

## 7 Essential Design Guidelines for Flex PCBs

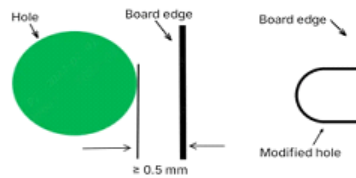
Aug 10, 2025



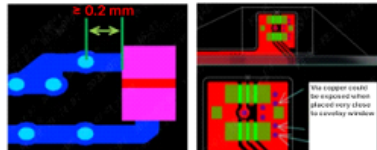
Flexible Printed Circuit Boards (Flex PCBs) play a crucial role in modern electronics by enabling compact and lightweight designs. [Flexible PCBs](#) are not new in the market, they are the part of almost every electronics item, which include small dimensions and fittings. Because bending the wires, fitting the components and enabling power in small handheld devices are not that much easy with rigid PCBs. They are specially designed and used for routing the power and display lines. Although the heavy electronics like CPU and GPU are still placed on [rigid PCBs](#) due to several design and reliability issues. The design and fabrication process requires engineers to follow specialized design principles. Based on JLCPCB's in-depth recommendations, here are 10 key guidelines to ensure reliable, manufacturable, and high-performance Flex PCB designs.

### 1. Ensure Adequate Hole and Via Clearances

**Through-Hole to Board Outline:** Under the DRC we have to maintain a minimum clearance of 0.5 mm between through-holes and the board. For a more practical approach, use U-shaped slots open to the frame to prevent structural failures.

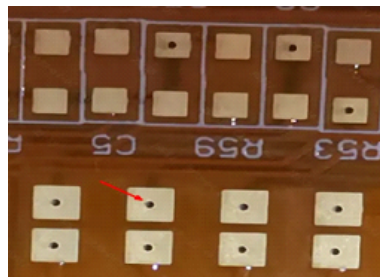


**Via-to-Solder Mask:** Keep vias at least 0.2 mm away from solder mask openings to avoid copper exposure. Exposure of copper may lead to short circuits and corrosion.



### 2. Avoid Via-in-Pad Designs

JLCPCB has via in pad technology with rigid and flexible boards. Via in Pad is possible in rigid structures because there is no issue of reliability. Rigid boards also contain BGA packages which increase the need of via in pad. But unlike rigid PCBs, Flex PCBs cannot be resin plugged, which can result in solder wicking and unreliable solder joints.



### 3. Prevent Oxidation on Large Copper Surfaces

## Rec

- Desi Man Aug
- How Boai Aug
- Sign Desi Aug
- SMT Beti Aug
- 7 Es Aug

If copper remained in air without a [solder mask](#), solid copper regions may trap air during the lamination of the coverlay, leading to oxidation under heat and pressure. To avoid this problem:

- Use hatched copper patterns to reduce surface area.
- Add solder mask windows to vent trapped air.



## 4. Reinforce and Strategically Place Pads

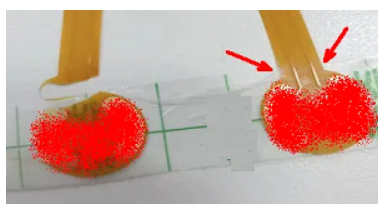
Independent pads, especially those overlapping on both sides, can easily detach since the FPC core is only 25 $\mu$ m thick. It's recommended to add copper reinforcement around the pad and connect pad corners to the copper area, and offset pads on opposite sides for better adhesion. The method is to design soldermask defined pads to prevent this, as the solder mask coverage on the pad's rim provides mechanical strength.

- Avoid isolated or overlapping pads, especially on thin 25 $\mu$ m cores.
- Connect pad corners to copper areas to enhance mechanical adhesion.



