SIERRA CIRCUITS

FLEX DESIGN GUIDE

Table of Content

Table of Content

Table of Content

Chapter 1: Flex PCBs and its materials

1.1 What is a flex PCB?

Flex PCBs are flexible circuit boards with very thin substrates and high level of bendability, tensile strength, and physical flexibility. These boards are also called flex boards, flex circuit boards and flexible electronics. Flex boards can be found in almost any high-end electronic device. These boards are widely used in medical devices and fitness wearables. Cameras and smartphones also incorporate flex circuits.

1.2 Types of flex PCBs

Flexible PCBs are classified into single-sided, double-sided, and multi-layered boards.

1.2.1 Single-sided flexible circuit boards

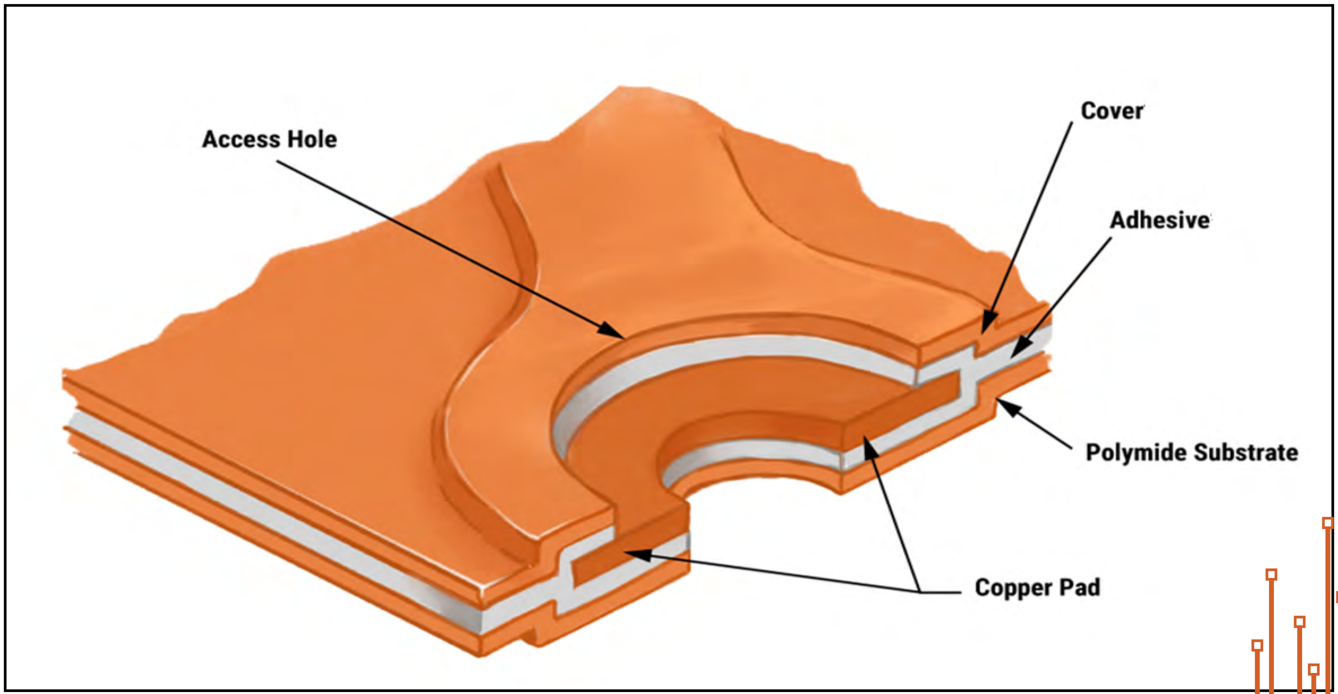


Image 1: Single-sided flex circuit board

Single-sided flex boards are the most basic type. These boards consist of a single conducting layer on a flex substrate. A flexible polyimide is laminated to a thin sheet of copper. Holes may be drilled through the substrate to allow lead of the components to pass during the soldering process. A polyimide coverlay can be used for the insulation and environmental protection of the circuit.

1.2.2 Double-sided flexible circuit boards

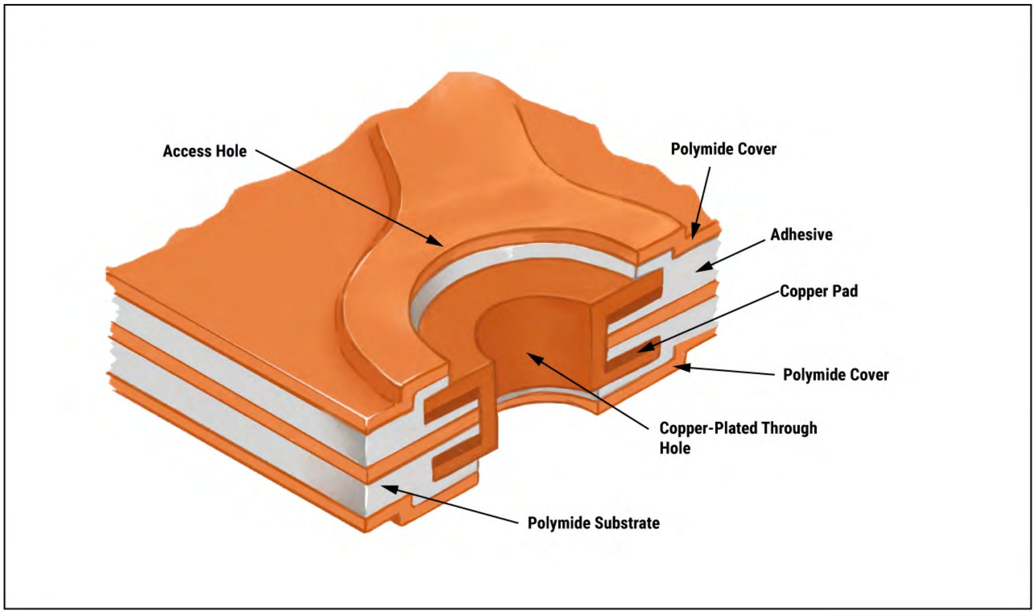


Image 2: Double-sided flex circuit board

Double-sided flex PCBs have two conductive layers (one on each side of the flex substrate). Plated through holes or vias establish electrical connection between the layers.

1.2.3 Multilayer flexible circuit boards

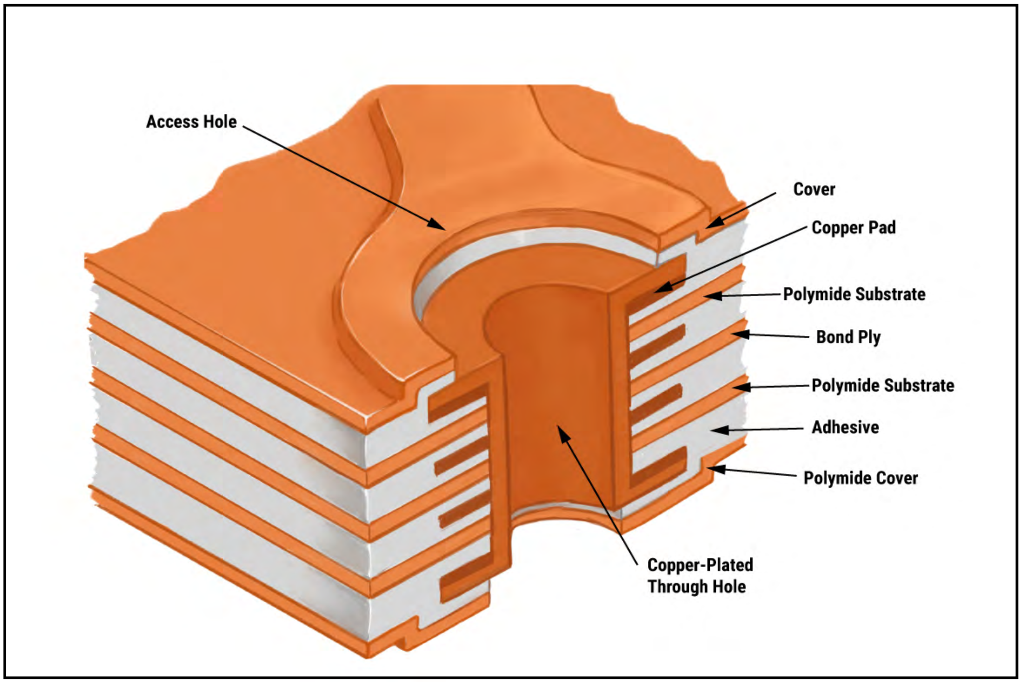


Image 3: Multilayer flex circuit board

Multi-layered flex boards consist of more than two copper conductors. Like double-sided flex PCBs, the conductive layers are interconnected through PTHs or vias. Multilayer boards are an effective solution when confronted with design challenges such as: unavoidable crossovers, specific impedance requirements, elimination of crosstalk, additional shielding and high component density.

1.3 Advantages of flex PCBs

These boards have several advantages over rigid PCBs.

- Flex boards can be fabricated in different shapes. This eases out the electronic device assembly process.
- Flexible circuitry minimizes the connection points and simplifies the assembly. Therefore, it eliminates the chances of interconnection defects like poor solder joints. This makes them more reliable when compared to rigid boards.
- Thinner and lightweight than their rigid counterparts.
- Offers superior resistance to vibrations and other disruptions within harsh environments.
- Flex PCBs make use of HDI technology for the miniaturization of devices.
- Better airflow and heat dissipation than many other PCBs.

1.4 Flex PCB materials

Polyimide is the material used for both flex core layers and coverlay layers. Flex substrates offer better material properties when compared to standard FR4 rigid materials. The thickness of flex materials is uniform throughout the substrate. These materials also offer improved DK value ranging between 3.2 and 3.4. The lack of woven glass reinforcement reduces variations in Dk. Typically, the thickness of the flex layers range between 1 and 5 mils.

Polyimide flex cores are cladded with rolled annealed copper. This copper is very thin and is suitable for both dynamic and static applications. 0.5oz (0.7mils) or less copper is more commonly used in these boards.

There are two major types of flex materials:

1. **Adhesive-based -** where the copper is bonded to the polyimide with acrylic adhesive.
2. **Adhesive-less -** where the copper is cast directly onto the polyimide.

