;  
X1. (Linear move to position 10) ;  
Y1. (Linear move to position 11) ;  
X2. (Linear move to position 12, Close Pocket Loop) ;  
M99 (Exit to Main Program) ;  
%
```

G154 Select Work Coordinates P1-P99 (Group 12)

This feature provides 99 additional work offsets. G154 with a P value from 1 to 99 activates additional work offsets. For example G154 P10 selects work offset 10 from the list of additional work offsets.


NOTE:

G110 to G129 refer to the same work offsets as G154 P1 through P20; they can be selected by using either method.

When a G154 work offset is active, the heading in the upper right work offset will show the G154 P value.


NOTE:

The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.

G154 work offsets format

#14001-#14006 G154 P1 (also #7001-#7006 and G110)
#14021-#14026 G154 P2 (also #7021-#7026 and G111)
#14041-#14046 G154 P3 (also #7041-#7046 and G112)
#14061-#14066 G154 P4 (also #7061-#7066 and G113)
#14081-#14086 G154 P5 (also #7081-#7086 and G114)
#14101-#14106 G154 P6 (also #7101-#7106 and G115)
#14121-#14126 G154 P7 (also #7121-#7126 and G116)
#14141-#14146 G154 P8 (also #7141-#7146 and G117)
#14161-#14166 G154 P9 (also #7161-#7166 and G118)
#14181-#14186 G154 P10 (also #7181-#7186 and G119)
#14201-#14206 G154 P11 (also #7201-#7206 and G120)
#14221-#14226 G154 P12 (also #7221-#7226 and G121)
#14241-#14246 G154 P13 (also #7241-#7246 and G122)
#14261-#14266 G154 P14 (also #7261-#7266 and G123)
#14281-#14286 G154 P15 (also #7281-#7286 and G124)
#14301-#14306 G154 P16 (also #7301-#7306 and G125)
#14321-#14326 G154 P17 (also #7321-#7326 and G126)
#14341-#14346 G154 P18 (also #7341-#7346 and G127)
#14361-#14366 G154 P19 (also #7361-#7366 and G128)
#14381-#14386 G154 P20 (also #7381-#7386 and G129)
#14401-#14406 G154 P21
#14421-#14426 G154 P22
#14441-#14446 G154 P23
#14461-#14466 G154 P24
#14481-#14486 G154 P25
#14501-#14506 G154 P26
#14521-#14526 G154 P27
#14541-#14546 G154 P28

```
#14561-#14566 G154 P29  
#14581-#14586 G154 P30  
#14781-#14786 G154 P40  
#14981-#14986 G154 P50  
#15181-#15186 G154 P60  
#15381-#15386 G154 P70  
#15581-#15586 G154 P80  
#15781-#15786 G154 P90  
#15881-#15886 G154 P95  
#15901-#15906 G154 P96  
#15921-#15926 G154 P97  
#15941-#15946 G154 P98  
#15961-#15966 G154 P99
```

G156 Broaching Canned Cycle (Group 09)

- * **Z** - Z-axis absolute location total pecking depth.
- * **X** - X-axis absolute location of furthest slice cycle.
- * **Y** - Y-axis absolute location of furthest slice cycle.
- * **I** - size of increment between slice cycles.
- * **K** - Z-axis size of increment between pecks in a cycle.
- * **F** - Feed rate per minute.
- * **C** - C-axis position.
- * **D** - Tool clearance when returning to starting plane.

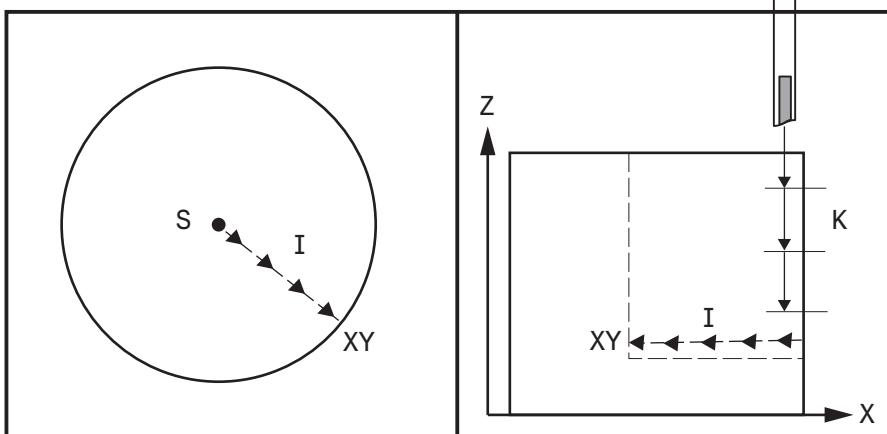
*indicates optional



NOTE:

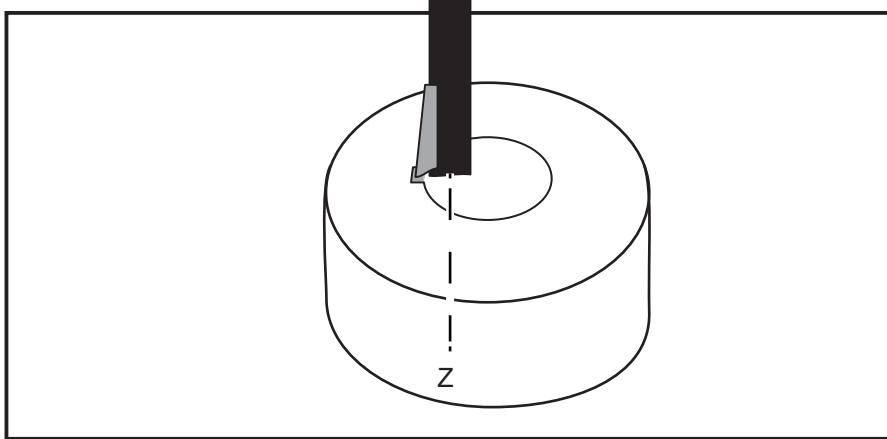
The machine must have software version 100.20.000.1000 or higher to use this G-code.

F7.44: G156 Broaching Canned Cycle: [S] Start position, [I] Slices, [K] Pecks.



Broaching operations canned cycle can be done on Inner and outer grooves in different position using the G156 code with an optional address codes which define the tool Pecks and the cut paths.

F7.45: Broaching operations canned cycle can be done using a single tool path (with adjustable programmable Pecks) when using a special broaching shape tool.



Spindle orient command M19 P/R code which used to position the spindle before broaching must be used first before executing G156 broaching canned cycle.



NOTE:

M19, P, or R should not be on the same line as G156. M19, P, or R value should not be negative. See

M19

for more information.



NOTE: When moving the program and broach tool from one machine to another, the R value in the M19 (spindle orientation) line may change.

When K is added to a G156 block, then pecking will be performed at each interval specified by K.

The D code can be used for grooving and turning to provide material clearance when returning to starting plane S.

The D code command is used to provide a tool clearance shift, in the X-axis before returning in the Z-axis to the "C" clearance point.

When an X, or Y code is added to a G156 block and the current (i.e. actual) X, Y-axis machine position not at the position defined by G156 X or Y, then a minimum of two pecking canned cycles will occur, one at the current (actual) location of the machine X Y-axis, and another one at the X Y location set by G156.

The I code is the incremental distance (slices cycle) between the start position S toward the target X/Y Axes positions (i.e. the thickness of each slice).

Adding an I will performs multiple, evenly spaced, slice cycles between the starting position S and X/Y target position.

When the distance between S and target X/Y is not evenly divisible by I, then the last interval is less than I.

The tool clearance shift direction (i.e. shift in + X or - X, ZX plane,) will be depend on the sign of the D value (Positive/Negative).

When the D has a positive value, the tool moves in the positive direction of the X-axis.

When the D has a negative value, the tool moves further in the negative direction of the X-axis.

When D has a zero value or it not specified with the G156 lathe broaching canned cycle, then there shall be no tool clearance shift in the X-axis before returning in the Z-axis to the "C" clearance point.i.e. the tool returning in the Z axis to the "C" clearance point will use the same peck path in the opposite direction.



NOTE: An alarm will be generated if any spindle commanded to turn (other than spindle orient/ c axis position) within broaching canned cycle, or the broaching canned cycle is starting the Pecks cycle and the spindle is still turning.

C_{xx} (position the C axis) could be used before or on the same line of G156 broaching canned cycle.

**NOTE:**

The broaching canned cycle is a modal, and should be repeated on each line within the canned cycle uses any broaching address code. Refer to example below:

T7.2: Mill Broaching: Example 1

G-Code	Program Sequence
<pre>% G156 X-10. Y-5.0 Z-12.0 I0.5 K0.5 F10.; X-5. Y-2.0 Z-10.0 I2.0 K2.0; I1.0 K1.0; C90.; C45.0 X-5. Y-3.0 Z-10.0 I2.0 K2.0; M19; G00 G80; %</pre>	<ul style="list-style-type: none"> • Mill Broaching Operations Canned Cycle Start. • Perform another broach with new XYZIK address codes values. • Perform another broach with new IK address codes values. • Perform another broach at new angle using the last specified address codes. • Rotate to new angle then broach using the new address codes values. • Orient the spindle. • End of Canned Cycle.

G167 - Modify Setting (Group 00)

P - This code specifies the setting number.

Q - specifies the setting value. This can be a decimal numeric value or an enumerated representation for unit-less settings.

K - the number after the K code specifies a guard code for permanent changes.

This G-code allows the user permanently make changes to settings during program execution.

The control will generate an Alarm when:

- the P or Q code is missing.
- the P code is not a valid setting number.
- the Q code is not a valid for the setting number.

Examples of G167:**Example #1**

```
G167 P250 Q1 K10755;
```

Turns on setting 250 Mirror Image C Axis.

Example #2

```
G167 P84 Q3 K10755;
```

Sets setting 84 Tool Overload Action to Autofeed.

Example #3

```
G167 P142 Q1.25 K10755;
```

Sets setting 142 Offset Chng Tolerance to 1.25.

G174 CCW / G184 CW Non-Vertical Rigid Tap (Group 00)

F - Feedrate

X - X position at bottom of hole

Y - Y position at bottom of hole

Z - Z position at bottom of hole

***S** - Spindle Speed

* indicates optional

A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded. This position is used as the Start position.

This G code is used to perform rigid tapping for non-vertical holes. It may be used with a right-angle head to perform rigid tapping in the X or Y Axis on a three-axis mill, or to perform rigid tapping along an arbitrary angle with a five-axis mill. The ratio between the feedrate and spindle speed must be precisely the thread pitch being cut.

It is not necessary to start the spindle before this canned cycle; the control does this automatically.

G187 Accuracy Control (Group 00)

G187 is an accuracy command that can set and control both the smoothness and max corner rounding value when cutting a part. The format for using G187 is G187 Pn Ennnn.

P - Controls the smoothness level, P1(rough), P2(medium), or P3(finish). Temporarily overrides Setting 191.

E - Sets the max corner rounding value. Temporarily overrides Setting 85.

Setting 191 sets the default smoothness to the user specified ROUGH, MEDIUM, or FINISH when G187 is not active. The Medium setting is the factory default setting.



NOTE:

Changing Setting 85 to a low value may make the machine operate as if it is in exact stop mode.



NOTE:

Changing setting 191 to FINISH will take longer to machine a part. Use this setting only when needed for the best finish.

G187 Pm Ennnn sets both the smoothness and max corner rounding value. G187 Pm sets the smoothness but leaves max corner rounding value at its current value. G187 Ennnn sets the max corner rounding but leaves smoothness at its current value. G187 by itself cancels the E value and sets smoothness to the default smoothness specified by Setting 191. G187 will be canceled whenever [RESET] is pressed, M30 or M02 is executed, the end of program is reached, or [EMERGENCY STOP] is pressed.

G234 Tool Center Point Control (TCPC) (Group 08)

G234 Tool Center Point Control (TCPC) lets a machine correctly run a contouring 4- or 5-axis program when the workpiece is not located in the exact location specified by the CAM-generated program. This eliminates the need to repost a program from the CAM system when the programmed and the actual workpiece locations are different.

VR/GM-2-5AX - G234 - Tool Center Point Control (TCPC)

G234 Tool Center Point Control (TCPC) is a software feature in the Haas CNC control that allows a machine to correctly run a contouring 4- or 5-axis program when the workpiece is not located in the exact location specified by a CAM-generated program.

This eliminates the need to repost a program from the CAM system when the programmed and the actual workpiece locations are different. The Haas CNC control combines the known centers of rotation for the rotary axes (MRZP) and the location of the workpiece (e.g., active work offset G54) into a coordinate system.

TCPC makes sure that this coordinate system remains fixed relative to the table; when the rotary axes rotate, the linear coordinate system rotates with them. Like any other work setup, the workpiece must have a work offset applied to it. This tells the Haas CNC control where the workpiece is located on the machine table.

TCPC is activated with G234. G234 cancels the previous H-code. An H-code must therefore be placed on the same block as G234. G234 is canceled by G49, G42, and G44.

**NOTE:**

The rotary axes must be at 0 before commanding G234.

TCPC G-code is programmed from the tool tip. The control knows the centers of rotation for the rotary axes (MRZP), the location of the workpiece (active work offset), and the tool length offset. The control uses this data to calculate the position of tool tip relative to the active work offset and maintains a static tool tip position through rotary feed moves.

**NOTE:**

Tool tip position is not maintained during rapid rotary moves. Do not program rapid moves while TCPC is active.

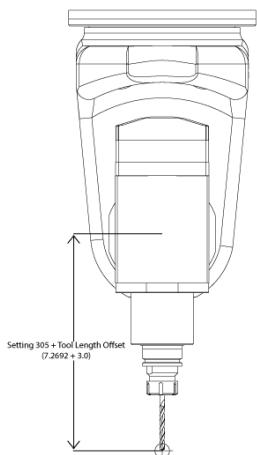
The diagram below illustrates TCPC positioning.

F7.46: GM-2-5AX TCPC

1

MDI:

T1 M06
G00 G90 G54 X0.Y0.
B0. C0.
G43 H01 Z6.

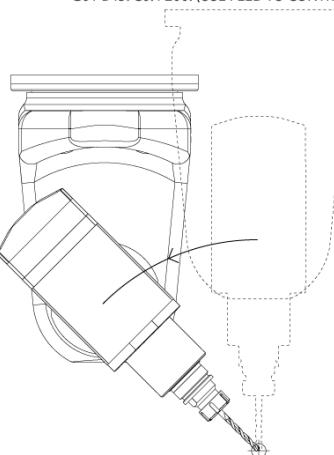
Program Position: \oplus

X = 0.0000
Y = 0.0000
Z = 6.0000
B = 0.000
C = 0.000

2

MDI:

G234 H01 Z6. (TCPC ON WITH LENGTH OFFSET 1)
G00 G54 X0.Y0.
G01 B45. C0. F200. (USE FEED TO CONTROL TOOL TIP)

Program Position: \oplus

X = 0.0000
Y = 0.0000
Z = 6.0000
B = 45.000
C = 0.000

TCPC Program Example



NOTE:

When starting TCPC it is safest to establish a new XYZ using a rapid move. The example below has a rapid G00 XYZ move after the G234 H code.

```
%  
O00003 (TCPC SAMPLE);  
G20;  
G00 G17 G40 G80 G90 G94 G98;  
G53 Z0.;  
T1 M06;  
G00 G90 G54 B0. C0. (POSITION ROTARY AXES);  
G00 G90 X-0.9762 Y1.9704 S10000 M03 (POSITION LINEAR AXES);  
G234 H01(TCPC ON WITH LENGTH OFFSET 1);  
G00 X-0.9762 Y1.9704;  
G00 Z1.0;  
G01 X-0.5688 Y1.1481 Z0.2391 B43.421 C115.162 F40.;
```

```

X-0.4386 Y0.8854 Z-0.033 B43.421 C116.576;
X-0.3085 Y0.6227 Z-0.3051 B45.421 C116.654;
X-0.307 Y0.6189 Z-0.3009 B46.784 C116.382;
X-0.3055 Y0.6152 Z-0.2966 B46.43 C116.411;
X-0.304 Y0.6114 Z-0.2924 B46.076 C116.44;
X-0.6202 Y0.5827 Z-0.5321 B63.846 C136.786;
X-0.6194 Y0.5798 Z-0.5271 B63.504 C136.891;
X-0.8807 Y0.8245 Z-0.3486;
X-1.1421 Y1.0691 Z-0.1701;
X-1.9601 Y1.8348 Z0.3884G49 (TCP/C OFF);
G00 G53 Z0.;
G53 B0. C0.;
G53 Y0.;
M30;
%

```

UMC - G234 - Tool Center Point Control (TCPC) (Group 08)

G234 Tool Center Point Control (TCPC) is a software feature in the Haas CNC control that allows a machine to correctly run a contouring 4- or 5-axis program when the workpiece is not located in the exact location specified by a CAM-generated program. This eliminates the need to repost a program from the CAM system when the programmed and the actual workpiece locations are different.

The Haas CNC control combines the known centers of rotation for the rotary table (MRZP) and the location of the workpiece (e.g., active work offset G54) into a coordinate system. TCPC makes sure that this coordinate system remains fixed relative to the table; when the rotary axes rotate, the linear coordinate system rotates with them. Like any other work setup, the workpiece must have a work offset applied to it. This tells the Haas CNC control where the workpiece is located on the machine table.

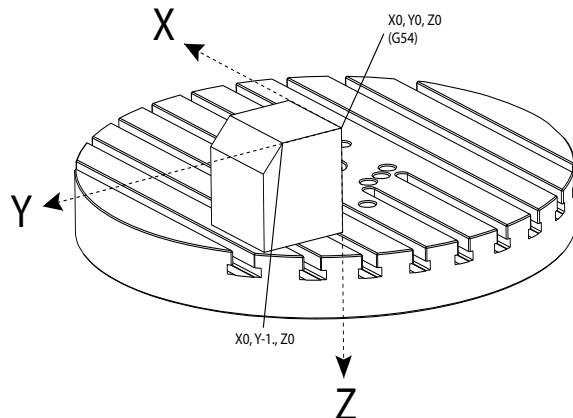
The conceptual example and illustrations in this section represent a line segment from a full 4- or 5-axis program.



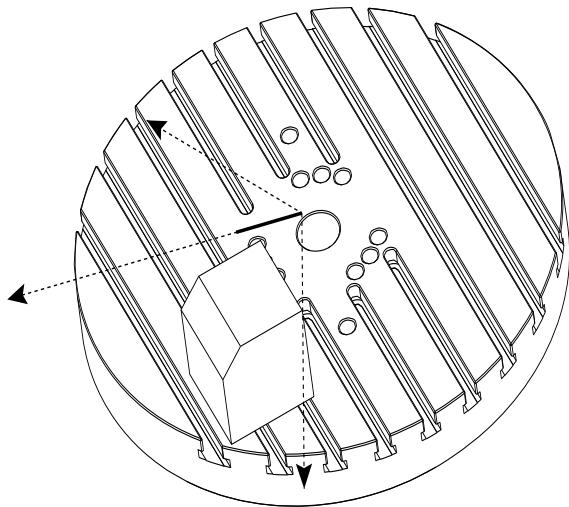
NOTE:

For clarity, the illustrations in this section do not depict workholding. Also, as conceptual, representative drawings, they are not to scale and may not depict the exact axis motion described in the text.

The straight line edge highlighted in Figure F7.47 is defined by point (X0, Y0, Z0) and point (X0, Y-1., Z0). Movement along the Y Axis is all that is required for the machine to create this edge. The location of the workpiece is defined by work offset G54.

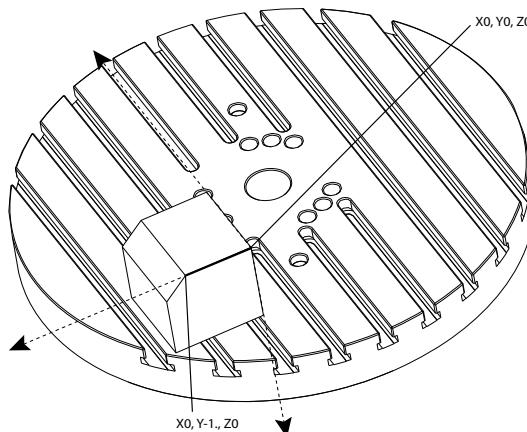
F7.47: Location of Workpiece Defined by G54

In Figure **F7.48**, the B and C Axes have been rotated 15 degrees each. To create the same edge, the machine needs to make an interpolated move with the X, Y, and Z Axes. Without TCPC, you would need to repost the CAM program for the machine to correctly create this edge.

F7.48: G234 (TCPC) Off and the B and C Axes Rotated

TCPC is invoked in Figure **F7.49**. The Haas CNC control knows the centers of rotation for the rotary table (MRZP), and the location of the workpiece (active work offset G54). This data is used to produce the desired machine motion from the original CAM-generated program. The machine follows an interpolated X-Y-Z path to create this edge, even though the program simply commands a single-axis move along the Y Axis.

F7.49: G234 (TCPC) On and the B and C Axes Rotated



G234 Programmer's Notes

These key presses and program codes cancel G234:

- [EMERGENCY STOP]
- [RESET]
- [HANDLE JOG]
- [LIST PROGRAM]
- M02 – Program End
- M30 – Program End and Reset
- G43 – Tool Length Compensation +
- G44 – Tool Length Compensation -
- G49 – G43 / G44 / G143 Cancel

These codes will NOT cancel G234:

- M00 – Program Stop
- M01 – Optional Stop

These key presses and program codes impact G234:

- G234 invokes TCPC and cancels G43.
- When using tool length compensation, either G43 or G234 must be active. G43 and G234 cannot be active at the same time.
- G234 cancels the previous H-code. An H-code must therefore be placed on the same block as G234.
- G234 cannot be used at the same time as G254 (DWO).

These codes ignore 234:

- G28 – Return to Machine Zero Through Optional Reference Point

- G29 – Move to Location Thru G29 Reference Point
- G53 – Non-Modal Machine Coordinate Selection
- M06 – Tool Change

Invoking G234 (TCPC) rotates the work envelope. If the position is close to the travel limits, the rotation can put the current work position outside of travel limits and cause an over travel alarm. To solve this, command the machine to the center of the work offset (or near the center of the table on a UMC), and then invoke G234 (TCPC).

G234 (TCPC) is intended for simultaneous 4- and 5-axis contouring programs. An active work offset (G54, G55, etc.) is required to use G234.

G253 Orient Spindle Normal To Feature Coordinate System (Group 00)

G253 is a 5 axis G-Code used to orient the spindle normal the feature coordinate system. This code can only be used while G268 is active.

