1270 Class notes.

* **Although Eagle is loaded on all of the machines in 1223 and 1211 you should probably create an account on the Autodesk site(**[**https://www.autodesk.com/education/free-software/featured#free-software**](https://www.autodesk.com/education/free-software/featured%23free-software)**) with your Pitt email and download it to your individual laptops.**
* Printed Circuit Board (PCB) Design rules
* All circuit boards will be sized to 3000 mils x 3000 mils which is 3inches by 3inches square. This board is approximately the size of a post it note when made on the milling machine.
* Mils are the default unit in Eagle. 1 mil is 1/1000 of an inch.
* Do not place your design in one small area of the board. Spread the components out over the area of the board.
* Trace widths should be at least 25 mils width. This is also the same amount of spacing that should be between your traces to avoid some other problems.
* The PCB you will be designing is based on a 555 timer circuit from ECE/COE - 501 with a few modifications. You can find this circuit on Courseweb or at <https://sercpitt.weebly.com/> under beginner level.
* The 470 Ohm resistor, 1 uF cap., .01 uf cap. and LED are located in the component drawers in the 1223 labs. All other components you will pick up in 1214 BEH. NOTE: [**These components are in the 555 Timer using Eagle (1270 Class) on the SERC website**](https://sercpitt.weebly.com/). This you will need to properly

1270 Class notes.

design your board. All components are through hole packages except for the 1K Ohm resistor which is a surface mounted device.

* You will upload all designs for this project to Courseweb.
* You will place the fuse holder in line with the supply voltage (5 volts) coming from the center pin on the DC power jack, and the three pin header will be used as test points for +5V, ground, and the output of the 555 timer.
* If you have any questions or need assistance stop by 1214/1217 BEH. No appointments necessary.