FLEXing LIKE A PCB EXPERT

EMA Design Automation®

FLEX CIRCUITS
AN OVERVIEW

Flex PCB design has many advantages, as well as new design challenges.

Unlike “standard” 2D circuit boards, there are specific design issues that affect PCB Design Engineers working with flex PCBs. Knowing the quirks of flex design will help catch problems before they occur and speed up the production process.

What is Flex Design?

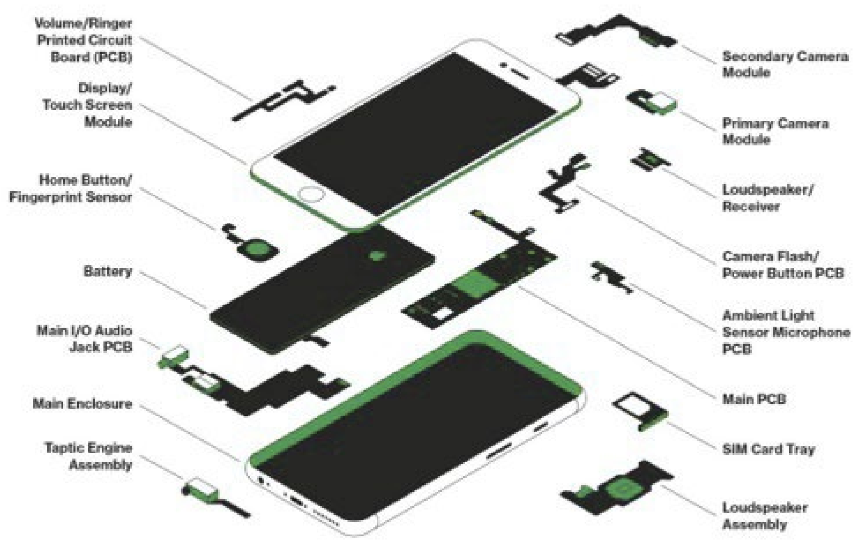
It is no secret almost all electronic devices require a PCB to function. However, as technology has improved, the need for smaller packaging requirements and lighter weight restrictions has led designers to embrace flex design. This method pairs conductors with flexible insulating film which allows it to carry the signals needed to function and remain true to its name. Its ability to bend and flex makes it as much a mechanical device as it is an electrical one. First developed by the space program to save weight and room on missions, the method soon spread to use in military and consumer endeavors.

CHOOSING THE RIGHT TYPE
FOR YOUR APPLICATION

Over time, two kinds of flex circuitry have developed: Flex and Rigid-Flex.

The former holds the ability to mount a device on flexible plastic substrate, while the latter is a hybrid of flexible and rigid circuits. Using the combination of Rigid-Flex will give you the best of both worlds, where the rigid boards carry the bulk of the components and the flexible sections connect them.

The move towards flex and rigid-flex design has grown exponentially in recent years, with the need for circuits in smaller, higher-performance electronics. A prime example of this use is in in many of our smart phones and tablets. These capabilities are also finding their way into many other high-end consumer electronics, with no end in sight.



An artist's rendering of an iPhone deconstruction that was performed by the analyst firm IHS.

Flex circuits have two designations of usage class: *static* and *dynamic*.

Static flex circuits are meant for minimal flexing, usually during service and assembly cases.

Dynamic circuits are those meant for an end-use in frequent flexing, as in a laptop hinge or printer head. The distinction in what your product will ultimately be used for is imperative to ensuring your circuit is constructed using appropriate materials and stack-up methodology.

MATERIAL SELECTION

CHOOSE WISELY

A wide variety of materials are used in creating a flex circuit.

These materials include **films, foils,** and **adhesives.** More common favorites are **polyester** and **polyimide,** and the choice of the material depends on how and where the circuit will be assembled and used. Certain materials may have added benefits or drawbacks, depending on your specific design intent. Carefully consider your choice of materials and consult with your fabricator to make the best decision possible in creating your flex circuit.

