Section 27
Position Event

Topics Covered in this Section:

- Introduction
- Position Event Data
- Cartesian or Polar Endpoint Description
- Cartesian Endpoint Description
- Polar Endpoint Description
- Quill Cycles
- Position Event Example

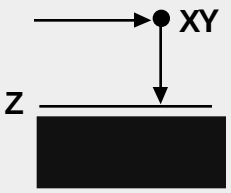
AUTOCON

PROGRAM MODE	N1800
0-POSITION	POSITION
1-LINEAR MILL	
2-ARC MILL	P0
3-FRAME MILL	X
4-CIRCLE MILL	Y
5-BOLT CIRCLE	Z
6-REPEAT	A
7-SUBROUTINE	B
8-DWELL	
9-M FUNCTION	V
	D
C-CAVITY MILL	
E-EIA	F
G-GRAPHICS	G
M-MACRO CALL	W
P-PROBE	K
R-ROTATE	Q
S-SET UP	L
T-TEXT	

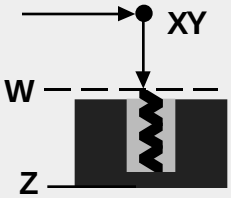
DEMOPART

G0-DrillCyclesOff

G0 Modal



Quill Cycle Modal



X = X Endpoint
Y = Y Endpoint
Z = Z Endpoint

V = Vector Angle
D = Distance

Quill Cycle Parameters:
G = G Cycle
F = Feedrate
K = Peck
Q = Peck Clearance
W = 1st Z Reference Plane
O = 2nd Z Reference Plane
L = Dwell

VIEW 1 OF 1

FRONT END GRAPHICS		DELETE EVENT	COPY/ STORE	PROGRAM DISPLAY	RUN CALC ASSIST	

PROGRAM MODE	N1800
0-POSITION	POSITION
1-LINEAR MILL	
2-ARC MILL	P0
3-FRAME MILL	X
4-CIRCLE MILL	Y
5-BOLT CIRCLE	Z
6-REPEAT	A
7-SUBROUTINE	B
8-DWELL	
9-M FUNCTION	V
	D
C-CAVITY MILL	
E-EIA	F
G-GRAPHICS	G
M-MACRO CALL	W
P-PROBE	K
R-ROTATE	Q
S-SET UP	L
T-TEXT	

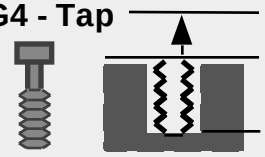
DEMOPART

G

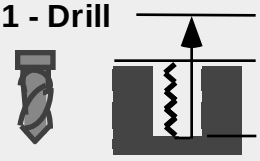
G0

Drill Cycle Off

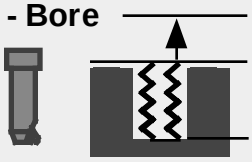
G4 - Tap



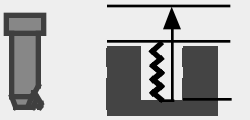
G1 - Drill



G5 - Bore

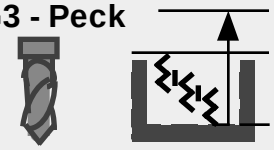


G2 - Counterbore

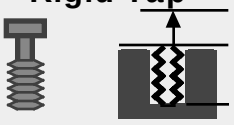


G7 - Dead Spindle

G3 - Peck



G8 - Rigid Tap



= Feed **= Rapid**

VIEW 1 OF 1

FRONT END GRAPHICS		DELETE EVENT	COPY/ STORE	PROGRAM DISPLAY	RUN CALC ASSIST	

POSITION EVENT - INTRODUCTION

This event positions the tool at rapid rate to an endpoint designated by Cartesian or polar coordinates. Programmable G Codes can add drill, tap, counterbore, peck, bore, or rigid tapping (with option) cycles to this event. A special G6 code lets you shift part coordinates.

POSITION EVENT DATA

Parameter	Description	Entry
N	Sequence number	Required
		Cartesian Endpoint
X or X/	X axis endpoint	Endpoint, see topic below
Y or Y/	Y axis endpoint	Endpoint, see topic below
Z or Z/	Z axis endpoint or G cycle depth	Endpoint, see topic below
		Polar Endpoint
P	Plane select P0=XY, P1=YZ, P2=ZX	Select P0 for G1-G5, or G7-G8
V	Vector Angle	Endpoint, see topic below
D	Distance	Endpoint, see topic below

G CYCLES and G CYCLE PARAMETERS (refer to the **lower** screen display)

G	G Cycle	G0-G5, G7-G8 Modal; G6 Nonmodal
F	Feedrate	Modal
K or K/	Peck	Modal
Q	Peck Clearance	Modal
W	1st Z Reference Plane	Modal
O	2nd Z Reference Plane	Modal
L	Dwell	Modal

G Cycle	Description	Required Parameters
G0	Turn off G Cycles	none
G1	Drill	W, Z, F
G2	Counterbore	W, Z, F, L
G3	Peck Drill	W, Z, F, K or K/, Q
G4	Tap	W, Z, F, L
G5	Bore	W, Z, F
G7	Dead Spindle Bore	W, Z, F
G8	Rigid Tap (option)	W, Z, F (see text)
G6	Load Position	see text

