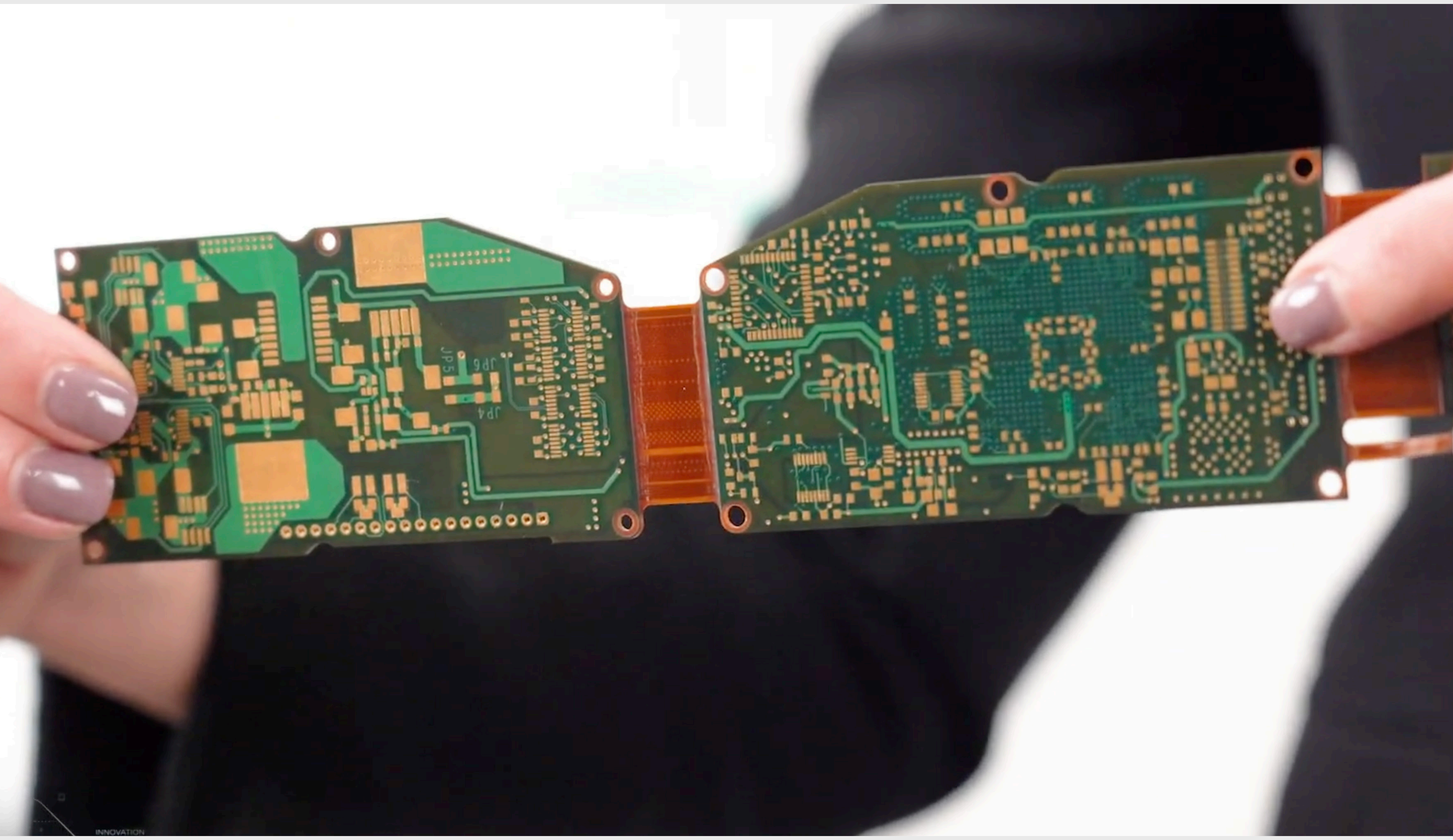
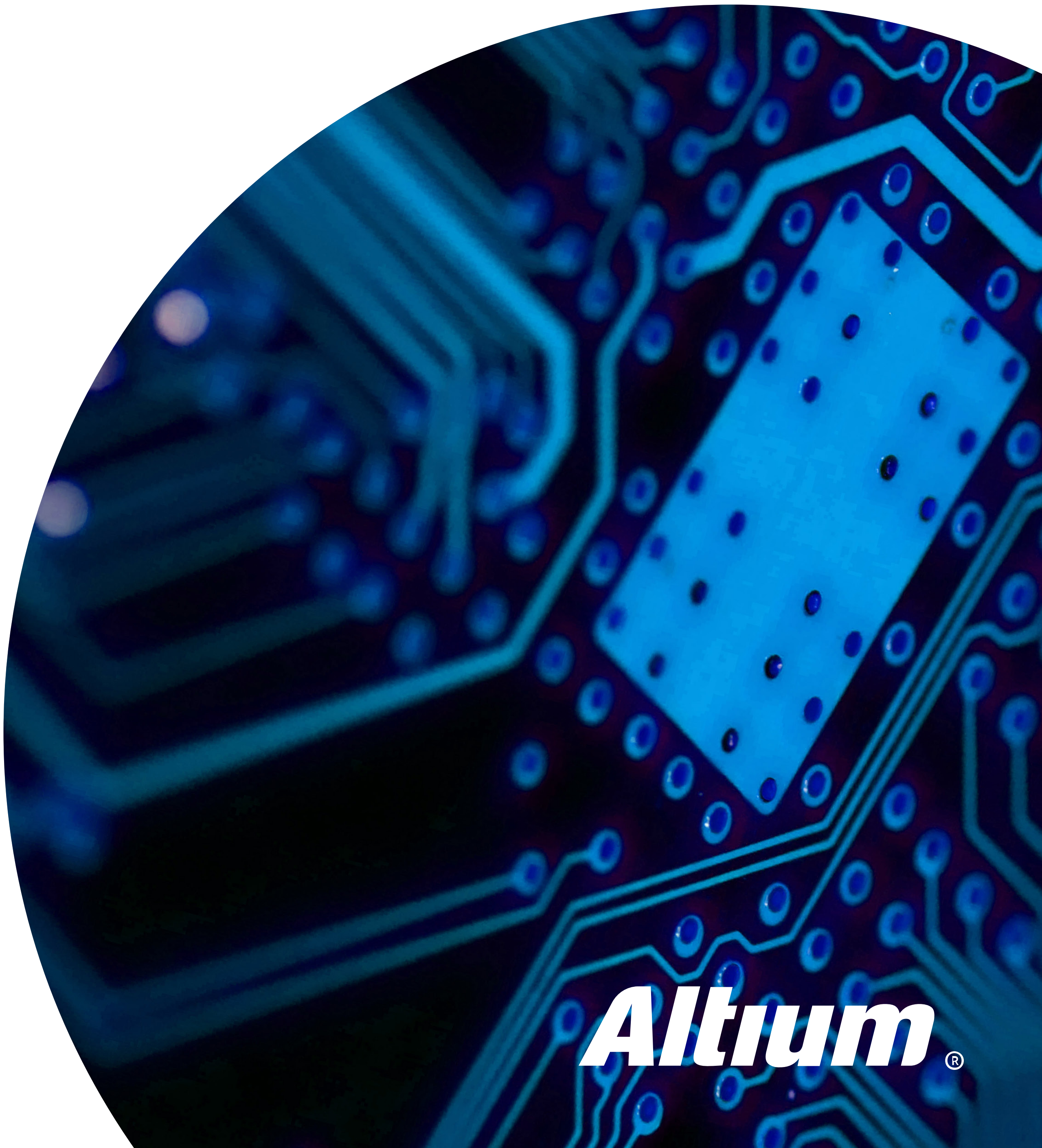


Rigid-Flex PCB Design Guidelines

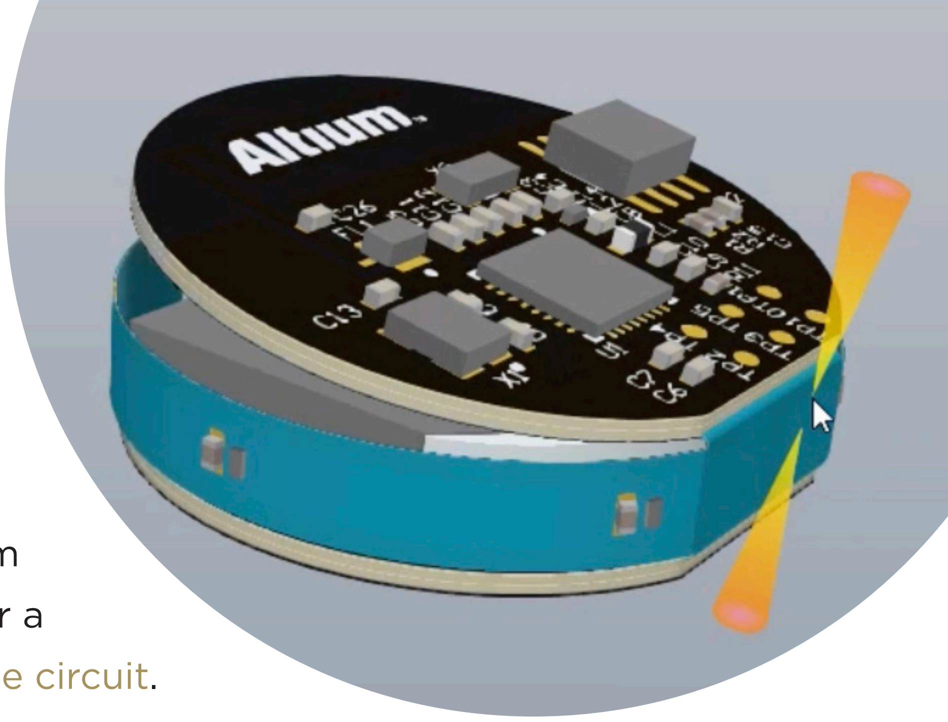
Reliable and Manufacturable Flex PCB

Altium®

Table of Contents



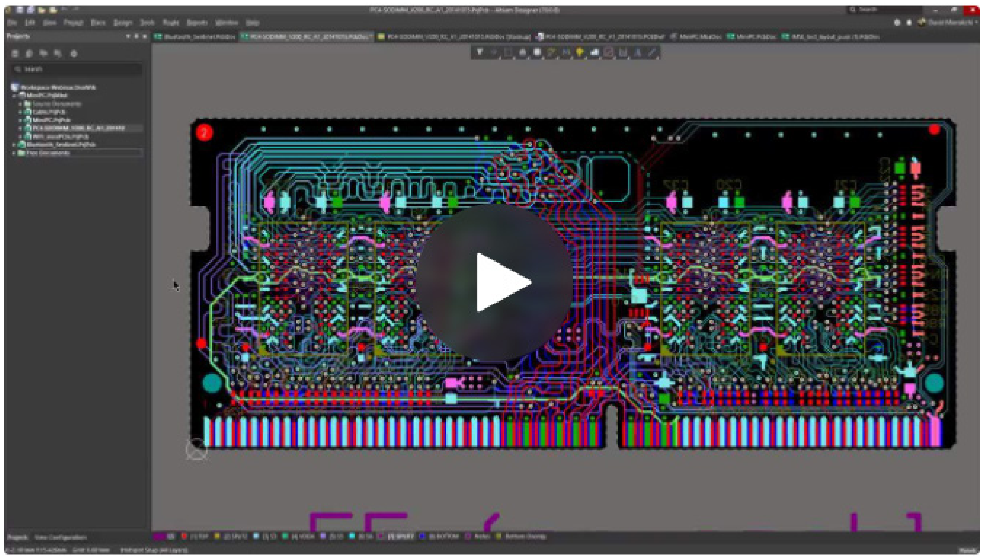
Flexible Circuits: Getting Started



Speaking to a group of students about printed circuit board fabrication recently, I asked for a show of hands asking how many had designed a rigid printed circuit board. Roughly 75% of the room raised their hands. The follow-on question asked for a show of hands for those that had designed a flexible circuit. The number of raised hands dropped dramatically. I suppose this shouldn't be surprising. While flex circuits are a growing segment of the printed circuit board industry, many seasoned PCB designers have not yet worked with flex materials. Thinking back to my college days, I think my knowledge of flexible circuits extended only as far as "that white ribbon that moves back in forth in my printer."

Those flexible circuits on polyester are certainly a significant portion of the flex market. There is no denying the volume driven by RFID tags, which often use aluminum on flexible polyester materials. However, when I am asked to speak about flex circuit fabrication or design, it is nearly always addressing the market segment utilizing flexible polyimide materials. These materials are standard in the flex and rigid-flex world as they integrate with a PCB stackup, allowing simplified routing between rigid and flexible regions with standard interconnects. In this guide, we'll look at how flex PCBs are made, starting with design and covering the various materials and processes used in fabrication.

- The Flex PCB Design Process
- Materials for Flex/Rigid-Flex PCBs
- Benefits of Flex PCB and Rigid-flex PCB Designs



The Flex PCB Design Process

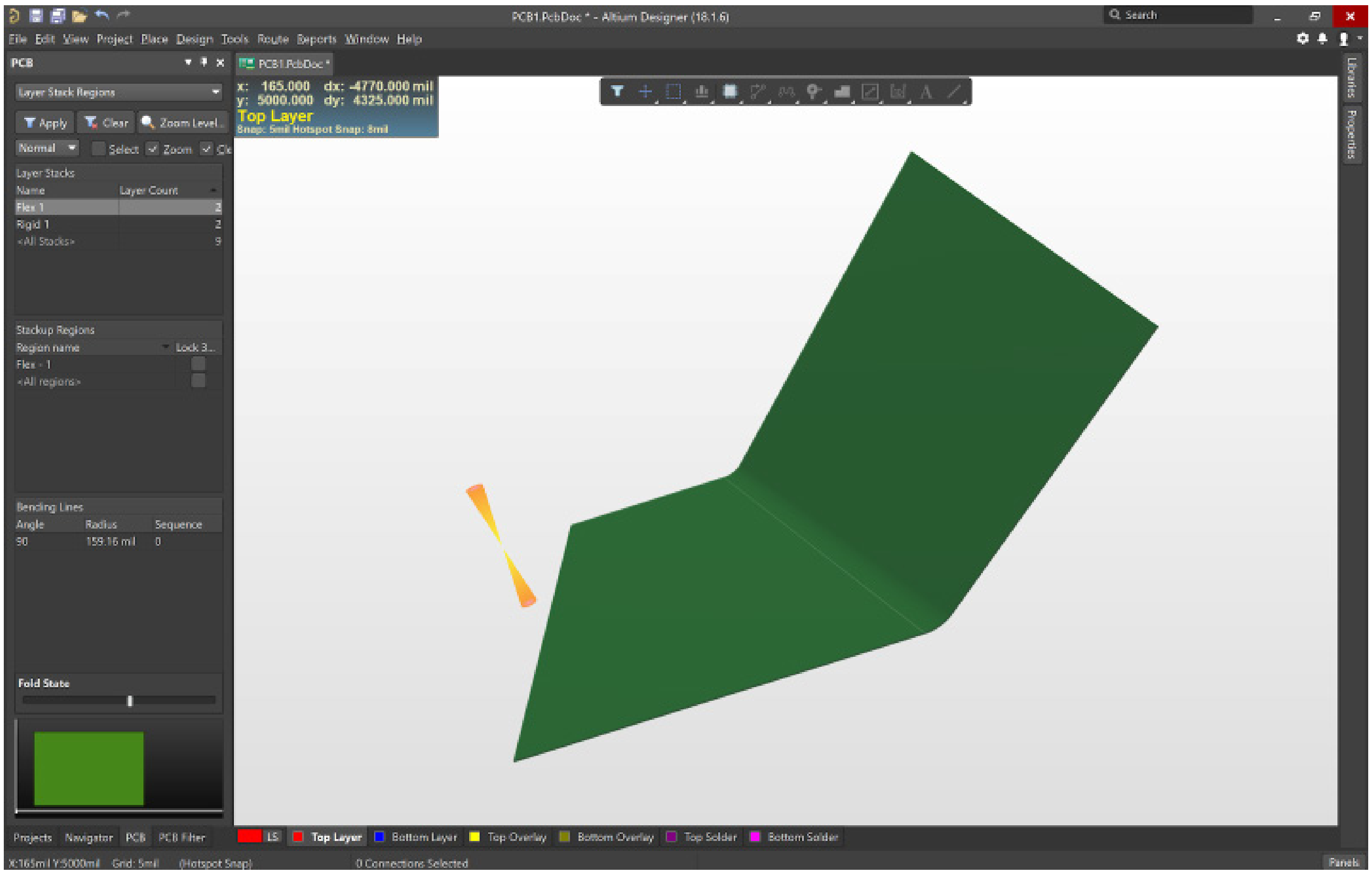
Flex PCB and rigid-flex PCB design and layout are interrelated; the major difference between flex and rigid-flex is an interconnect spanning into the rigid ends of the board. Much like a standard rigid PCB, the flex design process starts with designing a stackup, placing components, planning and routing paths for traces, and finally cleaning up the layout to prepare for manufacturing. Some additional steps are required to ensure the flex section has the required mechanical properties, such as adding stiffener, selecting a coverlay or photoimageable solder mask material, or setting a static bend in the design.

Because so much of the flex ribbon design process centers around the stackup, as well as its integration with any rigid section, stackup design is a critical part of the flex/rigid-flex PCB design process. In addition, things like copper weight and component/routing density in certain regions of the board can limit flexibility, making a rigid-flex board preferable. To get started with a flex or rigid-flex design, think about the following aspects of the final assembly and how the design will be integrated into its enclosure:

- **Floorplanning:** Where does the flex ribbon need to bend in the PCBA, and how will the design mount into its enclosure? The bending and mounting areas will need to be free of components, and you might want to use a rigid section for certain high pin-count ICs or high density circuitry. Decide where the flex ribbon will sit with respect to areas of the board with high component count.
- **Layer count:** How many layers are needed for routing, and will some of these layers be needed in a rigid section? If there is a rigid section, the flex ribbon will need to fit in the center of the stackup, which can make routing difficult when a larger number of planes are required (e.g., in a high-speed design).
- **Stiffness:** Can you accept a stiffer board, or will you need a highly flexible assembly? This can be controlled with component density, copper weight, or the addition of stiffeners to certain portions of the flex ribbon. If you need a very stiff polyimide section, it might be worth placing a fully rigid section instead. Stiffeners can be used to support higher component density on a flex ribbon and they will be less expensive than integrating a rigid section.
- **High-pin density components:** Smaller SMD components like passives can be easily placed on a polyimide flex ribbon. Other components in BGA packages require a more advanced via structure, as well as more internal layers to support routing. For high-speed or RF designs, hatched plane layers may be needed to provide required impedance or ground continuity (e.g., for single-ended signals), as well as isolation.

Your application requirements could dictate balancing any number of these requirements. Meanwhile, designers need to ensure their rigid-flex or flex design is manufacturable and will be reliable in the field.

Learn more about designing a rigid-flex PCB stackup in your design tools



x: 165.000 dx: -4770.000 mil
y: 5000.000 dy: 4325.000 mil
Top Layer
Snap: 5mil Hotspot Snap: 8mil

Layer Stacks

Name	Layer Count
Flex 1	2
Rigid 1	2
<All Stacks>	9

Stackup Regions

Region name	Lock 3...
Flex - 1	☐
<All regions>	☐

Bending Lines

Angle	Radius	Sequence
90	159.16 mil	0

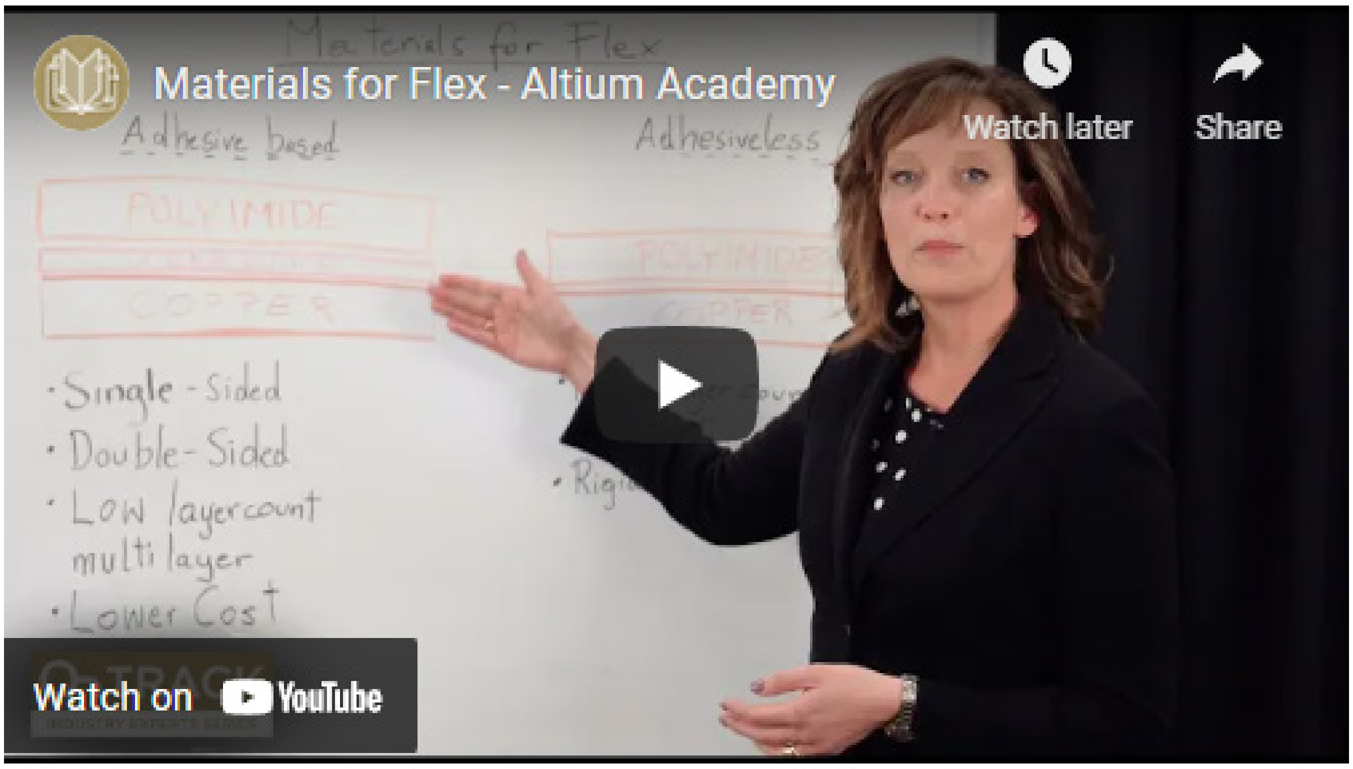
Fold State

X:165mil Y:5000mil Grid: 5mil (Hotspot Snap) 0 Connections Selected

Defining flex regions is an important part of building your stackup.

