

## G-Codes:

G00 - Rapid Move  
 G01 - Linear Feed Move  
 G02 - Clockwise Arc Feed Move  
 G03 - Counter Clockwise Arc Feed Move  
  
 G04 - Dwell  
  
 G05 - Cubic Spline  
 G05.1 - Quadratic Spline  
 G05.2 - NURBS Block  
 G05.3 - NURBS Block End  
  
 G07 - Lathe Mode - Diameter  
 G08 - Lathe Mode - Radius  
  
 G09 - Stop & Sync  
  
 G10 - Settings  
  
 G12 - Clockwise Circle  
 G12.1 - Clockwise Spiral Pocket  
 G13 - Counter Clockwise Circle  
 G13.1 - Counter Clockwise Spiral Pocket  
  
 G15 - Polar Coordinate Cancel  
 G16 - Polar Coordinate Enable  
  
 G17 - XY Plane  
 G18 - ZX Plane  
 G19 - YZ Plane  
  
 G20 - Inch Units  
 G21 - Millimeter Units  
  
 G28 - Go To Home  
 G28.1 - Set Home  
 G30 - Go To Home  
 G30.1 - Set Home  
  
 G32 - Spindle Synchronized Motion  
 G33 - Spindle Synchronized Motion  
 G33.1 - Spindle Synchronized Motion With Return  
  
 G31 - Probe  
 G38.1 - Probe  
 G38.2 - Probe  
 G38.3 - Probe  
 G38.4 - Probe  
 G38.5 - Probe  
  
 \*G40 - Tool Compensation Cancel  
 \*G41 - Tool Compensation Left  
 \*G41.1 - Tool Compensation Dynamic Left  
 \*G42 - Tool Compensation Right  
 \*G42.1 - Tool Compensation Dynamic Right  
  
 G43 - Tool Offset+ Enable  
 G43.1 - Tool Offset+ Enable  
 G44 - Tool Offset- Enable  
 G44.1 - Tool Offset- Enable  
 G49 - Tool Offset Cancel  
  
 G50 - Axes Scale Cancel  
 G51 - Axes Scale Enable  
  
 G52 - Axes Offset  
 G52.1 - Axes Offset Cancel

G53 - Machine Coordinate System  
  
 G54 - Coordinate System 1  
 G54.1 - Coordinate System P  
 G55 - Coordinate System 2  
 G56 - Coordinate System 3  
 G57 - Coordinate System 4  
 G58 - Coordinate System 5  
 G59 - Coordinate System 6 (or P)  
 G59.1 - Coordinate System 7  
 G59.2 - Coordinate System 8  
 G59.3 - Coordinate System 9  
  
 G61 - Blend Cancel  
 G64 - Blend Enable  
  
 \*G65 - Macro Call  
  
 G68 - Axes Rotate Enable  
 G69 - Axes Rotate Cancel  
  
 G70 - Inch Units (G20)  
 G71 - Millimeter Units (G21)  
  
 G73 - Drilling, Chip Break  
 \*G74 - Tapping, Left-Hand  
 \*G76 - Threading  
  
 G80 - Cancel Motion  
  
 G81 - Drilling  
 G82 - Drilling, Dwell  
 G83 - Drilling, Peck  
 \*G84 - Tapping, Right-Hand  
 G85 - Boring, Retract at Feed, Spindle On  
 G86 - Boring, Retract at Rapid, Spindle Off  
 G87 - Back Boring  
 G88 - Boring, Manual Retract, Spindle Off  
 G89 - Boring, Dwell, Retract at Feed, Spindle On  
  
 G90 - Distance Mode - Absolute  
 G90.1 - Distance Mode - IJK Absolute  
 G90.2 - Distance Mode - ABC Absolute  
 G91 - Distance Mode - Incremental  
 G91.1 - Distance Mode - IJK Incremental  
 G91.2 - Distance Mode - ABC Incremental  
  
 G92 - Working Offset  
 G92.1 - Working Offset Cancel  
  
 G93 - Feed Mode - Inverse Time  
 G94 - Feed Mode - Units per Minute  
 G95 - Feed Mode - Units per Revolution  
  
 G96 - Spindle Mode - CSS  
 G97 - Spindle Mode - RPM  
  
 G98 - Cycle Return - Initial Z Point  
 G99 - Cycle Return - R Point

## M-Codes:

M0	- Pause
M1	- Pause (optional)
M2	- Program End
M3	- Spindle CW
M4	- Spindle CCW
M5	- Spindle Stop
M6	- Tool Change
M61	- Tool Select
M7	- Mist On
M17	- Mist Off/On
M8	- Flood On
M18	- Flood Off/On
M9	- Mist & Flood Off
M48	- Enable Override Feed & Spindle
M49	- Disable Override Feed & Spindle
M50	- Enable/Disable Override Feed
M51	- Enable/Disable Override Spindle
M53	- Enable/Disable Pause
M54	- Enable/Disable THC
M55	- Enable/Disable Transformations
M56	- Enable/Disable Custom Scripts
M62	- Output
M63	- Output PWM
M64	- Output ExtOut
M65	- Output TX
M66	- Output I2C
M67	- Input
M68	- Input ExtIn
M69	- Input Aux
M59	- Wait For Input
M70	- Modal State Store
M71	- Modal State Invalidate
M72	- Modal State Restore
M73	- Modal State Store & Autorestore

## Other Codes:

F	- Feed Speed
S	- Spindle Speed
T	- Select Tool
N	- Line Number
O	- O-Word
( )	- Comment
;	- Comment
//	- Comment
/	- Optional line
#n	- Numbered Parameter
#<n>	- Named Parameter
[]	- Expression
!	- G-Script

## O-Words:

	• sub..endsub subroutine block
Osub	- Begin subroutine
Oendsub	- End subroutine
Oreturn	- Exit subroutine
Ocall	- Call subroutine
	• do..while loop block
Odo	- Begin block
Owhile	- End block [condition evaluation]
Obreak	- Exit block immediately
Ocontinue	- Skip to next condition evaluation
	• while..endwhile loop block
Owhile	- Begin block [condition evaluation]
Oendwhile	- End block
Obreak	- Exit block immediately
Ocontinue	- Skip to next condition evaluation
	• repeat..endrepeat loop block
Orepeat	- Begin block [count]
Oendrepeat	- End block
	• if..endif condition block
Oif	- Begin block [condition evaluation]
Oendif	- End block
Oelseif	- Else [condition evaluation]
Oelse	- Else

