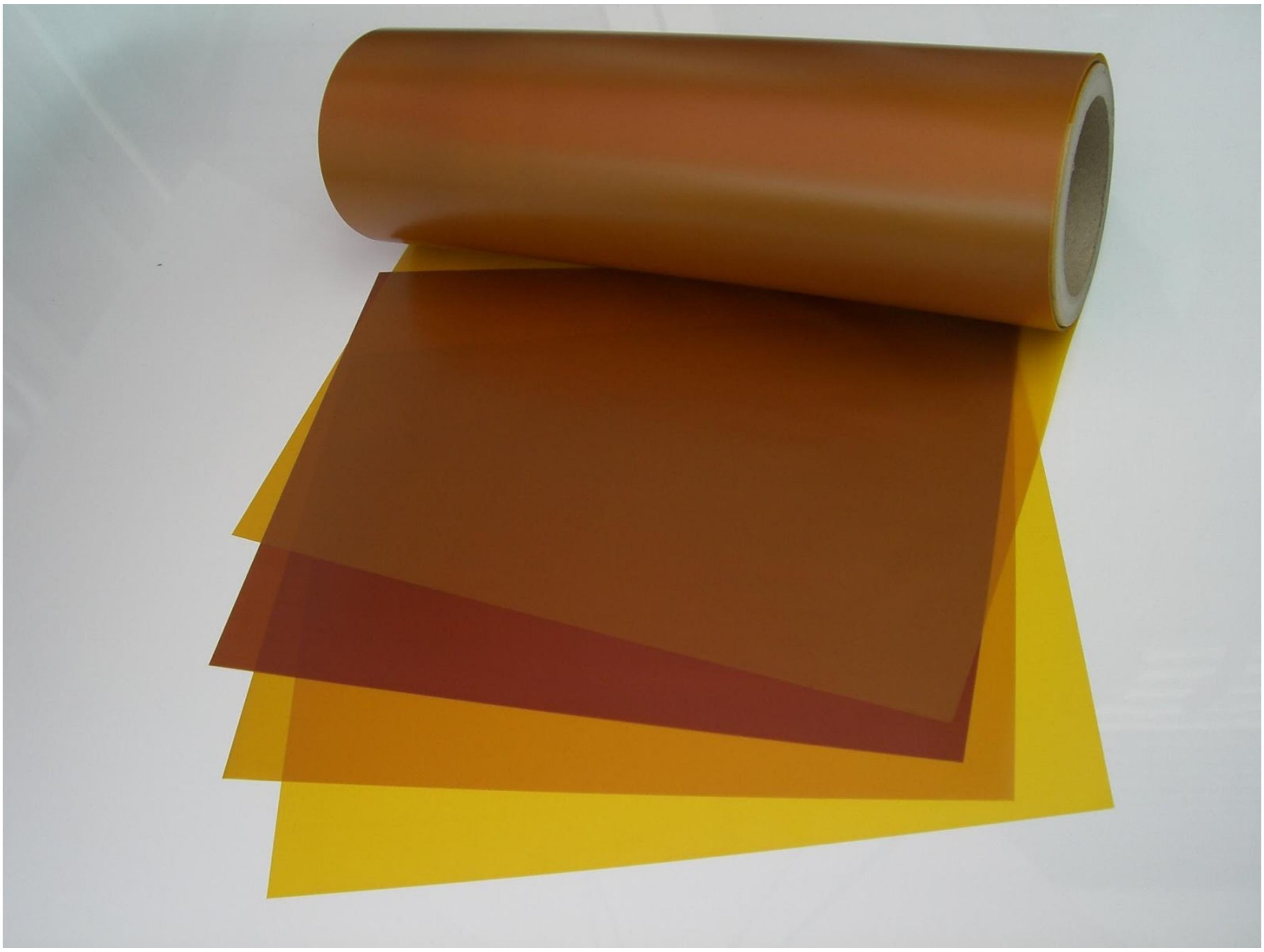
Rigid Flex In Depth

RIGID FLEX IN DEPTH

More and more designers are facing the need to reduce size and cost of the products they design, while increasing density and simplifying assembly. Rigid-Flex circuits (those which incorporate flexible portions between separate rigid sections) are becoming a more common solution. This eBook will cover the materials, fabrication, and design methods for using rigid-flex technology. Rigid-Flex can have many benefits, and there are many designers today considering it today who previously did not have to; the market is changing rapidly. It seems that more designers are facing higher pressures to build ever more densely populated electronics, and with that also comes pressure to reduce costs and time in manufacturing. Well, this is really nothing new of course. It's just that the scope of engineers and designers having to respond to these pressures is continuously broadening.

But there are aspects of rigid-flex which could be pot-holes in the road for newcomers to the technology. So it's wise to first understand how flex circuits and rigid-flex boards are actually made. From there we can look at the design issues and find a clear path forward. For now, let us consider what basic materials go into these boards.

FLEX-CIRCUIT MATERIALS



SUBSTRATE AND COVERLAY FILMS

Start by thinking of a normal rigid PCB - the base material is typically fibreglass and 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. This is not flexible enough for many applications though for simple assemblies where there's not going to be constant movement it is suitable.

For the majority of applications, more flexible plastic than the usual network epoxy resin is needed. The most common choice is polyimide, because its very flexible, very tough (you can't tear or noticeably stretch it by hand, making it tolerant in product assembly), and also incredibly heat resistant. This makes it highly tolerant of multiple 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 of high temps and less dimensionally sound that Polyimide (PI) films. 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. I seem to remember that the display in this product (a clock radio) in question 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 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 ▯ 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 el-cheapo 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.

