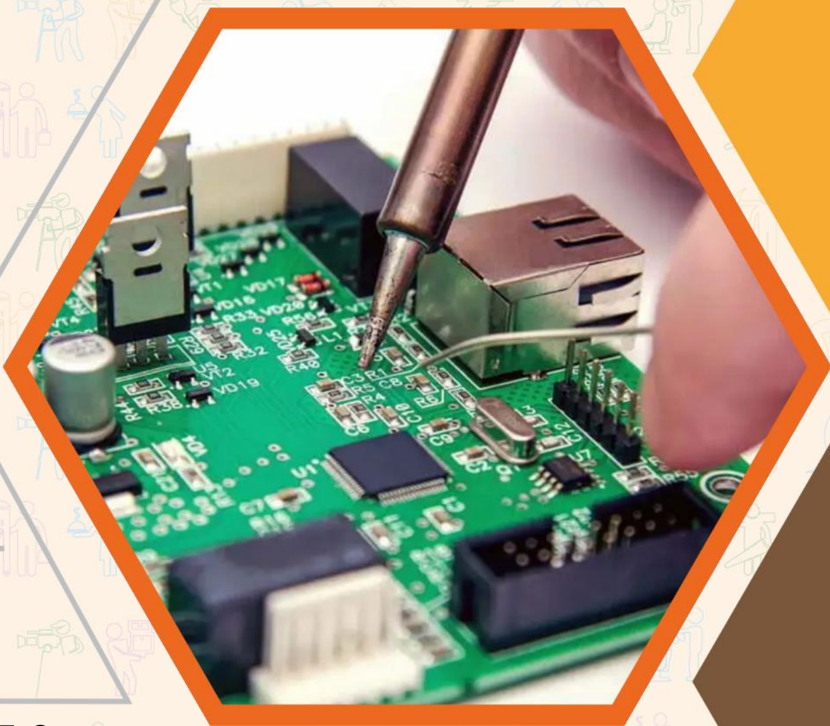
Telecom Sector Skill Council

Participant Handbook



Sector
Telecom

Sub-Sector
Handset

Occupation
Communication Electronics

Reference ID: **TEL/Q2501,** Version **5.0**
NSQF Level **4**

Telecom Surface Mount Technology (SMT) Technician

This book is sponsored by
Telecom Sector Skill Council
Estel House,3rd Floor, Plot No: - 126, Sector-44
Gurgaon, Haryana 122003
Phone: 0124-2222222
Email: tssc@tsscindia.com
Website: www.tsscindia.com

Second Edition, November 2025

Skilling is building a better India.
If we have to move India towards development then Skill Development should be our mission.

Shri Narendra Modi
Prime Minister of India

Certificate

COMPLIANCE TO QUALIFICATION PACK– NATIONAL OCCUPATIONAL STANDARDS

is hereby issued by the

TELECOM SECTOR SKILL COUNCIL

for

SKILLING CONTENT : PARTICIPANT HANDBOOK

Complying to National Occupational Standards of

Job Role/ Qualification Pack: "**Telecom Surface Mount Technology (SMT) Technician**"

QP No. "**TEL/Q2501, NSQF level 5.0**"

Date of Issuance: 8th May 2025

Valid up to*: 30th April 2028

**Valid up to the next review date of the Qualification Pack or the 'Valid up to' date mentioned above (whichever is earlier)*

Authorised Signatory
(Telecom Sector Skill Council)

Acknowledgements

Telecom Sector Skill Council would like to express its gratitude to all the individuals and institutions who contributed in different ways towards the preparation of this “Participant Handbook”. Without their contribution it could not have been completed. Special thanks are extended to those who collaborated in the preparation of its different modules. Sincere appreciation is also extended to all who provided peer review for these modules.

The preparation of this handbook would not have been possible without the Telecom Industry’s support. Industry feedback has been extremely encouraging from inception to conclusion and it is with their input that we have tried to bridge the skill gaps existing today in the industry.

This participant handbook is dedicated to the aspiring youth who desire to achieve special skills which will be a lifelong asset for their future endeavours.

About this book

India is currently the world's second-largest telecommunications market, with a total subscriber base of approximately 1.188 billion as of October 2024. As of June 30, 2025, the total number of telephone subscribers reached 1.218 billion, with wireless (mobile plus 5G FWA) subscriptions at 1.171 billion, illustrating continued robust growth.

Over the past decade and a half, the telecom industry has witnessed exponential expansion. Internet (broadband) connections surged nearly 386 million between March 2014 and June 2024, while broadband connections themselves grew more than fourteenfold. This surge reflects not only growth in subscriber numbers but also dramatic improvements in access and service affordability.

The telecom sector has been instrumental in India's socioeconomic development—playing a key role in bridging the rural–urban digital divide, enhancing connectivity, and fuelling digital inclusion.

According to data from around 2020–2021, the industry contributed about 6% of India's GDP and supported 2.2 million direct jobs and 1.8 million indirect jobs. While more recent estimates on GDP contribution specifically from telecom aren't available, the broader ICT and digital economy sector contributes over 13% of India's GDP, underscoring the telecom sector's growing economic significance.

Looking ahead, India's upcoming National Telecom Policy (NTP) aims to further catalyse growth. Among its goals are doubling telecom exports, achieving universal affordable connectivity through terrestrial and satellite networks, and creating one million new jobs by 2030. The policy also seeks to increase the telecom sector's GDP share from around 7.8% to 11% by 2030 and attract ₹1.5 lakh crore annually in telecom infrastructure investment.

This Participant Handbook delivers both theoretical knowledge and practical skill training for aspiring Surface Mount Technology (SMT) Technician in the telecom sector.

Key responsibilities include:

- Execute screen printing of solder paste onto telecom PCBs in line with prescribed process parameters.
- Perform accurate component placement on PCBs using automated and manual techniques, ensuring alignment and correct orientation.
- Operate the reflow soldering process to achieve defect-free solder joints by maintaining precise temperature profiles.
- Conduct thorough cleaning and inspection of PCBs to detect soldering defects, component misalignment, or other faults, ensuring quality and compliance.
- Carry out functional testing and segregation of PCBs, identifying and rectifying defective units by performing repairs and rework where necessary.
- Operate and maintain SMT equipment and tools, following standard operating procedures to ensure uninterrupted production.
- Apply sustainability practices in telecom production and assembly processes, focusing on energy efficiency, waste minimization, and proper handling of electronic materials.

Aligned with the latest and approved version of Surface Mount Technology (SMT) Technician (TEL/Q2501, v5.0), the handbook includes the following National Occupational Standards (NOSs):

1. TEL/N2503: Screen Printing of PCBs
2. TEL/N2504: Component Placement on PCBs
3. TEL/N2505: Re-flow soldering on PCBs
4. TEL/N2502: Cleaning and Inspection of PCBs
5. TEL/N9107: Follow sustainability practices in telecom production and assembly line processes
6. DGT/VSQ/N0101: Employability Skills (30 Hours)

Upon completion, participants will be equipped to:

- Execute the complete SMT assembly process in the telecom sector while maintaining high-quality production standards.
- Perform screen printing, component placement, reflow soldering, PCB cleaning, and inspection with precision and in alignment with industry practices.
- Identify defective PCBs through inspection and perform corrective actions to ensure functional telecom hardware.
- Operate and maintain advanced SMT machinery, optimizing production throughput and adhering to safety and quality standards.
- Implement sustainable practices during production to minimize environmental impact and reduce energy consumption.
- Communicate effectively with team members and supervisors, applying problem-solving skills under pressure to maintain smooth production flow.
- Exhibit professionalism, discipline, and strong workplace ethics, contributing to a productive and efficient work environment.
- Collaborate proactively in teams, supporting shared operational goals, and demonstrating responsibility and accountability.
- Manage time efficiently and focus on process standardization to enhance productivity without compromising on quality.

We trust this Participant Handbook will offer strong learning support and help budding professionals carve out engaging and rewarding careers in India's dynamic telecom industry.

Symbols Used

Key Learning Outcomes | Steps | Notes | Practical | Unit Objectives

Table of Contents

It is recommended that all trainings include the appropriate Employability skills Module. Content for the same is available here: **https://www.skillindiadigital.gov.in/content/list**

1. Introduction to Role and Responsibilities of a Telecom Surface Mount Technology (SMT) Technician

Unit 1.1 – Fundamentals of Electronics

Unit 1.2 – Basic Functionality of Common Electronic Components

Unit 1.3 – Printed Circuit Board (PCB)

TEL/N2503

Key Learning Outcomes

By the end of this module, the participants will be able to:

1. Identify basic electrical quantities: voltage, current, resistance, and power with their units.
2. List factors affecting resistance: material, length, cross-section, and temperature.
3. Define Ohm's Law and its relation between voltage, current, and resistance.
4. Differentiate series and parallel connections by their current, voltage, and resistance behavior.
5. List types of electronic components: resistors, capacitors, diodes, transistors, ICs, etc.
6. Identify applications of diodes and transistors in rectification, amplification, and switching.
7. Demonstrate functions of basic logic gates using truth tables and examples.
8. Assess applications of transmitters and receivers in communication systems.
9. Distinguish types of switches: toggle, push-button, DIP, and slide switches.
10. Identify different power supplies: linear, switching, UPS, and batteries.
11. Classify amplifiers: voltage, current, power, and operational amplifiers.
12. List applications of multiplexers and demultiplexers in data routing and signal management.
13. Identify PCB parts: tracks, vias, pads, silkscreen, solder mask, and substrate.
14. Differentiate PCB layers: component, solder mask, silkscreen, power, ground, and signal layers.
15. List three PCB types: single-layer, double-layer, and multi-layer.
16. Analyse PCB performance requirements: electrical, mechanical, thermal, and manufacturability aspects.
17. List PCB design steps: schematic capture, placement, routing, DRC, and Gerber generation.
18. Identify PCB inspection checks: visual, electrical, solder joints, layer alignment, and dimensions.
19. Execute PCB cleaning before pattern transfer to remove dust, grease, and oxidation.
20. List IPC standards: IPC-A-600, IPC-A-610, and IPC-2221.

UNIT 1.1: Fundamentals of Electronics

Unit Objectives

By the end of this unit, the participants will be able to:

1. Identify and explain basic electrical quantities, including voltage, current, resistance, and power, along with their units.
2. List and describe factors affecting resistance of an element, such as material type, length, cross-sectional area, and temperature.
3. Define Ohm's Law and explain the relationship between voltage, current, and resistance in an electrical circuit.
4. Differentiate between series and parallel connections in terms of current flow, voltage distribution, and overall resistance.

1.1.1 Electric Charge

An electric charge is a fundamental property of matter, rooted in the atoms that make up all substances. Atoms contain positively charged protons, negatively charged electrons, and neutral neutrons. An object becomes electrically charged when the number of electrons is either more or less than the number of protons. If an object has more protons than electrons, it has a net positive charge. Conversely, if it has more electrons than protons, it has a net negative charge.

The electric charge is a conserved quantity, meaning it cannot be created or destroyed, only transferred. It is represented by the symbol Q and is measured in Coulombs (C). One Coulomb is a very large amount of charge, equivalent to the charge of approximately 6.24×1018 electrons.

1.1.2 Electric Current

Electric current is the flow of electric charge, typically in the form of electrons, through a conductive material. It's the rate at which these charges pass a specific point in a circuit. In simpler terms, it's a measure of how many electrons are flowing per second.

The unit of electric current in the International System of Units (SI) is the Ampere (A), often shortened to amp. Constant current is denoted by the symbol I. One ampere is defined as one Coulomb of charge flowing past a point per second.

The relationship between current, charge, and time is given by the formula:

I=tQ

- I is the average current in Amperes.
- Q is the total charge in Coulombs.
- t is the time in seconds.

By convention, the direction of current flow is defined as the direction a positive charge would move, from the positive terminal to the negative terminal of a power source. This is opposite to the actual flow of electrons, which move from the negative terminal to the positive terminal.

1.1.3 Voltage (Potential Difference)

Voltage, also known as potential difference, is the difference in electric potential energy between two points in an electric field. It's the "pressure" that pushes electric charges through a circuit. Voltage is determined by the amount of work required to move a unit of charge (one Coulomb) from one point to another

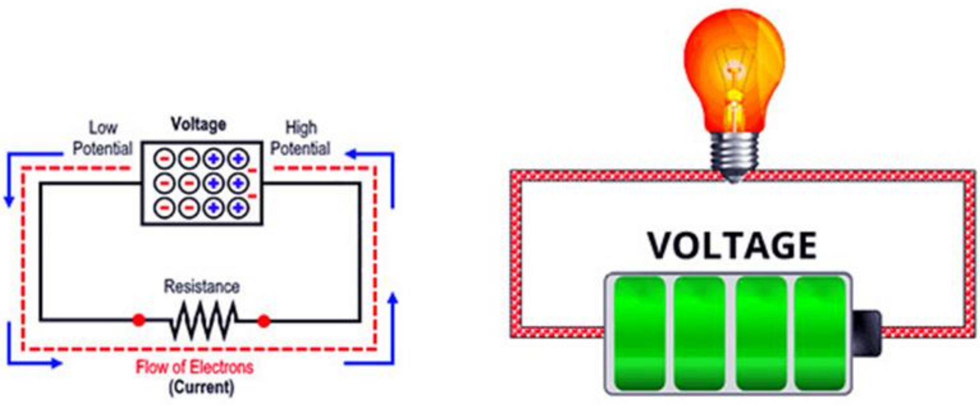


Fig 1.1.1 How Voltage Works

The unit of voltage is the volt (V). One volt is defined as one Joule of work per Coulomb of charge. The formula for voltage is:

$V=W/Q$

- V is the voltage in Volts.
- W is the work done in Joules.
- Q is the total charge in Coulombs.

Voltage can be constant, known as DC (Direct Current) voltage, or it can vary over time, known as AC (Alternating Current) voltage. DC voltage is produced by sources like batteries and solar cells (e.g., 12 VDC), while AC voltage is typically generated by power plants and is what comes from standard wall outlets (e.g., 240 VAC).

1.1.4 Resistance

Resistance is the opposition a material or a circuit component offers to the flow of electric current. It's a measure of how much a material resists the movement of electrons. The more resistance a circuit has, the more voltage is required to push a given amount of current through it.

Resistance is measured in Ohms (Ω). Materials with low resistance are called conductors, while materials with very high resistance are called insulators.

Several factors influence the resistance of a material:

- Length: Resistance is directly proportional to the length of the material. A longer wire offers more resistance.
- Cross-sectional Area: Resistance is inversely proportional to the cross-sectional area. A thicker wire has less resistance.
- Material Type: Different materials have different properties that affect resistance. Conductors, like copper, have low resistance, while insulators, like rubber, have high resistance.
- Temperature: For most conductors, resistance increases as temperature rises.

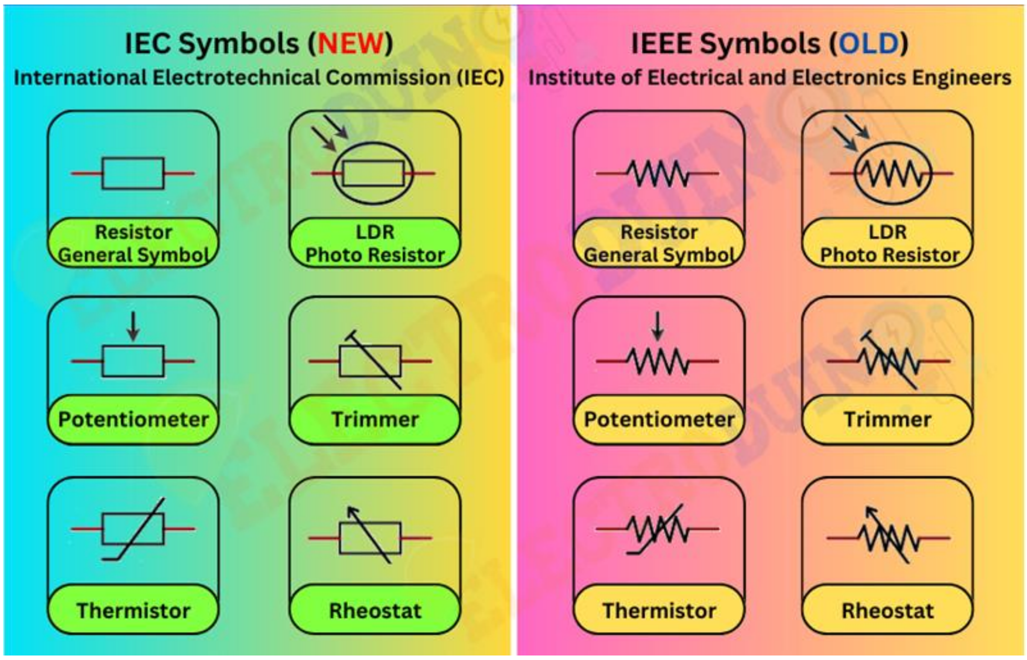


Fig 1.1.2 Resistor symbols

The resistance of a material with uniform cross-sectional area A and length l can be calculated using the formula:

R=ρl/A

- R is the resistance in Ohms.
- ρ (rho) is the resistivity of the material, a fundamental property of the material itself.
- l is the length of the material.
- A is the cross-sectional area.

1.1.5 Ohm's Law

Ohm's Law is a fundamental principle in electronics that describes the relationship between voltage, current, and resistance in an electrical circuit. It states that the voltage across a resistor is directly proportional to the current flowing through it.

The mathematical expression of Ohm's Law is: V=I×R

- V is the voltage in Volts.
- I is the current in Amperes.
- R is the resistance in Ohms.

This formula can be rearranged to find any of the three variables if the other two are known. For example, to find the current, the formula is I=V/R, and to find the resistance, it is R=V/I

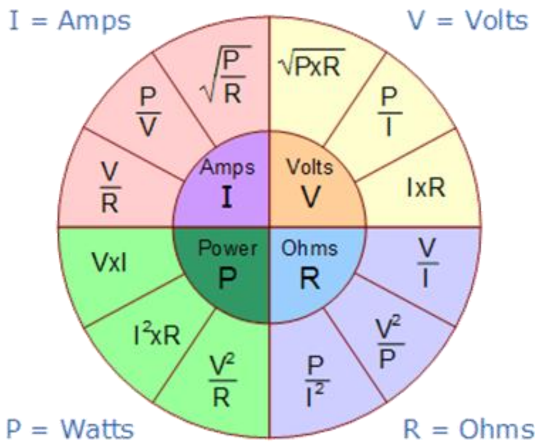


Fig 1.1.3 Ohm's Law

1.1.6 Power

Electrical power is the rate at which electrical energy is transferred or consumed in a circuit. It is measured in Watts (W). One watt is equal to one Joule of energy per second.

The most common formula for electrical power is:

P=V×I

- P is the power in Watts.
- V is the voltage in Volts.
- I is the current in Amperes.

This formula shows that power is directly related to both voltage and current. In a resistive circuit, power can also be calculated using other forms of Ohm's Law:

P=I2×R

P=V2/R

A resistor's power rating, or wattage, indicates the maximum amount of power it can safely dissipate as heat without being damaged. If the power flowing through a resistor exceeds its rating, it will overheat and fail.

1.1.7 Inductance

Inductance is the property of an electrical conductor to resist any change in the electric current flowing through it. This happens because a changing current creates a changing magnetic field, which in turn induces an opposing voltage (back electromotive force, or back EMF) in the conductor. This induced voltage works against the change in current.

This effect is most prominent in a coil of wire, known as an inductor or choke. When current flows through the coil, it creates a magnetic field. If the current increases, the magnetic field strengthens, and the induced back EMF opposes this increase. If the current decreases, the magnetic field weakens, and the back EMF opposes the decrease by trying to maintain the current flow.

The unit of inductance is the Henry (H). Smaller units like millihenry (mH) and microhenry (μH) are more commonly used. The inductance of a coil depends on its physical properties: L=N2μA/l

- L is the inductance in Henries.
- N is the number of turns in the coil.
- μ (mu) is the permeability of the core material, which measures its ability to support a magnetic field.
- A is the cross-sectional area of the core.
- l is the length of the core.

Inductors are designed to concentrate this magnetic field, often by wrapping the wire around a core made of a magnetic material like powdered iron or ferrite.

1.1.8 Capacitance

A capacitor is an electronic component that stores electrical energy in an electric field. It typically consists of two conductive plates separated by an insulating material called a dielectric. When a voltage is applied across the plates, one plate accumulates a positive charge and the other a negative charge, storing energy.

The ability of a capacitor to store charge is called capacitance. The unit of capacitance is the Farad (F). One Farad is a large amount of capacitance, so smaller units like microfarad (μF) and picofarad (pF) are more common.

Capacitors are categorized by the type of dielectric material used, as this greatly affects their performance. Common dielectric materials include ceramic, paper, mica, and plastic films. Examples of capacitors include ceramic discs, which are often used in high-frequency circuits, and electrolytic capacitors, which are polarized and can store a large amount of charge.

1.1.9 Series and Parallel Connections

When electronic components like resistors are connected, they can be arranged in either a series or a parallel configuration. The arrangement determines how voltage and current are distributed throughout the circuit.

Series Connection

In a series circuit, components are connected end-to-end, forming a single path for the current to flow. The same current flows through every component in the circuit. The total resistance of a series circuit is the sum of the individual resistances.

Current: The current is the same through all resistors:

Itotal=I1=I2=I3...

Voltage: The total voltage is the sum of the voltage drops across each resistor:

Vtotal=V1+V2+V3...

Resistance: The total resistance is the sum of all resistances:

Rtotal=R1+R2+R3...

Parallel Connection

In a parallel circuit, components are connected across each other, creating multiple paths for the current. The voltage across each component is the same, but the total current is divided among the branches.

Current: The total current is the sum of the currents flowing through each branch:

Itotal=I1+I2+I3...

Voltage: The voltage is the same across all resistors:

Vtotal=V1=V2=V3...

Resistance: The reciprocal of the total resistance is the sum of the reciprocals of the individual resistances:

Rtotal1=R11+R21+R31...

Notes

UNIT 1.2: Basic Functionality of Common Electronic Components

Unit Objectives

By the end of this unit, the participants will be able to:

1. List and classify common electronic components.
2. Explain the working principles and applications of diodes and transistors.
3. Demonstrate the logic operations of basic logic gates (AND, OR, NOT, NAND, NOR, XOR, XNOR).
4. Assess the role and applications of transmitters and receivers in communication systems.
5. Differentiate between various types of switches based on their operation and use.
6. Identify and describe different types of power supplies (linear, switching).
7. Classify amplifiers by type (e.g., voltage, current, operational) and application.
8. Explain the function and applications of multiplexers and demultiplexers in data routing.

1.2.1 Types of Electronic Components

Basically, there are two types of electronic components; active components and passive components.

1. Active Components

Active components refer to those components which require external power supply to operate them such as diode, transistor, integrated circuits, mosfets and so on.

2. Passive Components

As their name suggests, passive components do not require any electrical power to operate, unlike the active devices that need to be powered to make them work.

Passive devices do not offer amplification, gain or directionality to a circuit. But they provide attenuation to the circuit as they have a gain less than unity. Hence, the passive devices cannot generate, amplify or oscillate an electrical signal.

Passive devices can be used individually. They are used to control complex circuits or signals by getting connected together in series or in parallel combination. They are also used for generating a phase shift to signal or providing some form of feedback. But, since these devices have no power gain, they cannot multiply a signal by more than one.

Passive devices consume power in a circuit. They act like attenuators, whereas the active devices provide power to a circuit.

Passive devices are bi-directional components. Thus, in a circuit, they can be connected either way if they don't have a polarity marking; for example, electrolytic capacitors. The flow of the current from the positive to negative terminal determines the polarity of the voltage across the passive devices.

Some basic passive elements are resistor, capacitor and inductor.

1.2.2 Diode

A diode is an electrical semiconductor device. It works as a one-way switch to permit the current to flow through it in one direction only. A diode shows an exponential I-V relationship and hence, it does not have a linear relationship with the applied voltage. Therefore, its operation cannot be explained using equations such as Ohm's law. The following figure shows the I-V characteristics of a diode:

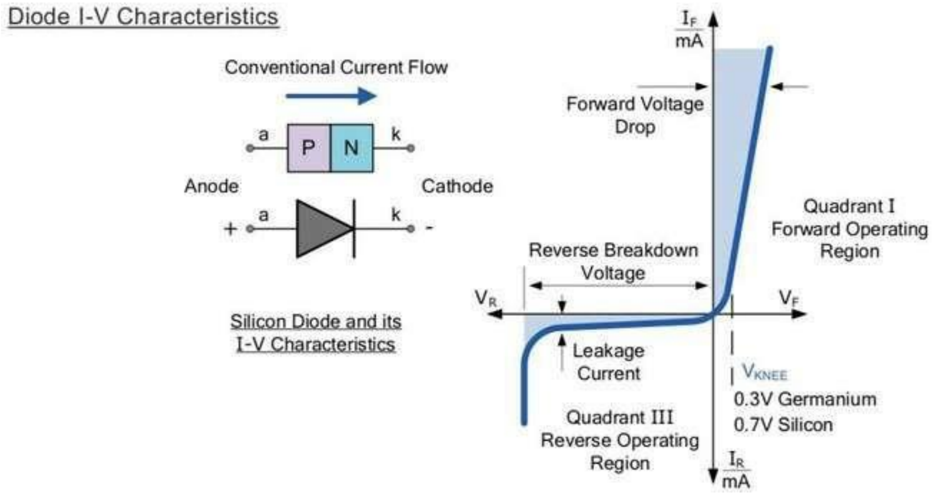


Fig. 1.2.1: I-V characteristics of a diode

The arrow in the above figure points in the direction in which current flows. Anode (A) and cathode (K) are the two terminals of the diode in which the cathode is the negative end and can be identified by the band marked on it.

A diode can be either 'forward biased' or 'reversed biased' and can be explained as follows:

Forward biased	Reversed biased
The anode terminal of the diode is more positive than the cathode terminal; minimum 0.7 volts in case of a silicon device.	The anode terminal of the diode is more negative than the cathode.
It permits the current to flow through the device.	It resists the flow of the current till its reverse breakdown voltage is reached.
	The diode becomes unstable at this point.

Fig. 1.2.2: Forward biased or reversed biased condition of diode

Diodes are made of semiconductor materials which include silicon and germanium. Unlike in silicon diodes, the conduction in germanium diodes occurs if the forward biasing voltage is 0.2 volts or more. Diodes can be classified as follows:

- **Small signal diodes:** Utilised in multiple low voltage applications
- **Power diodes:** Utilised in mains powered circuits and in rectifying

Since diodes are devices in which current flows in one direction only, they cannot be haphazardly connected in series.

The following figure shows the combination of diodes in series connection:

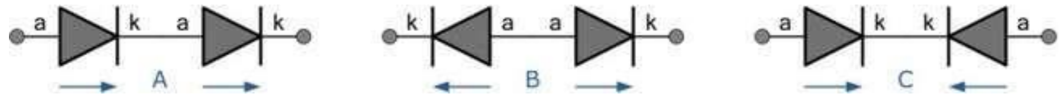


Fig. 1.2.3: Combination of diodes in series connection

In the above combination, only circuit A will conduct current.

Applications of Diode

Diodes can be used in many ways. They can work as a half wave rectifier, a full wave rectifier, a switch and so on.

Diode as a Switch

In case of forward bias, if the voltage of the diode exceeds the threshold level, generally 0.6V, then the diode performs as a short circuit and permits the current to pass. In case of reverse bias, that is if the polarity of voltage is changed, the diode performs as an open circuit and does not permit the current to flow. The following figure shows a diode as a switch:

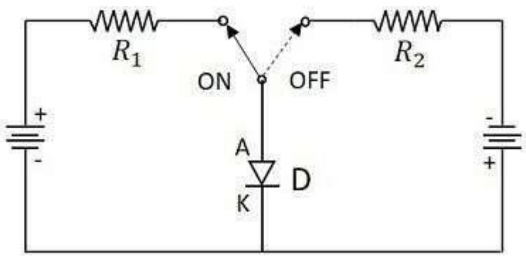


Fig. 1.2.4: A diode as a switch

Diode as a Half Wave Rectifier

The following figure shows a diode as a half wave rectifier:

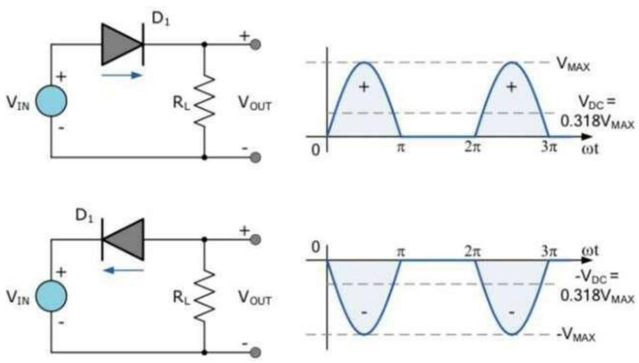


Fig. 1.2.5: A diode as half wave rectifier

The following figure explains the working of a diode as a half-wave rectifier (from above figure):

During the positive half cycle of the input supply (sinusoidal wave):	During the negative half cycle of the input supply (sinusoidal wave):
• Anode is made positive with respect to cathode so the diode gets forward biased. These results in flow of a current to the load. • Since the load is resistive, the voltage across the load resistor will be same as the supply voltage; that means, the input sinusoidal voltage will appear at the load. • The load current flow is proportional to the voltage applied.	• Anode is made negative with respect to cathode so the diode gets reverse-biased. • No current flows to the load. The circuit becomes open circuit and no voltage appears across the load.

Fig. 1.2.6: Working of diode as a half wave rectifier

The polarity of the current at the load is identical to the polarity of the voltage of the load. The output voltage is hence a periodic current, that is, pulsating DC. A capacitor is used in the rectification circuit. It is connected across the load so that a stable, continuous DC current devoid of ripples is produced.

Diode as a Full Wave Rectifier

The following figure shows a diode as a full wave rectifier:

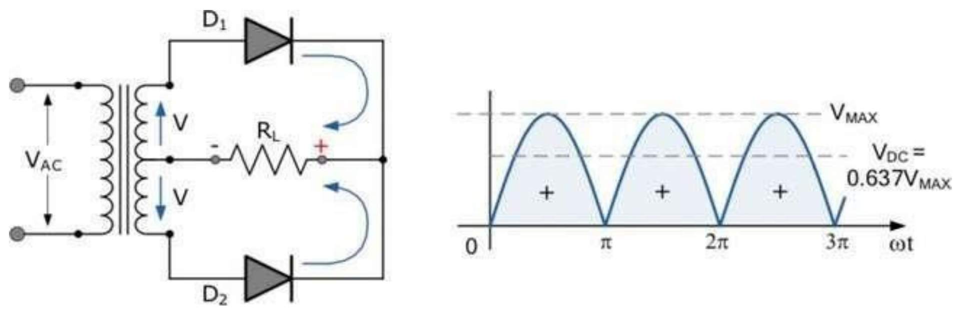


Fig. 1.2.7: Working of diode as a full wave rectifier

Diode as a Full Wave Bridge Rectifier

The figure shows a diode as a full wave bridge rectifier:

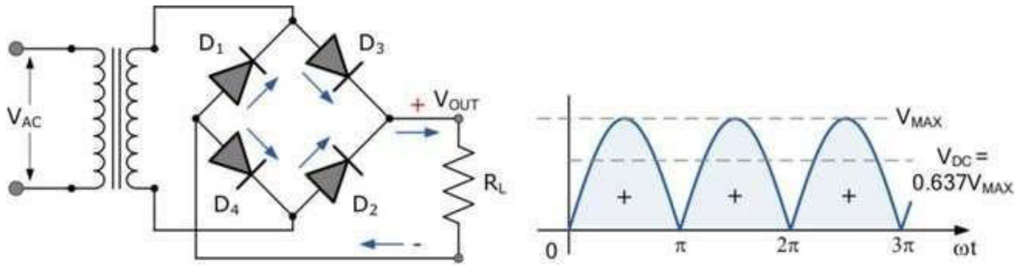


Fig. 1.2.8: Working of diode as a full wave bridge rectifier

Zener Diode

A zener diode is a specialised diode that can be used in place of a semiconductor junction diode. The latter is used to block current flow in reverse direction. At times, when the voltage in this type of diode exceeds a limit, the PN-junction of the diode breaks down and it gets damaged. The current begins to flow in the circuit at this breakdown voltage which is also called peak reverse voltage. In such cases a zener diode can be used instead. Th following figure shows the characteristics of Zener diode:

- It is designed to operate in the reverse biased mode in such a way that at a certain breakdown voltage point, the reverse voltage causes the diode to conduct in a controlled way, thereby allowing a reverse current to flow through a series limiting resistor (Rz). This breakdown voltage is called the zener voltage (Vz).
- The breakdown voltage point of a zener diode, Vz, is determined by the doping technique used during its manufacture. Zener breakdown voltages range from 2.7V to about 200V.
- The voltage, Vz, across the zener diode remains reasonably constant over a wide range of reverse currents passing through the diode.

The following figure shows the working of a zener diode:

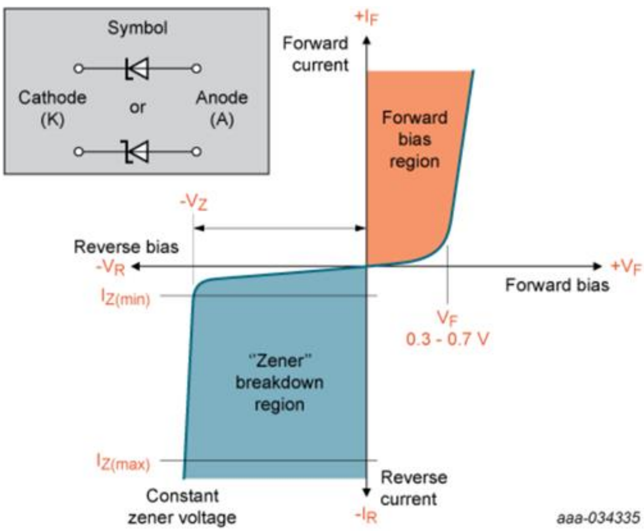


Fig. 1.2.9: Working of zener diode

In the above figure, a zener diode is shown with its reverse bias characteristics. A region can be observed where nearly constant negative voltage has been depicted. The region is unaffected by the magnitude of the current flowing across it. The changes in the magnitude of the load current does not affect the voltage of this region.

This special characteristic of a zener diode to remain in control at all times can be employed to keep a voltage source stabilised against the variations in the load or supply magnitude.

1.2.3 Transistor

A transistor is an electronic semiconductor device. It is made by including a semiconductor layer in a PN-junction diode. Whenever there a minute change in the current flowing in one lead, the transistor functions as a conductor or an insulator. It produces a considerable change in the current, voltage and power with its other two leads. The two fundamental functions of a transistor within an electrical circuit are as follows:

- Amplification (in analog circuits)
- Switching (in digital circuits)

The following figure shows the basic construction of a bipolar transistor:

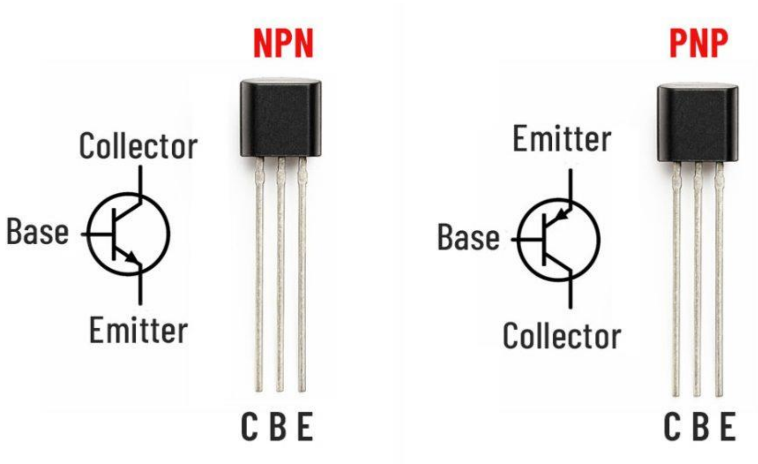


Fig. 1.2.9: Basic construction of a bipolar transistor

Bipolar transistors are used as current regulating devices. They act like current-controlled switches by controlling the amount of current across them in proportion to the amount of biasing voltage employed to their base terminal. There are basically two types of transistors; NPN and PNP. Their operations are based on the same principle but the difference lies in their biasing as well as the polarity of the power supply applied to them. The following figure shows the symbols and circuit diagram of NPN and PNP transistors:

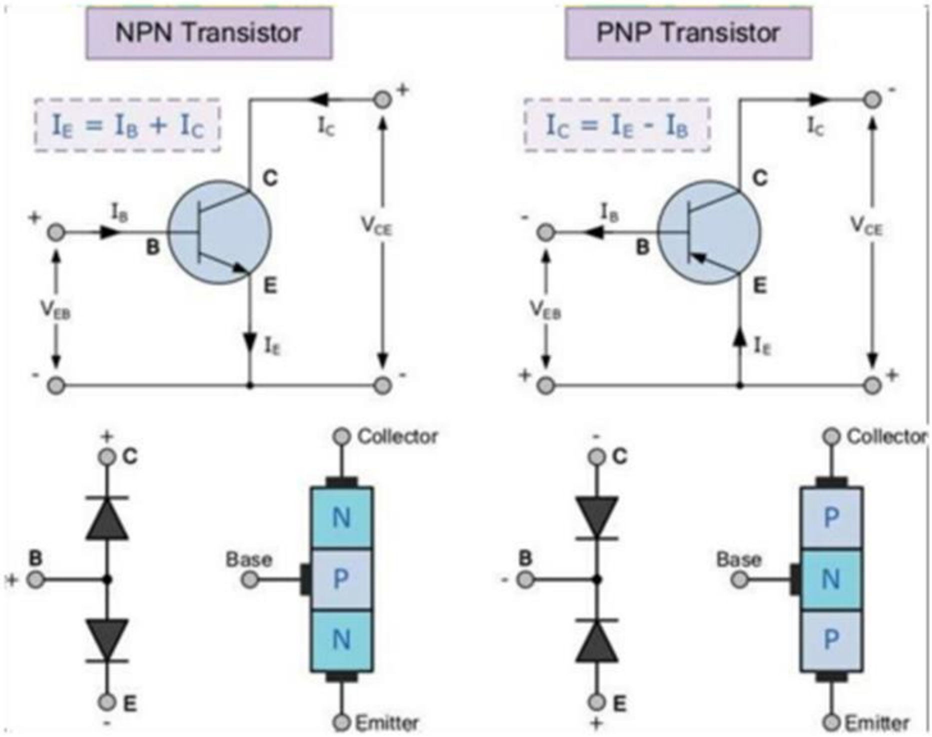


Fig. 1.2.10: Symbols and circuit diagram of NPN and PNP transistors

- The arrows in the circuit in the above diagram represent the direction of the conventional current flow between the base and the emitter terminal.
- For both transistor types, the direction of the arrow points from positive P-type region to negative N-type region, which is same as for a standard diode symbol.

Bipolar transistors can operate within three different regions. The following figure lists the three regions:

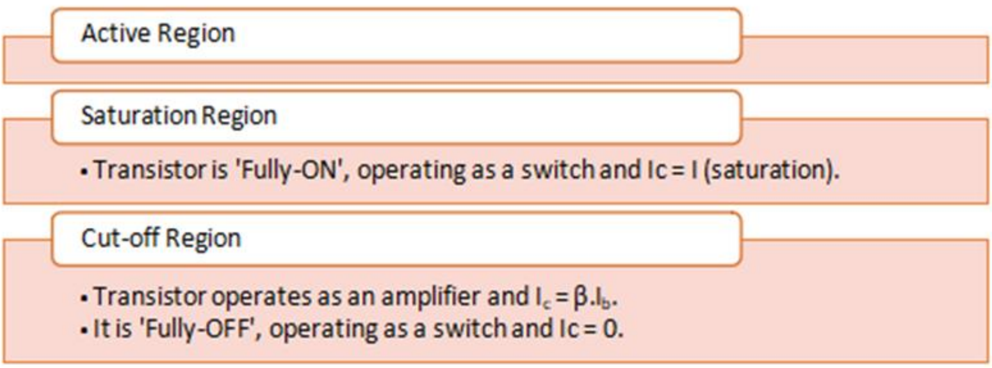


Fig. 1.2.11: Three regions of a bipolar transistor

Transistor as an Amplifier

The following figure shows working of a transistor as an amplifier:

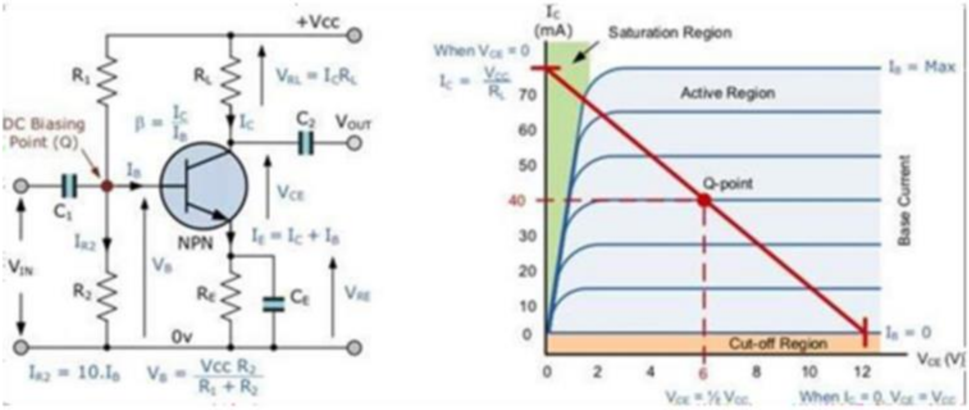


Fig. 1.2.12: Working of a transistor as an amplifier

In the above figure, a change of 0.1 V in the input voltage produces a change of 1 mA in the emitter current. This emitter current then produces a change of 1 mA in the collector current.

