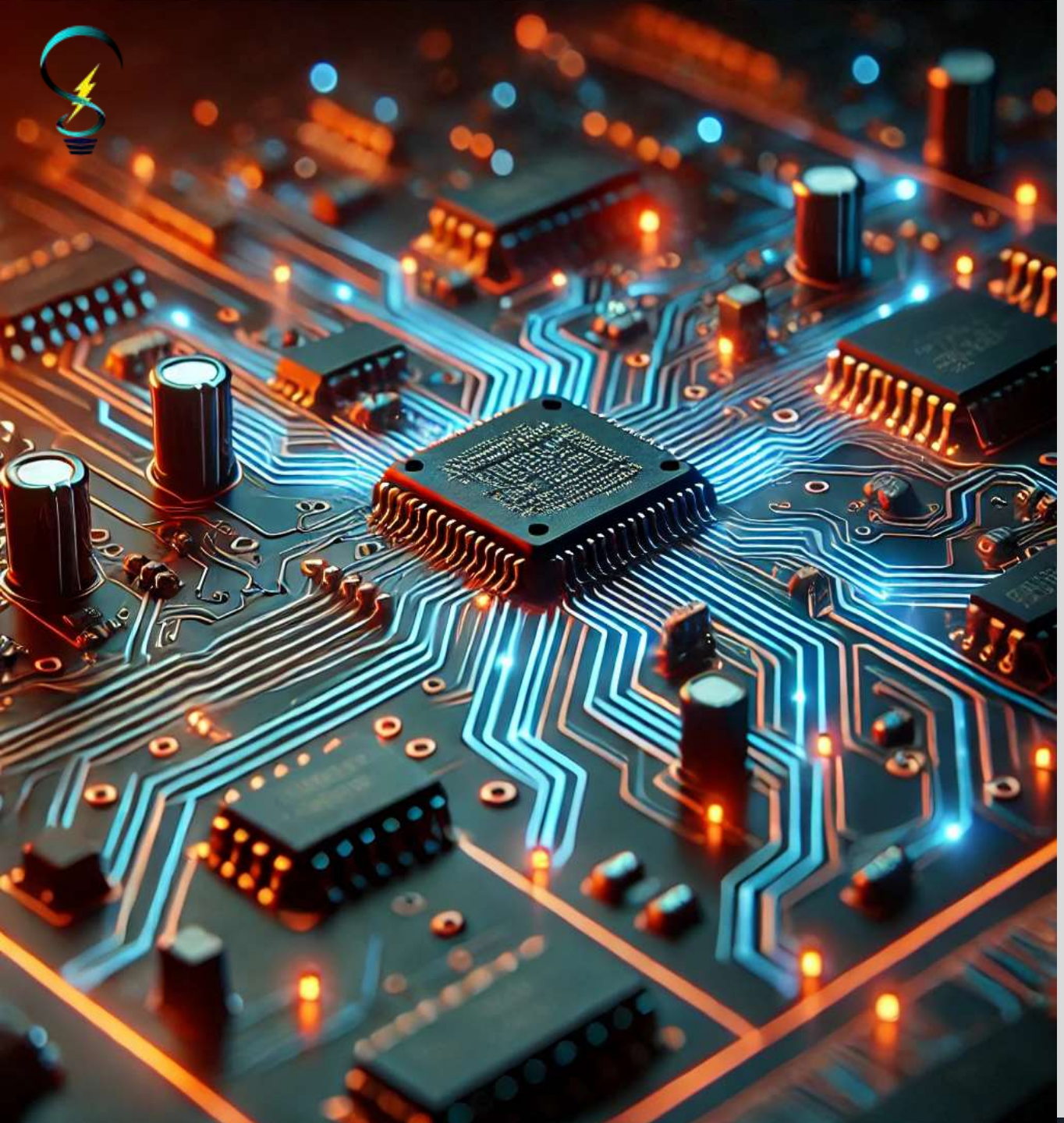
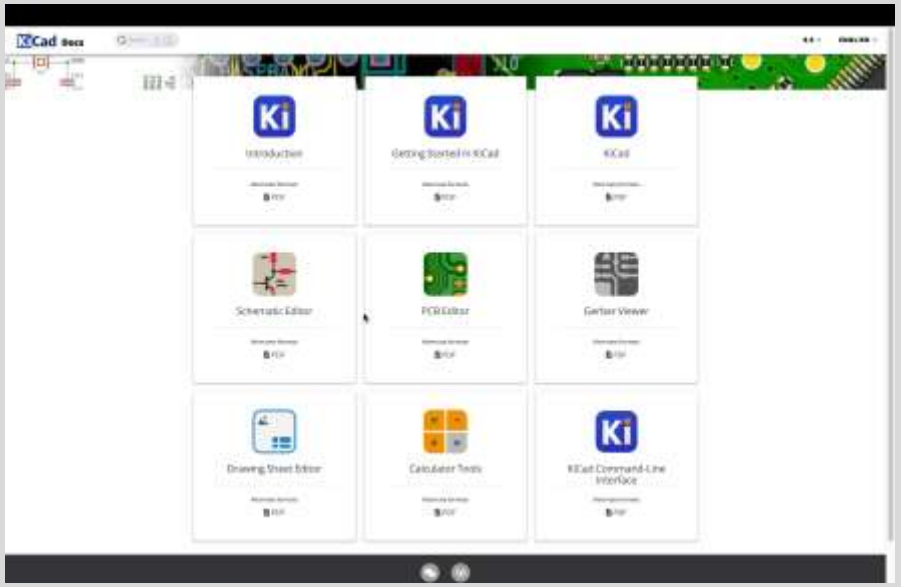


Month of Tech : PCB-designing

PCB Design 101: From Schematic to Prototype

Explore and Learn: Workshop for Beginners



LINKS

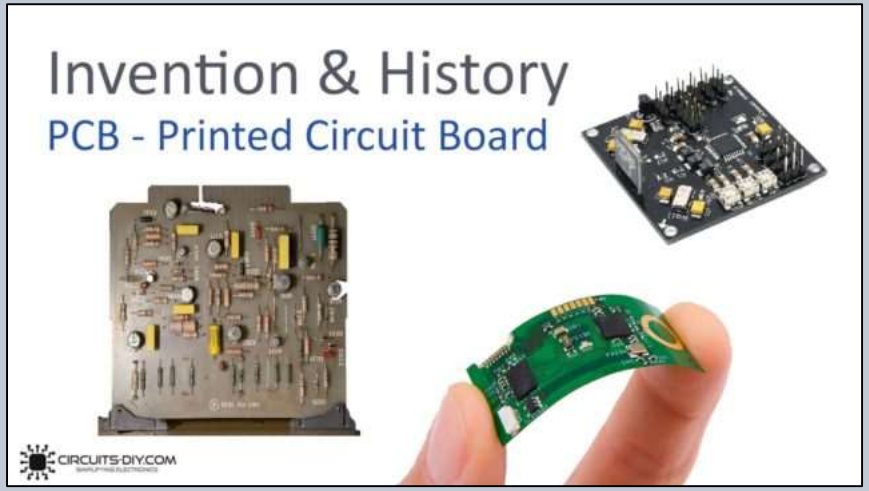
MAC – YOUTUBE

WINDOWS – YOUTUBE

DOCUMENTATIONS

Interesting Facts About PCBs:

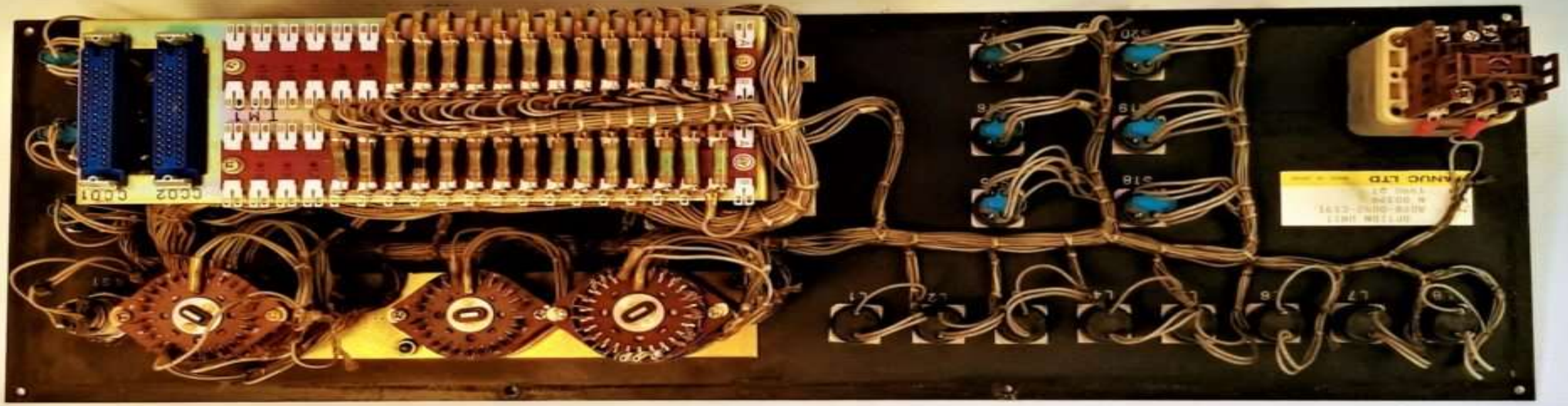
- The ***first PCBs*** were developed in the 1940s for military equipment.

