

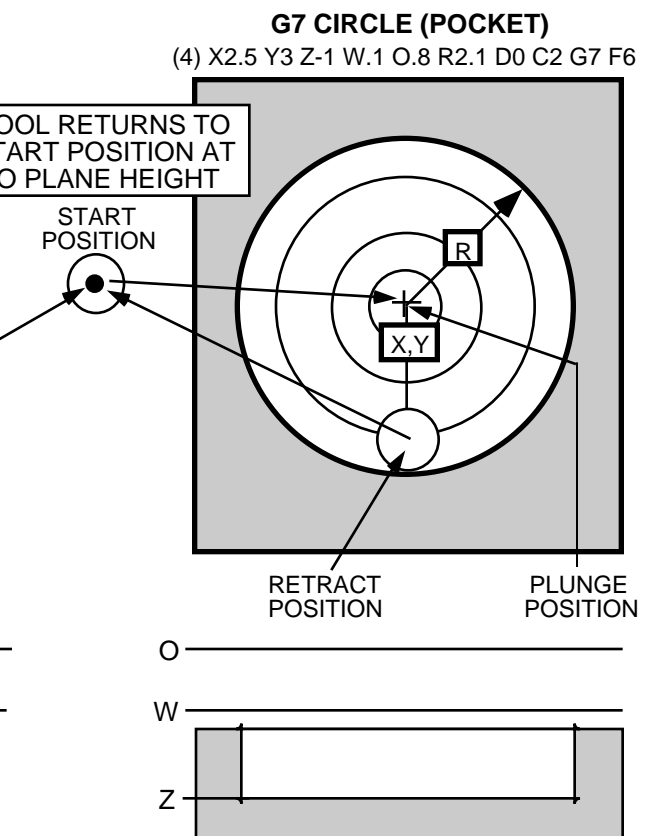
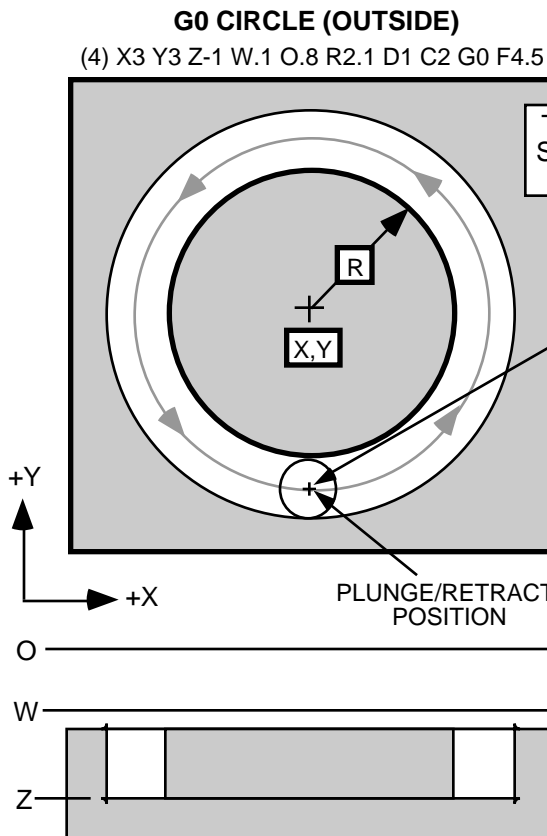
# Section 33

# Circle Mill Event

## Topics Covered in this Section:

- Introduction
- Circle Mill Data
- Required Entries
- Optional or Modal Entries
- G0 Circle Mill Types
- G7 Circle Pocket Mill Types