Adhesives are used to laminate the copper layer with the polyimide. The use of adhesives may cause cracks in the copper plating within via holes because acrylic adhesives can become soft when heated. Consequently, when designing for adhesive-based materials, it’s important to incorporate anchors and teardrops in your design.

Here are a few disadvantages of using adhesive based materials:

1. Cause cracks in via holes.
2. Presence of adhesives makes the copper clad laminate thicker. So, eliminating the adhesive bond layers make way for thinner laminates.
3. Adhesive based materials are prone to absorb moisture from the environment. Hence, it is not suitable to use this type of material in a system that is exposed to the outside environment.
4. The core thickness of adhesive based material can reduce after the fabrication proess. This leads to dimensional errors.

To address the issues associated with adhesive based material, adhesiveless materials are used.

Following are the features of adhesiveless materials:

1. Reduced flex thickness due to the elimination of the adhesive layers.
2. Improved flexibility due to reduced finished flex thickness.
3. Improved controlled impedance signal characteristics.
4. Better temperature ratings when compared to adhesive-based material.
5. Well-suited for harsh environment applications.

1.5 Applications of flex PCBs

Flexible and rigid-flex PCBs were originally used within the military industry as they require durable, reliable, lightweight 3D circuitry. Now, flex boards are found in nearly every industry. They are used in devices we use on a daily basis—from phones to computers. They are also found in cars, trains, and airplanes to satellites, missiles and radios. In fact, NASA’s Mars Rover which is 140 million miles away from the earth has flexible circuits within it.

Chapter 2: How to prepare the board outline

2.1 Board outline:

The shape of a PCB depends on the design of the device that it goes into. Once the shape is finalized, test your ideas by cutting out a piece of paper in the shape of your proposed board. Use cardboard to represent stiffeners and rigid areas.

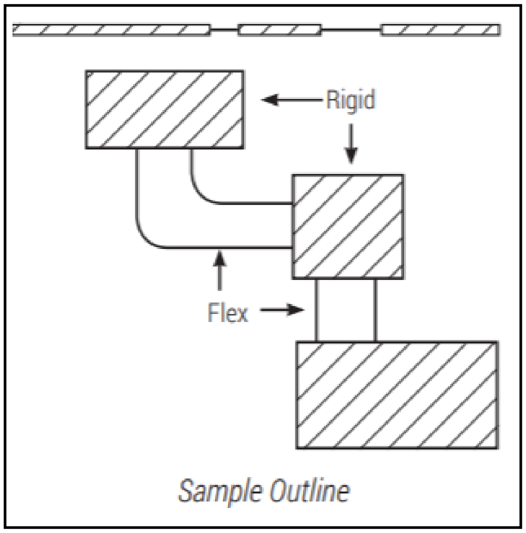


Image 4: Sample board outline

Start your layout by drawing the board outline on a piece of paper. Mark the location of varied thicknesses. Now, think about preliminary component placement and determine whether those components require stiffeners. Next, mark the stiffeners and rigid areas. This will give you a rough outline of your upcoming board design. It is essential to calculate the bend radius so that thicknesses could be marked precisely. If not planned properly, it will affect your board's flexing capabilities.

2.2 Factors affecting the thickness of flex board

Avoid unnecessary circuit thickness, which hinders flexible capabilities. The thickness of the flex area is determined by the bend radius needed. If part of the flex circuit needs to be be thicker, add a stiffener.

The following factors determine the required thickness of a circuit.

- Material thickness
- Copper layer count
- Base copper weight
- Adhesive thickness
- Dielectric thickness

Chapter 3: Calculating the bend radius

Bend radius is the measurement of the degree upto which the flex area of a circuit board can bend. The minimum degree of bendability for the flex area must be properly identified early in the design phase. This ensures your design can survive for the required number of bends without damaging the copper. Bend radius is calculated based on the number of layers the flex stack-up has.

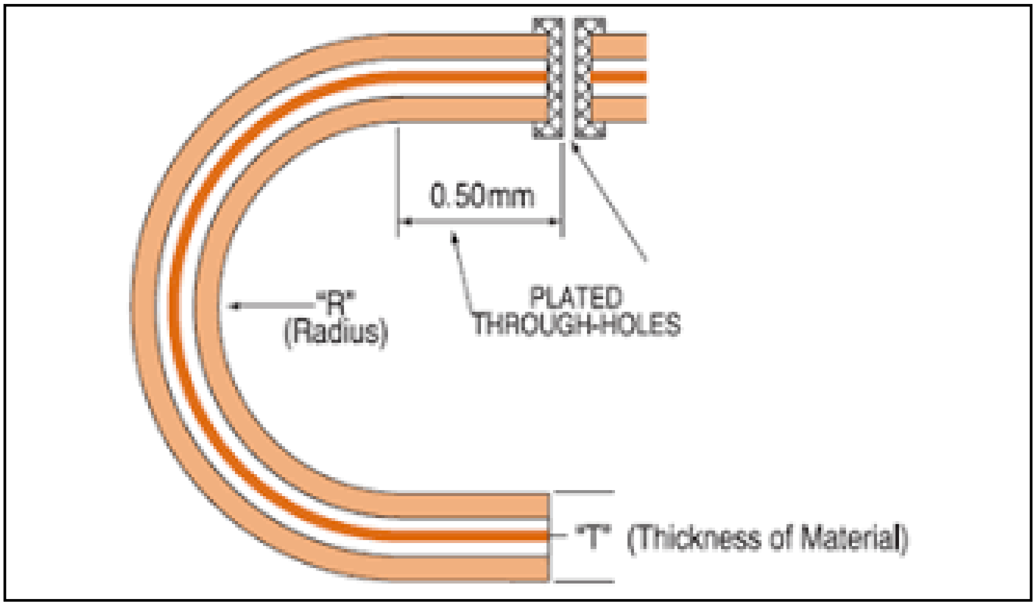


Image 5: Bend radius

3.1 Calculation of bend radius

Number of layers	Bend radius
1 (single-sided)	Flex thickness x 6
2-layer board (double-sided)	Flex thickness x 12
Multilayer board	Flex thickness x 24

***thickness in mils/mm**

It is vital to know two things about flexibility: how many times the PCB will be flexing, and to what extent the PCB will flex. The amount of times the flex board can bend determines whether the board will be a static or dynamic board. Note: Plated through-holes (PTH) should be at least 0.5mm away from the bend area as shown in Image 5.

3.2 Static flex PCBs

A static board is considered bend-to-install and will flex less than 100 times in its lifetime.

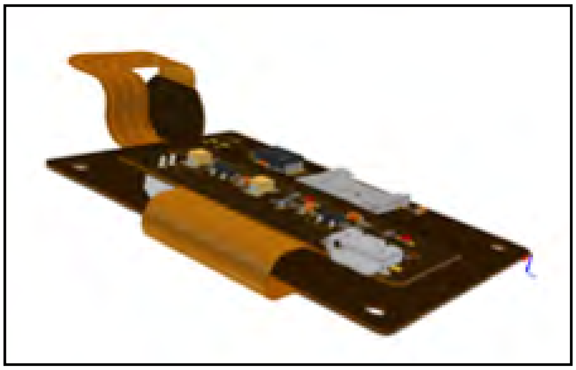


Image 6: Static flex board

It is generally bent during the assembly process. These boards are not intended to flex during the operation of the end product. Static boards may have a slight bend that allows the board to fit into its packaging. The image below shows a board with static ribbon (flex PCB).

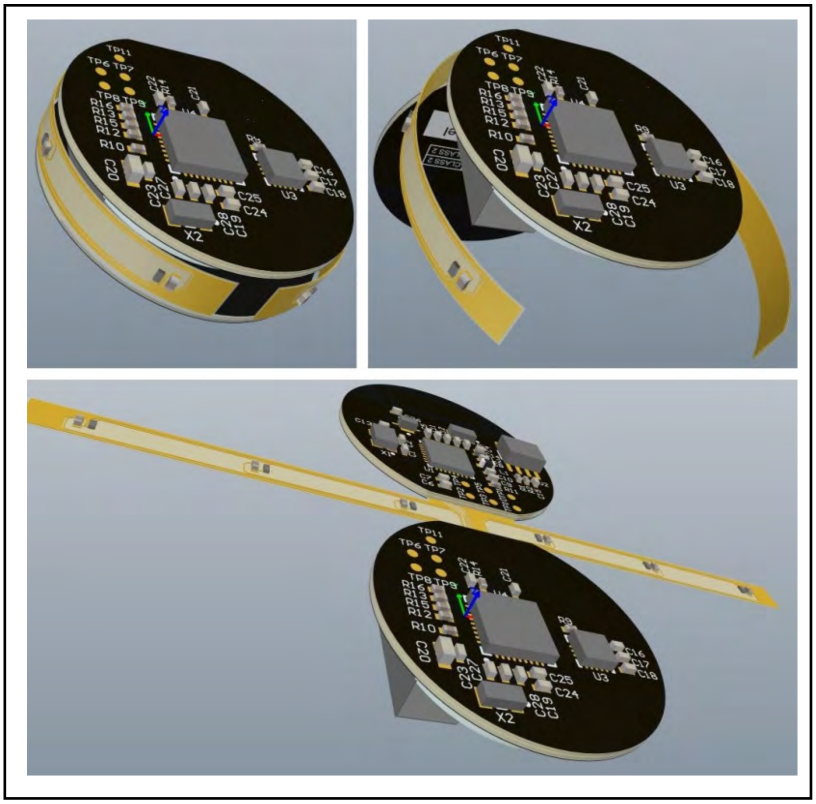


Image 7: PCB with a static ribbon

The flex ribbon in the above example is only bent during the assembly process of the electronic device. Once the board is installed on to the end product, the board will remain static.

3.3 Dynamic flex PCBs

A dynamic flex is a board that is regularly flexed (during the system operation). A dynamic board's design needs to be more robust in nature, as flexing will occur on a regular basis—and will need to withstand tens of thousands of bends. These PCBs are used in very harsh conditions such as spacecraft and military applications.

Image 8: Dynamic flex board

3.4 Challenges in dynamic flex PCB design

Dynamic flex boards are capable of solving many interconnect and packaging challenges in the applications that require repetitive flexing. The designers should ensure that the copper circuit on these boards should not fracture during its operation. These fractures in the copper circuits result in an open circuit.

