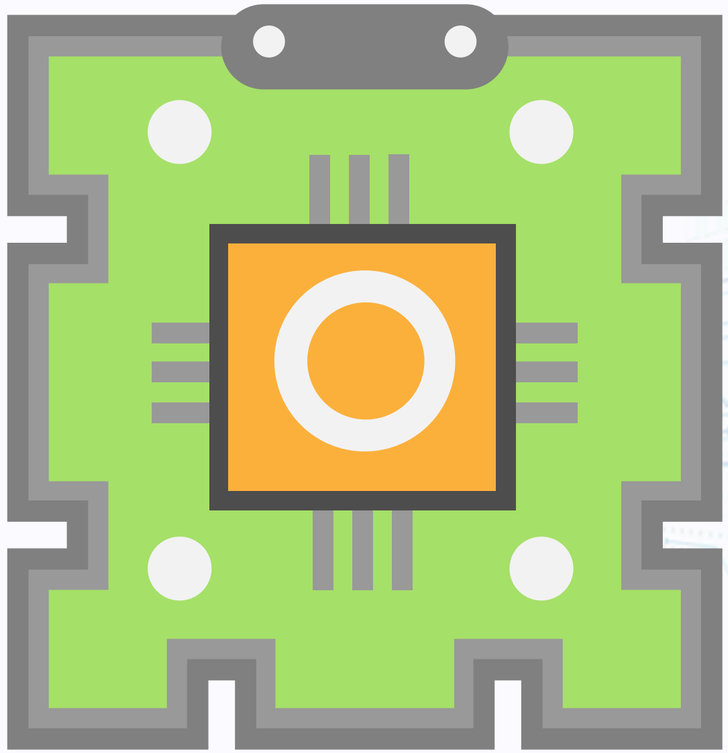
UNIT - IV

Agenda

1. What is Polygon
2. power plane and ground plane
3. Purpose of Polygon
4. Advantages of Polygon
5. Introduction to Thermal Management
6. Thermal Management Techniques

Polygon in PCB Design

What is Polygon PCBs?



A polygon in PCB (Printed Circuit Board) design is a closed shape (usually a filled area) that is used to create large conductive regions or copper planes on a PCB. These polygons are typically used for power distribution, ground planes, or shielding.

Purpose of Polygons in PCB

Ground and Power Planes:

- Provides a large, low-impedance path for power or ground connections.
- Improves circuit stability and reduces noise.

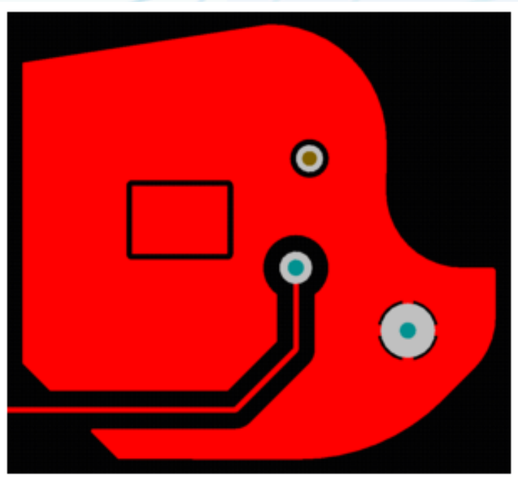
Thermal Management:

- Helps in dissipating heat from high-power components by spreading it over a large area.

Types of Polygon

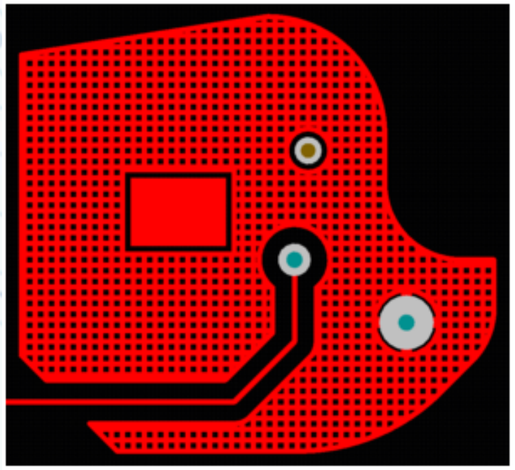
1. Solid Fill Polygons:

- Fully filled copper areas.
- Commonly used for ground or power planes.



2. Hatched Polygons:

- Filled with a mesh pattern instead of solid copper.
- Used to reduce copper usage and manage impedance.



Advantages of Using Polygons

Improved Electrical Performance

Reduces voltage drop and noise in power distribution.

Better Thermal Conductivity

Helps dissipate heat from high-power components.

