PCB Project Description

Karl Ventayen

June 23, 2022

Contents

1 Introduction

In the development of hardware today most if not all hardware requires the use of Printed Circuit Boards or PCBs. Compared to other methodologies for assembling circuits such as breadboards and freeform circuit assemblies the usage of PCBs provides solutions that have a better formfactor, mechanical properties, and asthetic.

However since the process for producing PCBs is a very involed process this project has been designed to step through how to produce a PCB from Schematic to Assembly to Production.

1.1 The Analog Multiplier

This project revolves around an analog amplifier design, which is attached to this sheet. The objective of an analog multiplier is to take two voltages and produce an output voltage, which is the product of the two voltages.

Applications for this device include radar, communications, and industrial controls. This is due to the nature of the scenarios needing a real time response.

For additional information see the links below:

- https://www.analog.com/en/product-category/analog-multipliers-dividers.html

- https://www.analog.com/media/en/training-seminars/tutorials/MT-079.pdf

- https://en.wikibooks.org/wiki/Electronics/Analog_multipliers

2 Project Schematic Diagram

On the following page the Schematic Diagam for the PCB Project is provided.
Note: This design originates from an assignment written by Dr. Meghdad (Amin) Hajimorad. The assignment name is MiniProject_Description_EECE211Fa2022_UPDATED.pdf
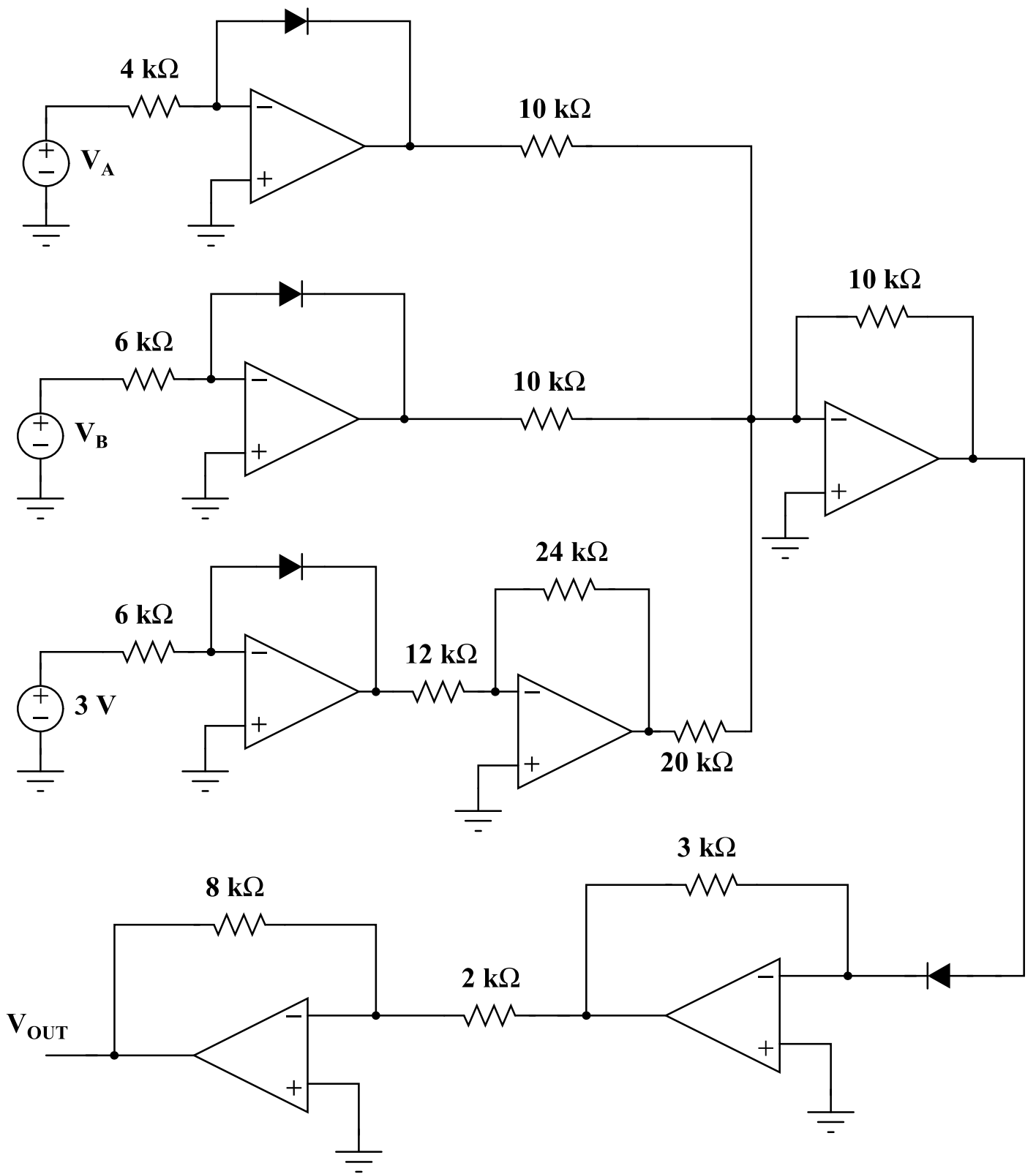
3 Bill of Materials

