

CNC PROGRAMMING IV.



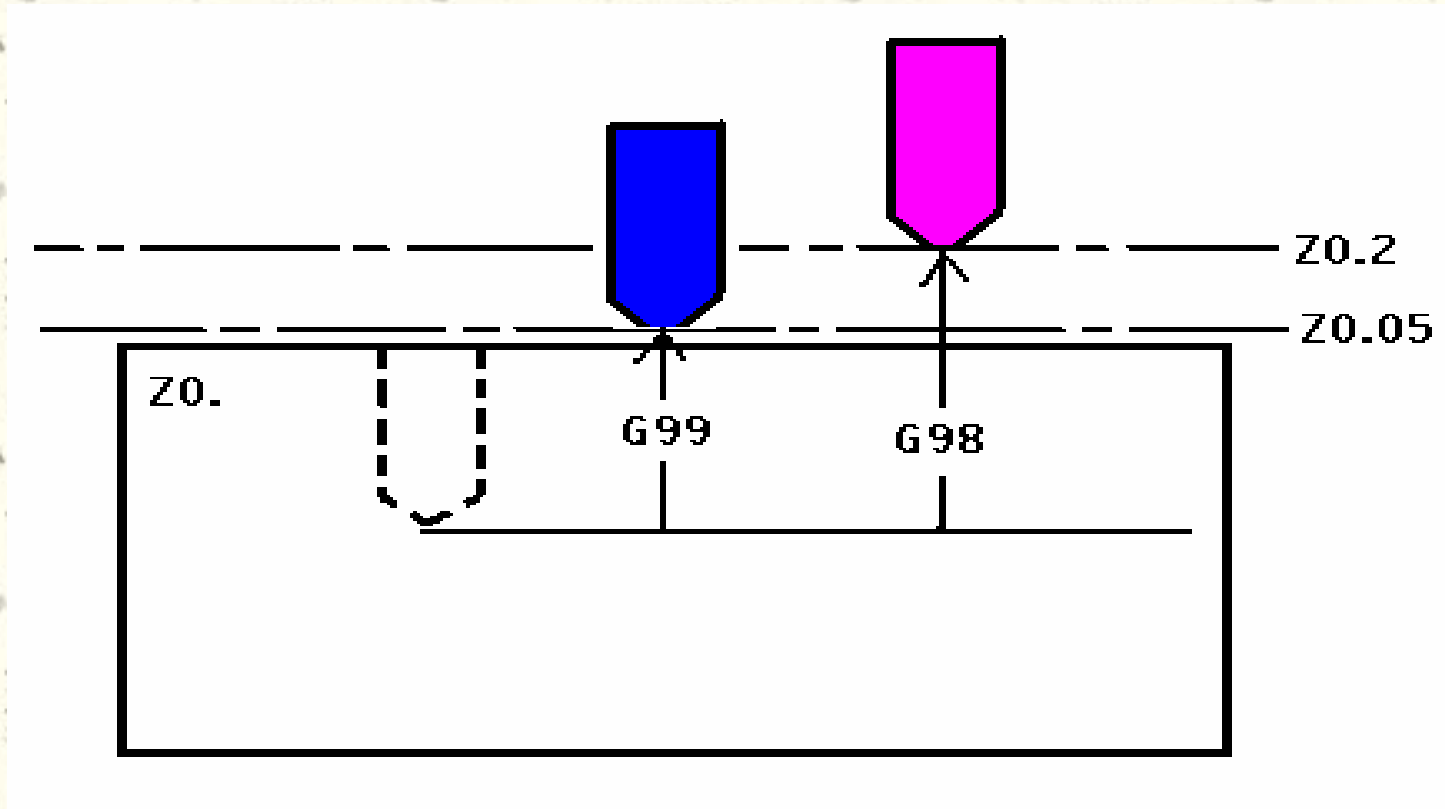
CANNED CYCLE
(MAKING HOLES)