3.4.1 Minimum bend radius

The minimum bend radius is one of the critical aspects to achieve design success. Recommended minimum bend radius is 100 times the finished thickness of the dynamic flex circuit. For example, a flex circuit with a finished thickness of 0.006” will need a 0.600” minimum bend radius or 1.200” minimum bend diameter to ensure its reliability.

3.4.2 Copper type and grain orientation

Flex PCBs generally use rolled annealed copper. Rolled annealed copper is created by subjecting electro-deposited copper to the rolled annealed process. The grain structure is transformed from vertical to elongated horizontal structure. This improves the ductility of copper, making it suitable for dynamic applications.

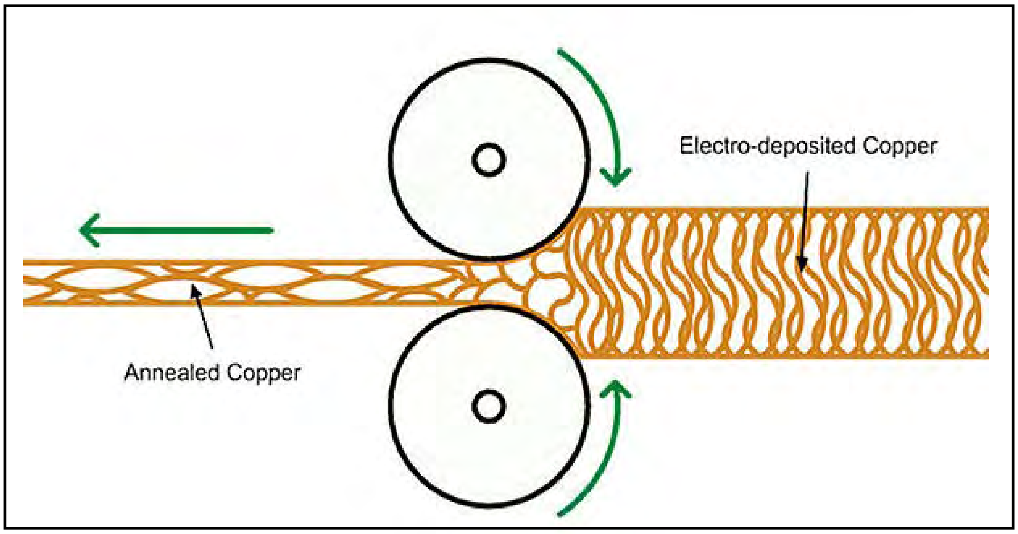


Image 9: Rolled annealed copper

RA copper is available in a variety of thicknesses ranging from ¼ ounces up to 2 ounces with ½ ounce and 1 ounce being the most commonly used.

3.4.3 Layer count and neutral bend axis

Layer count of dynamic flex boards are limited as per the IPC 2223 standard. The optimum configuration is a 1-layer construction. It allows the copper circuits to be close to the neutral bend axis (The neutral bend axis is the plane where there is minimum tension or compression when the circuit is flexed). This ensures that the copper is subjected to the least amount of compressive and tensile forces possible.

A 2-layer construction is permissible provided that a thin adhesiveless flex core of thickness 0.001" or lesser is used between the two layers. This ensures that the distance between the circuits and the neutral bend axis is minimum.

3.5 Best practices for designing bend areas

- Avoid 90 degree bends if possible. Tighter bends increase the chance of circuit damage. Gradual bends are safer for the circuit.
- Always measure the bend radius from the inside surface of the bend.
- Place conductors smaller than 10 mils inside the neutral bend axis, as they tolerate compression better than stretching.
- Avoid plated through-holes within the bend area.
- Conductors running through a bend need to be perpendicular to the bend axis.
- Use staggered conductors in multilayer circuits to increase the effectiveness of the circuit.

3.6 Methods for increasing flexibility of flex boards

There are a couple of different methods of increasing the flexibility of a flexible circuit. The most common method is to reduce the overall thickness of the flexible dielectric material because its thickness directly influences the flexibility.

The second method is to reduce the copper thickness of the traces and moreover the thickness of the plane layer. One way of reducing copper on a plane layer is by cross hatching the plane. Typically, we recommend 0.015” wide signals with 0.025” spacing for the cross hatched plane layers.

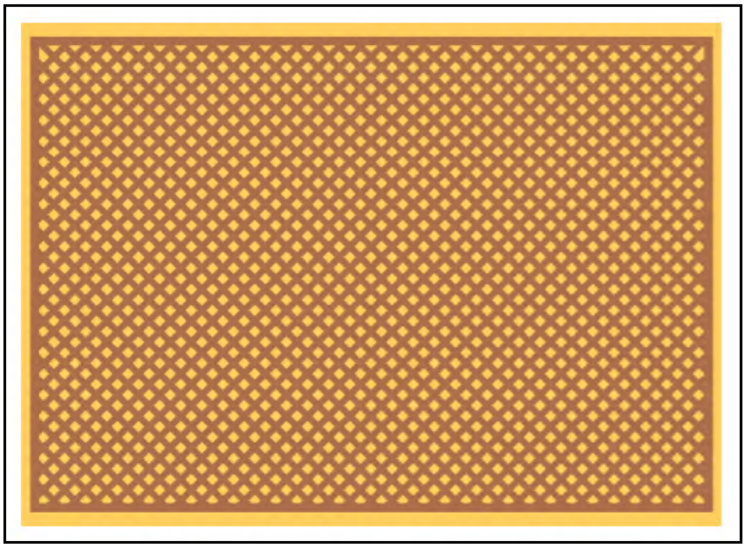


Image 10: Cross-hatched copper plane

Ground and power planes are usually cross hatched in flex PCBs in order to maintain or increase the flexibility. A ground or power plane that is completely flooded doesn’t bend.

Chapter 4: Flex routing

Routing is the process of laying copper traces between the nodes. This conductive path is defined by placing tracks, arcs, and vias to establish a connection between two nodes.

The performance and longevity of a flexible circuit board can be directly attributed to the layout of the circuitry. Keep the following points in mind while routing a flex PCB.

4.1 Guidelines for routing flex PCBs

- Always opt for a larger bend radius.

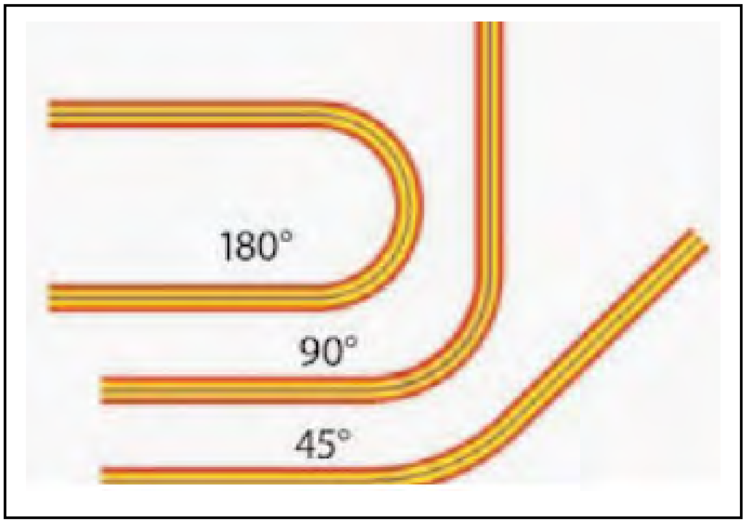


Image 11: Bend radius

- When designing multilayer flexible PCBs, stagger traces on the front and back. Stacked traces will not only reduce the flexibility of your circuit, it will increase stress contributing to the thinning of copper traces at the bend radius.

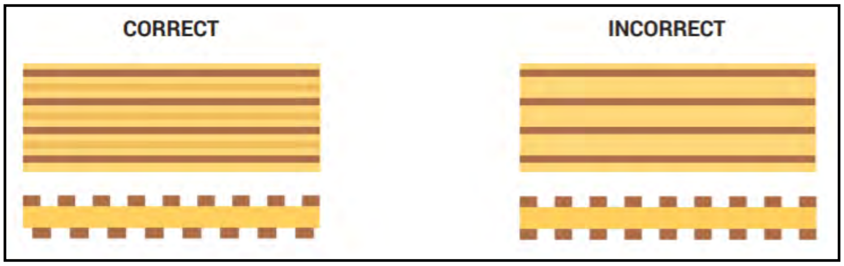


Image 12: Stacked and staggered

- Use curved traces instead of traces with corners.

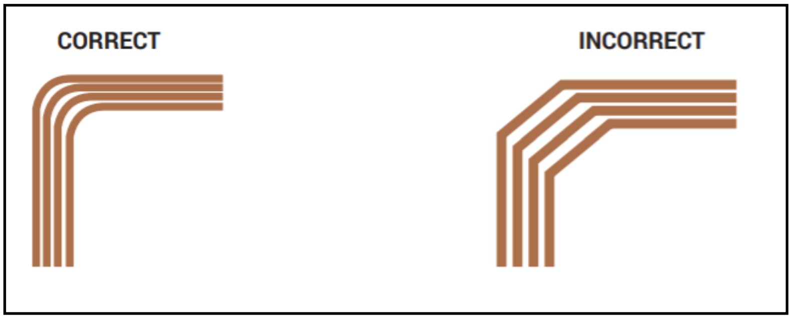


Image 13: Curved and sharp edged

- Traces should be perpendicular to the bend area.

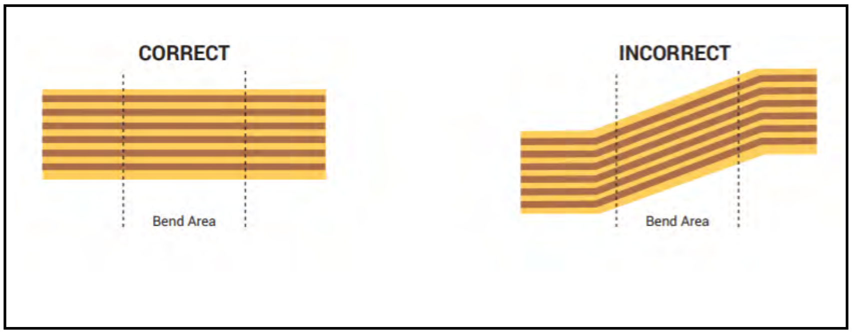


Image 14: Traces perpendicular to bend area

- Trace entering a pad forms a weak spot in which the copper might get fatigued over a point of time. It is always recommended to taper down the pads (as shown below) towards the end in which they are connected to the traces.

