GITAM

(DEEMED TO BE UNIVERSITY)

TITLE: PCB WORKSHOP

SUBTITLE: ANALOG CIRCUIT

NAME: VARSHINI V
PROGRAM: ECE-REGULAR
REGISTER NO: BU22EECE0100206
ACADEMIC YEAR: 2022-2026

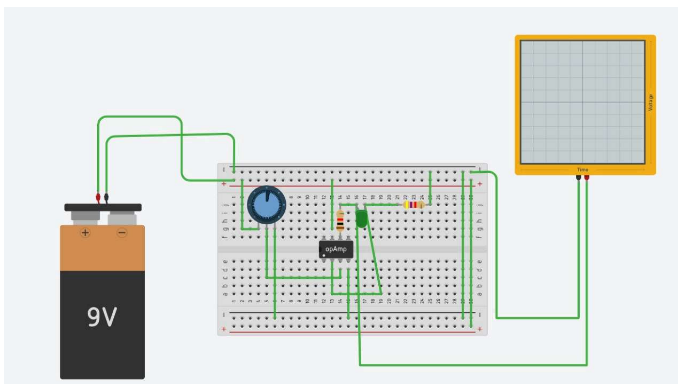
1.Prototype Title: Transconductance Amplifier

2. Simulation Results

a. Detailed Description of the Simulation Results

The simulation of the transconductance amplifier was conducted using a Tinker cad and EasyEDA - based simulation tool. The following parameters were examined: input/output waveforms, frequency response, and transient response.

b. Include Screenshots or Diagrams

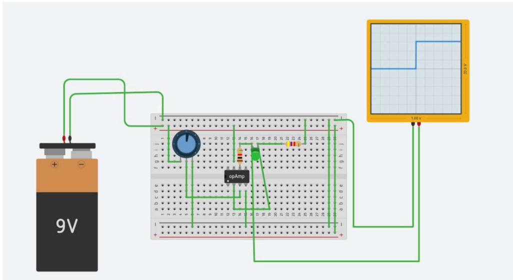


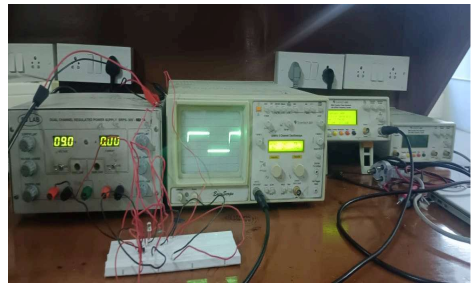
c. Illustrate Key Aspects of the Simulation

<u>i. Input/Output Waveforms</u>

Input Square Waveform: The input square wave will have abrupt transitions between high and low states. It will consist of alternating periods of high voltage (logic high) and low voltage (logic low). The frequency of the square wave will determine how quickly these transitions occur.

Output Square Waveform: The output square wave should ideally replicate the input waveform, albeit with some changes due to the characteristics of the amplifier. The output waveform may exhibit some distortion, such as overshoot, ringing, or delay, depending on the bandwidth and transient response of the amplifier.





ii.Frequency Response Plot:

The frequency response plot displays the gain of the transconductance amplifier over a range of frequencies, typically from low frequencies to high frequencies**.**

iii. Transient Responses:

The transient response was examined by applying a step input to the amplifier. The output waveform indicated the response time and settling time of the amplifier.

3. Hardware Results

a. Present Hardware Implementation of the Prototype on Breadboard

i. Include Measurements, Observations

The prototype was built on a breadboard using discrete components. Measurements were taken using an oscilloscope and a signal generator. Key parameters such as gain, bandwidth, and linearity were recorded.

ii. Deviations from the Expected Behavior

Some deviations were observed in the hardware implementation compared to the simulation results. Notable discrepancies included slightly lower gain and limited bandwidth.

4. Comparison of Simulation and Hardware Results

a. Analyze and Compare the Simulation Results with the Hardware Results

The simulation results closely matched the hardware results with minor deviations. The gain in the hardware was slightly lower, and the bandwidth was somewhat narrower.

b. Identify Any Discrepancies or Differences

Discrepancies were primarily due to parasitic elements and non-idealities in the components used for the hardware prototype.

c. Discuss Possible Reasons for Variations

Variations can be attributed to factors such as parasitic capacitance, resistance, and the quality of the discrete components. Additionally, breadboard connections may introduce noise and additional resistance.