Efficient PCB Layout

Simplifies connections and reduces routing complexity.

Introduction to Thermal Management

What is Thermal Management in PCBs?

Thermal management in PCBs refers to the design strategies and techniques used to control heat generation, dissipation, and distribution to ensure reliable operation and longevity of electronic components.

Why is Thermal Management Important?

Heat Generation in Electronics

High-performance electronic components like processors, power regulators, and LEDs generate significant heat during operation.

Impact of Heat on PCBs

- Excessive heat can cause component failure, signal degradation, and mechanical stress.
- Uneven heat distribution may lead to thermal expansion and reliability issues.

Sources of Heat in PCBs

Active Components

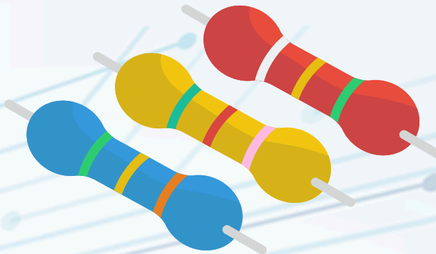
- ICs, power transistors, and LEDs.

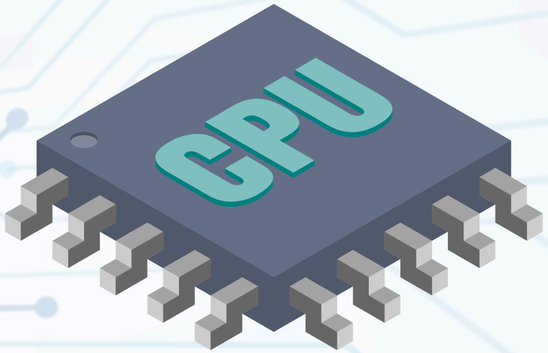
Passive Components

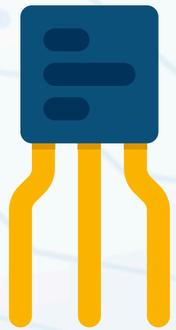
- Resistors, inductors, and transformers.

Environmental Factors

- Ambient temperature and enclosure constraints.







Key Goals of Thermal Management

Efficient Heat Dissipation:

- Transfer heat away from components to avoid overheating.

Uniform Temperature Distribution:

- Minimize hotspots on the PCB.

Optimized Design:

- Integrate thermal considerations early in the PCB design process.

Effective Thermal Management Techniques

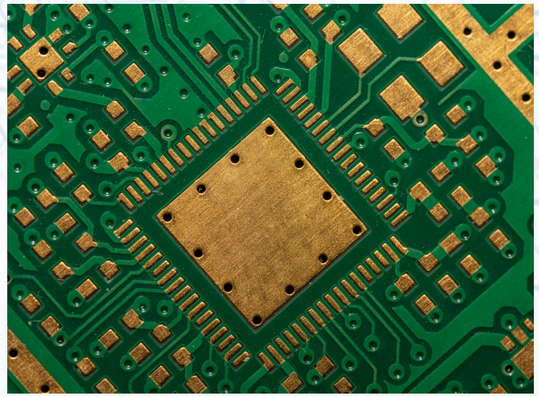
- Use of thermal vias.
- Copper pours and thicker copper layers
- Heat sinks and thermal interface materials.
- Component placement and thermal materials.

#enthutech®

1. Use of Thermal Vias

Best for high-power components.

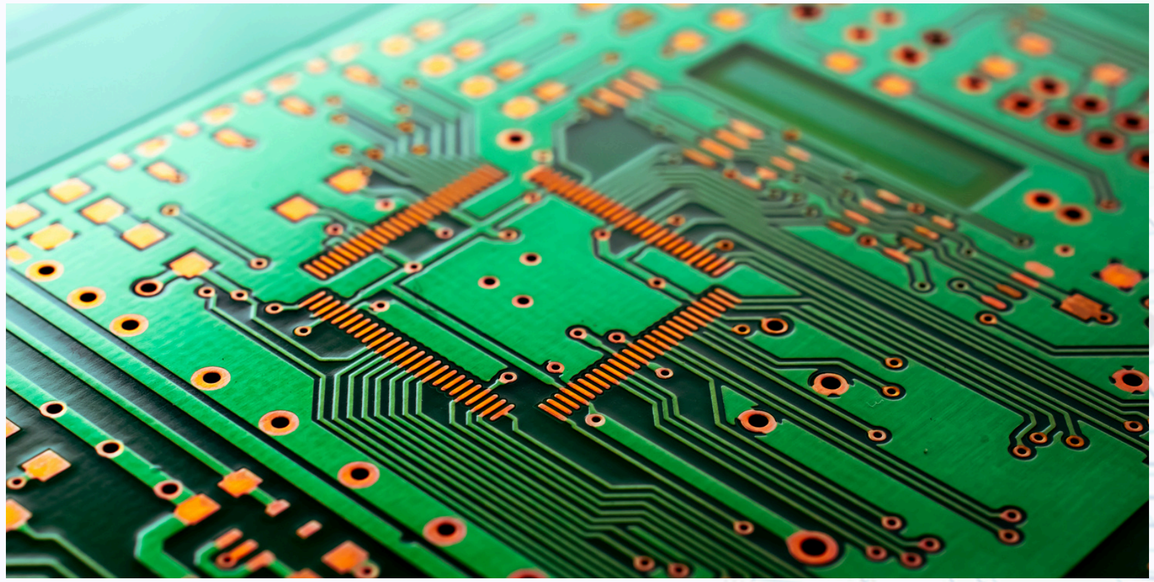
Transfers heat between PCB layers.



