

Datum Shift Q+A

➔ Answers to questions:

1. Work offsets G54-G59 can be considered, but the real datum shift is done through G52 (local coordinate system) and G10 (programmable datum shift).
2. G53 is a preparatory command for **Machine Coordinate System**. Its main application is to reach a specific position on the machine table, regardless of where the program zero is. G53 is program independent, always measured from machine zero.
3. G53 is a non-modal command, so there is no command that would cancel it. It can be said that G53 cancels itself.
4. Either G92 or G50 had been used for years as means of registering the tool position into the control system. They have been replaced by the modern G54-G59 work offset commands (work coordinate system).
5. The following program is equivalent to the one shown in the original exercise:


```

N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X3.5 Y2.0 S1200 M03
N4 G43 Z1.0 H01 M08
N5 G99 G81 R0.1 Z-0.5 F8.0
N6 Y4.0
N7 X1.5
N8 Y2.0
N9 G80 Z1.0 M09
N10 G28 Z1.0 M05
N11 M01
      
```
6. G10 is a command that is very control dependent, although there are similarities. The main purpose of G10 is to input offset values into the control system, from the CNC program. Of course, these values must be known. Also, G10 may be used to update current setting through the program, even if the actual settings are not known.
7. The block structure is G10 L2 P. X. Y. Z...:

For example, `G10 L2 P2 X-305.556 Y-287.412` will set G55 work offset to X-305.556 Y-287.412, while Z setting remains the same.
8. Local coordinate system (G52), provided it is available on the control, allows the programmer to shift zero from a convenient setup location (for example, a corner of a part, to a convenient program location (for example, a center of a bolt hole center) in the same setup.