

## Homework 1.2: Layout PCB Design

Homework is due Feb. 11<sup>th</sup> 11:59pm on Canvas. Again don't put this off to the last minute! You'll receive feedback on your design if submitted before Feb. 10<sup>th</sup>.

### 1. Introduction

Congratulations! You successfully created your first schematic using EAGLE and you are now ready to move to the next step – the layout of the board! For this purpose, we'll mainly use the PCB design tool included in EAGLE and called: *Generate/Switch to board*. This is usually done in 3 steps: (1) Open the schematic and switch from the schematic editor to the related board; (2) the placement of the components – where the different devices are physically organized on the PCB surface to minimize the wire routing complexity (you'll need to use your brain); and (3) the wire routing itself.

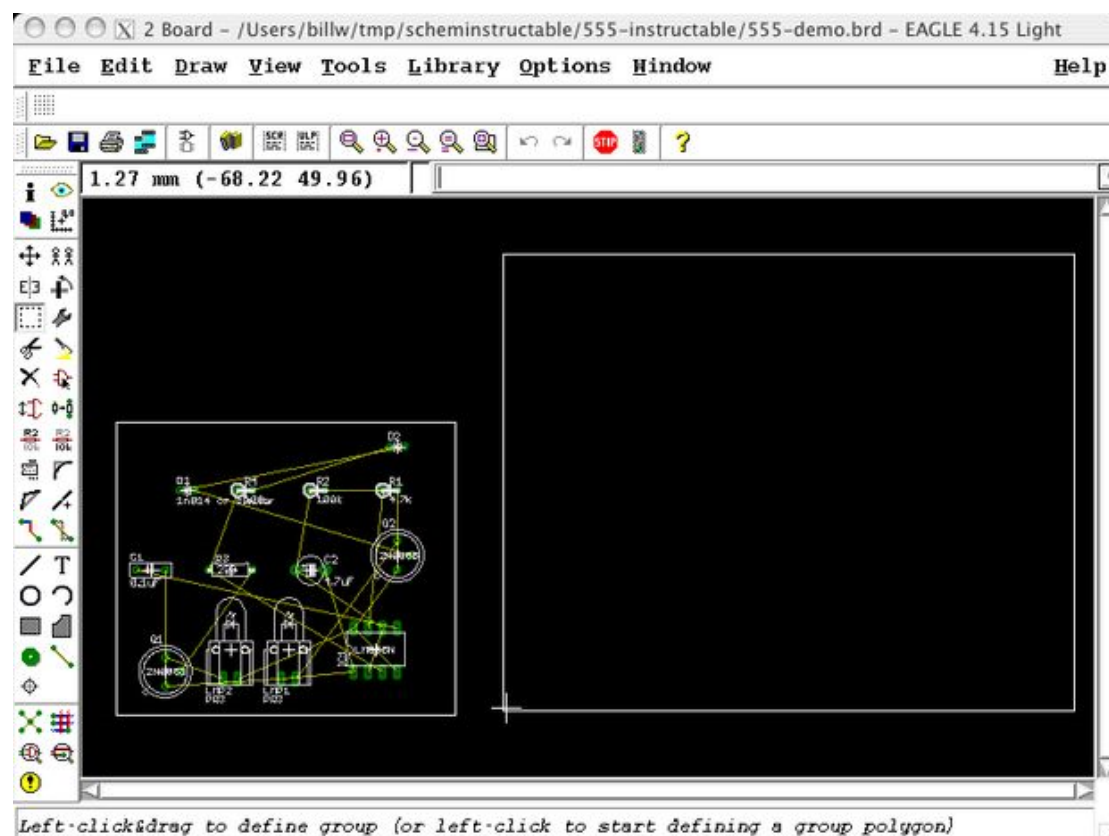
**Try to make the smallest possible PCB – as the smallest the cheapest ;-)** – As mentioned in HW1.1, it would be nice to have a board of 1"×2".

