Introduction to Printed Circuit Board Design and Layout using Ki-CAD

INTRODUCTION TO PRINTED CIRCUIT BOARD DESIGN AND LAYOUT USING KI-CAD

NEHA KARDAM

OpenWA
Olympia, WA

CONTENTS

Acknowledgments 1
Licenses and Permissions 2
About Introduction to Printed Circuit Board Design and Layout iii
About this Pressbook vi

Part I. Printed Circuit Board Fundamentals

Printed Circuit Board Fundamentals 9
PCB Design Considerations: Placement and Routing Techniques 15
Reducing Crosstalk in PCB Routing: Techniques and Best Practices 18
Lecture Video on Introduction to PCB Design 20
Lecture Video on Consideration for design Placement and routing techniques 21

Part II. PCB Terminology and Requirements

Routability Continued: Schematic Fundamentals and Traces 25
IPC Standards for PCB Design and Manufacturing: A Comprehensive Guide 27
Lecture Video on Routability continued Schematic Fundamentals Traces 29
Lecture Video on PCB requirements based on IPC standards 30
Lecture Video on Grounding Techniques 31

Part III. Thermal Management in PCB

Thermal Management in Printed Circuit Boards: Part 1 35

Thermal Management in Printed Circuit Boards: Part 2 38
Short reading and Video Lecture on Thermal Management in PCB 41
Learn to use the milling the PCB 43

Part IV. Design PCB using Ki-CAD Software

Brief Introduction to the LPKF ProtoMat S63 55
KiCad Tutorial: A Comprehensive Guide to Designing PCB Circuits 57
Understanding and designing NRF24 Transceiver Circuit using Ki-CAD 60
Video Lecture 1: Intro to NRF24 circuit design 63
Video Lecture 2: Creating a symbol Part-1 64
Video Lecture 3: Creating a symbol Part-2 65
Video Lecture 4: Associate component to the footprint for connector 01X08 66
Video Lecture 5: Create custom footprint nRF24 67
Video Lecture 6: Saving footprints 68
Video Lecture 7: Footprint placement and 3D view 69
Video Lecture 8: Routing 70
Video Lecture 9: Addtext 71
Video Lecture 10: Updating layout by adding the capacitor 72
Video Lecture 11: Track width control 73
Video Lecture 12: Automatic Trackwidth Calculator 74
Video Lecture 13: Adding Copper Layers 75
Video Lecture 14: Creating Gerber Files 76
Supplement Lecture Video on Annotation and Track Width Calculation using Ki-CAD Software 77

Appendix 79

ACKNOWLEDGMENTS

The Lake Washington Institute of Technology is grateful for the critical support of the National Science Foundation (NSF) Grant #2100136. This grant has been instrumental in propelling our research and development efforts in engineering through the creation of Open Educational Resources (OER) course materials.

I would also like to extend my sincere gratitude to Katherine Kelley and Priyanka Pant. Their expertise and dedication were invaluable in this project. Their encouragement played a key role in motivating me to develop these OER materials and the accompanying pressbook.

LICENSES AND PERMISSIONS

For Further Information

If you have any questions about using the content of this pressbook or require permission beyond the scope of the Creative Commons license, please feel free to contact the author, Neha Kardam (neha.kardam@lwtech.edu) and Katherin Kelley (katherine.kelley@lwtech.edu).

ABOUT INTRODUCTION TO PRINTED CIRCUIT BOARD DESIGN AND LAYOUT

Course Highlights:

In this comprehensive guide to PCB design and layout, students will learn the fundamentals of creating printed circuit boards, including:

- **PCB Design Fundamentals**: Understanding the basics of PCB design, including component placement, routing, and layout.
- **KiCad Tutorial**: A step-by-step guide to using KiCad software for designing and creating PCBs.
- **Component Association**: Learning how to associate components with footprints and create custom footprints.
- **Footprint Creation**: Understanding how to create and save custom footprints for use in PCB design.
- **Routing and Layout**: Mastering the art of routing and layout, including track width control and automatic track width calculation.
- **Copper Layers**: Learning how to add copper layers to a PCB design.
- **Gerber Files**: Understanding how to create and export Gerber files for PCB manufacturing.
- **Thermal Management**: Learning how to manage thermal issues in PCB design.
- **Grounding Techniques**: Understanding how to implement effective grounding techniques in PCB design.
- **IPC Standards**: Learning about IPC standards for PCB design and manufacturing.

Practical Applications:Throughout the course, students will work on practical applications, including:

- **Creating a Symbol**: Learning how to create a symbol for a component using KiCad software.
- **Associating a Component with a Footprint**: Understanding how to associate a component with a footprint.
- **Creating a Custom Footprint**: Learning how to create a custom footprint for a component.
- **Saving and Exporting Designs**: Understanding how to save and export PCB designs for manufacturing.

Learning Outcomes:By the end of this course, students will be able to:

- Design and create printed circuit boards using KiCad software.
- Understand the fundamentals of PCB design, including component placement, routing, and layout.
- Create and save custom footprints for use in PCB design.
- Implement effective grounding techniques in PCB design.
- Manage thermal issues in PCB design.
- Understand IPC standards for PCB design and manufacturing.

Investing in Your Future Success

- By choosing LWTech's "Introduction to Printed Circuit Board Design and Layout" course, you'll gain valuable skills like:
    - **Implementing and managing automated systems**: Learn how to design and implement automated systems using printed circuit boards, including the use of sensors, motors, and other components.
    - **Designing and analyzing PCBs**: Understand the fundamentals of PCB design, including placement, routing techniques, and terminology.
    - **Creating and editing PCB layouts**: Gain hands-on experience with PCB design software, including KiCad, and learn how to create and edit PCB layouts.
    - **Troubleshooting and debugging PCBs**: Develop skills in troubleshooting and debugging PCBs, including identifying and fixing common errors and issues.
    - **Understanding IPC standards and best practices**: Learn about IPC standards and best practices for PCB design and manufacturing, including safety protocols and quality control measures.

Join Us in Shaping the Future

LWTech is dedicated to preparing students with the skills needed to excel in the ever-changing tech environment. Get started today, and discover how our *"Introduction to Printed Circuit Board Design and Layout"* course can empower you to thrive in the automated workforce.

Visit https://www.lwtech.edu/ to learn more about LWTech's *" Introduction to Printed Circuit Board Design and Layout"* course and how it can prepare you for a successful career in the age of automation.

ABOUT THIS PRESSBOOK

This publication, "Introduction to Printed Circuit Board Design and Layout," was developed in Spring 2024 and was created as part of the National Science Foundation Advanced Technological Education grant titled, *Creation and Modernization of Technological Education in Electronics and Welding through Open Educational Resources that are Free to Share, Use, and Revise* (NSF ATE #2100136), awarded to Lake Washington Institute of Technology in 2021. In this three-year grant, three Electronics Technology courses were updated to accessible OER materials with hybrid delivery elements where possible. Faculty Librarian Katherine Kelley, and Dean of Instruction Priyanka Pant led the project. Electronics materials were authored by Electronics Professor, Neha Kardam.

This material is based upon work supported by the National Science Foundation under Grant No. 2100136. Any opinions, findings, and conclusions or recommendations expressed in this material are those of the author(s) and do not necessarily reflect the views of the National Science Foundation.

PART I

PRINTED CIRCUIT BOARD FUNDAMENTALS

PRINTED CIRCUIT BOARD FUNDAMENTALS

Introduction

Printed Circuit Boards (PCBs) are the backbone of modern electronics, providing a platform for mounting and connecting various components to create complex electronic circuits. In this comprehensive guide, we will cover the fundamentals of PCBs, including their definition, basic components, fabrication processes, and applications across various electronics.

What is a Printed Circuit Board?

A Printed Circuit Board (PCB) is a flat, rigid board made of insulating material, such as fiberglass or composite epoxy, with a conductive layer of copper or other metals. The conductive layer is etched to create a pattern of tracks, pads, and vias that connect various components, such as resistors, capacitors, and integrated circuits.

