

DeskCNC G-code Syntax

Note: Sections in RED are preliminary, untested, under development, or future features.

All words can be entered as upper or lower case. X9.999 or x9.999, either is acceptable to DeskCNC g-code interpreter. Decimals shown are optional, but are recommended to insure numeric intent.

Axis words:

2D-3D milling and routing

Linear Axes units in inches (or mm with 9.999)

X9.9999

Y9.9999

Z9.9999 Spindle

4th axis rotary

A9.999 Rotary axis units in degrees

4-axis foam cutting

Linear Axes units in inches (or mm with 9.999)

X9.9999

Y9.9999

U9.9999

V9.9999

Lathe

Linear Axes units in inches (or mm with 9.999)

X9.9999 Radius or diameter

Z9.9999 Longitudinal

Other words:

Line Numbers - N99999 unique names for lines of the program. Line numbers are optional on each line (integer)

Spindle Speed - S9999 in revolution per minute (integer)

Feed Rate - F99.99 in inches per minute or (mm/min) Modal

Pitch - P99.9 in threads per inch or threads per mm

I Incremental arc center position with respect to start position in X I9.999

J Incremental arc center position with respect to start position in Y J9.999

K Incremental arc center position with respect to start position in Z K9.999

Radius of arcs - R used in G02 and G03 R9.999

Tool Height - H, used with tool length offset (G43) H01 (integer)

Dwell time - P in seconds is used with G04 and canned cycles G81-G88. Also used for index in G10 command settings.

Program Name O%1 Specifies the name of a subprogram in a G65 subroutine call. Integer value in the range 01-99

Peck depth - Q is used in Drilling cycles Q.25

Tool selection - T to select a tool from the tool table T01 (integer), up to 50 tools may be used.

G - codes

G00 rapid positioning

G00 X9.999 Y9.999 Z9.999 A9.999

Positioning at maximum steps per unit, axes are not synchronized to reach destination at the same time. G00 or G0 may be used. Modal, the G00 may be left off lines with ensuing G00 format.

G01 linear interpolation

G01 X9.999 Y9.999 Z9.999 A9.999 F10.5

Coordinated linear positioning at programmed feed rate, axes are synchronized to reach destination at the same time. G01 or G1 may be used. Modal, after first usage, the G01 may be left off immediately following lines with G01 format. An initial feed rate must be programmed, with an MDI command, or with a feed rate as shown above.

Example showing modality for F and G01:

G01 X9.999 Y9.999 Z9.999 A9.999 F10.5
X8.999 Y9.999 Z9.999 A360.0
X7.999 Y7.999 Z6.999 A10.0
X6.999 Y3.999 Z3.999 A152.37

G02 circular interpolation Clockwise

G02 X9.9999 Y9.999 I9.999 J9.999 F10.0

Coordinated arc positioning at programmed feed rate, axes are synchronized to reach destination at the same time, maintaining uniform arc radius throughout movement. G02 or G2 may be used. Modal, after first usage, the G02 may be left off immediately following lines with G02 format. An initial feed rate must be programmed, with an MDI command, or with a feed rate as shown above.

X and Y are the end points of the arc.

I and J are the relative distance from the arc start to the center of the defined arc of X and Y respectively.

Controller defaults to G17, X-Y plane arcs at startup. Use G18 and G19 for formatting arcs in X-Z and Y-Z planes respectively.

G17 arc format: G02 X9.9999 Y9.999 I9.999 J9.999 F10.0

G18 arc format: G02 X9.9999 Z9.999 I9.999 K9.999 F10.0

G19 arc format: G02 Y9.9999 Z9.999 J9.999 K9.999 F10.0

Alternate arc format: R arc format is also supported

G02 X9.9999 Y9.999 R.250 F10.0

Where R is the radius of the defined arc and X and Y are the end points.

G03 circular interpolation Counter-Clockwise

Same as G02, except arc cutting direction is Counter clockwise instead of clockwise.

G04 Dwell

G04 P999.99

Upon execution the controller pauses for 999.99 seconds

G10 coordinate system origin setting

To set the coordinate values for the origin of a coordinate system, program

G10 L2 P99 X9.999 Y9.999 Z9.999 A9.999,

where the P number must evaluate to an integer in the range 1 to 9 (corresponding to G54 to G59.3) and all axis words are optional. The coordinates of the origin of the coordinate system specified by the P number are reset to the coordinate values given (in terms of the absolute coordinate system). Only those coordinates for which an axis word is included on the line will be reset.