If a load resistance of 5 kΩ is placed at collector, it will produce a voltage of 5 V (calculated as 5 kΩ × 1 mA). Therefore, a change of 0.1 V in input produces a change of 5v in output, amplifying the voltage level of the signal.

The following table explains the current gain and voltage gain:

Current Gain	Voltage Gain
The gain in terms of current when changes in input and output currents are observed, is called as Current gain.	The gain in terms of voltage when changes in input and output currents are observed, is called as Voltage gain.
By definition, it is the ratio of the change in collector current (ΔIC) to the change in base current (ΔIB). Current gain, β=ΔIC/ΔIB	By definition, it is the ratio of change in output voltage (ΔVCE) to the change in input voltage (ΔVBE). Voltage gain, AV=ΔVCE/ΔVBE = Change in output current × effective load Change in input current × input resistance
The value of β ranges from 20 to 500. The current gain indicates that input current becomes β times in the collector current.	

1.2.4 Logic Gates

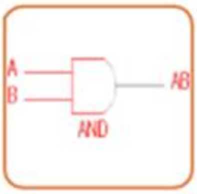
Logic gates are electronic circuits that are used to process signals which represent true or false. Normally, the positive supply voltage (typically +5 V) is represented true and 0 V is represented as false.

The following figure lists the terms related to logic states:

Logic States	
True	False
1	0
High	Low
On	Off
+5V	0V

Fig. 1.2.13: Terms related to logic states

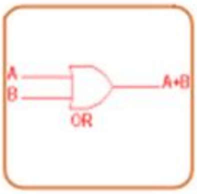
The following figure lists the basic logic gates, their symbols and truth tables:



AND Gate

The AND gate generates a high output (1) only if all its inputs are high. The AND operation is represented by a dot (.); for example, A.B. Sometimes, the dot is omitted and AND operation is represented as AB.

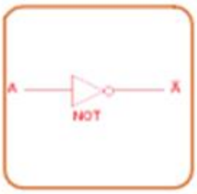
2 Input AND gate		
A	B	A.B
0	0	0
0	1	0
1	0	0
1	1	1



OR Gate

The OR gate generates a high output (1) if at least one input is high. A plus (+) is used to represent the OR operation.

2 Input OR gate		
A	B	A+B
0	0	0
0	1	1
1	0	1
1	1	1



NOT gate

The NOT gate, also known as an inverter, produces an inverted version of the input at its output. If the input variable is A, the inverted output is known as NOT A, shown as A', or A with a bar over the top.

NOT gate	
A	$\bar{A}$
0	1
1	0

Fig. 1.2.14: Basic logic gates, their symbols and truth tables

The following figure lists the universal gates, their symbols and truth tables:

NAND Gate

This is a combination of an AND gate and a NOT gate. NAND gates produce high outputs if any of the inputs is low.

It is represented by the symbol of an AND gate with a small circle at the output representing inversion.

2 Input NAND gate		
A	B	$\overline{A.B}$
0	0	1
0	1	1
1	0	1
1	1	0

NOR Gate

The NOT-OR gate is a combination of an OR gate and a NOT gate. NOR gates produce low outputs if any of the inputs is high.

It is represented by the symbol of an OR gate with a small circle at the output representing inversion.

2 Input NOR gate		
A	B	$\overline{A+B}$
0	0	1
0	1	0
1	0	0
1	1	0

Fig. 1.2.15: Universal logic gates, their symbols and truth tables

The following figure lists the combinational gates, their symbols and truth tables:

X-OR Gate

The 'Exclusive-OR' gate produces a high output when either of its two inputs, but not both, are high. The X-OR operation is represented as an encircled plus sign (⊕).

2 Input EXOR gate		
A	B	$A \oplus B$
0	0	0
0	1	1
1	0	1
1	1	0

X-NOR Gate

The 'Exclusive-NOR' gate circuit does the opposite to the X-OR gate. It generates a low output when either of its two inputs, but not both, are high. It is represented as an X-OR gate with a small circle on the output representing inversion.

2 Input EXNOR gate		
A	B	$\overline{A \oplus B}$
0	0	1
0	1	0
1	0	0
1	1	1

Fig. 1.2.16: Combinational logic gates, their symbols and truth tables

1.2.5 Transmitter

A Transmitter is defined as a combination of one or multiple electronic circuits or devices which converts the source information, also called as baseband signal, to a form suitable for transmission. Transmitters are used in the system where the sender encodes the information. AM radio transmitters and mobile phones are some of the examples. The following figure lists some characteristics of transmitters:

- It must generate a signal of desired frequency.
- It must provide some form of modulation that allows the information signal to modify a signal of higher frequency, also known as the carrier signal. Amplitude Modulation (AM) and Frequency Modulation are commonly used in broadcasting.
- It must provide power amplification to ensure that the signal level is high. This should be in such a way that it will carry over the desired distance for which the signal is to be sent.

1.2.6 Receiver

A receiver is a collection of various electronic circuits and devices which accepts the signals transmitted from the transmission medium and then converts them back to their original form understandable by humans. Television is a good example of a receiver. The following figure lists the primary requirements for a communication receiver:

Selectivity	Sensitivity
It is the ability of a receiver to select a signal of a desired frequency while rejecting those on closely adjacent frequencies.	It is the ability of a receiver to pick up weak signals.
With good selectivity, the receiver can select the desired signal and eliminate all other RF signals.	Sensitivity is directly related to receiver's gain. Gain is the factor by which an input signal is multiplied to produce the output.
Tuned circuits or LC circuits are used to obtain selectivity.	This gain can be increased by having a series of amplifications.
	Better sensitivity is attained with higher gain.

The following figure shows an overall communication system:

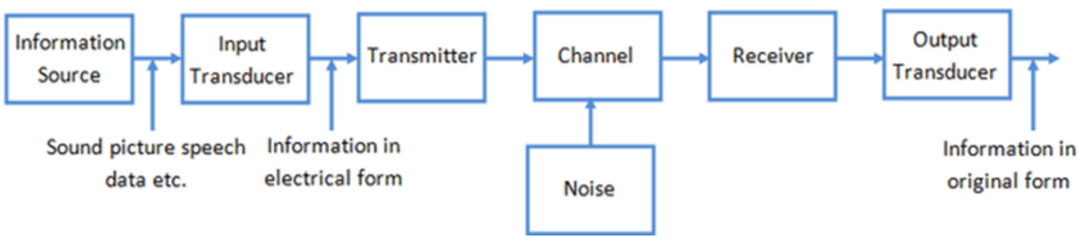


Fig. 1.2.17: An overall communication system

1.2.7 Switches

A switch can automatically or manually connect or break an electrical circuit. It mainly works with an ON (open) and OFF (closed) mechanism. The following figure lists the basic types of switches:

- Single Pole Single Throw (SPST)
- Single Pole Double Throw (SPDT)
- Double Pole Single Throw (DPST)
- Double Pole Double Throw (DPDT)

SPST Switch

An SPST, also known as toggle switch, is a basic on/off switch. It just breaks or connects the connection between two terminals. It is used for switching the power supply to a circuit. The following figure shows a simple SPST switch:

Fig. 1.2.18: A simple SPST switch

The following figure shows an SPST circuit diagram and its working:

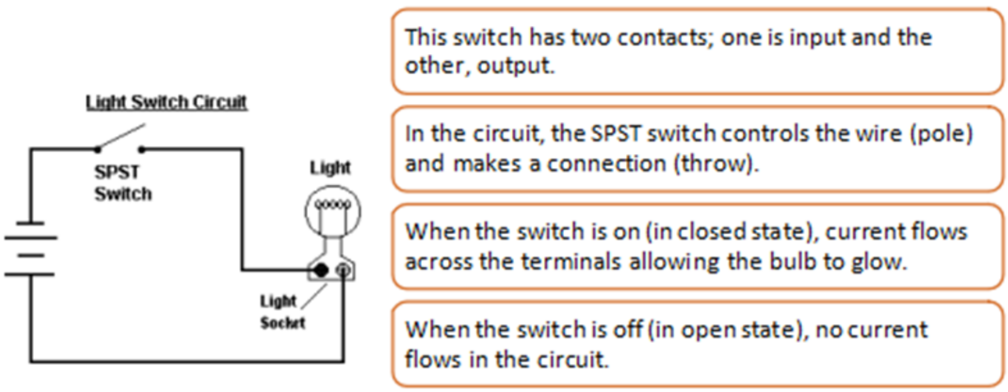


Fig. 1.2.19: SPST circuit diagram and its working

SPDT Switch

It is a three-terminal switch, with one input terminal that is common to one or both of the two output terminals.

To use an SPDT as an SPST switch, COM terminal needs to be used instead of other terminals. For example, COM with A or COM with B is used. The following figure shows an SPDT switch:

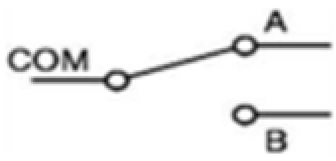
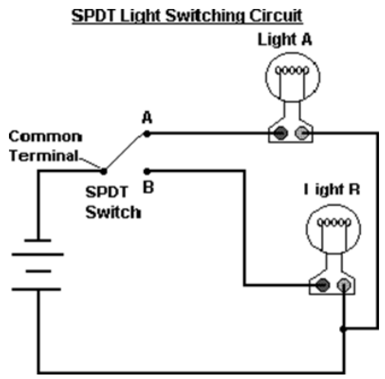


Fig. 1.2.20: SPDT switch

The following figure shows an SPDT circuit diagram and its explanation:



The above circuit demonstrates what happens when the SPDT switch is moved back and forth.

- These switches are used in a three-way circuit, where it is required to switch on/off a light from two locations; for example, from the bottom as well as top of a stairway.
- When switch A is closed only light A will be in ON position and the light B will in OFF position. Only light B will be in ON position, when the switch B is closed.
- Here, two circuits or paths can be controlled via one way or one source.

DPST switch has four terminals with two for inputs and two for outputs.

A DPST switch is used to control a 240-volt appliance, where it is essential to switch both supply lines, keeping the neutral wire permanently connected.

When this switch is in ON position, current flows through two circuits and when it is in OFF position, the flow is interrupted.

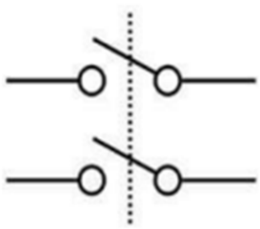


Fig. 1.2.21: DPST switch

SPDT Switch

It is a three-terminal switch, with one input terminal that is common to one or both of the two output terminals.

To use an SPDT as an SPST switch, COM terminal needs to be used instead of other terminals. For example, COM with A or COM with B is used. The following figure shows an SPDT switch:

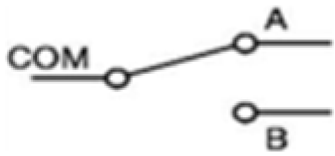
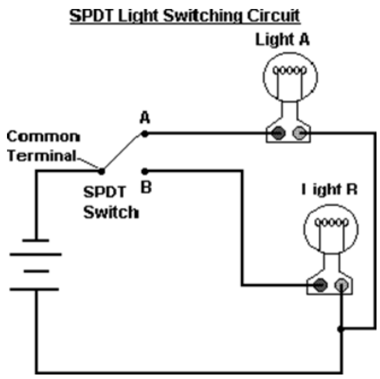


Fig. 1.2.20: SPDT switch

The following figure shows an SPDT circuit diagram and its explanation:



The above circuit demonstrates what happens when the SPDT switch is moved back and forth.

- These switches are used in a three-way circuit, where it is required to switch on/off a light from two locations; for example, from the bottom as well as top of a stairway.
- When switch A is closed only light A will be in ON position and the light B will in OFF position. Only light B will be in ON position, when the switch B is closed.
- Here, two circuits or paths can be controlled via one way or one source.

DPST switch has four terminals with two for inputs and two for outputs.

A DPST switch is used to control a 240-volt appliance, where it is essential to switch both supply lines, keeping the neutral wire permanently connected.

When this switch is in ON position, current flows through two circuits and when it is in OFF position, the flow is interrupted.

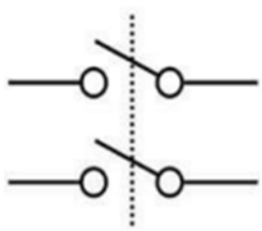


Fig. 1.2.21: DPST switch

DPDT Switch

A DPDT switch is equivalent to two SPDT switches operated by the same actuator. Two separate circuits are controlled by connecting each of the two inputs to one of the two outputs. At a time, only two loads can be in ON position. The number of ways each of the two contacts can be contacted depends on the position of the switch. The following figure shows a DPDT switch:

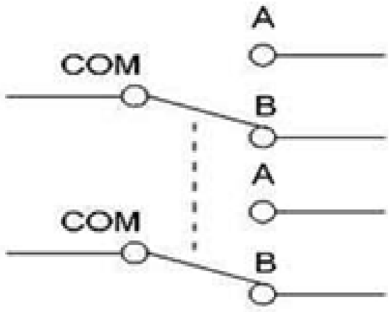


Fig. 1.2.22: DPDT switch

This can be used in an application requiring an open and closed wiring system. DPDT switch is used in railroad modeling that makes use of small scaled bridges, trains and railways as well as cars.

The closed state allows the system to be continuously in ON position while the open state allows the other piece to be activated or turned into ON position through the relay.

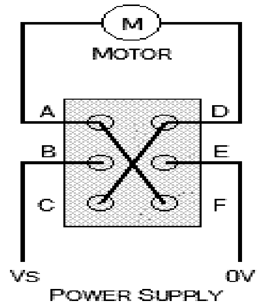


Fig. 1.2.23: Working of a DPDT circuit

In the circuit, connections A, B and C form one pole of the switch and connections D, E and F form the other pole. Connections B and E are common in each of the poles.

If the positive power supply (Vs) enters at connection B and the switch is set to the top most position, connection A becomes positive and the motor will rotate in one direction.

If the switch is set to the lower most position, the power supply is reversed, connection D becomes positive and the motor will rotate in the opposite direction.

In the centre position, the power supply is not connected to the motor and it does not rotate. These circuits are used in motor controllers.

1.2.8 Power Supply

A power supply supplies electric power to electrical loads by converting the current from a source to the correct voltage, current and frequency. There are two types of power supplies: AC and DC power supply. The following figure lists the specifications of various electrical devices to use AC power or DC power:

	OUTPUT = DC	OUTPUT = AC
INPUT = AC	• Wall wart • Bench power supplies • Battery charger	• Isolation transformer • Variable AC supply • Frequency changer
INPUT = DC	• DC-DC converter	• Inverter • Generator • UPS

Fig. 1.2.24: Specifications of various electrical devices to use AC power or DC power

Variable AC Power Supply

The following figure lists the characteristics of variable AC power supply:

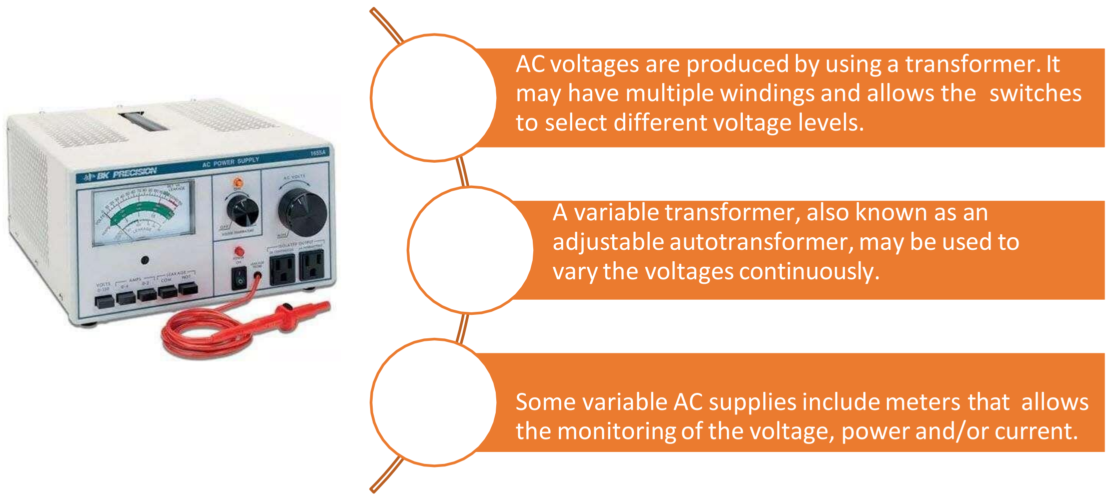


Fig. 1.2.25: Characteristics of variable AC power supply

Unregulated Linear Power Supply

The following figure lists the components of unregulated linear power supply:

Input Transformer

- The input transformer is used to convert the incoming line voltage down to the required level of the power supply.
- It also isolates the output circuit from the line supply. Here a step-down transformer is being used.

Rectifier

- The rectifier used to convert the incoming signal from an AC format into raw DC.

Filter Capacitor

- The pulsated DC from the rectifier is fed to the smoothing capacitor.
- It will remove the unwanted ripples in the pulsated DC.

Bleeder Resistor

- Bleeder Resistor is also known as a power supply drain resistor.
- It is connected across the filter capacitors to drain their stored charge so that the power system supply does not become dangerous.

Fig. 1.2.26: Components of unregulated linear power supply

This type is very simple and the most reliable source when low power is required. It is the least costly power source but the output voltage varies with the load current and input voltage. It is one of its disadvantages. Also, the ripple is not acceptable for electronic applications. If the filter capacitor is replaced by a resonance circuit (inductor-capacitor or LC), the ripple can be reduced, but it increases the cost. The following figure shows the circuit diagram of an unregulated linear power supply:

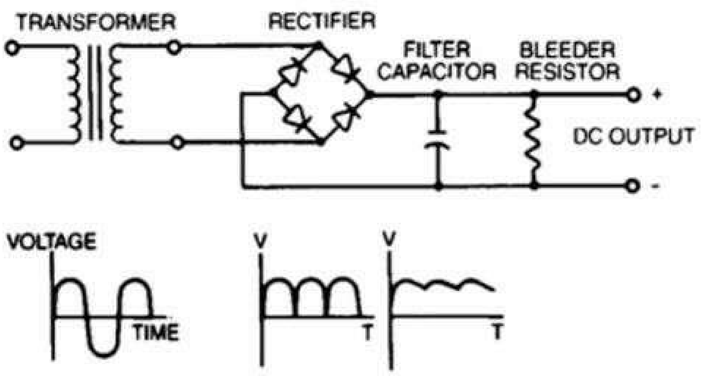


Fig. 1.2.27: A circuit diagram of an unregulated linear power supply

Regulated Linear Power Supply

Regulated linear power supplies include three-terminal regulators in place of the bleeder resistor which is used in unregulated supplies. This supply is intended to provide the required level of DC power to the load. Varying DC voltage when converted to a constant low DC voltage by a linear voltage regulator, the function of the regulator prevents the power supply or an overcurrent load from limiting the current.

Most power supply applications require an output voltage that is constant but the energy sources providing voltage vary with load impedance changes. Moreover, the output voltage varies with changes in input voltage when a DC power supply as the source of energy is unregulated. To avoid this, linear voltage regulator is used by a few power supplies for maintenance of output value at a constant value, free from any variance in load impedance and input voltage. Ripple's magnitude and output voltage noise can be reduced by linear regulators. The following figure shows the block diagram of regulated linear power supplies:

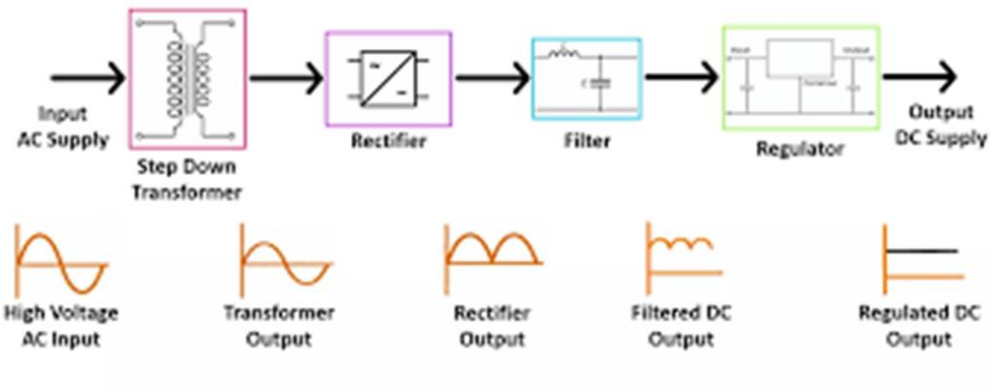


Fig. 1.2.28: A block diagram of regulated linear power supplies

The following figure shows the basic circuit diagram of regulated linear power supply:

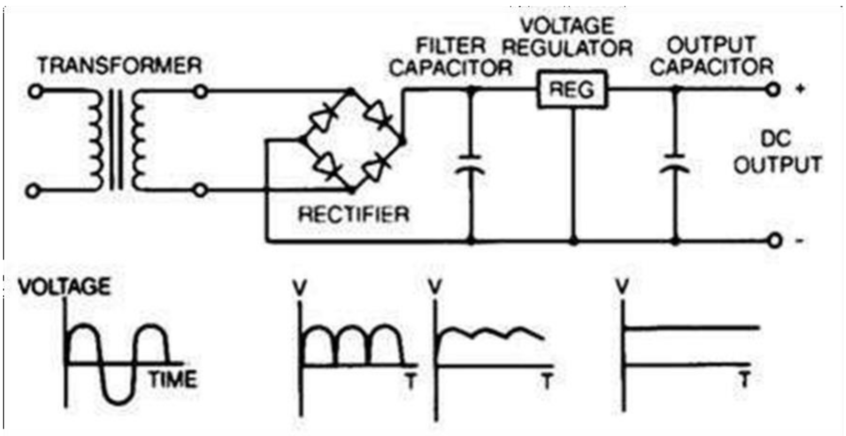


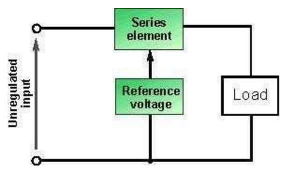
Fig. 1.2.29: A circuit diagram of regulated linear power supply

The following figure explains the working of smoothing filter and voltage regulator:

Smoothing	Voltage Regulator
Once rectified from an AC signal, the DC needs to be smoothed to remove the varying voltage level.	A linear regulator has an active (BJT or MOSFET) pass device (series or shunt) controlled by a high gain differential amplifier.
Large value capacitors are generally used for this purpose.	It compares the output voltage with a precise reference voltage and adjusts the pass device to maintain a constant level output voltage.

Table 1.2.1 Working of smoothing filter and voltage regulator

The following figure lists two types of voltage regulators:



Series regulator

- This is the most widely used regulators for linear power supplies.
- A series element is placed in the circuit and its resistance is varied via the control electronics to ensure that the correct output voltage is generated for the current taken.

Shunt regulator

- The shunt regulator is less widely used as the main element within a voltage regulator.
- In this, a variable element is placed across the load as shown in the figure.
- A source resistor is placed in series with the input and the shunt regulator is varied to make sure that the voltage across the load remains constant.

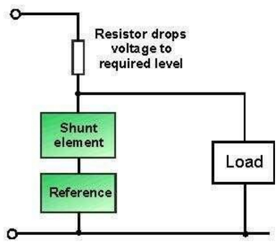


Fig. 1.2.30: Two types of voltage regulators

Switch Mode Power Supply (SMPS)

An SMPS includes filter capacitor, regulator, rectifier, series transistor and transformer. The following figure shows the block diagram of an SMPS and its working:

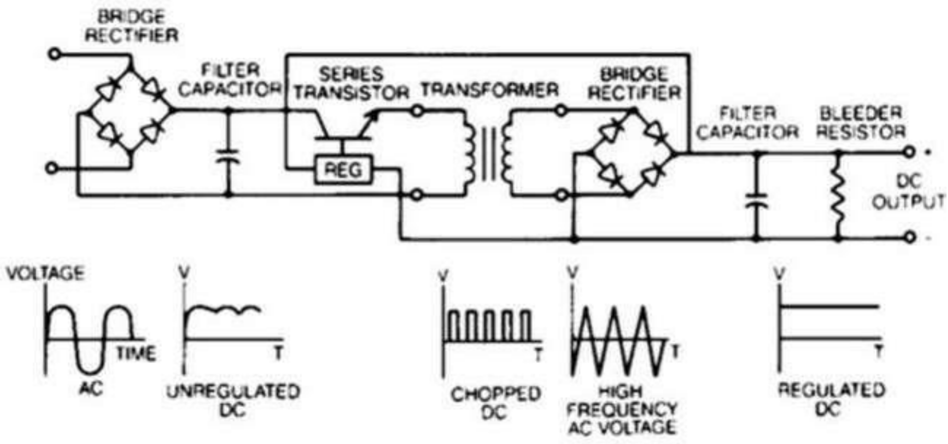


Fig. 1.2.31 (a): Block diagram of SMPS

The AC voltage is converted to an unregulated DC voltage by passing it through a rectifier, a series transistor and a regulator.

This DC is then chopped to a constant voltage of high frequency that enables the size of the transformer to be reduced dramatically, thereby allowing a smaller power supply.

Fig. 1.2.32 (b): Working of an SMPS

The disadvantages of SMPS are listed in the following figure:

It requires all the transformers to be custom-made.

The complexity of the power supply could lead to low production.

Fig. 1.2.42: Disadvantages of SMPS

The following image shows an SMPS:

Fig. 1.2.33: SMPS

Uninterruptible Power Supply (UPS)

UPS is known as a backup power source. It allows sufficient time for a standby generator or any system to shut down in an orderly manner in the case of a power fluctuation or a failure. A UPS comprises of power sensing and conditioning circuitry and a set of rechargeable batteries. The following image shows a UPS:

Fig. 1.2.34: UPS

1.2.9 Amplifier

An amplifier is used for increasing the magnitude of the signal that is applied to its input. The following figure shows the concept of an amplifier circuit:

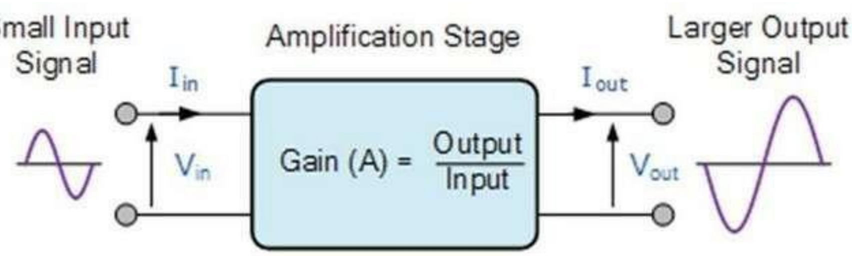


Fig. 1.2.35: Concept of an amplifier circuit

The following figure shows the concept amplifier gain:

The difference between the magnitude of the input and output signal is referred to as the amplifier's gain.

The gain measures the extent to which the input signal is amplified.

For example, an amplifier is provided input signal of 1 volt and it generates an output of 50 volts. The gain of the amplifier is 50. It means, the input signal has been amplified by a factor of 50.

Fig. 1.2.36: Concept amplifier gain

Amplifier gain is basically the ratio of output to input. The following figure lists three types of amplifier gains:

Voltage Amplifier Gain

$$Voltage\,Gain\,(A_V) = \frac{Output\ Voltage}{Input\ Voltage} = \frac{Vout}{Vin}$$

Current Amplifier Gain

$$Current\,Gain\,(A_i) = \frac{Output\ Current}{Input\ Current} = \frac{Iout}{Iin}$$

Power Amplifier Gain

$$Power\,Gain\,(A_p) = A_V \ x \ A_i$$

Fig. 1.2.37: Types of amplifier gain

The working of Class A amplifier:

- Class A amplifiers use the same transistor for each half of the output waveform.
- The output transistor has current flowing through it constantly even when there is no input signal.
- It is because of its biasing arrangement.
- The nature of the output transistor of always being in ON position makes the Class A operation very inefficient.
- This is because the conversion of DC power to AC power delivered to the load is very low.
- The output transistor gets very hot even if input signal is present.
- Hence, heat sinking is required. The DC flowing through the output transistor in absence of any output signal is equal to the current passing through the load.
- Thus, Class A amplifier is inefficient because most of the DC power gets converted to heat.

Class B Amplifier Operation: The following figure lists the characteristics of Class B amplifier:

- The Class B Amplifier uses two different set of transistors, either an NMOS and a PMOS or an NPN and a PNP for each half of the output waveform.
- One of the two transistors conducts for one-half of the signal waveform while the other one conducts for its opposite half.
- Each transistor amplifies only 50% of the input signal as it remains half of its time in active region and the other half in the cut-off region.
- The transistor conducts only when the input signal voltage is greater than the base-emitter voltage (VBE).
- For the silicon devices the VBE is about 0.7 V. Hence, only half of the input signal gets amplified, thereby giving a greater amount of amplifier efficiency.

The following figure shows output waveform of Class B amplifier:

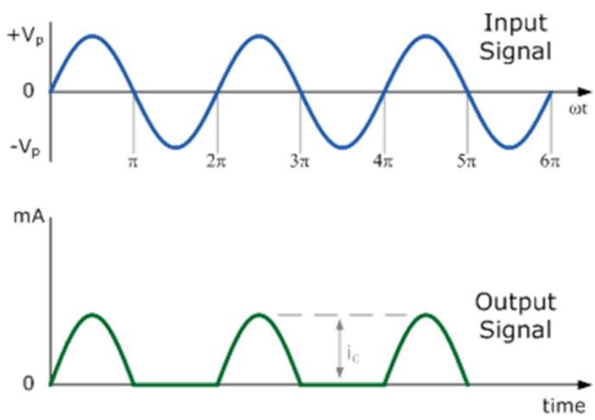


Fig. 1.2.38: Output waveform of Class B amplifier

In a Class B amplifier, to bias the transistors, no DC voltage is used. For the output transistors, to start conducting in both the halves (positive and negative) of the waveform, the VBE needs to be more than 0.7 V. This is a must for the bipolar transistor for conducting.

The lower part of the output waveform below 0.7 V is not accurately reproduced as one transistor is in OFF position and is waiting for the other one to come back to ON state. Hence, it results in a deformed area of the output waveform with a small part of it distorted at the zero-voltage cross over point. This is known as Crossover Distortion.

- Class AB Amplifier Operation: The following figure lists the working of class AB amplifier:
- Two complementary transistors are used in their output stage and a small biasing voltage is applied to bias the transistor at its base.
- This is done so that the cut-off region is reached even in the absence of any input signal.
- Any crossover distortion is eliminated because an input signal causes the transistor to operate normally as it operates in its active region.
- In absence of any input signal, a small Collector current flows.
- The transistor remains in 'ON' position for more than half a cycle of the waveform.
- This configuration improves the efficiency and linearity of the amplifier circuit as compared to the configuration of Class A.

The following figure shows the output waveform of Class AB amplifier:

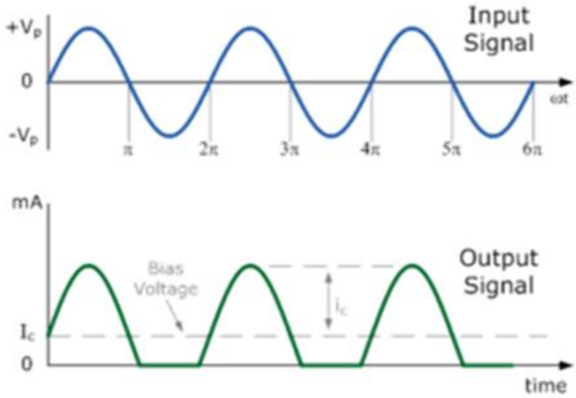


Fig. 1.2.39: Output waveform of Class AB amplifier

The class of operation depends on the amplitude required for input signal and the amount of transistor bias needed for the operation. The classification of the amplifier considers the following points:

- Segment of the input signal in which the transistor conducts
- Efficiency
- Amount of power that is consumed and dissipated in terms of heat by the switching transistor

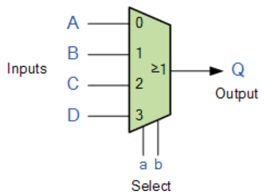
See below a table comparing the common classes of a power amplifier:

Class	A	B	C	AB
Conduction Angle	360°	180°	Less than 90°	180 to 360°
Position of the Q-point	Centre Point of the Load Line	Exactly on the X-axis	Below the X-axis	In between the X-axis and the Centre Load Line
Overall Efficiency	Poor 25 to 30%	Better 70 to 80%	Higher than 80%	Better than A but less than B 50 to 70%
Signal Distortion	None if Correctly Biased	At the X-axis Crossover Point	Large Amounts	Small Amounts

Table 1.2.2: Comparison between the common classes of power amplifier

1.2.10 Multiplexer (MUX) and Demultiplexer (Demux)

The following figure explains the multiplexer and demultiplexer:



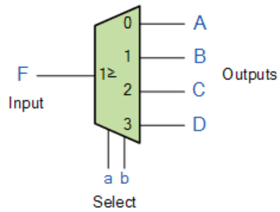
The **multiplexer** is a combinational logic circuit designed to switch one of several input lines to a single common output line by the application of a control logic.

Multiplexers can be either digital circuits made from high speed logic gates used to switch digital or binary data.

They can be analogue types using transistors, MOSFET's or relays to switch one of the voltage or current inputs to a single output.

The **demultiplexer** is a combinational logic circuit designed to switch one common input line to one of several output lines by the application of a control signal.

The data distributor, known more commonly as a Demultiplexer, is the exact opposite of the Multiplexer.



1.2.11 Coupler

Coupler is used to transfer electrical energy from a part of a circuit to another part. For example, conductive coupling transfers the energy from a power source to a load. The following figure lists the types of coupling:

Direct Coupling

- Direct coupling is essential for very low frequency applications Such as photoelectric current.
- It has got advantages of having simple and cheap circuit arrangement and outstanding ability to amplify low frequency signals.
- The drawbacks include poor temperature stability and unsuitability for amplification of high frequency signals. Direct coupled amplifiers are used when the load is directly in series with the output terminal of the active circuit element.

Impedance Coupling or Transformer Coupling

- Impedance coupling results in more efficient amplification because no signal power is wasted in Inductor L.
- Such coupling has the drawback of being larger, heavier and costlier than the RC coupling. Impedance coupling is rarely used beyond audio range.

RC Coupling

- RC coupling is the most commonly used coupling between the two stages of a cascaded or multistage amplifier because it is cheaper in cost, has very compact circuit and provides excellent frequency response.

1.2.12 Registers and Memories

The following figure shows a 4-bit register and its characteristics:

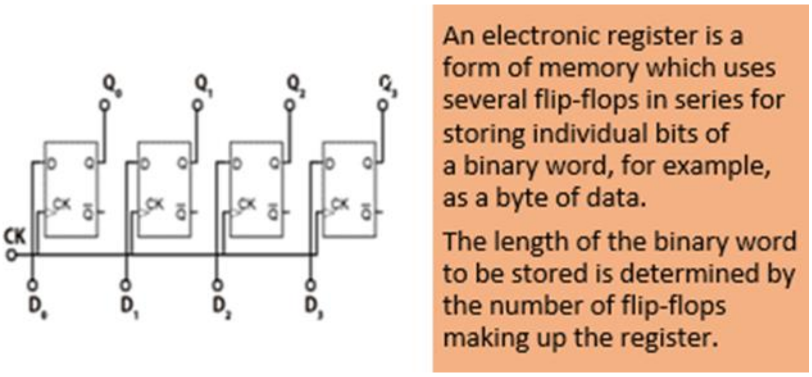


Fig. 1.2.40: A 4-bit register and its characteristics

The working of a 4-bit register shown in the above figure:

- The above 4-bit register is comprised of four D-type flip-flops sharing a common clock input and providing synchronous operation to ensure that all the bits are stored promptly.
- The binary word is applied to the D inputs. The word is remembered by the flip-flops at the rising edge of the next clock (CK) pulse.
- At any time, the stored data can be read from Q outputs as long as the power is maintained, or until there is a change of data on the inputs stored by a further clock pulse, overwriting the previous data.

The registers are typically classified according to the storage method and the readout used. The basic form of register is Parallel In/Parallel Out (PIPO) register.

Types of Registers

- Serial In Serial Out (SISO): Data bits are entered serially and output is also taken serially.
- Serial In Parallel Out (SIPO): Data bits are loaded serially but output is taken parallelly.
- Parallel In Parallel Out (PIPO): Data bits are loaded parallelly and output is also taken parallelly.
- Parallel in Serial out (PISO): Data bits are loaded parallelly and the output is taken serially.

The following figure shows the configuration of SISO registers:

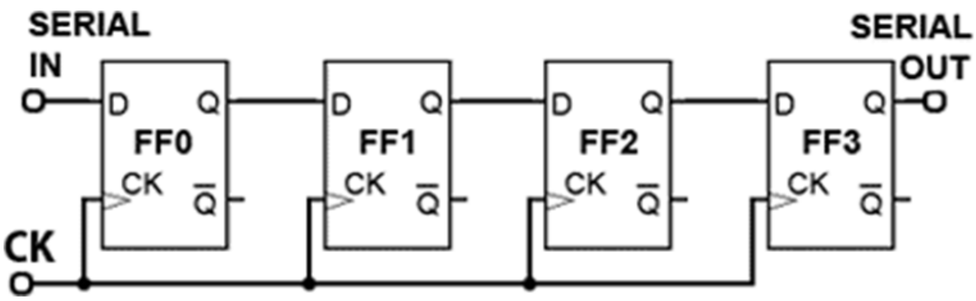


Fig. 1.2.41: Configuration of SISO registers

The following figure shows the configuration of SIPO registers:

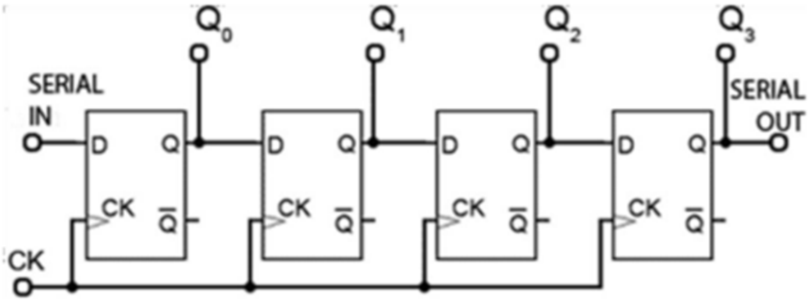


Fig. 1.2.42: Configuration of SIPO registers

Shift Registers

The structure of shift registers is similar to that of the PIPO registers but they have the ability to perform shifting of the stored word, one bit at a time, to right or left. These registers are used as a necessary component in communication systems to handle serial data. They are used to convert the data to parallel form and then to convert it back again to serial form. The following figure shows the application of shift registers:

Shift registers are also important for arithmetic circuits to shift the binary numbers to right (by dividing by two) or to left (by multiplying by two) as part of a calculation.

These can also be used for delaying the passage of data at a specific point in a circuit.

- At a time, one bit of data is shifted from the input towards the output.
- The amount of delay depends on the number of flip-flops and the frequency of the clock pulses driving the shift register.
- The action of storing several bits of data after they are inputted and then recovering them from the output at a later time is known as a digital delay line.

1.2.13 Radio Frequency (RF) Circuits in Telecom

RF range is used in radio. The frequency ranges between audio frequencies (with the upper limit of 20 kHz) and infrared frequencies (with lower limit of around 300 GHz).

The following figure shows the building blocks of a typical RF communication system:

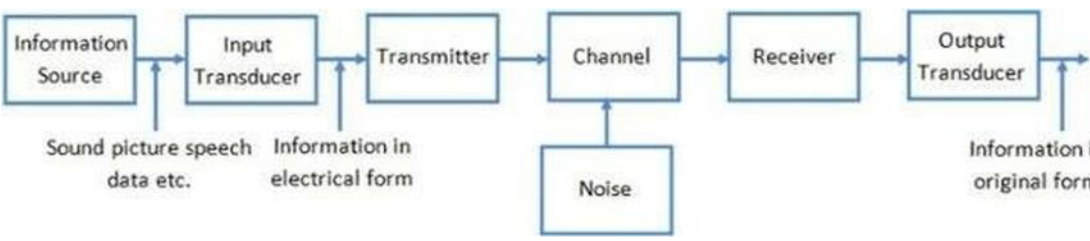


Fig. 1.2.43: Building blocks of a typical RF communication system

Notes

UNIT 1.3: Printed Circuit Board (PCB) Construction and Types

Unit Objectives

By the end of this unit, the participants will be able to:

1. Identify and describe the four main layers of a typical PCB.
2. Explain the function of each layer, including the substrate, copper, solder mask, and silkscreen.
3. Differentiate between rigid and flexible PCBs based on the substrate material.
4. Classify different types of PCBs (single-sided, double-sided, and multi-layer) based on their construction.
5. Describe the use of through-hole and surface-mount technologies for component mounting.
6. Recognize common applications for each type of PCB.

