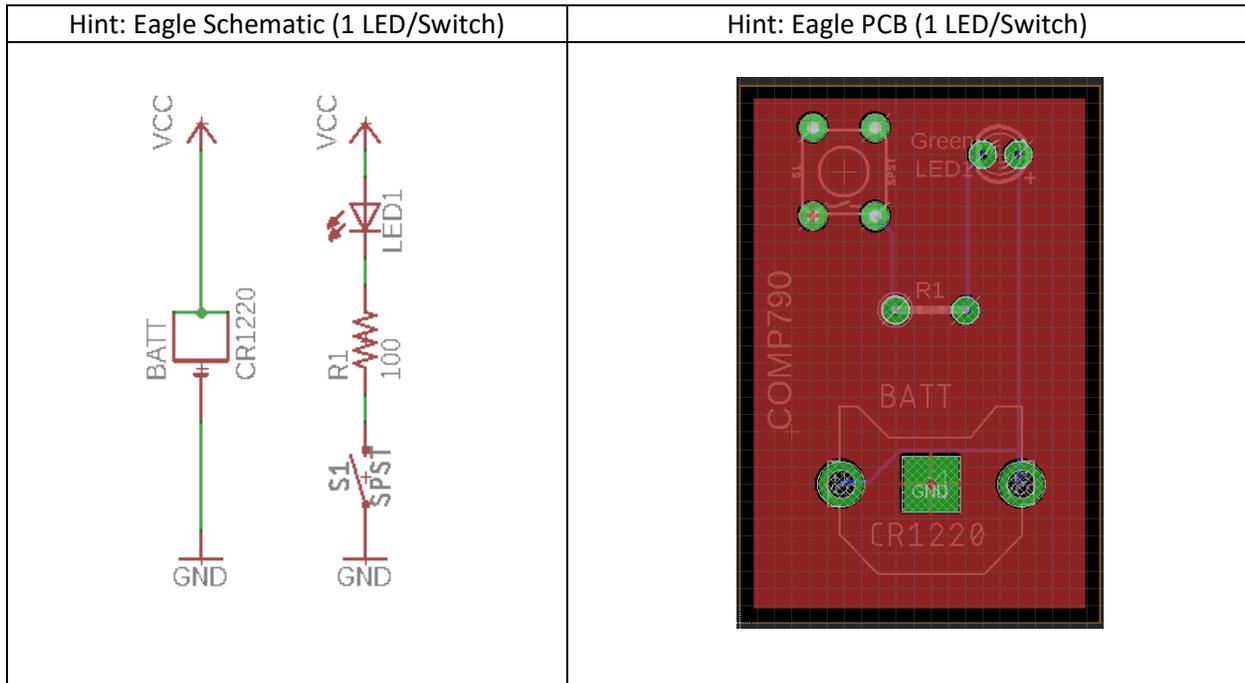




Lab 5: Three LEDs (Eagle PCB)

Duration: End of class.

1. We will design a PCB from the schematic of our last class in Eagle CAD software.



Placing Elements:

1. Move all elements inside the rectangular board, resize the rectangle, and arrange elements.
2. Be familiar with different layers, e.g., top, bottom, un-routed wires, name/value etc.
3. Always hit ratsnet after any changes.
4. Make a GND plane by drawing a polygon surrounding the board and renaming it to GND.
5. (Optional) Reposition Attribute and delete name/label that you don't like.
6. (Optional) Edit Info to place objects at fixed positions.
7. (Optional) Use tPlace layer to add your name and revision number to the board.

Automatic Routing:

1. Edit/Net Classes: edit the parameters of the default Net Class: W = 12, D = 20, C = 10.
2. Autorouter (grid size = 10)
3. Place "vias" as needed. Use D = 23.

(Often Required) Manual Routing:

1. Use Eye to check connections.
2. Use Ripup to remote connections
3. Manually route wires (W = 12, D = 30) and switch layers (top/bottom) when needed.

Design Rule Check:

1. Set min D = 20 (which is by default larger)
2. Apply DRC.
3. **Make sure there is no error.**

(Optional) User Language Programs:

1. Try a ULP script (e.g., auto numbering elements).

(Optional) Design Link to Generate Bill of Materials:

1. Find appropriate parts.
2. Create the bill of materials text file.
3. You can directly order these from Eagle. (Note: since you already have the parts from our last lab where you built the same circuit on a bread board, you don't have to order these for real).

Generate GERBER Files:

1. Run CAM processor (use 2 layer template of EAGLE, or download and use external ones).
2. Save the generated files in a folder and zip it.
3. We can send these gerber files to any PCB manufacturer.
4. (Optional) There are free online tools to visualize gerber files (e.g. Seed studio)

(Good Practice) Designed For Manufacturability (DFM) Check:

1. For free DFM, go to 4pcb.com and upload your gerber files.
2. Set 2 payers, X/Y dimensions (you get the dimensions from Eagle's PCB quote tool on the top).
3. Check your email for reports.

PCB Quote and Ordering:

1. Use Eagle's PCB quote tool (top middle) to order your PCB.
2. Alternatively, find some other company online.