

People's Democratic Republic of Algeria

Ministry of Higher Education and Scientific Research

University Mustapha Stambouli of Mascara

Faculty of Science and Technology

Department of Electrical Engineering

Course Handout

Pre-Project Works

Travaux Avant-Projet

Techniques of Electronic Drawing, PCB Realization, Wiring, and Troubleshooting

Level:	Third-Year Bachelor Degree in Electronics
Semester:	Semester 5
Teaching Unit:	UEM 3.1 – Subject 4
Total hourly volume:	37h30
Course / Practical work:	1h30 / 1h00
Credits:	3
Coefficient:	2
Evaluation mode:	Final exam 60%, Continuous assessment 40%

Prepared by: ..ZOUGGARET Abdelhak.....

Academic year: Academic Year 2025–2026

General Introduction

This course handout is intended for third-year Bachelor students in Electronics, Semester 5. It is prepared for the module *Pre-Project Works (Travaux Avant-Projet)*, which belongs to the teaching unit UEM 3.1. The module aims to prepare students for the practical design, realization, testing, and presentation of simple electronic circuits.

At this stage of their training, students have already acquired basic knowledge in electronic components, electrical measurements, and elementary electronic circuits. This module allows them to transform this knowledge into practical skills. The student is progressively guided from the analysis of an electronic function to the choice of components, the preparation of technical drawings, the realization of a printed circuit board, the soldering of components, and the troubleshooting of the final prototype.

The main objective of the module is not only to obtain a working circuit, but also to teach a complete engineering approach. A successful electronic project must be correctly documented, technically justified, carefully assembled, and experimentally verified. For this reason, the course combines theoretical reminders, technological rules, practical examples, and mini-project preparation guidelines.

This handout is organized into four chapters. The first chapter introduces the techniques of drawing in electronics, including block diagrams, detailed schematics, equivalent circuits, component placement drawings, wiring diagrams, definition drawings, and bills of materials. The second chapter focuses on the technology of realizing electronic schematics, with emphasis on the international grid, preliminary prototypes, and component placement rules. The third chapter presents wiring techniques and printed circuit board realization, including PCB materials, artwork preparation, transfer to copper, etching, drilling, inspection, repair, and surface-mount devices. The fourth chapter introduces the basic principles of troubleshooting electronic circuits, component failures, causes of faults, measuring instruments, and systematic testing methods.

By the end of this module, the student should be able to read and prepare basic electronic documentation, select suitable components, design a simple circuit layout, realize a small printed circuit board, solder and test components, detect common faults, and present the completed mini-project in a clear technical report or oral presentation.

Contents

General Introduction	i
1 Techniques of Drawing in Electronics	1
1.1 Introduction	1
1.2 Review of Electronic Components	2
1.2.1 Passive Components	2
1.2.2 Active Components	4
1.3 Principles and Technological Characteristics	5
1.3.1 Nominal Value and Tolerance	5
1.3.2 Power Rating	5
1.3.3 Voltage and Current Limits	6
1.3.4 Package Type	6
1.4 Fields of Application of Electronic Components	6
1.5 Introduction to Drawing in Electronics	6
1.6 Block Diagram	7
1.6.1 Rules for Drawing Block Diagrams	8
1.7 Detailed Circuit Schematic	8
1.7.1 Rules for Drawing Circuit Schematics	8
1.8 Equivalent Circuit Diagram	9
1.9 Component Placement Drawing	9
1.10 Wiring Diagram and PCB Routing	10
1.11 Definition Drawing	11
1.12 Bill of Materials	11
1.13 Example: From Function to Documentation	12
1.13.1 Step 1: Functional Need	12
1.13.2 Step 2: Block Diagram	13
1.13.3 Step 3: Circuit Schematic	13
1.13.4 Step 4: Component Calculation	13
1.13.5 Step 5: Practical Remarks	13
1.14 Common Mistakes in Electronic Drawing	13
1.15 Extended Design Documentation Guidelines	14

1.15.1	Hierarchy of Electronic Documents	14
1.15.2	Document Version Control	15
1.16	Standard Graphic Conventions in Schematics	15
1.16.1	Power and Ground Symbols	16
1.16.2	Net Labels	16
1.16.3	Reference Designators	16
1.17	Functional Specification Before Drawing	17
1.17.1	Example of a Functional Specification	17
1.18	Datasheets as Technical Drawing Support	18
1.18.1	Information Usually Found in a Datasheet	18
1.18.2	Example: Integrated Circuit Pinout	19
1.19	Detailed Example: Documentation of an LED Driver	19
1.19.1	Functional Description	19
1.19.2	Block Diagram	19
1.19.3	Detailed Schematic	20
1.19.4	Component Calculation	20
1.19.5	Documentation Remarks	20
1.20	Design Notes and Technical Annotations	20
1.20.1	Node Labels and Test Points	21
1.20.2	Consistency Between Documents	22
1.21	Using Datasheets During Electronic Drawing	22
1.21.1	Maximum Ratings and Recommended Conditions	22
1.21.2	Preparing a Drawing File	23
1.22	Chapter Summary	24
1.23	Exercises	24
1.24	Mini-Project Preparation Questions	25
2	Technology for Realizing Electronic Schematics	26
2.1	Introduction	26
2.2	From Schematic to Practical Realization	27
2.3	International Grid in Electronics	27
2.3.1	Importance of the Grid	28
2.4	Preliminary Prototypes	28
2.4.1	Breadboard Prototyping	29
2.4.2	Advantages and Limitations of Breadboards	29
2.5	Component Placement: General Rules	29
2.6	Placement of Passive Components	30
2.6.1	Resistors	30
2.6.2	Capacitors	30

2.6.3	Inductors	31
2.7	Placement of Active Components	31
2.7.1	Diodes	31
2.7.2	Transistors	31
2.7.3	Voltage Regulators	31
2.8	Placement of Integrated Circuits	32
2.8.1	Pin Numbering and Orientation	32
2.8.2	Decoupling of Integrated Circuits	32
2.9	Heat Sinks and Thermal Considerations	33
2.10	Placement of Transformers and Power Elements	33
2.10.1	Transformers	33
2.10.2	Power Components	34
2.11	Mechanical and Practical Constraints	34
2.12	Design Example: 5 V Regulated Power Supply	35
2.12.1	Functional Block Diagram	35
2.12.2	Simplified Schematic	35
2.12.3	Recommended Placement	35
2.13	Checklist Before PCB Routing	36
2.14	Common Mistakes in Component Placement	36
2.15	Footprint Selection and Package Verification	37
2.15.1	Difference Between Symbol and Footprint	37
2.15.2	Footprint Verification Checklist	38
2.16	Component Placement Strategies	38
2.16.1	Functional Placement	38
2.16.2	Electrical Placement	38
2.16.3	Thermal Placement	39
2.16.4	Mechanical Placement	39
2.17	Preliminary Prototyping Methodology	39
2.17.1	Prototype Planning	39
2.17.2	Prototype Validation Table	40
2.17.3	Progressive Testing	40
2.18	Power Distribution and Decoupling Rules	40
2.18.1	Power Track Organization	40
2.18.2	Decoupling Capacitors	40
2.19	Case Study: Placement of a Small Regulated Supply	41
2.19.1	Functional Zones	41
2.19.2	Placement Rules for the Example	42
2.20	Advanced Component Placement Considerations	42
2.20.1	Functional Zones	42

2.20.2	Placement According to Energy Flow	42
2.20.3	Design for Assembly	43
2.21	Thermal Derating and Reliability	43
2.21.1	Design for Testability	44
2.21.2	Elementary EMC Rules	44
2.22	Chapter Summary	45
2.23	Exercises	45
3	Wiring Techniques for Electronic Circuits	47
3.1	Introduction	47
3.2	Electronic Wiring Methods	48
3.3	Printed Circuit Board Technology	48
3.3.1	Main Parts of a PCB	48
3.3.2	Single-Sided and Double-Sided PCBs	49
3.4	PCB Materials and Properties	49
3.4.1	Phenolic Paper Boards	49
3.4.2	Fiberglass Epoxy Boards	50
3.4.3	Important PCB Properties	50
3.5	Establishing the PCB Layout	50
3.5.1	General PCB Layout Procedure	50
3.5.2	Track Width	51
3.5.3	Clearance Between Tracks	52
3.6	Artwork and Negative Preparation	52
3.6.1	Manual and Software Methods	52
3.6.2	Example of a Simple PCB Artwork	52
3.7	Transfer to Copper by Photolithography	52
3.7.1	Main Steps of Photolithography	53
3.8	Copper Etching	53
3.8.1	Etching Principle	54
3.8.2	Precautions During Etching	54
3.9	Post-Etching Treatment	54
3.9.1	Cleaning	54
3.9.2	Drilling	54
3.9.3	Tinning	55
3.9.4	Visual Inspection	55
3.10	Electrical Verification of the PCB	55
3.10.1	Continuity Test	55
3.10.2	Short-Circuit Test	55
3.11	Modification and Repair of Printed Circuits	56

3.11.1	Common PCB Faults	56
3.11.2	Repair Techniques	56
3.12	Surface-Mount Device Circuits	57
3.12.1	Advantages of SMD Technology	57
3.12.2	Limitations of SMD Technology	57
3.13	Example: PCB Realization of a Simple LED Indicator	58
3.13.1	Circuit Principle	58
3.13.2	PCB Layout Example	58
3.14	Safety Precautions	58
3.15	Detailed PCB Design Rules	59
3.15.1	Track Width Selection	59
3.15.2	Clearance and Creepage	60
3.15.3	Pad and Hole Dimensions	60
3.16	Grounding and Return Current Paths	60
3.16.1	Ground Return Principle	61
3.16.2	Ground Plane	61
3.16.3	Star Ground	61
3.17	PCB Routing Strategy	62
3.17.1	Recommended Routing Order	62
3.17.2	Avoiding Routing Problems	62
3.17.3	Use of Jumpers on Single-Sided Boards	62
3.18	Manufacturing Files and Real-Scale Verification	63
3.18.1	Real-Scale Printing	63
3.18.2	Gerber and Drill Files	63
3.19	Soldering Quality and Common Defects	64
3.19.1	Good Solder Joint	64
3.19.2	Common Soldering Defects	64
3.19.3	Inspection After Soldering	64
3.20	Design Example: PCB Documentation File	64
3.21	Detailed Design Rule Check Before Fabrication	65
3.21.1	Unconnected Nets and Short Circuits	66
3.22	Copper Areas and Thermal Reliefs	66
3.23	Drilling, Pads, and Mechanical Tolerance	66
3.24	Soldering Quality and First Article Verification	67
3.25	Chapter Summary	68
3.26	Exercises	68
4	Basic Principles of Troubleshooting Electronic Circuits	70
4.1	Introduction	70

4.2	General Troubleshooting Approach	71
4.3	Component Failures	72
4.3.1	Open-Circuit Failure	72
4.3.2	Short-Circuit Failure	72
4.3.3	Parameter Drift	73
4.3.4	Intermittent Failure	73
4.4	Failures of Common Components	73
4.4.1	Resistor Failures	73
4.4.2	Capacitor Failures	74
4.4.3	Diode Failures	74
4.4.4	Transistor Failures	74
4.4.5	Integrated Circuit Failures	75
4.5	Causes of Electronic Circuit Failures	75
4.5.1	Electrical Stress	75
4.5.2	Thermal Stress	76
4.5.3	Environmental Stress	76
4.5.4	Assembly Errors	76
4.6	Measuring Instruments for Troubleshooting	77
4.6.1	Digital Multimeter	77
4.6.2	Oscilloscope	77
4.6.3	Laboratory Power Supply	78
4.6.4	Function Generator	78
4.6.5	Logic Probe	78
4.7	Measurement Methods	78
4.7.1	Visual Inspection	78
4.7.2	Continuity Test	78
4.7.3	Resistance Measurement	79
4.7.4	Voltage Measurement	79
4.7.5	Current Measurement	79
4.7.6	Signal Measurement with Oscilloscope	80
4.8	Systematic Troubleshooting Methods	80
4.8.1	Input-to-Output Method	80
4.8.2	Output-to-Input Method	80
4.8.3	Half-Split Method	81
4.8.4	Substitution Method	81
4.9	Troubleshooting Example: LED Indicator Circuit	81
4.9.1	Possible Causes	81
4.9.2	Diagnostic Procedure	82
4.9.3	Expected Measurements	82

4.10	Troubleshooting Example: 5 V Regulated Power Supply	82
4.10.1	Possible Causes	83
4.10.2	Testing Strategy	83
4.11	Safety Rules During Troubleshooting	84
4.12	Troubleshooting Checklist	84
4.13	Common Troubleshooting Mistakes	85
4.14	First Power-Up Procedure	85
4.14.1	Before Applying Power	85
4.14.2	Using Current Limitation	86
4.14.3	Progressive Power-Up	86
4.15	Fault Classification by Circuit Function	86
4.15.1	Power Supply Faults	86
4.15.2	Amplifier Faults	87
4.15.3	Digital Circuit Faults	87
4.15.4	Sensor Interface Faults	87
4.16	Fault Tree Reasoning	87
4.16.1	Example: LED Does Not Turn ON	87
4.16.2	Prioritizing Causes	88
4.17	Measurement Records and Diagnosis Tables	89
4.17.1	Interpreting Measurements	89
4.17.2	Avoiding Measurement Errors	89
4.18	Troubleshooting Analog and Digital Signals	89
4.18.1	Analog Signals	89
4.18.2	Digital Signals	90
4.19	Maintenance Report After Repair	90
4.20	Detailed Diagnosis of Power Supply Faults	91
4.21	Signal Tracing in Analog Circuits	91
4.21.1	Clipping and Saturation	92
4.22	Signal Tracing in Digital Circuits	92
4.23	Intermittent Faults and Preventive Maintenance	92
4.24	Chapter Summary	94
4.25	Exercises	94

I Practical Teaching Materials and Project Support 96

5 Technical Appendix 97

5.1	Appendix A: Common Component Reference Designators	97
5.2	Appendix B: Practical Checklist Before Powering a Circuit	97
5.3	Appendix C: Common Laboratory Instruments	98

5.4	Appendix D: Student Self-Evaluation Sheet	98
5.5	Appendix E: Recommended Rules for Good Documentation	99
5.6	Appendix F: Resistor Color Code	99
5.7	Appendix G: Preferred Values and Practical Component Choice	100
5.8	Appendix H: Common Schematic and PCB Checklist	100
5.9	Appendix I: Glossary of Basic Terms	101
5.10	Appendix J: Common Symbols Used in Basic Electronic Drawings	102
5.11	Appendix K: Practical Reading Guide for Component Markings	102
5.12	Appendix L: Suggested Structure of a Technical Report	103
5.13	Appendix M: Vocabulary for Practical Electronics	103
5.14	Appendix N: Additional Verification Tables	104
5.15	Appendix O: Useful Formula Sheet for Pre-Project Work	106
5.15.1	Basic Electrical Relations	106
5.15.2	Filtering and Time Constants	106
5.15.3	Linear Regulator Dissipation	107
5.16	Appendix P: Quick Diagnosis Matrices	107
	General Conclusion	109
	References	111

Chapter 1

Techniques of Drawing in Electronics

Learning objectives

- Recall the main passive and active electronic components and their basic functions.
- Understand the difference between block diagrams, circuit schematics, equivalent diagrams, layout drawings, wiring diagrams, definition drawings, and bills of materials.
- Learn the basic rules used in electronic drawing and documentation.
- Prepare students for the design and realization of simple electronic circuits on printed circuit boards.

1.1 Introduction

Electronic design is not limited to choosing components and connecting them together. A good electronic project must be clearly represented, documented, checked, and prepared for practical realization. For this reason, technical drawing in electronics is an essential step between the theoretical idea and the physical circuit.

In electronic engineering, several types of drawings and documents are commonly used. Each one has a specific role. A block diagram explains the general function of a system. A circuit schematic shows the electrical connections between components. A layout drawing shows the physical position of components on a board. A wiring diagram helps during assembly and troubleshooting. Finally, a bill of materials lists all the components required for the realization of the circuit.

This chapter introduces the basic techniques of electronic drawing. It also reviews the main categories of electronic components and their technological characteristics.

1.2 Review of Electronic Components

Electronic components are generally classified into two main families: passive components and active components.

1.2.1 Passive Components

Passive components do not generate energy and do not provide power gain. They can store, dissipate, or transfer electrical energy. The most common passive components are resistors, capacitors, inductors, switches, connectors, and transformers.

Resistors

A resistor is used to limit current, divide voltage, polarize active components, or dissipate energy as heat. The ideal relationship between voltage, current, and resistance is given by Ohm's law:

$$V = RI \tag{1.1}$$

where V is the voltage in volts, I is the current in amperes, and R is the resistance in ohms.

The main technological characteristics of a resistor are:

- nominal resistance value, expressed in ohms Ω ;
- tolerance, expressed as a percentage;
- rated power, expressed in watts;
- temperature coefficient;
- package type, such as through-hole or surface-mount device.

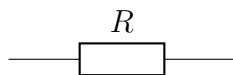


Figure 1.1: Electrical symbol of a resistor.

Capacitors

A capacitor stores electrical energy in an electric field. It is widely used for filtering, coupling, decoupling, timing circuits, and energy storage.

The current-voltage relationship of an ideal capacitor is:

$$i(t) = C \frac{dv(t)}{dt} \quad (1.2)$$

where C is the capacitance in farads.

The main characteristics of a capacitor are:

- capacitance value;
- rated voltage;
- tolerance;
- dielectric material;
- polarity for electrolytic capacitors;
- equivalent series resistance, especially in power and filtering applications.



Figure 1.2: Symbols of a non-polarized capacitor and a polarized capacitor.

Inductors

An inductor stores energy in a magnetic field. It is used in filters, oscillators, power supplies, converters, and electromagnetic applications.

The voltage-current relationship of an ideal inductor is:

$$v(t) = L \frac{di(t)}{dt} \quad (1.3)$$

where L is the inductance in henrys.

The main characteristics of an inductor are:

- inductance value;
- maximum current;
- winding resistance;
- core material;
- saturation current;
- operating frequency range.



Figure 1.3: Electrical symbol of an inductor.

1.2.2 Active Components

Active components can control current or voltage and may provide amplification or switching. They usually require a power supply to operate. Examples include diodes, transistors, operational amplifiers, integrated circuits, voltage regulators, sensors, and microcontrollers.

Diodes

A diode is a semiconductor component that allows current to flow mainly in one direction. It is commonly used for rectification, protection, signal detection, and voltage regulation.



Figure 1.4: Symbols of a diode and a Zener diode.

The main diode characteristics are:

- forward voltage drop;
- maximum forward current;
- reverse breakdown voltage;
- switching speed;
- power dissipation.

Transistors

Transistors are used as switches or amplifiers. The two most common families are bipolar junction transistors and field-effect transistors.



Figure 1.5: Symbols of NPN and PNP bipolar transistors.

Important transistor parameters include:

- maximum collector-emitter voltage;
- maximum collector current;
- current gain β for bipolar transistors;
- threshold voltage for MOSFETs;
- power dissipation;
- switching frequency.

Integrated Circuits

An integrated circuit is a complete electronic function integrated into a small package. It may contain analog circuits, digital circuits, or mixed-signal circuits. Examples include operational amplifiers, timers such as the NE555, logic gates, voltage regulators, drivers, converters, and microcontrollers.

When using an integrated circuit, the designer must carefully consult the datasheet. The datasheet provides the pinout, electrical limits, recommended operating conditions, typical application circuits, timing diagrams, and package dimensions.

1.3 Principles and Technological Characteristics

Choosing a component is not only based on its nominal value. The designer must also consider technological characteristics, environmental constraints, and practical availability.

1.3.1 Nominal Value and Tolerance

The nominal value is the theoretical value indicated by the manufacturer. The real value may be slightly different due to manufacturing tolerances. For example, a resistor of 1 k Ω with a tolerance of 5% may have a real value between:

$$950 \, \Omega \leq R \leq 1050 \, \Omega \quad (1.4)$$

Tolerance is important because it affects the accuracy and stability of the circuit.

1.3.2 Power Rating

Every component has a maximum power that it can dissipate without damage. For a resistor, the dissipated power is:

$$P = RI^2 = \frac{V^2}{R} \quad (1.5)$$

A safety margin is generally required. For example, if a resistor must dissipate 0.3 W, it is better to choose a resistor rated at 0.5 W or 1 W.

1.3.3 Voltage and Current Limits

Components must be selected according to the maximum voltages and currents present in the circuit. Exceeding these limits may lead to overheating, malfunction, or permanent damage.

1.3.4 Package Type

The package is the physical form of the component. The two common mounting technologies are through-hole technology and surface-mount technology.

