CubeSat Temperature Monitoring System using LM35 Sensor

SaiPrabha C Y

May 9, 2026

Contents

1 Abstract

Temperature monitoring is one of the essential subsystems in embedded electronics, aerospace systems, satellites, and CubeSat missions. Electronic components operating in space and embedded environments are highly sensitive to thermal variations. Therefore, a reliable and compact temperature sensing system becomes an important part of the spacecraft health monitoring process.

This project presents the design and implementation of a PCB-based temperature monitoring system using the LM35 analog temperature sensor. The complete schematic design, PCB layout generation, routing, footprint assignment, electrical verification, and 3D visualization were successfully developed using KiCad.

The system includes an LM35 temperature sensor, LED indication circuit, current limiting resistor, and output connector header for interfacing with external embedded systems such as Arduino, ESP32, STM32, or CubeSat onboard computers.

The project demonstrates practical skills in PCB design workflow, schematic development, hardware layout planning, track routing, and embedded hardware preparation relevant to aerospace electronics and embedded system engineering.

2 Introduction

CubeSats are miniature satellites widely used for research, communication, Earth observation, and educational purposes. Since these satellites operate in extreme environmental conditions, temperature monitoring becomes critical for maintaining subsystem reliability and preventing component failure.

Thermal monitoring systems are used to:

- Monitor onboard electronics temperature
- Detect overheating conditions
- Improve subsystem reliability
- Assist in thermal management strategies
- Provide telemetry data for ground stations

The LM35 sensor is a widely used analog temperature sensor due to its simplicity, low cost, and linear voltage output characteristics. It provides an output voltage directly proportional to temperature in degrees Celsius.

In this project, a compact PCB was designed using KiCad to implement a temperature sensing module that can later be integrated into embedded systems or CubeSat-oriented electronics.

3 Objectives

The main objectives of this project are:

- To design a PCB-based temperature monitoring system
- To learn practical schematic capture techniques
- To understand PCB layout and routing procedures
- To implement footprint assignment in KiCad
- To verify design correctness using ERC and DRC
- To visualize the PCB in 3D format
- To prepare a foundation for future CubeSat subsystem development

4 Components Used

S.No	Component	Purpose
1	LM35-LP Temperature Sensor	Temperature sensing
2	LED	Status / power indication
3	Resistor	Current limiting for LED
4	Conn_01x03 Header	External interfacing connection
5	PCB Board	Hardware implementation platform
6	KiCad Software	PCB design and visualization

5 Software Used

The following software tools and modules were used during the project development process:

5.1 KiCad Software Suite

KiCad is an open-source Electronic Design Automation (EDA) software suite widely used for schematic capture, PCB design, routing, footprint assignment, and hardware visualization.

In this project, KiCad was selected because it provides a complete PCB development workflow suitable for embedded systems and CubeSat-oriented hardware design.

The following KiCad modules were used throughout the project:

- Schematic Editor
- PCB Editor
- Footprint Assignment Tool
- Electrical Rules Checker (ERC)
- Design Rules Checker (DRC)
- Gerber Plot Generator
- Drill File Generator
- 3D Viewer

5.2 Purpose of Each Tool

Software Tool	**Purpose**
Schematic Editor	Used to create the electronic circuit diagram and establish electrical connections between components.
Footprint Assignment Tool	Used to assign physical PCB packages to schematic symbols before PCB generation.
PCB Editor	Used to arrange components, draw PCB boundaries, and perform copper routing.
ERC (Electrical Rules Checker)	Used to verify electrical correctness of the schematic and detect wiring issues.
DRC (Design Rules Checker)	Used to verify PCB routing rules and detect physical layout problems.

3D Viewer	Used to visualize the final PCB in realistic 3D format.
Gerber Plot Tool	Used to generate manufacturing files required for PCB fabrication.
Drill File Generator	Used to create drilling information for PCB manufacturing.

5.3 Future Software Enhancements

In future versions of the project, LTspice simulation software can be used to:

- Simulate LM35 sensor output
- Analyze voltage response
- Verify analog behavior
- Observe filtering performance

6 Working Principle

The LM35 sensor operates as an analog temperature sensor that generates an output voltage proportional to temperature.