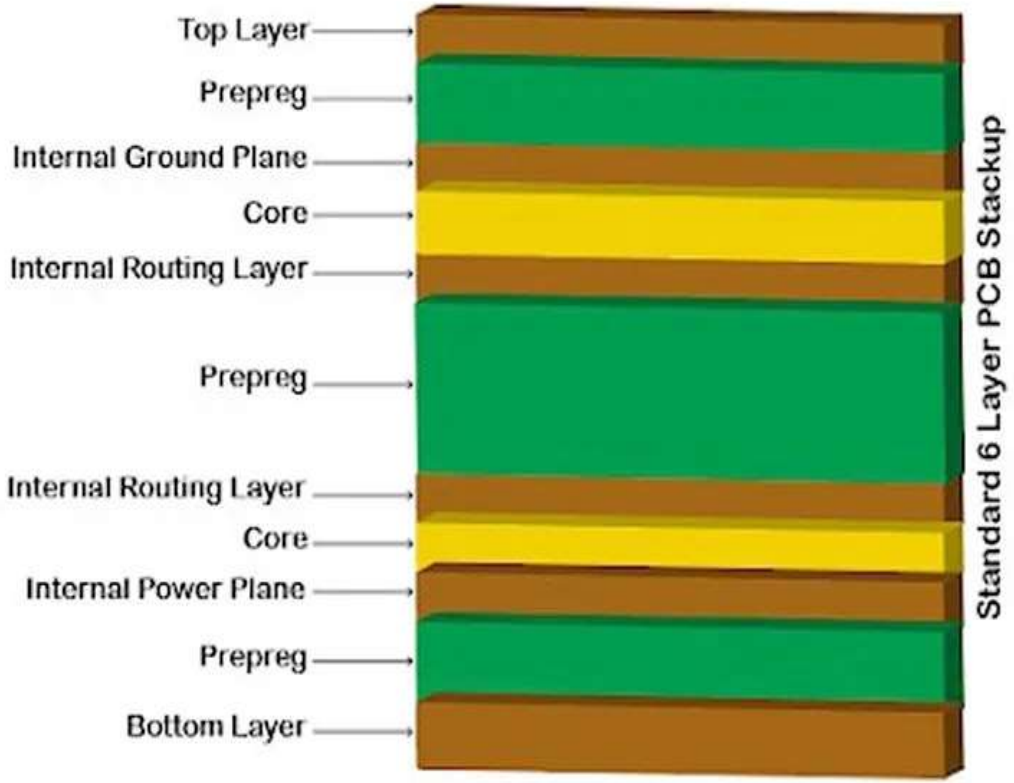


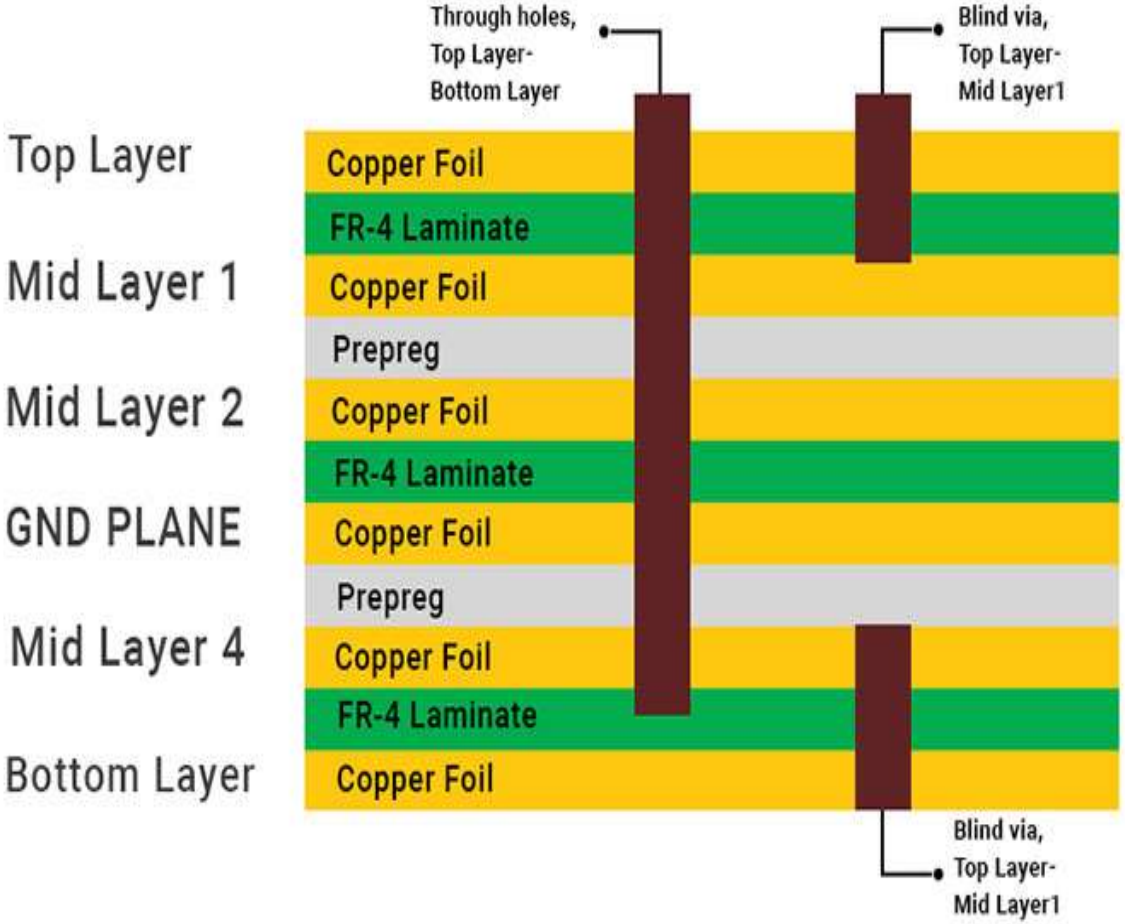
- Originally called **Printed Wiring Boards (PWBs);** the name changed as technology advanced to include more than just wiring

- ***Green solder mask*** on PCBs is iconic but not mandatory—colors can vary (e.g., red, blue, black).

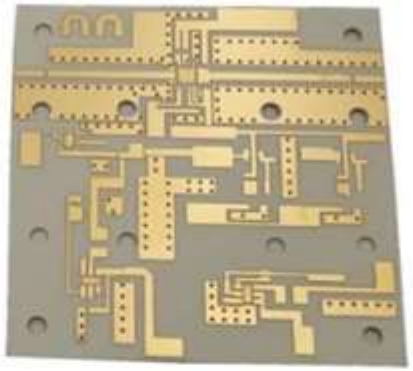
- Modern PCBs support multi-layer designs with up to ***20+ layers*** for high-performance systems.


CCD1
CCD2
IM1
FANUC LTD


Top Layer
Prepreg
Internal Ground Plane
Core
Internal Routing Layer
Prepreg
Internal Routing Layer
Core
Internal Power Plane
Prepreg
Bottom Layer
Standard 6 Layer PCB Stackup


Through holes, Top Layer- Bottom Layer
Blind via, Top Layer- Mid Layer1
Top Layer
Mid Layer 1
Mid Layer 2
GND PLANE
Mid Layer 4
Bottom Layer
Copper Foil
FR-4 Laminate
Copper Foil
Prepreg
Copper Foil
FR-4 Laminate
Copper Foil
Prepreg
Copper Foil
FR-4 Laminate
Copper Foil
Blind via, Top Layer- Mid Layer1

single-sided PCB

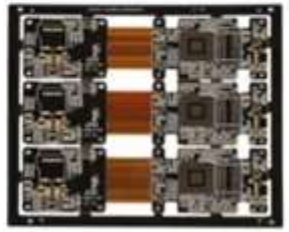


double-sided PCB

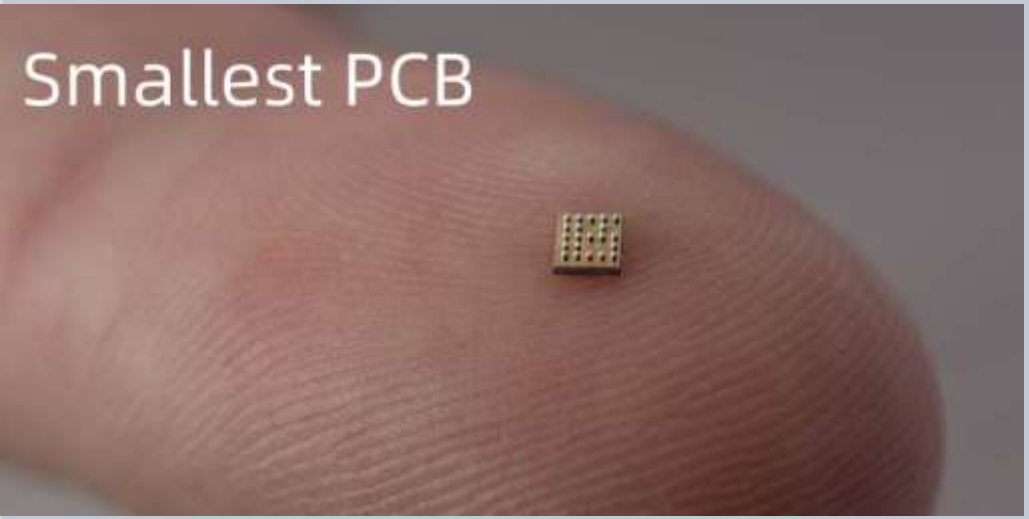
multilayer PCB

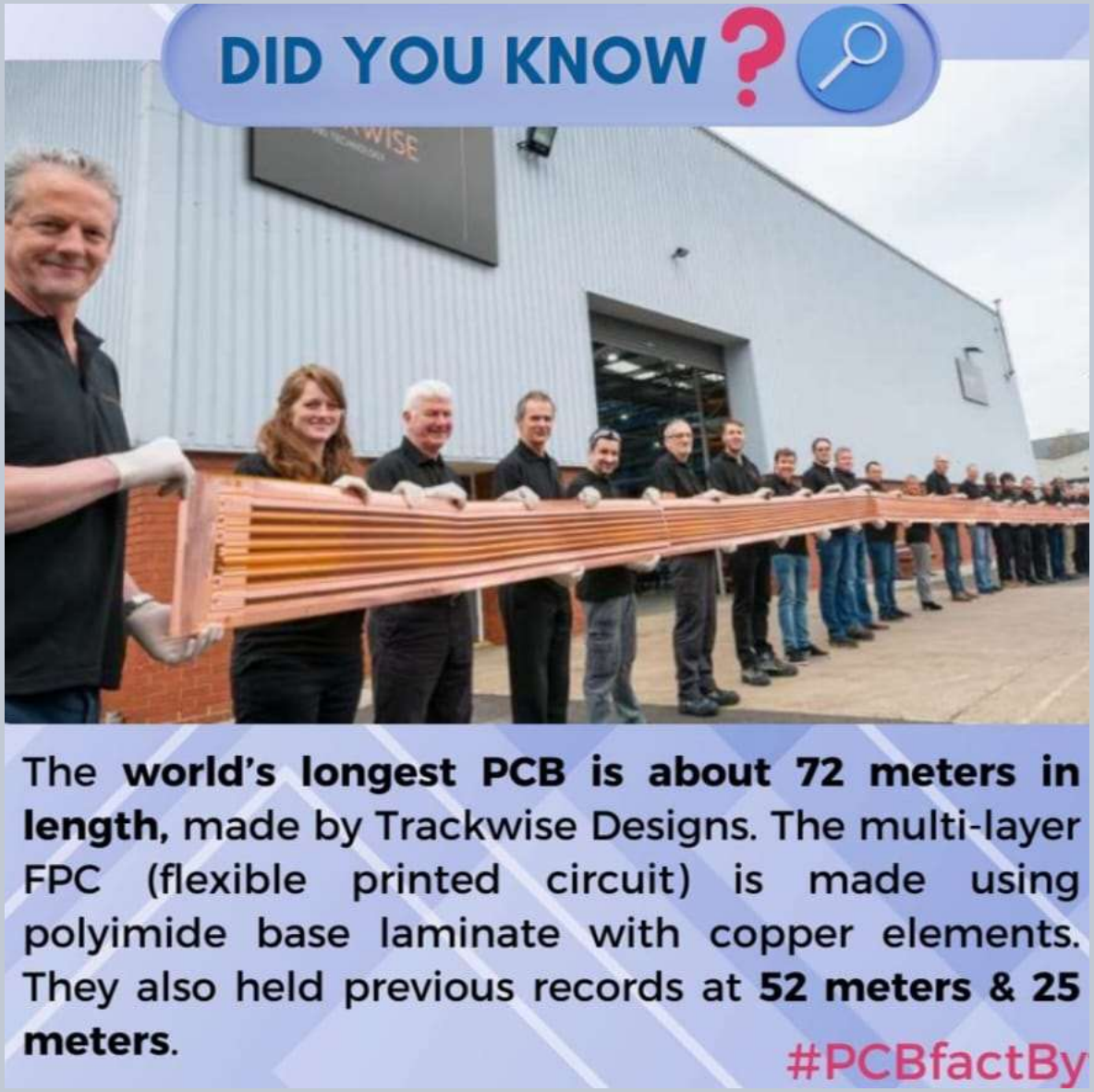
rigid PCB

flexible PCB



rigid-flex PCB


Smallest PCB


DID YOU KNOW ?
The world's longest PCB is about 72 meters in length, made by Trackwise Designs. The multi-layer FPC (flexible printed circuit) is made using polyimide base laminate with copper elements. They also held previous records at 52 meters & 25 meters.
#PCBfactBy