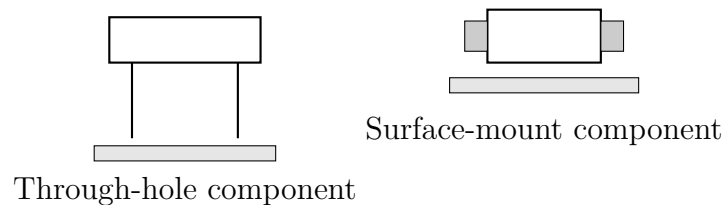


Figure 1.6: Simplified comparison between through-hole and surface-mount components.

Through-hole components are easier to solder manually and are often used for training and prototyping. Surface-mount components allow compact circuits and are widely used in modern electronic products.

1.4 Fields of Application of Electronic Components

Electronic components are used in many fields. The same component may have different roles depending on the circuit. Table 1.1 gives examples of common applications.

1.5 Introduction to Drawing in Electronics

Electronic drawing is the graphical representation of circuits and systems. It allows the designer to communicate ideas, analyze operation, prepare manufacturing, and facilitate troubleshooting.

A complete electronic project generally contains several documents:

- a block diagram;

Table 1.1: Examples of component applications.

Component	Common applications
Resistor	Current limitation, voltage divider, biasing, pull-up and pull-down networks.
Capacitor	Filtering, coupling, decoupling, timing circuits, energy storage.
Inductor	Filters, DC-DC converters, electromagnetic devices.
Diode	Rectification, protection, voltage regulation, signal detection.
Transistor	Switching, amplification, motor driving, signal control.
Operational amplifier	Signal amplification, filtering, comparison, instrumentation circuits.
Integrated circuit	Logic functions, timing, regulation, control, signal processing.

- a detailed circuit schematic;
- an equivalent circuit when needed for analysis;
- a component placement drawing;
- a wiring diagram or PCB routing drawing;
- a definition drawing for mechanical details;
- a bill of materials.

Each document must be clear, readable, and consistent with the others.

1.6 Block Diagram

A block diagram is a simplified representation of a system. Each block represents a function, and arrows show the flow of signals or energy. The block diagram is useful at the beginning of a project because it helps the designer understand the general structure before drawing the detailed schematic.

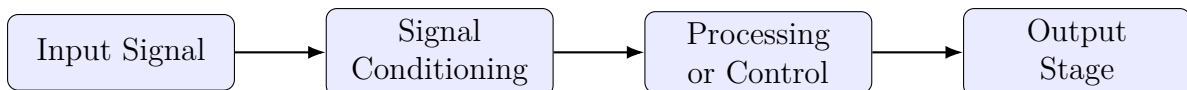


Figure 1.7: Example of a simple electronic system block diagram.

1.6.1 Rules for Drawing Block Diagrams

The following rules should be respected:

- use simple rectangular blocks;
- write the function of each block, not the component reference;
- arrange blocks according to the direction of signal flow;
- use arrows to indicate the direction of information or energy;
- avoid excessive details.

1.7 Detailed Circuit Schematic

A detailed circuit schematic shows the electrical connections between components using standardized symbols. It is the main document used for analysis, simulation, troubleshooting, and PCB design.

Unlike a physical drawing, a schematic does not necessarily show the real position of components. Its purpose is to make the electrical operation clear.

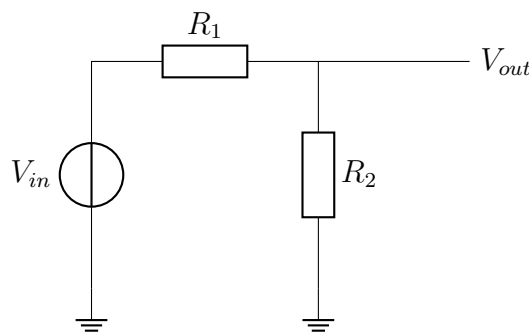


Figure 1.8: Example of a voltage divider schematic.

For the voltage divider shown in Figure 1.8, the output voltage is:

$$V_{out} = V_{in} \frac{R_2}{R_1 + R_2} \quad (1.6)$$

1.7.1 Rules for Drawing Circuit Schematics

A good schematic must follow these principles:

- use standard symbols;
- place the power supply at the top and ground at the bottom when possible;
- arrange the signal flow from left to right;

- avoid crossing wires unnecessarily;
- use component references such as R_1 , C_1 , D_1 , Q_1 , and U_1 ;
- indicate component values clearly;
- label important nodes and signals;
- separate functional blocks visually.

1.8 Equivalent Circuit Diagram

An equivalent circuit is a simplified model used for analysis. It replaces a real component or a part of a circuit by an ideal or simplified representation.

Equivalent diagrams are useful because real components are not ideal. For example, a real capacitor may be modeled by an ideal capacitor in series with a small resistance called equivalent series resistance.

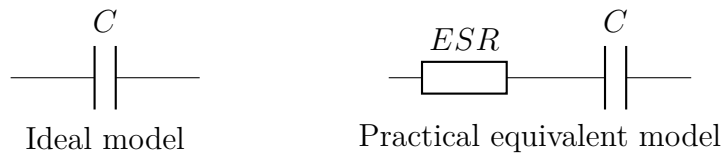


Figure 1.9: Ideal and practical equivalent model of a capacitor.

Another important example is the Thevenin equivalent circuit, where a complex linear network is replaced by a voltage source in series with a resistance.

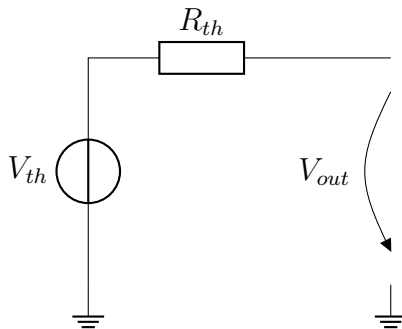


Figure 1.10: Thevenin equivalent circuit.

1.9 Component Placement Drawing

The component placement drawing, also called the implantation drawing, shows the physical position of the components on the printed circuit board. It is used during assembly, soldering, verification, and maintenance.

This drawing must contain:

- the board outline;
- the position of each component;
- the component references;
- the orientation of polarized components;
- connector positions;
- holes, mounting points, and mechanical constraints.

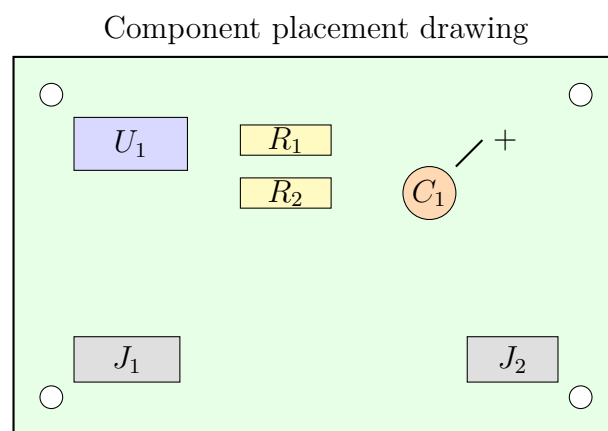


Figure 1.11: Simplified example of a component placement drawing on a PCB.

1.10 Wiring Diagram and PCB Routing

A wiring diagram shows how the physical electrical connections must be made. In a printed circuit board, the wiring is replaced by copper tracks. PCB routing represents the paths of these copper tracks.

The routing must respect electrical and technological constraints such as:

- track width according to current;
- distance between tracks according to voltage;
- short and direct paths for sensitive signals;
- separation between power and low-level signal tracks;
- correct grounding strategy;
- thermal considerations for power components.

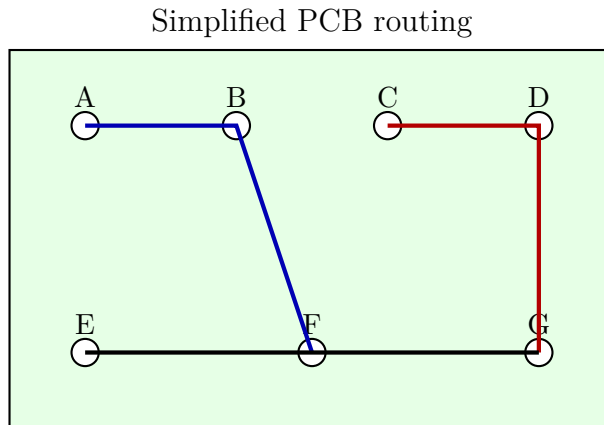


Figure 1.12: Simplified representation of copper tracks on a PCB.

1.11 Definition Drawing

A definition drawing gives precise mechanical and dimensional information. In electronic projects, it may be used to define the printed circuit board dimensions, hole diameters, connector positions, enclosure dimensions, or mounting constraints.

A definition drawing must be accurate and should include:

- dimensions in millimeters;
- hole diameters;
- tolerances if needed;
- board thickness;
- mechanical references;
- scale and title block.

1.12 Bill of Materials

The bill of materials, usually abbreviated as BOM, is a table that lists all components and materials required for the realization of the circuit. It is essential for purchasing, assembly, verification, and maintenance.

A good BOM must contain:

- component reference;
- quantity;
- component value;

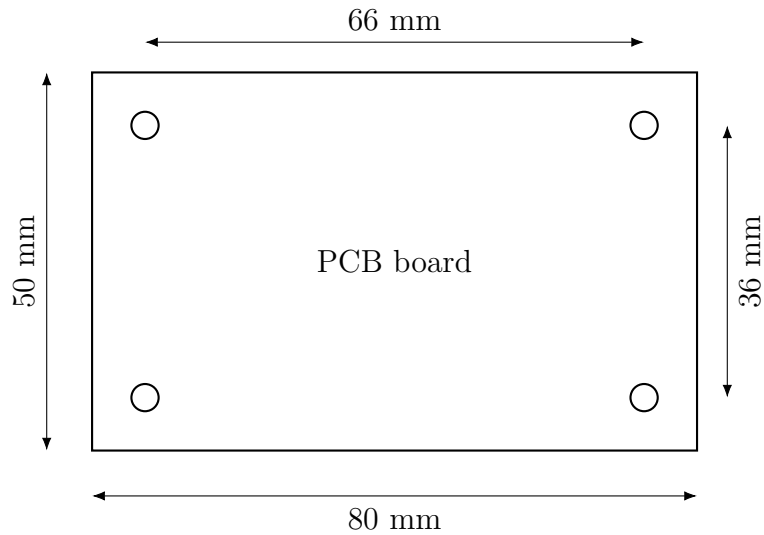


Figure 1.13: Example of a simple PCB definition drawing with dimensions.

- description;
- package;
- voltage, current, or power rating if necessary;
- manufacturer reference when available.

Table 1.2: Example of a bill of materials for a simple voltage divider circuit.

Reference	Qty.	Value	Description	Package
R_1	1	10 k Ω	Resistor, 1/4 W, 5%	Through-hole
R_2	1	4.7 k Ω	Resistor, 1/4 W, 5%	Through-hole
J_1	1	2 pins	Input connector	Terminal block
J_2	1	2 pins	Output connector	Terminal block

1.13 Example: From Function to Documentation

Let us consider a simple electronic function: obtaining a lower DC voltage from a higher DC voltage using a resistive divider.

1.13.1 Step 1: Functional Need

The objective is to obtain an output voltage V_{out} lower than the input voltage V_{in} .

1.13.2 Step 2: Block Diagram

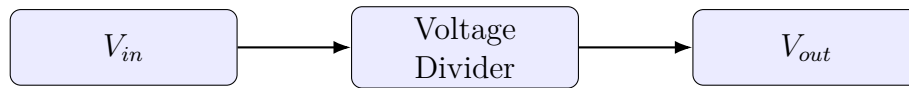


Figure 1.14: Block diagram of the voltage divider function.

1.13.3 Step 3: Circuit Schematic

The electrical schematic is shown in Figure 1.8. It contains two resistors connected in series.

1.13.4 Step 4: Component Calculation

If the desired output voltage is 5 V and the input voltage is 12 V, the resistor ratio must satisfy:

$$\frac{R_2}{R_1 + R_2} = \frac{5}{12} \quad (1.7)$$

A possible choice is $R_1 = 6.8 \text{ k}\Omega$ and $R_2 = 4.7 \text{ k}\Omega$. Then:

$$V_{out} = 12 \times \frac{4.7}{6.8 + 4.7} = 4.90 \text{ V} \quad (1.8)$$

This result is close to 5 V and may be acceptable depending on the application.

1.13.5 Step 5: Practical Remarks

A voltage divider is suitable for signal conditioning but it should not be used as a power supply for a load requiring significant current. When a load is connected to the output, it modifies the equivalent resistance and changes the output voltage.

1.14 Common Mistakes in Electronic Drawing

Students often make mistakes that reduce the clarity or reliability of their project documentation. The most common mistakes are:

- drawing components without references;
- forgetting ground symbols or power supply labels;
- using non-standard symbols;
- crossing too many wires in the schematic;

- confusing schematic drawing with physical layout;
- forgetting the polarity of diodes and electrolytic capacitors;
- not indicating component values;
- preparing an incomplete bill of materials;
- ignoring current, voltage, and power limits;
- placing connectors without clear pin numbering.

1.15 Extended Design Documentation Guidelines

In a complete electronic project, documentation must be considered as part of the design itself. A circuit that works experimentally but is not documented cannot be easily reproduced, corrected, or improved. For this reason, students must learn to prepare documents that are clear enough for another person to understand the circuit without asking the designer for explanations.

Good documentation answers four basic questions:

1. What is the function of the circuit?
2. How is this function obtained electrically?
3. Which components are used and why were they selected?
4. How can the circuit be assembled, tested, and repaired?

The level of detail depends on the objective. A preliminary notebook may contain sketches and approximate values, while a final technical report must contain clean schematics, justified calculations, component references, measurements, and clear conclusions.

1.15.1 Hierarchy of Electronic Documents

Electronic documents can be organized according to their level of abstraction. The block diagram is the highest-level document. It explains the function of each part without showing component details. The schematic is more precise because it shows components and electrical connections. The PCB layout is physical because it represents the real board. The bill of materials is logistical because it identifies what must be purchased and assembled.

Table 1.3: Hierarchy of electronic project documents.

Document	Main purpose	Typical use
Functional specification	Defines the need and operating conditions	Project definition and validation
Block diagram	Shows the main functions	General understanding of the system
Schematic diagram	Shows electrical connections	Analysis, simulation, and troubleshooting
Equivalent model	Simplifies a circuit for calculation	Theoretical study and dimensioning
Placement drawing	Shows component positions	Assembly and inspection
PCB layout	Shows copper tracks and pads	Manufacturing and routing verification
Bill of materials	Lists all components	Purchasing, assembly, and maintenance
Test sheet	Records measurements	Experimental validation

1.15.2 Document Version Control

During a project, the circuit may change several times. For example, a resistor value may be modified after testing, a connector may be replaced, or an error in the PCB layout may be corrected. If these changes are not recorded, the documentation becomes inconsistent.

A simple version table should be included in serious project files. It may contain the date, the version number, the author, and a short description of the modification.

Table 1.4: Example of a simple version history table.

Version	Date	Author	Modification
V1.0	10/02/2026	Student group	First schematic proposal
V1.1	17/02/2026	Student group	Correction of LED polarity and resistor value
V1.2	24/02/2026	Student group	Addition of input protection diode
V2.0	03/03/2026	Student group	Final PCB layout prepared

1.16 Standard Graphic Conventions in Schematics

A schematic is not a drawing of the physical circuit. It is a logical representation of electrical connections. Therefore, its readability depends on conventions. A clear schematic must be organized, labeled, and drawn according to common habits used in electronics.

1.16.1 Power and Ground Symbols

Power supply symbols should be placed clearly. Positive supplies are usually shown at the top of the schematic, while ground symbols are usually shown at the bottom. This convention helps the reader understand the direction of energy distribution.

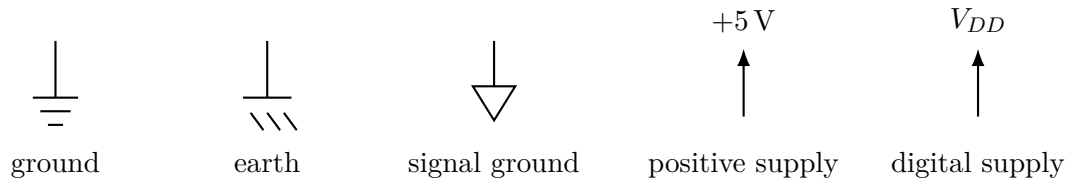


Figure 1.15: Examples of supply and ground symbols used in electronic schematics.

In mixed circuits, the designer may distinguish between analog ground, digital ground, and power ground. These grounds may be connected at a single point in order to reduce noise circulation.

1.16.2 Net Labels

A net label is a name assigned to an electrical node. It is useful when two points are electrically connected but not drawn with a continuous wire. Net labels reduce schematic complexity and avoid too many wire crossings.

Common examples include:

- V_{CC} , +5 V, +12 V for power supplies;
- GND for the reference node;
- IN, OUT, CLK, RESET, ENABLE for signals;
- SDA and SCL for an I2C communication bus;
- TX and RX for serial communication.

Important note. Net labels must be used carefully. If two labels have exactly the same name, most CAD software tools consider them electrically connected even if no wire is drawn between them. A spelling error may therefore create a missing connection or an unwanted connection.

1.16.3 Reference Designators

Each component in a schematic must have a unique reference designator. The reference designator identifies the component in the schematic, PCB layout, bill of materials, and test procedure.

Table 1.5: Common reference designators in electronics.

Prefix	Component family	Examples
R	Resistor	R_1, R_2, R_{LED}
C	Capacitor	C_1, C_2, C_{dec}
L	Inductor	L_1, L_2
D	Diode or LED	$D_1, D_2, LED1$
Q	Transistor	Q_1, Q_2
U	Integrated circuit	U_1, U_2
J	Connector	J_1, J_2
SW	Switch	SW_1, SW_2
F	Fuse	F_1
TP	Test point	TP_1, TP_2

1.17 Functional Specification Before Drawing

Before drawing a schematic, the designer should write a short functional specification. This document does not need to be long, but it must define the project clearly. It avoids designing a circuit without knowing the required input, output, supply voltage, current, precision, and operating conditions.

For a student project, the functional specification may include:

- title of the circuit;
- objective of the circuit;
- input quantities;
- output quantities;
- supply voltage and maximum current;
- expected operating range;
- main components to be used;
- safety restrictions;
- expected tests and validation criteria.

1.17.1 Example of a Functional Specification

Consider a simple temperature alarm circuit. The function is to turn ON an LED and activate a buzzer when the temperature exceeds a chosen threshold.

A block diagram can then be prepared before the detailed schematic.

Table 1.6: Example of functional specification for a temperature alarm.

Item	Specification
Function	Detect an excessive temperature and generate visual and audible alarms
Input	Temperature sensor voltage
Output	LED indicator and buzzer activation
Supply	+5 V DC
Threshold	Adjustable using a potentiometer
Main components	Sensor, comparator, potentiometer, transistor, LED, buzzer
Validation	LED and buzzer must activate when temperature exceeds the threshold

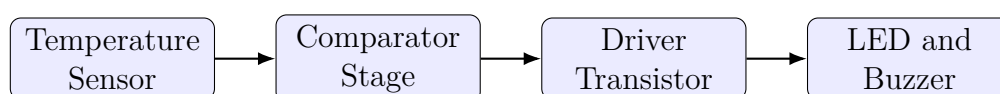


Figure 1.16: Block diagram of a simple temperature alarm circuit.

1.18 Datasheets as Technical Drawing Support

A datasheet is an official technical document provided by the component manufacturer. It is not only useful for electrical calculations; it is also necessary for drawing and layout. The datasheet gives the symbol, pinout, package dimensions, recommended external components, thermal limitations, and typical application circuits.

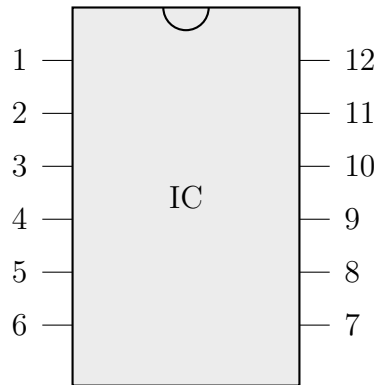
1.18.1 Information Usually Found in a Datasheet

A student should be able to locate at least the following information:

- component name and family;
- pin configuration;
- maximum ratings;
- recommended operating conditions;
- typical electrical characteristics;
- package dimensions;
- application circuits;
- timing diagrams for digital components;
- thermal resistance for power components.

1.18.2 Example: Integrated Circuit Pinout

When an integrated circuit is drawn in the schematic, each pin must correspond to the real pin number of the physical package. Reversing two pins in the schematic may produce a PCB that is electrically wrong even if the drawing looks clean.



pin numbering must match the datasheet

Figure 1.17: Example of pin numbering for a dual in-line integrated circuit.