G17 xy plane selection

The controller defaults to X-Y plane arcs see g02. Command is modal.

G18 xz plane selection

X-Z plane arcs see g02. Command is modal.

G19 yz plane selection

Y-Z plane arcs see g02. Command is modal.

G20 inch system selection

Following commands are in inch units

G21 millimeter system selection

Following commands are in mm units

G27 Verify Home

G27 is used to verify table position and perform an Emergency Stop if table position tolerances are beyond specification. G27 will home to a switch, record the switch location, and calculate the difference from this measured Home Location to the set Home Position. G27 should only be used after the machine has been homed using G30. No Home Offset is performed during the G27 call.

Syntax of G27 is...

G27 X0.001 Y0.002 Z0.003 F10

Where the axis values define the acceptable tolerance for table position error. The example above will home all 3 axes simultaneously to the Home Switches. G27 is only available when doing a Hard Home to physical switches. If an axis is not called out in the G27 command, then that axis is ignored during execution. For example, G27 X0.001 Z0.005 will only home/verify the XZ axes. The axis value defines the acceptable tolerance. In the example, the X-axis tolerance is set to 0.001. If the table position is off by more than 0.001 then the G27 command will put the machine in E-Stop.

G28 Home Return (Soft Home)

G28 is used to move to a known location. The G28 position is set in the Work Coordinate Offset Table under menu Setup - Work Coordinate Offsets. G28 will simply move to this location.

G30 Home

G30 is used to Home to physical Home Switches. G30 will home to a switch, set the Home position, and then move to an Offset Location. If an axis is not called out in the G30 command, then that axis is ignored during execution.

For example: G30 X.5 Z.25 F20

Homes the X and Z axes simultaneously at a feed rate of 20. The position of the XZ coordinates will be set from values entered in menu Setup - Machine Setup - Axes Setup - Home Position. The axis values set the Home Offset. After the machine finds the Home Switches, the XZ axis will move to position 0.5 and 0.25 respectively.

Using the Homing Script:

Any valid G-Code Block sequence can be executed when a Home Button is pressed. The Homing sequence is defined for each axis separately as well as for the 'All' button. Homing Scripts are defined in menu Setup - Machine Setup - Home. One example of Homing using the All button follows...

```
M5          (Turn Off Spindle)
G92.2 G54   (Zero Work Coordinate Offsets)
G30 X.5 F100 (Home the X Axis at a fast feed rate and offset by 0.5)
G30 X.5 F1   (Re-Home the X Axis at a slower feed rate and offset)
G30 Y.5 F100 (Home the Y Axis at a fast feed rate and offset by 0.5)
G30 Y.5 F1   (Re-Home the Y Axis at a slower feed rate and offset)
G30 Z.5 F100 (Home the Z Axis at a fast feed rate and offset by 0.5)
G30 Z.5 F1   (Re-Home the Z Axis at a slower feed rate and offset)
```

G33 Lathe Thread Cycle

The single pass threading cycle requires a quadrature encoder synchronized to spindle rotation. The format of the thread is:

```
P33 X9.999 Z9.999 P25
```

Where X and or/or Z can be specified to determine the end point of the thread cycle. Angled threads are cut when X and Z vary from the start location.

P is the pitch of the thread in threads/unit, ie P20= 20 threads per inch

G38.2 Straight Probe (Under development)

Program G38.2 X- Y- Z- A- to perform a straight probe operation. The rotational axis words are allowed, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move. The linear axis words are optional, except that at least one of them must be used. The tool in the spindle must be a probe.

It is an error if:

- the current point is less than 0.254 millimeter or 0.01 inch from the programmed point.
- G38.2 is used in inverse time feed rate mode,
- any rotational axis is commanded to move,
- no X, Y, or Z-axis word is used.

In response to this command, the machine moves the controlled point (which should be at the end of the probe tip) in a straight line at the current feed rate toward the programmed point. If the probe trips, the probe is retracted slightly from the trip point at the end of command execution. If the probe does not trip even after overshooting the programmed point slightly, an error is signaled.

After successful probing, parameters 5061 to 5066 will be set to the coordinates of the location of the controlled point at the time the probe tripped.

Using the straight probe command, if the probe shank is kept nominally parallel to the Z-axis (i.e., any rotational axes are at zero) and the tool length offset for the probe is used, so that the controlled point is at the end of the tip of the probe:

- without additional knowledge about the probe, the parallelism of a face of a part to the XY-plane may, for example, be found.
- if the probe tip radius is known approximately, the parallelism of a face of a part to the YZ or XZ-plane may, for example, be found.
- if the shank of the probe is known to be well-aligned with the Z-axis and the probe tip radius is known approximately, the center of a circular hole, may, for example, be found.
- if the shank of the probe is known to be well-aligned with the Z-axis and the probe tip radius is known precisely, more uses may be made of the straight probe command, such as finding the diameter of a circular hole.