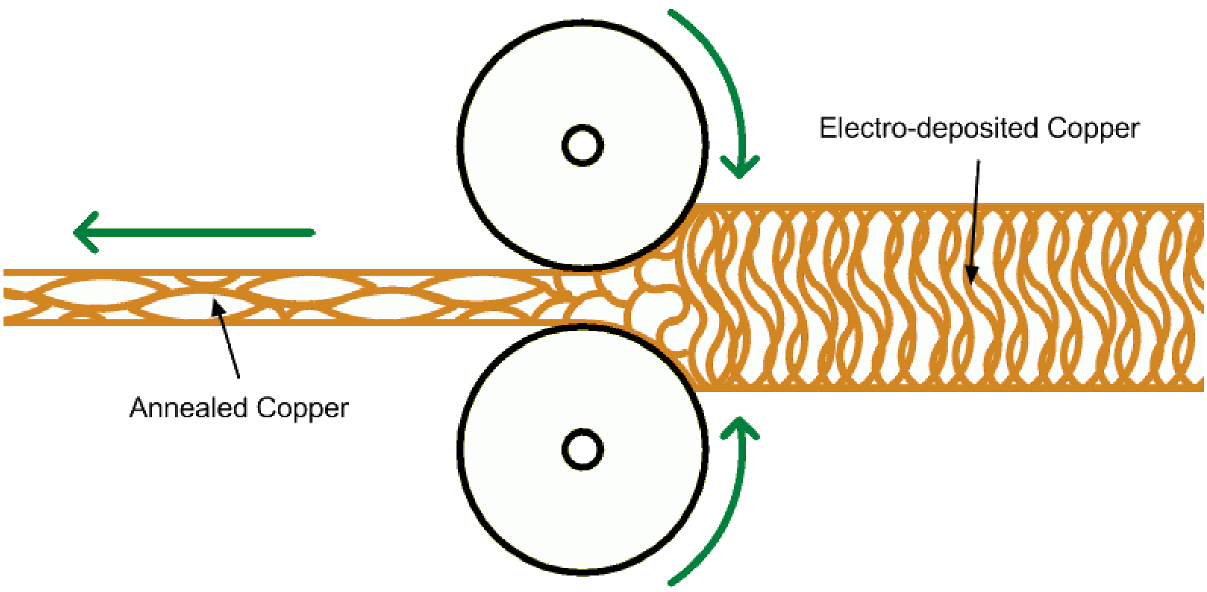


Figure 2: Exaggerated illustration of the annealing process, obviously not to scale. The copper foil passes between high-pressure rollers which elongate the grain structure in a planar orientation, making the copper much more flexible and springy in the z-deflection.

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.

Examples of such an application would be gantry connections to a CNC router head, or laser pickup for a Blu-Ray drive (as shown below).

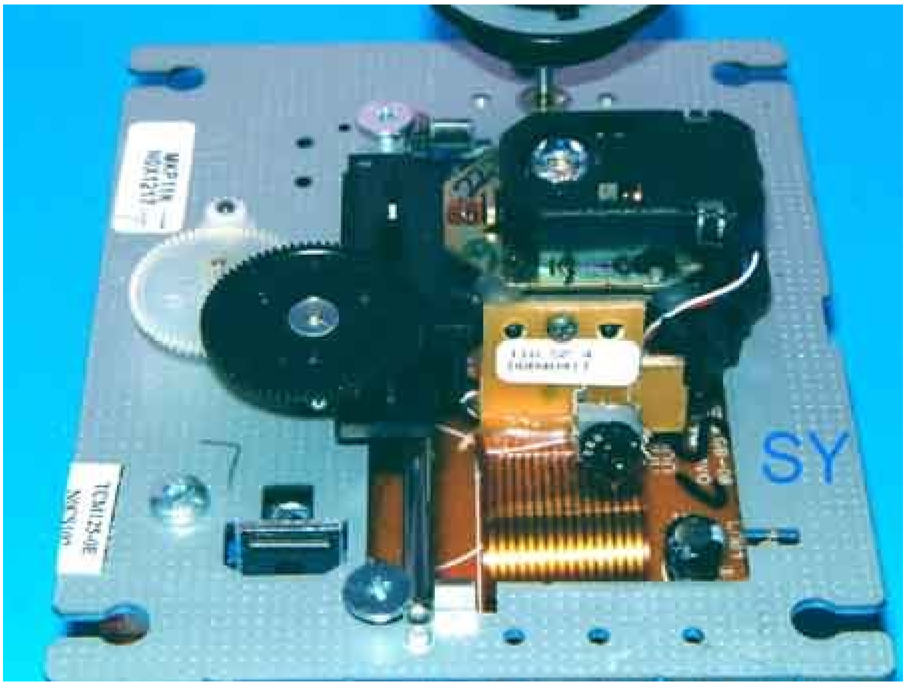


Figure 3: Flex-circuit used to link the laser pickup to the main board assembly in a Blu-Ray mechanism. Notice that the PCB on the laser head has the flexible portion bent at right angles, and an adhesive bead has been added for strengthening the flex circuit at the join.

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 as DuPont 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. An example of this is shown in Figure 3 above.

