HOW TO PERFORM A TOOL CHANGE AND LEARN A TOOL LENGTH (TOOL TOUCH OFF)
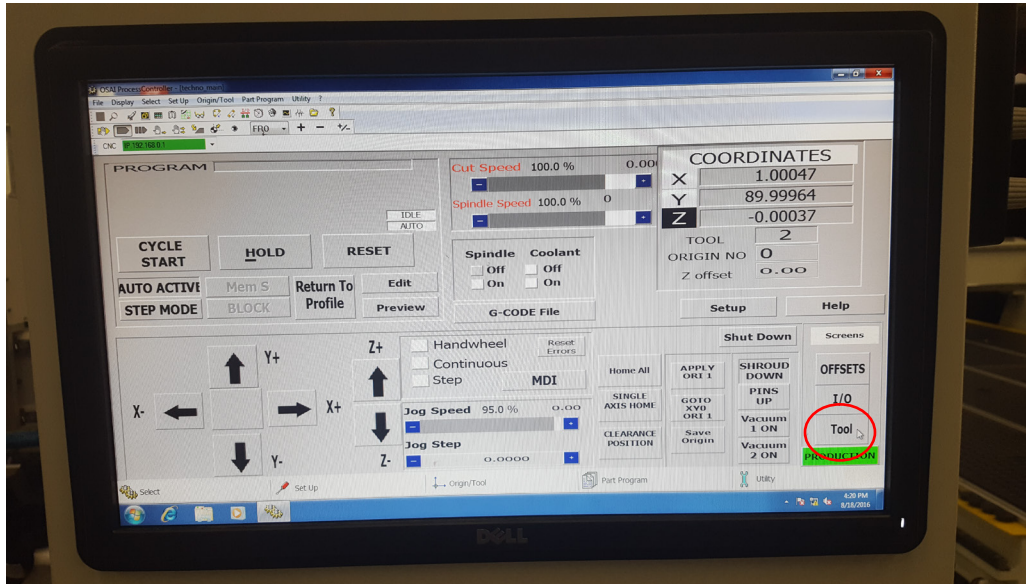
This Step by Step Guide Includes:

- How to load a program and run it
- How to perform tool change and tool touch offs

Step 1 – From the main screen on the interface, click on the "Tool" button located at the bottom right hand side of the screen.


