

# CNC PROGRAMMING IV.



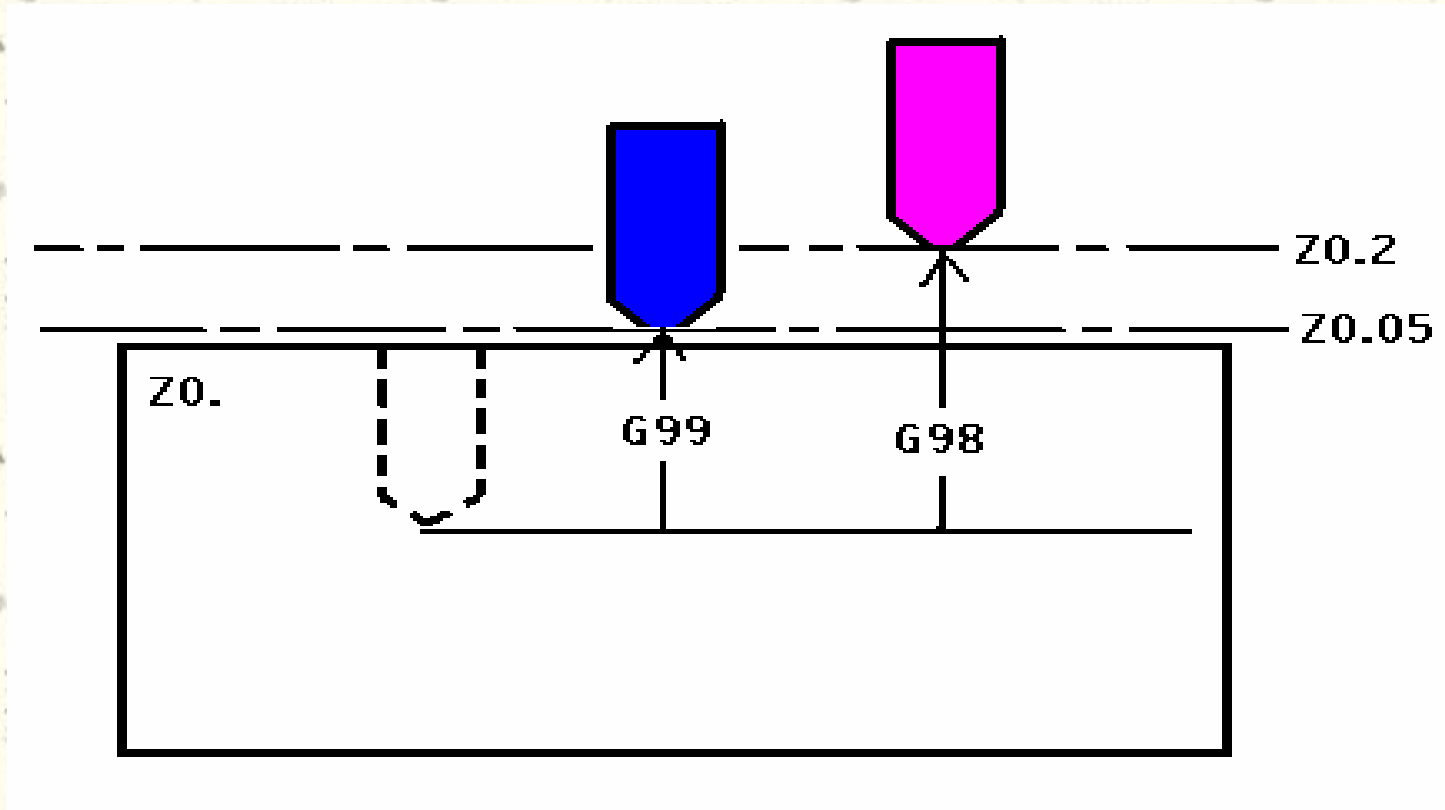
CANNED CYCLE  
(MAKING HOLES)

# 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

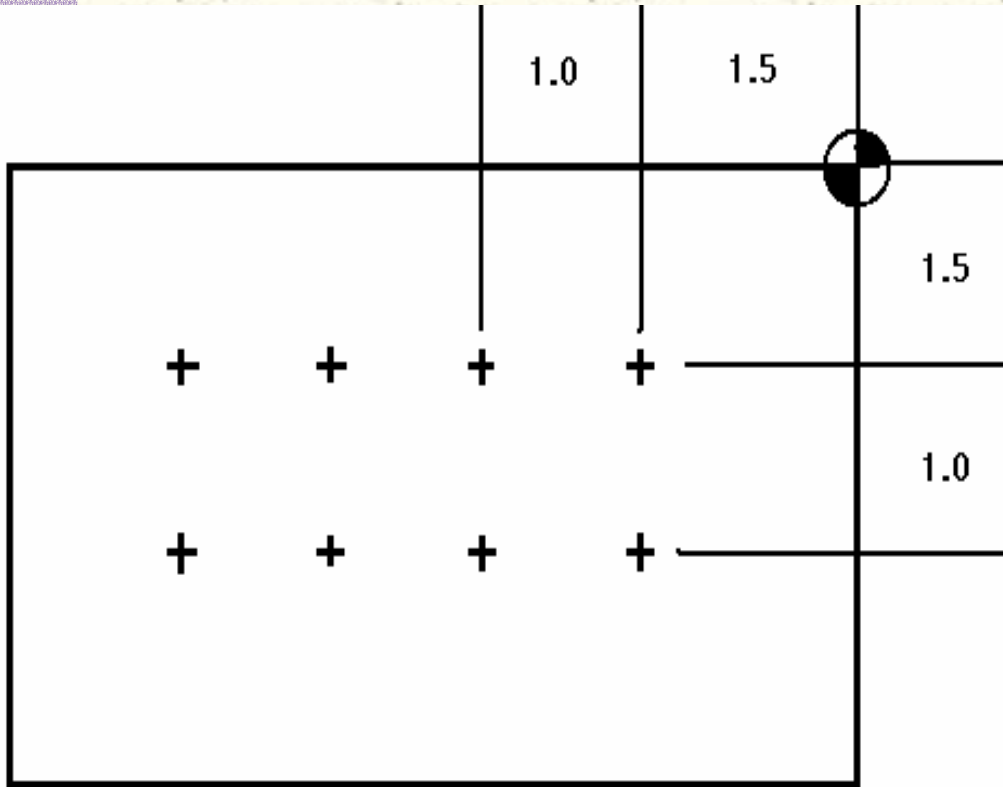
- Q; Peck Depth

- 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.