#### Comments:

MSG	- shows message in status bar
MSGDLG	- shows dialog with message
PRINT	- prints message to software log
LOGOPEN	- creates new log file
LOGAPPEND	- opens existing log file
LOGCLOSE	- closes log file
LOG	- writes message to log
PROBEOPEN	- opens new probe file
PROBECLOSE	- closes probe file
EXEC	- executes external program
EXECWAIT	- executes external program and waits
ARRAY	- creates parameters from data
ARCHR	- creates parameters from characters

#### Operators:

+	- Addition
-	- Subtraction
*	- Multiplication
/	- Division
MOD	- Modulus
**	- Power
EQ	- Relational equality
NQ	- Relational inequality
GT	- Relational strictly greater than
LT	- Relational strictly less than
GE	- Relational greater than or equal to
LE	- Relational less than or equal to
AND	- Logical AND
OR	- Logical non-exclusive OR
XOR	- Logical exclusive OR

#### Operator Precedence:

\*\*  
\* / MOD  
+ -  
EQ NE GT GE LT LE  
AND OR XOR

## Functions:

Abs	- Absolute value
Sqrt	- Square Root
Sqr	- Square
Sin	- Sine
Cos	- Cosine
Tan	- Tangent
ASin	- Inverse sine
ACos	- Inverse cosine
ATan	- Inverse tangent
ATan2	- Four quadrant inverse tangent
Pi	- Pi value
Deg	- Radians to degrees
Rad	- Degrees to radians
Pow	- Power
Exp	- e raised to the given power
Exp10	- 10 raised to the given power
Exp2	- 2 raised to the given power
Log	- Base e logarithm
Log10	- Base 10 logarithm
Log2	- Base 2 logarithm
DateTime	- Current time
Year	- Year from DateTime value
Month	- Month from DateTime value
Day	- Day from DateTime value
Hour	- Hour from DateTime value
Minute	- Minute from DateTime value
Second	- Second from DateTime value
Min	- Minimum
Max	- Maximum
And	- Bitwise AND
Or	- Bitwise non-exclusive OR
Xor	- Bitwise exclusive OR
Not	- Bitwise complement
Shl	- Bitwise shift left
Shr	- Bitwise shift right
Hex	- Converts string to number
Bin	- Converts string to number
Round	- Round to nearest integer
Floor	- Round down to integer
Ceil	- Round up to integer
Trunc	- Truncate to integer
ToMachine	- Work to machine coordinate
ToWork	- Machine to work Coordinate
Exists	- Check if named parameter exists

# G-Codes

## G00 - Rapid Move

Usage: G00 <X..W@^>

Rapid move machine to programmed position.

## G01 - Linear Feed Move

Usage: G01 <X..W@^>

Move machine to programmed position with feed speed.

## G02 - Clockwise Arc Feed Move

## G03 - Counter Clockwise Arc Feed Move

Usage: G02 <X..W@^> <IJK> <P>

G03 <X..W@^> <IJK> <P>

G02 <X..W@^> <R> <P>

G03 <X..W@^> <R> <P>

Move machine to programmed position with feed speed in arc motion.

G02 - clockwise arc

G03 - counterclockwise arc

Arc can be in 3 different planes, depending of G17, G18 or G19 modal state.

XY plane, G17 state, Z rotation axis

ZX plane, G18 state, Y rotation axis

YZ plane, G19 state, X rotation axis

If motion in direction of rotation axis is specified then helical motion is generated.

Arc is specified in center format using IJK words or in radius format using R word.

Center format – arc center is defined with IJ (in XY plane), KI (in ZX plane) or JK (in YZ plane) words.

In incremental arc distance mode (G91.1) arc center is set as offset from start position.

In absolute arc distance mode (G90.1) arc center is set as distance from zero position.

Radius format – arc is defined with arc radius. This format is depreciated because it can in certain conditions produce cuts that are out of tolerances.

Number of rotations can be set with optional P word. If, for example, P3 is set then we have two full circles before final arc.

Example:

G17 G21 G90 G91.1 G94

F1000

G0 X0 Y0

G1 X80 Y0

G3 X100 Y20 I0 J20

G1 X100 Y80

G3 X80 Y100 I-20 J0

G1 X20 Y100

G3 X0 Y80 I0 J-20

G1 X0 Y0

## **G04 - Dwell**

Usage: G04 P

Delays execution for duration P seconds.

## **G05 - Cubic Spline**

Usage: G05 XYZ IJK PQR X – end position X  
Y – end position Y  
Z – end position Z  
I – incremental offset X from start point to first control point  
J – incremental offset Y from start point to first control point  
K – incremental offset Z from start point to first control point  
P – incremental offset X from end point to second control point  
Q – incremental offset Y from end point to second control point  
R – incremental offset Z from end point to second control point

Cubic B-spline move.

Spline can be in 3 different planes, depending of G17, G18 or G19 modal state.

XY plane, G17 state, XY, IJ and PQ words used

ZX plane, G18 state, ZX, KI and RP words used

YZ plane, G19 state, YZ, JK and QR words used

Example:

G17 G21 G90 G91.1 G94

F1000

G0 X0 Y0

G1 X80 Y0

G5 X100 Y20 I20 J0 P0 Q-20

G1 X100 Y80

G5 X80 Y100 P20 Q0

G1 X20 Y100

G5 X0 Y80 P0 Q20

G1 X0 Y0

## **G05.1 - Quadratic Spline**

Usage:            G05.1 XYZ IJK        X – end position X  
    Y – end position Y  
    Z – end position Z  
    I – incremental offset X from start point to control point  
    J – incremental offset Y from start point to control point  
    K – incremental offset Z from start point to control point

Quadratic B-spline move.

Spline can be in 3 different planes, depending of G17, G18 or G19 modal state.

XY plane, G17 state, XY and IJ words used

ZX plane, G18 state, ZX and KI words used

YZ plane, G19 state, YZ and JK words used

Example:

G17 G21 G90 G91.1 G94

F1000

G0 X0 Y0

G1 X80 Y0

G5.1 X100 Y20 I20 J0

G1 X100 Y80

G5.1 X80 Y100 I0 J20

G1 X20 Y100

G5.1 X0 Y80 I-20 J0

G1 X0 Y0

### G05.3- NURBS Block End

G05.3                      end NURBS block

Spline can be in 3 different planes, depending of G17, G18 or G19 modal state.

