

The background of the entire page is a vibrant orange. In the upper left, there is a white circle with a thin orange border. Inside this circle, the words "SIERRA" and "CIRCUITS" are stacked in a bold, black, sans-serif font. To the right of the circle, a stylized circuit board pattern is visible, featuring a series of parallel lines that curve and branch out, resembling a flex circuit. The pattern is rendered in shades of orange and yellow, creating a sense of depth and movement.

SIERRA
CIRCUITS

FLEX

DESIGN GUIDE



Table of Contents

1. Why design a flex PCB?	
1.1 Flexible PCB advantages	4
1.2 Preparing the board outline and thickness	5
2. Calculating the bend radius	
2.1 Designing bend areas	7
2.2 Increasing flexibility in ground planes	7
3. Controlled impedance in flex PCB designs	
3.1 Routing flex traces	9
4. Annular rings in flex PCBs	
4.1 Pad design for outer layers	12
5. Vias in flex PCB designs	
5.1 Flex vias	13
5.2 Rigid-flex vias	14
5.3 Drill-to-copper	14
6. Fab drawing for flex	
6. 1 Rigid PCBs vs flex PCBs	16.
7. From design to assembly	
7.1 Designing rigid-flex PCBs	17
7.2 Assembling flex PCBs	18
7.3 Stiffeners and tear guards	18
8. Flex PCB stack-ups	
8.1 One-layer flex stack-up	20
8.2 Two-layer flex stack-up	21
8.3 Multilayer flex stack-up	22
8.4 Rigid-flex stack-ups	23
9. Drawing requirements	
9.1 What to include in flex drawing notes	24
10. More information on flex PCBs	
10.1 Flex PCB standards	26
10.2 Manufacturing classes	27

Why Design A Flex PCB?

Flexible and rigid-flex circuits were originally used within the military industry, as they required 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. Wearable technology almost always requires flexible circuits: PCBs need to be durable, tiny, and conform to a myriad of shapes and movements. They are also used in everything from cars, trains, and airplanes to satellites, missiles and radios. In fact, NASA's Mars Rover has flexible circuits within it. It is, at 140 million miles away, the furthest fully functioning circuit board from Earth.

Flexible PCBs are lightweight, easy to install, durable, and compact. The wide range of motion makes it ideal for nearly all applications. Use flexible circuits in “bend to install” applications or dynamic applications, where the circuits are continuously in motion. Flexible circuits are also advantageous for the design packages where space is a primary concern.

The differences between flex PCBs and rigid are few, but significant. Here, we outline the advantages of flex PCBs, the differences between flex and rigid PCBs, and what to keep in mind when designing your first flex PCB. We hope you find this flex PCB design guide useful. If you have any questions, we have engineers and designers who would be more than happy to address any concerns.

1.1 Flexible PCB Advantages

Flexible printed circuit boards have several advantages over rigid PCBs. They include, but are not limited to, ease of form, fit and function.

Ease of Use:

- Design constraints are minimal. PCBs can be designed to fit any device shape.
- Range of motion allows PCBs to suit nearly any application.
- Less mass reduces risk in environments with regular vibrations.
- Reduces errors found in standard PCB assemblies.
- Reduces weight through the elimination of additional wires, cables and connectors.

Reducing PCB Size:

- Thinner and more lightweight than their rigid alternatives.
- Durable against motion and bending.
- HDI allows for the miniaturization of devices.
-

Rigid-flex PCBs:

- Blends flexible and rigid PCBs. Commonly formed with flexible circuits connecting several rigid flex boards.

HDI for Flex:

- Smaller package size increases the need of HDI.
- Allows additional space for other features on the PCB.

Cost Reduction:

- Total cost of installation is reduced.
- Flex circuits eliminate several steps within the production process, shortening overall turntime and reducing cost.

Notes

1.2 Preparing The Board Outline & Thickness

Board Outline:

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.

- Begin your layout—but do not route yet!
- Draw the board outline.
- Draw the locations of different thicknesses.
- Do preliminary component placement.
- Determine whether those components need stiffeners.
- Draw the stiffeners and rigid areas.

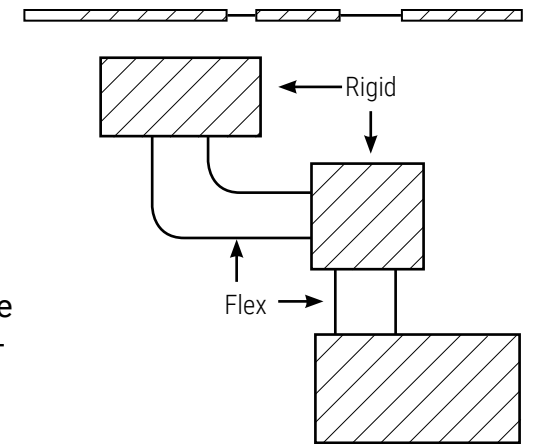
Thickness:

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 thicker, add a stiffener.

The following factors determine the required thickness of a circuit.

