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

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

* **D** - Tool radius or diameter selection**
F - Feedrate
I - Radius of first circle (or finish if no *K*). *I* value must be greater than Tool Radius, but less than *K* value.
* **K** - Radius of finished circle (if specified)
* **L** - Loop count for repeating deeper cuts
* **Q** - Radius increment, or stepover (must be used with K)
Z - Depth of cut or increment
*indicates optional

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

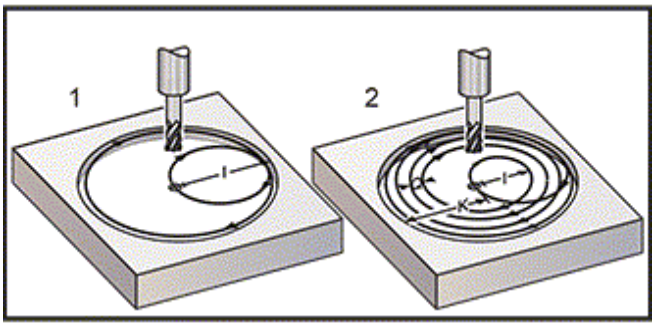
NOTE	Specify D00 if you do not want to use cutter compensation. If you do not specify a D value in the G12 / G13 block, the control uses the last commanded D value, even if it was previously canceled with a G40 .

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

```
%
O60121(SAMPLE G12 AND G13) ;
(G54 X0 Y0 is center of first pocket) ;
(Z0 is on top of the part) ;
(T1 is a .25 in. dia endmill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X0 Y0 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Tool offset 1 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G12 I0.75 F10. Z-1.2 D01 (Finish pocket CW) ;
G00 Z0.1 (Retract) ;
X5.(Move to center of next pocket) ;
G12 I0.3 K1.5 Q1. F10. Z-1.2 D01 ;
(Rough & finish CW) ;
G00 Z0.1 (Retract) ;
X10.(Move to center of next pocket) ;
G13 I1.5 F10. Z-1.2 D01 (Finish CCW) ;
G00 Z0.1 (Retract) ;
X15. (Move to center of the last pocket) ;
G13 I0.3 K1.5 Q0.3 F10. Z-1.2 D01 ;
```

(Rough & finish CCW) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%

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



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

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

Single Pass: Use *I* only.

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

Multiple Pass: Use *I*, *K*, and *Q*.

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

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

Applications: Deep rough and finish pocketing.

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

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

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

The *G91* and *L* count can also be used in a *G13I* only line.

%
O60131 (G13 G91 CCW EXAMPLE) ;
(G54 X0 Y0 is center of 1st pocket) ;
(Z0 is on top of the part) ;
(T1 is a 0.5 in. dia endmill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X0 Y0 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;

```
G43 H01 Z0.1 (Activate tool offset 1) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G13 G91 Z-.5 I.400 K2.0 Q.400 L4 D01 F20. ;
(Rough & finish CCW) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G90 Z0.1 M09 (Rapid retract, coolant off) ;
G53 G49 Z0 M05 (Z home, spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```