XY plane, G17 state, XY and IJ words used

ZX plane, G18 state, ZX and KI words used

YZ plane, G19 state, YZ and JK words used

Example:

G17 G21 G90 G91.1 G94

F1000

G0 X0 Y0

G1 X80 Y0

G5.2 L3

G5.2 X100 Y0 P2

G5.2 X100 Y20 P2

### G5.3

G1 X100 Y80

G5.2 L3

G5.2 X100 Y100 P2

G5.2 X80 Y100 P2

## G5.3

G1 X20 Y100

G5.2 L3

G5.2 X0 Y100 P2

G5.2 X0 Y80 P2

## G5.3

G1 X0 Y0



## **G07 - Lathe Mode – Diameter**

Usage: G07

Use diameter mode for axis X on a lathe. When in the diameter mode the X axis moves on a lathe will be 1/2 the distance to the center of the lathe.

Parameters:

#<\_lathemode> - (RW) current lathe mode

## **G08 - Lathe Mode – Radius**

Usage: G08

Use radius mode for axis X on a lathe. When in radius mode the X axis moves on a lathe will be the distance from the center.

Parameters:

#<\_lathemode> - (RW) current lathe mode

## **G09 - Stop & Sync**

Usage: G09

Stop motion any synchronize controller and interpreter state. Usually it is used before reading inputs from controller. For example:

G00 X0

G01 X100

G09 ;wait for motion to stop before reading \_hw\_input parameter.

(print, Value of input is #<\_hw\_input>)

## **G10 L9 - Settings – Controller Position**

Usage: G10 L9 <X..W>

Set controller position without move

Sends axis words to controller. Controller will set its current motor position to this value.

Parameters:

#<\_hw\_motor\_x..w> - (R) motor position

## **G10 L3 - Settings – Transformation**

Usage: G10 L3 P A B C I J K

- P – 1 for XY, 2 for UV system
- A – Transformation A0 parameter
- B – Transformation B0 parameter
- C – Transformation C0 parameter
- I – Transformation A1 parameter
- J – Transformation B1 parameter
- K – Transformation C1 parameter

Set transformation parameters.

P word selects coordinate system.

If any of ABCIJK words are missing or values are invalid transformation is reset.

Reset state is A0=1, B0=0, C0=0, A1=0, B1=1, C1=0.

See also: M55

Parameters:

#<_trans_en>	- (RW) true if transformations are enabled
#<_transxy_a0>	- (R) transformation A0 parameter in XY system
#<_transxy_b0>	- (R) transformation B0 parameter in XY system
#<_transxy_c0>	- (R) transformation C0 parameter in XY system
#<_transxy_a1>	- (R) transformation A1 parameter in XY system
#<_transxy_b1>	- (R) transformation B1 parameter in XY system
#<_transxy_c1>	- (R) transformation C1 parameter in XY system
#<_transuv_a0>	- (R) transformation A0 parameter in UV system
#<_transuv_b0>	- (R) transformation B0 parameter in UV system
#<_transuv_c0>	- (R) transformation C0 parameter in UV system
#<_transuv_a1>	- (R) transformation A1 parameter in UV system
#<_transuv_b1>	- (R) transformation B1 parameter in UV system
#<_transuv_c1>	- (R) transformation C1 parameter in UV system

## G10 L1, L10 - Settings – Tool Table

Usage:	G10 L1 P <X..W> <D or R>	<ul style="list-style-type: none"><li>- P – tool number</li><li>- X..W (optional) – tool offset</li><li>- D (optional) – tool diameter or</li><li>- R (optional) – tool radius</li></ul>
	G10 L10 P <X..W> <D or R>	<ul style="list-style-type: none"><li>- P – tool number</li><li>- X..W (optional) – tool offset</li><li>- D (optional) – tool diameter or</li><li>- R (optional) – tool radius</li></ul>

Use L1 or L10 to set tool in tool table.

L1 will set tool offset as entered. L10 will set offset so that current working position becomes entered value.

Parameters:

#<_tooloff>	- (R) true if tool offset is currently set
#<_tooloff_x..w>	- (R) current tool offset
#<_current_tool>	- (RW) current tool number
#<_current_tool_dia>	- (R) current tool diameter from tool table
#<_current_tool_off_x..w>	- (R) current tool offset from tool table
#<_selected_tool>	- (RW) selected tool number
#<_selected_tool_dia>	- (R) selected tool diameter from tool table
#<_selected_tool_off_x..w>	- (R) selected tool offset from tool table

## G10 L2, L20 - Settings – Coordinate System

Usage:	G10 L2 P <X..W> <R>	<ul style="list-style-type: none"><li>- P – coordinate system number</li><li>- X..W (optional) – coordinate system offset</li><li>- R (optional) – rotation in XY plane</li></ul>
	G10 L20 P <X..W> <R>	<ul style="list-style-type: none"><li>- P – coordinate system number</li><li>- X..W (optional) – coordinate system offset</li><li>- R (optional) – rotation in XY plane</li></ul>

Use L2 or L20 to set coordinate system.

L2 will set coordinate system offset as entered. L20 will set offset so that current working position becomes entered value.

Optionally coordinate system rotation in XY plane is set with R word.

300 different coordinate systems are available.

Parameters:

#<_coord_system_num>	- (R) current coordinate system number
#<_coord_system_x..w>	- (R) current coordinate system offset
#<_coord_system_rot>	- (R) current coordinate system rotation

## **G12 - Clockwise Circle**

## **G13 - Counter Clockwise Circle**

Usage:        G12 I J <P> <Q> <H>        I – circle lead-in direction X  
   J – circle lead-in direction Y  
   P (optional) – stepover  
   Q (optional) – inner circle radius  
   H (optional) – if true spirals are used

Cut a circle with current position as circle center.

G12 – circle is in clockwise direction.

G13 – circle is in counter clockwise direction.

I word and J word define lead-in direction and radius. Radius can be calculated as “ $\sqrt{I^2 + J^2}$ ”.

If optional P word is used then series of circles will be generated with P distance between them.

If optional Q word is used then inner circle will have radius Q.

If optional H word is true then spirals will be used instead of arcs.

This code is only valid in XY plane.

Example:

(clockwise spiral with center at X0 Y0, radius 20, inner radius 10 and step over 3)

G00 X0 Y0

G12 I14.1 J14.1 P3 Q10 H1

## **G12.1- Clockwise Spiral Pocket**

## **G13.1- Counter Clockwise Spiral Pocket**

Usage:        G12.1 X Y Z P D I K E H        X – pocket center X  
   Y – pocket center Y  
   Z – start height  
   P – pocket depth  
   R – pocket radius  
   D – tool diameter  
   I – stepover XY  
   K – stepover Z  
   E – plunge speed  
   H – number of finish passes

Cut a spiral pocket.

G12.1 – spiral is in clockwise direction.

G13.1 – spiral is in counter clockwise direction.

X and Y words define pocket center. Z word defines start height. P word defines pocket depth.

D word is used for tool diameter compensation.