Materials for Flex/Rigid-Flex PCBs

Once you have decided to move ahead with a flexible circuit design, what are the key items to keep in mind? We will review these in more detail in future blogs, but you'll need to consider your material set in order to get the process started. The materials required for flex PCBs include base materials (polyimide, copper foils), coverlay materials (polyimide or liquid photoimageable), and finally stiffener materials.



Flexible Base and Coverlay Materials

It is crucial to understand the base materials themselves as well as the coverlay. Will your application require adhesiveless materials? Or adhesive-based materials? Does the application have any UL requirements to consider? What copper weight and polyimide thickness is best for your application? What materials does your fabricator routinely stock? (Hint: if you have flexibility in materials, routinely stocked flexible materials can help keep costs down)

Some applications will best be suited to flexible liquid photo imageable coverlay, which offers the ability to form very tight pad openings. If the design requires dynamic flexing, the application should use a film-based polyimide coverlay material. Film-based could have either laser cut openings or drilled openings, depending on the required pad size.

Stiffeners

Will your application be supporting heavier components and could benefit from the robustness of an added stiffener? Different components and applications need different stiffeners to support mechanical behavior in the flex ribbon or PCBA. Common stiffeners include:

- **FR4 stiffener:** Rigid FR4 epoxy-impregnated fiberglass is probably the simplest stiffener, effectively applied as a laminate over the flex area.
- **Rigid polyimide:** A rigidized polyimide layer can be used to stiffen a region by simply stacking additional polyimide sheets to produce a local thickness increase. This is normally used when terminating with a ZIF (Zero Insertion Force) connector, where a polyimide stiffener is used to build to the appropriate thickness.
- **Connector/component stiffeners:** Material options for this application include metals, FR4, or polyimide, which are thermally bonded with a flex adhesive or pressure-sensitive adhesive.

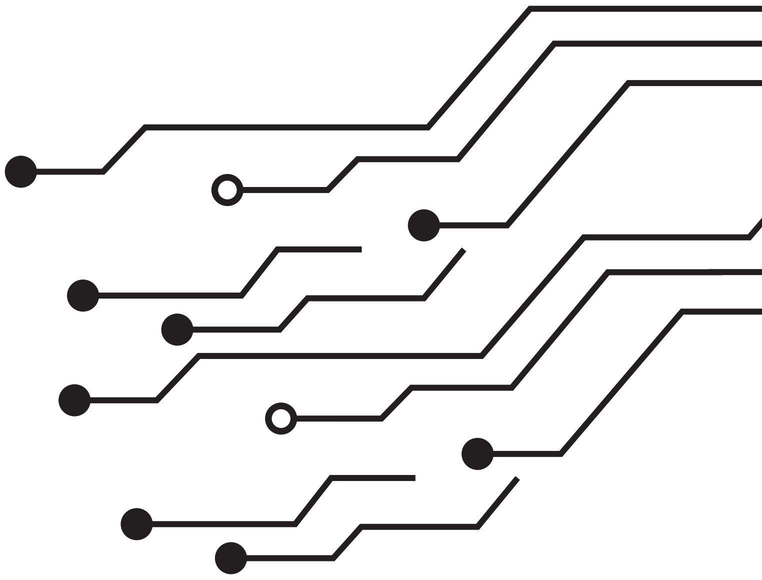
Fabricators' Capabilities

Do your existing preferred fabricators work with flexible materials on a regular basis? Will the complexity of your design match well with your preferred fabricators' capabilities, or are you stretching beyond their area of expertise? This is certainly not an all-inclusive list of things to consider but is intended to start the thought process of designing with a flexible circuit. Your PCB fabricator will be a vital resource, and you should contact them with a proposed stackup to ensure it can be reliably fabricated. They understand the materials and the fabrication process and will always be happy to help guide a customer along the learning curve with flexible PCBs.

Benefits of Flex PCB and Rigid-flex PCB Designs

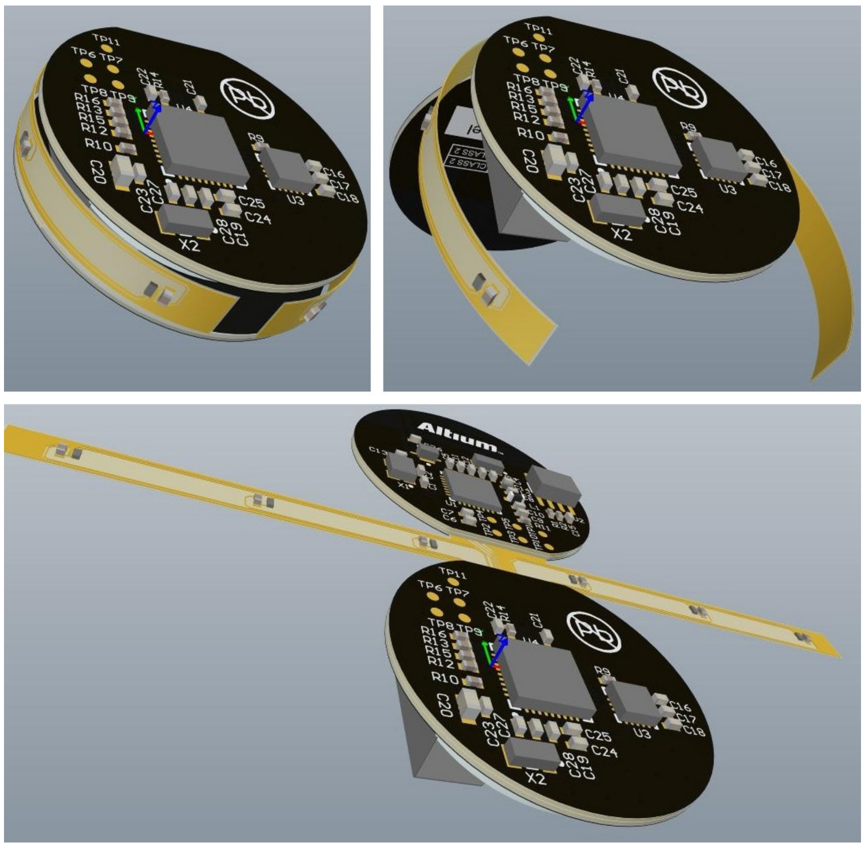
For those that are not experienced in flexible circuit design, let’s look at a few of the key benefits driving designers to use flexible materials. Although there are some particular design considerations needed to ensure a flex or rigid-flex PCB design is manufacturable as intended, these designs provide many benefits that aren’t seen in rigid circuit boards.

- **Space and weight:** Many of today’s electronic designs are challenged to reduce the size and weight of the end product. Flexible materials can replace bulkier wire and solder connections and a frequently touted statistic places that savings as high as 60% in the right circumstances. That is significant.
- **Packaging Benefits:** There is no denying that the ability to bend and fold a flexible circuit around a corner and the ability to provide a three-axis connection gives the PCB designer a significant advantage over both traditional wire and cable and rigid printed circuit board materials. In fact, the ability to best take advantage of these benefits is limited more by the imagination than anything else.
- **Simplified Assembly:** When compared with bulky wire and cable, a simple flexible circuit can dramatically shorten the assembly effort. One flex circuit can replace multiple line items on a bill of materials.
- **Thermal Management:** Polyimide materials can withstand high heat applications, and thin polyimide dissipates heat much better than thicker, less thermally conductive materials making this a top contender for higher power, higher frequency designs.
- **Biocompatibility:** Polyimide and LCP are two flexible materials that are an excellent choice for biocompatibility and are regularly used for that reason in both medical and wearable applications. Interestingly, I have been seeing increasing interest in flexible circuits that use gold as a conductor rather than copper, providing a fully bio-compatible option.



After fabrication, the flex portion of the circuit board assembly can be placed in its enclosure with static bending, or it can be allowed to flex dynamically with the enclosure or other mechanical elements. Sometimes a flexible circuit is designed to be “flex to install” or in other words, bent or folded with the intention of remaining stationary once the circuit is installed into the final electronics. Other times, the application requires a circuit to be flexed for hundreds, thousands, or even millions of times. Disk drives are a commonly touted example of a dynamically flexing application as is the flexible circuits in the hinges of our laptop computers. The design process varies slightly with static and dynamic flex assemblies, with the major consideration being deformation that can occur during bending.

Learn more about designing static and dynamic flex ribbons



Flex and rigid-flex PCB design engineers, layout engineers, and SI/PI engineers trust the advanced design tools in Altium Designer® for their design and layout needs. When you’ve finished a design and are ready to release it to manufacturing, the Altium 365™ platform makes it easy to collaborate and share your projects. Altium Designer also integrates with popular MCAD and simulation applications, giving you the ability to better understand power and signal behavior in multi-board systems.

When a design is finished and ready to be released to manufacturing, the Altium 365™ platform makes it easy to collaborate and share your projects.

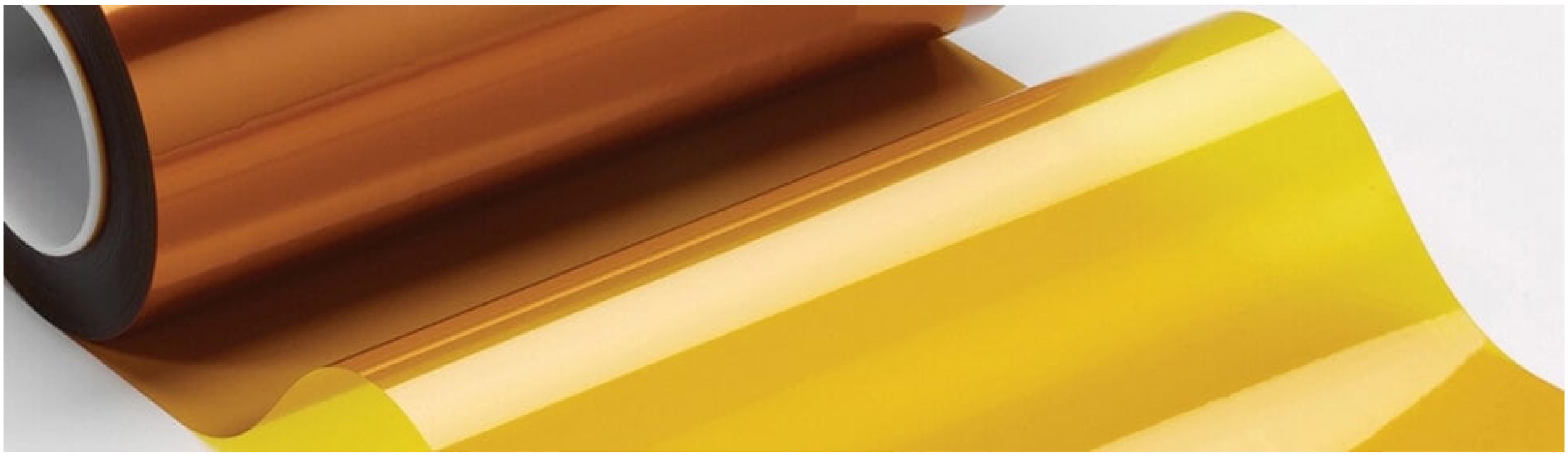
Introduction to Flex PCB Materials

Flex PCB materials need to support multiple design and operational goals: static or dynamic flexing, ability to pass through standard assembly processes, and support for simple fabrication procedures with high yield. Flex PCB materials may seem exotic at first, but a relatively small material set is used to produce flex and rigid-flex PCBs at high volume. In this guide, we'll examine some of the basic properties of flex PCB materials and how they are used to build flex/rigid-flex PCBs.

Substrate and Coverlay Films

The base material used in most common rigid printed circuit boards is woven fibreglass impregnated in epoxy resin. It's actually a fabric, and although we term these "rigid" if you take a single laminate layer they have a reasonable amount of elasticity. It's the cured epoxy which makes the board more rigid. Because of the use of epoxy resins, they are often referred to as organic rigid printed circuit boards. This is not flexible enough for many applications though for simple assemblies where there's not going to be constant movement it can be suitable.

The most common material choice used as a flex PCB substrate is polyimide. This material is very flexible, very tough, and incredibly heat resistant.



Flexible Polyimide film (source: Shinmax Technology Ltd.)

For the majority of flex circuit applications, more flexible plastic than the usual network epoxy resin is needed. The most common choice is polyimide, because it's very flexible, very tough (you can't tear or noticeably stretch it by hand, making it tolerant of product assembly processes), and also incredibly heat resistant. This makes it highly tolerant of multiple solder reflow cycles, and reasonably stable in expansion and contraction due to temperature fluctuations.

Polyester (PET) is another commonly used flex-circuit material, but it's not tolerant enough of high temperatures to survive soldering. I have seen this used in very low cost electronics where the flexible part had printed conductors (where the PET could not handle the heat of lamination), and needless to say nothing was soldered to it - rather, contact was made by crude pressure with an isotropic conductive elastomer.

The display in the product in question (a clock radio) never really worked too well due to the low quality of the flex circuit connection. So for rigid-flex we'll assume we're sticking to the PI film. (Other materials are available but not often used).

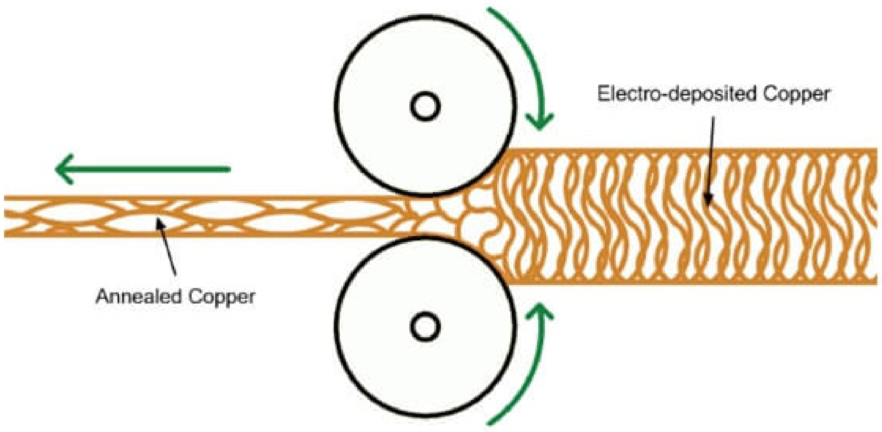
PI and PET films, as well as thin flexible-epoxy-and-glass-fibre cores, form common substrates for flex circuits. The circuits must then use additional films (usually PI or PET, sometimes flexible solder mask ink) for coverlay. Coverlay insulates the outer surface conductors and protects from corrosion and damage, in the same way solder mask does on the rigid board. Thicknesses of PI and PET films range from ⅓ mil to 3 mils, with 1 or 2 mils being typical. Glass fibre and epoxy substrates are sensibly thicker, ranging from 2 mils to 4 mils.

Conductors

While the above-mentioned cheap electronics may use printed conductors - usually some kind of carbon film or silver based ink - copper is the most typical conductor of choice. Depending upon the application different forms of copper need to be considered. If you are simply using the flexible part of the circuit to reduce manufacturing time and costs by removing cabling and connectors, then the usual laminated copper foil (Electro-Deposited, or ED) for rigid board use is fine. This may also be used where heavier copper weights are desired to keep high-current carrying conductors to the minimum viable width, as in planar inductors.

But copper is also infamous for work-hardening and fatigue. If your final application involves repeated creasing or movement of the flex circuit you need to consider higher-grade Rolled Annealed (RA) foils. Obviously the added step of annealing the foil adds to the cost considerably. But the annealed copper is able to stretch more before fatigue cracking occurs, and is springier in the Z deflection direction - exactly what you want for a flex circuit that will be bending or rolling all the time. This is because the rolling annealing process elongates the grain structure in the planar direction.

If you are simply using the flexible part of the circuit to reduce manufacturing time and costs by removing cabling and connectors, then the usual laminated copper foil for rigid board use is fine.



Exaggerated illustration showing the annealing process applied to flex PCB materials, obviously not to scale. The copper foil passes between high-pressure rollers which elongate the grain structure in a planar orientation, making the copper much more flexible and springy in the normal.

Adhesives

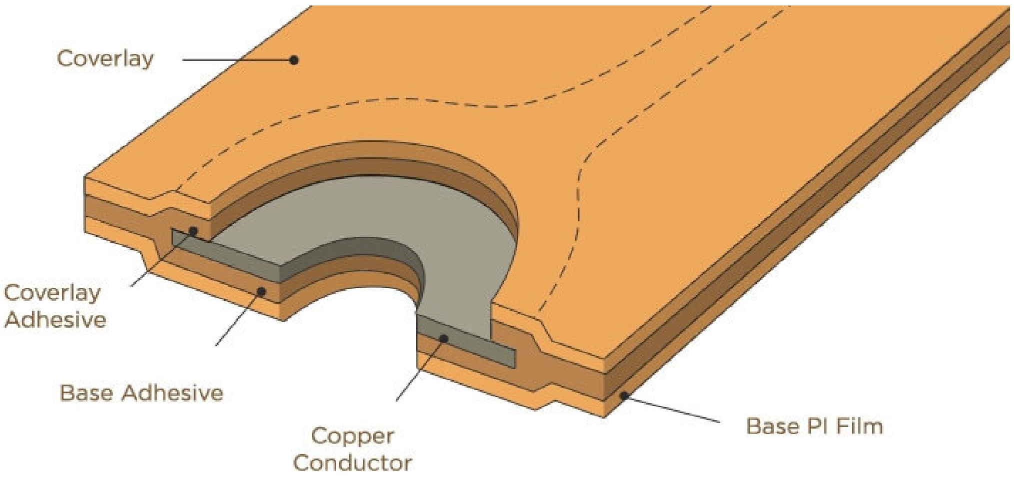
Traditionally, adhesives are required for bonding the copper foil to PI (or other) films, because unlike a typical FR-4 rigid board, there's less "tooth" in the annealed copper, and heat & pressure alone are not enough to form a reliable bond. Manufacturers such offer pre-laminated single- and double-sided copper clad films for flexible circuit etching, using acrylic or epoxy based adhesives with typical thicknesses of ½ and 1 mil. The adhesives are specially developed for flexibility.

"Adhesiveless" laminates are becoming more prevalent due to newer processes that involve copper plating or deposition directly onto the PI film. These films are chosen when finer pitches and smaller vias are needed as in HDI circuits.

Silicones, hot-melt glues, and epoxy resins are also used when protective beads are added to the flex-to-rigid joins or interfaces (i.e. where the flexible part of the layer stack leaves the rigid part). These offer mechanical reinforcement to the fulcrum of the flex-to-rigid join which otherwise would rapidly fatigue and crack or tear in repeated use.

Single Layer Flex Circuits

An example of a typical single layer flex circuit cutaway view is illustrated below. This is the same construction used for most common off-the-shelf FFC (Flexible Flat Connector) cables, which are an alternative to using rigid-flex PCBs where FFC connectors can be accommodated and cost is the primary factor driving design decisions. In single-layer flex circuits, the copper is pre-laminated on to the PI film by the material vendor, then the copper is etched and drilled with a rigid backing plate. This is finally laminated finally with an adhesive-based PolyImide coverlay that is pre-punched to expose the copper pads. The adhesives used in this arrangement for coverlay can squeeze out in the process, but this can be accommodated by enlarging the pads in the exposed areas.



Typical single-layer Flex Circuit stack-up.

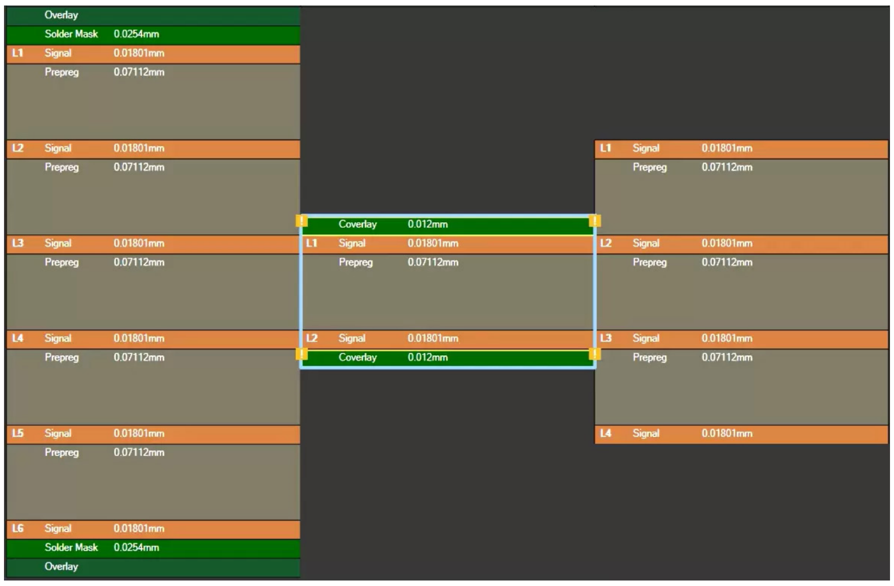
It's important to be aware of the materials used in flexible and rigid-flex circuits. Even though you may generally give a fabricator freedom to select the materials to ensure yield, you should remain aware of the factors that can cause a flex PCB to fail in the field. Knowing the material properties will also help in the mechanical design, evaluation and test of your product. If you are working on automotive products for instance; heat, moisture, chemicals, shock & vibration - all need to be modelled with accurate material properties to determine the product's reliability and minimum allowed bending radius. The irony is that the driving needs that cause you to choose flexible and rigid-flex are often tied to harsh environments. For example, low-cost consumer personal electronic devices are often subjected to vibrations, dropping, sweat and worse.

One great resource with far more detail than introduction shown in this article can be found in Coombs' 2008 textbook:

Coombs, C. F. (Editor, 2008) *The Printed Circuits Handbook,* 7th Ed. 2008 McGraw Hill.

