

Design and Implementation of an Optocoupler PCB Layout Using CadSoft Eagle Software

Ritu Raj Sondhiya¹, sondhiyar2rj@gmail.com, Orcid Id - <https://orcid.org/0009-0009-4349-7559>
Chinmay Pandey², chinmaypandey9@gmail.com, Orcid Id - <https://orcid.org/0009-0002-0878-9942>
Prashant Agrawal³, Pagrawal54@gmail.com, Orcid Id - <https://orcid.org/0009-0006-4763-9946>
Prof. (Dr.) Vikash Kumar Singh⁴, Professor, drvksingh76@gmail.com,
Orcid Id - <https://orcid.org/0009-0003-1438-149X>

^{1,2,3,4}Department of Computer Science, Indira Gandhi National Tribal University (IGNTU – A Central University), Amarkantak Distt. Anuppur Madhya Pradesh – 484887,

Abstract

The design process of an optocoupler circuit Printed Circuit Board (PCB) layout is presented through the implementation of CadSoft Eagle software. An optocoupler functions as an optoisolator to transmit electrical signals between two separate circuits by using light. The PCB layout stands as a critical element for optocoupler circuit functionality because it determines signal integrity and noise immunity and overall reliability. The paper demonstrates the Eagle software application for PCB design through schematic capture followed by component placement and routing and concludes with design rule checking (DRC). The design effectiveness is confirmed by simulation testing of the final layout.

1. Introduction

1.1 Background

The requirement to isolate electronic circuits at modern times remains crucial because high voltages combined with noise and ground loops affect sensitive components. Optocouplers serve as opto isolators which find extensive use in solving this problem. The signal transfer between isolated circuits occurs through light signals which use an LED and phototransistor or photodiode. Optocouplers function as essential components in power supplies and motor control systems and communication systems and industrial automation because they separate input and output circuits to block electrical interference and voltage spikes and ground loops. The performance quality of optocoupler circuits depends significantly on how their Printed Circuit Board (PCB) is designed. The design quality of a PCB enables proper signal integrity while reducing noise and improving circuit reliability. The functionality of an optocoupler suffers degradation when PCB design quality falls short because it creates crosstalk and electromagnetic interference (EMI) and signal degradation. The PCB layout process stands as a fundamental step during the development of optocoupler-based circuits.

1.2 Importance of PCB Design in Optocoupler Circuits

An optocoupler circuit requires a PCB layout which solves multiple essential problems.

1. Electrical Isolation stands as the main purpose of an optocoupler. The PCB design needs to preserve the isolation barrier through adequate spacing and creepage distances between input and output circuits.

2.The layout of signal traces needs proper design to achieve minimal noise and interference. High-speed signals need special attention during layout because reflections and crosstalk should be avoided.

3.Optocouplers produce heat like other electronic components because of their operational activity. The PCB design needs to enable effective heat removal through its layout to prevent component damage and maintain operational reliability.

4.The design needs to follow manufacturing limitations which include minimum trace widths together with drill sizes and layer stack-up requirements to achieve reliable and cost-effective PCB fabrication.

1.3 Role of CadSoft Eagle Software in PCB Design

The widely used software tool CadSoft Eagle (Easily Applicable Graphical Layout Editor) serves as a popular solution for PCB design. The software solution offers complete functionality which includes schematic capture and component placement and routing and design rule checking (DRC) capabilities suitable for optocoupler circuit design. The interface of Eagle provides an efficient design process due to its component libraries that contain optocouplers and resistors and connectors together with its intuitive interface which enables designers to create complex layouts efficiently.

The Eagle design rule checking (DRC) feature verifies that all PCB layout elements comply with established quality restrictions regarding trace widths and spacing rules and via dimensions. The feature enables designers to detect typical mistakes during design which guarantees that the end product can be manufactured. Simulation tools embedded in Eagle let designers check circuit functionality prior to manufacturing which decreases the possibility of producing expensive mistakes.

