This is where I ended up with my M6 remap. I made changes so that It could be used with G54 through G59.2 offsets. The one thing that was throwing me for a loop and giving strange results was that the G-Code file generated by Fusion360 was
calling the M6 before calling G54 etc. Everything worked fine if I used only G4 but when I started testing the same G-Code file where I had just changed G54 to G55/G56 I would get unexpected results. Then I realized that if it didn't know the work coordinate before the M6 then it couldn't retrieve the correct offset value to add the tool length to. Once I added the appropriate call for the coordinate system it started working correctly. I am going to look at the changing the post processor to fix that. I would also like to integrate this a little tighter with the QTDragon UI so that everything can be done with pushing buttons rather than manually running files.
Objective

My objective was to be able to use a mix of fixed length tools and variable length tools. During the tool change I only wanted to measure the variable length tools like drill bits or other rarely user tools that I don't keep in a dedicated tool holder. My setup is a Sieg X3 mill running LinuxCNC 2.8 using the QTDragon screen with an ASUS 1920x1080 touch screen and 3D probe. I based my remap on the work done by Peter Dam that I found through this post
https://forum.linuxcnc.org/qtvcp/39423-qtdragon-new-tool-length-after-a-manual-tool-change?start=20#190243.

Workflow

Start the machine and home all.
Touch off work piece X,Y,Z, using the 3D probe. See note below.
Load the G-Code file.
Cycle start
At each tool change the user is prompted to load the tool and indicate when the tool change is complete.
If the tool is a fixed length tool indicated by the presence of a Z value other than zero in the tool table machining will proceed using the offset from the tool table.
If the tool is variable length, the tool will move to the tool setter location and will probe the sensor to determine the tool's length and then machining will proceed.

Note: When the work piece is probed in the Z direction with the 3D probe, the value of the touch-off minus the length of the probe is stored in parameters #4001-4008 for G54-G59.2 respectivly. This measurement would be as if the spindle nose is sitting on the work piece. The length of the probe is stored in the .ini file.

Setup

Place the m6remap.ngc file in the ~linuxcnc/nc_files/probe/basic_probe/macros directory.
Place the measure_tool.ngc file in the ~linuxcnc/nc_files director.
Place the measure_toolsetter.ngc file in the ~linuxcnc/nc_files directory.

Add the following lines to the .ini file under section [RS274NGC]
REMAP=M6 modalgroup=6 ngc=m6remap.ngc
SUBROUTINE_PATH = macros

Add the following lines to the .ini file where <probe length> is the effective length of the probe..
[PROBE]
LENGTH=<probe length>

Add the following in the [TOOLSENSOR] section of the .ini file.
X_LOC = <ABS X location of tool setter>
Y_LOC = <ABS X =Ylocation of tool setter>

Z_PROBE_START = <ABS Z location to begin probing>
This value should keep your longet tool just above the tool setter. Otherwise the long tool will rapid into the tool setter.

Add the following bold line to the probe_z_minus_wco.ngc file which is located in the ~linuxcnc/nc_files/probe/basic_probe/macros directory.
This will save the Z touch-off value minus the probe length for G54 - G59.2 in parameters #4001 - #4008 respectively.

G10 L2 P#5220 Z[#<z_minus_probed> + #<workspace_z>]
#[4000 + [#5220]] = [#<z_minus_probed> + #<workspace_z> - #<_ini[PROBE]LENGTH>]

Add the following lines to linuxcnc.var file. This will make parameters #4001-#4008 persistent.
```
4001   0.000000
4002   0.000000
4003   0.000000
4004   0.000000
4005   0.000000
4006   0.000000
4007   0.000000
4008   0.000000
```

Place the nose of th spindle within .25 inch of the toolsetter and run the measure_toolsetter_height.ngc file. This will set the G59.3 coordinate system Z zero at the top of the toolstter.

For every fixed length tool either enter the tool length in the tool table or insert the tool and select the tool from the tool table using the M6 Qn button. Move the tool to .25 inch above the tool-setter and run the measure_tools inch.ngc file. This will measure the tool and write the value into the tool table.