Stackup Example

Just like in rigid PCBs, flex PCBs and rigid-flex PCBs can have complex stackups as more conductive layers are added. These stackups can involve multliple flex sections in the same PCB, such as in the example shown below. For a pure flex circuit (as opposed to rigid-flex) the layer stack planning is simplified, including in each section of the PCB. However, there may still be points requiring placement of stiffeners in areas where components are mounted or where the circuit is terminated.



	Overlay	
	Solder Mask	0.0254mm
L1	Signal	0.01801mm
	Prepreg	0.07112mm
L2	Signal	0.01801mm
	Prepreg	0.07112mm
L3	Signal	0.01801mm
	Prepreg	0.07112mm
L4	Signal	0.01801mm
	Prepreg	0.07112mm
L5	Signal	0.01801mm
	Prepreg	0.07112mm
L6	Signal	0.01801mm
	Solder Mask	0.0254mm
	Overlay	

	Coverlay	0.012mm
L1	Signal	0.01801mm
	Prepreg	0.07112mm
L2	Signal	0.01801mm
	Coverlay	0.012mm

L1	Signal	0.01801mm
	Prepreg	0.07112mm
L2	Signal	0.01801mm
	Prepreg	0.07112mm
L3	Signal	0.01801mm
	Prepreg	0.07112mm
L4	Signal	0.01801mm

In your design software, each of these sections is defined as its own stack and applied to different regions in the PCB layout. When it's time to manufacture the board, each board section will need to be clearly shown in fabrication drawings to illustrate the layer arrangement and materials in the board. We'll discuss this important aspect of flex design and production in a later section.

When you're ready to select and specify the flex PCB materials you need, use the complete set of CAD features and autoamted drawing tools in the Draftsman package inside Altium Designer®. Once you're ready to release your design data to your manufacturer, you can easily share and collaborate on your designs through the Altium 365™ platform. Everything you need to design and produce advanced electronics can be found in one software package.

The Rigid-Flex PCB Fabrication Process

If you thought the rigid PCB fabrication process was complex, then flex PCB manufacturing might seem to be on another level of complexity. However, many of the same steps used in standard rigid board fabrication conceptually transfer flex PCB fabrication. This guide provides an overview of all the steps implemented in the flex PCB manufacturing process. These processes are very similar to rigid PCB fabrication processes, but involving different material sets as illustrated in an earlier guide.

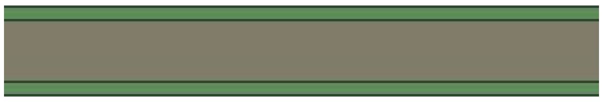
Flex Build-Ups

At first glance, a typical flex, or rigid-flex board, looks straightforward. However the nature of these requires several additional steps in the build-up process. The beginning of any rigid flex board is always the single or double-sided flex layers. The fabricator may begin with pre-laminated flex with foil, or may begin with unclad PI film, and then laminate or plate up the copper for the initial cladding. Laminating the film requires a thin layer of adhesive, whereas adhesiveless cladding requires a “seed” layer of copper. This seed layer is initially planted using vapor deposition techniques (i.e. sputtering), and provides the key to which chemically deposited copper is plated upon. This one or two-sided flex circuit is drilled, plated through, and etched in much the same steps as typical 2-sided cores in rigid boards.

Flex Fab Steps

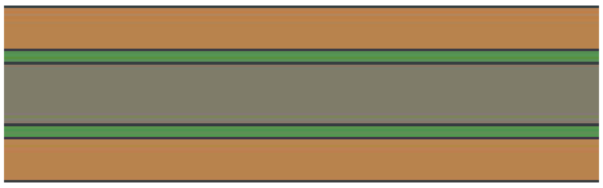
The steps below show Flex-Circuit creation for a typical double-sided flex circuit.

Step 1: Adhesive/Seed coating applied



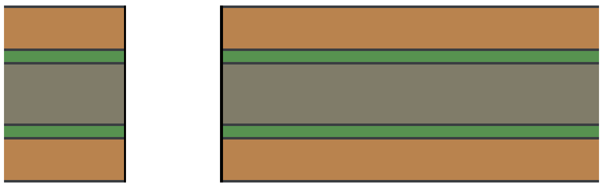
Either an epoxy or acrylic adhesive is applied, or sputtering is used to create a thin copper layer for a plating key.

Step 2: Copper foil added



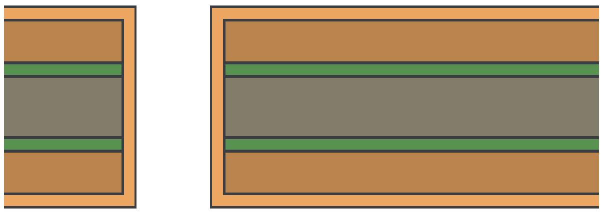
Copper foil is added, either by lamination to the adhesive (the more mainstream approach) or chemical plating onto the seed layer. Newer fabrication processes by materials vendors allow adhesiveless lamination of rolled annealed copper as an alternative.

Step 3: Drilling



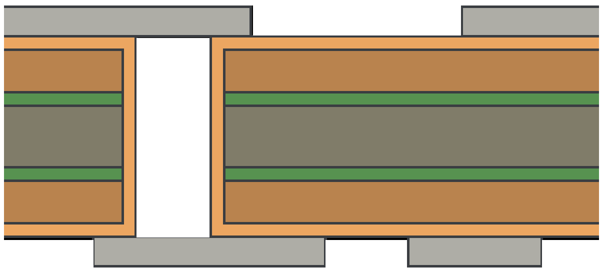
Holes to vias and pads are most often mechanically drilled. Multiple plated flex substrates can be drilled simultaneously by combining them from multiple reels on drums, drilling between work plates, then rolling out to separate reels on the other side of the drilling machine. Pre-cut flex panels can be combined and drilled between rigid blanks in the same way rigid cores are drilled as well, though it requires more careful registration and the alignment accuracy is reduced. For ultra-small holes, laser drilling is available, though at much added cost because each film has to be drilled separately. This would use Excimer (ultraviolet) or YAG (Infrared) lasers for higher accuracy (microvias), and CO2 lasers for medium holes (4+ mils). Large holes and cutouts are punched, but this is a separate process step.

Step 4: Through-hole plating



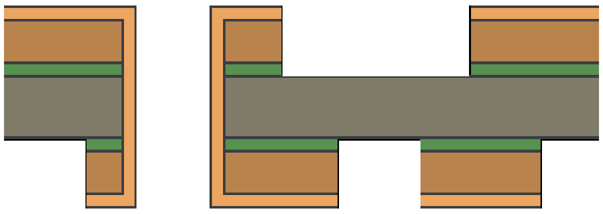
Once the holes are made, copper is deposited and chemically plated in the same way as rigid PCB cores (commonly referred to as Cuposit). Through-hole plating in flex circuits is recommended to be at least 1 mil in plating thickness to add mechanical support to the pad or via, whereas a typical low-cost rigid PCB may only have ½ mil cuposit.

Step 5: Etch-resist printing



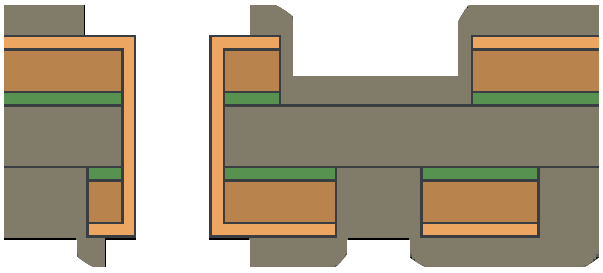
A photosensitive etch resist is coated onto the film surfaces, and the desired mask pattern is used to expose and develop the resist prior to chemical etching of the copper.

Step 6: Etching and stripping



After exposed copper is etched, the etch resist is chemically stripped from the flex circuit.

Step 7: Coverlay or covercoat



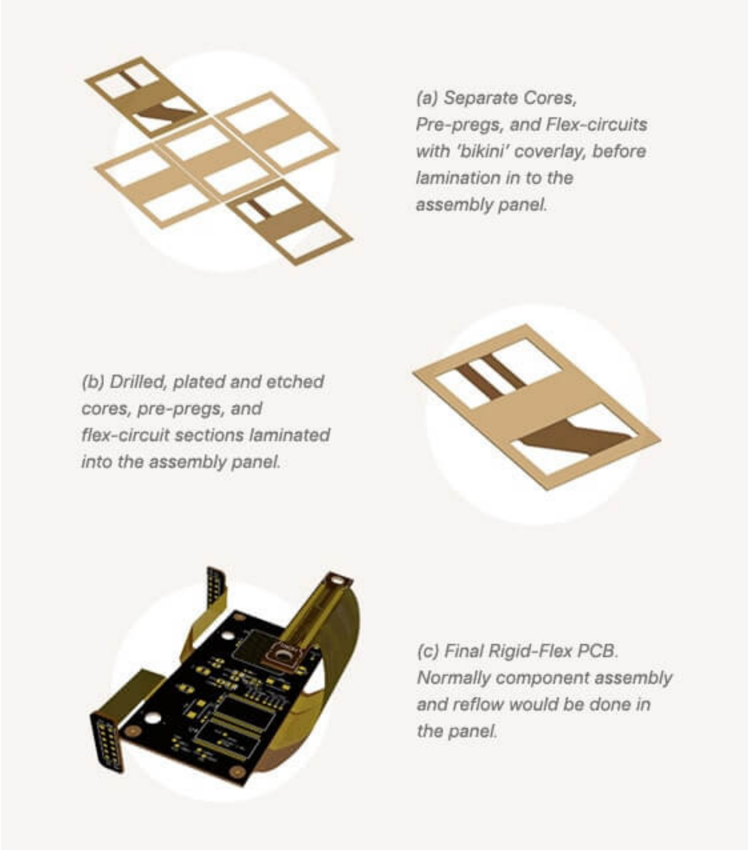
Top and bottom areas of the flex circuit are protected by coverlay layers which are cut to shape. There may be components actually mounted on sections of the flexible circuit, in which case the coverlay is also acting as a solder mask. The most common coverlay material is additional polyimide film with adhesive, though adhesiveless processes are available. In the adhesiveless process, photoimageable solder mask (the same as used on rigid board sections) is used, essentially printing the coverlay onto the flex circuit. For coarser cheaper designs screen printing is also an option with final curing of this covercoat film by UV exposure. Basically, the difference is that coverlay is a laminated film, whereas covercoat is an applied material coating which then needs to be cured.

Step 8: Cutting out the flex

The final step in creating the flex circuit is cutting it out. This is often referred to as “blanking”. The high-volume cost-effective approach to blanking is by a hydraulic punch and die set, which involves reasonably high tooling costs. However, this method allows punching out of many flex circuits at the same time. For prototype and low-volume runs, a blanking knife is used. The blanking knife is basically a long razor blade, bent into the shape of the flex circuit outline and affixed into a routed slot in a backing board (MDF, plywood or thick plastic such as teflon). The flex circuits are then pressed into the blanking knife to be cut out.

Lamination and Routing

If the flex circuit is to form a part of a rigid/flex combined stack-up (which is what we are interested in), the process doesn’t stop there. We now have a flex circuit that needs to be laminated in between the rigid sections. This is the same as an individual drilled, plated and etched core layer pair, only much thinner and more flexible due to the lack of glass fibre. As noted previously though, a less flexible layer could be made with PI and glass depending on the target application. Because this is being laminated in between rigid sections, it ultimately has to be framed in a panel that mates with the rigid board panel sections as well. Flex circuits that are not being combined with rigid sections are adhered temporarily to a rigid backing board of MDF or FR-4 style materials.



(a) Separate Cores, Pre-pregs, and Flex-circuits with ‘bikini’ coverlay, before lamination in to the assembly panel.

(b) Drilled, plated and etched cores, pre-pregs, and flex-circuit sections laminated into the assembly panel.

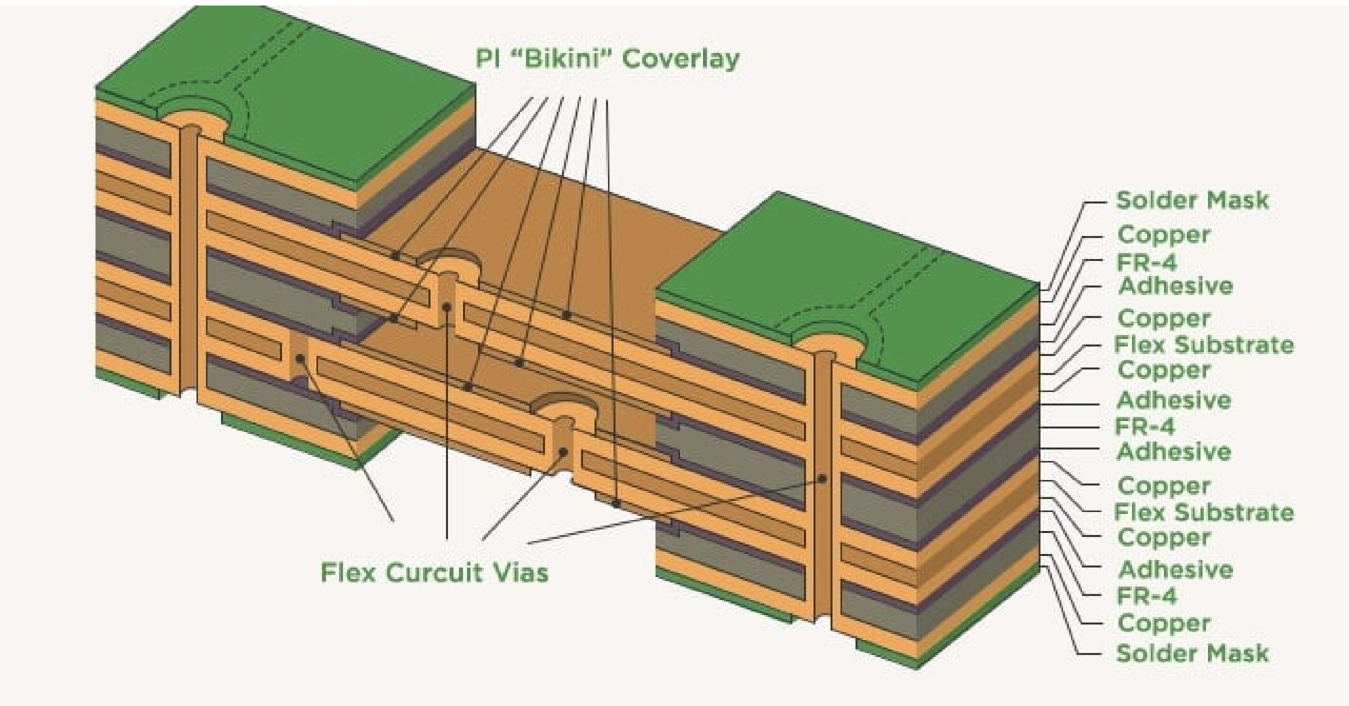
(c) Final Rigid-Flex PCB. Normally component assembly and reflow would be done in the panel.

How the etched, plated, coverlayed, and blanked flex panels are combined with the glass-epoxy rigid panels.

The flex circuit is laminated into the panel along with the rigid and any other flexible sections, with additional adhesive, heat and pressure. Multiple flex sections are not laminated adjacent to each other unless you are designing multi-layer flex. This generally means each flex section has a maximum copper layer count of 2, so that flexibility is maintained. These flex sections are separated by rigid pre-pregs and cores or PI bonding sheets with epoxy or acrylic adhesives.

Essentially, each rigid panel is separately routed out in the areas where the flex is going to be allowed to, well, flex. Here is an example process of laminating into a rigid-flex board, with two, 2-layer flex circuits embedded between three rigid sections. The layer stack up would look like that shown below.

Note: *Many designers are shying away from using adhesives, due to unacceptable z-axis expansion during reflow soldering.*



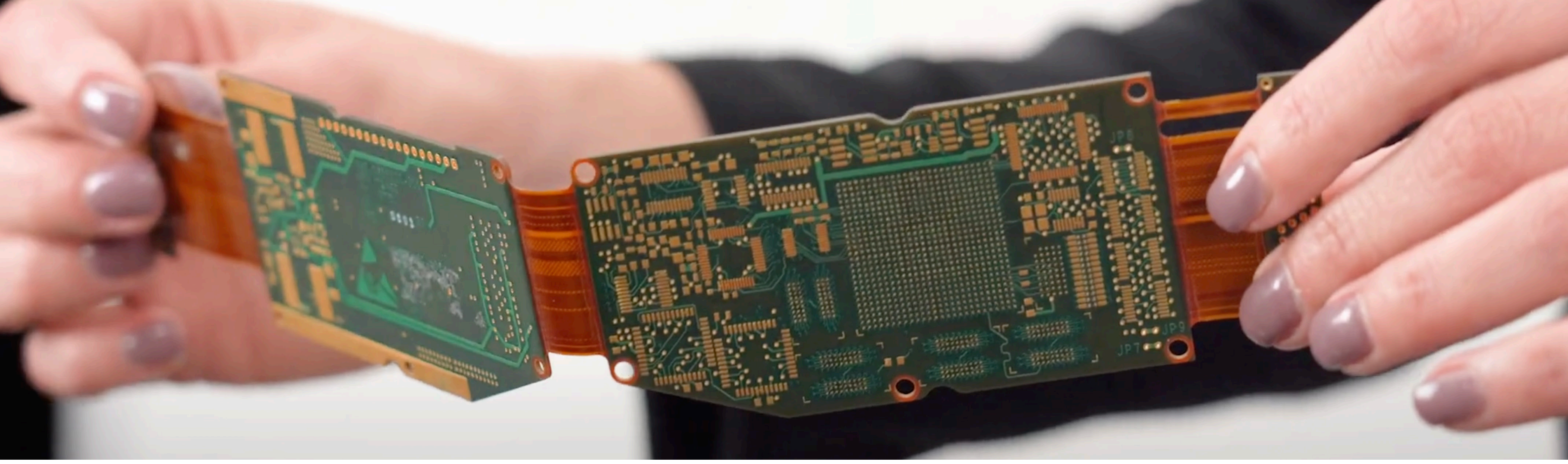
Detailed Stack Diagram including plated-through holes for each flex section, as well as final through-plated holes in the rigid section.

In the example stack up shown above, we have two pre-etched and cut flex circuits, each double sided and plated through. The flex circuit has been blanked into a final assembly panel including borders for framing - this will keep the flex circuit flat during final assembly after lamination with the rigid panel sections. There are certainly some potential hazards with inadequate support of flex circuit elbows and large open sections during assembly - especially in the heat of a reflow oven.

While this example does show adhesive layers it's important to note that many designers are shying away from using adhesives, due to unacceptable z-axis expansion in reflow. However FR-4 prepreg and thermosetting epoxies effectively achieve the desired result and are for all intents and purposes here considered 'adhesive' layers. Additional adhesion can be achieved through treatment of the copper on the flex layers to improve the 'tooth' into the laminated prepregs. Adhesiveless double-sided flex laminates are shown here. These are entirely polyimide film with a bondable polyimide coating which the copper foil is bonded to. DuPont™ Pyralux® and Rogers Corp. R/Flex® are examples of popular adhesiveless laminates.

The coverlay is also applied - like stickers laminated on with adhesive, or by a photo-printing process as mentioned earlier. Once the final flex and rigid panels in this 6-layer stackup are placed together, they are laminated with the outermost (top and bottom) final copper foil layers. Then another drilling for top-to-bottom plated through holes is done. Optionally, laser drilled blind vias (top to first flex, bottom to last flex) could also be made, again adding expense to the design. The holes are plated through from top to bottom, and blind vias if there are any, and the final outer layer copper patterns are etched. The final steps are the printing of the top and bottom solder mask, top and bottom silkscreen and preservative plating (such as ENIG) or hot air leveling (HASL).

When you're ready to create documentation for the rigid-flex PCB fabrication process and plan your new design for manufacturing, use the complete set of CAD features and automated drawing tools in the Draftsman package inside Altium Designer®. Once you're ready to release your design data to your manufacturer, you can easily share and collaborate on your designs through the Altium 365™ platform. Everything you need to design and produce advanced electronics can be found in one software package.



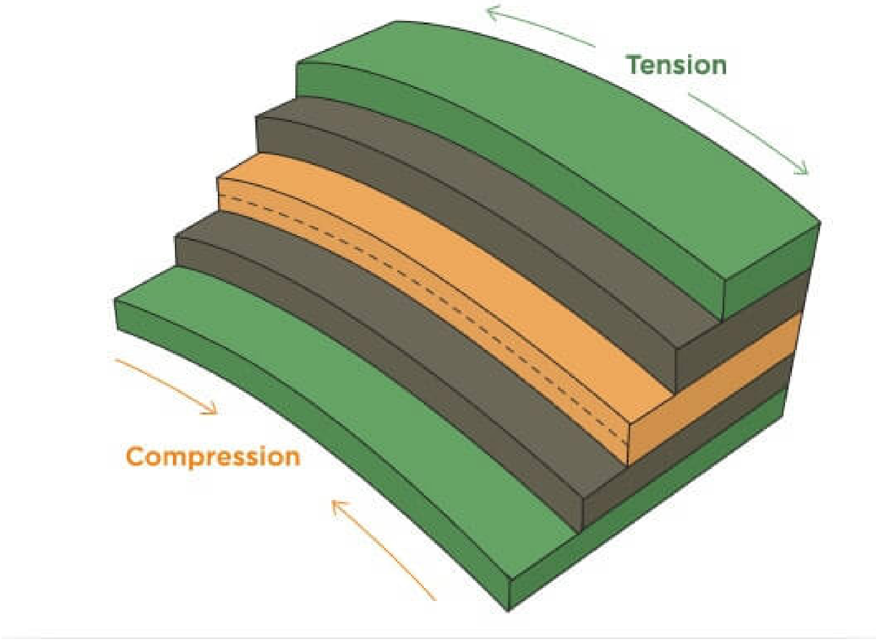
Flexible Printed Circuit Design Best Practices

In order to produce reliable rigid-flex based products, there are many considerations relating the fabrication and the end-use of the flex circuit, to the design of the copper pattern. Before you start placing and routing circuits in your flex/rigid-flex PCB, make sure you follow these flexible printed circuit engineering tips to ensure high yield and durability. These tips will help you balance durability in flex designs with the need to place components and route traces in flex boards or regions in advanced PCBs.