The sensor provides:

$$10\text{mV output per degree Celsius}$$

For example:

- 25°C produces approximately 250mV
- 30°C produces approximately 300mV
- 50°C produces approximately 500mV

The PCB includes:

- Power input connection
- Temperature sensor module
- LED indication system

- Output header for embedded interface

The analog output from the LM35 can later be connected to:

- Arduino ADC pins
- ESP32 analog input
- STM32 ADC channels
- CubeSat telemetry systems

7 Circuit Description

The complete circuit consists of:

7.1 LM35 Sensor Section

The LM35 sensor contains three pins:

- VCC (+5V Supply)
- VOUT (Analog Temperature Output)
- GND (Ground)

7.2 LED Indicator Section

An LED along with a resistor was added to indicate power or activity status.

The resistor protects the LED from excessive current.

7.3 Connector Header

A 3-pin connector header was added for:

- +5V
- Temperature Output
- Ground

This enables external interfacing with microcontrollers and embedded systems.

8 Schematic Design Procedure

The following steps were performed during schematic creation:

1. Opened KiCad schematic editor
2. Added LM35 sensor symbol
3. Added resistor and LED components
4. Added connector header
5. Connected all components using wiring tools
6. Added power and ground symbols
7. Added output labels
8. Performed Electrical Rules Check (ERC)

After verification, the schematic showed:

$$\text{ERC Errors} = 0$$

9 PCB Design Procedure

After successful schematic completion, PCB design was started.

9.1 Footprint Assignment

Footprints were assigned to all components:

- LM35 → TO-92 Inline Package
- LED → LED THT Footprint
- Resistor → Axial Through-Hole Footprint
- Connector → Pin Header Footprint

9.2 PCB Layout

The PCB editor was used to:

- Import netlist connections
- Arrange components
- Draw PCB board outline
- Route copper tracks
- Optimize component placement

9.3 Routing

Copper traces were routed between:

- Sensor pins
- LED section
- Connector header
- Power and ground connections

9.4 Design Verification

The Design Rules Checker (DRC) was executed.

Results obtained:

$$\text{DRC Errors} = 0$$

10 3D Visualization

KiCad 3D Viewer was used to visualize the final PCB model.

The following components were successfully visualized:

- LM35 Sensor
- LED
- Resistor
- Connector Header

- PCB Board

The 3D visualization confirms:

- Correct footprint assignment
- Proper component orientation
- Successful PCB generation

11 Results

The project was successfully completed with the following results:

- Complete schematic generated
- PCB layout designed successfully
- Copper routing completed
- ERC verification completed
- DRC verification completed
- 3D PCB visualization completed
- Ready for PCB manufacturing

12 Applications

This project can be used in:

- CubeSat thermal monitoring
- Embedded systems projects
- Aerospace electronics
- IoT environmental monitoring
- Industrial temperature monitoring
- Educational embedded hardware training

13 Advantages

- Simple and compact design
- Low-cost implementation
- Easy microcontroller integration
- Lightweight PCB structure
- Suitable for embedded systems learning
- Expandable for future CubeSat projects

14 Limitations

- Analog output may require filtering
- Basic single-layer routing
- No wireless communication included
- Requires external controller for telemetry

15 Future Enhancements

The following improvements can be implemented in future versions:

- Add capacitor filtering
- Integrate ESP32 or STM32
- Add wireless telemetry transmission
- Add onboard display system
- Add data logging capability
- Implement CubeSat communication interface
- Add LTspice simulation analysis
- Develop multi-sensor thermal monitoring system

16 Screenshots and Workflow Documentation

The following screenshots were captured throughout the PCB development workflow to document each important stage of the project.