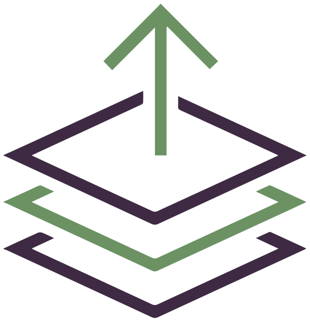
Conductor
Copper
Cupro-Nickel
Silver Epoxy
Stiffener
FR4
Polymide
Adhesive
Modified Acrylic
Modified Butyral
Modified Epoxy
Pressure-Sensitive Adhesive (PSA)
Pre-Impregnated Material: FR4, Polymide
Insulator
Polymide Film

DESIGN CONSIDERATIONS

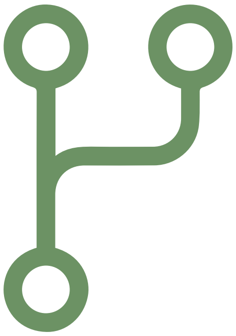
FLEXING THE RIGHT WAY

When designing a rigid-flex PCB, there are many key design areas that must be considered.

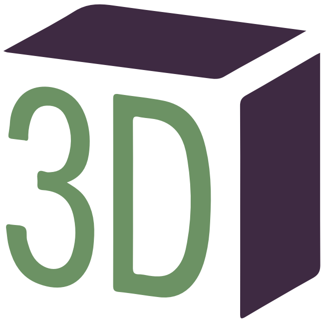
Each topic is an area ripe for critical failure, so your choices matter. All too often, projects are far in the process before a large error is discovered, and the team is forced to backtrack and rework the problem. This is why is it imperative to find a flex-aware tool that is easy-to-use and versatile to suit your needs. Doing so will help you quickly spot problems in complex design spots, and it will get you on the right path to design success.



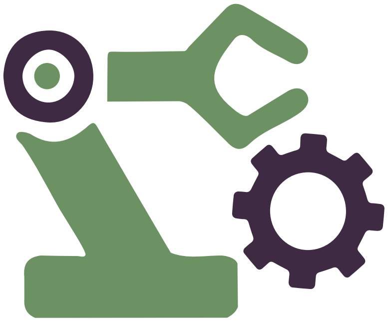
Stack-Up & Placement



Routing



3D Design & Analysis

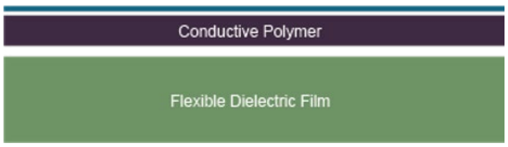


DFM & Flex-Specific Design Checks

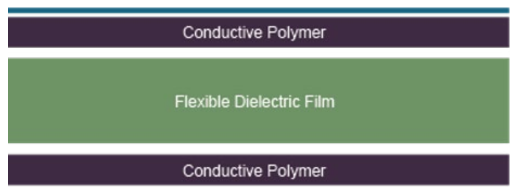
STACKUPS
DESIGNATIONS & CONFIGURATIONS

There are several flex stack-up configurations, each with its own purpose.

Single-layer circuits are mounted on a single conductor layer made of metal or conductive polymer over a flexible dielectric film. They are currently the most commonly used type of circuit- and the least expensive. With a thin construction, single-layer circuits are most effective in applications where constant motion is expected.

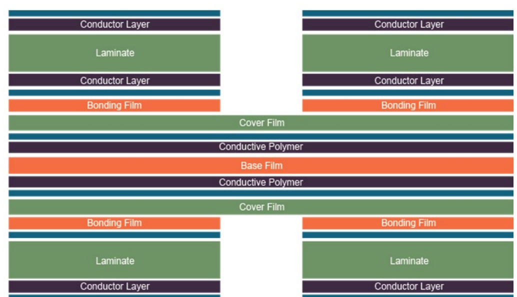


Single-layer



Double-layer

Multi-layer



Rigid-flex

Double-layer circuits are placed on two conductor layers, with the major benefit of the ability to have component assemblies on both sides of the board. Use this type when shielding applications is important to your design.

Multi-layer circuits span three or more layers which may not be continuously laminated together throughout the structure and may have openings or cavities. This type is useful in dense circuit mount assembly situations.