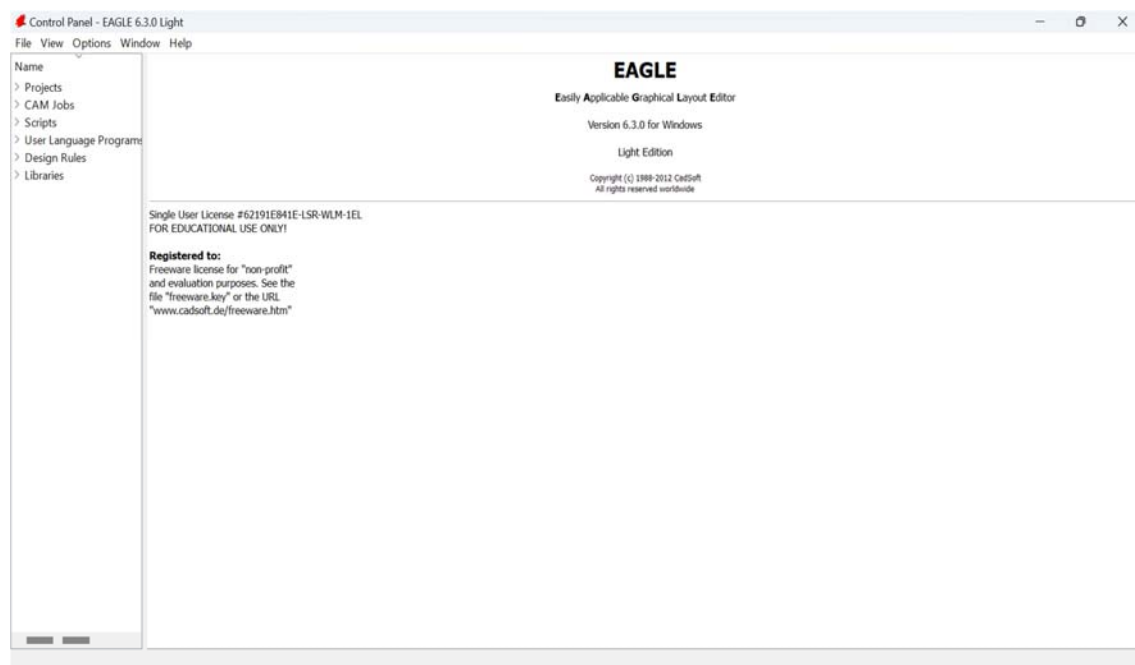


Figure 1 Open and work in the CadSoft Eagle Software

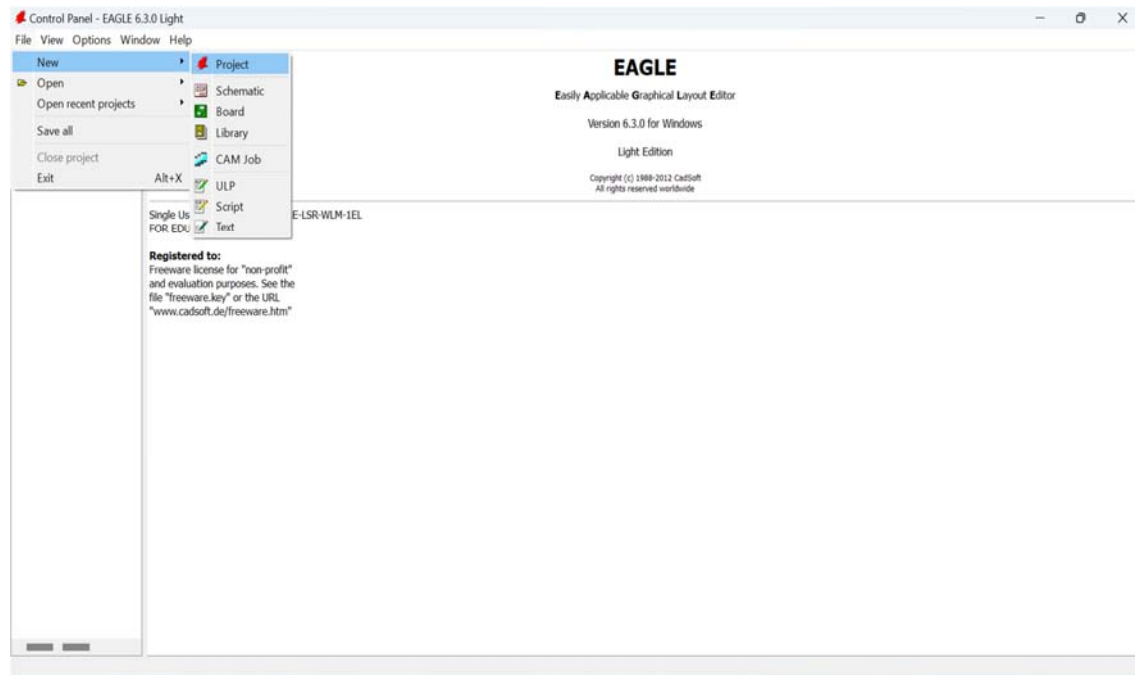


Figure 2 Open New Project

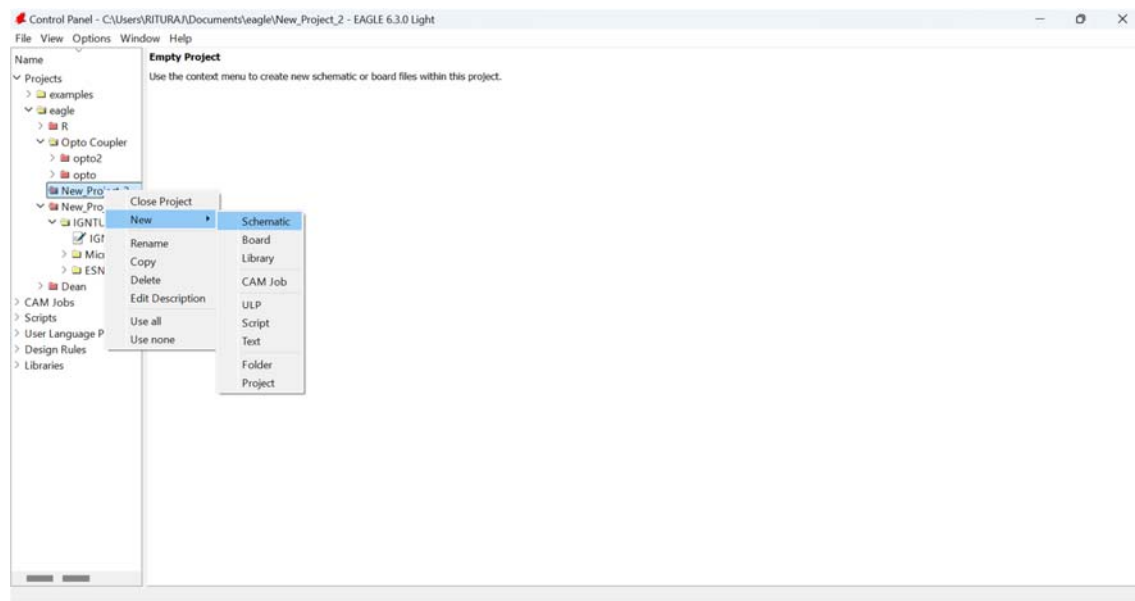


Figure 3 Open Schematic Layout

1.4 Objective of the Research

The main research goal involves designing and building an optocoupler circuit PCB through CadSoft Eagle software implementation. The design procedure follows a step-by-step process which consists of:

- 1.The first step involves drawing the circuit diagram through Eagle's schematic editor.
- 2.The placement of components on the PCB follows an arrangement for maximizing signal quality and thermal performance.
- 3.The design process requires the connection of components through copper traces which must follow established design rules.
- 4.The process of Design Rule Checking (DRC) verifies manufacturing compliance within the layout.
- 5.Simulation and Testing: Validating the circuit's functionality through simulation and real-world testing.

The study demonstrates why appropriate PCB layout methods are essential for maintaining reliable optocoupler circuit performance. This paper presents a practical design guide for engineers and designers through a demonstration of CadSoft Eagle design process.

2. Methodology

2.1 Schematic Capture

Circuit designers begin their project through schematic capture from within Eagle's schematic editor to produce the circuit diagram. The optocoupler circuit contains these elements:

- Optocoupler (PC817)
- Resistors (2.2k Ω and 3.3k Ω)
- Connectors (WAGO Screw Clamps)

The schematic is created by placing the components and connecting them according to the circuit diagram. The components are selected from Eagle's library, and their properties (e.g., resistance values) are defined.

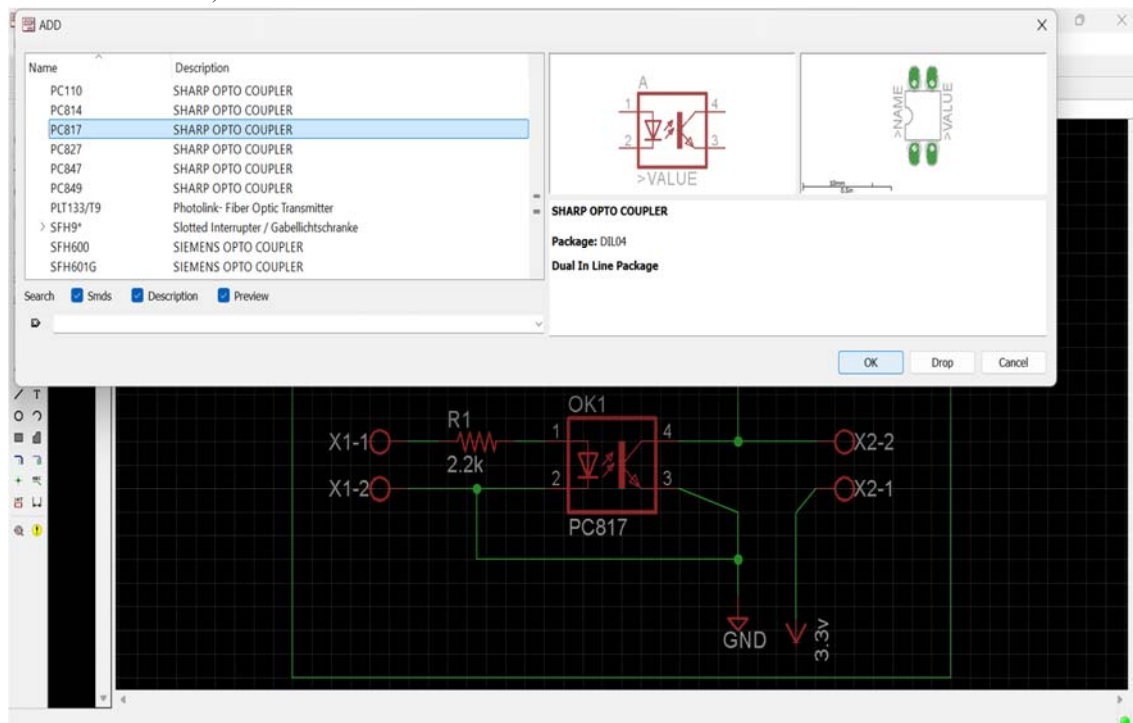


Figure 4 Select all Components

2.2 Component Placement

After finishing the schematic design, the following step involves placing components onto the PCB. Considering component placement in an appropriate manner proves essential for reduction of signal disturbances while simultaneously reducing electromagnetic noise through proper routing practices. The board placement begins with the following arrangement of components:

- The optocoupler (OK1) receives placement in the center section of the board to achieve shorter trace lengths.
- The optocoupler receives support from resistors R1 and R2 which minimize parasitic inductance and capacitance.
- The connectors (X1 and X2) receive placement at board edges to ensure easy accessibility.