Use an array for efficient heat transfer.

#enthutech®

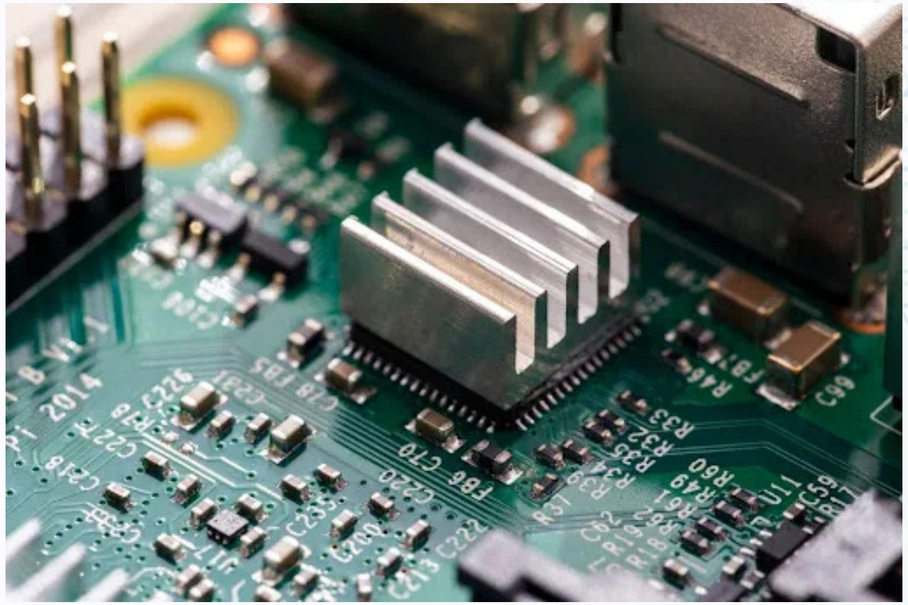
2. Copper Pours and Planes

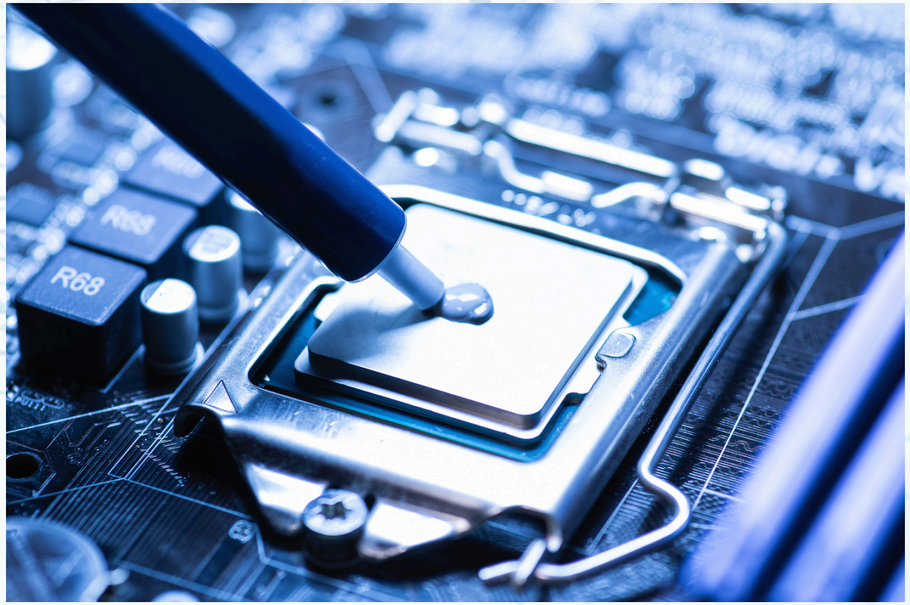


- Spreads heat across the PCB.
- Thicker copper improves heat conduction.
- Acts as a natural heat sink.