All of these materials are distributed by the CSU Chico IEEE, therefore component prices provided on the table below are according to them.

- Resistor: $0.25

- Diode: $0.50

- Pin Headers: $1.00

- Operational Amplifier (IC): $1.00

- 14 Pin Socket: $1.00

4 Scematic Layout

For the schematic layout ensure that you have the following parameters included into your diagrams and netlists.

1. All of these components are Through-Hole Components.

2. Footprint IDs refer to names that are provided in the KiCad 7 Library

Component	Identification	Footprint
Resistors	R_Small	Resistor_THT:R_Axial_DIN0309_L9.0mm_D3.2mm_P15.24mm
Diode	1N4148	Diode_THT:D_DO-35_SOD27_P7.62_Horizontal
Pin Headers	Conn_01x02_Pin	Connector_PinHeader_1x02_P2.54mm_Vertical
Pin Headers	Conn_01x03_Pin	Connector_PinHeader_1x03_P2.54mm_Vertical
Operational Amplifier	LF347N	Package_DIP:DIP-14_W7.62mm_LongPads
Operational Amplifier	TL084CN	Package_DIP:DIP-14_W7.62mm_LongPads
14 Pin Socket	TL084	Package_DIP:DIP-14_W7.62mm_Socket_LongPads

Note: There are two Operational Amplifiers in the Materials. Either one can be used for this project.

5 PCB Board Layout

For the PCB Board Layout ensure that you have the following parameters included in your design.

5.1 Trace Widths

General Trace Widths:

- 0.3 mm

  Power Trace Widths:

- 0.5 mm

- 1.0 mm

5.2 Via Diameters

- 0.3 mm

- 0.5 mm

- 1.0 mm

6 Production

The selected group to produce the Boards for this project is OSH Park. Be sure to upload the file with the right parameters addressed to make sure that the board works appropriately.

When ordering the board:

- Use either Default PCB or After Dark

7 Assembly

Collect all of the components and make sure that you collect all of the appropriate components and assemble them in accordance to your design. If you have any descrpancies with understanding your board you can consult your schematic diagram on KiCad.

8 Testing

Testing will happen with the use of An Oscilloscope, Waveform Generator, DC Power Supply, and Multimeter.

9 Appendix A: KiCad Download Link

KiCad is an Open-Source Software developed to design Printed Circuit Boards (PCBs).

- Official Website: https://www.kicad.org

- Download Page: https://www.kicad.org/download/

10 Appendix B: OSH Park Link

OSH Park is a PCB Manufacturing Group in Portland, Oregon.

- Official Website: https://oshpark.com

- Sign In / Create an Account: https://oshpark.com/users/sign_in