CARTESIAN or POLAR ENDPOINT DESCRIPTION

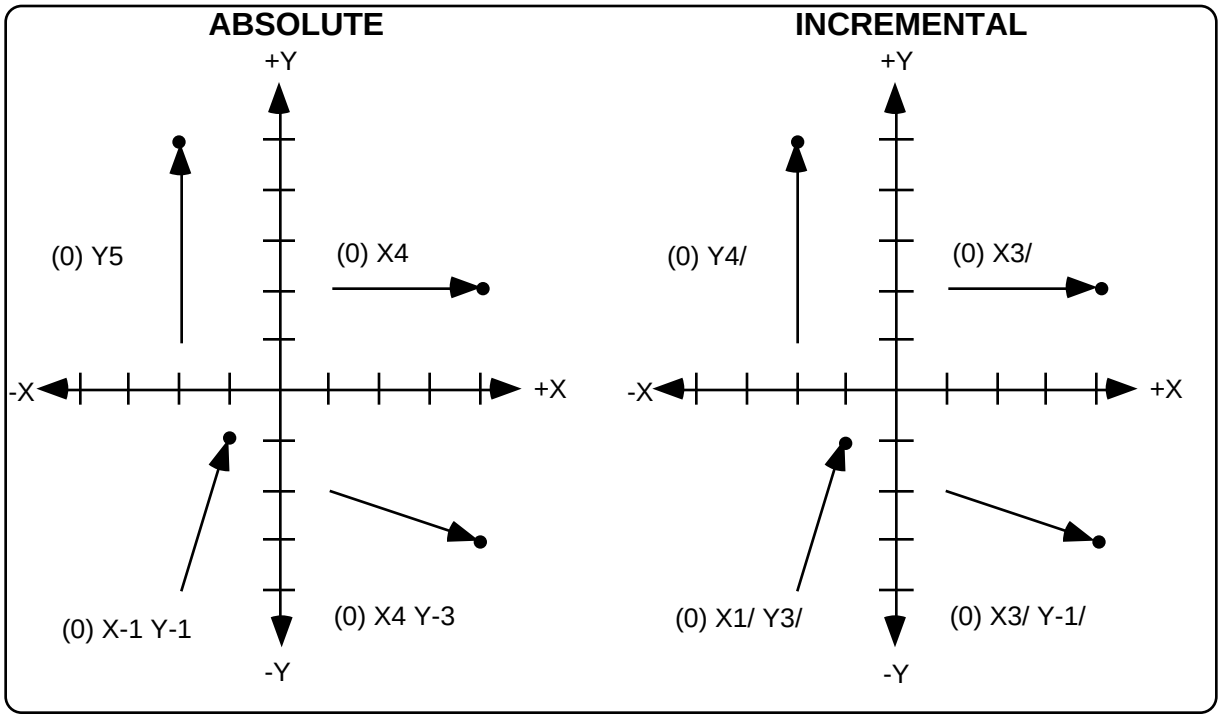
The G0 cycle, active at startup, is used for simple positioning. The event's endpoint can be completely described with Cartesian or polar entries, or with one Cartesian endpoint (X, Y, or Z) and an angle. If you run your part program through Calc-Assist, further endpoint descriptions can be programmed by combining Cartesian and polar data. Calc-Assist is explained in Part 5 of this manual.

G0 - POSITION WITH G CYCLES OFF

G0 is always active at start up. This causes the control to position without adding a fixed cycle. G0 remains in effect until a G1-G5 or G7-G8 executes in a Position or Bolt Circle event. To return to normal positioning after G1-G5 or G7-G8 events, program G0 in the next Position event. Executing an M02 or M30 - End of Program M code or pressing RESET when the control is not in cycle will also reestablish the G0 condition.

X, Y, Z - CARTESIAN ENDPOINT DESCRIPTION

Enter dimensions for each axis that must move to reach the endpoint. An axis endpoint may be programmed as a signed absolute or signed incremental entry. Examples of absolute and incremental Position events are shown **below**.

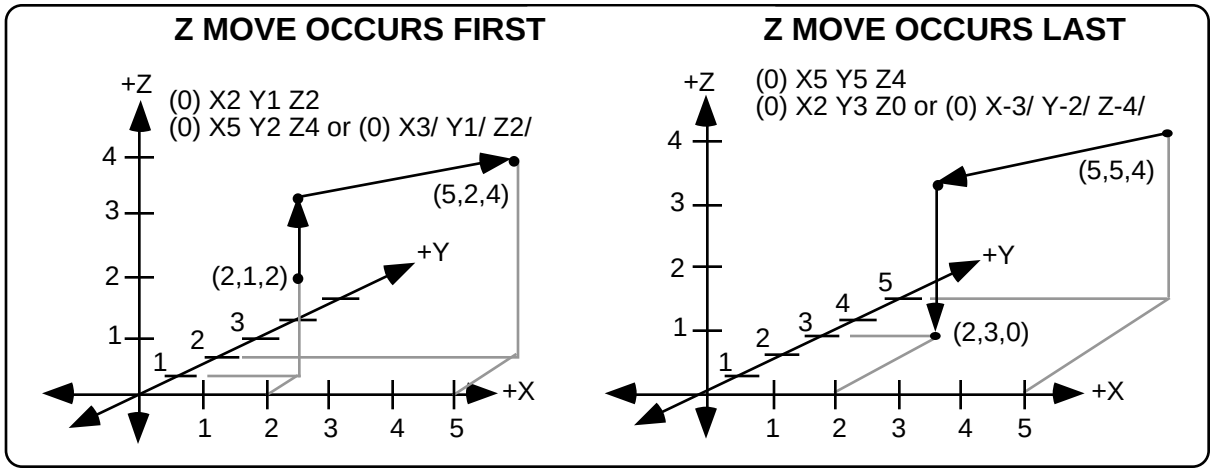


Absolute and incremental entries may be programmed in the same event. The move from X1, Y-2 to X4, Y-3 (lower right quadrant in upper illustrations), for example, can be programmed with any of the four events listed below.

(0) X4 Y-3	X and Y are absolute dimensions
(0) X4 Y-1/	X is absolute; Y is incremental
(0) X3/ Y-3	X is incremental; Y is absolute
(0) X3/ Y-1/	X and Y are incremental dimensions

If XY motion is programmed, the tool moves at rapid rate in a direct path from start point to the XY endpoint. In most systems, when Z is programmed with another axis, the Z motion occurs first for rapid Z motion away from the work, and last for rapid Z motion towards the work. Refer to **lower** examples.

Some controls are set to command simultaneous XYZ tool motion when a G0 Position event executes. However, all controls will move the Z axis independent of all other axes when a quill cycle (G1-G5, G7-G8) is modal, or is programmed in the event.