Physical Constraints in Flexible Printed Circuit Design Multiple Flex Sub-Stacks

While it's possible to build just about any stack-up with rigid and flex sections, it can get ridiculously expensive if you're not careful to consider the production steps and the material properties involved. One important aspect of flex circuits to remember is the stresses within the materials occurring as the circuit bends. Copper, being a non-ferrous metal, is known to suffer work-hardening, and fatigue fractures will occur eventually with repeated flex cycling and tight radii. One way to mitigate this is to only use single-layer flex circuits, in which case the copper resides at the center of the median bend radius and therefore the film substrate and coverlay are in the greatest compression and tension, as shown below.

Along the same lines, having multiple separate flex circuits is often necessary, but it's best to avoid having bends at overlapping sections where the length of the flex sections limits the bend radius. Since the polyimide is very elastic this is not a problem, and will last much longer under repeated movement than multiple copper layers will. The copper resides at the center of the median bend radius and therefore the film substrate and coverlay are in the greatest compression and tension.



For highly repetitive bending circuits, it's best to use RA copper in single-layer flex to increase the fatigue life (in cycles before failure) of the copper in the circuit.

Adhesive Beads, Stiffeners, and Terminations

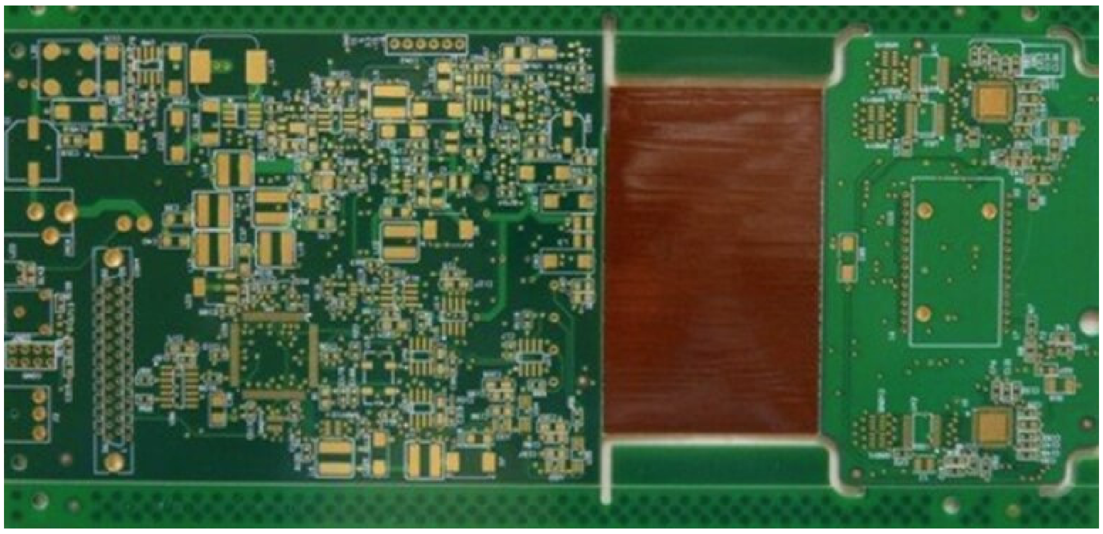
There are times when you need to consider using strengtheners where the flex circuit exits the rigid board. Adding a bead of epoxy, acrylic or hot-melt will help improve the longevity of the assembly. But dispensing these liquids and curing them can add laborious steps to the production process, increasing cost. As always with PCB design, there are trade-offs.

Automated fluid dispensing can be used, but you need to be really careful to collaborate with the assembly engineers to make sure you don't end up with globs of glue dripping under the assembly. In some instances the glue must be applied by hand which adds time and cost. Either way, you need to provide clear documentation for the fabrication and assembly folks.

Extreme ends of flex circuits typically terminate to a connector if not to the main rigid board assembly. In these cases, the termination can have a stiffener applied (more thick Polyimide with adhesive, or FR-4). Generally then, it's convenient to leave the ends of the flex embedded within the rigid-flex sections as well.

Rigid-flex PCB Panels

The rigid flex circuit stays together in it's panel for the assembly process, so components can be placed and soldered onto the rigid termination sections. Some products require components to be mounted also on flex in some areas, in which case the panel has to be put together with additional rigid areas to support the flex during assembly. These areas are not adhered to the flex and are routed out with a controlled-depth router bit (with "mouse-bites") and finally punched out by hand after assembly.



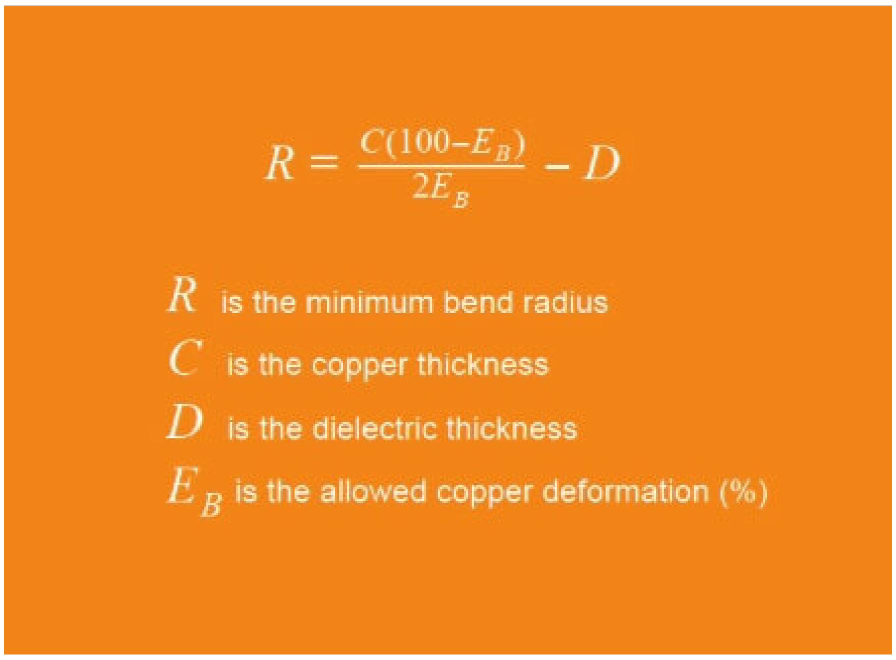
Example rigid-flex PCB panel. Notice that this one has front and back board edges, and flex circuit, routed out. The rigid sides are V-grooved for snapping off later. This will save time in assembly into the enclosure (source: YYUXING Shenzhen Electronics Co., LTD.).

It's easy to look at the problems of layer stack design, parts placement, and cutouts and think we've got the issues down. But remember how flex circuits have some gnarly material quirks. Quirks ranging from relatively high z-axis expansion coefficients of adhesives, to the lower adhesion of copper to PI substrate and coverlay, to copper's work hardening and fatigue. These can be compensated for largely by following some Dos and Don'ts.

Do Keep Flex Flexible

This may seem obvious, but it's worth saying. Decide just how much flex is needed up front, and whether flexing needs to be repeatable, or if the design will have a static bend. If your flex-circuit sections are only going to be folded during assembly and then left in a fixed position - such as in a handheld ultrasound device - then you are a lot freer in the number of layers, the type of copper (RA or ED) and so on you can use. On the other hand, if your flex-circuit sections are going to be continually moving, bending or rolling, then you should reduce the number of layers for each sub-stack of flex, and choose adhesiveless substrates.

Then, you can use the equations found in IPC-2223 (Eq. 1 for single-sided, Eq. 2 for double, etc.) to determine what is your minimum allowable bending radius for the flex section, based on your allowed deformation of copper and the characteristics of the other materials.



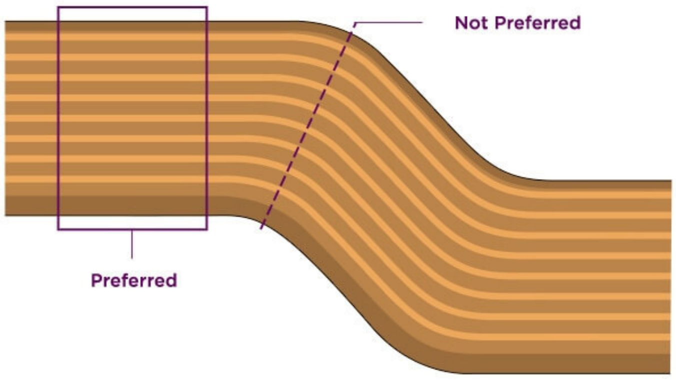
$$R = \frac{C(100-E_B)}{2E_B} - D$$

R is the minimum bend radius
C is the copper thickness
D is the dielectric thickness
E_B is the allowed copper deformation (%)

This example equation is for a single-sided flex section. It can be used with an assembled flex PCB, although you might stress solder points on component leads if the bending line is mis-located. You need to choose EB based on the target application, with 16% for single-crease installation of RA copper, 10% "flex-to-install" and 0.3% for "dynamic" flex designs (Source: IPC-2223B, 2008 http://www.ipc.org/TOC/IPC-2223B.pdf). Here, dynamic means continuous flex and roll during use of the product, such as a TFT panel connection on a mobile DVD player.

Don't Bend at Corners and Use Curved Traces

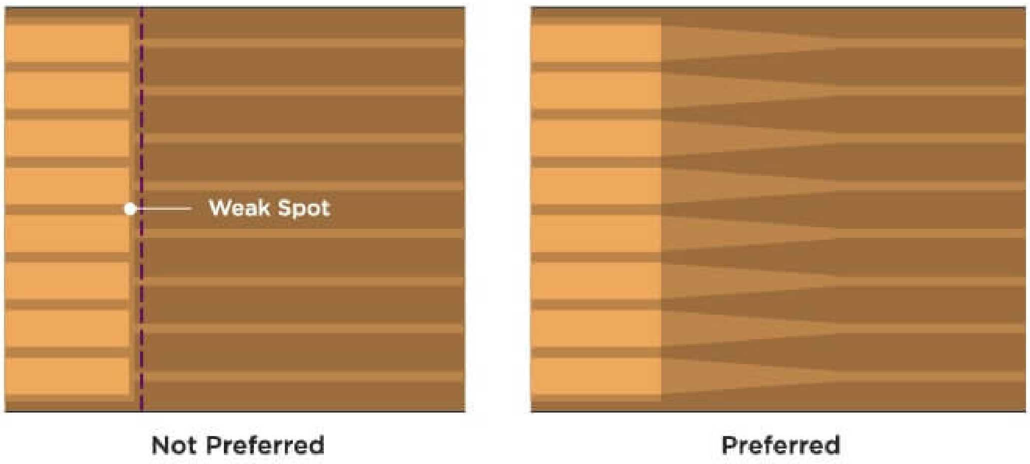
It is generally best to keep copper traces at right-angles to a flex-circuit bend. However there are some design situations where it's unavoidable. In those cases keep the track work as gently curving as possible, and as the mechanical product design dictates, you could use conical radius bends. Also referring to the image below, it's best to avoid abrupt hard right-angle trackwork, and even better than using 45° hard corners, route the tracks with arc corner modes. This reduces stresses in the copper during bending.



Preferred bend locations.

Don’t Abruptly Change Widths

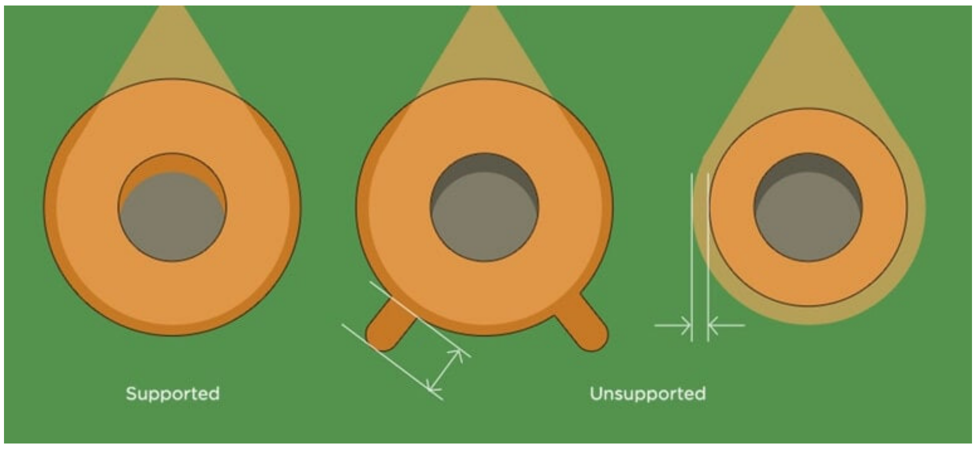
Whenever you have a track entering a pad, particularly when there is an aligned row of them as in a flex-circuit terminator (shown below), this will form a weak spot where the copper will be fatigued over time. Unless there is going to be stiffener applied or a one-time crease near the trace width transition, it’s advisable to taper down from the pads (hint: place teardrops on the pads and vias in the flex circuit!)



Trace width change and pad entries can cause weak spots.

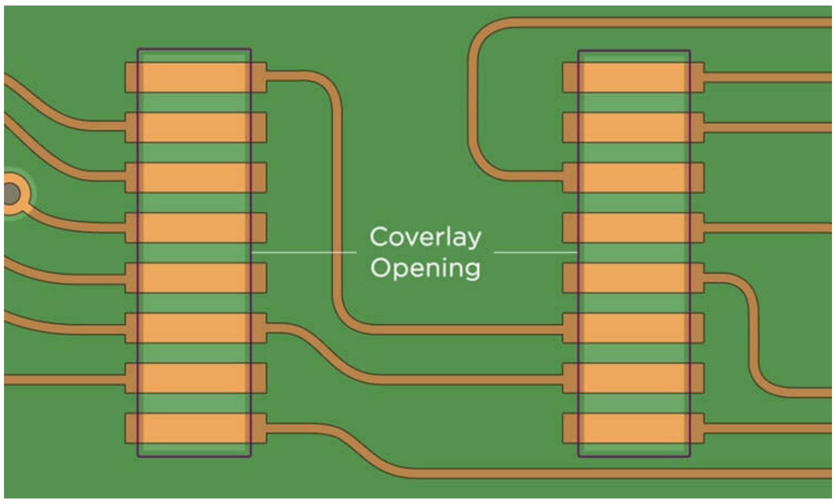
Do Add Support for Pads

Copper on a flex circuit is more likely to detach from a polyimide substrate due to the repeated stresses involved in bending, as well as the lower adhesion of copper to the substrate (relative to FR-4). It is especially important therefore to provide support for exposed copper. Vias are inherently supported because the through-hole plating offers a suitable mechanical anchor from one flex layer to another. For this reason (as well as z-axis expansion) many fabricators will recommend additional through-hole plating of up to 1.5 mils for rigid-flex and flex circuits, in addition to the conventional plating in rigid circuit boards. Surface mount pads and non-plated-through pads are referred to as unsupported, and need additional measures to prevent detachment.



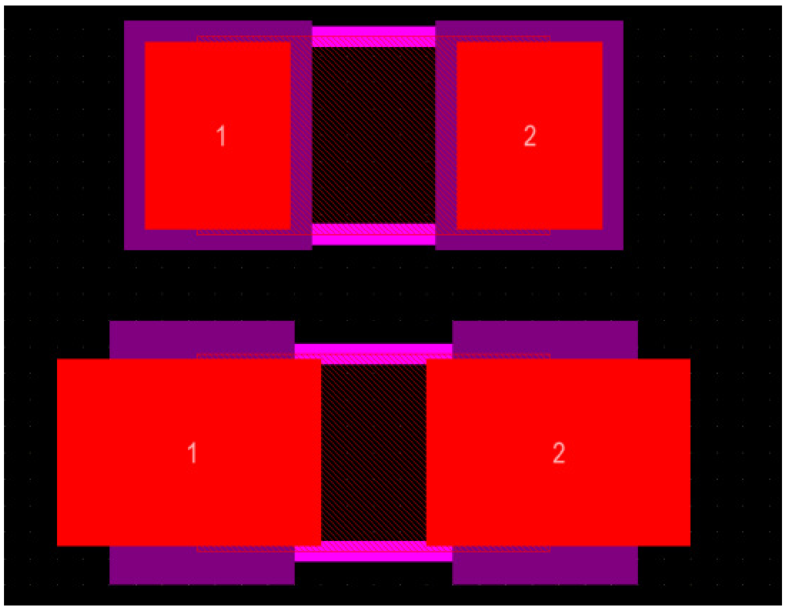
Supporting through-hole pads in flex with plating, anchoring stubs, and reduced coverlay access openings.

SMT component pads are among the most vulnerable, especially as the flex circuit may bend under the component's rigid pin and solder fillet. The pad and trace arrangemeny below shows how using the coverlay "mask" openings to anchor pads one 2 sides will solve the problem. To do this while still allowing the right amount of solder the pads have to be somewhat larger than typical rigid-board footprints would have. This obviously reduces the density of flex circuit component mounting, but by nature flex circuits cannot be very dense compared with rigid.



Coverlay openings for an SOW package showing anchoring at each end of each pad.

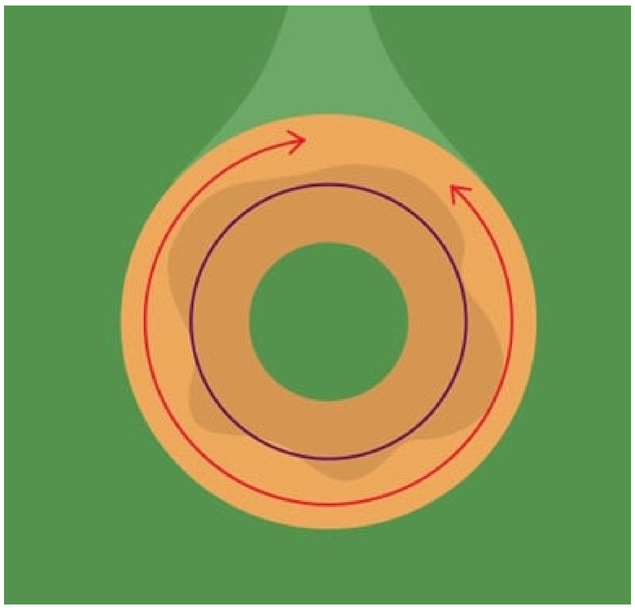
Inside your PCB design software, there is no "coverlay" layer specifically; you'll have to use a mask layer to define the coverlay opening around the pads. This can be done in the top solder layer inside the flex section; simply place an opening in the mask layer to define the coverlay opening, just as you would with solder mask. The pads on the footprint will also need to be modified to ensure accurate assembly and to add just enough extra covering for anchoring. An example for a 0603 component footprint is shown below.



In this footprint, the pad sizes and and top solder layer are used to show how the pads for an SMD passive and the coverlay opening should be placed for mounting on a rigid-flex PCB. The top land pattern is for a nominal 0603 package, whereas the bottom is a footprint from the same component, but with a modified coverlay opening.

Allow for Squeeze-Out

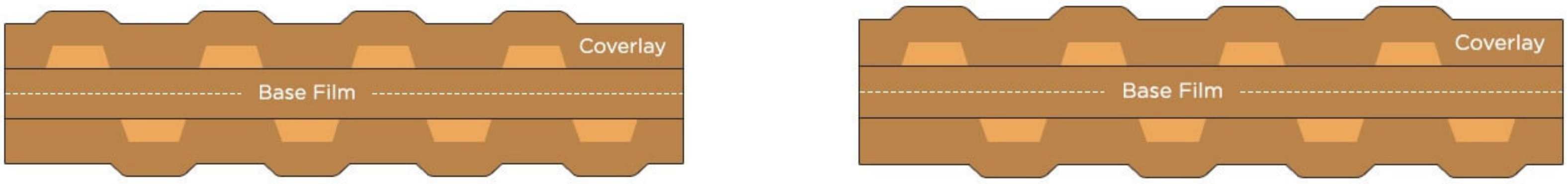
When a coverlay is laminated over the copper and substrate, some adhesive will exhibit "squeeze out" from any coverlay openings around pads when the coverlay is applied. To allow for squeeze out, the pad land and the access opening must be large enough to allow some adhesive leakage while still leaving enough exposed copper for a strong solder fillet. IPC-2223 recommends 360° solder wetting around the hole for high reliability designs and 270° for moderate reliability flex designs.



Size pads and coverlay openings to allow for adhesive squeeze out.

Double-Sided Flex Routing

For dynamic double-sided flex circuits, try to avoid laying traces over each other on the same direction. Instead, stagger trces across adjacent layers so that they do not overlap. This reduces tension stress on the traces when copper is more evenly distributed between copper layers (see below). In the case where the traces are overlapped, one of the layers will experience more stress during bending as the layers push against each other. Staggering spreads the stress across the flex substrate so that the stress distribution on the traces is closer to being uniform.

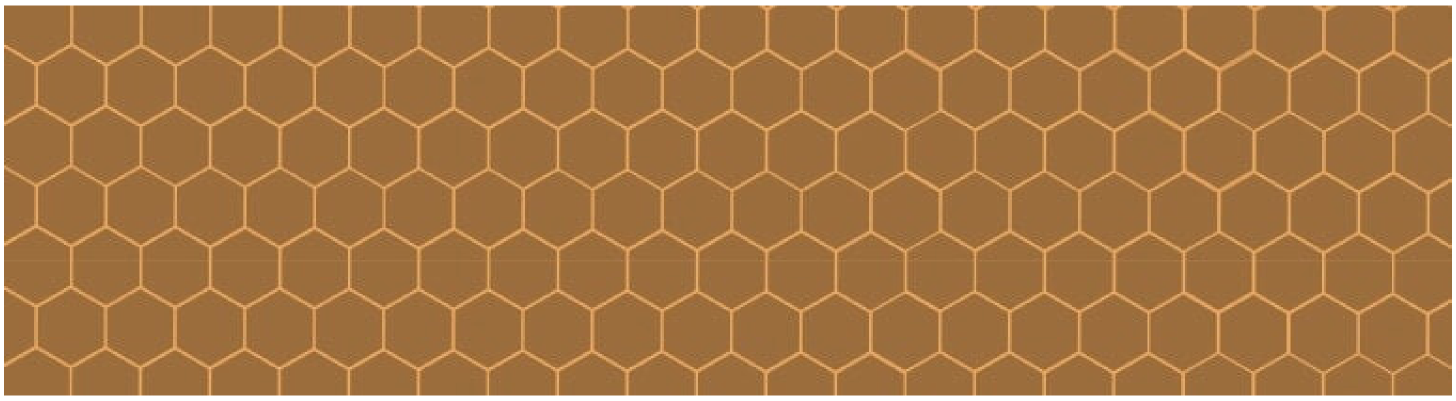


Adjacent-layer copper traces (top image) are not recommended. Instead, stagger traces in different layers so that stress on the traces is reduced when the assembly is flexed.