- Material thickness
- Design/layout of the materials
- Copper layer count
- Base copper weight
- Adhesive thickness
- Dielectric thickness

Notes

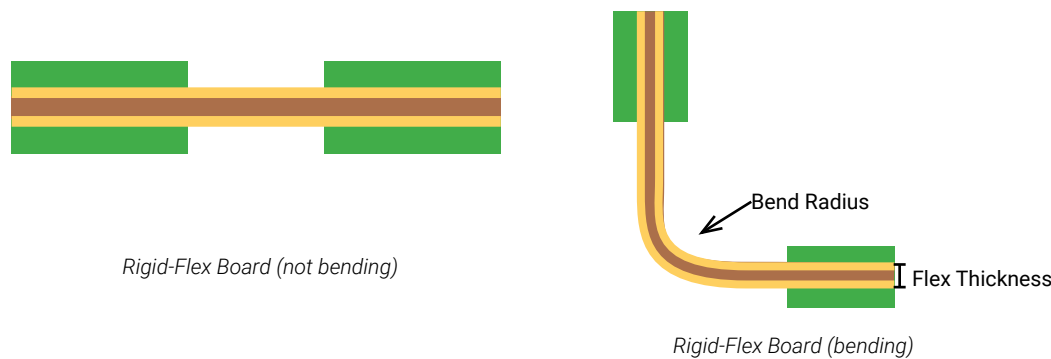


Sample Outline

Calculating The Bend Radius

Bend radius is the minimum amount the flex area can bend. Use the chart below to calculate bend radius on your next design.

1 Layer (single-sided)	Flex Thickness x 6
2 Layer (double-sided)	Flex Thickness x 12
Multi-Layer	Flex Thickness x 24



Knowing the amount of times your flex PCB will bend is crucial to your design. If a PCB is bent more times than the design allows for, the copper will begin to stretch and crack.

Semi-static: A PCB that flexes a maximum of 20 times.

Dynamic: A PCB that is regularly flexed and twisted.

Neutral Bend Radius

The neutral bend radius is the plane where there is no tension or compression when the circuit is flexed. Designing a balanced construction is the best way to center the neutral bend radius. Placing copper plane layers near the center of the material stack also keeps the neutral bend radius centered.

2.1 Designing Bend Areas

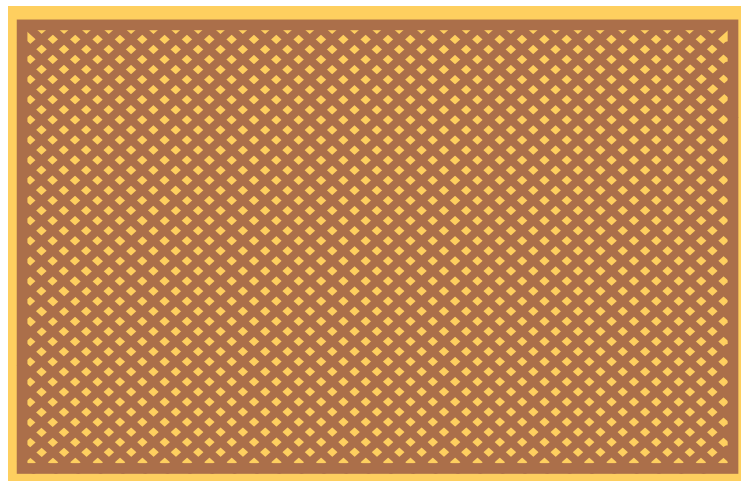
- When possible, avoid 90° bends. Tighter bends increase circuit damage. Gradual bends are safer for the circuit.
- Bend radius is measured from the inside surface of the bend.
- Place conductors smaller than 10 mils inside the neutral bend axis, as they tolerate compression better than stretching.
- Avoid plated through-holes within the bend area.
- Conductors running through a bend need to be perpendicular to the bend.
- Stagger conductors in multilayer circuits to increase the effectiveness of the circuit.

2.2 Increasing Flexibility in Ground

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 it's thickness is directly related to the flexibility of the circuit.

The second most common method is to reduce the copper thickness of the traces and more over the thickness of the plane layer. One way of reducing copper, on a plane layer, is by cross hatching the plane. Typically we recommend .015" wide signals with .025" spacing for the cross hatched plane layers.

Ground and power planes are usually cross hatched in flexible printed circuit boards in order to maintain or increase the flexibility of a circuit board. A ground or power plane that is completely flooded doesn't bend.



A cross hatch plane can be characterized by the ratio of cross hatch conductor width to the cross hatch pitch. The lesser the ratio, the greater the percentage of copper being removed. A 50% copper removal would be achieved if the ratio was about 0.293. For a pitch of 30 mils, this would be a cross hatch conductor width of about 9 mils.

Notes

Controlled Impedance in Flex Designs

A cross hatched reference plane has a significant impact on the controlled impedance value. Since cross hatching means the plane is no longer solid copper, but has a significant percentage of copper removed from it, the plane no longer provides 100% shielding to the signal traces. As such, the controlled impedance of the signal traces increases.

Impedance is a measure of how much the circuit impedes the flow of current. It is like resistance, but it also takes into account the effects of capacitance and inductance. Impedance is measured in ohms.

Impedance is very important to transmission lines and is used to determine the performance of a high-speed circuit. Impedance can be controlled with several different configurations including Characteristic, Differential, and Coplanar Impedance models. The first type is a micro-strip configuration where a conductor is above a ground plane. The second type is the stripline configura-

tion where a conductor is running between two ground planes. The varying of conductor width, dielectric thickness' and material selections controls impedance. (Dielectric constant). Impedance, on our flexible circuits can be reliably reproduced time and time again because of the relatively low dielectric constant and the roll annealed copper structure.