PROGRAM MODE	N1800	<b>Circle Mill Event</b>				
0-POSITION	CIRCLE EVENT					
1-LINEAR MILL						
2-ARC MILL	X					
3-FRAME MILL	Y					
4-CIRCLE MILL	Z					
5-BOLT CIRCLE	W					
6-REPEAT	O					
7-SUBROUTINE	R					
8-DWELL	D					
9-M FUNCTION	C					
	G					
C-CAVITY MILL	K	<b>G0 = Frame, G7 = Pocket</b> <b>D - Milling Direction</b> <b>C - Cutter Radius Compensation</b> <b>L - Finish Dimension</b> <b>K - Peck Mill</b> <b>J - Corner Roughing Width</b> <b>P, F, V - Feedrates</b>				
E-EIA						
G-GRAPHICS	J					
M-MACRO CALL	L					
P-PROBE						
R-ROTATE	P					
S-SET UP	F					
T-TEXT	V					
DEMOPART						
FRONT END GRAPHICS		DELETE EVENT	COPY/ STORE	PROGRAM DISPLAY	RUN CALC ASSIST	



# CIRCLE MILL EVENT - INTRODUCTION

This event cuts a circle to a specified depth. As shown, **facing page**, you can choose to mill around the circle's perimeter, or remove (pocket) all material within the circle's perimeter. It is not necessary to position to the circle prior to this event unless O reference plane clearance cannot be assured along a direct path for tool return between the circle and the tool's start position. **This clearance must be observed since the cutter returns to the O plane and then rapids in a direct path to its starting XY coordinates at the event's conclusion.**

## CIRCLE MILL DATA

Parameter	Description	Entry
N	Sequence Number	Required
G	G Cycle G0=circle, G7=pocket	Defaults to G0
X	Circle Center, X	Required
Y	Circle Center, Y	Required
Z	Depth	Required
W	1st Z Reference Plane	Modal
O	2nd Z Reference Plane	Modal
R	Radius	Required
D	Milling Direction D0=rough CW/finish CW D1=rough CCW/finish CCW D2=rough CW/finish CCW D3=rough CCW/finish CW	Defaults to D0
C	Cutter Radius Compensation C0=off, C1=left, C2=right	Required
K	Peck Mill	Optional
J	Roughing Width (G7 only)	Optional
L	Finish Dimension L=finish bottom and sides, L/= finish sides only	Optional
P	Plunge Feedrate	Defaults to F
F	Feedrate	Modal
V	Finish Feedrate	Defaults to F

## REQUIRED ENTRIES

Program G0 to mill around the circle's perimeter, or G7 to pocket the circle. A pocketing operation removes all material within the circle. Describe the circle's shape with center X and Y, radius R, depth Z, and reference planes W and O. Next program direction D to choose conventional or climb milling for the rough/finish passes. After selecting the direction, choose the cutter compensation direction C to place the cutter inside, outside, or on the circle's profile. The compensation direction (left, right, or center) is based on the roughing direction.

## OPTIONAL ENTRIES

After completing the required entries, choose your machining parameters with peck depth K, finish dimension L, and roughing width J (G7 only). You can program up to three independent feedrates with plunge feedrate P, roughing feedrate F, and finish feedrate V. Any feedrate that is not programmed defaults to the modal feedrate.

# **CIRCLE GEOMETRY - REQUIRED ENTRIES**

All Circle Mill events **must** program these entries. Refer to illustrations on **page 1**.

## **G0 or G7 - PERFORM THE CIRCLE or CIRCLE POCKET G CYCLE**

G selects a circle or pocket circle mill cycle. G0 is the default cycle.

## **X, Y - CIRCLE CENTER**

X and Y are the circle's center coordinates. Enter the signed absolute X and Y axes coordinates, or the signed incremental distance from the cutter's position at the start of the event. Absolute and incremental entries may appear in the same event. X and Y must be entered in all Circle Mill events.

## **Z - DEPTH**

Z is the final depth the cutter reaches while milling the circle. Depth is entered as the signed Z axis coordinate, or the signed incremental distance from the W reference plane. Z must be entered in all Circle Mill events.

## **R - RADIUS**

R is the circle's radius. R is an unsigned entry, and is required in all Circle Mill events.

## **W - 1st Z AXIS REFERENCE PLANE**

W is a reference plane, parallel to the XY plane, and typically .1" (2 mm) above the part surface. Z advance slows from rapid to a programmed feedrate when the cutter crosses W. W is a signed absolute coordinate, and is modal with all following event types which use 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 prior to making the final XY rapid move to the tool's position prior to the Circle Mill event. O is an optional entry; if not programmed, it will default to the value of the W plane.