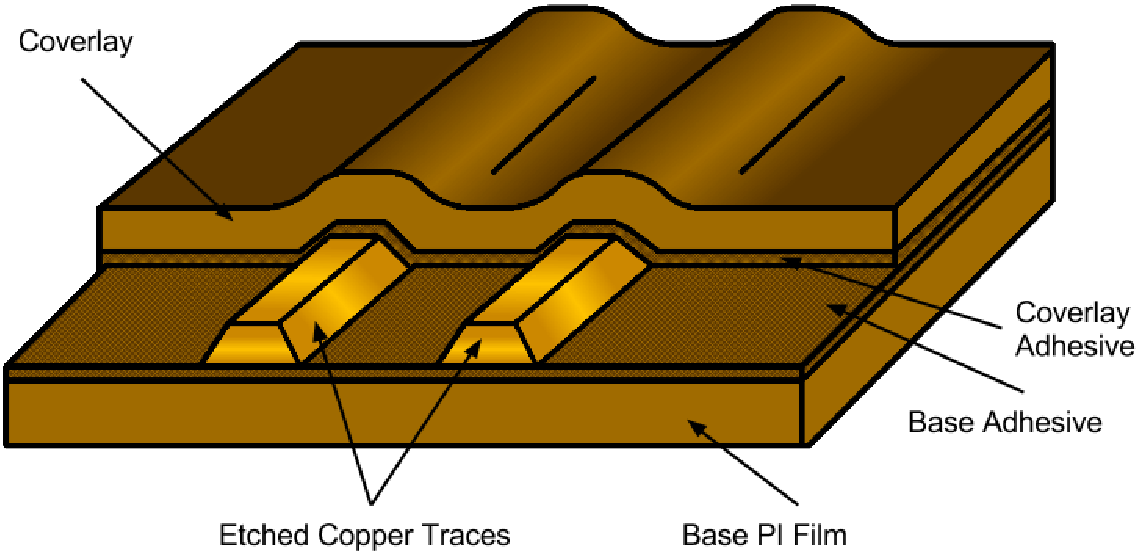


Figure 4: Typical single-layer Flex Circuit stack-up.

FLEX MATERIAL SUMMARY

It's important to be aware of the materials used in flexible and rigid-flex circuits. Even though you may generally allow the fabricator freedom to select the materials based on your application, ignorance will not protect you from field-failures of the final product. A really good resource which contains far more detail than my brief introduction here is Coombs, C. F. (Editor, 2008) The Printed Circuits Handbook, 6th Ed. 2008 McGraw Hill, pp 61.3 0 - 61.24.

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 & vibe - 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.

FABRICATION STEPS IN RIGID FLEX CIRCUITS

How are flex, and rigid flex PCBs manufactured? In this article I discuss how the materials are combined, laminated and cut out to create the final product, which will lead to better understanding of the design considerations.

FLEX AND RIGID-FLEX FABRICATION PROCESSES

I began our discussion of Rigid-Flex PCBs by discussing the materials used in fabricating these bendy little beasties. Now, I want to discuss how these materials are combined, laminated and cut out to create the final product. Next, we'll consider all these steps and address the design challenges associated with them.

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. As mentioned previously, the fabricator may begin with pre-laminated flex 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 GIF animation below shows the Flex-Circuit creation steps for a typical double-sided flex circuit.

Figure 1: GIF animation showing flex-circuit build-up process.

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.

2. COPPER FOIL ADDED

Either by RA/ED copper foil lamination to the adhesive (the more mainstream approach) or chemical plating onto the seed layer.

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, 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 (ultra-violet) or YAG (Infra-Red) lasers for higher accuracy (microvias), CO2 lasers for medium holes (4+ mils). Large holes and cutouts are punched, but this is a separate process step.

4. THROUGH-HOLE PLATING

Once the holes are made, copper is deposited and chemically plated in the same way as rigid board cores.

5. ETCH-RESIST PRINTING

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.

6. ETCHING AND STRIPPING

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

7. COVERLAY

Top and bottom areas of the flex circuit are protected by coverlay which is 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 by UV exposure.