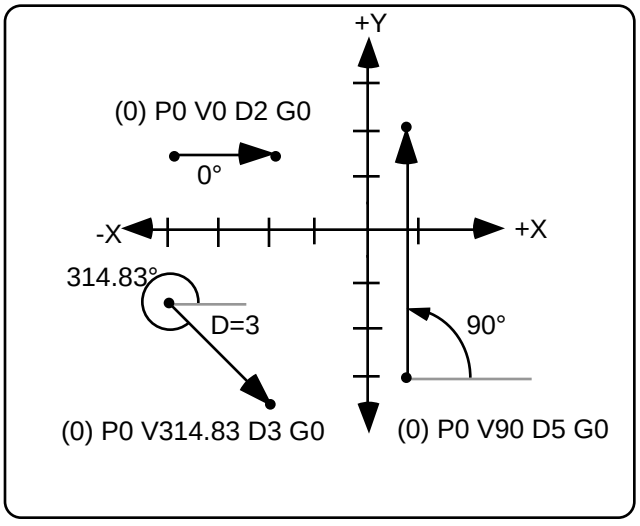
P , V, D - POLAR ENDPOINT DESCRIPTION

P - PLANE SELECT

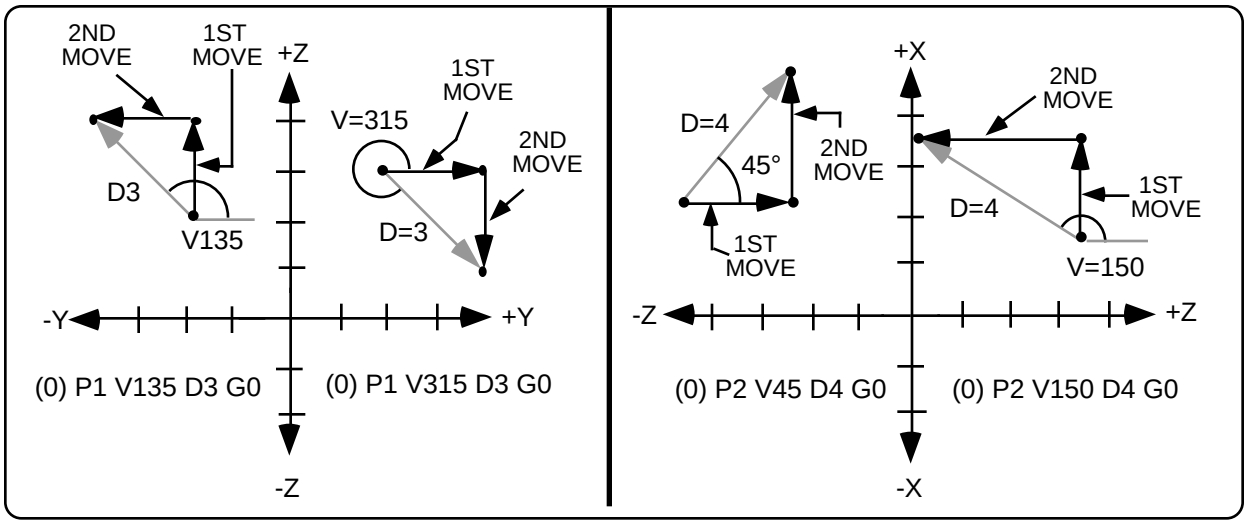
Polar coordinates define a move which occurs only in the selected plane. P is initially set to the XY plane P0 in each new event. Enter P1 to position in the YZ plane, or P2 to position in the ZX plane. Refer to examples, **following page**.

D, V - POLAR ENDPOINT DESCRIPTION

The tool moves in the selected plane P a distance D in direction V. D is the unsigned length of the move. V is an angle that opens counterclockwise about the start point from standard position. Standard position is drawn from the tool's start point in the +X direction in P0, +Y direction in P1, and +Z direction in P2. The counterclockwise direction is determined by looking at the plane from the positive axis that is perpendicular to the selected plane (you will be looking in perpendicular axis' negative direction). Example: to determine counterclockwise in the XY plane, you must view the plane from the +Z axis, looking in the -Z direction.



When a polar move is in the P0 (XY) plane, the tool moves directly along the programmed path at rapid rate. Although a YZ or ZX polar move is similarly programmed, Z axis motion does not occur simultaneously with the other axes.[1]



QUILL CYCLES G1-G5, G7-G8

A Position event with Cartesian or Polar coordinates or a Bolt Circle event may program G1-G5 or G7-G8 cycles The G cycle is modal, repeating in a following Position or Bolt Circle event if the following event does not program G.

The G cycles are:

G1 - drill	G4 - tap	G7 - dead spindle bore
G2 - counterbore	G5 - bore	G8 - rigid tapping (option)
G3 - peck drill		

If a G1-G5 or G7-G8 Position event programs XY motion, the XY motion occurs first (to position the tool over the center of the hole) then the Z axis advance and G cycle motion occur. **If no XY motion is required, enter X0/ or Y0/ to allow execution of the G cycle.**[2]

MODAL EIA/ISO QUILL CYCLES

Press the ISO ASSIST key to display a list of quill cycles available in the EIA event (they are generally in the G73-G89 range). There are some cycles in the EIA event that have capabilities beyond the G cycles in the Position event. It is important to note that the EIA quill cycles and the Position event quill cycles are interactively modal. In other words, a quill cycle established in an EIA event will execute in a following Position event and vice versa. Program G0 in the Position event if you must cancel the active quill cycle.