1.19 Detailed Example: Documentation of an LED Driver

This example shows how a simple circuit can be documented progressively. The chosen function is to control an LED using a transistor. The input is a low-current control signal and the output is the LED current.

1.19.1 Functional Description

The circuit must turn ON an LED when the control input is HIGH. The transistor is used as a switch because the control source may not be able to provide enough current directly to the LED.

1.19.2 Block Diagram

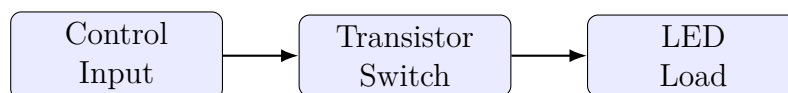


Figure 1.18: Block diagram of a transistor LED driver.

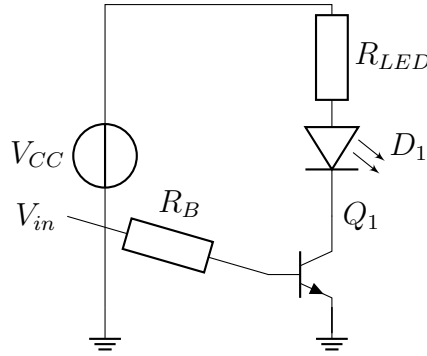


Figure 1.19: Schematic of a transistor LED driver.

1.19.3 Detailed Schematic

1.19.4 Component Calculation

If $V_{CC} = 5$ V, the LED forward voltage is approximately 2 V, and the desired current is 10 mA, the LED resistor is:

$$R_{LED} = \frac{V_{CC} - V_F - V_{CE(sat)}}{I_{LED}} \quad (1.9)$$

Assuming $V_{CE(sat)} = 0.2$ V:

$$R_{LED} = \frac{5 - 2 - 0.2}{0.01} = 280 \, \Omega \quad (1.10)$$

A standard value of $330 \, \Omega$ may be selected to limit the current slightly below 10 mA.

1.19.5 Documentation Remarks

The final documentation must mention the selected standard resistor value, the transistor reference, the LED type, and the test conditions. If the measured LED current is different from the calculated value, the difference should be explained by component tolerances and transistor saturation voltage.

1.20 Design Notes and Technical Annotations

A schematic becomes more useful when it is accompanied by design notes. These notes explain why each part of the circuit is used and how the values of the components were selected. For students, this step is very important because it connects the theoretical course with practical realization. A correct drawing must therefore include not only symbols and wires, but also the reasoning behind the design.

A design note generally contains the function of the circuit, the input and output quantities, the main equations, the selected standard component values, the expected

measurements, and the limits of operation. For example, a resistor used with a LED must be justified by the desired current and by the forward voltage of the LED. A capacitor used for filtering must be justified by the ripple reduction or by the frequency range of the signal.

Table 1.7: Typical information to add to an electronic drawing file.

Information	Purpose
Functional objective	Defines what the circuit must do.
Input and output values	Allows the reader to know the operating range.
Component calculations	Justifies resistor, capacitor, and protection values.
Expected measurements	Helps during testing and troubleshooting.
Component references	Links schematic, PCB, and bill of materials.
Operating limits	Prevents the use of the circuit outside safe conditions.

1.20.1 Node Labels and Test Points

Node labels are essential in a clear schematic. They reduce the number of long wires in the drawing and help the reader identify important signals. Labels such as V_{in} , V_{out} , GND, CLK, RESET, TP1, and TP2 are widely used. In practical work, the labels TP1, TP2, and TP3 can also correspond to real test points on the PCB.

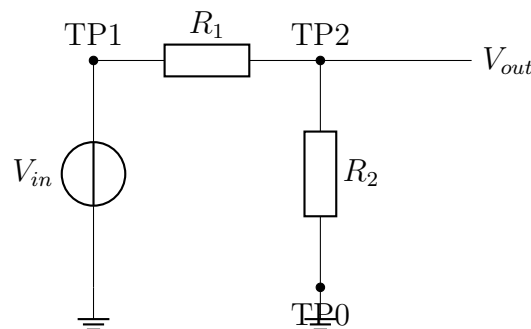


Figure 1.20: Example of labeled nodes and test points in a simple circuit.

The choice of test points must be logical. A test point is useful if it helps verify one function of the circuit. In a power supply, important test points are the input voltage, the regulator output voltage, and ground. In an amplifier, they are the input signal, bias voltage, and output signal. In a digital circuit, they are the clock signal, reset signal, and selected logic outputs.

1.20.2 Consistency Between Documents

The same component reference must be used in all documents. If a resistor is called R_4 in the schematic, it must also be called R_4 in the placement drawing, the PCB drawing, and the bill of materials. Inconsistency between documents is a frequent source of assembly errors.

Important note. Any modification of the schematic must be followed by an update of the bill of materials, the placement drawing, and the PCB layout. A project document must always describe the latest validated version of the circuit.

1.21 Using Datasheets During Electronic Drawing

A datasheet is a technical document provided by the manufacturer of a component. It gives the information needed to use the component correctly. During electronic drawing, the datasheet is necessary for identifying the pinout, package, maximum ratings, recommended operating conditions, typical circuits, and mechanical dimensions.

Students often make the mistake of using a symbol without checking the real pinout of the component. This is especially dangerous for transistors, voltage regulators, operational amplifiers, and integrated circuits. Two components may have similar names but different pin arrangements.

Table 1.8: Important datasheet sections for schematic and PCB preparation.

Datasheet section	Use in the project
Pin configuration	Correct connection of component terminals.
Absolute maximum ratings	Avoiding destructive operating conditions.
Recommended conditions	Choosing normal voltage, current, and temperature ranges.
Electrical characteristics	Estimating gains, thresholds, drops, and currents.
Typical application	Understanding recommended external components.
Package dimensions	Verifying the footprint and mechanical placement.

1.21.1 Maximum Ratings and Recommended Conditions

Maximum ratings are not design values. They indicate limits that must not be exceeded. A component used continuously near its maximum rating may heat excessively and fail early. Recommended operating conditions are more suitable for design.

For example, if a capacitor is used on a 12 V supply, a 16 V rated capacitor may work, but a 25 V rated capacitor gives a better safety margin. In the same way, a resistor dissipating 0.2 W should not be chosen with only a 0.25 W rating when long operation is expected.

1.21.2 Preparing a Drawing File

A complete drawing file should allow another person to reproduce the circuit. It should contain a block diagram, schematic, calculation notes, bill of materials, placement drawing, PCB drawing, and expected measurements. This documentation is a key part of engineering work and should be evaluated together with the physical prototype.

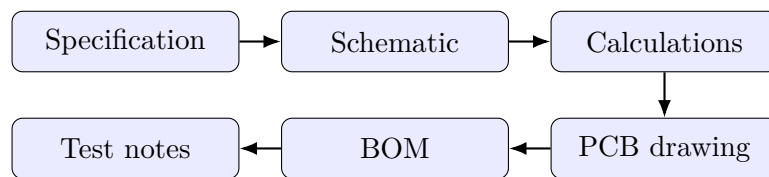


Figure 1.21: Recommended organization of a complete drawing file.

1.22 Chapter Summary

Electronic drawing is a key step in the design and realization of electronic circuits. In this chapter, the main passive and active components were reviewed, including their functions and technological characteristics. The chapter also introduced several types of electronic documents: block diagrams, detailed schematics, equivalent diagrams, component placement drawings, wiring diagrams, definition drawings, and bills of materials.

A well-prepared electronic drawing helps the student understand the circuit, calculate component values, prepare the PCB, solder components correctly, and troubleshoot the final prototype. These skills are essential for mini-projects and for the realization of practical electronic systems.

1.23 Exercises

Exercise 1: Component Identification

For each component listed below, indicate whether it is passive or active and give two typical applications:

1. Resistor
2. Capacitor
3. Diode
4. Transistor
5. Operational amplifier

Exercise 2: Reading a Schematic

Consider the voltage divider circuit in Figure 1.8. Let $V_{in} = 15\text{ V}$, $R_1 = 10\text{ k}\Omega$, and $R_2 = 5\text{ k}\Omega$.

1. Calculate V_{out} .
2. Calculate the current flowing through the two resistors.
3. Calculate the power dissipated in each resistor.

Exercise 3: Documentation of a Simple Circuit

Choose a simple LED indicator circuit powered by 5 V. Prepare:

1. a block diagram;

2. a circuit schematic;
3. a component placement drawing;
4. a bill of materials.

Exercise 4: Drawing Quality

List five rules that must be respected when preparing a clean electronic schematic.

1.24 Mini-Project Preparation Questions

Before starting a mini-project, the student should answer the following questions:

1. What is the main function of the circuit?
2. What are the input and output quantities?
3. What are the required voltage and current levels?
4. Which components are necessary?
5. Are there any polarized components?
6. Is the circuit analog, digital, or mixed?
7. What instruments are needed for testing?
8. What safety precautions must be respected?
9. What documents must be prepared before realization?

Chapter 2

Technology for Realizing Electronic Schematics

Learning objectives

- Understand the technological steps required to transform an electronic schematic into a practical circuit.
- Learn the role of the international grid in electronic layout and printed circuit board design.
- Identify the main rules for preliminary prototyping and component placement.
- Apply good practices for arranging passive components, active components, integrated circuits, connectors, heat sinks, transformers, and power components.
- Prepare students for the practical realization of simple electronic circuits.

2.1 Introduction

After drawing and understanding an electronic schematic, the next step is to prepare its practical realization. A circuit schematic represents electrical connections, but it does not directly define the physical position of components, the dimensions of the board, or the technological constraints related to soldering, heat dissipation, insulation, and accessibility.

The realization of an electronic circuit requires a methodical approach. The designer must first verify the schematic, choose suitable components, prepare a preliminary prototype, define the board dimensions, arrange the components, and finally route the electrical connections. This chapter focuses on the technological rules that help transform a theoretical circuit into a practical and reliable electronic assembly.

2.2 From Schematic to Practical Realization

The transition from an electronic schematic to a physical circuit is usually performed through several steps:

1. analysis of the circuit function;
2. identification of input and output terminals;
3. selection of component values and packages;
4. preliminary test on breadboard or experimental board;
5. component placement planning;
6. PCB routing or wiring plan;
7. verification of electrical and mechanical constraints;
8. manufacturing, soldering, and testing.

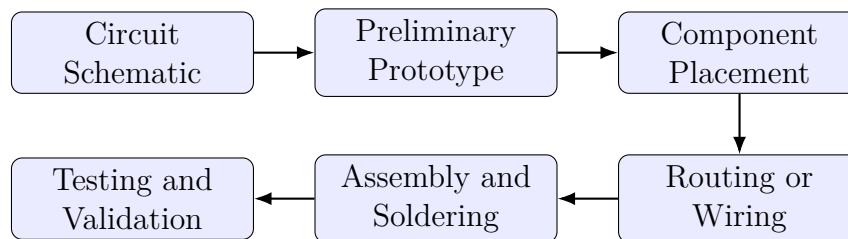


Figure 2.1: General process from electronic schematic to practical realization.

A common mistake is to directly start soldering components without checking the circuit or preparing the layout. This often leads to wiring errors, unstable operation, difficult troubleshooting, or damaged components.

2.3 International Grid in Electronics

In electronic design, the position of component terminals and holes is often based on a standard grid. The most widely used pitch is:

$$2.54 \text{ mm} = 0.1 \text{ inch} \quad (2.1)$$

This spacing is very common for through-hole components, dual in-line integrated circuits, pin headers, breadboards, and experimental boards.

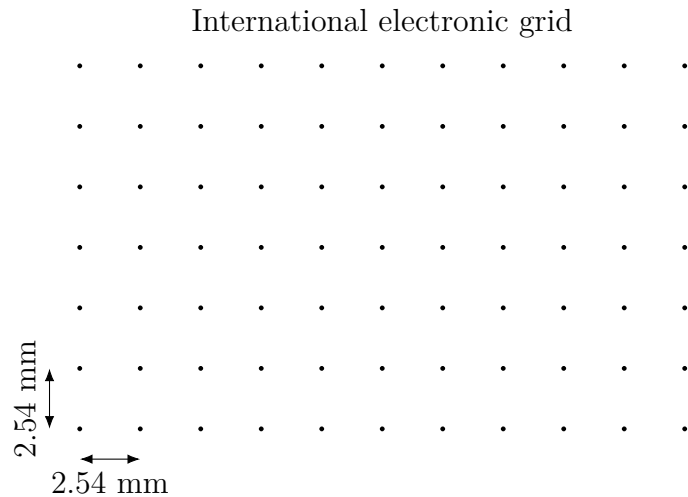


Figure 2.2: Representation of the standard 2.54 mm electronic grid.

2.3.1 Importance of the Grid

The international grid is important because it helps the designer to:

- place components in a regular and organized way;
- ensure compatibility with standard packages;
- simplify routing and soldering;
- reduce design errors;
- facilitate manual prototyping and educational work.

For educational projects, using components compatible with the 2.54 mm grid is highly recommended because they are easier to mount, solder, test, and replace.

2.4 Preliminary Prototypes

Before designing a printed circuit board, it is often necessary to build a preliminary prototype. The prototype allows the student to verify the circuit operation, measure voltages and currents, detect errors, and adjust component values.

Several methods can be used for preliminary prototyping:

- breadboard assembly;
- perforated board assembly;
- stripboard assembly;
- temporary wired assembly;
- simulation before hardware testing.

2.4.1 Breadboard Prototyping

A breadboard is a solderless prototyping board. It is useful for testing low-frequency and low-power circuits. Components can be inserted and removed easily, which makes it suitable for educational use.

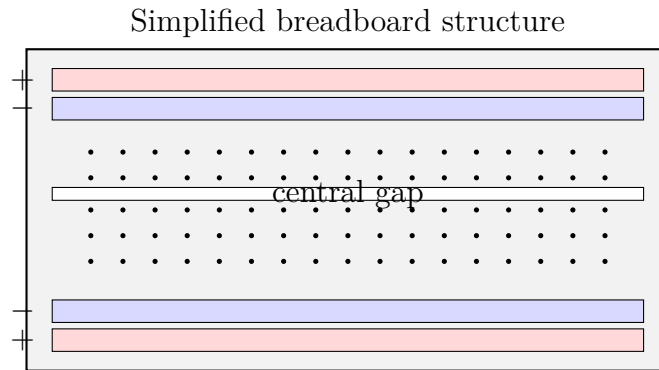


Figure 2.3: Simplified representation of a solderless breadboard.

2.4.2 Advantages and Limitations of Breadboards

Table 2.1: Advantages and limitations of breadboard prototyping.

Advantages	Limitations
No soldering required	Not suitable for high-current circuits
Fast modification	Not suitable for high-frequency circuits
Good for learning and testing	Loose contacts may create faults
Reusable	Parasitic capacitance and inductance exist
Easy measurement access	Not mechanically robust

Important note. A breadboard is useful for testing the principle of operation, but it is not equivalent to a final printed circuit board. A circuit that works on a breadboard may still require modifications before PCB realization, especially for power, high-frequency, or noise-sensitive circuits.

2.5 Component Placement: General Rules

Component placement is one of the most important steps in electronic realization. Good placement simplifies routing, improves reliability, reduces noise, facilitates soldering, and makes troubleshooting easier.

The following general rules should be respected:

- place input connectors near the input side of the board;
- place output connectors near the output side;
- arrange components according to the signal flow;
- place components that are electrically connected close to each other;
- keep power components away from sensitive low-level signals;
- respect component orientation;
- leave enough space for heat sinks and large components;
- make test points accessible;
- avoid placing components too close to the board edge;
- keep the layout readable and easy to inspect.

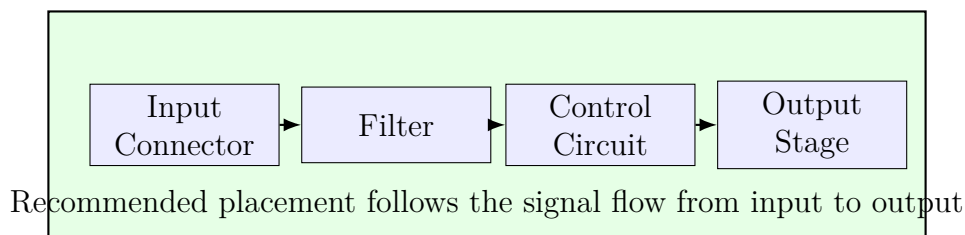


Figure 2.4: Example of component placement according to signal flow.

2.6 Placement of Passive Components

Passive components are usually small and easy to place. However, their position can strongly affect the behavior of the circuit.

2.6.1 Resistors

Resistors should be placed near the components they are connected to. In analog circuits, feedback resistors should be placed close to operational amplifiers. Pull-up and pull-down resistors should be placed close to the digital input pins they control.

2.6.2 Capacitors

Capacitor placement is very important, especially for filtering and decoupling. A decoupling capacitor must be placed as close as possible to the power supply pins of an integrated circuit.

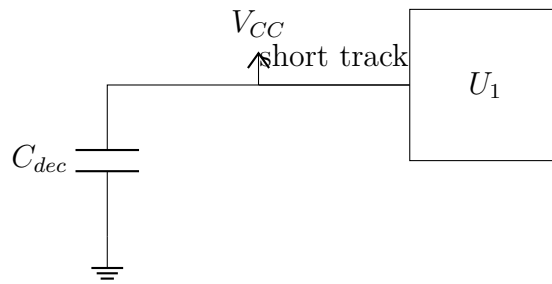


Figure 2.5: Decoupling capacitor placed close to the integrated circuit supply pin.

2.6.3 Inductors

Inductors may generate magnetic fields and should be placed carefully. In switching power supplies, inductors must be kept away from sensitive analog signals. Their current rating and heating must also be considered.

2.7 Placement of Active Components

Active components such as diodes, transistors, voltage regulators, and operational amplifiers must be placed according to their function, power dissipation, and sensitivity.

2.7.1 Diodes

The orientation of diodes is essential. The cathode and anode must be clearly identified on both the schematic and the PCB. Protection diodes should be placed close to the points they protect.

2.7.2 Transistors

Transistors used as small-signal amplifiers should be placed near the related input and output components. Transistors used as power switches should be placed near the load or power connector, while respecting heat dissipation requirements.

2.7.3 Voltage Regulators

Voltage regulators should be placed close to the power input and near their input and output capacitors. If the regulator dissipates significant power, space must be reserved for a heat sink or copper area.

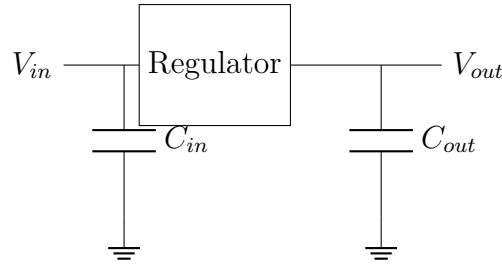


Figure 2.6: Input and output capacitors placed near a voltage regulator.

2.8 Placement of Integrated Circuits

Integrated circuits are central elements in many electronic systems. Their placement must consider pin orientation, power supply pins, signal paths, programming access, and thermal constraints.

2.8.1 Pin Numbering and Orientation

Integrated circuits usually have a mark that identifies pin 1. This mark may be a notch, a dot, or a bevel. The PCB layout must clearly indicate this orientation to avoid incorrect soldering.

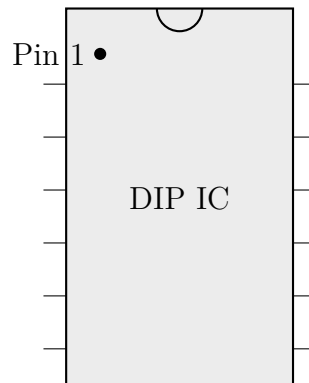


Figure 2.7: Orientation mark and pin 1 identification on a dual in-line integrated circuit.

2.8.2 Decoupling of Integrated Circuits

Each integrated circuit should generally have at least one decoupling capacitor connected between its supply pin and ground. This capacitor reduces voltage fluctuations caused by fast current variations.

For digital circuits, a common practical choice is:

$$C_{dec} = 100 \text{ nF} \quad (2.2)$$

This capacitor should be placed very close to the integrated circuit pins.

2.9 Heat Sinks and Thermal Considerations

Some components dissipate heat during operation. Examples include voltage regulators, power transistors, power diodes, triacs, MOSFETs, and power resistors. If heat is not properly evacuated, the component temperature may exceed its limit and cause failure.

The power dissipated by a linear voltage regulator can be estimated by:

$$P_D = (V_{in} - V_{out})I_{out} \quad (2.3)$$

where P_D is the dissipated power, V_{in} is the input voltage, V_{out} is the output voltage, and I_{out} is the output current.

