

Fixed Cycles Q+A

➔ Answers to questions:

1. Fixed cycles are used for machining holes. As the sequence of operations is always predictable, it can be pre-programmed, and each hole requires only one block of data, rather than many, if cycles were not applied
2. The initial level is the last tool Z-position in the program, before a fixed cycle is used
3. R-level is a clearance, usually close the part, from which the cutting feedrate starts
4. Z-depth is a tool position at the end of each hole
5. Dwell in a fixed cycles uses the address **P**, which accepts only milliseconds as input. For example, 0.5 seconds is 500 milliseconds, programmed as P500 (*also see answer 10*)
6. The Q-address can be used either as a peck drill amount in G73 and G83 cycles, or as a shift amount in G76 and G87 cycles
7. Only G87 cycle - it is a back boring cycle, where the tool starts below the hole and feeds up
8. G85 cycle has five built-in sequences:
 - a - Rapid motion to the XY position
 - b - Rapid motion to the R-level
 - c - Feedrate motion to the Z-depth
 - d - Feedrate motion back to the R-level
 - e - Rapid retract to the initial level (with G98) or remain at the R-level (with G99)
9. The Z-address in any fixed cycles always indicates the final depth - the tool will never cut past this position
10. The **P** address indicates a dwell, programmed in milliseconds (*also see answer 5*)
11. The shift amount in G76 cycle is normally programmed by the **Q** address, but some controls may be set to accept **I** and **J** motions instead
12. To return to the initial level after the hole has been completed, program the cycle in G98 mode (*also see answer 18*)
13. L0 (or K0) in a fixed cycle means - '**store the provided data, but do not apply them at this hole location**'
14. Two cycles - G83 and G73 - are normally used for peck drilling. G83 returns to the R-level after each peck, whereby in G73, the tool only clears by a small amount, set by the control
15. The initial level must be set higher than the height of any obstacle above the holes

16. Preparatory command G80 should be used as the correct method to cancel any fixed cycle
17. Any cycle can be repeated more than once, by programming the **L** or the **K** address with the number of repetitions. For example, L5 will repeat the active cycle five times.
The repetitive count is usually programmed in incremental mode
18. To return to the R-level after the hole has been completed, program the cycle in G99 mode (*also see answer 12*)
19. Most programmers would agree that G85 cycle (feed-in / feed-out) has the most advantages.
Other cycles may be used as well, if a certain method of reamer retract is desired
20. In everyday use, the dwell (address **P**) is programmed for G82 and G89 cycles.
Other cycles, namely G88, G76, even G74 and G84 also accept dwell, but not as a normal requirement
21. Cutting feedrate always starts from the R-level, never from the initial level, unless both positions are identical
22. Only two cycles use the shift from centerline (programmed by the **Q** address) - G76 and G87.
Orientation of the tool tip (of the boring bar) must be set correctly at the machine
23. Left hand tapping is programmed with G74 cycle, in M04 mode (spindle rotation CCW)
24. All cycles feed to the Z-depth, but not all cycles feed back to the R-level.
Those that do feed back are G74, G84, G85 and G89
25. Chipbreaking is a type of peck drilling. G73 peck drilling cycle is more efficient (therefore more useful) for chipbreaking than G83, but either cycle can be used, if the proper conditions exist