```
%  
O00005 (G268 WITH G81 DRILL CANNED CYCLE) (COMMAND ANGLE WITH  
IJK BEFORE MOVING TO OFFSET);  
T1 M06 (TOOL CHANGE);  
G54 G00 G40 G80 G17 G90 (GENERAL SAFE STARTUP LINE);  
X0 Y0 S1500 M03 (INITIAL XYZ LOCATION);  
G43 Z06. H01 (ENACT TOOL LENGTH COMP.);  
G268 X2. Y2. Z0 I0 J30. K45. Q123 (SET TILTED PLANE);  
G253 (MOVE SPINDLE PERPENDICULAR TO TILTED PLANE);  
G00 X0 Y0 Z.5 (MOVE TO START LOCATION);  
G81 G98 R0.1 Z-1. F75.;  
G80;  
G269 (CANCEL TILTED PLANE);  
G00 G53 Z0 M05;  
G53 B0 C0;  
G53 X0 Y0;  
M30;  
%
```

G254 Dynamic Work Offset (DWO) (Group 23)

G254 Dynamic Work Offset (DWO) is similar to TCPC, except that it is designed for use with 3+1 or 3+2 positioning, not for simultaneous 4- or 5-axis machining. If the program does not use the B and C Axes, there is no need to use DWO.

UMC - G254 - Dynamic Work Offset (DWO) (Group 23)

G254 Dynamic Work Offset (DWO) is similar to TCPC, except that it is designed for use with 3+1 or 3+2 positioning, not for simultaneous 4- or 5-axis machining. If the program does not make use of the tilt and rotary Axes, there is no need to use DWO.



CAUTION: *The B-Axis value of the work offset you use with G254 MUST be zero.*

With DWO, you no longer need to set the workpiece in the exact position as programmed in the CAM system. DWO applies the appropriate offsets to account for the differences between the programmed workpiece location and the actual workpiece location. This eliminates the need to repost a program from the CAM system when the programmed and actual workpiece locations are different.

The control knows the centers of rotation for the rotary table (MRZP) and the location of the workpiece (active work offset). This data is used to produce the desired machine motion from the original CAM-generated program. Therefore, it is recommended that G254 be invoked after the desired work offset is commanded, and after any rotational command to position the 4th and 5th axes.

After G254 is invoked, you must specify an X, Y, and Z Axis position before a cutting command, even if it recalls the current position. The program should specify the X- and Y-Axis position in one block and the Z Axis in a separate block.



CAUTION: *Before rotary motion, use a G53 Non-Modal Machine Coordinate motion command to safely retract the tool from the workpiece and allow clearance for the rotary motion. After the rotary motion finishes, specify an X-, Y-, and Z-Axis position before a cutting command, even if it recalls the current position. The program should specify the X- and Y-Axis position in one block and the Z-Axis position in a separate block.*



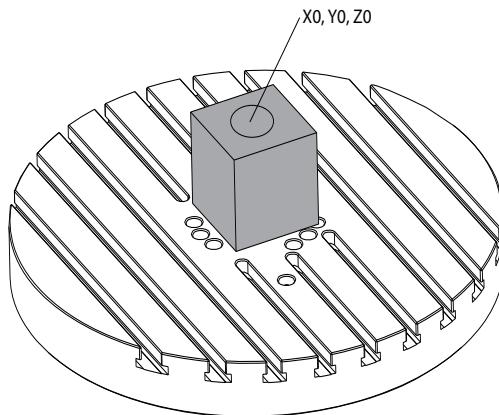
CAUTION: *Be sure to cancel G254 with G255 when your program does simultaneous 4- or 5-axis machining.*



NOTE: *For clarity, the illustrations in this section do not depict workholding.*

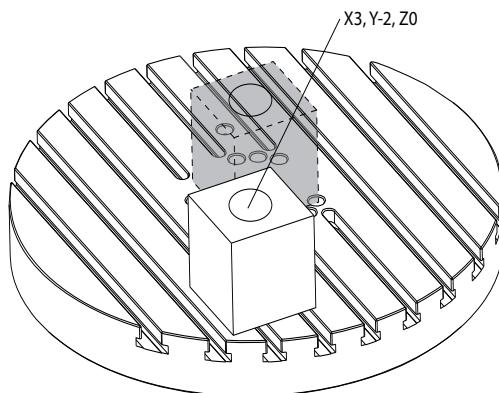
The block in the figure below was programmed in the CAM system with the top center hole located at the center of the pallet and defined as X0, Y0, Z0.

F7.50: Original Programmed Position

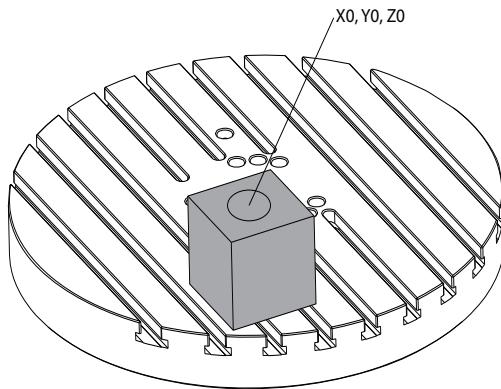


In the figure below, the actual workpiece is not located in this programmed position. The center of the workpiece is actually located at X3, Y-2, Z0, and is defined as G54.

F7.51: Center at G54, DWO Off



DWO is invoked in the figure below. The control knows the centers of rotation for the rotary table (MRZP), and the location of the workpiece (active work offset G54). The control uses this data to apply the appropriate offset adjustments to make sure that the proper toolpath is applied to the workpiece, as intended by the CAM-generated program. This eliminates the need to repost a program from the CAM system when the programmed and actual workpiece locations are different.

F7.52: Center with DWO On**G254 Program Example**

```
%  
O00004 (DWO SAMPLE) ;  
G20 ;  
G00 G17 G40 G80 G90 G94 G98 ;  
G53 Z0. ;  
T1 M06 ;  
G00 G90 G54 X0. Y0. B0. C0. (G54 is the active work offset  
for) ;  
(the actual workpiece location) ;  
S1000 M03 ;  
G43 H01 Z1. (Start position 1.0 above face of part Z0.) ;  
G01 Z-1.0 F20. (Feed into part 1.0) ;  
G00 G53 Z0. (Retract Z with G53) ;  
B90. C0. (ROTARY POSITIONING) ;  
G254 (INVOKE DWO) ;  
X1. Y0. (X and Y position command) ;  
Z2. (Start position 1.0 above face of part Z1.0) ;  
G01 Z0. F20. (Feed into part 1.0) ;  
G00 G53 Z0. (Retract Z with G53) ;  
B90. C-90. (ROTARY POSITIONING) ;  
X1. Y0. (X and Y position command) ;  
Z2. (Start position 1.0 above face of part Z1.0) ;  
G01 Z0. F20. (Feed into part 1.0) ;  
G255 (CANCEL DWO) ;  
B0. C0. ;  
M30 ;  
%
```

G254 Programmer's Notes

These key presses and program codes will cancel G254:

- [EMERGENCY STOP]
- [RESET]
- [HANDLE JOG]
- [LIST PROGRAM]
- G255 – Cancel DWO
- M02 – Program End
- M30 – Program End and Reset

These codes will NOT cancel G254:

- M00 – Program Stop
- M01 – Optional Stop

Some codes ignore G254. These codes will not apply rotational deltas:

- *G28 – Return to Machine Zero Through Optional Reference Point
- *G29 – Move to Location Through G29 Reference Point
- G53 – Non-Modal Machine Coordinate Selection
- M06 – Tool Change

*It is strongly recommended that you not use G28 or G29 while G254 is active, nor when the B and C Axes are not at zero.

1. G254 (DWO) is intended for 3+1 and 3+2 machining, where the B and C Axes are used to position only.
2. An active work offset (G54, G55, etc.) must be applied before G254 is commanded.
3. All rotary motion must be complete before G254 is commanded.
4. After G254 is invoked, you must specify an X-, Y-, and Z-Axis position prior to any cutting command, even if it recalls the current position. It is recommended to specify the X and Y Axes in one block, and the Z Axis in a separate block.
5. Cancel G254 with G255 immediately after use and before ANY rotary motion.
6. Cancel G254 with G255 any time simultaneous 4- or 5-axis machining is performed.
7. Cancel G254 with G255 and retract the cutting tool to a safe location before the workpiece is repositioned.

G255 Cancel Dynamic Work Offset (DWO) (Group 23)

G255 cancels G254 Dynamic Work Offset (DWO).

G266 Visible Axes Linear Rapid %Motion (Group 00)

E - Rapid rate.

P - Axis parameter number. Example P1 = X, P2 = Y, P3 = Z.

I - Machine coordinate position command.

The below example commands the X-axis to move to X-1. at 10% rapid rate.

```
%  
G266 E10. P1 I-1  
%
```

G268 / G269 Feature Coordinate System (Group 02)

X - Feature coordinate system origin X coordinate in the WCS.

Y - Feature coordinate system origin Y coordinate in the WCS.

Z - Feature coordinate system origin Z coordinate in the WCS.

***I** - Rotation of feature coordinate system about working coordinate system X axis.

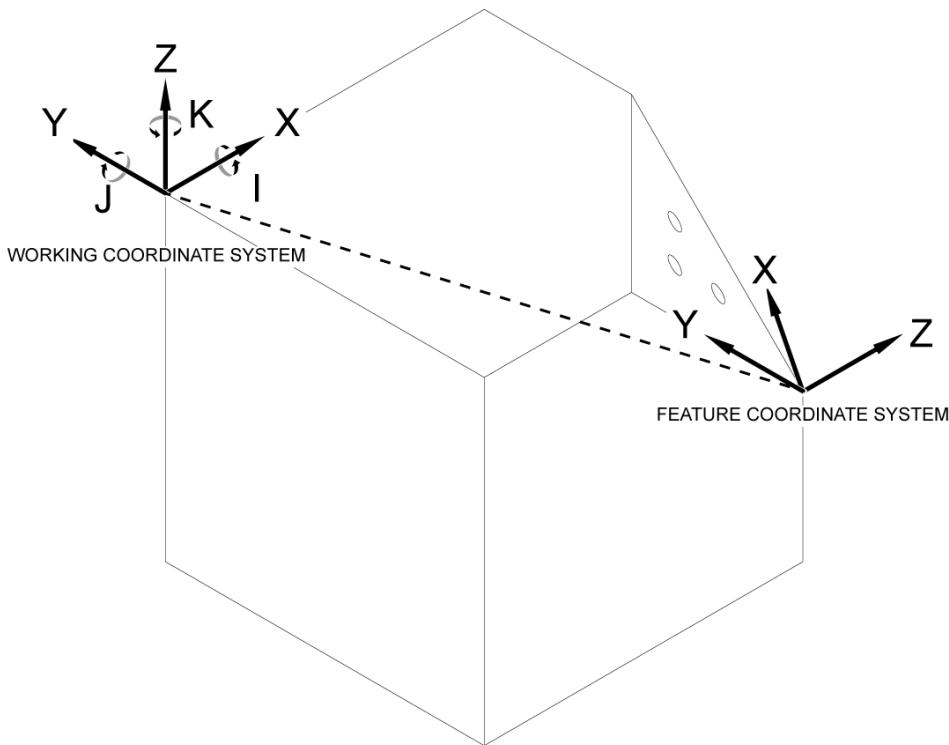
***J** - Rotation of feature coordinate system about working coordinate system Y axis.

***K** - Rotation of feature coordinate system about working coordinate system Z axis.

***Q** - Qnnn is used to define the order in which the I,J,K rotations will be applied. The default value used if Q is omitted, Q321 rotates about Z, then Y, then X. Q123 rotates about X, then Y, then Z.

* indicates optional

F7.53: G268 Feature Coordinate System



G268 is a 5 axis G-Code used to define a tilted feature coordinate system relative to the working coordinate system. Canned cycles and G-codes work normally within the feature coordinate system. Before activating G268, G43 Tool Length Compensation, must be activated. However, the transformation from the working coordinate system to the feature coordinate system is performed independently from the tool length offset. Calling G268 only establishes the feature coordinate system. It does not cause motion in any axis. After calling G268 the current position of the spindle must be recalled. G269 is used to cancel G268 and revert back the WCS.

There are two ways to define a feature coordinate system using G268. The first is to command the B and C axis to the desired angle and specify only the feature coordinate system origin using G268. The feature coordinate system plane will be the plane normal to the spindle axis at the moment G268 is called.

```
%  
O00001 (G268 WITH G81 DRILL CANNED CYCLE) (ANGLE FROM SPINDLE  
POSITION);  
T1 M06 (TOOL CHANGE);  
G54 G00 G40 G80 G17 G90 (GENERAL SAFE STARTUP LINE);  
X0 Y0 S1500 M03 (INITIAL XYZ LOCATION);
```

```
G00 B30. C45. (SET SPINDLE ANGLE);  
G43 Z6. H01 (ENACT TOOL LENGTH COMP.);  
G268 X2. Y2. Z0 (SET TILTED PLANE);  
G00 X0 Y0 Z.5 (RECALL POSITION);  
G81 G98 R0.1 Z-1. F75.;  
G80;  
G269 (CANCEL TILTED PLANE);  
G00 G53 Z0 M05;  
G53 B0 C0;  
G53 X0 Y0;  
M30;  
%
```

The second way to define a feature coordinate system using G268 is to use the optional I, J, K, and Q address codes to specify rotation angles relative to the WCS and rotation order. By using this method, a feature coordinate system that is not normal to the spindle axis may be defined.

```
%  
000002 (G268 WITH G81 DRILL CANNED CYCLE) (COMMAND ANGLE WITH  
IJK & Q);  
T1 M06 (TOOL CHANGE);  
G54 G00 G40 G80 G17 G90 (GENERAL SAFE STARTUP LINE);  
X0 Y0 S1500 M03 (INITIAL XYZ LOCATION);  
G00 B30. C45. (SET SPINDLE ANGLE);  
G43 Z06. H01 (ENACT TOOL LENGTH COMP.);  
G268 X2. Y2. Z0 I0 J30. K45. Q123 (SET TILTED PLANE);  
G00 X0 Y0 Z.5 (RECALL POSITION);  
G81 G98 R0.1 Z-1. F75.;  
G80;  
G269 (CANCEL TILTED PLANE);  
G00 G53 Z0 M05;  
G53 B0 C0;  
G53 X0 Y0;  
M30;  
%
```

More Information Online

For updated information, including tips, tricks, maintenance procedures, and more, visit the Haas Service page at www.HaasCNC.com.

For the most current Operator's and Service Manuals scan the code below with your mobile device:



Chapter 8: M-codes

8.1 Introduction

This chapter gives detailed descriptions of the M-codes that you use to program your machine.

For the most current M-code information scan the code below with your mobile device.

F8.1: Mill - M-Codes Guide



8.1.1 List of M-codes

This chapter gives detailed descriptions of the M-codes that you use to program your machine.



CAUTION:

The sample programs in this manual have been tested for accuracy, but they are for illustrative purposes only. The programs do not define tools, offsets, or materials. They do not describe workholding or other fixturing. If you choose to run a sample program on your machine, do so in Graphics mode. Always follow safe machining practices when you run an unfamiliar program.



NOTE:

The sample programs in this manual represent a very conservative programming style. The samples are intended to demonstrate safe and reliable programs, and they are not necessarily the fastest or most efficient way to operate a machine. The sample programs use G-codes that you might choose not to use in more efficient programs.

M-codes are miscellaneous machine commands that do not command axis motion. The format for an M-code is the letter M followed by two to three digits; for example M03.

Only one M-code is allowed per line of code. All M-codes take effect at the end of the block.

Setting	Description	Page
M00	Stop Program	412
M01	Optional Program Stop	412
M02	Program End	413
M03	Spindle Commands	413
M04	Spindle Commands	413
M05	Spindle Commands	413
M06	Tool Change	413
M07	Shower Coolant On	414
M08 / M09	Coolant On / Off	414
M10 / M11	Engage / Release 4th Axis Brake	415
M12 / M13	Engage / Release 5th Axis Brake	415
M16	Tool Change	415
M19	Orient Spindle	415
M21–M25	Optional User M Function with M-Fin	416
M29	Set Output Relay with M-Fin	418
M30	Program End and Reset	418
M31	Chip Conveyor Forward	418
M33	Chip Conveyor Stop	418
M34	Coolant Increment	418
M35	Coolant Decrement	418
M36	Pallet Part Ready	419
M39	Rotate Tool Turret	420

Setting	Description	Page
M41 / M42	Low / High Gear Override	420
M46	Qn Pmm Jump to Line	420
M48	Validate That The Current Program is Appropriate for Loaded Pallet	420
M50	Pallet Change Sequence	420
M51–M55	Set Optional User M-codes	421
M59	Set Output Relay	421
M61–M65	Clear Optional User M-codes	421
M69	Clear Output Relay	422
M70/M71	Workholding Clamp / Unclamp	422
M73 / M74	Tool Air Blast (TAB) On / Off	423
M75	Set G35 or G136 Reference Point	423
M78	Alarm if Skip Signal Found	423
M79	Alarm if Skip Signal Not Found	423
M80 / M81	Auto Door Open / Close	424
M82	Tool Unclamp	424
M83 / M84	Auto Air Gun On / Off	424
M86	Tool Clamp	424
M88 / M89	Through-Spindle Coolant On / Off	424
M90 / M91	Fixture Clamp Input On / Off	425
M95	Sleep Mode	425
M96	Jump If No Input	426
M97	Local Sub-Program Call	427
M98	Sub-Program Call	427

Setting	Description	Page
M99	Sub-Program Return or Loop	428
M104 / M105	Probe Arm Extend/ Retract	429
M109	Interactive User Input	429
M116 / M117	Vise Air Chips Blast ON/OFF	431
M130 / M131	Display Media / Cancel Display Media	431
M138 / M139	Spindle Speed Variation On/Off	433
M158 / M159	Mist Condenser On/Off	434
M160	Cancel Active PulseJet	434
M161 Pnn	PulseJet Continuous Mode	434
M162 Pnn	PulseJet Single Event Mode	434
M163 Pnn	PulseJet Modal Mode	435
M180 / M181	Automatic Window Open/Close	436
M183/M184	Auto Air Hole Blast On/OFF	436
M199	Pallet / Part Load or Program End	436
M300	M300 - APL/Robot Custom Sequence	436

M00 Stop Program

The M00 code stops a program. It stops the axes, spindle, and turns off the coolant (including optional Through Spindle Coolant, Through Tool Air Blast, and Auto Air Gun / Minimum Quantity Lubrication). The next block after the M00 is highlighted when viewed in the program editor. Press **[CYCLE START]** to continue program operation from the highlighted block.

M01 Optional Program Stop

M01 works the same as M00, except the optional stop feature must be on. Press **[OPTION STOP]** to toggle the feature on and off.

M02 Program End

M02 ends a program.



NOTE:

The most common way of ending a program is with an M30.

M03 Spindle Fwd / M04 Spindle Rev / M05 Spindle Stop

M03 turns the spindle on in the forward direction.

M04 turns the spindle on in the reverse direction.

M05 stops the spindle, and waits for it to stop.

Spindle speed is controlled with an S address code; for example, S5000 commands a spindle speed of 5000 RPM.

If your machine has a gearbox, the spindle speed you program determines the gear that the machine uses, unless you use M41 or M42 to override gear selection. Refer to page 420 for more information on the gear select override M-codes.

M06 Tool Change

T - Tool number

The M06 code is used to change tools. For example, M06 T12 puts tool 12 into the spindle. If the spindle is running, the spindle and coolant (including TSC) is stopped by the M06 command.



NOTE:

The M06 command automatically stops the spindle, stops coolant, moves the Z Axis to the tool change position, and orients the spindle for the tool change. You do not need to include these commands for a tool change in your program.



NOTE:

M00, M01, any work offset G-code (G54, etc.), and block delete slashes before a tool change stop look-ahead, and the control does not pre-call the next tool to the change position (for a side-mount tool changer only). This can cause significant delays to program execution, because the control must wait for the tool to arrive at the change position before it can execute the tool change. You can command the carousel to the tool position with a T code after a tool change; for example:

```
M06 T1 (FIRST TOOL CHANGE) ;  
T2 (PRE-CALL THE NEXT TOOL) ;
```

Refer to page **150** for more information on side-mount tool changer programming.

M07 Shower Coolant On

M07 starts the optional shower coolant. M09 stops the shower coolant and also stops the standard coolant. The optional shower coolant stops automatically before a tool change or a pallet change. It automatically starts again after a tool change if it was **ON** before a tool change command.



NOTE:

Some machines use optional relays and optional M-codes to command shower coolant, such as M51 on and M61 off. Check your machine configuration for the correct M-code programming.

M08 Coolant On / M09 Coolant Off

P - M08 Pn

M08 starts the optional coolant supply and M09 stops it.

An optional P-Code can now be specified along with an M08.



NOTE:

The machine is equipped with a Variable Frequency Drive for the coolant pump

As long as no other G-Codes are in the same block, and t, this P-Code can be used to specify the desired pressure level of the coolant pump: P0 = Low Pressure P1 = Normal Pressure P2 = High Pressure



NOTE:

If no P-Code is specified, or the specified P-Code is out of range, then normal pressure will be used.



NOTE:

If the machine is not equipped with a Variable Frequency Drive for the coolant pump, then the P-Code will have no effect.

**NOTE:**

The control checks the coolant level only at the start of a program, so a low coolant condition will not stop a running program.

**CAUTION:**

Do not use straight or “neat” mineral cutting oils. They cause damage to rubber components in the machine.

**NOTE:**

Use M88/M89 to start and stop the optional Through-Spindle Coolant.

**NOTE:**

Use M34/M35 to start and stop the optional Programmable Coolant (P-Cool).

M10 Engage 4th Axis Brake / M11 Release 4th Axis Brake

M10 applies the brake to the optional 4th axis and M11 releases the brake. The optional 4th axis brake is normally engaged, so the M10 command is only required when an M11 has released the brake.

M12 Engage 5th Axis Brake / M13 Release 5th Axis Brake

M12 applies the brake to the optional 5th axis and M13 releases the brake. The optional 5th axis brake is normally engaged, so the M12 command is only required when an M13 has released the brake.

M16 Tool Change

T - Tool number

This M16 behaves the same as M06. However M06 is the preferred method for commanding tool changes.

M19 Orient Spindle (Optional P and R Values)

P - Number of degrees (0 - 360)

R - Number of degrees with two decimal places (0.00 - 360.00).

M19 adjusts the spindle to a fixed position. The spindle only orients to the zero position without the optional M19 orient spindle feature. The orient spindle function allows P and R address codes. For example:

M19 P270. (orients the spindle to 270 degrees) ;

The R-value allows the programmer to specify up to two decimal places; for example:

M19 R123.45 (orients the spindle to 123.45 degrees) ;



NOTE:

The range for M19 is 0 to 360 degrees. If a negative value is given, it will be ignored and spindle will orient to 0 degrees.

M21-M25 Optional User M Function with M-Fin

M21 through M25 are for user-defined relays. Each M-code closes one of the optional relays and waits for an external M-Fin signal. [RESET] stops any operation waiting for a relay-activated accessory to finish. Also, refer to M51 - M55 and M61 - M65.

Only one relay is switched at a time. A typical operation is to command a rotary product. The sequence is:

1. Run the machining portion of a CNC part program.
2. Stop CNC motion and command a relay.
3. Wait for a finish (M-Fin) signal from the equipment.
4. Continue the CNC part program.

The M-Fin connector is at P8 on the I/O PCB. Refer to the description below for a diagram and pinouts.

M-code Relays

The M-code relays are in the lower-left corner of the I/O PCB.

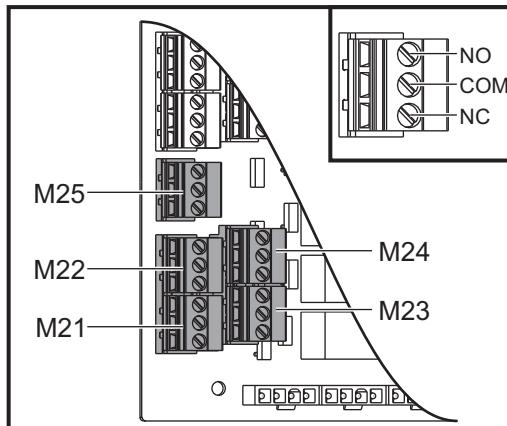


NOTE:

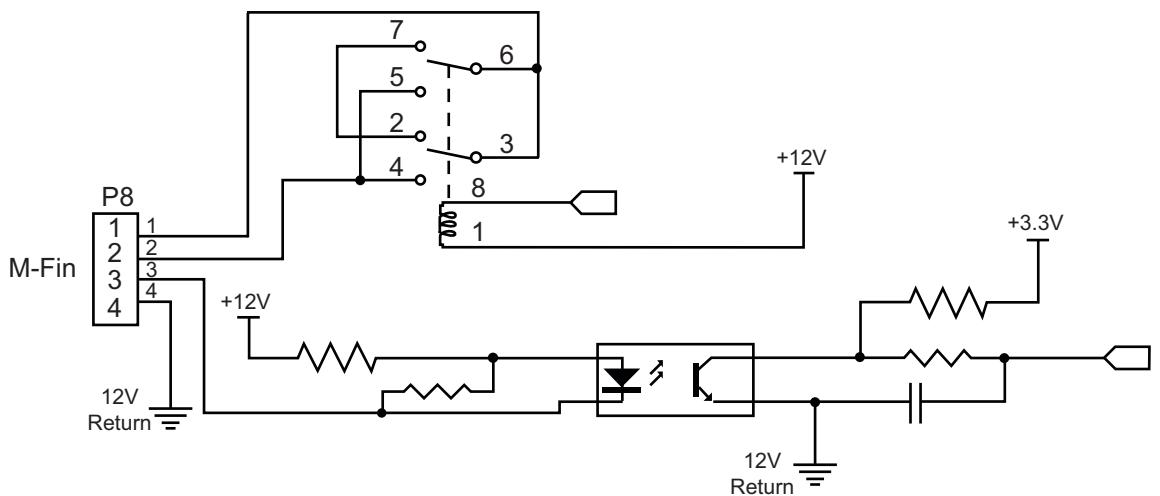
These M-Code relays are sold separately (P/N 93-2889)

These relays can activate probes, auxiliary pumps, clamping devices, etc. Connect these auxiliary devices to the terminal strip for the individual relay. The terminal strip has positions for, Normally Open (NO), Normally Closed (NC), and Common (COM).

F8.2: Main I/O PCB M-code Relays.



F8.3: M-Fin Circuit at P8 on the Main I/O PCB. Pin 3 is the M-Fin input and interacts with input number 18 in the control. Pin 1 is the M-Fin output and interacts with output number 4 on the control.



Optional 8M-code Relays

You can purchase additional M-code relays in banks of 8.

Only the outputs on the I/O PCB are addressable with M21-M25, M51-M55, and M61-M65. If you use an 8M relay bank, you must use M29, M59, and M69 with P codes to activate the relays on the bank. The P codes for the first 8M bank are P90-P97.

M29 Set Output Relay with M-Fin

P - Discrete output relay from 0 to 255.

M29 turns on a relay, pauses the program, and waits for an external M-Fin signal. When the control receives the M-Fin signal, the relay turns off and the program continues. **[RESET]** stops any operation waiting for a relay-activated accessory to finish.

M30 Program End and Reset

M30 stops a program. It also stops the spindle, turns off the coolant (including TSC), and returns the program cursor to the start of the program.



NOTE:

As of software version 100.16.000.1041, M30 no longer cancels tool length offsets.

M31 Chip Conveyor Forward / M33 Chip Conveyor Stop

M31 starts the optional chip removal system (auger, multi-auger, or belt-style conveyor) in the forward direction; the direction that moves the chips out of the machine. You should run the chip conveyor intermittently, as this allows piles of larger chips to collect smaller chips and carry them out of the machine. You can set the chip conveyor duty cycle and run time with Settings 114 and 115.

The optional conveyor coolant washdown runs while the chip conveyor is on.

M33 stops conveyor motion.

M34 Coolant Increment / M35 Coolant Decrement

P - **M34 Pnn** moves the P-Cool spigot to specific position away from home.**M35 Pnn** moves the P-Cool spigot to specific position towards home.

Example: The P-Cool spigot is at position P5 and you need to go to P10, you can use:

M34 P10

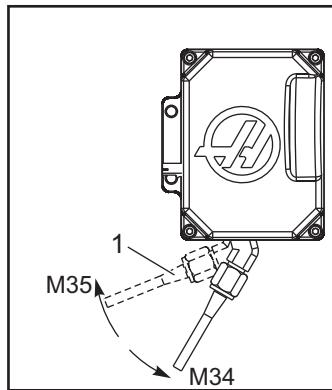
or

M35 P10



NOTE:

The P address value shall be keyed in without a decimal point.

F8.4: P-Cool Spigot

M34 moves the optional P-Cool spigot one position away from the current position (farther from home).

M35 moves the coolant spigot one position towards the home position.



CAUTION: *Do not rotate the coolant spigot by hand. Serious motor damage will occur.*

M36 Pallet Part Ready

Used on machines with pallet changers. M36 delays the pallet change until **[PART READY]** is pushed. A pallet change occurs after **[PART READY]** is pushed and the doors are closed. For example:

```
%  
Onnnnn (program number) ;  
M36 (Flash "Part Ready" light, wait until the button is  
pressed) ;  
M01 ;  
M50 (Perform pallet change after [PART READY] is pushed) ;  
(Part Program) ;  
M30 ;  
%
```

M39 Rotate Tool Turret

M39 is used to rotate the side mount tool changer without a tool change. Program the tool pocket number (T_n) before M39.

M06 is the command to change tools. M39 is normally useful for diagnostic purposes, or to recover from a tool changer crash.

M41 Low Gear Override / M42 High Gear Override

On machines with a transmission, M41 holds the machine in low gear and M42 holds the machine in high gear. Normally, the spindle speed (S_{nnnn}) determines which gear the transmission should be in.

Command M41 or M42 with the spindle speed before the spindle start command, M03. For example:

```
%  
S1200 M41 ;  
M03 ;  
%
```

The gear state reverts to default at the next spindle speed (S_{nnnn}) command. The spindle does not have to stop.

M46 Qn Pmm Jump to Line

Jump to line mm in the current program if pallet n is loaded, otherwise go to the next block.

M48 Validate That The Current Program is Appropriate for Loaded Pallet

Checks in the Pallet Schedule Table that the current program is assigned to the loaded pallet. If the current program is not in the list or the loaded pallet is incorrect for the program, an alarm is generated. M48 can be in a program listed in the PST, but never in a subroutine of the PST program. An alarm will occur if M48 is incorrectly nested.

M50 Pallet Change Sequence

*P - Pallet number

*indicates optional

This M-code is used to call a pallet change sequence. An M50 with a P command will call a specific pallet. M50 P3 will change to pallet 3, commonly used with Pallet Pool machines. Refer to the Pallet Changer section of the manual.

M51-M56 Turn On Built-In M-code Relay

The M51 through M56 codes are built-in M-Code relays. They activate one of the relays and leave it active. Use M61-M66 to turn these off. [RESET] turns off all of these relays.

Refer to M21 through M26 on page 416 for details on the M-code relays with M-Fin.

M59 Turn On Output Relay

P - Discrete output relay number.

M59 turns on a discrete output relay. An example of its usage is M59 Pnnn, where nnn is the relay number being turned on.

When using Macros, M59 P90 does the same thing as using the optional macro command #12090=1, except that it is processed at the end of the line of code.

Built-In M-Code Relays	8M PCB Relay Bank 1 (JP1)	8M PCB Relay Bank 2 (JP2)	8M PCB Relay Bank 3 (JP3)
P114 (M21)	P90	P103	P79
P115 (M22)	P91	P104	P80
P116 (M23)	P92	P105	P81
P113 (M24)	P93	P106	P82
P112 (M25)	P94	P107	P83
P4 (M26)	P95	P108	P84
-	P96	P109	P85
-	P97	P110	P86

M61-M66 Turn Off Built-In M-code Relay

M61 through M65 are optional and turn off one of the relays. The M number corresponds to M51 through M55 that turned on the relay. [RESET] turns off all of these relays. Refer to M21-M25 on page 416 for details on the M-code relays.

M69 Turn Off Output Relay

P - Discrete output relay number from 0 to 255.

M69 turns off a relay. An example of its usage is M69 P12nnn, where nnn is the number of the relay being turned off.

When using Macros, M69 P12003 does the same thing as using the optional macro command #12003=0, except that it is processed in the same order as axis motion.

Built-In M-Code Relays	8M PCB Relay Bank 1 (JP1)	8M PCB Relay Bank 2 (JP2)	8M PCB Relay Bank 3 (JP3)
P114 (M21)	P90	P103	P79
P115 (M22)	P91	P104	P80
P116 (M23)	P92	P105	P81
P113 (M24)	P93	P106	P82
P112 (M25)	P94	P107	P83
P4 (M26)	P95	P108	P84
-	P96	P109	P85
-	P97	P110	P86

M70 Workholding Clamp / M71 Workholding Unclamp

Use M70/M71 to clamp or unclamp the vises setup on the workholding page.

P - The P code defines the vise number. The range is P1 - 8.

Q - The Q code enables or disables the vise unclamp check when commanding the spindle.

Q = 0 or not specified: Enable Unclamp Check. Q = 1: Disable Unclamp Check.

Example: M70 P1 clamps Vise 1 and M71 P1 unclamps it.

Example: M71 P1 Q1 disables the unclamp check for Vise 1.



NOTE:

The Vise Safety Check will be restored after Power Cycle, Program End(M02,M30), End of Program, Alarms, Reset is pressed, or workholding commands.

**NOTE:**

The system will default to Vise 1, if no P-Code is specified with M70 or M71.

M73 Tool Air Blast (TAB) On / M74 Tool Air Blast Off

These M-codes control the Tool Air Blast (TAB) option. M73 turns on TAB, and M74 turns it off.

M75 Set G35 or G136 Reference Point

This code is used to set the reference point for G35 and G136 commands. It must be used after a probing function.

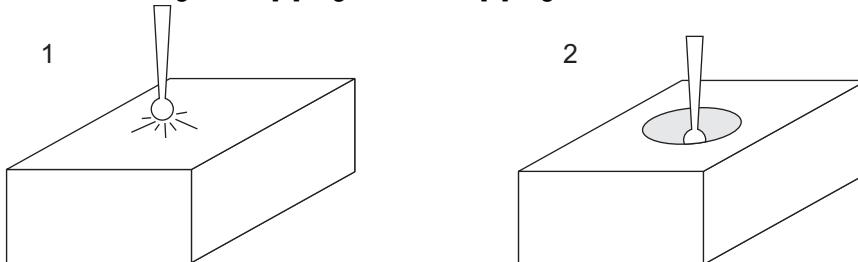
M78 Alarm if Skip Signal Found

M78 is used with a probe. An M78 generates an alarm if a programmed skip function (G31, G36 or G37) receives a signal from the probe. This is used when a skip signal is not expected, and may indicate a probe crash. This code can be placed on the same line as the skip G-code or in any block after.

M79 Alarm if Skip Signal Not Found

M79 is used with a probe. An M79 generates an alarm if a programmed skip function (G31, G36, or G37) did not receive a signal from the probe. This is used when the lack of the skip signal means a probe positioning error. This code can be placed on the same line as the skip G-code or in any block after.

F8.5: Probe Positioning Error: [1] Signal Found. [2] Signal not Found.



M80 Auto Door Open / M81 Auto Door Close

M80 opens the Auto Door and M81 closes it. The control pendant beeps while the door is in motion.

**NOTE:**

The M-codes work only while the machine receives a cell-safe signal from a robot or if it has a light curtain installed. For more information refer to Robot Integration Aid - NGC reference document.

M82 Tool Unclamp

M82 is used to release the tool from the spindle. It is used as a maintenance/test function only. Tool changes should be done using an M06.

M83 Auto Air Gun On / M84 Auto Air Gun Off

M83 turns the Auto Air Gun (AAG) option on, and M84 turns it off. M83 with a Pnnn argument (where nnn is in milliseconds) turns the AAG on for the specified time, then turns it off. You can also press [SHIFT] and then [COOLANT] to turn on the AAG manually.

M86 Tool Clamp

M86 clamps a tool into the spindle. It is used as a maintenance/test function only. Tool changes should be done using an M06.

M88 Through-Spindle Coolant On / M89 Through-Spindle Coolant Off

M88 turns on through-spindle coolant (TSC), and M89 turns off TSC.

The control automatically stops the spindle before it executes M88 or M89. The control does not automatically start the spindle again after M89. If your program continues with the same tool after an M89 command, be sure to add a spindle speed command before further motion.

**CAUTION:**

You must use proper tooling, with a through-hole, when you use the TSC system. Failure to use proper tooling can flood the spindle head with coolant and void the warranty.

Sample Program

**NOTE:**

The M88 command should be before the spindle speed command. If you command M88 after the spindle speed command, the spindle starts, then stops, turns on TSC, and then starts again.

```
%  
T1 M6 (TSC Coolant Through Drill) ;  
G90 G54 G00 X0 Y0 ;  
G43 H01 Z.5 ;  
M88 (Turn TSC on) ;  
S4400 M3 ;  
G81 Z-2.25 F44. R.1 ;  
M89 G80 (Turn TSC off) ;  
G91 G28 Z0 ;  
G90 ;  
M30 ;  
%
```

M90 Fixture Clamp Input ON / M91 Fixture Clamp Input OFF

The M90 M-code enables fixture clamp input monitoring when setting 276 has a valid input number greater than 0. If variable #709 = 1 and the spindle is commanded on, the machine will generate alarm: 973 Fixture Clamp Incomplete.

The M91 M-code disables the fixture clamp input monitoring.

M95 Sleep Mode

Sleep mode is a long dwell. The format of the M95 command is: M95 (hh:mm).

The comment immediately following M95 must contain the duration, in hours and minutes, that you want the machine to sleep. For example, if the current time were 6 p.m. and you want the machine to sleep until 6:30 a.m. the next day, command M95 (12:30). The line(s) after M95 should be axis moves and spindle warm-up commands.

M96 Jump If No Input

P - Program block to go to when conditional test is met

Q - Discrete input variable to test (0 to 255)

M96 is used to test a discrete input for 0 (off) status. This is useful for checking the status of automatic work holding or other accessories that generate a signal for the control. The Q value must be in the range 0 to 255, which corresponds to the inputs found on the diagnostic display I/O tab. When this program block is executed and the input signal specified by Q has a value of 0, the program block Pnnnn is performed (the Nnnnn that matches the Pnnnn line must be in the same program). The M96 sample program uses input #18 M-FIN INPUT

Example:

```
%  
000096 (SAMPLE PROGRAM FOR M96 JUMP IF NO INPUT) ;  
  (IF M-FIN INPUT #18 IS EQUAL TO 1 THE PROGRAM WILL JUMP TO  
  N100) ;  
  (AFTER JUMPING TO N100 THE CONTROL ALARMS OUT WITH A MESSAGE)  
  ;  
  (M-FIN INPUT=1) ;  
  (IF M-FIN INPUT #18 IS EQUAL TO 0 THE PROGRAM JUMPS TO N10) ;  
  (AFTER JUMPING TO N10 THE CONTROL DWELLS FOR 1 SECOND THEN  
  JUMPS TO N5) ;  
  (THE PROGRAM CONTINUES THIS LOOP UNTIL INPUT #18 IS EQUAL TO  
  1) ;  
  
G103 P1 ;  
... ;  
... ;  
N5 M96 P10 Q18 (JUMP TO N10 IF M-FIN INPUT #18 = 0) ;  
... ;  
M99 P100 (JUMP TO N100) ;  
N10 ;  
G04 P1. (DWELL FOR 1 SECOND) ;  
M99 P5 (JUMP TO N5) ;  
... ;  
N100 ;  
#3000= 10 (M-FIN INPUT=1) ;  
M30 ;  
... ;  
%
```

M97 Local Subprogram Call

- P** - Program line number to go to when conditional test is met
L - Repeats subprogram call (1-99) times.

M97 is used to call a subprogram referenced by a line number (**N**) within the same program. A code is required and must match a line number within the same program. This is useful for simple subprograms within a program; does not require a separate program. The subprogram must end with an M99. **Lnn** code in the M97 block repeats the subprogram call **nn** times.



NOTE:

The subprogram is within the body of the main program, placed after the M30.

M97 Example:

```
%  
000001 ;  
M97 P100 L4 (CALLS N100 SUBPROGRAM) ;  
M30 ;  
N100 (SUBPROGRAM) ; ;  
M00 ;  
M99 (RETURNS TO MAIN PROGRAM) ;  
%
```

M98 Subprogram Call

- P** - The subprogram number to run
L - Repeats the subprogram call (1-99) times.
(<PATH>) - The Subprogram's directory path

M98 calls a subprogram in the format M98 **Pnnnn**, where **Pnnnn** is the number of the program to call, or M98 (<path>/Onnnnn), where <path> is the device path that leads to the subprogram.

The subprogram must contain an M99 to return to the main program. You can add an **Lnn** count to the M98 block M98 to call the subprogram **nn** times before continuing to the next block.

When your program calls an M98 subprogram, the control looks for the subprogram in the main program's directory. If the control cannot find the subprogram, it then looks in the location specified in Setting 251. Refer to page 220 for more information. An alarm occurs if the control cannot find the subprogram.

M98 Example:

The subprogram is a separate program (000100) from the main program (000002).

```
%  
000002 (PROGRAM NUMBER CALL);  
M98 P100 L4 (CALLS 000100 SUB 4 TIMES) ;  
M30 ;  
%  
%  
000100 (SUBPROGRAM);  
M00 ;  
M99 (RETURN TO MAIN PROGRAM) ;  
%  
  
%  
000002 (PATH CALL);  
M98 (USB0/000001.nc) L4 (CALLS 000100 SUB 4 TIMES) ;  
M30 ;  
%  
%  
000100 (SUBPROGRAM);  
M00 ;  
M99 (RETURN TO MAIN PROGRAM) ;  
%
```

M99 Subprogram Return or Loop

P - Program line number to go to when conditional test is met

M99 has three main uses:

- M99 is used at the end of a subprogram, local subprogram, or macro to return back to the main program.
- An M99 Pnn jumps the program to the corresponding Nnn in the program.
- An M99 in the main program causes the program to loop back to the beginning and execute until [RESET] is pressed.

Haas	
calling program:	00001 ;
	...

Haas	
	N50 M98 P2 ;
	N51 M99 P100 ;
	...
	N100 (continue here) ;
	...
	M30 ;
subprogram:	O0002 ;
	M99 ;

M99 jumps to a specific block with or without the macro option.

M104 / M105 Probe Arm Extend/Retract (Optional)

The optional tool setting probe arm is extended and retracted using these M-codes.

M109 Interactive User Input

P - A number in the range (500-549 or 10500-10549) representing the macro variable of the same name.

M109 lets a G-code program place a short prompt (message) on the screen. You must use a P code to specify macro variable in the range 500-549 or 10500 through 10549. The program can check for any character that can be entered from the keyboard by comparing with the decimal equivalent of the ASCII character (G47, Text Engraving, has a list of ASCII characters).

**NOTE:**

Macro Variables 540-599 and 10549-10599 are reserved for the WIPS (probe) option. If your machine is equipped with WIPS only use P500-539 or P10500-10599.

The following sample program asks the user a **Y** or **N** question, then waits for either a **Y** or an **N** to be entered. All other characters are ignored.

```
%  
O61091 (M109 INTERACTIVE USER INPUT) ;  
(This program has no axis movement) ;  
N1 #10501= 0. (Clear the variable) ;  
N5 M109 P10501 (Sleep 1 min?) ;  
IF [ #10501 EQ 0. ] GOTO5 (Wait for a key) ;  
IF [ #10501 EQ 89. ] GOTO10 (Y) ;  
IF [ #10501 EQ 78. ] GOTO20 (N) ;  
GOTO1 (Keep checking) ;  
N10 (A Y was entered) ;  
M95 (00:01) ;  
GOTO30 ;  
N20 (An N was entered) ;  
G04 P1. (Do nothing for 1 second) ;  
N30 (Stop) ;  
M30 ;  
%
```

The following sample program asks the user to select a number, then wait for a **1**, **2**, **3**, **4** or a **5** to be entered; all other characters are ignored.

```
%  
O00065 (M109 INTERACTIVE USER INPUT 2) ;  
(This program has no axis movement) ;  
N1 #10501= 0 (Clear Variable #10501) ;  
(Variable #10501 will be checked) ;  
(Operator enters one of the following selections);  
N5 M109 P10501 (1,2,3,4,5) ;  
IF [ #10501 EQ 0 ] GOTO5 ;  
(Wait for keyboard entry loop until entry) ;  
(Decimal equivalent from 49-53 represent 1-5) ;  
IF [ #10501 EQ 49 ] GOTO10 (1 was entered go to N10) ;  
IF [ #10501 EQ 50 ] GOTO20 (2 was entered go to N20) ;  
IF [ #10501 EQ 51 ] GOTO30 (3 was entered go to N30) ;  
IF [ #10501 EQ 52 ] GOTO40 (4 was entered go to N40) ;  
IF [ #10501 EQ 53 ] GOTO50 (5 was entered go to N50) ;
```

```

GOTO1 (Keep checking for user input loop until found) ;
N10 ;
(If 1 was entered run this sub-routine) ;
(Go to sleep for 10 minutes) ;
#3006= 25 (Cycle start sleeps for 10 minutes) ;
M95 (00:10) ;
GOTO100 ;
N20 ;
(If 2 was entered run this sub routine) ;
(Programmed message) ;
#3006= 25 (Programmed message cycle start) ;
GOTO100 ;
N30 ;
(If 3 was entered run this sub routine) ;
(Run sub program 20) ;
#3006= 25 (Cycle start program 20 will run) ;
G65 P20 (Call sub-program 20) ;
GOTO100 ;
N40 ;
(If 4 was entered run this sub routine) ;
(Run sub program 22) ;
#3006= 25 (Cycle start program 22 will be run) ;
M98 P22 (Call sub program 22) ;
GOTO100 ;
N50 ;
(If 5 was entered run this sub-routine) ;
(Programmed message) ;
#3006= 25 (Reset or cycle start will turn power off) ;
#12006= 1 ;
N100 ;
M30 (End Program) ;
%
```

M116 Vise Air Chips Blast On / M117 Vise Air Chips Blast Off

M116 turns the Vise Air Chips Blast option on, and M117 turns it off.

M130 Display Media / M131 Cancel Display Media

M130 Lets you display video and still images during program execution. Some examples of how you can use this feature are:

- Providing visual cues or work instructions during program operation
- Providing images to aid part inspection at certain points in a program
- Demonstrating procedures with video

The correct command format is M130 (file.xxx), where file.xxx is the name of the file, plus the path, if necessary. You can also add a second comment in parentheses to appear as a comment at the top of the media window.



NOTE:

M130 uses the subprogram search settings, Settings 251 and 252 in the same way that M98 does. You can also use the Insert Media File command in the editor to easily insert an M130 code that includes the filepath. Refer to page 183 for more information.

Permitted file formats are MP4, MOV, PNG, and JPEG.



NOTE:

For the fastest loading times, use files with pixel dimensions divisible by 8 (most unedited digital images have these dimensions by default), and a maximum pixel size of 1920 x 1080.

Your media appears in the Media tab under Current Commands. The media displays until the next M130 displays a different file, or M131 clears the media tab contents.

F8.6: Media Display Example - Work Instruction during a Program



M138 / M139 Spindle Speed Variation On/Off

Spindle Speed Variation (SSV) lets you specify a range within which the spindle speed continuously varies. This is helpful in suppressing tool chatter, which can lead to an undesirable part finish and/or damage to the cutting tool. The control varies the spindle speed based on Settings 165 and 166. For example, to vary spindle speed +/- 100 RPM from its current commanded speed with a duty cycle of 1 seconds, set Setting 165 to 100 and Setting 166 to 1.

The variation you use depends on the material, tooling, and the characteristics of your application, but 100 RPM over 1 second is a good starting point.

You can override the values of settings 165 and 166 using P and E address codes when used with M138. Where P is SSV Variation (RPM) and E is the SSV Cycle (Sec). See example below:

```
M138 P500 E1.5 (Turn SSV On, vary the speed by 500 RPM, cycle  
every 1.5 seconds);
```

```
M138 P500 (Turn SSV on, vary the speed by 500, cycle based on  
setting 166);
```

```
M138 E1.5 (Turn SSV on, vary the speed by setting 165, cycle  
every 1.5 seconds);
```



NOTE:

If you have an M138 Enn on one line and a G187 Enn on another line, the E codes would be unique to the line they are on. The Enn Code for the G187 would apply to G187 only and not affect the active SSV behavior.

M138 is independent of spindle commands; once commanded, it is active even when the spindle is not turning. Also, M138 remains active until canceled with M139, or at M30, Reset, or Emergency Stop.

M140 MQL On Continuous Mode / M141 MQL On Single Squirt Mode / M142 Stop MQL

M140 turns the Minimum Quantity Lubrication (MQL) option on, and M142 turns it off. M141 turns on the MQL on for the specified time, then turns it off.

M158 Mist Condenser On / M159 Mist Condenser Off

M158 turns on the Mist Condenser, and M159 turns off the Mist Condenser.



NOTE:

The mist condenser will run continuously if a program is running with axis motion.



NOTE:

An M158 in MDI will activate the mist condenser for 10 seconds, after this the mist condenser will turn OFF. If continuous operation is required the program will require feed motion of at least one axis.



NOTE:

If you want the mist condenser to remain ON, then go to CURRENT COMMANDS>DEVICES>MECHANISMS>MIST CONDENSER and press [F2] to turn it on

M160 Cancel Active PulseJet

Use M160 to cancel an active PulseJet M-code.

M161 Pulse Jet Continuous Mode

*P - Pnn is the interval at which oil pulses occur (Min = 1 / Max = 99 seconds). For example P3 means there will be a pulse every 3 seconds.

*indicates optional

M161 will turn PulseJet on whenever a feed move is active in a program.

Refer to setting “369 - PulseJet Injection Cycle Time” on page 491 to set the PulseJet oil flow duty cycle.

M162 PulseJet Single Event Mode

*P - Pnn is how many pulses (Min= 1 / Max= 99 squirts).

*indicates optional

M162 will turn on PulseJet for a defined number of pulses. Best used for drilling and tapping or to manually lubricate a tool.

**NOTE:**

M162 is a non-blocking code. Anything after the code will be executed immediately.

Refer to setting “370 - PulseJet Single Squirt Count” on page 491 to set the squirt count.

M163 PulseJet Modal Mode

*P - Pnn is how many pulses for each hole (Min= 1 / Max = 99).

*indicates optional

M163 activates PulseJet to turn on during any canned drill, tap or bore cycles.

**NOTE:**

When a canned cycle is canceled by a method such as a G80 or a Feed. It will also cancel the M163 modal command.

M163 Program Example:

```
G90 G54 G00 G28;  
S100 M03;  
M163 P3;  
G81 F12. R-1. Z-2.;  
X-1.;  
X-2.;  
G80;  
G00 X-3.;  
G84 F12. R-1. Z-2.;  
X-4.;  
G80;  
M30;
```

**NOTE:**

The PulseJet M163 P3 in this program is canceled by G80 and will only run the first cycle.

Refer to setting “370 - PulseJet Single Squirt Count” on page 491 to set the squirt count.

M180 / M181 Automatic Window Open / Close

M180 opens the Auto Window and M181 closes it.

**NOTE:**

The M-codes work only while the machine receives a cell-safe signal from a robot or if it has a light curtain installed. For more information refer to Robot Integration Aid - NGC reference document.

M183 Auto Air Hole Blast On / M184 Auto Air Hole Blast Off

M183 turns the Hole Blast option on, and M184 turns it off. M183 with a Pnnn argument (where nnn is in seconds) turns the Hole Blast on for the specified time, then turns it off.

M183 P2. ;

**NOTE:**

This will turn the Hole Blaster on for 2 seconds.

The Hole Blast can also be turned on/off by going to the Mechanisms tab. Current Commands> Devices> Mechanisms.

M199 Pallet / Part Load or Program End

M199 takes the place of an M30 or M99 at the end of a program. When running in Memory or MDI mode, pressing **Cycle Start** to run the program, the M199 will behave the same as an M30. It will stop and rewind the program back to the beginning. While running in Pallet Change mode, pressing **INSERT** while on the Pallet Schedule Table to run a program, the M199 behaves the same as an M50 + M99. It will end the program, get the next scheduled pallet and associated program, then continue to run until all scheduled pallets are completed.

M300 - APL/Robot Custom Sequence

The M300 code is used to call APL or Robot Custom Sequence programs. To call a specific program use the P code followed by the alias code, found in the APL custom sequence tab.

M300 Pn Q0 Plays the sequence in the background to the program.

M300 Pn R0 Plays the Sequence in reverse.

**NOTE:**

M300 Pn R0 ignores all non-robot related commands when the motion file is ran in reverse. Example of some commands ignored: Machine Dwells, Auto Door commands, Workholding commands, and Spindle Orients.

8.1.2 More Information Online

For updated information, including tips, tricks, maintenance procedures, and more, visit the Haas Service page at www.HaasCNC.com.

For the most current Operator's and Service Manuals scan the code below with your mobile device:



Chapter 9: Settings

9.1 Introduction

This chapter gives detailed descriptions of the settings that control the way that your machine works.

For the most current Setting information scan the code below with your mobile device.

F9.1: Mill - Settings Guide



9.1.1 List of Settings

Inside the **SETTINGS** tab, the settings are organized into groups. Use the **[UP]** and **[DOWN]** cursor arrow keys to highlight a setting group. Press the **[RIGHT]** cursor arrow key to see the settings in a group. Press the **[LEFT]** cursor arrow key to return to the setting group list.

To quickly access a single setting, make sure the **SETTINGS** tab is active, type the setting number, and then press **[F1]** or, if a setting is highlighted, press the **[DOWN]** cursor.

Some settings have numerical values that fit in a given range. To change the value of these settings, type the new value and press **[ENTER]**. Other settings have specific available values that you select from a list. For these settings, use the **[RIGHT]** cursor to display the choices. Press **[UP]** and **[DOWN]** to scroll through the options. Press **[ENTER]** to select the option.

Setting	Description	Page
1	Auto Power Off Timer	448
2	Power Off at M30	448
4	Graphics Rapid Path	448

Setting	Description	Page
5	Graphics Drill Point	448
6	Front Panel Lock	449
8	Prog Memory Lock	449
9	Dimensioning	449
10	Limit Rapid at 50%	450
15	H and T Code Agreement	450
17	Opt Stop Lock Out	450
18	Block Delete Lock Out	450
19	Feedrate Override Lock	450
20	Spindle Override Lock	450
21	Rapid Override Lock	451
22	Can Cycle Delta Z	451
23	9xxx Progs Edit Lock	451
27	G76 / G77 Shift Dir.	451
28	Can Cycle No Linear Axes	451
29	G91 Non-modal	452
31	Reset Program Pointer	452
32	Coolant Override	452
33	Coordinate System	453
34	4th Axis Diameter	453
35	G60 Offset	453
36	Program Restart	453
39	Beep @ M00, M01, M02, M30	454

Setting	Description	Page
40	Tool Offset Measure	454
42	M00 After Tool Change	454
43	Cutter Comp Type	454
44	Min F Radius CC%	454
45	Mirror Image X Axis	455
46	Mirror Image Y Axis	455
47	Mirror Image Z Axis	455
48	Mirror Image A Axis	455
52	G83 Retract Above R	456
53	Jog w/o Zero Return	456
56	M30 Restore Default G	456
57	Canned Cycle Exact Stop	456
58	Cutter Compensation	456
59	Probe Offset X+	457
60	Probe Offset X-	457
61	Probe Offset Y+	457
62	Probe Offset Y-	457
63	Tool Probe Width	457
64	Tool Offset Measure Uses Work	457
71	Default G51 Scaling	457
72	Default G68 Rotation	457
73	G68 Incremental Angle	458
74	9xxx Progs Trace	458

Setting	Description	Page
75	9xxx Progs Single BLK	458
76	Tool Release Lockout	458
77	Scale Integer F	459
79	5th-Axis Diameter	459
80	Mirror Image B Axis	459
81	Tool At Power Up	459
82	Language	460
83	M30/Resets Overrides	460
84	Tool Overload Action	460
85	Maximum Corner Rounding	461
86	M39 Lockout	462
87	Tool Change Resets Override	462
88	Reset Rests Override	462
90	Max Tools To Display	462
101	Feed Override -> Rapid	462
103	Cyc Start/Fh Same Key	463
104	Jog Handle to SNGL BLK	463
108	Quick Rotary G28	463
109	Warm-Up Time in Min.	463
110	Warmup X Distance	464
111	Warmup Y Distance	464
112	Warmup Z Distance	464
113	Tool Change Method	464

Setting	Description	Page
114	Conveyor Cycle Time (minutes)	464
115	Conveyor On-Time (minutes)	458
117	G143 Global Offset	465
118	M99 Bumps M30 Cntrs	465
119	Offset Lock	465
120	Macro Var Lock	465
130	Tap Retract Speed	465
131	Auto Door	465
133	Repeat Rigid Tap	466
142	Offset Chng Tolerance	466
143	Machine Data Collection Port	466
144	Feed Override -> Spindle	466
155	Load Pocket Tables	467
156	Save Offsets with Program	467
158	X Screw Thermal Comp%	467
159	Y Screw Thermal Comp%	467
160	Z Screw Thermal Comp%	467
162	Default To Float	467
163	Disable .1 Jog Rate	467
164	Rotary Increment	468
165	Ssv Variation (RPM)	468
166	Ssv Cycle	468
188	G51 X Scale	468

Setting	Description	Page
189	G51 Y Scale	468
190	G51 Z Scale	468
191	Default Smoothness	468
196	Conveyor Shutoff	468
197	Coolant Shutoff	469
199	Backlight Timer	469
216	Servo and Hydraulic Shutoff	469
238	High Intensity Light Timer (minutes)	469
239	Worklight Off Timer (minutes)	469
240	Tool Life Warning	469
242	Air Water Purge Interval	466
243	Air Water Purge On-Time	469
245	Hazardous Vibration Sensitivity	470
247	Simultaneous XYZ Motion in Tool Change	470
249	Enable Haas Startup Screen	470
250	Mirror Image C Axis	470
251	Subprogram Search Location	470
252	Custom Subprogram Search Location	471
253	Default Graphics Tool Width	472
254	5-Axis Rotary Center Distance	472
255	MRZP X Offset	473
256	MRZP Y Offset	474
257	MRZP Z Offset	475

Setting	Description	Page
261	DPRNT Store Location	476
262	DPRNT Destination File Path	477
263	DPRNT Port	477
264	Autofeed Step Up	478
265	Autofeed Step Down	478
266	Autofeed Minimum Override	478
267	Exit Jog Mode After Idle Time	478
268	Second Home Postion X	478
269	Second Home Position Y	478
270	Second Home Position Z	478
271	Second Home Position A	478
272	Second Home Position B	478
273	Second Home Position C	478
276	Workholding Input Monitor	481
277	Lubrication Cycle Interval	481
291	Main Spindle Speed Limit	481
292	Door Open Spindle Speed Limit	481
293	Tool Change Mid Position X	481
294	Tool Change Mid Position Y	481
295	Tool Change Mid Position Z	481
296	Tool Change Mid Position A	481
297	Tool Change Mid Position B	481
298	Tool Change Mid Position C	481

Setting	Description	Page
300	MRZP X Offset Master	484
301	MRZP Y Offset Master	484
302	MRZP Z Offset Master	484
303	MRZP X Offset Slave	484
304	MRZP Y Offset Slave	484
305	MRZP Z Offset Slave	484
306	Minimum Chip Clear Time	487
310	Min User Travel Limit A	487
311	Min User Travel Limit B	487
312	Min User Travel Limit C	488
313	Max User Travel Limit X	488
314	Max User Travel Limit Y	488
315	Max User Travel Limit Z	488
316	Max User Travel Limit A	488
317	Max User Travel Limit B	488
318	Max User Travel Limit C	488
323	Disable Notch Filter	490
325	Manual Mode Enabled	490
330	MultiBoot Selection Time out	490
335	Linear Rapid Mode	490
356	Beeper Volume	491
357	Warm Up Cycle Start Idle Time	491
369	PulseJet Injection Cycle Time	491

Setting	Description	Page
370	PulseJet Single Squirt Count	491
372	Parts Loader Type	492
375	APL Gripper Type	492
376	Light Curtain Enable	492
377	Negative Work Offsets	493
378	Safe Zone Calibrated Geometry Reference Point X	493
379	Safe Zone Calibrated Geometry Reference Point Y	493
380	Safe Zone Calibrated Geometry Reference Point Z	493
381	Enable Touchscreen	493
382	Disable Pallet Changer	493
383	Table Row Size	493
389	Vise Check For Part Hold At Cycle Start	494
396	Enable / Disable Virtual Keyboard	494
397	Press and Hold Delay	494
398	Header Height	494
399	Header Tab	494
400	Pallet Ready Beep Type	494
403	Change Popup Button Size	495
408	Exclude Tool From Safe Zone	495
409	Default Coolant Pressure	495
416	Media Destination	495
420	ATC Button Behavior	496
421	General Orient Angle	496

Setting	Description	Page
422	Lock Graphics Plane	496
423	Help Text Icon Size	496
424	Mist Extractor Condenser Time Out	496

1 - Auto Power Off Timer

This setting is used to automatically power-down the machine after a period of idle time. The value entered in this setting is the number of minutes the machine remains idle until it is powered down. The machine does not power down while a program is running, and the time (number of minutes) starts back at zero anytime a button is pressed or the **[HANDLE JOG]** control is used. The auto-off sequence gives the operator a 15-second warning before power down, at which time pressing any button stops the power down.

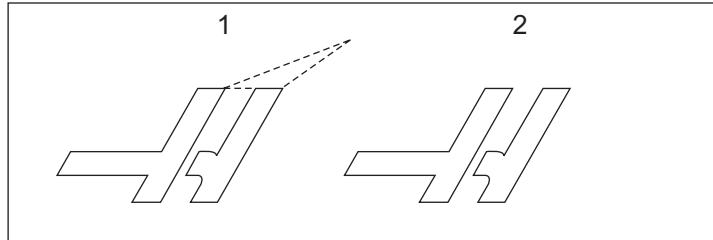
2 - Power Off at M30

If this setting is set to **ON**, the machine powers down at the end of a program (**M30**). The machine gives the operator a 15-second warning once an **M30** is reached. Press any key to interrupt the power-off sequence.

4 - Graphics Rapid Path

This setting changes the way a program is viewed in the Graphics mode. When it is **OFF**, rapid, non-cutting tool motions do not leave a path. When it is **ON**, rapid tool motions leave a dashed line on the screen.

F9.2: Setting 4 -Graphics Rapid Path:[1] All Rapid Tool Motions Shown with a Dashed Line When **ON**. [2] Only Cut Lines Shown When **OFF**.



5 - Graphics Drill Point

This setting changes the way a program is viewed in Graphics mode. When it is **ON**, canned cycle drill locations leave a circle mark on the screen. When it is **OFF**, no additional marks are shown on the graphics display.

6 - Front Panel Lock

When set to **ON**, this setting disables the following keys:

Front pendant:

- Spindle [**FWD**] / [**REV**]
- [**ATC FWD**] / [**ATC REV**]
- [**NEXT TOOL**]
- [**TOOL RELEASE**]

Remote Jog Handle (RJH):

- [**NEXT**]
- [**PREV**]

8 - Prog Memory Lock

This setting locks out the memory editing functions (**[ALTER]**, **[INSERT]**, etc.) when it is set to **ON**. This also locks out MDI. Editing functions are not restricted by this setting.

9 - Dimensioning

This setting selects between inch and metric mode. When it is set to **INCH**, the programmed units for X, Y, and Z are inches, to 0.0001". When it is set to **MM**, programmed units are millimeters, to 0.001 mm. All offset values are converted when this setting is changed from inches to metric, or vice versa. However, changing this setting does not automatically translate a program stored in memory; the programmed axis values must be changed for the new units.

When set to **INCH**, the default G-code is G20, when set to **MM**, the default G-code is G21.

	Inch	Metric
Feed	in/min	mm/min
Max Travel	Varies by axis and model	
Minimum programmable dimension	.0001	.001

Axis jog key	Inch	Metric
.0001	.0001 in/jog click	.001 mm/jog click
.001	.001 in/jog click	.01 mm/jog click
.01	.01 in/jog click	.1 mm/jog click
.1	.1 in/jog click	1 mm/jog click

10 - Limit Rapid at 50%

Turning this setting **ON** limits the machine to 50% of its fastest non-cutting axis motion (rapids). This means, if the machine can position the axes at 700 inches per minute (ipm), it is limited to 350 ipm when this setting is **ON**. The control displays a 50% rapid override message, when this setting is **ON**. When it is **OFF**, the highest rapid speed of 100% is available.

15 - H and T Code Agreement

Turning this setting **ON** has the machine check to ensure that the **H** offset code matches the tool in the spindle. This check can help to prevent crashes.



NOTE:

*This setting does not generate an alarm with an **H00**. **H00** is used to cancel the tool length offset.*

17 - Opt Stop Lock Out

The Optional Stop feature is not available when this setting is **ON**.

18 - Block Delete Lock Out

The Block Delete feature is not available when this setting is **ON**.

19 - Feedrate Override Lock

The feedrate override buttons are disabled when this setting is turned **ON**.

20 - Spindle Override Lock

The spindle speed override keys are disabled when this setting is turned **ON**.

21 - Rapid Override Lock

The axis rapid override keys are disabled when this setting is turned **ON**.

22 - Can Cycle Delta Z

This setting specifies the distance to retract the Z Axis to clear chips during a G73 canned cycle.

23 - 9xxx Progs Edit Lock

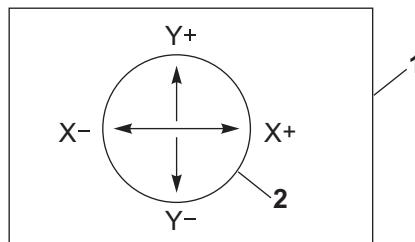
When this setting is **ON**, the control does not let you view or alter the files in the 09000 directory in **Memory/**. This protects macro programs, probing cycles, and any other files in the 09000 folder.

If you attempt to access the 09000 folder while Setting 23 is **ON**, you get the message *Setting 23 restricts access to folder*.

27 - G76 / G77 Shift Dir.

This setting specifies the direction to move to clear the boring tool during a G76 or G77 canned cycle. Selections are **X+**, **X-**, **Y+**, or **Y-**. For more information on how this setting works, refer to G76 and G77 cycle in the G-code section on page **353**.

- F9.3:** Setting 27, Direction the Tool is Shifted to Clear Boring Tool: [1] Part, [2] Bored hole.



28 - Can Cycle No Linear Axes

This is an **ON/OFF** setting. The preferred setting is **ON**.

When it is **OFF**, the initial canned cycle definition block requires an **X** or **Y** code for the canned cycle to be executed.

When it is **ON**, the initial canned cycle definition block causes one cycle to be executed even when there is no **X** or **Y** code in the block.



NOTE:

*When an **L0** is in that block, it will not execute the canned cycle on the definition line. This setting has no effect on G72 cycles.*

29 - G91 Non-modal

Turning this setting **ON** uses the **G91** command only in the program block it is in (non-modal). When it is **OFF**, and a **G91** is commanded, the machine uses incremental moves for all axis positions.

**NOTE:**

*This setting must be **OFF** for **G47** engraving cycles.*

31 - Reset Program Pointer

When this setting is **OFF**, **[RESET]** does not change the position of the program pointer. When it is **ON**, pressing **[RESET]** moves the program pointer to the beginning of the program.

32 - Coolant Override

This setting controls how the coolant pump operates. When Setting 32 **NORMAL**, you can press **[COOLANT]**, or you can use M-codes in a program, to turn the coolant pump on and off.

When Setting 32 is **OFF**, the control gives the message **FUNCTION LOCKED** when you press **[COOLANT]**. The control gives an alarm when a program commands the coolant pump on or off.

When Setting 32 is **IGNORE**, the control ignores all programmed coolant commands, but you can press **[COOLANT]** to turn the coolant pump on or off.

**NOTE:**

Starting in software version 100.22.000.1000 and higher the control will monitor the coolant level.

**NOTE:**

*When Setting 32 is set to **NORMAL** and **IGNORE** the control will allow Coolant Level Monitoring to take place.*

**NOTE:**

*When Setting 32 is set to **OFF** it will disable the monitor and all the notifications of Low Coolant condition. Only the coolant gauge will be shown indicating the remaining level.*

33 - Coordinate System

This setting changes the way the Haas control recognizes the work offset system when a G52 or G92 is programmed. It can be set to **FANUC** or **HAAS**.

Set to **FANUC** with G52:

Any values in the G52 register are added to all work offsets (global coordinate shift). This G52 value can be entered either manually or through a program. When **FANUC** is selected, pressing **[RESET]**, commanding an M30, or machine power down clears out the value in G52.

Set to **HAAS** with G52:

Any values in the G52 register are added to all work offsets. This G52 value can be entered either manually or through a program. The G52 coordinate shift value is set to zero (zeroed) by manually entering zero, or by programming it with G52 X0, Y0, and/or Z0.

34 - 4th Axis Diameter

This is used to set the diameter of the 4th axis, which the control uses to determine the angular feedrate. The feedrate in a program is always inches or millimeters per minute (G94); therefore, the control must know the diameter of the part being machined in the 4th axis in order to compute angular feedrate. Refer to Setting 79 on page **459** for information on the 5th axis diameter setting.

35 - G60 Offset

This setting is used to specify the distance an axis travels past the target point prior to reversing. Also see G60.

36 - Program Restart

When this setting is **ON**, restarting a program from a point other than the beginning directs the control to scan the entire program to make sure that the tools, offsets, G- and M-codes, and axis positions are set correctly before the program starts at the block where the cursor is positioned.

When Setting 36 is **ON**, an alarm will be generated if the program is started on a line of code where Cutter Compensation is active. It is mandatory to start the program before a line of code with G41/G42 or after a line of code with G40.



NOTE:

The machine first goes to the position and changes to the tool specified in the block before the cursor position. For example, if the cursor is on a tool change block in the program, the machine changes to the tool loaded before that block, then it changes to the tool specified in the block at the cursor location.

The control processes these M-codes when Setting 36 is enabled:

M08 Coolant On

M09 Coolant Off

M41 Low Gear

M42 High Gear

M51-M58 Set User M

M61-M68 Clear User M

When Setting 36 is **OFF**, the control starts the program, but it does not check the conditions of the machine. Having this setting **OFF** may save time when running a proven program.

39 - Beep @ M00, M01, M02, M30

Turning this setting **ON** causes the keyboard beeper to sound when an **M00**, **M01** (with Optional Stop active), **M02**, or an **M30** is found. The beeper continues until a button is pressed.

40 - Tool Offset Measure

This setting selects how tool size is specified for cutter compensation. Set to either **RADIUS** or **DIAMETER**. The selection also effects the Tool Diameter geometry and wear values displayed on the **TOOL OFFSETS** table. If setting 40 is changed from **RADIUS** to **DIAMETER**, the displayed value is twice the value entered before.

42 - M00 After Tool Change

Turning this setting **ON** stops the program after a tool change and a message is displayed stating this. **[CYCLE START]** must be pressed to continue the program.

43 - Cutter Comp Type

This controls how the first stroke of a compensated cut begins and the way the tool is cleared from the part. The selections can be **A** or **B**; see the Cutter Compensation section on page 199.

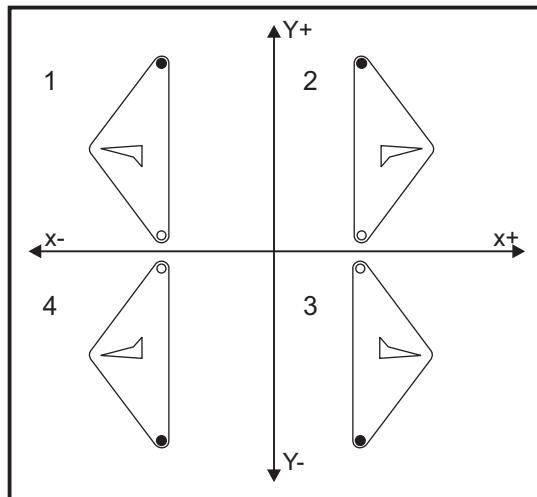
44 - Min F In Radius CC %

Minimum feedrate in radius cutter compensation percent setting affects the feed rate when cutter compensation moves the tool towards the inside of a circular cut. This type of cut slows down to maintain a constant surface feed rate. This setting specifies the slowest feed rate as a percentage of the programmed feed rate.

45, 46, 47 - Mirror Image X, Y, Z Axis

When one or more of these settings is **ON**, axis motion is mirrored (reversed) around the work zero point. See also G101, Enable Mirror Image.

- F9.4:** No Mirror Image [1], Setting 45 **ON** - X Mirror [2], Setting 46 **ON** - Y Mirror [4], Setting 45 and Setting 46 **ON** - XY Mirror [3]



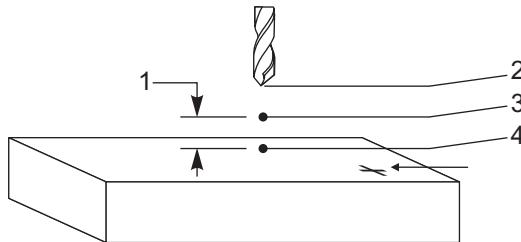
48 - Mirror Image A Axis

This is an **ON/OFF** setting. When it is **OFF**, axis motions occur normally. When it is **ON**, A-Axis motion may be mirrored (or reversed) around the work zero point. Also, see G101 and Settings 45, 46, 47, 80, and 250.

52 - G83 Retract Above R

This setting changes the way G83 (peck drilling cycle) behaves. Most programmers set the reference (R) plane well above the cut to ensure that the chip clearing motion actually allows the chips to get out of the hole. However this wastes time as the machine drills through this empty distance. If Setting 52 is set to the distance required to clear chips, the R plane can be set much closer to the part being drilled.

- F9.5:** Setting 52, Drill Retract Distance: [1] Setting 52, [2] Start Position, [3] Retract Distance Set by Setting 52, [4] R Plane



53 - Jog w/o Zero Return

Turning this setting **ON** allows the axes to be jogged without zero returning the machine (finding machine home). This is a dangerous condition as the axis can be run into the mechanical stops and possibly damage the machine. When the control is powered up, this setting automatically returns to **OFF**.

56 - M30 Restore Default G

When this setting is **ON**, ending a program with **M30** or pressing **[RESET]** returns all modal G-codes to their defaults.

57 - Canned Cycle Exact Stop

When this setting is **OFF**, the axes may not get to the programmed X, Y position before the Z-Axis starts moving. This may cause problems with fixtures, fine part details or workpiece edges.

Turning this setting **ON** makes the mill reach the programmed X,Y position before the Z-Axis moves.

58 - Cutter Compensation

This setting selects the type of cutter compensation used (FANUC or YASNAC). See the Cutter Compensation section on page 199.

59, 60, 61, 62 - Probe Offset X+, X-, Y+, Y-

These settings are used to define the displacement and size of the spindle probe. They specify the travel distance and direction from where the probe is triggered to where the actual sensed surface is located. These settings are used by G31, G36, G136, and M75 codes. The values entered for each setting can be either positive or negative numbers, equal to the radius of the probe stylus tip.

You can use macros to access these settings; for more information, refer to the Macro section of this manual (starting on page 248).

**NOTE:**

These settings are not used with the Renishaw WIPS option.

63 - Tool Probe Width

This setting is used to specify the width of the probe used to test tool diameter. This setting only applies to the probing option; it is used by G35. This value is equal to the diameter of the tool probe stylus.

64 - T. Ofs Meas Uses Work