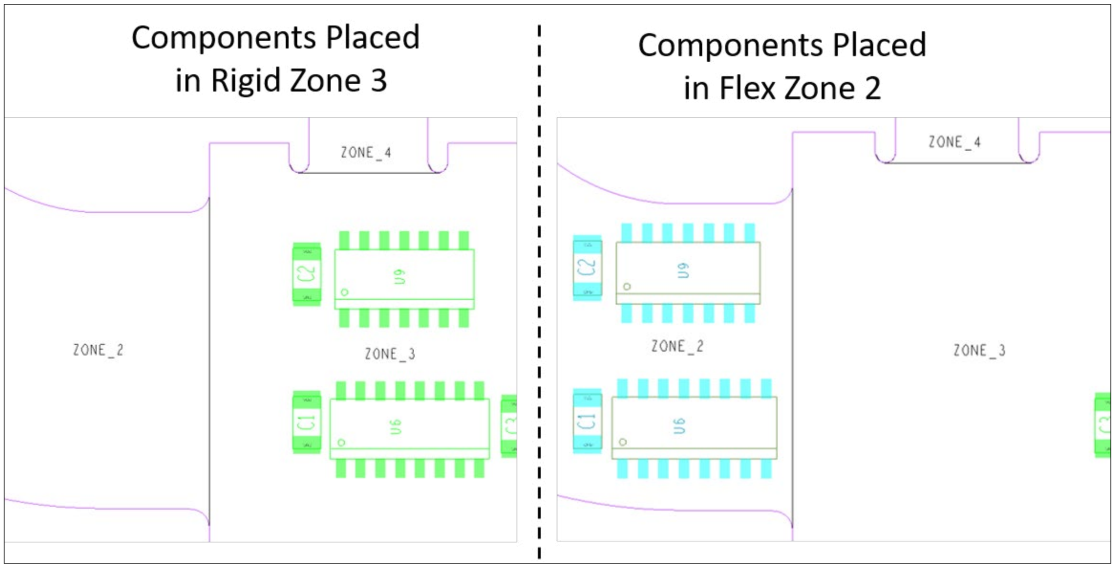
Lastly, the **rigid flex structure** is a combination of flex circuits placed on both rigid and flexible substrates. These are then laminated together into a single structure, ideal for applications where stable and flex areas are needed.

PLACEMENT

BEND AREAS ARE A SPECIAL CASE SCENARIO

There are various placement requirements on the flex and rigid areas of your circuit board, unique to flex design.

Intelligent component placement is the ability to place a component in either a rigid or a flex area and have it automatically placed on the topmost component layer in that zone. You will also need to check the clearance between components in the z-axis, in both flex states.



Bend Areas and Layer Structures

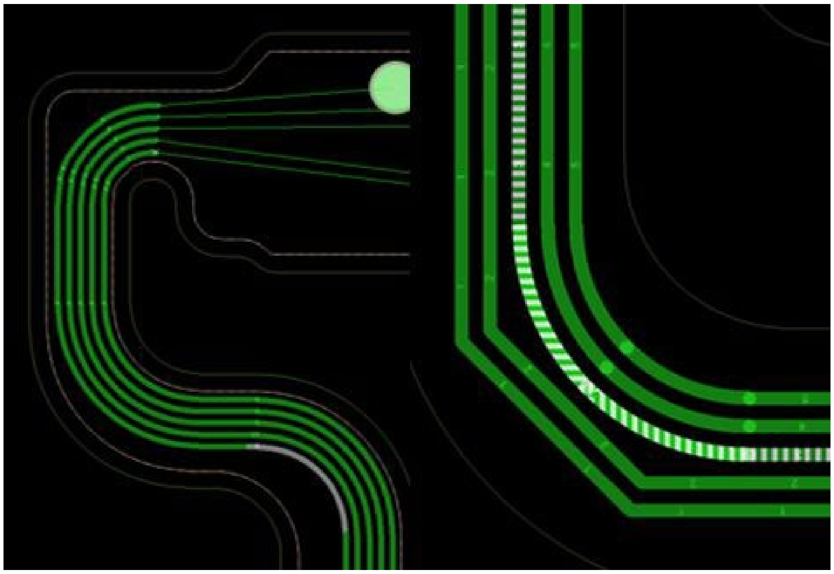
Bend areas take special consideration, especially across the bend lines of a flexible circuit. To reduce stress, traces need to cross the bend areas perpendicular to the bend line. Traces on adjacent layers should be offset to avoid the I-BEAM effect. The designer must have the ability to set the facing and overlap constraints between objects on different layers. For example, pins and vias should not be allowed within or near bend areas, to prevent cracking.