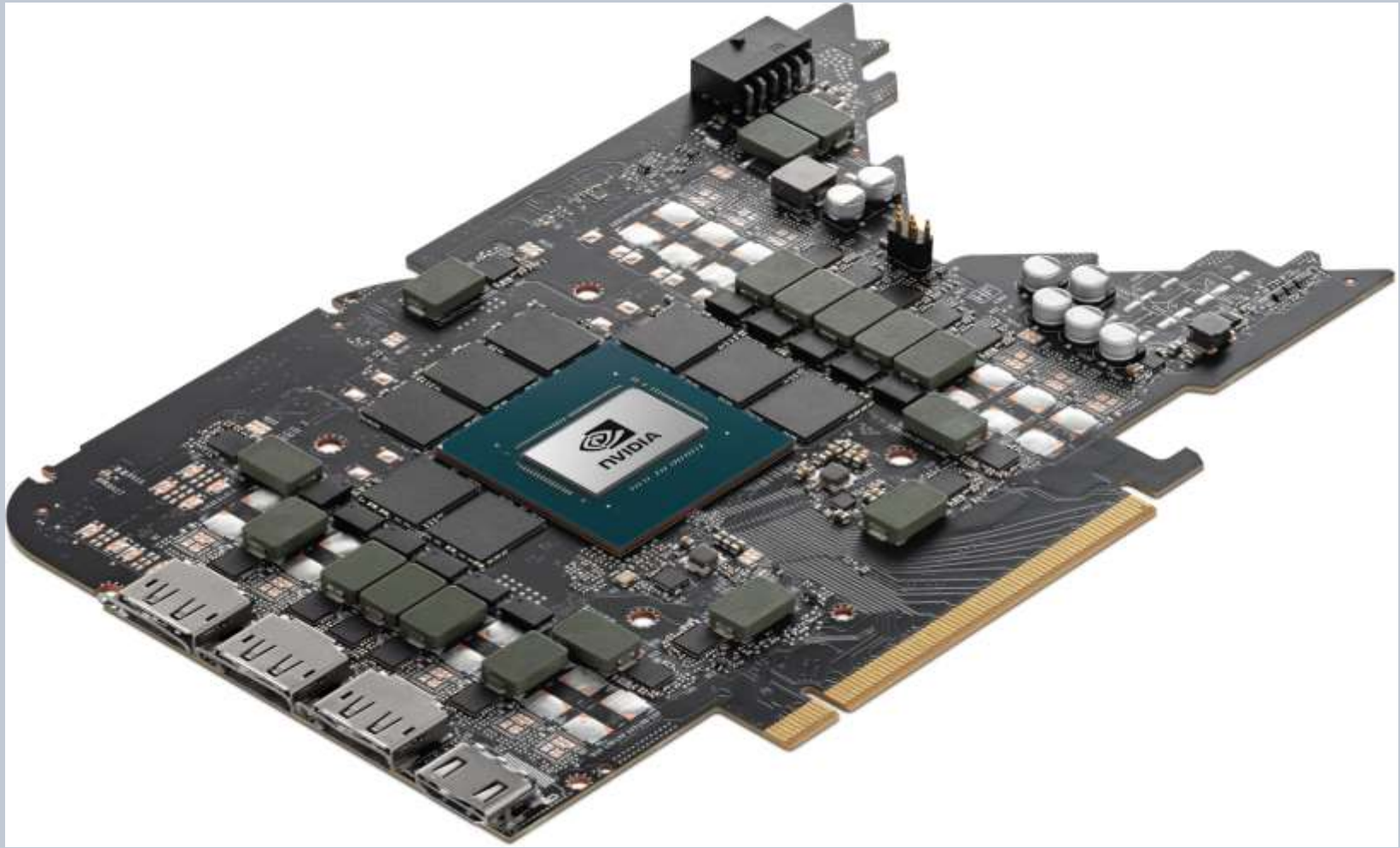
Why Learn PCB Design?

1. **Core Skill in Electronics**:

- PCBs are the backbone of modern devices, from smartphones to medical equipment.

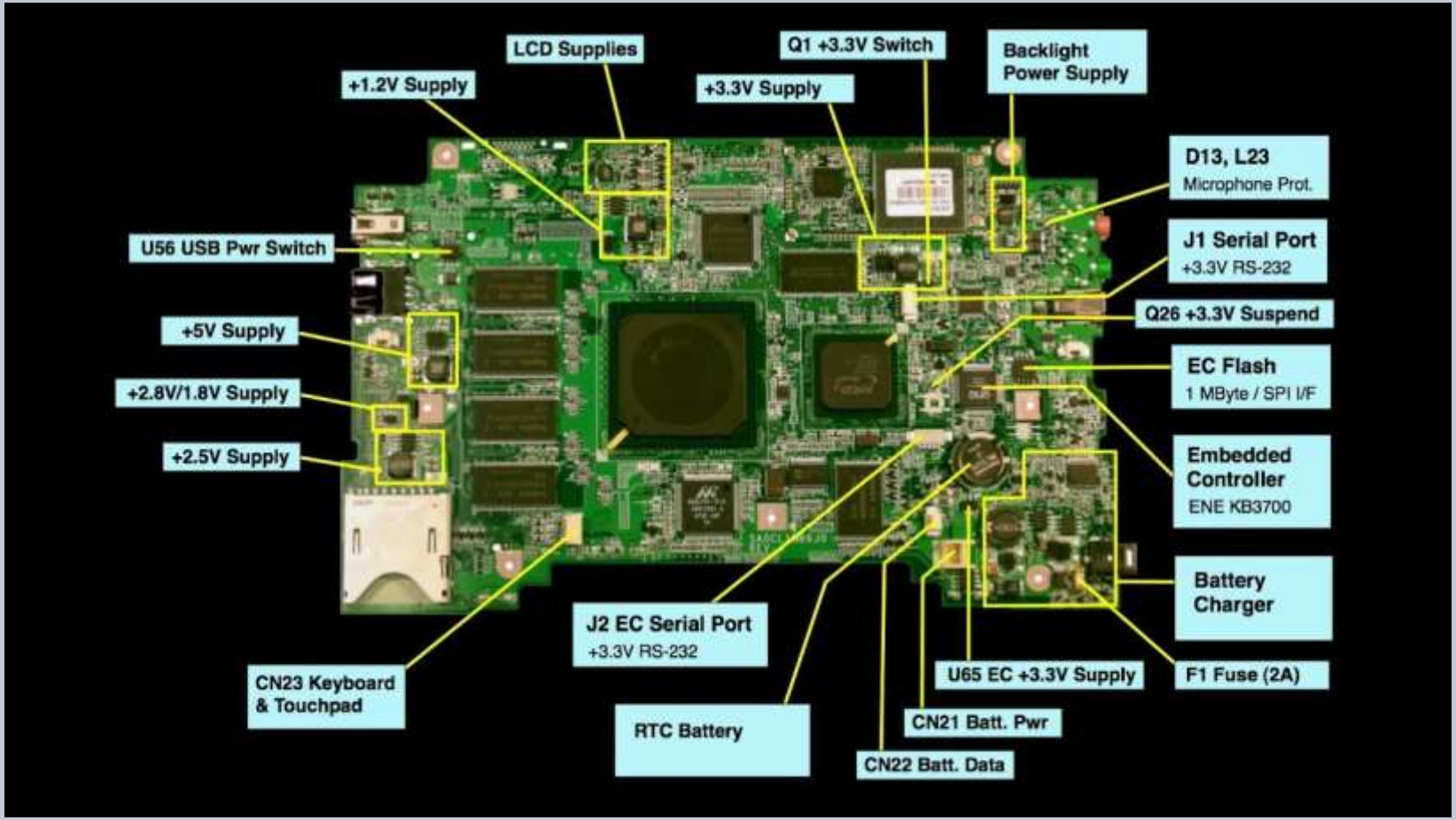
- Mastering PCB design is essential for understanding and *innovating* (Nvidia) in electronics.

Why Learn PCB Design?



Why Learn PCB Design?

Why Learn PCB Design?



Why Learn PCB Design?

Why Learn PCB Design?

Career Opportunities:

- Industries: Consumer electronics, IoT, robotics, aerospace and automotive.

- Job Roles: PCB Design Engineer, Hardware Developer, Embedded Systems Designer.

Why Learn PCB Design?

PCB Design-Specific Roles

1. PCB Design Engineer
2. PCB Layout Engineer
3. CAD PCB Designer
4. PCB Prototype Specialist

Hardware Development Roles Involving PCB Design

5. Hardware Engineer
6. Embedded Systems Engineer
7. Electronics Design Engineer
8. RF PCB Design Engineer
9. Analog and Mixed-Signal Design Engineer

Why Learn PCB Design?

Manufacturing and Quality Roles

10. PCB Manufacturing Engineer
11. PCB Assembly Engineer
12. DFM (Design for Manufacturing) Engineer
13. PCB Testing Engineer

Research and Development (R&D) Roles

14. R&D Hardware Engineer
15. Circuit Design Specialist
16. Power Electronics Engineer

Why Learn PCB Design?

Specialized Roles

17. Signal Integrity Engineer
18. High-Speed PCB Design Engineer
19. Flexible PCB Design Specialist
20. IoT Hardware Designer

Why Learn PCB Design?

Why Learn PCB Design?