I word is used for setting stepover in XY direction, K is used to set stepover in Z (plunge) direction.

E word defines plunge feed speed.

H word defines number of finish passes.

Example:

(clockwise spiral pocket with center at X0 Y0)

(start height at Z0, pocket depth 10 and radius 20)

(tool diameter 6 will be compensated, step over is 2.4 in XY and 1 in Z direction)

(plunge speed is 100, two finish passes)

G12.1 X0 Y0 Z0 P10 R20 D6 I2.4 K1 E100 H2

## **G15 - Polar Coordinate Cancel**

Usage: G15

Cancel polar mode coordinate positioning.

Parameters:

#<\_polarmode> - (RW) current polar mode

## **G16 - Polar Coordinate Enable**

Usage: G16

Enable polar mode coordinate positioning. In the polar coordinate mode movement end points are specified as a radius and angle. Origin is determined by the absolute/incremental position mode setting.

The current plane setting determines which word is radius and which is angle.

G17 – XY Plane – X is radius, Y is angle

G18 – ZX Plane – Z is radius, X is angle

G19 – YZ Plane – Y is radius, Z is angle

Parameters:

#<\_polarmode> - (RW) current polar mode

Example:

(square with corners -30,-30 and 30,30)

G0 X0 Y0

G16

G00 X42.4264 Y225

G01 X42.4264 Y135

G01 X42.4264 Y45

G01 X42.4264 Y315

G01 X42.4264 Y225

G15

## **G17 - XY Plane**

Usage: G17

Set XY plane. Planes are used with arcs, circles and cycles.

#<\_plane> - (RW) current plane

## **G18 - ZX Plane**

Usage: G18

Set ZX plane. Planes are used with arcs, circles and cycles.

Parameters:

#<\_plane> - (RW) current plane

## **G19 - YZ Plane**

Usage: G19

Set YZ plane. Planes are used with arcs, circles and cycles.

Parameters:

#<\_plane> - (RW) current plane

## **G20 - Inch Units**

## **G70 - Inch Units**

Usage: G20

G70

Set units to inches.

Parameters:

#<\_units> - (RW) current units

## **G21 - Millimeter Units**

## **G71 - Millimeter Units**

Usage: G21

G71

Set units to millimeters.

Parameters:

#<\_units> - (RW) current units

## **G28 - Go To Home 1 Position**

Usage:        G28  
              G28 <X..W>

Rapid move machine to Home 1 position. If intermediate move is specified then machine will rapid move to intermediate position first and only axes specified will move to final position.

This command is often used with G91. For example: G91 G28 Z0

Parameters:

#<\_home1\_x..w>                - (RW) home 1 position

### **G28.1 - Set Home 1 Position**

Usage:        G28.1

Set current position as Home 1 position.

Parameters:

#<\_home1\_x..w>                - (RW) home 1 position

## **G30 - Go To Home 2 Position**

Usage:        G30  
              G30 <X..W>

Rapid move machine to Home 2 position. If intermediate move is specified then machine will rapid move to intermediate position first and only axes specified will move to final position.

This command is often used with G91. For example: G91 G30 Z0

Parameters:

#<\_home1\_x..w>                - (RW) home 1 position

### **G30.1 - Set Home 2 Position**

Usage:        G30.1

Set current position as Home 2 position.

Parameters:

#<\_home2\_x..w>                - (RW) home 2 position

### G33 - Spindle Synchronized Motion / Threading

Move machine with speed synchronized to spindle speed.

Move machine with speed synchronized to spindle speed. When machine reaches end position, spindle reverses direction and machine moves back to start position.



## **G31 - Probe**

### **G38.1- Probe**

### **G38.2- Probe**

### **G38.3- Probe**

### **G38.4- Probe**

### **G38.5- Probe**

Usage:        G31 <X..W>  
              G38.1 <X..W>  
              G38.2 <X..W>  
              G38.3 <X..W>  
              G38.4 <X..W>  
              G38.5 <X..W>

Move machine with feed speed towards position. Immediately stop when sensor is signaled and store position to probe parameters.

G31 – axis words are position, stop on contact, error if failure

G38.1 – axis words are direction, stop on contact, error if failure

G38.2 – axis words are position, stop on contact, error if failure

G38.3 – axis words are position, stop on contact

G38.4 – axis words are position, stop on loss of contact, error if failure

G38.5 – axis words are position, stop on loss of contact

Parameters:

#<_probe>	- (R) true if probe tripped
#<_probe_x..w>	- (R) probe position

## **G40 - Tool Compensation Cancel**

Usage: G40

Cancel tool compensation.

Parameters:

#<_toolcomp>	- (R) tool compensation side
#<_toolcomp_dia>	- (R) tool compensation diameter

## **G41 - Tool Compensation Left**

## **G42 - Tool Compensation Right**

Usage: G41 <D> - D (optional) = tool number

Start tool compensation with diameter from tool table.

If D word is not specified then current tool number is used.

Parameters:

#<_toolcomp>	- (R) tool compensation side
#<_toolcomp_dia>	- (R) tool compensation diameter

## **G41.1- Tool Compensation Dynamic Left**

## **G42.1- Tool Compensation Dynamic Right**

Usage: G41.1 D - D = tool diameter

Start tool compensation.

D word specifies tool diameter.

Parameters:

#<_toolcomp>	- (R) tool compensation side
#<_toolcomp_dia>	- (R) tool compensation diameter

## **G43 - Tool Offset+ Enable**

## **G44 - Tool Offset- Enable**

Usage: G43 <H> - H (optional) = tool number

Set tool offset to value from tool table.

If H word is not specified then current tool number is used.

Parameters:

#<_tooloff>	- (R) true if tool offset is currently set
#<_tooloff_x..w>	- (R) current tool offset
#<_current_tool>	- (RW) current tool number
#<_current_tool_dia>	- (R) current tool diameter from tool table
#<_current_tool_off_x..w>	- (R) current tool offset from tool table
#<_selected_tool>	- (RW) selected tool number
#<_selected_tool_dia>	- (R) selected tool diameter from tool table
#<_selected_tool_off_x..w>	- (R) selected tool offset from tool table

## **G43.1 - Tool Offset+ Enable**

## **G44.1 - Tool Offset- Enable**

Usage: G43.1 X..W - X..W = tool offset

Set tool offset to value from axis words.

Parameters:

#<_tooloff>	- (R) true if tool offset is currently set
#<_tooloff_x..w>	- (R) current tool offset

## **G49 - Tool Offset Cancel**

Usage: G49

Cancel tool offset

Parameters:

#<_tooloff>	- (R) true if tool offset is currently set
#<_tooloff_x..w>	- (R) current tool offset

## **G50 - Axes Scale Cancel**

Usage: G50

Cancel axis scale.