Do Use Hatched Polygons

Sometimes it's necessary to carry a power or ground plane on a flex circuit. Using solid copper pours is okay, as long as you don't mind significantly reduced flexibility, and possible buckling of the copper under tight-radius bends. Generally it's best to use hatched polygons to retain a high level of flexibility.

A normal hatched polygon still has heavily biased copper stresses in 0°, 90°, and 45° angle directions, due to alignment of hatch traces and 'X'es. A more statistically optimal hatch pattern would be hexagonal. This could be done using a negative plane layer and an array of hexagonal anti-pads, but you can quickly build the hatch shown below by cutting and pasting sections.



Using hexagonal hatched polygons can spread the tension biases evenly among three angles.

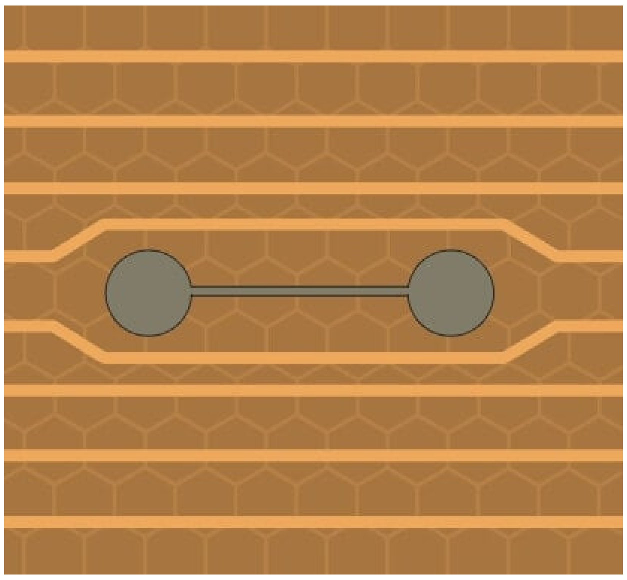
Via Placement

For multi-layer flex areas, it may sometimes be necessary to place vias to transition between layers. If possible it's recommended not to place vias, as these can suffer fatigue rapidly in flexing movement. It is also necessary to keep at least 20 mils (about a ½ mm) clearance between the copper annulus of the nearest via to the rigid-to-flex board interface. Board edge clearance rules can take care of this automatically in the PCB CAD editor.

As for the need to place vias - if you must have vias in a flex circuit, use "rooms" to define regions where you know there will be no bends and use the PCB editor's design rules to allow via placement only in those stationary areas. An alternative is to use the layer stack manager to define "rigid" sections that are ultimately flex but with a rigid dielectric stiffener material adhered to them.

Defining Flex Cutouts and Corners

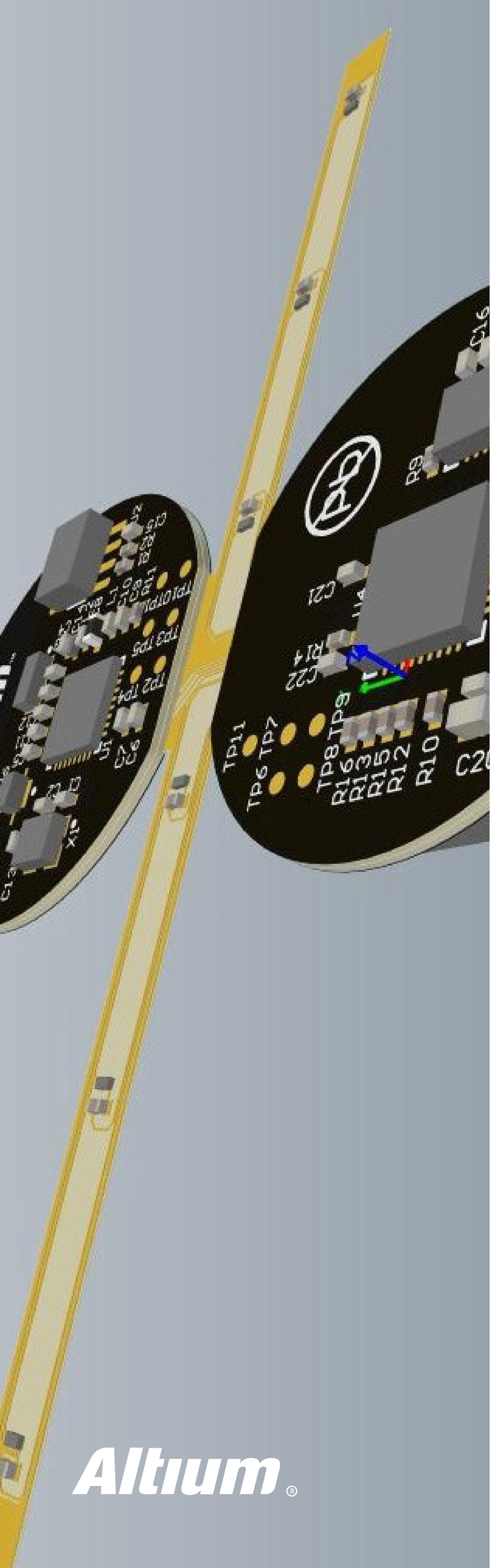
If you need to place a cutout or slot in the flex section of a board, the cutout should be terminated properly. IPC recommends terminating with circular sections with radii greater than 1.5 mm (about 60 mils) to reduce the risk of tearing the flex substrate materials at the corners. The rule here, in essence, is that whenever you have an inside corner (a flex-circuit edge corner with angle less than 180°), always use a tangential curved corner with radius greater than 1.5 mm. If the corner is much less (more acute) than 90° then have a circular curve punched out of it. The same goes for slots and slits in the flex - make sure there's a designed-in relief hole at each end of diameter 3 mm (1/8") or more. An example of this is shown below.



Slots, slits, and inside corners should have tear-relief holes or tangent curves with minimum 1.5mm radius.

This is by no means a complete set of flexible printed circuit engineering guidelines, but these tips should help you get started for many products. If you're ever unsure, your fabrication house should be available to give you DFM guidelines for your flexible board, or for flex sections in a rigid-flex PCB.

When you're ready to start designing flexible printed circuits for your next product, use the complete set of CAD features in Altium Designer®. Once you're ready to release your design data to your manufacturer, you can easily share and collaborate on your designs through the Altium 365™ platform. Everything you need to design and produce advanced electronics can be found in one software package.



Preparing Rigid-Flex PCB Documentation for Manufacturing

Anytime you design a new product that will be manufactured at scale, your job suddenly shifts from knowing everything about the design to knowing everything about the manufacturing process. You'll have one last set of tasks to complete before your job as a designer is complete: prepare the manufacturing documentation your manufacturer needs to begin production on your rigid-flex circuit board. For rigid-flex, the fabrication concerns and expectations are a bit different from those expected in rigid PCBs. In the coming sections, we'll provide some of the information you should specify in your fabrication documentation, specifically in your fabrication notes and drawings so that you can prevent misunderstandings when you transition to production.

Documenting Your Flex PCB for Manufacturing and Assembly

When you’re preparing documentation, you’re essentially telling the fabricator what you want to see in the rigid-flex assembly; it’s the most likely part of the process where errors or misunderstandings can cause costly delays. Fortunately, there are standards we can reference to make sure we are communicating clearly to the fabricator, in particular IPC-2223B (which I am referencing in writing this).

It could boil down to a few golden rules:

- Make sure your fabricator is capable of building your rigid-flex design.
- Make sure they collaborate with you on designing your layer stack to fit their particular processes.
- Use IPC-2223 as your point of reference for design, making sure the fabricator uses the same & related IPC standards - so they are using the same terminology as you.
- Involve them as early as possible in the process.

Careful attention needs to be paid to layer pair planning and documentation for drilling and through-hole plating, because blind vias from a rigid surface layer down to an opposing flex-circuit layer will have to be back-drilled and add significant cost and lower yield to the fab process.

Output Data Set

In interviewing a handful of rigid-flex capable board houses locally, we found that many designers still present Gerber files to the board house. However, ODB++ v8.1 or later is preferred, since it has specific layer types added to the job matrix that enable clear flex- documentation for GenFlex® and similar CAM tools. A subset of the data included is shown in the following table.

Layer Type	Base Type	Description
Coverlay	solder_mask	Clearances of a coverlay layer
Covercoat	solder_mask	Clearances of a covercoat layer
Punch	route	Pattern for die-punching of the flex
Stiffener	mask	Shapes and locations of stiffeners to be adhered
Bend Area	mask	Labelling of areas that will be bent while in use
PSA	mask	Pressure Sensitive Adhesive shapes and locations
Area	document	An area definition (Rigid, Flex, or arbitrary)t
Exposed Area	document	An exposed area of an inner layer and it's associated coverlay (could also be used for embedded components)
Signal Flex	signal	A signal layer for a flex
Power Ground Flex	pg	A power of ground layer for a flex
Mixed Flex	mixed	Mixed layer for a flex
Plating_mask	mask	A mask for defining which areas within a layer should be masked off from plating process
Immersion_Mask	mask	A mask for defining which areas within a layer should be masked off for immersion gold

Subset of Layer Types in ODB++ (v8.1 and later) used for GenFlex [Source: ODB++ v8.1 Specification]

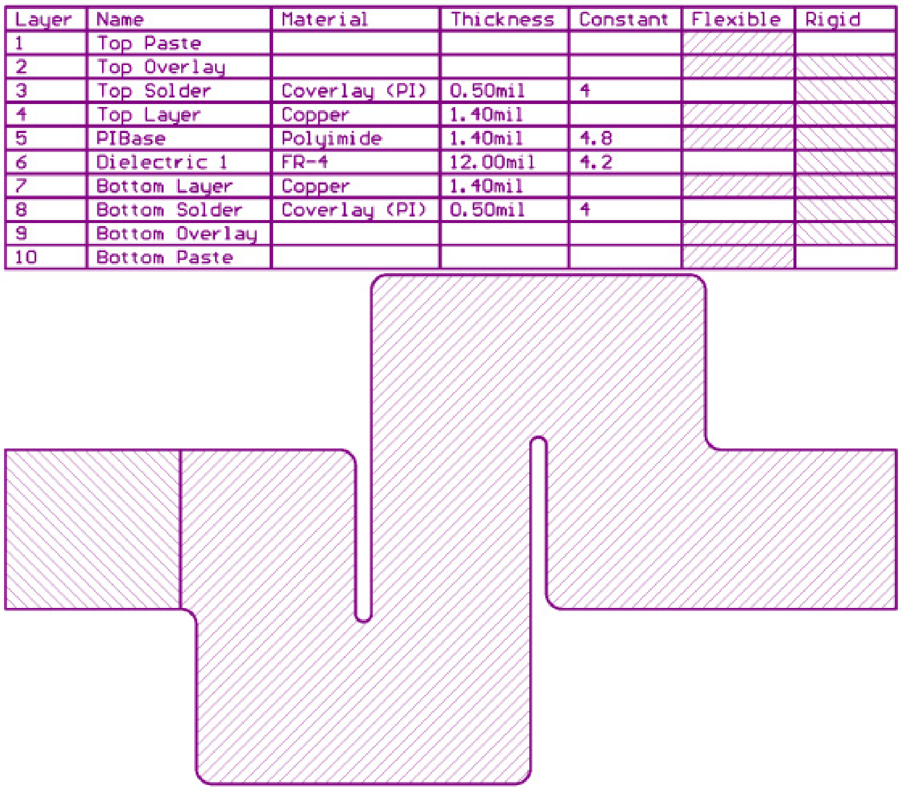
There are some issues we face if using Gerber for the output data set, or earlier versions of ODB++. Namely, the fabricator will need separate route tool paths and die cut patterns for each rigid and flex section in the layer stack. Effectively, mechanical layer films would need to be produced to show where voids need to be in the rigid areas, and more to show where coverlay or covercoat will be on the exposed flex areas. The coverlay or covercoat also has to be considered a mask for component pads for those components that may be mounted on flex areas.

In addition, careful attention needs to be paid to layer pairs for drilling and through-hole plating, because blind vias from a rigid surface layer down to an opposing flex- layer will have to be back-drilled and add significant cost and lower yield to the fab process.

As a designer, the question is really then, how can I define these areas, layers and stacks?

Define the Stack by Area Using a Table

The most important documentation you can provide your fabricator is arguably the layer stack design. Along with this, if you're doing rigid-flex, you have to provide different stacks for different areas, and somehow mark those very clearly. A simple way to do this is make a copy of your board outline on a mechanical layer, and lay down a layer stack table or diagram with a pattern-fill legend for the regions containing the different layer stacks. An example of this is shown below.



Layer	Name	Material	Thickness	Constant	Flexible	Rigid
1	Top Paste				▨	
2	Top Overlay				▨	▧
3	Top Solder	Coverlay (PI)	0.50mil	4		▧
4	Top Layer	Copper	1.40mil		▨	▧
5	PIBase	Polyimide	1.40mil	4.8	▨	▧
6	Dielectric 1	FR-4	12.00mil	4.2		▧
7	Bottom Layer	Copper	1.40mil		▨	▧
8	Bottom Solder	Coverlay (PI)	0.50mil	4		▧
9	Bottom Overlay				▨	▧
10	Bottom Paste				▨	

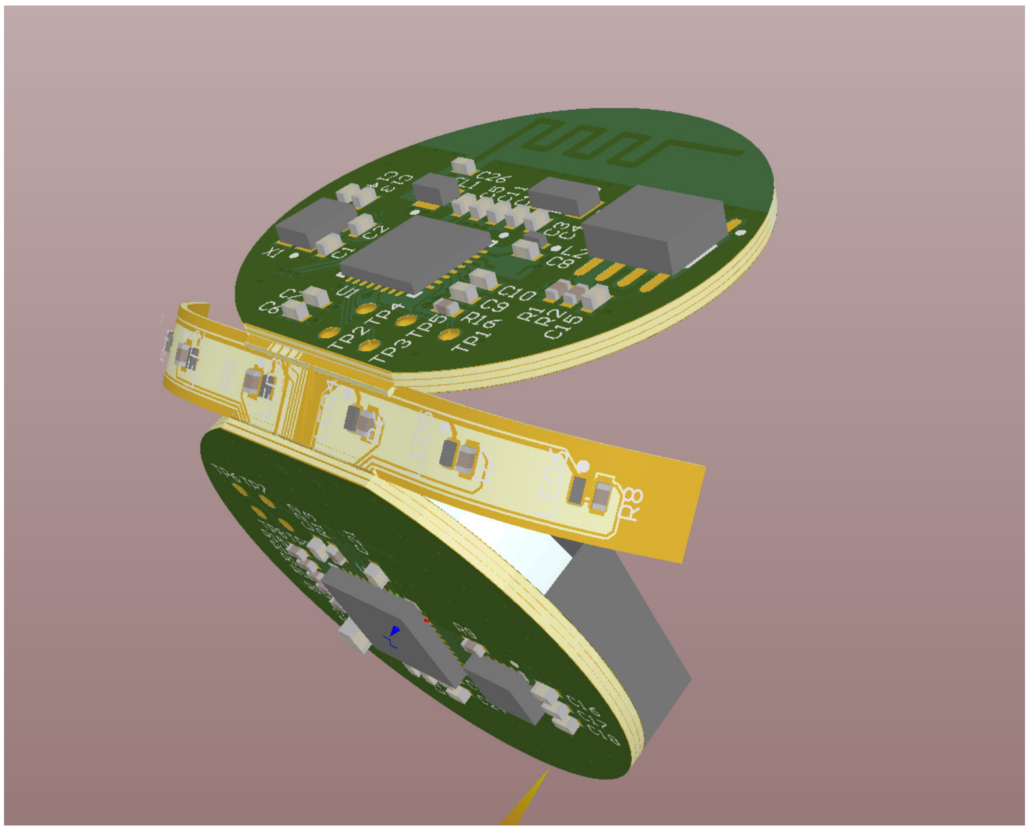
An example of a stack diagram showing fill patterns for rigid and flex areas.

In this example, I used matching fill patterns for different stack areas to indicate which stackup layers are included in the Flexible part or the Rigid part. You can see here the layer item I named "Dielectric 1" is actually an FR-4 core, which could alternatively be considered a stiffener.

Conveying the PCB Design Intent

This poses a new problem, in that you also have to define in 2D space where bends and folds can be, and where you will allow components and other critical objects to cross the boundaries of rigid and flexible sections. This all needs to be indicated in a fabrication drawing, an assembly drawing, or both. A 3D image showing flexible and rigid areas will help the fabricator understand your intent more clearly. Many people do this currently with MCAD software, after having imported the STEP files of the PCB layout. The image below shows an example of this concept.

Creating some kind fo graphic from an MCAD application can have the added benefit of detecting flex-to-flex and flex-to-rigid interferences before fabrication. As is the case with Altium Designer, the online Design Rule Checker (DRC) interactively shows where interferences occur. When it comes to final product assembly, it is even more desirable to provide the assembly manager and staff with a 3D animated movie of how the rigid-flex board will fold for installation into the product enclosure or assembly. This is where having a screen capture or 3D movie straight for the CAD tool can be very helpful as part of the final assembly documentation package.



Example rigid-flex PCB in Altium Designer

Placement

You can see also from the image above, that rigid-flex designs imply that components might exist in layers other than top and bottom. This is a bit tricky in the PCB design software, because normally components must exist on top or bottom. So we need some ability to place components on inner layers.

Interestingly, Altium Designer® has always supported pad objects on any layer, so this is not impossible. There's also an implication that silkscreen could exist on flex layers as well. This is not a problem, since coverlay material can adhere well to the silkscreen ink. The trick is more to make sure there's adequate contrast for the color of ink chosen against the coverlay material. Also, resolution is affected since the ink has to traverse a small gap beyond the screen to land on the flex coverlay. Again, this is something that needs to be discussed with the fabricator to determine what's possible and economical.

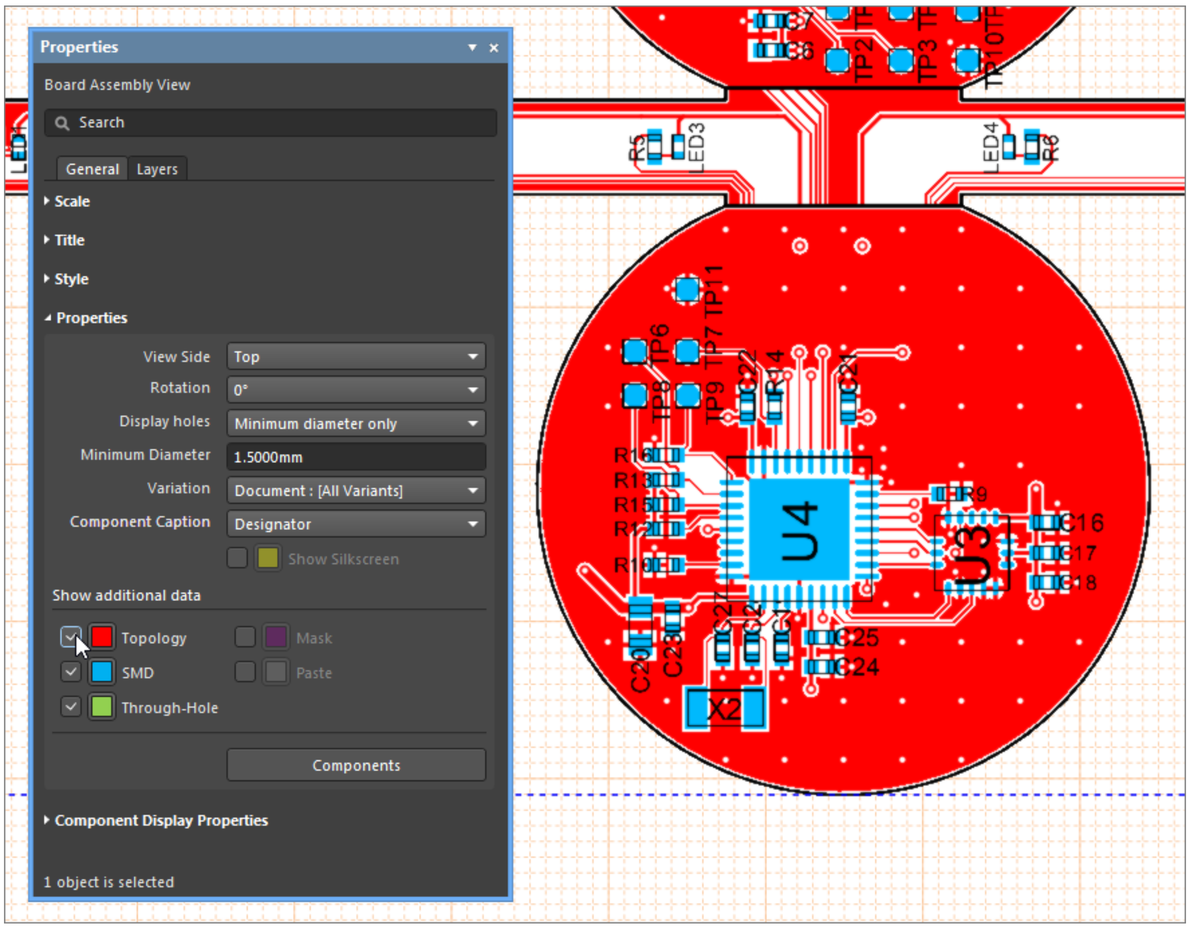
Note: If you're going to the effort of drawing the regions of the PCB which are exposed flex layers, and placing components on those regions, this also makes a reasonable method for placing embedded components into cutout regions of the board. You need to generate a set of very clear documents that show where the cutouts are and in which sections of the layer stack they apply. This is going to be limited depending on the fabricators methods - either back-drilling or multiple laminated stack-ups can be used. So communicating your intent and minimizing the number of separate cutout stack sections is important. It's best to completely avoid having intersecting cutouts from opposite sides of the board.