5. Design Finalization

a. Document the Final Design Parameters

Final design parameters included optimized component values to achieve desired performance metrics.

b. Fix Specifications Based on the Simulation and Hardware Results

Specifications were refined to match the achievable performance in hardware, including gain, bandwidth, and linearity.

c. Discuss Any Modifications or Optimizations Made to the Initial Design to Improve Performance

Modifications included using precision resistors and capacitors, and improving the layout to minimize parasitic effects.

d. Address Issues Encountered During Testing

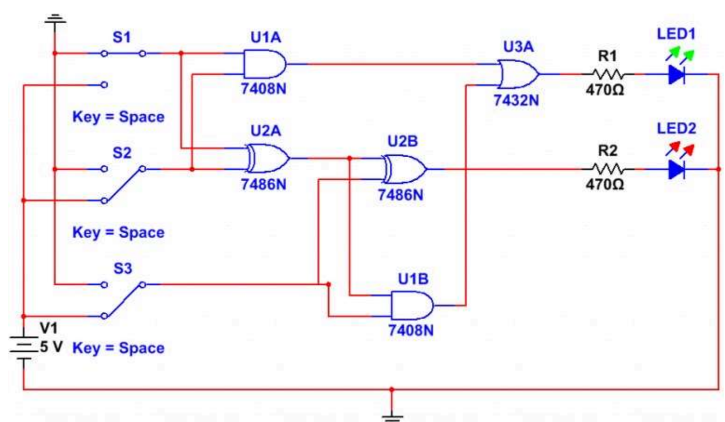
Issues such as noise and unexpected signal attenuation were addressed by improving the power supply decoupling and using shielded cables for signal connections.

6. Circuit Building on EasyEDA Tools

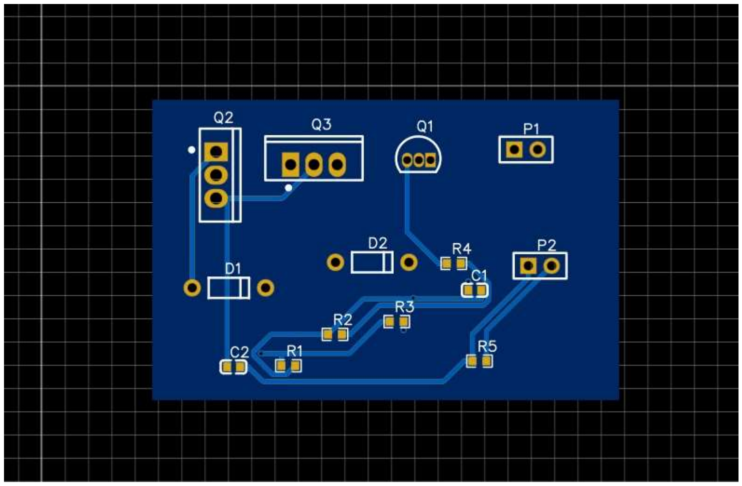
a. Outline the Process of Translating the Finalized Design into a Circuit Layout

i. Provide Step-by-Step Instructions, Screenshots

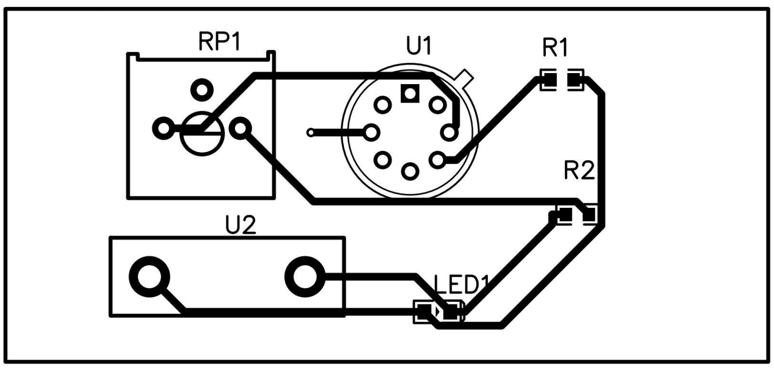
- Create a New Project: Open EasyEDA and create a new project.
- Schematic Capture: Draw the schematic using EasyEDA's component library.



- Annotate and Netlist: Annotate the schematic and generate the netlist.
- PCB Layout: Import the netlist into the PCB layout tool and place components.



- Route Traces: Route the traces ensuring minimal path lengths and proper grounding.



- Design Rule Check: Perform a design rule check to ensure there are no errors.

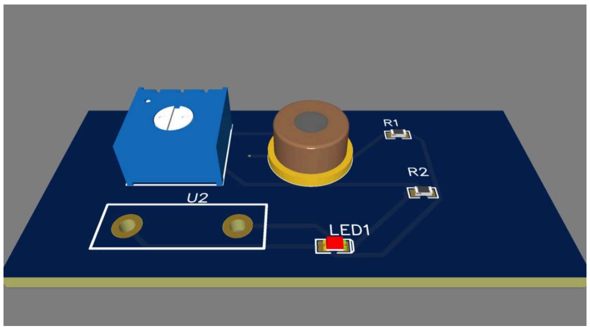
7. PCB Designing on EasyEDA Tools
a. Describe the Process of Designing the PCB Layout Based on the Circuit Layout
i. Explain How to Place Components, Route Traces, and Optimize the PCB Layout for Manufacturability and Performance