---

1. Your machine supplier may configure your system to simultaneously rapid X, Y, and Z during G0 Position events. Z axis motion will always occur independent of other axis motion when a quill cycle (G1-G5, G7-G8) executes. Refer to Section 2.
2. While a quill cycles is active, execution of a Position event that does not contain an X or Y endpoint will not cause any tool motion. All modal values programmed in this event, however, will be updated by the control and will be used in the next Position event that programs an X or Y endpoint. This programming method lets

PARAMETERS USED IN G CYCLES

W - 1st Z AXIS REFERENCE PLANE

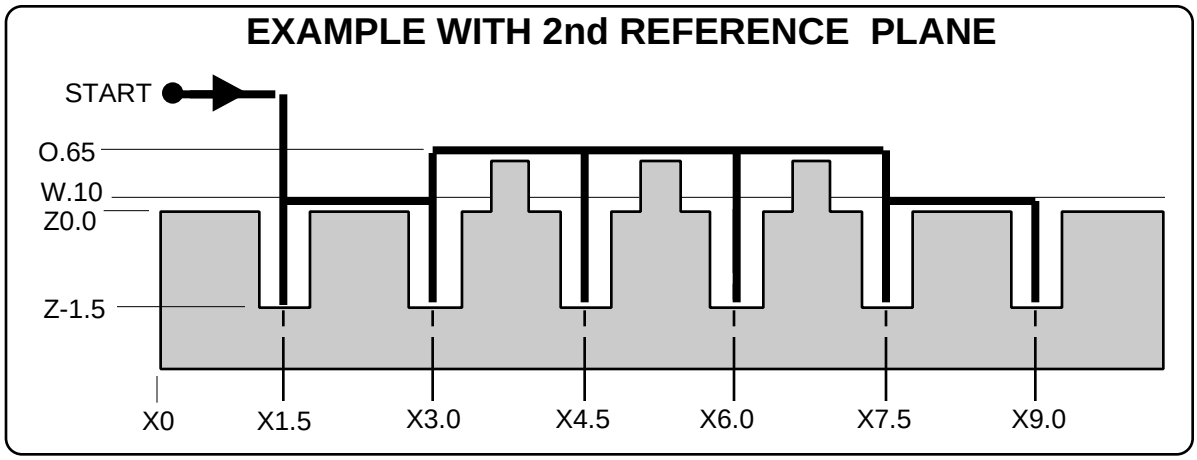
W is a Z axis reference plane that is parallel to the XY plane. Program W as the signed, absolute coordinate that locates the reference plane slightly above the part surface (we typically suggest a value of .100" or 2.0 mm assuming that the top of the work surface has been set to Z0). Z axis advance motion will slow from rapid to feedrate F as soon as the tool tip reaches W. The tool also returns to this plane at the end of each G cycle whenever a 2nd reference plane (letter O) is not active. W is modal with all following events that require a reference plane W.

Note: This entry is programmed with the letter "I" if your control is equipped with an auxiliary axis that is programmed with the letter "W". Refer to Section 2.

O - 2nd Z AXIS REFERENCE PLANE

O is a 2nd Z axis reference plane parallel to the XY plane. Program O as the signed, absolute coordinate that locates the Z axis position where you want the tool to return to at the end of every G cycle. O is an optional entry; if not programmed, it will default to the value of the W plane.

The easiest way to program the 2nd reference plane is to establish a rule that you will **always** program W in **every** Position or EIA event that programs the letter O. Although you are not required to follow this rule, it will make your program easier to understand since you will not have to scan through your entire program to learn the relationship between the two reference planes. Complete rules for programming the 2nd reference plane appear after the example below.



N010	(9)	T02 H02 D02 M06 E01	• use tool #2 and fixture offset #1
N020	(9)	M03 S1250	• turn spindle on
N030	(0)	X0 Y1 Z2 G0	• position tool above work
N040	(0)	X1.5 G1 Z-1.5 F12 W.1	• drill hole, 2nd ref. plane defaults to .10"
N050	(0)	X3.0 O.65 W.1	• drill hole #2, tool returns to O.65"
N060	(0)	X4.5	• drill hole #3, tool returns to O.65"
N070	(0)	X6.0	• drill hole #4, tool returns to O.65"
N080	(0)	X7.5 O.1 W.1	• drill hole #5, set O to .10"
N090	(0)	X9.0	• drill hole #6, tool returns to O.1
N100	(9)	M30	• end of program

Rules for programming a 2nd reference plane - if you decide not to follow the rule of programming W in every event that programs the 2nd reference plane, you need to be aware of the following rules that govern the O plane.

- The O reference plane is canceled when an M02, M30, Position event G0, or EIA event G80 command executes, or when the RESET button is pressed when the control is not in cycle. The control cancels the 2nd reference plane by setting it equal to the modal value of the 1st reference plane W.

- Until O is established for the first time in a part program (or established for the first time following a G0/G80 quill cycle cancel command), the control will always set the 2nd reference plane equal to the 1st reference plane. The 2nd reference plane is considered to be established when it has been programmed to a different value than W.

- Once O is established, it will be maintained independent of the W plane (i.e., programming W does not affect O; programming O does not affect W). As an exception to this rule, programming W and O to the same value will, in effect, cancel O.

F - FEEDRATE

The feed portion of a quill cycle occurs at feedrate F. Feedrate F is modal with all events that program feedrate.

Z - DEPTH

Z is the end depth of the G cycle. Enter Z as a signed Z axis coordinate or Z/ as a signed distance from W. Z or Z/ is modal with following G1-G5 or G7-G8 events.

L - DWELL

Tap and counterbore cycles can program L seconds of dwell. L is modal with following G2 and G4 Position and Bolt Circle events.