Cross-hatching planes give poor electrical quality for controlled impedance and other applications. In some applications, a wide, solid strip under critical traces is acceptable.

The higher the copper percentage being removed in the cross hatch, the higher the increase in controlled impedance when compared to the solid copper plane.

A 50% reduction in copper due to cross hatching may effect the controlled impedance as much as 7% – 16% (when compared to the case of the solid plane).

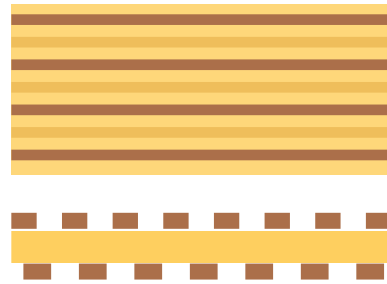
Notes

3.1 Routing Flex Traces

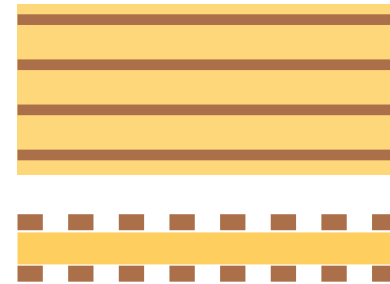
The performance and longevity of a flexibly circuit board can be directly attributed to the layout of the circuitry. Signals should never be routed at sharp angles. Sierra recommends the largest radius possible for your design. In addition, I-beaming will not only reduce flexibility of your circuit, it will increase stress contributing to the thinning of copper circuits at the bend radius.

When design flex with 2 layers+ stagger traces on the front and back

CORRECT



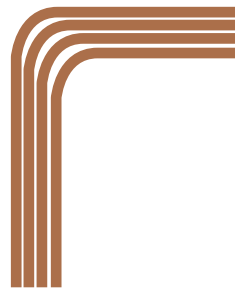
INCORRECT



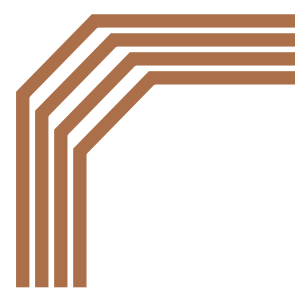
Notes

Use curved traces instead of traces with corners

CORRECT

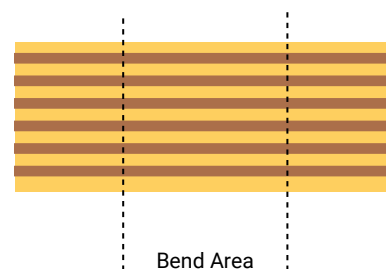


INCORRECT

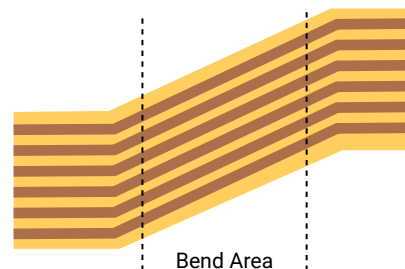


Traces should be perpendicular to the bend.

CORRECT



Bend Area



Bend Area

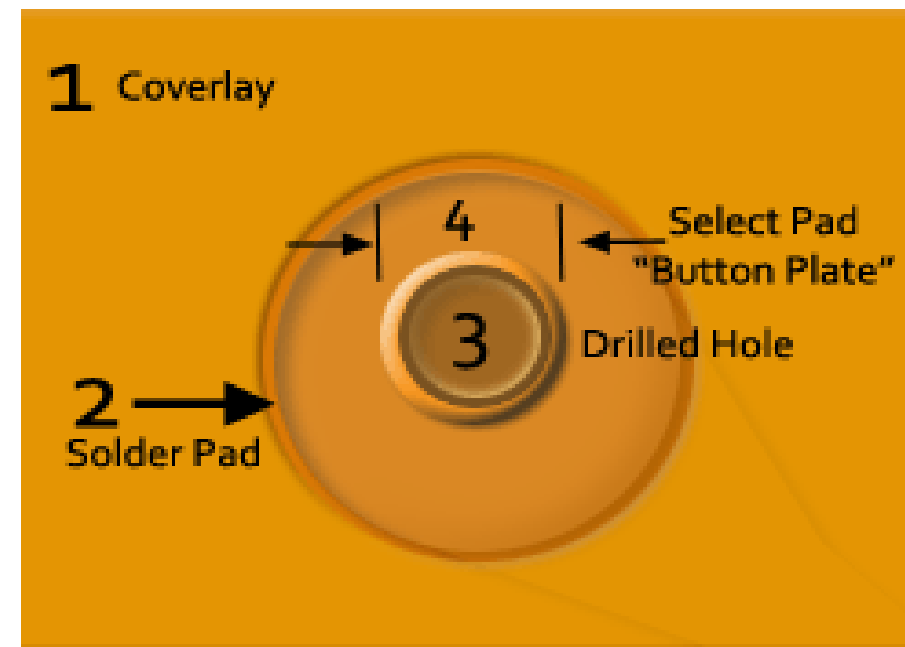
Annular Ring in Flex PCBs

Although squeeze-out only affects external layers, internal annular rings in rigid-flex and flex multilayer are often compromised, especially in places where tight hole to pad ratios are demanded. This is mainly due to the dimensional stability (1000ppm) of the flexible material. It is common to allow zero break-out (tangency) of the hole from the internal pad, and on some commercial parts there is sometimes agreement to allow a 270° minimum contact ring.