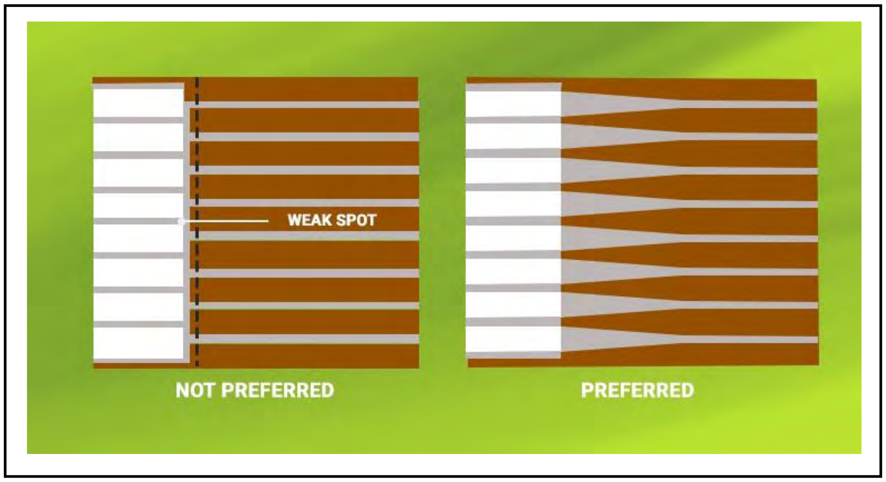


Image 15: Non tapered and tapered pads

- Copper is prone to be detached from the substrate due to the bending of the flex circuit. It is vital to provide mechanical support for the exposed copper. Through-hole plating in the vias inherently provides mechanical support to the vias in the flex region. For this reason, additional through-hole plating of up to 1.5 mils is recommended for rigid-flex and flex circuits. Usually, SMT pads and non-plated through-holes are referred to as unsupported, and they require additional measures to prevent detachment.

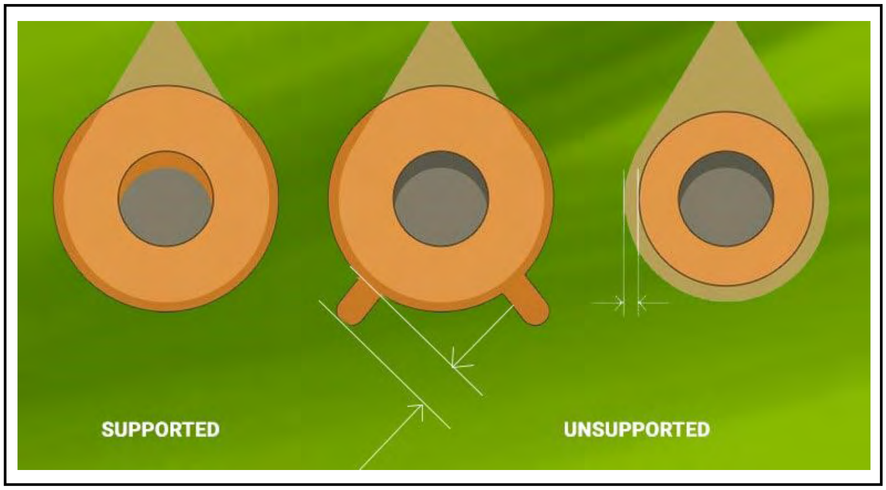


Image 16: Supported and unsupported pads

4.2 Pad design for outer layers

Pads are the exposed region of metal on a circuit board on which the component leads are soldered. Pads are prone to lift off due to the flexible nature of the substrate. This can be avoided by incorporating anchors or spurs encapsulated with coverlay. It is also highly recommended to make the pads as large as possible. During dynamic bending, anchors help stabilize the outer layer.

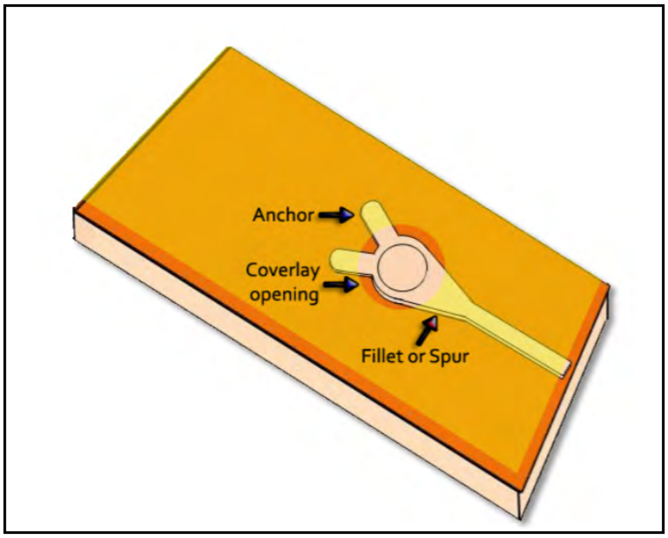


Image 17: Anchored trace on outer flex layer

18

It is advisable to use teardrops on all flexible PCBs. A teardrop is extra copper at the junction of a pad and a trace. It is also used when there is a transition of copper trace from thick to thin. For example, if a part of your trace width is changing from 10 mils to 4 mils, then a teardrop is added at the transition point to reduce any stress or hairline cracks. Teardrops can reduce and even eliminate potential stress concentration points on the PCB.

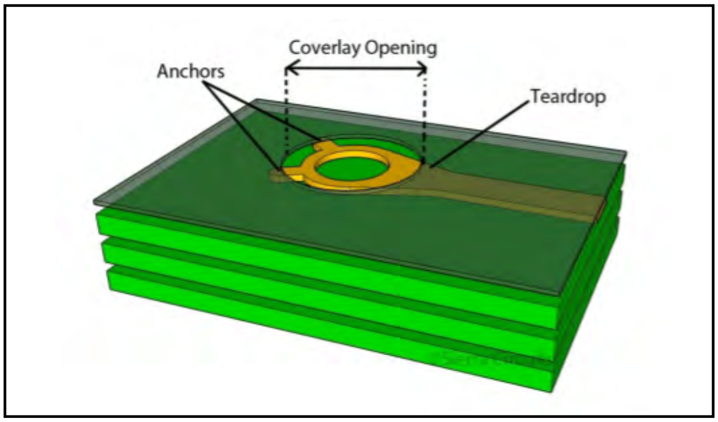


Image 18: Tear dropped trace

Chapter 5: Designing and manufacturing vias in flex PCBs

Most of the flex PCBs have multiple conductive layers that are electrically connected. The connection through the dielectric layers are established through vias that are either mechanically drilled or laser drilled. IPC 6013 standard defines various considerations related to flex vias. Here are a few points to consider:

5.1 Flex via design

Vias are at greater risk of peeling when implemented in flex designs.To reduce this risk:

- Make the annular rings as large as possible
- Vias should be tear dropped
- Adding tabs or anchors to vias will also help prevent peeling

5.2 Via locations

- Vias are not reliable in areas that will flex and bend
- In a dynamic application, flexed vias can crack very quickly
- Vias are okay over a stiffener, but vias just off the edge of a stiffener are at risk of cracking. A minimum of 50 mils is kept from the edge of the stiffener

5.3 Rigid-flex vias

Hole to flex distance is the distance between vias and the rigid-flex transition area. The hole to flex distance should be 50 mils for the boards that require high reliability. Sometimes, this hole to flex distance can be 30 mils for commercial applications.

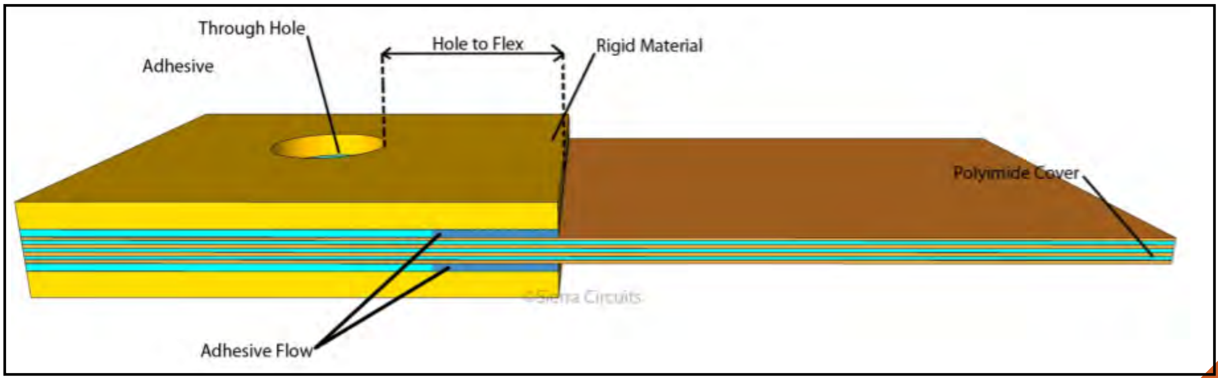


Image 19: Hole to flex distance distance in rigid-flex board

5.4 Annular ring in flex PCBs

Annular ring is the area of copper pad around a drilled and finished hole (copper plated via). There should be enough copper to form a solid connection between the copper traces and the via in a multilayer PCB.

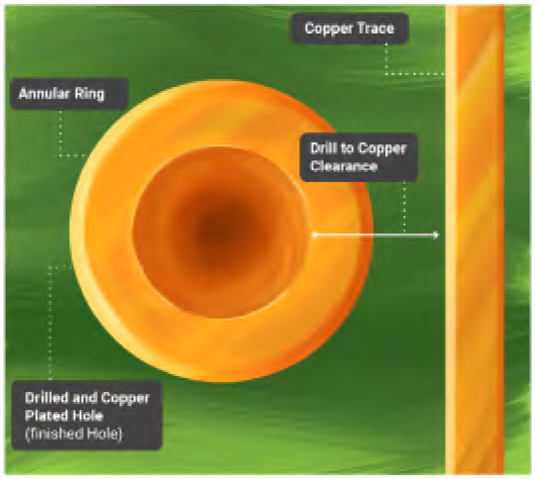


Image 20: Annular ring

The main purpose of an annular ring is to establish a good connection between a via and the copper trace.