Conventional wisdom calls for stiffeners or rigid sections where components will be mounted on flex anyway for dynamic applications. This is to prevent dry solder joints and copper cracks due to fatigue caused by circuit movement around rigid component pins. In some circumstances, this is not necessary because the flexible region will be installed into a fixed backing alongside the rest of the board, thereafter it will be static. This situation is referred to in IPC-2223 as 'Flex to Install'.

Creating Documentation

Documentation for rigid flex PCBs, including fabrication and assembly drawings, is simple with an automated drawing utility inside your ECAD software. Some of the important points to document in your rigid-flex or flex fabrication drawing include:

- Dimensions for each region in the bare board
- A stackup drawing that shows all materials and layer arrangements
- A drill table with hole sizes and tolerances
- An impedance table if controlled impedance is required
- Component outlines for major parts with reference designators
- Complete fabrication and assembly notes


Properties
Board Assembly View
View Side: Top
Rotation: 0°
Display holes: Minimum diameter only
Minimum Diameter: 1.5000mm
Variation: Document : [All Variants]
Component Caption: Designator
☐ Show Silkscreen
Show additional data
☑ Topology ☐ Mask
☑ SMD ☐ Paste
☑ Through-Hole
1 object is selected

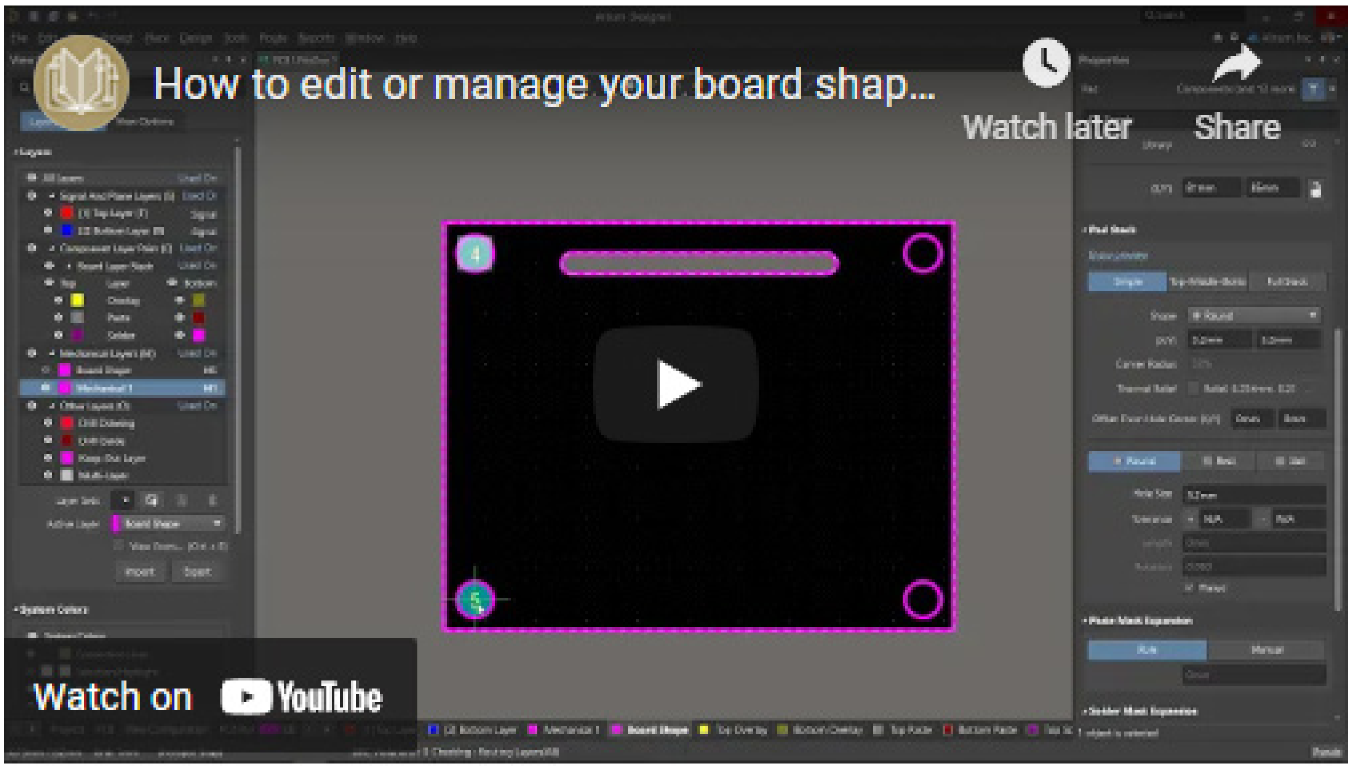
You can speed your way through the rigid-flex documentation process with the complete set of CAD features and automated drawing tools in the Draftsman package inside Altium Designer®. Once you're ready to release your design data to your manufacturer, you can easily share and collaborate on your designs through the Altium 365™ platform. Everything you need to design and produce advanced electronics can be found in one software package.

Flex and Rigid-Flex PCB Applications

There are two basic reasons for designing a flex circuit into your product: to build a compact and efficiently assembled device, or to make the circuit dynamically integrated with the mechanical function of the product. You may, of course, lean on both of these reasons for justifying the use of flex circuits. On this note, let's look at some rigid-flex PCB applications and design examples to see the issues that spring to mind when designing flex circuits.

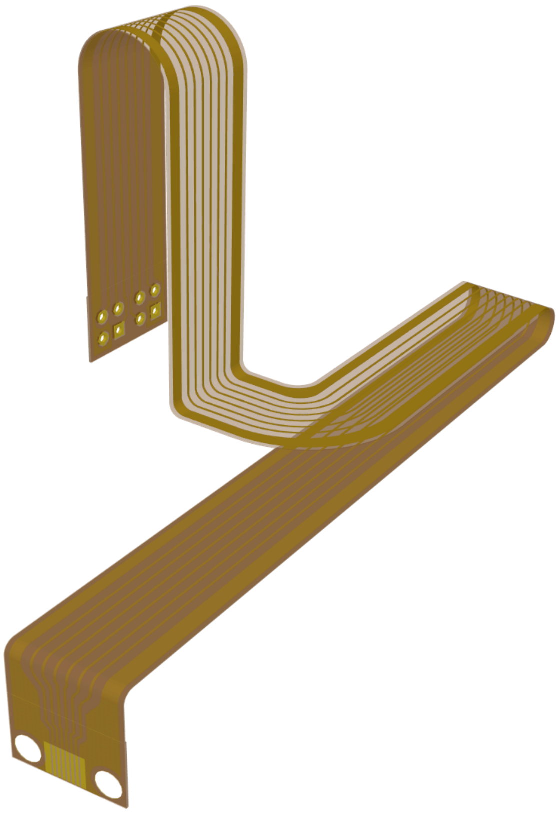
Dynamic Flex Application Examples Mechatronic Gantry

A very typical dynamic flex application, such as you might find in a 3D printer or CNC machine head, is a mechantronic gantry. In physically larger systems where electronic components need to follow the same motion as a mechanical element, this would be accomplished with separate rigid boards or modules, and these would be connected with cables. In smaller, sleeker packages, a flex ribbon makes more sense as it provides a low-profile assembly as well as the required motion.



Naturally, the example below would be laid along the X-axis gantry, and the z-axis tool head travels along it. The example below only shows two axes of motion here, and the gantry itself would move in the Y-axis.

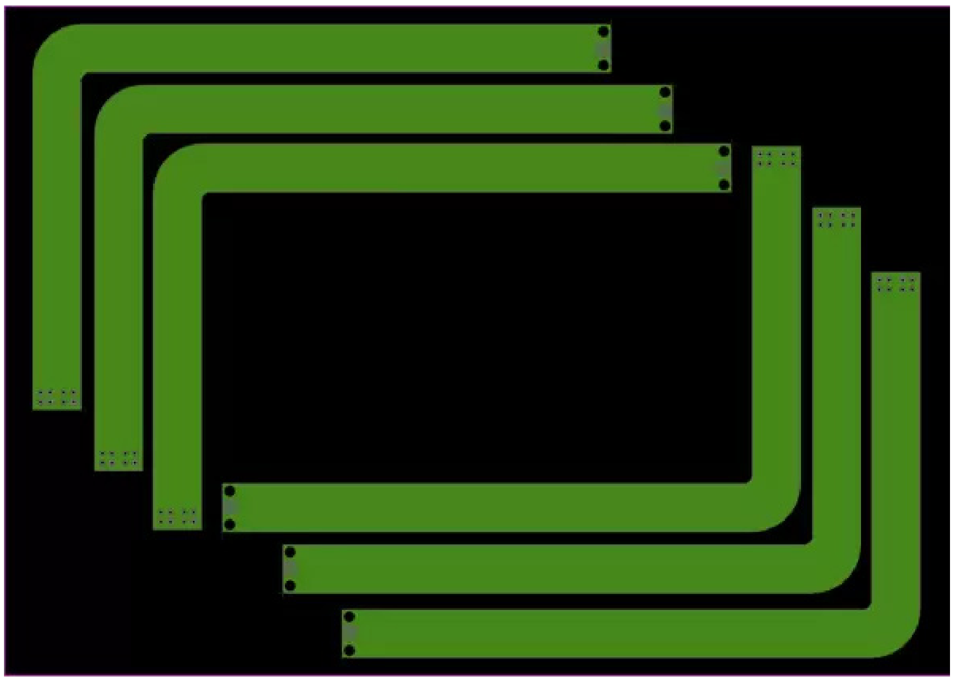
The total length of the flex ribbon is the most extreme end-distance required in addition to the corners and bends. The corner that sits behind the moving z-axis tool head would adhere to the x-axis shuttle that moves along the gantry (probably on sleeve bearings). The ends would have stiffener added to terminate the flex ribbon section. For this type of application, it's best to stick to single layer rolled-annealed copper and keep the bend radii as large as practically possible. This will help maximize the lifetime as the bend region is rolled along the length of the flex ribbon.



Example gantry flex design.

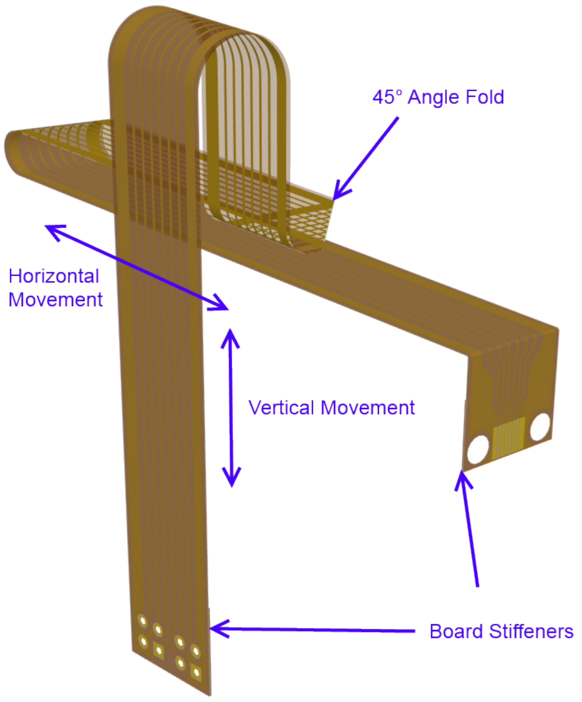
Gantry Fab Consideration: Panelizing

The example above raises a good question about fabrication and cost. Using a right-angled L-shaped circuit like this we could, for argument's sake, fit six identical flex ribbons on a fabrication panel. This results in roughly 50% waste of the panel space, and if components were to be mounted on this particular flex circuit, also add to the tooling cost and time. An example panel made from this particular flex circuit in an embedded board array is shown below.



Embedded Board Array panelization of the CNC gantry flex circuit.

The good thing about flex is if we use the right materials and plan the overall assembly right, we can also create low-radius installation folds. Placing a static flex section with a permanent crease is a good alternative to using curved flex circuits as shown in the panel above, but only in certain circumstances. The following figure shows the same gantry design but with a creased 45° fold to replace the 90° corner shown in the previous version.



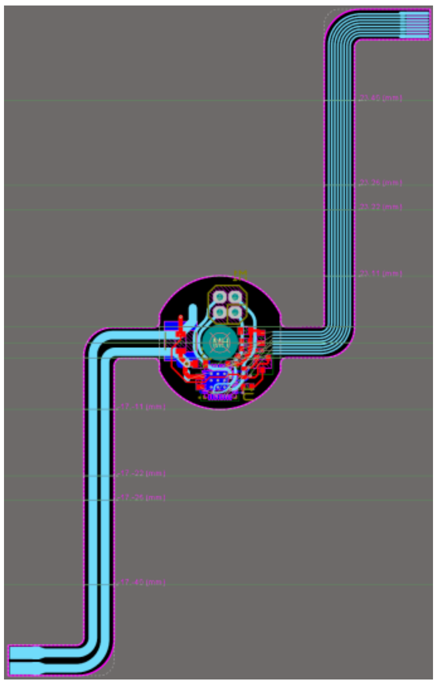
Gantry flex re-designed with static crease.

The fold becomes clearly useful once we look at the panel (shown below). To fabricate a flex circuit with this type of crease, we don't need to design a bend into the board. Instead, we can use a straight section in the flex PCB, so we can now line up an entire array of flex ribbons in a single panel. In this way, the yield increases significantly. The total cost per board will decrease due to the incrased yield per panel and then ease of tooling for pick-and-place assembly. However you may have this counteracted by having to place components on the opposite side at one end of the assembly, due to the fold.

Panel with the redesigned gantry board.

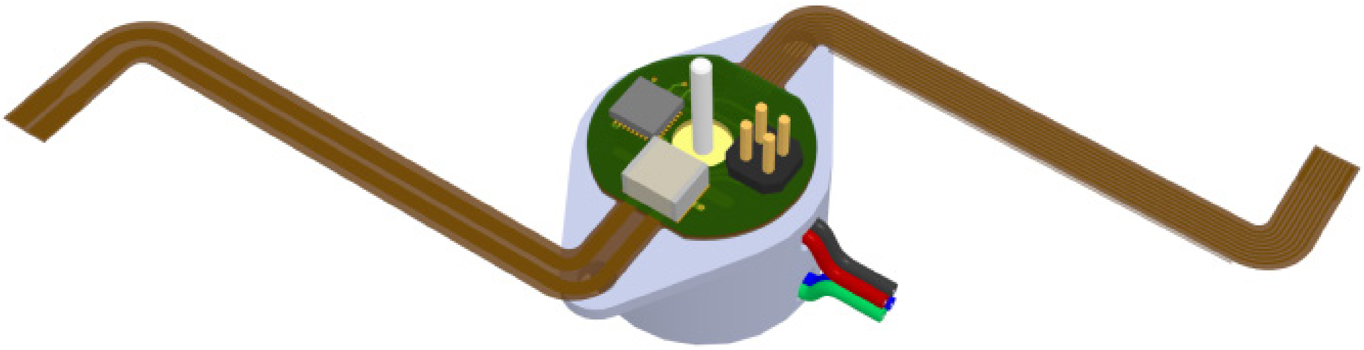
Rotational Devices

Take a look at the rigid-flex PCB layout shown below. In this layout, the flex layers are created using bends rather than a permanent crease. Notice the use of horizontal work guides in the PCB editor; this enables accurate design of the board outline based on curved circumferences of in-situ flex circuit sections. It also allows exact placement of flex circuit bending lines in the Board Planning Mode inside the PCB editor, which allows for accurate flex circuit bend simulations in 3D mode.

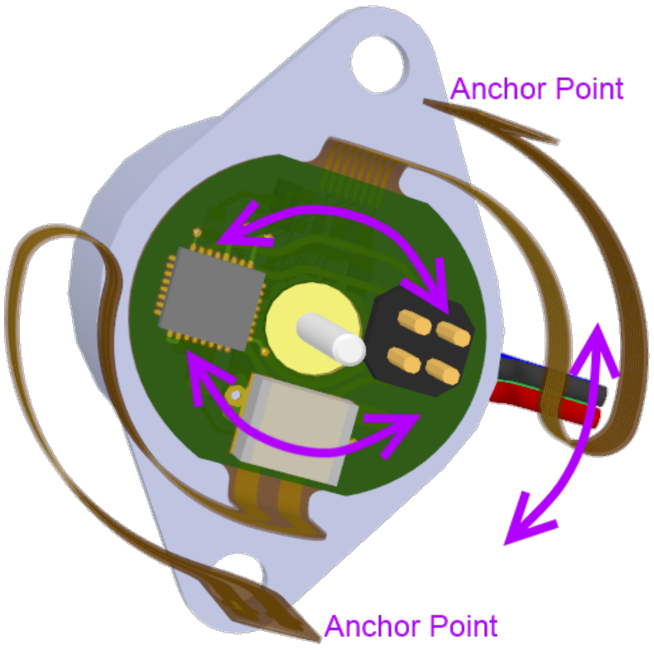


PCB layout for a rotational dynamic flex design. The flex ribbons can attached to a fixed housing or another component that will rotate with the central shaft in the assembly.

In this example, a stepper motor is to be mounted to an assembly such that the motor and it's control printed circuit board will be in motion, while the shaft will be stationary. The flex circuits are designed to be terminated at the extreme ends to a fixed base assembly and folded into a cylinder shape, doubling back to allow bi-directional movement. 3D views of this design are shown below.

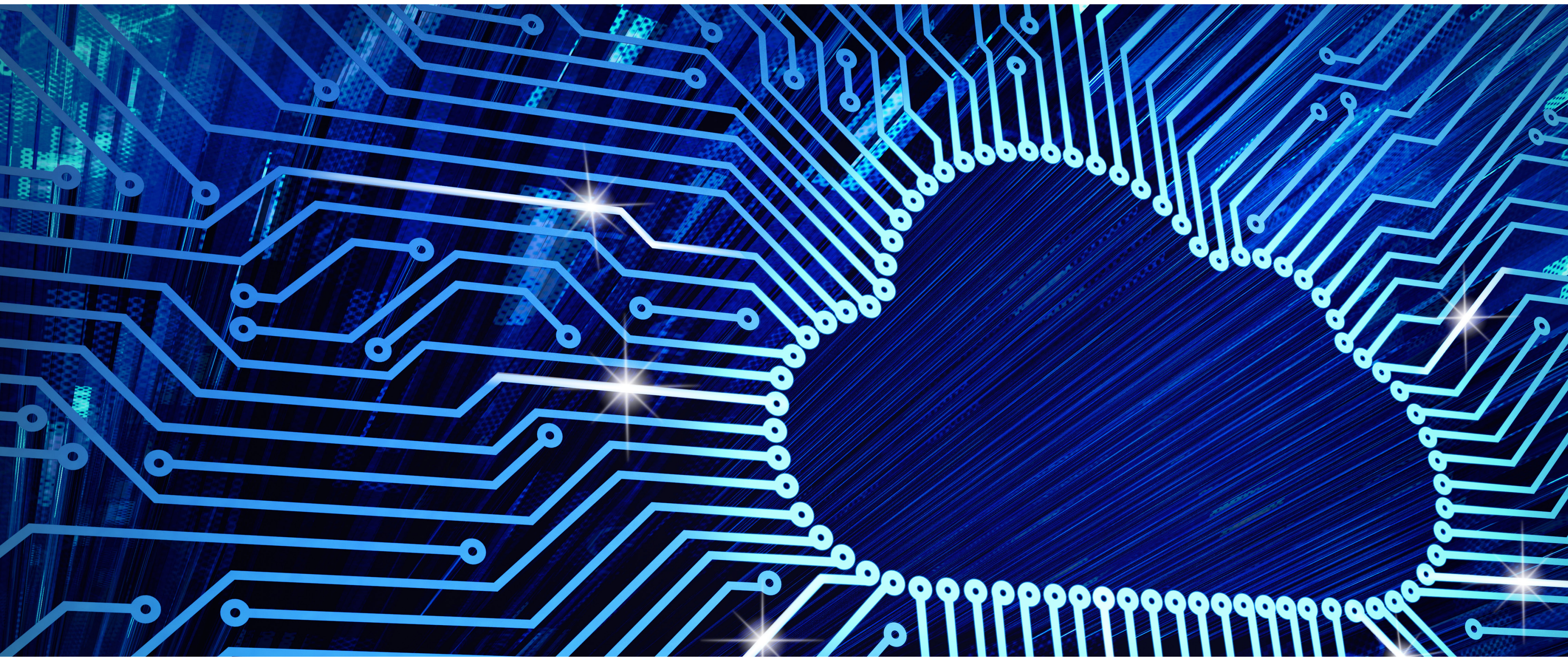


3D view of a rotating stepper motor control board. Longer "arms" would allow greater than 360° rotation of the motor and it's control board.



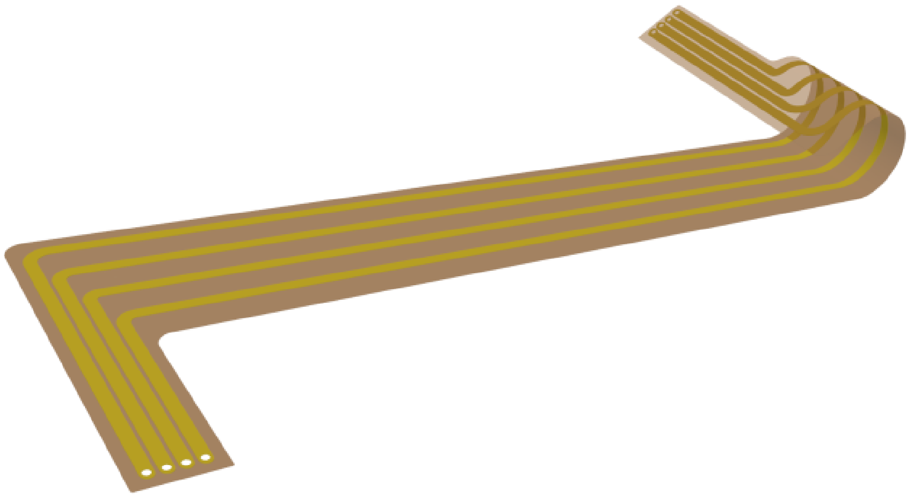
The fully folded view of the assembly, including the 3D body of the stepper motor.