If the straightness of the probe shank cannot be adjusted to high accuracy, it is desirable to know the effective radii of the probe tip in at least the +X, -X, +Y, and -Y directions. These quantities can be stored in parameters either by being included in the parameter file or by being set in an RS274/NGC program.

As a usable example, the code for finding the center and diameter of a circular hole is shown in Table 6. For this code to yield accurate results, the probe shank must be well-aligned with the Z-axis, the cross section of the probe tip at its widest point must be very circular, and the probe tip radius (i.e., the radius of the circular cross section) must be known precisely. If the probe tip radius is known only approximately (but the other conditions hold), the location of the hole center will still be accurate, but the hole diameter will not.

In the following example, an entry of the form <description of number> is meant to be replaced by an actual number that matches the description of number. After this section of code has executed, the X-value of the center will be in parameter 1041, the Y-value of the center in parameter 1022, and the diameter in parameter 1034. In addition, the diameter parallel to the X-axis will be in parameter 1024, the diameter parallel to the Y-axis in parameter 1014, and the difference (an indicator of circularity) in parameter 1035. The probe tip will be in the hole at the XY center of the hole.

The example does not include a tool change to put a probe in the spindle. Add the tool change code at the beginning, if needed.

N010 (probe to find center and diameter of circular hole)
N020 (This program will not run as given here. You have to)

N030 (insert numbers in place of <description of number>.)
 N040 (Delete lines N020, N030, and N040 when you do that.)
 N050 G0 Z <Z-value of retracted position> F <feed rate>
 N060 #1001=<nominal X-value of hole center>
 N070 #1002=<nominal Y-value of hole center>
 N080 #1003=<some Z-value inside the hole>
 N090 #1004=<probe tip radius>
 N100 #1005=[<nominal hole diameter>/2.0 - #1004]
 N110 G0 X#1001 Y#1002 (move above nominal hole center)
 N120 G0 Z#1003 (move into hole - to be cautious, substitute G1 for G0 here)
 N130 G38.2 X[#1001 + #1005] (probe +X side of hole)
 N140 #1011=#5061 (save results)
 N150 G0 X#1001 Y#1002 (back to center of hole)
 N160 G38.2 X[#1001 - #1005] (probe -X side of hole)
 N170 #1021=[[#1011 + #5061] / 2.0] (find pretty good X-value of hole center)
 N180 G0 X#1021 Y#1002 (back to center of hole)
 N190 G38.2 Y[#1002 + #1005] (probe +Y side of hole)
 N200 #1012=#5062 (save results)
 N210 G0 X#1021 Y#1002 (back to center of hole)
 N220 G38.2 Y[#1002 - #1005] (probe -Y side of hole)
 N230 #1022=[[#1012 + #5062] / 2.0] (find very good Y-value of hole center)
 N240 #1014=[#1012 - #5062 + [2 * #1004]] (find hole diameter in Y-direction)
 N250 G0 X#1021 Y#1022 (back to center of hole)
 N260 G38.2 X[#1021 + #1005] (probe +X side of hole)
 N270 #1031=#5061 (save results)
 N280 G0 X#1021 Y#1022 (back to center of hole)
 N290 G38.2 X[#1021 - #1005] (probe -X side of hole)
 N300 #1041=[[#1031 + #5061] / 2.0] (find very good X-value of hole center)
 N310 #1024=[#1031 - #5061 + [2 * #1004]] (find hole diameter in X-direction)
 N320 #1034=[[#1014 + #1024] / 2.0] (find average hole diameter)
 N330 #1035=[#1024 - #1014] (find difference in hole diameters)
 N340 G0 X#1041 Y#1022 (back to center of hole)
 N350 M2

G40 cancel cutter diameter compensation

End cutter diameter compensation. A linear movement is required to properly end cutter compensation.

G41 start cutter diameter compensation left

Start Cutter compensation to the left of the contour. A linear movement is required for the first line of motion. A tool and Diameter offset must be in effect either prior to execution of a G41-command, or on the same line. Tool will be offset by one radius on the left side of ensuing cuts. Corners sharper than the cutter radius will error. G17 – X-Y plane must be active to use Cutter compensation.

```

T01 D01
G00 X0 Y.5
G41 G01 X1.0 Y1.0 F15.0

```

G42 start cutter diameter compensation right

G42 is the same as G41, except compensation to the right side of the contour.