The (Tool Offset Measure Uses Work) setting changes the way the [TOOL OFFSET MEASURE] key works. When this is ON, the entered tool offset is the measured tool offset plus the work coordinate offset (Z-Axis). When it is OFF, the tool offset equals the Z machine position.

71 - Default G51 Scaling

This specifies the scaling for a G51 (See G-code Section, G51) command when the P address is not used. The default is 1.000.

72 - Default G68 Rotation

This specifies the rotation, in degrees, for a G68 command when the R address is not used.

73 - G68 Incremental Angle

This setting allows the G68 rotation angle to be changed for each commanded G68. When this switch is **ON** and a G68 command is executed in the Incremental mode (G91), the value specified in the Raddress is added to the previous rotation angle. For example, an R value of 10 causes the feature rotation to be 10 degrees the first time commanded, 20 degrees the next time, etc.

**NOTE:**

This setting must be OFF when you command an engraving cycle (G47).

74 - 9xxx Progs Trace

This setting, along with Setting 75, is useful for debugging CNC programs. When Setting 74 is **ON**, the control displays the code in the macro programs (09xxxx). When the setting is **OFF**, the control does not display the 9000 series code.

75 - 9xxxx Progs Single BLK

When Setting 75 is **ON** and the control is operating in Single Block mode, then the control stops at each block of code in a macro program (09xxxx) and waits for the operator to press **[CYCLE START]**. When Setting 75 is **OFF** the macro program is run continuously, the control does not pause at each block, even if Single Block is **ON**. The default setting is **ON**.

When Setting 74 and Setting 75 are both **ON**, the control acts normally. That is, all blocks executed are highlighted and displayed, and when in Single-Block mode there is a pause before each block is executed.

When Setting 74 and Setting 75 are both **OFF**, the control executes 9000 series programs without displaying the program code. If the control is in Single-Block mode, no single-block pause occurs during the running of the 9000 series program.

When Setting 75 is **ON** and Setting 74 is **OFF**, 9000 series programs are displayed as they are executed.

76 - Tool Release Lock Out

When this setting is **ON**, the **[TOOL RELEASE]** key on the keyboard is disabled.

77 - Scale Integer F

This setting allows the operator to select how the control interprets an F value (feedrate) that does not contain a decimal point. (It is recommended that you always use a decimal point.) This setting helps operators run programs developed on a control other than Haas.

There are 5 feedrate settings. This chart shows the effect of each setting on a given F10 address.

INCH		MILLIMETER	
Setting 77	Feedrate	Setting 77	Feedrate
DEFAULT	F0.0010	DEFAULT	F0.0100
INTEGER	F10.	INTEGER	F10.
.1	F1.0	.1	F1.0
.01	F0.10	.01	F0.10
.001	F0.010	.001	F0.010
.0001	F0.0010	.0001	F0.0010

79 - 5th-Axis Diameter

This is used to set the diameter of the 5th axis, that the control uses to determine the angular feedrate. The feedrate in a program is always inches or millimeters per minute; therefore, the control must know the diameter of the part being machined in the 5th-axis in order to compute angular feedrate. Refer to Setting 34 on page 453 for more information on the 4th axis diameter setting.

80 - Mirror Image B Axis

This is an **ON/OFF** setting. When it is **OFF**, axis motions occur normally. When it is **ON**, B-Axis motion may be mirrored (or reversed) around the work zero point. Also, see G101 and Settings 45, 46, 47, 48, and 250.

81 - Tool At Power Up

When **[POWER UP]** is pressed, the control changes to the tool specified in this setting. If zero (0) is specified, no tool change occurs at power up. The default setting is 1.

Setting 81, causes one of these actions to occur after you press **[POWER UP]**:

- If Setting 81 is set to zero, the carousel rotates to pocket #1. No tool change is performed.
- If Setting 81 contains the tool #1, and the tool currently in the spindle is tool #1, and **[ZERO RETURN]** then **[ALL]** are pressed, the carousel remains at the same pocket and no tool change is performed.
- If Setting 81 contains the tool number of a tool not currently in the spindle, the carousel rotates to pocket #1 and then to the pocket containing the tool specified by Setting 81. A tool change is performed to change the specified tool into the spindle.

82 - Language

Languages other than English are available in the Haas control. To change to another language, choose a language with the **[LEFT]** and **[RIGHT]** cursor arrows, then press **[ENTER]**.

83 - M30/Resets Overrides

When this setting is **ON**, an **M30** restores any overrides (feedrate, spindle, rapid) to their default values (100%).

84 - Tool Overload Action

When a tool becomes overloaded, Setting 84 designates the control response. These settings cause specified actions (refer to the Advanced Tool Management Introduction on page 124):

- **ALARM** causes the machine to stop.
- **FEEDHOLD** displays the message *Tool Overload* and the machine stops in a feedhold situation. Press any key to clear the message.
- **BEEP** causes an audible noise (beep) from the control.
- **AUTOFEED** causes the control to automatically limit the feedrate based on the tool load.

**NOTE:**

When tapping (rigid or floating), the feed and spindle overrides are locked out, so the AUTOFEED setting is ineffective (the control appears to respond to the override buttons, by displaying the override messages).

**CAUTION:**

Do not use the AUTOFEED setting when thread milling or auto reversing tapping heads, as it can cause unpredictable results or even a crash.

The last commanded feedrate is restored at the end of program execution, or when the operator presses [RESET] or turns OFF the AUTOFEED setting. The operator can use [FEEDRATE OVERRIDE] while the AUTOFEED setting is selected. These keys are recognized by the AUTOFEED setting as the new commanded feedrate as long as the tool load limit is not exceeded. However, if the tool load limit has already been exceeded, the control ignores [FEEDRATE OVERRIDE].

85 - Maximum Corner Rounding

This setting defines the machining accuracy tolerance around corners. The initial default value is 0.0250". This means that the control keeps the radii of corners no bigger than 0.0250".

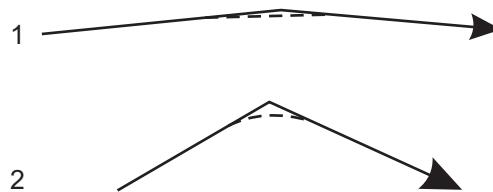
Setting 85 causes the control to adjust feeds around corners in all 3 axes to meet the tolerance value. The lower the value of Setting 85, the slower the control feeds around corners to meet the tolerance. The higher the value of Setting 85, the faster the control feeds around corners, up to the commanded feedrate, but it could round the corner off to a radius up to the tolerance value.

**NOTE:**

The angle of the corner also affects the change to the feedrate. The control can cut shallow corners within tolerance at a higher feedrate than it can with tighter corners.

F9.6:

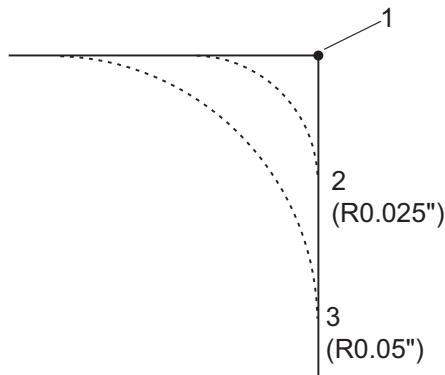
The control can cut corner [1] within tolerance at a higher feedrate than it can cut corner [2].



If Setting 85 has a value of zero, the control acts as if exact stop is active in each motion block.

Refer also to Setting 191 on page 468 and G187 on page 394.

- F9.7:** Assume that the commanded feedrate is too high to achieve corner [1]. If Setting 85 has a value of 0.025, then the control slows the feedrate enough to achieve corner [2] (with a radius of 0.025"). If Setting 85 has a value of 0.05, then the control slows the feedrate enough to achieve corner [3]. The feedrate to achieve corner [3] is faster than the feedrate to achieve corner [2].



86 - M39 (Rotate Tool Turret) Lockout

When this setting is **ON**, the control ignores **M39** commands.

87 - Tool Change Resets Override

This is an **ON/OFF** setting. When a **M06** is executed and this setting is **ON**, any overrides are canceled and set to their programmed values.



NOTE:

This setting only affects programmed tool changes, it does not affect [ATC FWD] or [ATC REV] tool changes.

88 - Reset Resets Overrides

This is an **ON/OFF** setting. When it is **ON** and **[RESET]** is pressed, any overrides are canceled and set to their programmed values or defaults (100%).

90 - Max Tools To Display

This setting limits the number of tools displayed on the Tool Offsets screen.

101 - Feed Override -> Rapid

Pressing **[HANDLE FEED]**, with this setting **ON**, will cause the jog handle to affect both the feedrate and the rapid rate overrides. Setting 10 affects the maximum rapid rate. The rapid rate cannot exceed 100%. Also, **[+10% FEEDRATE]**, **[- 10% FEEDRATE]**, and **[100% FEEDRATE]** change the rapid and feed rate together.

103 - Cyc Start/Fh Same Key

The **[CYCLE START]** button must be pressed and held to run a program when this setting is **ON**. When **[CYCLE START]** is released, a feed hold is generated.

This setting cannot be turned on while Setting 104 is **ON**. When one of them is set to **ON**, the other automatically turns off.

104 - Jog Handle to SNGL BLK

The **[HANDLE JOG]** control can single-step through a program when this setting is **ON**. Reversing the **[HANDLE JOG]** control direction generates a feed hold.

This setting cannot be turned on while Setting 103 is **ON**. When one of them is set to **ON**, the other automatically turns off.

108 - Quick Rotary G28

If this setting is **ON**, the control returns the rotary axes to zero in +/-359.99 degrees or less.

For example, if the rotary unit is at +/-950.000 degrees and a zero return is commanded, the rotary table rotates +/-230.000 degrees to the home position if this setting is **ON**.

**NOTE:**

The rotary axis returns to the machine home position, not the active work coordinate position.

**NOTE:**

This function only works when used with a G91 and not a G90.

109 - Warm-Up Time in MIN.

This is the number of minutes (up to 300 minutes from power-up) during which the control applies the compensations specified in Settings 110-112.

Overview – When the machine is powered on, if Setting 109, and at least one of Settings 110, 111, or 112, are set to a nonzero value, the control gives this warning:

CAUTION! Warm up Compensation is specified!

Do you wish to activate

Warm up Compensation (Y/N) ?

If you answer **Y** to the prompt, the control immediately applies the total compensation (Setting 110, 111, 112), and the compensation begins to decrease as the time elapses. For instance, after 50% of the time in Setting 109 has elapsed, the compensation distance is 50%.

To restart the time period, power the machine off and on, and then answer **YES** to the compensation query at start-up.



CAUTION: *Changing Setting 110, 111, or 112 while compensation is in progress can cause a sudden movement of up to 0.0044 inch.*

110, 111, 112 - Warmup X, Y, Z Distance

Settings 110, 111, and 112 specify the amount of compensation (max = +/- 0.0020" or +/- 0.051 mm) applied to the axes. Setting 109 must have a value entered for settings 110-112 to have an effect.

113 - Tool Change Method

This setting selects how a tool change is executed.

A selection of **Auto** defaults to the automatic tool changer on the machine.

A selection of **Manual** allows for manual tool change operation. When a tool change is executed in a program, the machine will stop at a tool change and prompt you to load the tool into the spindle. Insert the spindle and press **[CYCLE START]** to continue the program.

114 - Conveyor Cycle (minutes)

Setting 114 Conveyor Cycle Time is the interval that the conveyor turns on automatically. For example, if setting 114 is set to 30, the chip conveyor turns on every half an hour.

On-time should be set no greater than 80% of cycle time. Refer to Setting 115 on page **458**.

NOTE: *The **[CHIP FWD]** button (or **M31**) starts the conveyor in the forward direction and starts the cycle.*

*The **[CHIP STOP]** button (or **M33**) stops the conveyor and cancels the cycle.*

115 - Conveyor On-time (minutes)

Setting 115 Conveyor On-Time is the amount of time the conveyor runs. For example, if setting 115 is set to 2, the chip conveyor runs for 2 minutes, then turns off.

On-time should be set no greater than 80% of cycle time. Refer to Setting 114 Cycle Time on page **464**.

NOTE: *The **[CHIP FWD]** button (or **M31**) starts the conveyor in the forward direction and starts the cycle.*

*The **[CHIP STOP]** button (or **M33**) stops the conveyor and cancels the cycle.*

117 - G143 Global Offset (VR Models Only)

This setting is provided for customers who have several 5-axis Haas mills and want to transfer the programs and tools from one to another. The pivot-length difference is entered into this setting, and it is applied to the G143 tool length compensation.

118 - M99 Bumps M30 CNTRS

When this setting is ON, an M99 adds one to the M30 counters (these are visible after pressing [CURRENT COMMANDS]).

**NOTE:**

M99 only increases the counters as it occurs in a main program, not in a sub-program.

119 - Offset Lock

Turning the setting ON does not allow the values in the Offset display to be altered. However, programs that alter offsets with macros or G10 are permitted to do so.

120 - Macro Var Lock

Turning this setting ON does not allow the macro variables to be altered. However, programs that alter macro variables can do so.

130 - Tap Retract Speed

This setting affects the retract speed during a tapping cycle (The mill must have the Rigid Tapping option). Entering a value, such as 2, commands the mill to retract the tap twice as fast as it went in. If the value is 3, it retracts three times as fast. A value of 0 or 1 has no effect on the retract speed.

Entering a value of 2 is the equivalent of using a J address code value of 2 for G84 (tapping canned cycle). However, specifying a J code for a rigid tap overrides Setting 130.

131 - Auto Door

When this setting is set to ON the door closes when [CYCLE START] is pressed and opens when the program reaches an M00, M01 (with Optional Stop turned ON), M02, or M30 and the spindle has stopped turning.

**NOTE:**

The Autodoor button will Open/Close the door regardless if setting 131 is set to On or OFF, except if the machine has the HE patch installed.

To use M80/ M81 Auto Door Open / Close M-codes. Refer to M80 / M81 (Auto Door Open / close M-codes).



NOTE:

The M-codes work only while the machine receives a cell-safe signal from a robot or if it has a light curtain installed. For more information refer to <https://www.haascnc.com/service/troubleshooting-and-how-to/reference-documents/robot-integration-aid---ngc.html> reference document.

133 - Repeat Rigid Tap

This setting (Repeat Rigid Tap) ensures that the spindle is oriented during tapping so that the threads line up when a second tapping pass is programmed in the same hole.



NOTE:

This setting must be ON when a program commands peck tapping.

142 - Offset Chng Tolerance

This setting is intended to prevent operator errors. It generates a warning message if an offset is changed by more than the setting's value, 0 to 3.9370 inches (0 to 100 mm). If you change an offset by more than the entered amount (either positive or negative), the control prompts: *XX changes the offset by more than Setting 142! Accept (Y/N) ?*

Press [Y] to continue and update the offset. Press [N] to reject the change.

143 - Machine Data Collection Port

When this setting has a non-zero value, it defines the network port that the control uses to send machine data collection information. If this setting has a value of zero, the control does not send machine data collection information.

144 - Feed Override->Spindle

This setting is intended to keep the chip load constant when an override is applied. When this setting is ON, any feedrate override is also applied to the spindle speed, and the spindle overrides are disabled.

155 - Load Pocket Tables

This setting is used when a software upgrade is performed and/or memory has been cleared and/or the control is re-initialized. In order to replace the contents of the side-mount tool changer pocket tool table with the data from the file, the setting must be **ON**.

If this setting is **OFF** when loading an Offset file from a hardware device, the contents of the **Pocket Tool** table is unaltered. Setting 155 automatically defaults to **OFF** when the machine is turned on.

156 - Save Offsets with Program

When this setting is **ON**, the control includes the offsets in the program file when you save it. The offsets appear in the file before the final % sign, under the heading 0999999.

When you load the program back into memory, the control prompts *Load Offsets (Y/N?)*. Press **Y** if you want to load the saved offsets. Press **N** if you do not want to load them.

158, 159, 160 - X, Y, Z Screw Thermal COMP%

These settings can be set from -50 to +50 and adjust the existing screw thermal compensation by -50% to +50% accordingly.

162 - Default To Float

When this setting is **ON**, the control will interpret the integer code as if it had a decimal point. When the setting is **OFF**, values given after address codes that do not include decimal points are taken as machinist's notation; for example, thousandths or ten-thousandths. The feature applies to these address codes: X, Y, Z, A, B, C, E, I, J, K, U, and W.

	Value entered	With Setting Off	With Setting On
In Inch mode	X-2	X-.0002	X-2.
In MM mode	X-2	X-.002	X-2.



NOTE:

This setting affects the interpretation of all programs. It does not alter the effect of setting 77 Scale Integer F.

163 - Disable .1 Jog Rate

This setting disables the highest jog rate. If the highest jog rate is selected, the next lower rate is automatically selected instead.

164 - Rotary Increment

This setting applies to the [PALLET ROTATE] button on the EC-300 and EC-1600. It specifies the rotation for the rotary table in the load station. It should be set to a value from 0 to 360. The default value is 90. For example, entering 90 rotates the pallet 90 degrees each time the rotary index button is pressed. If it is set to zero, the rotary table does not rotate.

165 - Main Spindle SSV Variation (RPM)

Specifies the amount by which to allow the RPM to vary above and below the commanded value during use of the Spindle Speed Variation feature. This must be a positive value.

166 - Main Spindle SSV Cycle

Specifies the duty cycle, or the rate of change of the Main Spindle Speed. This must be a positive value.

188, 189, 190 - G51 X, Y, Z SCALE

You can scale the axes individually with these settings (the value must be a positive number).

Setting 188 = G51 X SCALE

Setting 189 = G51 Y SCALE

Setting 190 = G51 Z SCALE

If setting 71 has a value, then the control ignores Settings 188 - 190, and it uses the value in setting 71 for scaling. If the value for setting 71 is zero, then the control uses Settings 188 - 190.



NOTE:

When settings 188-190 are in effect, only linear interpolation, G01, is allowed. If G02 or G03 is used, alarm 467 is generated.

191 - Default Smoothness

This setting's value of ROUGH, MEDIUM, or FINISH sets the default smoothness and a maximum corner rounding factor. The control uses this default value unless a G187 command overrides the default.

196 - Conveyor Shutoff

This specifies the amount of time to wait without activity prior to turning off the chip conveyor (and washdown coolant, if installed). Units are minutes.

197 - Coolant Shutoff

This setting is the amount of time to wait without activity before Coolant flow stops. Units are minutes.

199 - Backlight Timer

This setting is the time in minutes after which the machine display backlight turns off when there is no input at the control (except in JOG, GRAPHICS, or SLEEP mode or when an alarm is present). Press any key to restore the screen (**[CANCEL]** is preferred).

216 - Servo and Hydraulic Shutoff

This setting specifies the duration of idle time, in seconds, before Power Save Mode starts. Power Save Mode shuts down all servo motors and hydraulic pumps. The motors and pumps start up again when needed (axis/spindle motion, program execution, etc.).

238 - High Intensity Light Timer (minutes)

Specifies the duration in minutes that the High Intensity Light option (HIL) remains turned on when activated. The light turns on when the door is opened and the work light switch is on. If this value is zero, then the light will remain turned on while the doors are open.

239 - Worklight Off Timer (minutes)

Specifies the amount of time in minutes after which the work light will turn off automatically if there are no key presses or **[HANDLE JOG]** changes. If a program is running when the light turns off, the program will continue running.

240 - Tool Life Warning

This value is a percentage of tool life. When tool wear reaches this threshold percentage, the control displays a Tool Wear Warning icon.

242 - Air Water Purge Interval (minutes)

This setting specifies the interval, in minutes, between condensate purges from the system air reservoir.

243 - Air Water Purge On-Time (seconds)

This setting specifies the duration, in seconds, of condensate purges from the system air reservoir.

245 - Hazardous Vibration Sensitivity

This setting has (3) levels of sensitivity for the hazardous vibration accelerometer in the machine's control cabinet: **Normal**, **Low**, or **Off**. The value defaults to **Normal** at each machine power-up.

You can see the current g force reading on the **Gauges** page in **Diagnostics**.

Depending on the machine, vibration is considered hazardous when it exceeds 600 - 1,400 g. At or above the limit, the machine gives an alarm.

If your application tends to cause vibration, you can change Setting 245 to a lower sensitivity to prevent nuisance alarms.

247 - Simultaneous XYZ Motion in Tool Change

Setting 247 defines how the axes move during a tool change. If Setting 247 is **OFF**, the Z Axis retracts first, followed by X- and Y-Axis motion. This feature can be useful in avoiding tool collisions for some fixture configurations. If Setting 247 is **ON**, the axes move simultaneously. This may cause collisions between the tool and the workpiece, due to B- and C-Axis rotations. It is strongly recommended that this setting remain **OFF** on the UMC-750, due to the high potential for collisions.

249 - Enable Haas Startup Screen

Enable Haas Startup Screen. If this setting is **ON**, the Haas Startup Screen will be displayed every time the machine is powered on.

This setting can be disabled through the Setting page or by pressing **[F1]** at the Haas Startup Screen.

250 - Mirror Image C Axis

This is an **ON/OFF** setting. When it is **OFF**, axis motions occur normally. When it is **ON**, C-Axis motion may be mirrored (or reversed) around the work zero point. Also, see G101 and Settings 45, 46, 47, 48, and 80.

251 - Subprogram Search Location

This setting specifies the directory to search for external subprograms when the subprogram is not in the same directory as the main program. Also, if the control cannot find an M98 subprogram, the control looks here. Setting 251 has (3) options:

- **Memory**
- **USB Device**
- **Setting 252**

For the **Memory** and **USB Device** options, the subprogram must be in the root directory of the device. For the **Setting 252** selection, Setting 252 must specify a search location to use.

**NOTE:**

When you use M98:

- The P code (nnnnn) is the same as the program number (Onnnnn) of the subprogram.
- If the subprogram is not in memory, the file name must be Onnnnn.nc. The file name must contain the O, leading zeros and .nc for the machine to find the subprogram.

252 - Custom Subprogram Search Location

This setting specifies the subprogram search locations when Setting 251 is set to **Setting 252**. To make changes to this setting, highlight Setting 252 and press the **[RIGHT]** cursor. The Setting 252 popup explains how to delete and add search paths and lists existing search paths.

To delete a search path:

1. Highlight the path listed in the Setting 252 popup.
2. Press **[DELETE]**.

If there is more than one path to delete, repeat steps 1 and 2.

To set a new path:

1. Press **[LIST PROGRAM]**.
2. Highlight the directory to add.
3. Press **[F3]**.
4. Select **Setting 252 add** and press **[ENTER]**.

To add another path, repeat steps 1 through 4.

**NOTE:**

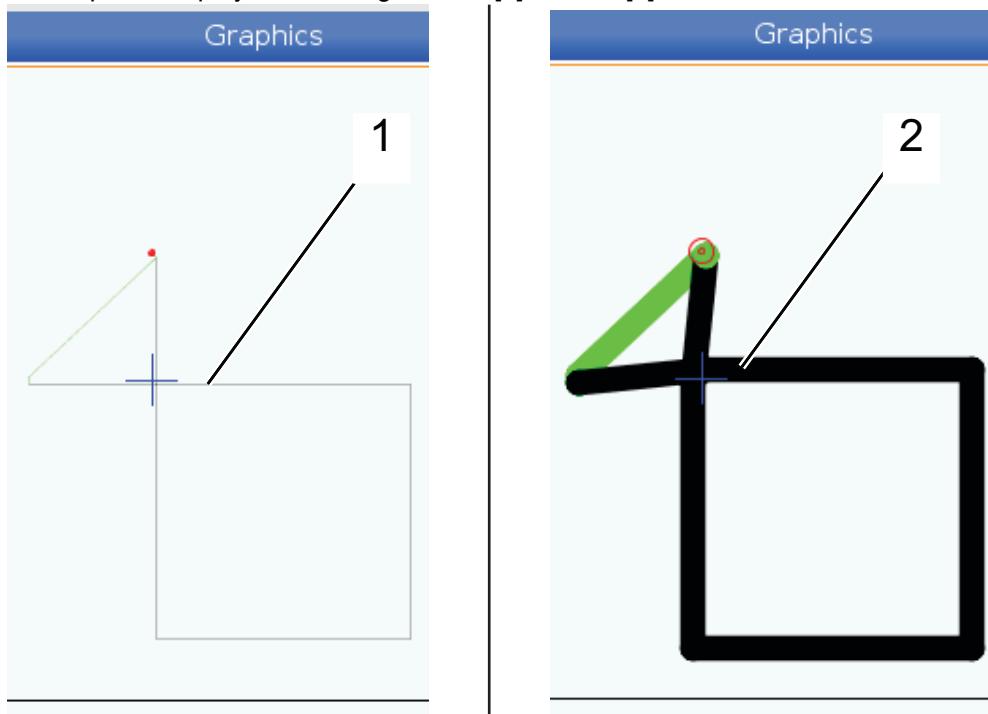
When you use M98:

- The P code (nnnnn) is the same as the program number (Onnnnn) of the subprogram.
- If the subprogram is not in memory, the file name must be Onnnnn.nc. The file name must contain the O, leading zeros and .nc for the machine to find the subprogram.

253 - Default Graphics Tool Width

If this setting is **ON**, Graphics mode uses default tool width (a line) [1]. If this setting is **OFF**, Graphics mode uses the Tool Offset Diameter Geometry specified in the **Tool Offsets** table as the graphics tool width [2].

F9.8: Graphics Display with Setting 253 On [1] and Off [2].



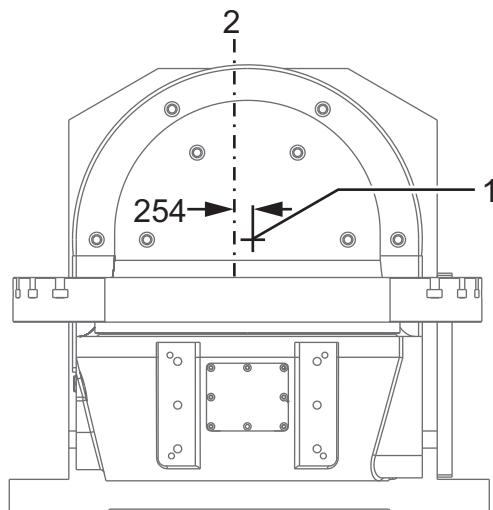
254 - 5-Axis Rotary Center Distance

Setting 254 defines the distance, in inches or millimeters, between the rotary centers of rotation. The default value is 0. The maximum allowed compensation is ± 0.005 in (± 0.1 mm).

When this setting is 0, the control does not use 5-axis rotary center distance compensation.

When this setting has a non-zero value, the control applies 5-axis rotary center distance compensation to the appropriate axes during all rotary motion. This aligns the tool tip with the programmed position when the program invokes **G234**, Tool Center Point Control (TCPC).

F9.9: Setting 254. [1] Tilt Axis Center of Rotation, [2] Rotary Axis Center of Rotation. This illustration is not to scale. Distances are exaggerated for clarity.



NOTE:

When touching off tools or working on the stationary table, make sure the tilt axis is at 0 degrees ($A0^\circ$ or $B0^\circ$).

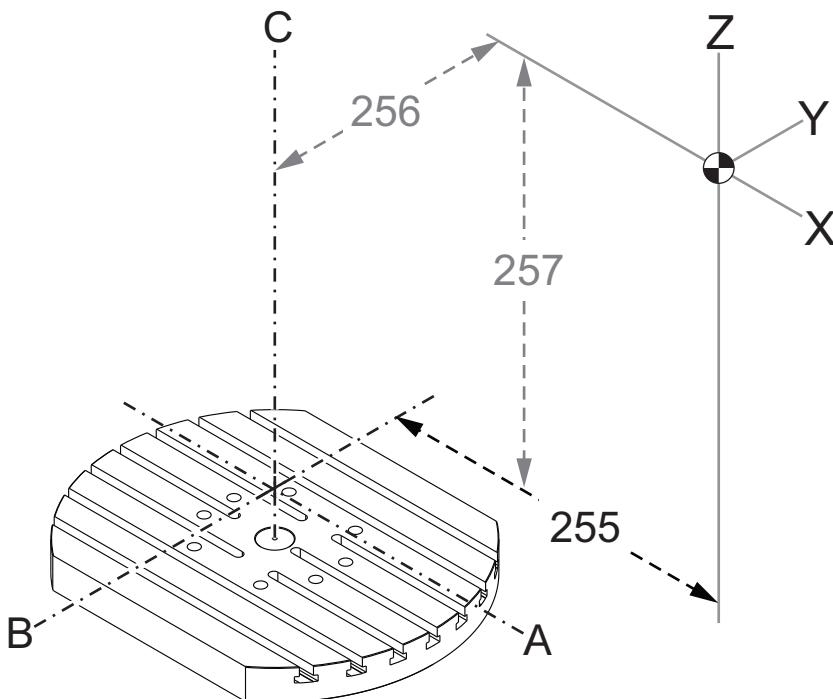
255 - MRZP X Offset

Setting 255 defines the distance, in inches or millimeters, between the

- B tilt axis centerline and the X-Axis home position for a B/C axis UMC, or
- C rotary axis centerline and the X-Axis home position for an A/C axis trunnion.

Use macro value #20255 to read the value of Setting 255.

F9.10: [B] Tilt Axis, [C] Rotary Axis. On a UMC-750 (shown), these axes intersect approximately 2" above the table. [255] Setting 255 is the distance along the X Axis between machine zero and the [B] tilt axis centerline. For [A] Tilt Axis, [C] Rotary Axis on a Trunnion, [255] Setting 255 is the distance along the X Axis between machine zero and the [C] axis centerline. This illustration is not to scale.



NOTE:

When touching off tools or working on the stationary table, make sure the tilt axis is at 0 degrees ($A0^\circ$ or $B0^\circ$).

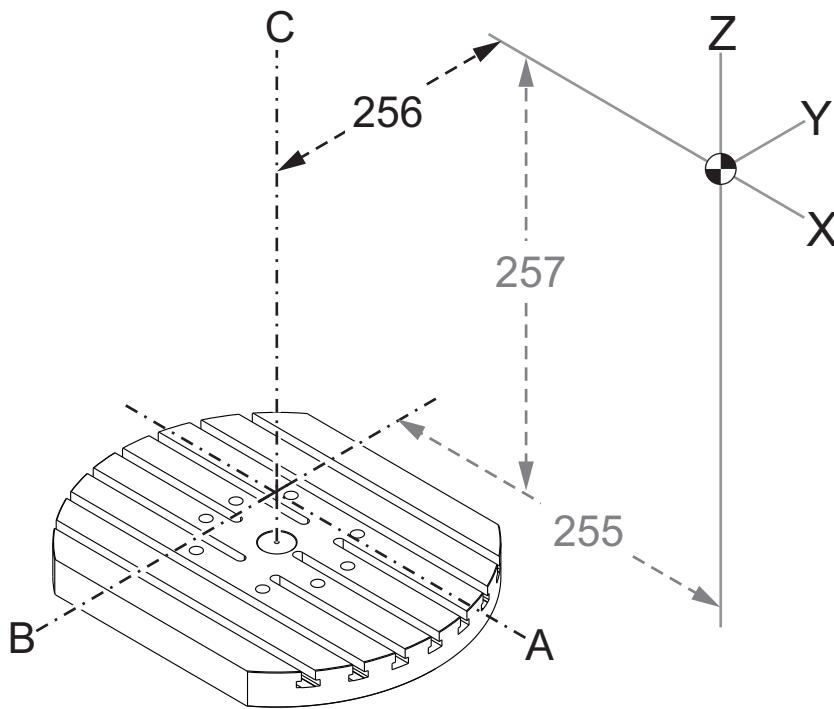
256 - MRZP Y Offset

Setting 256 defines the distance, in inches or millimeters, between the

- C rotary axis centerline and the Y-Axis home position for a B/C axis UMC, or.
- A tilt axis center line and the Y-Axis home position for an A/C axis trunnion.

Use macro value #20256 to read the value of Setting 256.

F9.11: [B] Tilt Axis, [C] Rotary Axis. [256] Setting 256 is the distance along the Y Axis between machine zero and the [C] rotary axis centerline. For [A] Tilt Axis, [C] Rotary Axis on a Trunnion, [256] Setting 256 is the distance along the Y Axis between machine zero and the [A] Tilt Axis centerline. This illustration is not to scale.



NOTE:

When touching off tools or working on the stationary table, make sure the tilt axis is at 0 degrees ($A0^\circ$ or $B0^\circ$).

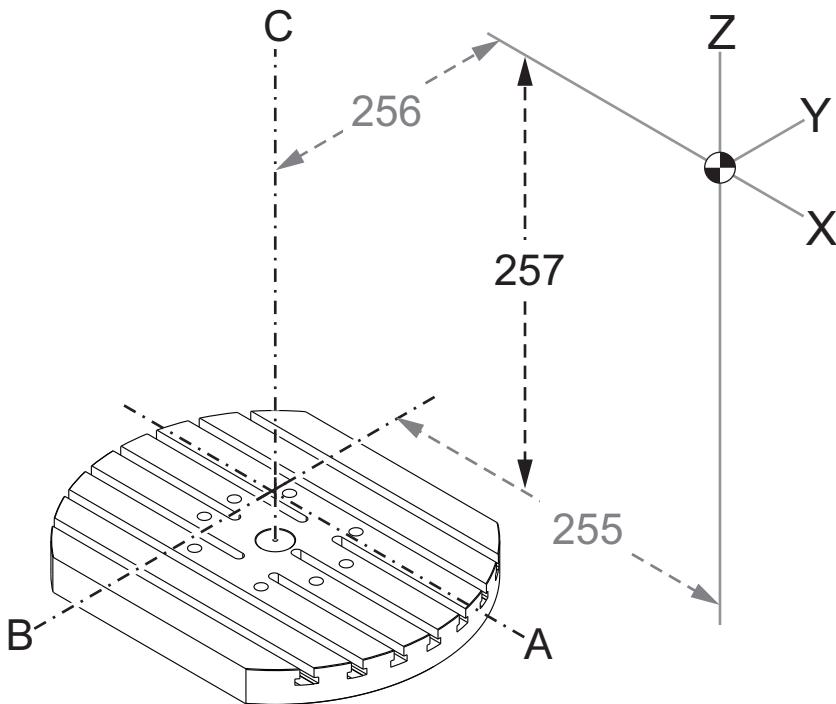
257 - MRZP Z Offset

Setting 257 defines the distance, in inches or millimeters, between the

- B tilt axis and the Z-Axis home position for a B/C axis UMC, or
- A tilt axis and the Z-Axis home position for an A/C axis trunnion

Use macro value #20257 to read the value of Setting 257.

F9.12: [B] Tilt Axis, [C] Rotary Axis. On a UMC-750 (shown), these axes intersect approximately 2" above the table. [257] Setting 257 is the distance along the Z Axis between machine zero and the [B] tilt axis. For [A] Tilt Axis, [C] Rotary Axis on a Trunnion, [257] Setting 257 is the distance along the Z Axis between machine zero and the [A] tilt axis. This illustration is not to scale.



NOTE:

When touching off tools or working on the stationary table, make sure the tilt axis is at 0 degrees ($A0^\circ$ or $B0^\circ$).

261 - DPRNT Store Location

DPRNT is a macro function that lets the machine control communicate with external devices. The Next-Generation Control (NGC) lets you output DPRNT statements over a TCP network, or to a file.

Setting 261 lets you specify where the DPRNT statement output goes:

- **Disabled** - The control does not process DPRNT statements.
- **File** - The control outputs DPRNT statements to the file location specified in setting 262.
- **TCP Port** - The control outputs DPRNT statements to the TCP port number specified in setting 263.

- **Messages** - The control outputs DPRNT statements to the Messages box. Press [ALARMS] and then navigate to the [MESSAGES] tab.

262 - DPRNT Destination File Path

DPRNT is a macro function that lets the machine control communicate with external devices. The Next-Generation Control (NGC) lets you output DPRNT statements to a file, or over a TCP network.

If setting 261 is set to **File**, setting 262 lets you specify the file location where the control sends DPRNT statements.

263 - DPRNT Port

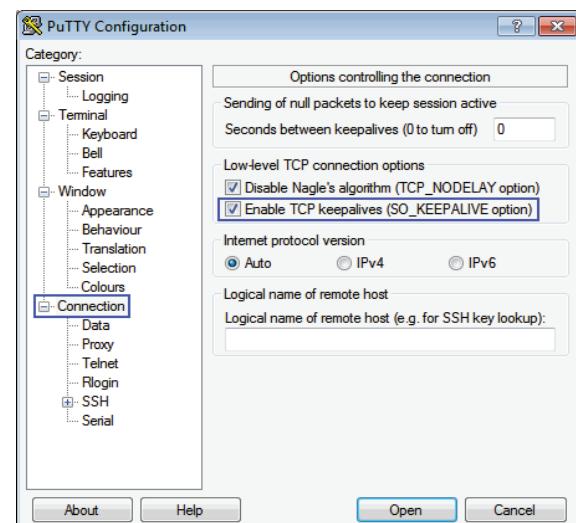
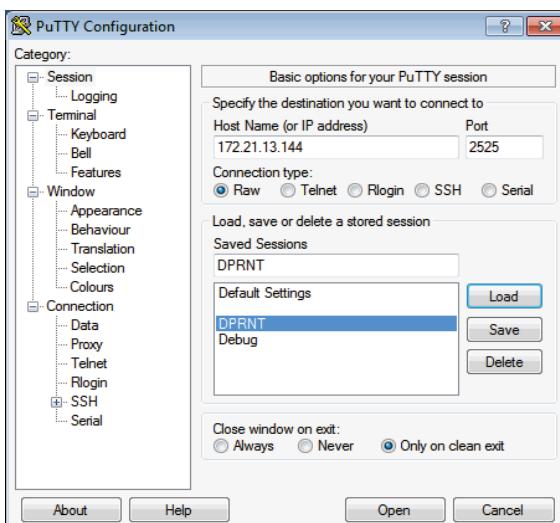
DPRNT is a macro function that lets the machine control communicate with external devices. The Next-Generation Control (NGC) lets you output DPRNT statements over a TCP network.

If setting 261 is set to **TCP Port**, setting 263 lets you specify the TCP port where the control sends DPRNT statements. On the PC, you can use any terminal program that supports TCP.

Use the port value along with the machine's IP address in the terminal program to connect to the machine's DPRNT stream. For example, if you use the terminal program PUTTY:

1. In the basic options section, type the machine's IP address and the port number in Setting 263.
2. Select the Raw or Telnet connection type.
3. Click "Open" to start the connection.

F9.13: PUTTY can save these options for subsequent connections. To keep the connection open, select "Enable TCP keepalives" in the "Connection" options.



To check the connection, type ping in the PUTTY terminal window and press enter. The machine sends a pingret message if the connection is active. You can establish up to (5) simultaneous connections at a time.

264 - Autofeed Step Up

While autofeed is active, this setting defines the percentage amount by which the feedrate increments after tool overload stops.

265 - Autofeed Step Down

When autofeed is active, this setting defines the percentage amount by which the feedrate decrements during a tool overload.

266 - Autofeed Minimum Override

This setting defines the minimum percentage to which autofeed can reduce the feedrate.

267 - Exit Jog Mode after Idle Time

This setting defines the maximum duration, in minutes, that the control remains in jog mode with no axis motion or keyboard activity. After this duration, the control automatically changes to **ZERO RETURN** mode. A value of zero disables this automatic change to **ZERO RETURN** mode from **JOG** mode.

268 - Second Home Position X

This setting defines the X-Axis position for second home, in inches or millimeters. The value is limited by the travel limits for the specific axis.

Press the **[ORIGIN]** button to set this setting to inactive or set the complete group to inactive.

**NOTE:**

This setting is in the User Positions tab under Settings. Refer to the tab description on page 512 for more information.

**CAUTION:**

Incorrectly set user positions can cause machine crashes. Set user positions with caution, especially after you have changed your application in some way (new program, different tools, etc.). Verify and change each axis position separately.

269 - Second Home Position Y

This setting defines the Y-Axis position for second home, in inches or millimeters. The value is limited by the travel limits for the specific axis.

Press the **[ORIGIN]** button to set this setting to inactive or set the complete group to inactive.

**NOTE:**

This setting is in the User Positions tab under Settings. Refer to the tab description on page 512 for more information.

**CAUTION:**

Incorrectly set user positions can cause machine crashes. Set user positions with caution, especially after you have changed your application in some way (new program, different tools, etc.). Verify and change each axis position separately.

270 - Second Home Position Z

This setting defines the Z-Axis position for second home, in inches or millimeters. The value is limited by the travel limits for the specific axis.

Press the **[ORIGIN]** button to set this setting to inactive or set the complete group to inactive.

**NOTE:**

This setting is in the User Positions tab under Settings. Refer to the tab description on page 512 for more information.

**CAUTION:**

Incorrectly set user positions can cause machine crashes. Set user positions with caution, especially after you have changed your application in some way (new program, different tools, etc.). Verify and change each axis position separately.

271 - Second Home Position A

This setting defines the A-Axis position for second home, in degrees. The value is limited by the travel limits for the specific axis.

Press the **[ORIGIN]** button to set this setting to inactive or set the complete group to inactive.



NOTE:

This setting is in the User Positions tab under Settings. Refer to the tab description on page 512 for more information.



CAUTION:

Incorrectly set user positions can cause machine crashes. Set user positions with caution, especially after you have changed your application in some way (new program, different tools, etc.). Verify and change each axis position separately.

272 - Second Home Position B

This setting defines the B-Axis position for second home, in degrees. The value is limited by the travel limits for the specific axis.

Press the [ORIGIN] button to set this setting to inactive or set the complete group to inactive.



NOTE:

This setting is in the User Positions tab under Settings. Refer to the tab description on page 512 for more information.



CAUTION:

Incorrectly set user positions can cause machine crashes. Set user positions with caution, especially after you have changed your application in some way (new program, different tools, etc.). Verify and change each axis position separately.

273 - Second Home Position C

This setting defines the C-Axis position for second home, in degrees. The value is limited by the travel limits for the specific axis.

Press the [ORIGIN] button to set this setting to inactive or set the complete group to inactive.



NOTE:

This setting is in the User Positions tab under Settings. Refer to the tab description on page 512 for more information.

**CAUTION:**

Incorrectly set user positions can cause machine crashes. Set user positions with caution, especially after you have changed your application in some way (new program, different tools, etc.). Verify and change each axis position separately.

276 - Workholding Input Number

This setting specifies the input number to monitor for workholding fixture clamping. If the control receives a spindle start command while this input indicates that the workholding is not clamped, the machine gives an alarm.

277 - Axis Lubrication Interval

This setting defines the interval, in hours, between cycles for the axis lubrication system. The minimum value is 1 hour. The maximum value is between 12 and 24 hours, depending on the machine model.

291 - Main Spindle Speed Limit

This setting defines a top speed for the main spindle. When this setting has a nonzero value, the spindle will never exceed the designated speed.

292 - Door Open Spindle Speed Limit

This setting specifies the maximum spindle speed allowed while the machine door is open.

293 - Tool Change Mid Position X

This setting lets you define a safe position for the X axis at a tool change command, before the axes go to their final tool change positions. Use this position to avoid collisions with fixtures, trunnions, and other potential obstacles. The control uses this position for every tool change, no matter how it is commanded (M06, [NEXT TOOL], etc.)

Press the [ORIGIN] button to set this setting to inactive or set the complete group to inactive.

**NOTE:**

This setting is in the User Positions tab under Settings. Refer to the tab description on page 512 for more information.



CAUTION:

Incorrectly set user positions can cause machine crashes. Set user positions with caution, especially after you have changed your application in some way (new program, different tools, etc.). Verify and change each axis position separately.

294 - Tool Change Mid Position Y

This setting lets you define a safe position for the Y axis at a tool change command, before the axes go to their final tool change positions. Use this position to avoid collisions with fixtures, trunnions, and other potential obstacles. The control uses this position for every tool change, no matter how it is commanded (M06, [NEXT TOOL], etc.)

Press the [ORIGIN] button to set this setting to inactive or set the complete group to inactive.



NOTE:

This setting is in the User Positions tab under Settings. Refer to the tab description on page 512 for more information.



CAUTION:

Incorrectly set user positions can cause machine crashes. Set user positions with caution, especially after you have changed your application in some way (new program, different tools, etc.). Verify and change each axis position separately.

295 - Tool Change Mid Position Z

This setting lets you define a safe position for the Z axis at a tool change command, before the axes go to their final tool change positions. Use this position to avoid collisions with fixtures, trunnions, and other potential obstacles. The control uses this position for every tool change, no matter how it is commanded (M06, [NEXT TOOL], etc.)

Press the [ORIGIN] button to set this setting to inactive or set the complete group to inactive.



NOTE:

This setting is in the User Positions tab under Settings. Refer to the tab description on page 512 for more information.

**CAUTION:**

Incorrectly set user positions can cause machine crashes. Set user positions with caution, especially after you have changed your application in some way (new program, different tools, etc.). Verify and change each axis position separately.

296 - Tool Change Mid Position A

This setting lets you define a safe position for the A axis at a tool change command, before the axes go to their final tool change positions. Use this position to avoid collisions with fixtures, trunnions, and other potential obstacles. The control uses this position for every tool change, no matter how it is commanded (M06, [NEXT TOOL], etc.)

Press the **[ORIGIN]** button to set this setting to inactive or set the complete group to inactive.

**NOTE:**

This setting is in the User Positions tab under Settings. Refer to the tab description on page User Positions for more information.

**CAUTION:**

Incorrectly set user positions can cause machine crashes. Set user positions with caution, especially after you have changed your application in some way (new program, different tools, etc.). Verify and change each axis position separately.

297 - Tool Change Mid Position B

This setting lets you define a safe position for the B axis at a tool change command, before the axes go to their final tool change positions. Use this position to avoid collisions with fixtures, trunnions, and other potential obstacles. The control uses this position for every tool change, no matter how it is commanded (M06, [NEXT TOOL], etc.)

Press the **[ORIGIN]** button to set this setting to inactive or set the complete group to inactive.

**NOTE:**

This setting is in the User Positions tab under Settings. Refer to the tab description on page 512 for more information.

**CAUTION:**

Incorrectly set user positions can cause machine crashes. Set user positions with caution, especially after you have changed your application in some way (new program, different tools, etc.). Verify and change each axis position separately.

298 - Tool Change Mid Position C

This setting lets you define a safe position for the C axis at a tool change command, before the axes go to their final tool change positions. Use this position to avoid collisions with fixtures, trunnions, and other potential obstacles. The control uses this position for every tool change, no matter how it is commanded (M06, [NEXT TOOL], etc.)

Press the [ORIGIN] button to set this setting to inactive or set the complete group to inactive.

**NOTE:**

This setting is in the User Positions tab under Settings. Refer to the tab description on page 512 for more information.

**CAUTION:**

Incorrectly set user positions can cause machine crashes. Set user positions with caution, especially after you have changed your application in some way (new program, different tools, etc.). Verify and change each axis position separately.

300 - MRZP X Offset Master

This setting defines the distance, in inches or mm, between the master rotary axis center and the X-Axis machine zero position. This is similar to Setting 255, except that a value in this setting also specifies that the value refers to the master rotary axis. This setting overrides Setting 255.

Master/Slave axis definition: Typically, when (2) rotary axes control the orientation of a table, one rotary mechanism (for example, a rotary table) sits on top of another rotary mechanism (for example, a tilting trunnion). The rotation mechanism at the bottom comprises the “master” axis (which remains parallel to one of the machine’s linear axes at all times), and the rotation mechanism at the top comprises the “slave” axis (which has a varying orientation to the machine’s axes).

**NOTE:**

When touching off tools or working on the stationary table, make sure the tilt axis is at 0 degrees (A0° or B0°).

301 - MRZP Y Offset Master

This setting defines the distance, in inches or mm, between the master rotary axis center and the Y-Axis machine zero position. This is similar to Setting 256, except that a value in this setting also specifies that the value refers to the master rotary axis. This setting overrides Setting 256.

Master/Slave axis definition: Typically, when (2) rotary axes control the orientation of a table, one rotary mechanism (for example, a rotary table) sits on top of another rotary mechanism (for example, a tilting trunnion). The rotation mechanism at the bottom comprises the “master” axis (which remains parallel to one of the machine’s linear axes at all times), and the rotation mechanism at the top comprises the “slave” axis (which has a varying orientation to the machine’s axes).

**NOTE:**

When touching off tools or working on the stationary table, make sure the tilt axis is at 0 degrees ($A0^\circ$ or $B0^\circ$).

302 - MRZP Z Offset Master

This setting defines the distance, in inches or mm, between the master rotary axis center and the Z-Axis machine zero position. This is similar to Setting 257, except that a value in this setting also specifies that the value refers to the master rotary axis. This setting overrides Setting 257.

Master/Slave axis definition: Typically, when (2) rotary axes control the orientation of a table, one rotary mechanism (for example, a rotary table) sits on top of another rotary mechanism (for example, a tilting trunnion). The rotation mechanism at the bottom comprises the “master” axis (which remains parallel to one of the machine’s linear axes at all times), and the rotation mechanism at the top comprises the “slave” axis (which has a varying orientation to the machine’s axes).

**NOTE:**

When touching off tools or working on the stationary table, make sure the tilt axis is at 0 degrees ($A0^\circ$ or $B0^\circ$).

303 - MRZP X Offset Slave

This setting defines the distance, in inches or mm, between the slave rotary axis center and the X-Axis machine zero position. This is similar to Setting 255, except that a value in this setting also specifies that the value refers to the slave rotary axis. This setting overrides Setting 255.

Master/Slave axis definition: Typically, when (2) rotary axes control the orientation of a table, one rotary mechanism (for example, a rotary table) sits on top of another rotary mechanism (for example, a tilting trunnion).

The rotation mechanism at the bottom comprises the “master” axis (which remains parallel to one of the machine’s linear axes at all times), and the rotation mechanism at the top comprises the “slave” axis (which has a varying orientation to the machine’s axes).



NOTE:

When touching off tools or working on the stationary table, make sure the tilt axis is at 0 degrees ($A0^\circ$ or $B0^\circ$).

304 - MRZP Y Offset Slave

This setting defines the distance, in inches or mm, between the slave rotary axis center and the Y-Axis machine zero position. This is similar to Setting 256, except that a value in this setting also specifies that the value refers to the slave rotary axis. This setting overrides Setting 256.

Master/Slave axis definition: Typically, when (2) rotary axes control the orientation of a table, one rotary mechanism (for example, a rotary table) sits on top of another rotary mechanism (for example, a tilting trunnion). The rotation mechanism at the bottom comprises the “master” axis (which remains parallel to one of the machine’s linear axes at all times), and the rotation mechanism at the top comprises the “slave” axis (which has a varying orientation to the machine’s axes).



NOTE:

When touching off tools or working on the stationary table, make sure the tilt axis is at 0 degrees ($A0^\circ$ or $B0^\circ$).

305 - MRZP Z Offset Slave

This setting defines the distance, in inches or mm, between the slave rotary axis center and the Z-Axis machine zero position. This is similar to Setting 257, except that a value in this setting also specifies that the value refers to the slave rotary axis. This setting overrides Setting 257.

Master/Slave axis definition: Typically, when (2) rotary axes control the orientation of a table, one rotary mechanism (for example, a rotary table) sits on top of another rotary mechanism (for example, a tilting trunnion). The rotation mechanism at the bottom comprises the “master” axis (which remains parallel to one of the machine’s linear axes at all times), and the rotation mechanism at the top comprises the “slave” axis (which has a varying orientation to the machine’s axes).



