The CNC Controller is a software Program called Mach 3 and provides for controlling all the movements and associated commands. The CNC controller (Mach) does not know a practical Machine Reference Point to work from, thus, the controllers “0,0” is somewhat meaningless and has unlimited movement capability. When the controller is turned on it monitors axis movement starting from it’s “0,0” value for each axis and displays the absolute movement. Machine X & Z=0 can be anywhere and can be set by the user manually, via switches, and other ways.

Machine Zero is a fixed point within the machine travel limits and does not normally change. It is typically called Machine Reference point, machine zero, or simply Home Position

Program Zero, or a better description Part Zero, is a reference point defined during coding of a program. It is not known by the machine until defined. It is the origin for all coding in the program.

BASIC WORK OFFSET DESCRIPTION

The controller does not know where Part Zero is and that needs to be defined.
The controller knows where the machine was Referenced / Homed / Machine Zero point is located and a Work offset is used to define where the part is in relation to Home. The distance is always defined from Home to the Part. Note that home is a point on the cutting tool!

G54 is the default work offset when Mach is started. The G54…to G5x is called Global and there are other offsets. G52 is an offset Local to the G54. G52 is a temporary work offset. And there are G code commands to change the work offset or tool offsets.

Milling machines use Work Offsets and they are called Fixture Offsets. The fixture offset has a zero reference point (Datum) and a part within the fixture can have a temporary offset whose reference point (Datum) is relative to the fixture reference point. The programmer can make use of the offsets depending on what and how something will be machined. This is graphically portrayed in FIGURE 2.

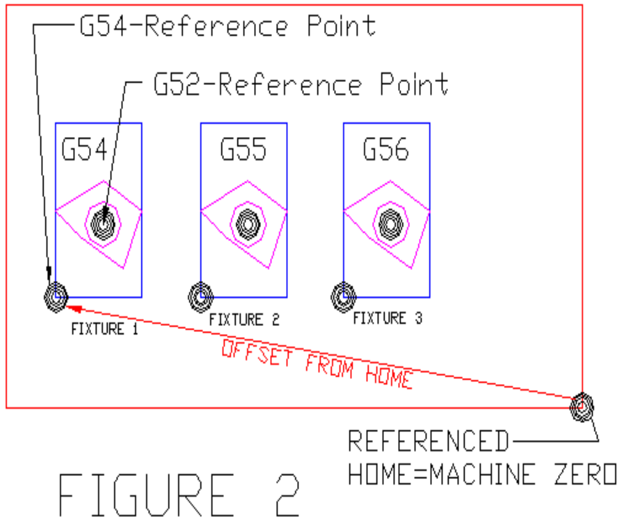

FIGURE 2

G54-G59 and G59 P1-254 Work Offsets

The G54 – G59 offset is a defined Datum point with their offsets from "HOME to Work/ Part" and can be used to relate two coordinate systems. ie; Home position (machine coordinates) to part position shown in Figure 3. Not to confuse, but, the reader may also find work offsets called fixture or work coordinate systems.

Work offsets provide for changing / adjusting an offset without affecting the other offsets. Initial offset of G54 are and should be based on Part zero and allow movement / enough offset value to a tool change location.

Here is my simplified version of Mach's 255 Work Offsets:

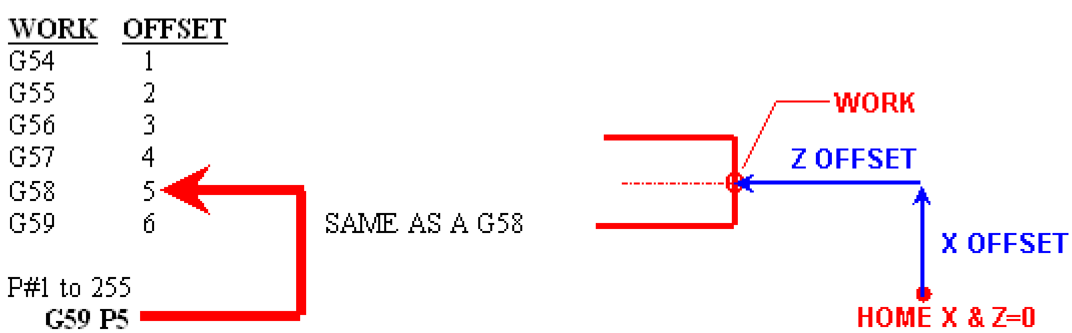

FIGURE 4

<u>Mach Code description – G59 Command (shown in FIGURE 4)</u>

To select work offset #1, program G54, and similarly for the first six offsets. The system-number-G-code pairs are: (1-G54), (2-G55), (3-G56), (4-G57), (5-G58), (6-G59)
To access any of the 254 work offsets (1 - 254) program G59 P~ where the P word gives the required offset number. Thus G59 P5 is identical in effect to G58.
It is an error if:

- one of these G-codes is used while cutter radius compensation is on.