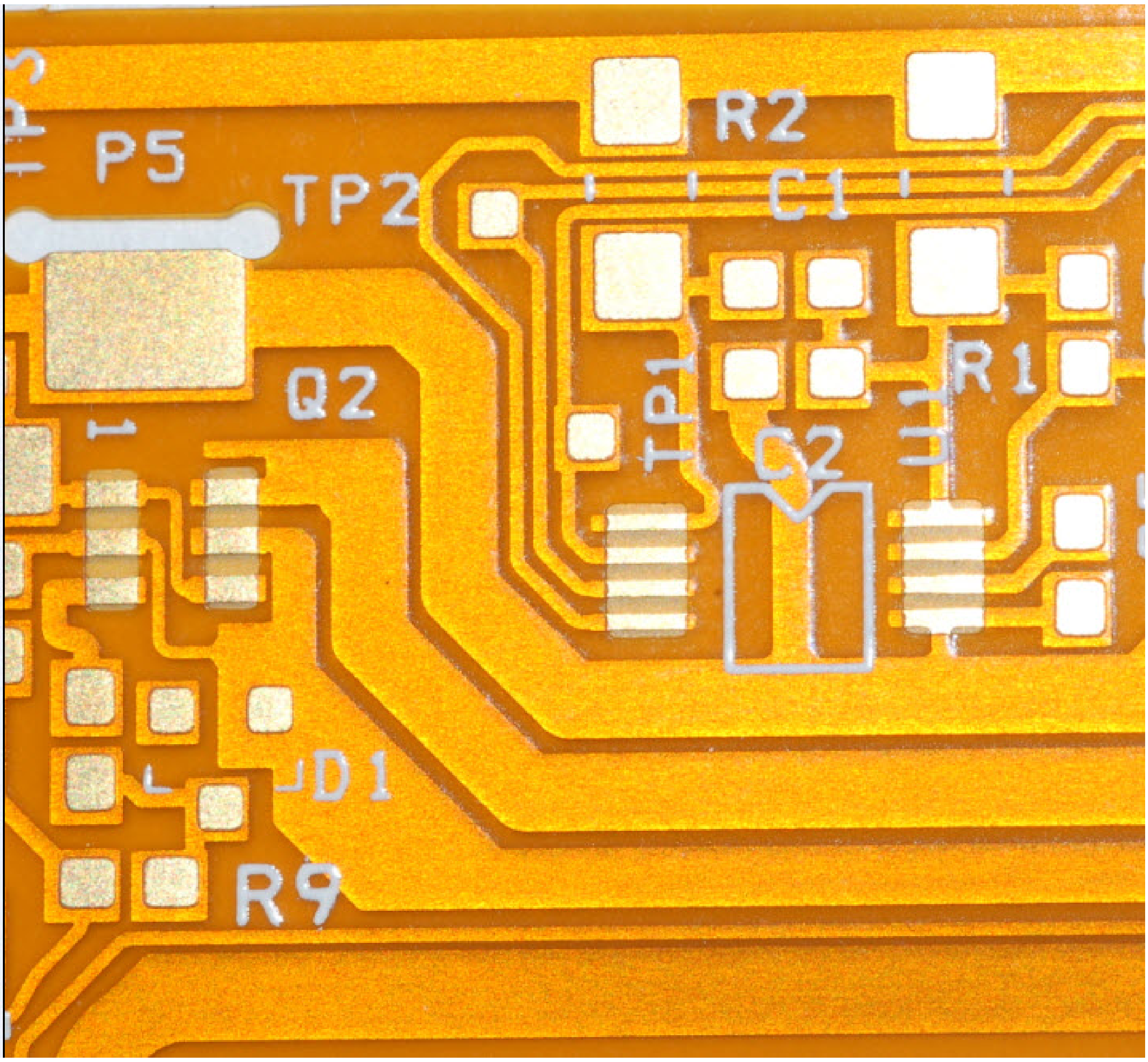


Figure 2: An example of flex-circuit with Coverlay - notice that the openings in the coverlay are generally smaller than the component pads.

An important note to make about coverlay is that it is typically only placed on parts of the flex circuit that are ultimately to be exposed. For rigid-flex boards, this means the coverlay is not placed where rigid sections will be, apart from a small overlap - usually about ½ a mm. Coverlay can be included throughout the rigid section, though it adversely affects adhesion and z-axis stability of the rigid board to do so. This kind of selective coverlay is referred to as "bikini coverlay" by the board fabricators that use this process because it just covers the bare essentials. Also, cutouts for component or connection pads in the coverlay leave at least two sides of the pad land to anchor under it. We'll revisit this in the next section.

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 using 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 looong 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. For even smaller prototype runs, X/Y cutters (similar to those used in vinyl sign making) could possibly be used.

LAMINATION AND ROUTING

If the flex circuit is to form a part of a rigid/flex combined stackup (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 although, a less flexible layer could be made with PI and glass depending on the target application. Because this is being laminated in with rigid sections, it ultimately has to be framed in a panel that mates with the rigid board panel sections as well.

LAMINATED STACKUPS

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. 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 in figures 3 & 4.

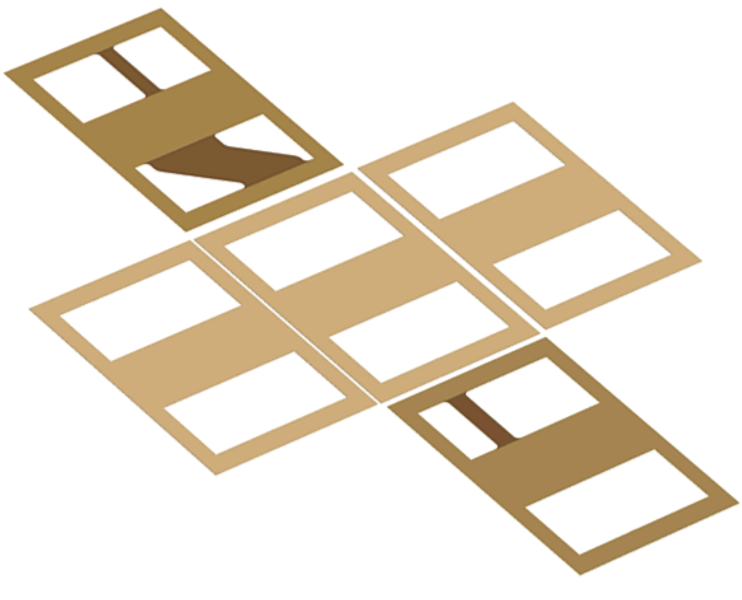


Figure 3: How the Etched, plated, coverlayed and blanked flex panels are combined with the glass-epoxy rigid panels.