Step 2 – Once you are on your tool menu page, you want to IDENTIFY the tool in the chuck.
This is the tool that is currently in the spindle. This is also the tool that the machine will put away before picking up a new tool during the tool change process.
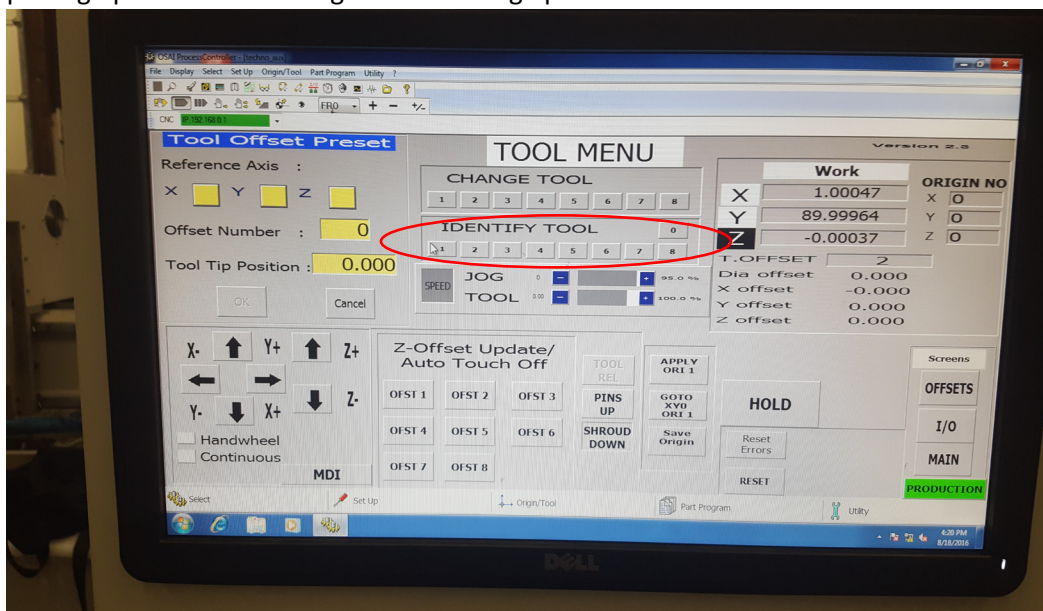
Step 3 – After identifying the tool in the chuck, you will then change to the tool number that you want in the spindle. Once you choose a number to change to, the machine will put the tool you identified away, and pick up the tool number it was told to change to.
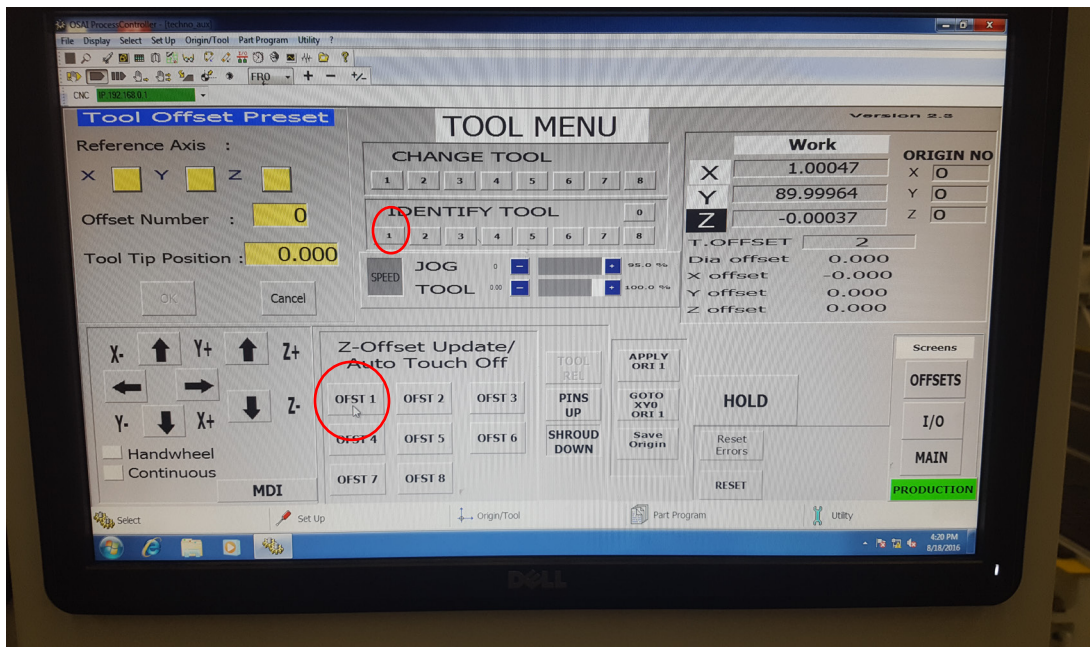
Example: If I have tool 1 in the spindle and I want to have tool 2 in the spindle,
I will identify tool 1 and change to tool 2.
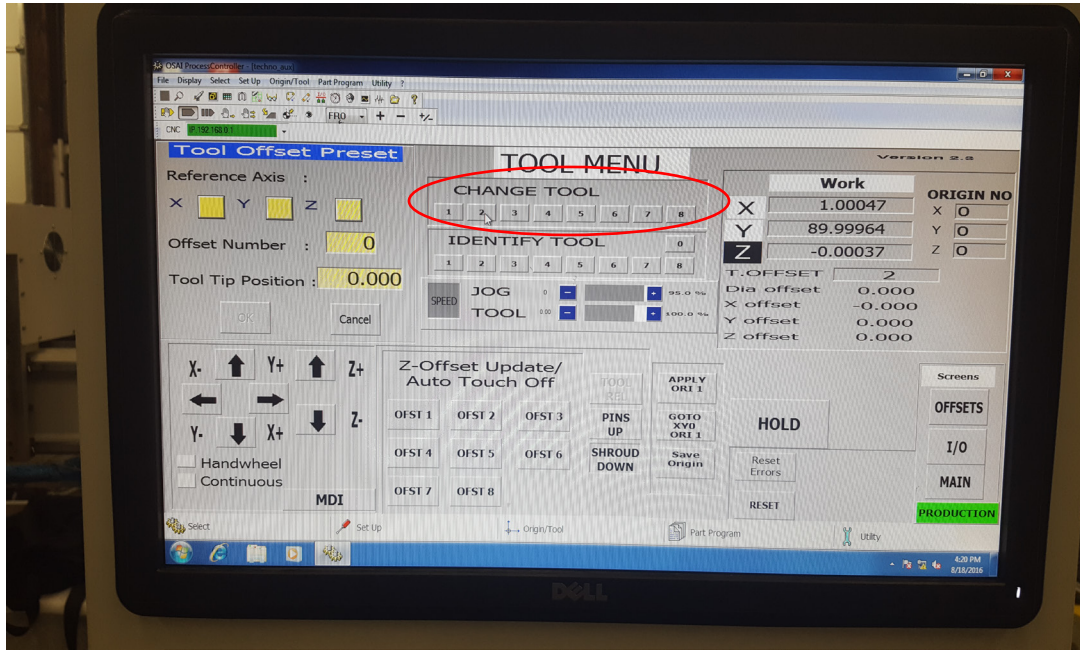
Step 4 – Identify the current tool in the spindle.

