EAGLECAD ASSIGNMENT

ECE445
University of Illinois at Urbana Champaign
Department of Electrical and Computer Engineering

Assignment

Your task is to create Eagle schematic and PCB (printed circuit board) files for the LED driver circuit shown in Fig. 1 below. The completed assignment should be sent to your TA in the form of **.sch** and **.brd** files. If you've never used Eagle before, you will find the following series of videos helpful: Tutorial

Your design must meet the following criteria:
1. Use any off-board connector for the 5V input (imagine the power supply is on a different PCB and you simply need a way to connect the signals (5V_in, GND) to your LED circuit.
2. Use any through-hole resistor and LED.
3. TPS799 is a linear regulator, included in the circuit to have you practice designing custom parts. Design a part footprint and schematic symbol for the TPS79930 (use the DDC package, 5-pin SOT, fixed voltage at 3V). The datasheet from the manufacturer can be found at http://www.ti.com/lit/ds/symlink/tps799.pdf.

Note the box "TPS799" below is not a single component, but a small circuit itself. Refer to the datasheet's **Typical Applications diagram and the **Pin Functions** table.
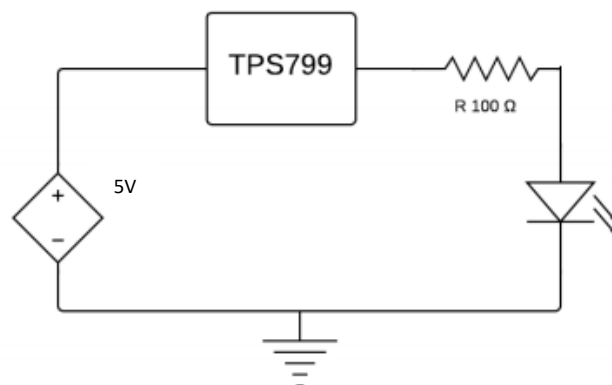

Figure 1. LED driver circuit

ECE Shop Requirements

The shop has a number of special requirements for board files in order to ensure that the board gets manufactured properly. In order to familiarize yourself with the requirements imposed by the ECE shop, read through their website: eshop.ece.illinois.edu/pcbdesign/pcbdesign.php

The following are some suggestions and requirements from the shop, listed in no particular order:

1. Always keep both the schematic and board open when working
2. Vias will not be plated through.
3. Make gaps between parts/traces as wide as possible, absolute min. spacing = **10 mils**.
4. Make traces as wide as possible, absolute min. trace width = **10 mils**.
5. Traces have current limits. The thicker the trace, the more current it can handle.
6. No vias under SMT parts .
7. Do not use autorouter unless you absolutely have to or have experience using the autorouter. If you are going to use the autorouter and plan to have the board made by the Electronics Shop, use the parameters provided on their website.
8. If you have a GND plane (using "polygon") make the "isolate" parameter **50 mils**.

Rubric

Grading for Schematic:
1. Schematic contains correct components (1 point)
2. Schematic wiring matches provided circuit (1 point)

Grading for Custom Part:
1. Symbol has properly labeled pins (1 point)
2. Package contains correct pad dimensions + spacing (1 point)
3. Device has correct symbol <-> package connections (1 point)

Grading for PCB:
1. Board has two layers with traces on both sides (1 point)
2. Board contains at least one via in layout (1 point)
3. All traces greater than 10 mils (1 point)
4. At least 10mils of spacing between all adjacent part, traces, etc (1 point)
5. Board has at least one ground plane/fill with an isolation of >=50 mils (1 point)