Example. A linear regulator supplies 5 V from a 12 V input with an output current of 0.5 A. The dissipated power is:

$$P_D = (12 - 5) \times 0.5 = 3.5 \text{ W}$$

This power is significant. A heat sink is probably required.

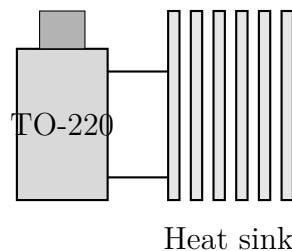


Figure 2.8: Simplified representation of a power component mounted on a heat sink.

When placing components with heat sinks, the designer must leave enough space around them and avoid placing heat-sensitive components nearby.

2.10 Placement of Transformers and Power Elements

Transformers, relays, power resistors, triacs, MOSFETs, and large capacitors require special attention because they may be heavy, hot, noisy, or connected to dangerous voltages.

2.10.1 Transformers

A transformer is usually one of the largest components on the board. It should be fixed mechanically and placed away from sensitive low-level analog circuits because it may generate magnetic interference.

2.10.2 Power Components

Power components should be placed according to the following rules:

- use wider tracks for high-current paths;
- provide sufficient spacing for high-voltage areas;
- separate power and control sections;
- consider heat dissipation;
- use screw terminals or suitable connectors for external loads;
- ensure mechanical stability.

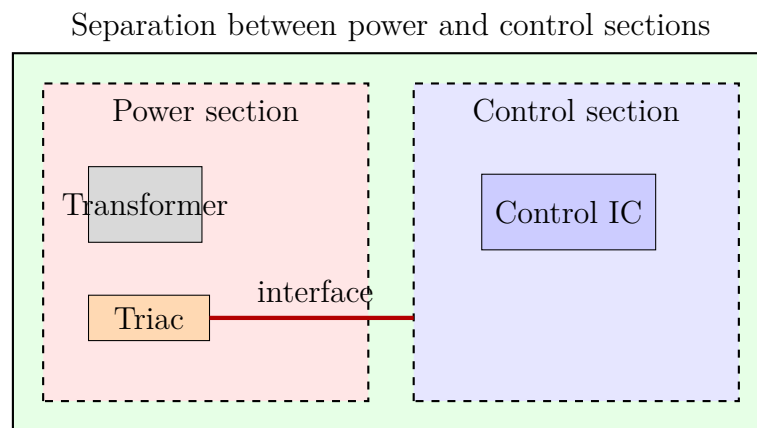


Figure 2.9: Example of separation between power and control sections on a PCB.

2.11 Mechanical and Practical Constraints

A good electronic layout must also respect mechanical constraints. These constraints are often ignored by beginners, but they are important for final assembly and maintenance.

The designer should check:

- board dimensions;
- connector accessibility;
- mounting hole positions;
- height of components;
- distance between hot components and plastic parts;
- cable entry direction;

- enclosure dimensions;
- possibility of measuring signals during testing.

Important note. A circuit may be electrically correct but mechanically difficult to assemble. Practical realization must consider both electrical and mechanical requirements.

2.12 Design Example: 5 V Regulated Power Supply

This example shows how to organize a simple regulated DC power supply. The circuit contains an input connector, a protection diode, a voltage regulator, input and output capacitors, an LED indicator, and an output connector.

2.12.1 Functional Block Diagram

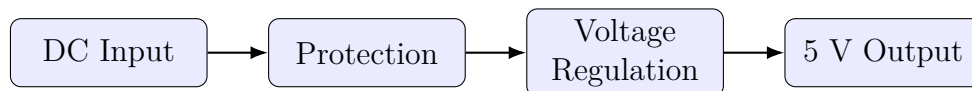


Figure 2.10: Block diagram of a regulated 5 V power supply.

2.12.2 Simplified Schematic

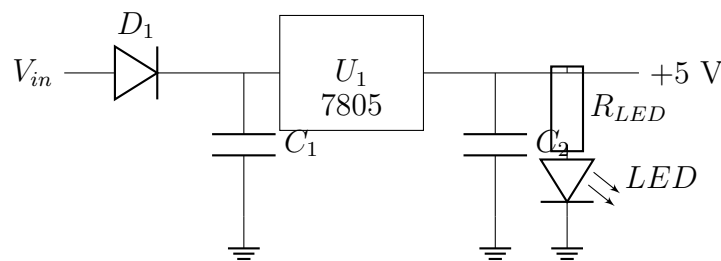


Figure 2.11: Simplified schematic of a 5 V regulated power supply.

2.12.3 Recommended Placement

The input connector should be placed near the input side of the board. The protection diode and input capacitor should be near the input connector and regulator. The output capacitor should be close to the regulator output pin. The LED indicator may be placed near the output connector or at the board edge to be visible.

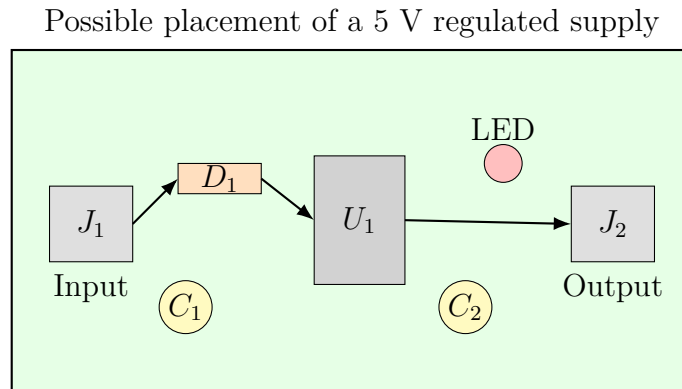


Figure 2.12: Simplified component placement for a 5 V regulated supply.

2.13 Checklist Before PCB Routing

Before starting the routing of a printed circuit board, the designer should check the following points:

1. Are all components correctly selected?
2. Are the component packages available and compatible with the board technology?
3. Are the input and output connectors correctly placed?
4. Are polarized components clearly oriented?
5. Are decoupling capacitors close to integrated circuits?
6. Are power tracks planned to be wide enough?
7. Are high-voltage and low-voltage parts separated?
8. Are heat-generating components placed with enough space?
9. Are test points accessible?
10. Is the board mechanically compatible with its enclosure?

2.14 Common Mistakes in Component Placement

The most frequent mistakes made by beginners are:

- placing components randomly without following signal flow;
- placing decoupling capacitors far from integrated circuits;
- ignoring the orientation of diodes, electrolytic capacitors, and integrated circuits;

- placing connectors in inaccessible positions;
- using tracks that are too narrow for high current;
- mixing power and low-level signal sections;
- placing heat-sensitive components near hot components;
- forgetting mounting holes;
- choosing component packages that are difficult to solder manually;
- starting PCB routing before verifying the schematic.

2.15 Footprint Selection and Package Verification

In PCB design, a footprint is the physical representation of a component on the board. It defines pad positions, hole diameters, mechanical outline, and orientation marks. A correct schematic with an incorrect footprint can produce a board that cannot be assembled. Therefore, footprint selection is a critical technological step.

2.15.1 Difference Between Symbol and Footprint

The symbol is used in the schematic to represent the electrical function. The footprint is used in the PCB layout to represent the real mechanical component. One electrical symbol may correspond to several possible footprints. For example, a resistor may be through-hole, SMD 1206, SMD 0805, or a power resistor with larger spacing.

Table 2.2: Examples of symbols and possible footprints.

Component	Schematic symbol	Possible footprints
Resistor	Two-terminal resistor	Axial through-hole, 1206, 0805, 0603
Capacitor	Non-polarized or polarized capacitor	Radial electrolytic, ceramic disc, 1206, 0805
Diode	Diode symbol	DO-41, DO-35, SOD-123, SMA
Transistor	BJT or MOSFET symbol	TO-92, TO-220, SOT-23, DPAK
Integrated circuit	Functional or multi-pin symbol	DIP, SOIC, TSSOP, QFN
Connector	Pin connector	Screw terminal, pin header, JST connector

2.15.2 Footprint Verification Checklist

Before starting the PCB layout, the designer should verify:

- pin spacing;
- pad diameter or pad size;
- hole diameter for through-hole components;
- body dimensions;
- orientation of pin 1;
- polarity marks for diodes and capacitors;
- mechanical clearance around the component;
- compatibility with manual soldering if the project is educational.

Important note. For student projects, through-hole components are often preferable because they are easier to handle, solder, test, and replace. SMD components may be used progressively after students master basic soldering techniques.

2.16 Component Placement Strategies

Component placement is not random. It must follow functional, electrical, thermal, and mechanical logic. A well-placed circuit is easier to route and less likely to present noise, overheating, or troubleshooting difficulties.

2.16.1 Functional Placement

Functional placement means that components are grouped according to their role. For example, an input filter should be placed close to the input connector, a regulator should be close to the power input, and an output driver should be close to the output connector.

2.16.2 Electrical Placement

Electrical placement aims to minimize undesirable effects. Sensitive signals must be kept short and separated from high-current or switching tracks. Feedback components of an operational amplifier must be close to the amplifier. Decoupling capacitors must be very close to supply pins. Oscillator components must be placed close to the oscillator or microcontroller pins.

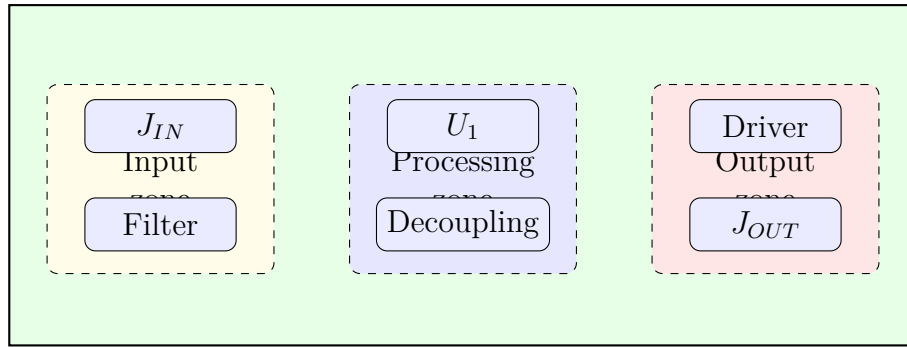


Figure 2.13: Functional zoning of a printed circuit board.

2.16.3 Thermal Placement

Power components must be placed where heat can be evacuated. They should not be surrounded by heat-sensitive capacitors or plastic connectors. If a heat sink is used, enough space must be reserved for it before routing the PCB.

2.16.4 Mechanical Placement

Connectors, switches, potentiometers, LEDs, and displays are often constrained by the enclosure or by user access. Their position must be decided early in the design. Mounting holes must also be considered before placing other components.

2.17 Preliminary Prototyping Methodology

Preliminary prototyping is more than assembling components quickly. It is an experimental phase used to verify assumptions and reduce design risk. During this phase, students should record their observations and not only check whether the circuit works.

2.17.1 Prototype Planning

Before building a prototype, the following points should be prepared:

- the simplified schematic;
- the list of required components;
- the power supply voltage and current limit;
- expected voltage levels at important points;
- the measurement instruments needed;
- possible risks such as reversed polarity or excessive current.

2.17.2 Prototype Validation Table

A validation table helps the student compare theoretical and measured values.

Table 2.3: Example of validation table for a preliminary prototype.

Test point	Expected value	Measured value	Comment
Input supply	5 V		Check polarity
Regulated out-put	5 V		No load condition
LED voltage	2 V approx.		Depends on LED color
Transistor base	0.7 V approx.		When ON
Output signal	HIGH/LOW		Depends on command

2.17.3 Progressive Testing

A prototype should be tested progressively. The power section can be tested first without connecting the sensitive control section. Then each functional block can be added and verified. This progressive method helps locate faults quickly and prevents damaging the complete circuit.

2.18 Power Distribution and Decoupling Rules

A circuit may fail even when the schematic is correct if the power distribution is poorly designed. Power tracks have resistance and inductance. When current changes quickly, voltage drops and noise may appear.

2.18.1 Power Track Organization

Power tracks should be wide enough and as short as possible. A star distribution may be used when several loads draw significant current. Sensitive analog circuits should not share the same narrow ground return path with high-current loads.

2.18.2 Decoupling Capacitors

Decoupling capacitors act as local energy reservoirs. They supply fast transient currents demanded by integrated circuits and reduce voltage disturbances on the supply line. A common practice is to place a 100 nF ceramic capacitor close to each integrated circuit power pin. Larger capacitors may be added near regulators or connectors.

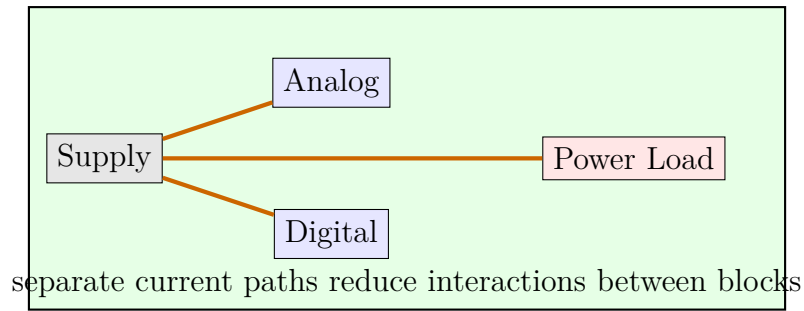


Figure 2.14: Simplified star-like power distribution.

Table 2.4: Typical capacitor roles in electronic circuits.

Capacitor type	Typical value	Role
Local decoupling	100 nF	Reduce high-frequency supply noise near ICs
Bulk capacitor	10 μ F to 1000 μ F	Store energy and reduce low-frequency ripple
Timing capacitor	nF to μ F range	Define time constants in RC or NE555 circuits
Coupling capacitor	application-dependent	Pass AC signal and block DC component
Filter capacitor	application-dependent	Smooth rectified or noisy voltage

2.19 Case Study: Placement of a Small Regulated Supply

A small regulated supply is a useful example because it includes power input, protection, filtering, regulation, indication, and output connection. The placement must respect current paths and thermal constraints.

2.19.1 Functional Zones

The circuit can be divided into four zones:

1. input zone: connector and protection diode;
2. filtering zone: input capacitor;
3. regulation zone: regulator and heat sink if necessary;
4. output zone: output capacitor, LED indicator, and connector.

2.19.2 Placement Rules for the Example

The input capacitor must be close to the regulator input. The output capacitor must be close to the regulator output. The regulator should be placed so that heat can be evacuated. The LED should be visible from the board edge if it is used as a power indicator. The input and output connectors should not be confused; labels such as IN, OUT, +, and GND should be printed or written clearly.

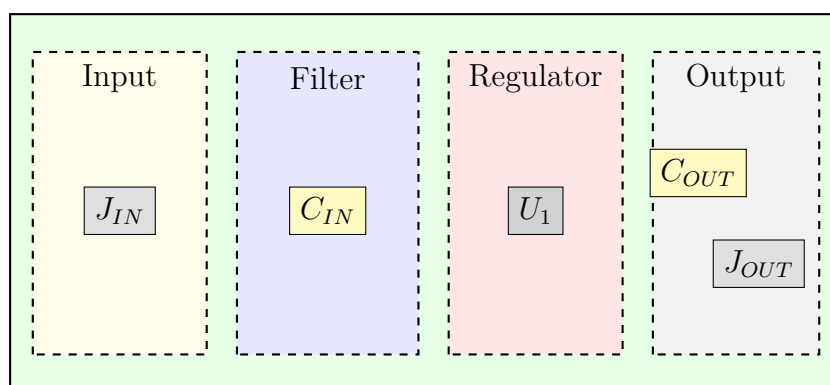


Figure 2.15: Functional placement zones of a small regulated power supply.

2.20 Advanced Component Placement Considerations

Component placement has a strong influence on circuit reliability. The same schematic may work correctly or incorrectly depending on the physical position of the components. Good placement reduces wiring complexity, improves thermal behavior, reduces noise, and makes testing easier.

2.20.1 Functional Zones

A printed circuit board may be divided into functional zones: power input, protection, analog conditioning, digital processing, output interface, and connectors. This organization makes the board easier to read and reduces errors during routing.

Functional zoning does not mean that the sections are completely separated. It means that the designer controls the position of sensitive signals, noisy signals, and power paths. For example, an analog input should not be placed near a relay coil, transformer, motor driver, or high-current switching transistor.

2.20.2 Placement According to Energy Flow

In power circuits, the current path must be short and logical. For a linear regulated supply, the input connector, protection diode, input capacitor, regulator, output capacitor, and

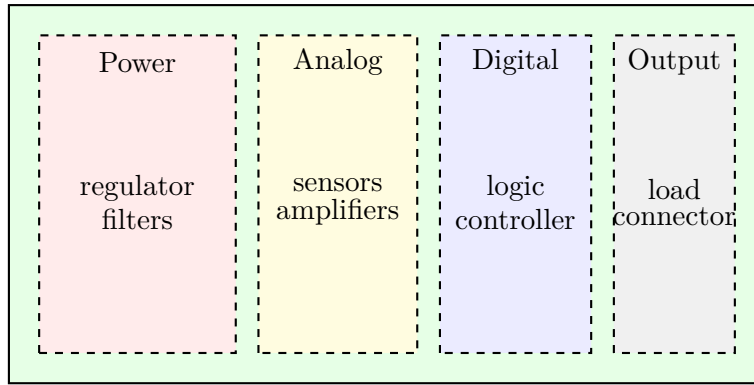


Figure 2.16: Example of functional zoning on a printed circuit board.

output connector should be placed in the same direction as the energy flow. This reduces unnecessary loops and voltage drops.

Important note. A short track is not always sufficient. The designer must also check the return path of the current. A supply current always flows in a loop: from the source to the load and back to the source through ground or return conductors.

2.20.3 Design for Assembly

The board must be easy to assemble. Components with polarity must have visible marks. Connectors must remain accessible. Large components must not hide small components. Test points must be reachable by measurement probes. If the board is difficult to assemble, the risk of error becomes high even if the schematic is correct.

2.21 Thermal Derating and Reliability

Derating consists of using components below their maximum limits in order to increase reliability. A resistor, diode, regulator, or capacitor may work at its maximum rating during a short test, but long-term operation near the limit is not recommended.

Table 2.5: Examples of practical derating choices.

Component	Calculated stress	Recommended choice
Resistor	0.20 W	0.5 W or higher
Electrolytic capacitor	12 V supply	25 V rating
Diode	0.7 A current	1 A or higher
Regulator	1.5 W dissipation	heat sink or better regulator
PCB track	1 A current	wide and short track

The temperature rise can be estimated using:

$$\Delta T = P_D R_\theta \quad (2.4)$$

where P_D is the dissipated power and R_θ is the thermal resistance. If a regulator dissipates 2 W and its thermal resistance is 50 °C/W, the temperature rise may reach 100 °C. This is too high for safe operation without additional cooling.

2.21.1 Design for Testability

Design for testability means adding practical details that make the circuit easier to verify after assembly. These details include test points, accessible connectors, clear ground references, and enough space for measurement probes.

Recommended test points for a small board are:

- power input;
- regulated supply output;
- ground reference;
- input signal;
- output signal;
- clock or control signal when present.

2.21.2 Elementary EMC Rules

Electromagnetic compatibility can be introduced through simple layout rules. High-current loops should be small. Sensitive analog inputs should be far from switching paths. Decoupling capacitors should be placed near integrated circuits. Ground paths should be clear and direct.

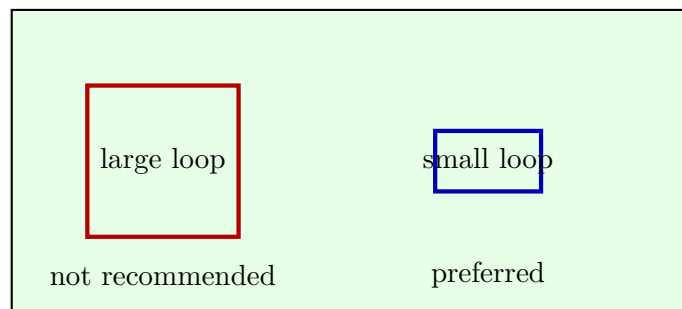


Figure 2.17: Comparison between a large current loop and a compact current loop.

2.22 Chapter Summary

This chapter presented the technological steps required to transform an electronic schematic into a practical electronic circuit. The international grid, preliminary prototyping methods, and component placement rules were discussed. Special attention was given to passive components, active components, integrated circuits, heat sinks, transformers, and power elements.

Good component placement is essential for reliable circuit operation. It reduces wiring complexity, improves thermal behavior, simplifies troubleshooting, and prepares the circuit for successful printed circuit board realization.

2.23 Exercises

Exercise 1: International Grid

Answer the following questions:

1. What is the standard pitch commonly used in through-hole electronic components?
2. Convert 0.1 inch into millimeters.
3. Why is this grid useful in educational electronic projects?