Step 5 – click OFST # (The # depends on what tool you identified).
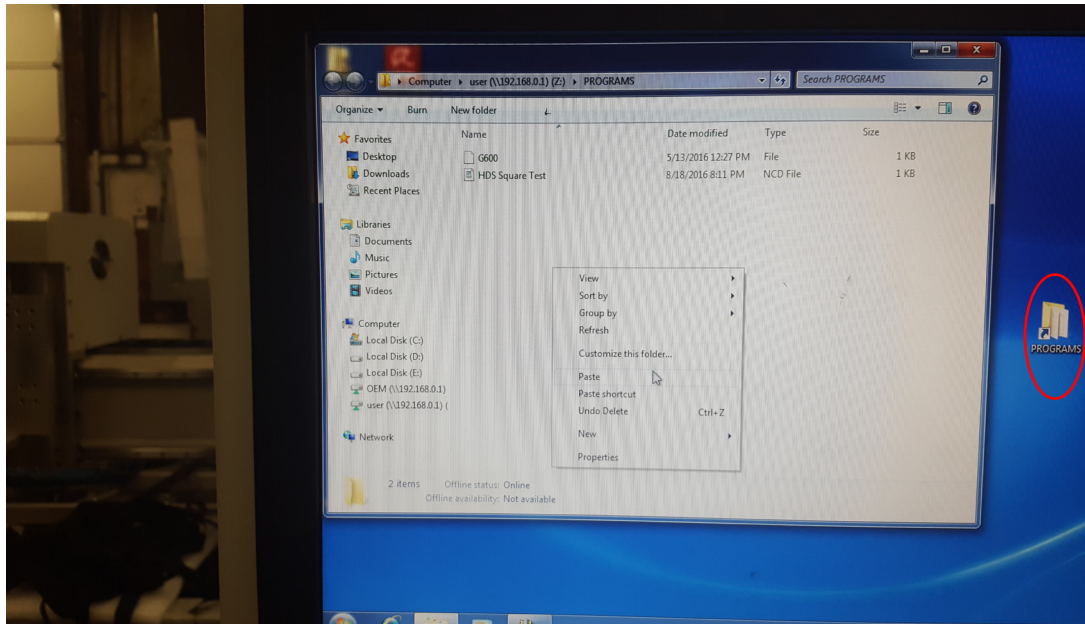Example: If you identify tool 1, you want to OFST 1.

HOW TO LOAD A PROGRAM, SET AN ORIGIN, AND RUN SAID PROGRAM:

Step 1 – Insert your thumb drive into the front of the CNC controller directly below the computer monitor.

Step 2 – Copy the desired file from the memory stick.