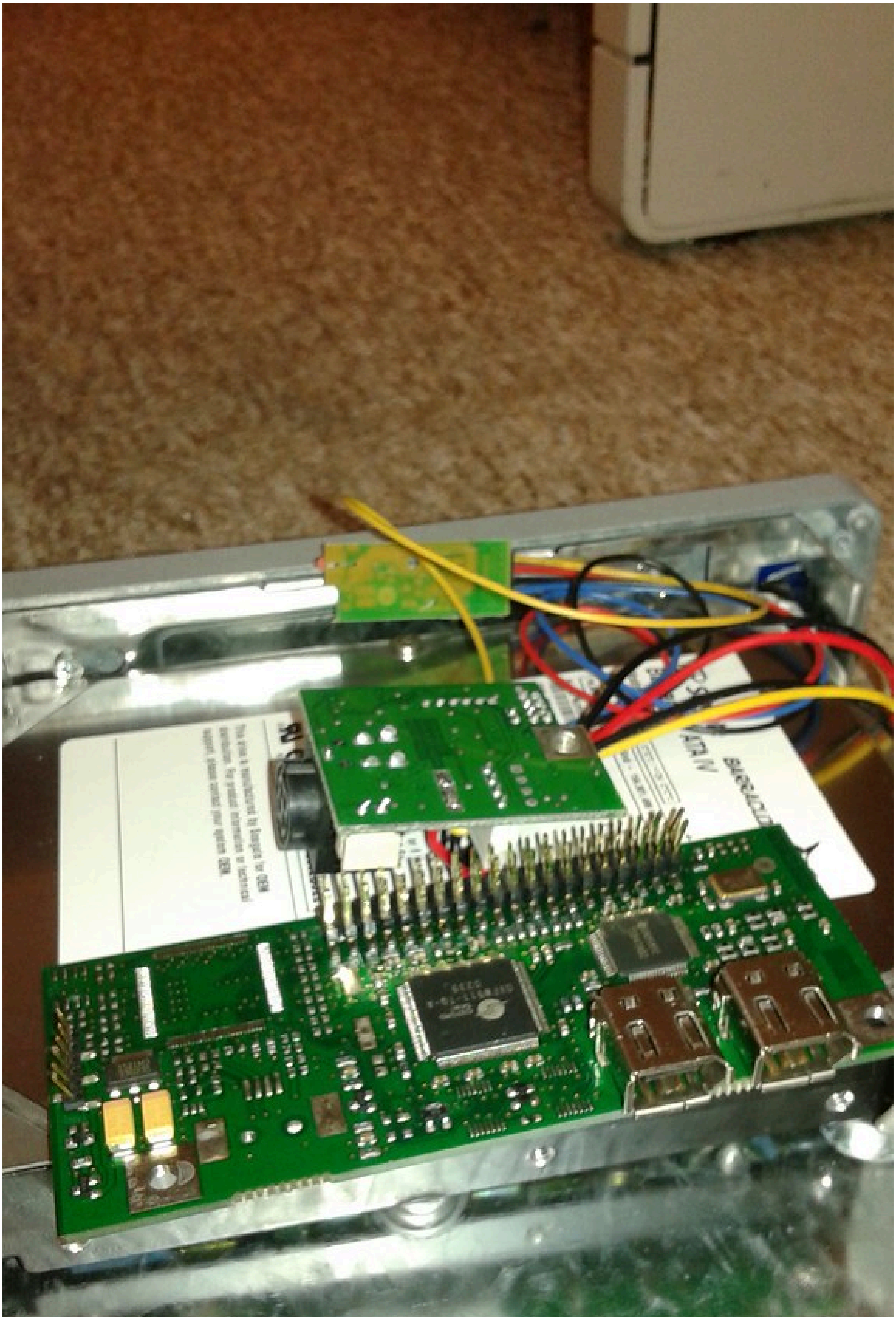


Figure: Printed Circuit Board ("Printed Circuit Board rough shots for logo" by Ken_Mayer is licensed under CC BY 2.0.)

History and Evolution

- **Early Development:** The concept of PCBs was first developed in the early 20th century. Paul Eisler, an Austrian engineer, is credited with developing the first PCB in 1936.
- **Post-War Expansion:** After World War II, PCBs became more widespread, driven by the need for compact and reliable circuits in consumer electronics, military, and aerospace applications.
- **Modern Era:** Today, PCBs are ubiquitous in electronic devices, ranging from simple single-layer boards to complex multi-layer boards with advanced materials and manufacturing techniques.

Basic Components of a PCB

A PCB consists of several basic components, including:

1. **Substrate**: The insulating material that provides the base for the PCB.
2. **Copper Layer**: The conductive layer that carries electrical signals.
3. **Solder Mask**: A layer of polymer that protects the copper layer from oxidation and prevents short circuits.
4. **Silkscreen**: A layer of ink that provides labels and markings for the components.
5. **Components**: The various electronic components, such as resistors, capacitors, and integrated circuits, that are mounted on the PCB.

Types of PCBs

There are several types of PCBs, including:

1. **Single-Layer PCBs**: PCBs with a single layer of copper.
2. **Double-Layer PCBs**: PCBs with two layers of copper.
3. **Multi-Layer PCBs**: PCBs with multiple layers of copper.
4. **Flexible PCBs**: PCBs that can be bent or flexed.
5. **Rigid-Flex PCBs**: PCBs that combine rigid and flexible sections.

PCB Design Process

1. **Schematic Capture:**
    - Create a schematic diagram representing the electronic circuit.
    - Use design software (e.g., KiCad, Eagle) to draw the schematic and define the electrical connections between components.
2. **Component Placement:**
    - Position the components on the PCB layout according to the schematic.
    - Consider factors like signal integrity, thermal management, and physical constraints.
3. **Routing:**
    - Draw the electrical traces connecting the components.
    - Follow design rules to ensure signal integrity, manufacturability, and reliability.
4. **Design Rule Check (DRC):**
    - Perform automated checks to ensure the design adheres to manufacturing and electrical rules.
5. **Gerber File Generation:**
    - Generate Gerber files containing the information needed for PCB fabrication, including copper layers, solder mask, and silkscreen layers.

PCB Fabrication Process

The PCB fabrication process involves several steps, including:

1. **Design**: The PCB design is created using specialized software.
2. **Printing**: The design is printed onto the substrate material.
3. **Etching**: The copper layer is etched to create the pattern of tracks, pads, and vias.
4. **Drilling**: Holes are drilled into the PCB for mounting components.
5. **Plating**: A layer of metal is applied to the PCB to provide a conductive surface.
6. **Solder Mask Application**: The solder mask is applied to the PCB.

7. **Silkscreen Application**: The silkscreen is applied to the PCB.
8. **Component Mounting**: The components are mounted onto the PCB.
9. **Soldering**: The components are soldered onto the PCB.

Applications of PCBs

PCBs are used in a wide range of applications, including:

1. **Consumer Electronics**: PCBs are used in smartphones, laptops, and other consumer electronics.
2. **Industrial Control Systems**: PCBs are used in industrial control systems, such as motor control systems and power distribution systems.
3. **Medical Devices**: PCBs are used in medical devices, such as pacemakers and X-ray machines.
4. **Aerospace and Defense**: PCBs are used in aerospace and defense applications, such as satellite systems and missile guidance systems.
5. **Automotive Systems**: PCBs are used in automotive systems, such as engine control systems and infotainment systems.

Safety and Best Practices

- **ESD Protection:** Use anti-static wrist straps and mats to prevent electrostatic discharge damage to sensitive components.
- **Proper Handling:** Handle PCBs with care to avoid physical damage and contamination.
- **Documentation:** Maintain detailed documentation of the design and manufacturing process for troubleshooting and future reference.

Conclusion

In conclusion, PCBs are a fundamental component of modern electronics, providing a platform for mounting and connecting various components to create complex electronic circuits. Understanding the basics of PCBs, including their definition, components, fabrication process, and applications, is essential for anyone working in the field of electronics.

References

1. MCL. (n.d.). An Ultimate Guide To The PCB Manufacturing Process. Retrieved

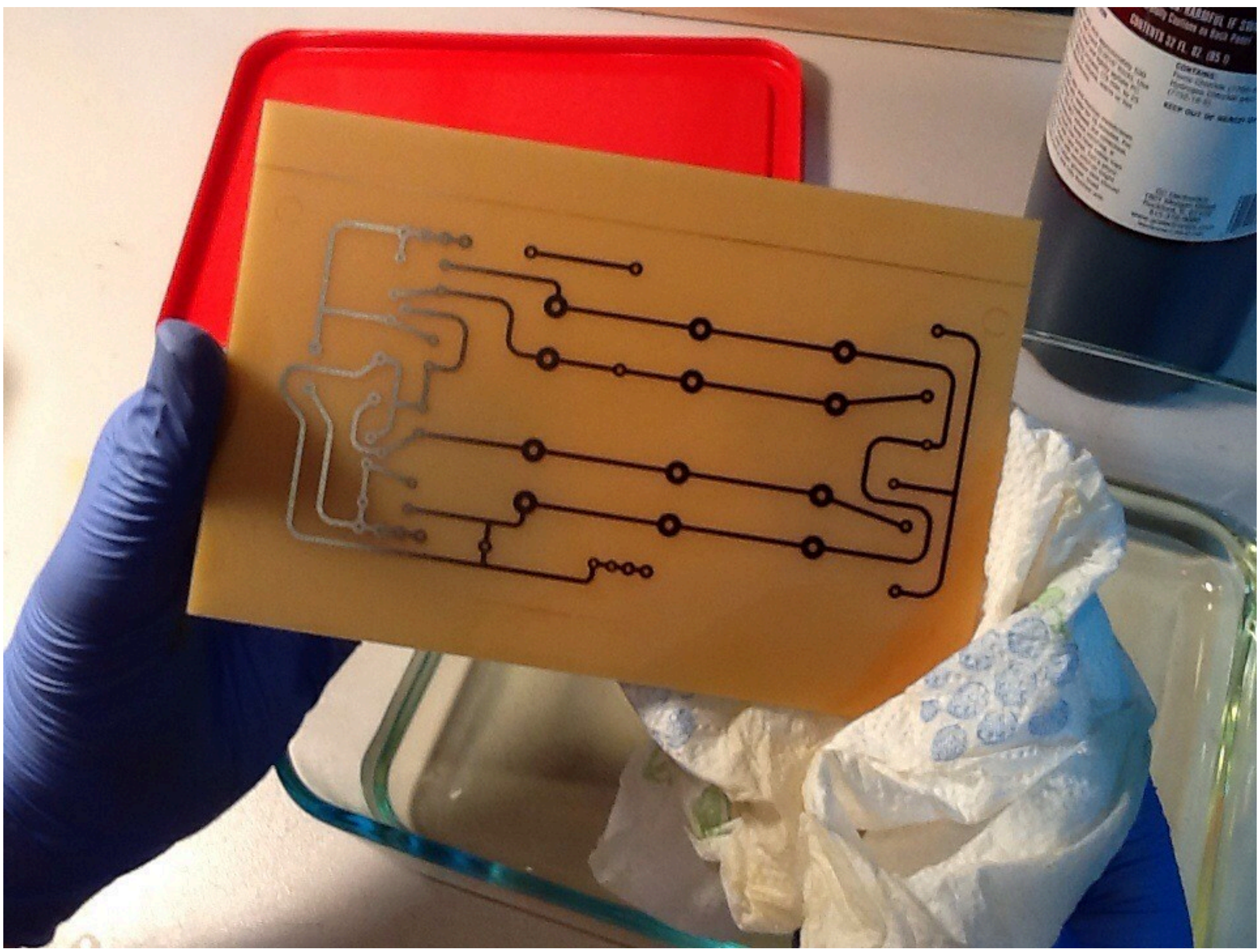
from https://www.mclpcb.com/blog/pcb-manufacturing-process/