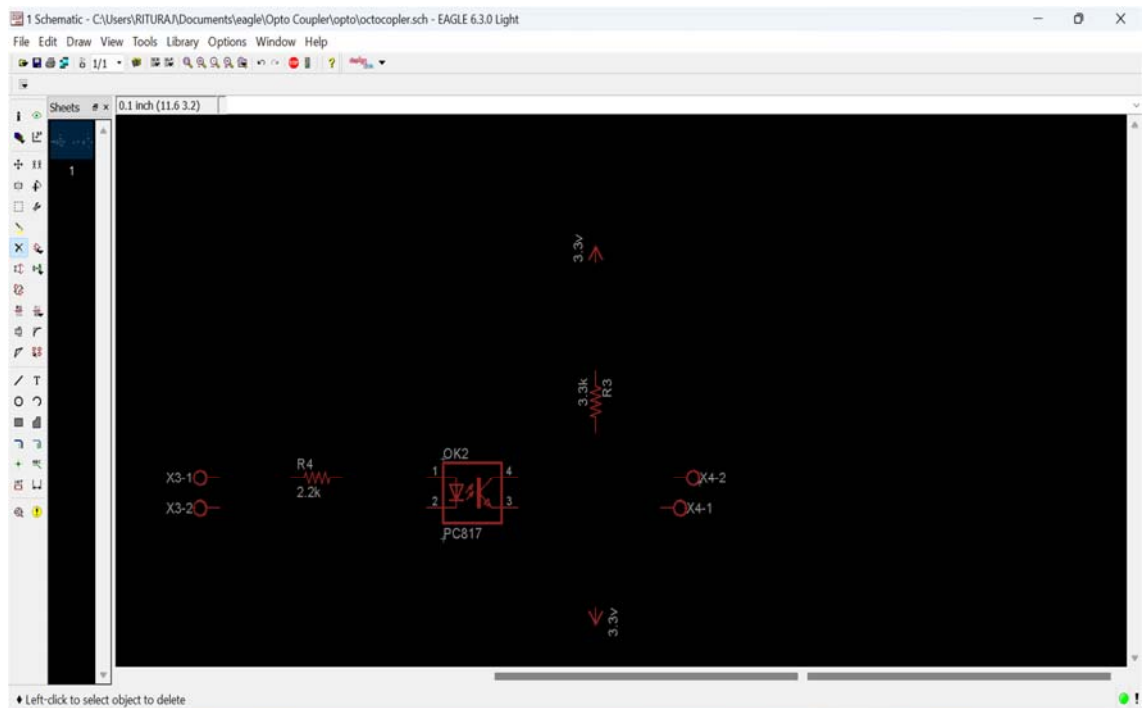


Figure 5 Placement of all selected components

2.3 Routing

The PCB components are connected through copper traces during the routing process. A series of design regulations monitors the routing process to help maintain signal quality and manufacturing readiness. The PCB implementation uses these routing methods:

- The top layer (Layer 1) receives signal traces to minimize interference between components.
- The bottom layer (Layer 16) contains all ground and power traces to establish a low-impedance pathway.

- The routing process requires the use of vias to establish connections between layers when needed.

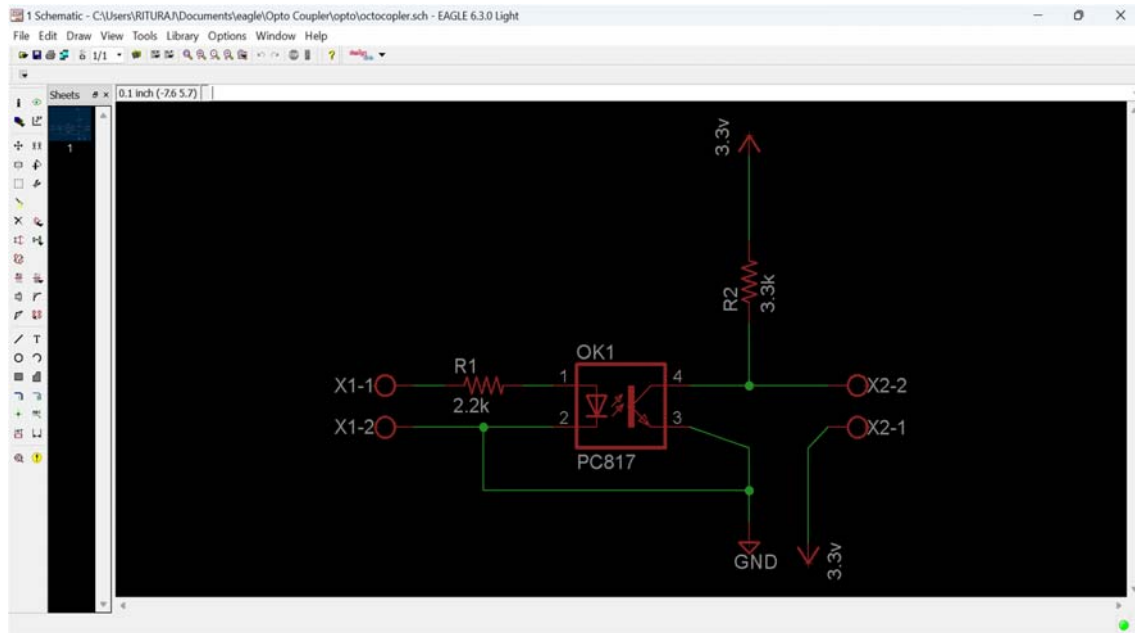


Figure 6 Route all components using wire tool

2.4 Design Rule Checking (DRC)

Design Rule Checking (DRC) is performed to ensure that the PCB layout adheres to the design rules specified in Eagle. The DRC process checks for issues such as minimum trace width, clearance between traces, and via sizes. Any violations are corrected before proceeding to the next step.