When designing flexible printed circuits, allow for some misregistration between the internal pads and the drilled hole. Consider minimum space between the tracks and the drilled holes.

Plating Flex Circuit Boards

For double-sided flexible products we use a selective plating process "pads only plate" to plate copper in the through-holes. In order to accomplish this, we drill the flexible copper clad dielectric material and then image the "pads plate" image around the drilled holes at (drilled diameter) $D + .003$ " or better. After plating the holes with approximately .001" of copper we image again with the finished circuitry pattern and the etch the desired pattern. An additional $D + .014$ " of pad is needed for this etching process, which puts our minimum pad size on double-sided flexible products at a drilled diameter + .017." These pads should always be as large as your design will allow.



Notes

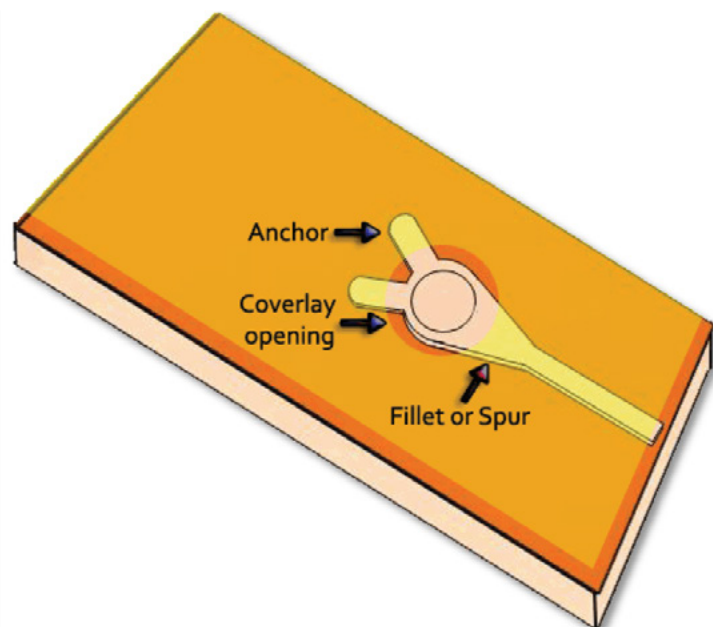
Layer Type	Standard (mils)	Advanced (mils)
Flex	drill + 14	drill + 10
Outer Layer Rigid	drill + 10	drill + 6
Inner Layer Rigid	drill + 14	drill + 10

4.1 Pad Design for Outer Layers

Notes

The materials used to manufacture flexible circuits typically contain acrylic adhesive. When heated, this acrylic adhesive can become soft and pliable. However, when designed properly, these circuits are extremely dependable. Making the pads as large as possible is key. On single circuits, we recommend the use of anchors or spurs. During dynamic bending, anchors help stabilize the outer layer. Flex PCBs with multilayers do not have any issues with adhesives.

Sierra also highly recommends using teardrops on all flexible PCBs. Teardrops can reduce and even eliminate potential stress concentration points on the PCB.



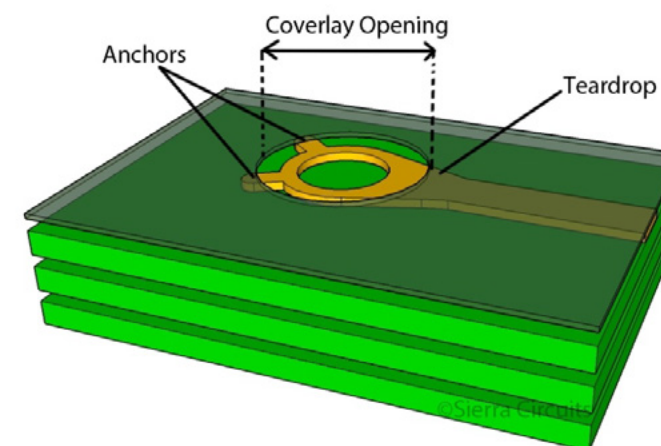
Vias in Flex PCB Designs

Notes

5.1 Flex vias

• Via Design

- Vias are at greater risk for peeling on flex designs.
- To reduce this risk:
 - Make annular rings as large as possible.
 - Vias should be teardropped.
 - Adding tabs or anchors to vias, as shown, will also help prevent peeling.



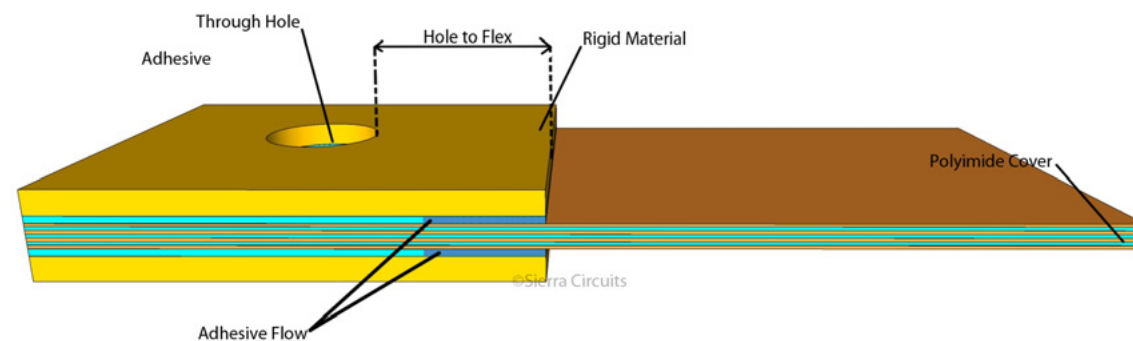
• Via Locations

- Vias are not reliable in areas that will flex.
- 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 for cracking.