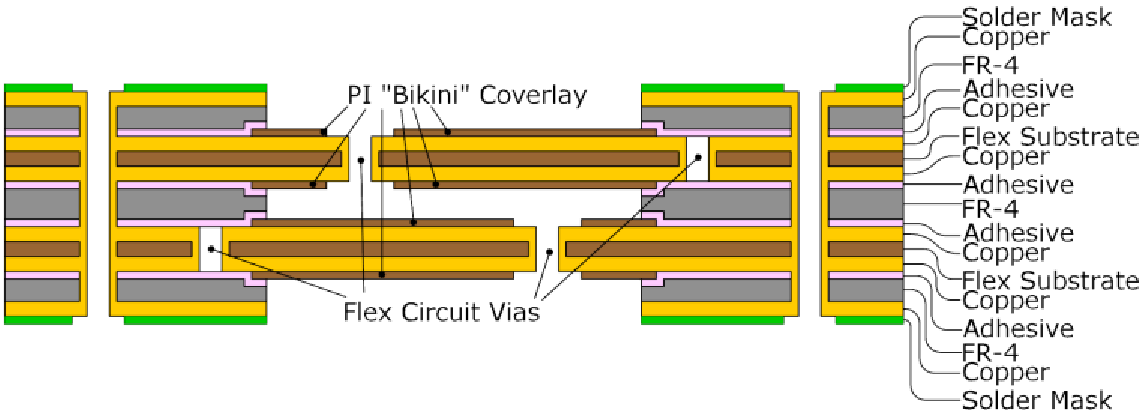


Figure 4: Detailed Stack Diagram including plated-through holes for each flex section, as well as final through-plated holes in the rigid section.

In the example stack up shown in figure 4, 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 boarders 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. I'll address some of these issues when looking at the design aspects in the next section.

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 final steps are the printing of the top and bottom soldermask, top and bottom silkscreen and preservative plating (such as ENIG) or hot air leveling (HASL).

PHYSICAL CONSTRAINTS

MULTIPLE FLEX SUB-STACKS

While it's possible to build just about any stackup 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. Again, copper is known to be work-hardened 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 in figure 5. Since the PolyImide is very elastic this is not a problem, and will last much longer under repeated movement than multiple copper layers will.

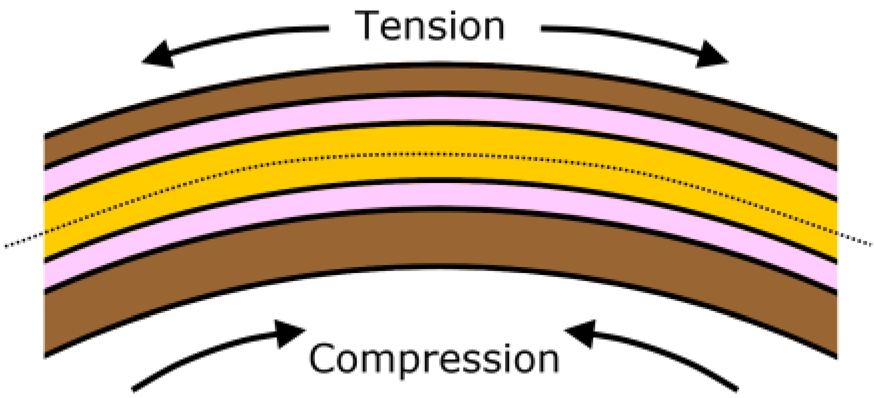


Figure 5: For highly repetitive bending circuits, it's best to use RA copper in single-layer flex to increase the fatigue life (in cycles before failure) of the copper in the circuit.

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. Ah! I'm getting ahead of myself - I'll write more about these design considerations next.

ADHESIVE BEADS

As I mentioned previously, 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.

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.

STIFFENERS & TERMINATIONS

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.