Step 3 – Paste the file into the "PROGRAMS" folder located on the desktop of the computer.
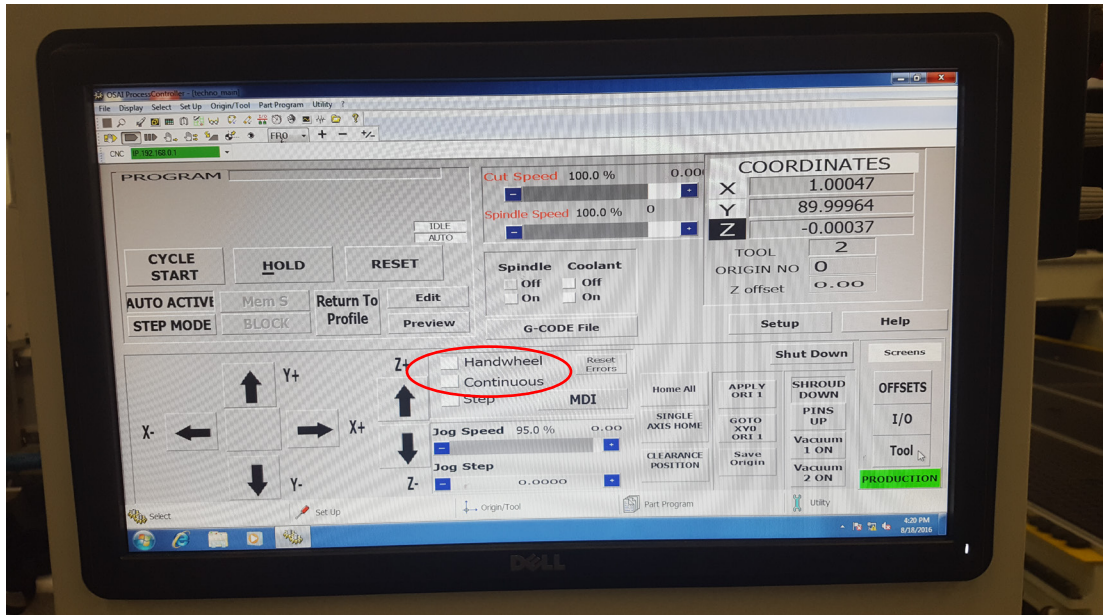

Step 4 – On your Techno Interface, jog the machine to the desired location on the table.
"Continuous" mode is the correct mode to be in when manually jogging the machine.
