

Switching to G28 from G53

April 2, 2009

1.0 G28 Simulation and Posting

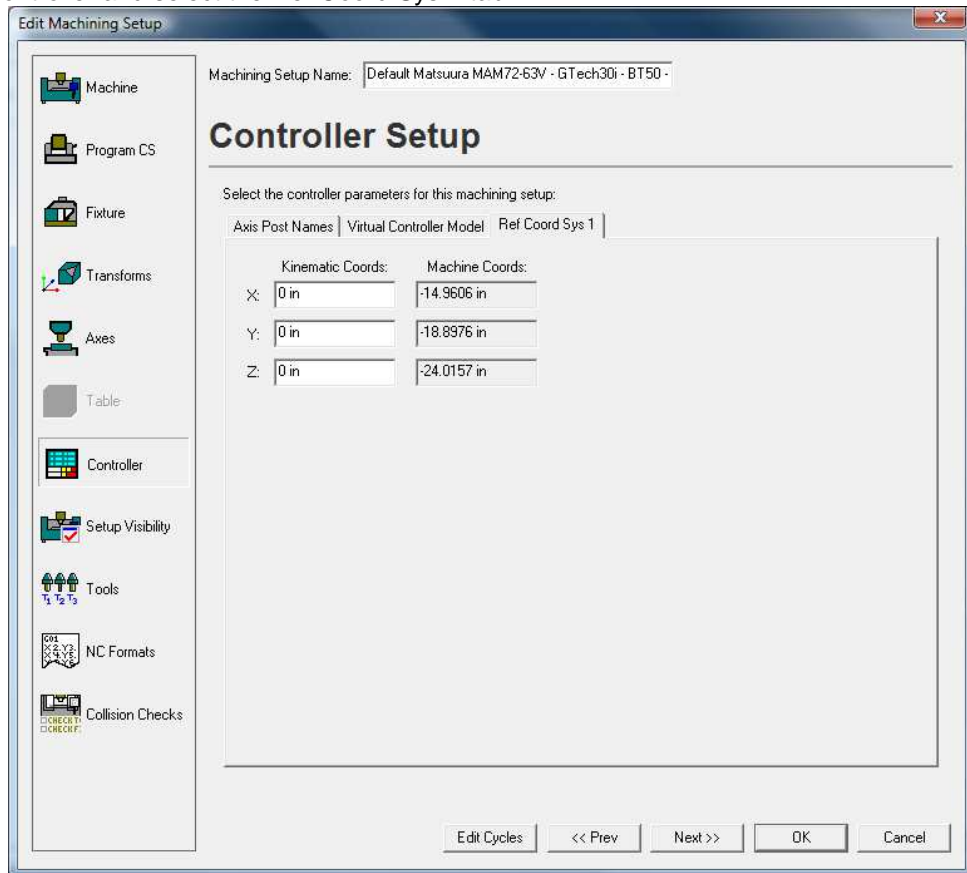
To setup the G28 NC Format, set the format up with the following codes:

End 3+2 Orientation Lines:

Index	Default Settings	Code Layout
1	<input checked="" type="radio"/> Rapid <input type="radio"/> Linear (Cuttin: <input type="radio"/> Circular Interp: <input type="radio"/> Circular Interp:	G00
2	<input checked="" type="radio"/> Cancel Tool L <input type="radio"/> Tool Len Corr: <input type="radio"/> Tool Len Corr: <input type="radio"/> None	G49
3	WP CS Set Plane Spatial Reset	G69
4	<input type="radio"/> Abs Coords <input checked="" type="radio"/> Incr Coords <input type="radio"/> Incr Coords (H <input type="radio"/> None <input checked="" type="radio"/> Rapid <input type="radio"/> Linear (Cuttin: <input type="radio"/> Circular Interp: <input type="radio"/> Circular Interp: <input checked="" type="radio"/> First Datum <input type="radio"/> Additional Datum <input type="checkbox"/> Set X 0.000000 <input type="checkbox"/> Set Y 0.000000 <input checked="" type="checkbox"/> Set Z 0.0 mm	G91 G00 G28 Machine Pos:XYZ
5	<input checked="" type="radio"/> Abs Coords <input type="radio"/> Incr Coords <input type="radio"/> Incr Coords (H <input type="radio"/> None	G90

This will output the correct G28 G-Code. You also need to set up the location of the G28 reference coordinate system.

1. Go to Resources > Machining Setup Editor
2. Hold the SHIFT key and click EDIT
3. Click Controller and select the Ref Coord Sys 1 tab



4. This offset is measured relative to the kinematic coord system. So to set the reference to the top of the Z-axis travel, set this value to 610 mm for Z (for example for the 63V). You can also set it to the opposite of the value shown in the Machine Coords, which will make the machine coords value become 0. Different machines will have different numbers
5. Click OK to close and commit this change