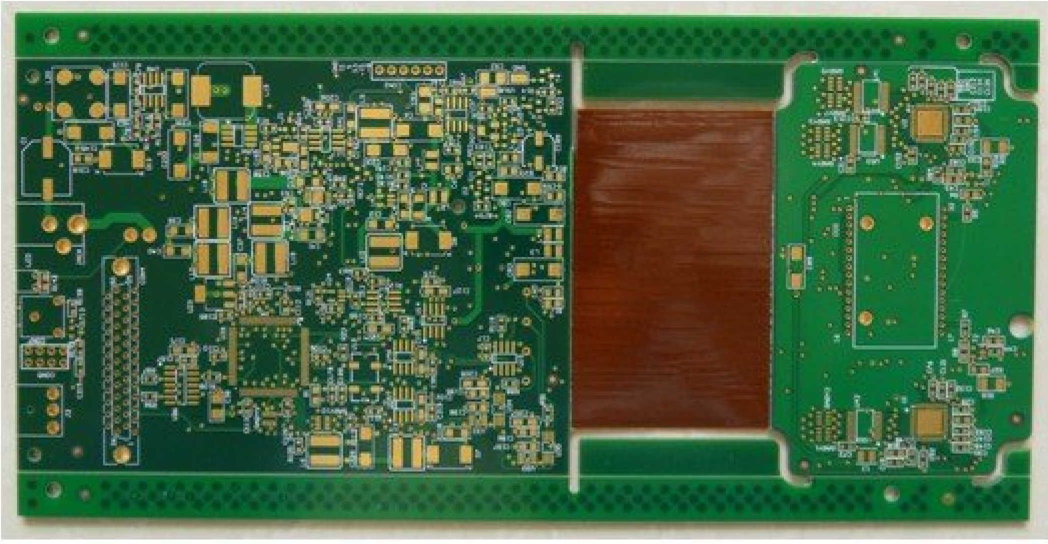
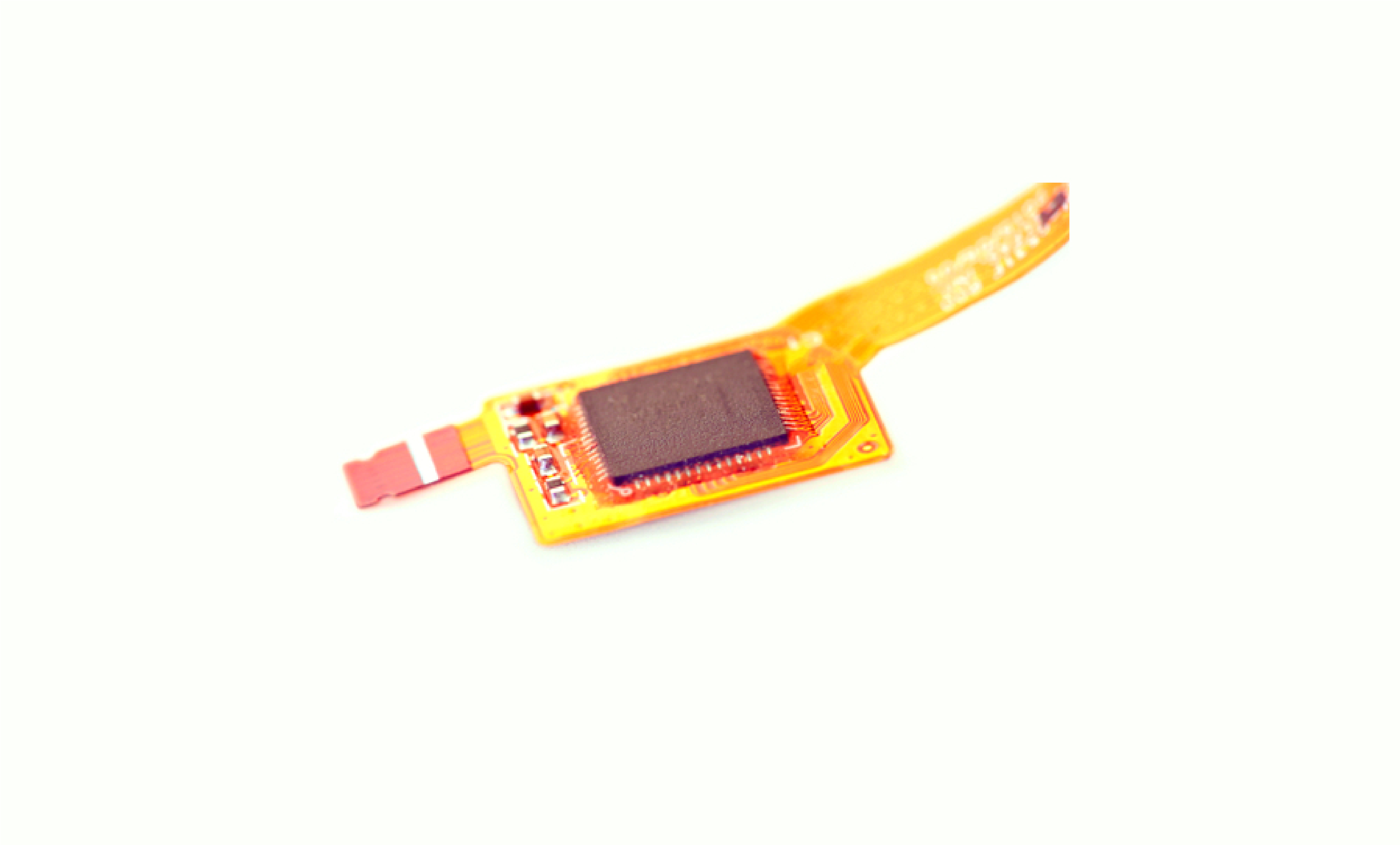


Figure 6: Final Rigid-Flex panel example. Notice that this one has front and back board edges, and flex circuit, routed out. The rigid sides are V-grooved for snapping off later. This will save time in assembly into the enclosure.

THE PANEL

The rigid flex circuit stays together in it's panel for the assembly process, so components can be placed and soldered on to the rigid terminations. 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.

DOCUMENTATION REQUIREMENTS FOR FLEX FABRICATION



In this third section on Rigid-Flex design, I review a few of the documentation requirements needed to get a flex or rigid-flex circuit board fabricated. Along with that, there are a few flex-circuit related issues to watch out for that I review here.

FAB DOCUMENTATION FOR FLEX CIRCUITS AND RIGID-FLEX BOARDS

In the last section on rigid-flex PCBs, I talked about the fabrication processes typically used by board houses. It's important to understand the steps required to build up a rigid-flex or flex circuit PCB because it has a big effect on how you need to design the board. And it also affects what needs to be included in your fabrication data set to send the design to successful fabrication.

DOCUMENTATION

Let's talk about documentation then. This is essentially where we tell the fabricator what we want, and it's probably the most likely part of the process where errors or misunderstandings can make costly delays happen. 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:

1. Make sure your fabricator is capable of building your rigid-flex design.
2. Make sure they collaborate with you on designing your layer stack to fit their particular processes.
3. 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.
4. Involve them as early as possible in the 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++ v7.0 or later is preferred, since it has specific layer types added to the job matrix that enable clear flex-circuit documentation for GenFlex® and similar CAM tools. A subset of the data included is shown in table 1.