## **D - MILLING DIRECTION**

The four D entries specify the cutting direction for roughing and finishing the circle. D0 and D1 maintain the same cutting direction while D2 and D3 reverse it before the finish pass. If L is not programmed, the finish direction is ignored. When not programmed, D defaults to D0.

## C - CUTTER RADIUS COMPENSATION

C selects the cutter's orientation to the circle's profile. C0 sets the cutter center on the profile. C1 or C2 selects a radius offset left or right of the profile<sup>1</sup>. Choose the left or right offset by looking along the roughing direction D and selecting the direction that puts the tool on the correct side of the profile. G0 cycles allow outside, tool center, or inside milling. A G7 pocket cycle must choose D then a C entry that places the tool inside the profile. Both cycles automate the cutter's positioning to the profile.

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

## OPTIONAL or MODAL ENTRIES

### F - FEEDRATE

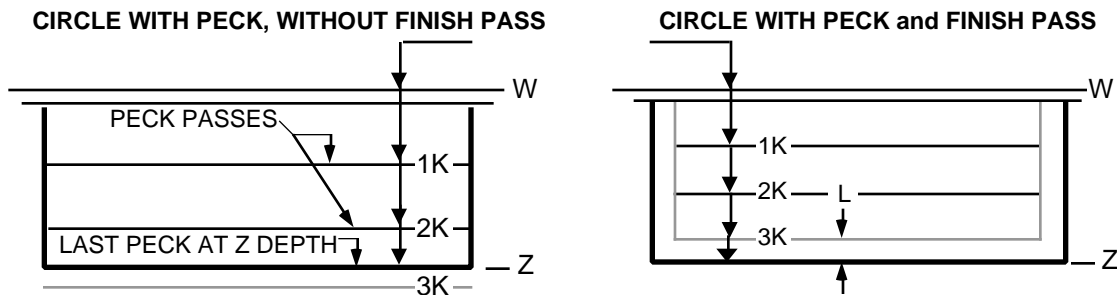
F programs the roughing feedrate. If P or V is not programmed, their respective operations will default to feedrate F. F is modal with all following event types which use feedrate F.

### P - PLUNGE RATE

P is a Z axis feedrate which is operative as the cutter plunges from reference plane W to depth Z. A Circle Mill event with P not programmed plunges at feedrate F.

### K - PECK INTERVAL

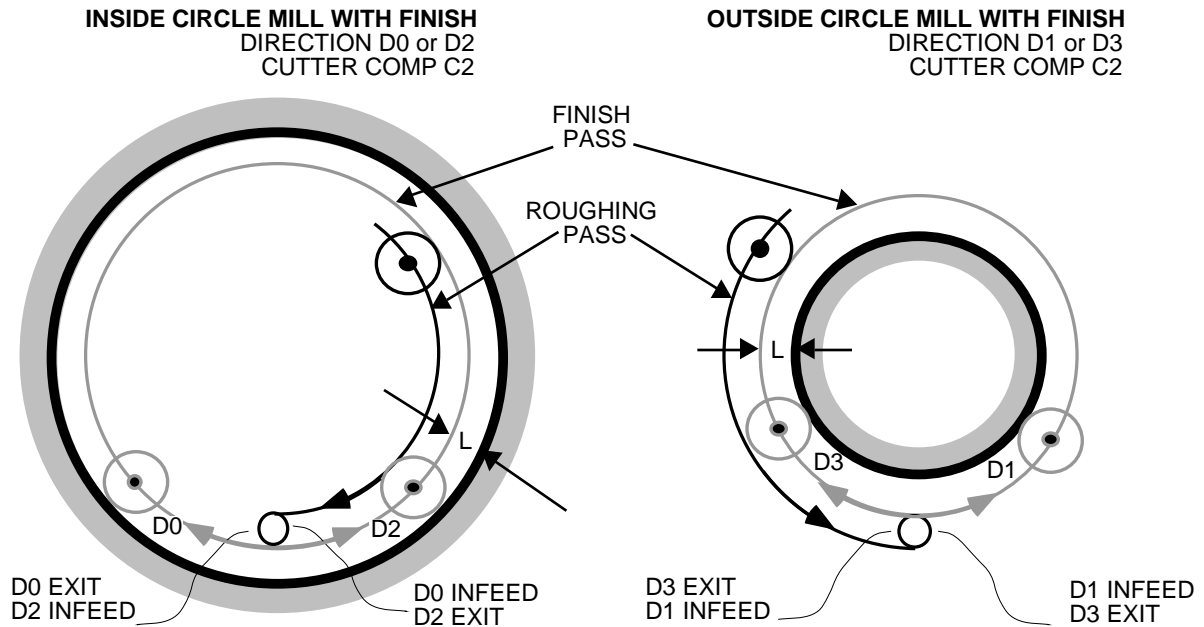
K creates a deep milling routine by specifying a maximum depth per plunge. The control will calculate the number of circle mill passes that are required to reach the programmed depth. Passes are performed at each successive interval of K from W, with the last pass interval reduced to your final Z depth. When L (absolute) requests a bottom finish, the final plunge depth is further reduced by finish dimension L to leave room for the bottom finish pass. K must be programmed in every Circle Mill event that requires a deep milling cycle, and is entered as an unsigned value. Refer to the illustrations **below**.