1.3.1 Composition of a PCB

A Printed Circuit Board (PCB) is a composite structure made of different layers that are bonded together using heat and adhesive to form a single, rigid or flexible object. The following sections describe the primary layers of a PCB.

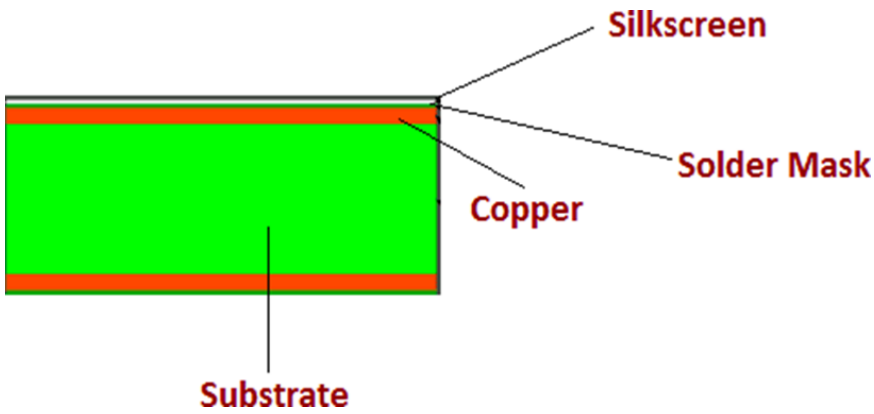


Fig. 1.3.1: Layers of a PCB

Substrate Layer

- This is the base layer, typically composed of a dielectric material (an electrical insulator). The nature of the substrate material determines whether the board will be a Rigid PCB or a Flexible PCB.
- FR-4 (Flame Retardant 4) is the most common substrate material used today. It is a woven fiberglass fabric bound with an epoxy resin, providing a solid core that gives the PCB its rigidity and thickness.
- Some less expensive PCBs use substrates made of phenolics and epoxies. These are not as durable as FR-4, have a distinct odor, and are commonly found in low-end consumer electronics.
- Phenolics have a low decomposition temperature, which can cause them to delaminate or blister if exposed to soldering heat for an extended period.

Copper Layer

- This is a thin layer of copper foil laminated onto the substrate using heat and an adhesive. This layer is etched to form the conductive tracks (traces) and pads that connect the electronic components.
- Most PCBs have a copper layer on at least one side. Double-sided PCBs have copper on both sides of the substrate.
- The thickness of the copper layer is measured in ounces per square foot (oz/ft2). One ounce of copper spread evenly over one square foot results in a thickness of approximately 35 micrometers (µm).
- A thickness of 1 oz/ft2 is standard for most PCBs. However, applications requiring higher current capacity may use thicker copper layers, such as 2 or 3 oz/ft2.

Solder Mask Layer

- This is a protective polymer layer applied over the copper traces to insulate them. Its primary function is to prevent short circuits by protecting the copper from accidental contact with other conductive materials, as well as to prevent oxidation of the copper.
- The solder mask is typically green, but it is available in various colors, including red, blue, black, and white.
- The solder mask has openings for the pads where components are to be soldered.

Silkscreen Layer

- The silkscreen is the final layer applied on top of the solder mask. It is used to print text, symbols, and outlines that provide important information for assembly and troubleshooting.
- Silkscreen labels can indicate component designators (e.g., R1, C2), polarity markings, component outlines, pin functions, and other useful information.
- The most common silkscreen color is white, but other colors like yellow, black, and red are also used.
- Using a single color for the silkscreen is standard practice to ensure clarity and readability.

1.3.2 Types of PCBs

PCBs are categorized based on the number and arrangement of their conductive layers. The following are the most common types:

1. Single-Sided PCB

- This is the simplest type of PCB, with conductive copper traces and electronic components on only one side of a single substrate layer.
- Components are inserted from the non-copper side and their pins are soldered to the copper pads on the other side.
- These are used in low-cost, low-complexity electronic devices such as calculators, coffee machines, and simple control boards.

2. Double-Sided PCB

- This type of PCB has conductive copper layers on both the top and bottom sides of the substrate.
- Holes are drilled through the board and plated with copper (vias) to electrically connect circuits on one side to the other.
- Components can be mounted using two main technologies:
- Through-Hole Technology (THT): Component leads are inserted through holes in the board and soldered to pads on the opposite side. This creates a strong mechanical connection.
- Surface-Mount Technology (SMT): Components are soldered directly to pads on the surface of the board, without using through-holes. This allows for smaller components and higher component density.
- Double-sided PCBs are widely used in applications like amplifiers, power supplies, and HVAC systems.

Multi-Layer PCB

- Multi-layer PCBs consist of three or more double-sided PCBs (or more) laminated together, separated by layers of insulating material (prepreg).
- These boards are bonded using specialized adhesives and heat, creating a complex, integrated structure.
- Vias can connect layers on the top, bottom, or any internal layers, allowing for very high-density and complex circuit designs.
- The thickest multi-layer PCB developed was reportedly 50 layers thick.
- These boards are essential for high-performance and complex applications such as satellite systems, data storage devices, GPS technology, and high-speed computing. Their multi-layered structure helps in reducing electromagnetic interference and allows for shorter signal paths, enabling faster operation.

1.3.3 Copper Clad Laminates (CCL)

A Copper Clad Laminate (CCL) is the fundamental base material used to manufacture PCBs. It consists of a reinforcing material (such as fiberglass cloth or wood pulp paper) that is impregnated with a resin and then laminated with a copper foil on one or both sides. This raw material is then processed to create the final PCB.

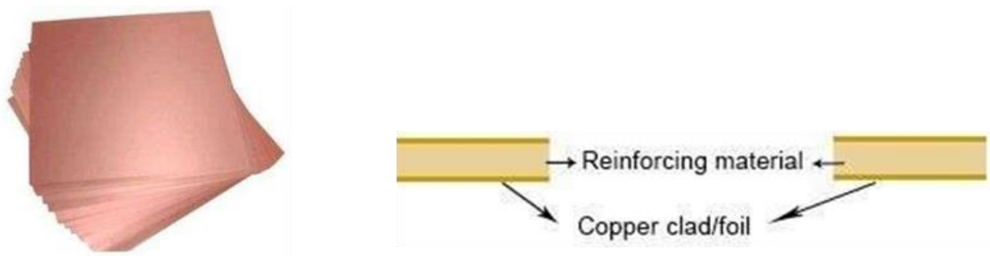


Fig. 1.3.2: A set of CCL boards and the construction of a CCL board

The following figure lists the performance requirements for CCLs:

Appearance

- Issues may be caused on copper foil due to unexpected elements in manufacturing process such as dent, scratch, resin point, wrinkle, pinhole, bubble and so on.
- All these problems will lead to low performance of the CCL and consequently, the PCB. Therefore, an excellent CCL should be flat and smooth in appearance.

Size

- Since CCLs are base material of PCBs, they have to conform to size requirements corresponding to the PCBs.
- Parameters concerning the size of CCLs include length, width, diagonal deviation and warpage, each of which has to meet specific requirement.

Electric Performance

- The factors affecting electric performance of PCBs which are to be carefully designed include dielectric constant (Dk), dielectric breakdown voltage, dielectric loss tangent (Df), surface resistance, arc resistance, volume resistance, insulation resistance, electric strength, Comparative Tracking Index (CTI) and so on.

Physical Performance

- Parameters concerning physical performance of a CCL include dimensional stability, peel strength (PS), bending strength, heat resistance (including thermal stress, Td, T260, T288, T300), punching quality and so on.

Chemical Performance

- Chemical performance of a CCL has to meet the requirements of flammability, chemical reagents resistance, Tg, Z-axis coefficient of thermal expansion (Z-CTE), dimensional stability and so on.

Environmental Performance

- It has to cater to the requirements in terms of water absorption amongst others.

Fig. 1.3.3: Performance requirements for CCLs

Notes

UNIT 1.4: PCB Designing Concepts

Unit Objectives

By the end of this unit, the participants will be able to:

1. List the steps involved in PCB designing process.
2. Identify various inspection checks to be performed during PCB design and fabrication.
3. Perform cleaning of printed circuit boards before the pattern transfer process.
4. List key IPC standards relevant to PCB designing and manufacturing.

1.4.1 Steps of PCB Designing

The following figure lists the steps of designing a PCB:

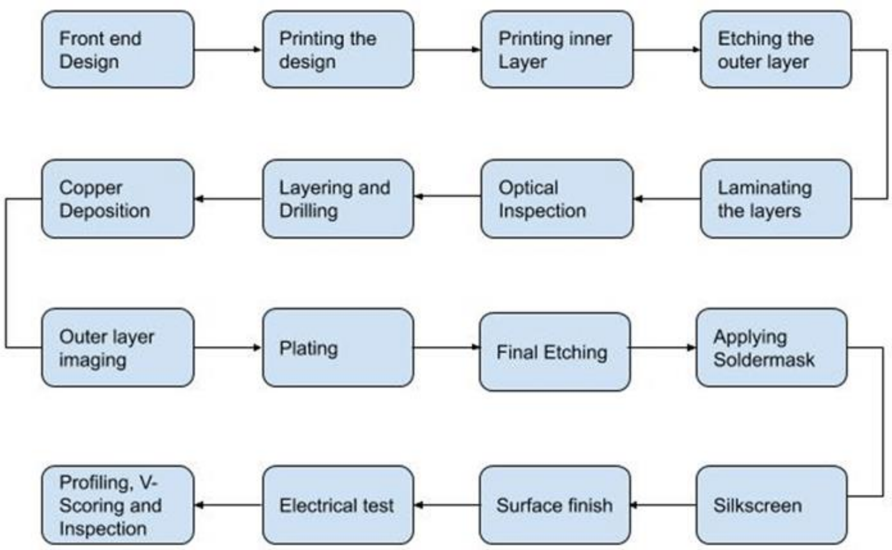


Fig. 1.4.1: Steps of designing a PCB

Designing the PCB

A blueprint is prepared by the designer for the PCB, considering all the specifications.

The following are the steps of designing the PCB using software:

- The software encodes all information required by the designer, such as the amount of solder masks required, the number of copper layers needed and the pieces of component notation required. The design software most widely used by the PCB designers is Extended Gerber, also known as IX274X.
- Once the design blueprint gets encoded by the software, the design is checked, considering all aspects, to ensure that there are no errors.
- When the checking is done, the design is sent out to a PCB fabrication house to be built.
- The fabricator checks the PCB design plan, known as a Design for Manufacture (DFM) check, to verify that the design meets the tolerances required for the manufacturing process.

Printing the Design on PCB Layers

When all required checks are complete, a plotter printer prints the design on the PCB layers by making a “film” of the PCB which looks similar to a photo negative of the board itself. The layers are represented in two different ink colours. The following figure lists the colours to represent the layers of the PCB:

Inner Layers

- Black ink denotes the copper traces and circuits of the PCB.
- Clear Ink denotes the non-conductive areas of the PCB such as the fibreglass base.

Outer Layers

- Clear ink denotes the line of copper pathways.
- Black ink denotes the areas from where copper would be removed.

There are separate films for each PCB layer and its solder mask. For a two-layer PCB, there will be four sheets:

- One for each layer
- One for the solder mask of each layer

The films are lined up after being printed. A hole is punched through the films. It is referred as registration hole and it helps in aligning the films in later process.

Printing Copper for the Inner Layers

Once the design gets printed on the laminate, copper is pre-bonded to it which acts as the structure for the PCB. The following figure lists the steps of the printing process:

- The copper is etched away so that the earlier blueprint is revealed.
- The laminate panel is then covered with a photo-sensitive film called resist, which consists of a layer of photo-reactive chemicals.
- These chemicals harden when they are exposed to ultraviolet light.
- Resist helps to obtain a perfect match between the photos of the blueprint and what gets printed to the photo resist.
- Once the resist, along with the laminate, is lined up, ultraviolet light is passed through it.
- The photo resist gets hardened when the ultraviolet light passes through the translucent parts of the film, thereby indicating the areas of copper meant for the pathways.
- The black ink prevents the light from reaching the areas which are not meant to harden and they are removed later.
- The board is then washed with an alkaline solution so that any leftover from the photo resist can be removed.
- After that, the board is pressure-washed for removing any leftover from the surface and is left to dry.
- When the PCB is dried, the resist should only be left on top of the copper, which remains as part of the PCB.

A technician needs to check the PCB to ensure that there are no errors.

Getting Rid of the Unwanted Copper

The unwanted copper (not protected by the photo resist) is removed using another powerful chemical like an alkaline solution. After this the hardened photo resist should be removed using another solvent, leaving the copper necessary for the PCB.

Inspection and Alignment of the Layers

After cleaning the layers of the PCB, they are aligned using the holes. The layers are placed on an optical punch. The punch aligns the holes by driving a pin down through them.

The layers are then passed through an optical inspection to ensure that they are free of defects. The PCB is inspected by automated optical inspection (AOI) machine and the design on it is compared with the specified design received from Extended Gerber software. After the PCB passes the inspection, it is moved on to the lamination process.

Laminating the layers of the PCB

Once the layers pass the AOI inspection, they are ready to be joined together. The following figure explains the two-step process of laminating:

Step 1: Lay-up

- The outside of the PCB is constructed of fibreglass and pre-coated with an epoxy resin.
- The substrate is covered with a thin copper foil containing etchings for copper traces.
- The sandwiching of the layers and the substrate is done using metal clamps on a special press table where each layer is fit using a specialised pin.
- The layer of pre-coated epoxy resin is known as pre-impregnated or prepreg. It is placed on the alignment basin of the table.
- The substrate is then placed over it, followed by a copper foil layer, which is then followed by more sheets of pre-impregnated resin.
- Then a last piece of copper known as a press plate is placed.
- The stack of layers is pressed together using a mechanical press. Pins are punched down through the layers ensuring that they are fixed properly.

Step 2: Laminate

- The stack with fixed layers is then taken to the laminating press that uses a pair of heated plates for applying heat and pressure to the layers. The epoxy melts due to the heat and the pressure and fuses the layers of PCB together.
- It is required to remove the top press plate and the pins and then pull the actual PCB free.

Drilling

Following are the steps for drilling holes on the PCB:

- Before drilling, the drill spots are located using an X-ray machine.
- Then, the registration holes are drilled to secure the PCB stack before drilling the more specific holes.
- The specific holes are drilled automatically by using a computer- guided drill, considering the Extended Gerber design file as a guide.

The additional copper, left at the edges, needs to be scraped off after drilling is completed.

PCB Plating

The plating process fuses the layers of the PCB together through bathing process in which it is passed through various chemicals. A micron-thick copper layer is used to coat the panel. The copper layer is deposited into the just-drilled holes and over the top most layer. The fibre glass substrates inside the panel are exposed through the holes before they are filled with copper. The walls of the holes get covered after being bathed in copper.

Imaging and Plating Outer Layer

The outside layers then need to be imaged.

The process of plating the outer layers:

- They are coated in photo resist and imaged.
- Then they are plated in the same way as the interior layers.
- The outer layers are plated with tin to protect the copper of the layers during the etching process.

The Last Etching

In this process, the unwanted copper is removed using copper solvent. The tin plating protects the required copper of the etching area.

The PCB panels are sent for solder masking when the removal of unwanted copper and the establishment of connections are done properly.

Application of Solder Mask

Before the panels are masked, they are cleaned. The following figure shows the steps of applying solder mask:

- An ink epoxy in combination with solder mask film is then applied to the cleaned PCB panels.
- The boards are passed through ultraviolet light and certain portions of the solder mask are marked for removal.
- After the unwanted solder mask is removed, the PCB is kept inside an oven and baked to allow the solder mask to be cured.

Finishing the PCB and Silk-screening

In the finishing process, in order to protect the copper, the board is plated with hot air solder levelling (HASL), gold or silver. The plating is also done to enable the soldering of the components to pads.

Once the plating is done, all the important information, for example, company ID numbers manufacturer marks and warning labels, are printed on the boards using the silk-screening process. After silk-screening the PCB appropriately, it is passed through final curing stage.

Electrical Reliability Testing

Once the board has been cured, the technician needs to ensure the functionality of the PCB by conducting electrical tests such as circuit continuity and isolation tests, on different areas of the PCB. See below the purpose of the electrical tests:

- Circuit continuity test: It checks for any disconnections in the PCB, known as “opens.”
- Circuit isolation test: It checks the isolation values of various parts of the PCB to check if there are any shorts.

The electrical tests also check how well the initial PCB design will perform and react to the manufacturing process.

Besides the basic electrical reliability testing, some other tests are performed to determine whether a PCB is functional. The following figure shows an example of such functionality test:

The “bed of nails” test is done to check the PCB’s performance under high-pressure contact. Several spring fixtures are attached to the test points on the board and then the test points are subjected to up to 200g of pressure.

If the PCB passes all the tests, it can be moved for cutting.

Cutting and Profiling

Cutting and scoring is the last stage that involves cutting out PCBs from the original panel. The following figure lists the two ways of cutting the PCBs from their original panels:

- Using a CNC machine or router, which cuts out small tabs around the edges of the PCB
- Using a V-groove, which cuts a diagonal channel along the sides of the board

Usually, the individual boards from the PCB panels are routed out and scored. This process enables the separation of the panels from the construction board when they are assembled. Once the boards are separated, a final inspection is performed. Following are the inspection checks to be performed:

- The boards are checked for general cleanliness to ensure that there are no sharp edges, burrs or other manufacturing hazards.
- Slots, chamfers, bevels and countersinks are added during the routing and fabrication process, as necessary.
- If any shorts are repaired, the shorted boards are re-tested using the electrical reliability tests.
- A visual inspection can be conducted, if necessary, to ensure that boards meet industry specifications and match up to the details laid out in Gerber data.
- The visual inspection can also be done to verify the hole sizes and the physical dimensions of the PCB.

1.4.2 Cleaning of Boards before Pattern Transfer

It is usually assumed that the boards are clean enough to start the printing process and often paper rolls are used to clean the boards. However, this should not be done as the boards get microfibres or lint due to cleaning with cheap wipes. These microfibers cause various defects in the boards, thus increasing the cost of production. Hence, it is highly recommended that the boards are properly cleaned before the printing process.

The boards can be cleaned in a dip-tank cleaner that has a special ultrasonic cleaning capability. If the volumes of the boards are high, then it is recommended to use the vapour degreaser as it has a high throughput and uses less electricity compared to the aqueous system.

1.4.3 IPC Standards

Following are the standards published by IPC:

General documents

- IPC-T-50 Terms and Definitions
- IPC-2615 Printed Board Dimensions and Tolerances
- IPC-D-325 Documentation Requirements for Printed Boards
- IPC-A-31 Flexible Raw Material Test Pattern
- IPC-ET-652 Guidelines and Requirements for Electrical Testing of Unpopulated Printed Boards

Design specifications

- IPC-2612 Sectional Requirements for Electronic Diagramming Documentation (Schematic and Logic Descriptions)
- IPC-2221 Generic Standard on Printed Board Design
- IPC-2223 Sectional Design Standard for Flexible Printed Boards
- IPC-7351B Generic Requirements for Surface Mount Design and Land Pattern Standards

Material specifications

- IPC-FC-234 Pressure Sensitive Adhesives Assembly Guidelines for Single-Sided and Double-Sided Flexible Printed Circuits
- IPC-4562 Metal Foil for Printed Wiring Applications
- IPC-4101 Laminate Prepreg Materials Standard for Printed Boards
- IPC-4202 Flexible Base Dielectrics for Use in Flexible Printed Circuitry
- IPC-4203 Adhesive Coated Dielectric Films for Use as Cover Sheets for Flexible Printed Circuitry and Flexible Adhesive Bonding Films
- IPC-4204 Flexible Metal-Clad Dielectrics for Use in Fabrication of Flexible Printed Circuitry

Performance and inspection documents

- IPC-A-600 Acceptability of Printed Boards
- IPC-A-610 Acceptability of Electronic Assemblies
- IPC-6011 Generic Performance Specification for Printed Boards
- IPC-6012 Qualification and Performance Specification for Rigid Printed Boards
- IPC-6013 Specification for Printed Wiring, Flexible and Rigid-Flex
- IPC-6018 Qualification and Performance Specification for High Frequency (Microwave) Printed Boards
- IPC- 6202 IPC/JPCA Performance Guide Manual for Single and Double Sided Flexible Printed Wiring Boards
- PAS-62123 Performance Guide Manual for Single & Double Sided Flexible Printed Wiring Boards
- IPC-TF-870 Qualification and Performance of Polymer Thick Film Printed Boards

IPC, a standard developing organisation by the American National Standards Institute (ANSI), is globally recognised for its standards. The acceptability standards, widely used in the electronics manufacturing industry are published by IPC.

IPC-A-610, Acceptability of Electronic Assemblies, is used by EMS companies and the original equipment manufacturers across the world.

Flex assembly and materials standards

- IPC-FA-251 Assembly Guidelines for Single and Double Sided Flexible Printed Circuits
- IPC-3406 Guidelines for Electrically Conductive Surface Mount Adhesives
- IPC-3408 General Requirements for Anisotropically Conductive Adhesives Films

IPC standards related to PCB design and manufacturing flow:

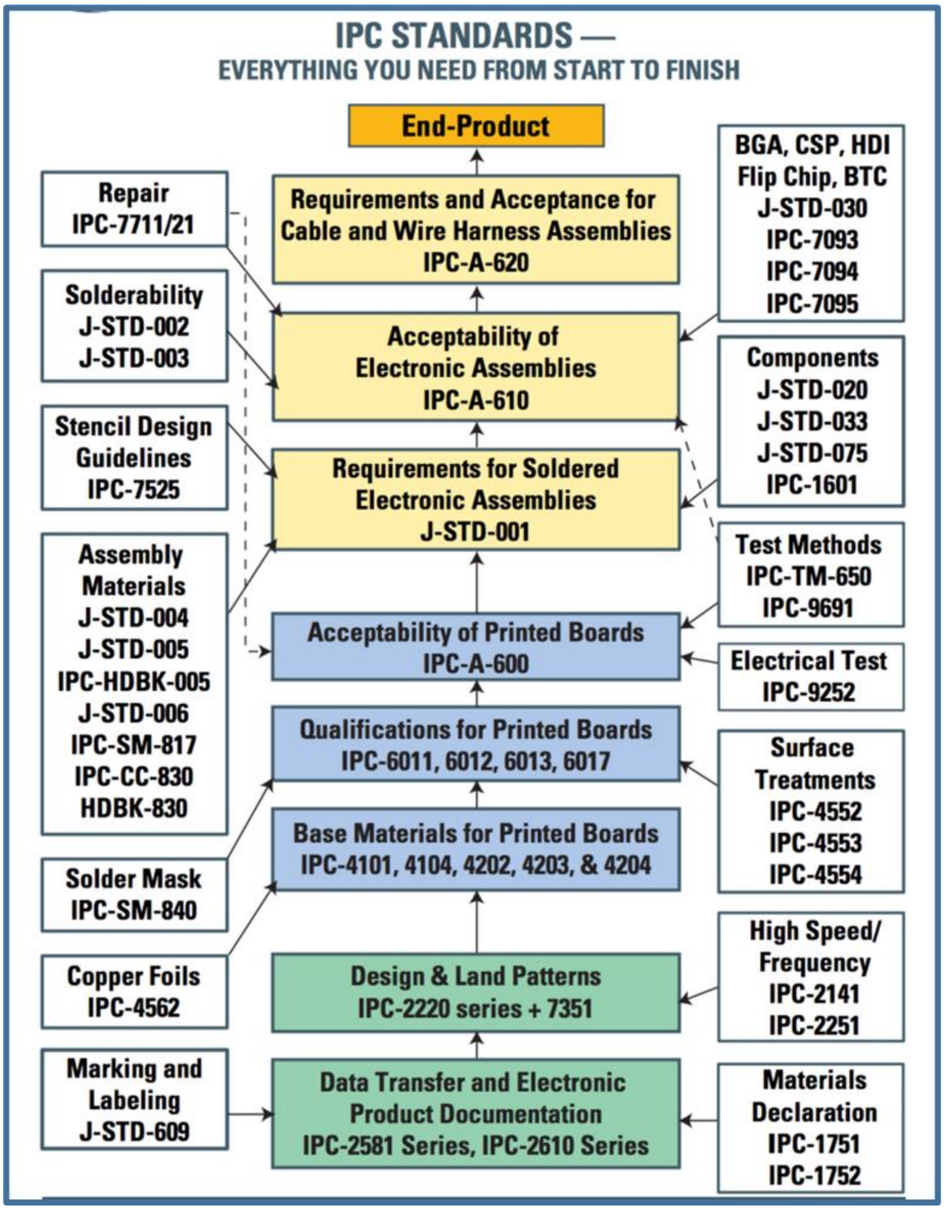


Fig. 1.4.2: IPC standards related to PCB design and manufacturing flow

Exercise

Short Questions:

1. Define the four basic electrical quantities and their units.
2. Explain two factors affecting the resistance of a conductor.
3. Differentiate between series and parallel circuits in terms of current and voltage distribution.
4. State two applications of transistors in electronics.
5. List any three types of power supplies and briefly describe their use.

Fill in the Blanks:

1. The unit of electric current is ___________.
2. The resistance of a conductor is directly proportional to its ___________ and inversely proportional to its ___________.
3. Ohm's Law is mathematically expressed as ___________.
4. The truth table is used to represent the function of ___________.
5. A toggle, push-button, and DIP are types of ___________.

Multiple Choice Questions (MCQs):

1. Which of the following is not a basic electrical quantity?
   a) Voltage
   b) Current
   c) Resistance
   d) Frequency

2. The resistance of a conductor increases with:
   a) Increase in length
   b) Decrease in temperature
   c) Increase in cross-sectional area
   d) Use of high-conductivity material

3. Which law defines the relationship between voltage, current, and resistance?
   a) Faraday's Law
   b) Ohm's Law
   c) Ampere's Law
   d) Coulomb's Law

4. A diode is commonly used for:
   a) Switching only
   b) Rectification
   c) Amplification
   d) Signal Transmission

5. Which type of power supply is commonly used for energy efficiency in modern electronics?
   a) Linear Power Supply
   b) Switching Power Supply
   c) UPS
   d) Battery

Notes

2. Screen Printing of Telecom Boards

TEL/N2503

Key Learning Outcomes

By the end of this module, the participants will be able to:

1. Demonstrate the correct procedure for applying solder paste on a PCB.
2. Identify and list the tools and accessories required for screen printing.
3. Explain the key parameters influencing the screen-printing process.
4. Perform the stencil cleaning process effectively.
5. Analyze the importance of the baking process in PCB assembly.
6. Identify oven specifications and requirements for performing baking operations.
7. List and explain the factors that influence baking time and temperature for effective results.
8. Differentiate between various types of screen-printing machines based on functionality and application.
9. List the key features of equipment used for detecting solder paste printing defects.
10. Differentiate between different inspection methods used in solder paste printing.
11. Analyze the causes of common solder paste print inspection outcomes.
12. List solder paste printing defects along with their acceptance and rejection criteria.

UNIT 2.1: Screen Printing Process

Unit Objectives

By the end of this unit, the participants will be able to:

1. Demonstrate the step-by-step process of applying solder paste accurately on a PCB.
2. Identify and list the essential tools and accessories used in the screen-printing process.
3. Explain the critical parameters that influence the quality and effectiveness of screen printing.
4. Perform the stencil cleaning process using standard procedures to ensure defect-free printing.

2.1.1 Applying Solder Paste

When a bare PCB enters the SMT assembly line, the first step is the application of solder paste. Screen printing is the most commonly used method for this purpose. In this process, a stencil (typically made of stainless steel) with predefined apertures is placed over the PCB. Solder paste is then deposited onto the copper pads by moving a squeegee across the stencil. The squeegee, held at an angle of approximately 45° to 60°, pushes and rolls the solder paste into the stencil apertures, ensuring precise deposition on the pads. This step is critical, as accurate paste application directly influences the quality of solder joints in the subsequent reflow process.

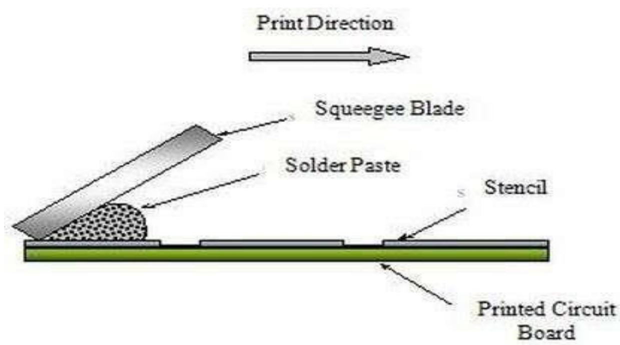


Fig. 2.1.1: The way of applying the solder paste on the board

2.1.2 Parameters of Printing Process

The following figure lists the key parameters of an effective solder paste printing:

- Squeegee speed
- Squeegee pressure
- Stencil separation speed
- Stencil cleaning
- Stencil and squeegee condition
- PCB support
- Print stroke
- Type, storage and handling

Fig. 2.1.7: Key parameters of an effective solder paste printing

Squeegee Speed

The available time for the solder paste to get rolled into the stencil apertures and then onto the PCB pads depends on the speed of the squeegee. Typically, squeegee speed is set at 25mm per second but this may vary on the basis of the aperture's size and the type of solder paste used.

Squeegee Pressure

Sufficient pressure is required to be applied along the length of the squeegee blade during the print cycle to achieve a clean wipe of the stencil. The following figure lists the effect of insufficient squeegee pressure on the PCB:

- Too little pressure can cause smearing of the paste on the stencil, poor deposition and incomplete transfer to the PCB.
- Too much pressure can cause scooping of the paste from larger apertures, excess wear on the stencil and squeegees and bleeding of the paste between the stencil and the PCB.

Fig. 2.1.8: Effect of insufficient squeegee pressure on the PCB

Typically, the pressure is set at 0.5Kg per 25mm of squeegee blade.

Stencil Separation Speed

It is the speed at which the PCB gets separated from the stencil after printing. The aperture size governs the speed. Typically, the speed of up to 3mm per second needs to be used. If separation is too fast, it will not allow the solder paste to be fully released from the apertures. Also, high speed will cause dog-ears, which is the formation of high edges around the deposits.

The following image shows dog-ears formation in solder paste printing:

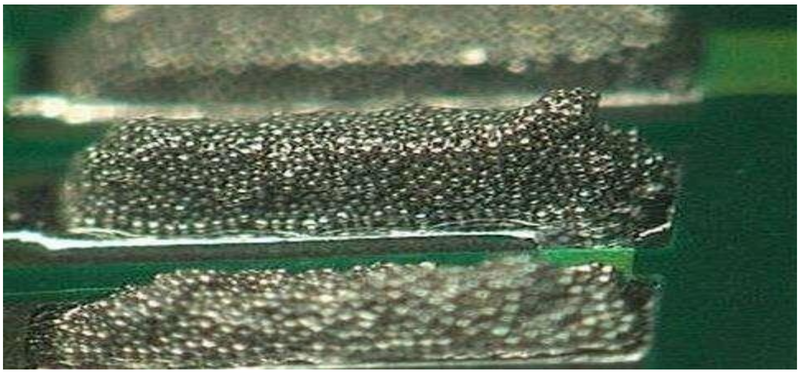


Fig. 2.1.9: Dog-ears formation in solder paste printing

Condition of Stencil and Squeegee

Squeegees and stencils need appropriate storage facilities and need to be maintained carefully. If any mechanical damage occurs to them it may cause undesired results. Before they are used, they need to be checked properly. After being used, they should be thoroughly cleaned to eliminate any solder paste residue. An automated cleaning system is ideally used to clean them. If any damage to the squeegees or stencils is detected, they should be promptly replaced. This is required to ensure that the process is reliable and repeatable.

PCB support

During the printing process, the PCB is required to be held flat against the stencil. It needs to be well supported and secured otherwise printing defects can occur such as smudging and improper paste deposit. The PCB supports that come with the printing machines generally permit a certain fixed height. These supports have the provision for programmable positions to ensure a consistent process. These adaptable PCB supports are available in different designs. These are useful for the double-sided assemblies and can mould themselves as per the PCB. The following image shows an adaptable PCB support in use:

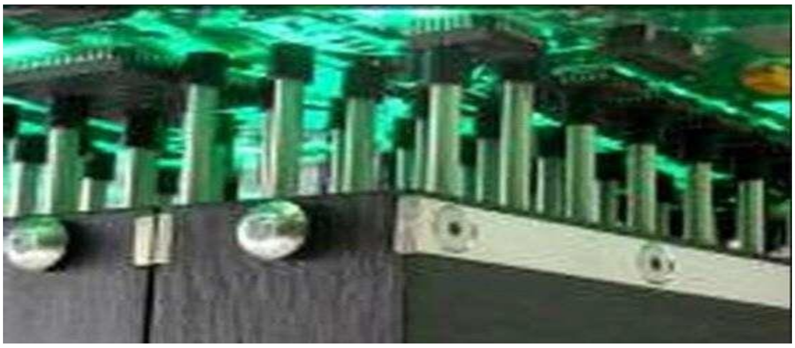


Fig. 2.1.10: An adaptable PCB support in use

Print Stroke

The distance that is travelled by a squeegee across a stencil is known as a print stroke. The distance recommended for a stroke past the furthest aperture is at least 20mm. This distance is essential as it gives the required space to the paste to roll on its return stroke. The downward force that is generated due to the rolling of the solder paste bead propels the paste inside the apertures.

Type of Solder Paste, its Storage and Handling

The solder paste of correct type should be selected depending on the size of the aperture. The particle size within the solder paste affects the release from the apertures. The following table lists the available particle sizes:

Particle size in microns	Particle type
75-45	2
45-25	3
38-20	4
25-15	5
15-5	6

Table 2.1.1: Available particle sizes

There is a '5 ball rule' stating that at least 5 solder particles should span across the width of the smallest aperture. The following figure shows the 5-solder ball rule:

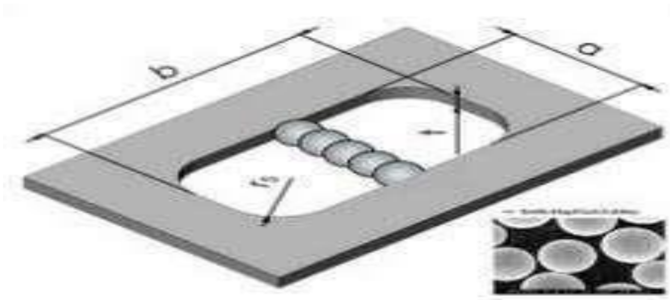


Fig. 2.1.11: 5 solder ball rule

Both types of solder paste, Tin-Lead and Lead-Free, need to be stored in a refrigerated place to maintain their shelf-life. But before using the paste, it must be brought to room temperature for at least eight hours for maintaining the quality. The paste should be mixed well manually or automatically for one to three minutes before being used. This needs to be done to achieve even distribution of the components throughout the paste. The following figure explains the expiry of solder paste:

Solder paste that has been in use for more than 8 hours should be disposed off.

Solder paste which has been in use for up to 4 hrs can be stored for up to 24 hours in a sealed container at room temperature before being re-used depending upon data sheet of solder paste used

Fig. 2.1.12: Expiry of solder paste

The working environment that includes relative humidity and ambient temperature affects the performance of the paste. A basic coalescence test may be performed to check the paste condition.

2.1.4 Stencil Cleaning Process

A regular cleaning of the stencil is a must. It can be done manually or automatically. Most of the automatic printing machines provide a system that can be set up in such a way that it cleans the stencil after a pre-defined number of prints. It does this by wiping with a lint-free material along with using a cleaning chemical such as IPA. The system carries out following two functions:

- Cleaning the underside of stencil in order to stop smudging
- Cleaning the apertures using vacuum in order to stop blockages

The following images show the defects that occur when the stencil is not cleaned:

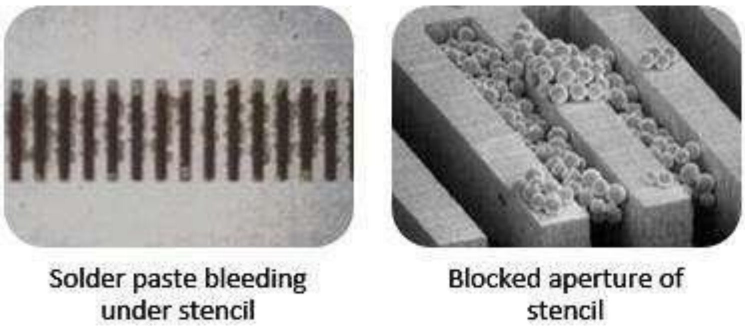


Fig. 2.1.13: Defects when the stencil is not cleaned

The following image shows the stencil cleaning rolls:

Fig. 2.1.14: Stencil cleaning rolls

Notes

UNIT 2.2: Pre-baking of Board

Unit Objectives

By the end of this unit, the participants will be able to:

1. Analyze the purpose and importance of the baking process in PCB assembly.
2. Identify the oven specifications and requirements necessary to perform effective baking.
3. List and explain the factors that influence baking time and temperature for achieving reliable results.

2.2.1 Need of Baking Process

Baking process may be essential for the removal of any residual moisture which may get absorbed into the PCB during the period between its fabrication completion and exposure to assembly soldering. Ideally, the PCBs should be dry packed by the manufacturers adhering to the printing board handling and storage guidelines (IPC-1601), where the moisture content is controlled as per the specifications. However, there may be chances that lead to increased moisture contents in the PCB once it is opened from the packaging for the screen-printing process. The increased moisture content in the PCB may lead to delamination of the PCB once it is put into the soldering process. The following image shows delamination and blistering defects:

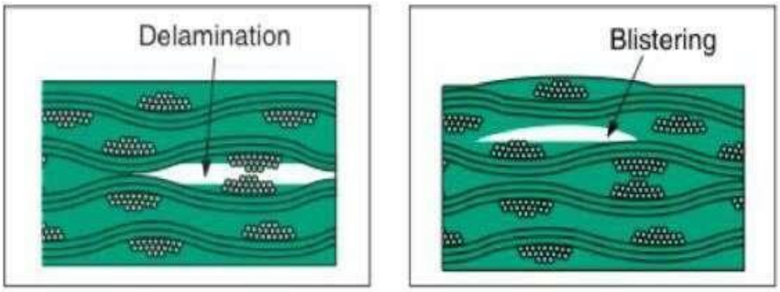


Fig. 2.2.1: Delamination and blistering defects

The moisture content can be checked with a humidity indicator card. The following image shows a humidity indicator card:

Fig. 2.2.2: Humidity indicator card

It is highly recommended that a printed board fabricator should control the moisture content of the PCB during the fabrication process before placing it in protective packaging. This is needed to avoid the baking process. The baking of board should be done only when it is required as excess baking reduces solderability. It also increases the cost as well as the cycle time.

Baking should be performed in a forced air recirculating oven, though effectiveness may be improved by reducing the relative humidity or vapour pressure in the oven, such as through baking in vacuum or nitrogen atmosphere. The oven that is used for baking, should not only be vented but also be able to maintain the required temperature along with the relative humidity (RH) as less than 5%. The oven should be free of contamination. Contaminants such as silicones can deposit on the surface of the board. Sufficient space should be maintained between the printed boards for heated air to circulate and remove the moisture.

The following figure lists the factors that affect the time and temperature required for effective baking:

- Moisture content of the printed board
- Desired degree of dryness after the bake
- Moisture desorption characteristics of the laminate
- Overall printed board thickness
- Locations and structures of copper plane layers
- Design features such as plated edges

Fig. 2.2.3: Factors that affect the time and temperature required for effective baking

The following table shows the typical recommendations for oven settings:

Final Finish	Temperature	Time	Comments
Tin	105 – 125 °C	4-6 Hours	Higher temperature may reduce solderability
Silver	105 – 125 °C	4-6 Hours	Silver may tarnish, but solderability should not be affected
Nickel/Gold	105 – 125 °C	4-6 Hours	No issue with extended bake on Nickel/Gold finish
HASL/HAL	105 – 125 °C	4-6 Hours	Final surface thickness below 0.77 μm [30.0 μin] may turn into pure intermetallics and render the printed board unsolderable

Table 2.2.1: Typical recommendations for oven settings

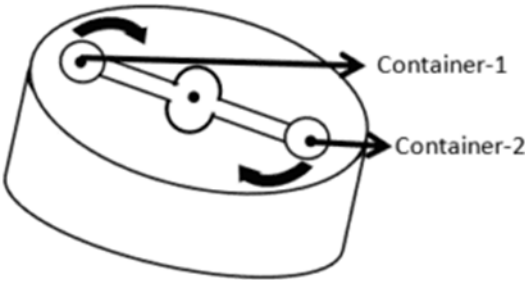
If the boards are baked in stacks, the maximum recommended stack height is 25.4mm (1.0in) to ensure that heating is uniform throughout the stack. Stacks are typically baked for two hours at 105-120 ºC. The material should be cooled as fast as possible in a dry environment as hot material in a humid atmosphere will reabsorb moisture rapidly.