We can see the directions of mtoion and the anchored flex-circuit terminators to give you an idea of how this assembly will operate. This kind of arrangement makes it relatively easy to achieve greater than 360° of rotation. This example is hypothetical and shows a stepper motor, though this kind of design would be well suited to rotary sensor applications. The terminated rigid-flex sections could also mount to some components on the enclosure, as long as the enclosure was rotating, giving a simple way to provide a connection back to the rigid control board section.



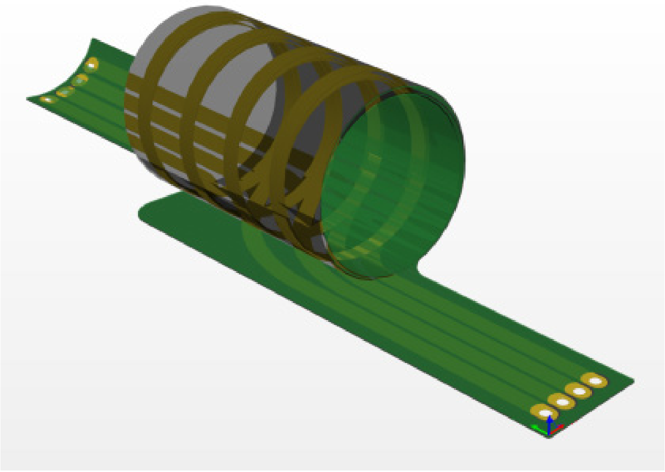
Static Flex Application Examples Planar Magnetics (Transformers and Inductors)

The use of flex and rigid-flex circuits for integrated planar magnetics is rising in popularity. Using flex-circuits for planar magnetics has some distinct advantages. The polyImide film comes in thicknesses that allow very high isolation of windings, as well as high temperature stability that makes it suitable for hot enamel potting processes. From a loss standpoint; using etched copper traces requires the traces be wider, but this can easily reduce eddy-current losses because the additional impedance from the skin effect will be reduced.



The un-rolled solenoid turns of a four winding inductor.

An interesting entry and exit scheme for a rolled air-core inductor is shown below. In this rolled flex PCB assembly, the end of each winding overlaps with the beginning of the next winding. This could be done to increase the number of turns versus simply having multiple separate windings.

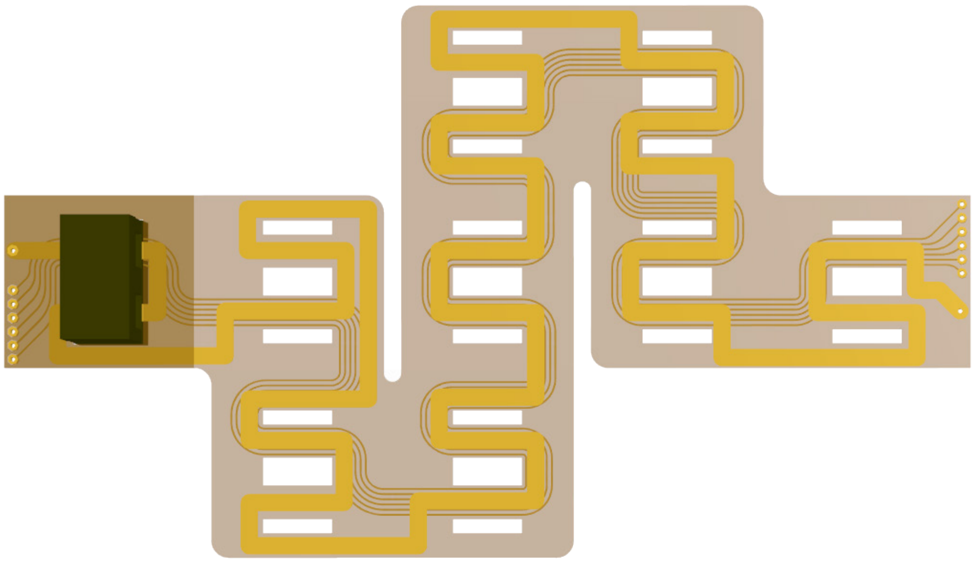


Rolled inductor windings.

18 Layers for the Price of 2

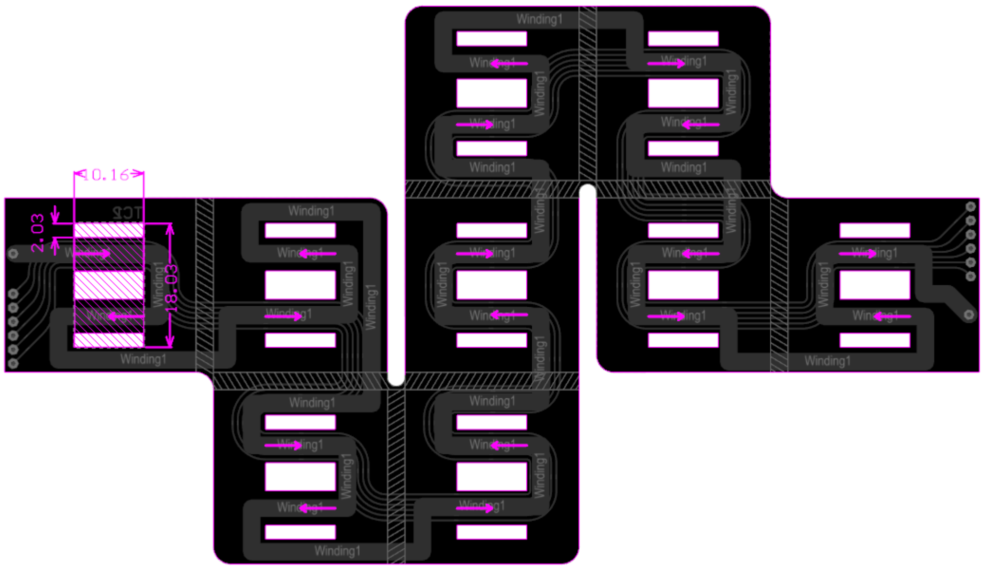
The natural extension of this concept is to include some flex layers in your converter design with the intent of folding them over each other. In the example shown below, a 2-layer flex circuit transformer design is shown, where a single E18 planar ferrite core protrudes through cutouts in the end terminator region (on the left side). This idea could be arbitrarily extended (albeit with practical limits of the thickness of the final folded board). In figure 11, the top and bottom copper layers on double-sided flex yield 18 usable layers for the transformer windings.

Around each of the core center-leg cutouts, you can make a single turn for an inductor winding. Snaking the track around a side-leg will give you half a turn, while the return path provides the other half turn in a transformer coil; together, the folded conductor sections form a set of stacked current loops that can generate and receive a magnetic field.



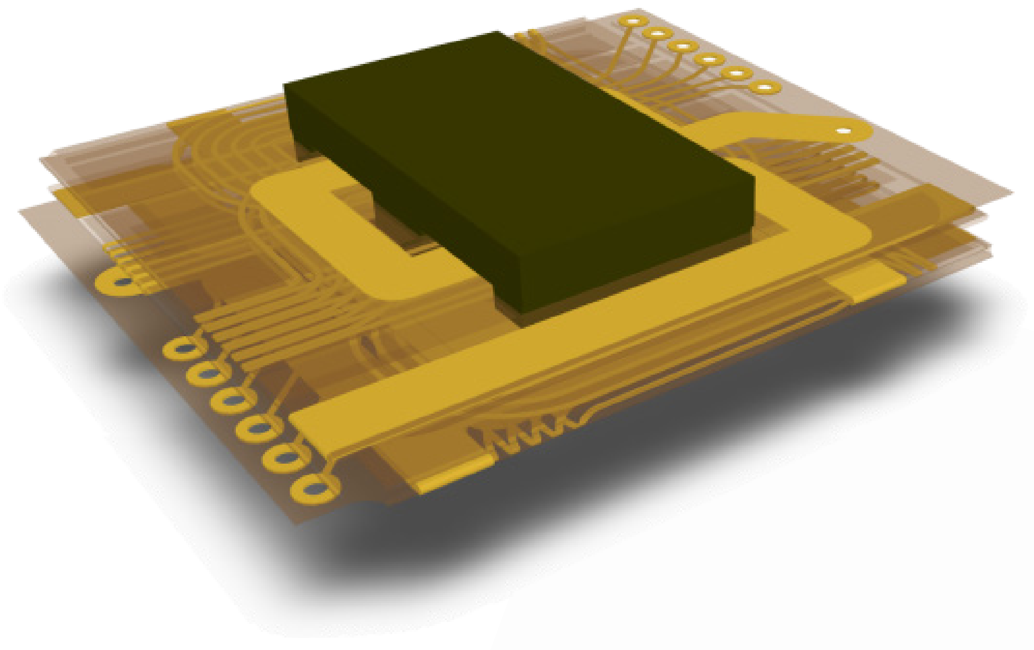
Overhead view of a flex-circuit transformer. A single heavy current winding is shown on the top layer, and six lighter current windings are routed on the bottom layer.

This could be confusing though, because you have to keep track of the proper winding directions with respect to each folded section's relation to the ferrite core geometry. Given that this whole flex circuit will fold orthogonally, I've added arrows on the Mechanical 1 layer of the design facing opposite to each adjacent winding layer to remind me which way to route the copper. This is shown below for clarity.



Mechanical 1 layer showing the board outline and winding direction arrows for guidance.

The final core-and-flex assembly is shown below. Note that this could be integrated within a rigid-flex design where the majority of the circuit is on a rigid 2-layer Printed Circuit Board, with the flex part being used to get the additional layers needed for all the core windings. Of course, there is going to be a cost trade-off between using a large flex area versus just adding heaps of layers to a rigid-only design.

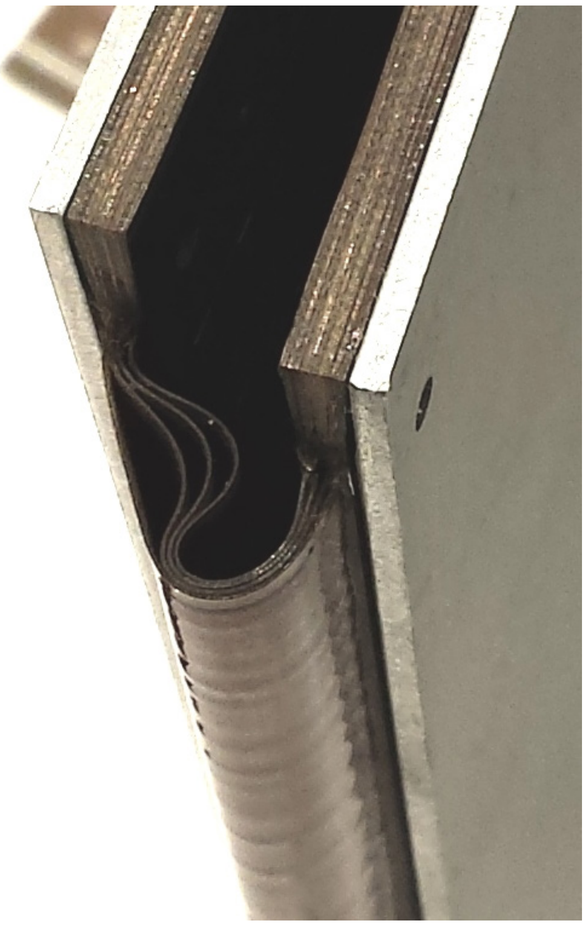


The final completely folded-up transformer, with 3D model of the Ferroxcube E18 ferrite magnetic core through the cutouts.

Multi-Layer Rigid-Flex

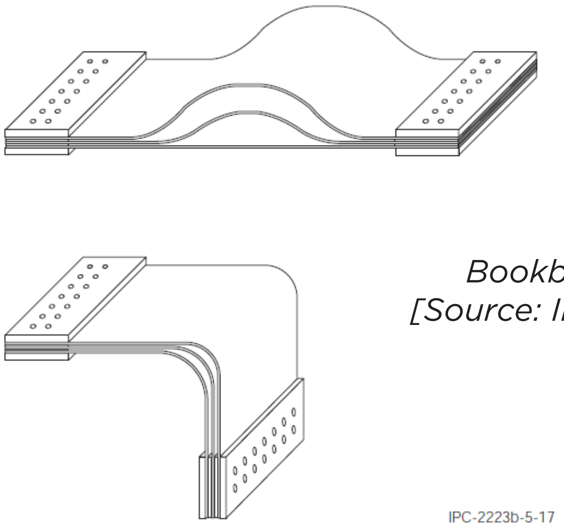
For many military, aerospace, or similar high-density designs that require compact, reliable assemblies in tight spaces, it's hard to avoid using several layers of flex circuits between rigid board areas. Even more so, this is necessary with high-speed digital designs, due to the need for shielding or plane layers between busses traversing the flex regions. The challenge here is that to maintain a good degree of flexibility. The number of flex circuit layers has to be kept to a minimum, usually two copper layers over a single polyimide substrate with polyimide coverlays.

In "normal" designs, the length of the flex-circuit sections is the same for overlapping flex regions. This means you end up with the situation shown below, where the folds can produce significant tension in the flex areas between rigid boards once placed into the final assembly.



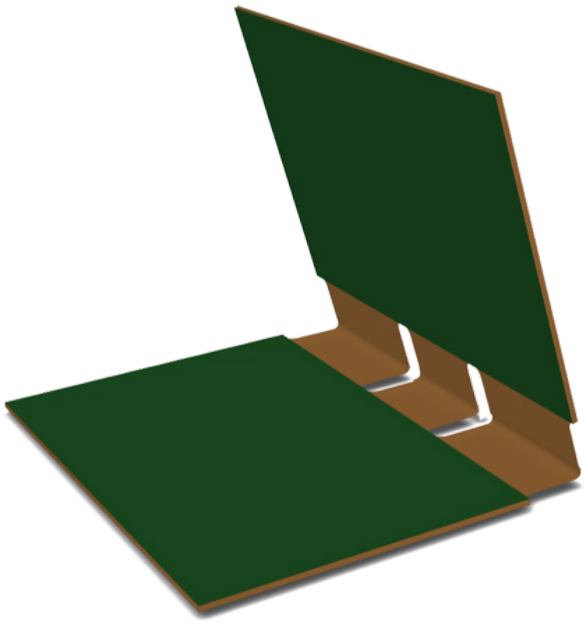
Tension in the outer flex circuit, and compression of the inner circuit, will result when multiple overlapping flex layers are designed with the same length. Notice the "squeeze out" of the adhesive bead used in this design, right where the flex enters the rigid section.

Most specialist rigid-flex board fabricators at this point would tell you to use "bookbinder" construction. Bookbinder construction is a viable method where the in-situ radii of the flex circuit bends are used to determine the correct length for each flex circuit and substrate combination in the layer stack. An example illustration of the concept is shown in the IPC-2223b excerpt below.



Bookbinder construction [Source: IPC-2223B, 2008 p26].

You can tell immediately this method is going to cost money and increase the challenge of design. Often, a better alternative is to use the same length and radius flex-circuits, but separate the different flex circuit layers so that they do not overlap each other. An example of this is shown below.

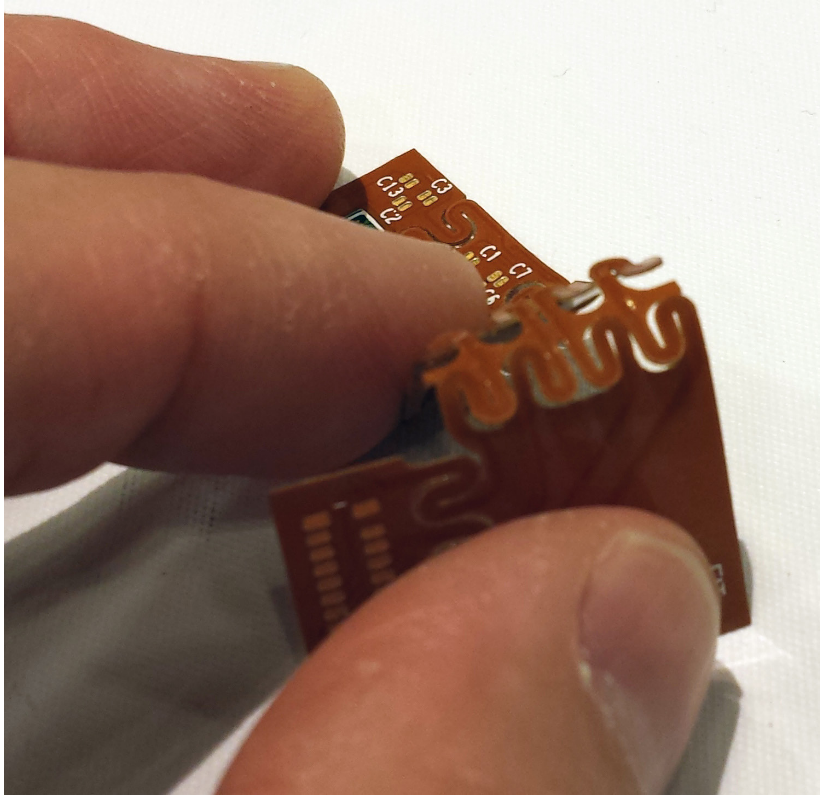


Alternative bookbinder construction. Normally, the flex sections might overlap and would require different lengths to maintain low tension/compression. In this alternative, the flex sections are placed in different regions across the edge of the rigid sections so that they no longer need to overlap.

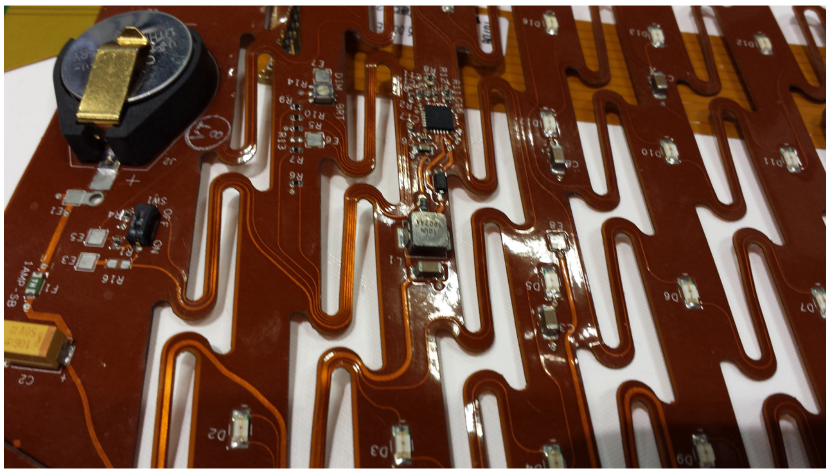
Ultra-Tight Bends Without Sacrificing Layer Count

With some creative design choices along the bend region, it's possible to get very tight bends without losing copper layers.

The tiny board shown below uses an "S" shaped ribbon to define bends and decrease the minimum bend radius along the edge of the stiffened regions. It's not visible in this photo, but there are components mounted on sections that had thin stiffener adhered on the back side of the board.



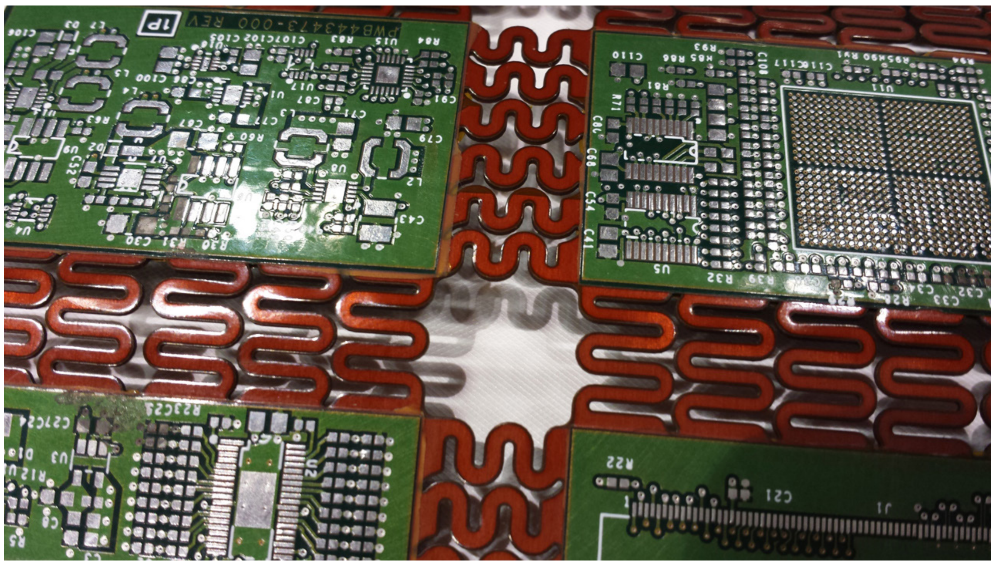
Getting essentially 180° bend radius with multiple copper layers.



X-Y S-bend flex array.

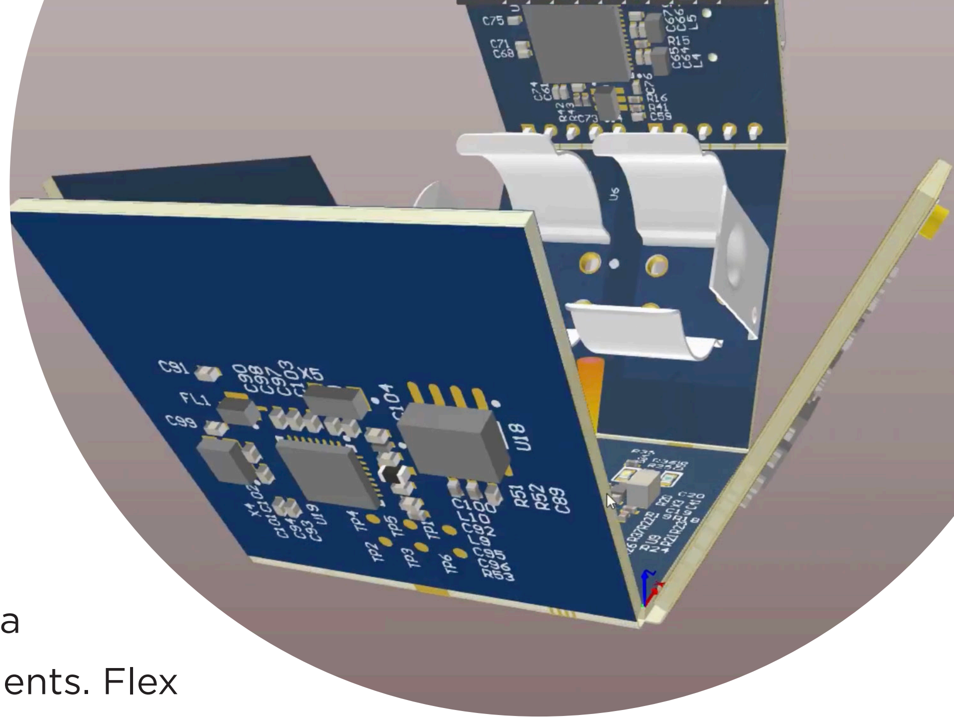
This concept can be extended in multiple directions. The PCB design shown below is an ultra-flexible PCB display board. You can see the many LEDs in a matrix on the wider, stiffer sections. The whole assembly is rigid in those sections only because of the sheer number of copper and PI film layers laminated together. Again, using S-bends between those LED matrix regions allows this assembly to more easily bend into a curved housing.

