BASIC CNC LATHE INFORMATION

The CNC Controller is a software Program called Mach 3 Lathe and provides for controlling all the movements and associated commands. The axis directions are shown next to the controller in Figure 1 and are about the center of the lathe spindle. Some lathes are front and some are back thus the X direction moves are reversed. 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. (Note, do not confuse 0,0 with a switch position, as the switch just provides for a repeatable carriage location.)

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. Machine zero is located at the furthest distance from the lathe centerline and also as far as possible from spindle head (because different chucks, faceplate etc can be used). The position provides for full available machine usage and is the usual setup. A user may decide to define a different location for Home.

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. For the lathe, Part Zero is the X axis Zero X=0 which is always the center line of the spindle. Z axis Zero can be anywhere, but, usually the front of the stock / machined surface after a facing cut is made is used for Z zero.

  So the blue grid portrays an exaggerated possible carriage movement. The green grid shows the possible tool travel. The user can set home at a more convenient location and can also use that position as a tool change position (dark blue). Stock in the chuck is shown and the location of the part away from Home is called Part Zero. The above is graphically depicted in Figure 1.
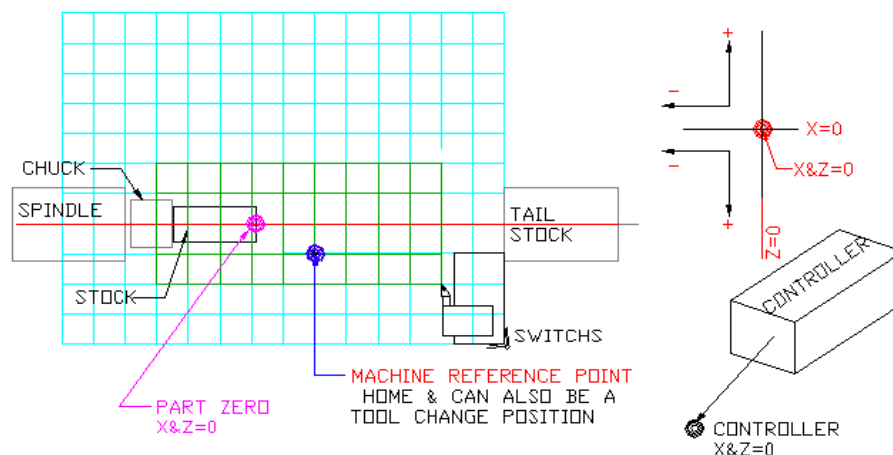

FIGURE 1

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 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
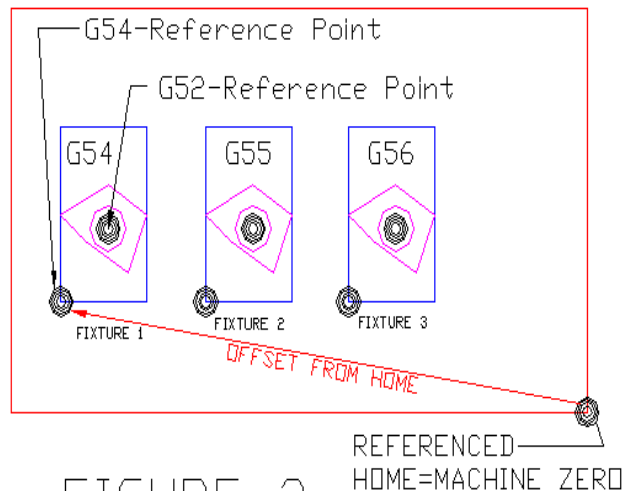be machined. This is graphically portrayed in FIGURE 2.