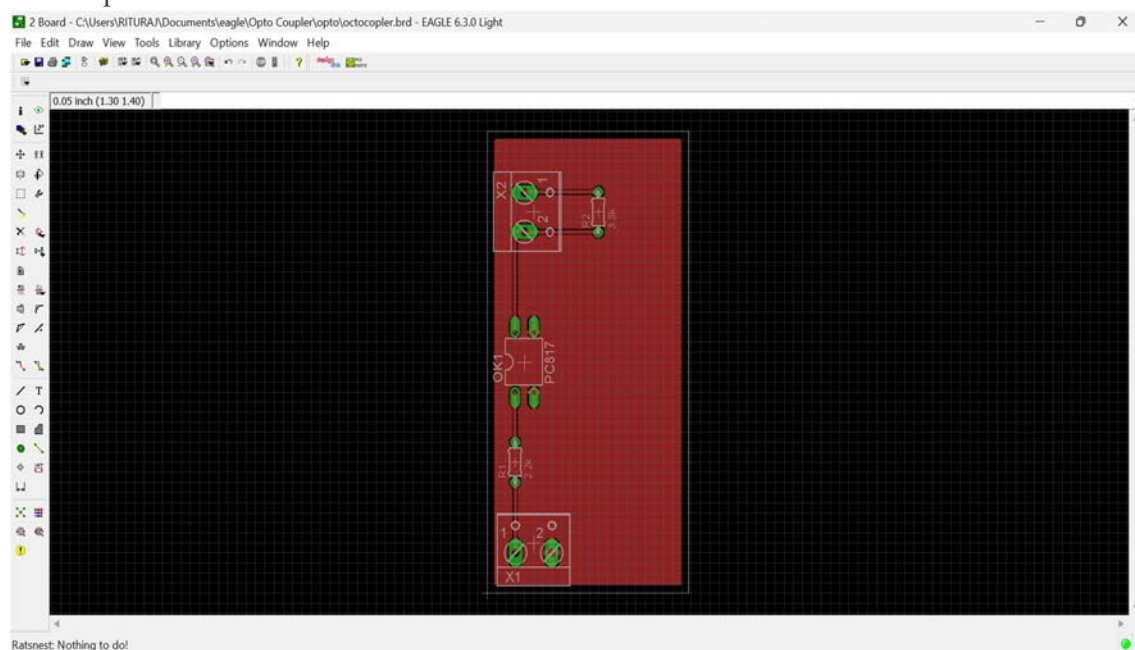


Figure 7 DRC checking of PCB using Ratsnest tool

2.5 Simulation and Testing

The final printed circuit board layout undergoes both simulation and testing for validation purposes. The built-in simulation tools from Eagle verify the circuit functionality through a simulation process. The PCB production process concludes with testing it in actual conditions to verify its operational functionality.

3. Results and Discussion

3.1 Schematic Capture

The schematic capture process finished successfully while Eagle's schematic editor generated the circuit diagram. The circuit components were properly positioned and linked in accordance with the original design.