16.1 Title Page and Initial Project Setup

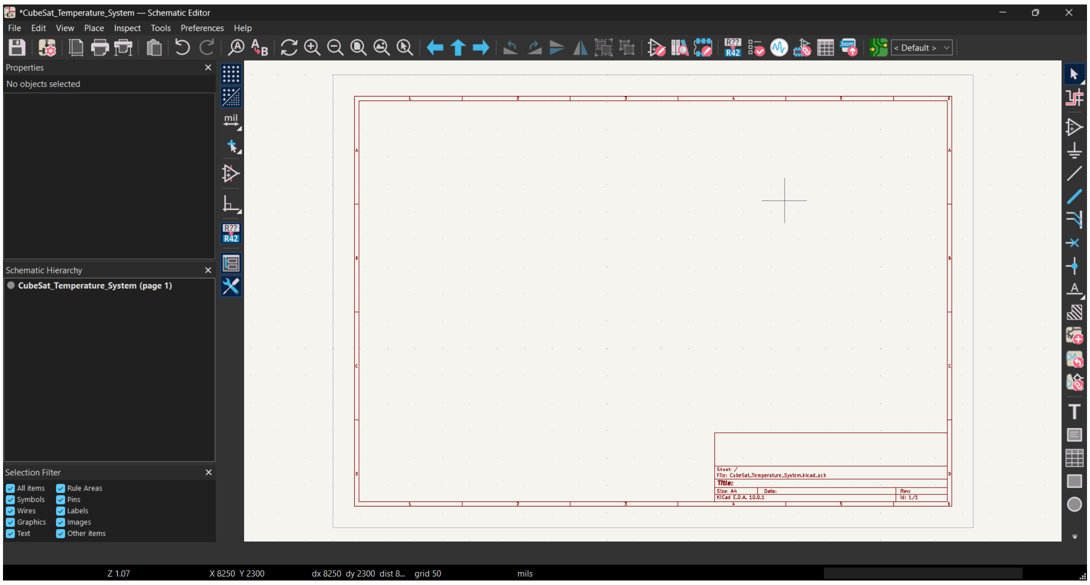

Figure 1: Initial Project Setup in KiCad

This stage involved creating a new KiCad project and preparing the workspace for schematic design.

16.2 Schematic Diagram

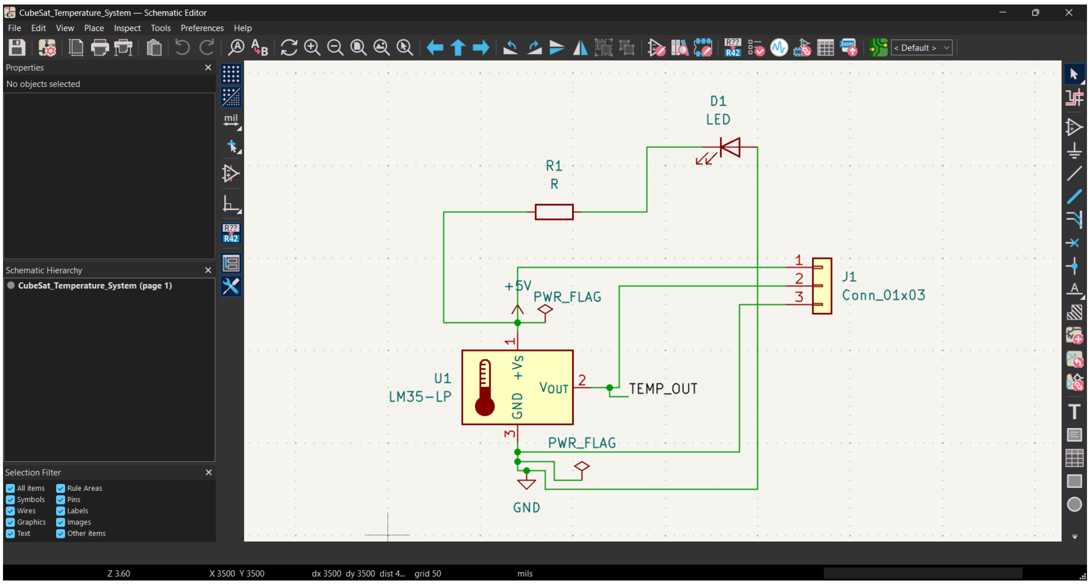

Figure 2: LM35 Temperature Monitoring Schematic

The schematic diagram contains the LM35 temperature sensor, resistor, LED, connector header, and power connections.