G43 tool length offset (plus)

G43 H01

To use a tool length offset, program G43 H-, where the H number is the desired index in the tool table. It is expected that all entries in this table will be positive. The H number should be, but does not have to be, the same as the slot number of the tool currently in the spindle. It is OK for the H number to be zero; an offset value of zero will be used.

G49 cancel tool length offset

Cancel all tool length offsets in effect

G53 motion in machine coordinate system

For linear motion to a point expressed in absolute coordinates, program G1 G53 X- Y- Z- A- (or use G0 instead of G1), where all the axis words are optional, except that at least one must be used. The G0 or G1 is optional if it is the current motion mode. G53 is not modal and must be programmed on each line on which it is intended to be active. This will produce coordinated linear motion to the programmed point. If G1 is active, the speed of motion is the current feed rate (or slower if the machine will not go that fast). If G0 is active, the speed of motion is the current traverse rate (or slower if the machine will not go that fast).

G54 use preset work coordinate system 1

G55 use preset work coordinate system 2

G56 use preset work coordinate system 3

G57 use preset work coordinate system 4

G58 work coordinate 5

G59 work coordinate 6

G59.1 work coordinate 7

G59.2 work coordinate 8

G59.3 work coordinate 9

Work coordinate offsets names. Each offset can store an offset distance for X-Y-Z-A coordinates. You can select one of the nine systems by using G54, G55, G56, G57, G58, G59, G59.1, G59.2, or G59.3

G65 Subroutine

Formatting for G65 is as follows:

N100 (Subroutine sample)

N110 G65 P1

N120 M02

N130 O%1

N140 (Subroutine Body)

N150 M99

Explanation:

N110 G65 P1 (The next line to be executed is the first line of Subroutine O%1)

-Additional program instructions may be placed between N110 & N120

N120 M02 (End Program)

Subroutine names currently must be Integers only, between 1-99 of the form...

N130 O%1 (Subroutine 1, Use the letter O "OH", not zero for the word address)

-place the subroutine instruction lines between N130 and N150, ie G01 X1.5 Y1.25, etc

N150 M99 (Return to the line following the G65 that called this subroutine.

Multiple subroutines may be placed within the same file.

```
N101 O%1
-Additional program instructions between N101 & N107
N107 M99
N109 O%2
-Additional program instructions between N109 & N111
N111 M99
N121 O%3
-Additional program instructions between N121 & N127
N127 M99
N131 O%4
-Additional program instructions between N131 & N137
N137 M99
```

Here is a program that drills 4 holes, 1.0 deep, using a subroutine..

```
N100 G90
N110 G0 X.5 Y0.0 Z.5
N120 G65 P1
N130 G0 X1.0
N140 G65 P1
N150 G0 X1.5
N160 G65 P1
N170 G0 X2.0
N180 G65 P1
N190 M02
N200 O%1
N210 G1 Z-1.0 F1.0
N220 G0 Z.5
N230 M99
```

G80 cancel motion mode (including any canned cycle)

Any canned cycle is finished (cancelled) by this command. A G00 will also serve the same function.

All canned cycles are performed with respect to the currently selected plane. Any of the three planes (XY, YZ, ZX) may be selected. Throughout this section, most of the descriptions assume the XY-plane has been selected. The behavior is always analogous if the YZ or XZ-plane is selected.

Rotational axis words are allowed in canned cycles, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move.

All canned cycles use X, Y, R, and Z numbers in the NC code. These numbers are used to determine X, Y, R, and Z positions. The R (usually meaning retract) position is along the axis perpendicular to the currently selected plane (Z-axis for XY-plane, X-axis for YZ-plane, Y-axis for XZ-plane). Some canned cycles use additional arguments.

For canned cycles, we will call a number "sticky" if, when the same cycle is used on several lines of code in a row, the number must be used the first time, but is optional on the rest of the lines. Sticky numbers keep their value on the rest of the lines if they are not explicitly programmed to be different. The R number is always sticky.

In incremental distance mode: when the XY-plane is selected, X, Y, and R numbers are treated as increments to the current position and Z as an increment from the Z-axis position before the move involving Z takes place; when the YZ or XZ-plane is selected, treatment of the axis words is analogous. In absolute distance mode, the X, Y, R, and Z numbers are absolute positions in the current coordinate system.