5.2 Rigid-Flex Vias

- For rigid-flex designs, hole to flex distance is important. That is, the distance between vias and the rigid-flex transition area.
- Avoid going below 50 mils for high reliability applications.
- Keep in mind the rule most broken in rigid-flex designs: most manufacturers will not allow less than 30 mils for commercial applications.

Notes

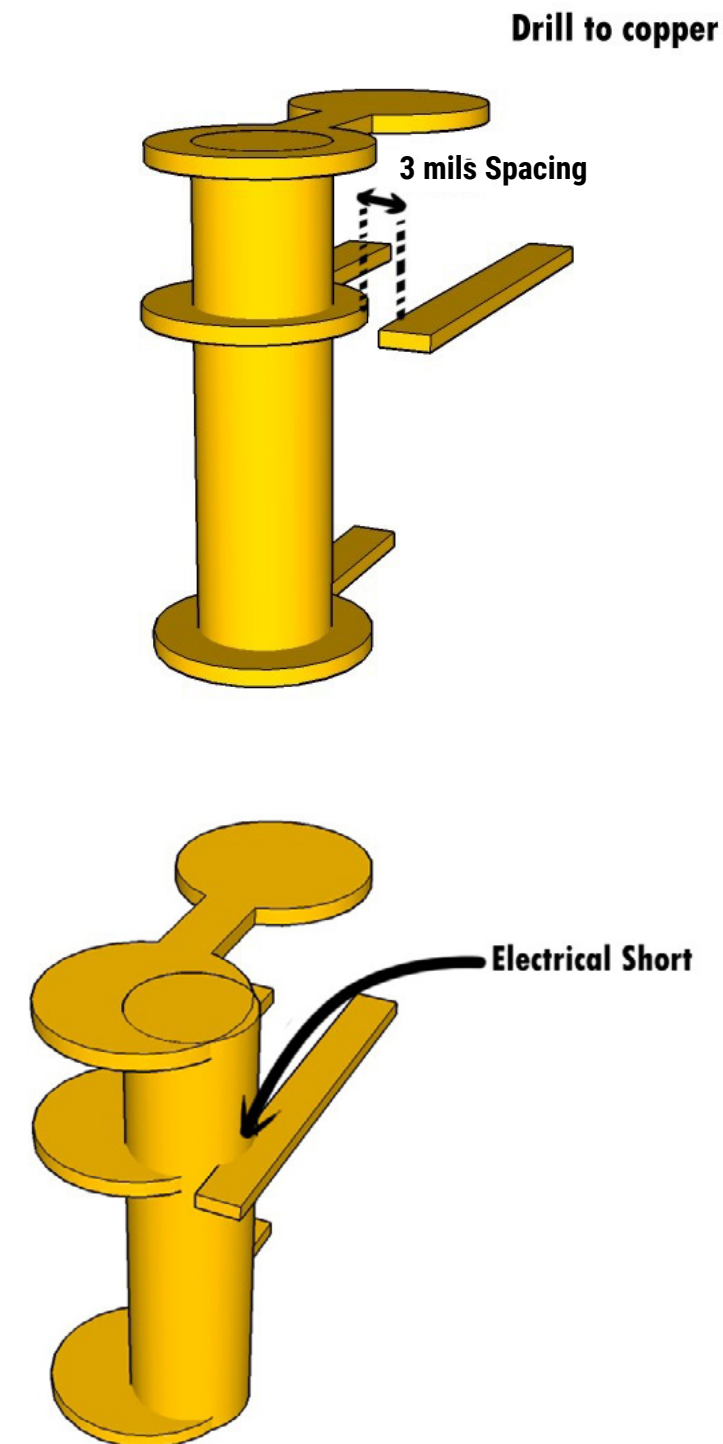


5.3 Drill to Copper

When designing flexible products it is crucial to keep drill to copper in mind. That is, the distance between a hole (via or nonplated) and a copper feature. The flexible dielectric materials that are used to manufacture flex product are not as dimensionally stable as standard rigid material. This material moves, shrinks and contracts during the manufacturing processes which makes drill to copper critical to a successful product.

Tighter hole to copper is possible, but usually requires a first article to measure the material movement. A longer lead time is needed.