16.3 Electrical Rules Checker (ERC)

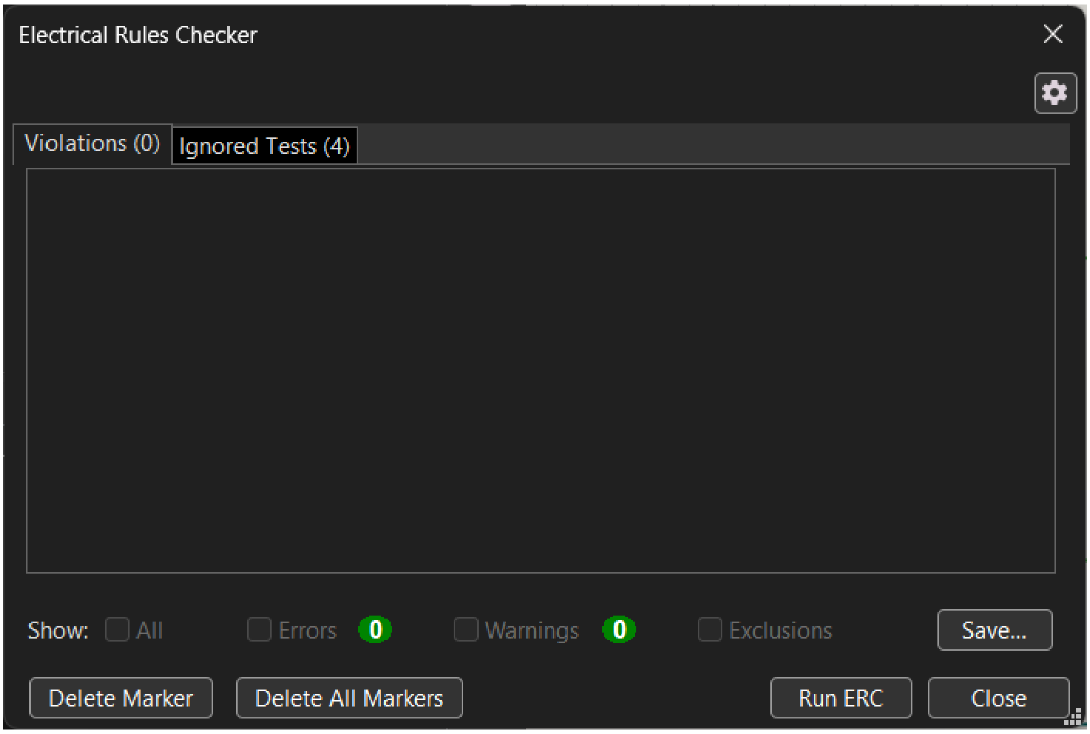

Figure 3: Electrical Rules Checker Verification

The ERC verification confirmed that all schematic connections and electrical rules were correct without major errors.