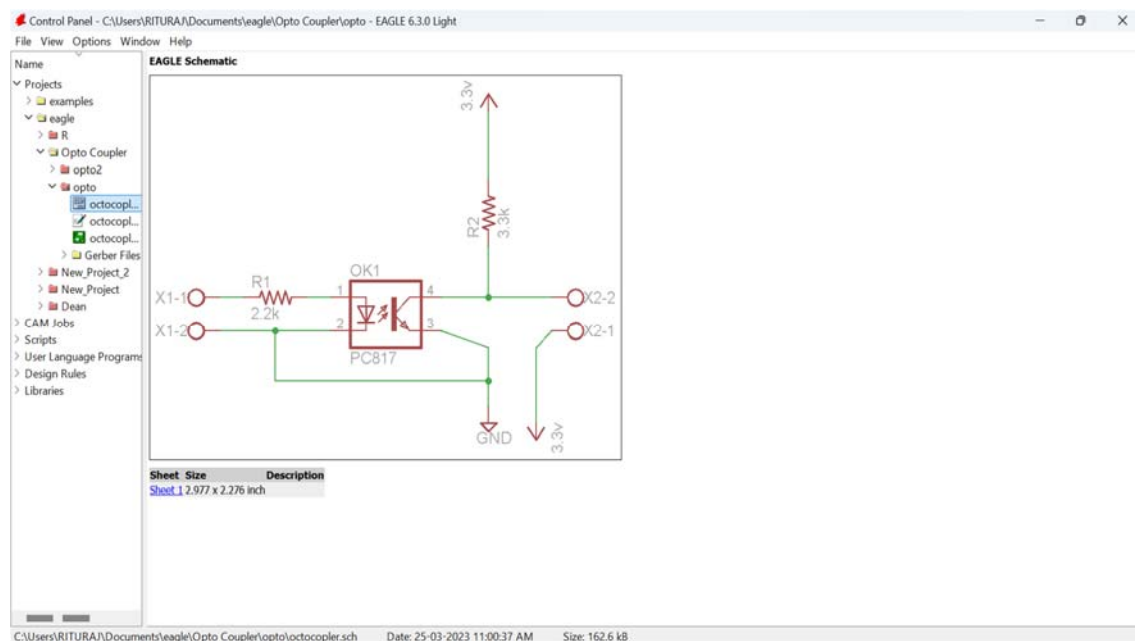


Figure 8 Export Schematic Sheet of optocoupler

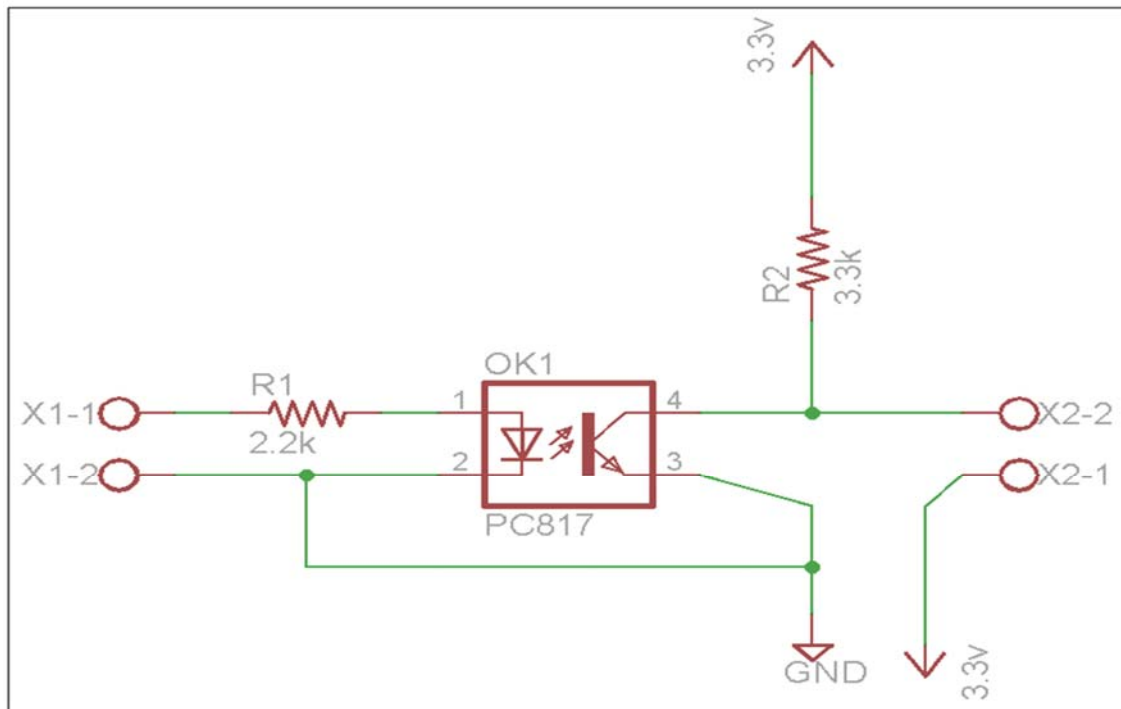


Figure 9 Schematic Diagram of optocoupler

3.2 Component Placement

The components were placed on the PCB in a manner that minimized trace lengths and reduced noise. The optocoupler was placed near the centre of the board, while the resistors and connectors were placed close to their respective connections.

3.3 Routing

The routing process finished successfully because all components received connection through copper traces. Vias enabled effective connection pathways between layers and the design specifications were applied across the entire fabrication process.

3.4 Design Rule Checking (DRC)

The DRC process detected minor violations that were resolved before the layout reached its final stage. The last layout design followed all design rules to achieve both signal integrity and manufacture readiness.

3.5 Simulation and Testing

The simulation output demonstrated proper functionality of the circuit design. The fabricated PCB underwent testing in an actual environment where the optocoupler circuit maintained reliable performance to achieve required input-output circuit isolation.



