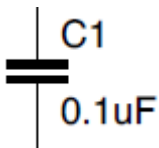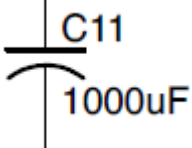# Final Project Checklist

The aim of this checklist is to assist the ENGR 301 students for their Final Project board design. All the labeling refers to the project schematic that can be downloaded from iLearn.

☐ Be familiar with EAGLE software
- o Watch YouTube tutorials (former instructor's tutorial link).

☐ Download the EAGLE resource files, the project manual, the project schematic, the how-to-import-library document, the LTspice simulations, and the sample component list from iLearn. Review them.

☐ Starting off with the Power Supply unit
- o In EAGLE, whenever you select a component, you may refer to the sample component list as a guide. Make sure that the component you use in EAGLE can be bought online, and it has the same dimensions.
- o Browse to the engr301 library to find the positive/negative regulators.
- o The fuse, switch, and the power jack can be replaced with two-pin terminal blocks (con-phoenix-254) as placeholder to connect external circuit to you board. One pin is connected to the diodes (D3 and D4), and the other pin to GND.
- o The schematic uses two capacitor (cap) symbols to represent electrolytic (polarized) capacitor and ceramic capacitor. Electrolytic cap tends to have more capacitance than ceramic cap. Make sure that the polarized cap is oriented correctly; (+) side points to higher voltage than the (-) side.

Ceramic capacitor	Electrolytic (polarized)
C1 0.1uF	C11 1000uF

☐ Next is the left channel.
- o You will need to use the same schematic sheet as the power supply unit; in other words, you will be connecting the channels to the PSU.
- o In EAGLE, browse to the engr301 library to find the OP275. This dual Operational Amplifier (OpAmp) is an IC chip that has two OpAmps inside. There will be three symbols: one for left, one for right, and one for the power pins. You have to use

these three symbols. The number on the symbol represents the pin number of the chip. See Figure 1.

- o You will also find the transistors inside the engr301 library.
- o For the input, the potentiometer and the audio jack can be replaced with three-pin terminal block: one connects to diodes D5 and D6, the other one to diodes D14 and D15 (right channel), and the other one to GND. See Figure 1.
- o For the output, the 56-ohm resistor can be soldered onto the board. You can replace the audio jack with three-pin terminal block: one for left, one for right, and one for GND. See Figure 1.

- ☐ Right channel
  - o Copy the left channel schematic over. Connect the opAmp's power and input/output pin accordingly. Connect the input and output terminal blocks accordingly. See Figure 1.

- ☐ Board Layout
  - o Load the Design Rule (.dru) in EAGLE files downloaded from iLearn. The design rule is the design constraints that the manufacturer imposed, and we have to comply with it so that our boards get printed properly.
    - ▪ Go to Tools > DRC > Load > Open the engr301_drc.dru file > Apply. You should see under the "Clearance" tab that the minimum distance from wire-to-wire is now 8 mil, and pad-to-wire is 10 mil.
  - o Make sure you have a Ground Plane on the board, typically drawn on the bottom layer. Polygon > draw around the board > rename the polygon to GND.
  - o Make sure you don't have any airwires (unconnected wires). Use "Ratsnest" tool. The status info located on the lower left corner should display "Nothing to do!". If you have airwires left, you must connect them.
  - o You should aim to manually route the power signals for practice. You may use auto-router tool to automatically route the remaining airwires.

- ☐ Gerber Files
  - o Gerber files are the files that are sent to the manufacturer.
  - o In the board layout window, go to File > CAM Processor > Load job file (top left corner) > Open CAM file > Open ENGR301-Gerber.cam > Process Job > select the folder you want to save your files in.
  - o You should have 9 files in total. Zip them.

- ☐ What to turn in on iLearn
  - o You need to turn in the .sch, .brd, and the Gerber files zipped.

