

Machine Zero Q+A

➡ Answers to questions:

1. **G27** Machine zero reference position return check
 ... confirms that axes are at machine zero - was very handy with the old G92 position register command
- G28** Return to the PRIMARY machine zero reference position
 ... this is the normal and most common method - through an intermediate point
- G29** Return from the machine reference position
 ... opposite of G28 - returns axes to their original position through an intermediate point
- G30** Return to the SECONDARY machine zero reference position
 ... used mainly with special equipment, such as pallets - also through an intermediate point (similar to G28)

2. Secondary machine zero position is an addition to G28, uses G30 command (or G30 P. for more than one).
 Its main application is to provide another reference on the machine, to align additional equipment, such as pallets

3. Intermediate point is the axes location through which the tool will travel before making the actual move
 to machine zero. Used as part of G28/G30 block, mainly to bypass obstacles.

4. There are three methods that will move the machine axes to the zero position:
 - 1 - Manually, via the machine operation panel (control panel) - usually one axis at a time
 - 2 - Through MDI, by inputting the required program block(-s)
 - 3 - In the CNC program, using program commands

5. Machine zero reference point, also known as the 'home position', is a fixed position on the CNC machine,
 that can be reached repeatedly, on request, with always the same results - it is the start point for various settings

6. **N37 G28 X220.0 Y135.0 Z10.0** (absolute mode was assumed)

7. G28/G30 are non-modal commands, and must be repeated in every block, when required

8. Block for item A: **G90 G28 X13.5 Y8.75 Z1.0**
 Block for item B: **G91 G28 X0 Y0 Z2.25**

9. These are machine zero return commands used on CNC lathes - absolute (top) and incremental (bottom)
 USE CARE - X0 Z0 means the part origin is the intermediate point !!!

10. These are machine zero return commands used on CNC machining centers - absolute (top) and incremental (bottom)
 USE CARE - X0 Y0 in absolute mode means the part origin is the intermediate point !!!