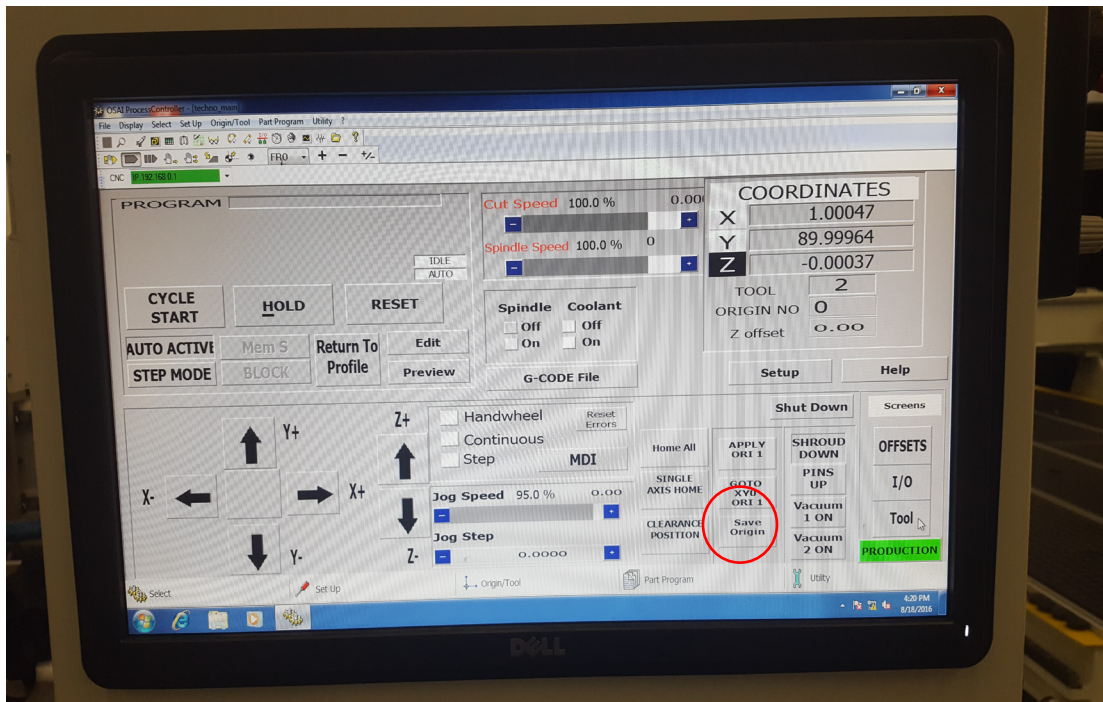To accurately position the machine, you can switch from continuous mode to handwheel mode in order to use the manual pulse generator.
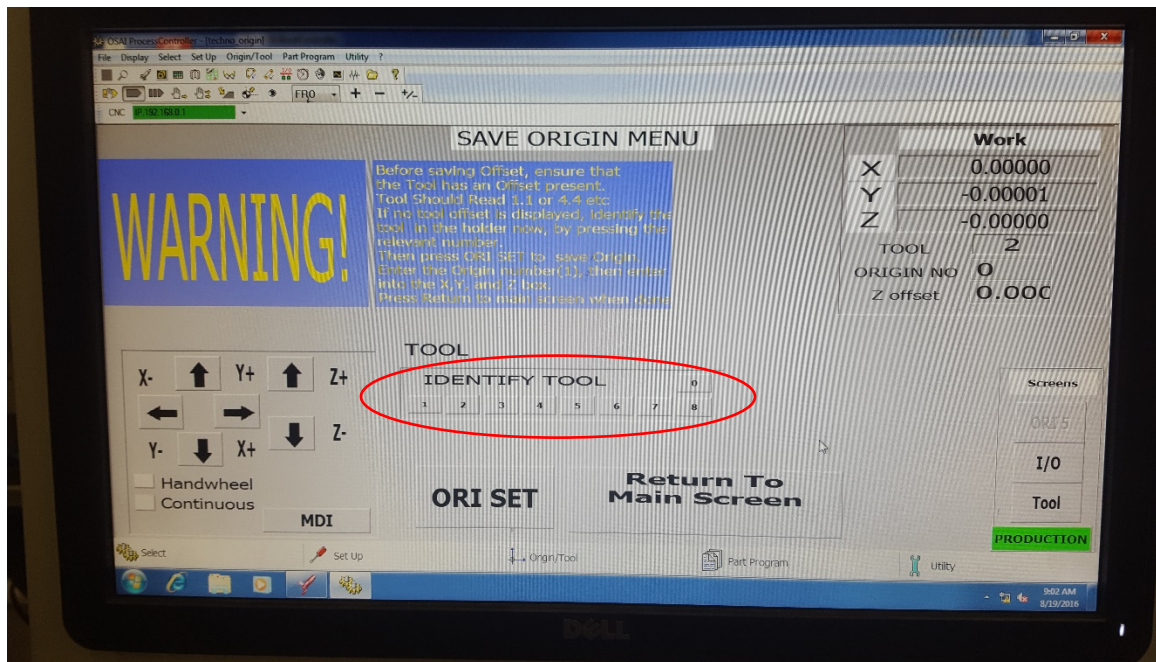
Step 5 – Once you have your X, Y, and Z axes positioned in the desired origin position,
you must take the machine out of handwheel mode, by switching to continuous mode or step mode.

