**Pcb guideline SMPS**

As a general rule of thumb, this tutorial will provide some detailed aspects of important **PCB design layout guidelines** that are essential for any kind of switch-mode power supply-based PCB design. You can also check out the [Design Techniques for EMI Reduction](https://circuitdigest.com/article/design-techniques-for-reducing-emi-in-smps-circuits) in SMPS Circuits.

First thing first, for designing a switch-mode power supply, one needs to have a clear indication of the circuitry requirement and specifications. The power supply has four important portions.

1. Input and output filters.
2. Driver circuitry and associated components for the driver especially control circuit.
3. Switching inductors or Transformers
4. Output Bridge and the associated filters.

In a PCB design, these all segments need to be separated in PCB and require special attention. We will discuss each segment in detail in this article

**Guidelines for Input and Associated filters**

The input and the filter section is where the noisy or unregulated supply lines get connected into the circuit. Therefore, the **input filter capacitors** need to be situated in an evenly spaced distance from the input connector and the driver circuit. It is essential to always use a short length of connection for connecting the Input section with the driver circuit.

**Guidelines for Switching Inductors and Transformers**

 One best way to **increase creepage** is by applying a PCB cutoff using a milling layer. Never use any kind of routing between the transformer leads.