HQ NextPCB
www.nextpcb.com

viasion

J@LC JLCPCB

PCBWay

深圳市三辉创电子
SHC Electronics Limited

Topscom

GLOBALPCB
The BEST PCB Solution Service Provider

UET
Union Electronic Technology

PCBgogo

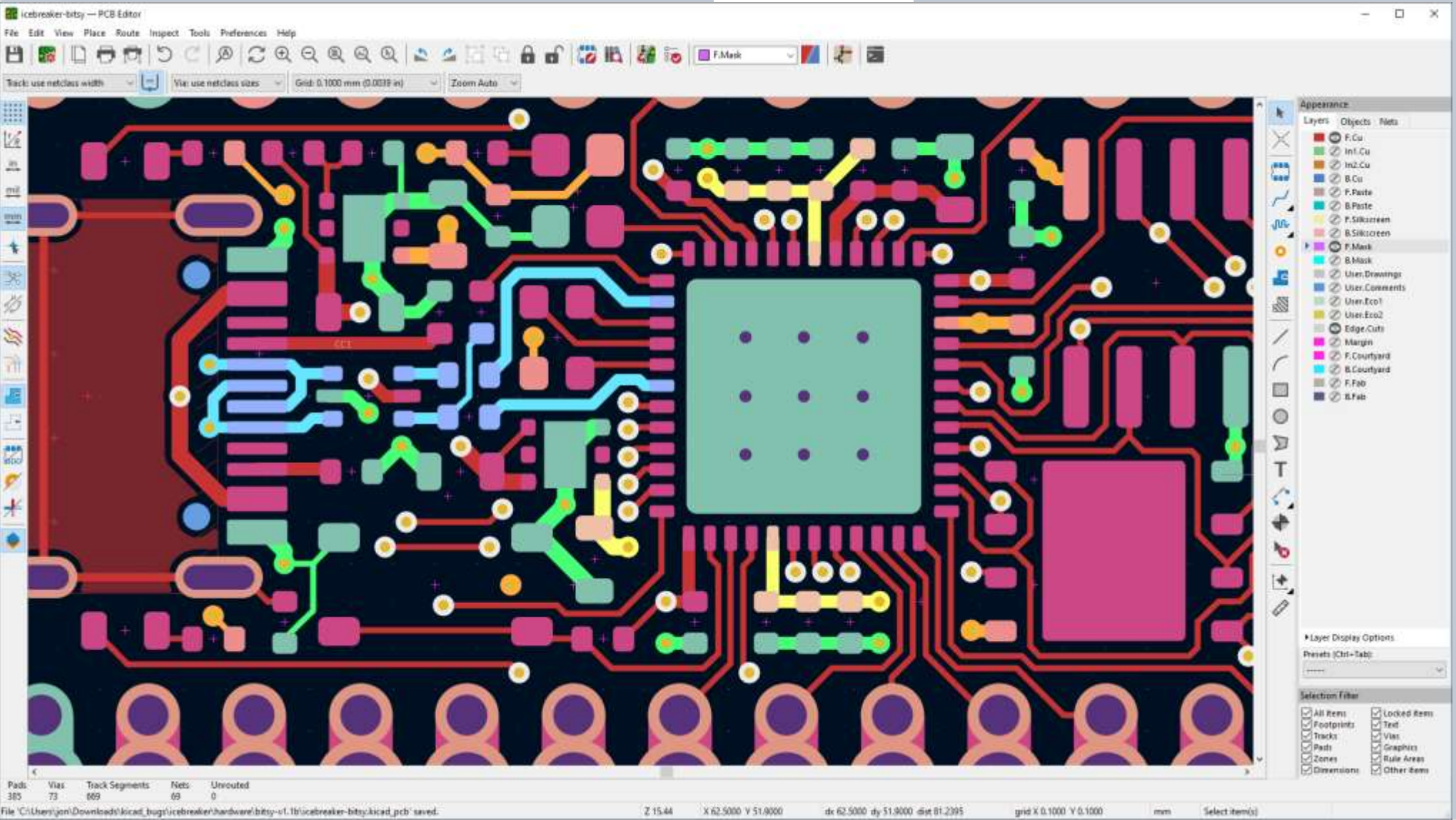
PCBWIN

Why Learn PCB Design?

Key Benefits of PCBs

- **Compact Designs**: Replaces messy wiring with organized, durable layouts.
- **Scalability**: Enables mass production and consistent quality.
- **Enhanced Performance**: Reliable connections, minimal interference and durability.

Why Ki-CAD?


icebreaker-bitsy — PCB Editor
File Edit View Place Route Inspect Tools Preferences Help
Appearance
Layers Objects Nets
F.Cu
In1.Cu
In2.Cu
B.Cu
F.Paste
B.Paste
F.Silkscreen
B.Silkscreen
F.Mask
B.Mask
User.Drawings
User.Comments
User.Eco1
User.Eco2
Edge.Cuts
Margin
F.Courtyard
B.Courtyard
F.Fab
B.Fab
Selection Filter

Why Ki-CAD?

- Free, open-source, and widely used in both academia and industry.

- Extensive community support and compatibility with fabrication services.

How Learning PCB Design Helps You:

- **Boost Creativity**: Transform circuit ideas into functional prototypes.
- **Develop Problem-Solving Skills**: Address challenges like routing and component placement.
- **Hands-On Experience**: Bridge the gap between theory and practical application.

When to use PCB Designing?

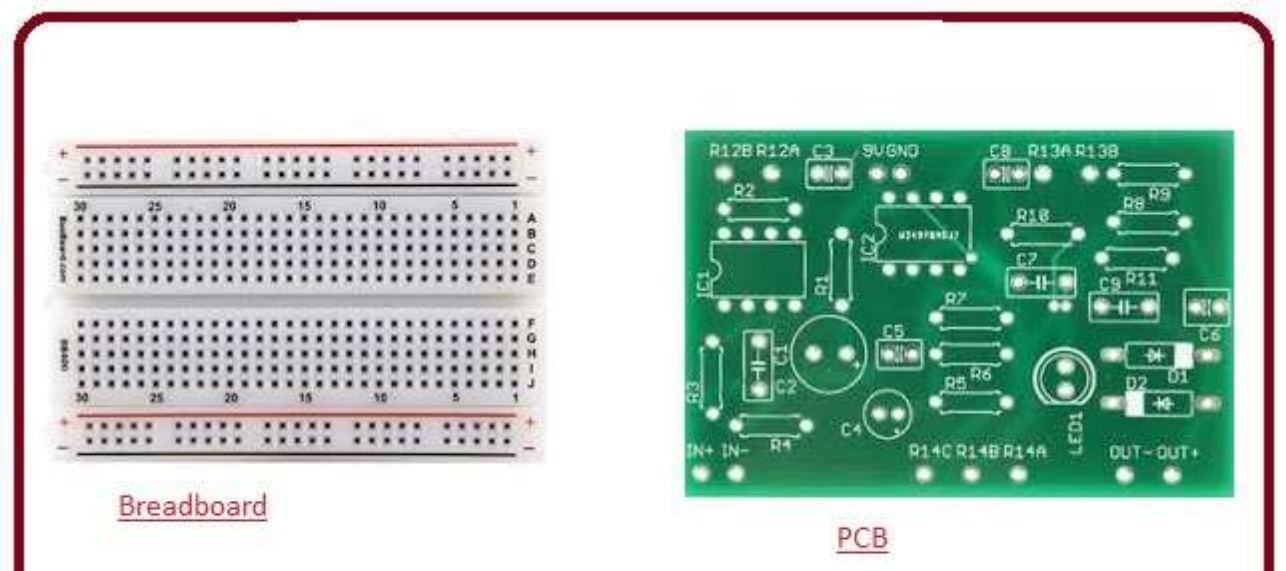
1. Prototype to Production

• **From Breadboard to PCB**: Transitioning from temporary breadboard setups to permanent, robust, and scalable designs.

• **Why Use PCBs Here?**:

• Breadboards are for testing concepts and quick experiments.

• PCBs ensure reliability, repeatability, and durability for production.



When to use PCB Designing?

2. Complex Circuits

- **For Multi-Component Circuits**: When your circuit requires many components (e.g., microcontrollers, sensors, actuators) and complex interconnections.

- **Why PCBs Are Ideal**:
- Allows precise placement of components.
- Clean, organized routing that avoids wire clutter.

When to use PCB Designing?

3. Custom Applications

- **Tailored Designs**: Create PCBs for unique and specialized devices, such as IoT applications, robotics, or wearable tech.

- **Why Custom PCBs?**:
- Meets specific design requirements, including form factor, power, and functionality.
- Enables mass production of custom circuits.

When to use PCB Designing?

4. Signal Integrity Requirements

• **High-Speed & RF Circuits**: For circuits that operate at high frequencies or with precise timing (e.g., RF circuits, communication systems).

• **Why Use PCBs for Signal Integrity?**:

• Minimizes signal loss, reflection and interference by controlling trace geometry.

• Ensures stable operation of high-speed signals (e.g., HDMI, USB).

When to use PCB Designing?

5. Durability

• **Mechanical & Environmental Stress**: When the circuit will face physical stress (e.g., vibration, temperature changes) or harsh environments (e.g., industrial applications, automotive).

• **Why PCBs Are Stronger?**:

• PCBs are more resistant to damage and wear compared to breadboards or point-to-point wiring.

• They can be made with protective coatings, making them resistant to moisture, heat, and other environmental factors.

Basic Terminology.

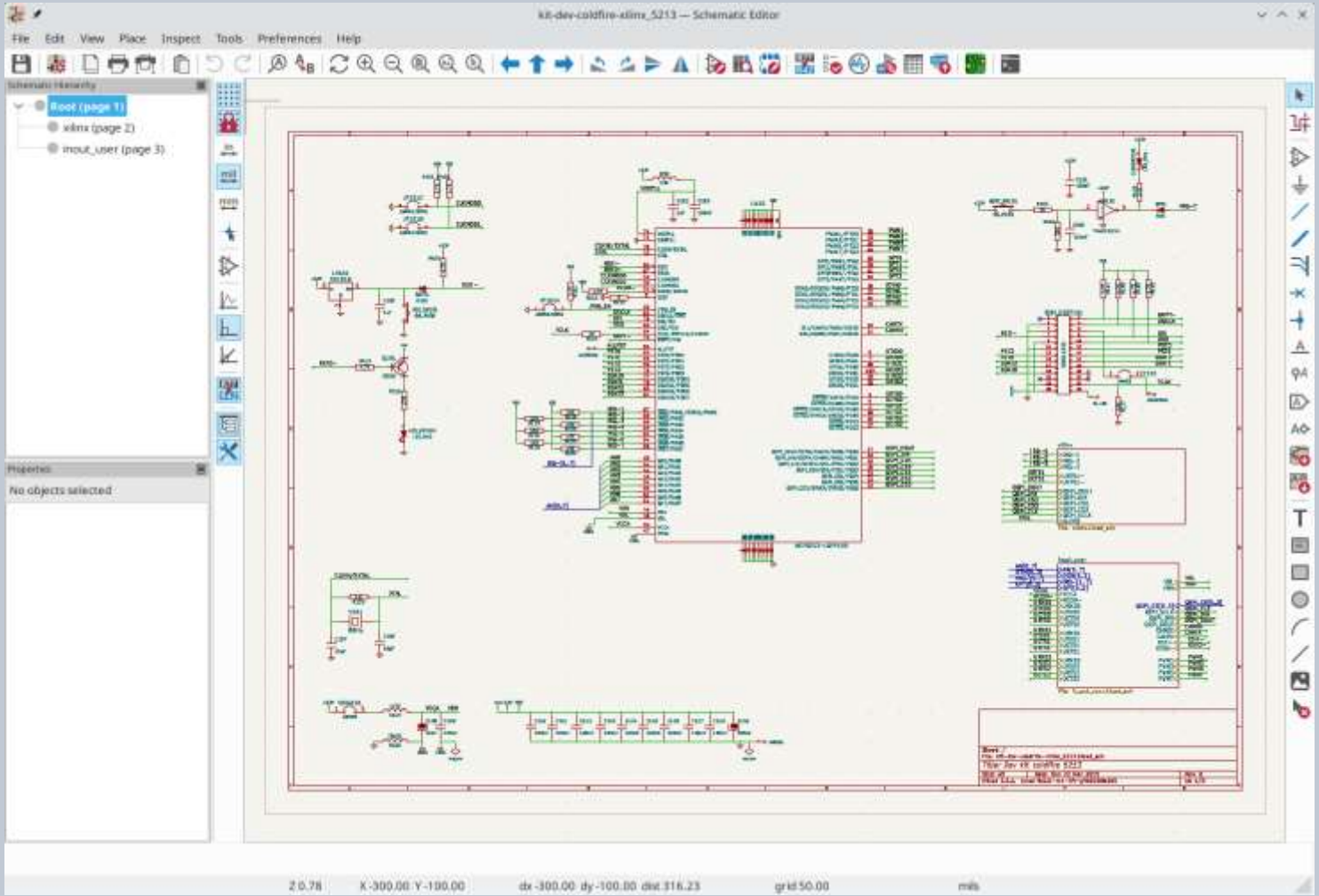
- *schematic*
- *hierarchical schematic*
- *symbol*
- *Netlist*