Exercise 2: Breadboard Limitations

Give three advantages and three limitations of breadboard prototyping. Explain why a breadboard is not suitable for high-current circuits.

Exercise 3: Component Placement

For a circuit containing an input connector, a voltage regulator, two capacitors, an LED, and an output connector:

1. propose a suitable component placement strategy;
2. indicate where the input capacitor and output capacitor should be placed;
3. explain where the LED should be placed for practical use.

Exercise 4: Thermal Calculation

A linear regulator converts 15 V to 5 V and supplies a load current of 0.3 A.

1. Calculate the power dissipated by the regulator.
2. Explain whether thermal precautions are necessary.

Exercise 5: Power and Control Separation

Explain why it is recommended to separate the power section from the control section in a circuit containing a triac or a relay.

Chapter 3

Wiring Techniques for Electronic Circuits

Learning objectives

- Understand the basic principles of electronic circuit wiring and printed circuit boards.
- Identify the main materials and properties of printed circuit boards.
- Learn the general methodology for designing a PCB layout from an electronic schematic.
- Understand the basic steps of PCB manufacturing: artwork preparation, transfer, etching, drilling, inspection, and finishing.
- Recognize common PCB faults and learn basic repair and modification techniques.
- Introduce surface-mount device circuits and compare them with through-hole technology.

3.1 Introduction

The practical realization of an electronic circuit requires reliable electrical connections between components. In simple prototypes, these connections can be made using wires on a breadboard or a perforated board. However, for a more robust and professional realization, the circuit is usually implemented on a printed circuit board, commonly called a PCB.

A printed circuit board supports the electronic components mechanically and connects them electrically using copper tracks. PCB technology improves reliability, reduces wiring errors, allows compact circuits, and facilitates repeated production. This chapter presents

the main techniques used in the wiring and realization of electronic circuits, with special attention to printed circuit boards.

3.2 Electronic Wiring Methods

Several wiring methods can be used depending on the complexity of the circuit, available equipment, cost, and educational objectives.

Table 3.1: Comparison of common electronic wiring methods.

Method	Advantages	Limitations
Breadboard	Fast, reusable, no soldering	Not robust, not suitable for high current or high frequency
Perforated board	Simple, low cost, suitable for soldering practice	Wiring may become complex and difficult to inspect
Stripboard	Easier wiring than perforated board	Requires track cutting and careful planning
Printed circuit board	Reliable, compact, professional	Requires design, manufacturing, and verification steps

Important note. The choice of wiring method depends on the objective. A breadboard is suitable for testing, while a printed circuit board is more suitable for final realization.

3.3 Printed Circuit Board Technology

A printed circuit board is usually made of an insulating substrate covered by one or more copper layers. The copper is selectively removed to create tracks, pads, and connection areas.

3.3.1 Main Parts of a PCB

A basic single-sided PCB contains the following parts:

- an insulating substrate;
- a copper layer;
- copper tracks;
- pads for component soldering;

- holes for through-hole components;
- sometimes a solder mask and silkscreen layer.

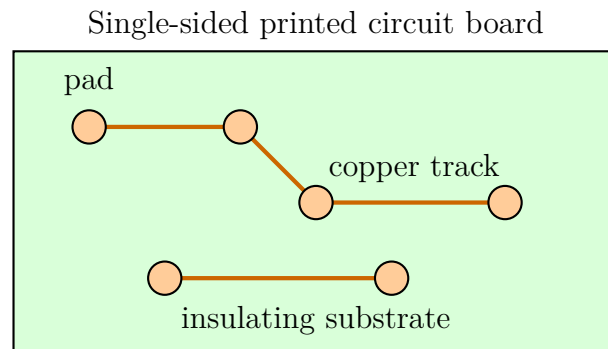


Figure 3.1: Simplified structure of a single-sided printed circuit board.

3.3.2 Single-Sided and Double-Sided PCBs

A single-sided PCB has copper tracks on only one side of the board. It is simple and suitable for educational and low-complexity circuits. A double-sided PCB has copper on both sides and allows more compact and complex routing.

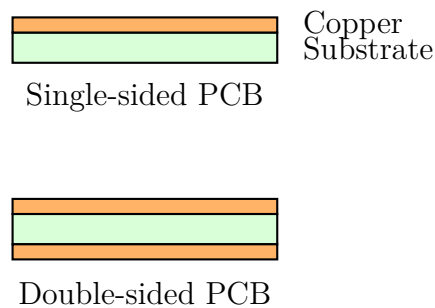


Figure 3.2: Simplified comparison between single-sided and double-sided PCBs.

3.4 PCB Materials and Properties

The most common PCB substrate materials are phenolic paper and fiberglass epoxy.

3.4.1 Phenolic Paper Boards

Phenolic paper boards are low-cost and easy to drill. They are often used for simple educational circuits. However, they are less mechanically robust and less suitable for demanding applications.

3.4.2 Fiberglass Epoxy Boards

Fiberglass epoxy boards, often known as FR-4 boards, are stronger, more stable, and more commonly used in professional electronics. They provide better mechanical strength and better electrical insulation.

3.4.3 Important PCB Properties

The main properties to consider when selecting a PCB material are:

- mechanical strength;
- insulation resistance;
- copper thickness;
- thermal resistance;
- ease of drilling and cutting;
- compatibility with soldering;
- cost and availability.

Copper thickness is often expressed in micrometers or ounces per square foot. A common copper thickness is approximately:

$$35 \mu\text{m} \tag{3.1}$$

3.5 Establishing the PCB Layout

PCB layout is the process of transforming the electronic schematic into a physical arrangement of components and copper tracks. This step must be performed carefully because it strongly affects the reliability and performance of the circuit.

3.5.1 General PCB Layout Procedure

A typical PCB layout procedure includes:

1. checking the schematic;
2. assigning correct footprints to all components;
3. defining the board dimensions;
4. placing connectors and mechanical elements;

5. placing main components;
6. placing passive components close to related active components;
7. routing power tracks;
8. routing signal tracks;
9. adding ground connections;
10. checking clearances, track widths, and orientation;
11. generating manufacturing files or artwork.

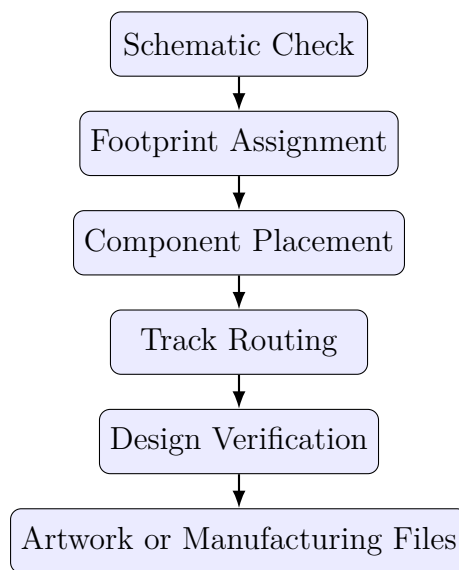


Figure 3.3: Main steps of PCB layout design.

3.5.2 Track Width

Track width must be selected according to the current flowing through the track. Power tracks must be wider than signal tracks. If a track is too narrow, it may overheat or cause a voltage drop.

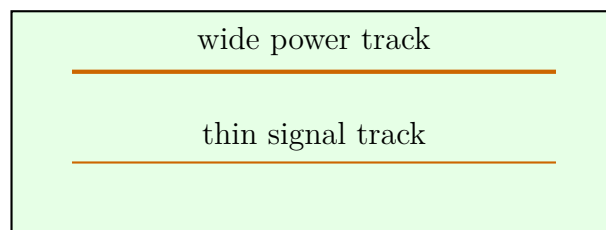


Figure 3.4: Power tracks are generally wider than signal tracks.

3.5.3 Clearance Between Tracks

Clearance is the minimum distance between two copper conductors. It must be sufficient to avoid short circuits, especially when the circuit operates at high voltage.

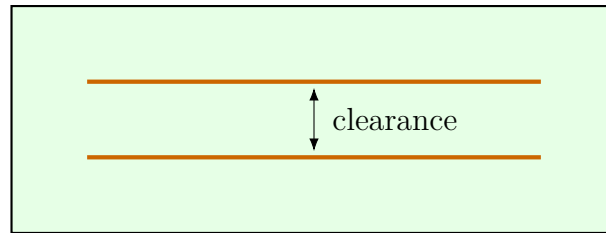


Figure 3.5: Clearance between two copper tracks.

3.6 Artwork and Negative Preparation

In traditional PCB manufacturing, the PCB layout is printed on a transparent film. This film is used to transfer the copper pattern onto a photosensitive copper-clad board. Depending on the process, the artwork may be positive or negative.

3.6.1 Manual and Software Methods

PCB artwork can be prepared manually for very simple circuits, but modern design uses software tools. Common PCB design software includes KiCad, Proteus ARES, EasyEDA, Eagle, Altium Designer, and other CAD tools.

The software allows the designer to:

- draw or import the schematic;
- assign footprints;
- place components;
- route tracks;
- check design rules;
- generate printing files, Gerber files, and drilling files.

3.6.2 Example of a Simple PCB Artwork

3.7 Transfer to Copper by Photolithography

Photolithography is a process used to transfer the PCB pattern onto a photosensitive copper-clad board. The board is covered by a light-sensitive layer. The artwork film is

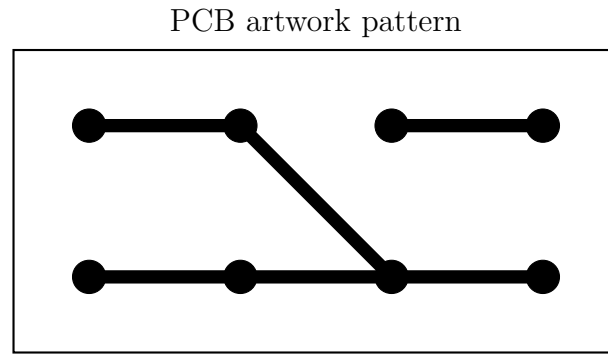


Figure 3.6: Simplified example of a PCB artwork used for transfer.

placed over the board, and the assembly is exposed to ultraviolet light.

3.7.1 Main Steps of Photolithography

The main steps are:

1. clean the copper-clad board;
2. place the printed artwork film on the photosensitive surface;
3. expose the board to ultraviolet light;
4. develop the exposed board using a developer solution;
5. rinse and inspect the pattern;
6. etch the copper;
7. clean the remaining protective layer.

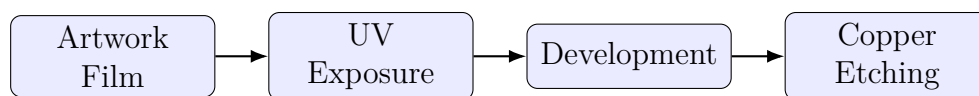


Figure 3.7: Basic photolithography steps for PCB fabrication.

Important note. Photolithography involves chemicals and ultraviolet light. It must be performed with appropriate protection, ventilation, and laboratory supervision.

3.8 Copper Etching

Etching is the chemical process that removes unwanted copper from the board. Only the copper protected by the resist remains after etching and forms the final tracks and pads.

3.8.1 Etching Principle

The board is immersed in an etching solution. The exposed copper reacts chemically with the solution and is removed. The protected copper remains on the substrate.

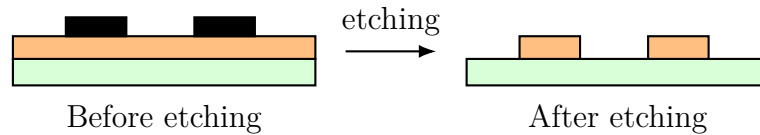


Figure 3.8: Simplified principle of copper etching.

3.8.2 Precautions During Etching

The following precautions are necessary:

- use gloves and protective glasses;
- avoid skin contact with chemical solutions;
- work in a ventilated area;
- avoid over-etching;
- rinse the board carefully after etching;
- dispose of used chemicals according to laboratory rules.

3.9 Post-Etching Treatment

After etching, the board is not yet ready for assembly. Several finishing operations are required.

3.9.1 Cleaning

The remaining protective layer must be removed. The copper surface must be cleaned to improve solderability.

3.9.2 Drilling

Through-hole components require drilling. The drill diameter depends on the component leads and connector pins. Typical small component holes may have diameters between 0.8 mm and 1.2 mm.

Drilled pads for through-hole components

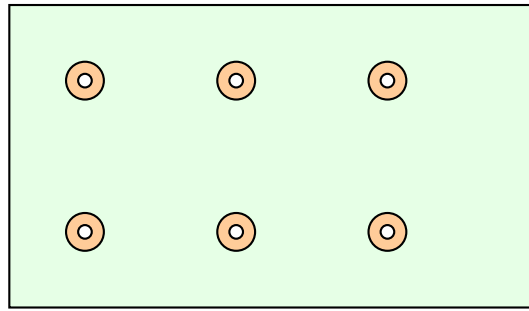


Figure 3.9: Pads after drilling.

3.9.3 Tinning

Tinning is the process of covering copper tracks with a thin layer of solder. It improves solderability and protects copper from oxidation.

3.9.4 Visual Inspection

The board must be visually inspected before soldering. The designer should verify:

- continuity of tracks;
- absence of short circuits between tracks;
- correct drilling;
- cleanliness of the copper surface;
- correct dimensions of the board.

3.10 Electrical Verification of the PCB

Before soldering components, the PCB must be checked electrically using a multimeter in continuity mode.

3.10.1 Continuity Test

A continuity test verifies that points that should be connected are electrically connected.

3.10.2 Short-Circuit Test

A short-circuit test verifies that points that should be separated are not accidentally connected.

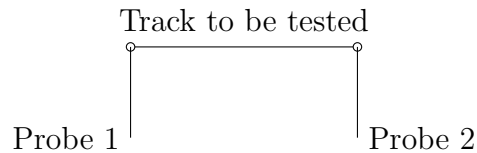


Figure 3.10: Principle of a continuity test using two probes.

Important note. It is easier to correct a PCB before soldering the components. For this reason, visual and electrical verification must always be performed before assembly.

3.11 Modification and Repair of Printed Circuits

During practical realization, errors may occur. A PCB track may be cut, a short circuit may appear, a pad may be damaged, or the designer may need to modify the circuit.

3.11.1 Common PCB Faults

Common PCB faults include:

- broken copper track;
- unwanted solder bridge;
- damaged pad;
- incorrect footprint;
- reversed component orientation;
- missing connection;
- excessive etching;
- insufficient etching.

3.11.2 Repair Techniques

Simple repair techniques include:

- removing solder bridges using solder wick;
- adding a small insulated wire to replace a broken track;
- cutting an unwanted track with a cutter;
- cleaning pads before resoldering;

- adding external jumpers when necessary.

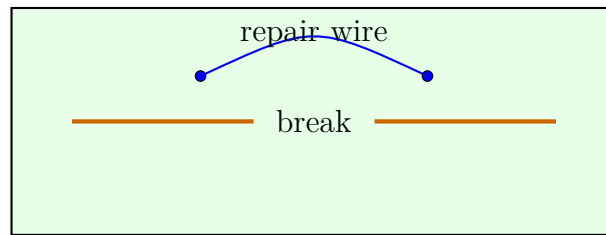


Figure 3.11: Repair of a broken track using a jumper wire.

3.12 Surface-Mount Device Circuits

Surface-mount device technology, usually abbreviated as SMD or SMT, uses components soldered directly onto the surface of the PCB. Unlike through-hole components, SMD components do not require holes for their leads.

3.12.1 Advantages of SMD Technology

SMD technology offers several advantages:

- compact circuits;
- reduced parasitic inductance and capacitance;
- compatibility with automated assembly;
- possibility of placing components on both sides of the board;
- lower mass and smaller product size.

3.12.2 Limitations of SMD Technology

However, SMD technology also has limitations:

- more difficult manual soldering for beginners;
- smaller markings on components;
- requires more precise tools;
- more difficult repair and replacement;
- sensitive to overheating during soldering.

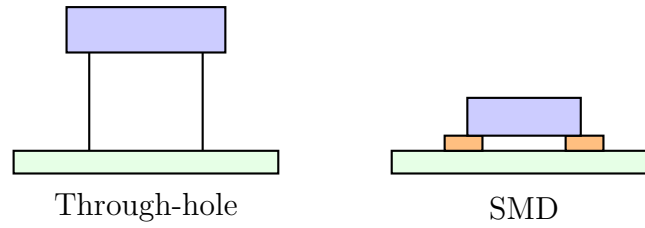


Figure 3.12: Comparison between through-hole and SMD component mounting.

3.13 Example: PCB Realization of a Simple LED Indicator

This example illustrates the realization of a simple LED indicator powered by a DC voltage source.

3.13.1 Circuit Principle

The LED must be connected in series with a current-limiting resistor. The resistor protects the LED by limiting the current.

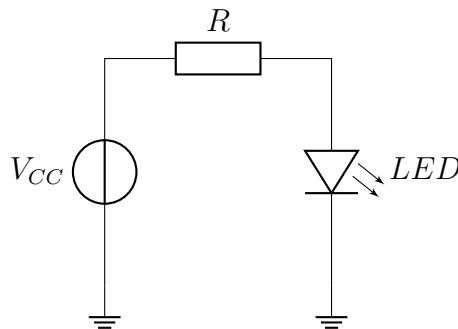


Figure 3.13: Simple LED indicator circuit.

The resistor value can be estimated by:

$$R = \frac{V_{CC} - V_F}{I_F} \quad (3.2)$$

where V_F is the LED forward voltage and I_F is the desired LED current.

3.13.2 PCB Layout Example

3.14 Safety Precautions

PCB realization involves tools, heat, drilling, and chemical products. Safety rules must always be respected.

Important precautions include:

PCB layout of a simple LED indicator

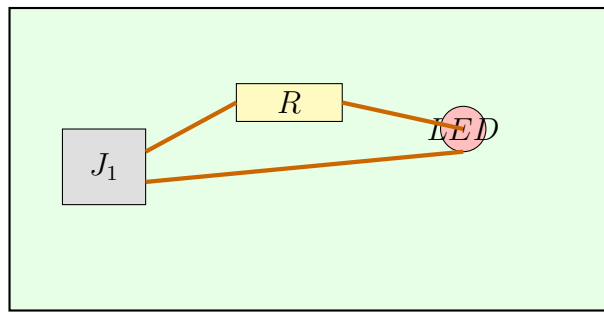


Figure 3.14: Simplified PCB placement and routing for an LED indicator.

- wear safety glasses during drilling and chemical processing;
- use gloves when handling chemical products;
- work in a ventilated area;
- avoid touching hot soldering iron tips;
- keep liquids away from electrical equipment;
- check power supplies before connecting a circuit;
- never work alone on circuits connected to dangerous voltages;
- follow laboratory safety instructions.

3.15 Detailed PCB Design Rules

PCB design rules define the technological limits that must be respected during routing. These rules depend on the manufacturing process, the available tools, the voltage level, the current level, and the soldering method. In an educational laboratory, the rules are usually less aggressive than in industrial production because the objective is reliability and ease of fabrication.

3.15.1 Track Width Selection

A copper track behaves like a small resistance. When current flows through it, a voltage drop and heating appear. For low-current signal tracks, a small width is sufficient. For power tracks, motor tracks, relay tracks, and supply tracks, a larger width is required.

In student work, it is preferable to use generous track widths when board space allows. This makes the board easier to etch and more tolerant to small manufacturing defects.

Table 3.2: Indicative track width choices for educational PCB design.

Application	Typical current	Recommended approach
Logic signal	few mA	thin track acceptable
Analog signal	few mA	short and separated from noise sources
LED supply	10 to 30 mA	medium track acceptable
Relay coil	50 to 150 mA	wider track recommended
Motor or power load	hundreds of mA or more	wide track and short path required
Main supply input	depends on total current	widest practical track

3.15.2 Clearance and Creepage

Clearance is the shortest distance through air between two conductors. Creepage is the shortest distance along the surface of the insulating material. For low-voltage educational circuits, small clearances are usually sufficient. However, when the circuit contains higher voltages, mains voltage, or triac circuits, separation must be increased significantly.

Important note. Circuits connected to mains voltage are not recommended for unsupervised student realization. If a triac dimmer or any AC mains circuit is studied, the practical work must be performed under strict supervision and with appropriate isolation.

3.15.3 Pad and Hole Dimensions

Through-hole pads must be large enough to resist drilling and soldering. If the pad is too small, it may detach during repair. If the hole is too small, the component lead will not enter. If the hole is too large, soldering may become weak.

3.16 Grounding and Return Current Paths

Many beginners consider ground as an ideal point that has the same voltage everywhere. In a real PCB, ground is made of copper tracks or planes that have resistance and inductance. When current returns through ground, small voltage differences may appear between different ground points. These differences can disturb sensitive signals.

Table 3.3: Indicative hole diameter ranges for common through-hole components.