2.2.2 Solder Paste Thawing

Solder paste is a critical material in surface-mount technology (SMT), consisting of tiny solder spheres suspended in a flux medium. For optimal performance, it must be stored under specific conditions and properly prepared before use.

Solder Paste Storage and Handling

Solder paste is typically stored in a refrigerated environment at a low temperature, usually around 5°C, to maintain its consistency and extend its shelf life. To ensure freshness and prevent using expired material, a First-In, First-Out (FIFO) inventory management system should always be followed.



Thawing Process:

Before being used in a solder paste printer, the cold solder paste must be brought to the ambient temperature of the shop floor. This process, known as thawing, is essential to ensure the paste's viscosity is correct for printing. Using cold paste can lead to improper flow, poor stencil release, and defects.

To thaw the paste, it's typically removed from cold storage and allowed to sit in a cool, controlled area. For more precise and accelerated thawing, a solder paste thawing apparatus is often used. This machine gently agitates the solder paste container, ensuring a uniform temperature and consistency throughout the paste. The speed (RPM) and duration of the agitation are determined by factors such as:

- The initial cold storage temperature.
- The size of the solder spheres in the paste.
- The manufacturer's recommendations.

Post-Thawing Inspection

Once thawed, the solder paste must be inspected to confirm it is ready for use. Technicians check for:

- Solid Lumps: Presence of solid clumps indicates improper thawing or degradation.
- Air Bubbles: Bubbles can lead to voids in the solder joints after reflow.
- Flux Separation: Visible separation of the solder spheres from the flux indicates the paste is not a homogeneous mixture.

If any of these issues are detected, the paste should either be re-thawed (if the issue is minor) or discarded to prevent printing defects.

Types of Thawing Equipment

Modern thawing equipment comes in various designs, offering different features to suit production needs:

- Container Capacity: Machines can thaw multiple containers at once, improving efficiency.
- Speed Control: Some machines offer variable speed settings, allowing for precise control based on different paste types.
- Maintenance: Equipment is designed for low maintenance, as solder paste can sometimes spill and contaminate internal components.

Proper thawing and inspection are critical steps in the SMT assembly process, directly impacting the quality and reliability of the solder joints.

Solder Paste Stencils

A solder paste stencil is a thin sheet of material, typically stainless steel, with precision-cut openings (apertures) that correspond to the surface mount pads on a PCB. Created from the PCB's CAD data, the stencil is used in the printing process to deposit a precise amount of solder paste onto the pads.

Fabrication and Use

Stencils are often laser-cut to achieve the high precision required for modern electronics. The material can be coated or plated to improve paste release and ensure a smooth application. During the printing process, the stencil is aligned with the PCB, and a squeegee blade is used to spread solder paste across its surface. The paste is forced through the apertures, leaving a uniform deposit on each pad.

The process typically takes 15-45 seconds per board

After printing, the board is often inspected using an automated optical inspection (AOI) system to verify the quality and volume of the paste deposit. The stencil is then cleaned before being used for the next board.

Advantages of Using a Solder Paste Stencil

Using a solder paste stencil offers numerous benefits that are crucial for efficient and high-quality PCB assembly:

- Uniformity and Precision: Stencils ensure that the correct amount of solder paste is applied uniformly to every pad, resulting in consistent and reliable solder joints. This is a significant advantage over manual application.
- Efficiency: They enable the entire board to be printed with solder paste in a single step, drastically reducing the time and effort required compared to applying paste to each pad individually.
- High Accuracy: Stencils provide excellent positional accuracy and are essential for fine-pitch components, which have very small pads and tight spacing. Their use significantly reduces the risk of defects like bridging (solder shorts) and insufficient paste (open circuits).
- Process Control: By controlling the volume and shape of the paste deposit, stencils help to make the assembly process reliable and repeatable, leading to higher yields and reduced rework.

2.2.3 Temperature and Humidity Control in SMT Manufacturing

Proper environmental control is critical in Surface Mount Technology (SMT) manufacturing to prevent defects and ensure product quality. Uncontrolled temperature and humidity can negatively affect solder paste performance and moisture-sensitive components.

Relative Humidity (Rh)

Relative humidity is the measure of water vapor in the air relative to the maximum amount the air can hold at a given temperature. Controlling it is essential for the reliability of solder joints and components.

High Humidity

High humidity can cause solder paste to absorb excessive moisture, leading to:

- Slumping: The paste becomes too fluid and spreads out, causing bridging (unwanted solder connections) between adjacent pads during reflow.
- Solderballing: Trapped moisture rapidly boils during reflow, violently expelling solder particles and creating small spheres of solder away from the joint.
- Popcorning: This is a term used to describe the effect on Moisture Sensitive Components (MSCs). Moisture absorbed by the component package expands rapidly during reflow, causing the package to crack or even explode.

Low Humidity

Low humidity can cause the flux in solder paste to dry out too quickly. This results in:

- Poor Stencil Release: The dried paste can stick to the stencil, leading to incomplete deposition on the PCB pads.
- Insufficient Solder Joints: The lack of proper paste volume can result in weak or open solder joints.

Temperature

Temperature directly affects the viscosity of solder paste and the behavior of components.

I. High Temperature

High temperatures reduce the viscosity of solder paste, causing it to become too thin. This can lead to:

- Slumping and Smearing: The paste becomes overly fluid and spreads, potentially leading to bridging.
- Voiding: Excessive heat can cause gasses in the paste to escape too quickly, creating voids (empty spaces) within the finished solder joint.
- Oxidation: Higher temperatures accelerate the oxidation of both solder paste and component pads, which can inhibit proper wetting and create weak solder connections.

II. Low Temperature

Low temperatures increase solder paste viscosity, making it too thick. This can cause:

- Printing Defects: The paste is too stiff to roll properly, leading to inconsistent paste volume, incomplete aperture filling, and poor stencil release.

Acceptable Conditions and Control

While expert opinions on the optimal ranges vary, a generally accepted range for most SMT processes is a temperature of 68-78°F (20-25°C) and a relative humidity of 40-60%. However, the ideal conditions can depend on the specific solder paste and components used.

- Monitoring: Use high-quality humidity and temperature sensors placed strategically on the manufacturing floor. Regular calibration and inspection of these sensors are crucial.
- Environmental Control: Invest in a robust HVAC system that can effectively manage temperature and humidity. A dedicated dehumidifier is essential in humid climates.
- Nitrogen in Ovens: Using nitrogen gas in the reflow oven can help reduce oxidation, especially in environments with high humidity.
- Moisture-Sensitive Components (MSCs): These components are graded based on their sensitivity to moisture. They must be stored in dry bags with a desiccant and exposed to the atmosphere for a limited time before assembly to prevent popcorning during reflow. Adhering to the manufacturer's moisture exposure guidelines is critical.

Notes

UNIT 2.3: Screen Printing Machines

Unit Objectives

By the end of this unit, the participants will be able to:

1. Differentiate between the different types of screen-printing machines.

2.3.1 Types of Screen-Printing Machines

In screen-printing and component placement, there are four key dimensions that must be controlled for accuracy:

- X-axis – Horizontal movement.
- Y-axis – Vertical movement.
- Z-axis – Height or pressure control.
- Θ (Theta) – Rotational alignment.

The precision with which these parameters are adjusted directly impacts solder paste deposition quality and overall assembly accuracy. Based on the level of automation available to control these variables, three main types of screen-printing machines are used in PCB assembly.

Manual Screen-Printing Machine

In a manual screen printer, all four parameters (X, Y, Z, and Θ) are adjusted manually by the operator. This type of machine is generally used for low-volume production, prototyping, or training purposes, where flexibility and cost-effectiveness are more important than high throughput.

The image shows the different parts of a manual screen-printing machine:

Fig. 2.3.1: Parts of a manual screen-printing machine

Semi-Automatic Screen-Printing Machine

A semi-automatic stencil printer combines manual setup with automated functions to improve consistency and reduce operator effort. Typical features include:

- Automatic board clamping and release for easier loading and unloading, reducing operator fatigue.
- Controlled squeegee pressure to ensure consistent paste deposition.
- Programmable squeegee movement and speed, improving repeatability across multiple prints.
- Visual alignment/assist tools to support accurate PCB positioning against the stencil.
- Semi-automatic machines are well-suited for medium-volume production, balancing cost and efficiency.

Fig. 2.3.2: A semi-automatic screen-printing machine

Automatic Screen-Printing Machine

An automatic screen printer provides the highest level of precision, repeatability, and throughput. These machines integrate advanced features such as:

- Automatic fiducial alignment to control positioning in X, Y, Z, and Θ axes.
- Automated board loading and unloading, minimizing operator intervention.
- Optimized process control for high-speed, high-volume production with consistent quality.

For production environments handling 2,000 or more boards per day, automatic systems offer significant benefits in terms of labor cost reduction, improved yield, and greater control over the printing process.

Fig. 2.3.3: An automatic screen-printing machine

The following table lists the parameters of an automatic screen-printing machine:

Parameter	Specification	Notes
Printing Throughput	~ 4,000 – 6,000 PCB/hr (depending on PCB size & complexity)	High-speed models achieve >6,000/hr with inline automation.
Printing Accuracy	± 20–25 µm @ 6 Sigma	Critical for fine-pitch SMT components (<0.5 mm pitch).
Repeatability	± 12–15 µm	Ensures uniform solder paste deposition.
Max PCB Size	Typically up to 510 × 510 mm	Mid- to high-range machines; compact machines may support 300 × 300 mm.
PCB Thickness Range	0.4 – 6.0 mm	Compatible with standard FR-4 and specialty boards.
Stencil Frame Size	Adjustable, typically 584 × 584 mm (23″ × 23″)	Can support larger with adapter kits.
Paste Rolling Speed (Squeegee Stroke)	30 – 200 mm/sec	Programmable, depends on paste type.
Squeegee Pressure Control	Automatic, programmable	Ensures consistent paste fill across apertures.
Cleaning System	Automatic (dry, wet, vacuum)	Cleans underside of stencil & apertures to prevent defects.
Fiducial Alignment	Automatic vision alignment (X, Y, Z, Θ)	Uses camera to detect PCB fiducials for precise registration.
Power Supply	220–415 VAC, 50 Hz, 3-Phase, 2–6 kW	Indian shop-floor compatible.
Compressed Air Requirement	~ 0.5–0.6 MPa (5–6 bar)	For pneumatic actuators & clamping.
Machine Dimensions	~ 1.5 – 2.0 m (W) × 1.2 – 1.5 m (D) × 1.5 – 1.8 m (H)	Mid-range PCB printers; inline systems may be longer.
Weight	~ 800 – 1,200 kg	Heavy-duty frame for vibration-free precision.
Control System	Touchscreen + PC-based GUI	Often Windows/Linux based with SPC (statistical process control).
Environment Requirement	Temp: 23 ± 3 °C, RH: 40–60%	Controlled environment avoids solder paste drying/moisture issues.

Table 2.3.1: Typical parameters of an automatic screen-printing machine

2.3.2 Setting Up a Screen-Printing Machine

The following table lists the standard value of different parameters of a screen-printing machine:

Parameter	Standard Values (Corrected & Verified)	Notes / Clarifications
Squeegee		
Speed	20 – 150 mm/sec (≈1–6 in/sec)	150 mm/sec (6 in/sec) is correct, but range depends on solder paste type & aperture size.
Pressure	5 – 15 N (≈0.5 – 1.5 kg) per 25 mm of blade length	“0.5 kg” alone is too low — pressure is proportional to blade length. Typical motion speed is set separately, not as “20 mm/sec.”
Angle	45° – 60° typical; up to 75° in some cases	60° is most common for fine-pitch SMT printing.
Solder Paste		
Thickness	100 – 150 µm (4 – 6 mils)	145 µm is within standard range.
Viscosity	300,000 – 1,600,000 cP	Range given is correct (per IPC-TM-650 & paste datasheets).
Composition (Sn-Pb)	Sn 63% / Pb 37%	Correct for eutectic solder.
Composition (Lead-Free)	Sn 96.5% / Ag 3% / Cu 0.5%	Correct (SAC305 alloy).
Stencil		
Type	Laser-cut stainless steel (framed or frameless, nano-coated optional)	Matches industry practice.
Thickness	100 – 150 µm (≈0.004 – 0.006 in / 4 – 6 mils)	Your values (0.02–0.16 in = 20–160 mils) are incorrect by factor of 10. Standard stencil thickness is 0.004–0.006 in, not 0.02 in+.
Environment		
Temperature	22 – 25 °C	Correct, stable room temperature required.
Humidity	40 – 60% RH	Correct range, though keeping closer to 50% is ideal. Below 30% risks paste drying; above 60% risks moisture absorption.

Table 2.3.2: Standard value of different parameters of a screen-printing machine

The following image shows the user interface of a screen-printing machine:

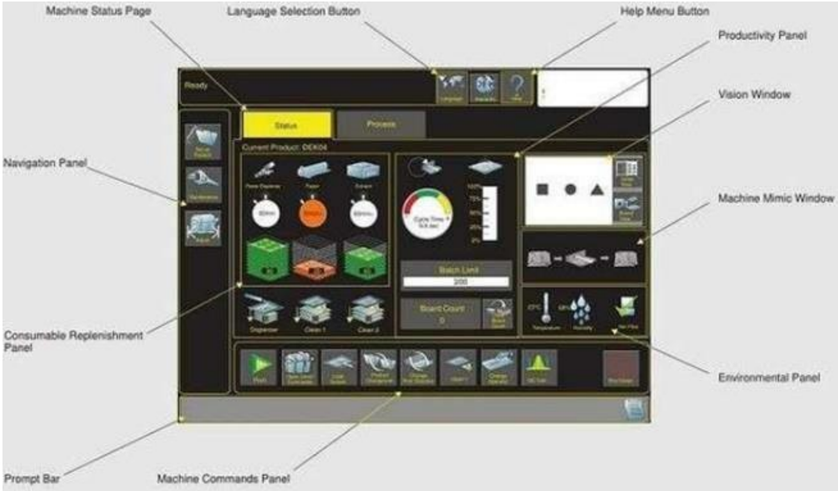


Fig. 2.3.4: User interface of a screen-printing machine

Safety Measures for Screen-Printing Machine Operation

1. Personal Safety (Operator Precautions)
   - Always wear ESD-safe gloves, shoes, and wrist straps to prevent electrostatic discharge that can damage PCBs.
   - Use safety glasses to protect eyes from solder paste splashes or cleaning solvents.
   - Avoid loose clothing, jewelry, or long hair near moving parts (squeegee, conveyor, stencil lift).
   - Wash hands after handling solder paste, flux, or cleaning chemicals to prevent ingestion or skin irritation.

2. Machine Safety
   - Ensure all guards, covers, and interlocks are in place before operation.
   - Do not attempt to reach into the machine while it is running (squeegee or stencil mechanism can cause injury).
   - Follow proper lockout/tagout procedures when servicing or repairing the machine.
   - Regularly inspect pneumatic and electrical connections for leaks, wear, or loose fittings.
   - Avoid overloading the machine with oversized or warped PCBs, as this can damage alignment systems.

3. Solder Paste & Chemical Handling
   - Store solder paste in refrigerated conditions (1–10 °C); warm to room temperature before use.
   - Mix solder paste in a closed environment (manual or automatic mixer) to avoid fumes.
   - Handle solvents (e.g., IPA for stencil cleaning) in a well-ventilated area or under fume extraction.
   - Dispose of expired solder paste and cleaning materials according to hazardous waste guidelines.

4. Environmental & Ergonomic Safety
    - Maintain room temperature 22–25 °C and RH 40–60% to ensure stable printing and reduce moisture absorption.
    - Provide adequate lighting around the stencil area for visual inspection.
    - Ensure proper ventilation / fume extraction near the printing and cleaning zones.
    - Set machine at appropriate working height to reduce operator fatigue.

5. Emergency Preparedness
    - Train operators to immediately press the Emergency Stop (E-Stop) button in case of abnormal sounds, motion, or hazards.
    - Keep a spill kit for solder paste or solvent leaks.
    - Provide nearby fire extinguishers rated for electrical and chemical fires.
    - Maintain first-aid kits and train staff in basic response to chemical exposure.

6. Operational Best Practices
    - Perform daily cleaning of machine surfaces to avoid solder paste buildup.
    - Clean stencils at recommended intervals to prevent smudging/bridging defects.
    - Use lint-free wipes and only approved solvents for stencil and underside cleaning.
    - Train operators to handle PCBs with care to avoid bending, contamination, or ESD damage.

Notes

UNIT 2.4: Solder Paste Inspection (SPI)

Unit Objectives

By the end of this unit, the participants will be able to:

1. Describe the importance of Solder Paste Inspection (SPI) in the PCB assembly process.
2. Differentiate between 2D and 3D inspection methods for solder paste.
3. Identify common solder paste printing defects and their root causes.
4. Explain how SPI equipment helps in process control and defect prevention.

2.4.1 Identification of Solder Paste Printing Defects

Solder paste printing is the most critical process in Surface Mount Technology (SMT) assembly, as printing defects account for a significant majority (often cited as over 60%) of all end-of-line faults. Solder Paste Inspection (SPI) is an automated process used to monitor and control the quality of the solder paste deposit on the PCB pads before component placement. Implementing SPI at this stage is a crucial strategy for preventing costly rework, improving first-pass yield, and achieving a faster return on investment.

Fig. 2.4.1: An SPI Equipment

Key Features of Modern SPI Systems

Modern SPI systems, such as 5D post-print inspection systems, use advanced sensor technology and combine both 3D and 2D imaging methodologies simultaneously to provide comprehensive defect detection. This combined approach offers:

- Topographical 3D Zero Referencing: The system uses the board's surface as a reference plane, allowing for precise height measurements of the solder paste deposit.
- Shadow-Free Measurement: By using multiple light sources, 3D systems can eliminate shadows, ensuring accurate and precise measurement of paste volume, shape, and height.

Types of Inspection (2D/3D)

Automated SPI systems can verify the accuracy of solder paste deposits. The primary types of inspection available are 2D and 3D.

- 2D Inspection: This method uses a single camera and a light source to capture a top-down, two-dimensional image of the solder paste deposit. It can measure the area and position of the paste. However, it cannot measure the height or volume of the paste, making it susceptible to false readings from slumped or smeared paste.
- 3D Inspection: This method uses structured light (e.g., lasers or digital projectors) and multiple cameras to create a 3D topographical map of the solder paste deposit. This allows for precise measurement of volume, height, shape, and area. 3D inspection is considered the industry standard as it provides the most critical data for process control and is highly effective at detecting subtle defects that 2D systems cannot.

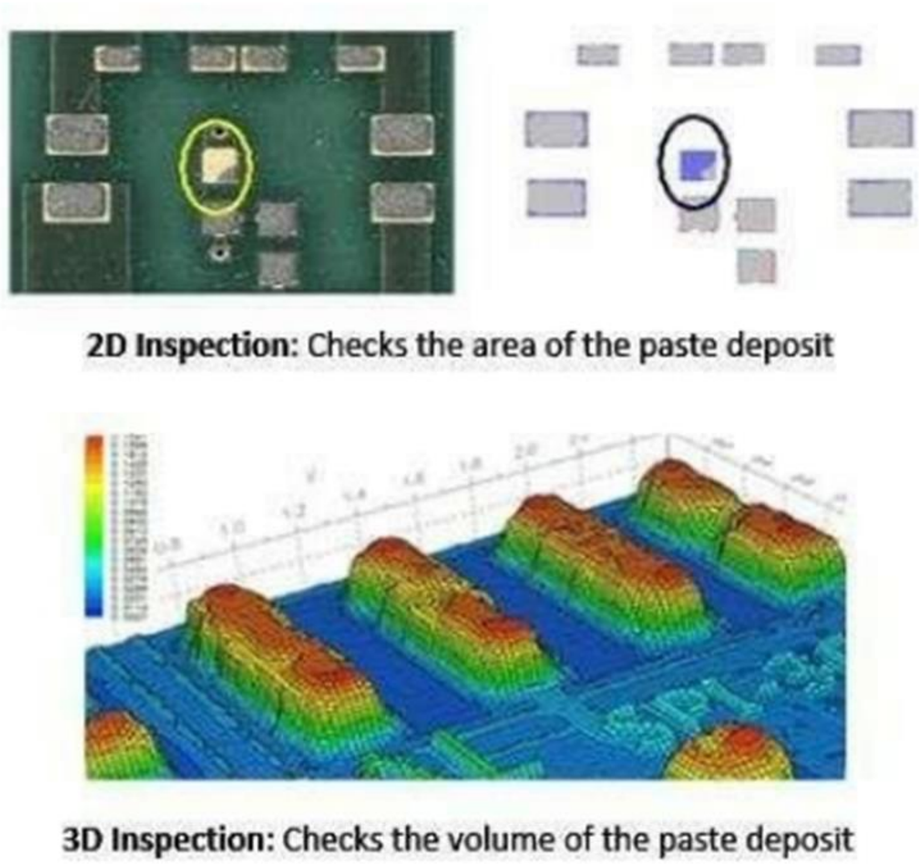


Fig. 2.4.2: Types of inspection available

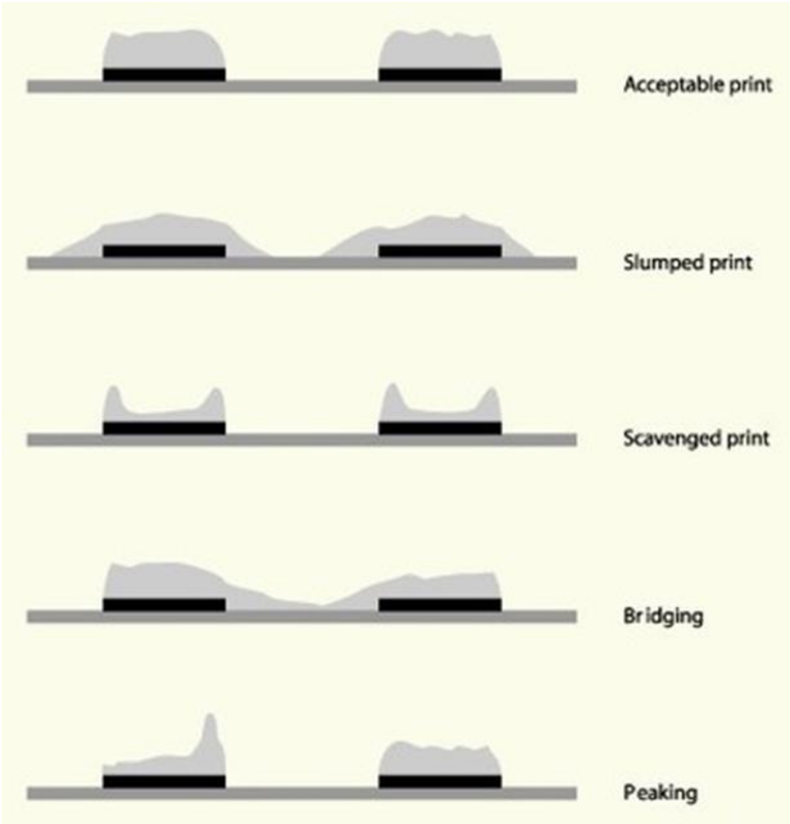


Fig. 2.4.3: Possible solder paste print inspection results

Common Solder Paste Printing Defects

Here are some common solder paste printing defects and their primary causes:

- Slumping: This defect occurs when the solder paste spreads or collapses after printing. It is often caused by an overly high ambient temperature on the production floor, which reduces the paste's viscosity.
- Scooping/Scavenging: This happens when the solder paste is partially or completely removed from the stencil apertures. The most frequent cause is excessive squeegee pressure, which scoops the paste out of the openings rather than pushing it through.
- Bridging: This is the unwanted connection of solder paste between adjacent pads. It can be caused by a variety of issues, including:
- Improper board support, which allows the PCB to flex.
- A dirty or damaged stencil, which prevents a clean print.
- Slumping due to high temperatures or humidity.
- Peaking/Dog-Earing: This defect refers to the formation of small peaks or "dog-ears" on the top of the solder paste deposit. It is typically caused by a stencil separation speed that is too fast, which pulls the paste upward as the stencil separates from the board.

The following figure lists other solder paste printing defects and their criteria in a table:

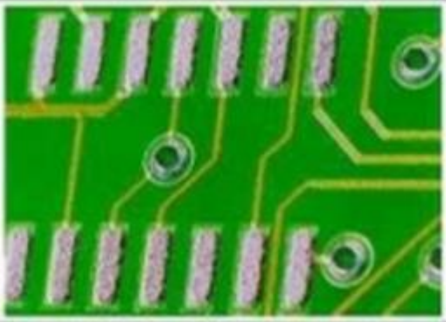
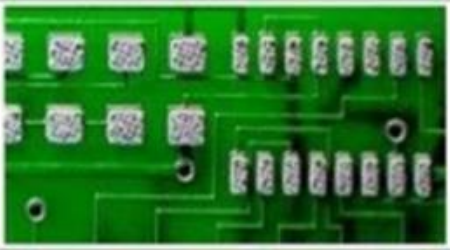
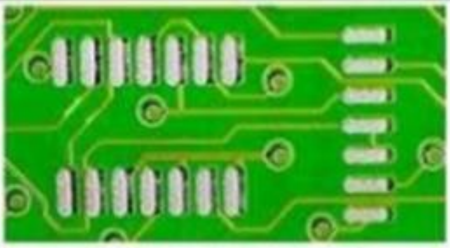
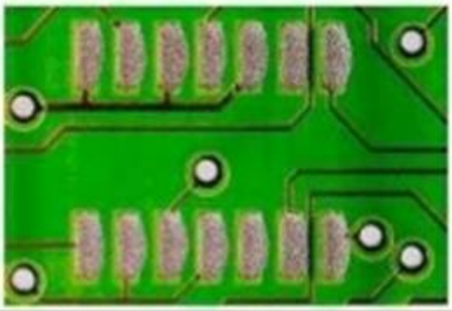
Solder paste printing shifted in x/y direction	Criteria
	**Preferred**
	**Acceptable** For Components like 0805, 1206 : Displacement ≤ 0.2 mm. For multi-lead components like array resistor with a pitch ≤ 0.8 mm : Displacement ≤ 0.1 mm.
	**Rejected** For Components like 0805, 1206 : Displacement > 0.2 mm. For multi-lead components like array resistor with a pitch ≤ 0.8 mm : Displacement >0.1 mm
	**Rejected** Contamination of paste, too little paste and no paste
	**Rejected** Paste outflow more than 0.2mm outside the solder land caused by too much solder paste or too great a placing force, (for multi-lead components with pitch ≤ 0.8 mm : outflow more than 0.1mm)

Fig. 2.4.4: Other solder paste printing defects and their criteria

Exercise

Short Questions:

1. Explain the role of solder paste in SMT assembly.
2. List three critical parameters of the screen-printing process and describe their impact.
3. Differentiate between manual, semi-automatic, and automatic screen-printing machines.
4. Why is pre-baking of PCBs sometimes required before assembly?
5. Compare 2D and 3D solder paste inspection methods with examples of defects they can detect.

Fill in the Blanks:

1. The ___________ is a thin sheet of stainless steel used to transfer solder paste onto PCB pads.
2. Typical stencil thickness ranges from ___________ to ___________ microns.
3. The process of gradually separating the PCB from the stencil after printing is called ___________ speed.
4. The ___________ rule states that at least five solder particles should span across the width of the smallest aperture.
5. Delamination and blistering in PCBs during soldering are caused by ___________.

Multiple Choice Questions (MCQs):

1. Which tool is used to spread solder paste evenly across the stencil?
   a) Brush
   b) Squeegee blade
   c) Roller
   d) Heater
2. The recommended angle of a squeegee blade during screen printing is:
   a) 10°–20°
   b) 25°–35°
   c) 45°–60°
   d) 80°–90°
3. Excessive squeegee pressure can cause:
   a) Better solder paste transfer
   b) Scooping of paste and stencil damage
   c) Improved paste uniformity
   d) Reduced cleaning requirement

4. The baking process of a PCB before assembly is mainly done to:
    a) Reduce pad size
    b) Remove absorbed moisture
    c) Improve stencil release
    d) Reduce copper oxidation
5. Which inspection method can measure solder paste height and volume accurately?
    a) 2D Inspection
    b) Manual visual inspection
    c) 3D Inspection
    d) None of the above

Notes

सत्यमेव जयते
GOVERNMENT OF INDIA
MINISTRY OF SKILL DEVELOPMENT
& ENTREPRENEURSHIP

3. Component Placement on Telecom Boards

TEL/N2504

Key Learning Outcomes

By the end of this module, the participants will be able to:

1. List the key components of the pick-and-place process.
2. Define common terms used in pick-and-place operations.
3. Identify the major parts of a pick-and-place machine.
4. Demonstrate the correct procedure for starting up and shutting down the machine.
5. List the different types of feeders used in SMT assembly.
6. Demonstrate the steps for operating tape feeders.
7. Recognize the application of trays for loading chip components.
8. Explain the functions of a chip shooter and fine-pitch placer.

UNIT 3.1: Components Pick and Place Operations

Unit Objectives

By the end of this unit, the participants will be able to:

1. List the key components of an automated pick and place machine.
2. Define essential terms used in pick and place operations, such as fiducial marks and component feeders.

3.1.1 Components of the Pick and Place Process

The pick and place process is the core of modern electronics manufacturing, where electronic components are precisely placed onto a PCB. This step is crucial for creating a functional circuit by ensuring that component leads are correctly positioned within the solder paste on the pads.

The pick and place process is executed by a sophisticated machine that assembles electronic components onto a printed circuit board (PCB) with high speed and precision. The key components of this process include:

Pick and Place Machine:

The core of the operation. This is a robotic system with a moving head that can access component feeders and the PCB.

- Nozzle Head: The part of the machine that picks up and places components. It uses vacuum pressure to hold the components securely.
- Component Feeders: These are dispensing units that hold the components in an organized manner (e.g., on tapes, in tubes, or in trays) and present them to the nozzle head for picking.
- Conveyor System: This moves the PCB into and out of the machine's work area, ensuring a continuous and automated assembly line.
- Vision System (Cameras): Cameras are used for two primary functions:
- Fiducial Recognition: To precisely align the PCB before placement.
- Component Recognition: To inspect components for correct orientation and to verify they have been picked correctly.
- Control Software: The software that controls all machine functions, including the placement coordinates (from the CAD data), speed, and vision system operations.

Automated Placement

To achieve the high speed and accuracy required for mass production, automated pick and place machines are used. These machines use vacuum nozzles to pick components from feeders or trays and then place them onto the PCB. During this process, a camera often performs a visual inspection to ensure the correct component has been picked and to verify its orientation. The machine precisely adjusts the component's position using a coordinate system before placing it on the board.

For double-sided boards or components that may shift before soldering, a small amount of Surface Mount Technology (SMT) glue can be applied to temporarily secure the component to the PCB.

Coordinate System

The precise positioning of components is controlled by a multi-axis coordinate system:

- X-axis: Controls horizontal movement (left and right), with a positive value typically representing movement to the right.
- Y-axis: Controls vertical movement (forward and back), with a positive value typically representing movement forward.
- Z-axis: Controls vertical height (up and down), with a positive value typically representing upward movement.
- C-axis (or Theta, Θ): Controls the rotational movement of the component, with a positive value typically representing a counter-clockwise rotation.

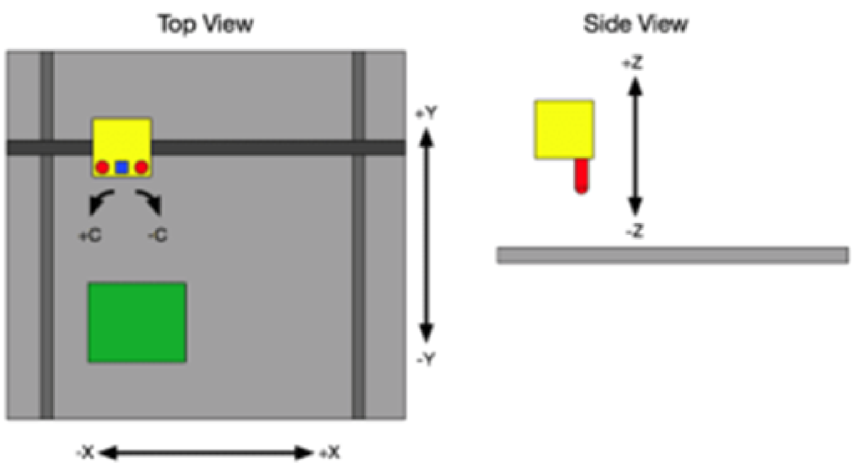


Fig. 3.1.2: Top view and side view of the positioning of the components

The units for the axes are set in the Graphic User Interface (GUI). The default unit for X, Y and Z axes is millimeters and for C axis is degree.

3.1.2 Common Terms Used in Pick and Place Operations

Board:

In SMT manufacturing, a board refers to the physical Printed Circuit Board (PCB) onto which electronic components are mounted. Each revision or design update of a PCB is considered a new board. In pick-and-place operations, the board data (often from CAD or Gerber files) contains placement information that guides the machine software on where and how to position each component accurately.

Fiducial:

A fiducial is a reference marker on the PCB used by the vision system of the pick-and-place machine to align the board with high precision. Fiducials are typically small, round copper pads with a clear, solder mask–free area around them (known as a keep-out zone) to ensure high-contrast recognition. They allow the machine to correct for slight shifts, rotations, or distortions in PCB positioning during assembly. Most PCBs include global fiducials (for overall board alignment) and local fiducials (near fine-pitch components for localized accuracy).

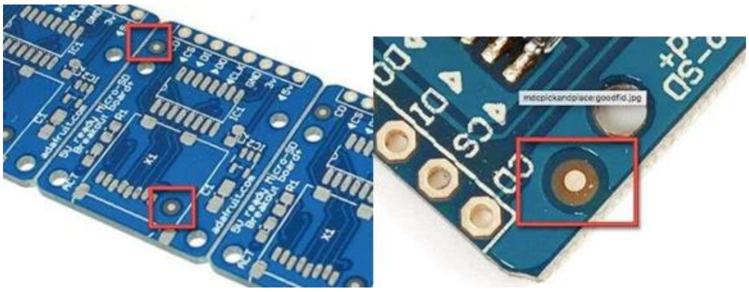


Fig. 3.1.3: Fiducials on a PCB

The following images show fiducial alignments:

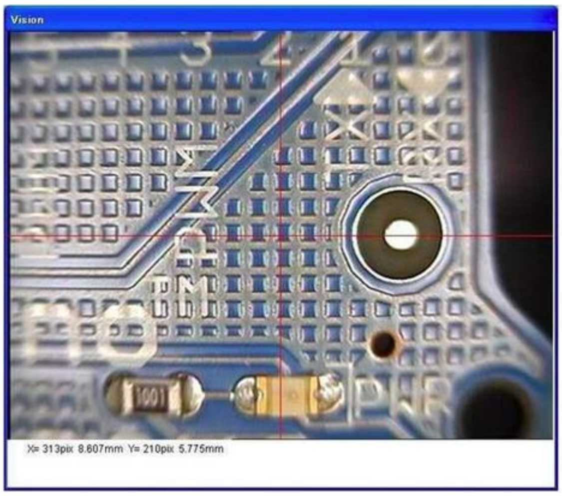


Fig. 3.1.4a: Before fiducial alignment

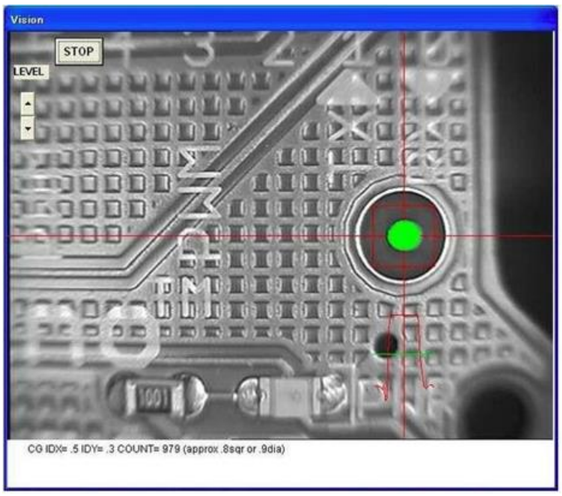


Fig. 3.1.4b: After fiducial alignment

Customers are generally recommended to provide three fiducials on a PCB for optimal alignment. If space constraints do not allow three, then two fiducials are sufficient, provided they are placed at diagonally opposite corners of the board to ensure proper scaling and rotation compensation.

However, the use of three fiducials is considered best practice, as each one serves a distinct purpose:

1. First Fiducial
    - Defines the reference point of the PCB in the X and Y dimensions.
    - Allows the pick-and-place machine to establish the board's location in its workspace.
2. Second Fiducial
    - Defines the orientation of the PCB and compensates for skew or rotation in the clamps.
    - Even a small rotation (e.g., 0.1°) can affect accurate placement, but fiducials allow the machine to detect and correct such deviations.
    - Modern machines can measure rotation with precision down to 0.01° and adjust all placements accordingly.
3. Third Fiducial
    - Compensates for PCB stretch, shrinkage, or warpage that can occur during fabrication or reflow.
    - Particularly critical for large PCBs where dimensional variation accumulates over longer distances.
    - In double-sided SMT assemblies, after the first side is reflowed, the board may shrink, expand, bow, or flex; a third fiducial helps correct alignment for the second-side assembly.

The figures below illustrate:
- The typical measurement and size specifications of a fiducial.
- The three types of fiducials commonly used on a PCB (global fiducials, local fiducials, and panel fiducials).

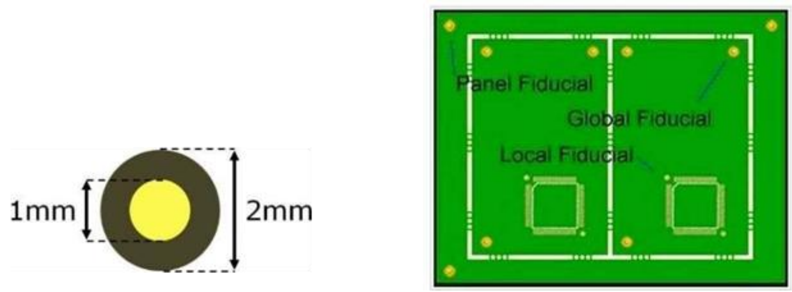


Fig. 3.1.5: Measurement of a fiducial and three types of fiducials

Part
- Refers to a specific component defined for placement on a PCB.
- It is linked to the manufacturer's part number and represents both the function and the value of the component.
- Example: A 22 kΩ 0603 resistor and a 10 kΩ 0603 resistor are treated as different parts, even though they share the same package size.

Package
- Describes the physical attributes of a part, including dimensions, pin configuration, and footprint.
- Different components can share the same package type.
- Examples include 0603 (chip package), SOIC-8, TQFP-32, QFN, and BGA.

Placement

- Refers to the specific location on the PCB where a part must be mounted.
- Defined by X and Y coordinates relative to the board origin, along with rotation (θ).
- Each placement is associated with a part assignment so the machine software knows exactly which component goes where.

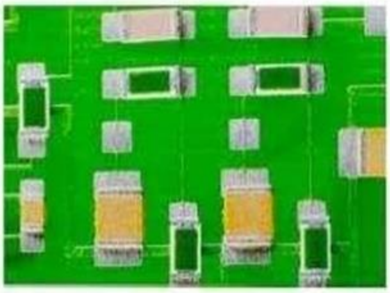
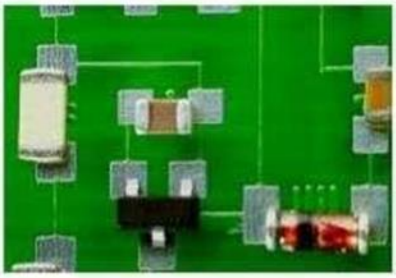
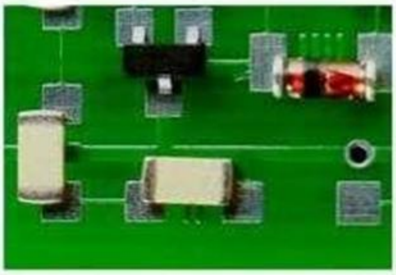
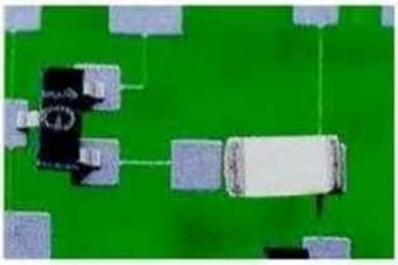
Components shifted in x/y direction	Criteria
	**Preferred**
	**Acceptable** Half or more of the width of the component is situated on the solder land. Only acceptable when the conductor is covered with insulating lacquer.
	**Rework** Less than half the width of the component is situated on the solder land.
	**Rework** The metallisation must at least be partly positioned on the solder land.