Parameters:

#<_axisscale>	- (R) true if axis scale is enabled
#<_axisscale_x..w>	- (R) current axis scale value

## **G51 - Axes Scale Enable**

Usage: G51 X..W

Set axis scale.

Parameters:

#<_axisscale>	- (R) true if axis scale is enabled
#<_axisscale_x..w>	- (R) current axis scale value

## **G52 - Axes Offset**

Usage: G52 X..W

Set axis offset.

Parameters:

#<_axisoff>	- (R) true if axis offset is enabled
#<_axisoff_x..w>	- (R) current axis offset value

## **G52.1 - Axes Offset Cancel**

Usage: G52.1

Cancel axis offset.

Parameters:

#<_axisoff>	- (R) true if axis offset is enabled
#<_axisoff_x..w>	- (R) current axis offset value

## **G53 - Machine Coordinate System**

Usage: G53

Use G53 in same line with motion code to execute motion in machine coordinates.

## **G54 - Coordinate System 1**

## **G55 - Coordinate System 2**

## **G56 - Coordinate System 3**

## **G57 - Coordinate System 4**

## **G58 - Coordinate System 5**

## **G59 - Coordinate System 6**

## **G59.1 - Coordinate System 7**

## **G59.2 - Coordinate System 8**

## **G59.3 - Coordinate System 9**

Usage:	G54	- select coordinate system 1
	G55	- select coordinate system 2
	G56	- select coordinate system 3
	G57	- select coordinate system 4
	G58	- select coordinate system 5
	G59	- select coordinate system 6
	G59.1	- select coordinate system 7
	G59.2	- select coordinate system 8
	G59.3	- select coordinate system 9

Select coordinate system.

## **G54.1 - Coordinate System P**

## **G59 P - Coordinate System P**

Usage:	G54.1 P	- P = select coordinate system P+6
	G59 P	- P = select coordinate system P

Select coordinate system.

Coordinate system can be set with G10 L2 code.

300 different coordinate systems are available.

Parameters:

#<_coord_system_num>	- (R) current coordinate system number
#<_coord_system_x..w>	- (R) current coordinate system offset
#<_coord_system_rot>	- (R) current coordinate system rotation

## **G61 - Blend Cancel**

Usage:        G61

In canceled blend mode movement is exactly as programmed. Moves will slow or stop as needed to reach every programmed point. If two sequential moves are exactly co-linear movement will not stop.

Parameters:

#<\_blendmode>                      - (RW) current blend mode

## **G64 - Blend Enable**

Usage:        G64 <P> <Q>        - P (optional) = tolerance  
   - Q (optional) = CAM tolerance

In blend mode sharp corners will be rounded and the machine may never reach the programmed point before a direction change. Optional tolerance can be set.

Parameters:

#<\_blendmode>                      - (RW) current blend mode  
#<\_blend\_tol>                       - (R) current blend tolerance  
#<\_blend\_camtol>                   - (R) current blend CAM tolerance

## **G68 - Axes Rotate Enable**

Usage:        G68   X..Z R        - X..Z – rotation center  
                                     - R – rotation angle

Set axis rotation.

Parameters:

#<_axisrot>	- (R) true if axis rotation is enabled
#<_axisrot_ang>	- (R) current axis rotation angle
#<_axisrot_x..z>	- (R) current axis rotation center

## **G69 - Axes Rotate Cancel**

Usage:        G69

Cancel axis rotation.

Parameters:

#<_axisrot>	- (R) true if axis rotation is enabled
-------------	--

### **G73 - Drilling, Chip Break**

Usage: G73 X Y Z R L P Q

- X, Y – coordinate of drill center
- Z – drill position
- R – retract plane
- L – repetition
- P – dwell time
- Q – delta increment

### **G74 - Tapping, Left-Hand**

### **G76 - Threading**

### **G80 - Cancel Motion**

Usage: G90

Cancels all motion.

### **G81 - Drilling**

Usage: G81 X Y Z R L

- X, Y – coordinate of drill center
- Z – drill position
- R – retract plane
- L – repetition

### **G82 - Drilling, Dwell**

Usage: G82 X Y Z R L P

- X, Y – coordinate of drill center
- Z – drill position
- R – retract plane
- L – repetition
- P – dwell time

### **G83 - Drilling, Peck**

Usage: G83 X Y Z R L P Q

- X, Y – coordinate of drill center
- Z – drill position
- R – retract plane
- L – repetition
- P – dwell time
- Q – delta increment



## G84 - Tapping, Right-Hand

### G85 - Boring, Retract at Feed, Spindle On

Usage:        G85    X Y Z R L P        - X, Y – coordinate of boring center  
   - Z – bore position  
   - R – retract plane  
   - L – repetition  
   - P – dwell time

**G86 - Boring, Retract at Rapid, Spindle Off**

Usage:        G86    X Y Z R L P        - X, Y – coordinate of boring center  
    - Z – bore position  
    - R – retract plane  
    - L – repetition  
    - P – dwell time

## G87 - Back Boring

Usage:        G87    X Y Z R L P        - X, Y – coordinate of boring center  
    - Z – bore position  
    - R – retract plane  
    - L – repetition  
    - P – dwell time

(not fully implemented)

## G88 - Boring, Manual Retract, Spindle Off

Usage:        G88    X Y Z R L P        - X, Y – coordinate of boring center  
    - Z – bore position  
    - R – retract plane  
    - L – repetition  
    - P – dwell time

(manual retract not implemented)

**G89 - Boring, Dwell, Retract at Feed, Spindle On**

Usage:	G89	X Y Z R L P	<ul style="list-style-type: none"> <li>- X, Y – coordinate of boring center</li> <li>- Z – bore position</li> <li>- R – retract plane</li> <li>- L – repetition</li> <li>- P – dwell time</li> </ul>
--------	-----	-------------	--

## **G90 - Distance Mode – Absolute**

Usage: G90

Set absolute distance mode. In absolute position mode machine will move to the commanded position in the active user coordinate system.

Parameters:

#<\_distancemode> - (RW) current distance mode

## **G91 - Distance Mode – Incremental**

Usage: G91

Set incremental distance mode. In incremental position mode commanded moves are interpreted as distance and direction from the current position.

Parameters:

#<\_distancemode> - (RW) current distance mode

## **G90.1 - Distance Mode – IJK Absolute**

Usage: G90.1

Set absolute distance mode for I, J, K arc values. In absolute arc center mode the I, J, K values designate the position of the arc center in the user coordinate system.