2. PCBCart. (n.d.). Introduction to PCB and Different Types of Circuit Boards. Retrieved from https://www.pcbcart.com/article/content/PCB-introduction.html
3. Vector Blue Hub. (n.d.). A Beginner's Guide to Understanding Basic Circuit Board Components. Retrieved from https://vectorbluehub.com/circuit-board-components
4. TechTarget. (n.d.). What is printed circuit board (PCB)? Retrieved from https://www.techtarget.com/whatis/definition/printed-circuit-board-PCB
5. Wikipedia. (n.d.). Printed circuit board. Retrieved from https://en.wikipedia.org/wiki/Printed_circuit_board

PCB DESIGN CONSIDERATIONS: PLACEMENT AND ROUTING TECHNIQUES

Introduction

Printed Circuit Boards (PCBs) are a crucial component of modern electronics, and their design plays a significant role in determining the overall performance and reliability of a system. In this lecture, we will explore the key considerations for PCB design, focusing on placement and routing techniques.



Placement Techniques

Component placement is a critical aspect of PCB design, as it directly affects the performance, reliability, and manufacturability of the board. Here are some key placement techniques to consider:

1. **Component Placement Guidelines**: Components should be placed in a way that minimizes signal interference, thermal issues, and difficulties in assembly and testing.
2. **Floor Planning**: The floor plan of the PCB should be carefully designed to ensure that components are placed in a logical and efficient manner.
3. **Component Placement Order**: Components should be placed in a specific order, starting with the most critical components, such as connectors and main functional chips.

Routing Techniques

Routing is another critical aspect of PCB design, as it directly affects the performance and reliability of the board. Here are some key routing techniques to consider:

1. **Autorouting**: Autorouters can be used to save time and effort in routing, but they should not be relied upon entirely.
2. **Manual Routing**: Manual routing is often necessary to ensure that the routing is done correctly and efficiently.
3. **Trace Width and Spacing**: Trace width and spacing are critical factors in determining the performance and reliability of the board.
4. **Ground and Power Traces**: Ground and power traces should be carefully designed to ensure that they are properly connected and do not interfere with other signals.
5. **Thermal Vias and Pads**: Thermal vias and pads should be used to dissipate heat and ensure that the board operates within a safe temperature range.

Best Practices

Here are some best practices to keep in mind when designing a PCB:

1. **Avoid 90° Angles**: 90° angles should be avoided in routing, as they can cause signal reflections and other issues.
2. **Use Ground and Power Planes**: Ground and power planes should be used to provide a solid reference point for measuring voltages and to reduce noise immunity.

3. **Keep Enough Space between Traces**: Enough space should be kept between traces and pads to avoid short circuits and other issues.
4. **Alternate Trace Direction**: Trace direction should be alternated to reduce crosstalk and other signal integrity issues.

References

1. MCL. (n.d.). Top 10 Best Practices for PCB Routing. Retrieved from https://www.mclpcb.com/blog/pcb-routing-best-practices/
2. Proto-Electronics. (n.d.). Our Top 10 PCB Routing Tips. Retrieved from https://www.proto-electronics.com/blog/top-10-pcb-routing-tips
3. Candor Industries. (n.d.). PCB Design Considerations: Placement and Routing Techniques. Retrieved from https://www.candorind.com/pcb-design-considerations-placement-routing-techniques/
4. Sierra Circuits. (n.d.). Component Placement in PCB Design & Assembly. Retrieved from https://www.protoexpress.com/blog/component-placement-guidelines-pcb-design-assembly/
5. Cadence. (n.d.). Basic PCB Component Placement Guidelines. Retrieved from https://resources.pcb.cadence.com/blog/2022-basic-pcb-component-placement-guidelines

REDUCING CROSSTALK IN PCB ROUTING: TECHNIQUES AND BEST PRACTICES

Crosstalk is a significant issue in high-speed PCB design, and reducing it is important for maintaining signal integrity. In this summary, we will discuss the best techniques for reducing crosstalk in PCB routing, including increasing trace spacing, using ground planes, employing differential signaling, and maintaining a proper reference plane.

1. Increase Trace Spacing

Increasing the spacing between traces is an effective way to reduce crosstalk. A general rule of thumb is to maintain a minimum spacing of 3W (where W is the width of the trace) between signal lines. This can be achieved by using a wider trace width or by increasing the spacing between traces.

2. Use Ground Planes

Ground planes are an essential component in reducing crosstalk. By placing a ground plane between signal layers, you can reduce the coupling between traces and minimize crosstalk. It is recommended to place the ground plane 1 dielectric away from signal and power planes.

3. Employ Differential Signaling

Differential signaling is a technique that uses two complementary signals to transmit data. This technique can help reduce crosstalk by canceling out noise and interference.

4. Maintain a Proper Reference Plane

Maintaining a proper reference plane is crucial for reducing crosstalk. A reference plane provides a return path for signals and helps to minimize coupling between traces.

5. Use Guard Traces

Guard traces are used to isolate high-frequency signals from other traces. By placing a guard trace between a high-frequency signal and other traces, you can reduce crosstalk.

6. Reduce Parallelism

Reducing parallelism between traces can help minimize crosstalk. By using a 45-degree angle or curved traces, you can reduce the parallelism between traces.

7. Isolate High-Frequency Signals

Isolating high-frequency signals from other traces can help reduce crosstalk. By placing high-frequency signals on a separate layer or using a guard trace, you can minimize crosstalk.

References

1. NextPCB. (n.d.). Minimizing Ways of Crosstalk in PCB. Retrieved from https://www.nextpcb.com/blog/pcb-crosstalk
2. Cadence. (n.d.). How to Reduce Crosstalk in Your PCB Layout. Retrieved from https://resources.pcb.cadence.com/blog/2019-how-to-reduce-crosstalk-in-your-pcb-layout
3. Altium. (n.d.). Introduction to High-Speed PCB Designing: Techniques for Avoiding Crosstalk. Retrieved from https://resources.altium.com/p/introduction-high-speed-pcb-designing-how-eliminate-crosstalk
4. Proto-Electronics. (n.d.). The best crosstalk reduction techniques. Retrieved from https://www.proto-electronics.com/blog/best-crosstalk-reduction-techniques
5. Sierra Circuits. (n.d.). Handling Crosstalk in High-Speed PCB Design. Retrieved from https://www.protoexpress.com/blog/crosstalk-high-speed-pcb-design/

LECTURE VIDEO ON INTRODUCTION TO PCB DESIGN

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=5#oembed-1

LECTURE VIDEO ON CONSIDERATION FOR DESIGN PLACEMENT AND ROUTING TECHNIQUES

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=165#oembed-1

PART II

PCB TERMINOLOGY AND REQUIREMENTS

ROUTABILITY CONTINUED: SCHEMATIC FUNDAMENTALS AND TRACES

Introduction

In our previous lecture, we discussed the importance of routability in PCB design and the various factors that affect it. In this lecture, we will continue to explore the topic of routability, focusing on schematic fundamentals and traces.

Schematic Fundamentals

A schematic is a graphical representation of a circuit, showing the connections between components. A well-designed schematic is essential for ensuring the routability of a PCB. Here are some key considerations for schematic design:

- **Component placement**: Components should be placed in a logical and efficient manner, taking into account their function and connectivity.
- **Net naming**: Nets should be named clearly and consistently, making it easy to identify and connect components.
- **Component values**: Component values should be clearly labeled, making it easy to identify and verify the correct components.

Traces

Traces are the physical connections between components on a PCB. They are the "wires" that carry signals and power between components. Here are some key considerations for trace design:

- **Trace width**: The width of a trace determines its current-carrying capacity and impedance. Wider traces can carry more current, but may also increase the risk of crosstalk and noise.
- **Trace spacing**: The spacing between traces determines the risk of crosstalk and noise. Closer spacing can increase the risk of crosstalk, while wider spacing can reduce it.
- **Trace length**: The length of a trace determines its impedance and signal delay. Longer traces can increase signal delay and impedance, while shorter traces can reduce them.

Best Practices for Trace Design

Here are some best practices for trace design:

- **Use a consistent trace width**: Using a consistent trace width throughout the design can help to reduce errors and improve routability.
- **Use a consistent trace spacing**: Using a consistent trace spacing throughout the design can help to reduce crosstalk and noise.
- **Minimize trace length**: Minimizing trace length can help to reduce signal delay and impedance.
- **Avoid sharp angles**: Avoiding sharp angles in trace design can help to reduce the risk of crosstalk and noise.