Layer structures (aka cross-sections) are set up per the design requirements. With rigid-flex design, not all layers exist in all areas of the board. Generally, the flex areas have fewer routing layers than rigid areas. You must be sure to avoid routing in areas where the layer does not actually exist.

ROUTING
CONNECTING THE DOTS

Flex circuits have a unique set of challenges when routing.

The ability to bend means these circuits will have a special list of requirements. A list of routing considerations and feature requirements include: routing arced corners, multi-line routing, contour routing, slide functionality for arc editing, and converting corners to arcs. Below is a quick overview of these routing strategies, when to use them, and why they are important for flex design.



Routing arced corners is the ability to route traces that have curved corners instead of orthogonal (45°) corners. Flex requires that bus lines be routed using curved traces in order to reduce strain, so you must route with arcs.

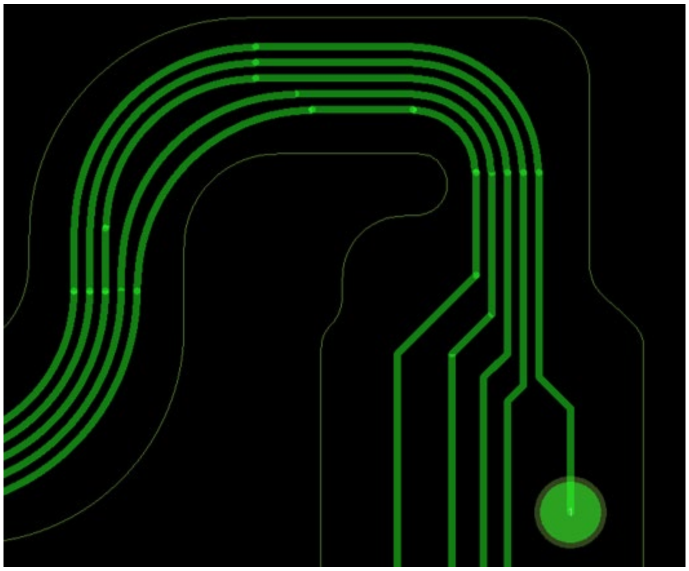
Multi-line routing is the ability to route a group of evenly spaced nets, side-by-side in a straight or arced pattern. This automation will save time and reduce errors in your layout.

Contour routing allows the designer to route a net while adhering to the board outline. This is especially needed when routing a group of nets along a complex board design outline with multiple curves. Flex board outlines often change during the design cycle, so quickly adjusting routes is key to efficiency.

ROUTING
CONNECTING THE DOTS

Slide functionality for arc editing is incredibly helpful when routing. The functionality allows the designer to slide arced traces once they are placed. Without this functionality, you would need to completely re-route the nets if an error was found, losing valuable production time.

Converting corners to arcs automatically is also necessary in flex design in order to reduce strain or shorts in the bent state. 45° or 90° corners can be converted, saving time in rerouting.



Pads have specific requirements to ensure sturdiness and longevity.

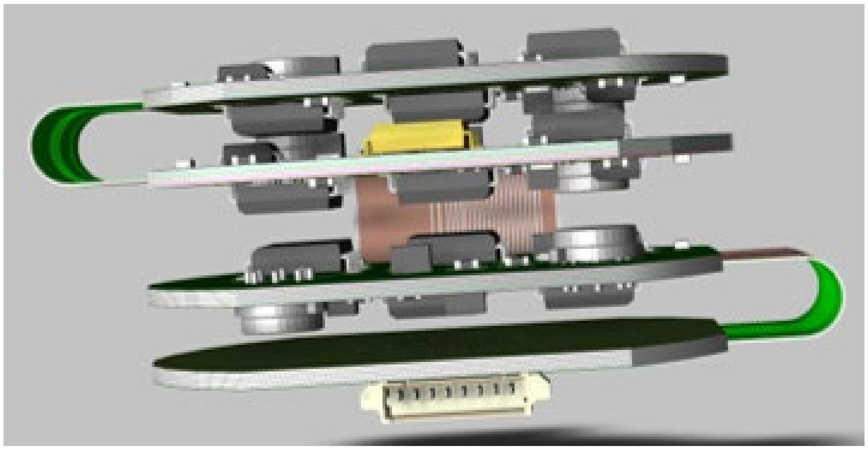
All pads, including surface mount and through-hole should have **tie downs**, preventing separation between the copper and base material. All pads should also have **fillets** to reduce stress points and eliminate breaking during flexing. The quality of these components will determine the life of the end-user product, so careful checking is necessary.