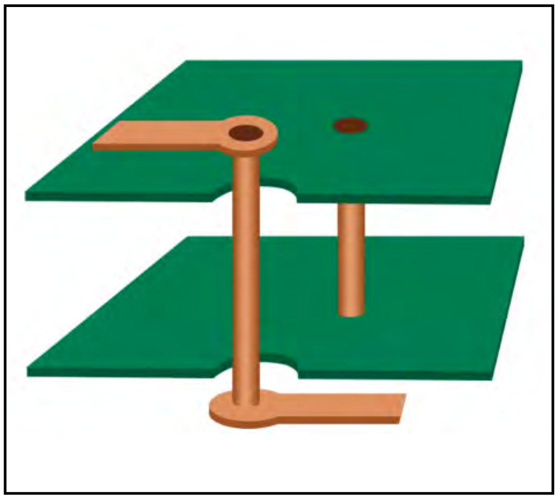


Image 21: Annular ring connection between via and trace

Minimum annular ring should be 8 mils for flex PCBs.

5.5 Plating flex circuit boards

For double-sided flexible boards, Sierra Circuits uses a pattern plating process called “pads only plate” for plating through-holes. In order to accomplish this, we drill the flexible copper clad dielectric material and then image the pads around the drilled holes at (drill diameter) D + .003” or better.

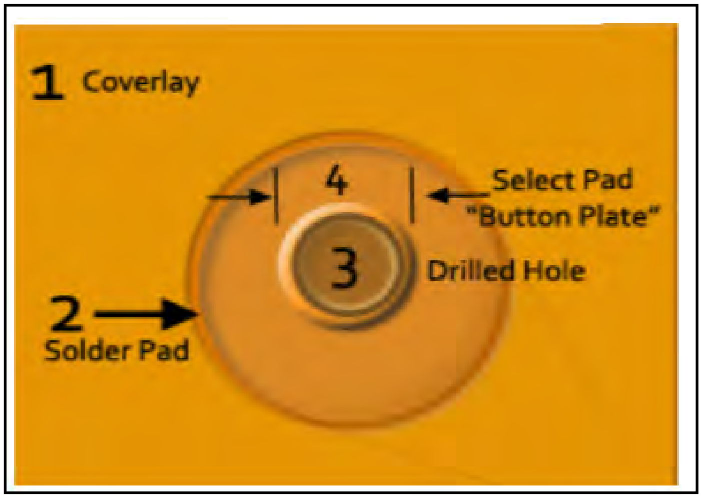


Image 22: Hole plating in flex PCBs

After plating the holes with approximately 0.001” of copper we again image the final circuitry pattern and the unwanted copper is etched. An additional D + 0.014” of pad is needed for this etching process. Below table helps in finalizing the drill size for different layer types.

Layer type	Standard boards (mils)	Advanced boards (mils)
Flex	drill + 14	drill + 10
Outer layer rigid	drill + 10	drill + 6
Inner layer rigid	drill + 14	drill + 10

5.6 Drill to copper

When designing flexible boards, it is crucial to keep drill to copper in mind. Drill to copper is the distance between a hole (via or non plated hole) and the nearest copper feature. The flexible dielectric materials used to manufacture flex products are not as dimensionally stable as standard rigid materials. Flexible material moves, shrinks, and contracts during the manufacturing process which makes drill to copper a critical factor when designing flex boards.

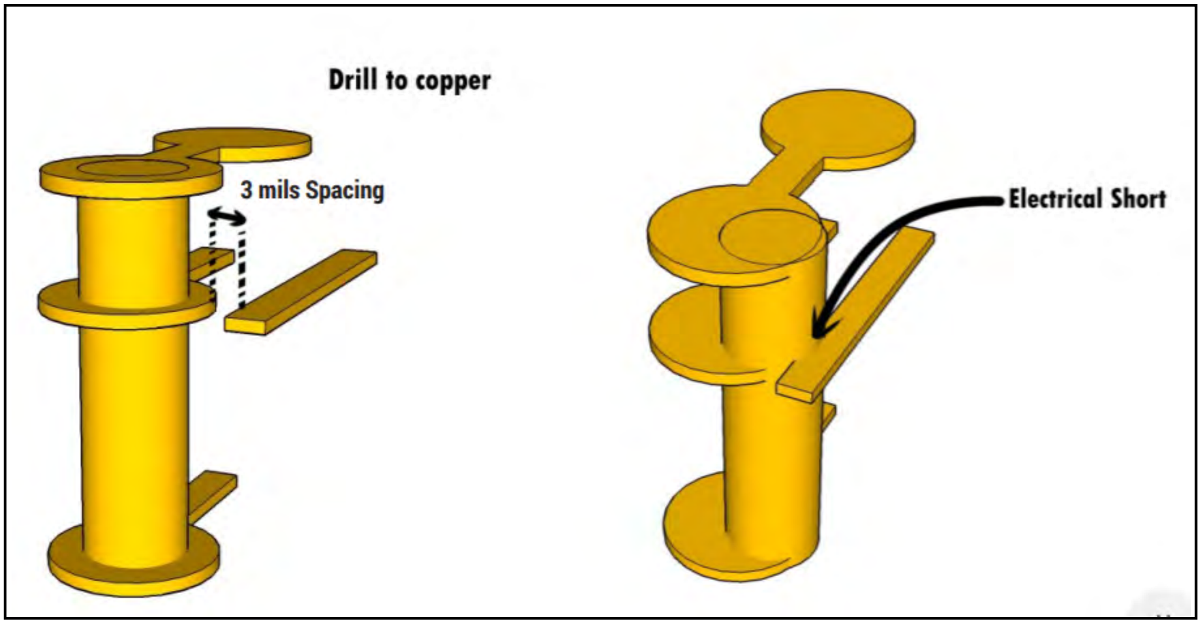


Image 23: Drill to copper

Finished hole is the final hole obtained after metallization and surface finish.
The finished hole to copper clearance will be the drill to copper clearance including the copper plating thickness. Typically, drill to copper clearance is 8 mils.

Finished hole to copper clearance = drill to copper clearance + plating thickness x 2.

So, if the drill diameter is 6 mil and the plating thickness is 1 mil, then the finished hole to copper clearance = 6 + 1 x 2 = 8 mils.

For drill to copper, always drilled hole edge is considered. In order to maintain accuracy in layer alignment, it is important to keep the drill to copper distance around 8 mils.

To quickly run a DFM check on your designs check out our **[Better DFM tool]**.

Chapter 6: Designing and assembling rigid-flex PCBs

A rigid-flex PCB is a combination of rigid and flexible materials. Here, one or more flex circuits are used to connect rigid sub-circuits. The flexible section of a rigid-flex board generally features more than one layer.

Rigid-flex designs may be expensive to fabricate, but they can essentially save costs during electronic system assembly.

6.1 How do rigid-flex PCBs cut assembly costs?

Using rigid-flex boards can make way for both direct and indirect cost savings. Direct cost savings mainly comes from reduced **BOM (bill of materials)** and inventory. Indirect cost savings are from reduced assembly costs and improved reliability.

You can quickly validate your BOM files with our **BOM Checker** tool.

Let us assume that a product has 6 interconnected rigid PCBs (a power board, two control boards, and three display boards). Interconnections among these boards would require wire harnesses and connector pairs. Now, let us have a look at how using rigid-flex PCB could reduce direct and indirect costs.

6.1.1 Direct cost savings

A single rigid-flex PCB with 6 rigid sections could be used to replace the entire assembly of 6 rigid boards within the electronic device. It also replaces wire harnesses and eliminates the requirement of connector pairs. This inventory reduction leads to direct cost savings.

6.1.2 Indirect cost savings

Since there are no wiring harnesses involved in rigid-flex PCBs, the cost incurred in procurement and assembly is saved. Also, no wiring harnesses mean there are no wiring errors which increases the reliability of the product. This eases out the testing procedure which also reduces the production costs.

6.2 Materials used in rigid-flex PCBs

Rigid-flex PCBs use a combination of standard rigid and flex materials. The materials include core, prepreg, copper foil, flexible laminates, cover layers, and bond plies. The PCB material used in flex sections can be just a few microns thick but can be reliably etched. This often makes them preferable over rigid PCBs in satellite and aerospace applications.

No flow prepregs are one of the most critical components in rigid-flex manufacturing. This type of prepregs prevents the flow of epoxy resin onto the flexible sections of your PCB. Flex materials are less dimensionally stable than the rigid materials that they are stacked with.

6.3 Rigid-flex design rules

All of the flex rules apply to the flex portion of rigid-flex. For the rigid portions, most of the rules are the same as a rigid PCB. The exceptions are defined below:

- Place the flex layers in the middle of the stack-up and use an even number of layers.
- Drill to copper should be at least 10 mils.
- To allow multilayer flex to bend in a tight radius without deformation, a technique called "bookbinding" is used and the layers are manufactured in progressively longer lengths around the outside bend radius.

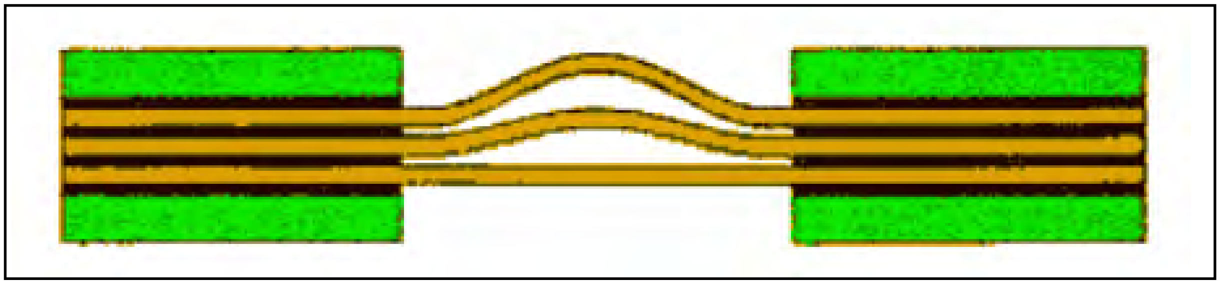


Image 24: Bookbinding construction

- Clearly name the rigid and flex sections.

