Scan scale test and adjust with editor instruction – 7/21/14

First: The menu needs to be adjusted so that the advance menu can be used. To do this:

1) Go to C:/HIMT/Exposure Wizard/System

2) Find the file: upg.cfg & wizard.cfg

3) Change the menu entry

MODE = 1 to MODE = 2

This will let you start up as advanced.

When the menu starts, the Password is dwl100

Quick note: Make sure that you ONLY change the Mode value. Changing anything else will cause major problems with the machine.

Ok, Next:

You can see that the machine starts up as normal in the advance mode. Once hardware is initialized, we'll start the APP, which is located on the upper menu bar.

Now, we have the script editor available, and the actual system control panel. First, we start by loading a wafer.

Note the "To Unload" button. The stage should be out in the unload position now. can you see that? yes, loading

Once you've loaded and closed the door, feel free to press the "To Center" button.

Next, we focus the writehead.

And the plate is physically set for writing.

Now, we need to set up the actual job, and that's where the script editor comes into play:

This part can be a little complex, especially over text, so if I go to fast, feel free to interrupt me.

First, we select the design, using the design window.

What's the design called? It might not be on the current list, since these are designs are generally installed by connecting directly to the linux box inside the upg.

Ok, just checked: adding your design to the app list is a long and difficult process that'll require us to connect direcly to the linux computer in the UPG.

Instead, we can test with the design "Square2500" or “GSCALDESIGN” and just write with an energy that takes us halfway through the resist. (Do you know what that energy value might be?) 4 mW 80% (adjust for resist)

Next part of setup: Do you know how far along the scanscale you'd like to test, and at what stepsize?

2.075MHz, to 2.125MHz, with step of 0.001MHz (adjust for resist)

Ok, first, let's set the energy:

Next, the scan scale itself: note that the white boxes are starting values, while the blue ones are step sizes.

And only 1 blue box can be active at a time.

Then, we need to figure out how many fields it will take to get to 2.125, and write that many fields. 50

Ok, we'll do a 10x5 matrix, then.

For GS set bidirectional to NO

Next, it's best to set a starting position, How big is the sample?3" x3"

Ok, i"m going to load the design real quick, and check the size.

See how the x/y increments changed?

Ok, so we can go ahead and move the design a little to the left, since we can see how it'll fit. 10 designs should be something like a centimeter or so.oK?

Now that our parameters are set up, we can create the actual script with the create button. Once we're happy with the script, we can use: Add to Script

Note how it's all there.

Finally, we can hit "Run Batch" to execute it.

Once you have the scan scale value, you need to connect to the Linux computer in the UPG to change it.

Open teraterm, and connect. The username and password are both upg101

Then, you emacs the file dwl.cfg located in the sys director: There, you can see the scanscale value.

Alt. use nano, it has command notes

From home directory type cd sys

Type nano dwl.cfg

Arrow down to scanscale and arrow over to value then backspace to delete and ype in correct value.

^ = control key

^O = save change

^X = exit

Then under file tab select disconnect

Make sure you ONLY change that.

Just remember to go to the upg.cfg file & wizard.cfg and change mode =1.

Note: scanscale for binary exposure is 2.112 (7-21-14), may need minor change to correct 8 um spaced bumps.