

A few comments on wording which is important.

A machine has extreme travel limits. A point within the limits must be defined to the controller ( Mach3) so that all other movements can be associated. That point is machine zero and as such is known, fixed and defined. Home is just a point within the limits and it can be anywhere, but, home is usually machine zero. So home is used interchangeably with machine zero many times.

Referencing is used to describe the condition where one has moved to some point and done something to tell the controller that the current position is machine zero or home. Homing can be done automatically, using the MDI, or manually and used to describe the motion of returning to machine zero. Fixture offsets are defined locations away from the home position. Fixture offsets are not to be confused with tool offsets. So an offset is just some distance from a point and needs to be always clarified. So much for the play on words!

G30 - behaves the same as G28 and is unique as compared to G28. G28 is associated with the primary machine zero. G30 allows for machine movements to additional machine zero's and requires use of other parameters along with the G30 command like P or whatever. Mach's G30 use is not defined in the manuals other than "or just use G30". So there is more to G30 and it's dialect of use may vary from manufacturer to manufacturer and can even be proprietary.

G28 intended use was to provide movement to an intermediate point before continuing on it's way to machine zero. It provided for eliminating some coding in a program and best example of use is to avoid an interference in the tools path to machine zero. G28 by itself is an incomplete command to some controllers but Mach ( and some other controllers ) provide for movement without axis definitions resulting in a transverse move to machine zero. As noted in Mach definitions, the intermediate point is the current point and only one movement is made when no axis words are given, where as in Smid's definition, one would need to add the current point / axis words as part of the G28 definition. One must be careful in it's use since G90, G91, and additionally Fixture Offsets can affect the resulting machine movements when G28 is commanded. So it's always interesting to see how different controllers use the commands but in the end the important definition is the one for the controller that is being used.

Smid reference says:

G28 X0.0 "will only send the X axis to the X axis zero reference position".

But as Smid points out the axis values associated with the G28 should always indicate an intermediate point and at least one axis must be specified.

In MACH Lathe:

G28 X0.0 - if a G54 / work offset exists and you are away from the exact offset value then you will have two movements. IE; it will first go to the intermediate value ( the work offset value ) and then go to X axis machine zero.

G28 X0.0 – if G54 exists and you are at the exact offset value then you will have one movement ie; since your at the intermediate value there is no need to move to it so one move only to the X axis machine zero.

G28 X0.0 – if no offset exists there is no intermediate movement and the axis just goes to X axis machine zero.

G91 G28 X0.0 – there is no movement since mode is changed to incremental and the request is for zero movement

G28 – only one movement back to machine zero irrelevant if there is a work offset

All of the above is with tool zero / master tool but it's worth going through the above with a call for a different tool. May as well try them with additional offsets.

RICH

OK with Mach3 ,

Machine Home (zero)

Can be set by switches,

OR REhome Button if you have no switches,

OR is set by mach when you OPEN it to the current position

OR can be zeroed to a position via CB

OR rumored to be settable via #vars 5161-5166 (untested as yet)

Moves to get to machine home,

G53 X0 Y0 Z0 ----- Makes a straight traverse run to Machine Zero

With G30 ALL axis go home

G30 ----- Makes a straight traverse move to machine zero

G30 Z0 -----Makes a single Z move to Zero THEN all AXIS make a straight traverse move to Machine zero

G30 Z0 X0-----Makes a Straight traverse move to Z0 X0 then all axis makes a straight traverse move to machine zero

ETC,ETC

Local Home(G28), Is an offset from Machine ZERO. With G28 without defined axis all axis goes home in the order of Z then all the rest. USED with a defined axis ONLY the defined axis goes home via the intermediate point.

It can be the same as Machine zero or different as defined

Moves to get to Local Home

G28 ----- Raises Z first then makes straight traverse moves in XY to Local Home(LH)

G28 Z-1 ----- Single axis move to intermediate point(-1.000) and then goes to Z0. Z axis ONLY goes to LH

G28 Z0 X0 ----- ONLY the defined AXIS will go to LH UNDEFINED axis remain as is . Makes a straight traverse move in ZX to 0 then ZX moves in a straight traverse move to LH.

ETC,ETC

Part HOME ( Part Origin, X0Y0 Z0)

G0 Z0 X0Y0 ----- Makes a straight traverse RAPID move to Z0 X0Y0

G1 Z0 X0 Y0 ----- Makes a straight traverse FEEDRATE move to Z0 X0 Y0

Fixture Home

G55 G0 Z0 X0 Y0 ----- Swithes the Fixture offset then goes to X0 Y0 Z0 as straight traverse move at rapid speed

G56 G1 Z0 X0 Y0 F100 ----- Swithes the fixture offset then goes to X0 Y0 Z0 as straight traverse move at feedrate speed

\*\*\*\*\*

You will note that the G28/30 in MACH3 is backwards to FANUC . In the fanuc world the G30 local Homes can be set in the parameters AND there can be MANY etc; G30 G30.1 G30.2 G30.3

 etc,etc

Also the Axis calls in G28/30 are different from each other. In g28 you can call a single axis to home, in G30 ALL axis go home regardless

Just a thought, (-)TP

*« Last Edit: March 12, 2012, 12:06:56 PM by BR549 »*

IF you use a G28 X0 Y0 call then you get a straight traverse move (combined) to home. the X will NOT move first both move at the same time.

To get what you need (IF the G28 and G30 positions are the same) I would use

G30 X0

That will Home X first THEN home Z. That way you have Safe clearance for all axis to travel.

Other wise you would need to do

G28 X0 (home X)

G28 Z0 (home Z)

Just a thought, (-) TP