Fig. 3.1.6: Criteria of component placements

Job

- A job is a machine-readable file that contains instructions for processing one or more PCBs in a production run.
- It includes details such as board design data, component placements, feeder assignments, and machine settings.
- A job may contain a mixture of different board types or multiple instances of the same board.

Footprint

- A footprint defines the layout of pads, land patterns, and outlines of a component on the PCB.
- It ensures that the component aligns correctly with the solder lands during placement and soldering.
- Footprints are usually created following IPC-7351 standards to maintain compatibility and manufacturability.

Key Parameters Influencing Pick and Place Accuracy

- Component dimensions and tolerances – As specified in the component datasheet, these determine how accurately the machine can grip and place the part.
- Board dimensional accuracy – Variations in PCB size or warpage can affect placement precision.
- Placement accuracy – The alignment of the component leads or terminals relative to the solder lands on the PCB.
- Solder paste position tolerances – For reflow soldering, the stencil-applied solder paste must align precisely with the pads to ensure good solder joints.
- Soldering process parameters – Reflow oven profiles, temperature settings, and conveyor speeds must be optimized for reliable solder joints.
- Solder resist (solder mask) position tolerances – The solder mask must be accurately registered to prevent covering pads or leaving unwanted exposed copper.
- Solder joint quality parameters – Factors such as wetting, fillet formation, and void percentage are critical for mechanical strength and electrical reliability of the joint.

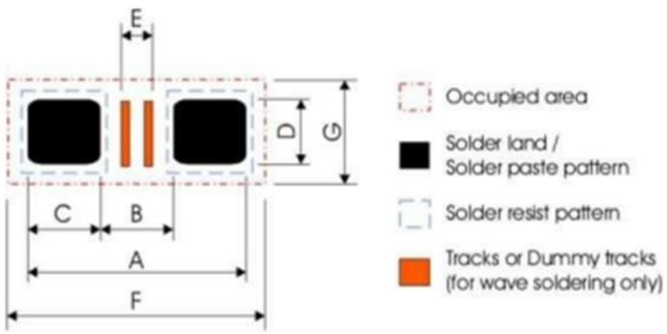


Fig. 3.1.7: Footprint details

■ Footprint design for Array Resistor :

Type	0603*4	0402*4	0402*2
Symbol / Item	WA06X / WA06T	WA04X	WA04Y, WA04P
A	2.85 +0.10/-0.05	1.80 +0.15/-0.05	1.20 ± 0.05
B	0.45 ± 0.05	0.30 ± 0.05	0.40 +0/-0.05
D	0.80 ± 0.10	0.50 ± 0.1	0.50 ± 0.05
P	0.80	0.50	0.65
F	3.10 ± 0.30	2.00 +0.40/-0.20	1.50 +0.20/-0.10

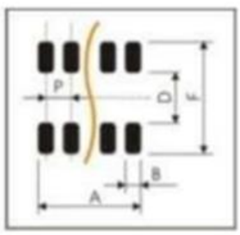


Fig. 3.1.8: Footprint design details for array resistor

■ Footprint design for discrete CHIP-R

◆ Reflow Soldering

SIZE	Footprint dimensions in mm							Processing remarks	Placement Accuracy
	A	B	C	D	E	F	G		
0201	0.75	0.30	0.30	0.30	0.20	1.10	0.50	IR reflow soldering	± 0.05
0402	1.50	0.50	0.50	0.60	0.10	1.90	1.00	IR or hot plate soldering	± 0.15
0603	2.10	0.90	0.60	0.90	0.50	2.35	1.45		± 0.25
0805	2.60	1.20	0.70	1.30	0.75	2.85	1.90		± 0.25
1206	3.80	2.00	0.90	1.60	1.60	4.05	2.25		± 0.25
1218	3.80	2.00	0.90	4.80	1.40	4.20	5.50		± 0.25
2010	5.60	3.80	0.90	2.80	3.40	5.85	3.15		± 0.25
2512	7.00	3.80	1.60	3.50	3.40	7.25	3.85		± 0.25

◆ Wave Soldering

SIZE	Footprint dimensions in mm							Proposed number & Dimensions of dummy tracks	Placement Accuracy
	A	B	C	D	E	F	G		
0603	2.70	0.90	0.90	0.80	0.15	3.40	1.90	1x (0.15x0.80)	± 0.25
0805	3.40	1.30	1.05	1.30	0.20	4.30	2.70	1x (0.20x1.30)	± 0.25
1206	4.80	2.30	1.25	1.70	1.25	5.90	3.20	3x (0.25x1.70)	± 0.25
1218	4.80	2.30	1.25	4.80	1.30	5.90	5.60	3x (0.25x4.80)	± 0.25
2010	6.30	3.50	1.40	2.50	3.00	7.00	3.60	3x (0.75x2.50)	± 0.25
2512	8.50	4.50	2.00	3.20	3.00	9.00	4.30	3x (1.00x3.20)	± 0.25

Fig. 3.1.9: Footprint design details for discrete CHIP-R

■ Footprint design for discrete MLCC :

◆ Reflow Soldering

SIZE	Footprint dimensions in mm							Processing remarks	Placement Accuracy
	A	B	C	D	E	F	G		
0402	1.50	0.50	0.50	0.50	0.10	1.75	0.95	IR or hot plate soldering	± 0.15
0508	2.50	0.50	1.00	2.00	0.15	2.90	2.40		± 0.20
0603	2.30	0.70	0.80	0.80	0.20	2.55	1.40		± 0.25
0612	2.80	0.80	1.00	3.20	0.20	3.08	3.85		± 0.25
0805	2.80	1.00	0.90	1.30	0.40	3.08	1.85		± 0.25
1206	4.00	2.20	0.90	1.60	1.60	4.25	2.25		± 0.25
1210	4.00	2.20	0.90	2.50	1.60	4.25	3.15		± 0.25
1808	5.40	3.30	1.05	2.30	2.70	5.80	2.90		± 0.25
1812	5.30	3.50	0.90	3.80	3.00	5.55	4.05		± 0.25
2220	6.50	4.70	0.90	5.60	4.20	6.75	5.85		± 0.25

◆ Wave Soldering

SIZE	Footprint dimensions in mm							Proposed number & Dimensions of dummy tracks	Placement Accuracy
	A	B	C	D	E	F	G		
0603	2.40	1.00	0.70	0.80	0.20	3.10	1.90	1x (0.20x0.80)	± 0.10
0805	3.20	1.40	0.90	1.30	0.36	4.10	2.50	1x (0.30x1.30)	± 0.15
1206	4.80	2.30	1.25	1.70	1.25	5.90	3.20	3x (0.25x1.70)	± 0.25
1210	5.30	2.30	1.50	2.60	1.25	6.30	4.20	3x (0.25x2.60)	± 0.25

Fig. 3.1.10: Footprint design details for discrete MLCC

Reticle

A reticle is a visual overlay—such as a crosshair, grid, or other geometric shape—displayed in the camera window of a pick-and-place or inspection system. It assists operators and machines in:

- Identifying the center of an image or component for alignment.
- Measuring distances using on-screen rulers.
- Displaying arbitrary shapes or outlines that correspond to component packages or fiducials.
- Ensuring accurate calibration of the vision system by providing a fixed reference point.

Reticles can be scaled to represent actual physical dimensions, making them a valuable tool for verifying placement accuracy, package orientation, and fiducial recognition during setup and operation.

Fig. 3.1.11: Reticle

Driver

- A driver is the part of the machine software that converts operator inputs from the Graphical User Interface (GUI) into commands that the pick-and-place machine can understand.
- It acts as a translator, ensuring that high-level user instructions are converted into low-level machine code (often geometric code or G-code) that controls the hardware.

G-code

- G-code is a standardized programming language used to control computerized machine tools such as CNC equipment, 3D printers, and in some cases pick-and-place systems.
- G-code provides precise instructions to the machine controller—the industrial computer that governs motors, actuators, and vision systems.

These instructions define:

- Movement paths (X, Y, Z axes, rotation)
- Speeds and feeds (how fast a motor should move)
- Process parameters (pick height, place pressure, nozzle control, etc.)
- In essence, G-code tells the machine not just where to move, but also how to execute the movement for reliable and repeatable results.

Letters in G-code and Their Meanings:

- A – Rotation around the X-axis (rotary axis)
- B – Rotation around the Y-axis (rotary axis)
- C – Rotation around the Z-axis (rotary axis)
- D – Tool radius compensation number / cutter offset
- E – Extrusion length (commonly used in 3D printing)
- F – Feed rate (movement speed)
- G – Preparatory function (defines motion or cycle, e.g., G00 = rapid move, G01 = linear move)
- H – Tool length offset number
- I – Incremental distance for arcs in the X direction (arc center X offset)
- J – Incremental distance for arcs in the Y direction (arc center Y offset)
- K – Incremental distance for arcs in the Z direction (arc center Z offset)
- L – Loop count or number of repetitions (varies by controller)
- M – Miscellaneous function (machine commands, e.g., spindle on/off, program stop)
- N – Line (block) number in the program
- O – Program number or subroutine label
- P – Dwell time (pause), or parameter for functions like subprogram call
- Q – Peck drilling depth (or incremental value for cycles)
- R – Arc radius or return point in cycles
- S – Spindle speed (RPM)
- T – Tool number (for tool change)
- U – Incremental movement along the X-axis (used in turning/lathe operations)
- V – Incremental movement along the Y-axis (less common)
- W – Incremental movement along the Z-axis (used in turning/lathe operations)
- X – Absolute or incremental movement along the X-axis
- Y – Absolute or incremental movement along the Y-axis
- Z – Absolute or incremental movement along the Z-axis

For pick-and-place systems, the most frequently used are X, Y, Z, A, B, C (movement axes), F (speed), G (motion commands), M (misc. functions), and T (tool/nozzle selection).

Alpha numeric codes are used for programming as they are a simple way to do the following:

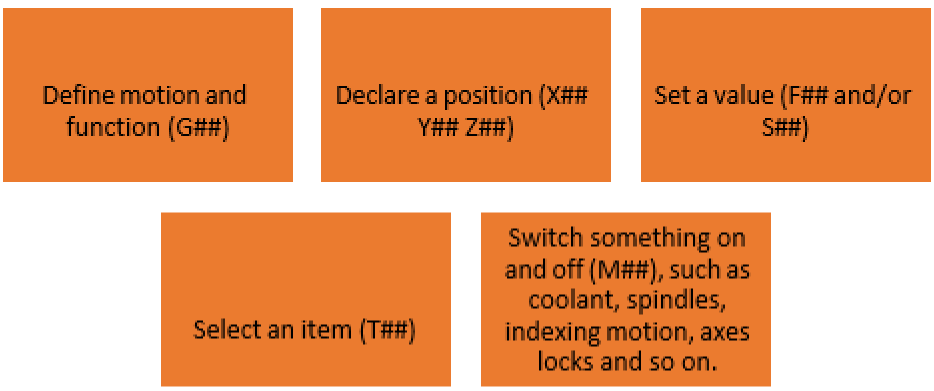


Fig. 3.1.12: Use of alphanumeric code in programming

The following figure shows an example of G-code:

G01 X1 Y1 F20 T01 M03 S500 would generally indicate:
A linear feed move (G01) to the given It is using Tool 1 and the spindle speed XY position at feed rate of 20. is 500.

Fig. 3.1.13: An example of G-code

The functions of G-codes and M-codes are not universal; they vary between different machine manufacturers. To determine what a specific M-code or G-code means and what action it will perform, you must consult the instruction manual for that particular machine.

Types of Machine Motions

All machine movements in CNC and pick-and-place systems are based on three fundamental types of motion:

1. Rapid Move (G00)
    - A linear move to a specified X, Y, or Z position at the maximum machine speed.
    - Used for positioning the tool/nozzle quickly without engaging in cutting, placing, or dispensing operations.
2. Feed Move (G01)
    - A controlled linear move to a specified position at a defined feed rate (F).
    - Used when accuracy and precision are required, such as during component placement or machining.
3. Circular Move (G02/G03)
    - A controlled move along a circular path at a defined feed rate.
    - G02: Clockwise arc.
    - G03: Counterclockwise arc.
    - Additional parameters such as I, J, and R specify arc centers and radii.

Key Programming Elements in G-code

- X, Y, Z – Cartesian coordinates that define horizontal, vertical, and depth positions.
- G (Preparatory commands) – Define the type of motion or machining cycle (e.g., G00 rapid move, G01 feed move, G02/G03 circular moves).
- F (Feed rate) – Defines the movement speed during feed or circular moves.
- S (Spindle speed) – Defines the spindle or rotational speed (in RPM).
- T (Tool selection) – Specifies the tool or nozzle to be used.
- I, J, K, R – Define arc centers or radii for circular moves (I = X offset, J = Y offset, K = Z offset, R = radius).

M-code (Miscellaneous Functions)

M-codes control all non-movement machine functions, such as:

- Starting or stopping the spindle.
- Turning on/off coolant or vacuum.
- Performing a tool change.
- Program start, pause, or end commands.

Together, G-codes (motion control) and M-codes (auxiliary functions) form the complete set of instructions that tell the machine what to do and how to do it.

M-Code relates to functions other than positioning such as speed and tool change. The following figure lists some

M00: Program stop	M01: Optional program stop	M02: End of program	M03: Spindle on clockwise
M04: Spindle on counter clockwise	M05: Spindle stop	M06: Tool change	M08: Flood coolant on
M09: Flood coolant off	M30: End of program/return to start	M41: Spindle low gear range	M42: Spindle high gear range

Fig. 3.1.14: Some M-code instructions

Bottom Vision:

Bottom vision refers to both an upwards facing camera and the process of using computer vision to automatically inspect parts before placing them. This inspection allows the SMT machine to place the part with greater accuracy than what would be achieved by going from feeder to placement directly.

The following image shows the bottom vision of a pick and place machine:

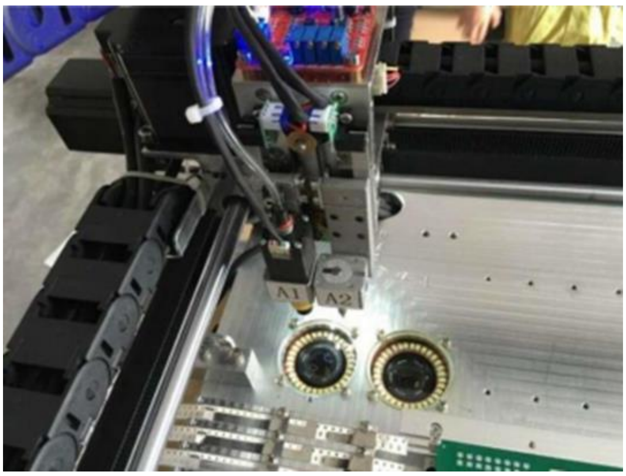


Fig. 3.1.15: Bottom vision of a pick and place machine

Top Vision

Top vision refers to the camera, facing downwards, which is mounted on the head. It is used for identifying the fiducials and for enabling computer vision on the vision assisted feeders.

Head

The head of the machine carries nozzles and often a top vision camera. It can move in both X and Y direction.

The following image shows the head of a machine:

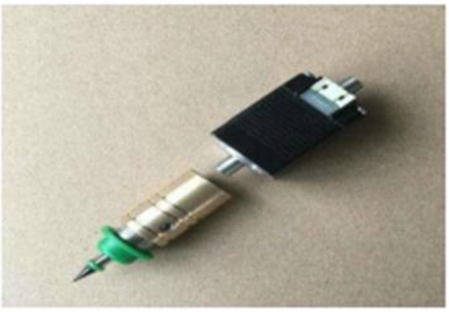


Fig. 3.1.15: Head of a machine

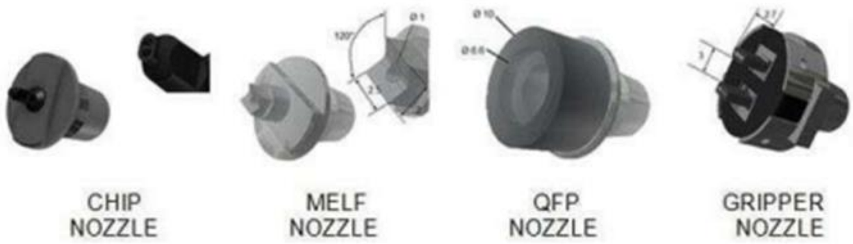


Fig. 3.1.16: Different types of nozzles

Nozzle Tip:

A nozzle tip is mounted on a nozzle. Multiple nozzle tips can be assigned to a nozzle. Generally, the tips are determined by the size of the part they are required to pick up. Many SMT machines can change nozzle tips automatically in order to pick up different parts.

Notes

UNIT 3.2: Pick and Place Operations

Unit Objectives

By the end of this unit, the participants will be able to:

1. Identify the major parts of a pick-and-place machine.
2. Demonstrate the correct procedure for starting up a pick-and-place machine.
3. Demonstrate the correct procedure for shutting down a pick-and-place machine.

3.2.1 Specifications of Pick and Place Machine

A pick-and-place machine is defined by several key specifications that determine its performance, capacity, and suitability for different production environments. The major specifications include:

1. Part Types
    - Refers to the range of components the machine can handle, from small chip resistors/capacitors (e.g., 01005, 0201) to large IC packages such as QFP, BGA, or connectors.
    - High-end machines can place ultra-fine pitch and odd-shaped components, while entry-level machines are limited to standard chip sizes.
2. PCB Loading Time
    - The time taken by the machine to load, align, and secure a PCB before beginning placement.
    - Shorter loading times contribute to higher throughput and overall productivity.
3. Placement Accuracy
    - Defines how precisely a component is positioned relative to its target solder pads.
    - Typically measured in micrometers (µm), e.g., ±30 µm at 3σ.
    - Critical for fine-pitch components such as BGAs and QFNs.
4. Productivity (Throughput in Components Per Hour – CPH)
    - Indicates the maximum number of components a machine can place per hour under ideal conditions.
    - Actual productivity may be lower depending on feeder setup, board complexity, and placement mix.
    - Modern high-speed chip shooters can exceed 100,000 CPH, while general-purpose machines range between 10,000–50,000 CPH.
5. Nozzle Quantity
    - Refers to the number of placement nozzles available on the machine head.
    - More nozzles enable simultaneous handling of multiple components, improving speed and efficiency.
    - Machines may feature single-head, multi-head, or turret-style designs with varying nozzle counts.

3.2.2 SMT Pick and Place Machine

The following figures show the parts of a pick and place machine:

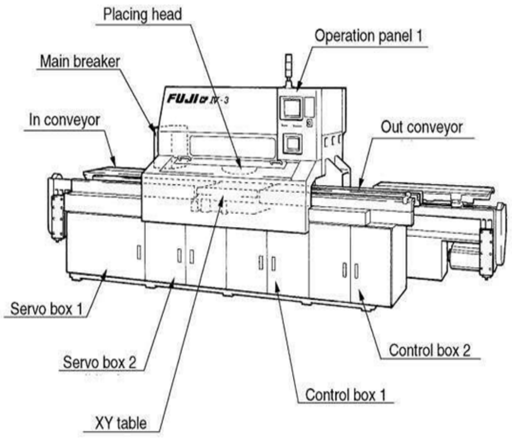


Fig. 3.2.2 (a): Parts of a pick and place machine

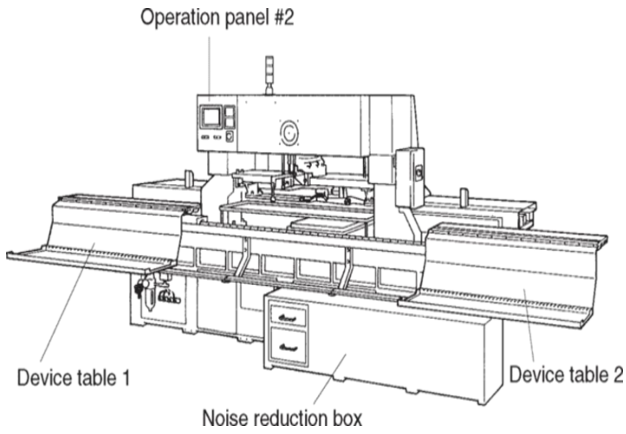


Fig. 3.2.2 (b): Parts of a pick and place machine

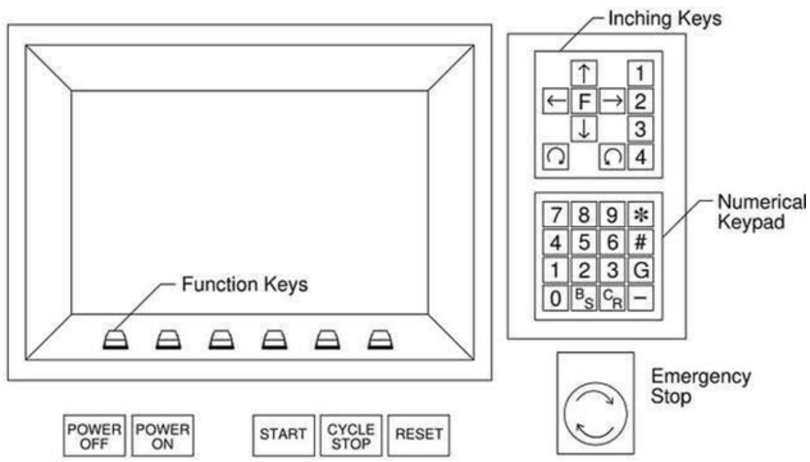


Fig. 3.2.3: Control keys of a pick and place machine

Function Keys:
The function keys (F1 to F6) are pressed to select commands. Each function key corresponds to a command in the function key menu section of the MACHINE display. To execute a command, the corresponding function key is pressed.

Inching Keys:
The inching keys are used to select the inching axes and to carry out inching along the selected axes.

Arrow Keys:
The arrow keys are used to perform the inching operation. Some arrow keys are invalid when certain inching axes are selected.

Function Keys:

The function keys (F1 to F6) are pressed to select commands. Each function key corresponds to a command in the function key menu section of the Machine display. To execute a command, the corresponding function key is pressed.

Inching Keys:

The inching keys are used to select the inching axes and to carry out inching along the selected axes.

Arrow Keys:

The arrow keys are used to perform the inching operation. Some arrow keys are invalid when certain inching axes are selected.

Inching Axis Selection Keys ([1] to [4])

These keys are used to select an axis for inching.

Rapid Inching Key ([F])

This key and an inching key are pressed simultaneously to inch the axis rapidly. Inching can be carried out at any time except during machine operation or Proper data measurement. However, it is not possible to inch all of the axes at the same time. Only those axes which are currently selected by the inching axis selection keys can be inched.

Numerical Input Keys

These keys are used to input numerical values. The following figure shows the numeric input keys:

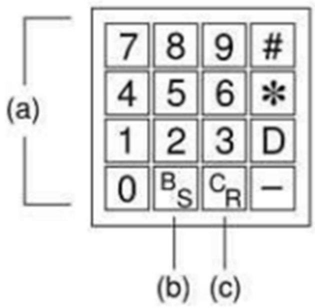


Fig. 3.2.4: Numeric input keys

The keys in the above image are as follows:

- Input keys (0 - 9, *, #, D, –): The numerical keys are used to input numerical values such as camera threshold levels or I.D. codes.
- Backspace key (BS): When inputting numerical values, this key is used to delete previously input values. Press this key once to move the cursor backwards by a single space.
- Carriage return key (CR): This key is used to complete the entry of a numerical value. It may also be pressed to take the machine out of numerical input mode.

Buttons

The machine is equipped with several pushbutton switches. The buttons are as follows:

1. Front Panel

The following figure shows the buttons on the front panel of a pick and place machine:

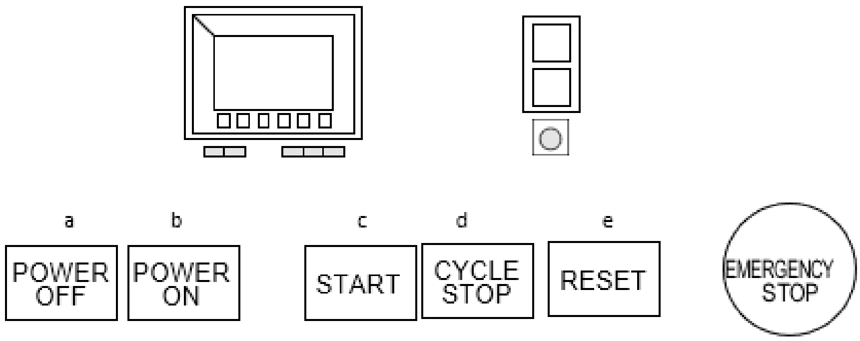


Fig. 3.2.5: Buttons on the front panel of a pick and place machine

The description of the buttons is as follows:

1. **[POWER OFF] button:** This button, when pressed after the [EMERGENCY STOP] button is pressed, turns off the power. Pressing this button alone does not turn off the power.
2. **[POWER ON] button:** This button turns on the machine. When the machine circuit breaker is turned on, this button is pressed to turn on the machine's power supply.
3. **[START] button:** This button starts the machine operation. It is pressed to start automatic operation and zero setting when the message 'Start SW' begins to blink in the second display area.
4. **[CYCLE STOP] button:** This button is for stop operation. It is pressed during automatic operation to cancel the automatic operation mode at the end of the current sequence and to return the machine to the [START] button wait mode.
5. **[RESET] button:** This button is used to reset an alarm which has occurred. When an alarm occurs, an error message is displayed and this button begins to flash. It needs to be pressed to clear the alarm.
6. **[EMERGENCY STOP] button:** This button is used to stop the machine instantly. When it is pressed, the machine's 200 V power supply is turned off. In addition, the button locks down when it is pressed. It needs to be turned in the clockwise direction of the arrow to get released.

2. Rear Operation Panel

The following figure shows the buttons on the front panel:

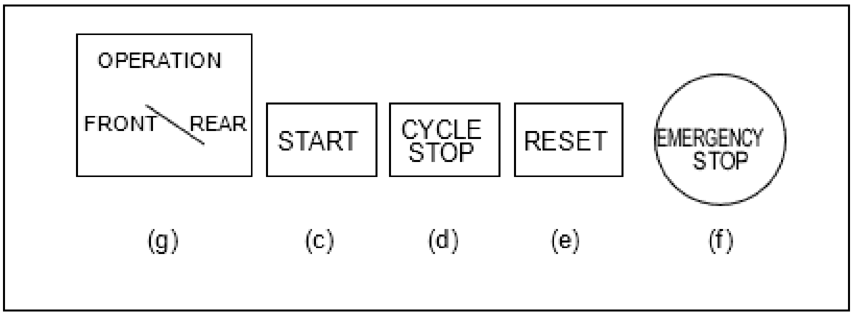


Fig. 3.2.6: Buttons on the rear panel

The description of the buttons is as follows:

1. **[POWER OFF] button:** This button, when pressed after the [EMERGENCY STOP] button is pressed, turns the power off.
2. **[POWER ON] button:** This button turns on the machine. When the machine circuit breaker is turned on, this button is pressed to turn on the machine's power supply.
3. **[START] button:** This button starts the machine operation. It is pressed to start automatic operation and zero setting when the message "Start SW" begins to blink in the second display area.
4. **[CYCLE STOP] button:** This button is used to stop operation. It is pressed during automatic operation to cancel the automatic operation mode at the end of the current sequence and to return the machine to the [START] button wait mode.
5. **[RESET] button:** This button is used to reset alarms. When an alarm occurs, an error message is displayed and this button begins to flash. It needs to be pressed to clear the alarm.
6. **[EMERGENCY STOP] button:** This button is used to stop the machine instantly. When it is pressed, the machine's 200 V power supply is turned off. In addition, the button locks down when it is pressed. It needs to be turned in the clockwise direction of the arrow to get released.
7. **Operation panel selection switch:** This selector switch is used to switch from the front operation panel to the rear operation panel. When this switch is set to a certain panel, the other panel cannot be used.

   It is to be noted that the [EMERGENCY STOP] and [CYCLE STOP] buttons on both panels are always operational.
8. **Inching keys:** It is possible to inch the X, D1, D2, C and F axes. The axes that can be inched depend on the inching axis selection keys mentioned in the previous section.

3.2.3 Operating the Pick and Place Machine

Operating the SMT pick and place machine includes its start-up, reset-start and I / O check start procedures.

Normal Operation

The following procedure is used to start the machine:

1. **Boot up the MCS 30 Supervisor Computer:** This computer compiles production information transmitted from the machine in Off Line mode.
2. **Set the Circuit Breaker:** The machine main circuit breaker must be set to ON position before the machine can be turned on.
3. **Turn on the Power:** Press the [POWER ON] button on the machine operation panel to turn the machine on. The following image shows the display that appears after pressing the [POWER ON] button:

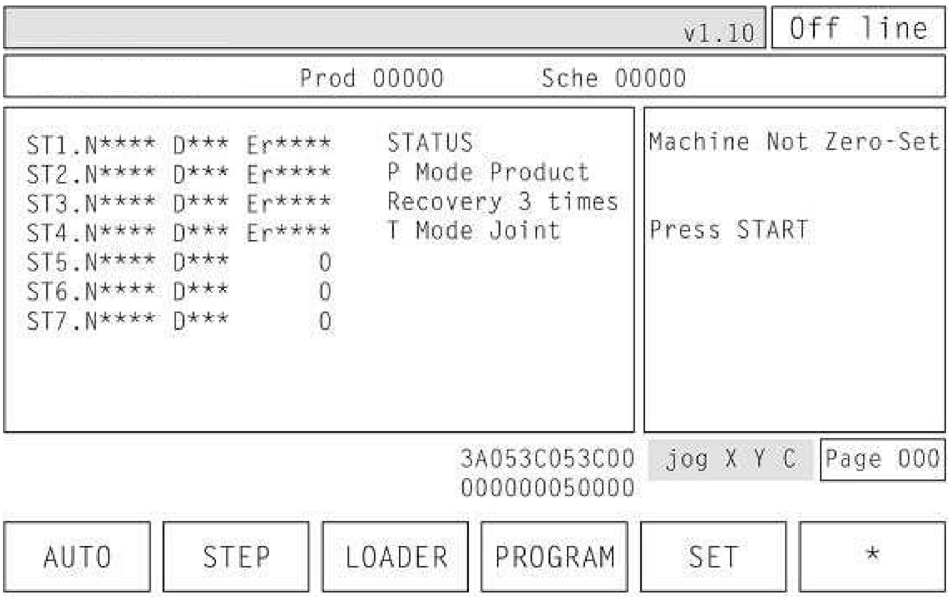


Fig. 3.2.7: Display that appears after pressing the [POWER ON] button

4. **Transmit Status Data to the Machine:** Status data is retained in the machine's memory even after the machine's power is turned off. Thus, once Status data has been transmitted to the machine, this step need not be repeated. However, it will be necessary to transmit the Status data to the machine after performing the reset-start procedure. The Status data can be modified either directly at the machine or indirectly using an MCS 30 computer. When the Status data is modified using an MCS 30 computer, it must be transmitted to the machine before operation.
5. **Transmit Proper Data to the Machine:** Proper data is retained in the machine's memory even after the machine's power is turned off. Thus, once it has been transmitted to the machine, this step need not be repeated. However, it will be necessary to transmit the Proper data to the machine after performing the reset-start procedure. When Proper data is modified at the MCS 30 computer, it must then be transmitted to the machine before operation. It is also possible to change a part of the Proper data at the machine. Once the change is made at the machine, if this data is transmitted to the MCS 30, then it will be saved on the MCS 30 computer.

6. **Transmit Programs to the Machine:** Programs are retained in the machine's memory even after the machine's power is turned off. Thus, once a program has been transmitted to the machine, this step need not be repeated. However, it will be necessary to transmit programs to the machine after performing the reset-start procedure. New programs are created using an MCS 30 computer; these must be transmitted to the machine before they can be used in production.
7. **Release the [EMERGENCY STOP] Button:** While the Machine [EMERGENCY STOP] switch is pressed, the 200V power line will be cut. If this switch is pressed, release it.
8. **Select the Production Program:** Using either MCS / 2H or the Machine [PROGRAM] command function, select the production program.
9. **Select the Quantity of the Production Run:** Press the [PROGRAM], [QTYSET] command function keys and set the number of boards to be produced. A setting of zero will make the machine run indefinitely.

   Note: Scheduled production quantity can be set in the program. Enter the production quantity into the Production_qty. field (in the Production_mode section of MCS/2H) and transmit to the MACHINE. When the production run is complete, the machine will stop automatically and the operator will be informed.
10. **Set the Operation Mode:** In either [AUTO] or [STEP], press the [MODE] command function key to change the operation mode. The operation mode changes among the three choices each time the [MODE] key is pressed. The following figure lists the three operation modes:

Product Mode	Simulate Mode	Idle Mode
Machine carries out normal production of boards.	This mode is not currently supported.	Machine carries out all functions except vision processing and fiducial mark reading. Board feed is not carried out.

Fig. 3.2.8: Three operation modes

The operation mode "Product" is automatically selected when the power is turned on.

Reset-Start

The machine memory contents have to be cleared whenever the machine's memory card and ROM chips are replaced to upgrade the software. It may also be necessary to clear the memory of erroneous data should the machine begin to operate abnormally.

All Status data, Proper data and production programs are deleted from the memory of the machine when the reset-start operation is carried out. Therefore, this data has to be transmitted from the MCS 30 to the machine again. It is important to restart the machine after transmitting the Proper data from the computer.

The following procedure explains how to reset-start the machine:

1. **Boot up the MCS 30 computer:** This computer compiles production information transmitted from the MACHINE in Off Line mode.
2. **Set the circuit breaker:** The machine's main circuit breaker must be set to ON position before the MACHINE can be turned on.
3. **Turn on the power:** While pressing the [RESET] button, the [POWER ON] button is pressed. When "Memory Backup NG" is displayed on the screen, the [RESET] button may be released.
4. **Release the [EMERGENCY STOP] button:** If the [EMERGENCY STOP] button has been pressed; the 200 V power will not come on when the operator presses [POWER ON]. It is important to release the [EMERGENCY STOP] button if it has been pressed.
5. **Transmit the Status data:** Transmit the correct Status data from the MCS 30 computer.
6. **Transmit the Proper data:** Transmit the Proper data from the MCS 30 computer to the machine. It is to be noted that the Proper data can be sent either before or after zero setting is done provided that it is transmitted before any command function keys are pressed. Valid Proper data must be transmitted to the machine. If invalid data is transmitted, the program check will not function normally and errors will occur during program transmission.
7. **Transmit the program:** Transmit the program from the MCS 30 computer to the machine.
8. **Cut the power and then restart the machine:** Even though the power has been cut, the Status data, the Proper data and the program will remain in the machine. After this, continue with normal operation.

I/O Check Start

To enter the I / O check mode, the [F1] function key is pressed and held down while pressing the [POWER ON] button. This brings the machine directly into the I / O check mode. It is to be noted that when I/O check start is used to start the machine, the 200V power does not come on. When the operator completes the I/O check and presses the [RETURN] function key, the 200V power will come on and the machine will be ready for zero setting.

3.2.4 Trace List Printout

If a problem occurs on the machine, having data from the machine that explains the conditions under which the problem occurred can help Fuji to find and eliminate the cause. A trace list is the raw data that can be obtained from the machine for this purpose.

The procedure to print out a trace list is as follows:

1. Stop the machine promptly after the trouble has occurred. Do not press the function keys or continue operating the machine as this may prevent the source of the problem from appearing in the printed trace list.
2. With the machine's power turned off, attach the printer cable from the MCS printer to the lowest of the three connectors (marked 'Parallel I / O') on the machine CPU board.

3. Press [CYCLE STOP] button along with [POWER ON] button. Release the [CYCLE STOP] button after the printer starts printing. After the printer has finished printing, the machine can be operated as normal.

3.2.5 Shutting Down the Pick and Place Machine

When shutting down the power, the following procedure should be used:

1. **Turn off the 200V power.** Press the [EMERGENCY STOP] button. When shutting down in the course of production, first press the [CYCLE STOP] button to halt automatic operation before pressing the [EMERGENCY STOP] button.
2. **Turn off the main power.** Press the [POWER OFF] button on the operation panel to turn the power supply off.
3. **Turn the circuit breaker off.** Turn off the power at the main circuit breaker if the machine is to be left idle for a longer duration or if work is to be carried out on the servo box or the control box.

It must be noted that if the power is cut while parts are being picked, the parts will drop off the nozzles and, during the next production run, vision processing will not work properly. Therefore, before shutting down the machine, press the [DUMP PARTS] command function key. The remaining parts will be placed before the board is unloaded.

Notes

UNIT 3.3: Loading of Components

Unit Objectives

By the end of this unit, the participants will be able to:

1. List the different types of feeders.
2. Demonstrate the steps to use tape feeders.
3. Identify the use of trays to load chip components.
4. Identify the use of chip shooter and fine-pitch placer.

3.3.1 Loading of Chip Component Rolls

Feeder

A feeder is a hardware attachment used to supply electronic components to an SMT pick-and-place machine. Feeders ensure a continuous and accurate flow of parts for placement.

SMT machines generally support multiple feeder formats such as:

- Cut tape strips
- Full reels (tape feeders)
- Component trays (JEDEC trays)
- Tubes (stick feeders)
- Drag-style feeders
- Bins of loose parts

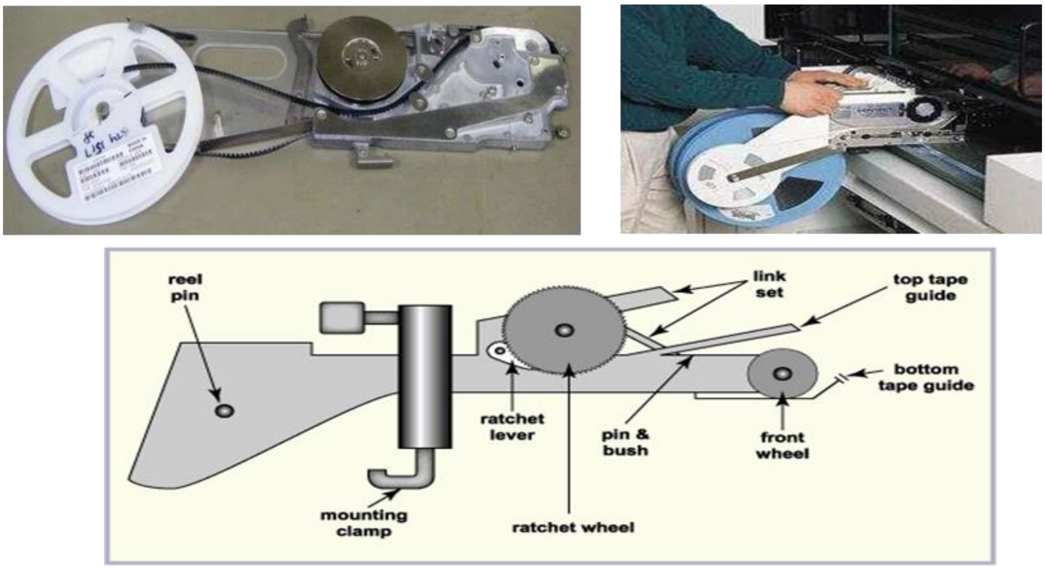


Fig. 3.3.1: Drag feeder and different parts of a drag feeder

- **Tape Feeders:**

  A tape feeder is used by performing the following two steps:

  - **Checking the feed pitch and action:** Press the manual feed lever to check that the tape is fed at a proper pitch. The following figure shows the manual feed lever of a machine:

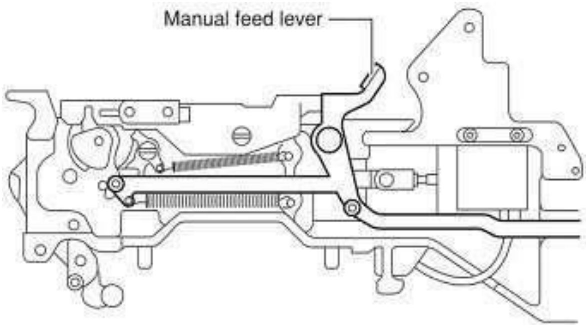


Fig. 3.3.2: Manual feed lever of a machine

  - **Setting the tape:** When fitting a new roll of tape to a tape feeder, the tape needs to be set. The procedure of using a tape feeder is as follows:

    A. Peel off the top tape:

    A tape consists of two layers; "carrier tape" that contains electronic components in the pockets and "top tape" that covers the upper side of components on the carrier tape. Peel off the top tape to separate these two layers.

    B. Lift the clamping lever lock handle:

    Lift the clamping lever lock handle to lift the tape guide.