Step 6 – Click on "Save Origin" once in continuous or step mode.

Step 7 – You will be brought to a new screen.
The first thing you need to do when on this page is IDENTIFY the tool in the spindle.

Step 8 – After identifying the tool in the spindle, click on the button labeled "ORI SET"

**Step 9 –** You will then be prompted to enter an origin number, and a 0 value for each axis.
NOTE: Your origin number must correspond to the origin number in your nc file.
You are able to save up to 8 origins on your HDS machine.
If you are using Vectric software, your origin number will always be number 1 and must be changed in the G-code in order to activate origin numbers 2-8.

Step 10 – After saving your origin, click on "Return To Main Screen".

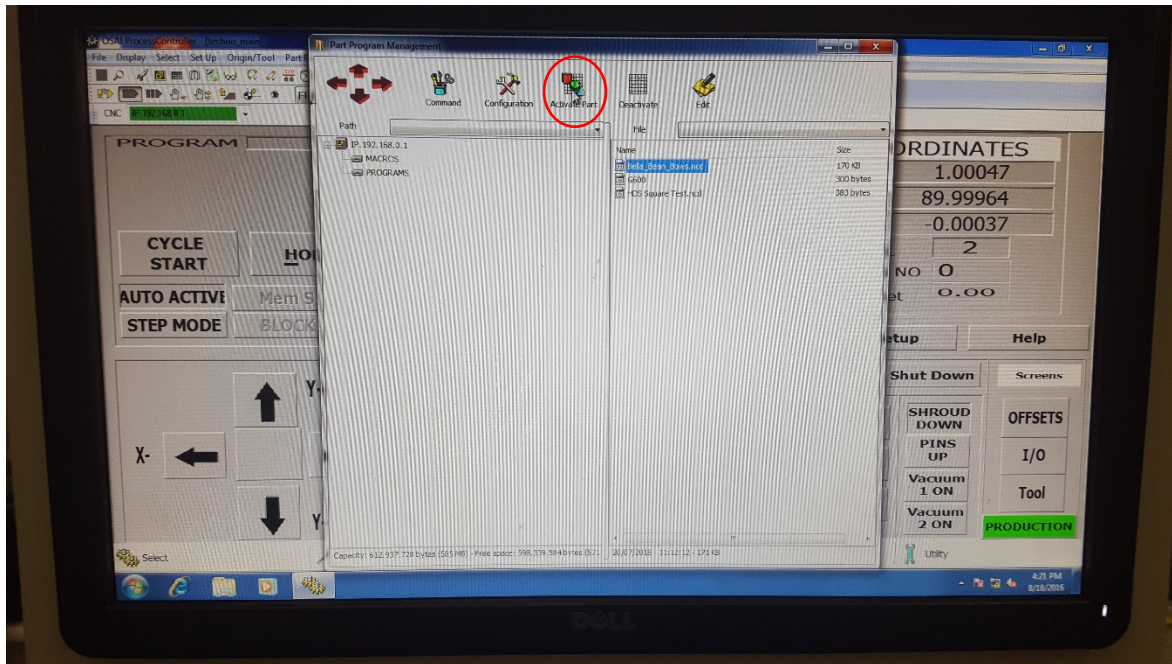**Step 11 –** Click on "G-Code File" on your main screen.
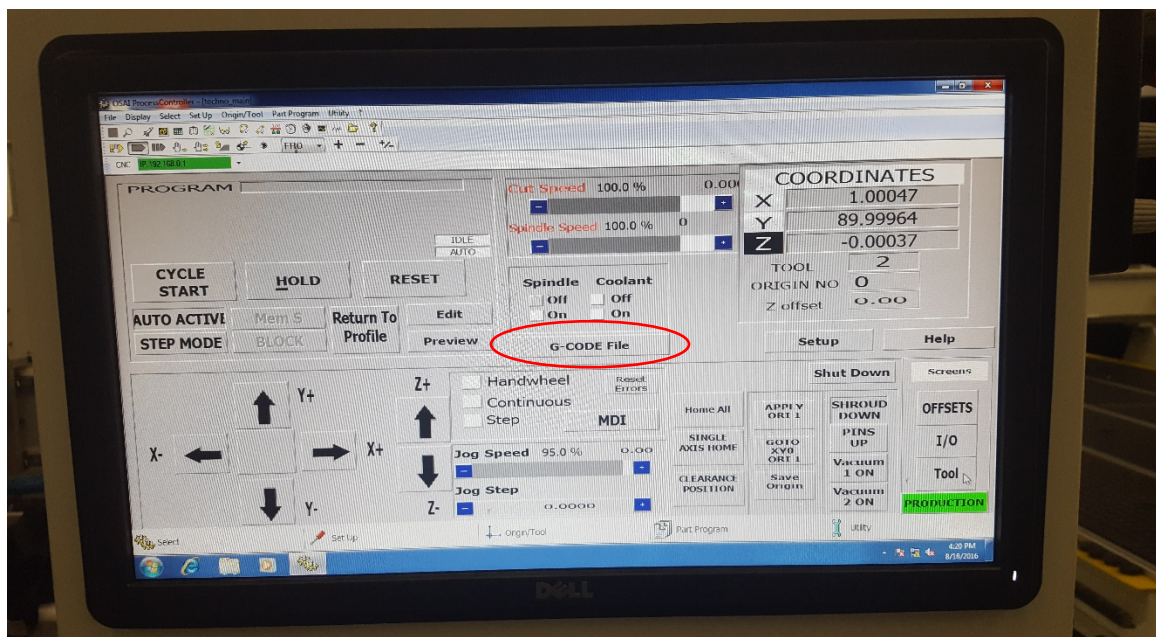
Step 12 – Your programs folder will then be opened on the main interface.

Choose the nc file you would like to run by clicking it once.

After highlighting the program, click "Activate Part" located at the top of the window.

Step 13 – After activating the part, you can exit the G-code file window. You may want to raise your Z-axis off the table before running the file.

Step 14 – Click "AUTO" then "CYCLE START" to initiate the program.