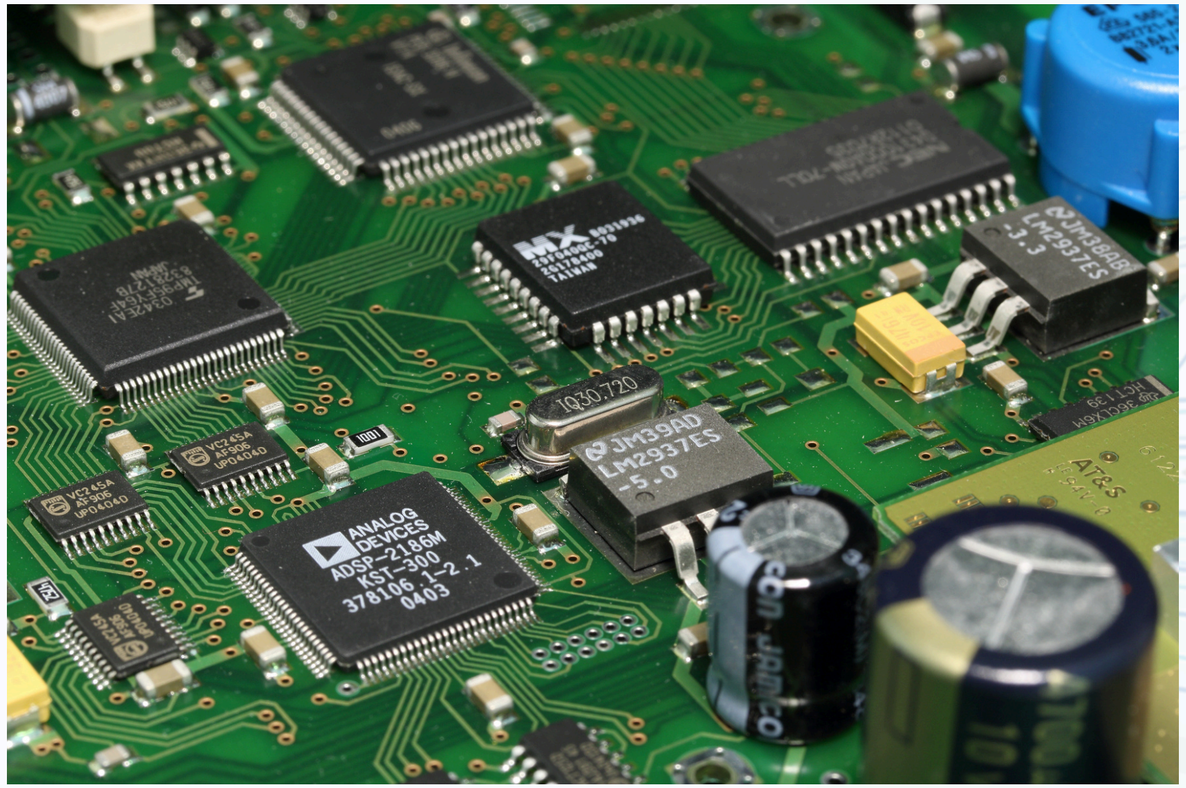
3. Heat Sinks and Thermal Interface Materials

- Heat sinks draw heat away from components.
- Thermal pads/paste fill gaps to improve conduction.





4. Optimal Component Placement



- Spread high-power components to avoid hotspots.
- Place heat-sensitive components away from heat sources.
- Position heat-generating components near edges for better ventilation.

Thank You

What is EMI (Electromagnetic Interference)?

Unwanted electromagnetic energy that disrupts the performance of electronic devices.

