Copy .scr (scripts) and .dru (design rules) files from Andrew's andrewgreenberg github account to Documents > EAGLE > Scripts or Design Rules folder. Paste just the files, don’t paste folders in the Scripts or Design Rules folder.

We’ll be learning command line interface and hot keys rather than using the GUI buttons.

Ctrl + right clicking

SCHEMATIC NAME MUST BE IDENTICAL TO THE BOARD FILE NAME, AND THEY MUST BE IN THE SAME DIRECTORY. If board background annotation is “killed”, stop what you’re doing and find out what happened.

SHEETS – right click on “ribbon” to check sheets.

Use black background for Layout (PCB board) so layers are properly “alpha blended” and appear correctly.

Commands (cmds)

**DO this first:** Grid (cmd) (set to 0.05in, multiple 2, so you can set in between lines. Don’t ever change this again.)

Add (cmd)

R-US\_R0805

C-EUC0805

Type script (cmd), select assign-hotkeys-for-schematics (andrew’s script)

Ctrl+m (move)

Ctrl+g (group)

After: Ctrl+right click to move group

Double click to start drawing lines – make an enclosed polygon, then right-click to close polygon and select everything inside. Then grab one of the components by the plus sign, and it’ll move the whole selected group.

Center mouse button mirrors the selected component(s)

Right click rotates

DELETE GROUP:

Select group

Ctrl+d

Ctrl+right click

Text (cmd) – use to write on the schematic

Type text, then, BEFORE PLACING TEXT, select layer and change to INFO Layer. You don’t want text on the green net layer!

Change (cmd) then select layer, then select INFO so that when you select text that’s already on the net layer, it’ll change to the INFO layer.

Junction command allows you to place junctions

Name command – change name of component you select

Value command – change value (ohms, henries, farads) of component you select

**Name all your nets! Use the name command and select the nets**

You can label (using the label command) your nets!

**Plus sign = THE ORIGIN**

Erc (cmd) ELECTRICAL RULES CHECK!

Check centered, then select each error to view where it occurs on the schematic!

DO NOT HAVE ANY ERRORS IN YOUR SCHEMATIC BEFORE YOU MAKE A PCB!

Click on the sch/brd button to generate a board from the schematic!

USE 45° angles where you can in PCB. The chemical etch can cut through a 90° PCB.

Drc (cmd) set the limitations, such as trace width minimum.

File > load > select OSH park design rules file

Can right-click > properties of component or trace, then can use the coordinates (to/from x/y coordinates) for component alignment.

Ctrl+alt+1 to view all layers? Notice the stop mask over the pads – it’s the white cross-hatched areas where copper will be exposed and not covered with the green or purple solder mask.

Ctrl+1, ctrl+2, and ctrl+3 change layers for moving components around!

Display (cmd) – Visible Layers menu

Layers:

torigins

borigins (“plus signs” on components for moving around)

IF THESE LAYERS ARE NOT DISPLAYED, YOU WON’T BE ABLE TO MOVE YOUR COMPONENTS!

trestrict alerts in DRC if components or traces (anything!) is in a restricted area of the board. Useful if some physical component of your system touches the PCB, in which case you don’t want components in the way.

tkeepout – similar to trestrict, but doesn’t restrict traces. Only restricts components.

Dim (cmd) – command for assigning dimensions. Normally goes in the dimensions layer, so it won’t be part of the physical board.

Display -bname (or any layer name) will subtract the layer bname from the view.