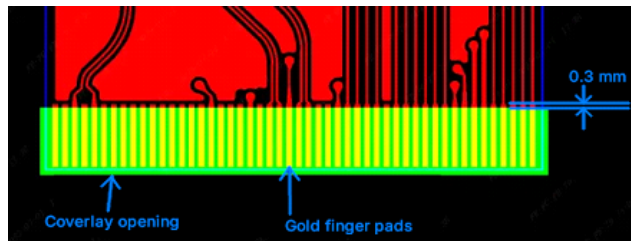
## 5. Design According to Coverlay Requirements

For flexible boards, [coverlay](#) acts as the solder mask. It needs to be pre-windowed before application. Ensure a 0.2mm gap between pads and adjacent traces, and a 0.5mm spacing between pads. Otherwise, bridged apertures should be used, accepting exposed traces. For tight spacing (<0.5 mm), expose connecting traces through a single window. Add anti tear routing lines if required.



## 6. Optimize Gold Finger and Connector Pad Designs

In the FPC usually a cable ends with a gold pad going to some connector, which is used to improve the signal integrity and in this way the cable can be easily replaced during repair. For this we can use solder mask-defined pads for connectors. Shrinking gold finger lengths by 0.2 mm helps and avoid micro-shorts from laser cutting. Ensure the coverlay overlaps connector pads by at least 0.3 mm for mechanical durability.



## 7. Validate Thickness and Impedance Requirements

FPC impedance calculations using simulation software can be inaccurate. JLCPCB's empirical experience for designing line widths is available for reference, but always validate with a prototype first. This parameter applies to double-sided boards with 0.11 mm thickness. The impedance calculations are altered with change in thickness and material used for fabrication. But to get a general idea here is the table:

- Be aware that Flex PCB thickness is affected by coverlay, copper layers, and PI material.
- For impedance-controlled designs, avoid relying solely on simulation tools. Always consult manufacturer data and prototype to verify performance.

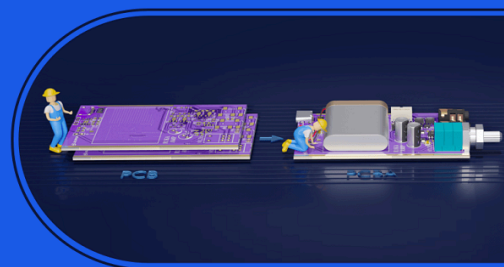
JLCPCB provides Flex PCB prototyping and expert manufacturing advice to help bring your designs to life.

Target Impedance	Trace width / spacing ( $\mu\text{m}$ )	Reference Layer Type	Cover Material
Single-Ended 50 $\Omega$	56 / 50	Solid	Coverlay
	80 / 150	Hatched	Coverlay
	50 / 150	Solid	Coverlay & EMI Film
	72 / 150	Hatched	Coverlay & EMI Film
Differential 90 $\Omega$	79 / 50 / 150	Solid	Coverlay
	108 / 150 / 150	Hatched	Coverlay
	63 / 150 / 150	Solid	Coverlay & EMI Film
	71 / 150 / 150	Hatched	Coverlay & EMI Film
Differential 100 $\Omega$	72 / 150 / 150	Solid	Coverlay
	95 / 150 / 150	Hatched	Coverlay
	52 / 150 / 150	Solid	Coverlay & EMI Film
	65 / 150 / 150	Hatched	Coverlay & EMI Film

## Affordable and High-Quality PCB Manufacturing

Save Time & Money by choosing All-in-One PCB fabrication, assembly, and components supplier.

[Get Quote Now >](#)



## Conclusion

Designing Flex PCBs requires careful attention to mechanical and thermal factors. By following these 7 critical guidelines, engineers can significantly improve the durability, and manufacturability of their flexible circuit designs. But these 7 guidelines are just a getting started guide to FPC, in future we will come with such amazing guides on FPC and design related issues. We will keep sharing in-depth guides related to PCB issues on our blog page.

GET FREE QUOTE ►

#### COMPANY

[About JLCPCB](#)

[News](#)

[Blog](#)

[How we work](#)

[Quality Management](#)

[Security](#)

[Certifications](#)

#### SUPPORT

[Help Center](#)

[Shipping Info](#)

[How To Order](#)

[How To Track](#)

[Contact Us](#)

#### NETWORK SITES

[EasyEDA - PCB Design Tool](#)

[JLC3DP - 3D Printing&CNC Machinir](#)

[JLCCMC - Mechatronic Parts](#)

[OSHWLAB - Open Source Hardware](#)

© 2025 JLCPCB.COM All Rights Reserved