Conclusion

In conclusion, schematic fundamentals and trace design are critical components of routability in PCB design. By following best practices for schematic design and trace design, designers can improve the routability of their PCBs and reduce the risk of errors and failures.

References

- Wevolver. (n.d.). PCB Trace: The Backbone of Modern Circuit Design. Retrieved from https://www.wevolver.com/article/trace-pcb-a-comprehensive-guide
- MokoTechnology. (n.d.). The Essential Guide to PCB Traces: Understanding the Basics. Retrieved from https://www.mokotechnology.com/pcb-traces/
- Rowsum. (n.d.). Transform Your PCB Design with These 31 Essential Routing Tips. Retrieved from https://www.rowsum.com/31-pcb-routing-tips/
- Proto-Electronics. (n.d.). Our Top 10 PCB Routing Tips. Retrieved from https://www.proto-electronics.com/blog/top-10-pcb-routing-tips
- Cadence. (n.d.). PCB Routing Essentials for the Modern Designer. Retrieved from https://resources.pcb.cadence.com/blog/pcb-routing-essentials-for-the-modern-designer

IPC STANDARDS FOR PCB DESIGN AND MANUFACTURING: A COMPREHENSIVE GUIDE

Introduction

The Institute for Printed Circuits (IPC) is a leading authority on PCB design and manufacturing standards. IPC standards provide a framework for ensuring quality, reliability, and consistency in the design and manufacturing of printed circuit boards (PCBs). In this comprehensive guide, we will explore the importance of IPC standards, the different types of IPC standards, and the benefits of complying with these standards.

Why IPC Standards Matter

IPC standards are essential for ensuring that PCBs meet the required quality, reliability, and performance standards. By complying with IPC standards, designers and manufacturers can ensure that their PCBs are:

- **Reliable**: IPC standards help ensure that PCBs are designed and manufactured to withstand the rigors of use and environmental conditions.
- **Consistent**: IPC standards provide a framework for consistency in design and manufacturing, reducing the risk of errors and variability.
- **High-quality**: IPC standards promote high-quality design and manufacturing practices, resulting in PCBs that meet or exceed customer expectations.

Types of IPC Standards

There are several types of IPC standards, including:

- **IPC-A-600**: Acceptability of Printed Boards
- **IPC-6012**: Qualification and Performance Specification for Rigid Printed Boards
- **IPC-A-610**: Acceptability of Electronic Assemblies
- **IPC-7711/7721**: Rework, Modification, and Repair of Electronic Assemblies
- **IPC-2581**: Generic Requirements for Printed Board Assembly Products Manufacturing Description

Data and Transfer Methodology

Benefits of Complying with IPC Standards

Complying with IPC standards offers numerous benefits, including:

- **Improved quality and reliability**: IPC standards help ensure that PCBs meet the required quality and reliability standards.
- **Increased efficiency**: IPC standards promote efficient design and manufacturing practices, reducing the risk of errors and variability.
- **Reduced costs**: Complying with IPC standards can help reduce costs associated with rework, repair, and replacement.
- **Enhanced customer satisfaction**: IPC standards promote high-quality design and manufacturing practices, resulting in PCBs that meet or exceed customer expectations.

Conclusion

IPC standards are essential for ensuring quality, reliability, and consistency in the design and manufacturing of PCBs. By complying with IPC standards, designers and manufacturers can ensure that their PCBs meet the required quality and reliability standards, while also improving efficiency, reducing costs, and enhancing customer satisfaction.

References

- Highleap Electronic. (n.d.). IPC Standards for PCB: Why is it Important？ Retrieved from https://hilelectronic.com/ipc-standards-for-pcb/
- Creative Hi-Tech. (n.d.). A Brief On IPC Standards of PCB. Retrieved from https://www.creativehitech.com/blog/ipc-standards-of-pcb-introduction-and-significance-in-quality-pcb-manufacturing/
- Altium. (n.d.). IPC Classes and Complying with IPC Standards for PCB Design. Retrieved from https://resources.altium.com/p/complying-with-ipc-standards-for-pcb-design
- PCBElec. (n.d.). IPC Standards: A Guide to Standards for PCB Manufacturing and Assembly. Retrieved from https://www.pcbelec.com/ipc-standards-for-pcb-manufacturing-and-assembly.html
- Electronic Design. (n.d.). IPC Standards for PCBs: What Are They and Why Do They Matter? Retrieved from https://www.electronicdesign.com/technologies/embedded/article/21216532/mer-mar-electronics-ipc-standards-for-pcbs-what-are-they-and-why-do-they-matter

LECTURE VIDEO ON ROUTABILITY CONTINUED SCHEMATIC FUNDAMENTALS TRACES

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=150#oembed-1

LECTURE VIDEO ON PCB REQUIREMENTS BASED ON IPC STANDARDS

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=152#oembed-1

LECTURE VIDEO ON GROUNDING TECHNIQUES

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=154#oembed-1

PART III
THERMAL MANAGEMENT IN PCB

THERMAL MANAGEMENT IN PRINTED CIRCUIT BOARDS: PART 1

Introduction

Thermal management is a critical aspect of printed circuit board (PCB) design, as it directly affects the performance, reliability, and lifespan of electronic components. With the increasing trend of miniaturization and high-power density in modern electronics, thermal management has become a significant challenge. In this two-part reading material, we will explore the fundamentals of thermal management in PCBs, its importance, and various techniques to manage heat effectively.

Why Thermal Management is Important

Thermal management is essential in PCB design because excessive heat can cause:

- Component failure or degradation
- Reduced system performance and reliability
- Increased risk of thermal runaway
- Decreased lifespan of components and the overall system

Heat Generation in PCBs

Heat is generated in PCBs due to various sources, including:

- Power dissipation from components (e.g., CPUs, GPUs, and power transistors)
- Resistance in electrical connections and copper traces
- Switching losses in digital circuits
- Ambient temperature and environmental conditions

Thermal Conductivity and Thermal Resistance

Thermal conductivity (K) is a measure of a material's ability to conduct heat. Thermal resistance (θ) is the opposition to heat flow between two points. Understanding these concepts is crucial in designing effective thermal management systems.

PCB Materials and Thermal Conductivity

Common PCB materials have varying thermal conductivity values:

- FR4 (0.2-0.5 W/mK)
- Copper (386 W/mK)
- Aluminum (237 W/mK)

Thermal Management Techniques

Several techniques can be employed to manage heat in PCBs, including:

- Heat sinks and thermal interfaces
- Thermal vias and plugged vias
- Thick copper and metal core PCBs
- Cooling fans and air flow management
- Advanced thermal management solutions (e.g., cavities and copper-filled thermal vias)

Conclusion

In this first part of the reading material, we have introduced the importance of thermal management in PCB design, heat generation sources, and thermal conductivity and resistance concepts. We have also touched on various thermal management techniques. In the second part, we will delve deeper into these techniques and explore advanced solutions for effective thermal management in PCBs.

References

1. Multi-Circuit-Boards. (n.d.). Thermal Management for Printed Circuit Boards. Retrieved from https://www.multi-circuit-boards.eu/en/support/articles/thermal-management.html
2. Altium. (n.d.). PCB Thermal Conductivity. Retrieved from https://resources.altium.com/p/pcb-thermal-conductivity
3. Millennium Circuits Limited. (n.d.). 5 PCB Thermal Management Techniques. Retrieved

from https://www.mclpcb.com/blog/thermal-pcb-management-techniques/

4. Please note that this is the first part of the reading material, and the second part will be provided separately.

THERMAL MANAGEMENT IN PRINTED CIRCUIT BOARDS: PART 2

Advanced Thermal Management Solutions

In addition to the techniques discussed in Part 1, several advanced thermal management solutions can be employed to manage heat in PCBs. These include:

1. Metal-Core PCBs

Metal-core PCBs are designed with a metal core, typically aluminum or copper, which provides excellent thermal conductivity. This allows for efficient heat dissipation and can be used in high-power applications.

2. Advanced Materials

New materials with improved thermal conductivity are being developed for use in PCBs. These include thermally conductive dielectrics, which can be used to create high-performance PCBs with improved thermal management capabilities.

3. Heat Pipe Technology

Heat pipes are sealed tubes filled with a working fluid that vaporizes at one end and condenses at the other, transferring heat efficiently. This technology can be used in PCBs to manage heat in high-power applications.

4. Simulations for Thermal Analysis

Simulations can be used to analyze the thermal performance of a PCB design before it is manufactured. This allows designers to identify potential thermal issues and make changes to the design to improve thermal management.

Strategies for Effective Thermal Management

To achieve effective thermal management in PCBs, several strategies can be employed:

1. Component Placement

Careful placement of components on the PCB can help to improve heat dissipation. Components that generate a lot of heat should be placed in areas with good airflow, and thermal pathways should be designed to direct heat away from critical components.

2. Thermal Vias

Thermal vias are small holes drilled through the PCB that allow heat to be transferred from one layer to another. These can be used to improve heat dissipation in high-power applications.

3. Heat Sinks

Heat sinks are components that are designed to absorb and dissipate heat. These can be used in conjunction with thermal vias to improve heat dissipation in high-power applications.

4. Advanced Cooling Systems

In some cases, advanced cooling systems such as liquid cooling or forced air cooling may be necessary to manage heat in high-power applications.

