6.2 USING G52 – TEMPORARY OFFSET

If a tool table was based on the cutting tip, and the user wanted to have a tool based on the center of the tip radius, a G52 command can used to accomplish it.

A G52 shifts the part zero within the current work offset by a specified amount. The G52 requires an active work offset. The G52 X….Z….. is placed before the two G0 moves and a G52 X 0.0 Z0.0 is placed after the moves to cancel the shift.

FIGURE 6.2.1

The temporary shift in the axis would cause the tool to be aligned upon the first axis move from home.

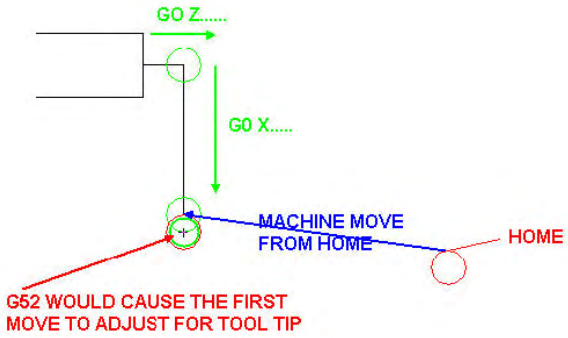


FIGURE 6.2.2

G52 EXAMPLE FOR X TOOL ALIGNMENT "only": TOOL RADIUS =.010"

G52 X-.010 (provides for adding the radius to the actual length traveled in the G0x....move)
G0 Z .....
G0 X.....
G52 X0.0 (cancels the G52 and returns to work offset)
G.....no difference from here as the code is still precompensted including the clearance.

COMMENTS:

1. If the same tool radius is used for all passes then only need to implement the G52 one time and home posting is only required once.
2. A G52 is always required on the first pass, and a G52 is always required if the tool radius changes from the previous one.

6.7 GCODE OPTIONS

LazyTurn provides all the code necessary to machine a turning. The user may find the following of value in setting up the lathe. Offsets were mentioned in prior sections of this appendix. This just provides some insight in offsets.

G54 – Work Offsets can relate two coordinate systems
G52 – Temporary Offsets work within the work offsets and can temporarily shift the G54 offset
G10 – Tool Offsets allows selection of a tool and will permanently change the G54 work offsets based on the tool
G92 – a legacy Gcode somewhat replaced by G52 and tells the controller where the current tool tip is from program zero. The values used are from program zero to current tool position along the axes.
G53 – Provides for a tool change position

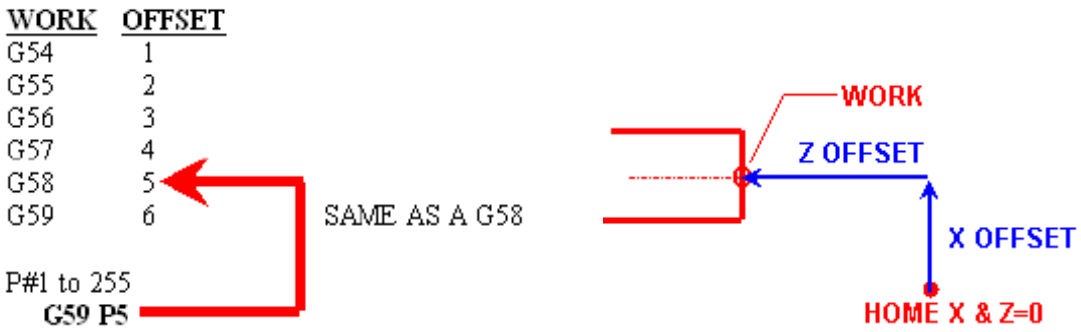
G10 Tool Offset
The G10 will permantely change the work offsets (G52 will not as it can be cancelled) and provides for selection of tool offsets. Note that the tool tip radius and wear offsets may be problematic in Mach (so I have read and frankly I don't use them in the tool table).

Mach Code description:
To set the offset values of a tool, program G10 L1 P~ X~ Z~ A~, where the P number must evaluate to an integer in the range 0 to 255 - the tool number - Offsets of the tool specified by the P number are reset to the given. The A number will reset the tool tip radius. Only those values for which an axis word is included on the line will be reset. The Tool diameter cannot be set in this way.
To set the coordinate values for the origin of a fixture coordinate system, program G10 L2 P~ X~ Y~ Z~ A~ B~ C~, where the P number must evaluate to an integer in the range 1 to 255 - the fixture number - (Values 1 to 6 corresponding to G54 to G59) and all axis words are optional. The coordinates of the origin of the coordinate system specified by the P number are reset to the coordinate values given (in terms of the absolute coordinate system). Only those coordinates for which an axis word is included on the line will be reset.
It is an error if:
¨ the P number does not evaluate to an integer in the range 0 to 255.
If origin offsets (made by G92 or G92.3) were in effect before G10 is used, they will continue to be in effect afterwards.
The coordinate system whose origin is set by a G10 command may be active or inactive at the time the G10 is executed.
The values set will not be persistent unless the tool or fixture tables are saved using the

