EE419/519 Project Part 2: PCB Design

(Due November 20, 2024)

Using Diptrace, generate a nice-looking schematic of your converter. Refer to the Diptrace Schematic Tutorial for details. Upload the schematic PDF to Moodle.

Design the PCB with dimensions of 2100 × 2100 mils. Refer to the Diptrace PCB Tutorial for details. The PCB should be one-sided. Only the bottom layer should contain the copper. All components must fit on the same side of the PCB with only SMD components on the back side. The components should not overlap with each other or extend beyond the board dimensions. Check the Gerber files using a Gerber viewer before you upload it. If I find errors, you need to redesign it. For each error, you will lose points.

1. Upload Diptrace schematic file: (LastnameSch.dch)
2. Upload Diptrace PCB file: (LastnamePCB.dip)
3. Upload a PDF file (LastnameSch.pdf) containing the schematic.
4. Upload a PDF file (LastnameBOM.pdf) containing the bill of materials.
5. Upload the zip file (Lastname.zip) containing the Gerber and drill files.