<sup>1</sup> The active tool's diameter must be stored in the Tool Tables mode, and must be made active by executing its

## L, V - FINISH DIMENSION AND FEEDRATE - G0 or G7 OPTIONS

L is an unsigned finish dimension. When entered as an absolute value, L adds bottom and side finish passes to the milling cycle. When L is incremental, only the side pass is performed. The side finish feedrate V is entered as an unsigned absolute value, and if not programmed, defaults to feedrate F.



The finish cut occurs in the direction specified with direction D. You can select to rough and finish in the same direction, or reverse cutting direction between passes. To eliminate dwell marks at the start of the finish pass, the tool tangentially feeds into the part along an arc with a diameter equal to the finish dimension L. The tool also tangentially exits the part along this arc at the end of the finish pass. Sample tangential entry and exit moves are shown **above**.

## J - ROUGHING WIDTH - G7 OPTION

J controls the maximum width of material that is removed on each pocket pass. J is a maximum roughing width that will not be exceeded on any pass while milling the pocket. Program J only when you wish to reduce the amount of material the cutter will remove per outward pass from the default value. If J is not entered, the control sets a default value equivalent to the cutter diameter  $-.010''$  ( $-.254$  mm).

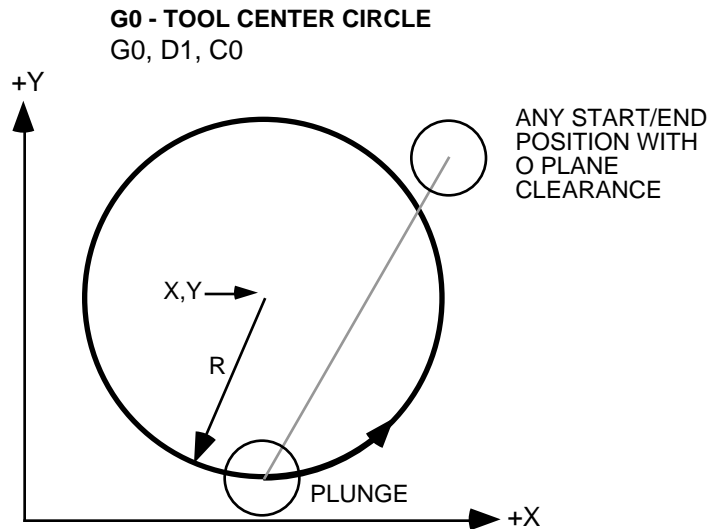
J should not exceed the cutter diameter. Be aware that the control will not use J directly but will calculate, under the constraint of J, the number of outward passes required to pocket the circle. To insure that all passes are of uniform width, the control rounds its calculations up to the next integer number of passes, and proportionally reduces the roughing width to fit the number of passes.