The L number is optional and represents the number of repeats. L=0 is not allowed. If the repeat feature is used, it is normally used in incremental distance mode, so that the same sequence of motions is repeated in several equally spaced places along a straight line. In absolute distance mode, L > 1 means "do the same cycle in the same place several times." Omitting the L word is equivalent to specifying L=1. The L number is not sticky.

When L>1 in incremental mode with the XY-plane selected, the X and Y positions are determined by adding the given X and Y numbers either to the current X and Y positions (on the first go-around) or to the X and Y positions at the end of the previous go-around (on the repetitions). The R and Z positions do not change during the repeats.

The height of the retract move at the end of each repeat (called "clear Z" in the descriptions below) is determined by the setting of the retract mode: either to the original Z position (if that is above the R position and the retract mode is G98, OLD_Z), or otherwise to the R position.

It is an error if:

- X, Y, and Z words are all missing during a canned cycle,
- a P number is required and a negative P number is used,
- an L number is used that does not evaluate to a positive integer,
- rotational axis motion is used during a canned cycle,
- inverse time feed rate is active during a canned cycle,
- cutter radius compensation is active during a canned cycle.

When the XY plane is active, the Z number is sticky, and it is an error if:

- the Z number is missing and the same canned cycle was not already active,
- the R number is less than the Z number.

When the XZ plane is active, the Y number is sticky, and it is an error if:

- the Y number is missing and the same canned cycle was not already active,
- the R number is less than the Y number.

When the YZ plane is active, the X number is sticky, and it is an error if:

- the X number is missing and the same canned cycle was not already active,
- the R number is less than the X number.

Preliminary and In-Between Motion

At the very beginning of the execution of any of the canned cycles, with the XY-plane selected, if the current Z position is below the R position, the Z-axis is traversed to the R position. This happens only once, regardless of the value of L.

In addition, at the beginning of the first cycle and each repeat, the following one or two moves are made:

1. a straight traverse parallel to the XY-plane to the given XY-position,
2. a straight traverse of the Z-axis only to the R position, if it is not already at the R position.

If the XZ or YZ plane is active, the preliminary and in-between motions are analogous.

G81 drilling canned cycle

The G81 cycle is intended for drilling. Program G81 X- Y- Z- A- R- L-

0. Preliminary motion, as described above.

1. Move the Z-axis only at the current feed rate to the Z position.
2. Retract the Z-axis at traverse rate to clear Z.

G82 chip-breaking drilling canned cycle

The G82 cycle is intended for drilling with dwell. Program G82 X- Y- Z- A- B- C- R- L- P-

0. Preliminary motion, as described above.

1. Move the Z-axis only at the current feed rate to the Z position.
2. Dwell for the P number of seconds.
3. Retract the Z-axis at traverse rate to clear Z.

G83 chip-breaking drilling canned cycle

The G83 cycle (often called peck drilling) is intended for deep drilling or milling with chip breaking. The retractions in this cycle clear the hole of chips and cut off any long stringers (which are common when drilling in aluminum). This cycle takes a Q number, which represents a "delta" increment along the Z-axis. Program G83 X- Y- Z- A- R- L- Q-

0. Preliminary motion, as described above.

1. Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.
2. Rapid back out to the clear_z.
3. Rapid back down to the current hole bottom, backed off a bit.
4. Repeat steps 1, 2, and 3 until the Z position is reached at step 1.
5. Retract the Z-axis at traverse rate to clear Z.

It is an error if:

- the Q number is negative or zero.

G84 right hand tapping cycle

The G84 cycle is intended for right-hand tapping with a tap tool.

Program G84 X- Y- Z- A- R- L-

0. Preliminary motion, as described above.

1. Start speed-feed synchronization.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Stop the spindle.
4. Start the spindle counterclockwise.
5. Retract the Z-axis at the current feed rate to clear Z.
6. If speed-feed synch was not on before the cycle started, stop it.
7. Stop the spindle.
8. Start the spindle clockwise.

The spindle must be turning clockwise before this cycle is used. It is an error if:

- the spindle is not turning clockwise before this cycle is executed.

With this cycle, the programmer must be sure to program the speed and feed in the correct proportion to match the pitch of threads being made. The relationship is that the spindle speed equals the feed rate times the pitch (in threads per length unit). For example, if the pitch is 2 threads per millimeter, the active length units are millimeters, and the feed rate has been set with the command F150, then the speed should be set with the command S300, since $150 \times 2 = 300$.

If the feed and speed override switches are enabled and not set at 100%, the one set at the lower setting will take effect. The speed and feed rates will still be synchronized.