NOTE:

When touching off tools or working on the stationary table, make sure the tilt axis is at 0 degrees ($A0^\circ$ or $B0^\circ$).

306 - Minimum Chip Clear Time

This setting specifies the minimum amount of time, in seconds, that the spindle remains at "chip clean speed" (the spindle RPM designated in a canned cycle E command). Add time to this setting if your commanded chip clean cycles do not completely remove the chips from the tool.

310 - Min User Travel Limit A

This setting lets you define a custom user travel limit (UTL) position for the A axis.

1. Make sure the work table is clear of any obstructions and clear all of the other user position settings.
2. Highlight the rotary axis travel limit setting and press **[F3]** to move the axis to the mount position. Do not move the axis until the part or fixture is mounted.
3. Mount the part or fixture to the table in the most NEGATIVE position possible for the selected axis.
4. Jog the axis in the POSITIVE direction to the desired travel limit location. Do not re-zero the machine until all UTL's are set.
5. Highlight the max rotary axis travel limit setting and press **[F2]** to set the travel limit. If the tool change offset is not between the Max Rotary UTL and the Min Rotary UTL, a popup will ask for confirmation about resetting the tool change offset for this axis. The minimum travel limit for this axis is calculated to ensure safe zero return and homing.

Press the **[ORIGIN]** button to set this setting to inactive or set the complete group to inactive.

311 - Min User Travel Limit B

This setting lets you define a custom user travel limit (UTL) position for the B axis.

1. Make sure the work table is clear of any obstructions and clear all of the other user position settings.
2. Highlight the rotary axis travel limit setting and press **[F3]** to move the axis to the mount position. Do not move the axis until the part or fixture is mounted.
3. Mount the part or fixture to the table in the most NEGATIVE position possible for the selected axis.
4. Jog the axis in the POSITIVE direction to the desired travel limit location. Do not re-zero the machine until all UTL's are set.
5. Highlight the max rotary axis travel limit setting and press **[F2]** to set the travel limit. If the tool change offset is not between the Max Rotary UTL and the Min Rotary UTL, a popup will ask for confirmation about resetting the tool change offset for this axis. The minimum travel limit for this axis is calculated to ensure safe zero return and homing.

Press the [ORIGIN] button to set this setting to inactive or set the complete group to inactive.

312 - Min User Travel Limit C

This setting lets you define a custom user travel limit (UTL) position for the C axis.

1. Make sure the work table is clear of any obstructions and clear all of the other user position settings.
2. Highlight the rotary axis travel limit setting and press [F3] to move the axis to the mount position. Do not move the axis until the part or fixture is mounted.
3. Mount the part or fixture to the table in the most NEGATIVE position possible for the selected axis.
4. Jog the axis in the POSITIVE direction to the desired travel limit location. Do not re-zero the machine until all UTL's are set.
5. Highlight the max rotary axis travel limit setting and press [F2] to set the travel limit. If the tool change offset is not between the Max Rotary UTL and the Min Rotary UTL, a popup will ask for confirmation about resetting the tool change offset for this axis. The minimum travel limit for this axis is calculated to ensure safe zero return and homing.

Press the [ORIGIN] button to set this setting to inactive or set the complete group to inactive.

313, 314, 315 - Max User Travel Limit X, Y, Z

This setting lets you define a custom travel limit position for the X, Y, and Z axis.

Press the [ORIGIN] button to set this setting to inactive or set the complete group to inactive.



NOTE:

This setting is in the User Positions tab under Settings. Refer to the tab description on page 512 for more information.

316 - Max User Travel Limit A

This setting lets you define a custom user travel limit (UTL) position for the A axis.

1. Make sure the work table is clear of any obstructions and clear all of the other user position settings.
2. Highlight the rotary axis travel limit setting and press [F3] to move the axis to the mount position. Do not move the axis until the part or fixture is mounted.
3. Mount the part or fixture to the table in the most POSITIVE position possible for the selected axis.

4. Jog the axis in the POSITIVE direction to the desired travel limit location. Do not re-zero the machine until all UTL's are set.
5. Highlight the max rotary axis travel limit setting and press [F2] to set the travel limit. If the tool change offset is not between the Max Rotary UTL and the Min Rotary UTL, a popup will ask for confirmation about resetting the tool change offset for this axis. The minimum travel limit for this axis is calculated to ensure safe zero return and homing.

Press the [ORIGIN] button to set this setting to inactive or set the complete group to inactive.

317 - Max User Travel Limit B

This setting lets you define a custom user travel limit (UTL) position for the B axis.

1. Make sure the work table is clear of any obstructions and clear all of the other user position settings.
2. Highlight the rotary axis travel limit setting and press [F3] to move the axis to the mount position. Do not move the axis until the part or fixture is mounted.
3. Mount the part or fixture to the table in the most NEGATIVE position possible for the selected axis.
4. Jog the axis in the POSITIVE direction to the desired travel limit location. Do not re-zero the machine until all UTL's are set.
5. Highlight the max rotary axis travel limit setting and press [F2] to set the travel limit. If the tool change offset is not between the Max Rotary UTL and the Min Rotary UTL, a popup will ask for confirmation about resetting the tool change offset for this axis. The minimum travel limit for this axis is calculated to ensure safe zero return and homing.

Press the [ORIGIN] button to set this setting to inactive or set the complete group to inactive.

318 - Max User Travel Limit C

This setting lets you define a custom user travel limit (UTL) position for the C axis.

1. Make sure the work table is clear of any obstructions and clear all of the other user position settings.
2. Highlight the rotary axis travel limit setting and press [F3] to move the axis to the mount position. Do not move the axis until the part or fixture is mounted.
3. Mount the part or fixture to the table in the most NEGATIVE position possible for the selected axis.
4. Jog the axis in the POSITIVE direction to the desired travel limit location. Do not re-zero the machine until all UTL's are set.
5. Highlight the max rotary axis travel limit setting and press [F2] to set the travel limit. If the tool change offset is not between the Max Rotary UTL and the Min Rotary UTL,

a popup will ask for confirmation about resetting the tool change offset for this axis. The minimum travel limit for this axis is calculated to ensure safe zero return and homing.

Press the **[ORIGIN]** button to set this setting to inactive or set the complete group to inactive.

323 - Disable Notch Filter

When this setting is **on**, the notch filter values are set to zero. When this setting is **off**, it uses the default values of the machine as set defined by parameters. Turning this setting **On** will improve circular accuracy and turning **Off** will improve surface finish.



NOTE:

You must cycle power for this setting to take effect.

325 - Manual Mode Enabled

Turning this setting **on** allows the axes to be jogged without zero returning the machine (finding machine home).

The jog limits imposed by setting 53 Jog W/O Zero Return will not apply. The jog rate will be defined by the eWheel switch or the jog rate buttons (if the eWheel is not connected).

With this setting **on** you can perform tool changes using the **[ATC FWD]** or **[ATC REV]** buttons.

When turning this setting **off** the machine will operate as normal, and will require to be zero return.

330 - MultiBoot Selection Time out

This is a simulator only setting. When a simulator is powered on, it displays a screen from where different simulator models can be chosen. This setting sets how long that screen is shown. If the user does nothing before the time expires, the software will load the last active simulator configuration.

335 - Linear Rapid Mode

This setting can be set to one of three modes. The description of these modes is as follows:

NONE The individual axis rapid to their endpoints independently of each other.

LINEAR (XYZ) The XYZ axes, when commanded to rapid, move linearly through 3D space. All other axis rapid with independent speeds/accelerations.

LINEAR + ROTARY Axes X/Y/Z/A/B/C reach their endpoints at the same time. Rotary axis may be slowed down compared to **LINEAR XYZ**.

**NOTE:**

All modes cause a program to run in the same amount of time (no increase or decrease in execution time).

356 - Beeper ON/OFF

This setting allows the user to turn ON and OFF the beeper located on the control pendant. Setting a value of 0 will turn OFF the beeper. Setting a value of 1 will turn ON the beeper.

**NOTE:**

This setting will only affect the pendant beeper, not any pallet change or other beeper.

357 - Warmup Compensation Cycle Start Idle Time

This setting defines an appropriate idle time, in hours, for warmup compensation to be restarted. When a machine has been idle longer than the amount of time in this setting, a **[CYCLE START]** will ask the user if he wishes to apply warmup compensation.

If the user answers with **[Y]** or **[ENTER]**, warmup compensation is applied a new, just as if the machine was powered up and **[CYCLE START]** commences. An **[N]** answer will continue cycle start with no warmup compensation. The next opportunity to apply warmup compensation will be after the setting 357 period has elapsed.

369 - PulseJet Injection Cycle Time

This setting works in conjunction with M161 code, it defines the PulseJet oil pulse cycle time.

Refer to “M161 Pulse Jet Continuous Mode” on page 434 for more information.

370 - PulseJet Single Squirt Count

This setting works in conjunction with M162 and M163, it defines the PulseJet squirt count.

Refer to “M162 PulseJet Single Event Mode” on page 434 and “M163 PulseJet Modal Mode” on page 435 for more information.

372 - Parts Loader Type

This setting turns on the Automatic Parts Loader (APL) in **[CURRENT COMMANDS]** under the Devices tab. Use this page to set up the APL.


DANGER:

If you turn this setting [OFF], the AW - axis will drop if not supported or disconnected. Do one of the following:

- Jog to a safe location
- Unplug all APL cables BEFORE releasing E-STOP
- Unplug AW brake BEFORE releasing E-STOP

375 - APL Gripper Type

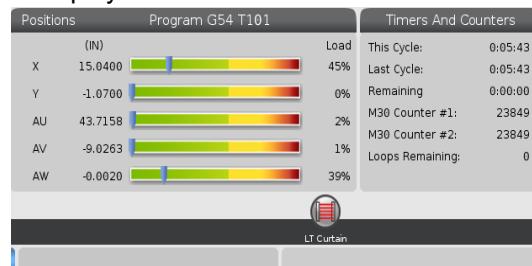
This setting chooses the type of gripper attached to the Automatic Parts Loader (APL).

APL Gripper has the functionality of gripping raw and finished parts on an outer diameter or inner diameter, in addition to being able to swap between them.

376 - Light Curtain Enable

This setting enables the Light Curtain. When the Light Curtain is enabled, it will prevent APL motion if it detects something in an area too close to the APL axes.

If the light curtain beam is obstructed the machine will go into a Light Curtain Hold condition; the CNC program will continue to run and the machine's spindle and axes will continue to move but the AU, AV and AW axes will not move. The machine will remain in Light Curtain Hold until the light curtain beam is unobstructed and the Cycle Start button is pressed.

F9.14: Light curtain Icon Display


When the light curtain beam is obstructed the machine will go into a Light Curtain Hold condition and the Light Curtain icon will appear on the screen. The icon will disappear when the beam is no longer obstructed.

**NOTE:**

You can operate the machine in standalone mode with the light curtain disabled. But the light curtain must be enabled in order to run the APL.

377 - Negative Work Offset

This setting selects the use of work offsets in the negative direction.

Set this setting to On to use negative work offsets to move the axis away from the home position. If set to OFF, then you must use positive work offsets to move the axes away from home position.

378, 379, 380 - Safe Zone Calibrated Geometry Reference Point X, Y, Z

These settings let you define the Safe Zone Calibrated Geometry Reference Point for X, Y and Z axis.

381 - Enable Touchscreen

This setting enables the touchscreen feature on machines built with a touchscreen. If the machine does not have a touch screen a alarm message will be generated at power on.

382 - Disable Pallet Changer

This setting enables/disables the pallet changer on the machine. Machine requires to be in **[E-STOP]** before you can change this setting, after the change you need to cycle the power before the setting can take effect.

If machine has Automatic Pallet Changer and Pallet Pool, setting options are:

- **None** - Nothing is disabled.
- **Pallet Pool:** - Disables only the Pallet Pool.
- **All** - Disables the Automatic Pallet Changer and Pallet Pool.

If machine only has an Automatic Pallet Changer, setting options are:

- **None** - Nothing is disabled.
- **All** - Disables the Automatic Pallet Changer.

If machine only has a Pallet Pool, setting options are:

- **None** - Nothing is disabled.
- **Pallet Pool:** - Disables the Pallet Pool.

383 - Table Row Size

This settings allows you to resize the rows, when using the touchscreen feature.

389 - Vise Unclamped Safety Check

When this setting is set to ON the user is not allowed to press [CYCLE START] with any workhold device that is unclamped.

Starting in software version 100.21.000.1200 or higher the user may command a *Q1* code, which ignores the safety check for a specified vise.

For example:

M71 P2 Q1

; This command unclamps Vise 2 and sets the ignore safety check to true. This allows the user to run the spindle while the specified vise is unclamped.



CAUTION:

*When this setting is turned OFF, any *Q1* code commanded with an M71 is ignored. Please use the *Q1* code with caution.*

396 - Enable / Disable Virtual Keyboard

This settings allows you to use a virtual keyboard on the screen, when using the touchscreen feature.

397 - Press and Hold Delay

This settings allows you to set the hold delay before the a pop shows up.

398 - Header Height

This setting adjust the header height for the pop-ups and display boxes.

399 - Tab Height

This setting adjust the height of the tabs.

400 - Pallet Ready Beep Type

This setting adjusts the length of beeps when the automatic pallet changer is in motion or when a completed pallet was dropped off at the load station.

There are three modes:

- Normal: Machines beeps normal.
- Short: Beeps three times and stops.
- Off: No beeps.

403 - Change Popup Button Size

This setting allows you to resize the popup buttons, when using the touchscreen feature.

408 - Exclude Tool From Safe Zone

This setting excludes the tool from the Safe Zone Calculation. Set this setting to On to machine the table for workholding.

**NOTE:**

When machining close to the edge of the table on a GM-2-5X machine, turn setting 408 On and rotate the C-axis 180 degrees away from the table edge.

**NOTE:**

This setting will revert back to Off after power cycle.

409 - Default Coolant Pressure

Some machine models are equipped with a variable frequency drive that allows the coolant pump to operate in different coolant pressures. This setting specifies the default coolant pressure when M08 is commanded. The choices are:

- 0 - Low Pressure
- 1 - Normal Pressure
- 2 - High Pressure

**NOTE:**

A P code can be used with M08 to specify the desired coolant pressure. Refer to the M08 Coolant On section for more information.

416 - Media Destination

This setting allows the user to view media files on a external monitor via the HDMI output on the Main Control PCB.

When this setting is to 0: Built-In Monitor the machine will behave as normal.

When this setting is set to 1: External Monitor a message “See External Monitor” will be displayed on the Media tab.

420 - ATC Button Behavior

This setting determines the behavior of the [ATC FWD] or [ATC REV] buttons on the control. When this setting is set to Conventional the [ATC FWD] or [ATC REV] buttons behave as normal.

When set to Enhanced mode the [ATC FWD] key will execute a tool change to the next higher tool number into the Pocket Tool Table. It will loop around to the lowest tool number if there is no higher tool number. When in enhanced mode the [ATC REV] key will execute a tool change to the next lower tool number into the Pocket Tool Table. It will loop around to the highest tool number if there is no lower tool number.

421 - General Orient Angle

On Mill/Turn machines when a user is using turning tools, the Live Tool Spindle may need to be rotated to a specific direction and the brake clamped after a tool change to properly use the tool. This setting is used on Mill/ Turn machines to save a general orientation offset that will be used by any turning tool.

422 - Lock Graphics Plane

This setting allows you to lock the graphic plane into one specific view. Turn this setting off if you want the control to switch planes in graphics automatically.

423 - Help Text Icon Size

This setting is used to change the size of the text displayed when you click on one of the icons.



NOTE:

This setting is only for machines equipped with touchscreen.

424 - Mist Extractor Condenser Time Out

This setting defines the time in seconds for the mist condenser to shut off after an M00, M01, M02, M30, or End of Program.

9.2 Network Connection

You can use a computer network through a wired connection (Ethernet) or a wireless connection (WiFi) to transfer program files to and from your Haas machine, and to let multiple machines access files from a central network location. You can also set up Net Share to quickly and easily share programs between the machines in your shop and the computers on your network.

To access the Network page:

1. Press **[SETTING]**.
2. Select the **Network** tab in the tabbed menu.
3. Select the tab for the network settings (**Wired Connection**, **Wireless Connection**, or **Net Share**) that you want to set up.

F9.15: Wired Network Settings Page Example

Wired Network Information			
Host Name	HAASMachine	DHCP Server	*
Domain		IP Address	*
DNS Server	*	Subnet Mask	*
Mac Address		Gateway	
DHCP Enabled	OFF	Status	UP

NAME	>	VALUE
Wired Network Enabled	>	On
Obtain Address Automatically	>	Off
IP Address		
Subnet Mask		
Default Gateway		
DNS Server		

Warning: Changes will not be saved if page is left without pressing [F4]!

F3 Discard Changes
 F4 Apply Changes



NOTE:

Settings with a > character in the second column have preset values that you select from. Press the [RIGHT] cursor arrow key to see the list of options. Use the [UP] and [DOWN] cursor arrow keys to choose an option, and then press [ENTER] to confirm your choice.

9.2.1 Network Icon Guide

The control screen shows icons to quickly give you information about the machine network status.

Icon	Meaning
	The machine is connected to the Internet via a wired network with an Ethernet cable.
	The machine is connected to the Internet via a wireless network and has 70 - 100% signal strength.
	The machine is connected to the Internet via a wireless network and has 30 - 70% signal strength.
	The machine is connected to the Internet via a wireless network and has 1 - 30% signal strength.
	The machine was connected to the Internet via a wireless network and is not receiving any data packets.

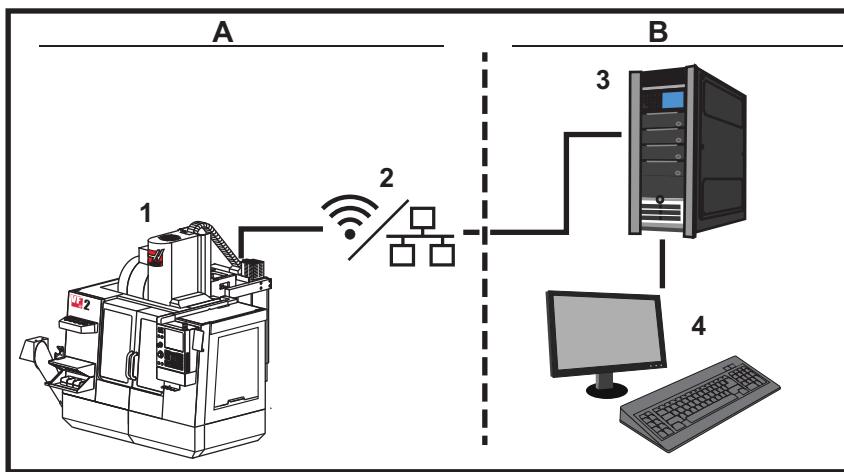
Icon	Meaning
	The machine is successfully registered with HaasConnect and is communicating with the server.
	The machine had previously registered with HaasConnect and has a problem connecting to the server.
	The machine is connected to a remote Netshare.

9.2.2 Network Connection Terms and Responsibilities

Networks and operating systems are different from company to company. When your HFO Service Technician installs your machine, they can attempt to connect it to your network with your information, and they can troubleshoot connection problems with the machine itself. If the problem is with your network, you need a qualified IT service provider to assist you, at your expense.

If you call your HFO for help with network problems, remember that the technician can help only as far as the machine software and networking hardware.

F9.16: Network Responsibility Diagram: [A] Haas Responsibility, [B] Your Responsibility, [1] Haas Machine, [2] Haas Machine Network Hardware, [3] Your Server, [4] Your computer(s).



9.2.3 Wired Connection Setup

Before you begin, ask your network administrator if your network has a Dynamic Host Configuration Protocol (DHCP) server. If it does not have a DHCP server, collect this information:

- The IP address that your machine will use on the network
 - The Subnet Mask address
 - The Default Gateway address
 - The DNS Server name
1. Connect an active Ethernet cable to the Ethernet port on your machine.
 2. Select the **Wired Connection** tab in the **Network** tabbed menu.
 3. Change the **Wired Network Enabled** setting to **ON**.
 4. If your network has a DHCP server, you can let the network assign an IP address automatically. Change the **Obtain Address Automatically** setting to **ON**, and then press **[F4]** to complete the connection. If your network does not have a DHCP server, go to the next step.
 5. Type the machine's **IP Address**, the **Subnet Mask** address, the **Default Gateway** address, and the **DNS Server** name into their respective fields.
 6. Press **[F4]** to complete the connection, or press **[F3]** to discard your changes.

After the machine successfully connects to the network, the **Status** indicator in the **Wired Network Information** box changes to **UP**.

9.2.4 Wired Network Settings

Wired Network Enabled - This setting activates and deactivates wired networking.

Obtain Address Automatically - Lets the machine retrieve an IP address and other network information from the network's Dynamic Host Configuration Protocol (DHCP) server. You can use this option only if your network has a DHCP server.

IP Address - The machine's static TCP/IP address on a network without a DHCP server. Your network administrator assigns this address to your machine.

Subnet Mask - Your network administrator assigns the subnet mask value for machines with a static TCP/IP address.

Default Gateway - An address to gain access to your network through routers. Your network administrator assigns this address.

DNS Server - The name of the Domain Name Server or DHCP server on the network.



NOTE:

The address format for Subnet Mask, Gateway, and DNS is XXX.XXX.XXX.XXX. Do not end the address with a period. Do not use negative numbers. 255.255.255.255 is the highest possible address.

9.2.5 Wireless Connection Setup

This option lets your machine connect to a 2.4 GHz, 802.11b/g/n wireless network. 5 GHz is not supported.