Component	Typical lead type	Indicative hole
Small resistor	thin axial lead	0.8 to 1.0 mm
Small diode	axial lead	0.8 to 1.0 mm
Electrolytic capacitor	radial leads	0.8 to 1.2 mm
Pin header	square pin	around 1.0 mm
Screw terminal	large pin	1.2 to 1.5 mm or more
Power component	thick lead	verify datasheet

3.16.1 Ground Return Principle

Every current that leaves the power supply must return to it. The path of the return current is as important as the path of the supply current. High-current return paths should not pass through sensitive analog reference points.

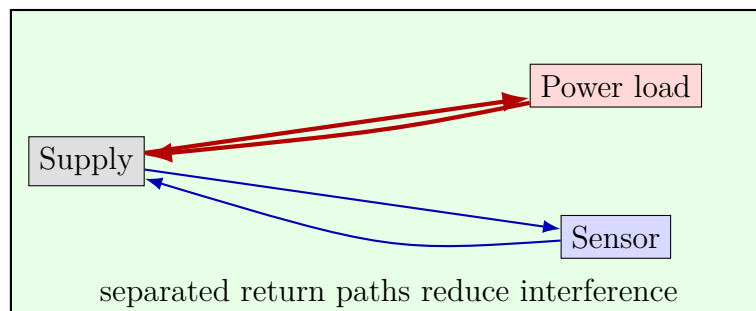


Figure 3.15: Illustration of separated return current paths.

3.16.2 Ground Plane

A ground plane is a large copper area connected to ground. It reduces impedance and can improve noise behavior. In simple single-sided educational boards, a full ground plane may not always be possible, but the designer can still use large ground tracks and short return paths.

3.16.3 Star Ground

In some circuits, especially audio or measurement circuits, several ground returns are connected to a single reference point. This method is called star grounding. It helps prevent high-current ground paths from disturbing low-level signals.

3.17 PCB Routing Strategy

Routing should be performed after careful placement. If routing becomes too complex, the first solution is not to force tracks everywhere, but to improve component placement.

3.17.1 Recommended Routing Order

A practical routing order is:

1. route power supply tracks;
2. route ground returns or prepare the ground plane;
3. route critical signals such as clocks, feedback, or sensor signals;
4. route remaining low-speed signals;
5. add test points;
6. inspect clearances and track widths;
7. print and verify the layout at real scale.

3.17.2 Avoiding Routing Problems

The following habits help reduce routing problems:

- avoid unnecessary wire crossings in the schematic;
- group components belonging to the same function;
- rotate components to reduce track length;
- keep connectors at board edges;
- avoid routing tracks between close pads when not necessary;
- use jumpers only when they simplify a single-sided board;
- label all connectors and test points.

3.17.3 Use of Jumpers on Single-Sided Boards

In single-sided PCB design, it may be impossible to route all connections without crossings. A jumper is a short insulated wire used to cross over other tracks. Jumpers should be minimized, but they are acceptable in educational circuits if clearly documented.

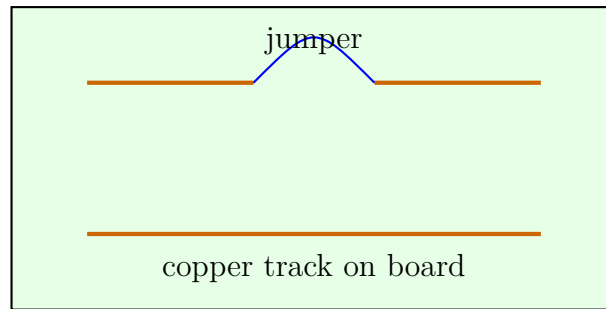


Figure 3.16: Use of a jumper wire on a single-sided PCB.

3.18 Manufacturing Files and Real-Scale Verification

Before manufacturing a PCB, the layout must be verified at real scale. This is especially important for connectors, integrated circuits, and mechanical mounting holes.

3.18.1 Real-Scale Printing

A real-scale print allows the student to place actual components over the printed layout. This simple step can reveal wrong footprints, inverted connectors, insufficient spacing, and mechanical conflicts.

3.18.2 Gerber and Drill Files

In professional fabrication, PCB software generates Gerber files for copper layers, solder mask, and silkscreen, as well as drill files for holes. In educational manual fabrication, a printed artwork may be sufficient, but students should understand that industrial fabrication requires standardized manufacturing files.

Table 3.4: Common PCB manufacturing outputs.

File or document	Purpose
Copper layer artwork	Defines the copper tracks and pads
Drill file	Defines hole positions and diameters
Silkscreen layer	Shows component references and text on the board
Solder mask layer	Defines areas protected by solder mask
Board outline	Defines the external shape and dimensions
Bill of materials	Lists components required for assembly
Pick-and-place file	Used in automated SMD assembly

3.19 Soldering Quality and Common Defects

A PCB may be well designed but fail because of poor soldering. Soldering quality is therefore part of circuit realization. A good solder joint must be shiny, mechanically stable, and electrically continuous.

3.19.1 Good Solder Joint

A good solder joint has a smooth shape and covers both the pad and the component lead. It should not contain excessive solder, cracks, or movement marks.

3.19.2 Common Soldering Defects

Table 3.5: Common soldering defects and their consequences.

Defect	Possible cause	Consequence
Cold solder joint	insufficient heating or movement during cooling	intermittent contact
Solder bridge	too much solder between close pads	short circuit
Insufficient solder	not enough solder or poor wetting	weak mechanical and electrical contact
Lifted pad	excessive heat or mechanical force	broken connection
Burnt board	overheating during soldering	damaged substrate and poor reliability
Wrong component height	poor assembly method	mechanical stress or enclosure conflict

3.19.3 Inspection After Soldering

After soldering, the board should be inspected with the naked eye and, if possible, using a magnifier. Solder bridges must be removed before powering the circuit. Component orientation must be checked again because visual inspection after soldering often reveals reversed LEDs, diodes, or electrolytic capacitors.

3.20 Design Example: PCB Documentation File

A complete PCB documentation file should contain enough information to reproduce the circuit. For a student project, the following structure is recommended:

1. project title and objective;
2. functional block diagram;
3. complete schematic;
4. calculation of important component values;
5. PCB placement drawing;
6. PCB routing drawing;
7. bill of materials;
8. manufacturing notes;
9. inspection checklist;
10. test results.

This structure trains students to think like engineers: design, justify, realize, verify, and document.

3.21 Detailed Design Rule Check Before Fabrication

Before a PCB is fabricated, the layout must be checked carefully. This verification is called design rule check, or DRC. The purpose of DRC is to detect errors that may prevent the board from being manufactured or assembled correctly.

Table 3.6: Examples of educational PCB design rules.

Rule			Reason	Typical value
Minimum width	signal	track	Avoid broken tracks during etching	0.4 to 0.6 mm
Minimum width	power	track	Reduce heating and voltage drop	1 mm or more
Minimum clearance			Avoid short circuits	0.4 to 0.8 mm
Pad diameter			Ensure soldering area	1.6 to 2.2 mm
Board edge clearance			Avoid copper damage	1 to 2 mm

A useful manual method is to print the schematic and the PCB layout, then mark each connection after verifying it. This prevents missing connections, especially when the circuit is routed manually.

3.21.1 Unconnected Nets and Short Circuits

An unconnected net is a connection present in the schematic but absent on the PCB. It produces an open circuit. A short circuit is the opposite problem: two nets that should be separated are accidentally connected. Both errors must be detected before soldering.

3.22 Copper Areas and Thermal Reliefs

Copper areas are large copper surfaces used for ground, power distribution, heat dissipation, or shielding. In educational circuits, a ground copper area can simplify routing and reduce the impedance of the ground connection.

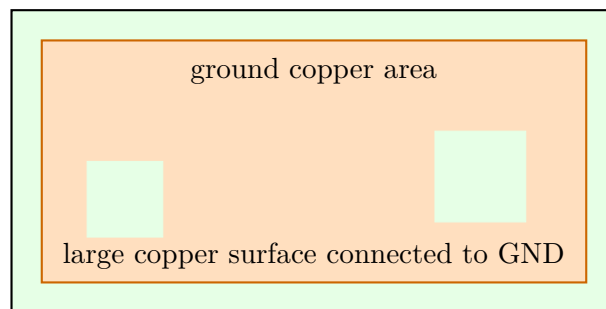
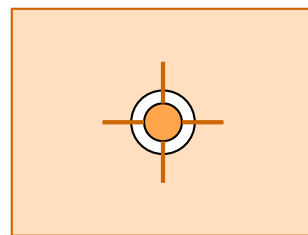


Figure 3.17: Simplified ground copper area on a PCB.

When a pad is connected directly to a large copper area, soldering may become difficult because the copper absorbs heat. Thermal reliefs solve this problem by connecting the pad through small copper spokes.



thermal relief connection

Figure 3.18: Principle of a thermal relief connection.

3.23 Drilling, Pads, and Mechanical Tolerance

Drilling is a critical operation for through-hole PCBs. If the hole is too small, the lead cannot be inserted. If it is too large, the pad becomes weak and the solder joint is less reliable.

Table 3.7: Indicative hole diameters for common through-hole components.

Component type	Lead diameter	Suggested hole
Small resistor	0.5 to 0.6 mm	0.8 mm
Small diode	0.6 to 0.8 mm	0.9 to 1.0 mm
Electrolytic capacitor	0.6 to 1.0 mm	0.9 to 1.2 mm
Pin header	about 0.6 mm	0.9 to 1.0 mm
Terminal block	1.0 to 1.3 mm	1.3 to 1.6 mm

A practical rule for manual PCB design is:

$$D_{pad} \geq D_{hole} + 0.8 \text{ mm.} \quad (3.3)$$

This keeps enough copper around the hole after drilling.

3.24 Soldering Quality and First Article Verification

A solder joint must provide both electrical continuity and mechanical strength. Poor soldering may create intermittent faults that are difficult to find. A good joint is smooth, well wetted, and free from bridges.

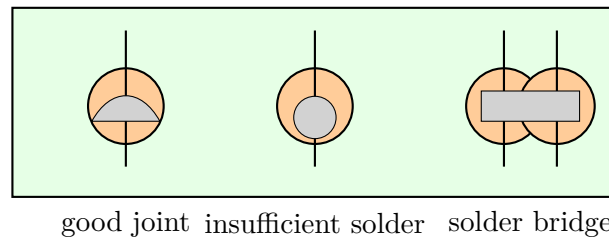


Figure 3.19: Simplified examples of solder joint quality.

The first assembled board must be verified carefully. This verification includes component fit, connector accessibility, polarity marks, continuity tests, first power-up with current limitation, and measurement of main voltages. A prototype that works once is not necessarily a good design. A good design must be reproducible, inspectable, and safe to assemble.

3.25 Chapter Summary

This chapter presented the main wiring techniques used in electronic circuit realization. It introduced printed circuit board technology, PCB materials, layout methodology, artwork preparation, photolithography, copper etching, post-etching treatment, electrical verification, repair techniques, and SMD circuits.

A good PCB must be electrically correct, mechanically reliable, easy to inspect, and safe to use. Careful preparation before manufacturing reduces errors and improves the quality of the final electronic project.

3.26 Exercises

Exercise 1: Wiring Methods

Compare breadboard wiring, perforated board wiring, and printed circuit board realization. Give one suitable application for each method.

Exercise 2: PCB Structure

Draw and describe the main parts of a single-sided printed circuit board. Explain the role of the substrate, copper tracks, pads, and holes.

Exercise 3: Track Width and Clearance

Answer the following questions:

1. Why must power tracks be wider than signal tracks?
2. Why is clearance between tracks important?
3. What may happen if two high-voltage tracks are too close?

Exercise 4: LED Indicator PCB

A red LED has a forward voltage of 2 V and the desired current is 10 mA. The supply voltage is 5 V.

1. Calculate the current-limiting resistor.
2. Choose a standard resistor value.
3. Draw a simple PCB placement with an input connector, the resistor, and the LED.

Exercise 5: PCB Inspection

List five points that must be checked before soldering components on a newly fabricated PCB.

Exercise 6: PCB Repair

Explain how to repair:

1. a broken copper track;
2. a solder bridge between two pads;
3. an unwanted copper connection.

Chapter 4

Basic Principles of Troubleshooting Electronic Circuits

Learning objectives

- Understand the meaning of troubleshooting in electronic circuits.
- Identify common component failures and their symptoms.
- Recognize the main causes of electronic circuit failures.
- Learn how to use basic measuring instruments for diagnosis.
- Apply systematic methods for testing and locating faults in electronic circuits.
- Develop safe and logical habits for maintenance and repair work.

4.1 Introduction

Troubleshooting is the process of finding, identifying, and correcting faults in an electronic circuit. Even if a circuit is correctly designed, practical problems may appear during assembly, testing, or operation. These problems may be caused by incorrect soldering, damaged components, wrong connections, power supply faults, excessive temperature, or incorrect use.

In electronic projects, troubleshooting is an essential skill. It helps the student understand the real behavior of circuits and compare it with the theoretical design. A good technician or engineer does not test randomly. Instead, troubleshooting must be performed using a logical method, appropriate measuring instruments, and safety precautions.

4.2 General Troubleshooting Approach

Troubleshooting should follow a systematic procedure. The objective is to move from general observations to precise fault localization.

1. Observe the circuit visually.
2. Check the power supply.
3. Verify component orientation and values.
4. Test continuity and detect short circuits.
5. Measure voltages at important nodes.
6. Compare measured values with expected values.
7. Isolate the faulty stage or component.
8. Correct the fault and test again.

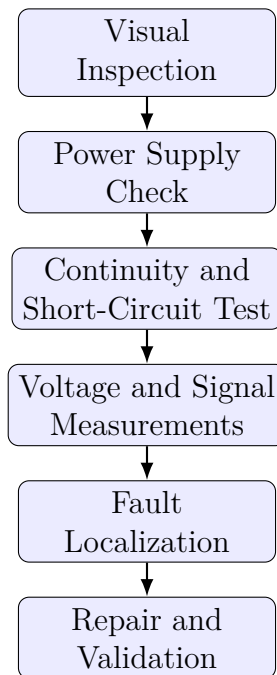


Figure 4.1: General troubleshooting procedure for electronic circuits.

Important note. Troubleshooting must always begin with simple checks. Many faults are caused by reversed components, missing connections, poor solder joints, or incorrect power supply polarity.

4.3 Component Failures

Electronic components may fail in different ways. Some failures are visible, such as a burnt resistor or an exploded capacitor. Other failures are not visible and require measurement.

4.3.1 Open-Circuit Failure

An open-circuit failure occurs when a component or connection is interrupted. In this case, current cannot flow through the component.

Examples include:

- broken copper track;
- disconnected wire;
- burnt resistor becoming open;
- broken component lead;
- damaged switch contact.

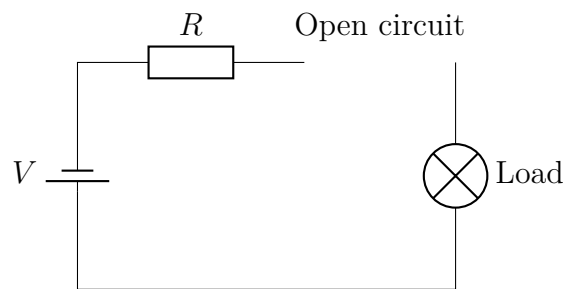


Figure 4.2: Example of an open-circuit fault.

4.3.2 Short-Circuit Failure

A short circuit occurs when two points that should be electrically separated become connected. It may cause excessive current, overheating, power supply shutdown, or component damage.

Common causes include:

- solder bridge between two pads;
- incorrect wiring;
- damaged insulation;
- metallic object on the board;
- wrong PCB routing.

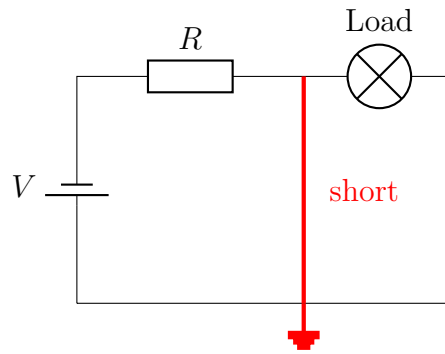


Figure 4.3: Example of a short-circuit fault.

4.3.3 Parameter Drift

Parameter drift occurs when the value of a component changes with time, temperature, humidity, or ageing. For example, a resistor may change value, or an electrolytic capacitor may lose capacitance.

4.3.4 Intermittent Failure

An intermittent failure appears only from time to time. It may be caused by poor soldering, loose connectors, vibration, or temperature variation. These faults are often difficult to locate because the circuit may work correctly during testing and fail later.

4.4 Failures of Common Components

4.4.1 Resistor Failures

Resistors may fail due to excessive current or excessive power dissipation. A resistor can become open, change value, or show visible burning.

Symptoms of resistor failure include:

- no current in part of the circuit;
- incorrect biasing voltage;
- visible overheating or color change;
- abnormal smell or smoke during operation.

The power dissipated in a resistor is:

$$P = RI^2 = \frac{V^2}{R} \quad (4.1)$$

If the dissipated power is greater than the resistor power rating, the resistor may overheat and fail.

4.4.2 Capacitor Failures

Capacitors, especially electrolytic capacitors, are common sources of faults. They may dry out, lose capacitance, leak current, or become short-circuited.

Symptoms of capacitor failure include:

- excessive ripple in power supplies;
- unstable circuit operation;
- delayed startup;
- swelling or leakage in electrolytic capacitors;
- noise in audio circuits.

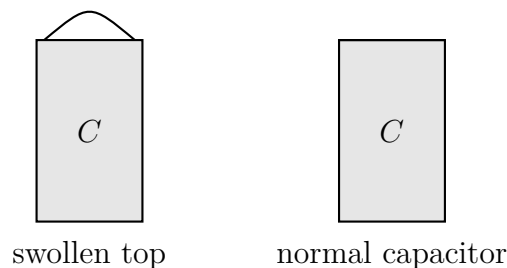


Figure 4.4: Visual comparison between a swollen electrolytic capacitor and a normal capacitor.

4.4.3 Diode Failures

A diode may fail as an open circuit or as a short circuit. It may also be damaged by excessive current, reverse voltage, or overheating.

A multimeter diode test can be used to check a diode. In the forward direction, a silicon diode typically shows a voltage drop around 0.6 to 0.7 V. In the reverse direction, it should not conduct under normal test conditions.

4.4.4 Transistor Failures

Transistors may fail due to excessive current, voltage, heat, or incorrect biasing. A damaged transistor may be shorted between terminals, open between terminals, or unable to amplify or switch correctly.

Symptoms of transistor failure include:

- load permanently ON or permanently OFF;
- no amplification;

- excessive heating;
- incorrect collector, drain, or output voltage.

4.4.5 Integrated Circuit Failures

Integrated circuits may fail due to wrong power supply polarity, electrostatic discharge, excessive temperature, overvoltage, incorrect input signals, or soldering problems.

Before replacing an integrated circuit, the following points must be checked:

- correct supply voltage;
- correct ground connection;
- correct orientation of the IC;
- presence of input signals;
- short circuits on output pins;
- decoupling capacitors.

4.5 Causes of Electronic Circuit Failures

Electronic faults may be caused by operating constraints, environmental constraints, assembly mistakes, or design errors.

4.5.1 Electrical Stress

Electrical stress occurs when a component is subjected to voltage, current, or power higher than its rated value. Examples include:

- overvoltage;
- overcurrent;
- reverse polarity;
- electrostatic discharge;
- short circuit at the output;
- excessive load current.

4.5.2 Thermal Stress

Heat is one of the main causes of electronic failure. Excessive temperature accelerates ageing and may destroy components.

Thermal stress may be caused by:

- insufficient heat sink;
- poor ventilation;
- excessive power dissipation;
- high ambient temperature;
- components placed too close to heat sources.

4.5.3 Environmental Stress

Environmental conditions may affect circuit reliability. Important factors include:

- humidity;
- dust;
- corrosion;
- vibration;
- mechanical shock;
- aggressive chemical atmosphere.

4.5.4 Assembly Errors

Assembly errors are very common in student projects. They include:

- reversed diode or LED;
- reversed electrolytic capacitor;
- wrong resistor value;
- wrong IC orientation;
- poor solder joint;
- solder bridge;
- missing jumper or missing connection;
- incorrect connector wiring.

4.6 Measuring Instruments for Troubleshooting

Troubleshooting requires appropriate instruments. The most common instruments are the multimeter, oscilloscope, laboratory power supply, function generator, and logic probe.

4.6.1 Digital Multimeter

The digital multimeter is the basic instrument for electronic troubleshooting. It can measure voltage, current, resistance, continuity, and sometimes capacitance, frequency, and diode voltage drop.