16.4 Footprint Assignment

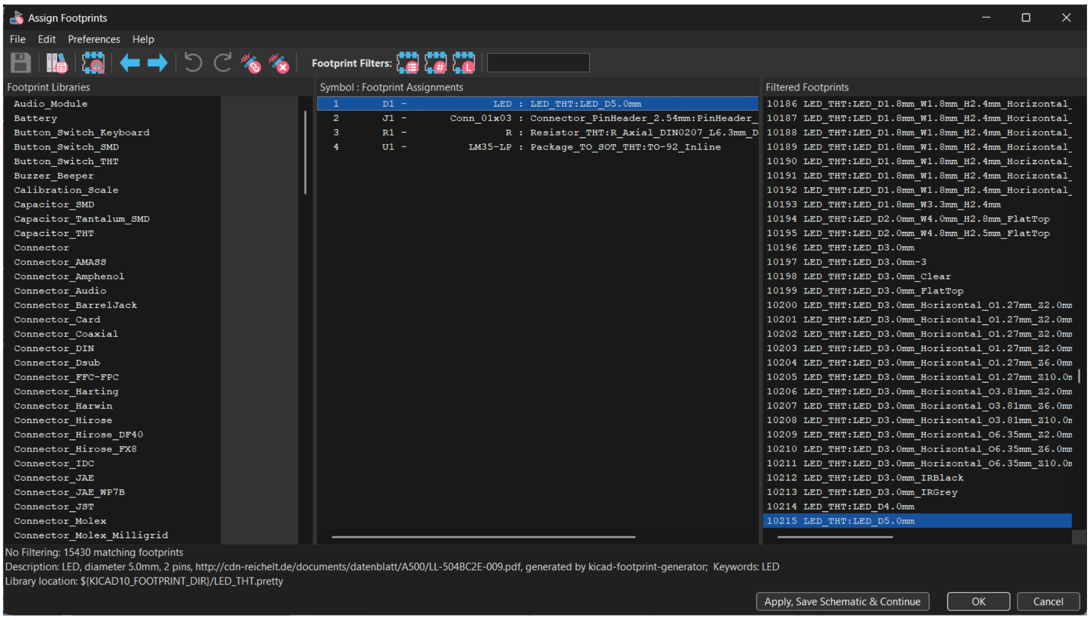

Figure 4: Footprint Assignment Process

Physical PCB footprints were assigned to all schematic symbols before PCB generation.

16.5 PCB Editor Layout

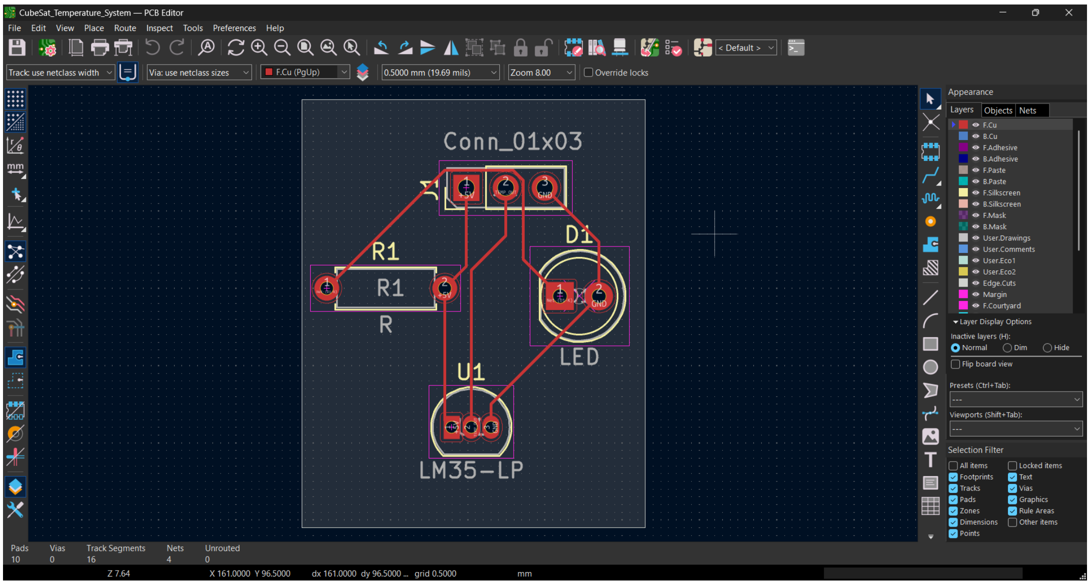

Figure 5: PCB Layout in PCB Editor

The PCB editor was used to arrange components, create board boundaries, and perform copper routing.