    C. Lift the tape guide:

    Lift the tape guide while pressing the lock lever. After the tape guide is lifted, return the lock lever to the original position and hold the tape guide in the raised position. The following image shows a technician lifting a tape guide:

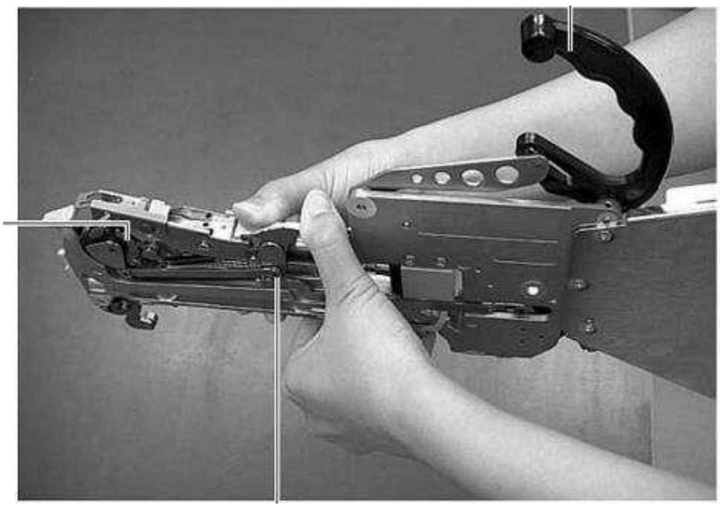
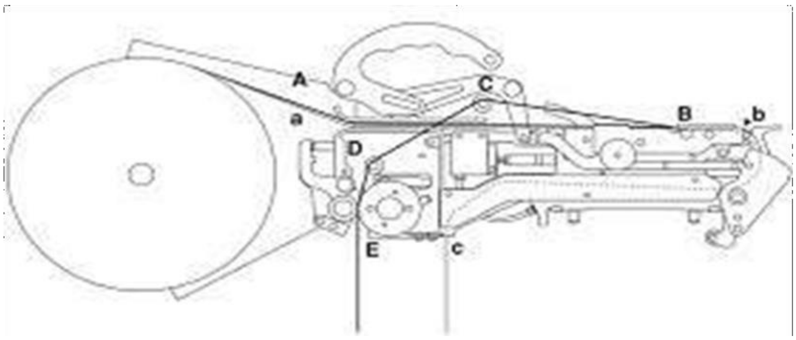


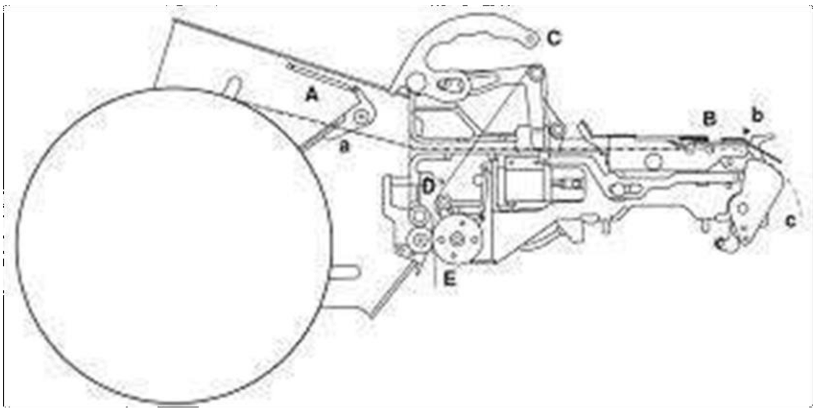
Fig. 3.3.3: A technician lifting a tape guide

D. **Set the tape in the tape feeder:**

Set the separated layers of the tape on the tape feeder through the tape paths. The following figure shows tape feeders of different measurements:



8mm tape feeder



12mm to 16mm tape feeder

Fig. 3.3.4: Tape feeders of different measurements

In the above figure, A ~ E represents the top tape and a ~ c represents the carrier tape. To turn the top tape, press the idle roller assembly lever and insert the top tape between the drive roller assembly and the idle roller assembly. The following figure shows a top tape and the notch of a tape feeder:

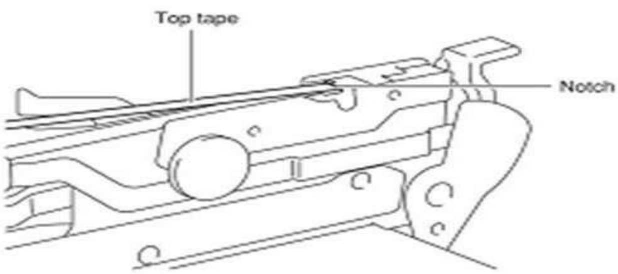


Fig. 3.3.5: Top tape and notch of a tape feeder

The following figure shows a drive roller assembly and an idle roller assembly:

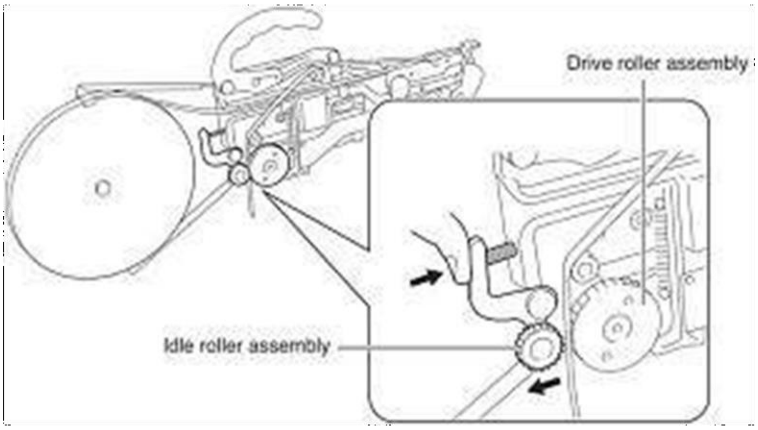


Fig. 3.3.6: Drive roller assembly and idle roller assembly

E. **Set the tape guide.**

After the tape is set, press the lock lever to lower the tape guide. The following figure shows setting the tape guide:

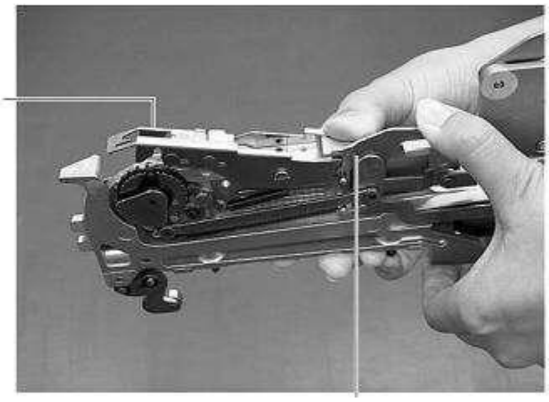


Fig. 3.3.7: Setting the tape guide

3.3.2 Loading of Chip Component Trays

Tray feeders are used to load the chip components. A tray feeder is a "virtual" feeder which picks the required parts from a uniform array of parts. The following image shows a uniform array of parts:

Fig. 3.3.8: Uniform array of parts

3.3.3 Chip Shooter and Fine Pitch Placer

A chip shooter machine in a PCB assembly has three movable mechanisms. The following figure lists the movable mechanisms:

An X-Y table carrying a PCB	A feeder carrier with several feeders holding components	A rotary turret with multiple assembly heads to pick up and place components

Fig. 3.3.9: Movable mechanisms of chip shooter

To get the minimal assembly time, all the components should be placed on the board in a perfect sequence. The components should be placed on a right feeder or feeders as two feeders can hold the components of same type. Additionally, the assembly head must pick up a component from the correct feeder. The following figure shows a chip shooter assembly:

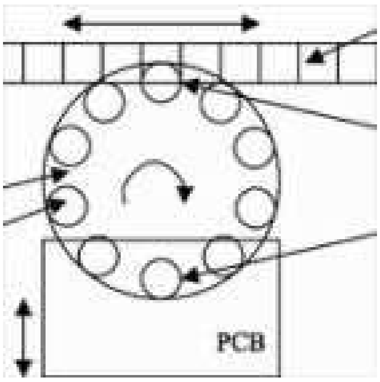


Fig. 3.3.10: Chip shooter assembly

Fine Pitch Placer

It is used when SMT components with a lead pitch of 25 mils or less are to be placed.

Exercise

Short Questions:

1. Explain the difference between global fiducials and local fiducials on a PCB.
2. Define the role of "bottom vision" in a pick-and-place machine.
3. What are the key specifications used to measure the performance of a pick-and-place machine?
4. Describe the step-by-step procedure for loading a tape feeder.
5. How does feeder placement optimization help in reducing assembly time in PCB production?

Fill in the Blanks:

1. The ______ system in pick-and-place machines ensures precise alignment of both PCBs and components.
2. The X, Y, and Z axes in a pick-and-place system control movement in ______, ______, and ______ directions, respectively.
3. The most widely used feeders in SMT assembly are ______ feeders.
4. A ______ is a visual overlay in the camera window, often in the shape of crosshairs, to aid alignment.
5. The use of ______ fiducials is recommended for optimal PCB alignment during pick-and-place operations.

Multiple Choice Questions (MCQs):

1. Which of the following is the main function of fiducial marks on a PCB?
   a) To hold the PCB in place
   b) To align the PCB during pick-and-place operations
   c) To connect components electrically
   d) To reduce soldering time
2. In a pick-and-place machine, the nozzle head functions to:
   a) Move the PCB
   b) Hold and place components using vacuum pressure
   c) Control feeder alignment
   d) Inspect solder joints
3. Which G-code command represents a rapid move?
   a) G01
   b) G02
   c) G00
   d) M06

4. The component "footprint" on a PCB defines:
    a) Solder joint quality parameters
    b) Layout of pads, land patterns, and outlines for a component
    c) Alignment of fiducials
    d) Component rotation angle only

5. A chip shooter is mainly used for:
    a) Placing fine-pitch ICs
    b) High-speed placement of standard chip components
    c) Manual soldering of components
    d) Performing AOI inspections

Notes

4. Reflow Soldering on Telecom Boards

Unit 4.1 – Reflow Soldering

Unit 4.2 – Operation of Reflow Machine

TEL/N2505

Key Learning Outcomes

By the end of this module, the participants will be able to:

1. List the factors that determine an effective reflow soldering process.
2. Identify the factors that influence the reflow profile.
3. Define the concept of footprint design.
4. List the different types of reflow soldering ovens.
5. Describe the stages of the reflow soldering process.
6. Identify the alarm messages generated during reflow soldering.
7. Analyse the significance of the N_2 conservation mode.
8. Identify common defects occurring in the reflow soldering process.

UNIT 4.1: Reflow Soldering

Unit Objectives

By the end of this unit, the participants will be able to:

1. List the key factors of an effective reflow soldering process.
2. Identify the factors that affect the reflow profile.
3. Define footprint design in the context of PCB assembly.
4. List the different types of reflow soldering ovens.

4.1.1 Reflow Soldering Process

Reflow soldering is the most common method for creating permanent electrical connections between surface-mount components and a PCB. The process involves temporarily attaching components to their pads with solder paste, a sticky substance made of solder and flux. The entire assembly is then exposed to a controlled heating profile, which melts the solder paste to form a secure and permanent solder joint.

The entire process is performed inside a specialized machine called a reflow oven, which precisely controls the temperature to ensure a successful soldering process without damaging the components.

Key Aspects for an Effective Reflow Soldering Process

Achieving a high-quality reflow soldering process depends on controlling several key aspects. Each factor contributes to creating strong, reliable solder joints and preventing defects.

1. Suitable Machine

The reflow oven is the heart of the process. An effective machine must have precise and repeatable temperature control across its heating zones. It should be able to accurately follow the programmed reflow profile, which dictates the temperature changes over time.

2. Acceptable Reflow Profile

The reflow profile is a time-temperature graph that guides the entire soldering process. It consists of four main stages:

- Pre-heat: The PCB and components are gradually heated to prevent thermal shock.
- Soak: The assembly is held at a consistent temperature to activate the flux and evaporate volatile solvents from the solder paste.
- Reflow: The temperature rises above the solder paste's melting point, allowing the solder to liquefy, wet the pads, and form the joint.
- Cooling: The solder solidifies, creating a permanent connection.

A correctly designed profile is essential for preventing defects like solder balls, tombstoning, and component damage.

Fig. 4.1.1: A reflow soldering oven

1. PCB and Component Footprint Design

The physical design of the PCB pads and component footprints directly impacts solder joint quality. Pads must be correctly sized and spaced to ensure the solder paste can form a proper joint without causing shorts. Non-symmetrical pads can lead to defects like tombstoning, where a component stands up on one end.

2. Carefully Printed PCB

The quality of the solder paste print is critical, as it determines the amount and position of solder on each pad. This requires a well-designed stencil and a precisely calibrated screen-printing machine. Defects such as insufficient paste, bridging, or slumping at this stage will likely result in soldering failures.

3. Repeatable Placement

Accurate and consistent placement of surface-mount components by the pick and place machine is vital. Components must be placed precisely on the solder paste deposits. Misalignment can lead to poor electrical connections or shorts during the reflow stage.

4. Good Quality Materials

The quality of the materials used—the PCB, components, and solder paste—is fundamental. A high-quality PCB with a clean surface and proper finish ensures good solderability. Components must be free of damage or oxidation. Finally, using fresh, properly stored solder paste with the correct composition is essential for a successful reflow process.

4.1.2 Suitable Machine

Various types of reflow soldering machines are available, with selection depending on the PCB design, materials, and required production line speed. The oven must be appropriately sized to handle the output of the pick and place machine.

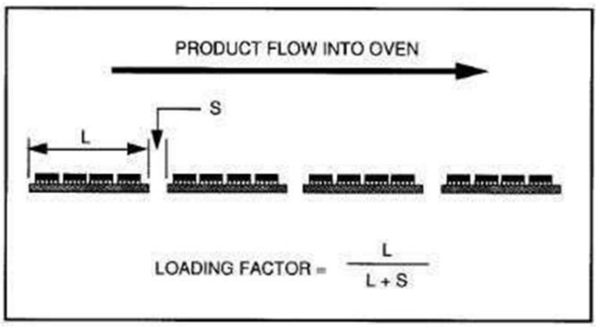


Fig. 4.1.2: Calculation of load factor

Calculating Reflow Oven Size

To select a reflow oven of the correct size, the oven's process speed must be greater than the minimum required line speed.

1. Calculate Minimum Line Speed

The minimum line speed is the rate at which boards must move through the oven to keep up with production.

Line Speed (minimum)=Load FactorBoards per Minute×Board Length

- Boards per Minute: The production rate of the pick and place equipment.
- Board Length: The length of a single PCB.
- Load Factor: A value, typically provided by the manufacturer, that accounts for the necessary space between boards on the conveyor for process repeatability.

2. Calculate Required Oven Heated Length

The process speed of an oven is a function of its heated length and the total time the board needs to spend inside (process dwell time). By rearranging this relationship, we can determine the minimum heated length required.

Process Speed=Process Dwell TimeOven Heated Length

Therefore:

Oven Heated Length=Process Speed×Process Dwell Time

Example Calculation

An SMT assembler needs to produce 8-inch boards at a rate of 180 per hour. The solder paste manufacturer recommends a 4-minute profile.

- Boards per Minute: 180 boards/hour÷60 min/hour=3 boards/min
- Board Length: 8 inches
- Load Factor: 0.8 (assuming a 2-inch space between boards, as an example)
- Process Dwell Time: 4 minutes

First, calculate the required minimum line speed:

Line Speed=0.8(3 boards/min)×(8 inches/board)=30 inches/minute

Next, determine the required oven heated length:

Oven Heated Length=30 inches/min×4 minutes=120 inches

Therefore, the reflow oven must have a minimum heated length of 120 inches (10 feet) to meet the production requirements. It's important to note that the total length of the oven, including the cooling and loading sections, will be greater than the heated length.

Common Reflow Oven Options

The selection of a reflow oven and its features is heavily influenced by the PCB assembly's design.

- Conveyor Type:
- Mesh Conveyor: A flat mesh belt suitable for single-sided boards.
- Edge Conveyor: A system that holds the PCB by its edges. This is typically preferred for in-line production and for processing double-sided assemblies without disturbing components on the bottom side. A center-board-support is often used with edge conveyors to prevent the board from sagging during the high-temperature reflow process.

- Closed-Loop Control of Convection Fans: Convection fans circulate heated air within the oven. For assemblies with lightweight components (e.g., small packages like SOD-323) that are susceptible to being blown out of position, a closed-loop control system is recommended. This feature automatically adjusts fan speed to minimize disturbances.
- Automatic Control of Conveyor and Center-Board-Support Widths: This option is highly beneficial for manufacturing lines that process a variety of PCBs with different widths. Instead of manual adjustment, the conveyor and center-board-support widths are automatically configured based on the assembly program, ensuring process consistency and reducing changeover time.

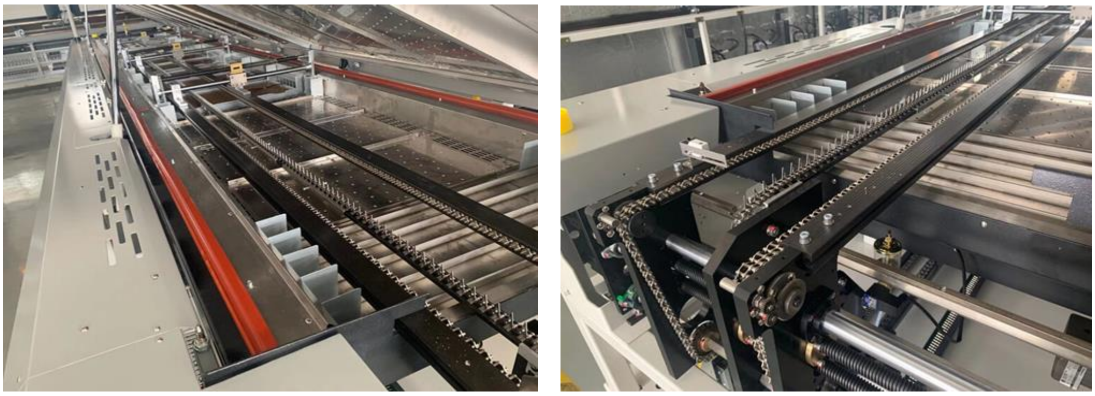


Fig. 4.1.3: Edge conveyor with a centre-board support

4.1.3 Acceptable Reflow Profile

An acceptable reflow profile is a time-temperature graph that guides the soldering process to ensure strong, reliable solder joints without damaging components. Because no two assemblies are exactly alike, a unique reflow profile must be developed for each one. The profile is programmed into the reflow oven and is affected by several critical factors:

- Type of Solder Paste: Different solder pastes have varying melting points and require specific temperature ramps and soak times. The profile must be tailored to the paste's properties to ensure proper wetting and joint formation.
- PCB Material, Thickness, and Number of Layers: PCBs with thicker material, more layers, or a higher copper content require more energy to heat up and cool down. The reflow profile must be adjusted to account for the thermal mass of the board.
- Amount of Copper: A higher density of copper planes and traces within the PCB increases its thermal mass. This requires a longer pre-heat stage to bring the entire board to a uniform temperature, preventing cold spots and uneven soldering.
- Number and Type of Surface Mount Components: The size, type, and density of components on the board significantly affect the thermal profile. Large or thermally massive components (e.g., BGAs, large capacitors) absorb more heat, requiring a longer soak time to reach the desired temperature. Smaller, more delicate components may be sensitive to rapid temperature changes, necessitating a gentler ramp rate.

4.1.4 PCB/Component Footprint Design

A PCB design can influence the efficiency of the assembly reflow in many ways. The size of tracks that connects to a component footprint is an example. If the track that is joined to one side of a footprint is bigger than that joined to the other, a thermal imbalance can happen. This leads to 'tombstone' formation.

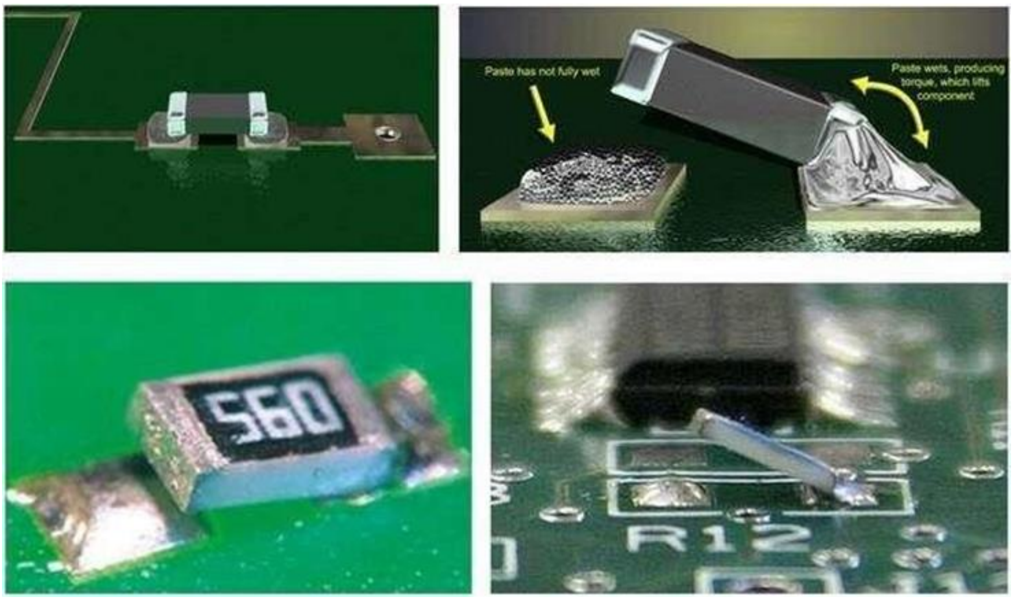


Fig. 4.1.4: Tombstone formation

Copper balancing is a critical consideration in reflow soldering, especially for panels. This technique is used to prevent the panel from warping as it goes through the high-temperature reflow oven.

When a PCB design has large, uneven copper areas, it creates a thermal imbalance. The copper-heavy sections heat up and cool down at a different rate than the areas with little or no copper. This differential thermal expansion and contraction can cause the entire panel to warp, leading to soldering defects such as tombstoning, shorts, and delamination.

To counteract this, the "copper balancing" technique involves adding a pattern of copper to the empty, non-functional (waste) areas of the panel. This helps to create a more even distribution of thermal mass across the entire panel, ensuring that it heats and cools more uniformly. The result is a flatter panel throughout the reflow process, which significantly improves the quality and yield of the finished PCBs.

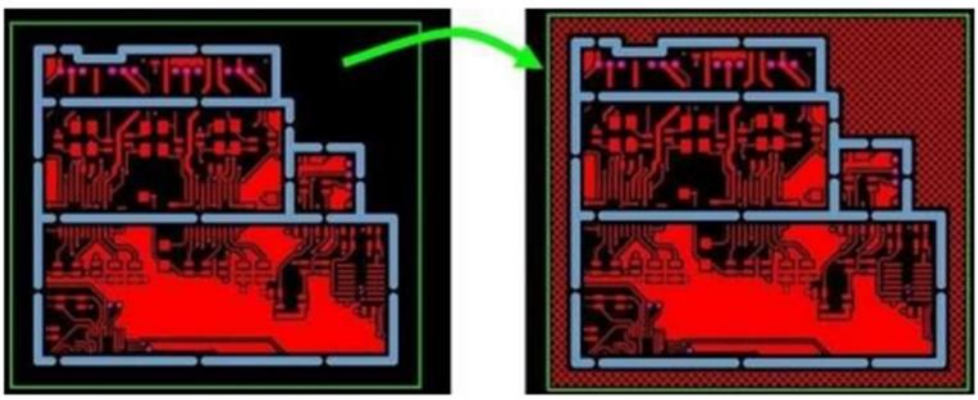


Fig. 4.1.5: Copper balancing process

4.1.5 Well-Designed Stencils and Printed PCBs

For an effective reflow soldering process, the quality of the solder paste printing is paramount. As discussed in earlier modules, the solder paste printing process is the first critical step in SMT assembly and directly impacts the final product quality. Any defects introduced at this stage—such as insufficient paste, slumping, or bridging—will lead to unacceptable results after reflow.

Therefore, ensuring a consistent and accurate solder paste deposit requires two key elements:

1. A well-designed stencil: The stencil must have precision-cut apertures that perfectly match the pads on the PCB.
2. Complete process control: The stencil printing machine must be correctly calibrated, including optimal squeegee speed, pressure, and stencil separation speed.

Controlling these factors ensures a clean, uniform solder paste deposit, which is a prerequisite for a successful and reliable soldering process.

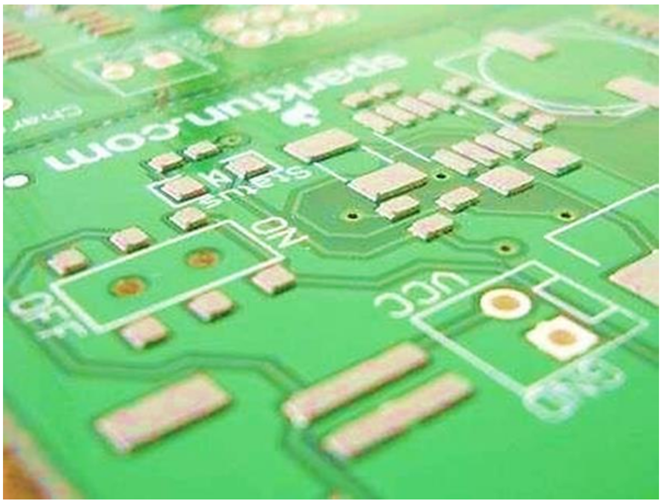


Fig. 4.1.6: A PCB after the printing process

4.1.6 Repeatable Placement of SMT Components

Repeatable and accurate component placement is crucial for a successful reflow soldering process. This is achieved using a well-maintained, reliable pick and place machine.

A key aspect of a machine's performance is its vision system, which must be able to consistently and accurately identify different component packages. If the vision system fails to recognize parts correctly, it can lead to inconsistent placement and a high risk of defects after reflow.

All professional pick and place machines have a defined "Placement Accuracy" specification, which indicates the precision of component placement. This is typically measured in micrometers (µm) at a 3-sigma (3σ) level. For example, an accuracy of 35µm for Quad Flat Packages (QFPs) or 60µm for smaller chip components at 3σ means that 99.7% of all placed components will be within that specified tolerance. This ensures that every component is placed precisely, leading to consistent and reliable solder joints.

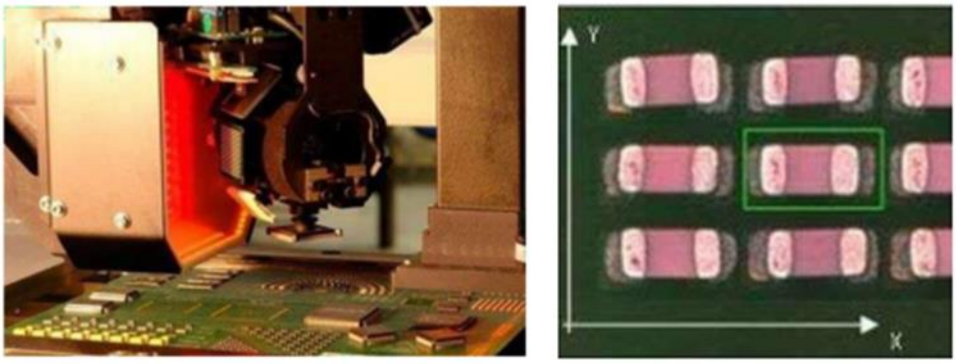


Fig. 4.1.7: Component placement variation

It is also imperative to choose the appropriate nozzle for the component type. The following image shows a range of nozzles for different component placement:

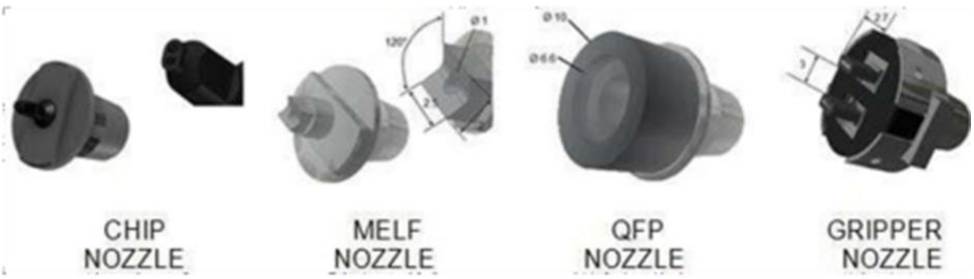


Fig. 4.1.8: A range of nozzles for different component placement

4.1.7 Good Quality PCB, Components, and Solder Paste

Using high-quality materials is fundamental to achieving a reliable and high-yield reflow soldering process. Even with a perfectly calibrated machine and a well-designed profile, poor-quality inputs will inevitably lead to defects.

- PCBs: The quality of the PCB is paramount. A poor surface finish, which can result from inaccurate manufacturing or improper storage, can lead to issues with solderability. Contaminants, oxidation, or moisture on the pads will prevent the solder from wetting the surface correctly, resulting in weak or incomplete joints.
- Components: Components must be free of defects and properly stored. Oxidized leads, for example, will not form a proper bond with the solder. Moisture-sensitive components must be handled and baked according to manufacturer specifications to prevent internal cracking or "popcorning" during the high heat of reflow.
- Solder Paste: Solder paste must be stored correctly, used within its shelf life, and properly prepared (thawed and mixed) before printing. Using old, improperly stored, or poorly mixed solder paste can lead to defects such as slumping, solder balls, and incomplete reflow.

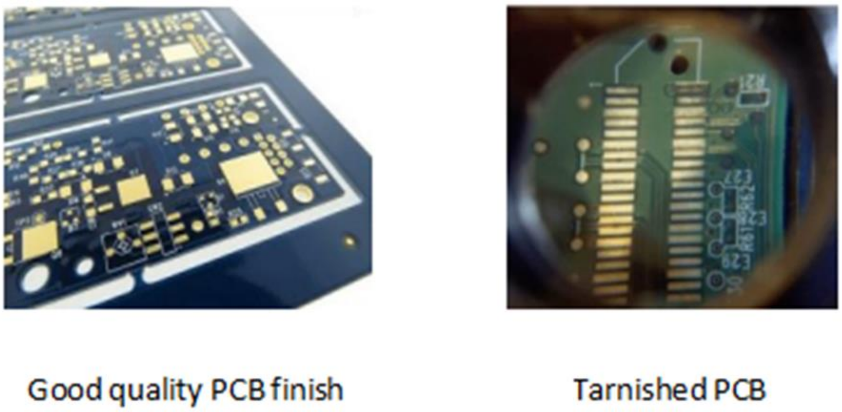


Fig. 4.1.9: An example of poor surface finishing of PCB

Also, depending on the manufacturing process and storage method, the quality of the leads of the surface mount component can be poor.

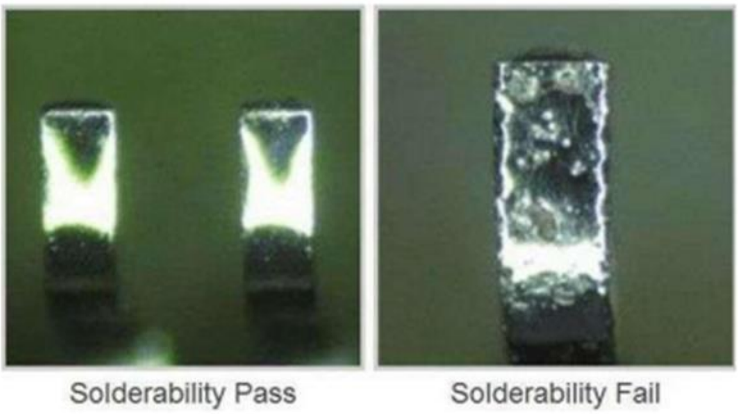


Fig. 4.1.10: Poor component leads with solderability

The storage process and the handling of the solder paste largely affect its quality. The following images show the result of using poor quality solder paste:

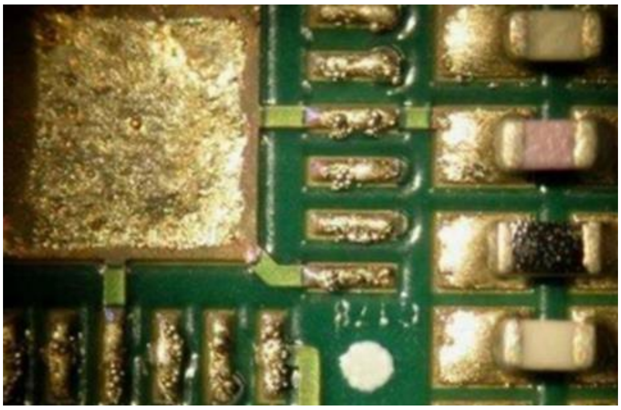


Fig. 4.1.11: Result of using poor quality solder paste

4.1.8 Types of Reflow Soldering Ovens

Two primary types of reflow soldering ovens are used in SMT manufacturing, distinguished by their method of heat transfer.

1. Infrared (IR) and Convection Type

This is the most common type of reflow oven used today. It combines two methods of heat transfer to achieve a precise reflow profile:

- Infrared (IR) Radiation: Heat is transferred directly from heating elements above and below the conveyor to the PCB and components.
- Forced Convection: Fans circulate hot air throughout the oven's chamber. This ensures even heat distribution across the entire board, preventing cold spots and providing a more uniform temperature profile. The forced convection element makes this method highly efficient and controllable.

2. Vapor Phase Type

Vapor phase reflow ovens use a different, more controlled heating process.

- Heating Medium: A non-flammable, inert liquid (typically a perfluoropolyether, such as Galden®) is heated to its boiling point, creating a saturated vapor.
- Soldering Process: The PCB assembly is lowered into this hot vapor layer. The vapor condenses on the cooler surfaces of the board and components, releasing latent heat of vaporization. This rapidly and uniformly heats the assembly to the boiling point of the fluid, which is fixed and stable. This prevents the board from overheating.

This method provides excellent thermal uniformity and is particularly useful for complex or high-density boards, as it eliminates the risk of component overheating or thermal shock.

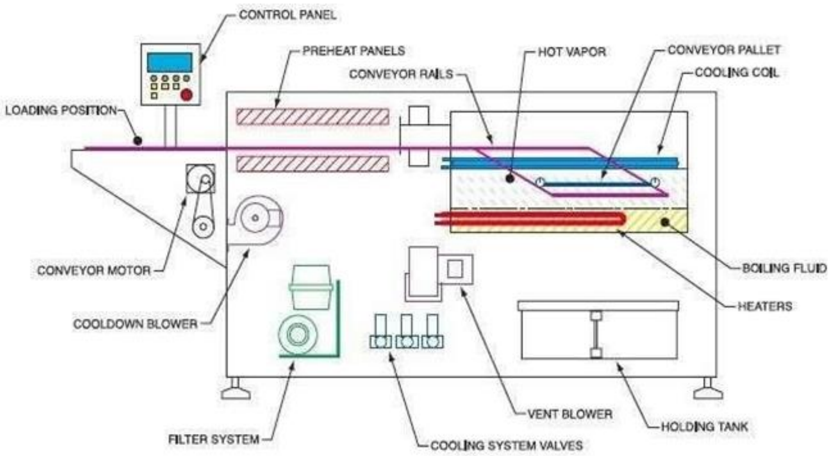


Fig. 4.1.12: Batch vapour phase reflow

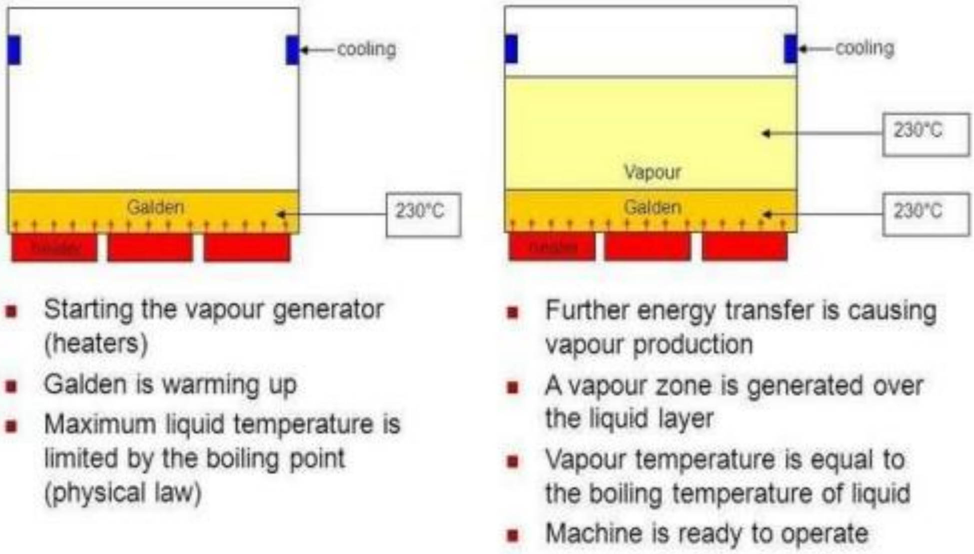


Fig. 4.1.13: Vapour phase soldering process

Broadband refers to high-speed, reliable, and "always-on" internet access, which is foundational for the services offered by CSCs. An Internet Service Provider (ISP) is the organization that provides this internet access to the CSCs and their customers.

Broadband technologies have advanced significantly, with Fiber to the Home (FTTH) becoming increasingly common. The BharatNet project by the Government of India aims to connect all Gram Panchayats with high-speed fiber-optic connectivity, which is critical for the success and expansion of the CSC network, ensuring seamless delivery of digital services even in the most remote parts of the country.

Notes

UNIT 4.2: Operation of Reflow Machine

Unit Objectives

By the end of this unit, the participants will be able to:

1. List and describe the four stages of a typical reflow soldering profile.
2. Identify common alarm messages that may be received during the reflow soldering process.
3. Explain the significance of the nitrogen (N2) conservation mode in reflow ovens.
4. Recognize common defects that occur during reflow soldering and their causes.

4.2.1 Stages of Reflow Soldering Profile

A reflow profile is a time-temperature graph that dictates how a PCB assembly is heated inside a reflow oven. To create and verify a profile, thermocouples are attached to a sample assembly at several locations. A high-temperature solder is typically used to secure the thermocouples to pads on the PCB to ensure an accurate measurement of the temperature range across the board, known as Delta T (ΔT). A minimum of one thermocouple should be placed on a pad near the edge and one near the center of the board to capture the temperature difference. Additional thermocouples can be used to measure ΔT on specific components or large ground planes.

A typical reflow soldering profile consists of four distinct stages, each serving a specific purpose.

1. Preheat

The preheat stage is the initial phase where the PCB and all its components are gradually heated. The temperature rises at a controlled rate, typically between 1-3°C per second. This stage prevents thermal shock to the components and activates the flux in the solder paste.

2. Soak

During the soak stage, the temperature is held at a consistent level, just below the melting point of the solder. This stage allows the entire assembly to reach a uniform temperature (minimizing ΔT) and gives the flux time to clean the pads and component leads of oxides. It also allows volatile solvents in the solder paste to evaporate.

3. Reflow

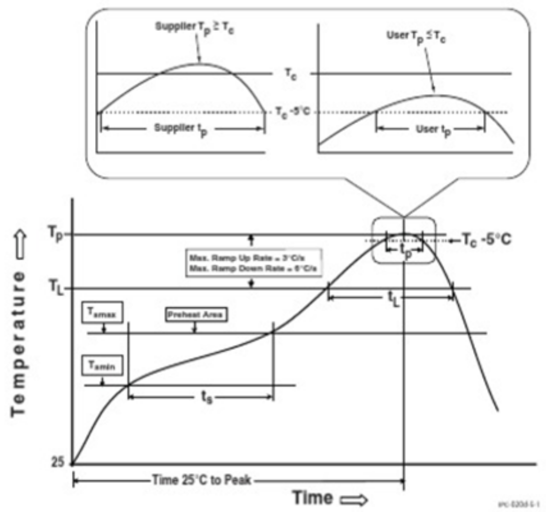
The reflow stage is when the temperature rises above the solder's melting point. The solder becomes liquid, or "reflows," creating a strong metallurgical bond between the component leads and the PCB pads. The peak temperature is reached in this stage and is held for a short period to ensure a complete joint.

4. Cooling

In the final cooling stage, the PCB is rapidly cooled to solidify the solder joints. A fast cooling rate is generally preferred as it creates a finer grain structure in the solder, resulting in a stronger, more reliable joint.

Refer below reflow profile guidelines as per J-STD-020:

Reflow Soldering Profile, Per J-STD-020D, Table 5-2, Pb-Free Devices



Profile Feature	Sn-Pb Eutectic Assembly	Pb-Free Assembly
Preheat/Soak Temperature Min (T_{smin}) Temperature Max (T_{smax}) Time (t_s) from (T_{smin} to T_{smax})	 100°C 150°C 60-120 seconds	 150°C 200°C 60-120 seconds
Ramp-up rate (T_L to T_p)	3°C/second max.	3°C/second max.
Liquidous temperature (T_L) Time (t_L) maintained above T_L	183°C 60-150 seconds	217°C 60-150 seconds
Time (t_p)* within 5°C of the specified classification temperature (T_c)**, see Reflow Soldering Profile above.	20* seconds	30* seconds
Ramp-down rate (T_p to T_L)	6°C/second max.	6°C/second max.
Time 25°C to peak temperature	6 minutes max.	8 minutes max.