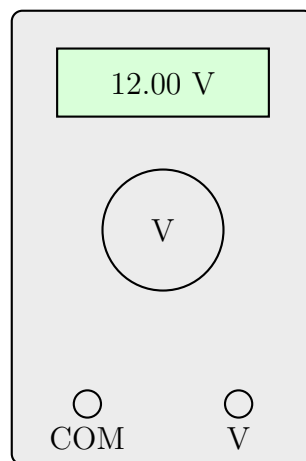


Figure 4.5: Simplified representation of a digital multimeter.

4.6.2 Oscilloscope

An oscilloscope displays voltage as a function of time. It is used to observe signals, waveforms, ripple, noise, oscillations, and timing problems.

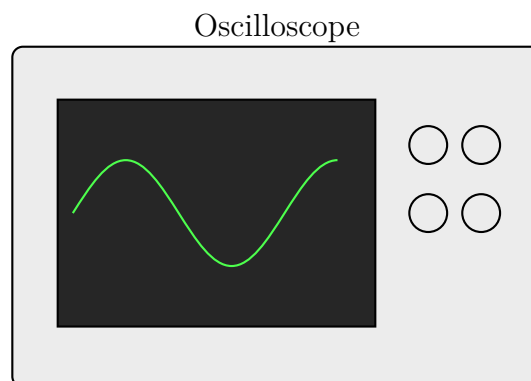


Figure 4.6: Simplified representation of an oscilloscope displaying a waveform.

4.6.3 Laboratory Power Supply

A laboratory power supply provides a controlled DC voltage. It is very useful because the voltage and current limit can be adjusted. Current limitation protects the circuit during first power-up.

4.6.4 Function Generator

A function generator produces test signals such as sine waves, square waves, and triangular waves. It is used to test amplifiers, filters, timing circuits, and signal conditioning circuits.

4.6.5 Logic Probe

A logic probe is used in digital circuits to detect logic levels. It indicates whether a point is at logic high, logic low, or changing state.

4.7 Measurement Methods

4.7.1 Visual Inspection

Visual inspection is the first step in troubleshooting. The technician should look for:

- burnt components;
- broken tracks;
- solder bridges;
- poor solder joints;
- reversed components;
- loose connectors;
- missing components;
- mechanical damage.

4.7.2 Continuity Test

The continuity test is performed with a multimeter. It verifies whether two points are electrically connected.

This test is useful for checking:

- PCB tracks;

- wires;
- switches;
- connectors;
- fuses;
- suspected broken connections.

4.7.3 Resistance Measurement

Resistance measurement is performed with the circuit powered OFF. It can be used to check resistors, detect short circuits, and verify open circuits.

Important note. Resistance and continuity measurements must not be performed on a powered circuit. The circuit must be disconnected from the power supply before using these modes.

4.7.4 Voltage Measurement

Voltage measurement is performed with the circuit powered ON. The black probe is usually connected to the reference ground, and the red probe is used to measure voltage at different nodes.

Typical voltage checks include:

- input supply voltage;
- regulated output voltage;
- voltage across a resistor;
- diode forward voltage;
- transistor bias voltages;
- IC supply pins.

4.7.5 Current Measurement

Current measurement requires inserting the multimeter in series with the circuit. Incorrect current measurement may damage the multimeter or the circuit.

For safe testing, it is often better to use the current limit of the laboratory power supply or measure voltage across a known resistor and calculate current using Ohm's law.

$$I = \frac{V_R}{R} \tag{4.2}$$

where V_R is the voltage measured across the resistor R .

4.7.6 Signal Measurement with Oscilloscope

The oscilloscope is used when the signal changes with time. It can reveal problems that a multimeter cannot detect, such as:

- ripple on a DC power supply;
- oscillation in an amplifier;
- missing clock signal;
- incorrect duty cycle;
- noise or unstable waveform;
- incorrect timing.

4.8 Systematic Troubleshooting Methods

4.8.1 Input-to-Output Method

In this method, the technician follows the signal path from the input to the output. At each stage, the signal is measured and compared with the expected signal.

This method is useful for amplifiers, filters, power supplies, and signal processing circuits.

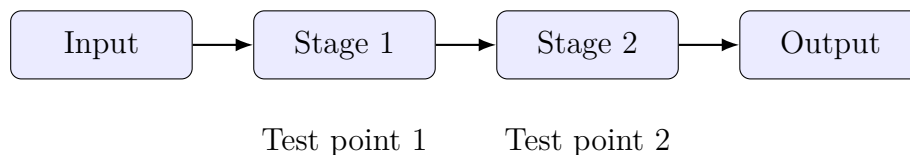


Figure 4.7: Input-to-output troubleshooting method.

4.8.2 Output-to-Input Method

In this method, the technician starts from the output and moves backward toward the input. It is useful when the output is incorrect or absent.

4.8.3 Half-Split Method

The half-split method consists of testing the circuit approximately in the middle. If the signal is correct at this point, the fault is probably after this point. If the signal is incorrect, the fault is probably before this point. This method reduces the search area quickly.

4.8.4 Substitution Method

The substitution method consists of replacing a suspected component with a known good component. This method should be used carefully and only after checking that the surrounding circuit will not damage the replacement component.

4.9 Troubleshooting Example: LED Indicator Circuit

Consider the LED indicator circuit shown in Figure 4.8. The LED does not turn ON when the supply is connected.

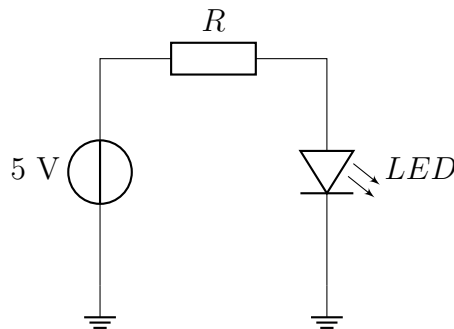


Figure 4.8: LED indicator circuit for troubleshooting example.

4.9.1 Possible Causes

The fault may be caused by:

- no supply voltage;
- reversed LED;
- wrong resistor value;
- open resistor;
- broken track or wire;

- poor solder joint;
- damaged LED.

4.9.2 Diagnostic Procedure

A logical diagnostic procedure is:

1. Measure the supply voltage.
2. Check the LED orientation.
3. Measure the voltage across the LED.
4. Measure the voltage across the resistor.
5. Power OFF the circuit and test continuity of the connections.
6. Test the LED using the diode mode of the multimeter.

4.9.3 Expected Measurements

For a red LED supplied by 5 V, the forward voltage is usually approximately 2 V. The remaining voltage appears across the resistor.

$$V_R = V_{CC} - V_F \quad (4.3)$$

If $V_{CC} = 5$ V and $V_F = 2$ V, then:

$$V_R = 5 - 2 = 3 \text{ V} \quad (4.4)$$

If the resistor is 330Ω , the current is:

$$I = \frac{3}{330} \approx 9.1 \text{ mA} \quad (4.5)$$

4.10 Troubleshooting Example: 5 V Regulated Power Supply

A 5 V regulated power supply is expected to produce a stable 5 V output, but the measured output voltage is zero.

4.10.1 Possible Causes

Possible causes include:

- no input voltage;
- reversed input polarity;
- damaged protection diode;
- incorrect regulator pinout;
- short circuit at the output;
- damaged regulator;
- poor solder joint;
- incorrect capacitor polarity.

4.10.2 Testing Strategy

The following steps may be used:

1. Disconnect the load from the output.
2. Set the laboratory power supply current limit to a safe low value.
3. Measure the input voltage at the regulator input pin.
4. Measure the regulator ground pin connection.
5. Measure the output voltage.
6. Check for short circuit between output and ground with power OFF.
7. Verify the regulator pinout using its datasheet.

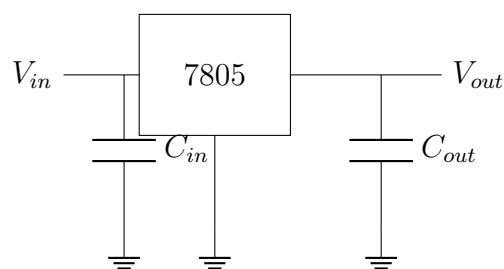


Figure 4.9: Basic measurement points of a 5 V regulated power supply.

4.11 Safety Rules During Troubleshooting

Troubleshooting must always be performed safely. The following rules are essential:

- disconnect the power supply before modifying connections;
- do not touch live circuits with bare hands;
- use current limitation during first power-up;
- check the polarity before connecting electrolytic capacitors;
- avoid shorting two points with the multimeter probe;
- use insulated tools when necessary;
- be careful with circuits connected to mains voltage;
- allow hot components to cool before touching them;
- respect laboratory safety instructions.

Important note. For student projects, it is strongly recommended to work with low-voltage circuits whenever possible. Circuits connected directly to mains voltage must only be tested under qualified supervision.

4.12 Troubleshooting Checklist

Before concluding that a component is damaged, the following checklist should be used:

1. Is the power supply voltage correct?
2. Is the polarity correct?
3. Are ground connections continuous?
4. Are all components correctly oriented?
5. Are resistor values correct?
6. Are electrolytic capacitors correctly mounted?
7. Are ICs inserted in the correct direction?
8. Are there solder bridges?
9. Are there poor solder joints?

10. Are input and output connectors correctly wired?
11. Are measured voltages close to expected values?
12. Is the load compatible with the circuit?

4.13 Common Troubleshooting Mistakes

Beginners often make mistakes during troubleshooting. The most common ones are:

- testing randomly without a logical method;
- replacing components before checking the power supply;
- measuring resistance on a powered circuit;
- using the multimeter current mode incorrectly;
- ignoring the datasheet pinout;
- forgetting the ground reference during voltage measurement;
- assuming that a new component is always functional;
- overlooking poor solder joints;
- touching or modifying the circuit while it is powered;
- not recording measured values.

4.14 First Power-Up Procedure

The first power-up is one of the most critical moments in electronic realization. Many circuits are damaged during the first connection to the power supply because of reversed polarity, short circuits, wrong component orientation, or excessive current. A careful first power-up procedure reduces the risk of damage.

4.14.1 Before Applying Power

Before applying power, the following checks should be performed:

1. visually inspect the circuit;
2. verify the polarity of electrolytic capacitors;
3. verify the orientation of diodes, LEDs, transistors, and integrated circuits;

4. check for solder bridges;
5. measure resistance between supply and ground;
6. verify that the power supply voltage is correctly adjusted;
7. set a current limit on the laboratory power supply;
8. disconnect the load if possible.

4.14.2 Using Current Limitation

A laboratory power supply with current limitation is very useful. If the circuit contains a short circuit, the current limit prevents excessive current and may protect components. For small low-voltage educational circuits, the current limit can be set to a low value at the beginning, then increased progressively if the circuit behaves normally.

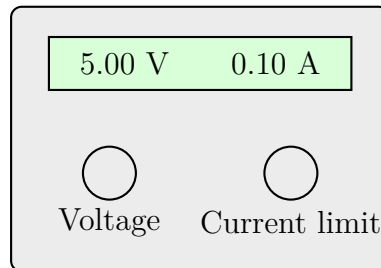


Figure 4.10: Laboratory power supply adjusted with voltage and current limit.

4.14.3 Progressive Power-Up

When possible, the circuit should be powered in stages. For example, the supply regulator can be tested first without the control circuit. Then the control circuit can be powered without the load. Finally, the load can be connected. This method helps identify the stage responsible for a fault.

4.15 Fault Classification by Circuit Function

Troubleshooting becomes easier when faults are classified according to the function of the circuit. The symptoms of a power supply fault are not the same as those of an amplifier, a digital circuit, or a sensor interface.

4.15.1 Power Supply Faults

Power supply faults are often the first faults to check because all other stages depend on the supply. Common symptoms include:

- no output voltage;
- output voltage too low;
- excessive ripple;
- regulator overheating;
- fuse blowing;
- output voltage correct without load but incorrect with load.

Possible causes include short circuits, wrong regulator pinout, insufficient input voltage, damaged rectifier diode, dry capacitor, or excessive load current.

4.15.2 Amplifier Faults

An amplifier fault may appear as no output signal, distorted signal, excessive noise, oscillation, or incorrect gain. The technician should check the supply voltage, biasing points, coupling capacitors, feedback network, input signal, and load impedance.

4.15.3 Digital Circuit Faults

Digital circuits may fail because of missing clock, floating inputs, wrong logic levels, incorrect reset signal, poor power decoupling, or wrong IC orientation. A multimeter may detect static logic levels, but an oscilloscope or logic probe is often required to observe changing signals.

4.15.4 Sensor Interface Faults

Sensor interface circuits may produce incorrect measurements because of wrong calibration, noise, poor grounding, insufficient filtering, or incorrect supply voltage. The technician must distinguish between a sensor problem and a signal conditioning problem.

4.16 Fault Tree Reasoning

Fault tree reasoning consists of starting from an observed symptom and listing possible causes in a logical order. This method avoids random testing and helps organize diagnosis.

4.16.1 Example: LED Does Not Turn ON

If an LED does not turn ON, possible causes can be organized as follows:

- no supply voltage;

Table 4.1: Fault symptoms according to circuit function.

Circuit type	Common symptom	First checks
Power supply	no voltage or unstable voltage	input voltage, regulator, short circuit, capacitors
Amplifier	no signal or distorted signal	supply, biasing, input signal, feedback
Digital logic	wrong state or no switching	supply, clock, reset, input levels
Sensor interface	incorrect measurement	sensor supply, calibration, noise, grounding
Power driver	load always ON or OFF	transistor, diode, command signal, load current

- LED reversed;
- resistor open or value too high;
- broken track;
- transistor not driven;
- damaged LED;
- ground connection missing.

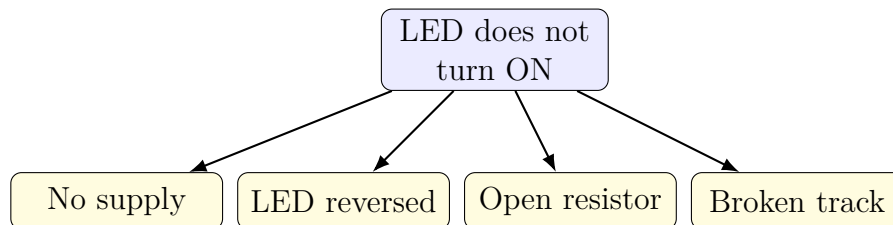


Figure 4.11: Simple fault tree for an LED that does not turn ON.

4.16.2 Prioritizing Causes

The most probable and easiest causes should be tested first. For example, measuring the supply voltage is faster than replacing the LED. Checking polarity is faster than desoldering a component. This priority saves time and reduces the risk of creating new faults.

4.17 Measurement Records and Diagnosis Tables

During troubleshooting, measured values should be written down. Without records, the student may repeat the same measurements or forget important observations. A diagnosis table helps compare theory and practice.

Table 4.2: Example of troubleshooting measurement record.

Point sured	mea-	Expected value	Measured value	Interpretation
Input supply		12 V		Supply source check
Regulator out- put		5 V		Regulator check
IC supply pin		5 V		PCB track or pin check
LED voltage		2 V approx.		LED conduction check
Transistor base		0.7 V approx.		Command check

4.17.1 Interpreting Measurements

A measurement has meaning only when it is compared with an expected value. For example, measuring 0 V at a regulator output may indicate a short circuit, a damaged regulator, no input voltage, or an incorrect ground reference. The interpretation depends on the other measurements.

4.17.2 Avoiding Measurement Errors

Measurement errors are common. The student should check the selected multimeter range, the connection of the probes, the reference ground, and the circuit state. In current measurement mode, the multimeter input is different and incorrect connection may create a short circuit.

4.18 Troubleshooting Analog and Digital Signals

Analog and digital circuits require different diagnostic habits.

4.18.1 Analog Signals

Analog signals may have any value within a range. The technician should check amplitude, offset, noise, distortion, bandwidth, and loading effects. An oscilloscope is very useful

because it shows the waveform shape.

4.18.2 Digital Signals

Digital signals are interpreted as logic LOW or logic HIGH. However, a digital signal may still be faulty if it has slow edges, noise, bouncing, missing clock pulses, or incorrect timing. A multimeter may show an average voltage that hides the real waveform.

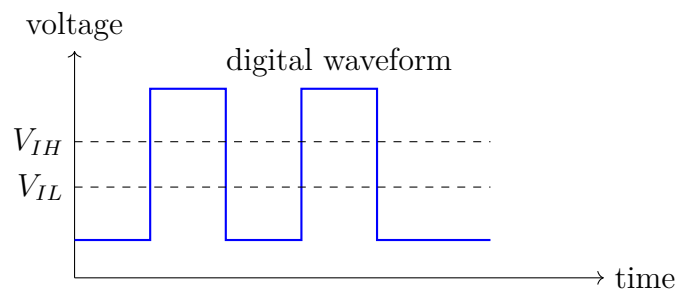


Figure 4.12: Digital signal levels and timing must sometimes be observed with an oscilloscope.

4.19 Maintenance Report After Repair

After repairing a circuit, the work should be documented. This habit is useful in laboratories and professional maintenance.

A simple maintenance report may contain:

- identification of the circuit;
- observed symptom;
- measurements performed;
- identified cause;
- repair action;
- replaced components;
- final test results;
- recommendations to avoid future failure.

Table 4.3: Example of a maintenance report structure.

Item	Description
Circuit name	
Symptom	
Initial measurement	
Faulty component or area	
Repair action	
Final validation	
Recommendations	

4.20 Detailed Diagnosis of Power Supply Faults

Many electronic faults are related to the power supply. For this reason, the supply voltage must be checked before analyzing complex functions. If the supply is wrong, all other measurements may become meaningless.

Table 4.4: Common power supply symptoms and possible causes.

Symptom	Possible causes
No output voltage	No input voltage, broken track, reversed diode, damaged regulator, output short circuit.
Low output voltage	Overload, insufficient input voltage, excessive voltage drop, wrong regulator.
Regulator overheating	Excessive load current, high input-output difference, missing heat sink.
High ripple	Faulty filter capacitor, insufficient capacitance, poor ground connection.
Supply shutdown	Short circuit, current limit activated, wrong polarity.

A safe procedure is to disconnect the load, set a current limit, verify the polarity, measure the input voltage, measure the regulator input and output pins, and check resistance between output and ground with the power off.

4.21 Signal Tracing in Analog Circuits

Signal tracing consists of following the signal from the input to the output. It is useful for amplifiers, filters, sensors, and conditioning circuits. At each stage, the measured signal is compared with the expected signal.

If the signal is correct at TP1 but absent at TP2, the fault is probably in the first stage. If the signal is correct at TP2 but distorted at TP3, the problem may be in the

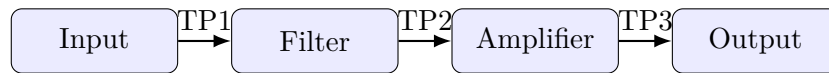


Figure 4.13: Signal tracing through successive analog stages.

amplifier or its biasing.

4.21.1 Clipping and Saturation

Amplifier saturation occurs when the circuit tries to produce an output voltage beyond the supply limits. This creates a clipped waveform. The cause may be excessive gain, wrong biasing, or an input signal that is too large.

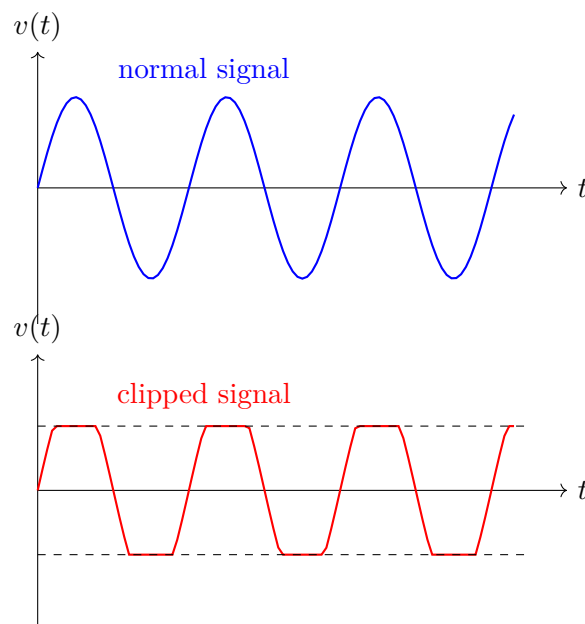


Figure 4.14: Normal waveform and clipped waveform caused by saturation.

4.22 Signal Tracing in Digital Circuits

Digital troubleshooting is based on logic levels and timing. A digital input must not be floating. It must be connected to a defined logic high or logic low state. Floating inputs are sensitive to noise and may produce random behavior.

4.23 Intermittent Faults and Preventive Maintenance

Intermittent faults appear only from time to time. They may be caused by cracked solder joints, loose connectors, broken wires, oxidized contacts, temperature variation,

Table 4.5: Examples of digital troubleshooting observations.