buttons on Tables screen.
Example: G10 L2 P1 x3.5 y17.2 sets the origin of the first coordinate system (the one selected by G54) to a point where X is 3.5 and Y is 17.2 (in absolute coordinates). The Z coordinate of the origin (and the coordinates for any rotational axes) are whatever those coordinates of the origin were before the line was executed.

G54-G59 and G59 P1-254 Work Offsets
The G54 – G59 offset is the distance from "machine zero to program zero" and can be used to relate two coordinate systems. ie; Home position (machine coordinates) to part position. This was shown in Section 3.0 of this appendix. G54 is the default system. Not to confuse, but, the reader may also find work offsets called fixture or work coordinate systems. Most projects on the lathe are one of a kind and datum shifts are not usually required but can be useful when doing multiple turnings.

Here is my simplified version of Mach's description:



Another example of use would be making multiple turnings using the same code by providing an offset to do the other turnings. ie; G54 offset used for the first turning and then G55 Z (offset amount) such that the cutter would start at a different point along the Z axis for the next turning.

Mach Code description
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.

G52 Coordinate System Offset
Use of G52 was covered in Section 6.2 of this appendix providing for a temporary offset within an offset. It basically provides a way of offsetting the original offset (G54) by some value and can be cancelled, thus, if cancelled the original work offsets are not changed as would happen with G10. I have read that it basically replaced the G92.

Mach Code description:

To offset the current point by a given positive or negative distance (without motion), program G52 X~ Y~ Z~ A~ B~ C~ , where the axis words contain the offsets you want to provide. All axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed.

It is an error if:

¨ all axis words are omitted.

G52 and G92 use common internal mechanisms in Mach3 and may not be used together. When G52 is executed, the origin of the currently active coordinate system moves by the values given.The effect of G52 is cancelled by programming G52 X0 Y0 etc.

The axis offsets are always used when motion is specified in absolute distance mode using any of the fixture coordinate systems. Thus all fixture coordinate systems are affected by G52

G92,G92.1,G92.2 & G92.3 Offsets

See the chapter on coordinate systems for full details. You are strongly advised not to use this legacy feature on any axis where there is another offset applied.

To make the current point have the coordinates you want (without motion), program G92 X~ Y~ Z~ A~ B~ C~ , where the axis words contain the axis numbers you want. All axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed. It is an error if:

¨ all axis words are omitted.

G52 and G92 use common internal mechanisms in Mach3 and may not be used together. When G92 is executed, the origin of the currently active coordinate system moves. To do this, origin offsets are calculated so that the coordinates of the current point with respect to the moved origin are as specified on the line containing the G92. In addition, parameters 5211 to 5216 are set to the X, Y, Z, A, B, and C-axis offsets. The offset for an axis is the amount the origin must be moved so that the coordinate of the controlled point on the axis has the specified value.

Here is an example. Suppose the current point is at X=4 in the currently specified coordinate system and the current X-axis offset is zero, then G92 X7 sets the X-axis offset to -3, sets parameter 5211 to -3, and causes the X-coordinate of the current point to be 7.

The axis offsets are always used when motion is specified in absolute distance mode using any of the fixture coordinate systems Thus all fixture coordinate systems are affected by G92.

Being in incremental distance mode has no effect on the action of G92.

Non-zero offsets may be already be in effect when the G92 is called. They are in effect discarded before the new value is applied. Mathematically the new value of each offset is A+B, where A is what the offset would be if the old offset were zero, and B is the old offset. For example, after the previous example, the X-value of the current point is 7. If G92 X9 is then programmed, the new X-axis offset is -5, which is calculated by [[7-9] + -3]. Put another way the G92 X9 produces the same offset whatever G92 offset was already in place.

To reset axis offsets to zero, program G92.1 or G92.2 G92.1 sets parameters 5211 to 5216 to zero, whereas G92.2 leaves their current values alone.
To set the axis offset values to the values given in parameters 5211 to 5216, program G92.3
You can set axis offsets in one program and use the same offsets in another program. Program G92 in the first program. This will set parameters 5211 to 5216. Do not use G92.1 in the remainder of the first program. The parameter values will be saved when the first program exits and restored when the second one starts up. Use G92.3 near the beginning of the second program. That will restore the offsets saved in the first program.