Parameters:

#<\_distancemode\_ijk> - (RW) current distance mode for IJK

## **G91.1 - Distance Mode – IJK Incremental**

Usage: G91.1

Set incremental distance mode for I, J, K arc values. In incremental arc center mode the I, J, K values designate the distance and direction to the arc center from the start point.

Parameters:

#<\_distancemode\_ijk> - (RW) current distance mode for IJK

## **G90.2 - Distance Mode – ABC Absolute**

Usage: G90.2

Set absolute distance mode for A, B, C axis values.

Parameters:

#<\_distancemode\_abc> - (RW) current distance mode for ABC

## **G91.2 - Distance Mode – ABC Incremental**

Usage: G91.2

Set incremental distance mode for A, B, C axis values.

#<\_distancemode\_abc> - (RW) current distance mode for ABC

## **G92 - Working Offset**

Usage:        G92 X..W                - X..W = desired position

Set working offset. Offset is calculated so that current position becomes desired position specified with axis words.

Parameters:

#<_workoff>	- (R) true, if working offset is set
#<_workoff_x..w>	- (R) current work offset

### **G92.1 - Working Offset Cancel**

Usage:        G92.1

Cancel work offset and set it to zero.

Parameters:

#<_workoff>	- (R) true, if working offset is set
-------------	--------------------------------------

### **G93 - Feed Mode - Inverse Time**

Usage: G93

In inverse time feed mode, an F word means the move should be completed in [one divided by the F number] minutes. When inverse time feed mode is active, an F word must appear on every motion line.

Parameters:

#<\_feedmode> - (RW) current feed mode

### **G94 - Feed Mode - Units per Minute**

Usage: G94

In units per minute feed mode, an F word is interpreted to mean the controlled point should move at a certain distance per minute.

Parameters:

#<\_feedmode> - (RW) current feed mode

### **G95 - Feed Mode - Units per Revolution**

Usage: G95

In units per revolution feed mode, an F word is interpreted to mean the controlled point should move at a certain distance per spindle revolution. This mode requires RPM feedback from the spindle.

Parameters:

#<\_feedmode> - (RW) current feed mode

## **G96 - Spindle Mode – CSS**

Usage: G96

In this mode surface speed is constant and based on cutting diameter. Surface speed is specified in units per minute. This requires special hardware and is usually not used on simple machines.

Parameters:

#<\_spindlemode> - (RW) current spindle mode

## **G97 - Spindle Mode – RPM**

Usage: G97

In this mode the spindle speed is specified in revolutions per minute.

Parameters:

#<\_spindlemode> - (RW) current spindle mode

## **G98 - Cycle Return - Initial Z Point**

Usage: G98

Specifies that a cycle ends at the initial Z level.

Parameters:

#<\_cyclereturnmode> - (RW) current cycle return mode

## **G99 - Cycle Return - R Point**

Usage: G99

Specifies that a cycle ends at the programmed R level.

Parameters:

#<\_cyclereturnmode> - (RW) current cycle return mode

# M-Codes

## Program

### **M0 - Pause**

Usage: M0

Pause a running program.

### **M1 - Optional pause**

Usage: M1

Pause a running program if „Optional Pause“ is enabled.

### **M2 - Program End**

Usage: M2

End the program. Code after M2 will not be executed. Wrapping G-Code between % does the same thing.

Customization is possible by modifying M2.gcode script file (for example, to turn spindle off and restore modal state at end of g-code execution).

# Spindle

## M3 - Spindle CW

Usage: M3

Start spindle clockwise with S speed. Customization is possible by modifying M3.gcode script file.

Parameters:

#<_spindle>	- (RW) spindle state
#<_spindle_on>	- (RW) true if spindle is running
#<_spindle_cw>	- (RW) true if spindle is running clockwise
#<_spindle_ccw>	- (RW) true if spindle is running counterclockwise

## M4 - Spindle CCW

Usage: M4

Start spindle counterclockwise with S speed. Customization is possible by modifying M4.gcode script file.

Parameters:

#<_spindle>	- (RW) spindle state
#<_spindle_on>	- (RW) true if spindle is running
#<_spindle_cw>	- (RW) true if spindle is running clockwise
#<_spindle_ccw>	- (RW) true if spindle is running counterclockwise

## M5 - Spindle Stop

Usage: M5

Stop spindle. Customization is possible by modifying M5.gcode script file.

Parameters:

#<_spindle>	- (RW) spindle state
#<_spindle_on>	- (RW) true if spindle is running
#<_spindle_cw>	- (RW) true if spindle is running clockwise
#<_spindle_ccw>	- (RW) true if spindle is running counterclockwise

# Tools

## M6 - Tool Change

Usage: M6

Change tool with change procedure. Tool change will set tool selected with T as current tool. Tool change customization is possible by modifying M6.gcode script file.

Parameters:

#<_selected_tool>	- (RW) selected tool number
#<_current_tool>	- (RW) current tool number

## M61 - Tool Select

Usage: M61 Q - Q = tool number

Select current tool number without tool change procedure.

Parameters:

#<_selected_tool>	- (RW) selected tool number
#<_current_tool>	- (RW) current tool number



## Coolant

### M7 - Mist On

Usage: M7  
Turn mist output On.

Parameters:

#<\_mist> - (RW) mist state

### M17 - Mist Off/On

Usage: M17  
M17 P - P = zero for Off, otherwise On

Turn mist output Off or On.

Parameters:

#<\_mist> - (RW) mist state

### M8 - Flood On

Usage: M8  
Turn flood output On.

Parameters:

#<\_flood> - (RW) flood state

### M18 - Flood Off/On

Usage: M18  
M18 P - P = zero for Off, otherwise On

Turn flood output Off or On.

Parameters:

#<\_flood> - (RW) flood state

### M9 - Mist & Flood Off

Usage: M9  
Turn mist and flood output Off

Parameters:

#<\_mist> - (RW) mist state

#<\_flood> - (RW) flood state

## Overrides and options

**M48 - Enable Override Feed & Spindle**

**M49 - Disable Override Feed & Spindle**

**M50 - Enable/Disable Override Feed**

**M51 - Enable/Disable Override Spindle**

**M53 - Enable/Disable Pause**

**M54 - Enable/Disable THC**

**M55 - Enable/Disable Transformations**

Usage: M55 P - P = zero for Off, otherwise On

Enable or disable transformations.

See also: G10 L3

Parameters:

#<\_trans\_en> - (RW) true if transformations are enabled

**M56 - Enable/Disable Warp**

Usage: M56 P - P = zero for Off, otherwise On

Enable or disable warp.