Observation	Possible explanation
Input always low	Missing pull-up resistor, short to ground, inactive sensor.
Input always high	Missing pull-down resistor, open switch, short to supply.
Output does not change	Missing clock, disabled output, wrong logic condition, damaged IC.
Unstable input	Floating input, poor contact, noise coupling.
Wrong sequence	Wiring error, timing problem, logic design mistake.

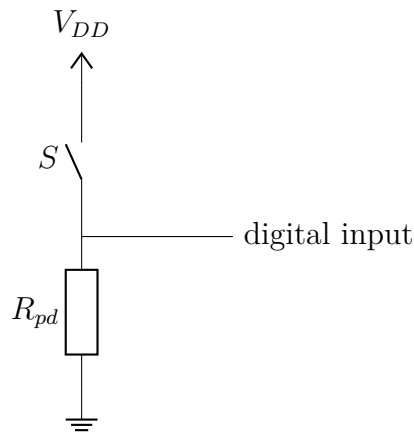


Figure 4.15: Pull-down resistor defining the input state when the switch is open.

or mechanical vibration. They are difficult to locate because the circuit may operate normally during measurement.

The technician should record the conditions under which the fault appears: after a long operating time, when the board is moved, when the temperature changes, with a specific load, or when a connector is touched. This information helps identify the physical cause of the problem.

Preventive maintenance reduces the probability of faults. It includes keeping boards clean and dry, checking connectors, avoiding mechanical stress on wires, respecting the nominal supply voltage, ensuring ventilation, and protecting circuits from dust, humidity, and electrostatic discharge.

4.24 Chapter Summary

Troubleshooting is an essential practical skill in electronics. It requires observation, measurement, logic, and safety. This chapter presented common component failures, causes of faults, measuring instruments, basic testing methods, and systematic troubleshooting approaches.

A good troubleshooting method starts with simple checks, such as visual inspection and power supply verification, before moving to detailed measurements. By comparing measured values with expected values, the technician can progressively locate the faulty stage or component and repair the circuit efficiently.

4.25 Exercises

Exercise 1: Fault Identification

For each fault below, indicate whether it is an open circuit, a short circuit, or a parameter drift:

1. A resistor is burnt and no current flows through it.
2. Two adjacent solder pads are connected by excess solder.
3. An electrolytic capacitor has lost most of its capacitance.
4. A copper track is cut.
5. A transistor has a short circuit between collector and emitter.

Exercise 2: LED Circuit Troubleshooting

A LED indicator connected to a 5 V supply does not turn ON. Give at least five possible causes and propose a logical testing procedure.

Exercise 3: Power Supply Diagnosis

A regulated power supply should provide 5 V, but the output voltage is 0 V. The input voltage at the regulator input pin is correct. Give three possible causes and explain how to test them.

Exercise 4: Multimeter Use

Answer the following questions:

1. Why must resistance be measured with the circuit powered OFF?

2. Why is current measurement more delicate than voltage measurement?
3. What is the role of continuity mode?

Exercise 5: Oscilloscope Use

Give three examples of faults that may be detected with an oscilloscope but not easily detected with a simple multimeter.

Exercise 6: Safety

List five safety precautions that must be respected during troubleshooting of electronic circuits.

Part I

Practical Teaching Materials and Project Support

Chapter 5

Technical Appendix

5.1 Appendix A: Common Component Reference Designators

Table 5.1: Common reference designators used in schematics and PCB layouts.

Designator	Component
R	Resistor
C	Capacitor
L	Inductor
D	Diode or LED
Q	Transistor
U or IC	Integrated circuit
J or CN	Connector
SW	Switch
F	Fuse
T	Transformer
RV	Variable resistor or potentiometer
X or Y	Crystal or resonator

5.2 Appendix B: Practical Checklist Before Powering a Circuit

1. Verify that the supply voltage is correct.
2. Verify supply polarity.
3. Check that ground is connected to all required points.
4. Check for short circuit between supply and ground.

5. Verify orientation of diodes and LEDs.
6. Verify orientation of electrolytic capacitors.
7. Verify IC orientation and pin 1.
8. Check resistor values.
9. Check connector wiring.
10. Inspect solder joints.
11. Remove metallic objects from the board.
12. Use a current-limited supply for first power-up.

5.3 Appendix C: Common Laboratory Instruments

Table 5.2: Common laboratory instruments and their uses.

Instrument	Main Use
Digital multimeter	Measures voltage, current, resistance, continuity, and diode drop.
Oscilloscope	Displays voltage waveform as a function of time.
Function generator	Produces test signals such as sine, square, and triangular waves.
DC power supply	Provides adjustable voltage and current limitation.
Soldering station	Used for soldering and desoldering components.
Logic probe	Detects logic high, logic low, and pulse activity.

5.4 Appendix D: Student Self-Evaluation Sheet

1. I can identify common passive and active components.
2. I can read a simple electronic schematic.
3. I can prepare a block diagram for a simple electronic system.
4. I can calculate a resistor for an LED circuit.
5. I can prepare a bill of materials.
6. I can use a multimeter safely.
7. I can solder through-hole components correctly.

8. I can verify a PCB before soldering.
9. I can identify common soldering defects.
10. I can troubleshoot a simple electronic circuit.
11. I can write a technical report for a mini-project.
12. I can present my work orally using technical vocabulary.

5.5 Appendix E: Recommended Rules for Good Documentation

- Use clear titles and subtitles.
- Number figures and tables.
- Refer to figures and tables in the text.
- Use standard symbols in schematics.
- Indicate all component values.
- Include component references in the schematic and BOM.
- Explain calculations step by step.
- Present measurements in tables.
- Discuss differences between theory and practice.
- Mention all sources and datasheets used.

5.6 Appendix F: Resistor Color Code

The resistor color code is used to identify the nominal resistance value and tolerance of through-hole resistors. Each color corresponds to a digit, a multiplier, or a tolerance value. Although digital multimeters are commonly available, students should still know how to read resistor color bands because it helps detect assembly mistakes quickly.

For example, a resistor with the bands brown, black, red, and gold has the value:

$$10 \times 10^2 = 1000 \, \Omega = 1 \, \text{k}\Omega \quad (5.1)$$

with a tolerance of approximately 5%.

Table 5.3: Basic resistor color code.

Color	Digit	Multiplier
Black	0	10^0
Brown	1	10^1
Red	2	10^2
Orange	3	10^3
Yellow	4	10^4
Green	5	10^5
Blue	6	10^6
Violet	7	10^7
Grey	8	10^8
White	9	10^9
Gold	—	10^{-1}
Silver	—	10^{-2}

5.7 Appendix G: Preferred Values and Practical Component Choice

Electronic components are not available in all possible values. Resistors and capacitors are manufactured according to preferred value series such as E6, E12, and E24. When a calculation gives a non-standard value, the designer selects the closest available value and verifies that the circuit still works correctly.

Table 5.4: E12 preferred values for one decade.

10	12	15	18	22	27	33	39	47	56	68	82
----	----	----	----	----	----	----	----	----	----	----	----

For example, if the calculated resistor value is $280\ \Omega$, possible standard choices are $270\ \Omega$ or $330\ \Omega$. The final choice depends on whether the designer wants a slightly higher or lower current.

5.8 Appendix H: Common Schematic and PCB Checklist

The following checklist may be used before submitting a mini-project file or before manufacturing a PCB.

1. The title and objective of the circuit are clearly written.
2. The block diagram is consistent with the schematic.
3. All components have unique reference designators.

4. All component values are indicated.
5. Power supply and ground symbols are clear.
6. All connectors are labeled.
7. Polarized components have visible orientation marks.
8. Integrated circuit pin numbers are checked with datasheets.
9. Footprints correspond to real components.
10. Decoupling capacitors are close to integrated circuits.
11. High-current tracks are wide enough.
12. Sensitive signals are separated from power switching paths.
13. Mounting holes and mechanical constraints are considered.
14. The PCB has been printed at real scale and checked with actual components.
15. The bill of materials is complete.

5.9 Appendix I: Glossary of Basic Terms

Block diagram: simplified representation of the main functions of a system.

Schematic: electrical representation showing components and connections.

Footprint: physical representation of a component on a PCB.

Pad: copper area used for soldering a component lead or terminal.

Track: copper conductor connecting two points on a PCB.

Via: plated hole connecting two copper layers in a double-sided or multilayer PCB.

Clearance: distance between two conductors through air.

Creepage: distance between two conductors along an insulating surface.

Decoupling: reduction of power supply disturbances using local capacitors.

Troubleshooting: logical process used to find and correct faults.

Continuity test: measurement used to verify that two points are connected.

Short circuit: unwanted low-resistance connection between two points.

Open circuit: interrupted path where current cannot flow.

5.10 Appendix J: Common Symbols Used in Basic Electronic Drawings

The following table summarizes common schematic symbols and their usual meanings. It is intended as a practical reference for students preparing simple electronic drawings.

Table 5.5: Common schematic symbols and practical meaning.

Symbol family	Practical meaning
Resistor	Current limitation, voltage division, biasing, pull-up or pull-down function.
Capacitor	Filtering, coupling, decoupling, timing, energy storage.
Inductor	Filtering, magnetic energy storage, switching power circuits.
Diode	Current flow in one direction, protection, rectification, voltage reference.
LED	Visual indication of current flow or circuit state.
Transistor	Switching, amplification, driving a load.
Operational amplifier	Amplification, comparison, filtering, signal conditioning.
Connector	Interface between the PCB and external wires or devices.
Switch	Manual or mechanical control of a circuit state.
Ground	Reference node for voltage measurements and current return path.

5.11 Appendix K: Practical Reading Guide for Component Markings

Small components often use abbreviated markings. For example, an SMD resistor marked 472 usually means:

$$47 \times 10^2 = 4700 \, \Omega = 4.7 \, \text{k}\Omega. \quad (5.2)$$

The marking 103 means:

$$10 \times 10^3 = 10000 \, \Omega = 10 \, \text{k}\Omega. \quad (5.3)$$

Small capacitors may use codes expressed in picofarads. For example, 104 usually means:

$$10 \times 10^4 \, \text{pF} = 100000 \, \text{pF} = 100 \, \text{nF}. \quad (5.4)$$

Electrolytic capacitors usually display capacitance and voltage directly, such as 470 μF and 25 V.

Important note. The voltage rating of a capacitor is not the voltage produced by the capacitor. It is the maximum voltage that the capacitor can withstand safely.

5.12 Appendix L: Suggested Structure of a Technical Report

A technical report must describe the work clearly and reproducibly. A suitable report structure for this module is:

1. Title page.
2. Introduction and objective of the circuit.
3. Functional block diagram.
4. Complete circuit schematic.
5. Explanation of the operating principle.
6. Component calculations.
7. Bill of materials.
8. PCB layout or wiring drawing.
9. Assembly procedure.
10. Measurements and test results.
11. Discussion of problems encountered.
12. Conclusion.

The report must not be limited to photographs of the final circuit. It must explain the reasoning, calculations, component choices, and validation method. A reader should be able to reproduce the circuit using only the report.

5.13 Appendix M: Vocabulary for Practical Electronics

Assembly: process of placing and soldering components on a board.

Biasing: applying DC voltages and currents to set the operating point of an active component.

Bypass capacitor: capacitor used to provide a local low-impedance path for noise or transient current.

Cold solder joint: poor solder joint caused by insufficient heating or movement during cooling.

Derating: using a component below its maximum rating to improve reliability.

Footprint mismatch: error where the PCB footprint does not correspond to the real component dimensions or pinout.

Load: circuit or device that receives power from another circuit.

Net: group of points connected electrically together in a schematic or PCB layout.

Prototype: first practical realization used for testing and improvement.

Ripple: residual AC variation superimposed on a DC voltage.

Test point: accessible node provided to simplify measurements.

Thermal relief: copper connection style used to make soldering easier when a pad is connected to a large copper area.

5.14 Appendix N: Additional Verification Tables

The following tables may be used by students as a quick reference during the preparation and verification of their electronic circuits.

Table 5.6: Pre-assembly verification table.

Question	Expected answer
Are all component references visible?	Yes, on schematic and placement drawing.
Are polarized components identified?	Yes, polarity marks are clear.
Are footprints checked?	Yes, with real components or datasheets.
Are power tracks wide enough?	Yes, according to expected current.
Are test points available?	Yes, on supply and key signals.

Table 5.7: First power-up verification table.

Step	Purpose
Use current limitation	Protect the circuit in case of short circuit.
Check input polarity	Avoid damage to polarized components.
Measure supply voltage	Confirm that the board receives correct voltage.
Measure regulator output	Verify the power section before the rest of the circuit.
Observe heating	Detect overload or incorrect assembly.

5.15 Appendix O: Useful Formula Sheet for Pre-Project Work

This formula sheet summarizes elementary relations frequently used during the preparation of simple electronic circuits. It is included as a quick reference for calculations, documentation, and verification.

5.15.1 Basic Electrical Relations

Table 5.8: Basic electrical formulas used in simple electronic design.

Quantity	Formula	Use
Ohm's law	$V = RI$	Resistors, current limitation, voltage drops.
Resistor power	$P = RI^2$	Verifying resistor power rating.
Alternative power	$P = V^2/R$	Power calculation when voltage is known.
Capacitor current	$i = C dv/dt$	Transient and filtering behavior.
Inductor voltage	$v = L di/dt$	Switching and magnetic circuits.
LED resistor	$R = (V_{CC} - V_F)/I_F$	LED current limitation.
Voltage divider	$V_o = V_i R_2/(R_1 + R_2)$	Signal scaling and biasing.

5.15.2 Filtering and Time Constants

For a resistor-capacitor circuit, the time constant is:

$$\tau = RC. \quad (5.5)$$

The time constant gives an estimation of the charging and discharging speed of the capacitor. After approximately 5τ , the transient response is usually considered close to its final value.

For a simple first-order RC low-pass filter, the cutoff frequency is:

$$f_c = \frac{1}{2\pi RC}. \quad (5.6)$$

This expression is useful when the student needs to choose a filtering capacitor or verify the expected bandwidth of a simple analog stage.

Example. If $R = 10\text{ k}\Omega$ and $C = 100\text{ nF}$, then

$$\tau = RC = 10^4 \times 100 \times 10^{-9} = 1\text{ ms}.$$

The approximate cutoff frequency is

$$f_c = \frac{1}{2\pi RC} \approx 159\text{ Hz}.$$

5.15.3 Linear Regulator Dissipation

For a linear regulator, the power dissipated as heat is:

$$P_D = (V_{in} - V_{out})I_{out}. \quad (5.7)$$

This equation is essential when deciding whether a heat sink is necessary. A regulator can deliver the correct voltage during a short test but become too hot during long operation.

Table 5.9: Examples of linear regulator dissipation.

V_{in}	V_{out}	I_{out}	P_D
9 V	5 V	0.2 A	0.8 W
12 V	5 V	0.5 A	3.5 W
15 V	5 V	0.3 A	3.0 W
24 V	12 V	0.2 A	2.4 W

5.16 Appendix P: Quick Diagnosis Matrices

The following matrices summarize common symptoms and first checks. They are useful during the first minutes of troubleshooting.

Table 5.10: Quick diagnosis matrix for supply circuits.

Symptom	First check	Instrument
No output voltage	Input voltage and polarity	Multimeter
Output too low	Load current and input voltage	Multimeter, power supply
Regulator hot	Dissipated power and load current	Multimeter, thermal observation
Ripple too high	Filter capacitor and ground path	Oscilloscope
Fuse opens	Short circuit at input or output	Continuity test

Table 5.11: Quick diagnosis matrix for signal circuits.

Symptom	First check	Instrument
No output signal	Input signal and supply voltage	Oscilloscope, multi-meter
Weak signal	Gain setting and component values	Oscilloscope
Clipped signal	Biasing and supply limits	Oscilloscope
Noisy signal	Ground path and decoupling	Oscilloscope
Unstable operation	Loose connection or floating input	Multimeter, logic probe

Table 5.12: Quick diagnosis matrix for assembly faults.

Symptom	Possible assembly cause	First action
Circuit does not start	Reversed polarity, missing supply wire	Visual inspection
One stage inactive	Broken track or missing solder joint	Continuity test
Intermittent operation	Cracked solder joint or loose connector	Mechanical inspection
Component overheats	Wrong value or wrong orientation	Power off and inspect
Unexpected short	Solder bridge or wrong wiring	Continuity test

General Conclusion

This course handout has presented the main theoretical and practical foundations required for the module *Pre-Project Works* for third-year Bachelor students in Electronics. The objective of this module is to help students move progressively from theoretical knowledge toward the practical realization of simple electronic systems.

Throughout the different chapters, students were introduced to the essential steps of an electronic project. The first chapter focused on electronic drawing techniques, including block diagrams, detailed schematics, equivalent circuits, component placement drawings, wiring diagrams, and bills of materials. These documents are necessary for understanding, communicating, and preparing an electronic design.

The second chapter dealt with the technological aspects of realizing electronic schematics. It emphasized the importance of preliminary prototyping, the international electronic grid, correct component placement, thermal considerations, and the organization of passive, active, integrated, and power components on a board.

The third chapter presented wiring techniques and printed circuit board realization. It explained the structure and materials of PCBs, the preparation of artwork, transfer methods, copper etching, drilling, inspection, repair techniques, and the basic principles of surface-mount device technology. These elements allow students to understand how a theoretical circuit can be transformed into a physical and reliable prototype.

The fourth chapter introduced the basic principles of troubleshooting electronic circuits. It highlighted the importance of visual inspection, measurement methods, fault identification, component failures, and the safe use of instruments such as the multimeter, oscilloscope, laboratory power supply, and function generator.

By completing this module, students should be able to analyze a simple electronic function, select suitable components, prepare the necessary technical documentation, realize a printed circuit board, solder and test electronic components, identify common faults, and present their work through a technical report or oral presentation.

In conclusion, this module represents an important bridge between theoretical electronics and practical engineering work. It prepares students for mini-projects, final-year projects, laboratory activities, and future professional tasks in the field of electronics. The skills developed in this module are not limited to circuit realization; they also include organization, documentation, technical reasoning, safety awareness, and problem-solving.

methodology.

Bibliography

- [1] P. Gueule, *Circuits imprimés et PC*. Paris, France: Dunod, 2004.
- [2] J. Alary, *Circuits imprimés en pratique: Méthodes de fabrication de circuits imprimés, détaillées et économiques*. Paris, France: Dunod, 1999.
- [3] P. Dunand, *Tracés des circuits imprimés, compatibilité électromagnétique*. Paris, France: ETSF.
- [4] H. Mostefai, *Le dépannage des circuits électroniques*. Alger, Algeria: Editions Lamine.
- [5] R. Besson, *Technologie des composants électroniques*. Paris, France: Editions Radio.
- [6] E. Lowenberg, *Electronique: Principes et applications*. New York, NY, USA: McGraw-Hill, 1978.
- [7] M. Fray, *Schémas d'électronique: Principes et méthodes*. Paris, France: Masson & Cie, 1967.
- [8] P. Horowitz and W. Hill, *The Art of Electronics*, 3rd ed. Cambridge, UK: Cambridge University Press, 2015.
- [9] T. L. Floyd, *Electronic Devices*, 9th ed. Boston, MA, USA: Pearson, 2012.
- [10] R. L. Boylestad and L. Nashelsky, *Electronic Devices and Circuit Theory*, 11th ed. Boston, MA, USA: Pearson, 2013.
- [11] A. S. Sedra and K. C. Smith, *Microelectronic Circuits*, 7th ed. Oxford, UK: Oxford University Press, 2015.
- [12] A. P. Malvino and D. J. Bates, *Electronic Principles*, 8th ed. New York, NY, USA: McGraw-Hill Education, 2015.
- [13] C. F. Coombs and H. T. Holden, *Printed Circuits Handbook*, 6th ed. New York, NY, USA: McGraw-Hill, 2008.

- [14] R. S. Khandpur, *Printed Circuit Boards: Design, Fabrication, Assembly and Testing*. New Delhi, India: Tata McGraw-Hill, 2005.
- [15] R. Mancini, Ed., *Op Amps for Everyone: Design Reference*. Dallas, TX, USA: Texas Instruments, 2002.
- [16] IPC, *IPC-2221: Generic Standard on Printed Board Design*. Bannockburn, IL, USA: IPC Association Connecting Electronics Industries.
- [17] IPC, *IPC-A-610: Acceptability of Electronic Assemblies*. Bannockburn, IL, USA: IPC Association Connecting Electronics Industries.
- [18] KiCad Development Team, *KiCad Documentation: Schematic Capture and PCB Design*. Available: <https://docs.kicad.org>
- [19] Texas Instruments, *Analog and Embedded Processing Application Notes*. Available: <https://www.ti.com>
- [20] Analog Devices, *Analog Dialogue and Application Notes*. Available: <https://www.analog.com>