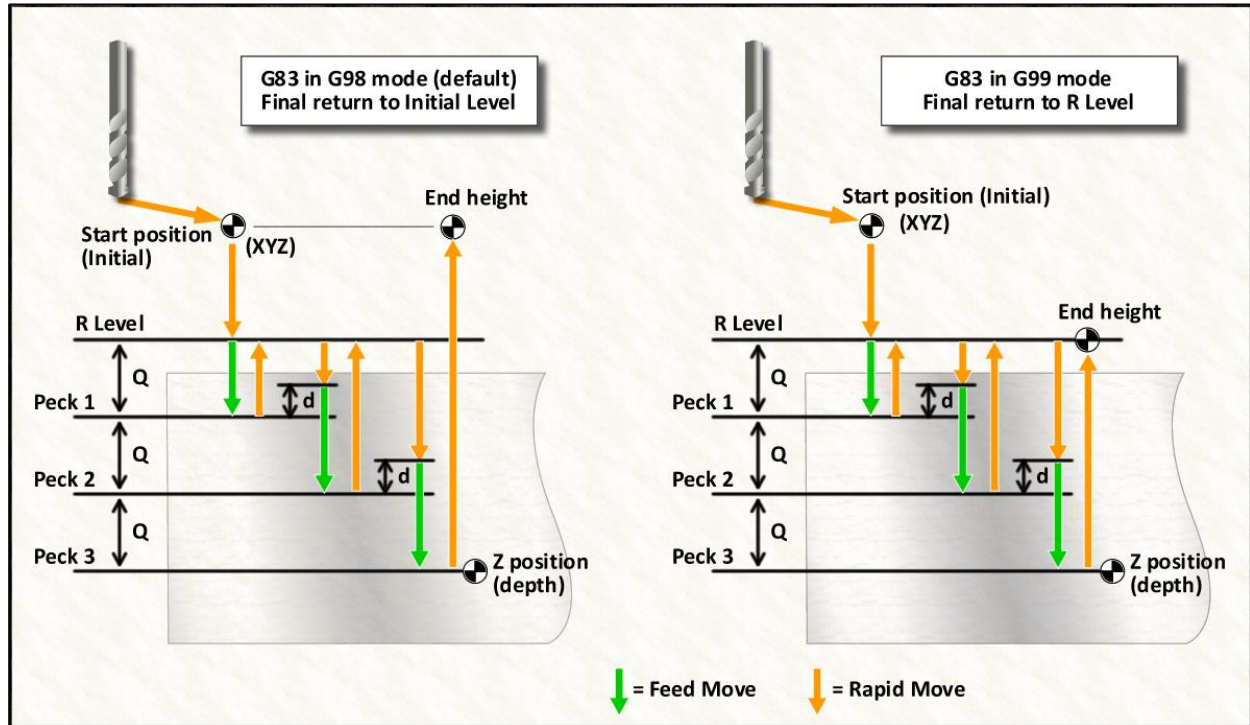


G83 – Peck Drilling Cycle (As of v5.33 VR Milling)

This cycle performs peck drilling – feeding down a hole and intermittently retracting out at rapid to remove material built up on the tool.



Command format:

G83 X... Y... Z... R... Q... F... K...

X,Y = The X,Y coordinates of the hole

Z = Position of the bottom of the hole

R = Start level of the cycle (should be above top surface of material)

Q = Depth of each peck cut below the R level (always taken as a positive value)

F = Feedrate of each cutting peck move

K = Number of repeats (ie, cut the same hole a number of times)

d = Distance above previous cut to rapid back down to. This value is configured in the file **cycles.ini** (explained further down) and is usually set to 1mm.

Example G code:

```
G00 Z50
G83 G98 X10 Y15 Z-4.8 R1.2 Q2 F300
X20 Y15
G80 G00 Z50
```

Program sequence for above example:

Description	Axes positions	Relevant G code
Set Initial Z Level	Z50	G00 Z50
Move to hole XY position	X10 Y15	G83 ... X10 Y15
Rapid to R level	Z1.2	R1.2
Feed to first depth (R-Q)	Z-0.8	Q2
Rapid up to R level	Z1.2	R1.2
Rapid to pos above last depth (R-Q) + d	Z0.2	d=1mm
Feed to next depth (R-2xQ)	Z-2.8	Q2
Rapid up to R level	Z1.2	R1.2
Rapid to pos above last depth (R-Q) + d	Z-1.8	d=1mm
Feed to next depth (R-3xQ)	Z-4.8 (final depth reached)	G83 ... Z-4.8
Rapid back to R Level (G98)	Z1.2	R1.2
OR		
Rapid back to Initial Level (G99)	Z50	G00 Z50
Rapid to next hole position	X20 Y15	X20 Y15
Repeat above pecking cycle and drill another hole		
Cancel modal G83 cycle		G80 G00 Z50

NB, the height above each peck to return to (value d) is set in the file **cycles.ini** which is normally installed with V5 VR Milling in this directory: *C:\Program Files\Denford\VRMilling5*

To alter the d value, open the file in a text editor (eg: Notepad)

Under the section for G83 [83], the command **Peck Z = 2** would set the d value to 2mm, so each peck down would start 2mm above the previous cut depth.