### 2. Generate The Board

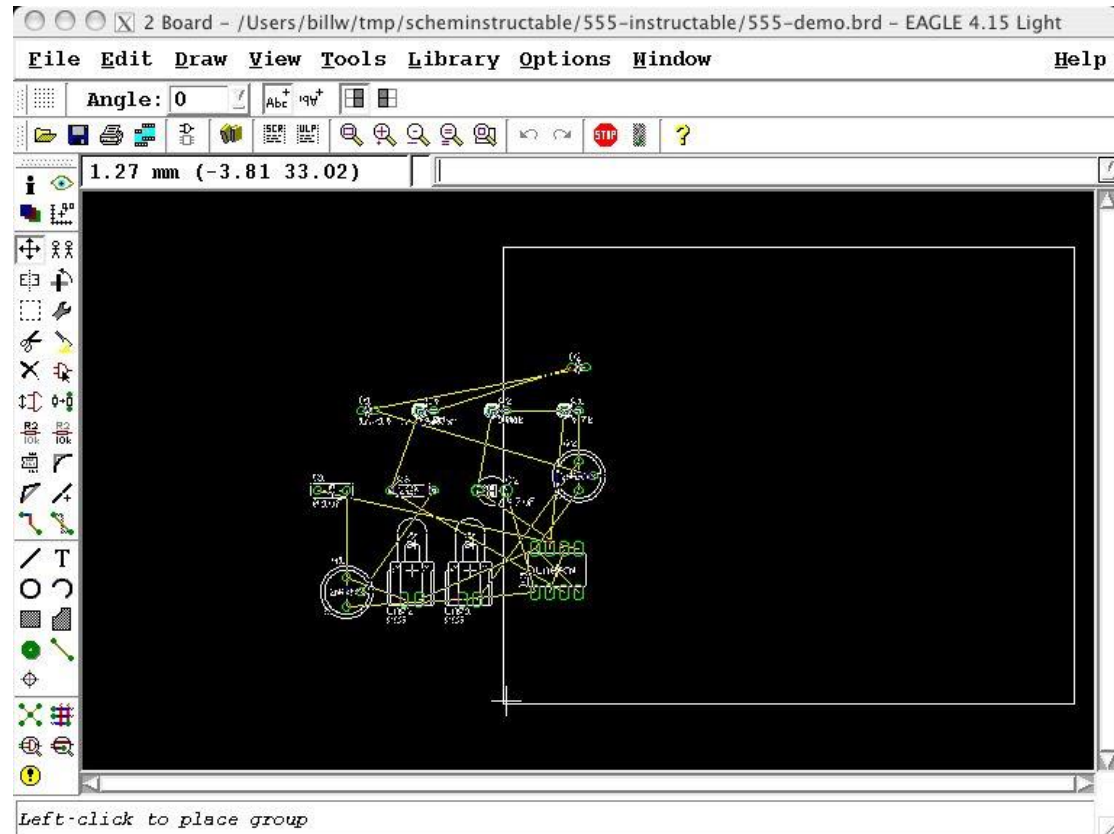
The schematic is now finished. We can now create PCB (printed circuit board) layout. For that, simply click the *Generate/Switch to board* command (on the top toolbar, or under the File menu) – which should prompt a new, **board** editor window to open.

### 3. Placing Components

After the board editor window is opened, it will look like the following. All your components will be in a clump over to the left of the origin.



The first thing you want to do is move at least some components into the legal board area where you can work with them. If you have a particularly large board with many components, you might want to do this a section at a time. You can move them all at once, using the group-move feature.



Select the GROUP icon, then click and drag to make a rectangle that goes all the way around the components. Then select the MOVE icon and RIGHT click (right clicking selects the group instead of a single component) and drag the set into the board outline. Use the ZOOM button to tighten the view.

The full legal side of the board is bigger than you need. Shrink the outline by using the MOVE tool. Click on the center of the top horizontal line (which selects the whole line instead of an endpoint) and move it down, then click on the center of the rightmost vertical line and move it leftward.

Now you need to move the components to (near) where you want them on the final board. OR you want to move them to sensible places that will make the placement of traces easier. A lot of the "ART" of making PCBs (and especially Single Sided Boards) lies in finding "good" places for the components. In general, you can start by placing the components similar to how they appear on the schematic. Another common practice is to put the pin headers on the edge of the board.

After placing all components in the board, click the Design Rule Check icon for checking DRC errors.

## 4. Routing

Routing is probably the hardest part of designing a PCB. Usually, one routes different nets according to their importance like this:

1. Analog

2. High-speed digital and differential signals
3. Digital Signals
4. Power/Ground

Do not use the autorouter!!! The autorouter doesn't know the Importance of your signals by default. You should first route critical traces by hand and then set the autorouter to do the boring bits. Only if necessary, use the autorouter to do the digital signals. Though, this is highly discouraged, as it often times messes up your design!

PCB Manufacturers usually have specific limits in how small of a trace or drill hole you can make. Our manufacturer has a minimum trace width of 8 mil (8/1000") and has a minimum spacing requirement of 8 mil (i.e. 8 mil trace and space). Also try not to use a drill smaller than 15 mil with an annular ring of 30 mil diameter.

## **5. Artwork & Making Gerbers**

**Artwork:** Once you finished routing your board, it is time to add some artwork. Use the "Draw →Text" tool to add text onto the "Top Overlay" layer. The most common things one adds are: Name of the board; Version and date; Your name and maybe company logo. Try adding some artwork to your board. Signatures and logos can be imported from bit map pictures. This won't be covered here, but there are many tutorials online that show you how to do it.

**Board Outline:** Your board can have any outline. However, we will keep to a rectangular shape. If you don't provide a board outline to the manufacturer, then they will just assume you want a rectangular board around all your components. However, to make sure that they know what the boarder is, it is better to provide one. After you generate/switch the schematic to your PCB, you will see there is a default board outline there. You can directly modify the property of each line into the length you want. Or, delete the default board outline and draw a rectangle (using "Dimension" layer (layer 20)) by yourself.

## **6. Deliverables**

Create a PDF document as you did for homework 1.1 with your board – and send it on Canvas along with a zipped version of your project folder. Use ZIP! (and NOT RAR, TAR, ...). Submit the board PDF and the folder. Do not send a unique zip file containing the PDF and the files.