3D
CHECKING, BENDING, AND FLEX

Inherently, Flex design has an added layer of physical complexity.

This complication is the process of scrutinizing what the circuit will ultimately look like in its final bended state. The components must all be placed precisely so as not to collide or otherwise interfere with each other during any part of the bending action.

Therefore, design tools are necessary to analyze a flex circuit's fit and clearance in 3D. Ideally, the analysis can be done by the designer, and is best done in two stages: both during and after the bend's completed state. This will aid in identifying issues in either state, as well as alert you to problems that arise anywhere in between.



Data sharing is a key element to consider in your project workflow.

The ability to run a simulation and identify errors on the spot and communicate them within a collaborative environment is invaluable to the overall design process in saving time and allowing for a faster, more accurate project turnaround.

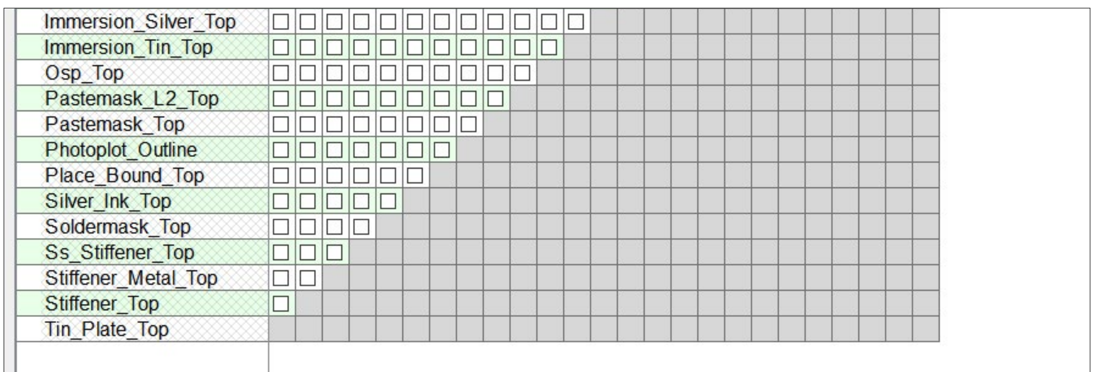
Typically, the bend area and flex radius will be defined by the mechanical requirements. Therefore, maintaining communication between MCAD and ECAD is important. **Bi-directional data sharing** of flex and bend information is crucial for keeping both sides updated and working as intended.

DFM CONSIDERATIONS

RIGHT FIRST TIME FLEX DESIGN

Flex requires unique DFM rules and checks as well as extra care during the manufacturer hand-off.

Flex designs require additional checking with constraints that check the clearance between objects on different layers. Throughout this part of the process, it is important to maintain clear communication with your manufacturer on their specific set of constraints and allowances in building the physical product.



Layer 1	Layer 2	Type	Value	Enabled	DRC label	DRC subclass	Description
Emi_Shield_Top	Coverlay_Bottom	Gap	25.000	☑	E	INTER_LAYER	Ensure EMI is
Gold_Hard_Top	Coverlay_Top	1 inside 2	1.000	☑	g	INTER_LAYER	Opening of
Transition_Zone	Via (and Pin)/TOP	Gap	15.000	☑	T	INTER_LAYER	Spacing from

Example of integrated flex-aware rule checks

If the conditions allow, the ground areas of flex circuits should be cross-hatched, reducing the weight, improving circuit flexibility, and preventing tearing. As you design, employ **inter-layer rule checking** to catch errors as they occur, rather than finding them later and causing valuable time to be lost. These rules become even more critical in densely-packed boards with many layers, as they are the most susceptible to multiple design errors.

