MOHAMED SAHUL HAMEED H

4TH YEAR, B.E.ECE,

CEG CAMPUS, ANNA UNIVERSITY.

PCB PLACEMENT & ROUTING NOTES

Project Description:

A 5 V, 3 A DC–DC buck converter was designed in **KiCad Software** using the LM2596S-5 IC, including schematic, PCB layout, and 3D verification to ensure stable performance and manufacturability

Board Structure and Size:

- A two-layer PCB is used to keep the design compact while maintaining good electrical performance.
- All components are placed on the top layer, which makes assembly, inspection, and debugging easier.
- The bottom layer is mainly used as a continuous ground plane to provide a stable return path for current and to reduce electrical noise.

Component Arrangement:

- The buck regulator IC is placed near the center of the board so that all major power components can be connected with short and direct tracks.
- Input capacitors are placed very close to the input pins of the IC to reduce input voltage ripple and noise.
- The diode, inductor, and output capacitors are placed close to the output pin of the regulator to minimize switching loop area and improve efficiency.

Power Trace Routing:

- All power traces that carry high current, such as the input supply, switch node, inductor path, and output voltage, are routed using wide copper tracks.
- These tracks are kept short and direct to reduce voltage drop, heat generation, and power loss.
- Smooth routing is followed to avoid sharp bends in high-current paths.

Ground Connections:

- A solid ground plane is provided on the bottom layer to ensure a low-impedance ground reference. Ground pins of the regulator IC, capacitors, and connectors are connected to this ground plane using multiple vias.
- This grounding method helps reduce ground noise and improves the overall stability of the circuit.

Noise Reduction:

- Small ceramic capacitors are placed close to the IC supply pins to filter high-frequency switching noise.
- The switching node is routed carefully and kept away from sensitive signal and output traces to prevent noise coupling.
- This helps in improving EMI performance and output signal quality.

Connectors and Mounting:

- Input and output connectors are placed at the edges of the PCB to allow easy wiring and access.
- Mounting holes are provided at the corners of the board to securely fix the PCB inside an enclosure and improve mechanical strength.

Layout Quality:

- The overall layout is kept clean and well-organized to improve readability and airflow.
- Clear silkscreen labels are added to mark input, output, ground, and polarity, which helps in correct connections and easier use of the board.

Summary:

- A 5 V, 3 A DC–DC buck converter was successfully designed using the LM2596S-5 regulator in KiCad.
- The schematic and PCB layout were developed based on proper design calculations and component selection.
- Placement and routing were optimized to support high current operation, low noise, and reliable performance.
- Protection and decoupling components were included to improve robustness.
- The final PCB layout and 3D verification confirm that the design is manufacturable, stable, and meets all specified electrical requirements.