References

1. ProtoExpress. (n.d.). 12 PCB Thermal Management Techniques to Reduce Heating. Retrieved from https://www.protoexpress.com/blog/12-pcb-thermal-management-techniques-to-reduce-pcb-heating/
2. All About Circuits. (n.d.). PCB Thermal Management Techniques. Retrieved from https://www.allaboutcircuits.com/technical-articles/pcb-thermal-management-techniques/
3. IPC. (n.d.). Advanced Thermal Management Solutions on PCBs for High Power Applications. Retrieved from https://www.ipc.org/system/files/technical_resource/E15%26S02_03%20-%20Best%20International%20Paper.pdf

4. PGF Technology Group. (n.d.). Innovations in Thermal Management for High-Performance PCBs. Retrieved from https://www.pgftech.com/innovations-in-thermal-management-for-high-performance-pcbs/
5. Sierra Assembly. (n.d.). PCB Thermal Management Techniques: Ensuring Reliability in High-Performance Applications. Retrieved from https://www.sierraassembly.com/blog/pcb-thermal-management-techniques/

SHORT READING AND VIDEO LECTURE ON THERMAL MANAGEMENT IN PCB

Thermal management is an important aspect of PCB design and manufacturing. Proper thermal management ensures that the PCB operates within the specified temperature range and minimizes the risk of damage to the components. In this reading material, we will cover the PCB requirements related to thermal management, including conduction, convention, radiation, board heat dissipation, board heat sink design, assembly of heatsink to boards, film type adhesives, special design considerations for SMT board heatsinks, PCB thermal pads, and thermal paste.

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=106#oembed-1

1. ***Thermal Management:*** Thermal management is the process of managing the temperature of the PCB. It involves controlling the heat generated by the components and dissipating it to the environment. There are three modes of heat transfer – conduction, convention, and radiation – that need to be considered in PCB design.
2. ***Conduction:*** Conduction is the transfer of heat through a material by direct contact. In PCBs, heat is transferred from the components to the copper traces and planes, and then to the PCB substrate. To improve conduction, the copper thickness and width should be optimized to provide maximum thermal conductivity.
3. ***Convention:*** Convention is the transfer of heat through a fluid or gas. In PCBs, convention is achieved through the use of heat sinks, fans, and vents. The design of the PCB should allow for proper airflow to ensure efficient convention.
4. ***Radiation:*** Radiation is the transfer of heat through electromagnetic waves. In PCBs, radiation can be minimized by using materials with low emissivity, such as copper or aluminum.
5. ***Board Heat Dissipation***: Board heat dissipation is the process of dissipating the heat generated by the PCB to the environment. This can be achieved through the use of heat sinks, fans, and vents.
6. ***Board Heat Sink Design:*** Board heat sink design involves the selection and placement of heat sinks on the PCB. The heat sink should be designed to maximize surface area and provide adequate heat dissipation.

7. ***Assembly of Heatsink to Boards:*** The assembly of heatsinks to boards involves the use of film type adhesives to bond the heat sink to the PCB. The adhesive should be selected based on the thermal conductivity and the operating temperature range.
8. ***Film Type Adhesives:*** Film type adhesives are used to bond the heat sink to the PCB. They are available in different thermal conductivities and operating temperature ranges. The adhesive should be selected based on the requirements of the PCB design.
9. ***Special Design Considerations for SMT Board Heatsinks:*** Surface Mount Technology (SMT) board heatsinks require special design considerations to ensure proper thermal management. The heatsink should be designed to fit within the SMT footprint and not interfere with other components.
10. ***PCB Thermal Pads:*** PCB thermal pads are used to improve the thermal conductivity between the components and the PCB substrate. They are available in different thicknesses and thermal conductivities and should be selected based on the requirements of the PCB design.
11. ***Thermal Paste:*** Thermal paste is used to improve the thermal conductivity between the components and the heat sink. It should be applied in a thin, even layer to ensure proper heat transfer.

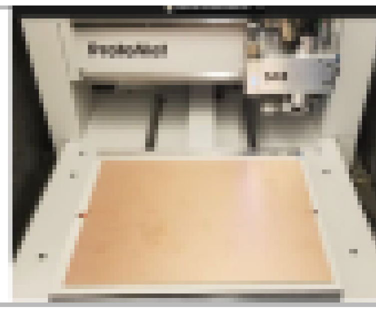
In conclusion, thermal management is an important aspect of PCB design and manufacturing. Conduction, convention, radiation, board heat dissipation, board heat sink design, assembly of heatsink to boards, film type adhesives, special design considerations for SMT board heatsinks, PCB thermal pads, and thermal paste are some of the important factors that need to be considered in PCB design and manufacturing. With the knowledge of these factors, students taking the printed circuit board design course online can design and manufacture functional and reliable PCBs that meet industry standards.

References:

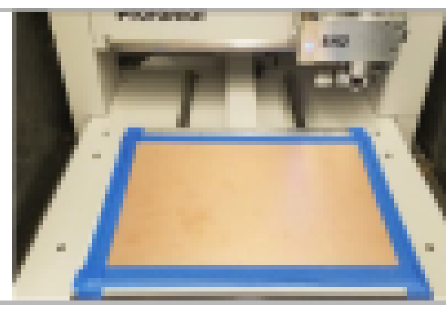
[1] Robertson, C. T. (2004). Printed circuit board designer's reference: basics. Prentice Hall Professional.

LEARN TO USE THE MILLING THE PCB

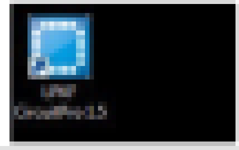
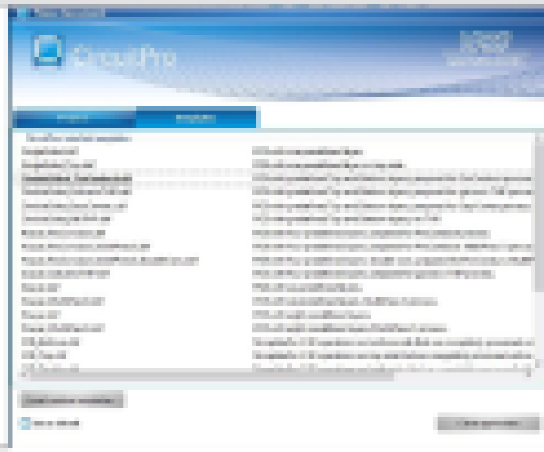
Board Setup

Open the S63 Protomat enclosure and place the 'Underlay Material' on the LPKF table attached to the S43 Protomat. If the 'Underlay Material' appears to be curved, flatten it out by affixing small amounts of paper Note: make sure not to lose the pins situated at the ends when removing the underlay material	
Select a Copper cladded FR4 board **9" x 12"** and place it on top of the underlay material. Make sure its properly aligned with the underlay material.	

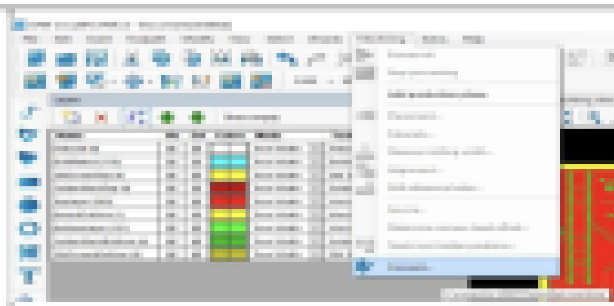
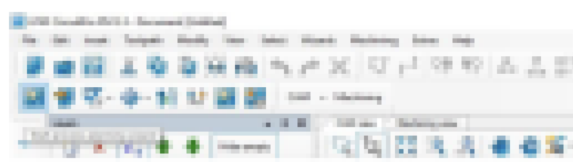
Board Setup..

Align the copper-clad board along the edges and use strips of paper tape to affix the board firmly onto the table	
Switch on the S63 Protomat and pull down the enclosure lid.	

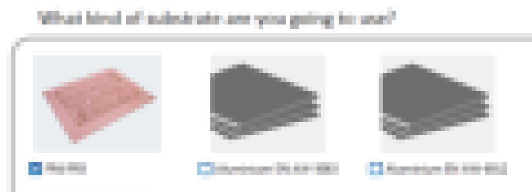
Circuit Pro Setup

Find LPKF's Circuit Pro application shortcut and launch it .	
Select FILE then New Document and choose the configurations template for the current project. Since we are milling a two-layer board, choose the "DoubleSided_NoTHP.cbf"	

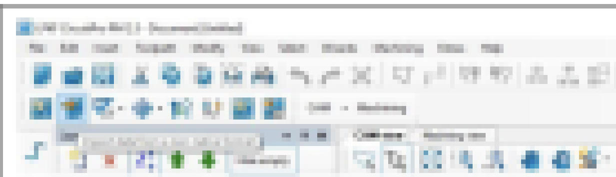
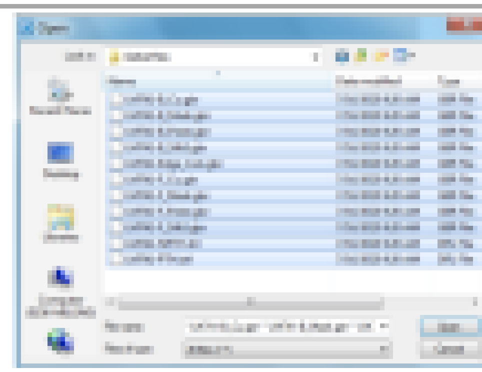
Circuit Pro Setup...

