USING G28 ON YOUR HURCO MILL

Instructions:

For this process to work you must have a G-Code program active

Step 1: Press the INPUT button

Step 2: Select PROGRAM PARAMETERS

Step 3: Select NC PARAMETERS

Step 4: Select NC CONFIGURATION PARAMETERS

Step 5: Set your X and Y REFERENCE POINTS to your desired values. (Example: If we wanted the table to come to the very front of the machine after the cycle is completed for loading and unloading parts, or automation purposes we would do the following steps. 1. For the X REFERENCE POINT, we would input a value that is equal to half of the machine's X-axis travel. 2. For the Y REFERENCE POINT, we would input a value equal to the machine's max Y-axis travel)

NOTE: The X and Y reference points automatically default to 0. NOTE: These values are always input as MACHINE RELATIVE values, and not part releative.

**We recommend you do not change the Z REFERENCE POINT, only change the Z-axis reference point if you are confident in what you are doing. We know there are times you will want to put your own value in the Z-axis field, but only do so for very specific reasons and do so with caution.

NOTE: The NC code G91 G28 X0 Y0 will always appear in the code as G91 G28 X0 Y0 or G91 G28 Z0 regardless of any custom values present in the NC CONFIGURATION PARAMETERS screen for XY REFERENCE POINT.