K or K/ - PECK

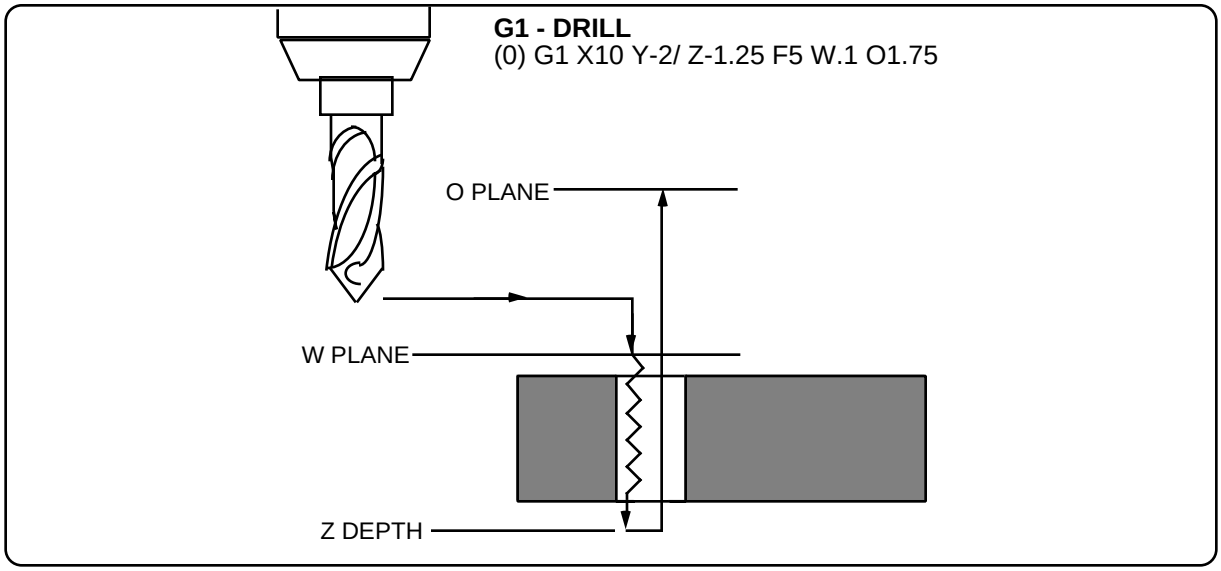
In the G3 cycle, a unit depth of K is drilled per peck. K is unsigned. If K is absolute, each retract motion will return the tool tip to W. If K is incremental, each retract motion will return the tool tip to Q. K or K/ is modal with following Position and Bolt Circle peck drill events.

Q - PECK CLEARANCE DIMENSION

Q is a clearance dimension above each K/ peck in the G3 peck cycle. If Q is not programmed, the default value is 0.1" (2.5mm). Q is used only when peck depth K is programmed as an incremental dimension.

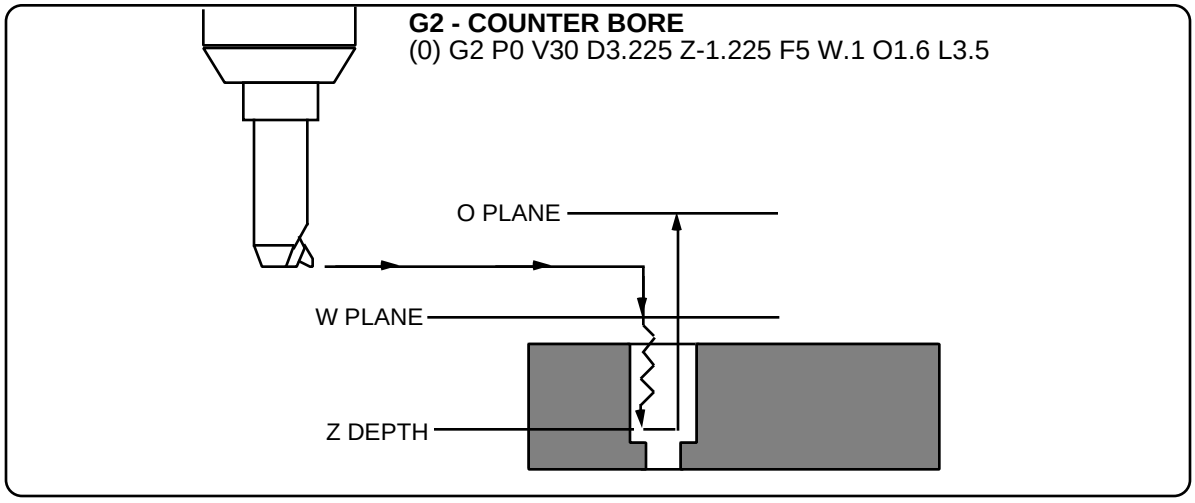
G1 - DRILL CYCLE

The drill tip slows from rapid to feedrate F at reference plane W, continues the feed to Z, then rapids to the O plane. (See illustration **below**).



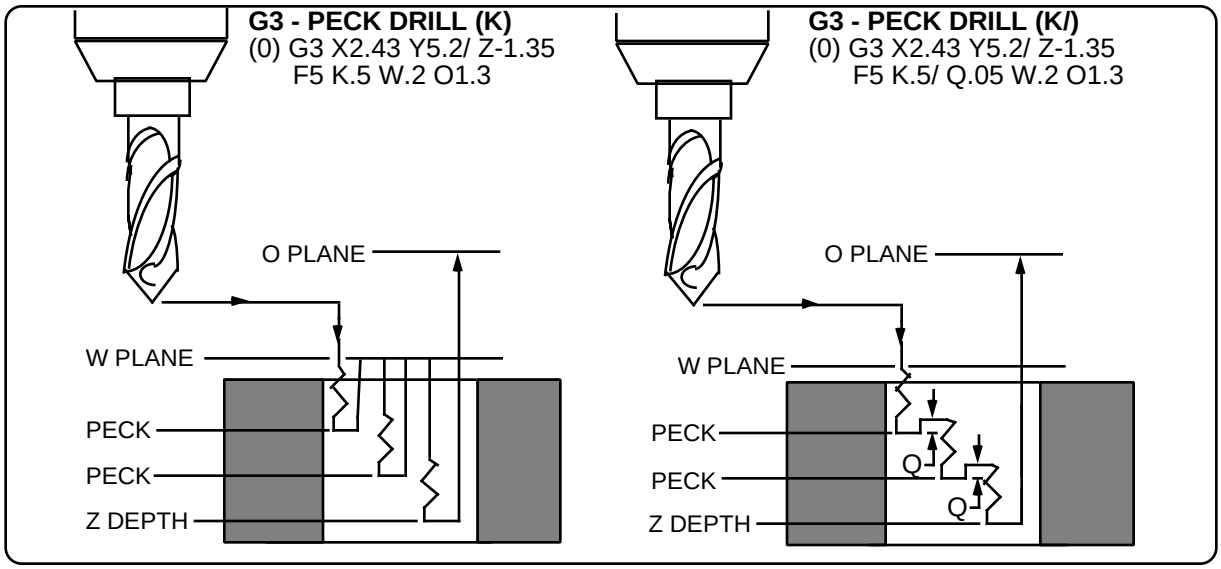
G2 - COUNTERBORE CYCLE

The counterbore tool tip slows from rapid to feedrate F at reference plane W, continues the feed to Z, dwells L seconds, then rapids to the O plane (See illustration, **below**).