Notes



Fab Drawing for flex

6.1 Rigid PCBs vs Flex PCBs

- **Rigid PCB Fab Drawings**
 - Board outline with dimensions
 - Any other critical information that is not standard
 - Drill map and drill legend for quality checks
 - All board part data, or indication for manufacturer to use their standard
 -
 -
- Flex Drawings
 - Dimensions and locations of stiffeners and rigid areas
 - Thicknesses of each portion of the board AND/OR materials to be used
 - Application: Static or Dynamic
 - Locations where board will flex and flex the most, if not obvious

Notes

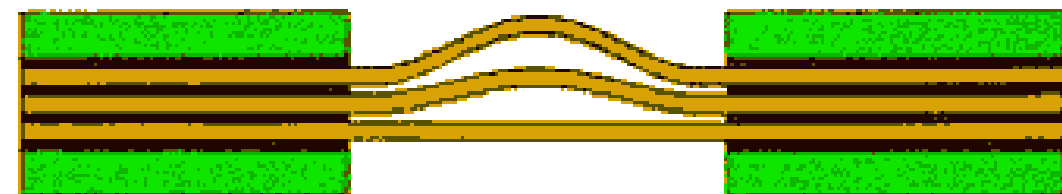
7 From Design to Assembly

Notes

7.1 Designing rigid-flex PCBs

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. Exceptions are defined below

- Drill to copper still has a 10 mil standard. Drill to copper is very important on rigid-flex.
- Hole to flex is the most commonly overlooked flex/rigid-flex design rule.
- Place the flex layers in the middle of the stackup and use an even number of layers.
- Vias in flex areas of a rigid-flex are considered buried.
- Like stiffeners, rigid portions of rigid flex add thickness, but circuitry can also be found in the rigid portion of a rigid-flex.
- The rigid portion of a rigid-flex is usually home to dense components that require a lot of circuitry.
- Sierra recommends putting flexible layers within rigid layers when designing rigid-flex boards as opposed to flex layers outside the rigid layers.
- 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.



7.2 Assembling Flex PCBs

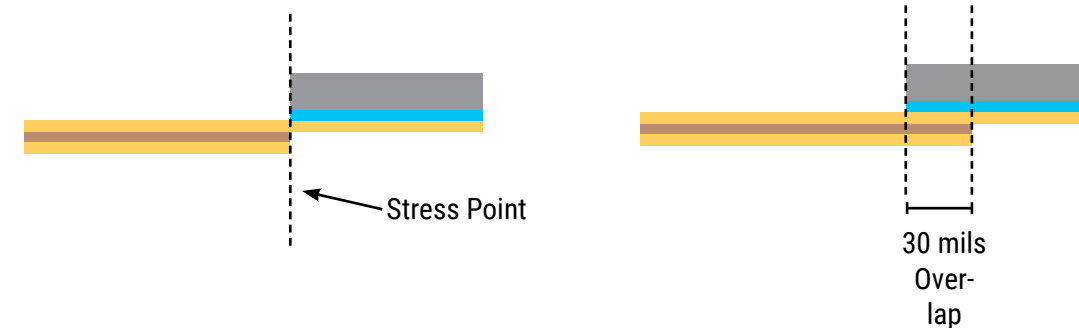
- Solder joints can weaken if components are in areas where the board will flex.
- If components need to be close to a flex area, consider adding stiffeners.
- If you have a stiffener or rigid area for mechanical reasons, try to place components over the stiffener or rigid area.
- Flex circuits typically have SMT components on one only side of the board.
- Rigid-flex circuits typically are typically 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. Because Sierra manufactures and assembles in-house, we eliminate the need for pre-bake.

Notes

7.3 Stiffeners & Tear Guards

Stiffeners

Single sided, Double sided, and Multilayered Flex circuits can be stiffened in specific area's by adding localized rigid material. This material can add support for mounting components, increasing strength, thickness and rigidity. The thickness of flex legs can be adjusted for component needs as well, such as the "Ziff" end of a flexible circuit. Kapton and FR4 materials are commonly used for stiffeners, this material can be attached with thermally cured acrylic adhesive or pressure sensitive adhesive. Stiffeners should overlap bare coverlay by .030" to relief stress.



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
- Stiffeners reinforce solder joints and increase abrasion resistance.
- Stiffeners can be used for strain relief and heat dissipation.