G85 boring, no dwell, feed out canned cycle

The G85 cycle is intended for boring or reaming, but could be used for drilling or milling.
Program G85 X- Y- Z- A- R- L-

0. Preliminary motion, as described above.
1. Move the Z-axis only at the current feed rate to the Z position.
2. Retract the Z-axis at the current feed rate to clear Z.

G86 boring, spindle stop, rapid out canned cycle

The G86 cycle is intended for boring. This cycle uses a P number for the number of seconds to dwell.
Program G86 X- Y- Z- A- R- L- P-

0. Preliminary motion, as described above.
1. Move the Z-axis only at the current feed rate to the Z position.
2. Dwell for the P number of seconds.
3. Stop the spindle turning.
4. Retract the Z-axis at traverse rate to clear Z.
5. Restart the spindle in the direction it was going.

The spindle must be turning before this cycle is used. It is an error if:
· the spindle is not turning before this cycle is executed.

G87 back boring canned cycle

The G87 cycle is intended for back boring.
Program G87 X- Y- Z- A- R- L- I- J- K-

The situation, as shown in Figure 1, is that you have a through hole and you want to counter bore the bottom of hole. To do this you put an L-shaped tool in the spindle with a cutting surface on the UPPER side of its base. You stick it carefully through the hole when it is not spinning and is oriented so it fits through the hole, then you move it so the stem of the L is on the axis of the hole, start the spindle, and feed the tool upward to make the counter bore. Then you stop the tool, get it out of the hole, and restart it.

This cycle uses I and J numbers to indicate the position for inserting and removing the tool. I and J will always be increments from the X position and the Y position, regardless of the distance mode setting. This cycle also uses a K number to specify the position along the Z-axis of the controlled point top of the counter bore. The K number is a Z-value in the current coordinate system in absolute distance mode, and an increment (from the Z position) in incremental distance mode.

0. Preliminary motion, as described above.
1. Move at traverse rate parallel to the XY-plane to the point indicated by I and J.
2. Stop the spindle in a specific orientation.
3. Move the Z-axis only at traverse rate downward to the Z position.
4. Move at traverse rate parallel to the XY-plane to the X,Y location.
5. Start the spindle in the direction it was going before.
6. Move the Z-axis only at the given feed rate upward to the position indicated by K.
7. Move the Z-axis only at the given feed rate back down to the Z position.
8. Stop the spindle in the same orientation as before.

9. Move at traverse rate parallel to the XY-plane to the point indicated by I and J.
10. Move the Z-axis only at traverse rate to the clear Z.
11. Move at traverse rate parallel to the XY-plane to the specified X,Y location.
12. Restart the spindle in the direction it was going before.

When programming this cycle, the I and J numbers must be chosen so that when the tool is stopped in an oriented position, it will fit through the hole. Because different cutters are made differently, it may take some analysis and/or experimentation to determine appropriate values for I and J.

G88 boring, spindle stop, manual out canned cycle

The G88 cycle is intended for boring. This cycle uses a P word, where P specifies the number of seconds to dwell. Program G88 X- Y- Z- A- R- L- P-

0. Preliminary motion, as described above.
1. Move the Z-axis only at the current feed rate to the Z position.
2. Dwell for the P number of seconds.
3. Stop the spindle turning.
4. Stop the program so the operator can retract the spindle manually.
5. Restart the spindle in the direction it was going.

G89 boring, feed-in, dwell, feed-out cycle

The G89 cycle is intended for boring. This cycle uses a P number, where P specifies the number of seconds to dwell. program G89 X- Y- Z- A- B- C- R- L- P-

0. Preliminary motion, as described above.
1. Move the Z-axis only at the current feed rate to the Z position.
2. Dwell for the P number of seconds.
3. Retract the Z-axis at the current feed rate to clear Z.

G90 absolute distance mode

Modal. All following code is programmed in machine coordinates. Each command is relative to the absolute position of the machine coordinates.

G91 incremental distance mode

Modal. All following code is programmed in relative motion. Each command is relative to the last end point.

G92 offset coordinate systems (add XYZA values)

Offset the current coordinate system using G92 or G92.3. This offset will then apply to all program coordinate systems. This offset may be cancelled with G92.1 or G92.2.

To make the current point have the coordinates you want (without motion), program G92 X- Y- Z- A- , where the axis words contain the axis numbers you want. All axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed. It is an error if: all axis words are omitted.