- *printed circuit board (PCB)*

- *footprint*
- *worksheet*
- *plotting*
- *ngspice*

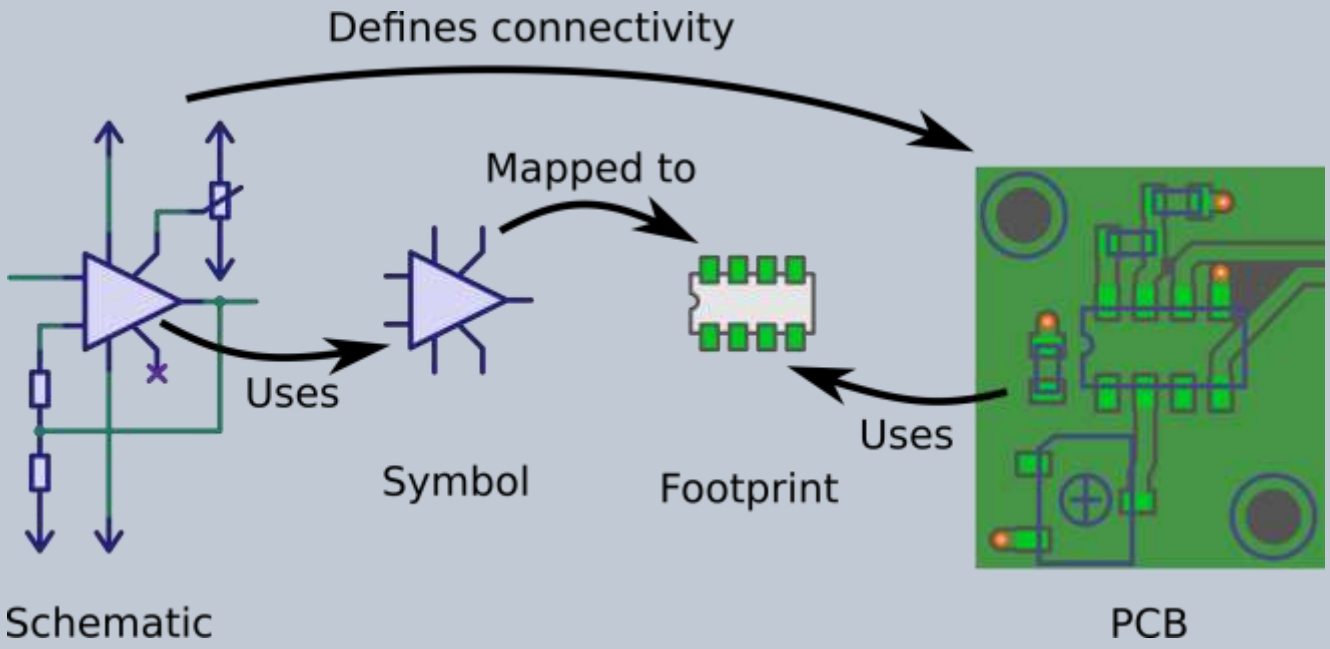
Basic Terminology. Schematic

A **schematic** is a collection of one or more pages (sheets) of circuit schematic drawings. Each Ki-Cad schematic file represents a single sheet.



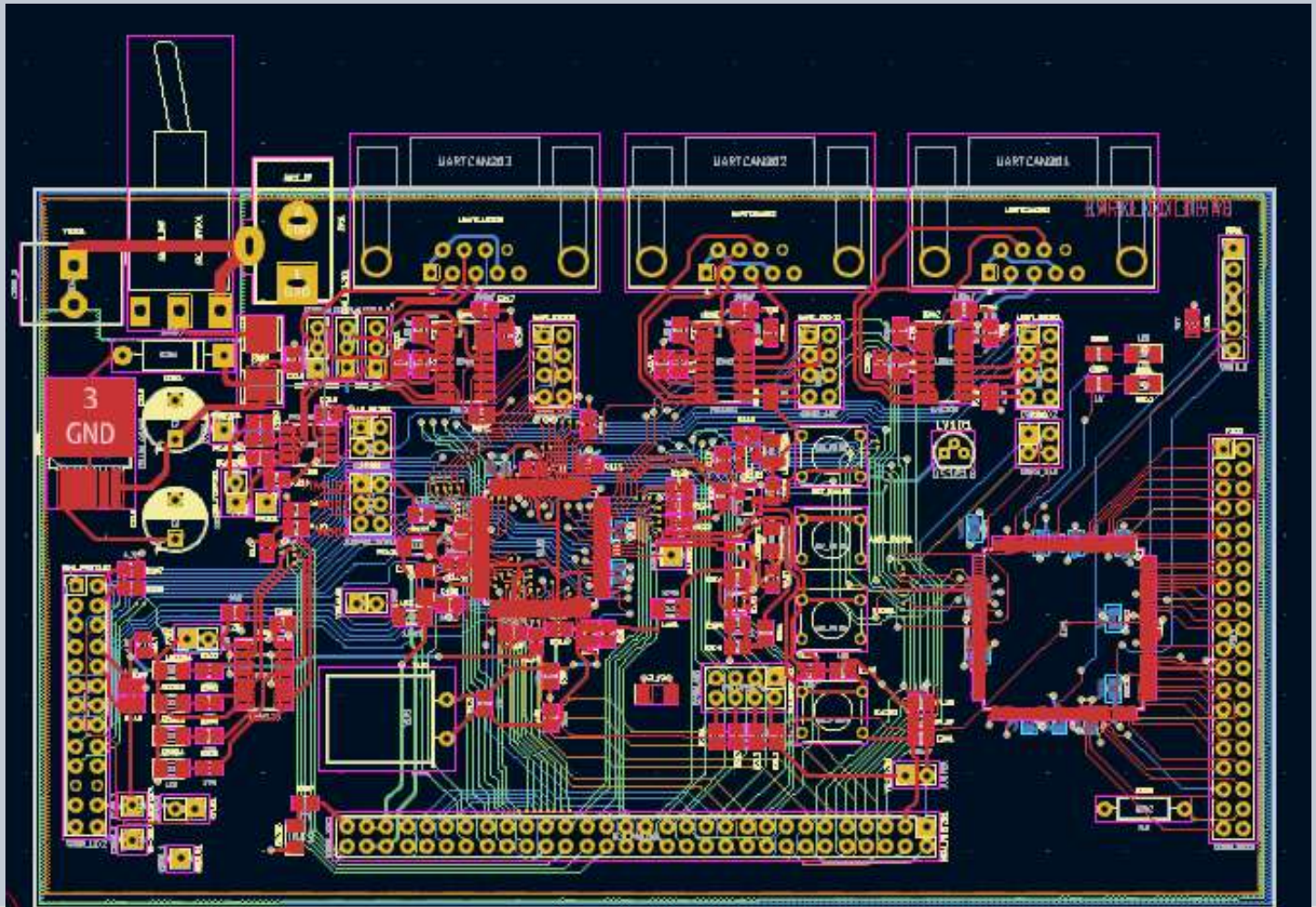
Basic Terminology. Symbol

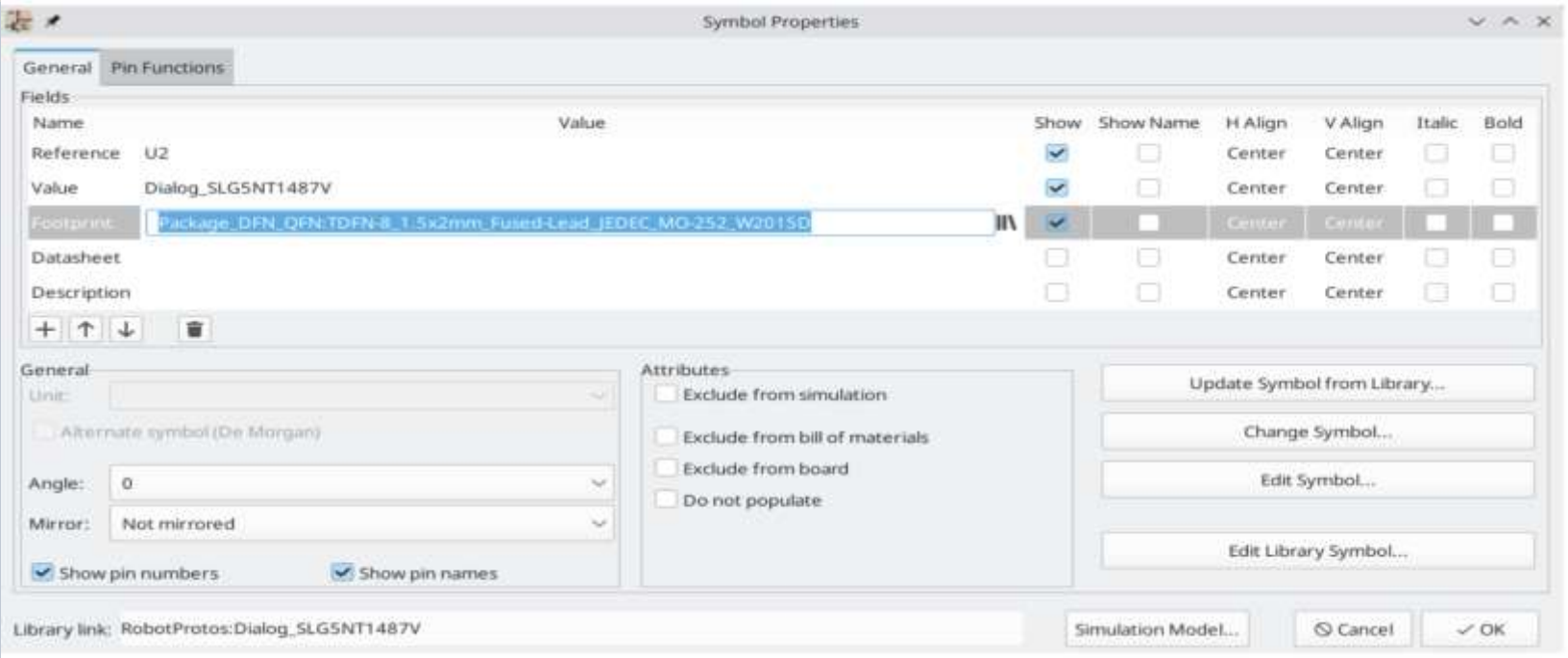
A **symbol** is a circuit element that can be placed on a schematic. Symbols can represent physical electrical components, such as a resistor or microcontroller, or non-physical concepts such as a power or ground rail. Symbols have **pins** which serve as the connection points that can be wired to each other in a schematic.