Wireless network setup uses a wizard to scan for available networks and then set up the connection with your network information.

Before you begin, ask your network administrator if your network has a Dynamic Host Configuration Protocol (DHCP) server. If it does not have a DHCP server, collect this information:

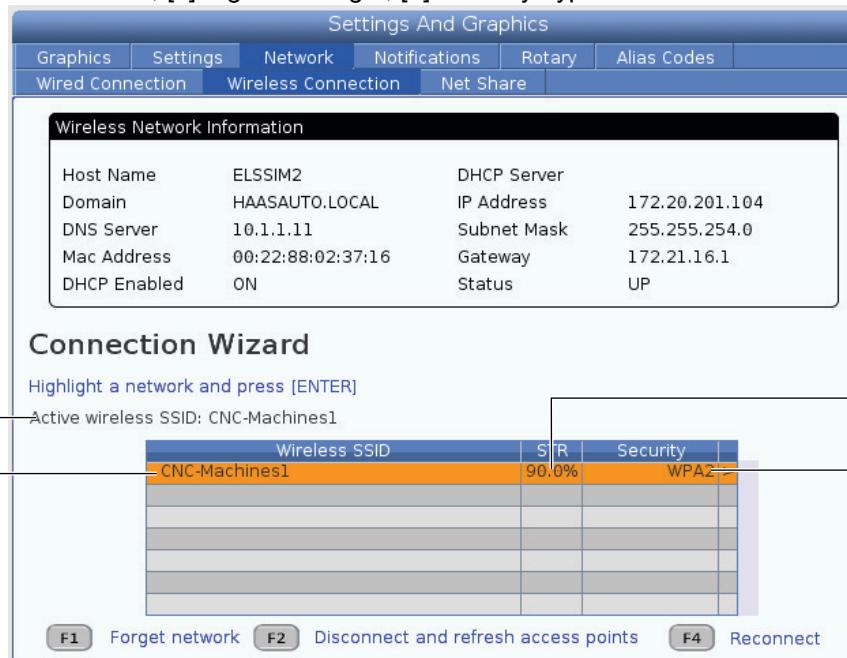
- The IP address that your machine will use on the network
- The Subnet Mask address
- The Default Gateway address
- The DNS Server name

You also need this information:

- The SSID for your wireless network
 - The password to connect to your secured wireless network
1. Select the **Wireless Connection** tab in the **Network** tabbed menu.
 2. Press **[F2]** to scan for available networks.

The Connection Wizard displays a list of available networks, with their signal strengths and security types. The control supports 64/128 WEP, WPA, WPA2, TKIP, and AES security types.

- F9.17:** Connection Wizard List Display. [1] Current Active Network Connection (if any), [2] Network SSID, [3] Signal Strength, [4] Security Type.



3. Use the cursor arrow keys to highlight the network you want to connect to.
 4. Press **[ENTER]**.

The network settings table appears.

- F9.18:** Network Settings Table. [1] Password Field, [2] DHCP Enable / Disable. Further options appear when you turn the DHCP Setting OFF.



5. Type the access point password in the **Password** field.

**NOTE:**

If you need special characters such as underscores (_) or carets (^) for the password, press [F2] and use the menu to select the special character you need.

6. If your network does not have a DHCP server, change the **DHCP Enabled** setting to **OFF** and type the IP Address, Subnet Mask, Default Gateway, and DNS Server Address into their respective fields.
7. Press **[F4]** to complete the connection, or press **[F3]** to discard your changes.

After the machine successfully connects to the network, the **Status** indicator in the **Wired Network Information** box changes to **UP**. The machine will also automatically connect to this network when it is available, unless you press F1 and confirm to “forget” the network.

The possible status indicators are:

- **UP** - The machine has an active connection to a wireless network.
- **DOWN** - The machine does not have an active connection to a wireless network.
- **DORMANT** - The machine is waiting for an external action (typically, waiting for authentication with the wireless access point).
- **UNKNOWN** - The machine cannot determine the connection status. A bad link or incorrect network configuration can cause this. You may also see this status while the machine transitions between statuses.

Wireless Network Function Keys

Key	Description
 F1	Forget network - Highlight a network and press [F1] to remove all connection information and prevent automatic reconnection to this network.
 F2	Scan for network and Disconnect and refresh access points - In the network selection table, press [F2] to disconnect from the current network and scan for available networks. Special Symbols - In the wireless network settings table, use [F2] to access special characters, such as carets or underscores, for password entry.
 F4	Reconnect - Connect again to a network the machine was previously connected to. Apply Changes - After you make changes to settings for a particular network, press [F4] to save the changes and connect to the network.

9.2.6 Wireless Network Settings

Wireless Network Enabled - This setting activates and deactivates wireless networking.

Obtain Address Automatically - Lets the machine retrieve an IP address and other network information from the network's Dynamic Host Configuration Protocol (DHCP) server. You can use this option only if your network has a DHCP server.

IP Address - The machine's static TCP/IP address on a network without a DHCP server. Your network administrator assigns this address to your machine.

Subnet Mask - Your network administrator assigns the subnet mask value for machines with a static TCP/IP address.

Default Gateway - An address to gain access to your network through routers. Your network administrator assigns this address.

DNS Server - The name of the Domain Name Server or DHCP server on the network.



NOTE:

The address format for Subnet Mask, Gateway, and DNS is XXX.XXX.XXX.XXX. Do not end the address with a period. Do not use negative numbers. 255.255.255.255 is the highest possible address.

Wireless SSID - The name of the wireless access point. You can enter this manually, or you can press the LEFT or RIGHT cursor arrow keys to select from a list of available networks. If your network does not broadcast its SSID, you must enter this manually.

Wireless Security - The security mode that your wireless access point uses.

Password - The password for the wireless access point.

9.2.7 Net Share Settings

Net Share lets you connect remote computers to the machine control over the network, to transfer files to and from the machine's User Data directory. These are the settings you need to adjust to set up Net Share. Your network administrator can give you the correct values to use. You must enable remote sharing, local sharing, or both to use Net Share.

After you change these settings to the correct values, press **[F4]** to begin Net Share.



NOTE:

If you need special characters such as underscores (_) or carets (^) for these settings, refer to page 71 for instructions.

CNC Network Name - The name of the machine on the network. The default value is **HAASMachine**, but you must change this so that each machine on the network has a unique name.

Domain / Workgroup Name - The name of the domain or workgroup the machine belongs to.

Remote Net Share Enabled - When this is **ON**, the machine shows the contents of the shared network folder in the **Network** tab in the Device Manager.

Remote Server Name - The remote network name or IP address of the computer that has the share folder.

Remote Share Path - The name and location of the shared remote network folder.



NOTE:

Do not use spaces in the shared folder name.

Remote User Name - The name to use to log in to the remote server or domain. User names are case-sensitive and cannot contain spaces.

Remote Password - The password to use to log in to the remote server. Passwords are case-sensitive.

Remote Share Connection Retry - This setting adjust the Remote NetShare connection retry behavior.



NOTE:

The higher levels of this setting can cause intermittent user interface to freeze. If not using Wi-Fi connection all the time always set this setting to Relaxed.

Local Net Share Enabled - When this is **ON**, the machine allows access to the **User Data** directory to computers on the network (password required).

Local User Name - Displays the user name to log into the control from a remote computer. The default value is **haas**; you cannot change this.

Local Password - The password for the user account on the machine.



NOTE:

You need the local user name and password to access the machine from an outside network.

Net Share Example

In this example, you have established a net share connection with the **Local Net Share Enabled** setting turned **on**. You want to view the contents of the machine's **User Data** folder on a networked PC.



NOTE:

This example uses a Windows 7 PC; your configuration may vary. Ask your network administrator for help if you cannot establish a connection.

1. On the PC, click the START menu and select the RUN command. You can also hold the Windows key and press R.
2. At the Run prompt, type (2) backslashes (\) and then the machine's IP address or CNC Network Name.
3. Click OK or press Enter.
4. Type the machine's **Local User Name** (haas) and **Local Password** in the appropriate fields, and then click OK or press Enter.
5. A window appears on the PC with the machine's **User Data** folder displayed. You can interact with the folder as you would with any other Windows folder.



NOTE:

If you use the machine's CNC Network Name instead of the IP address, you may need to type a backslash before the User Name (\haas). If you cannot change the username in the Windows prompt, select the "Use another account" option first.

9.2.8

Haas Drop

The HaasDrop application is used for sending files from a iOS or Android device to the control (NGC) on a Haas Machine.

The procedure is located on the website click on the following link: [Haas Drop - Help](#)

You can also scan the code below with your mobile device to go directly to the procedure



9.2.9 Haas Connect

HaasConnect is a web-based application that lets you monitor your shop with a web browser or mobile device. To use HaasConnect, you set up an account at myhaascnc.com, add users and machines, and designate the alerts you want to receive. For more information about HaasConnect, go to www.haascnc.com or scan the QR code below with your mobile device.



9.2.10 Remote Display View

This procedure tells you how to view the machine display on a computer. The machine must be connected to a network with an Ethernet cable or with a wireless connection.

Refer to the Networking Connection section on page **497** for information on how to connect your machine to a network.



NOTE:

You must download the VNC Viewer to your computer. Go to www.realvnc.com to download the free VNC Viewer.

1. Push the **[SETTING]** button.
2. Navigate to the Wired Connection or Wireless Connection tab in the Network tab.
3. Write down the IP Address for your machine.

4. Remote Display Tab

**NOTE:**

The Remote Display tab is available in software version 100.18.000.1020 or higher.

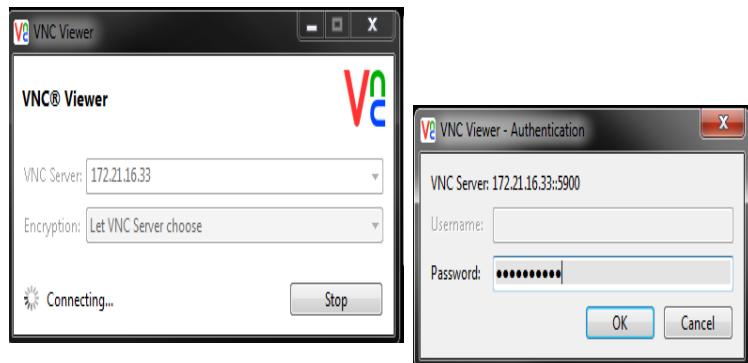
5. Navigate to the Remote Display tab in the Network tab.
6. Turn on the Remote Display.
7. Set the Remote Display Password.

**NOTE:**

The Remote Display feature requires a strong password, follow the guide lines on the screen.

- Press [F4] to apply settings.
8. Open the VNC Viewer application on your computer.

9. VNC Software Screen



Enter your IP Address in VNC Server. Select **Connect**.

10. At the login box enter the password you entered at the Haas control.
11. Select **OK**.
12. The machine display shows on your computer screen

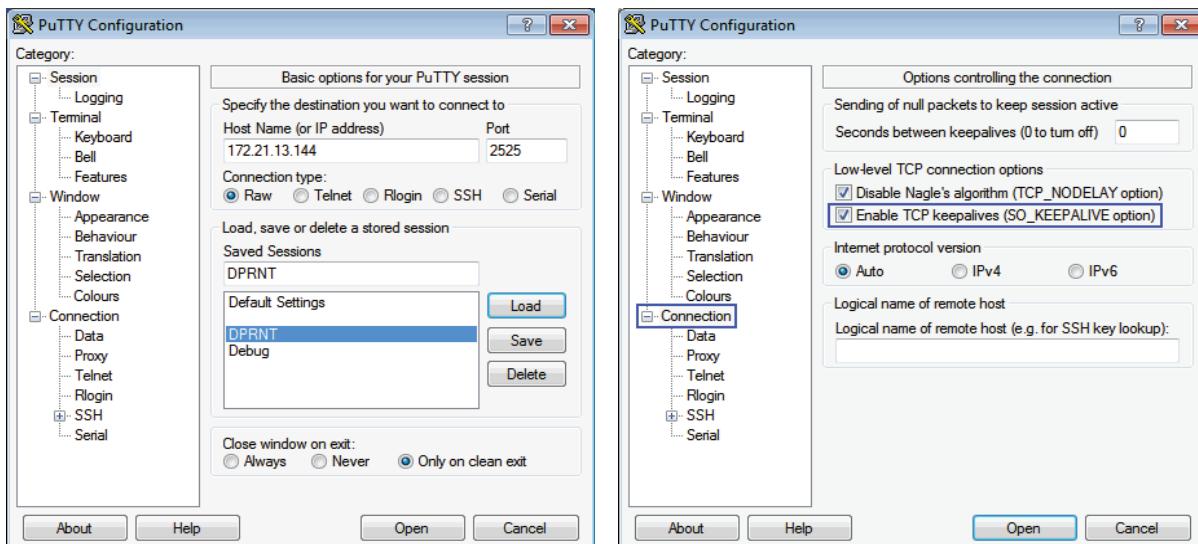
9.2.11 Machine Data Collection

Machine Data Collection (MDC) lets you use Q and E commands to extract data from the control through the Ethernet port or the Wireless Networking option. Setting 143 both enables the feature and specifies the data port that the control uses to communicate. MDC is a software-based feature that requires an additional computer to request, interpret, and store data from the control. The remote computer can also set certain Macro variables.

The Haas control uses a TCP server to communicate over networks. On the remote computer, you can use any terminal program that supports TCP; examples in this manual use PuTTY. Up to (2) simultaneous connections are allowed. Output requested by one connection is sent to all connections.

1. In the basic options section, type the machine's IP address and the port number in Setting 143. Setting 143 must have a nonzero value to use MDC.
2. Select the Raw or Telnet connection type.
3. Click "Open" to start the connection.

F9.19: PuTTY can save these options for subsequent connections. To keep the connection open, select "Enable TCP keepalives" in the "Connection" options.



To check the connection, type ?Q100 in the PuTTY terminal window. If the connection is active, the machine control responds with *SERIAL NUMBER, XXXXXX*, where *XXXXXX* is the machine's actual serial number.

Data Collection Queries and Commands

The control responds to a Q command only when Setting 143 has a nonzero value.

MDC Queries

These commands are available:

T9.1: MDC Queries

Command	Definition	Example
Q100	Machine Serial Number	>Q100 SERIAL NUMBER, 3093228
Q101	Control Software Version	>Q101 SOFTWARE, VER 100.16.000.1041
Q102	Machine Model Number	>Q102 MODEL, VF2D
Q104	Mode (LIST PROG, MDI, etc.)	>Q104 MODE, (MEM)
Q200	Tool Changes (total)	>Q200 TOOL CHANGES, 23

Command	Definition	Example
Q201	Tool Number in use	>Q201 USING TOOL, 1
Q300	Power-on Time (total)	>Q300 P.O. TIME, 00027:50:59
Q301	Motion Time (total)	>Q301 C.S. TIME, 00003:02:57
Q303	Last Cycle Time	>Q303 LAST CYCLE, 000:00:00
Q304	Previous Cycle Time	>Q304 PREV CYCLE, 000:00:00
Q402	M30 Parts Counter #1 (resettable at control)	>Q402 M30 #1, 553
Q403	M30 Parts Counter #2 (resettable at control)	>Q403 M30 #2, 553 STATUS, BUSY (if in cycle)
Q500	Three-in-one (PROGRAM, Oxxxxx, STATUS, PARTS, xxxx)	>PROGRAM, O00110, IDLE, PARTS, 4523
Q600	Macro or system variable	>Q600 801 MACRO, 801, 333.339996

You can request the contents of any macro or system variable with the **Q600** command; for example, **Q600 xxxx**. This shows the contents of macro variable **xxxx** on the remote computer.

Query Format

The correct query format is **?Q###**, where **###** is the query number, terminated with a new line.

Response Format

Responses from the control begin with **>** and end with **/r/n**. Successful queries return the name of the query, then the requested information, separated by commas. For example, a query of **?Q102** returns **MODEL, XXX**, where **XXX** is the machine model. The comma lets you treat the output as comma-separated variable (CSV) data.

An unrecognized command returns a question mark followed by the unrecognized command; for example, **?Q105** returns **?, ?Q105**.

E Commands (Write to Variable)

You can use an E command to write to macro variables **#1-33, 100-199, 500-699** (note that variables **#550-580** are unavailable if the mill has a probing system), **800-999** and **#2001** through **#2800**. For example, **Exxxxx yyyy.yyyy** where **xxxxx** is the macro variable and **yyyy.yyyy** is the new value.

**NOTE:**

When you write to a global variable, make sure that no other programs on the machine use that variable.

9.3 User Positions

This tab collects settings that control user-defined positions such as second home, tool change mid-positions, spindle center line, tailstock and travel limits. Refer to the Settings section of this manual for more information about these position settings.

F9.20: User Positions Tab

Settings

Settings Network Rotary **User Positions** Alias Codes

Search (TEXT) [F1], or [F1] to clear.

Group	
Second Home Position	>
Tool Change Mid Position	>
User Travel Limit	>

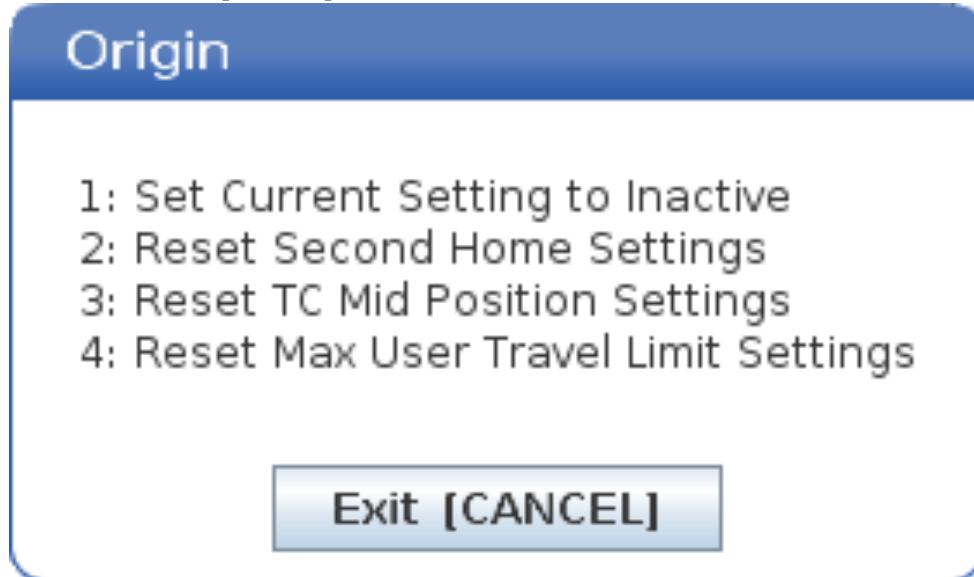
**CAUTION:**

Incorrectly set user positions can cause machine crashes. Set user positions with caution, especially after you have changed your application in some way (new program, different tools, etc.). Verify and change each axis position separately.

To set a user position, jog the axis into the position you want to use, and then press F2 to set the position. If the axis position is valid, a crash warning appears (except for user travel limits). After you verify that you want to make the change to the position, the control sets the position and makes the setting active.

If the position is not valid, the message bar at the bottom of the screen gives a message to explain why the position is not valid.

To inactivate and reset user position settings, press ORIGIN while the user positions tab is active, then choose from the menu that appears.

F9.21: User Positions [ORIGIN] Menu

1. Press **[1]** to remove the value of the currently selected position setting and make it inactive.
2. Press **[2]** to remove the values of all second home position settings and make them inactive.
3. Press **[3]** to remove the values of all Tool Change Mid-Position settings and make them inactive.
4. Press **[4]** to remove the values of all Max User Travel Limit settings and make them inactive.
5. Press **[CANCEL]** to exit the menu without making changes.

9.4 More Information Online

For updated information, including tips, tricks, maintenance procedures, and more, visit the Haas Service page at www.HaasCNC.com.

For the most current Operator's and Service Manuals scan the code below with your mobile device:



Chapter 10: Other Equipment

10.1 More Information Online

For updated information, including tips, tricks, maintenance procedures, and more, visit the Haas Service page at www.HaasCNC.com.

For the most current Operator's and Service Manuals scan the code below with your mobile device:



Index

#

5-Axis Tool Length Compensation + 378

A

absolute positioning (G90)

 versus incremental 190

Active Codes 64

active program 109

active tool display 65

Advanced Tool Management (ATM) 124

 macros and 128

 tool group usage 127

APL

 APL Enable 492

auto door (option)

 override 37

axis motion

 absolute versus incremental 190

 circular 198

 linear 197

B

background edit 181

beacon light

 status 37

Block Delete 42

block selection 179

BT tooling 123

C

Calculators

 Milling / Turning 59

 Standard 57

 Tapping 59

Canned Cycles

 Boring and Reaming 210

 Drilling 209

 R Planes 210

 Tapping 209

canned cycles

 general information 303

check box selection 109

circular interpolation 198

control display

 active codes 55

 active tool 65

 basic layout 50

 offsets 53

control pendant 36, 37

 USB port 37

coolant

 operator override 49

 setting 32 and 452

coolant gauge 66

counters

 reset 54

create a container

 unzip files 108

 zip files 108

CT tooling 123

Current commands 53

cutter compensation

 circular interpolation and 206

 entry and exit 203

 feed adjustments 204

 general description 200

 improper application example 204

 Setting 58 and 200

D	
device manager	
create new program	107
edit	110
file display	106
operation	105
device manager (List Program)	104
directory	
create new	111
display	
axis positions.....	69
settings	64
distance to go position	69
drilling canned cycles.....	209
dynamic work offset (G254).....	400
E	
edit keys	178
editing	
highlight code	178
editor.....	181
Edit menu	183
File menu.....	183
Modify menu.....	185
pull-down menu.....	182
Search menu	184
Error Report Shift F3	112
F	
Fanuc.....	200
Feature List	221
200-hour tryout	222
Enable/Disable.....	222
feed adjustments	
in cutter compensation.....	204
feed hold	
as override	49
file	
deletion	111
file display columns	106
file selection	
multiple	109
G	
G253	400
G268 / G269.....	405
G-codes	298
cutting.....	197
graphics mode	174
H	
Haas Connect.....	507
Haas Robot	
FANUC DCS.....	12
installation	12
HaasDrop	506
help function.....	82
high-speed SMTC	
heavy tools and	148
I	
incremental positioning (G91)	
versus absolute	190
input	
special symbols	112
input bar	70
interpolation motion	
circular.....	198
linear	197
J	
jog mode	164
K	
keyboard	
alpha keys	46
cursor keys	40
display keys	41
function keys.....	39
jog keys	47
key groups.....	38
mode keys	41
numeric keys.....	45
override keys.....	48
L	
LCD Touchscreen - maintenance	82
LCD Touchscreen - navigation	76
LCD Touchscreen - overview	74
LCD Touchscreen - program editing	81

LCD Touchscreen - selectable boxes	78
LCD Touchscreen - Virtual Keyboard	80
line numbers	
remove all	185
linear interpolation.....	197
LIST PROGRAM display	105
local subprograms (M97)	215
locate the last program error	120
 M	
M30 counters	67
machine data	
back up and recover.....	113
Machine Data Collection.....	509
machine position	69
machine power-up	103
machine restore	
full data	117
selected data.....	118
machine rotary zero point (MRZP)	240
macro variables	
axis position	269
Macros	
#3000 programmable alarm	266
#3001-#3002 timers	266
#3006 programmable stop.....	268
#3030 single block	268
#5041-#5046 current work coordinate posi-	
tion	269
1-bit discrete outputs	274
aliasing	292
arguments	253
block look ahead and block delete	250
DPRNT	288
DPRNT editing	290
DPRNT execution	290
DPRNT formatted output	288
DPRNT settings	289
G65 macro subprogram call	291
global variables	257
introduction	248
local variables	256
look ahead.....	250
macro variable display	251
macro variable table	257
round off	249
setting aliasing	293
system variables	257
system variables in-depth	264
timers and counters window	252
useful g- and m-codes.....	249
variable usage	275
macros	
M30 counters and	67
variables	255
main spindle display.....	73
manual data input (MDI).....	180
save as numbered program.....	180
material	
fire risk.....	8
M-code relays	
with M-fin	416
M-codes	410
coolant commands	197
program stop.....	196
spindle commands	196
media display	60
memory lock.....	37
mode display	51

N

Network Connection	497
Icons.....	498
Net Share Setting	504
Wired Connection	500
Wired Network Settings	501
Wireless Connection Setup.....	501
new program.....	107

O

offset	
tool.....	194
work	194
offsets	
display	53
operating modes.....	51
operation	
unattended.....	8
operator position.....	69
optional stop	412
overrides.....	49
disabling	49

P

Pallet Changer	
maximum weight	153
pallet schedule table	156
recovery	157
warnings.....	153
part setup.....	164
set a tool offset	169
set a work offset.....	171
setting offsets	164
work offsets.....	170
position display.....	69
positioning	
absolute vs. incremental	190
positions	
distance to go	69
machine	69
operator.....	69
work (G54).....	69
probe	
troubleshooting	232
probing	227

program

active	109
basic search	119
duplication	111
rename	111
programming	
background edit.....	181
basic example	186
safe startup line	188
subprograms.....	212

R

r plane.....	210
Rapid Mode.....	490
remote jog handle (RJH-Touch)	
manual jogging	161
mode menu	160
overview	159
tool offsets	162
work offsets	163
rotary	
axis disable / enable	238
configuring new	233
custom configuration.....	236
grid offset	237
tool change offset.....	237
rotary offset	
tilt center	246
run-stop-jog-continue	172

S

safe mode.....	120
safe startup line.....	188
safety	
decals.....	16
door interlock	6
during operation.....	5
electrical	4
glass window	6
inspection	6
introduction.....	1
part loading/unloading	6
robot cells.....	11

safety decals	
standard layout	16
symbol reference	17
safety information	21
search	
find / replace.....	184
second home	37
selection	
multiple blocks	179
Setting 28.....	304
setup mode	9
keyswitch.....	37
side-mount tool changer (SMTC)	
door panel.....	152
extra-large tools	149
moving tools	149
recovery	151
zero pocket designation	148
special G-codes	
engraving.....	211
mirror image	211
pocket milling.....	211
rotation and scaling.....	211
special symbols	112
spindle load meter.....	73
spindle orientation (M19)	226
spindle safety limit.....	14
spindle warm-up	104
subprograms.....	212
external	212
local.....	215
T	
tabbed menus	
basic navigation	73
table workholding.....	495
tapping canned cycles	209
text	
find / replace.....	184
selection	179
Through-Spindle Coolant.....	47
drilling cycle and.....	209
M-code.....	424
tilt axis	
center of rotation offset	246
timer and counters display	
reset.....	54
timers and counters display	67
Tool Center Point Control.....	394
tool center point control.....	394, 397
G54 and.....	398
rotary setup and.....	239
tool change offset	
rotary.....	237
tool changer	143
safety	152
tool management tables	
save and restore	129
tool offset.....	194
tool table.....	145
Tool Usage Display.....	130
tooling	
pull studs.....	124
Tnn code.....	196
tool holder care.....	124
tool holders.....	123
U	
umbrella tool changer	
loading.....	149
recovery	150
unattended operation	8
user positions	512
W	
work (G54) position.....	69
work offset	194
macros and.....	270
Workholding	
e-vise setup	134
hydraulic vise setup	137
overview	133
pneumatic vise setup	141
workholding.....	164
safety and	5