FIGURE 2

Mach 3 Lathe stores the Work Offsets for each G code position in the Work Offsets
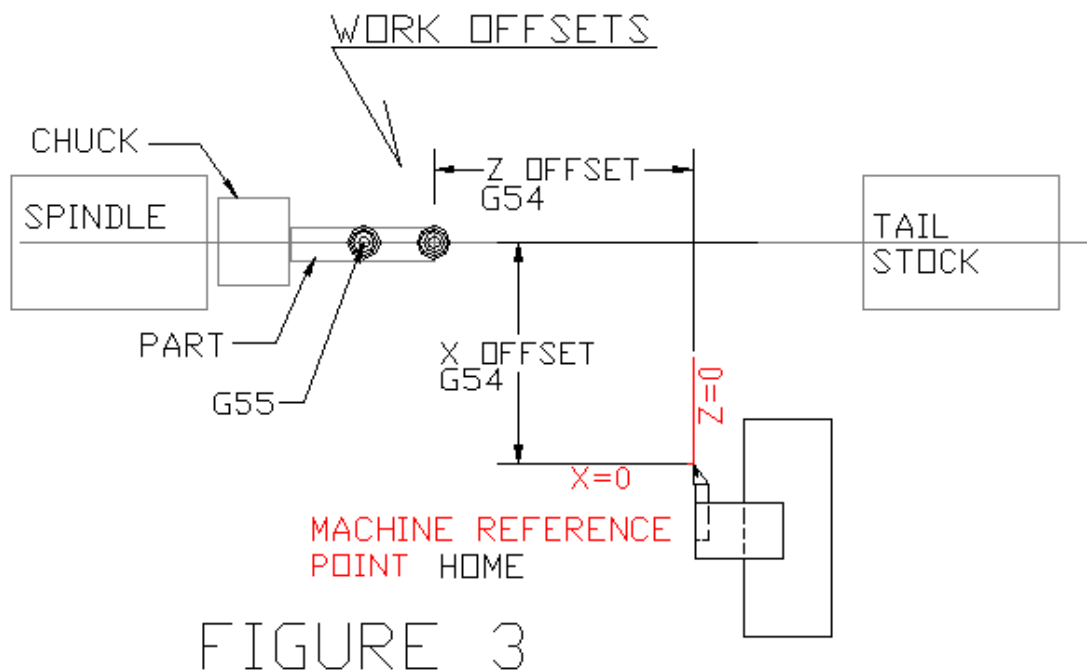Table shown below.

G-Code Pos	X	Y	Z	A	B	C	Name
G54	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	G54
G55	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G56	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G57	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G58	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G59	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G59P7	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G59P8	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	

The work offsets are similarly used for the lathe as portrayed in Figure 3.
The lathe dose not have fixtures (so to speak of !), rather work offset datum's are used in programming.

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.

FIGURE 3

NOTE:
There are different kinds of offsets used for different tasks and basically used in program coding and give the controller instructions. The word offset is used rather loosely and one needs to be distinct in the description when communicating. A good example would be saying that clicking a button on the lathe screen will create an offset.

LATHE COORDINATES / AXIS DRO'S GENERAL OVERVIEW

Think in terms of where the tool (controlled point) is relative to user defined Datums, Work offsets, Part, etc. The lathe carriage just moves the tool!

Three buttons provide for selecting what coordinate location will be displayed in the axis DRO. The coordinate systems are as follows:
- Machine Coordinates
- Part Coordinates
- Program Coordinates

The current position of the tool is defined by the following formula.

	Current Position		Machine Coord		WorkOffset		G92/ G52 Offset		Tool Offset
X Pos	+0.0000	=	+0.0000	-	+0.0000	-	+0.0000	-	+0.0000
Z Pos	+0.0000	=	+0.0000	-	+0.0000	-	+0.0000	-	+0.0000

The formula can be manipulated:

Current Position = Machine Coord - Work Offset - G92 Offset - Tool Offset
Machine Coord = Current Position + Work Offset + G92 Offset + Tool Offset
Work Offset= Current Position - Machine Coord - Work Offset - G92 Offset -Tool Offset
(typical for X & Z axis and values can be + or -)

Where the tool is (Current Position) relative to Machine Coordinates is influenced by the other factors of the formula. The Part Coordinates displays where the tool is when Work and Tool Offsets are in affect (doesn't include G92/G52) *and excludes machine coordinates.* The Program Coordinates are like Part Coordinates but include any G92 / G52 offsets. So if there are no G52 or G92 (one or the other / not both) offsets then Part = Program Coordinates. If there are no G54, G52, or tool offsets then Part Coordinates = Machine Coordinates.