Basic Terminology. Footprint

A **footprint** is a circuit element that can be placed on a PCB. Footprints often represent physical electrical components but can also be used as a library of design elements (silkscreen logos, copper antennas and coils, etc.).





Component Packaging Variants:

•Manufacturers produce components in various physical packages to suit different applications.

•Example: A 555 timer IC can come in DIP (Dual Inline Package), SOIC (Small Outline IC), or TSSOP (Thin Shrink Small Outline Package).

Design Constraints:

•Designers may choose specific footprints based on available space, heat dissipation requirements, or assembly processes.

•Example: A surface-mount package is often used in compact PCBs, while through-hole packages may be preferred for prototypes.

Standardization Across Libraries:

•KiCad libraries are designed to be generic, supporting a wide range of footprints for a single symbol to cater to diverse user needs.

Interchangeability:

•Some components, like resistors or capacitors, have standardized symbols but can have footprints for different mounting types (e.g., axial, radial, or surface mount).

Basic Terminology. Net-List

A **netlist** is a file that defines the connections (nets) between components in your schematic and translates them to the physical layout on the PCB.

Purpose:

- Maps the **logical connections** from the schematic to the PCB layout.
- Ensures accurate **electrical connectivity** between components in the design.
- Acts as a **bridge** between schematic and PCB layout modules.

Key Components of a Netlist:

1. **Components**: List of all components with their reference designators (e.g., R1, U1).
2. **Pins**: Details about the pins of each component.
3. **Nets**: Connections that specify which pins are electrically connected.

Basic Terminology. Net-List

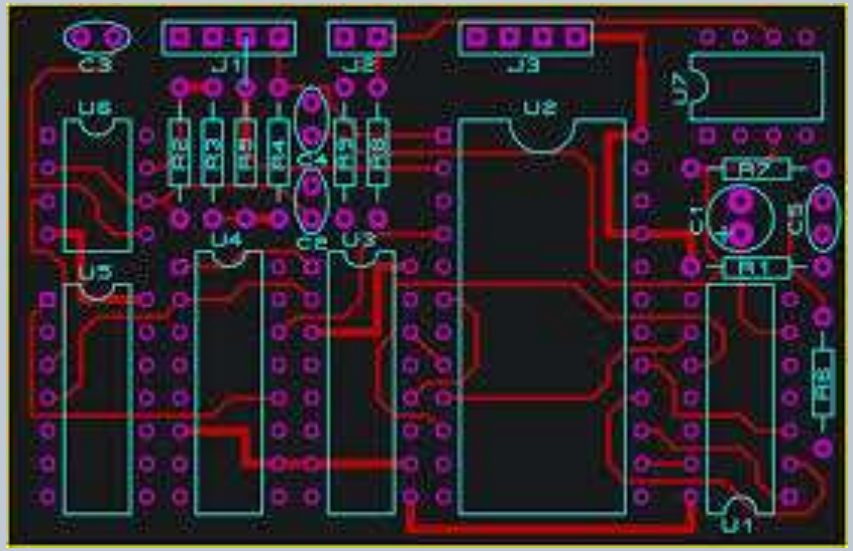

```xml
<?xml version="1.0" encoding="UTF-8"?>
<export version="D">
  <design>
    <source>C:/1_KiCad/Relay Board/Relay Board.sch</source>
    <date>2016-11-26 10:13:28</date>
    <tool>Eeschema 4.0.1-stable</tool>
    <sheet number="1" name="/" tstamps="/">
      <title_block>
        <title>Relay Board</title>
        <company/>
        <rev/>
        <date>2016-01-30</date>
        <source>Relay Board.sch</source>
        <comment number="1" value=""/>
        <comment number="2" value=""/>
        <comment number="3" value=""/>
        <comment number="4" value=""/>
      </title_block>
    </sheet>
  </design>
  <components>
    <comp ref="J2">
      <value>9Pos_Header</value>
      <footprint>Connectors:9_PIN_MATE-N-LOK</footprint>
      <libsource lib="Relay Board" part="9Pos_Header"/>
      <sheetpath names="/" tstamps="/"/>
      <tstamp>56AD5AE4</tstamp>
    </comp>
```

Basic Terminology. PCB LAYOUT

Schematic diagram contains 2-dimensional lines that arrange all the components connecting in an organized manner.

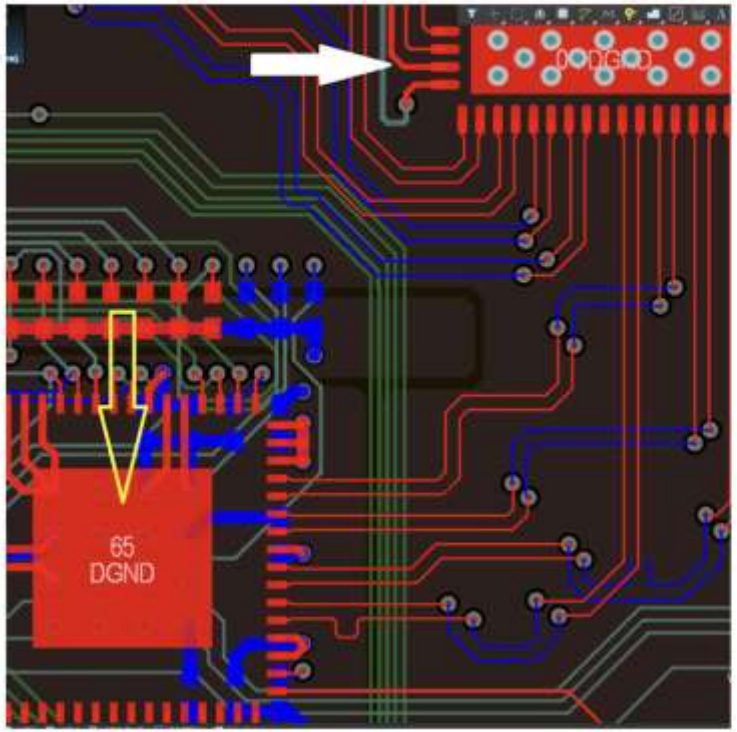
On the other hand, a PCB layout is like a virtual circuit board that is okay to consider as a digital copy of the circuit.

The layout is generated through a schematic. Without a schematic, it is almost impossible to design the layout as we will not have a defined pattern for component placement and developing signal tracks.