When G92 is executed, the origin of the currently active coordinate system moves. To do this, origin offsets are calculated so that the coordinates of the current point with respect to the moved origin are as specified on the line containing the G92. In addition, parameters 5211 to 5216 are set to the X, Y, Z, and A-axis offsets. The offset for an axis is the amount the origin must be moved so that the coordinate of the controlled point on the axis has the specified value.

Here is an example. Suppose the current point is at X=4 in the currently specified coordinate system and the current X-axis offset is zero, then G92 x7 sets the X-axis offset to -3, sets parameter 5211 to -3, and causes the X-coordinate of the current point to be 7.

The axis offsets are always used when motion is specified in absolute distance mode using any of the nine coordinate systems (those designated by G54 - G59.3). Thus all nine coordinate systems are affected by G92.

Being in incremental distance mode has no effect on the action of G92.

Non-zero offsets may be already be in effect when the G92 is called. If this is the case, the new value of each offset is A+B, where A is what the offset would be if the old offset were zero, and B is the old offset. For example, after the previous example, the X-value of the current point is 7. If G92 x9 is then programmed, the new X-axis offset is -5, which is calculated by $[[7-9] + -3]$.

To reset axis offsets to zero, program G92.1 or G92.2. G92.1 sets parameters 5211 to 5216 to zero, whereas G92.2 leaves their current values alone.

To set the axis-offset values to the values given in parameters 5211 to 5216, program G92.3.

You can set axis offsets in one program and use the same offsets in another program. Program G92 in the first program. This will set parameters 5211 to 5216. Do not use G92.1 in the remainder of the first program. The parameter values will be saved when the first program exits and restored when the second one starts up. Use G92.3 near the beginning of the second program. That will restore the offsets saved in the first program. If other programs are to run between the the program that sets the offsets and the one that restores them, make a copy of the parameter file written by the first program and use it as the parameter file for the second program.

G92.2 cancel offset coordinate systems

Cancels coordinate system offsets effected by G92 or G92.3. This offset will then apply to all program coordinate systems. This offset may be cancelled with G92.1 or G92.2.

G92.3 offset coordinate systems (use stored XYZA values)

Use stored offsets to offset the current coordinate system.

G93 inverse time feed mode (untested)

In inverse time feed rate mode, an F word means the move should be completed in [one divided by the F number] minutes. For example, if the F number is 2.0, the move should be completed in half a minute.

When the inverse time feed rate mode is active, an F word must appear on every line which has a G1, G2, or G3 motion, and an F word on a line that does not have G1, G2, or G3 is ignored. Being in inverse time feed rate mode does not affect G0 (rapid traverse) motions. It is an error if: inverse time feed rate mode is active and a line with G1, G2, or G3 (explicitly or implicitly) does not have an F word.

G94 feed per minute mode

In units per minute feed rate mode, an F word (no, not that F word; we mean feedrate) is interpreted to mean the controlled point should move at a certain number of inches per minute, millimeters per minute, or degrees per minute, depending upon what length units are being used and which axis or axes are moving.

G95 feed per rev mode (Future development)

In units per rev feed rate mode, an F word (no, not that F word; we mean feedrate) is interpreted to mean the controlled point should move at a certain number of inches per revolution, or millimeters per revolution, depending upon what length units are being used and which axis or axes are moving. Used for lathe feed rate programming. Requires lathe spindle encoder installation.

G98 initial level return in canned cycles

When the spindle retracts during canned cycles, there is a choice of how far it retracts: (2) retract perpendicular to the selected plane to the position that axis was in just before the canned cycle started (unless that position is lower than the position indicated by the R word, in which case use the R word position). Remember that the R word has different meanings in absolute distance mode and incremental distance mode.

G99 R-point level return in canned cycles

When the spindle retracts during canned cycles, there is a choice of how far it retracts: (1) retract perpendicular to the selected plane to the position indicated by the R word. Remember that the R word has different meanings in absolute distance mode and incremental distance mode.

Miscellaneous M codes

/ Block Skip (no option button presently available)

Line of code to the right of this character is ignored.

M00 program stop

Program execution stops and machine waits for operator to press continue or start button. Program is not rewound.

M01 optional program stop (no option button presently available)

Same as M00, except that the stop only occurs if option stop is turned on.

M02 program end

Stops and ends program execution. Program is reqound to beginning.

M03 Spindle on Clockwise

Turns on the Spindle relay corresponding to clockwise rotation (CW).

M04 Spindle on Counter Clockwise

Turns on the Spindle relay corresponding to counter-clockwise rotation (CCW).

M05 Spindle Off

Turns off both CW and CCW spindle relays.