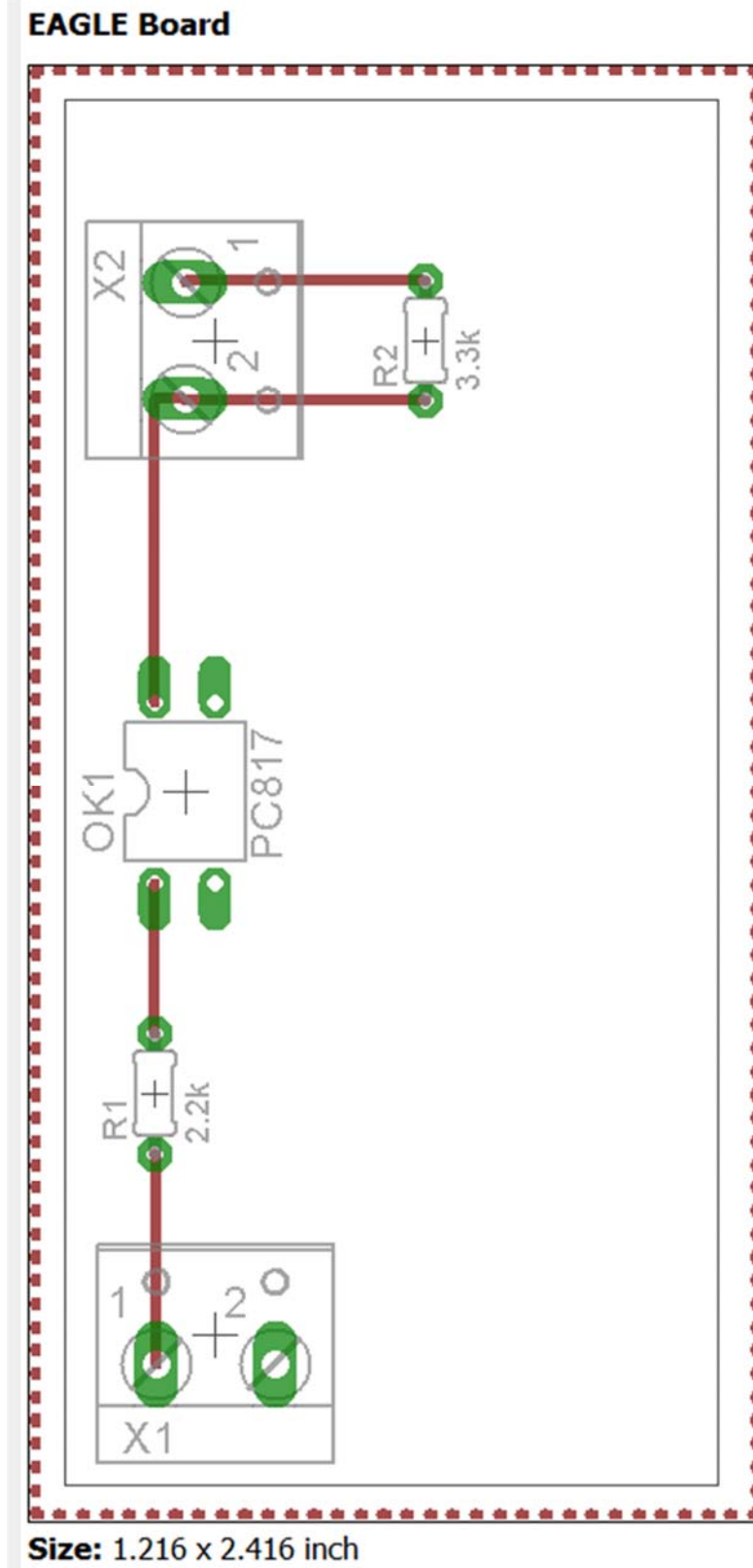
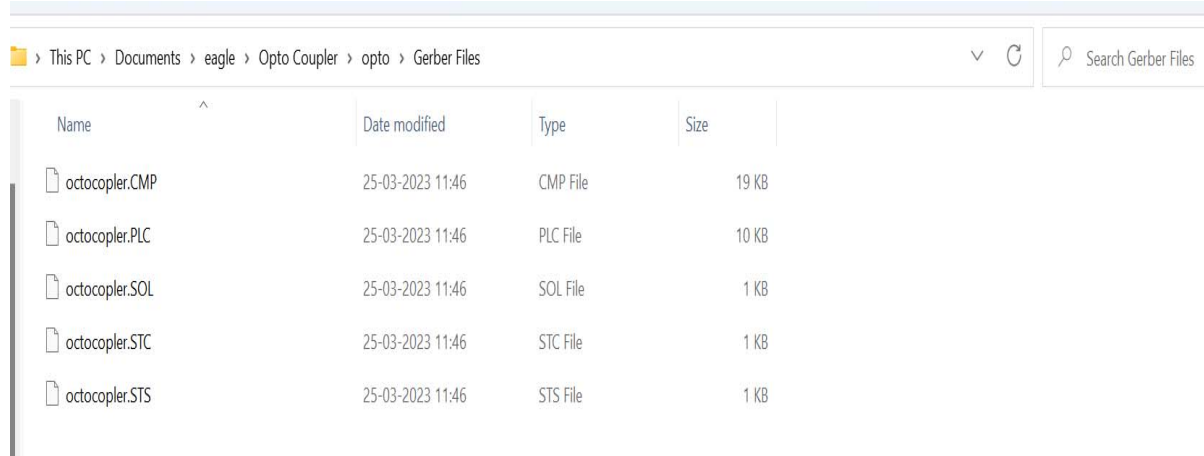


Figure 11 PCB Layout of optocoupler

3.6 Build Gerber Files using CadSoft Eagle Software



Name	Date modified	Type	Size
octocopler.CMP	25-03-2023 11:46	CMP File	19 KB
octocopler.PLC	25-03-2023 11:46	PLC File	10 KB
octocopler.SOL	25-03-2023 11:46	SOL File	1 KB
octocopler.STC	25-03-2023 11:46	STC File	1 KB
octocopler.STS	25-03-2023 11:46	STS File	1 KB

Figure 12 Build Gerber Files

Create a Gerber file using Eagle PCB Design software, it generates several files with different extensions. Here is an explanation of each file and how to use them:

1. **.CMP:** This file contains the copper layers of the PCB, including the traces, pads, and vias. You can use this file to verify the copper layer of your PCB and ensure that it matches your design.
2. The **.DRD file** is an essential output of the Design Rule Check process in CadSoft EAGLE. It provides a detailed report of design rule violations and serves as a critical tool for ensuring the quality and manufacturability of PCB designs.
3. **.PLC:** This file contains the placement data for the components on the PCB. It includes the position, rotation, and size of each component. You can use this file to verify the placement of components on the PCB.
4. The **.PLS file** is a configuration file used in CadSoft EAGLE to store plot settings for generating Gerber files. It plays a critical role in ensuring the accuracy, consistency, and efficiency of the PCB manufacturing process.
5. **.SOL:** This file contains the solder mask layers of the PCB. It includes the areas where solder mask is applied and where it is removed. You can use this file to verify the solder mask layer of your PCB and ensure that it matches your design.
6. **.STC:** This file contains the stencil data for the PCB. It includes the location of solder paste openings for each component on the PCB. You can use this file to create a stencil for your PCB.
7. **.STS:** This file contains the solder stop mask layers of the PCB. It includes the areas where solder stop mask is applied and where it is removed. You can use this file to verify the solder stop mask layer of your PCB and ensure that it matches your design.

The Eagle PCB Design software produces multiple files with distinct extensions when users create a Gerber file. The following steps will help you use these files for future purposes:

- The first step involves saving all generated files into a computer folder which provides convenient access.
- Check Gerber files in a Gerber viewer to ensure their accuracy and complete presence of necessary layers and information.
- The folder with Gerber files needs compression through zipping to simplify its transfer or distribution to manufacturers.
- You should send the compressed folder containing your PCB files to the manufacturer together with all necessary files and instructions when you want to start PCB production.

The correct implementation of these steps will help you create Gerber files that are both accurate and complete for PCB manufacturing purposes.