Tear Guards

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



Notes

Flex PCB Stack-ups

8.1 One-Layer Flex Stackups

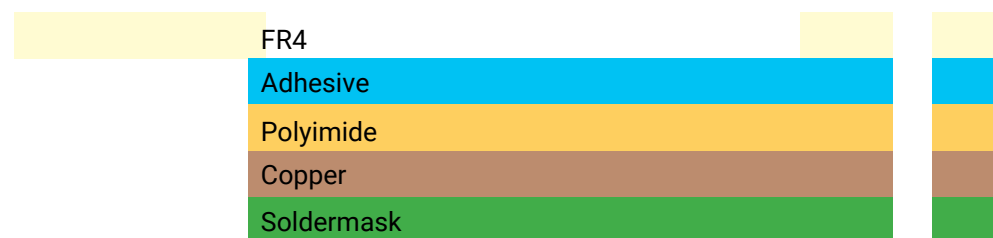
With Soldermask



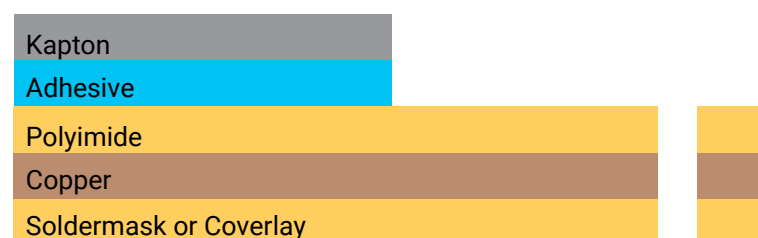
With Coverlay



With Stiffener



With Kapton Stiffener



8.2 Two-Layer Flex Stackups

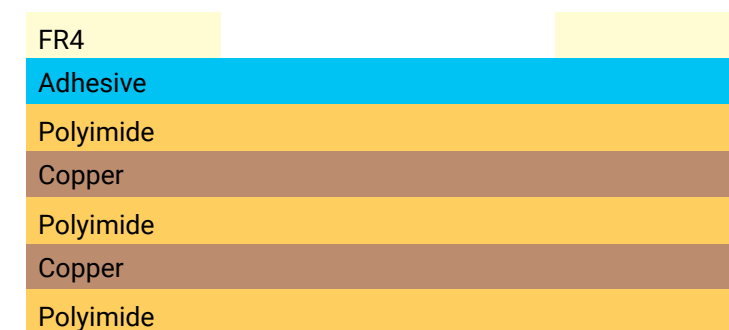
With Soldermask



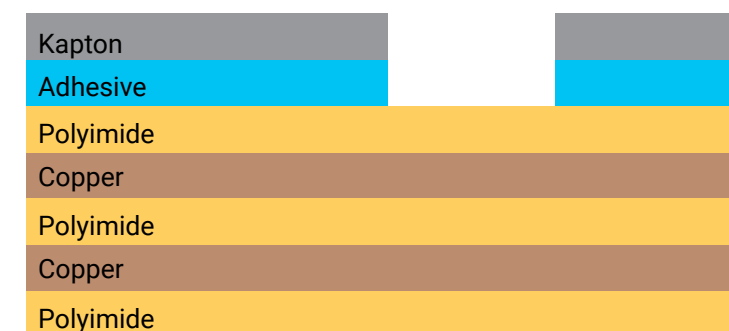
With Coverlay



With Stiffener



With Kapton Stiffener



Notes

Notes

8.3 Multilayer Flex Stackups

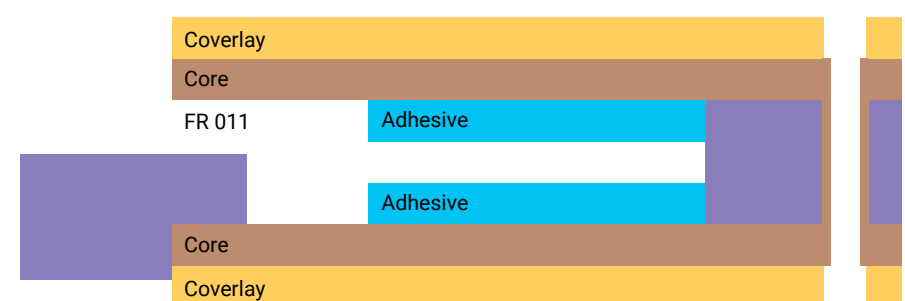
4 Layer



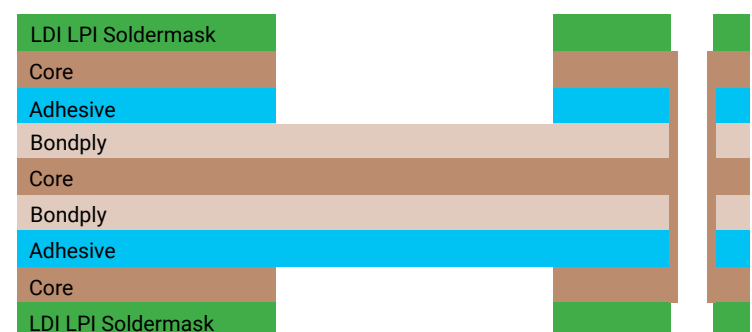
4 Layer With Stiffener



4 Layer Loose Leaf



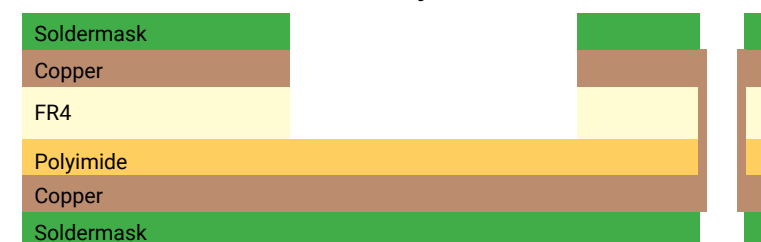
6 Layer with LDI LPI Soldermask



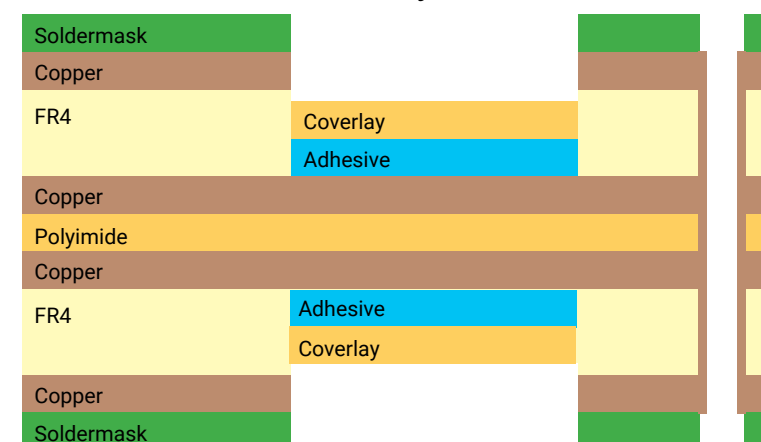
Notes

8.4 Rigid-Flex Stackups

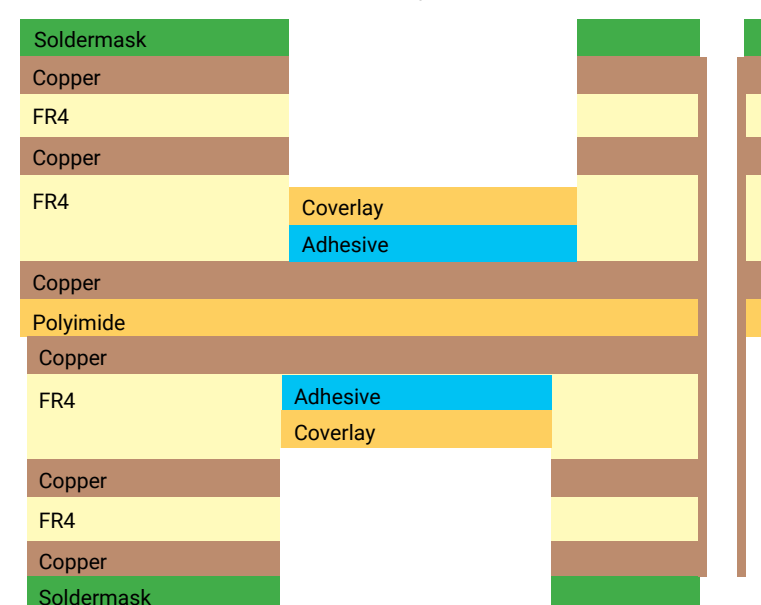
2 Layers



4 Layers



6 Layers



Notes

Drawing Requirements

- A drill symbol chart is required. This chart depicts your finished hole sizes and associated hole size tolerances. A finished hole size of +/- .003" is typical.
- A dimensional drawing is required. All critical dimensions must be noted and the rigid to flex interfaces (where the rigid material stops and the flexible material begins) must be defined. Typical outline tolerances are +/- .010". If a pallet or array is required, a dimension view must be supplied for this as well.
- A board construction and layer order is also required. This should show which layers are rigid material and which layers are flexible material including copper weights.
- A drawing revision table is advised. Your company name, part number, current revision should be in the title block area of the drawing.

Drawing Requirements:

- Cross-section diagram (material stackup)
- Outline drawing of circuit
- Material listing
- Specifications
- Hole chart
- Dimensional tolerances
- Feature chart (minimum conductor width and spacing, minimum spacing requirements)
- Special plating requirements
- Marking requirements
- Testing requirements
- Special packaging requirements

9.1 What to include in flex drawing notes

Notes

- The PCB shall be fabricated to IPC-6013, class (*your requirement here*), wiring type (*your requirement here*), and installation use (*your requirement here*) standards.
- The flexible copper clad material shall be IPC-FC-241/11 preferred (or /1) (flexible adhesiveless copper clad dielectric material). Example: Select FR / AP / LF
- The covercoat material shall be 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.
- Min. thickness of plated through-holes is .001", with a minimum annular ring of 2 mils.
- Vacuum press in autoclave or vacuum lamination
- Misregistration between any two layers shall not exceed $\pm .005$.
- Warpage shall not exceed .010 inch per inch.