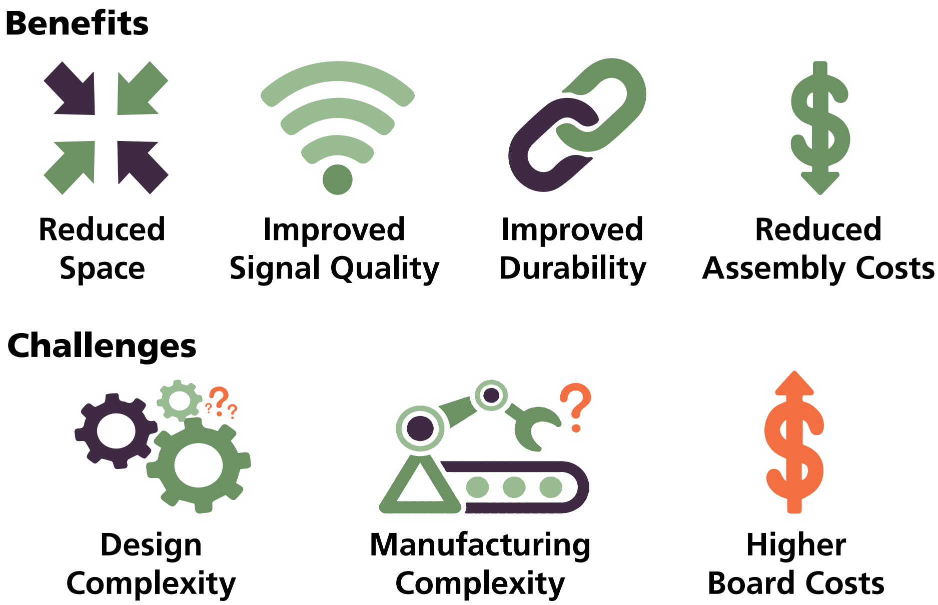
The distance between the bend area to stiffeners, pads, and vias all must be tracked, as well as from coverlay to pad. Using an intelligent interchange format such as IPC-2581 will enable your manufacturer to more closely capture your design intent. Engage with them early in the design cycle to provide you with the best possible feedback.

CHALLENGES & BENEFITS

IN DESIGNING FOR FLEX CIRCUITS

There are many benefits to flex design, but they don't come without new and unique design challenges.

Consumer, military, and medical devices are becoming much smaller, and so too are the casings that house the electronics within them. Secondary to packaging requirements are those of reliability, and designers have found a substantial improvement in shock and vibration performance within flex and rigid-flex PCBs. The improved durability, along with reduced assembly costs, make the flex PCB a great choice for designers.



It is imperative to start your design with a plan and hard goals in mind, as the design can quickly become very complex, and you must use the space wisely. Just as complex, the manufacturing of your design may be a challenge as you marry the traditional methods of rigid and flex design into a hybrid. These challenges may result in a higher initial board cost of the PCB but will ultimately serve you with an increased ability to produce a high-end flexible product for your company.

ORCAD
FLEX-AWARE PCB DESIGN

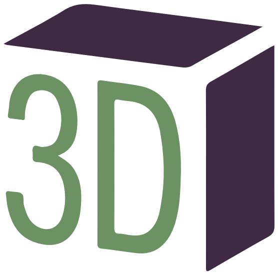
While flex design may be difficult, the benefits are often worth it.

It is a proven fact that today's technical innovations require a different kind of circuitry. Having a "flex aware" PCB design tool with rules-driven placement and routing, DFM checks, and real-time 3D design and analysis is critical to ensuring on-time design success.

Get on the cutting-edge of design technology and start designing your own rigid-flex circuits today. OrCAD will allow you to access powerful flex circuit abilities and much more.

Rules Driven Placement and Routing

Flex Aware and DFM Checks



Real-Time 3D Design and Analysis

Still want more information? Check out our Rigid-Flex page or contact us at info@ema-eda.com for inquiries.

EMA Design Automation®