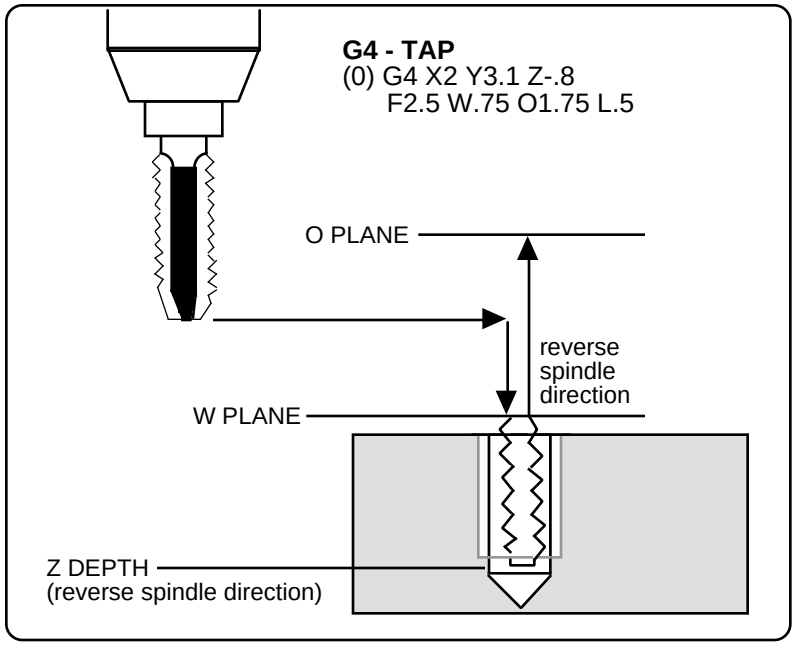
G3 - PECK DRILL CYCLE

The drill tip slows from rapid to feed F at reference plane W. Drill feeds K units from W into the part and retracts to finish the first peck. The retract motion is to W if K is absolute, or to clearance Q, if K is incremental. The next infeed begins at Q and continues through a second unit of K before another retract motion occurs. Each following peck similarly advances until the tip arrives at Z depth. The tool then rapids to O. If K is incremental and Q is not programmed, the backup will be .1" (2.5 mm) (See illustrations, **below**).



G4 - TAP CYCLE

You need a floating tap holder and programmable reversible spindle. The spindle speed to feedrate ratio must be proportional to tap threads per inch (or mm). Spindle speed is set prior to the event. The tap slows to feed F at reference plane W, feeds to Z, then the spindle will reverse direction. L seconds of added dwell may be programmed if more reversal time is required. The tap will then feed back to W where the spindle reverses again, restoring your initial rotary direction. If a 2nd reference plane is active, the tool will now rapid to the O plane. If spindle reversal or spindle speed is not programmable, we recommend that you use the G5 cycle with a reversing tap holder.



The feedrate that you program for this event must be selected based on the pitch or lead of the tap and the spindle speed that you have programmed. The formulas to calculate the tapping feedrate are shown below.

Tapping Formula for Inch Mode

$$\text{Feedrate} = \frac{\text{spindle speed}}{\text{threads/inch of tap}}$$

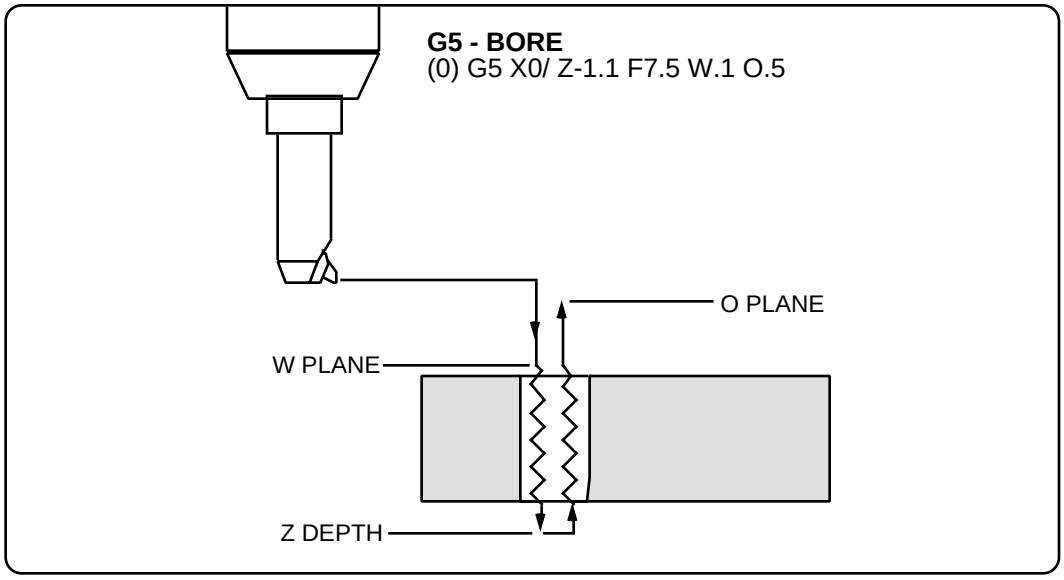
Tapping Formula for Metric Mode

$$\text{Feedrate} = \text{spindle speed} \times \text{mm/threads of tap}$$

After you calculate the suggested feedrate for the tapping cycle, most machinist handbooks teach that you can modify the calculated feedrate by ±10% to adjust for variances in the spindle speed, and different types of floating tap holders. If the CNC does not control the spindle speed on your machine, you must ensure that the spindle speed is set properly (by using a handheld tachometer) before you begin the tapping cycle.