MACHINE COORDINATES (MC)

MC GENERAL
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. For CNC machining the + or – are indicative of the direction as related to the distances from the fixed point of "0,0". Machine Coordinates are absolute distances from a fixed point (Datum) namely "0,0". Machine X & Z=0 can be anywhere and can be set by the user manually, via switches, and other ways.

<div align="center">SO</div>

MC are normally used to move to some location and declare that location as Home (Referencing).

<div align="center">BUT</div>

It can be used to advantage to move (ie ; G53) to an exact location irrelevant of other offsets that are current. Say, to a tool change position or to create a reference point to measure from.

- The machine coordinates always display position in absolute values exclusive of any Work / Global or local offsets.
 - X axis is in terms of Radii even if in diameter mode.
- The DRO value will continuously change when the axis is moving in any direction. Mach is keeping track of axis movement!
- Pressing the Zero World button will reset the DRO to zero and will also delete any current Work offset in affect.
- Do not press the ZERO axis button when in machine coordinates since it is to be used when in Part Coordinates. Do not put a value in the MC DRO and press the return key as that will create a work offset and the value in the DRO will not change.
- Pressing the Set Home button will reference the machine (LED turns green), the DRO value changes to zero, and any value that was in the DRO will become an axis offset value in the current Work Offset as can be seen by opening the Fixture table or selecting the Part DRO. It will only create the offset one time, and I will remark that this is not the way to create a work offset!
- The LED will stay Green unless you click the de-reference button.
- The Part and Program DRO's will be zero / same as the Machine coordinate if no offset values have been defined.

MACHINE COORDINATE USE

PART COORDINATES
Part Coordinates are distances from a fixed point which was defined in Machine Coordinates. For the X axis the DRO displays in terms of radii or diameter based on the users lathe configuration. The controller needs to know where the part is relative to machine coordinates. Part is the piece in the headstock to be machined as discussed here. The part zero can be located anywhere along the z axis. The part has coordinates associated with it and normally the center of the part is on the lathe X=0 axis and the end of the part is usually Z=0 in a gcode program. The software can't properly control the machining of the part if it doesn't know where the part is. Part coordinates can be equal to or even the same as Machine Coordinates. Thus defining the part location is very dependant on how a user sets up a job including how the gcode program was created. So part coordinates are used to relate or define where the part is relative to the machine coordinates.

In general, only the user can define where the part is located to the controller.

Part DRO for an unreferenced machine
Part Zero is a reference point defined during coding of a program. It is not known by the machine until defined. And is called Program Zero more accurately Part Zero. It is the origin for all coding in the program. So you have Part ORIGIN.

For the lathe Part Zero is the X axis Zero X=0 which is always the center line of the spindle. Z axis Zero can be anywhere, usually the front of the stock / machined surface after a facing cut is made.

- Part Coordinates are displayed in absolute values but unlike Machine Coordinates
   the X value will be in terms of Diameter OR Radii. The user determines the mode he
   wants to work in and configures Mach accordingly.
- Displays Work coordinates not inclusive of G92.
- The DRO value will continuously change when the axis is moving in any direction.
   Mach is keeping track of where the part relative to the machine coordinates
- The user must define to Mach where the part is relative to Home/ machine origin.
    If the part zero is the same as machine zero, unlikely, then no offset is required.
- If it is somewhere other than machine zero, then you need to define the dimensional
   distances to Mach. Note that the default G54 datum / reference of 0,0 can be anywhere
   along the Lathe bed. Normally part zero is at the center line of the X axis.

.

PROGRAM COORDINATES

The Program Coordinates are like Part Coordinates but include any G92 / G52 offsets. So if there are no G52 or G92 (one or the other / not both) offsets then Part = Program Coordinates. If there are G52 or G92 the DRO will display tool location to include them.

NOTE: The "Program" contains all the Gcode instructions for machining. The controller uses the program instructions to control tool movement and additionally other associated hardware.