## G0 CIRCLE MILL TYPES

There are three basic circles which can be programmed with G0 Circle Mill events. Each circle is described below. Note that the combination of cutting direction (D0-D3) and the cutter compensation direction (C0-C2) dictates which type of circle is cut.

### TOOL CENTER CIRCLE MILL

Enter C0 to program a tool center circle mill. When this type of Circle Mill event executes, the tool's center travels around the circle at radius R. Only one pass occurs at Z depth unless peck depth K causes multiple passes at increasing Z depths. No finish pass is allowed with tool center circle mills.



### TOOL CENTER CIRCLE TOOL MOTION

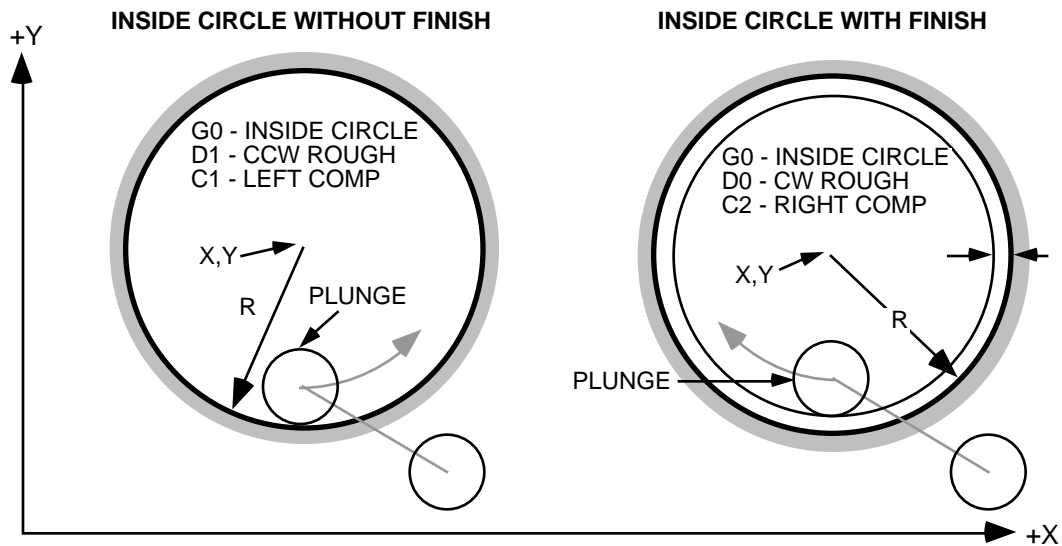
- The X and Y axes rapid to the plunge point determined by the control (see illustration, **above**). The Z axis rapids to the W plane, slows to feedrate P until the tool reaches the programmed depth, or peck increment.
- The tool center mills around the circle at feedrate F. If a peck is programmed, the tool feeds at P to the next depth, then repeats this step until the final depth is reached.
- When the tool has completed the roughing pass, it rapids to O, then XY rapid back to the tool's XY start point. End of event.

## GO - INSIDE CIRCLE MILL

Inside circle mills, **below**, maintain your programmed radius R on the inside of the circle. When L is not programmed, the tool will make one roughing pass around the circle. When L is programmed, the tool will make additional finish passes around the circle to remove L inches (or millimeters) from the sides and bottom<sup>2</sup>.

The total width of the material removed during an inside circle mill is equal to the tool's diameter plus the programmed finish dimension L.

You can program inside circle mill cycles with any of the following combinations of rough/finish directions and cutter compensation directions: (G0, D0, C2), or (G0, D2, C2), or (G0, D1, C1), or (G0, D3, C1).



### INSIDE CIRCLE TOOL MOTION - WITHOUT FINISH PASS

- The X and Y axes rapid to the plunge point. The plunge location is on the inside wall of the circle. Refer to the illustration **above left**.
- The Z axis rapids to the W plane, slows to feedrate P until the tool reaches the programmed depth, or peck increment.
- The tool feeds once around the circle at feedrate F. Repetitive passes occur at increasing Z depths if a peck cycle has been programmed.
- The tool rapids to O, then rapids back to the tool's XY start point. End of event.