G5 - BORE CYCLE

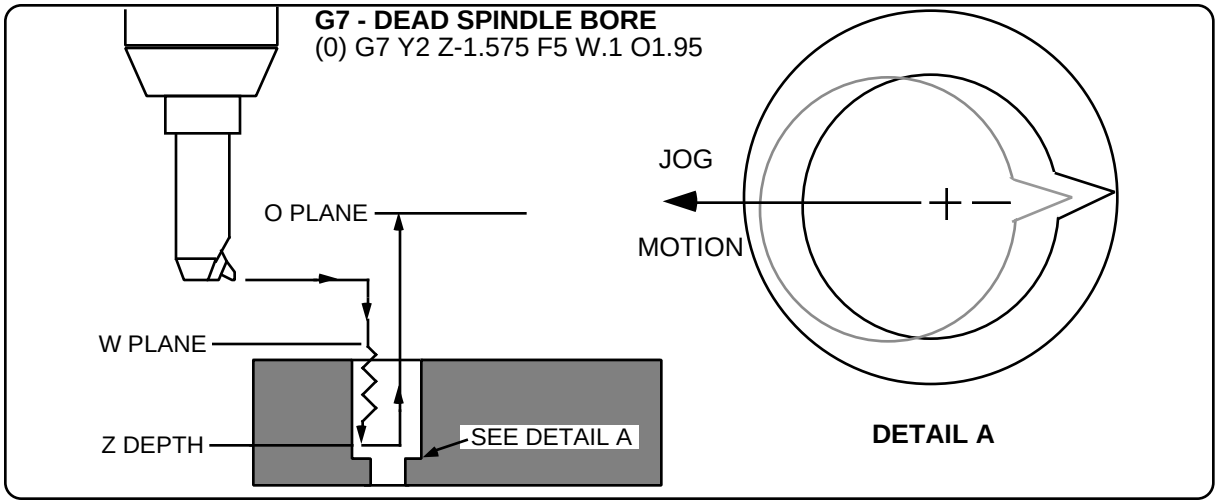
The bore tool tip slows to F at reference plane W, feeds to Z, feeds back to W, and rapids to the 2nd reference plane O. (See illustration **below**).



G7 - DEAD SPINDLE BORE CYCLE

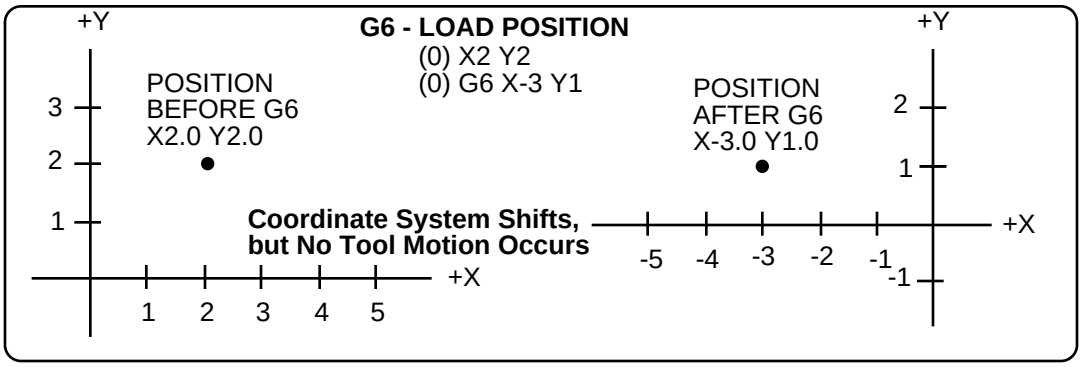
The bore tool tip slows to F at reference plane W, then feeds to Z (see illustration **below**). The cycle sends out M05 to halt the spindle, then M00 to halt the program. You can then press RETRACT to withdraw the tool from the hole. If the halted spindle turns freely, you can turn the tip to point along the X or Y axis. Then move to Jog mode and back the tip away from the hole's circumference (the clearance will depend on size of the the boring bar). Next, jog in Z, or move to Auto mode and press RETRACT to withdraw the tool from the hole.

To continue the cycle in Auto mode, press CYCLE START. The tool will rapid (+ or - direction) in Z to the O plane. If X or Y was jogged during the halt, the tool then rapids back to the XY hole center. The spindle then turns on in the original direction to end the cycle. If necessary, you can program a following Dwell event to give the spindle time to accelerate to the programmed speed



G6 - LOAD POSITION

A G6 event shifts axis origins and coordinates without causing any machine motion (See illustration **below**). The G6 event assigns its entered X, Y, Z coordinates to the current tool position. X, Y or Z entries must be absolute. Any axis not programmed keeps its last dimension. The axis shift is effective until replaced by a later G6 event or it is canceled by an EIA/ISO G99 block, End of Program M Code, or RESET when the control is not in cycle. A G6 event has no effect on the last G0-G5 or G7-G8 condition. The last G Code condition in this range remains modal.



A, B, C, U, V, W - AUXILIARY AXIS POSITION PROGRAMMING

Your machine supplier labels the optional Auxiliary axes A, B, C, U, V, or W.

A linear auxiliary axis endpoint may be a signed absolute or incremental entry.

A rotary axis endpoint is programmed in decimal degrees as an absolute or incremental angle. Use a + sign for ascending degrees rotation, or a - sign for descending degrees rotation, to a maximum 999.999 degrees incremental. (There is no windup.) The rotary display rolls over at the 359.999/000.000 degree division in either direction.

If a 1 or 5 degree (minimum increment) index axis is installed, a FORMAT ERROR alert appears if you try to enter an endpoint between the smallest index increments.

For G1-G5 or G7-G8, the G cycle will not begin until the X,Y and auxiliary axes (linear or rotary) are in position. A G cycle is always performed by the Z axis.

G8 - RIGID TAPPING

Rigid Tapping is an optional feature which allows tapping with a rigid tap holder. Your machine supplier can enable this feature if your machine's spindle can provide digital feedback signals to the control.

You must program the spindle speed in a prior M Function event. The control will calculate the tapping feedrate by using your last spindle speed and thread dimensions you program.

Rigid Tapping is a modal cycle which is programmed in Position or Bolt Circle event. A quick way to determine if your control has Rigid Tapping is by looking at the Position or Bolt Circle event screen. Rigid Tapping is installed if G8 RIGID TAP is listed on the screen when you press the G key.

REQUIREMENTS

To use the Rigid Tapping feature, you must have:

- A machine equipped with an AC spindle.
- The Rigid Tapping option.