Canned Cycle

- # Hole operation
- # Modal command
- # The five operations in a canned cycle
 - Positioning of the X and Y axes
 - Rapid traverse to the R plane
 - Drill, boring & tapping
 - Operation at the bottom of hole
 - Retraction to the R plane

G98 & G99

- # G98 (Initial Return Point; for HAAS)
- # G99 (Reference Plane Return)



Codes

- # G80, G81, G82, G83, G84, G98, G99
 - G80: Cancel canned cycle
 - G81: Drill
 - G82: Center Drill
 - G83: Peck drill
 - G84: Tapping
 - G98: Initial point return
 - G99: Reference plane return

G81, G82, G83 & G84

G81 (Drill)

- F, L, P, R, X, Y & Z

- L; # of repeats

- R; Reference plane

G82 (Center Drill)

- F, L, P, R, X, Y & Z

- P; Dwell operation time in seconds

G83 (Peck Drill)

- F, L, Q, R, X, Y & X

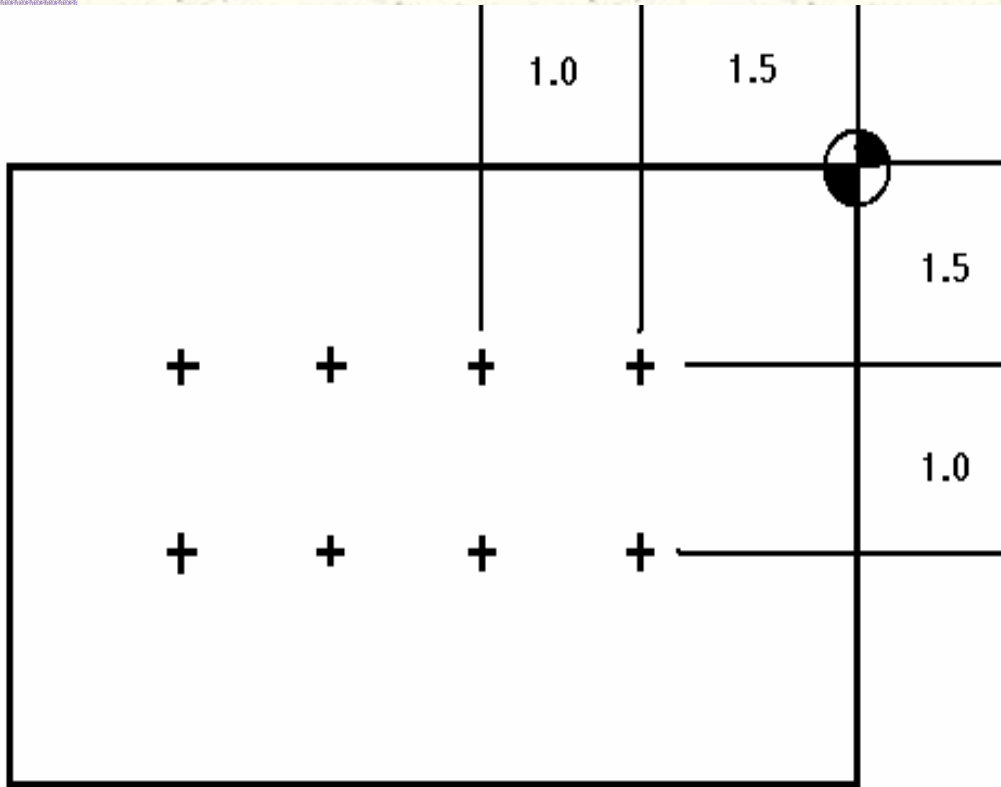
- Q; Peck Depth

- If the depth of hole is 1", Q.25 will peck 4 times.

G84 (Tapping)

- Same as G81 but be careful with F.

Simple Drilling Program



Tools:

T5; #4 Center Drill
T6; #7 Drill (=0.201)

RPMs:

T5 = 6000
T6 = 5000

Feeds:

T5 = 18.
T6 = 15.