Navigate to 'Machining' option on the main toolbar and select 'Connect'. The first time will create an error when trying to connect ignore it and retry connecting.	
Initiate the "Process Planning Wizard" by selecting the respective icon from the wizard's toolbar within CircuitPro	

Circuit Pro Setup…

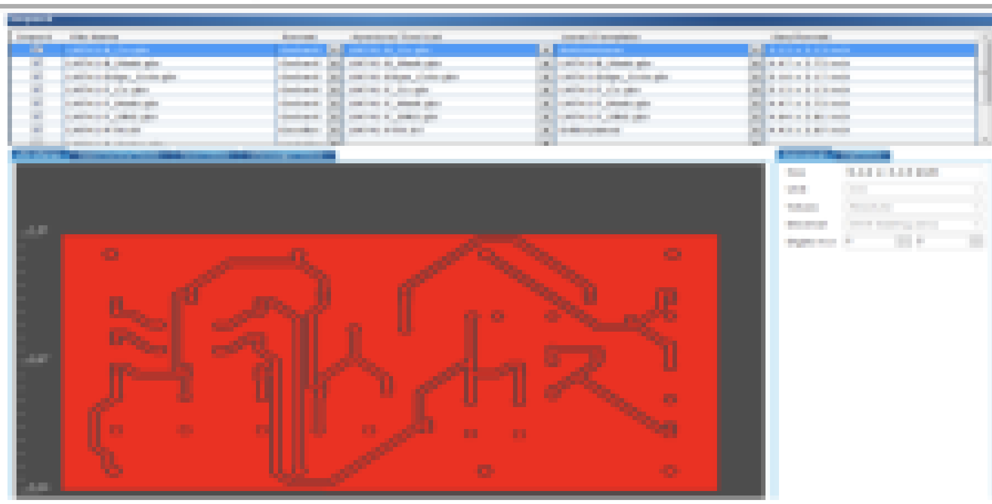
Checkout 'Process PCBs' from available choices and click on 'Next' to proceed. Checkout 'Double-sided' from available choices for the number of layers and click on 'Next' to proceed.	
Checkout 'FR4/FR5' from available choices for substrates and click on 'Next' to proceed. Then press Done	

Importing Gerber Files

Initiate the 'Import Wizard' by selecting the respective icon from the wizard's toolbar within CircuitPro. We will import the Gerber and drill files prepared prior to the milling process.	
Use the file selection wizard to browse to the location of your Gerber files and Control-click and drag to select all the layers and drill files. Click on 'Open' to proceed.	

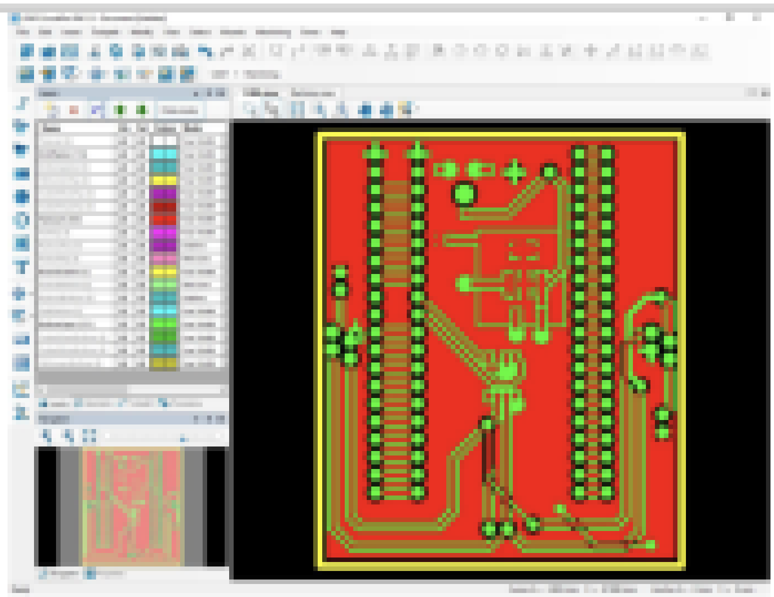
Importing Gerber Files...

Choose the appropriate label for each imported Gerber file from the drop-down list under "Layer/Template". For example, the bottom layer Gerber file will be assigned the label "BottomLayer", Outline Gerber file will be assigned "Boardoutline".

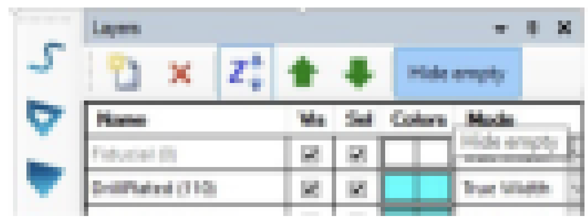


Importing Gerber Files...

Once the Gerber file import process is complete, click on the "OK" button to close the panel and return back to the main window. The layout for the design will be displayed in the "CAM View" panel. On the left, a listing on all the layers imported will be presented. The number of CAM routes will be displayed between the brackets beside each layer.

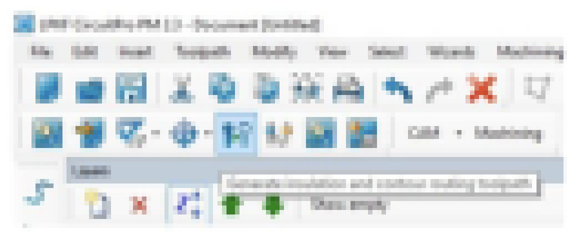
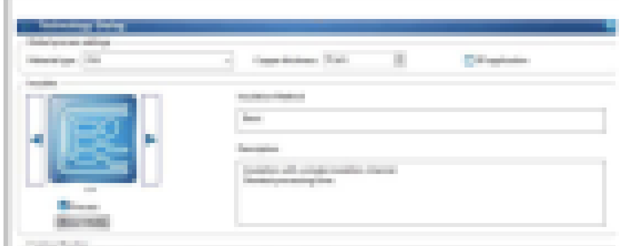


Importing Gerber Files…

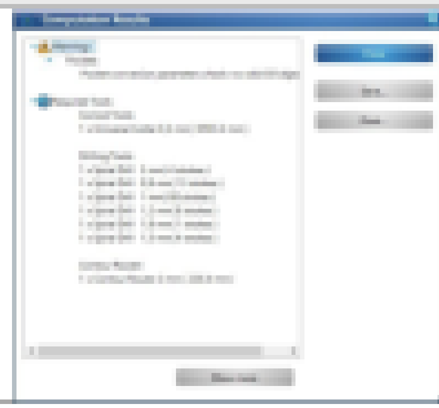
Click on "Hide Empty" to have a compact listing of only the necessary layers imported in the previous step.	

Save your project by selecting File > Save As
This will allow you to pick up where you left off, if you need to restart CircuitPro for any reason.
Saving is not mandatory, but if you do not save the project before beginning the wizards you will receve frequent prompts asking if you would like to save throughout the milling process.

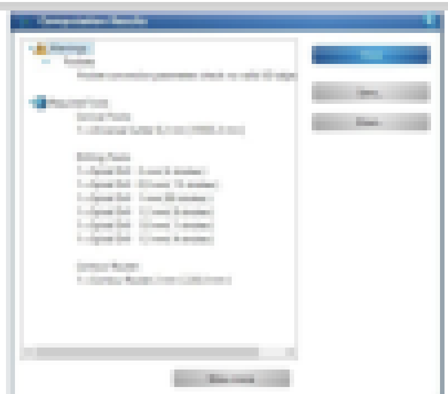
Generating CAM Toolpaths

Before we start the milling process, the routing toolpaths for insulation and contour of the imported design must be generated. The 'create insulation and contour routing toolpaths' wizard may be initiated by selecting the appropriate icon, as shown, from the wizard's toolbar.	
It is recommended to select the first option to save the life of the drill bits and to save time.	

Generating CAM Toolpaths…

This option is used for clipping off the design from the base copper plate. No gaps is recommended or "Horizontal Gaps", "Vertical Gaps". Select an option based on size of your project.	
Once the toolpaths have been generated, a summary of the Computation results are displayed. The required tools (drill bits) for the job is summarized. You may safely ignore the 'Pocket conversion check' warning. Click on 'Close' to exit the summary dialog.	

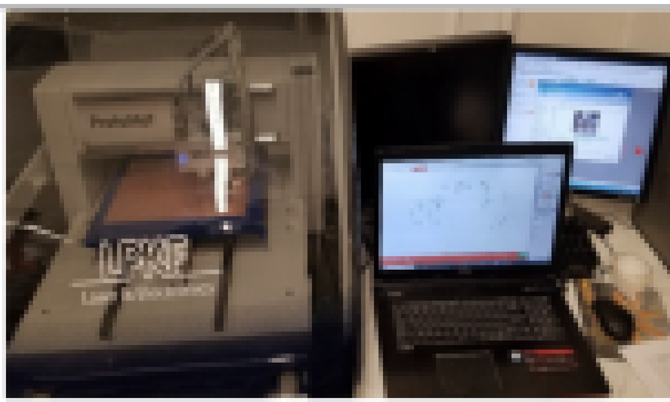
Generating CAM Toolpaths…

This option is used for clipping off the design from the base copper plate. No gaps is recommended or "Horizontal Gaps", "Vertical Gaps". Select an option based on size of your project.	
Once the toolpaths have been generated, a summary of the Computation results are displayed. The required tools (drill bits) for the job is summarized. You may safely ignore the 'Pocket conversion check' warning. Click on 'Close' to exit the summary dialog.	