6.4 Assembling flex and rigid-flex PCBs

- Solder joints can weaken if components are in areas where the board will flex.
- If components need to be close to a flex area, consider adding stiffeners.
- If you have a stiffener or rigid area for mechanical reasons, try to place components over the stiffener or rigid area.
- Flex circuits typically have SMT components on only one side of the board.
- Rigid-flex circuits typically are stiffened along most of their surface, with relatively small areas left unstiffened-the hinges or flexible arms.
- Depending on component size, surface mount areas do not always require a stiffener.
- Apply stiffeners to the opposite side of SMT components.
- Apply stiffeners to the same side as connector or through-hole components.
- The pre-bake cycle eliminates any moisture the board has retained, and allows for improved assembly yields and reliability. However, if boards are assembled immediately after manufacturing, there is no need for pre-bake. Since Sierra Circuits manufactures and assembles in-house, we eliminate the need for pre-bake.

6.5 Stiffeners

Single sided, double sided, and multilayer flex circuits can be stiffened in specific areas by adding localized rigid material called stiffener. This can add support for mounting components, increasing strength, thickness and rigidity. Kapton and FR4 materials are commonly used for stiffeners. These materials can be attached with thermally cured acrylic adhesive or pressure sensitive adhesive. Stiffeners should overlap bared coverlay by 0.030" to relieve stress.

Image 24.1 Stiffener considerations

6.5.1 Stiffener considerations

- Maintain the same stiffener thickness when using multiple stiffeners to lower cost.
- Stiffeners should come to at least two edges of the board.
- They reinforce solder joints and increase abrasion resistance.
- Stiffeners can be used for strain relief and heat dissipation.

6.6 Tear guards

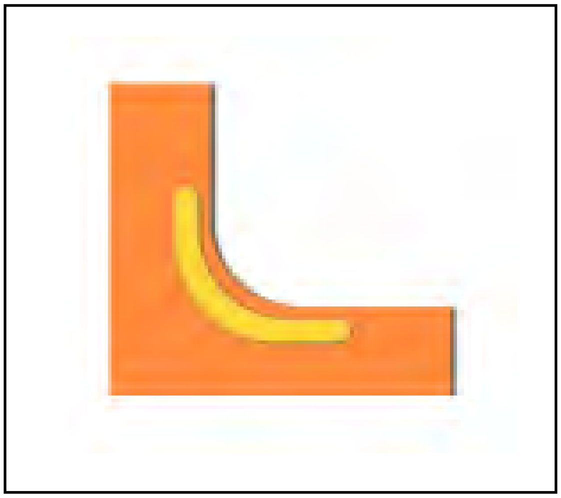


Image 24.2 Tear guards

We recommend the use of tear guards which will help to reinforce the flex material along the inside bend radius. This will help prevent tearing of the flex material. Avoid any discontinuation of materials close to the bend area and try to use a liberal bend radius avoiding sharp corners.

Chapter 7: Fab drawings for flex PCB

To successfully design a flexible PCB, it is important for the designers to have a basic understanding of the flex drawing requirements. Let us have a look at a few of these.

7.1 Flex PCB stack-up

Flexible PCB stack-up drawing will depict the thickness of each layer including the copper thicknesses of the conductive layers. This should also specify which layers are of rigid material and which layers are of flexible material including copper weights. Sierra Circuits can assist you in designing your flex stack-up.

You can check out our **Stackup Planner** to get an accurate and detailed PCB stack-up.

7.2 Dimensional drawing and tolerances

The dimensional drawing of a flexible PCB design provides the following information:

- Location of PCB stiffeners.
- Thicknesses of each section of the board and the materials to be used.
- Type of flex board (static or dynamic)
- Locations where the board rarely flexes and frequently flexes.

7.3 Design specifications a PCB manufacturer needs to know

A PCB manufacturer expects the following range of specific details from a designer:

- Class type (class 1, class 2, class 3), wiring type, and installation use requirements
- Flexible copper clad material to be used
- The cover coat material
- Minimum conductor width and spacing
- Maximum board thickness
- The minimum size of plated through holes
- Electrical test requirements
- Color of coverlay
- Color of silkscreen
- Board markings such as part number, version, and company logo
- Special packaging requirements (if any)
- Packaging and shipping needs

7.4 Drill symbol chart

The drill symbol chart summarizes the drill hole information of the board. An example of a drill symbol chart is shown below. The standard finished hole size is +/- 0.003" but this is never assumed, so this measurement must be stated on your design drawing.

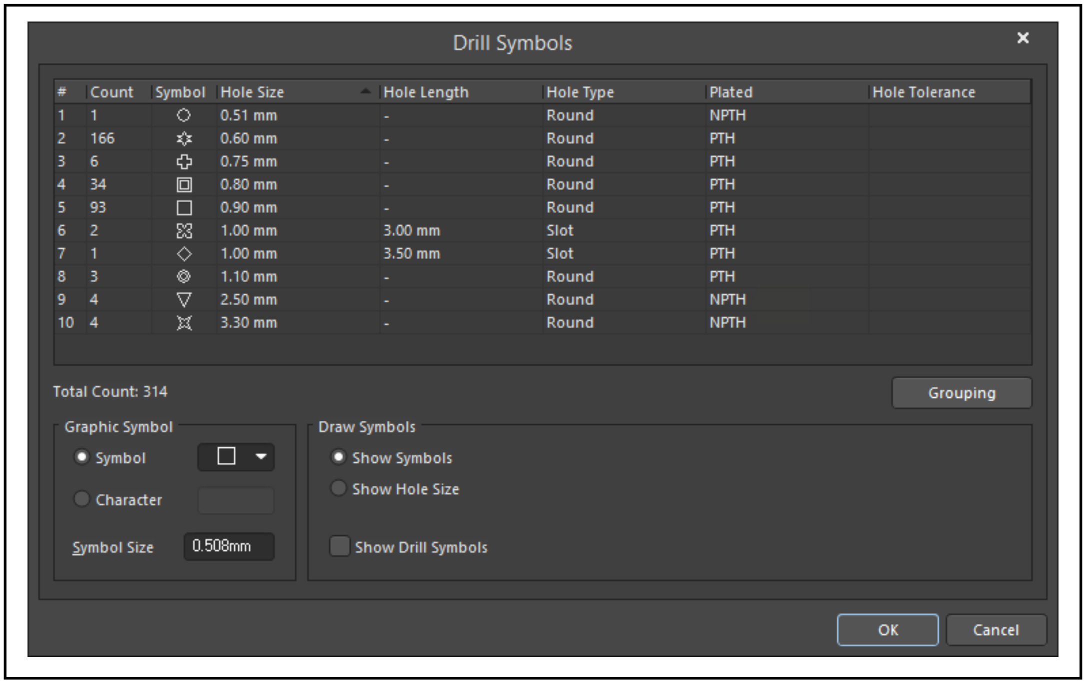


Image 25: Drill symbol chart

7.5 Flexibility (bend radius)

The flexibility of a flex PCB is determined by the bend radius of the flex material used. Bend radius is the minimum angle the flex region can bend. Knowing the number of times your flex PCB will bend is crucial in your design. If a PCB is bent more times than the design allows for, then the copper will begin to stretch and crack.

7.6 Plating requirements

Plating the through-holes or vias is a requirement for multilayer PCBs. It is very important to mention the type of plating that your flex board requires. The available types of plating are:

- **Panel plating:** This method of plating deposits copper on the entire panel. This type of plating is generally performed before circuit imaging.
- **Pattern plating:** This type of plating deposits copper on the selected areas of flexPCB.
- **Pads only plating:** It is a type of pattern plating. A photoresist covers the entire board panel exposing the pads surrounding the vias. As a result, only the vias and
- exposed pads get plated.

7.7 Testing requirements

PCBs undergo various tests before it reaches the designer. It is important to mention the physical and electrical testing requirements (test type and frequency). Over specifying the test requirements might increase the overall circuit cost. Basic tests include:

- Dimensional checks
- Electrical continuity
- Ionic cleanliness testing
- Flexibility check
- Plating thickness
- Insulation resistance

7.8 Marking requirements

Designers can specify the type of ink to be used for various board markings like serial numbering, component mounting location, stiffeners/cover locations, and panel-based marking. The types of ink include:

- Durable white ink
- Traditional epoxy ink

7.9 What to include in flex drawing notes

- The PCB shall be fabricated to IPC-6013, class <1/2/3>, wiring type <enter your requirement here>, and installation use <enter your requirement here>.
- The flexible copper clad material shall be IPC-FC-241/11 preferred (or /1) (flexible adhesiveless copper clad dielectric material). For example, FR / AP / LF.
- The covercoat material shall be as per IPC 4203/1 <your requirement here>. Example: LPI or Coverlay.
- The maximum board thickness shall not exceed <your requirement here> and applies after all lamination and plating processes.
- For Rigid-flex constructions, the thickness of acrylic adhesive through the rigid portion of the panel shall not exceed 10% of the overall construction.
- Minimum thickness of plated through-holes is 0.001”, with a minimum annular ring of 2 mils.
- Misregistration between any two layers shall not exceed ±0.005.
- Warpage shall not exceed 0.010 inch per inch.
- The thickness of copper plating in plated through-holes shall be at least 0.001mm.
- If polyimide material is used in a rigid section, the rigid material shall be GIN (Poly) as per IPC4101/40.
- If the thickness requirement is critical then mention it in the drawing notes.
- All external conductive surfaces not covered by solder mask shall be plated with ENIG.
- Marking and identification requirements
- Electrical test requirements

- Packaging and shipping requirements
- Apply a green LPI solder mask in the rigid sections of the board over bare copper on both sides. All exposed metal will be <surface finish requirement here>
- Add silkscreen marking on both sides of the board using white non-conductive epoxy ink
- The PCB shall be constructed to meet a minimum flammability rating of V-0.

7.10 Flex PCB design checklist

In order to get the most out of your flex PCB, you should have a clear vision of the circuit board's functionality and design rules. Below are some guidelines that we have discussed throughout the course of this design guide.

- Make annular rings as large as possible
- Vias should be tear-dropped. Teardrops can reduce potential stress concentration points on the PCB
- Adding tabs or anchors to vias will help prevent peeling
- Vias are not reliable in areas that will bend
- Avoid vias in the flex section of dynamic boards as they are at a risk of cracking
- Vias can be used over a stiffener, but they are at the risk of cracking if placed at the edge of a stiffener
- Vias should be placed at least 30 mils away from the rigid-flex/flex interface
- Always opt for a larger bend radius
- Use curved traces instead of traces with corners. Curved traces cause lower stress than angled ones
- When designing multilayer flexible PCBs, utilize staggered traces on the front and back. Stacked traces will not only reduce the flexibility of the circuit but also will increase stress contributing to the thinning of copper circuits at the bend radius.
- Traces should also be kept perpendicular to the overall bend.
- The rigid-flex fab notes must consist of rigid notes and flex notes separately.
- The thickness of acrylic adhesive through the rigid portion of the panel shall not exceed 10% of the overall construction.
- Misregistration between any two layers shall not exceed ±0.005”.
- Warpage shall not exceed 0.75%.
- Provide impedance trace details such as trace width, height, and impedance tolerance in the stack-up.