### INSIDE CIRCLE TOOL MOTION - WITH FINISH PASS

- The X and Y axes rapid to the plunge point. The plunge location is L units inside the circle's wall. Refer to the illustration **above right**.
- The Z axis rapids to the W plane, slows to feedrate P until the tool reaches the programmed depth, or peck increment.
- The tool makes one roughing pass, at feedrate F, leaving L inches (or millimeters) to be removed on the finish pass. If a peck is programmed, repetitive roughing passes occur until the correct depth is reached.
- If L absolute requests a bottom finish, the tool feeds L units to depth Z, and makes one bottom finish pass at feedrate F.
- The tool makes a final finish pass at feedrate V to remove L units from the sides of the circle. The tool tangentially feeds in and out of the part at the start and end of the finish pass. The side finish pass occurs at Z depth; the finish pass does not peck.
- The tool rapids to O, then rapids back to the tool's XY start point. End of event.

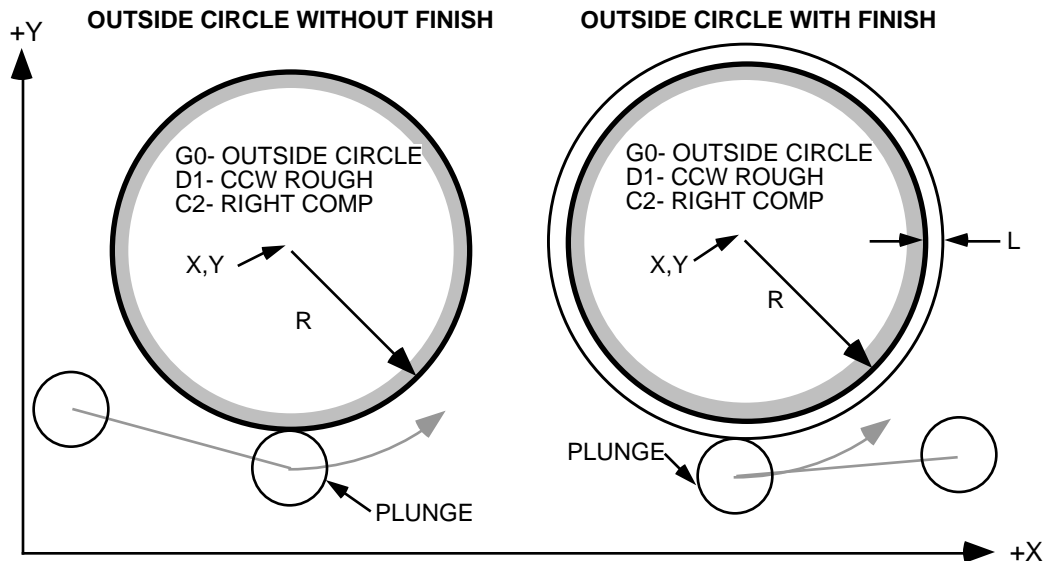
<sup>2</sup> A finish pass will occur on the bottom only when L is programmed as an absolute value. The bottom finish

## G0 - OUTSIDE CIRCLE MILL

Outside circle mills, **below**, maintain your programmed radius  $R$  on the outside of the circle's profile. When  $L$  is not programmed, the tool will make one roughing pass around the circle; when  $L$  is programmed, the tool will make additional finish passes around the circle to remove  $L$  inches (or millimeters) from the sides and bottom<sup>2</sup>.

The total width of the material removed during this event is equal to the tool's diameter plus the programmed finish dimension  $L$ .

You can program outside circle mill cycles with any of the following combinations of rough and finish directions, and cutter compensation direction: (G0, D0, C1), or (G0, D2, C1), or (G0, D1, C2), or (G0, D3, C2).