Parameters:

#<\_warp\_en> - (RW) true if transformations are enabled

**M57 - Custom Scripts Enable/Disable**

Usage: M56 P - P = zero for Off, otherwise On

Enable or disable custom scripts. This is useful inside script to prevent recursion.

Parameters:

#<\_custom\_en> - (RW) true if custom scripts are enabled

## Inputs & Outputs

### M59 - Wait For Input

Usage: M59 P - P = pin number (1-8)

Waits for input pin.

### M62 - Output

Usage: M62 Q - Q = value for all eight outputs  
M62 P Q - P = output number  
- Q = zero for Off, otherwise On

Set signals on output connector.

Parameters:

#<\_output> - (R) output signals state  
#<\_hw\_output> - (R) actual output signals on controller

### M63 - Output PWM

Usage: M63 P Q <E> - P = output number  
- Q = frequency  
- E (optional) = duty cycle (0% - 100%)  
M63 P R - P = output number  
- R = RC servo motor position (0% - 100%)

Set PWM signal on output connector. It is possible to set frequency, frequency with duty cycle of RC servo motor position.

Parameters:

#<\_output> - (R) output signals state  
#<\_hw\_output> - (R) actual output signals on controller  
#<\_hw\_output\_freq1> - (R) actual PWM frequency on first PWM output  
#<\_hw\_output\_duty1> - (R) actual PWM duty cycle on first PWM output  
#<\_hw\_output\_freq2> - (R) actual PWM frequency on second PWM output  
#<\_hw\_output\_duty2> - (R) actual PWM duty cycle on second PWM output  
#<\_hw\_output\_freq3> - (R) actual PWM frequency on third PWM output  
#<\_hw\_output\_duty3> - (R) actual PWM duty cycle on third PWM output

### M64 - Output ExtOut

Usage: M64 H Q <L> - H = ExtOut SSEL value (1 or 2)  
- Q = data send to ExtOut  
- L (optional) = number of bytes send to ExtOut  
M64 H Q E <L> - H = ExtOut SSEL value (1 or 2)  
- Q = parameter number of first data location  
- E = data count  
- L (optional) = number of bytes send to ExtOut  
M64 H P Q <L> - H = ExtOut SSEL value (1 or 2)  
- P = bit position  
- Q = zero for Off, otherwise On  
- L (optional) = number of bytes send to ExtOut  
M64 H L0 - H = ExtOut SSEL value (1 or 2)  
- L = zero bytes send to ExtOut

Send data to OutExt.

Parameters:

#<_hw_extout1>	- (R) actual ExtOut1 value on controller
#<_hw_extout2>	- (R) actual ExtOut2 value on controller

## **M65 - Output TX**

Usage:	M65 Q	- Q = data send to TX
	M65 Q E	- Q = parameter number of first data location - E = data count

Send data to TX.

## **M66 - Output I2C**

Usage:	M66 H Q	- H = I2C address in 8bit mode - Q = data send to I2C
	M66 H Q E <R> <D>	- H = I2C address in 8bit mode - Q = parameter number of first data location - E = data count - R (optional) = parameter number for returned data - D (optional) = returned data count

Send and receive data from I2C.

## **M67 - Input**

Usage:	M67 <P> Q	- P (optional) = pin number - Q = parameter number where result is stored
--------	-----------	--

Read Input port value.

## **M68 - Input ExtIn**

Usage:	M68 H <P> Q <E>	- H = ExtIn SSEL value (1 or 2) - P (optional) = pin number - Q = parameter number where result is stored - E (optional) = data count
--------	-----------------	--

Read ExtIn port value.

## **M69 - Input Aux**

Usage:	M69 P Q	- P = Aux pin number (1-4) - Q = parameter number where result is stored
--------	---------	---

Read AUX pin value.

# Modal State

## M70 - Modal State Store

Usage: M70

Store current modal state. Stored modal state can be restored with M72. Modal state is stored only in current call level. Storing/restoring modal state between call levels is not possible.

Stored modal state values are:

- Units (G20/G21)
- Plane (G17/G18/G19)
- DistanceMode (G90/G91)
- DistanceModeIJK (G90.1/G91.1)
- DistanceModeABC (G90.2/G91.2)
- PolarMode (G15/G16)
- CycleReturnMode (G98/G99)
- FeedMode (G93/G94/G95)
- FeedSpeed (F)
- SpindleMode (G96/G97)
- SpindleSpeed(S)
- Spindle (M3/M4/M5)
- Coolant (M7/M8/M9)
- Overrides (M48/M49/M50/51)
- Options (M53/M54/M55/M56)
- CoordinateSystem (G54-G59)
- LatheMode (G7/G8)
- AxisShift (G52)
- AxisScale (G50/G51)
- AxisRotation (G68/G69)
- ToolCompensation (G40/G41/G42)
- BlendMode (G61/G64)

It is also possible to store/restore modal state manually, one by one. For example:

#<tmp> = #<\_units> ; units are stored to tmp

#<\_units> = #<tmp> ; units are restored from tmp

Parameters:

#<\_level> - (R) current call level

## M71 - Modal State Invalidate

Usage: M71

Invalidates stored modal state. Modal state can no longer be restored.

## M72 - Modal State Restore

Usage: M72

Restore modal state stored with M70 or M73 in current call level.

## M73 - Modal State Store & Autorestore

Usage: M73

Store current modal state and automatically restore on subroutine end. If stored state was invalidated with M71 it will not be restored. If program ends with M2, M30 or % modal state will not be restored.

## Other Codes

### F - Feed Speed

Usage:        Fn                - F = feed speed

Set feed speed. Value depends on feed mode which can be „Units Per Minute“, „Units Per Revolution“ or „Inverse Time“. In inverse time F must be specified an every motion line.

Parameters:

#<_feedspeed>	- (RW) feed speed value dependant on current feed mode
#<_feedspeed_upm>	- (RW) feed speed value in units per minute
#<_feedspeed_upr>	- (RW) feed speed value in units per minute per revolution

### S - Spindle Speed

Usage:        Sn                - S = spindle speed

Set spindle speed. Value depends on spindle mode which can be „Revolution Per Minute“ or „Constant Surface Speed“.