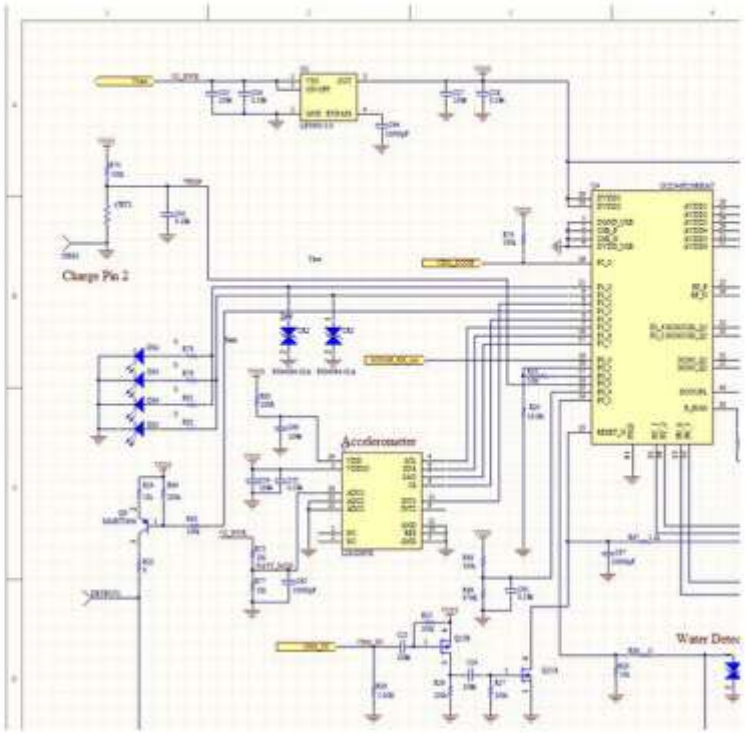


Basic Terminology. PCB LAYOUT

PCB Layout

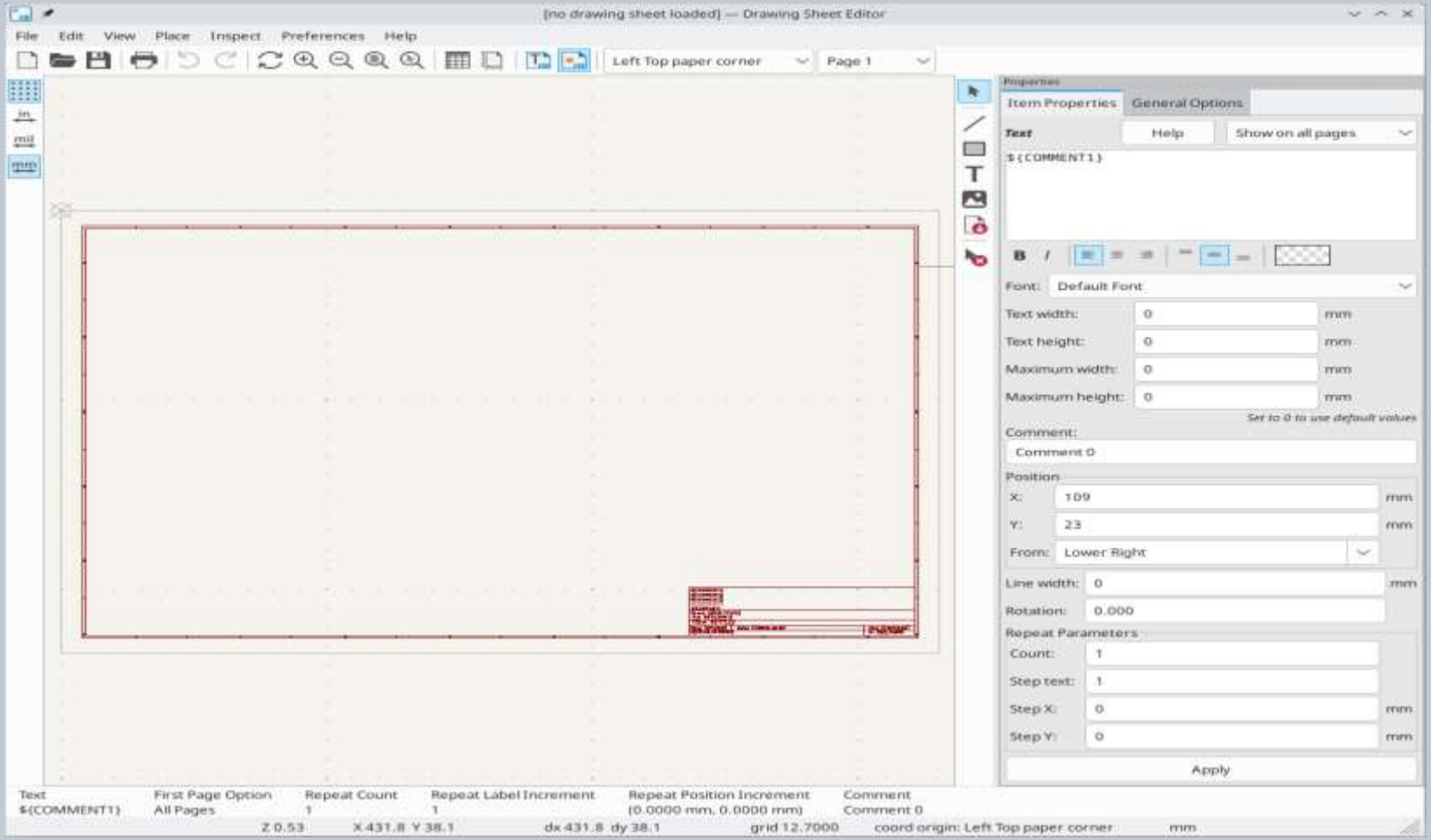


Schematic Diagram



Basic Terminology. WORKSHEET

The frame, title block, and other graphic items (logos) are collectively called a **drawing sheet**

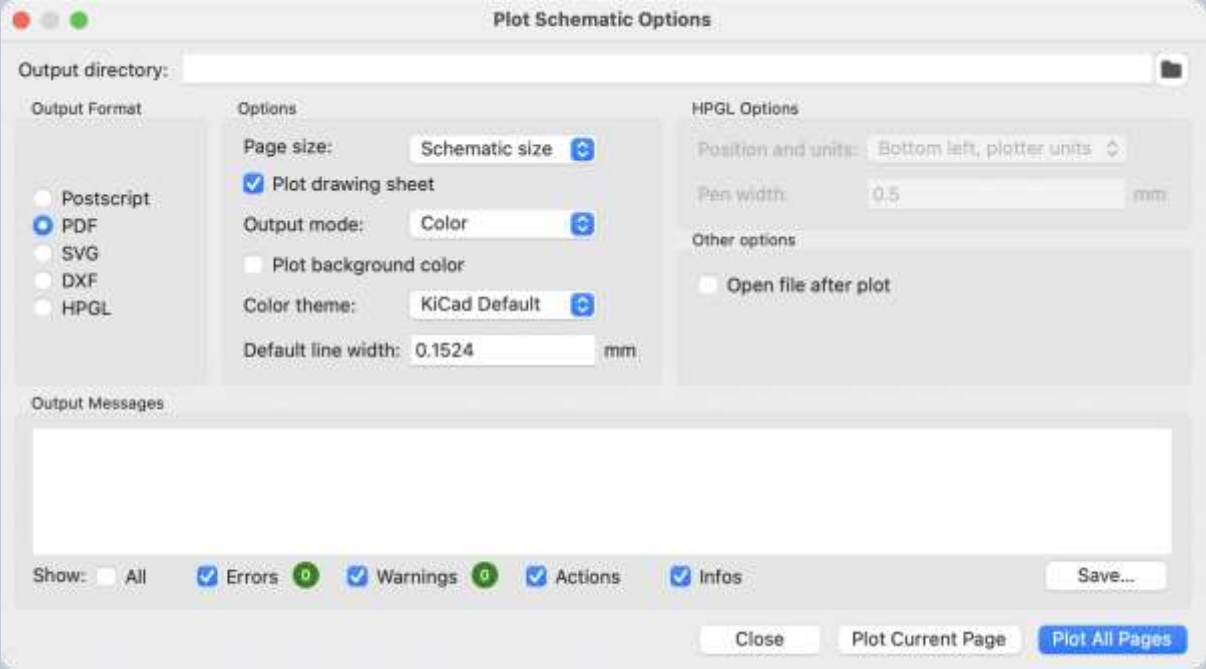


Basic Terminology. PLOTTING

Plotting generates files that represent the PCB design layers, ready for fabrication or inspection.

Purpose:

- To create **Gerber files,** which are standard for PCB manufacturing.
- To produce **drill files** for hole information.
- To export **design layers** for documentation or review.



Basic Terminology. GERBERFILES

Gerber files are standardized files used to represent different layers of a PCB design, such as copper layers, silkscreen, solder mask, and drill holes. These files are essential for manufacturing PCBs.

Purpose:

•To communicate the exact physical design of the PCB to the manufacturer.

•Each Gerber file represents a specific layer or detail of the PCB, ensuring clarity and precision in production.

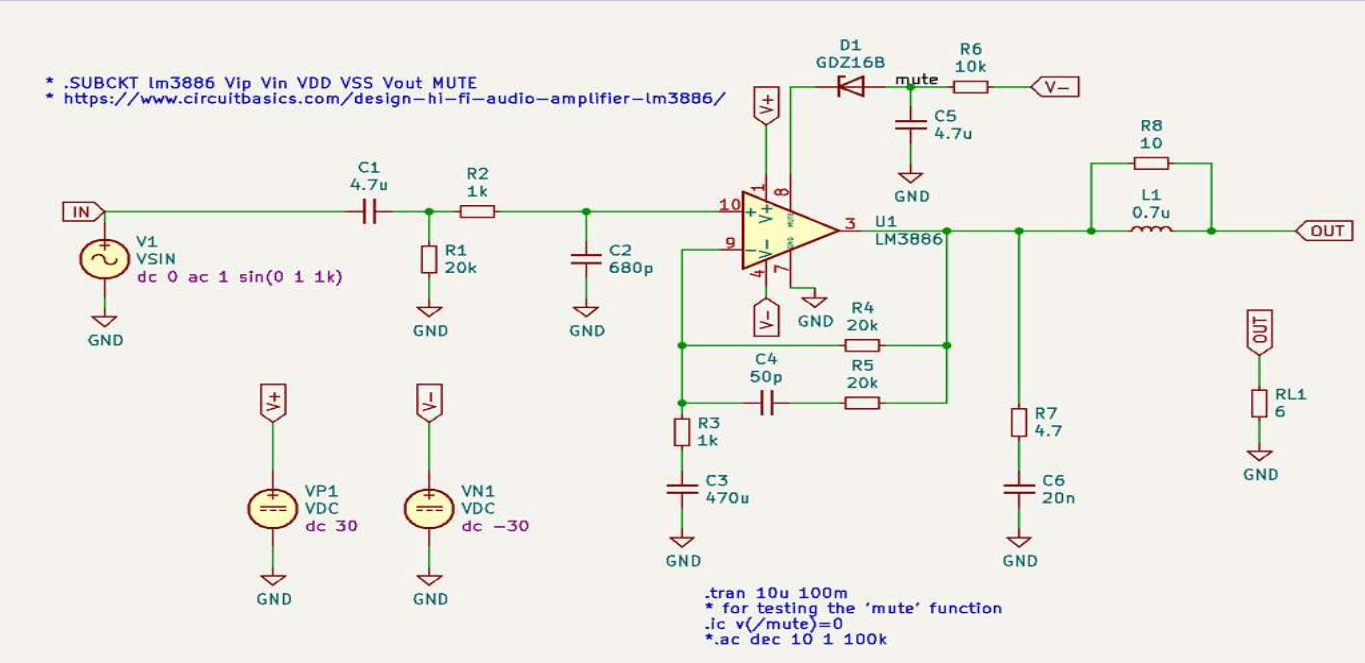
•Eliminates ambiguity by providing standardized data readable by fabrication machines.

Basic Terminology. NG-SPICE

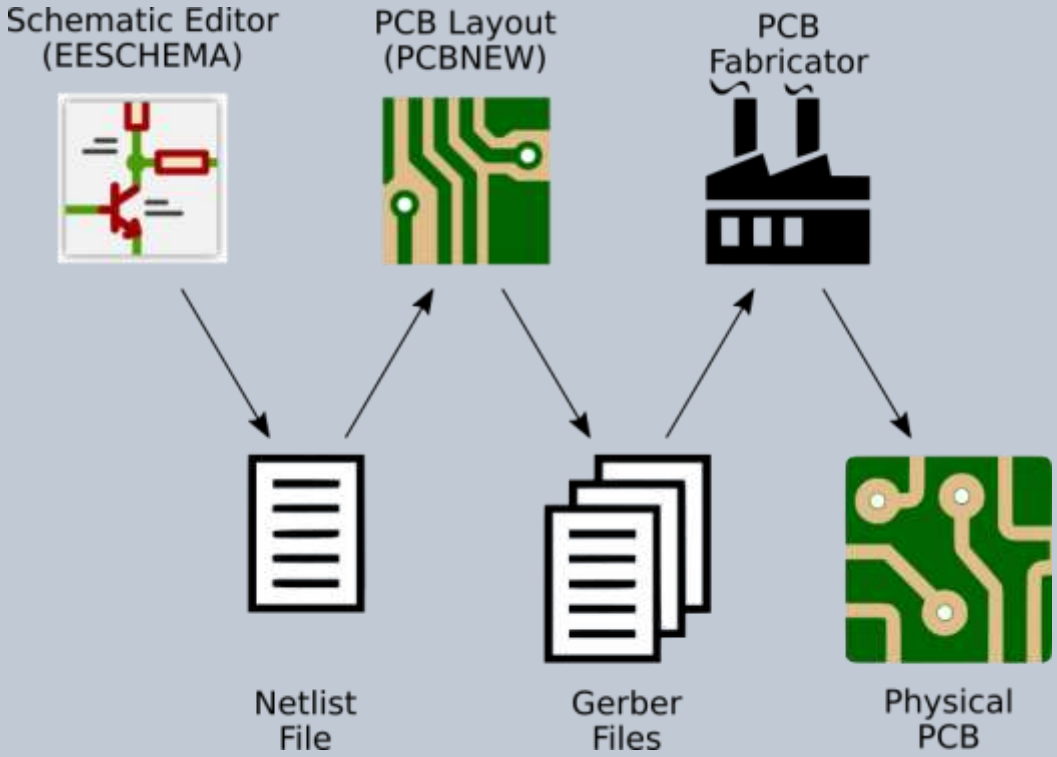
NGSpice is an open-source **circuit simulation tool** integrated with KiCad for analyzing schematic designs.