### OUTSIDE CIRCLE TOOL MOTION - WITHOUT FINISH PASS

- The X and Y axes rapid to the plunge point. The plunge location is on the outside wall of the circle. Refer to the illustrations **above left**.
- The Z axis rapids to the W plane, slows to feedrate  $P$  until the tool reaches the programmed depth, or peck increment.
- The tool feeds once around the circle at feedrate  $F$ . Repetitive passes occur at increasing  $Z$  depths if a peck cycle has been programmed.
- The tool rapids to O, then rapids back to the tool's XY start point. End of event.

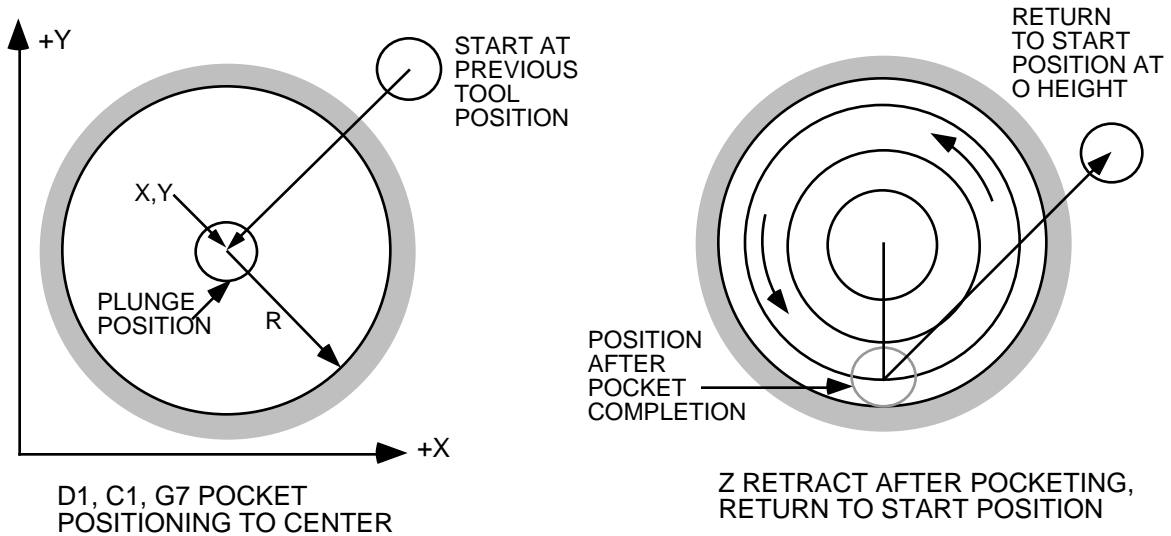
### OUTSIDE CIRCLE TOOL MOTION - WITH FINISH PASS

- The X and Y axes rapid to the plunge point. The plunge location is  $L$  units outside of the circle's profile. Refer to the illustrations **above right**.
- The Z axis rapids to the W plane, slows to feedrate  $P$  until the tool reaches the programmed depth, or peck increment.
- The tool makes one roughing pass, at feedrate  $F$ , leaving  $L$  inches (or millimeters) to be removed on the finish pass. If a peck is programmed, repetitive roughing passes occur until the correct depth is reached.
- If  $L$  absolute requests a bottom finish, the tool feeds  $L$  units to depth  $Z$ , and makes one bottom finish pass at feedrate  $F$ .
- The tool makes a final finish pass at feedrate  $V$  to remove  $L$  units from the side of the circle. This occurs in one pass; the finish pass does not peck. The tool tangentially feeds in and out of the part at the start and end of the finish pass. The side finish pass occurs at  $Z$  depth; the finish pass does not peck.
- The Z axis rapids to O, then rapids back to the tool's XY start point. End of event.

