G90 and G91   Absolute and Incremental mode selection

G90 and G91 set the mode of operation of the control. These commands are used in the program. Once one of these commands are used it stays in that mode until it is changed. There should be one of these commands in the first line of your program.

G90 - Absolute mode : distances given will move the tool relative to an absolute zero.

G91 - Incremental mode : distances given will move the tool relative to where the tool is.

NOTE: You will find that almost all (if not all) of your programs will be done in G90.

G92 is a position preset.

This is not a move command. When this command is executed the tool does not move, it changes where the control thinks it is.

After a tool change, a G92 can be used to set the location value to what you want the tool to be moved to. After the tool is called and it is in position use:

G92 X0 Z.2   This will set the position display to X=0, Z.2

The G92 command does not effect the reference zero, "HOME". Even after you preset a location with the G92 the control knows where the reference zero is.

NOTE: If the tool is not starting at X=0 then the values will be either radius(G73) or diameter(G72) depending on the mode you are in.

G94 and G95 Feedrate selections

These set the feedrate mode. After a feedrate is selected, it remains until it is changed.

G94 - feed set at inches per minute
Slowest: .00005" per minute
Fastest: 300" per minute

G95 - feed set at inches per revolution
Slowest: .00005" per revolution
Fastest: depends on spindle speed, the limit has to be calculated to 300 ipm

as an example, the fastest feed rate at 3000 rpm is .1"/rev