Chapter 8: Controlled impedance in flex PCBs

8.1 What is controlled impedance?

Controlled Impedance is the characteristic impedance of a transmission line formed by PCB traces and its associated reference plane. It becomes very important when high-speed signals propagate on circuit board transmission lines. In order to achieve good signal integrity, a uniform controlled impedance is required.

8.2 Factors affecting impedance control in flex

Controlled impedance is determined by the physical dimensions of the PCB traces and the properties of the dielectric material used. Below are the factors that affect impedance control in flex boards.

8.2.1 Physical dimensions of the traces

1. Height of the trace
2. Width of the trace at top
3. Width of the trace at bottom
4. Difference between the width at the top of the trace and the bottom of the trace

8.2.2 Dielectric properties of the material used

1. Dielectric constant
2. Dielectric height between the trace and the reference plane

8.3 Controlled impedance configurations for flex circuit boards

Common controlled impedance configurations for flex circuit boards are:

8.3.1 Single-ended microstrip

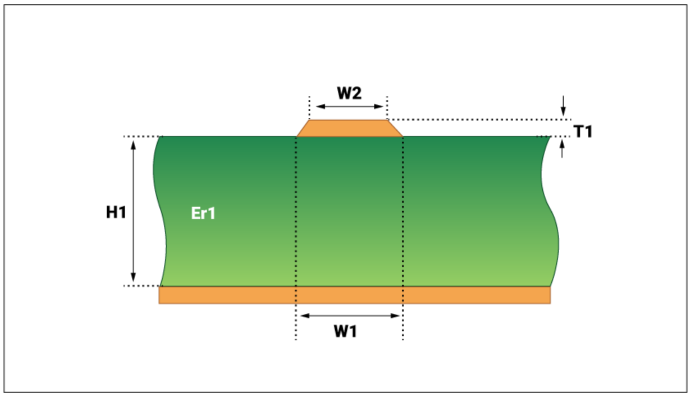


Image 26: Single ended microstrip

It is the most preferred controlled impedance configuration for flex circuit boards. It has a transmission line of a single uniform conductor on the outer layer of the board. The return path for the signal is provided by a reference plane separated by a dielectric layer of height H1. This configuration allows for thinner flex construction, improved bend capability and reduced cost.

8.3.2 Edge coupled coated differential pair microstrip

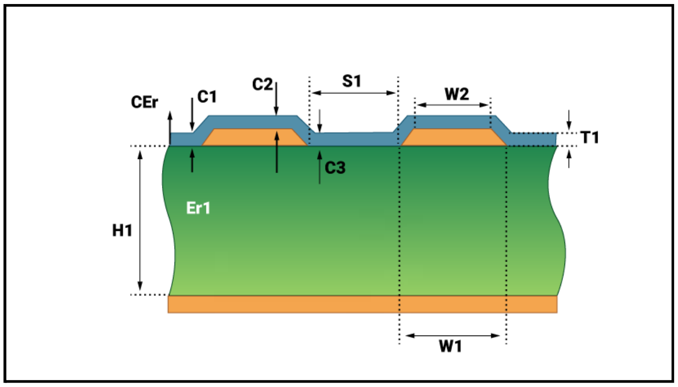


Image 27: Edge coupled coated differential pair microstrip

When a signal and its complement are transmitted on two separate traces, it is called differential signaling. These traces are called differential pairs. The traces are routed with a constant space between them. One of the primary advantages of having edge coupled differential pairs is that the noise on the reference plane is common to both the traces. This cancels out the noise at the receiver end.

This technique is used for routing differential pairs and has the same arrangement as regular microstrip routing. It is more complex due to the additional trace spacing required for the differential pair. It consists of a differential configuration with two controlled impedance traces on the surface, separated by a uniform distance and backed by a plane on the other side of the laminate.

8.3.3 Single-ended stripline

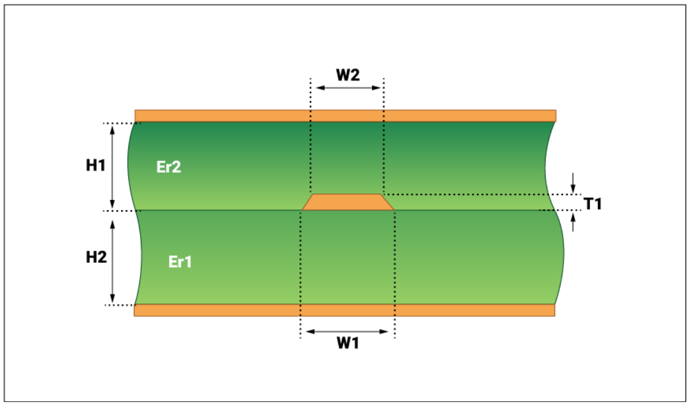


Image 28: Single ended stripline

It implements the signal trace between two ground planes in a multi-layer PCB. The return path for a high-frequency signal trace is located above and below the signal trace on the planes.

8.3.4 Edge coupled differential stripline

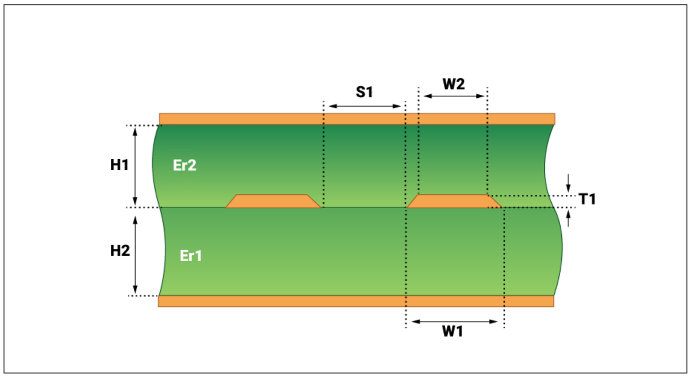


Image 29: Edge coupled differential stripline

It is similar to the single-ended stripline described above, except that we now have a pair of conductors separated by a uniform distance between them. It is a differential configuration with two controlled impedance traces sandwiched between two planes.

Reference planes in flex PCBs

Cross-hatched copper planes are used as reference planes in flex circuit boards. An image of a cross-hatched plane is shown below.

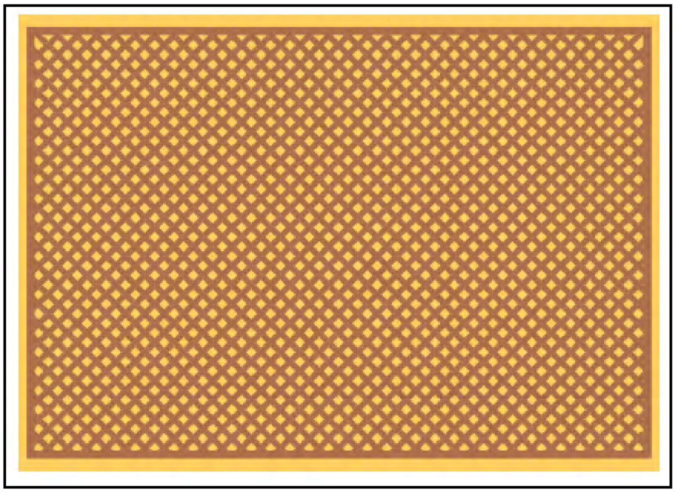


Image 30: Cross-hatched copper plane

A cross hatch plane can be characterized by the ratio of cross hatch conductor width (HW) to the cross hatch pitch (HP). The lesser the ratio, the greater the percentage of copper being removed. A 50% copper removal would be achieved if the ratio is about 0.293. Higher the copper percentage being removed in the cross hatch, the higher the increase in controlled impedance when compared to the solid copper plane.

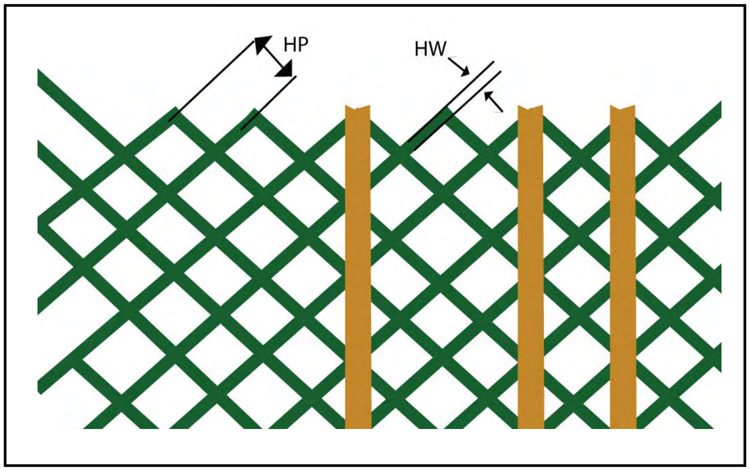


Image 31: Hatch pitch and hatch width in flex reference

8.4 Flex PCB materials for controlled impedance

Flex PCBs are often made of polyimide flex materials. These materials are well suited for controlled impedance designs as they offer lower Dk value, tightly controlled thickness, and homogeneous material construction.

There are two types of polyimide materials. They are adhesive based and adhesive less materials. Both adhesiveless and adhesive based materials can be used for the controlled impedance designs. However, adhesiveless materials are preferred for higher speed applications due to its consistent results.

Advanced materials like Teflon and Teflon/Polyimide hybrids are suitable for high speed applications. These materials are more expensive than polyimide materials.

It should also be noted that standard adhesiveless polyimide materials meet the controlled impedance design requirements while reducing the costs.

Sierra Circuits uses **Dupont** for flex PCBs.

