PCB layout practice

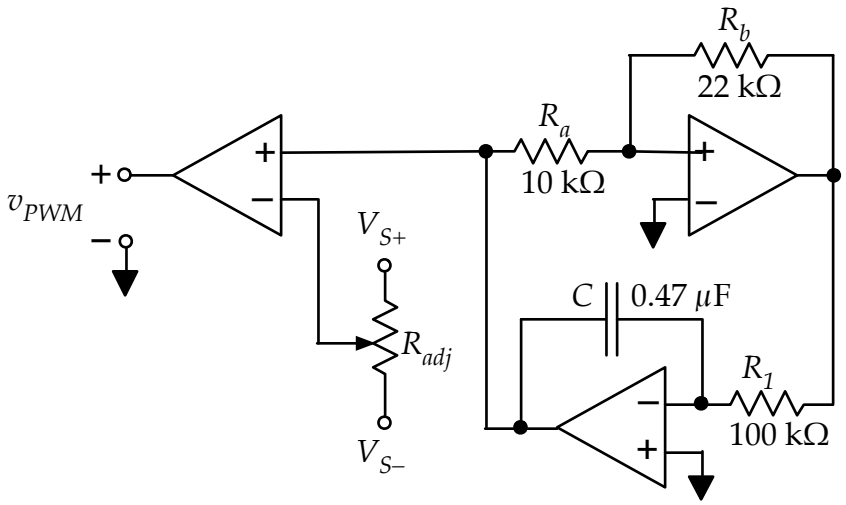
This is a required lab that should probably be done everyone. (If you think you should be excused, you can try to convince GT to allow it.) It consists of three parts:

1. Through-hole PCB tutorial

Read (and work) your way through the KiCad tutorial. We have gone through an example in class, but you should spend some hands-on time becoming familiar with the details of doing a layout with KiCad. This can be done before or during your lab time. There is nothing to submit — use the tutorials to get yourselves up to speed before starting the two exercises below.

2. PCB layout of the PWM circuit

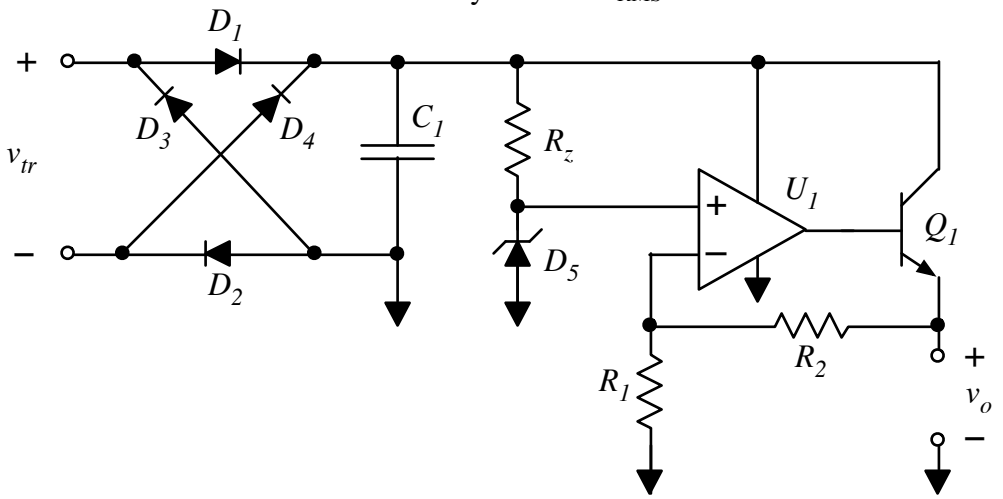
Design the layout for a through-hole PCB for the pulse-width modulation circuit that was used in the soldering lab. Use the parts listed in the bill of materials given below. Recall that the variable resistor consists of a 10-kΩ potentiometer in series with two 6.8-kΩ fixed resistors.



schematic label	description	part	DigiKey part number
U1	op amp	3/4 of LMC660*	LMC660CN/NOPB-ND
Ra	resistor	10 kΩ, 1/4 W, 1%	RNF14FTD10K0CT-ND
Rb	resistor	22 kΩ, 1/4 W, 1%	RNF14FTD22K0-ND
R1	resistor	100 kΩ, 1/4 W, 1%	RNF14FTD100KCT-ND
C	capacitor	0.47 μF, cer., 25 V	399-14039-1-ND
Radj	potentiometer	10 kΩ, single turn	3362P-103LF-ND
Radj	2 fixed resistors	6.8 kΩ, 1/4 W, 1%	RNF14FTD6K80-ND
power connector	three wire	0.1-in pin spacing	A98334-ND
output connector	two wire	0.1-in pin spacing	A98333-ND

3. 10-V power supply with a "built-from-scratch" regulator

Design the layout for a full-wave rectifier with a "built from scratch" voltage regulator. The input would be a sine wave from the secondary of a 12-V_{RMS} transformer.



schematic label	description	part	DigiKey part number
D1 – D4	rectifying diodes	1N4006	1N4006RLGOSCT-ND
D5	5-V Zener	1N4733	1N4733AFS-ND
C1	capacitor	1000 µF	493-1085-ND
R1, R2	feedback resistors	1/4 watt. 10 kΩ	RNF14FTD10K0CT-ND
RZ	Zener limiting resistor	1/4 watt. 1 kΩ	RNF14FTD1K00CT-ND
U1	op amp	1/2 of TL082	296-1780-5-ND
Q1	BJT	MJE180	MJE180STU-ND
input jack	2.1 mm barrel plug		CP-102A-ND
output connector	2-wire "greenie" from kit.		A98333-ND

Note: Other than the connectors, all components should be in EE201 or EE230 kits. If you want to substitute anything, check with the lab instructor or GT first.

Comments (Apply to both circuit layouts):

- These should be two-layer boards.
- Use a ground plane on at least one side of the PCB.
- Reduce the size of the board — big enough to fit the components comfortably, but no bigger.
- If KiCad does not have the specific part listed, choose one from the library that has a similar footprint. In particular for the first design, KiCad does not include the LMC660 in its standard library of parts. Instead, you can use an LM324, which has the same footprint as the LMC660. Note that "Unit E" for the LM324 has the positive and negative power supply connections.
- The connectors are standard — a set of two or three through holes spaced 0.1 inches apart suitable for male header pins or the "green" connectors included in your lab kits. These are similar to the connectors described in the tutorial.
- When the layout is finished, trying uploading the .kicad_pcb file to Oshpark to see if it passes muster and to learn how much the boards will cost.

Reporting

When finished, show your lab instructors the final designs. Be prepared to answer any questions they may have about your work. Then write a short lab report describing the two layouts and submit that to your instructors.