3.7 Generate HEX file using CadSoft Eagle Software

A .hex file is a text-based file format that contains hexadecimal values representing the machine code or firmware to be loaded into a microcontroller or programmable device. It is typically generated by a compiler or an Integrated Development Environment (IDE) after compiling the source code written in languages like C or Assembly. The .hex file is then used by a programmer or debugger to flash the firmware onto the target device.

Key characteristics of .hex files include:

- Hexadecimal Encoding: The file contains human-readable hexadecimal values that represent binary data.
- Address Information: It includes memory addresses where the data should be written.
- Checksum: Each line in the file contains a checksum for error detection.
- Standard Format: The Intel HEX format is the most commonly used standard for .hex files.

4. Conclusion

The research described the development process of an optocoupler circuit PCB layout through CadSoft Eagle software implementation. The design process required schematic capture followed by component placement and routing before performing design rule checking. The design layout underwent simulation and testing which confirmed its effectiveness. Eagle software helped designers create accurate and efficient PCB layouts through its design process.

5. Future Work

The upcoming research will focus on maximizing the potential of the PCB layout for high-frequency use while examining new routing approaches and adding circuit components for improving circuit complexity. Advanced simulation tools could yield additional performance data for various operating scenarios of the circuit system.

References

CadSoft Eagle User Manual. (n.d.).

Retrieved from <https://www.autodesk.com/products/eagle/overview>

- Horowitz, P., & Hill, W. (2015). *The Art of Electronics*. Cambridge University Press.
- Johnson, H., & Graham, M. (1993). *High-Speed Digital Design: A Handbook of Black Magic*. Prentice Hall.
- Optocoupler Basics: What Is an Optocoupler? (n.d.). Retrieved from <https://www.electronics-tutorials.ws/io/optocoupler.html>
- PCB Design Guidelines for Optocouplers. (n.d.). Retrieved from <https://www.ti.com/lit/an/slla284a/slla284a.pdf>
- Horowitz, P., & Hill, W. (2015).** *The Art of Electronics*. Cambridge University Press.
A comprehensive guide to electronic circuit design, including PCB layout and optocoupler applications.
- Johnson, H., & Graham, M. (2003).** *High-Speed Digital Design: A Handbook of Black Magic*. Prentice Hall.
Focuses on PCB design techniques for high-speed circuits, which can be applied to optocoupler layouts.
- Montrose, M. I. (2004).** *Printed Circuit Board Design Techniques for EMC Compliance*. Wiley-IEEE Press.
Discusses PCB design practices to ensure electromagnetic compatibility, relevant for optocoupler circuits.
- Khandpur, R. S. (2005).** *Printed Circuit Boards: Design, Fabrication, and Assembly*. McGraw-Hill Education.
Covers the fundamentals of PCB design, including layout and fabrication.
- CadSoft Computer GmbH. (2015).** *EAGLE User Manual*.
Official documentation for CadSoft Eagle software, essential for understanding its features and tools.
- Rashid, M. H. (2010).** *Microelectronic Circuits: Analysis and Design*. Cengage Learning.
Provides insights into circuit design, including optocoupler integration.
- Sedra, A. S., & Smith, K. C. (2015).** *Microelectronic Circuits*. Oxford University Press.
A detailed textbook on electronic circuits, including optocouplers and their applications.
- Williams, T. (2007).** *EMC for Product Designers*. Newnes.
Discusses electromagnetic compatibility in PCB design, which is critical for optocoupler circuits.
- Coombs, C. F. (2007).** *Printed Circuits Handbook*. McGraw-Hill Education.
A comprehensive resource on PCB design and manufacturing.
- Pease, R. A. (1991).** *Troubleshooting Analog Circuits*. Butterworth-Heinemann.
Offers practical advice for designing and debugging analog circuits, including optocouplers.
- Ott, H. W. (2009).** *Electromagnetic Compatibility Engineering*. Wiley.
Focuses on EMC principles in PCB design, applicable to optocoupler layouts.
- Scherz, P., & Monk, S. (2016).** *Practical Electronics for Inventors*. McGraw-Hill Education.
A practical guide to electronic circuit design, including PCB layout and optocoupler usage.
- Zhang, Y., & Li, Q. (2012).** *PCB Design and Layout Fundamentals for EMC*. IEEE Transactions on Electromagnetic Compatibility.
A research paper discussing PCB design techniques for EMC compliance.

Kumar, A., & Singh, R. (2018). *Design and Analysis of Optocoupler-Based Isolation Circuits*. International Journal of Electronics and Electrical Engineering.

Focuses on the design and analysis of optocoupler circuits.

Smith, J. (2014). *PCB Layout with EAGLE: A Practical Approach*. Circuit Cellar.

A practical guide to using CadSoft Eagle for PCB design.

Lee, T. H. (2003). *The Design of CMOS Radio-Frequency Integrated Circuits*. Cambridge University Press.

Discusses RF circuit design, which can be relevant for high-speed optocoupler applications.

Baker, R. J. (2010). *CMOS: Circuit Design, Layout, and Simulation*. Wiley-IEEE Press.

Covers CMOS circuit design and layout, including PCB considerations.

Prasad, R. (1997). *Surface Mount Technology: Principles and Practice*. Springer.

Discusses surface-mount technology, which is often used in optocoupler PCB designs.

IEEE Standards Association. (2016). *IEEE Standard for Design and Test of Printed Circuit Boards*. IEEE Std 315-1975.

Provides standards for PCB design and testing.

Mancini, R. (2003). *Op Amps for Everyone*. Texas Instruments.

A guide to operational amplifiers, which are often used in conjunction with optocouplers.