

CNC Programming Canned Cycles

G Code	Function
G80	Canned Cycle Cancellation
G73	Deep Hole Drilling Cycle (High Speed) AKA Peck Drilling Cycle – Chip Breaker
G81	Drilling Cycle (general)
G83	Peck Drilling Cycle
G84	Tapping Cycle (R/H)

G Code	Function
G98	Returns the tool to the initial Z position (last used before canned cycle was called)
G99	Returns the tool to the point specified by the address 'R'

Canned Cycle Data Sequence (Example)

G99 or **G98** (Z retract command)

G81 (or other canned cycle)

X position

Y Position

R Z axis start position R level is the position at which the cutting feed rate is applied.

Z Z axis end position or Z depth full. The point at which feed rate ends.

P Dwell time in Milliseconds. Therefore an input of 1 is equal to 1/1000 seconds.

Q For milling - depth of each peck. For Turning – amount of shift for boring. Q only applies when using G73 or G83 pecking cycles.

I Shift amount (when boring) X axis.

J Shift amount (when boring) Y axis.

F Feed Rate (applies to cutting motions only)

L Number of canned cycle repetitions (L1 is default if no L value is input). The range is from L0 to L9999

The Polar Coordinate System

G Code	Function
G15	Polar Coordinate System Cancel (Off)
G16	Polar Coordinate System (On)