16.6 Design Rules Checker (DRC)

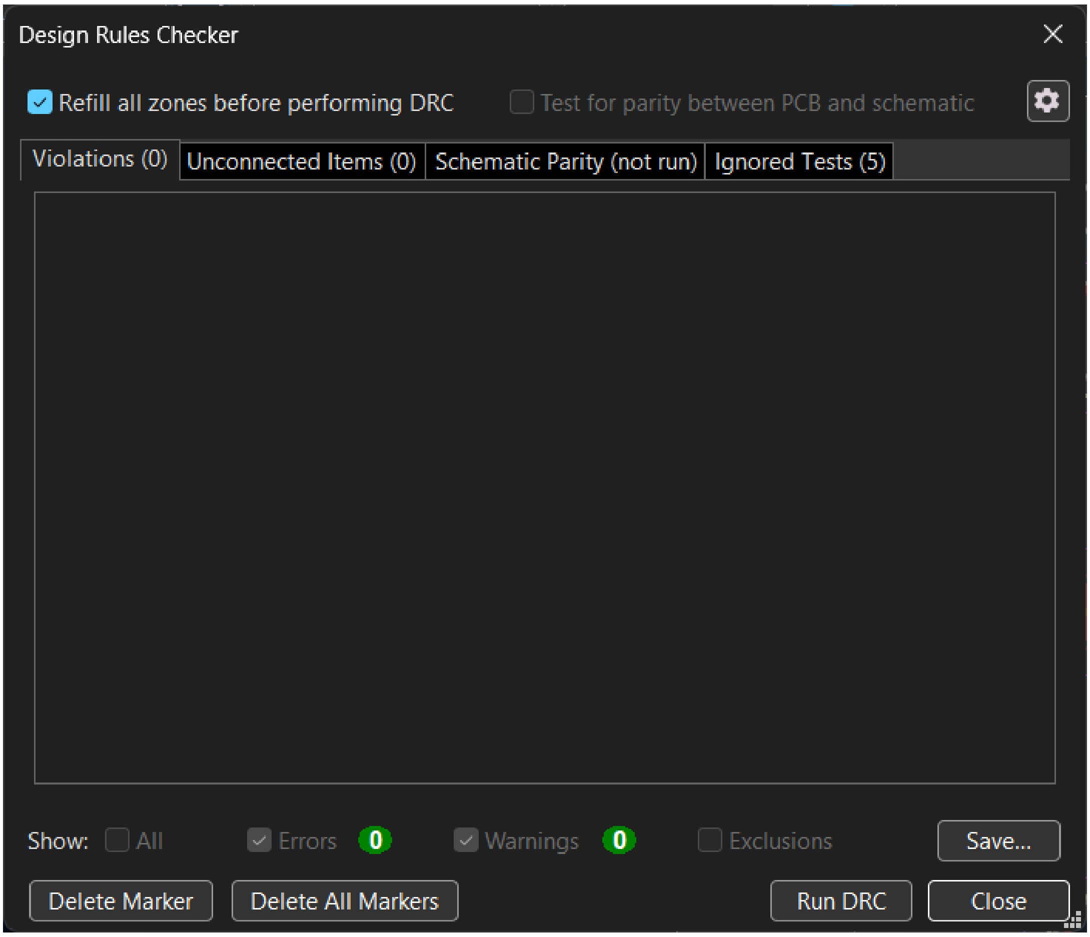

Figure 6: Design Rules Checker Verification

The DRC verification ensured that the PCB routing satisfied all required design constraints.

16.7 3D PCB Visualization

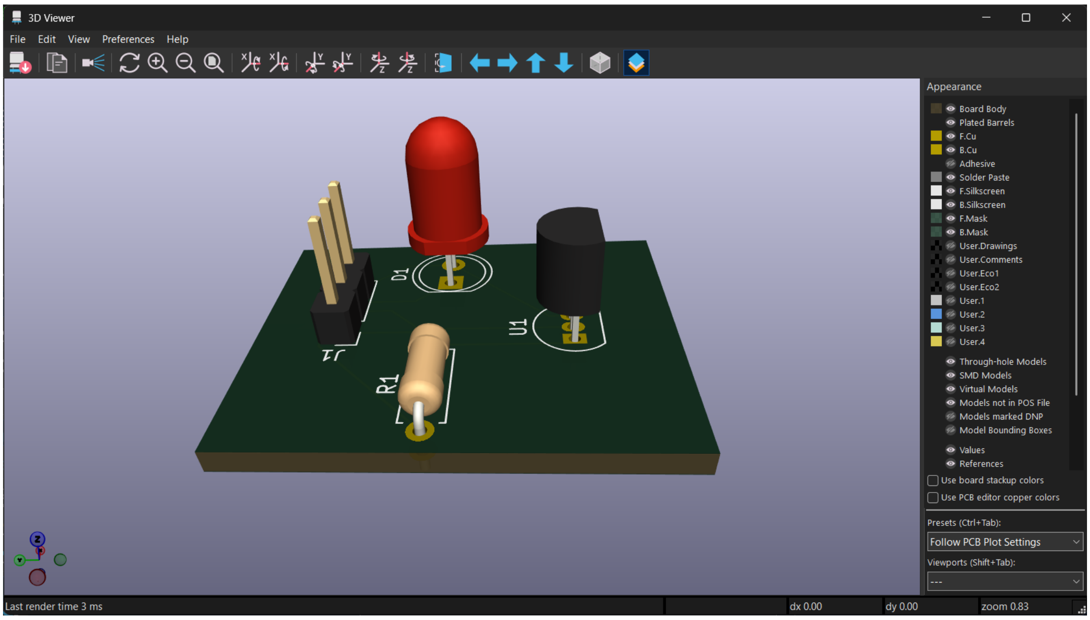

Figure 7: 3D Visualization of Final PCB

The 3D viewer provided a realistic visualization of the PCB including all placed components.