## G7 - CIRCLE POCKET MILL TYPES

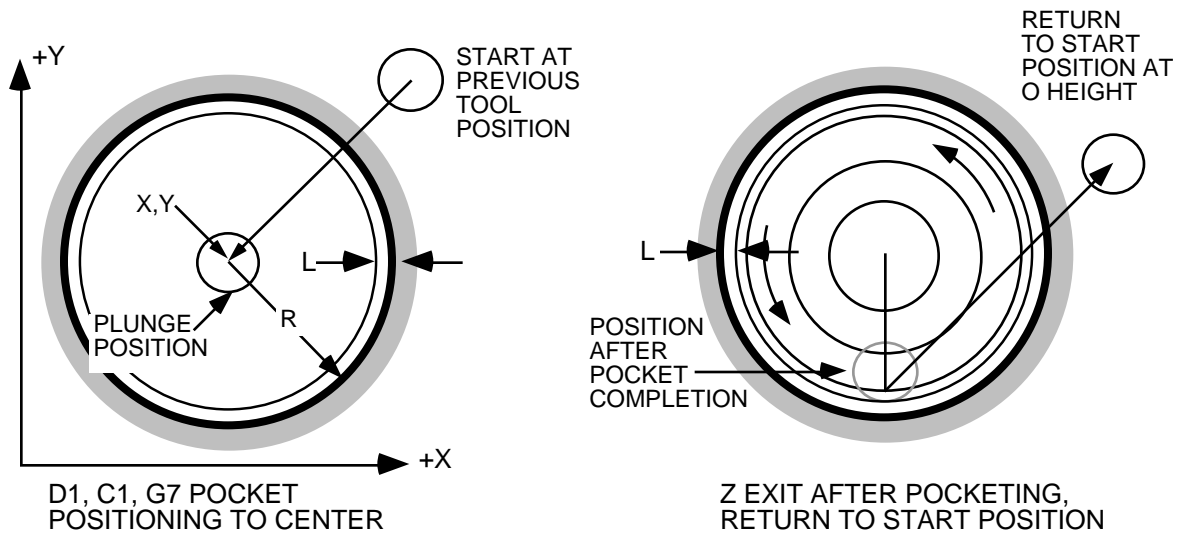
Circle Mill events with G7 programmed, **below**, will remove all material within the boundaries of the circle defined with X,Y and R. If desired, you can limit the tool overlap distance by programming the roughing width J.

You can program pocket mills with the following combination of cutting directions and cutter compensation directions: (G7, D0, C2), or (G7, D2, C2), or (G7, D1, C1), or (G7, D3, C1). Any other combination of C and D will cause a fault at run time. A finish dimension L and/or peck depth K can be added to this cycle.



### G7 POCKET MILL TOOL MOTION WITHOUT FINISH PASS

- The X and Y axes rapid the circle's center. Refer to the illustrations **above**.
- The Z axis rapids to W then feeds at P to depth Z, or peck increment K.
- The tool feeds a distance equal to the calculated roughing width in the - Y direction. Remember the control optimizes the roughing width J dimension to ensure constant metal removal.
- The tool cuts a circle, at feedrate F, around the arc center.
- The tool then moves outward and cuts another circle. This step repeats until the tool reaches the radius R.
- If a peck is programmed, the tool retracts .1" in Z then rapids back to the center and repeats the process until the final Z depth is reached.
- The tool rapids to O, then rapids back to the tool's XY start point. End of event.



### G7 POCKET MILL TOOL MOTION WITH FINISH PASS

- The X and Y axes rapid the circle's center. Refer to the illustrations **above**.
- The Z axis rapids to W then feeds at P to depth Z, or peck increment K.
- The tool feeds a distance equal to the calculated roughing width in the - Y direction. Remember the control optimizes the roughing width J dimension to ensure constant metal removal.
- The tool cuts a circle, at feedrate F, around the arc center.
- The tool then moves outward and cuts another circle. This step repeats until the tool is L inches (or millimeters) from radius R.
- If a peck is programmed, the tool retracts .1" in Z then rapids back to the center and repeats the process until the tool is L inches (or millimeters) from the Z depth. If L is programmed as an incremental dimension, no bottom finish pass occurs; therefore, the peck continues to depth Z.
- If L is programmed as an absolute dimension, the control will feed to the final Z depth and complete a pocket at feedrate F.
- The tool now tangentially feeds L units into the part, and make one more circular pass. At the end of the pass, the tool tangentially exits from the wall.
- The tool rapids to O, then rapids back to the tool's XY start point. End of event.