* Tolerance for peak profile temperature (Tp) is defined as a supplier minimum and a user maximum.

Fig. 4.2.1: Time-temperature profile of reflow soldering process

The primary purpose of these stages is to transfer the required heat into the assembly so that it can melt the solder and form the solder joints. However, this should be done without damaging the components or the PCB.

Preheat Stage

The preheat stage is the initial phase of the reflow soldering process, where the entire PCB assembly is gradually heated from ambient temperature to a target pre-reflow or soak temperature. The primary goal of this stage is to bring the assembly to a stable temperature in a controlled manner, preventing thermal shock to the components.

During this stage, the volatile solvents in the solder paste have time to evaporate and escape. It's crucial that the board is heated consistently and linearly to allow for proper outgassing. If the temperature rises too quickly, the solvents can outgas violently, causing the paste to splatter and form solder balls or other defects.

The rate of temperature increase is measured as the temperature slope rate, in degrees Celsius per second (∘C/s). This metric is a key control point for the preheat stage. The optimal slope rate depends on several factors:

- Component Sensitivity: Many components, particularly ceramic capacitors and large ICs, are sensitive to rapid temperature changes and can crack if the slope rate is too high. The maximum allowable slope rate is determined by the most sensitive component on the board.
- Solder Paste Volatility: Solder pastes with highly volatile solvents require a gentler, more controlled heating slope to prevent violent outgassing.
- Production Throughput: For assemblies without thermally sensitive components, manufacturers may use an aggressive slope rate, sometimes up to 3.0∘C/s, to reduce processing time and maximize throughput.

Once the board reaches the predetermined temperature at the end of the preheat stage, it is ready to enter the next phase: the soak.

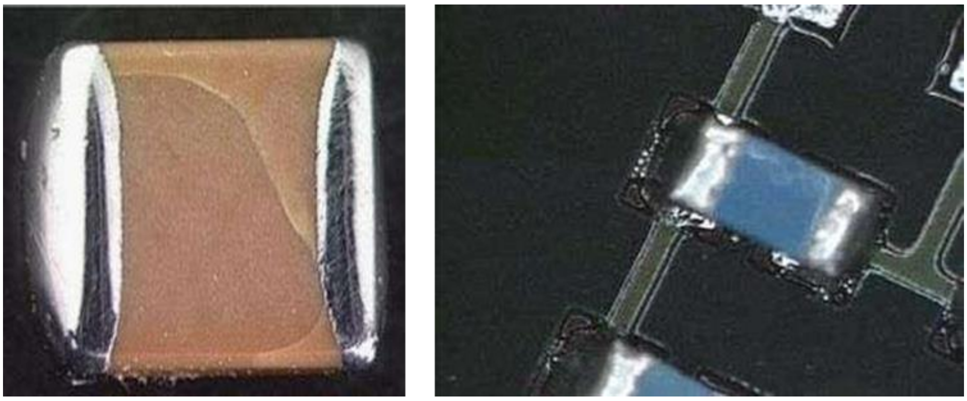


Fig. 4.2.2: Impact of too fast heating of the assemblies

Soak Stage

The soak stage is a crucial phase in the reflow soldering process where the PCB assembly's temperature is held in a plateau just below the solder's melting point. The main purpose is to bring the entire assembly to a uniform temperature, allowing for several key processes to occur before reflow.

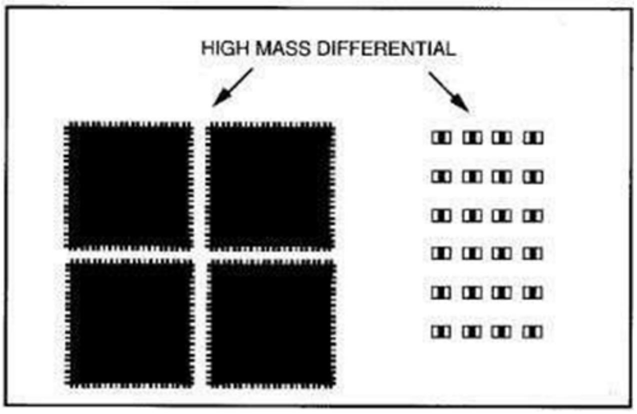


Fig. 4.2.3: High mass differential

Thermal Equilibrium: The soak period allows components of different sizes and thermal masses to reach a consistent temperature. This minimizes the Delta T (ΔT), the temperature difference across the board, which is essential for uniform reflow and preventing defects.

- Flux Activation: During this time, the flux in the solder paste becomes fully active. It chemically cleans the oxide layers from the component leads and the PCB pads, preparing the surfaces for a strong metallurgical bond.
- Volatiles Removal: The controlled, stable heat ensures that any remaining volatile solvents from the solder paste evaporate safely without causing spattering.

The soak stage typically lasts for 60 to 120 seconds, though this can be adjusted based on the board's thermal mass and component types. A proper soak ensures the entire assembly is ready for a uniform and rapid transition into the reflow stage, minimizing defects like voiding in area array packages (e.g., BGAs) and preventing solder spattering or oxidation from overly aggressive heating.

Key Characteristics of a Soak Profile

- Temperature Control: If the soak temperature is too high, it can lead to solder spattering and premature oxidation of the paste, pads, and component terminations. Conversely, if the temperature is too low, the flux may not fully activate, leading to poor wetting and weak solder joints.
- Purpose: The soak profile is highly recommended for large assemblies or those with a wide variety of component sizes to ensure thermal equilibrium and reduce ΔT.
- Void Reduction: A well-managed soak profile is effective in minimizing voids—small air pockets that can form within solder joints, especially under area array packages like BGAs, which can compromise reliability.

Reflow Stage

The reflow stage is the most critical phase of the soldering process, where the temperature inside the oven is raised above the solder paste's melting point. This causes the paste to transition from a solid to a liquid, creating the final metallurgical bond between the component leads and the PCB pads.

Key Parameters

- Time Above Liquidus (TAL): This is the duration for which the solder remains in its molten state. A sufficient TAL is crucial to allow proper wetting, where the molten solder flows and adheres to the pads and component leads. A commonly recommended TAL is between 30 to 60 seconds. If the TAL is too short, the solder may not fully reflow, leading to cold joints or voids. If it is too long, it can cause excessive intermetallic growth, which can make the solder joints brittle, or damage heat-sensitive components. The flux may also be consumed prematurely.
- Peak Temperature: This is the maximum temperature the PCB assembly reaches during the entire process. It is typically set 20-40°C above the liquidus temperature of the solder paste. This peak temperature is determined by the component on the board that has the lowest tolerance for high temperatures, often a delicate integrated circuit. As a safety measure, the peak temperature is often set at least 5°C below the maximum temperature that the most vulnerable component can withstand. Excessive temperatures (above 260°C) can damage component dies and cause a rapid, potentially harmful, growth of intermetallics.

Wetting and Flux Action

During reflow, the flux within the solder paste performs a vital function. It reduces the surface tension at the metal-to-metal interface and cleans the pads and leads of oxides, allowing the individual solder spheres to coalesce into a single, uniform solder joint.

Role of Nitrogen (N2)

In modern electronics manufacturing, nitrogen (N2) is often used in reflow ovens. The inert atmosphere of nitrogen displaces oxygen, preventing oxidation of the pads, leads, and solder paste during the heating process. This is particularly important for newer, more environmentally friendly solder pastes that may use less active fluxes. The absence of oxygen ensures that the surfaces remain clean, allowing the solder to wet properly and form a strong joint.

Cooling Stage

The cooling stage is the final phase of the reflow soldering process, where the temperature of the PCB assembly is brought down from the peak reflow temperature to a level safe for handling, typically between 30°C and 100°C.

The primary purpose of this stage is to solidify the molten solder joints and create a strong, reliable metallurgical bond. A proper cooling rate is essential to prevent two key issues:

- Excessive Intermetallic Growth: Solder joints can become brittle if they cool too slowly, as this allows for the growth of a thick intermetallic layer between the solder and the pads.
- Thermal Shock: While the most dramatic temperature changes happen during preheat and reflow, a sudden, uncontrolled drop in temperature can still stress components, particularly large ones.

A fast cooling rate, typically around 4°C/s, is recommended to produce a finer grain structure in the solder, which results in a more mechanically sound and durable joint. The maximum allowable temperature slope should be respected during both heating and cooling to protect sensitive components. The cooling rate is an important metric to analyze when evaluating the quality of the reflow process.

4.2.2 Alarm Messages during Reflow Soldering

The following table lists some examples of alarm messages:

ALARM	DESCRIPTION
EMERGENCY STOP	E-stop switch pushed (or electrical cabinet open)
CONVEYOR STOPPED	The conveyor is not moving
COMMUNICATIONS FAILURES -WITH CONTROLLER -PORT TO CONTROLLER COULD NOT BE OPENED -CONTROLLER GIVES ERROR	 The oven controller is not functioning / communicating The oven controller is not functioning / communicating Non specific communications error
BOARD DROPPED	An expected board did not arrive at the off load sensor
OIL LEVEL LOW	The oil level is low in the Lubrication Reservoir
COOLANT FLOW	The coolant flow low or stopped. (or tank level low)
COOLANT TEMP	The enhanced cool coolant temp is too high
GAS PRESSURE LOW	The gas pressure is below switch setpoint and too low to maintain the gas atmosphere in the tunnel
CONVECTION LOW	A cell fan(s) has stopped, Inverter failure, or exhaust sensing detects low facilities exhaust flow
EXHAUST FAILURE	The facilities exhaust is below specifications
RAIL WIDTH	Rail width position is outside the acceptable tolerance
BOARD JAM	A circuit board is under the exit sensor for a time period that exceeds the jam length in the product file
IAS	The IAS Unit detected a temperature alarm condition, or a High Temperature Switch has opened
SYSTEM READY TIME OUT	The countdown timer has expired and the machine has not reached the Process Ready state
UPS BATTERY LOW	The UPS battery is low an requires charging or replacement

Fig. 4.2.4: Some examples of alarm messages

4.2.3 N_2 Conservation (Optional)

The N_2 conservation mode is designed to minimize nitrogen consumption during idle periods of the reflow soldering oven. It allows the user to set a timer so that the main N_2 supply is automatically switched off when no boards are being processed. During this mode, a stand-by N_2 valve remains active, supplying a controlled amount of nitrogen through a regulator. This ensures the oven atmosphere remains partially protected while conserving N_2.

The objective of this mode is to optimize N_2 usage and reduce operating costs without compromising board quality.

Settings Parameters (found in Configuration → Atmosphere tab):

- Time-out [min]: Duration the oven must remain empty before conservation mode activates.

If set to 0, the feature is inactive:

- Fans [rpm]: Fan speed during conservation mode.
- Purge [min]: Duration of the N_2 purge cycle when switching back to normal operation.

Principle of Operation

The system relies on three sensors:

- S1 (Upstream sensor): Positioned before the oven load point.
- Detects approaching boards early enough to exit conservation mode and restore O_2 ppm levels before processing.
- Placement must minimize board queueing in front of the oven.
- S2 (Entrance sensor): Detects boards entering the oven.
- S3 (Exit sensor): Detects boards leaving the oven.

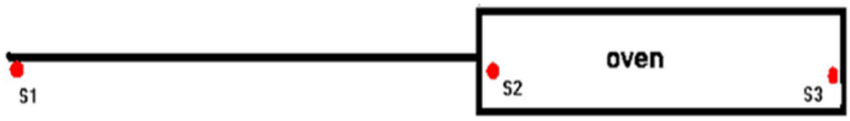


Fig. 4.2.5: Principle of operation in the conservation mode

Normal Startup with N_2 Recipe

- Main N_2 and purge valves are activated.
- Purge duration is determined by recipe settings.
- Once purge and stabilization are complete, the oven enters Ready status and allows board entry.

When Time-out Expires (No boards detected between S1 and the oven):

- Main N_2 supply shuts off.
- Stand-by N_2 supply is activated.
- “N_2 Conservation” indication appears on the system PC.
- Yellow lamp on the tower light turns on.
- Board entry is restricted.

When a Board is Detected at S1:

- Stand-by N_2 shuts off.
- Main N_2 supply reactivates.
- Re-purge cycle starts.
- Oven stabilizes atmosphere (O_2 ppm).
- Board entry resumes only when oven is ready.

Notes:

- If a board remains between S1 and S2, the system latches the event as “N_2 Conservation Inhibited” (displayed on the screen).
- This latch clears only when the oven is completely empty.
- The N_2 conservation time-out must always be set longer than the board travel time from S1 to S2, ensuring smooth transition without interruptions.

4.2.4 Common Defects During Reflow Soldering

Reflow soldering is a complex process, and slight deviations in material quality or process parameters can lead to a variety of defects. Understanding these common issues is the first step to preventing them.

1. Solder Balling

Solder balling is the formation of small, unwanted solder spheres near the intended solder joint. This happens when the solder paste doesn't properly melt and coalesce into a single mass.

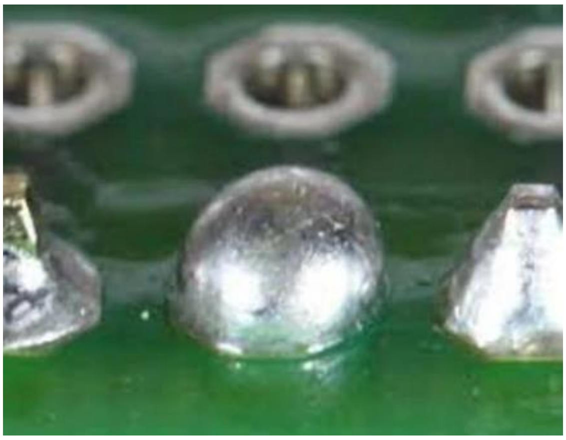


Fig 4.2.6: Solder Balling

Causes:

- Volatile Solvents: The primary cause is the rapid expulsion of solvents from the solder paste during the preheat stage. If the temperature ramp is too steep, the solvents boil and splatter the solder powder, which then melts and forms tiny balls.
- Oxidized Paste: If the solder paste is old or improperly stored, the metal powder can become oxidized. The flux in the paste may not be active enough to clean this oxidation, preventing the powder from fusing together.
- Wettability Issues: Poor solderability of the PCB pads or component leads due to contamination or oxidation can also cause the solder to bead up instead of flowing smoothly.

Prevention:

- Follow the solder paste manufacturer's recommended reflow profile, especially the preheat ramp rate.
- Ensure proper storage and handling of solder paste (FIFO, refrigeration, and thawing).
- Use high-quality PCBs and components to ensure good solderability.

1. Solder Beads

Solder beads are similar to solder balls but are typically larger and form a continuous ring around the edge of a component.

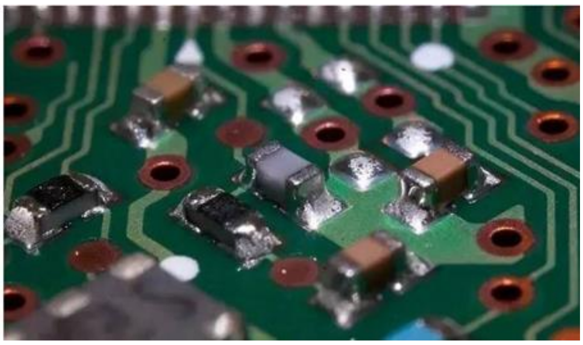


Fig 4.2.7: Solder Beads

Causes:

- Excessive Paste: This is often a stencil printing issue where too much solder paste is deposited on the pads.
- Component Placement: Excessive downward pressure from the pick and place machine can squeeze out paste from underneath the component.
- Paste Slump: Low-viscosity solder paste will spread excessively after printing, causing it to flow beyond the pad boundaries and form a bead during reflow.

Prevention:

- Optimize stencil aperture design and size to control paste volume.
- Calibrate the pick and place machine's placement pressure.
- Store and handle solder paste correctly to maintain its viscosity.

1. Poor Wetting

Poor wetting occurs when the molten solder fails to spread and adhere to the intended metallic surfaces. The solder remains as a ball or lump, creating a weak or non-existent electrical connection.

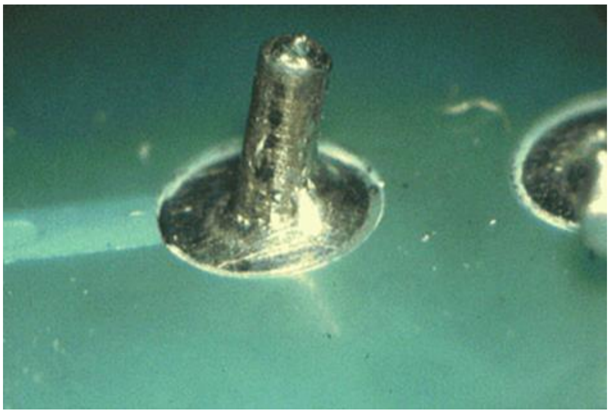


Fig 4.2.8: Poor Wettings

Causes:

- Oxidation: The most common reason is the presence of an oxide layer on the PCB pads or component leads. If the flux is not active enough or has been consumed prematurely, it cannot remove the oxidation.
- Insufficient Flux Activation: The reflow profile's soak or reflow stage temperature may be too low, preventing the flux from activating and cleaning the surfaces.
- Contamination: Residue, moisture, or other contaminants on the pads can act as a barrier to wetting.

Prevention:

- Use fresh, properly handled solder paste with active flux.
- Ensure the reflow profile meets the paste and component requirements for proper flux activation.
- Maintain a clean manufacturing environment and ensure PCBs are properly stored.

1. Open Solder Joints

An open solder joint is a complete failure to form an electrical connection. The solder may be attached to the pad but not the component lead, or vice versa.

Causes:

- Component Coplanarity: This is a major issue with large, multi-leaded components like QFPs and BGA. If the component's leads are not perfectly flat, some leads will not make contact with the solder paste.
- Insufficient Paste: An inadequate volume of solder paste may prevent the solder from bridging the gap between the lead and the pad.
- Poor Solderability: As with poor wetting, contaminated or oxidized surfaces can prevent the solder from bonding to one side of the joint.

Prevention:

- Inspect components for lead coplanarity before placement.
- Ensure the stencil prints the correct volume of paste.
- Control the reflow profile to ensure all surfaces reach the reflow temperature.

2. Short Circuits (Bridges)

A short circuit, or solder bridge, is an unwanted electrical connection between adjacent pads or traces.

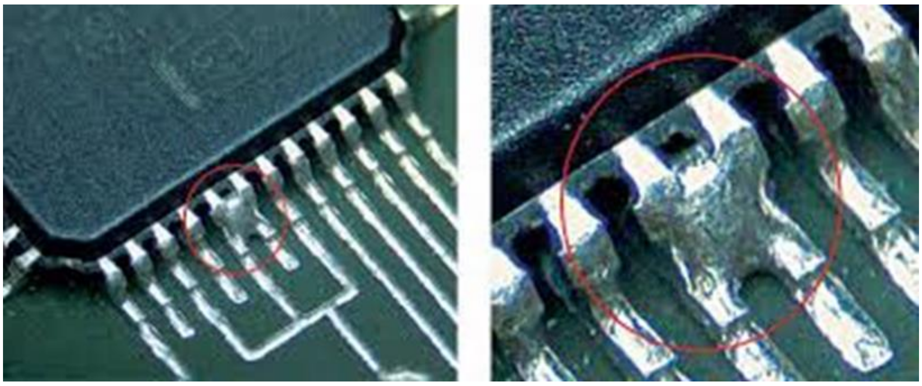


Fig 4.2.8: Poor Wettings

Causes:

- Excessive Paste: This is the most common cause. Too much paste on the pad can lead to it spreading and connecting to a neighboring pad.
- Stencil Issues: A dirty stencil or a mismatch between the stencil and the board can cause paste smearing.
- Paste Slump: If the paste viscosity is too low, it can spread and bridge the gap between pads before or during reflow.

Prevention:

- Optimize stencil aperture size to reduce paste volume.
- Ensure the stencil and PCB are perfectly aligned during printing.
- Control the preheat stage of the reflow profile to prevent excessive paste slump.

3. Tombstoning (Manhattan Effect)

This is a defect where a small, two-terminal component (like a chip resistor or capacitor) stands up on one end after reflow, resembling a tombstone.

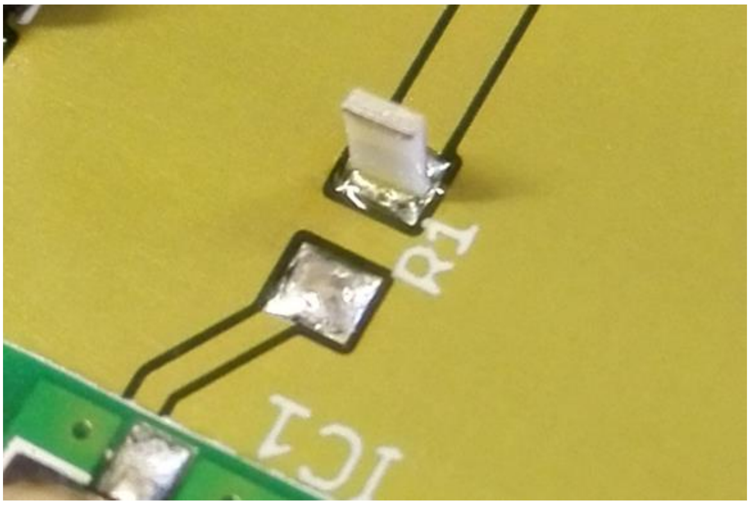


Fig 4.2.9: Tombstoning

Causes:

- Thermal Imbalance: The primary cause is uneven heating between the two pads. If one pad reflows before the other, the surface tension of the molten solder on the first pad pulls the component upright. This often happens with an aggressive reflow profile or uneven air flow.
- Uneven Paste Volume: A significant difference in the amount of solder paste on the two pads can also lead to an imbalance in surface tension.

Prevention:

- Use a gentler reflow profile with a longer soak stage to ensure thermal equilibrium.
- Ensure uniform solder paste printing on both pads.
- Optimize the air flow and heating elements of the reflow oven.

3. Wicking

Wicking is the phenomenon where molten solder travels up a component's lead away from the pad, leaving an insufficient amount of solder to form a proper joint.

Causes:

- Thermal Mass Difference: This occurs when the component lead heats up faster than the PCB pad. The molten solder is attracted to the hotter surface and climbs up the lead. This is common with components that have high thermal mass leads (e.g., J-leaded packages).

Prevention:

- Adjust the reflow profile to minimize the temperature difference between the component leads and the PCB pads.
- Use a solder paste with a suitable activity level to ensure the pad and lead are cleaned at the right moment.

Exercise

Short Questions:

1. Define footprint design in the context of PCB assembly and explain its significance in reflow soldering.
2. List any three factors that influence the reflow profile.
3. Differentiate between infrared + convection ovens and vapor phase ovens used in reflow soldering.
4. Explain the term time above liquidus (TAL) and why it is important in the reflow stage.
5. Identify any two common defects in reflow soldering and briefly state their causes.

Fill in the Blanks:

1. The ___________ is the heart of the reflow soldering process, ensuring controlled heating.
2. The stage in which the assembly is held at a constant temperature to activate flux is called the ___________ stage.
3. Poor-quality solder paste can lead to defects such as ___________ and ___________.
4. To prevent warping in PCBs during reflow, the ___________ balancing technique is used.
5. ___________ conveyors are preferred for double-sided assemblies as they hold the PCB by its edges.

Multiple Choice Questions (MCQs):

1. Reflow soldering is most commonly used to:
   - a) Connect through-hole components to a PCB
   - b) Create permanent connections between SMT components and a PCB
   - c) Remove defective solder joints
   - d) Clean the PCB surface
2. Which of the following is not a stage in a typical reflow profile?
   - a) Pre-heat
   - b) Soak
   - c) Electroplating
   - d) Cooling
3. Tombstoning of a component usually occurs due to:
   - a) Excess solder paste on the pad
   - b) Thermal imbalance between pads
   - c) Use of poor-quality stencil
   - d) Slow cooling rate

4. A well-designed stencil is important because:
    a) It prevents oxidation of solder paste
    b) It controls the amount and placement of solder paste
    c) It reduces oven length requirement
    d) It helps in cooling the PCB
5. Which type of reflow oven uses vapor condensation for heating?
    a) Convection oven
    b) Mesh conveyor oven
    c) Vapor phase oven
    d) Infrared oven

Notes

5. Cleaning and Inspection of Telecom Boards

Unit 5.1 – Cleaning of PCBs
Unit 5.2 – Inspection of PCBs

Key Learning Outcomes

By the end of this module, the participants will be able to:

1. Identify the different types and sources of contamination in PCBs.
2. Describe various methods of PCB cleaning and their applications.
3. Explain the causes and impact of white residue on PCB performance and reliability.
4. Demonstrate correct procedures for storing and handling PCBs after cleaning.
5. List and explain key points of importance in visual inspection of PCBs.
6. Identify the attributes, capabilities, and limitations of Automated Optical Inspection (AOI).
7. List the applications and advantages of Automated X-ray Inspection (AXI).
8. Apply in-circuit testing (ICT) methods to verify assembly integrity.
9. Compare different types of visual inspection systems and their suitability for specific applications.

UNIT 5.1: Cleaning of PCBs

Unit Objectives

By the end of this unit, the participants will be able to:

1. Identify different types and sources of contamination in PCBs.
2. List the common methods used for PCB cleaning.
3. Explain the impact of white residue on PCB quality and reliability.
4. Demonstrate correct methods for storage and handling of PCBs after cleaning.

5.1.1 Waste IPC Standards of Soldering

PCB Cleaning and Flux Residue

PCB cleaning refers to the process of removing solder flux residues and other contaminants from the printed circuit board (PCB) after the surface-mount technology (SMT) process. During reflow soldering, the flux in the solder paste reacts with metal oxides to enable proper wetting of solder and prevent further oxidation of solder joints. However, this reaction produces flux residues, which often remain trapped beneath components, solder joints, or the undersides of solder balls.

To remove these residues, cleaning is performed using aqueous or semi-aqueous solvents, often combined with external agitation (e.g., spray, ultrasonic, or mechanical methods). Effective cleaning is critical because flux residues can cause reliability issues such as electromigration.

Electromigration Mechanism

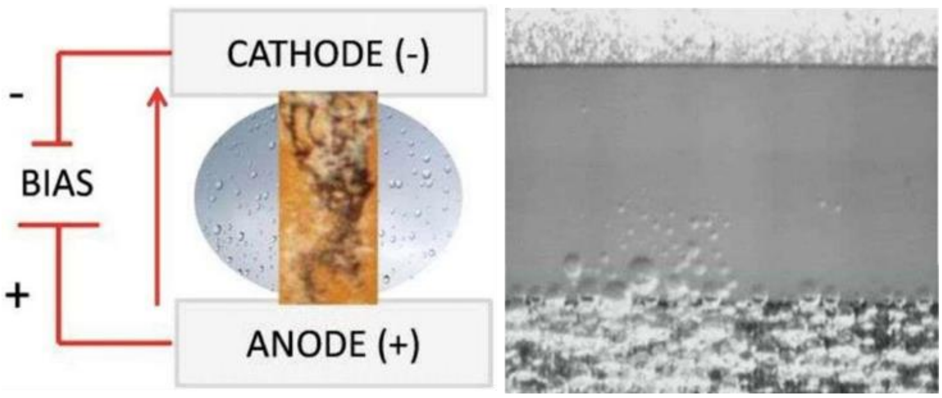


Fig. 5.1.1: Electromigration process

Residual flux, in the presence of humidity and an applied electric field, can initiate an electrochemical process known as electromigration. In this process:

- A thin film of moisture mixed with flux residue forms a conductive path between two adjacent conductors.
- At the positively biased anode, metal ions are generated and begin migrating toward the negatively charged cathode.
- Over time, these ions accumulate as metallic dendrites between conductors.
- The dendritic growth reduces the isolation distance and can eventually create a metal bridge.
- This conductive bridge may lead to short circuits or solder bridging, compromising the functionality and long-term reliability of the PCB assembly.

Importance of Cleaning

Proper PCB cleaning prevents dendritic growth, reduces the risk of electrical shorts, and ensures long-term reliability of electronic assemblies—particularly for high-density, fine-pitch, or safety-critical applications.

Introduction to IPC

IPC (Institute for Printed Circuits), officially known as the Association Connecting Electronics Industries, is a globally recognized trade association that sets standards for printed circuit boards (PCBs) and electronic assemblies. With over 4000 member organizations worldwide, IPC standards are applied across diverse sectors such as advanced microelectronics, aerospace, defense, automotive, computing, medical equipment, industrial systems, and telecommunications.

These standards are essential to ensure that PCB-related products are safe, reliable, consistent, and high-performing. They emphasize maintaining quality, focus, and adherence throughout the entire manufacturing and assembly process.

Importance of IPC Standards

Adopting IPC standards in PCB manufacturing and soldering provides several benefits:

- Ensures high-quality and reliable end products
- Improves communication across employees, vendors, and suppliers
- Helps in cost reduction through consistency and error prevention
- Enhances reputation and opens up new opportunities
- Standardizes terminology and practices, improving global trade and collaboration

Key IPC Terminology

Some important IPC terms relevant to PCB manufacturing and soldering include:

- Acceptance Tests – Verify if the product meets buyer or vendor requirements.
- Assembly – The process of joining and combining parts into a working product.
- Resist – A protective coating material used during plating, etching, or soldering.
- IC (Integrated Circuit) – Multiple electronic circuits integrated into a single unit.
- Flexural Strength – The ability of a material to bend or fold without breaking.
- Critical Operation – A process step or product feature essential for performance.

IPC Standards for Soldering

For soldering processes, the most widely followed IPC standard is:

- IPC J-STD-001 – Requirements for Soldered Electrical and Electronic Assemblies
- Covers materials, methods, and verification criteria for producing reliable soldered connections.
- Used globally for certification and training.
- IPC-A-620 – Requirements and Acceptance for Cable and Wire Harness Assemblies
- Provides guidelines for solder splices, including four approved methods:
  - Mesh
  - Wrap
  - Hook
  - Lap
- Each method is clearly defined with diagrams and acceptance/rejection criteria.

These standards also detail the correct mounting of components such as DIP pins, SIP pins, and sockets.

Critical Requirements in IPC J-STD-001 for Soldering

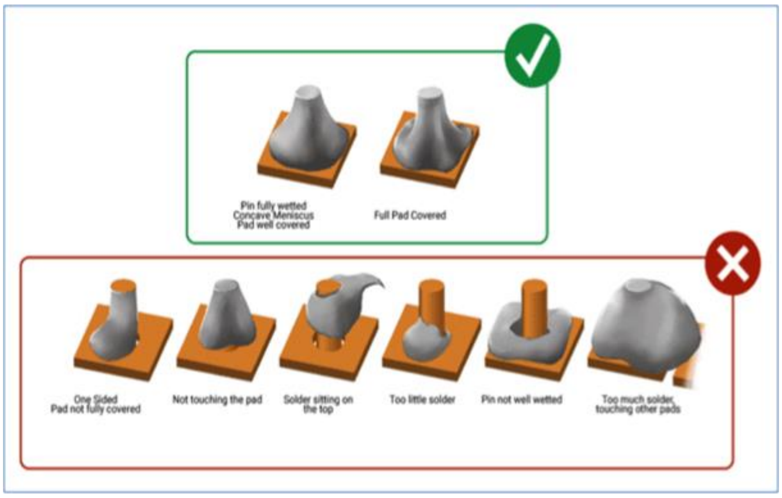


Fig. 5.1.2 Soldering according to J-STD-001 for Soldering

When soldering PCBs and assemblies, IPC emphasizes:

- Cleanliness – Tools, materials, and surfaces must be free of contaminants.
- Thermal Management – Heating and cooling rates should match manufacturer specifications to prevent thermal shock (especially in multilayer chip capacitors).
- Wire Handling – Strands should remain intact; the solder must fully wet the tinned portion of wires.
- Inspection – Cleanliness and solder joints should be inspected prior to applying conformal coatings or stacking.
- Defect Management – Any defect affecting form, fit, or function must be reworked or rejected based on customer requirements.
- Inspection Tools – Use Automated Optical Inspection (AOI) and Automated X-ray Inspection (AXI) for reliable quality checks.
- Exposed Base Metal – May be acceptable depending on conductor design, component leads, land patterns, and solder mask allowances.

Additional Considerations

IPC J-STD-001 also specifies requirements related to:

- Wire preparation and tinning
- Lead forming and shaping
- Material-class-specific flaws
- Plated through-holes and lamination
- Documentation and record-keeping of inspection outcomes

5.1.2 Types of Contamination in PCBs

Contaminants on a PCB can cause various issues, from cosmetic blemishes to severe electrical failures. These substances can interfere with soldering, lead to corrosion, or create unwanted electrical paths. They are broadly categorized as either ionic or non-ionic.

- Ionic Contamination: These are residues that can dissociate into ions in the presence of water, making the surface electrically conductive. Examples include:
- Salts: From fluxes and handling (e.g., salts from fingerprints).
- Flux Activators: Chemicals in flux that are meant to clean surfaces but can become corrosive if not properly removed.
- Non-Ionic Contamination: These residues do not conduct electricity. Examples include:
- Natural and Synthetic Rosins: The solid part of flux that remains after soldering.
- Oils and Greases: From machinery, human contact, or the manufacturing process.
- Solder Balls: Tiny spheres of solder that can create shorts.
- Fingerprints: A mix of both ionic (salts) and non-ionic (oils) contaminants.
- Particles: Dust, dirt, and fragments from the boards themselves.

5.1.3 Types of Cleaning Solvents

Choosing the right solvent is critical for effective cleaning. Solvents are selected based on the type of contamination (ionic or non-ionic) and the manufacturing process.

- Aqueous Solutions: Water-based cleaners. These are effective at removing ionic contaminants like salts and flux activators. They often contain additives like surfactants (to reduce surface tension) and saponifiers (to dissolve rosin-based residues).
- Semi-Aqueous Solutions: These are typically hydrocarbon-based solvents with added surfactants. They are very effective at dissolving non-ionic residues like rosins, oils, and greases. A water rinse is required after using these solvents.
- Hydrocarbon-based Solvents: Including petroleum distillates, alcohols, and ketones. These are flammable but highly effective at dissolving non-ionic contamination.
- Halogenated Solvents: Including brominated or fluorinated solvents (like HCFCs and HFCs). These are known for being non-flammable and very effective at dissolving a wide range of contaminants. However, their use is often regulated due to environmental concerns.

5.1.4 Board Cleaning Methods

Several methods are used to clean PCBs, ranging from simple manual techniques to complex automated systems.

1. Manual Cleaning Method

This method is suitable for low-volume production or for specific rework tasks. It is highly labor-intensive and can be inconsistent.

- Soaking: The board is soaked in a cleaning solvent (e.g., acetone or a specialized cleaning solution) for a few minutes to loosen contaminants.
- Brushing: The solder joints and contaminated areas are manually brushed with a stiff brush while submerged in a cleaning solution (e.g., ethanol) to physically remove residues.
- Rinsing: The board is thoroughly rinsed with deionized (DI) water to remove all remaining cleaning solution and ionic contaminants.
- Dehydration and Drying: The board is dehydrated with a solvent like absolute ethyl alcohol and then dried with a nitrogen gas gun to prevent water spots and corrosion.

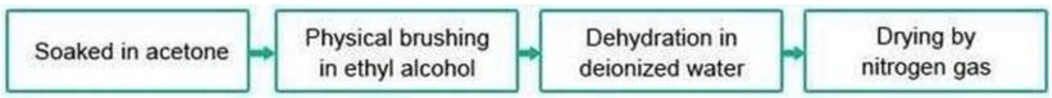


Fig. 5.1.3 Manual cleaning of a PCB

2. Ultrasonic Cleaning Method

This method uses high-frequency sound waves to create microscopic bubbles (cavitation) in a liquid, which effectively scrubs contaminants from the board's surface. It is more efficient than manual cleaning but can be damaging to some sensitive components.

- Soaking and Placement: The board is placed in a special basket and submerged in a solvent (e.g., ethyl alcohol or a dedicated ultrasonic solvent) inside a tank.
- Ultrasonic Action: An ultrasonic generator is activated for a set period (e.g., 5 minutes). The cavitation action removes contaminants from even hard-to-reach areas.
- Rinsing and Drying: The board is then rinsed with DI water, dehydrated with alcohol, and dried with a nitrogen gas gun.

Note: This method is not recommended for components like crystal oscillators, as the high-frequency vibrations can damage their internal structure.

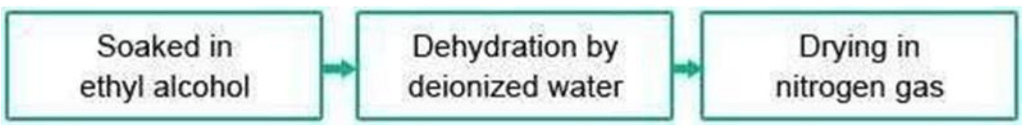


Fig. 5.1.4 Ultrasonic cleaning of a PCB

3. Vapor Degreasing

This is an automated, high-efficiency cleaning method that uses the condensation of a heated solvent vapor to clean parts.

- Vapor Generation: A solvent is heated in a sump at the bottom of the unit, creating a dense, hot vapor.
- Cleaning: The workpiece is lowered into this vapor. As the vapor touches the cooler surface of the part, it condenses, releasing heat and dissolving the contaminants. The contaminated solvent drips back into the sump.
- Purification: Since the contaminants typically have a higher boiling point than the solvent, the vapor remains pure, ensuring that the parts are always cleaned with a fresh, uncontaminated solvent.
- Drying: The parts dry rapidly upon removal from the vapor. A spray stage can be added to the process for parts with blind holes or stubborn contaminants.

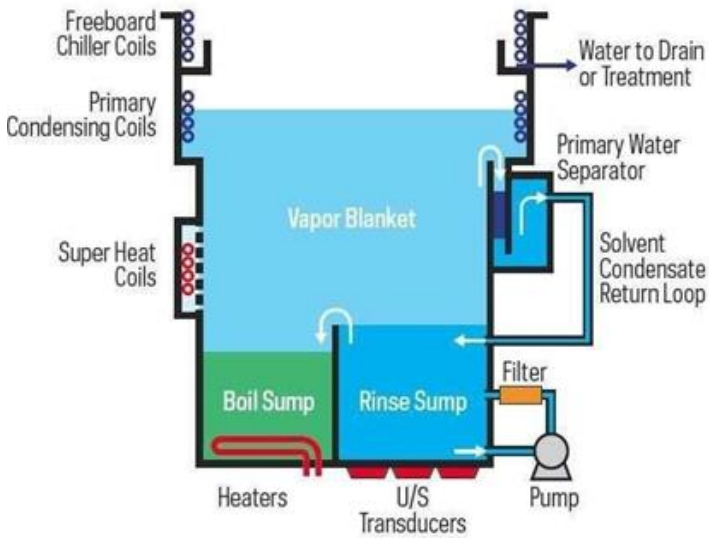


Fig. 5.1.5 A vapour degreasing unit

5.1.5 White Residues on PCBs

White residues are a common post-soldering defect. While often perceived as simple salts, they are complex mixtures of activator salts, flux resins, and other materials. When exposed to heat or certain chemicals, these residues can become corrosive and difficult to remove.

Causes:

- Flux Chemistry: The salts in flux, which act as "activators," can form residues if not properly rinsed. This problem is exacerbated with lead-free solder, which requires higher temperatures and different flux chemistries.
- Improper Process: Insufficient cleaning, incorrect solvent selection, or a reflow profile that doesn't properly activate and consume the flux can all lead to white residues.
- Material Interaction: Poorly cured solder mask or PCB substrates can interact with the flux and contribute to residue formation.
- Environmental Factors: Humidity, fingerprints, and other environmental factors can also be a source of the compounds that form these residues.

5.1.6 Storage and Handling of PCBs After Cleaning

Proper storage is crucial for maintaining the quality of a cleaned PCB. PCBs, especially multi-layer boards, are highly susceptible to moisture absorption due to their porous material structure. Accumulated moisture can cause delamination or "popcorning" during subsequent heating processes.

- Moisture Control: PCBs should be stored in a dry, low-humidity environment, often in moisture barrier bags with a desiccant.
- ESD Protection: The storage area must be ESD-free (Electrostatic Discharge) to protect sensitive components from static damage. This typically involves using ESD-safe racks and shelving.

Fig. 5.1.6 ESD-free racks are used for storage of PCBs

Notes

UNIT 5.2: Inspection of PCBs

Unit Objectives

By the end of this unit, the participants will be able to:

1. Explain the importance and key requirements of visual inspection in PCB assembly.
2. Identify the features and capabilities of Automated Optical Inspection (AOI) systems.
3. Describe the applications and benefits of Automated X-ray Inspection (AXI) methods.
4. Perform in-circuit testing to verify electrical functionality.
5. Compare the working principles, advantages, and limitations of different visual inspection machines.