Types of EMI:

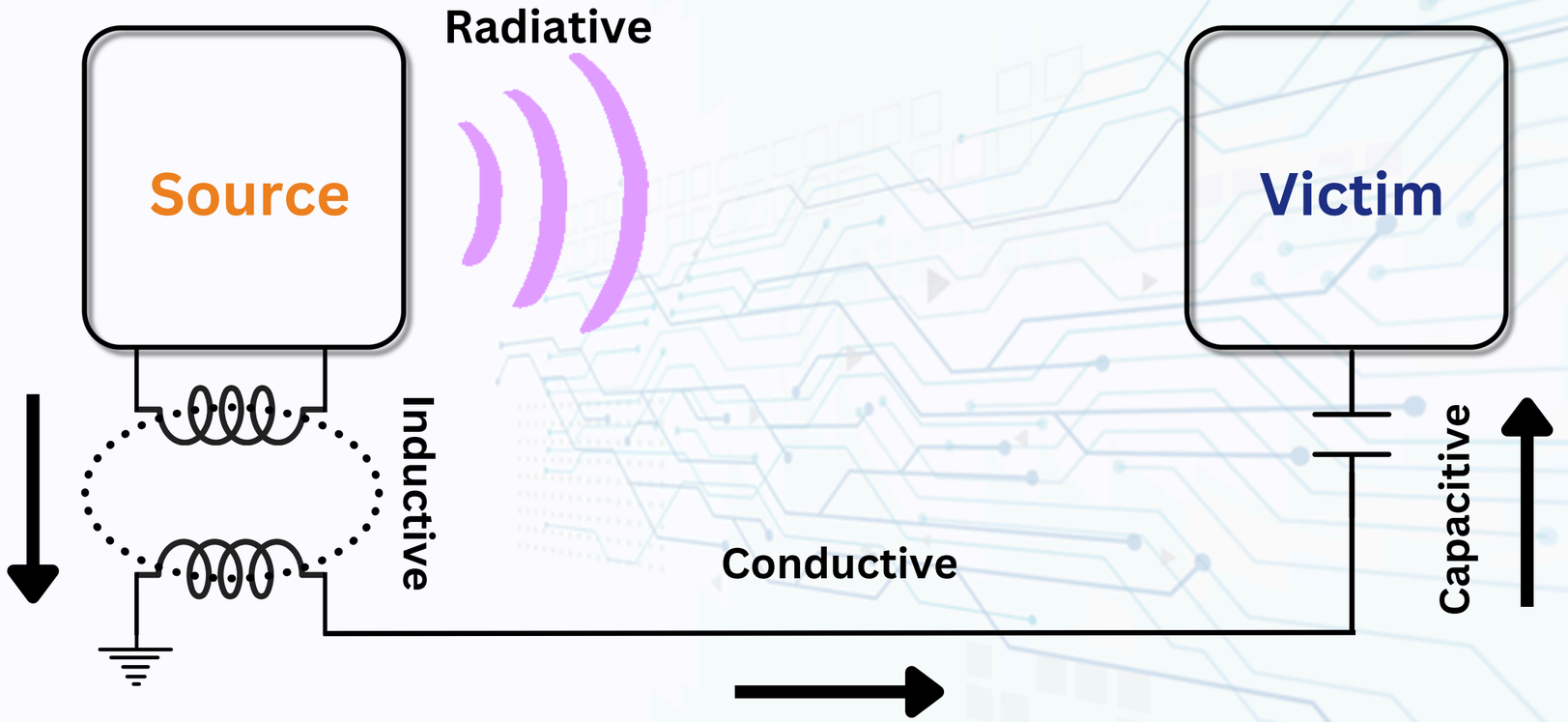
Radiated EMI:

Energy emitted into the air (e.g., antennas, high-speed traces).

Conducted EMI:

Interference transmitted through power or signal lines.

EMI & EMC



What is EMC (Electromagnetic Compatibility)?

- The ability of a device to function without causing or being affected by EMI.
- Ensures devices can coexist in the same environment without interference.

Sources of EMI in PCBs

- High-speed circuits and switching components.
- Poor grounding and PCB layout.
- Inductive and capacitive coupling between traces.

Strategies to Reduce EMI in PCBs

Use Ground Planes:

- Solid ground planes minimize loop areas and reduce noise.

Shielding:

- Enclose sensitive circuits with metal shields.

Trace Routing:

- Keep high-speed and noisy traces short.
- Avoid crossing signals over splits in the ground plane.

Decoupling Capacitors:

- Place capacitors near power pins of ICs to reduce noise.

Differential Signaling:

- Use paired signals to reduce radiated emissions.

Strategies to Achieve EMC

Component Placement:

- Separate noisy components (e.g., oscillators, power circuits) from sensitive analog components.

PCB Stack-Up:

- Use multi-layer PCBs with dedicated ground and power planes for better isolation.

Isolation and Filtering:

- Use optocouplers or isolation transformers for signal isolation.

EMI Testing:

- Test and tweak designs in compliance with regulatory standards.

Best Practices for EMI/EMC in PCB Design

- Minimize loop areas for return currents.
- Use controlled impedance for high-speed signals.
- Maintain consistent grounding and avoid floating grounds.
- Use thermal relief pads for grounding noisy components.