Purpose:

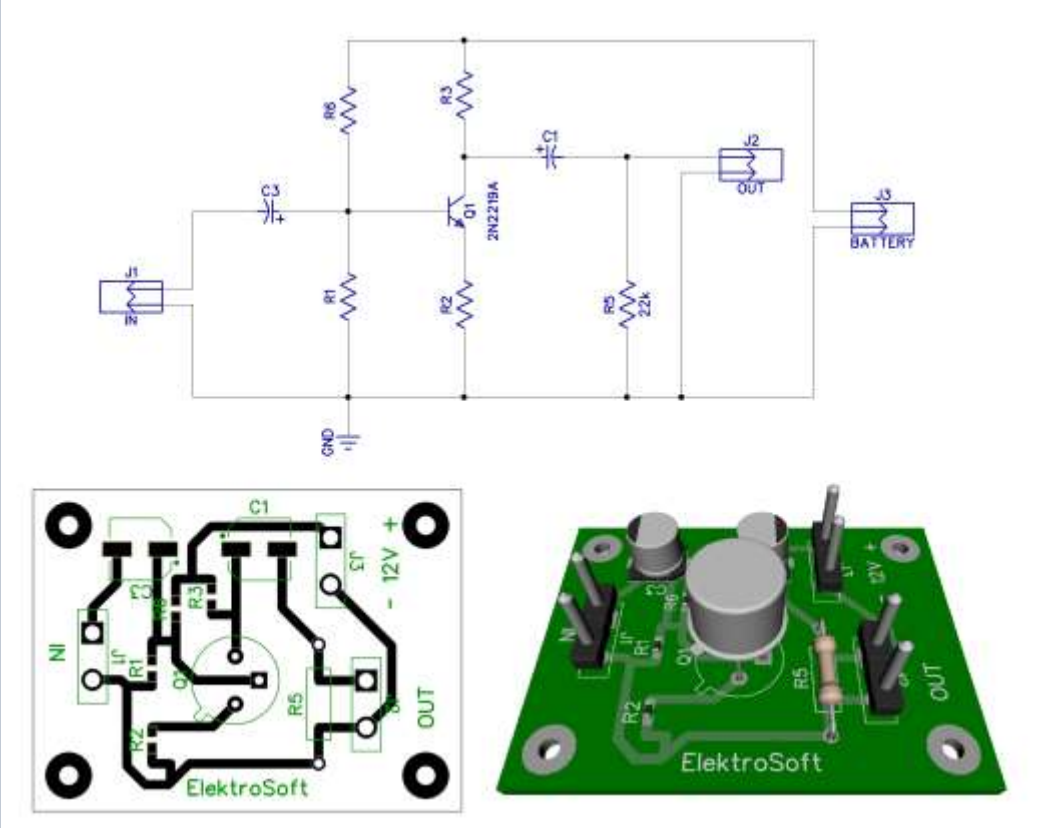
- To simulate and test circuit behavior before hardware implementation.
- To verify the functionality of designs under different conditions.



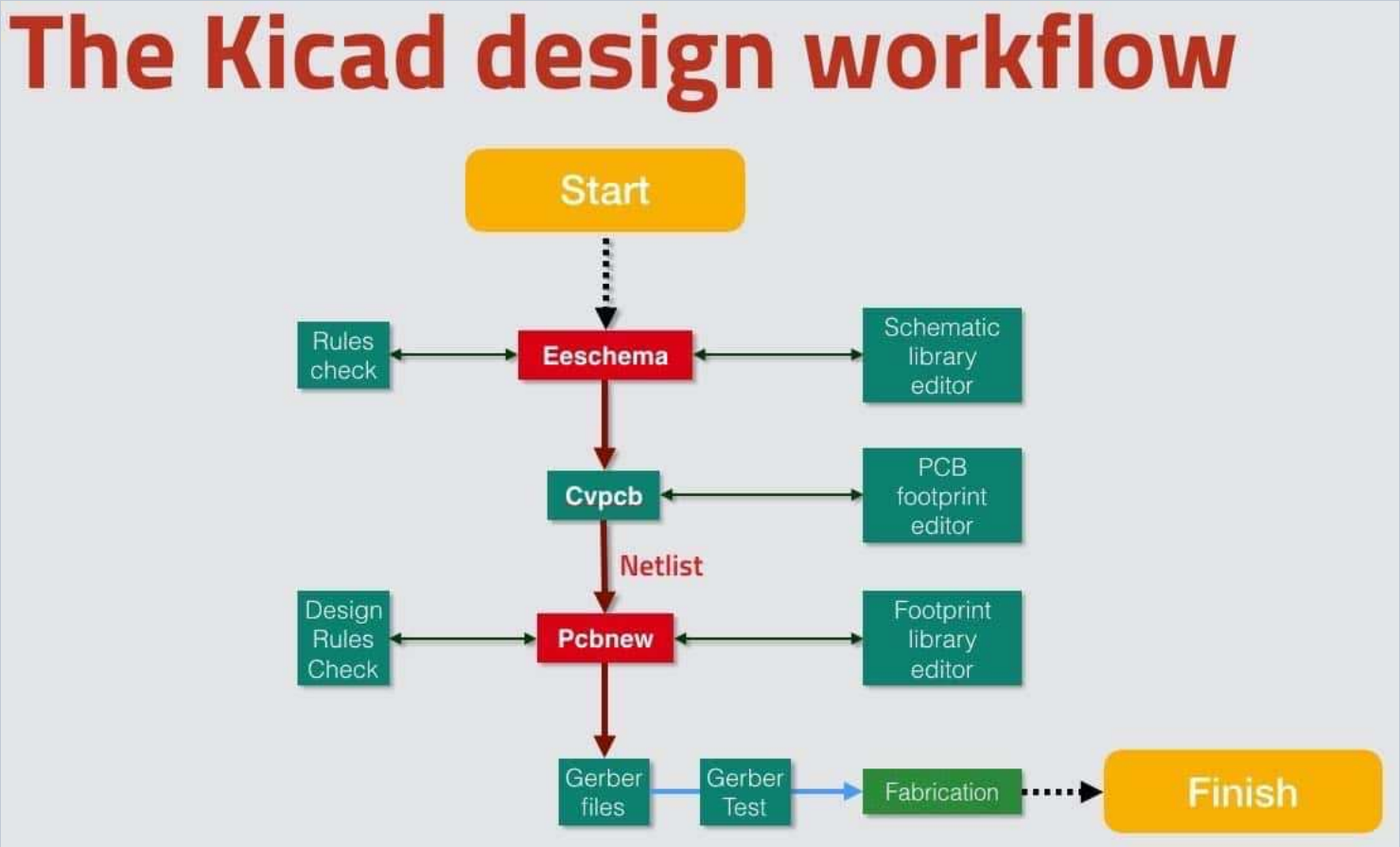
Basic Terminology.



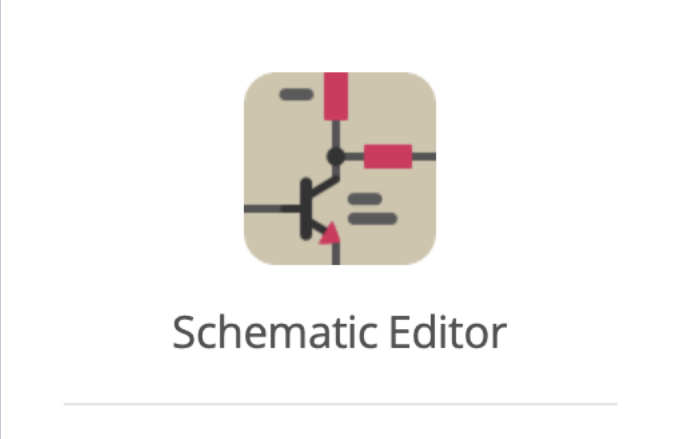
Basic Terminology.



Basic Terminology.



SCHEMATIC EDITOR



LINK

PCB EDITOR

LINK

GERBER VIEW

Gerber Viewer

LINK

DRAWING SHEET EDITOR

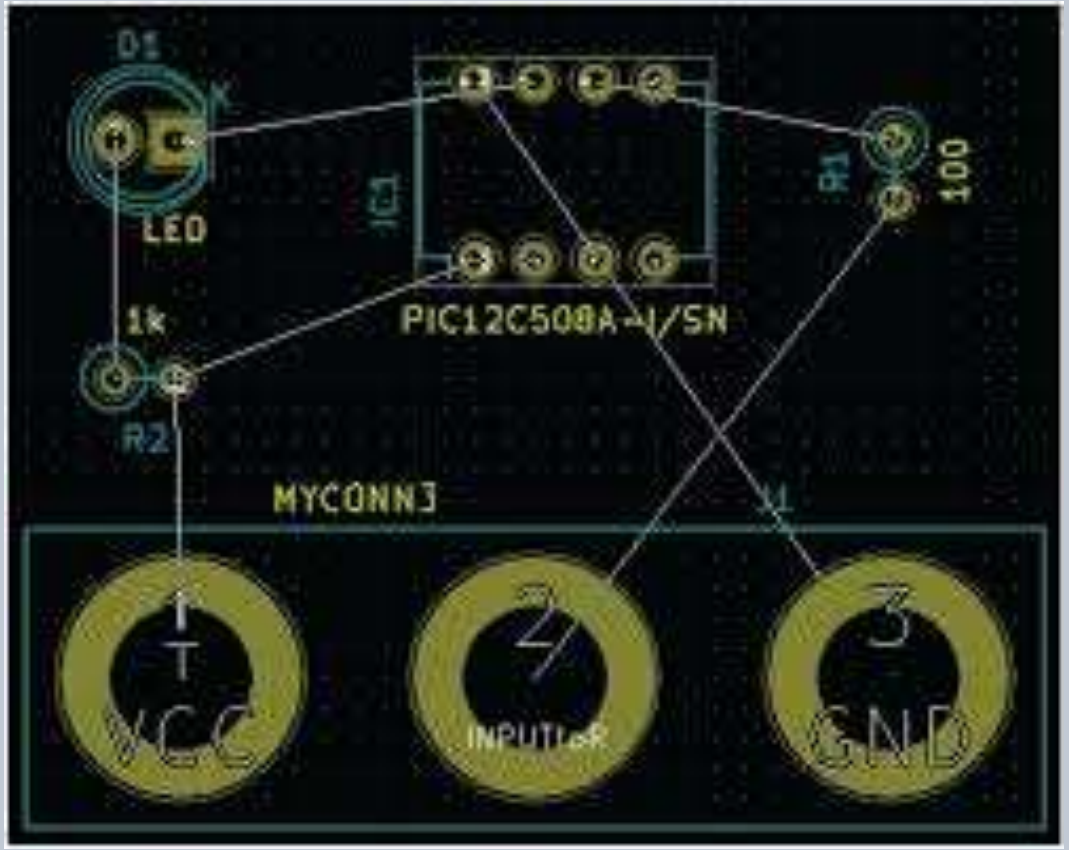
LINK

CALCULATOR TOOLS

Calculator Tools

LINK

DESIGNER RULES



LINK

THANK YOU