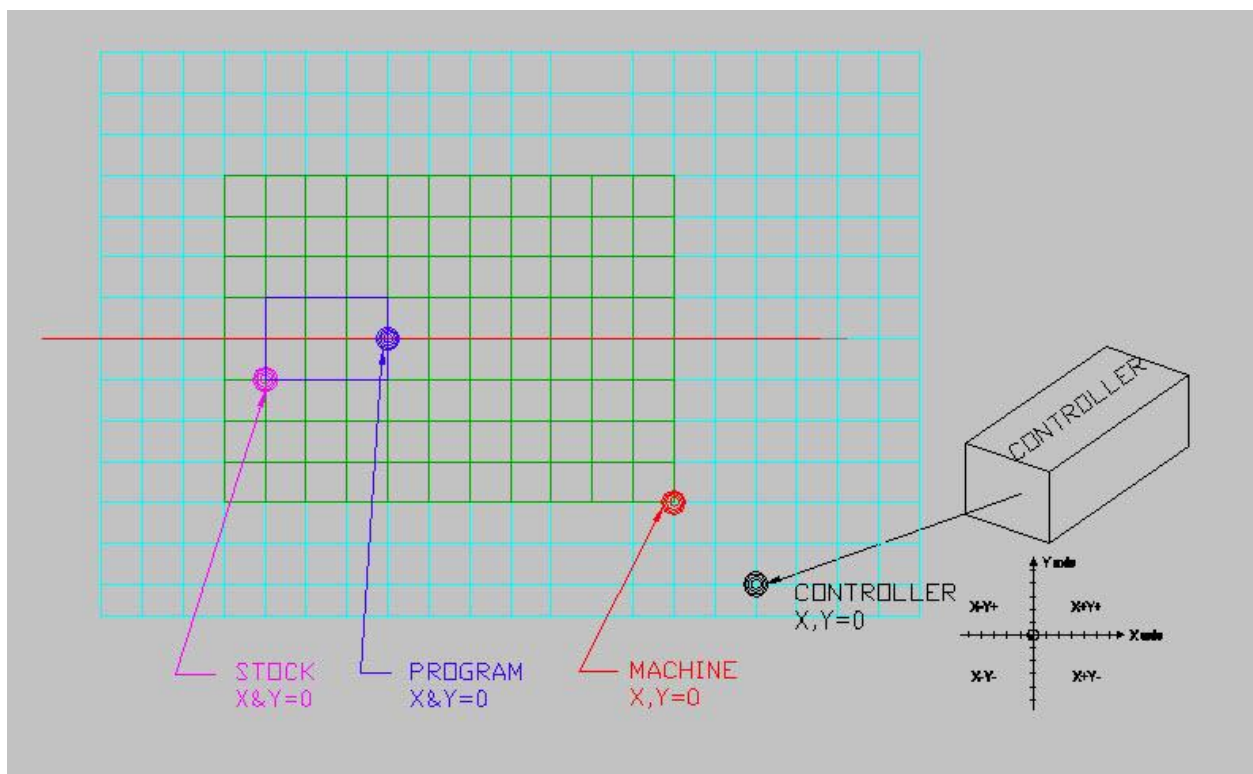


The CNC Controller is a software Program called Mach3 and provides for controlling all the movements and associated commands.. The CNC controller (Mach) does not know a practical Machine Reference Point to work from, thus, the controllers “0,0” is somewhat meaningless and has unlimited movement capability. 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 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.

So the blue grid portrays an exaggerated possible tool movement. The green grid shows the possible tool travel. The user can set home at a more convenient location. Stock on the machine is shown in purple and the location of the stock / part away from Home is called Part Zero. The above is graphically depicted in figure below.



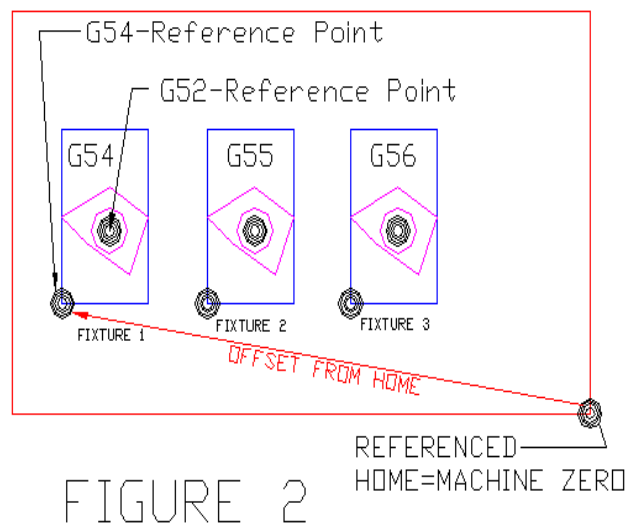
BASIC WORK OFFSET DESCRIPTION

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.

G54 is the default work offset when Mach is started. The G54...to G5x are 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.



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 is the G54 offset. Note that home is a point on the cutting tool!

G54 is the default work offset when Mach is started. A user can choose a different offset but do so with care and for reason.

To define where Part Zero is to the controller the tool is touched off to the part and with the proper button click a work offset is created from home to the part