Chapter 9: IPC standards for flex PCBs

9.1 Design

IPC-FC-2221	Generic standard on printed board design
IPC-FC-2222	Sectional design standard for rigid organic printed boards
IPC-FC-2223	Sectional design standard for flexible printed boards

9.2 Materials

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

9.3 Performance

IPC-6011	Generic performance specification for printed boards
IPC-6012	Qualification and performance specification for rigid printed boards
IPC-6013	Qualification and performance specification for flexible printed boards

9.4 Quality guidelines — circuits & assembly

IPC-A-600	Acceptability of circuit boards
IPC-A-610	Acceptability of printed circuit board assemblies
IPC/EIA J-STD001	Requirements for soldered electrical and electronic assemblies

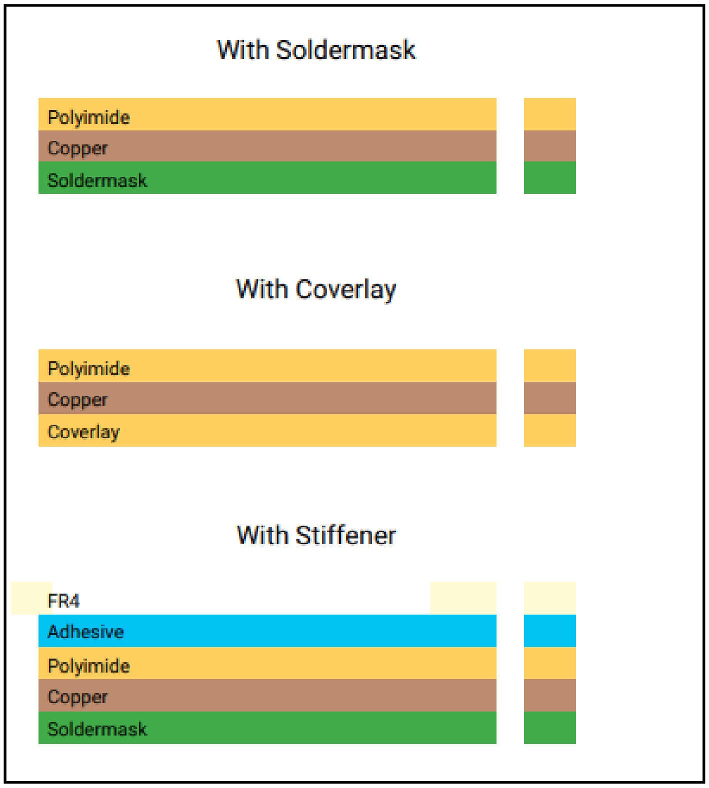
9.5 Military

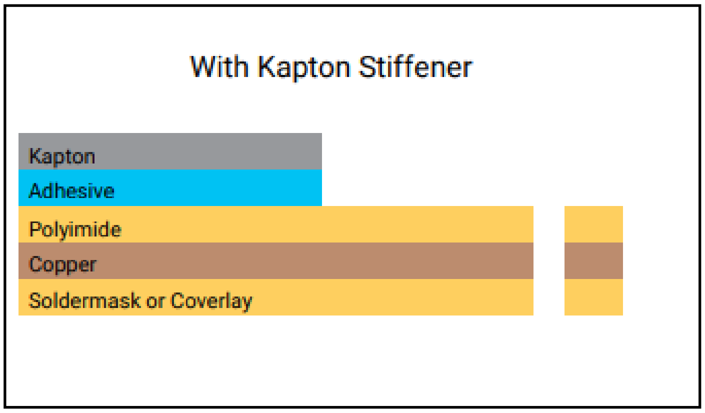
MIL-P-50884	Military specification: Printed wiring board, flexible or rigid-flex
MIL-PRF-31032	Performance specification: Printed circuit board/Printed wiring board, general specification

IPC-6013 Class C meets the same performance requirements as MIL-PRF-31032, and is accepted by government agencies as a COTS equivalent of the latter. If your flex circuit must meet performance requirements of MIL-P-50884, MIL-PRF-31032 or IPC6013, follow the IPC-2223 design specification recommendations.

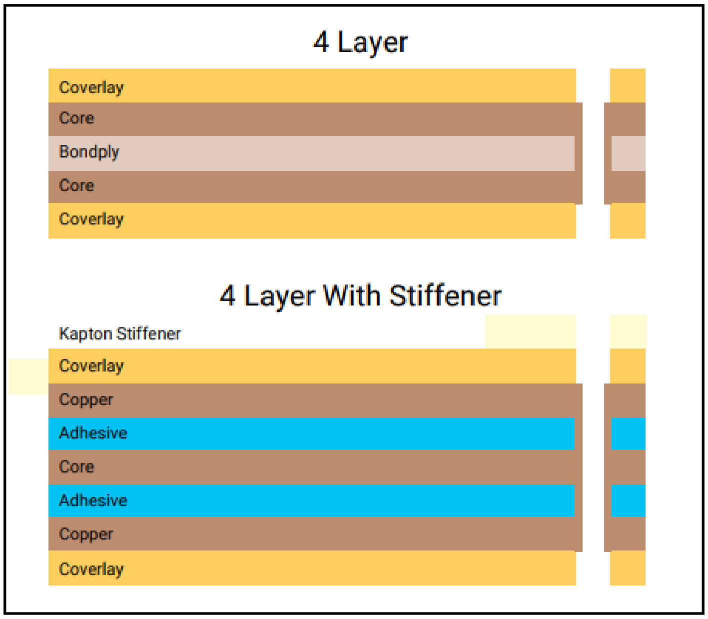
Chapter 10: Example flex stack-ups

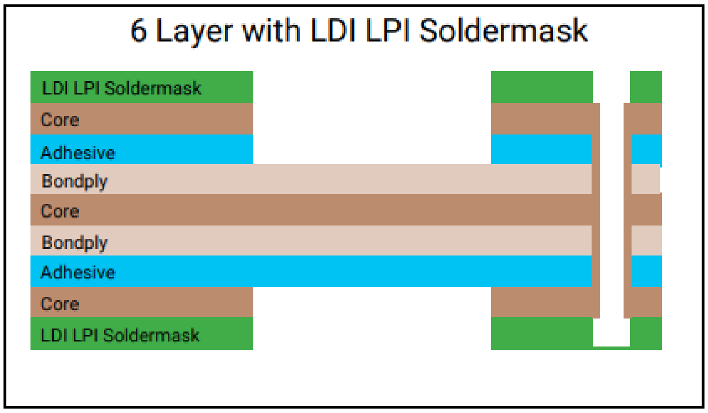
10.1 One layer flex stack-ups



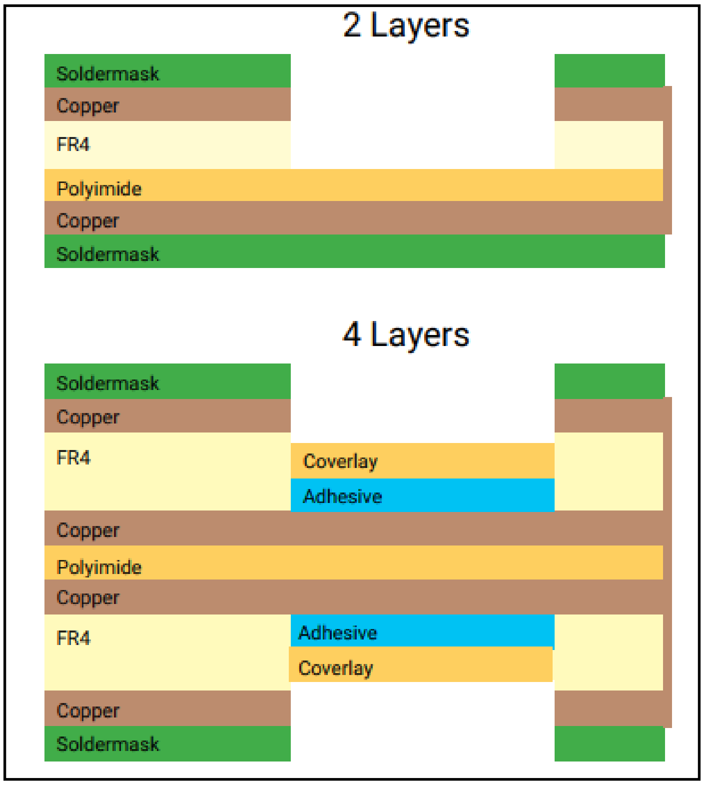


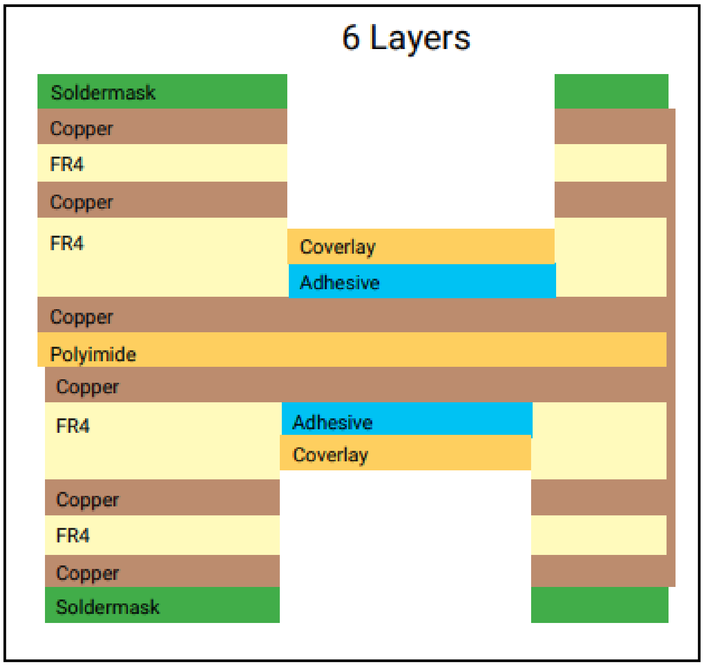
10.2 Multilayer flex stack-ups





10.3 Rigid-flex stack-ups





A well-designed flex PCB will be lightweight, durable, easy to install, and suitable for demanding applications such as wearable devices and satellites. The physical advantages of flex are that it offers improved resistance to vibrations and movement, and it is easier to prepare for harsh environments.

We provide our customers with unprecedented quality, reliability, and a single point of support. No more miscommunication between multiple vendors and no more delays.

We are **ISO-9001:2008, ISO 13485:2016** and **MilSpec MIL-P-55110** certified.

SIERRA CIRCUITS

Sierra Circuits
1108 West Evelyn Avenue Sunnyvale, CA 94086
+1 (408) 735-7137
www.protoexpress.com