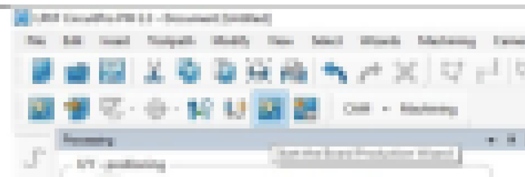
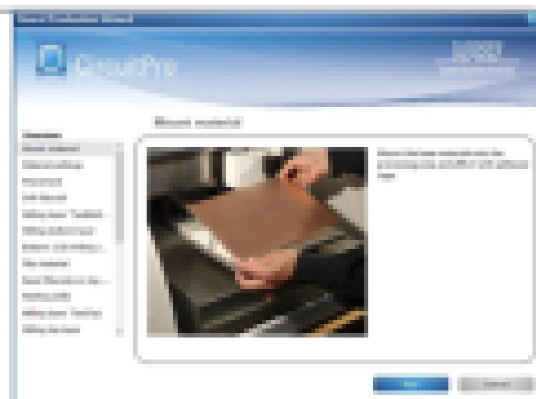
Board Production

Ensure that the copper clad board has been affixed firmly to the tabletop and the enclosure lid is in the closed position. Turn on the vacuum system

Note: When turning on the vacuum, flip the switch toward the front of the machine. This will set the vacuum to auto, which only operates the vacuum while the milling machine is in use.



Board Production ...

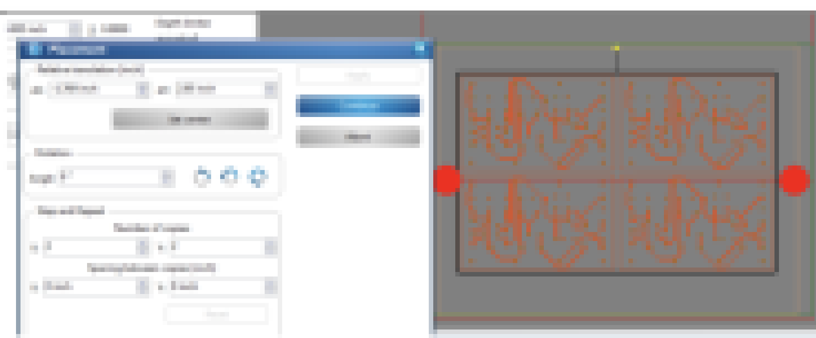
Initiate the 'Board Production' wizard by clicking on the appropriate icon as illustrated.	
This step was cover previously but if not make sure to mount the material and secure it properly. Then proceed by clicking on "Start".	

Board Production ...

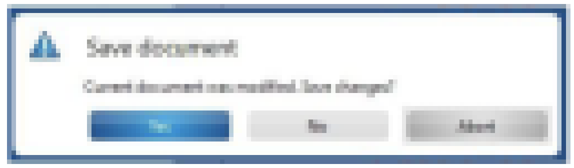
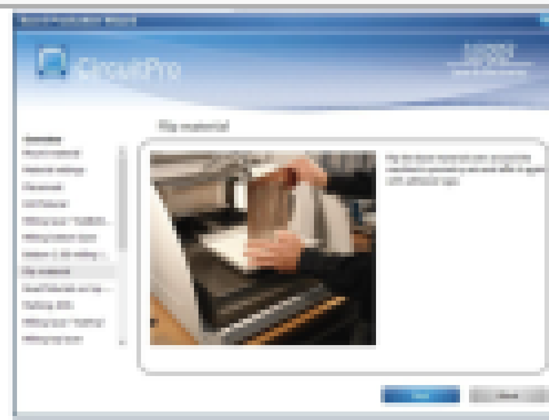
Click on 'Continue' to accept the configured settings and proceed. Make sure to use FR4 and cut to the appropriate size

Board Production ...

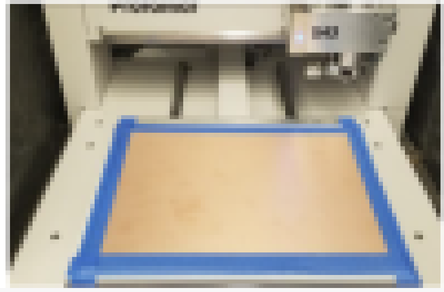
The placement dialog box (Which can be manually access from the 'Machining' option in the main toolbar), allows for relative positioning (based on an anchor point, rotation as well as creating multiple copies of the design along both the design's vertical and horizontal dimensions. You can also use the mouse to grab the design (along the black outline) within the 'Machining View' and move it to a desired milling location on the board.
It is advisable to preserve as much space possible on the copper board all the while maintaining a distance from the board extremities. Once you have chosen the preferred location on the copper board, click on the 'Continue' button.



Board Production …

CircuitPro offers the option to save the current configuration which comprises of the configuration settings, template, and the design. If you wish to save a copy of the same, you may do so at this stage.	
Once the milling for the bottom layer is complete, the machine will prompt to open the milling machine's enclosure lid in order to flip the board along it is symmetry axis. Note: It is advisable to keep the alignment as close to the initial board setup as possible.	

Board Production …

Mount the board with protection film onto the vacuum table and affix it firmly with paper tape. Close the enclosure lid and confirm it on CircuitPro. Note: Make sure to clean the underlay material and secure the copper board with masking tape once again. Also, ensure the PCB is properly aligned with the underlay material.	
Confirm the prompt from CircuitPro to complete the board production process. A sample board is shown for illustrative purposes. Any via holes that appear to be filled up can be pierced later if they are header holes. Clean the board using the PCB Cleaning solution. Test for ground and supply connectivity before soldering components onto the board.	

Board Production …

Click on 'File->Exit' to close CircuitPro. CircuitPro prompts the user for moving the milling head to a safe position (This position is randomly assigned). Switch off the milling machine and ensure that the enclosure lid is firmly shut.
 

PART IV
DESIGN PCB USING KI-CAD SOFTWARE

BRIEF INTRODUCTION TO THE LPKF PROTOMAT S63

The LPKF ProtoMat S63 is a high-performance circuit board plotter designed for rapid PCB prototyping. It is an ideal system for in-house PCB prototyping, offering high-quality results and efficient production.

Key Features of the ProtoMat S63

- Automatic tool change and milling width adjustment
- 60,000 rpm spindle motor for high-speed machining
- Precise and accurate milling and drilling capabilities
- Suitable for machining various materials, including PCBs, RF and microwave substrates, and housing materials
- Compact design with a small footprint

Performance Specifications

- Max. material size and layout area (X/Y/Z): 229 mm x 305 mm x 35/22 mm (9” x 12” x 1.4/ 0.9”)
- Resolution (X/Y): 0.5 µm (0.02 mil)
- Repeatability: ± 0.001 mm (± 0.04 mil)
- Travel speed (X/Y): Max. 150 mm/s (6”/s)

Applications of the ProtoMat S63

- Milling and drilling of 1- and 2-sided circuit boards
- Milling and drilling of RF and microwave substrates
- Milling and drilling of multilayers up to 8 layers
- Routing of printed circuit boards
- Engraving of front panels and signs
- Milling cut-outs in front panels
- Milling of SMD solder paste stencils
- Machining of housings

- Depaneling and reworking of printed circuit boards

Software and Control

The ProtoMat S63 comes with a comprehensive software package, LPKF CircuitPro, which allows for easy operation, high-quality results, and fast production. The software imports all common CAD data and transfers the production data to the structuring systems.

References

1. LPKF. (n.d.). ProtoMat S63. Retrieved from https://www.lpkf.com/en/industries-technologies/research-in-house-pcb-prototyping/products/lpkf-protomat
2. LPKF. (n.d.). CircuitPro. Retrieved from https://app.lpkfusa.com/products/pcb_prototyping/software/circuitpro_pm/
3. DirectIndustry. (n.d.). PCB rapid prototyping machine – ProtoMat S63. Retrieved from https://www.directindustry.com/prod/lpkf-laser-electronics/product-9183-435763.html
4. CDME. (n.d.). LPKF ProtoMat S63. Retrieved from https://cdme.osu.edu/equipment/lpkf-protomat-s63
5. SMTnet. (n.d.). LPKF ProtoMat S-Series. Retrieved from https://smtnet.com/company/index.cfm?catalog_id=18207&company_id=48010&component=catalog&fuseaction=view_company

KICAD TUTORIAL: A COMPREHENSIVE GUIDE TO DESIGNING PCB CIRCUITS

Introduction

KiCad is an open-source software suite for creating electronic circuit schematics, printed circuit boards (PCBs), and associated part descriptions. It is a powerful tool for electronics designers, engineers, and hobbyists. In this tutorial, we will cover the basics of KiCad and provide a step-by-step guide on how to use it for designing PCB circuits.