F - TAP THREAD DIMENSIONS - PITCH OR LEAD

Program F to specify the pitch or lead of your tap. When programming in inches, enter a sign positive F value to define the tap's pitch in threads/inch. When programming in millimeters, program a sign negative F value to define the tap's lead in millimeters/thread.

The Lead/Pitch dimension F is modal to Rigid Tapping cycles **only**, and **does not** cancel the last F feedrate. To maintain program clarity, you should program the Lead/Pitch F entry in every Rigid Tapping event.

The modal feedrate F will be reactivated when a non-Rigid Tap event or block executes.

FEEDRATE FORMULAS

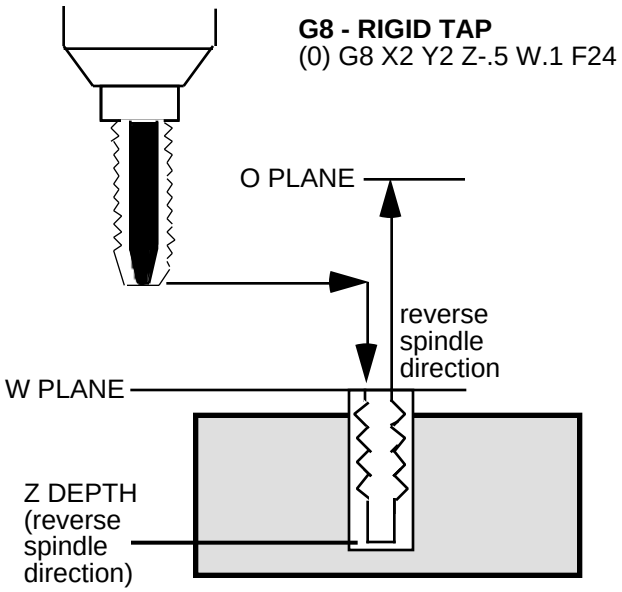
The control uses the following equations to calculate the tapping feedrate. The spindle speed in revs/min must be programmed in a prior M Function event.

Inch feedrate: +F defines the pitch in threads/inch:

Z feedrate = (spindle revs/min) / (threads/in) = in/min

Millimeter feedrate: -F defines the lead in mm/thread:

Z feedrate = (spindle revs/min)*(mm/thread) = mm/min



ORDER OF G8 CYCLE MOTION

1) Spindle direction and speed must be set in a previous M Function event.
2)X, Y, and any auxiliary axes rapid to their programmed endpoints.
3)The Z axis rapids to the reference plane W. During or immediately after this move, the spindle will stop then resume at the original speed and direction.
4) Z feeds to the programmed Z depth. The FEEDRATE OVERRIDE Pot is disabled at this time.
5) When Z reaches it's programmed depth, the spindle reverses direction.
6) Z retracts at feed to the reference plane W.
7) An M05 is output to stop the spindle.
8)M03 or M04 is output to start the spindle in original direction.
9) Tool rapids to O plane.

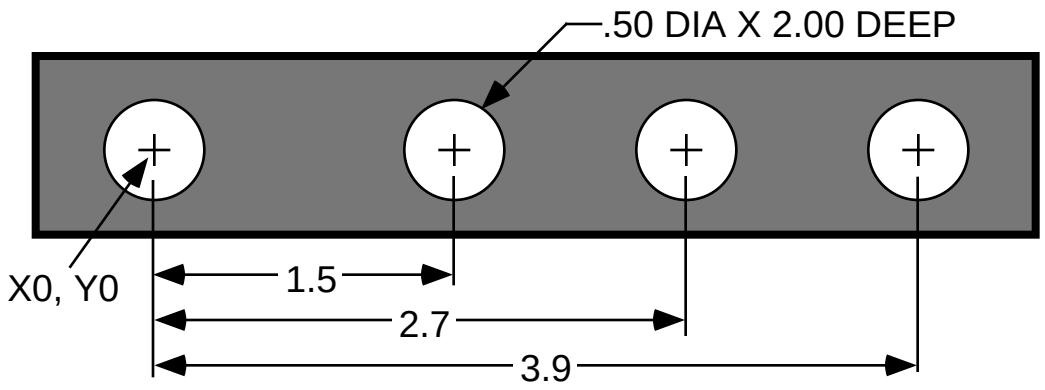
CANCELING RIGID TAPPING

The G8 Rigid Tapping cycle is canceled when G0-G5 or G7 executes in a Position or Bolt Circle event, M02 or M30 executes, or you press the RESET button when the control is not in cycle.

POSITION EVENT EXAMPLE

The **following** example drills four holes using the G1-Drill cycle in several Position events. Both part programs will cut the same part. The left program uses absolute dimensions to locate successive holes; the right program uses incremental dimensions to locate the holes. The program begins by activating the offsets for tool number 1 and turning the spindle on. Next, N20 positions the tool over the first hole. N30 then drills the first hole. Notice that the command X0/ causes no tool motion in XY, but allows the G1 cycle to be performed in Z. Next, N40 positions the tool over and drills the second hole. Since G1 is modal, all following Position events will perform the drill cycle programmed in N30. N40 and N50 drill the third and fourth holes. N60 ends the program with an M30 command. The M30 command automatically cancels the drill cycle.

Since the work surface is at a uniform height and there are no clamps to be concerned about, this program does not use a 2nd reference plane. The control will automatically set it equal to the W plane.



Absolute Dimensions

```
N05  (G)  X-1 Y-1 Z-2.5 H8
N10  (9)  T1 H1 D1 M3 S1600
N20  (0)  X0 Y0 Z.5 G0
N30  (0)  X0/ G1 Z-2 F10 W.1
N40  (0)  X1.5
N50  (0)  X2.7
N60  (0)  X3.9
N70  (9)  M30
```

Incremental Dimensions

```
N05  (G)  X-1 Y-1 Z-2.5 H8
N10  (9)  T1 H1 D1 M3 S1600
N20  (0)  X0 Y0 Z.5 G0
N30  (0)  X0/ G1 Z-2 F10 W.1
N40  (0)  X1.5/
N50  (0)  X1.2/
N60  (0)  X1.2/
N70  (9)  M30
```