Table 1: Subset of Layer Types in ODB++ (v7.0 and later) used for GenFlex
(Source: ODB++ v7.0 Specification)

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 circuit
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)
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 circuit
Power Ground Flex	pg	A power of ground layer for a flex circuit
Mixed Flex	mixed	Mixed layer for a flex circuit
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

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 circuit 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 circuit areas. The coverlay or covercoat also has to be considered a mask for component pads for those components that may be mounted on flex circuit 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-circuit 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 in figure 1.

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					

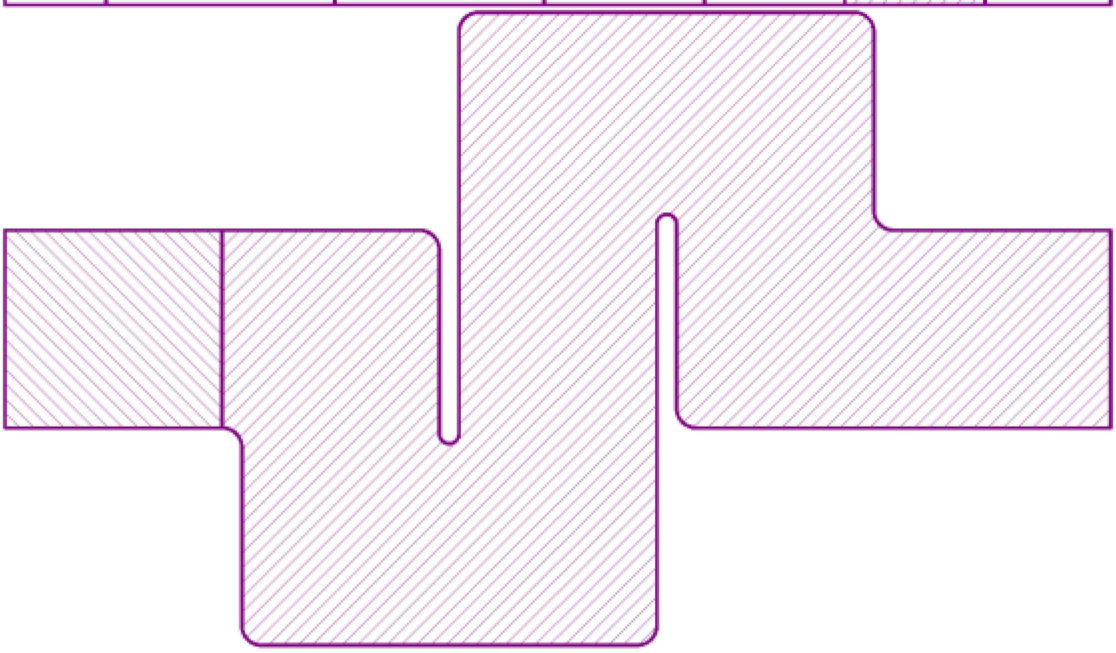


Figure 1: An example of a stack diagram showing fill patterns for rigid and flex circuit areas.

In this example, I used the 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.

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. I will discuss this a little more later on.

CONVEYING THE PCB DESIGN INTENT

We all know a picture is worth a thousand words. If you can generate a 3D image showing flexible and rigid areas this will help the fabricator understand your intent more clearly. Many people do this currently with the MCAD software, after having imported the STEP model from the PCB design. Figure 2 is an example of this concept.