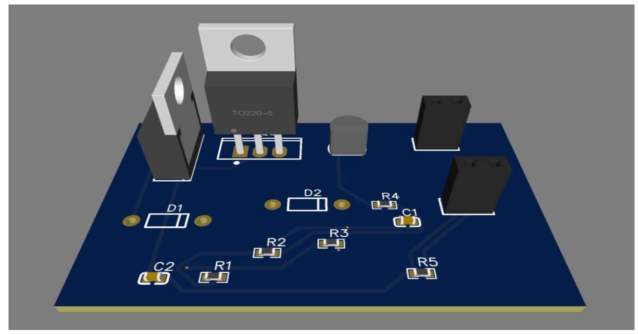
- Component Placement: Place components logically based on the schematic, keeping related components close.
- Trace Routing: Route traces while maintaining signal integrity. Use wider traces for power lines and critical signal paths.
- Optimization: Optimize the layout for manufacturability by ensuring clearances, reducing trace lengths, and avoiding right-angle traces.

8. Verification of the Final Design
a. Detail the Steps Taken to Verify the Final PCB Design

i. Include Checks for Electrical Connectivity, Design Rule Compliance, and Any Additional Verification

- Electrical Connectivity Check: Verify that all connections match the schematic.
- Design Rule Compliance: Ensure compliance with the design rules set by the PCB fabrication service.





9. Download the Gerber File

a. Provide Instructions for Downloading the Gerber File of the Final PCB Design from EasyEDA

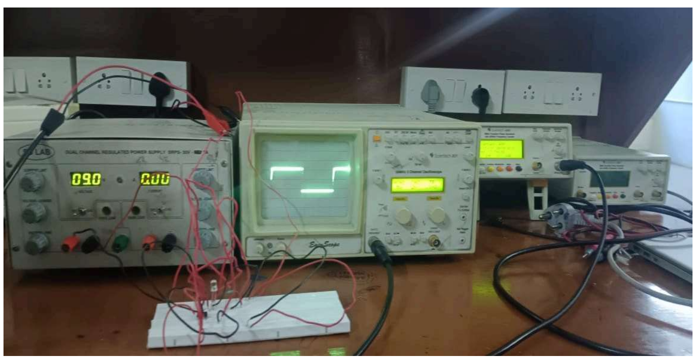
i. Explain How to Generate and Export the Gerber File for Submission to a PCB Fabrication Service

- Open your PCB project: Log in to your EasyEDA account and navigate to the project containing your transconductance amplifier PCB design.
- Open the PCB Editor: Click on the PCB icon or the "PCB Layout" option within your project to open the PCB editor.
- Verify the design: Before generating the Gerber files, ensure that your PCB design is complete and error-free. Double-check the component placement, routing, and design rules to avoid any issues during fabrication.
- Generate Gerber files:In the PCB editor, click on "File" in the menu bar. Select "Export" > "Gerber File..." from the dropdown menu. This will open the Gerber file generation dialog box.
- Configure Gerber settings:In the Gerber file generation dialog box, review and adjust the settings as needed. Ensure that all necessary layers are selected, including copper layers, solder mask, silkscreen, drill files, and any additional layers required for your PCB fabrication process.
- Generate Gerber files: Once you've configured the settings, click on the "Generate" or "Export" button to create the Gerber files for your PCB design.
- Download Gerber files:After the Gerber files are generated, EasyEDA will compile them into a zip file.Click on the download link or button to save the zip file containing the Gerber files to your computer.
- Verify Gerber files: Before submitting the Gerber files to the PCB fabrication service, it's a good practice to verify them using a Gerber viewer software to ensure that all layers and elements are correct.

- Submit Gerber files for fabrication: Once you've confirmed that the Gerber files are accurate, upload them to your chosen PCB fabrication service for manufacturing.

10. Appendix

The transconductance amplifier, supplementary materials are provided to offer a more comprehensive understanding of the design, simulation, and testing processes. Schematics detailing the circuit configuration, simulation setups showcasing the parameters under investigation, and simulation results elucidating the amplifier's performance across various scenarios are included. Additionally, raw measurement data obtained during hardware testing, such as gain measurements and frequency response analyses, are presented alongside detailed calculations for key amplifier parameters. Datasheets for crucial components, images of the printed circuit board (PCB) ensure transparency regarding the components utilized and their specifications. This appendix is further enriched with references to relevant literature and supporting documentation, augmenting the credibility and depth of the main report's findings.



<u>TEAM MEMBERS:</u>
Monisha H S(BU22EECE0100439)
Priyanka G(BU22EECE0100447)
Varshini V(BU22EECE0100206)

THANK YOU