Take this concept even further, and you have the very compact design shown below. The flex-circuit sections in this example contain 8 layers. Such flex circuits would normally not be flexible if placed as direct ribbons between rigid sections. However, using the myriad of S-bends (notice the top flexible material layers are all solid copper for shielding!) allows this to bend enough to go into the final mechanical housing, even with hundreds of high speed memory and display connections.



8 Layers of flex, plus 4 additional rigid PCB layers. Notice the top layer of flex is entirely copper pour for shielding. Notice also the protective adhesive around the edges of the rigid-to-flex interfaces.

Designing for any rigid-flex PCB application is easy with the complete set of PCB design and manufacturing features in Altium Designer®. Once you're ready to release your design data to your manufacturer, you can easily share and collaborate on your designs through the Altium 365™ platform. Everything you need to design and produce advanced electronics can be found in one software package.

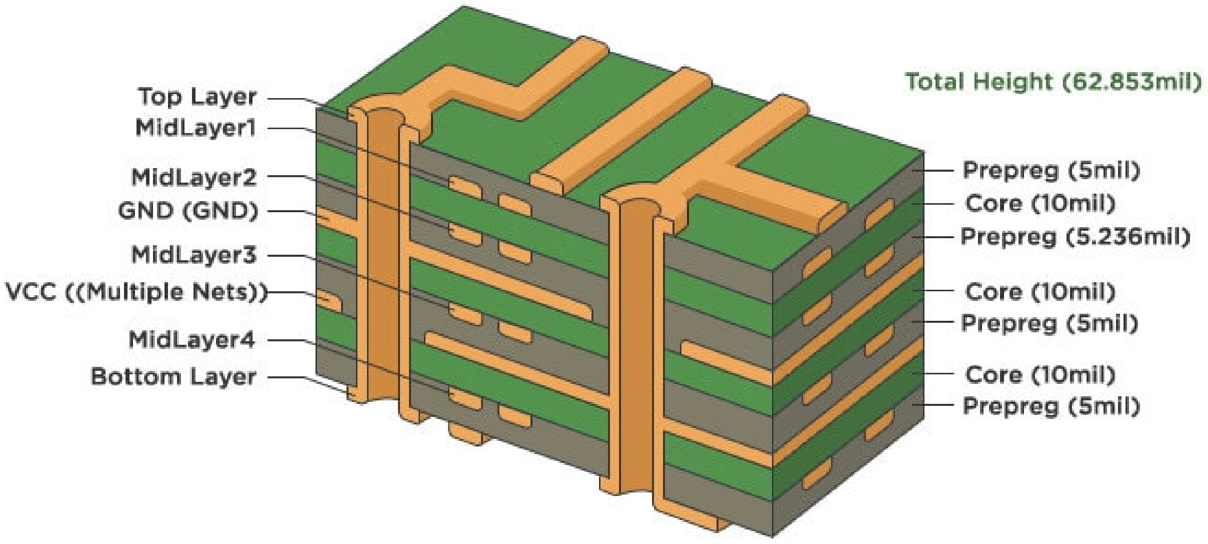


Support for Rigid-Flex in Altium Designer

Altium Designer's world-class PCB design features help users quickly get started with new rigid-flex designs and prepare them for manufacturing. Rigid-flex in Altium Designer starts with designing a manufacturable PCB layer stack complete with via transitions and any calculated impedance requirements. Flex sections also need to be placed in the layer stack before moving into the PCB layout. Once inside the PCB editor, bending lines can be clearly defined in the PCB layout, and these can be visualized in Altium Designer's 3D PCB design tools. Keep reading to see how Altium Designer supports your flex and rigid-flex designs.

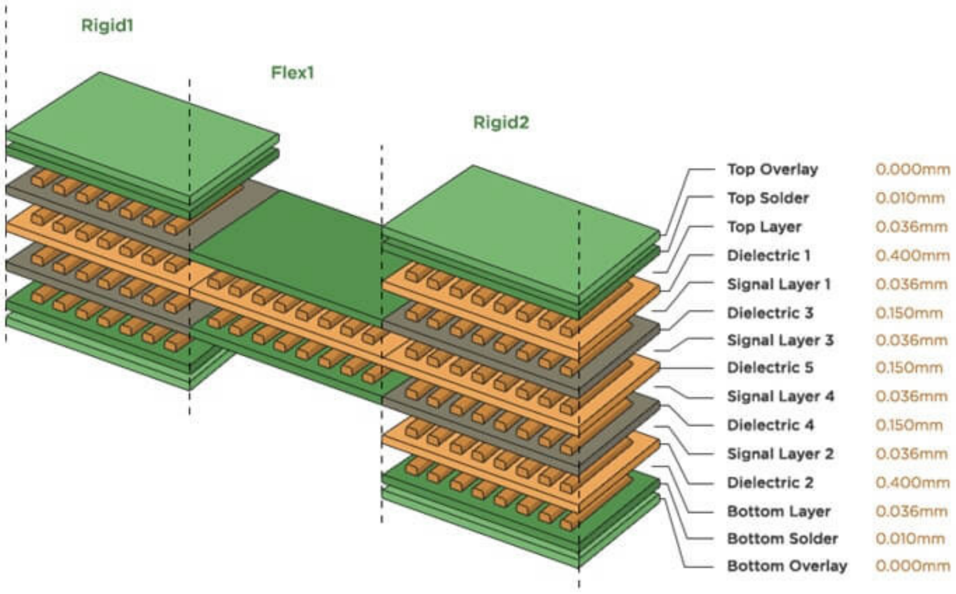
Support for Rigid-Flex in Altium Designer

Altium Designer's PCB editor is a layered design environment, supporting up to 32 signal layers and 16 plane layers. These copper layers are separated by insulation layers. In a traditional rigid PCB these insulating layers are typically fabricated using FR4 core and pre-preg, although there is a range of materials available, each with properties that suit different applications. For a traditional rigid PCB these copper and insulating layers exist across the entire PCB, so a single layer stack can be defined for the entire board area.

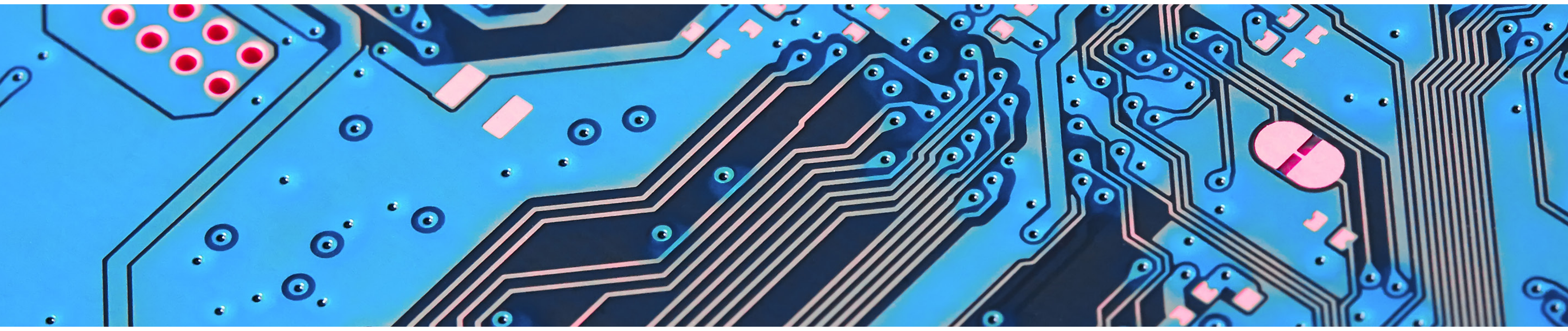


The layer stack for an eight layer rigid circuit, as it was configured in earlier versions of Altium Designer.

A rigid-flex design does not have a consistent set of layers across the entire circuit design, the rigid section of the board will have a different set of layers from the flexible section. And if the rigid-flex design has a number of rigid sections joined by a number of flex sections, then there may be a different set of layers used in each of these sections. A PCB editor with a single layer stack cannot support this design requirement. To support this, Altium Designer's layer stack management system has been enhanced to support the definition of multiple stacks, as shown below.



The Layer Stack Manager supports the definition of any number of layer stacks.



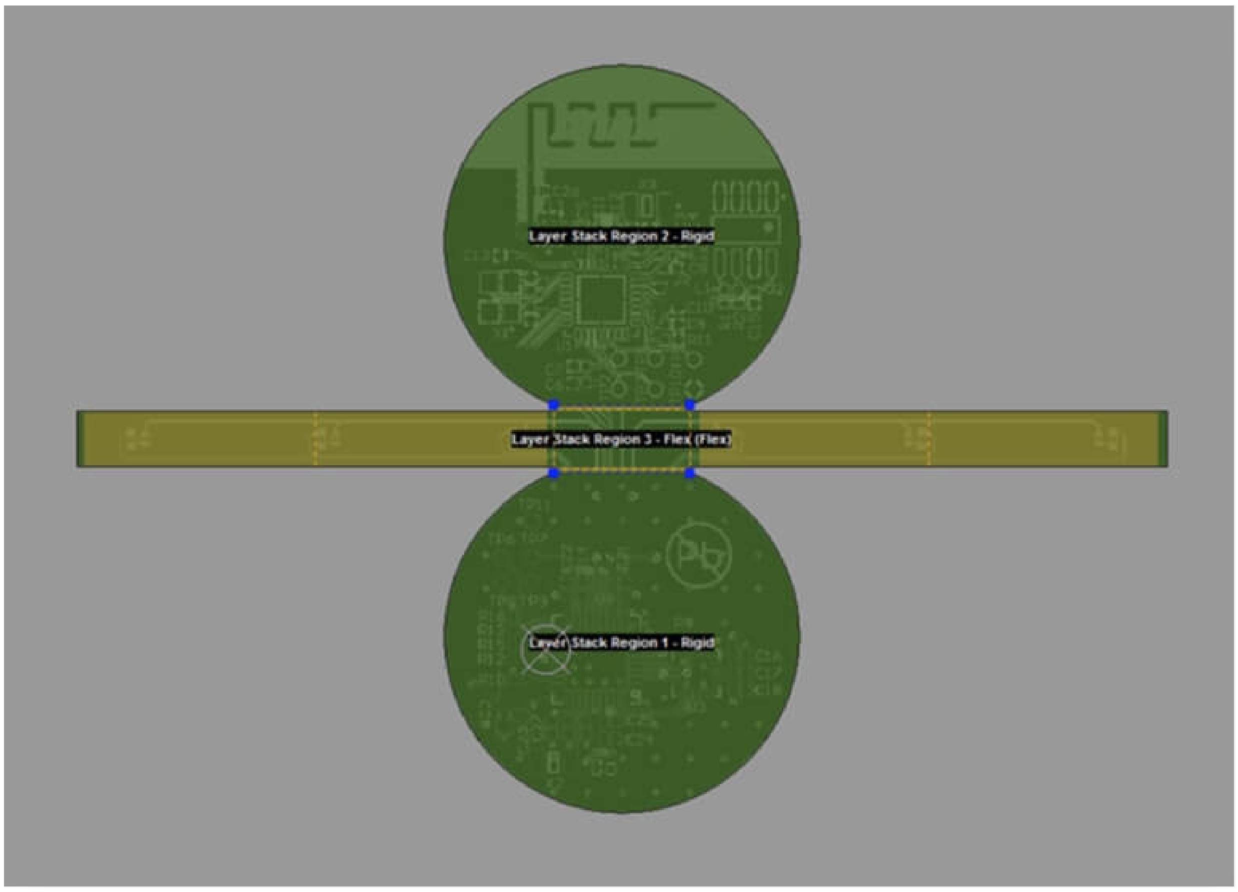
Multiple Layer Stacks

To support the need to define a different set of layers in different areas of the board design, Altium Designer supports the concept of multiple layer stacks. This is achieved by having an overall master layer stack that defines the total set of layers available to the board designer in this design. From this master layer stack, any number of sub-stacks can be defined, using any of the layers available in the master stack. Each sub-stack is defined and named, ready for use in the rigid-flex design.

The Board Shape

The layer stack defines the board design space in the vertical direction, or Z plane. In Altium Designer, the board space is defined in the X and Y planes by the Board Shape. The board shape is a polygonal region of any shape, with straight or curved edges that lie at any angle, which can also include cutouts (internal holes) of any shape. The board shape is a fundamental concept in Altium Designer's PCB editor, it defines the area available for design - where the components and routing can be placed - and all of Altium Designer's intelligent analysis engines, such as the design rule checker or the autorouter, operate with the boundaries of the board shape.

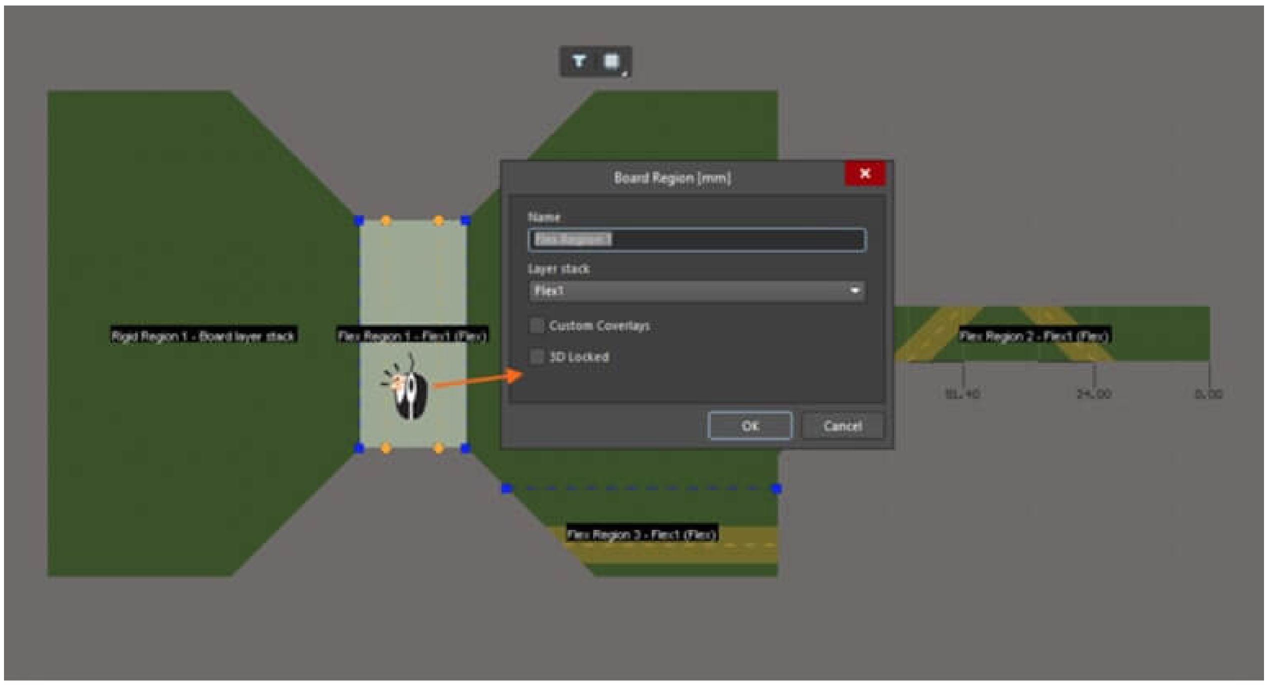
Note that there is a single, overall board shape for the entire circuit design, including rigid-flex. Within this board shape, any number of board regions can be defined by placing Split Lines to divide the board into separate regions. The image below shows a board shape that has been divided into 3 regions, by the placement of the 2 horizontal blue Split Lines. Use the links above to learn more about splitting a board into multiple regions.



An unusual rigid-flex board shape with two rigid sections and a wrap-around flex ribbon.Note the horizontal dashed blue Split lines; these divide the flex ribbon into 3 separate regions.

Assigning a Layer Stack to a Board Region

In a traditional rigid PCB, the copper and insulating layers exist across the entire PCB, so a single layer stack can be defined for the entire board shape. For a rigid-flex design made up of a number of rigid and flex regions, where each region needs a different layer stack, an alternative approach is needed. In Altium Designer, this is achieved by supporting the ability to assign a layer sub-stack to a specific region of the board shape. To do this, double click on the region to open the Board Region dialog, then select the required Layer stack in the dropdown, as shown in the image below.



Name: Flex Region 1

Layer stack: Flex1

☐ Custom Coverlays

☐ 3D Locked

Double-click on a region to open the Board Region dialog and assign the required layer stack.

If a region has a layer stack assigned, and that stack has the Flex option enabled, then Bending Lines can be placed across that region. Each Bending Line has a: Radius, Bend Angle and an Affected Area Width property, allowing them to be displayed in their folded state, as they would be in a real-world situation.

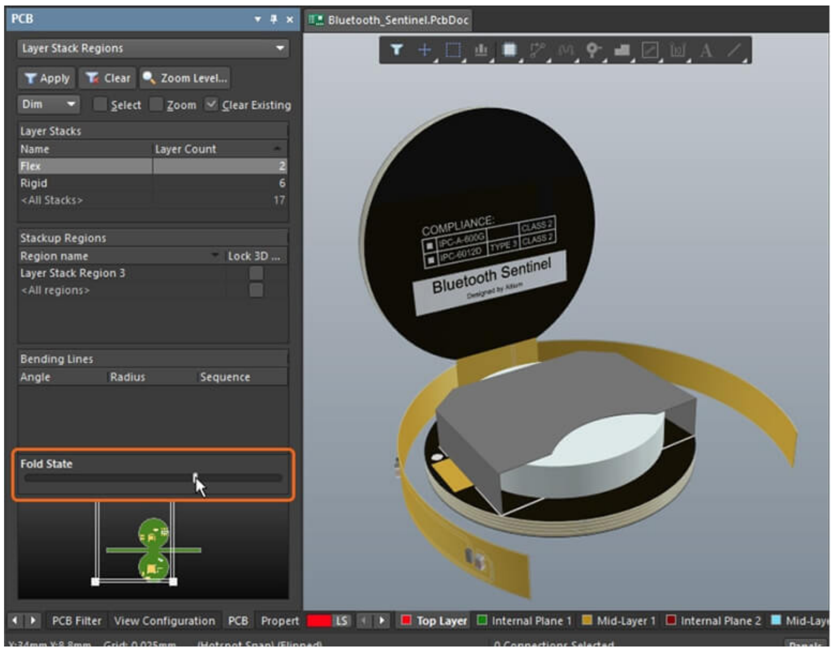
Bending Lines have been defined, allowing this rigid-flex board to be displayed in its folded state.

Displaying and Folding a Rigid-Flex Design in 3D

Altium Designer includes a powerful 3D rendering engine for 3D PCB layout, MCAD design tasks, and ECAD/MCAD collaboration. The engine enables presentation of a highly realistic 3 dimensional representation of the loaded circuit board. This engine also supports rigid-flex circuits, and when it is used in combination with the Fold State slider allows the designer to examine their rigid-flex design in the flat state, the fully folded state, and anywhere in between.

To switch to the 3D display mode, press the 3 shortcut key (press 2 to return to 2D, or 1 to return to Board Planning Mode). The board will be displayed in 3D, and if the component footprints include 3D Body Objects that define the mounted component, then these will also be displayed. In the image below you can see that the board includes a battery and a battery clip.

To apply all of the Bending Lines, slide the Fold State slider - it's in the PCB panel when set to Layer Stack Regions mode - as highlighted in the image below. Note that the bends are applied in the order defined by their sequence number. Bending Lines can share the same sequence number, it simply means that those bends will be folded at the same time when the Fold State slider is used. The board can also be folded/unfolded by running the View » Fold/Unfold command (press the 5 shortcut).



Name	Layer Count
Flex	2
Rigid	6
<All Stacks>	17

Region name	Lock 3D ...
Layer Stack Region 3	☐
<All regions>	☐

Use the Fold State slider (or the Fold/Unfold command) to apply all Bending Lines in the order defined by their sequence value (Fold Index).

You can only fold a board if one of the rigid sections has the 3D Locked checkbox enabled in the Board Region dialog. Altium Designer needs this to know which section of the board must remain fixed during the folding process.

When you're ready to create your next flex or rigid-flex design, use Altium Designer® and its complete set of PCB design and manufacturing features. Once you're ready to release your design data to your manufacturer, you can easily share and collaborate on your designs through the Altium 365™ platform. Everything you need to design and produce advanced electronics can be found in one software package.

ABOUT ALTIUM

Altium LLC (ASX: ALU) is a multinational software corporation headquartered in San Diego, California, that focuses on electronics design systems for 3D PCB design and embedded system development. Altium products are found everywhere from world leading electronic design teams to the grassroots electronic design community.

With a unique range of technologies Altium helps organisations and design communities to innovate, collaborate and create connected products while remaining on-time and on-budget. Products provided are Altium Designer®, Altium Vault®, CircuitStudio®, PCBWorks®, CircuitMaker®, Octopart®, Ciiva® and the TASKING® range of embedded software compilers.

Founded in 1985, Altium has offices worldwide, with US locations in San Diego, Boston and New York City, European locations in Karlsruhe, Amersfoort, Kiev and Zug and Asia-Pacific locations in Shanghai, Tokyo and Sydney. For more information, visit www.altium.com. You can also follow and engage with Altium via Facebook, Twitter and YouTube.