Notes:

1.) When working in EAGLE, make sure you have the schematic and board layout window open. Every time you change a component in the schematic sheet, it must reflect on the board design. Otherwise, you might get F/B annotation warning.
2.) Work with another group. It's easier to collaborate and design with a larger group of people.
3.) You should aim to have relatively clean board. To minimize crammed board, you may move around component names/values, delete certain component names, rename components, decrease text fonts, etc. See Figure 2.
4.) You **must** have your name and the date on your board. Use the text tool for this, and change the layer to top silkscreen layer.
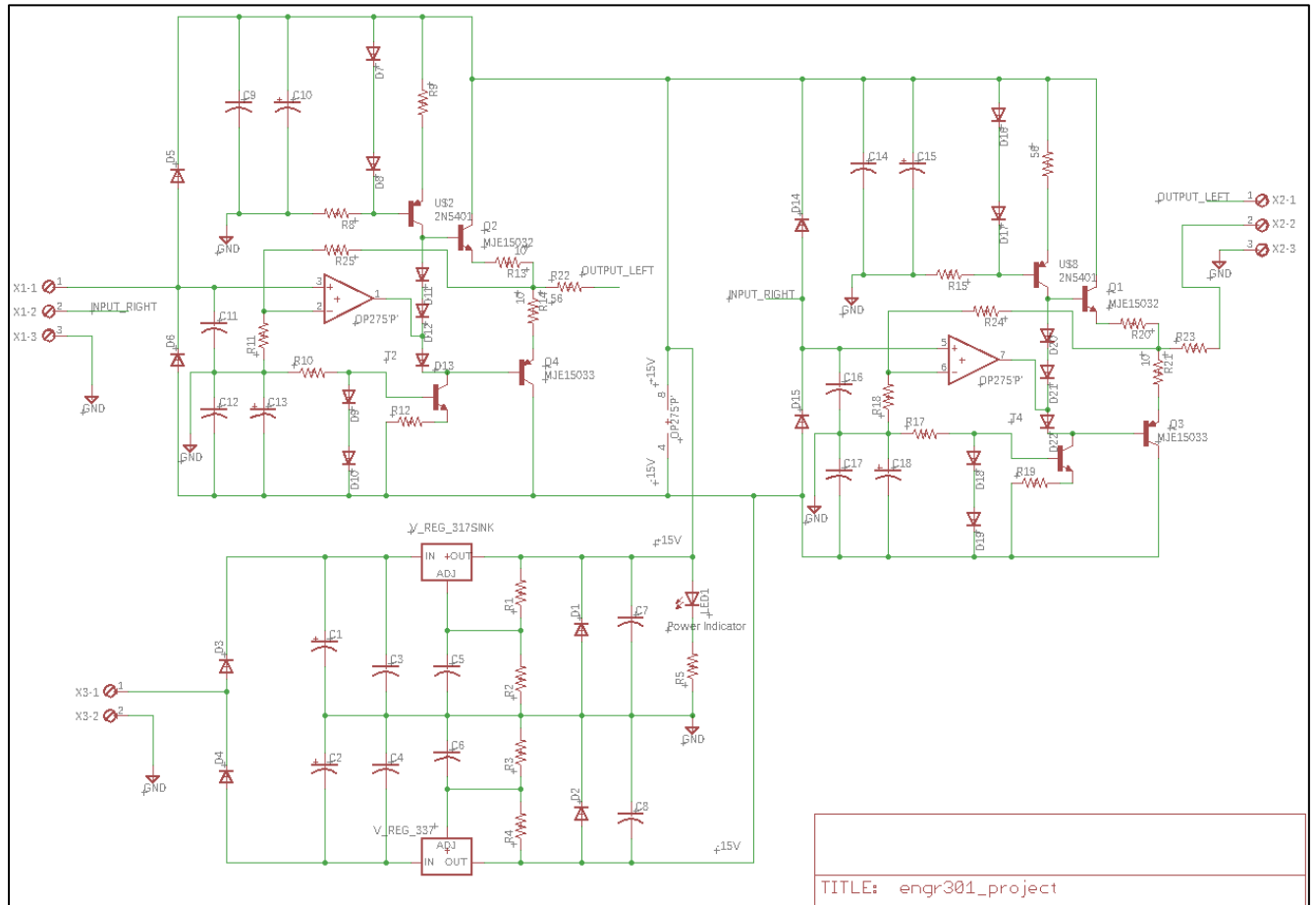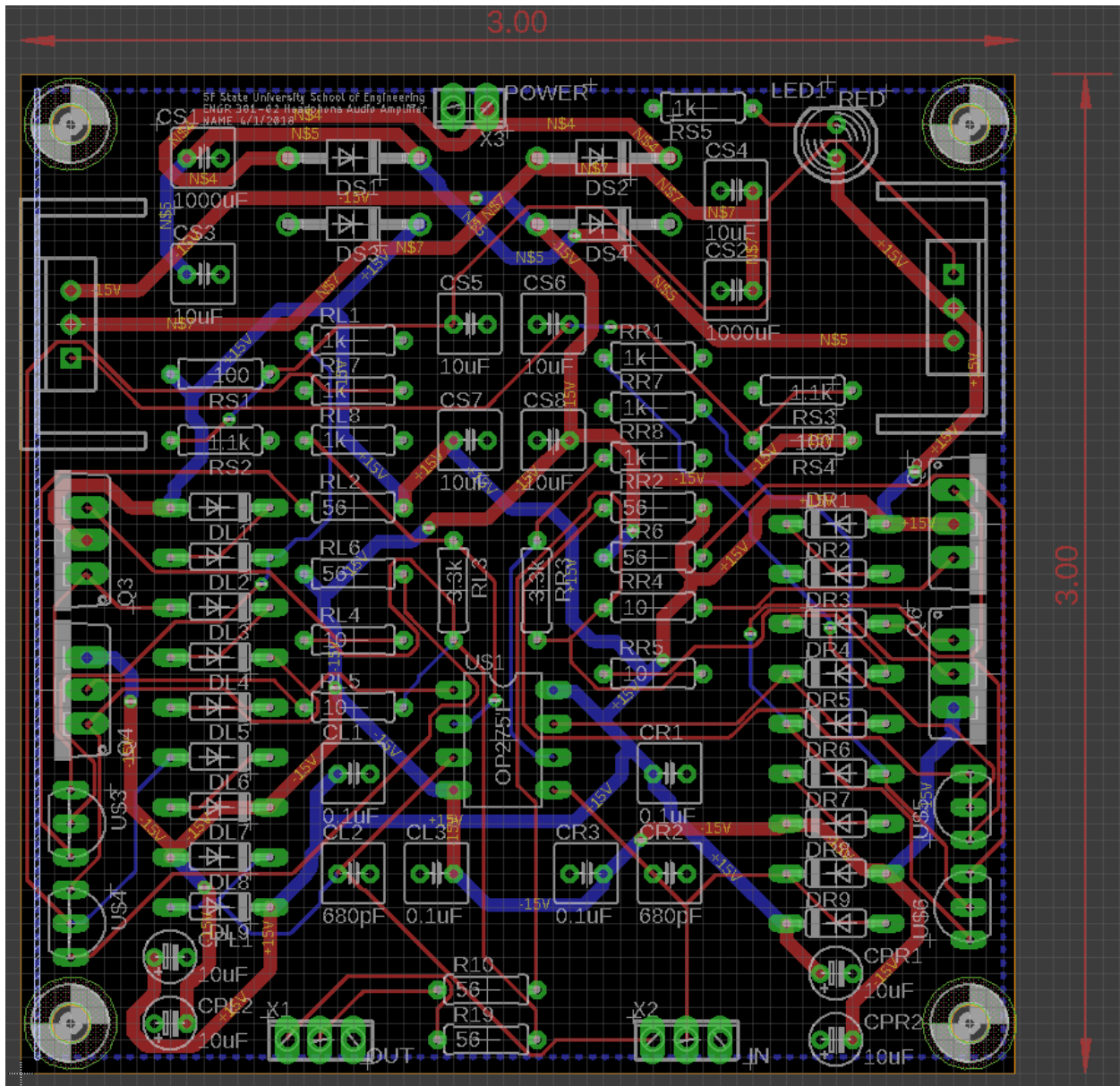

Figure 1: Final Project Schematic

Figure 2: Final Project Board Layout