Getting Started with KiCad

To get started with KiCad, follow these steps:

1. Download and install KiCad from the official website: https://www.kicad.org/download/
2. Create a new project by clicking on "File" > "New Project"
3. Familiarize yourself with the KiCad interface and features

Designing a Schematic

To design a schematic, follow these steps:

1. Open the schematic editor by clicking on "Schematic" in the KiCad project window
2. Add components to your schematic by clicking on "Add a symbol" and searching for the desired component
3. Connect the components by drawing wires between them
4. Save your schematic by clicking on "File" > "Save"

Designing a PCB

To design a PCB, follow these steps:

1. Open the PCB editor by clicking on "PCB" in the KiCad project window
2. Import your schematic by clicking on "File" > "Import" > "Schematic"
3. Place the components on the PCB by clicking on "Place" > "Component"
4. Route the tracks by clicking on "Route" > "Track"
5. Save your PCB by clicking on "File" > "Save"

Generating Gerber Files

To generate Gerber files, follow these steps:

1. Open the PCB editor by clicking on "PCB" in the KiCad project window
2. Click on "File" > "Fabrication Outputs" > "Gerbers"
3. Select the desired options and click on "Generate"

Learning Resources

For more information on KiCad and PCB design, check out the following resources:

- "KiCad Like a Pro, 3rd Edition" by Dr. Peter Dalmaris
- Udemy course: "The Complete Course of KiCad"
- Community-created tutorials on the KiCad website
- YouTube tutorials by Chris Gammell at Contextual Electronics

Conclusion

KiCad is a powerful tool for designing PCB circuits. With this tutorial, you should now have a basic understanding of how to use KiCad for designing schematics and PCBs. For more information and advanced topics, check out the learning resources listed above.

References

1. KiCad Documentation. (n.d.). Getting Started in KiCad. Retrieved from https://docs.kicad.org/6.0/en/getting_started_in_kicad/getting_started_in_kicad.html
2. KiCad. (n.d.). About KiCad. Retrieved from https://www.kicad.org/about/kicad/
3. YouTube. (n.d.). KiCAD 7 PCB Layout in 5 steps. Retrieved from https://www.youtube.com/

watch?v=3FGNw28xBr0

4. KiCad. (n.d.). Learning Resources. Retrieved from https://www.kicad.org/help/learning-resources/
5. Udemy. (n.d.). The Complete Course of KiCad. Retrieved from https://www.udemy.com/course/kicad-course/
6. PCBCart. (n.d.). KiCAD PCB Design Tutorial. Retrieved from https://www.pcbcart.com/article/content/KiCAD-PCB-design-tutorial.html

UNDERSTANDING AND DESIGNING NRF24 TRANSCEIVER CIRCUIT USING KI-CAD

Overview of NRF24 Transceiver

The NRF24 is a popular 2.4 GHz wireless transceiver module used in a variety of applications, including remote controls, home automation, and wireless sensor networks. It allows for wireless communication between microcontrollers and other devices over a short range, typically up to 100 meters indoors and 1,000 meters outdoors with line of sight.

Features of NRF24 Transceiver

- **Frequency Range:** 2.4 GHz ISM band
- **Data Rate:** Up to 2 Mbps
- **Power Consumption:** Low power consumption with power-down and standby modes
- **Communication Range:** Typically up to 100 meters indoors
- **Interface:** SPI (Serial Peripheral Interface)
- **Operating Voltage:** 1.9V to 3.6V

Circuit Design for NRF24 Transceiver

Designing a circuit with the NRF24 transceiver involves connecting it to a microcontroller (e.g., Arduino, ESP8266) and providing the necessary power and communication lines. Below is a basic schematic of how the NRF24 transceiver is connected in a circuit:

1. **Power Supply:**
    - VCC (Pin 2) -> 3.3V
    - GND (Pin 1) -> Ground
2. **SPI Interface:**
    - CE (Pin 3) -> Digital I/O pin on the microcontroller (e.g., D9)
    - CSN (Pin 4) -> Digital I/O pin on the microcontroller (e.g., D10)
    - SCK (Pin 5) -> SCK pin on the microcontroller (e.g., D13)

- MOSI (Pin 6) -> MOSI pin on the microcontroller (e.g., D11)
- MISO (Pin 7) -> MISO pin on the microcontroller (e.g., D12)

3. **Additional Connections:**

- IRQ (Pin 8) -> Can be connected to an interrupt pin on the microcontroller if needed.

Below is a simple schematic representation

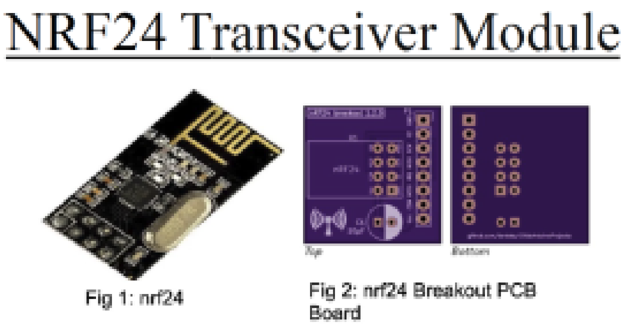


Fig 1: nrf24

Fig 2: nrf24 Breakout PCB Board

```
        NRF24
      +-------+
VCC --|2     1|-- GND
CE  --|3     4|-- CSN
SCK --|5     6|-- MOSI
MISO--|7     8|-- IRQ (optional)
      +-------+
```

PCB Layout for NRF24 Transceiver

Creating a PCB layout for the NRF24 transceiver involves placing the components on the board and routing the connections. The following steps outline the process using KiCad:

1. **Component Placement:**

- Place the NRF24 module in a suitable location on the PCB.
- Place the microcontroller and any other components needed for the circuit.

2. **Routing:**

- Route the power lines (VCC and GND) to the NRF24 module.
- Route the SPI lines (CE, CSN, SCK, MOSI, MISO) to the respective pins on the microcontroller.
- Ensure the traces are wide enough for power lines and appropriately spaced to avoid interference.

3. **Ground Plane:**

- Create a ground plane to reduce noise and improve signal integrity.

4. **Design Rules Check:**
    - Perform a design rules check (DRC) to ensure there are no errors in the PCB layout.

Below is an example of how the PCB layout may look:

Video Lecture: Designing the NRF24 Circuit on KiCad

To complement this reading, refer to the following video lectures demonstrating the design process of the NRF24 transceiver circuit on KiCad. These videos will guide you step-by-step through creating the schematic, placing components, routing, and finalizing the PCB layout.

VIDEO LECTURE 1: INTRO TO NRF24 CIRCUIT DESIGN

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=44#oembed-1

VIDEO LECTURE 2: CREATING A SYMBOL PART-1

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=49#oembed-1

VIDEO LECTURE 3: CREATING A SYMBOL PART-2

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=51#oembed-1

VIDEO LECTURE 4: ASSOCIATE COMPONENT TO THE FOOTPRINT FOR CONNECTOR 01X08

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=54#oembed-1

VIDEO LECTURE 5: CREATE CUSTOM FOOTPRINT NRF24

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=56#oembed-1

VIDEO LECTURE 6: SAVING FOOTPRINTS

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=58#oembed-1

VIDEO LECTURE 7: FOOTPRINT PLACEMENT AND 3D VIEW

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=60#oembed-1

VIDEO LECTURE 8: ROUTING

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=62#oembed-1

VIDEO LECTURE 9: ADDTEXT

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=64#oembed-1

VIDEO LECTURE 10: UPDATING LAYOUT BY ADDING THE CAPACITOR

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=68#oembed-1

VIDEO LECTURE 11: TRACK WIDTH CONTROL

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=71#oembed-1

VIDEO LECTURE 12: AUTOMATIC TRACKWIDTH CALCULATOR

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=73#oembed-1

VIDEO LECTURE 13: ADDING COPPER LAYERS

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=75#oembed-1

VIDEO LECTURE 14: CREATING GERBER FILES

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=77#oembed-1

SUPPLEMENT LECTURE VIDEO ON ANNOTATION AND TRACK WIDTH CALCULATION USING KI-CAD SOFTWARE

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://openwa.pressbooks.pub/nehakardam18/?p=157#oembed-1

A brief summary of PCB terminology:

- **Signal Integrity:** Signal integrity is the ability of the PCB to maintain the integrity of the signals being transmitted between the components. This involves proper signal routing, trace impedance matching, and minimizing signal reflections.
- **Power Integrity:** Power integrity is the ability of the PCB to maintain the integrity of the power being supplied to the components. This involves proper power routing, minimizing voltage drop, and minimizing noise in the power supply.
- **EMI/EMC:** Electromagnetic interference (EMI) and electromagnetic compatibility (EMC) are important considerations in PCB design. While EMC refers to the PCB's capacity to function without interference from other electromagnetic sources, EMI refers to the unwanted electromagnetic radiation that the PCB generates.
- **Design for Manufacturing (DFM):** Design for Manufacturing (DFM) is the process of designing the PCB in a way that is optimized for the manufacturing process. This involves considering factors such as component placement, trace routing, and panelization.
- **Design for Test (DFT):** Design for Test (DFT) is the process of designing the PCB in a way that makes it easier to test and diagnose issues. This involves considering factors such as test points, probe access, and fault coverage.
- **High–Speed Design:** High–speed design is the process of designing PCBs that can handle high–speed signals, such as those used in communication systems. This involves considerations such as signal integrity, power integrity, and EMI/EMC.
- **PCB Assembly:** PCB assembly is the process of assembling the components onto the PCB. This involves considerations such as soldering techniques, inspection, and testing.
- **PCB Testing:** PCB testing is the process of testing the PCB to ensure that it meets the required specifications. This involves considerations such as functional testing, environmental testing, and reliability testing.

References:

[1] Robertson, C. T. (2004). Printed circuit board designer's reference: basics. Prentice Hall Professional.