16.8 Gerber Plot Generation

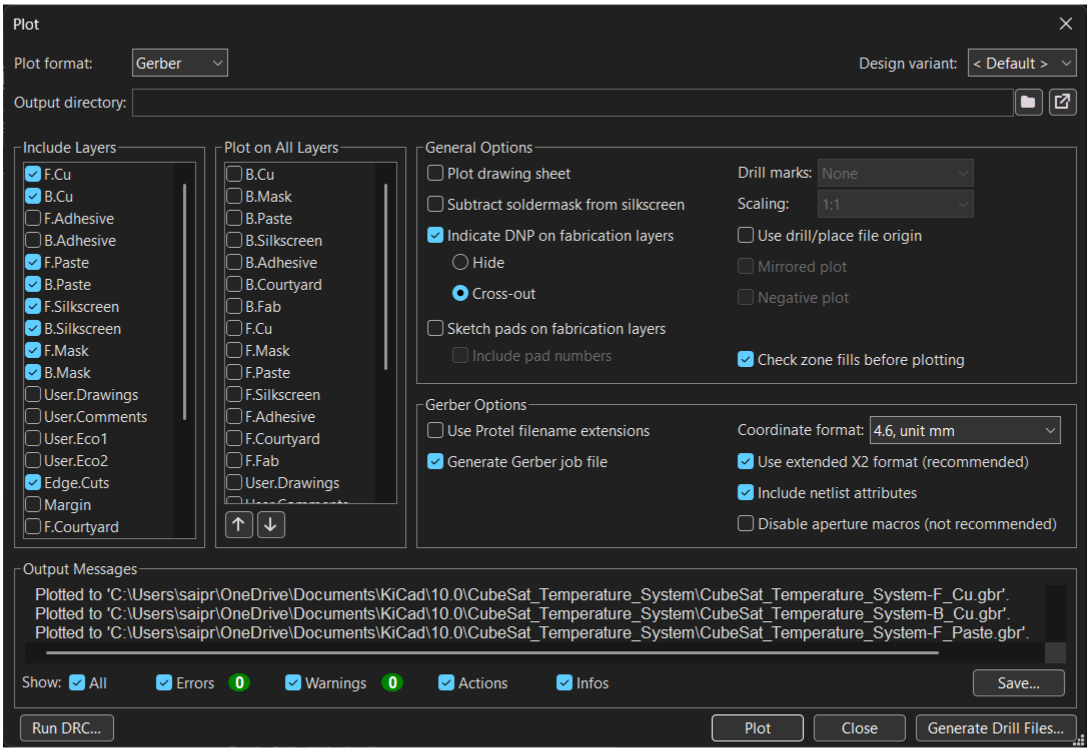

Figure 8: Gerber File Generation Process

Gerber files were generated for PCB manufacturing purposes.

16.9 Final Project Files

Figure 9: Final Project Files and Outputs

The final project files included schematic files, PCB layouts, Gerber files, drill files, and 3D visualizations.

17 Conclusion

A compact PCB-based temperature monitoring system using the LM35 sensor was successfully designed and verified using KiCad.

The project provided practical exposure to:

- Schematic capture
- PCB design workflow
- Footprint assignment
- Copper routing
- Design verification
- 3D hardware visualization

The project establishes a strong foundation for future embedded systems and CubeSat-oriented hardware development.

Although full analog simulation was not performed in this phase, the PCB design workflow and hardware implementation process were successfully demonstrated using KiCad. Future work can include LTspice-based simulation and hardware testing using physical components.

The system can later be expanded into a more advanced aerospace-grade thermal monitoring subsystem with wireless telemetry and intelligent onboard processing.

18 References

1. KiCad Documentation
2. LM35 Datasheet
3. Embedded Systems Design References
4. CubeSat Subsystem Design Concepts