- Copper Plating in plated through-holes shall be .001 minimum thickness.
- **If Using FR4 Material in Rigid Section:** The rigid material shall be GFN (FR4) per IPC-4101/24.
- **If Using Polyimide Material in Rigid Section:** The rigid material shall be GIN (Poly) per IPC-4101/40.
- **If Thickness Is Critical, Add:** The flexible section thickness shall be (*your requirement here*).
- **Finish:** All external conductive surfaces not covered by solder mask shall be plated with ENIG.

Additional Requirements:

- Marking and identification requirements here.
- Electrical test requirements here.
- Packaging and shipping requirements here.

If Required:

- Apply green LPI soldermask in the rigid sections of the board over bare copper on both sides. All exposed metal will be (*surface finish requirement here*).
- Silkscreen 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.

Notes

More Information on Flex PCBs

10.1 Flex PCB Standards

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

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

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

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

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 IPC-6013, follow the IPC-2223 design specification recommendations.

Notes

10.2 Manufacturing Classes

Class 1: General Electronic Products – Includes consumer products, some computer and computer peripherals suitable for applications where cosmetic imperfections are not important and the major requirement is function of the completed printed board.

Class 2: Dedicated Service Electronic Products – Includes communications equipment, sophisticated business machines, instruments where high performance and extended life is required and for which uninterrupted service is desired but not critical.

Class 3: High Reliability Electronic Parts – Includes equipment and products where continued performance on demand is critical. Equipment downtime cannot be tolerated and must function when required such as in life support items or flight control systems. Printed boards in this class are suitable for applications where high levels of assurance are required and service is essential.

Notes

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.

The logo consists of a large white circle with a thick orange border, resembling a magnifying glass. Inside the circle, the word "SIERRA" is written in a bold, orange, sans-serif font, and the word "CIRCUITS" is written below it in a smaller, orange, sans-serif font. The background of the entire page is orange. In the bottom right corner, there is a stylized graphic of a circuit board with orange and black lines.

SIERRA
CIRCUITS

Sierra Circuits, Inc.
1108 West Evelyn Avenue
Sunnyvale, CA 94086