M06 tool change

Effects tool change and executes tool change script corresponding to the tool in the last Txx command.

A tool change script defined in the ToolChange.cfg file handles tool changes. This file should be edited in a standard Windows Text Editor. During a Tool Change, any valid G-Code Block sequence can be executed. There are separate commands for picking up and putting back each tool so that each tool can be picked/placed at any location. If a tool is called for and there is not a valid tool section in the ToolChange.cfg file, then the last valid tool section defined will be executed.

Example of a basic tool change script for Tool 1.

```
[TOOL1]
PLACE0=M05           (Turn Off Spindle)
PLACE1=G4 P4         (Pause for Spindle wind down)
PLACE2=G0 Z0.5       (Raise Z above Zero)
PLACE3=G0 X7.000 Y4.770 (Move to a location for manual tool change)
PLACE4=M0            (Wait for user to change tool and continue)
PICKUP0=M03          (Turn on Spindle)
PICKUP1=G4 P3        (Wait for Spindle wind up)
```

M07 Mist Coolant On

Turns on the Mist relay

M08 Flood Coolant On

Turns on the Flood relay

M09 Coolant Off

Turns off both the flood and the Mist relays.

M83 M3 Off

Used to turn off just the M03 relay, rather than both the M03 and the M04.

M84 M4 Off

Used to turn off just the M04 relay, rather than both the M03 and the M04

The M83/M84 are only available when you select 'Independent spindle' in Machine Setup.

Additional relay on and off commands:

M87 M7 Off

M88 M8 Off

M90 Aux1 0N

M91 Aux1 OFF

M92 Aux2 0N

M93 Aux2 OFF

M94 Aux3 0N

M95 Aux3 OFF

M96 Dynamic Tool Length offset, with preposition in X-Y

M96 moves to the Tool Sensor Location defined in menu Setup - Machine Setup - Digitizing Probe. Then 'down' to the tool sensor. M96 is used when the tool sensor is semi-permanently mounted to a fixed location on the machines table. See discussion of tool setting under M97.

M97 Dynamic Tool Length Offset, Without preposition in X-Y

M97 does not move to location but rather moves directly 'down' to the tool sensor. M97 is used when the tool sensor is manually moved/placed under the tool for measurement.

TLO's are measured using the M96 or M97 codes. M96 moves to the Tool Sensor Location defined in menu Setup - Machine Setup - Digitizing Probe. M97 does not move to location but rather moves directly 'down' to the tool sensor. M97 is used when the tool sensor is manually moved/placed under the tool for measurement. M96 is used when the tool sensor is semi-permanently mounted to a fixed location on the machines table.

Measuring TLO's:

1. If using M96, set the tool sensor location in menu Setup - Machine Setup - Tool Sensor - X/Y Location along with the Tool Sensor Height. The Sensor Height is relative for this method of calculating TLO's so it does not have to be entered exactly. Make certain that the Default Lim Polarity is set to Normally Closed when using the Digitizing Probe or Tool Sensor.
2. Tool Changes are executed using a Tx M6 combination. A tool needs to be loaded for the M96/M97 commands to function. Enter T1M6 in the MDI box and press Enter. The Tool 1 'Tool Change Script' will be executed (Tool Change Scripting). Tool 1 should now be in the Spindle.
3. Place the Tool Sensor under the tool and Enter M97 in the MDI box. TLO's are measured using the current feedrate. Enter a new feedrate (F) if warranted.
4. The Spindle will lower the Tool to the Sensor and record the TLO. The TLO will be active. The TLO for Tool 1 will NOT be saved in the Tool Library until you save the Tool Library from menu Setup - Tool Library - Save. You can measure all tools and then save the entire table.
5. Once a TLO has been measured, it will need to be active when machining. This is done with the G43 Hx command where x is the tool number. The G43 Hx may be placed in the Tool Change Script.
6. When the Spindle is Zeroed, the Active TLO is used.

Using TLO's when machining:

1. Place the first tool used in your GCode file in the Spindle. Do this by entering T1M6 (assumes Tool 1) in the MDI box. The tool change script for Tool 1 will be executed. The Tool Change Script should include the G43 H1 command to make the TLO for tool 1 active. The TLO readout in DeskCNC will display the current TLO.
2. Zero the tool to the part material. The Z Coordinate will reflect the active TLO.
3. Run the GCode file. All subsequent tool calls will place the tip of each tool at the proper Z Height according to their TLO.

M99 Return from subroutine.

This command is the last line of a subroutine segment. Returns program execution to the next sequential line after the calling line in the main program that called this subroutine.