Parameters:

#<_spindlespeed>	- (RW) spindle speed value dependant on current spindle mode
#<_spindlespeed_rpm>	- (RW) spindle speed value in units per minute mode
#<_spindlespeed_css>	- (RW) spindle speed value in constant surface speed mode

### T - Select Tool

Usage:        Tn                - T = tool number

Select tool number that will be used in next tool change (M6)

Parameters:

#<_selected_tool>	- (RW) selected tool number
#<_current_tool>	- (RW) current tool number

# Table of Contents

<b>G-Codes.....</b>	<b>5</b>
G00 - Rapid Move.....	5
G01 - Linear Feed Move.....	5
G02 - Clockwise Arc Feed Move.....	5
G03 - Counter Clockwise Arc Feed Move.....	5
G04 - Dwell.....	6
G05 - Cubic Spline.....	6
G05.1 - Quadratic Spline.....	7
G05.2 - NURBS Block.....	8
G05.3 - NURBS Block End.....	8
G07 - Lathe Mode – Diameter .....	9
G08 - Lathe Mode – Radius.....	9
G09 - Stop & Sync.....	9
G10 L9 - Settings – Controller Position.....	10
G10 L3 - Settings – Transformation.....	10
G10 L1, L10 - Settings – Tool Table.....	11
G10 L2, L20 - Settings – Coordinate System.....	11
G12 - Clockwise Circle.....	12
G13 - Counter Clockwise Circle.....	12
G12.1 - Clockwise Spiral Pocket.....	12
G13.1 - Counter Clockwise Spiral Pocket.....	12
G15 - Polar Coordinate Cancel .....	13
G16 - Polar Coordinate Enable.....	13
G17 - XY Plane.....	14
G18 - ZX Plane.....	14
G19 - YZ Plane.....	14
G20 - Inch Units.....	14
G70 - Inch Units.....	14
G21 - Millimeter Units.....	14
G71 - Millimeter Units.....	14
G28 - Go To Home 1 Position.....	15
G28.1 - Set Home 1 Position.....	15
G30 - Go To Home 2 Position.....	15
G30.1 - Set Home 2 Position.....	15
G32 - Spindle Synchronized Motion / Threading.....	16
G33 - Spindle Synchronized Motion / Threading.....	16
G33.1 - Spindle Synchronized Motion With Return / Rigid Tapping.....	16
G31 - Probe.....	17
G38.1 - Probe.....	17
G38.2 - Probe.....	17
G38.3 - Probe.....	17
G38.4 - Probe.....	17
G38.5 - Probe.....	17
G40 - Tool Compensation Cancel.....	18
G41 - Tool Compensation Left.....	18
G42 - Tool Compensation Right.....	18
G41.1 - Tool Compensation Dynamic Left.....	18
G42.1 - Tool Compensation Dynamic Right.....	18
G43 - Tool Offset+ Enable.....	19
G44 - Tool Offset- Enable.....	19
G43.1 - Tool Offset+ Enable.....	19
G44.1 - Tool Offset- Enable.....	19
G49 - Tool Offset Cancel.....	19
G50 - Axes Scale Cancel.....	20

G51 - Axes Scale Enable.....	20
G52 - Axes Offset.....	20
G52.1 - Axes Offset Cancel.....	20
G53 - Machine Coordinate System.....	21
G54 - Coordinate System 1.....	21
G55 - Coordinate System 2.....	21
G56 - Coordinate System 3.....	21
G57 - Coordinate System 4.....	21
G58 - Coordinate System 5.....	21
G59 - Coordinate System 6.....	21
G59.1 - Coordinate System 7.....	21
G59.2 - Coordinate System 8.....	21
G59.3 - Coordinate System 9.....	21
G54.1 - Coordinate System P.....	21
G59 P- Coordinate System P.....	21
G61 - Blend Cancel.....	22
G64 - Blend Enable.....	22
G68 - Axes Rotate Enable.....	23
G69 - Axes Rotate Cancel.....	23
G73 - Drilling, Chip Break.....	24
G74 - Tapping, Left-Hand.....	24
G76 - Threading.....	24
G80 - Cancel Motion.....	24
G81 - Drilling.....	24
G82 - Drilling, Dwell.....	24
G83 - Drilling, Peck.....	24
G84 - Tapping, Right-Hand.....	25
G85 - Boring, Retract at Feed, Spindle On.....	25
G86 - Boring, Retract at Rapid, Spindle Off.....	25
G87 - Back Boring.....	25
G88 - Boring, Manual Retract, Spindle Off.....	25
G89 - Boring, Dwell, Retract at Feed, Spindle On.....	25
G90 - Distance Mode – Absolute.....	26
G91 - Distance Mode – Incremental.....	26
G90.1 - Distance Mode – IJK Absolute.....	26
G91.1 - Distance Mode – IJK Incremental.....	26
G90.2 - Distance Mode – ABC Absolute.....	26
G91.2 - Distance Mode – ABC Incremental.....	26
G92 - Working Offset.....	27
G92.1 - Working Offset Cancel .....	27
G93 - Feed Mode - Inverse Time.....	28
G94 - Feed Mode - Units per Minute.....	28
G95 - Feed Mode - Units per Revolution.....	28
G96 - Spindle Mode – CSS.....	29
G97 - Spindle Mode – RPM.....	29
G98 - Cycle Return - Initial Z Point .....	29
G99 - Cycle Return - R Point .....	29
<b>M-Codes.....</b>	<b>30</b>
Program.....	30
M0 - Pause.....	30
M1 - Optional pause.....	30
M2 - Program End.....	30
Spindle.....	31
M3 - Spindle CW.....	31
M4 - Spindle CCW.....	31
M5 - Spindle Stop.....	31



Tools.....	32
M6 - Tool Change.....	32
M61 - Tool Select.....	32
Coolant.....	33
M7 - Mist On.....	33
M17 - Mist Off/On.....	33
M8 - Flood On.....	33
M18 - Flood Off/On.....	33
M9 - Mist & Flood Off.....	33
Overrides and options.....	34
M48 - Enable Override Feed & Spindle.....	34
M49 - Disable Override Feed & Spindle.....	34
M50 - Enable/Disable Override Feed.....	34
M51 - Enable/Disable Override Spindle.....	34
M53 - Enable/Disable Pause.....	34
M54 - Enable/Disable THC.....	34
M55 - Enable/Disable Transformations.....	34
M56 - Enable/Disable Warp.....	34
M57 - Custom Scripts Enable/Disable.....	34
Inputs & Outputs.....	35
M59 - Wait For Input.....	35
M62 - Output.....	35
M63 - Output PWM.....	35
M64 - Output ExtOut.....	35
M65 - Output TX.....	36
M66 - Output I2C.....	36
M67 - Input.....	36
M68 - Input ExtIn.....	36
M69 - Input Aux.....	36
Modal State.....	37
M70 - Modal State Store.....	37
M71 - Modal State Invalidate.....	37
M72 - Modal State Restore.....	37
M73 - Modal State Store & Autorestore.....	37
<b>Other Codes.....</b>	<b>38</b>
F - Feed Speed.....	38
S - Spindle Speed.....	38
T - Select Tool.....	38