5.2.1 Visual Inspection

Visual inspection is one of the most basic yet essential steps in the PCB assembly quality control process. It is usually performed after critical stages such as solder paste printing, component placement, and reflow soldering. The type of equipment or method used depends on the inspection requirements and the position of the components on the board.

For example, inspection personnel may use the naked eye or simple magnifiers to detect obvious defects such as missing components, misaligned parts, solder bridges, or excessive/contaminated solder paste. One of the most common tasks is the examination of reflow solder joints, which can be performed by observing how light reflects at different angles from the solder surface (often with the help of prisms or microscopes). On average, an experienced operator can inspect around five joints per second using such methods.

The effectiveness of visual inspection depends on:

- The inspector's skill, knowledge, and consistency.
- The availability and application of well-defined inspection standards.
- Understanding of defect categories for different solder joint types.

Each solder joint type has multiple possible defects (sometimes up to eight defect classifications per joint). Since a PCB assembly may contain six or more different solder joint types, inspection requires thorough training and reference to IPC standards.

However, visual inspection has limitations:

- It cannot provide quantitative measurements required for statistical process control.
- It is unsuitable for hidden or complex joints, such as those found in BGA (Ball Grid Array), QFN (Quad Flat No-lead), J-leaded packages, or flip-chip assemblies.

Despite these limitations, visual inspection remains a cost-effective and easily accessible method for detecting large, visible defects, particularly in early production stages or for low-complexity boards.

5.2.2 Automated Optical Inspection (AOI)

Automated Optical Inspection (AOI) is an advanced inspection technique that uses high-resolution cameras and image-processing algorithms to automatically scan PCBs for defects. Unlike manual inspection, AOI offers speed, consistency, and higher accuracy, making it suitable for modern, high-density, and miniaturized PCB assemblies.

Fig. 5.2.1: AOI equipment

During AOI, the system captures multiple images of the PCB under different lighting conditions (fluorescent, LED, infrared, or UV). These images are compared against the golden board reference or CAD design data stored in the system. Any deviation—such as missing components, incorrect orientation, or defective solder joints—is flagged for review.

Necessity of AOI

- Modern PCBs are more complex, often with multiple layers, fine-pitch components, and extremely small packages such as 0402 or 0201.
- Manual inspection is insufficient for detecting hidden or microscopic defects.
- AOI ensures that defects are identified early in the production process, reducing rework costs and preventing defective products from reaching customers.
- It is critical for industries with high reliability requirements, including medical, military, aerospace, automotive, and telecommunications.

Range	Tolerance
Accuracy	±0.0024mm
Speed	5in^2/sec (60FOV/sec)
PCB Max. size	400*330mm
Available Component	0201 chip and fine pitch

Attributes of AOI Equipment

Fig. 5.2.2 Attributes of an AOI equipment

Attributes and Capabilities of AOI

1. Variety of Defects Detected
    - Surface defects: scratches, nodules, stains.
    - Electrical defects: opens, shorts.
    - Solder-related issues: insufficient or excessive solder, bridges, voids.
    - Component defects: incorrect part, missing part, wrong polarity, misalignment.
2. Multiple Inspection Objects
    - Bare PCB inspection: checks for open circuits, shorts, and surface damage.
    - PCBA inspection: verifies component placement, polarity, solder joints, and part values.
3. Flexibility
    - AOI can be integrated at various stages of the SMT line (post-solder paste printing, pre-reflow, post-reflow).
    - Most effective placement is after reflow soldering, as the majority of defects originate during soldering.
    - By identifying defects early, AOI reduces downstream failures and overall production costs.
4. Cost Efficiency

AOI is less expensive compared to Automated X-ray Inspection (AXI) for detecting surface solder defects such as bridges or cold joints.

It is widely used as the primary inspection method before final testing.

Working Principle of AOI

AOI systems rely on image capture and analysis:

1. The PCB is scanned using high-resolution cameras with multiple light sources.
2. Captured images are processed and compared against predefined design data.
3. Software algorithms highlight differences, abnormalities, or potential defects.
4. Results are displayed in real time for operator verification.

Algorithms Used in AOI

1. Template Matching
    - A reference “golden board” image is used.
    - Pixel-by-pixel comparison is performed using normalized cross correlation (NCC).
    - High correlation values indicate correct matches.
2. Object Recognition
    - Compares captured component images against stored ideal images.
    - Effective for verifying polarity, shape, and orientation.
3. Blob Analysis
    - Separates objects (e.g., solder deposits) from the background.
    - Groups pixels into “blobs” for analysis of size, shape, and continuity.
4. Vectoral Imaging Technology (newer approach)
    - Uses geometric feature extraction instead of relying on pixel values.
    - Reduces false calls caused by lighting or color variations.
    - Provides higher accuracy in identifying component placement and orientation.

5.2.3 Automated X-ray Inspection (AXI)

Automated X-ray Inspection (AXI) is a powerful inspection method that uses X-rays instead of visible light to detect hidden defects that cannot be observed using Automated Optical Inspection (AOI). It is especially important for modern high-density PCB assemblies with BGA (Ball Grid Array), CSP (Chip Scale Package), and other hidden-joint components.

Fig. 5.2.3: An AXI equipment

Unlike AOI, which checks surface-level defects, AXI penetrates the PCB and reveals internal features, providing a deeper level of quality assurance.

Goals of AXI

- Process Optimization: Inspection results are used to fine-tune manufacturing steps, such as soldering and reflow parameters.
- Anomaly Detection: Detects and rejects defective parts for scrap or rework, preventing them from advancing further in the production line.

Principle of Operation

- X-rays are generated by an X-ray tube and passed through the target object.
- A detector opposite the emitter captures the varying X-ray intensities caused by different material densities.

Two common detector methods are:

1. Scintillator + Camera: Converts X-rays into visible light, which is then recorded by a camera.
2. Direct Sensor Arrays: Detect X-rays directly and convert them into digital electronic images.

The resulting images highlight internal structures and solder joints. The degree of absorption depends on material density and thickness, allowing hidden features to be revealed.

Unlike AOI, which checks surface-level defects, AXI penetrates the PCB and reveals internal features, providing a deeper level of quality assurance.

Goals of AXI

- Process Optimization: Inspection results are used to fine-tune manufacturing steps, such as soldering and reflow parameters.
- Anomaly Detection: Detects and rejects defective parts for scrap or rework, preventing them from advancing further in the production line.

Principle of Operation

- X-rays are generated by an X-ray tube and passed through the target object.
- A detector opposite the emitter captures the varying X-ray intensities caused by different material densities.

Two common detector methods are:

1. Scintillator + Camera: Converts X-rays into visible light, which is then recorded by a camera.
2. Direct Sensor Arrays: Detect X-rays directly and convert them into digital electronic images.

The resulting images highlight internal structures and solder joints. The degree of absorption depends on material density and thickness, allowing hidden features to be revealed.

Applications of AXI

- Electronics: Inspection of BGAs, CSPs, hidden solder joints, voids, and solder bridges.
- Industrial: Detecting cracks in alloy wheels, welding seam defects in nuclear power stations.
- Food processing: Detection of contaminants like glass, metal, and bone fragments. Used also for process optimization (e.g., detecting hole patterns in cheese to optimize slicing yield).
- Medical & Aerospace: Ensures reliability and safety in life-critical and mission-critical systems.

Benefits of Early Defect Detection via AXI

- Provides feedback at the earliest stage if process parameters or materials are out of control.
- Prevents defective components from moving downstream, saving time and cost.
- Improves overall reliability of final products by catching defects not detectable with functional testing.

Advantages of AXI

- Can “see through” components, making it possible to inspect hidden solder joints (e.g., BGAs).
- Effective for densely packed, multilayer PCBs.
- Provides detailed analysis of solder joint integrity, including voids and bridging.

Disadvantages of AXI

- Equipment is expensive, and ROI is justified mainly in high-density, high-volume production.
- New or uncommon component types may not yet be fully supported by existing AXI algorithms.
- Slower than AOI due to the complexity of image capture and analysis.

5.2.4 In-Circuit Testing (ICT)

In-Circuit Testing (ICT) is an electrical test method that checks whether a populated PCB has been assembled correctly. Unlike AOI or AXI, which rely on visual or X-ray imaging, ICT directly measures electrical parameters to verify component placement, orientation, and connectivity.

ICT detects:

- Shorts and opens.
- Component values such as resistance, capacitance, and sometimes inductance.
- Whether components are present and correctly installed.

Methods of ICT

1. Bed-of-Nails Tester
   - A test fixture with hundreds or thousands of spring-loaded pogo pins aligned to contact specific nodes on the PCB (DUT – Device Under Test).
   - Each pin touches a test point, allowing simultaneous measurement of many nodes.
   - Advantages: Fast, accurate, reliable for high-volume testing.
   - Disadvantages: Expensive to develop, difficult to reconfigure, less effective for dense, fine-pitch boards with limited test points.
2. Flying Probe Tester
   - Uses a small number of movable probes that “fly” across the board, contacting test points sequentially.
   - Advantages: Lower cost, flexible, easily adapted to design changes (software-driven).
   - Disadvantages: Slower than bed-of-nails for large-volume testing.

Fig. 5.2.4: ICT setup

Key Points about ICT

- ICT is structural, not functional—it doesn’t test whether the board performs its intended task. Instead, it assumes that if all components are correct and properly soldered, the board will function as designed.
- Best suited for low-to-medium complexity boards or when combined with functional testing.
- Often used together with AOI/AXI to provide a comprehensive inspection and testing strategy.

5.2.5 Comparison between Visual Inspection Machines

The following table lists the comparison between the three visual inspection machines:

Defect Type	AXI	AOI	ICT
Soldering Defect			
Open circuits	YES	YES	YES
Solder Bridge	YES	YES	YES
Solder Short	YES	YES	NO
Insufficient solder	YES	YES (But not Heel)	NO
solder void	YES	NO	NO
Excess solder	YES	YES	NO
Solder Quality	YES	NO	NO
Components Lifted			
Lifted Lead	YES	YES	YES
Missing Component	YES	YES	YES
Misplaced Components	YES	YES	YES
Incorrect components value	NO	NO	YES
Faulty components	NO	NO	YES
BGA and CSP			
BGA short	YES	NO	YES
BCA open circuit Connection	YES	NO	YES

Table 5.2.1: Comparison between the three visual inspection machines

Exercise

Short Questions:

1. Explain the mechanism of electromigration and why cleaning PCBs is critical to prevent it.
2. List any three benefits of adopting IPC standards in PCB manufacturing.
3. Differentiate between aqueous and semi-aqueous PCB cleaning solvents.
4. What are white residues on PCBs, and why are they considered a reliability concern?
5. State one key advantage and one limitation each of AOI, AXI, and ICT.
6. Compare Bed-of-Nails Tester and Flying Probe Tester in In-Circuit Testing (ICT).

Fill in the Blanks:

1. __________ contamination includes salts and flux activators, while __________ contamination includes oils, greases, and solder balls.
2. The IPC standard __________ covers requirements for soldered electrical and electronic assemblies.
3. White residues on PCBs are complex mixtures of __________, __________, and other materials.
4. In ultrasonic cleaning, contaminants are removed through a process called __________.
5. __________ inspection uses high-resolution cameras and image-processing algorithms to detect surface defects on PCBs.
6. The two common ICT methods are __________ tester and __________ tester.

Multiple Choice Questions (MCQs):

1. The main purpose of PCB cleaning is to:
   - a) Remove copper layers
   - b) Prevent electromigration and improve reliability
   - c) Reduce PCB thickness
   - d) Remove solder mask
2. Which IPC standard is most widely used for soldered electrical and electronic assemblies?
   - a) IPC-A-620
   - b) IPC-2581
   - c) IPC J-STD-001
   - d) IPC-6012
3. White residues on PCBs are mainly caused by:
   - a) Overuse of nitrogen in reflow ovens
   - b) Flux chemistry and improper cleaning processes
   - c) High-quality solder paste
   - d) Proper storage of boards

4. Which cleaning method uses microscopic bubbles created by high-frequency sound waves?
   a) Manual cleaning
   b) Ultrasonic cleaning
   c) Vapor degreasing
   d) Brushing
5. Automated X-ray Inspection (AXI) is particularly useful for:
   a) Detecting scratches on PCB surface
   b) Inspecting hidden solder joints like BGAs and CSPs
   c) Measuring resistance and capacitance
   d) Checking solder mask alignment
6. In-circuit testing (ICT) is best described as:
   a) A method for cleaning PCBs after reflow
   b) A structural electrical test for verifying assembly integrity
   c) A visual inspection using microscopes
   d) A flux residue removal method

Notes

6. Sustainability Practices in Telecom Production and Assembly Lines

TEL/N9107

Key Learning Outcomes

By the end of this module, the participants will be able to:

1. Identify recyclable, refurbishable, and hazardous components in telecom production and assembly lines.
2. Segregate and label telecom components and e-waste in compliance with EPR and organizational guidelines.
3. Apply green manufacturing and sustainable assembly practices such as energy-efficient operations, lead-free soldering, and optimized resource usage.
4. Dispose of and recycle hazardous and non-hazardous materials responsibly as per environmental safety protocols.
5. Maintain documentation for waste management, disposal, and sustainability compliance audits.
6. Comply with national and international environmental and sustainability regulations, including ISO 14001 and e-waste management laws.
7. Participate in sustainability audits and assessments, and implement corrective actions based on findings.
8. Evaluate production processes to identify opportunities for reducing environmental impact while maintaining efficiency and quality.

UNIT 6.1: Identification and Segregation of Telecom Components

Unit Objectives

By the end of this unit, the participants will be able to:

1. Inspect telecom materials to identify recyclable, refurbishable, and hazardous components.
2. Categorize PCBs, cables, batteries, plastics, and other materials for proper processing.
3. Maintain inventory and storage systems for recyclable and hazardous materials.
4. Follow EPR guidelines for labeling and segregation of e-waste.
5. Identify and report improper disposal practices.

6.1.1 Identification and Segregation of E-Waste

The first and most crucial step in managing electronic waste is a thorough and accurate inspection of materials. This is not just about a quick glance; it requires a systematic approach to categorize every component as it enters or leaves the facility. The goal is to separate valuable resources from toxic substances and reusable items.

Fig 6.1.1 Types of E-waste

- Recyclable Materials: These are components with high material value that can be reprocessed. This category includes Printed Circuit Boards (PCBs), which are rich in precious metals like gold, silver, and palladium. Other recyclable materials include copper wires from cables and aluminum or steel from equipment casings.

- Refurbishable Materials: These are items that can be reconditioned and put back into use, extending their product lifecycle. This might include fully functional networking equipment, servers, or repairable sub-assemblies. This practice significantly reduces the need for new manufacturing and is a cornerstone of the circular economy.
- Hazardous Materials: These are components that contain toxic elements and must be handled with extreme care. This category includes all types of batteries (lead-acid, lithium-ion, nickel-cadmium), capacitors (especially older ones that may contain PCBs), and materials with heavy metals like mercury and cadmium.

Beyond visual inspection, technicians can use more advanced methods.

- Material Safety Data Sheets (MSDS): These documents provide detailed information on the composition of a product, including any hazardous materials.
- Component Markings: Many components are marked with symbols or codes that indicate their material composition (e.g., "Pb" for lead, "Cd" for cadmium).
- Training and Certification: Ongoing training is essential for staying current on new materials and regulations. Certification programs for e-waste handlers provide the necessary skills and knowledge for proper identification and handling.

6.1.2 Categorizing Telecom Components

A technician must be able to categorize specific components for the correct waste stream.

- PCBs: The primary material in most electronic devices. They are highly valuable and classified as recyclable, but also contain hazardous materials, so they must be sent to a specialized, certified recycling facility.
- Cables and Wires: These are primarily valuable for their metal content (copper and aluminum). The plastic insulation is often recycled separately.
- Batteries: All batteries, regardless of their type or size, are considered hazardous waste and must be stored in dedicated containers and disposed of through certified battery recyclers.
- Plastic Casings: These should be inspected for recycling symbols. If recyclable, they are shredded and reprocessed. If not, they are disposed of as general waste, though efforts are made to minimize this.

6.1.3 Inventory Management and Tracking

Maintaining a meticulous inventory is essential for accountability and compliance. The inventory log should detail:

- The type of material (e.g., "Lithium-ion batteries," "Mixed PCBs").
- The quantity or weight of the material.
- The date of collection and the date of shipment.
- The final destination of the waste (e.g., name of the recycling vendor, landfill).

This information is used to track the company's waste management performance against its sustainability goals and to prepare reports for regulatory agencies, such as those required under EPR guidelines.

6.1.4 Storage and Labelling

Proper storage of e-waste is critical to prevent environmental harm and ensure safety.

- Segregation: E-waste must be stored in a dedicated, secure area, away from other waste streams. Hazardous materials must be kept separate from non-hazardous ones.
- Labelling: All containers must be clearly and accurately labeled. The label should identify the contents as e-waste and list any hazardous materials present. This practice is mandated by EPR guidelines, which hold manufacturers responsible for the proper disposal of their products.
- Environment: The storage area should be well-ventilated and protected from weather to prevent corrosion or leaks. Containers should be robust enough to prevent spills.

6.1.5 Identifying and Reporting Improper Practices

Every individual on the job has a responsibility to act as a steward of the environment. This means being vigilant and reporting any observed improper disposal. For example, if a technician sees a hazardous component being thrown into a regular trash bin, they must report it to a supervisor or the designated authority. This vigilance helps uphold the company's commitment to legal and ethical standards.

Fig 6.1.2 E-waste Process

6.1.6 Sustainability Frameworks and Laws

A technician's competence is underpinned by a broader understanding of the policies and laws governing e-waste.

Fig 6.1.3 Circular Economy

- Sustainability Policies: Many companies have their own sustainability and corporate social responsibility (CSR) policies. These policies provide the internal motivation and framework for proper e-waste management.
- Extended Producer Responsibility (EPR): These are policies that make manufacturers financially and physically responsible for the collection and recycling of their products. This is a key policy that shifts the responsibility of a product's end-of-life disposal from the consumer to the manufacturer. It encourages companies to design more sustainable products and set up take-back systems. A technician must know how to identify products covered by these guidelines and ensure they are handled accordingly.
- ISO 14001: This is an international standard that provides a framework for an Environmental Management System (EMS). Companies certified under ISO 14001 have a structured approach to managing their environmental responsibilities, including waste management.

- E-Waste Laws: Laws such as the WEEE Directive in the European Union or the E-Waste Management Rules in India provide specific legal requirements for the collection, handling, and recycling of e-waste. A technician must be aware of these laws to ensure the company remains compliant and avoids legal penalties. Technicians must be familiar with local, national, and international e-waste laws (e.g., in the EU, the WEEE Directive; in India, the E-Waste Management Rules). These laws dictate how waste must be categorized, transported, and recycled.

A competent technician must understand the company's broader sustainability goals. This knowledge provides context for why e-waste management is so important. Companies may set specific targets for reducing landfill waste, increasing recycling rates, or achieving carbon neutrality. Understanding these goals motivates proper e-waste handling and reinforces a culture of environmental responsibility.

The proper identification and segregation of electronic waste are fundamental to a responsible and sustainable telecommunications industry. By mastering the competencies of inspection, categorization, and inventory management, and by understanding the relevant legal and policy frameworks, a technician can play a vital role in reducing environmental impact, conserving valuable resources, and ensuring their organization's compliance. E-waste management is not just about waste disposal; it's about transforming a linear product life cycle into a circular, sustainable one.

Notes

UNIT 6.2: Green Manufacturing and Sustainable Assembly Practices

Unit Objectives

By the end of this unit, the participants will be able to:

1. Operate energy-efficient tools and automated systems to minimize carbon footprint.
2. Apply low-emission soldering and lead-free assembly processes.
3. Optimize the use of materials, water, and electricity in production activities.
4. Perform maintenance and calibration of energy-efficient machinery.
5. Integrate ISO 14001 standards into telecom production workflows.

6.2.1 Use of Energy-Efficient Equipment, Tools, and Automation

Green manufacturing in telecom production focuses on reducing environmental impact while maintaining high-quality output. It integrates energy efficiency, pollution control, resource optimization, and international standards like ISO 14001. Sustainable assembly practices not only reduce costs but also help organizations meet regulatory requirements and corporate sustainability goals.

Fig 6.2.1 Greening Practices

Modern telecom manufacturing relies heavily on energy-efficient machines and automation to minimize power consumption and reduce the overall carbon footprint.

- Energy-efficient equipment: Machines with lower standby power, variable speed drives (VSDs), and smart sensors optimize electricity use.
- Automation: Robotic arms, automated pick-and-place machines, and conveyor systems reduce human error, increase speed, and minimize material waste.
- Smart monitoring systems: IoT-enabled sensors track real-time energy use and identify inefficiencies.

Example in telecom production: Automated SMT (Surface Mount Technology) lines that use optimized reflow ovens with controlled heating zones, reducing unnecessary power consumption.

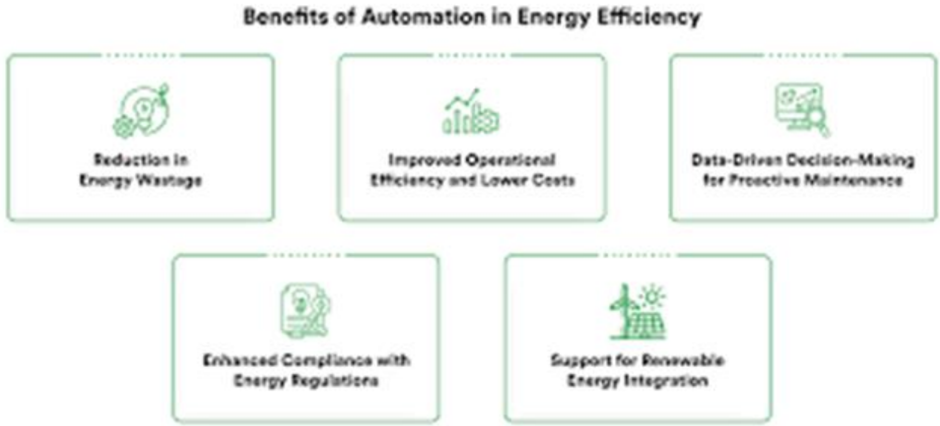


Fig 6.2.2 Benefits of Automation

6.2.2 Low-Emission Soldering and Lead-Free Assembly Processes

Traditional soldering relied on lead-based solder, which is hazardous to human health and the environment. Sustainable assembly practices promote low-emission and lead-free soldering:

- Lead-free solder alloys: Common compositions include Sn-Ag-Cu (SAC) alloys that comply with RoHS (Restriction of Hazardous Substances).
- Low-emission fluxes: Water-based or no-clean fluxes reduce volatile organic compounds (VOCs).
- Controlled soldering techniques: Selective soldering, vapor-phase soldering, and nitrogen-assisted reflow reduce oxidation and improve efficiency.

Benefit: Reduced workplace hazards, improved recyclability of PCBs, and compliance with global regulations such as RoHS and WEEE.

6.2.3 Optimization of Materials, Water, and Electricity

Efficient use of resources is at the core of green manufacturing:

1. Material optimization:
    - Adopt just-in-time (JIT) production to avoid overstock and waste.
    - Use PCB panelization to reduce unused board material.
    - Recycle scrap metals (copper, aluminum) from production.
2. Water optimization:
    - Reuse and recycle process water in PCB cleaning.
    - Install closed-loop water systems to prevent discharge.
    - Monitor for leaks to prevent wastage.
3. Electricity optimization:
    - Schedule high-power operations during off-peak hours.
    - Use LED lighting and motion sensors in production floors.
    - Employ renewable energy sources (solar panels, wind) where possible.

Example: A telecom assembly plant reduced electricity usage by 20% by upgrading to energy-efficient HVAC systems and optimizing reflow oven heating cycles.

Fig 6.2.3 Steps of Resource Optimization

6.2.4 Maintenance and Calibration of Energy-Efficient Machinery

Regular maintenance ensures machines continue to perform at optimal efficiency:

- Preventive maintenance: Cleaning, lubrication, and inspection prevent energy loss due to friction or clogging.
- Calibration: Ensures accuracy of measuring instruments and reduces rework or scrap.
- Software updates: Improve energy performance of automated systems.

Example: A misaligned pick-and-place machine can waste components, increasing material consumption. Proper calibration ensures reduced errors and better yield.

6.2.5 Adoption of ISO 14001 in Production Workflows

ISO 14001: Environmental Management System (EMS) is an international standard that guides organizations in managing their environmental responsibilities:

- Planning: Identify environmental aspects (energy use, emissions, waste) and legal requirements.
- Implementation: Integrate eco-friendly practices into assembly line processes.
- Monitoring: Use environmental audits and KPIs (e.g., energy saved, waste reduced) to track performance.
- Continuous improvement: Correct deviations, adopt new technologies, and set higher sustainability goals.

Impact in telecom: Ensures compliance with global sustainability regulations, improves corporate image, and reduces operational risks.

In conclusion, to practice sustainable assembly, individuals must understand:

- Principles of green manufacturing in telecom production.
- Role of energy-efficient tools and automation in reducing emissions.
- Benefits of lead-free soldering and low-emission assembly.
- Methods for optimizing resources such as materials, water, and electricity.
- Importance of maintenance and calibration in sustaining efficiency.
- Requirements of ISO 14001 EMS for compliance and global competitiveness.

Notes

UNIT 6.3: Waste Disposal and Recycling in Telecom Production

Unit Objectives

By the end of this unit, the participants will be able to:

1. Classify and handle hazardous waste (e.g., lithium batteries, chemical residues) in compliance with environmental safety protocols.
2. Segregate and recycle non-hazardous materials such as plastics, aluminum, and copper through designated collection and recycling channels.
3. Maintain accurate documentation of waste disposal and recycling activities for audits and compliance tracking.
4. Conduct periodic waste audits to identify opportunities for minimizing waste generation and improving material efficiency.
5. Collaborate with authorized e-waste recyclers to ensure proper processing, recovery of resources, and safe disposal of unusable components.
6. Apply safe handling practices for hazardous and non-hazardous materials to prevent health risks and environmental damage.
7. Implement techniques to reduce material wastage without compromising product quality or production efficiency.

6.3.1 Disposal of Hazardous Waste

Proper disposal and recycling of waste are critical sustainability practices in telecom production and assembly lines. With the increasing complexity of electronic devices and the growing amount of e-waste generated, manufacturers must adopt safe, systematic, and environmentally compliant methods to manage both hazardous and non-hazardous waste streams.

Hazardous waste in telecom production includes lithium batteries, chemical residues from soldering, solvents, heavy metals, and contaminated packaging. Mishandling these materials can cause serious health risks, fires, or environmental pollution.

Fig 6.3.1 Handling of Hazardous Waste

Safe Handling Procedures:

- Hazardous waste must be segregated from regular waste immediately after generation.
- Workers should use PPE such as gloves, masks, and safety glasses while handling toxic substances.
- Containers must be sealed, corrosion-resistant, and clearly labeled as per hazardous material classification standards.

Storage and Transport:

- Hazardous waste should be stored in dedicated areas with spill containment systems.
- Waste must be transported only through certified carriers who comply with environmental safety regulations.

Disposal Protocols:

- Lithium-ion batteries are sent to certified recyclers that recover valuable metals (cobalt, nickel, and lithium).
- Chemical residues and solvents must be neutralized or treated before disposal to prevent groundwater contamination.
- All disposal activities must align with local environmental laws and Extended Producer Responsibility (EPR) guidelines.

6.3.2 Recycling of Non-Hazardous Materials

Non-hazardous, recyclable waste forms a major portion of telecom assembly line by-products. This includes plastic casings, aluminum housings, copper wires, PCBs, and packaging materials.

Segregation Process:

- Separate bins should be designated for plastics, metals, PCBs, and paper-based packaging.
- Colour-coded labels help workers quickly identify the correct collection points.

Collection and Processing:

- Plastics (ABS, polycarbonate) are shredded and repurposed into new casings or packaging.
- Aluminum and copper are melted and reused in electronic and non-electronic industries.
- PCBs are processed using hydrometallurgical or pyrometallurgical methods to recover precious metals like gold and silver.

Circular Economy Integration:

- By recycling materials back into production cycles, manufacturers reduce dependency on virgin raw materials, cut costs, and lower the carbon footprint.

6.3.3 Documentation and Compliance Audits

Effective waste management is incomplete without thorough documentation. Records serve as evidence of compliance during audits and also help track sustainability performance.

Required Records:

- Type and quantity of hazardous and non-hazardous waste generated.
- Details of storage, transportation, and disposal methods used.
- Certificates from authorized recyclers confirming proper waste processing.

Compliance Tracking:

- Records should meet the standards of ISO 14001 (Environmental Management Systems).
- Digital documentation systems are increasingly being used to ensure transparency, traceability, and ease of access during inspections.

Audit Preparation:

- Organizations should conduct internal audits before official compliance checks.
- Any gaps in documentation or handling should be identified and corrected.

6.3.4 Conducting Waste Audits

Audit Steps:

- Collect data on material flow — what enters the production line versus what exits as waste.
- Categorize waste streams into recyclable, refurbishable, and non-recyclable.
- Evaluate the efficiency of segregation, storage, and disposal practices.
- Identify material wastage hotspots (e.g., solder paste spillage, excess packaging).

Benefits of Waste Audits:

- Highlight cost-saving opportunities by reducing unnecessary waste.
- Improve raw material utilization efficiency.
- Strengthen the company’s sustainability credentials and compliance standing.

6.3.5 Coordination with Authorized E-Waste Recyclers

Not all waste can be handled internally. Authorized e-waste recyclers ensure safe processing, recovery of valuable resources, and compliance with environmental norms.

Collaboration Requirements:

- Partner only with government-approved or certified recycling agencies.
- Provide recyclers with complete documentation on material classification and quantities.
- Track recycler certifications and audit their processes to ensure compliance with EPR and ISO standards.

Advantages of Third-Party Recyclers:

- Access to advanced recovery technologies (precious metal extraction, thermal treatment).
- Reduced environmental liability for manufacturers.
- Contribution to circular economy and reduced landfill dependency.

Takeaways:

- Workers must understand proper handling, storage, and disposal of both hazardous and non-hazardous materials.
- Knowledge of material wastage reduction techniques (e.g., reusing off-cuts, optimizing packaging, right-sizing components) is essential for minimizing resource loss.
- Familiarity with environmental and safety regulations, as well as the role of e-waste in sustainability, ensures responsible production practices.

Fig 6.3.2 E-waste Recycling Unit

Notes

UNIT 6.4: Compliance with Environmental and Sustainability Regulations

Unit Objectives

By the end of this unit, the participants will be able to:

1. Describe national and international environmental laws and standards relevant to telecom production.
2. Participate effectively in sustainability audits and assessments.
3. Monitor compliance with sustainability guidelines in production processes.
4. Identify, report, and address deviations from sustainability protocols.
5. Apply corrective actions from environmental audits to improve workflows.
6. Use basic environmental impact assessment techniques in telecom manufacturing.
7. Maintain accurate documentation for compliance and audit purposes.

6.4.1 National and International Environmental Laws

Compliance with environmental and sustainability regulations is an essential requirement in telecom production and assembly processes. Given the increasing global focus on reducing the environmental footprint of industrial operations, it is the responsibility of professionals to strictly adhere to applicable laws, internal policies, and international standards. Effective compliance ensures that the organization maintains its social responsibility, avoids legal penalties, and contributes toward long-term ecological balance.

Telecom production involves handling a wide range of materials, including plastics, metals, soldering alloys, printed circuit boards (PCBs), and batteries, many of which have potential environmental impacts if not properly managed. Workers must be familiar with and follow:

National Regulations:

- E-Waste Management Rules (India) or equivalent in other regions, mandating safe disposal, recycling, and Extended Producer Responsibility (EPR).
- Hazardous Waste Handling Regulations covering the storage, transport, and disposal of harmful substances.
- Energy Conservation Acts encouraging the use of efficient technologies in production.

International Regulations and Standards:

- ISO 14001: Environmental Management System (EMS) framework to minimize environmental impact.
- RoHS (Restriction of Hazardous Substances): Limiting toxic substances like lead, mercury, and cadmium in electrical and electronic products.
- REACH (Registration, Evaluation, Authorisation, and Restriction of Chemicals): Ensuring safe use of chemical substances.
- Basel Convention: Governing cross-border movement of hazardous wastes.

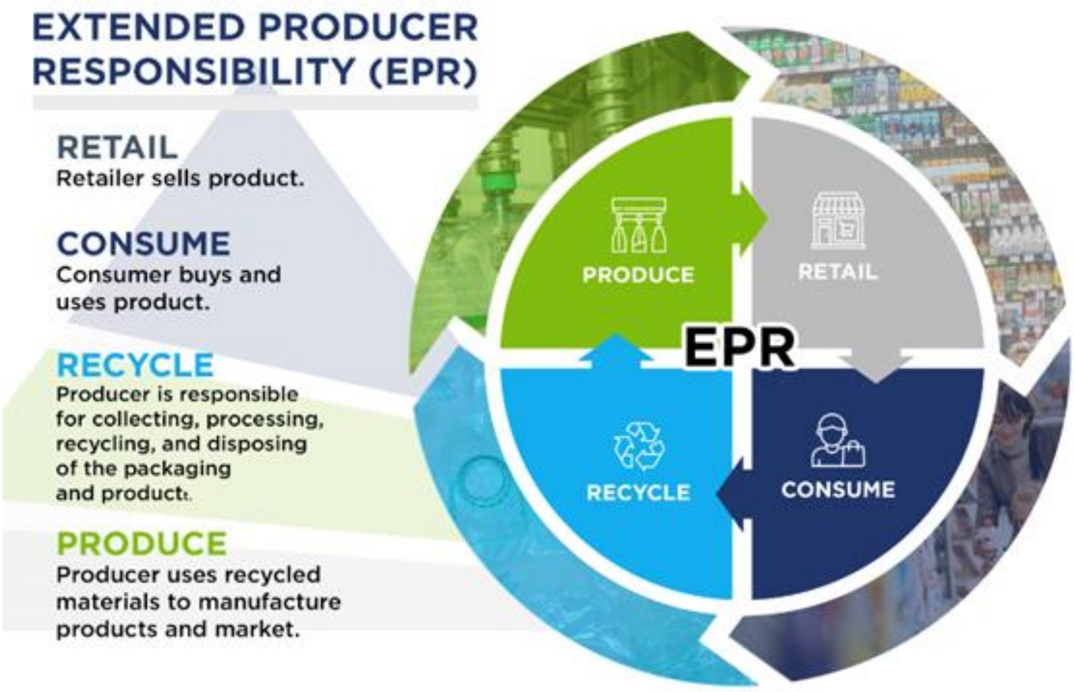


Fig 6.4.1 Extended Producer Responsibility (EPR)

Strict compliance ensures that telecom organizations remain aligned with both local and global sustainability goals.

6.4.2 Sustainability Audits and Assessments

Sustainability audits evaluate how well the organization is implementing eco-friendly practices. Participation involves:

- Providing Documentation: Supplying records of waste disposal, recycling, energy consumption, and material usage.
- On-Ground Support: Assisting auditors during inspections of production and assembly lines.
- Performance Monitoring: Tracking progress against internal sustainability goals, such as reduced energy use or lower carbon emissions.
- Corrective Action Plans: Contributing to discussions about improving existing practices based on audit results.

Regular participation in audits not only validates compliance but also builds a culture of accountability and continuous improvement.

Environmental Impact Assessment Techniques

Environmental Impact Assessment (EIA) is a systematic approach used to evaluate the potential environmental consequences of production activities. For telecom manufacturing, this may include:

- Material Impact Studies: Assessing how solder, plastics, and metals affect ecosystems.
- Energy Analysis: Measuring electricity and fuel consumption in production processes.
- Emission Monitoring: Tracking carbon dioxide and other emissions from assembly line activities.
- Waste Mapping: Identifying types, sources, and quantities of waste generated.

Fig 6.4.2 Environmental Impact Assessment (EIA)

Understanding EIA techniques helps employees anticipate risks and adopt preventive measures.

Documentation Requirements for Audits and Compliance

Proper documentation is crucial for demonstrating adherence during audits and inspections. Key records include:

- Waste disposal logs with details of hazardous and non-hazardous materials.
- Recycling records in compliance with EPR and ISO 14001.
- Calibration and maintenance logs for energy-efficient equipment.
- Audit reports, corrective action records, and sustainability performance reports.

Accurate documentation not only proves compliance but also enables continuous monitoring and long-term sustainability planning.

6.4.3 Adherence to Sustainability Guidelines

Employees play a vital role in ensuring that sustainability guidelines are followed on a day-to-day basis. Key practices include:

- Monitoring whether hazardous and non-hazardous waste is correctly segregated and disposed of.
- Checking for the use of lead-free soldering and low-emission assembly processes in line with green manufacturing policies.
- Ensuring that energy-efficient equipment is properly used and maintained.
- Reporting deviations such as unauthorized disposal, overuse of resources, or non-compliance with EPR labeling and storage requirements.

By identifying and reporting deviations early, employees help prevent regulatory breaches and environmental risks.

6.4.4 Corrective Actions from Environmental Assessments

When audits or inspections reveal non-compliance, corrective actions must be promptly implemented. This may involve:

- Retraining employees on sustainable handling of materials.
- Updating processes to align with new or revised environmental laws.
- Repairing or replacing faulty equipment that causes excessive emissions or energy wastage.
- Improving documentation practices for better tracking of recycling, disposal, and energy-saving measures.

Taking corrective actions promptly ensures continuous alignment with environmental goals and demonstrates the organization's commitment to sustainability.

In summary, complying with environmental and sustainability regulations requires telecom professionals to:

- Stay updated on applicable local and global laws.
- Participate actively in sustainability audits.
- Ensure daily adherence to eco-friendly practices.
- Follow corrective actions as per assessments.
- Maintain thorough documentation and use impact assessment techniques to guide decision-making.

This creates a workplace culture where sustainability is embedded into every stage of telecom production and assembly.

Exercise

Short Questions:

1. Differentiate between recyclable, refurbishable, and hazardous telecom components with examples.
2. State any three green manufacturing practices that reduce environmental impact in telecom assembly.
3. What are the key benefits of adopting ISO 14001 standards in telecom production workflows?
4. Explain why batteries are always categorized as hazardous waste and how they must be handled.
5. Describe the role of waste documentation in sustainability compliance and audits.
6. Identify two corrective actions that may be implemented if a sustainability audit reveals non-compliance.

Fill in the Blanks:

1. PCBs contain valuable metals like ___________ and ___________, making them recyclable but also requiring special handling.
2. The ___________ Directive in the EU and the ___________ Rules in India regulate e-waste management.
3. Lead-free soldering complies with ___________ regulations that restrict hazardous substances.
4. ___________ audits help organizations identify material wastage hotspots and improve raw material efficiency.
5. Hazardous waste containers must be ___________, ___________, and clearly labeled.
6. ___________ is the systematic process of evaluating potential environmental consequences of production activities.

Multiple Choice Questions (MCQs):

1. Which of the following is classified as hazardous waste in telecom production?

a) Copper wires

b) Lithium-ion batteries

c) Plastic casings

d) Aluminum housings

2. Extended Producer Responsibility (EPR) policies make:

a) Consumers responsible for recycling products

b) Manufacturers responsible for end-of-life product collection and recycling

c) Governments solely responsible for managing e-waste

d) Technicians responsible for storage of waste only

3. Lead-free soldering practices typically use:
    a) Tin-lead (Sn-Pb) alloys
    b) Mercury-based alloys
    c) Sn-Ag-Cu alloys
    d) Copper-tin alloys
4. Which ISO standard provides a framework for Environmental Management Systems (EMS)?
    a) ISO 45001
    b) ISO 9001
    c) ISO 14001
    d) ISO 27001
5. In In-circuit testing of sustainability compliance, waste audits are used primarily to:
    a) Repair damaged boards
    b) Identify opportunities to reduce waste generation
    c) Replace old machinery
    d) Check voltage across PCBs
6. Which international agreement governs cross-border movement of hazardous wastes?
    a) RoHS
    b) REACH
    c) Basel Convention
    d) Kyoto Protocol

Notes

GOVERNMENT OF INDIA
MINISTRY OF SKILL DEVELOPMENT
& ENTREPRENEURSHIP

Telecom Sector Skill Council

7. Employability Skills (30 Hours)

It is recommended that all training include the appropriate. Employability Skills Module. Content for the same can be accessed https://www.skillindiadigital.gov.in/content/list

DGT/VSQ/N0101

8. Annexure

Annexure I - QR Codes –Video Links

Annexure I

QR Codes –Video Links

NCVET
कौशल गुणवत्ता प्रगति

Skill India
कौशल भारत - कुशल भारत

सत्यमेव जयते
GOVERNMENT OF INDIA
MINISTRY OF SKILL DEVELOPMENT
& ENTREPRENEURSHIP

N·S·D·C
RE IMAGINE FUTURE

Telecom
Sector
Skill
Council

Telecom Sector Skill Council
Estel House,3rd Floor, Plot No: - 126, Sector-44
Gurgaon, Haryana 122003
Phone: 0124-2222222
Email: tssc@tsscindia.com
Website: www.tsscindia.com