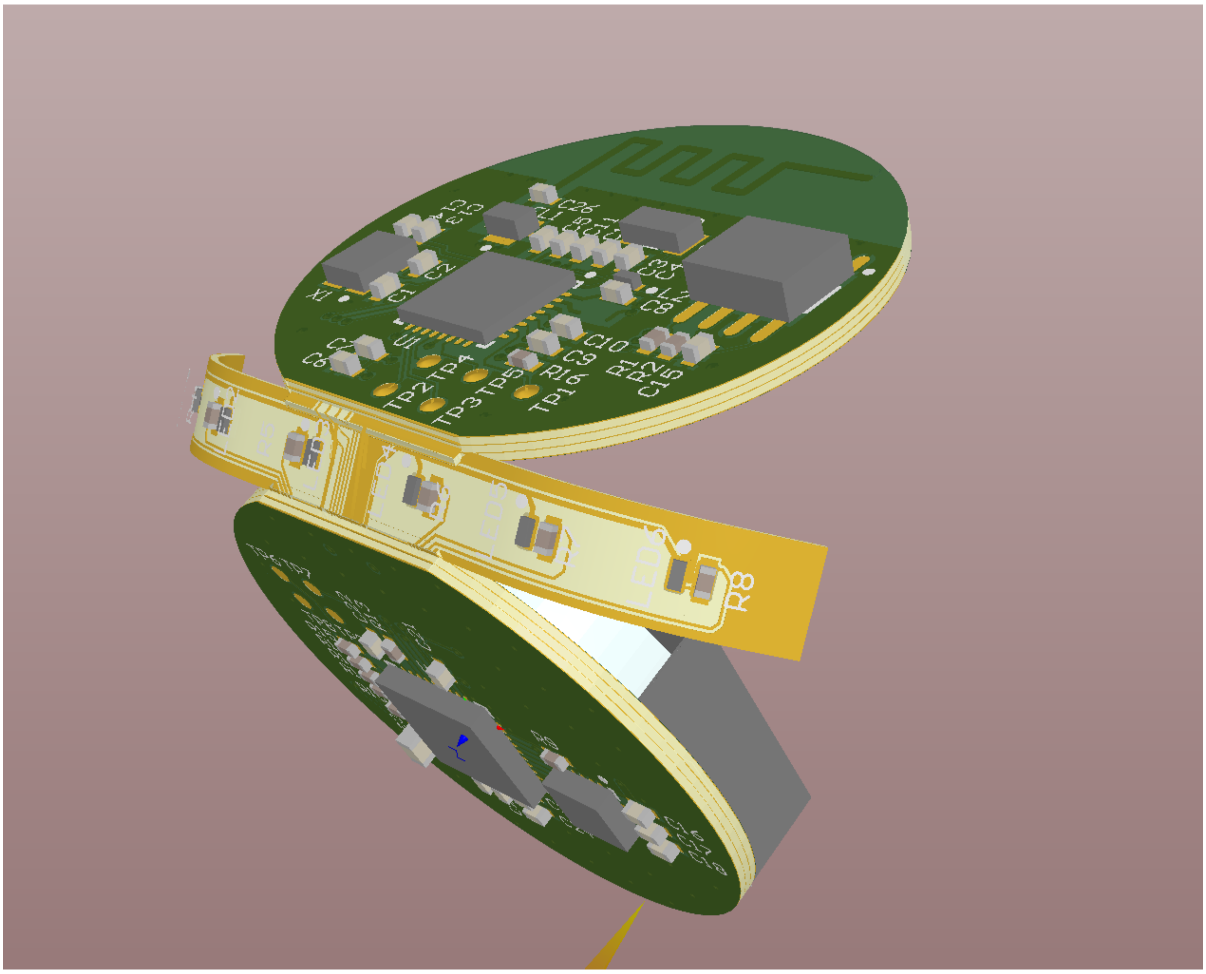


Figure 2: Bending up the mechanical model to show design intent.

This of course can have the added benefit of detecting flex to flex and flex to rigid interferences ahead of epic failure.

PARTS 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 circuit coverlay. Again, this is something that needs to be discussed with the fabricator to determine what's possible and economical.

Side 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.

SIDE-NOTE: DEFINING FLEX CUTOUT

Notice in figure 1 how there are no hard corners, but rather there's a minimum radius to each angle? IPC recommends greater radii than 1.5mm (about 60 mils), to reduce the risk of tearing of the flex circuit at corners. The same goes for slots and slits in the flex - make sure there's a designed-in relief hole at each end of diameter 3mm (⅛") or more. Another example of this is shown below.

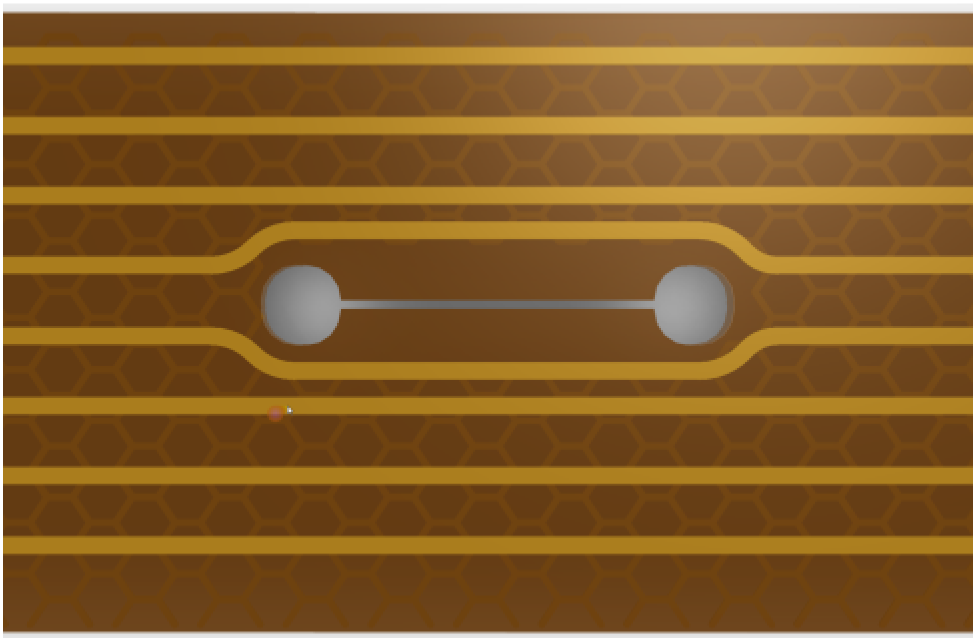


Figure 3: Slots, slits and inside corners should have tear-relief holes or tangent curves with minimum 1.5mm radius.

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. In the next section I discuss several of these do's and don'ts of Rigid-Flex.

DO'S AND DON'TS OF RIGID-FLEX DESIGN

Leading up to now, we've looked at rigid-flex circuit materials, fabrication, and some aspects of design. In this section, I want to show a handful of important rules to follow when routing copper for flex and rigid-flex circuits, that not only increase the fabrication yield but also the reliability and lifespan of the flex circuit.

DOS AND DON'TS OF FLEX CIRCUIT COPPER

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 in my first art in this series 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. What I mean is; 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 single-sided flex. 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

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.

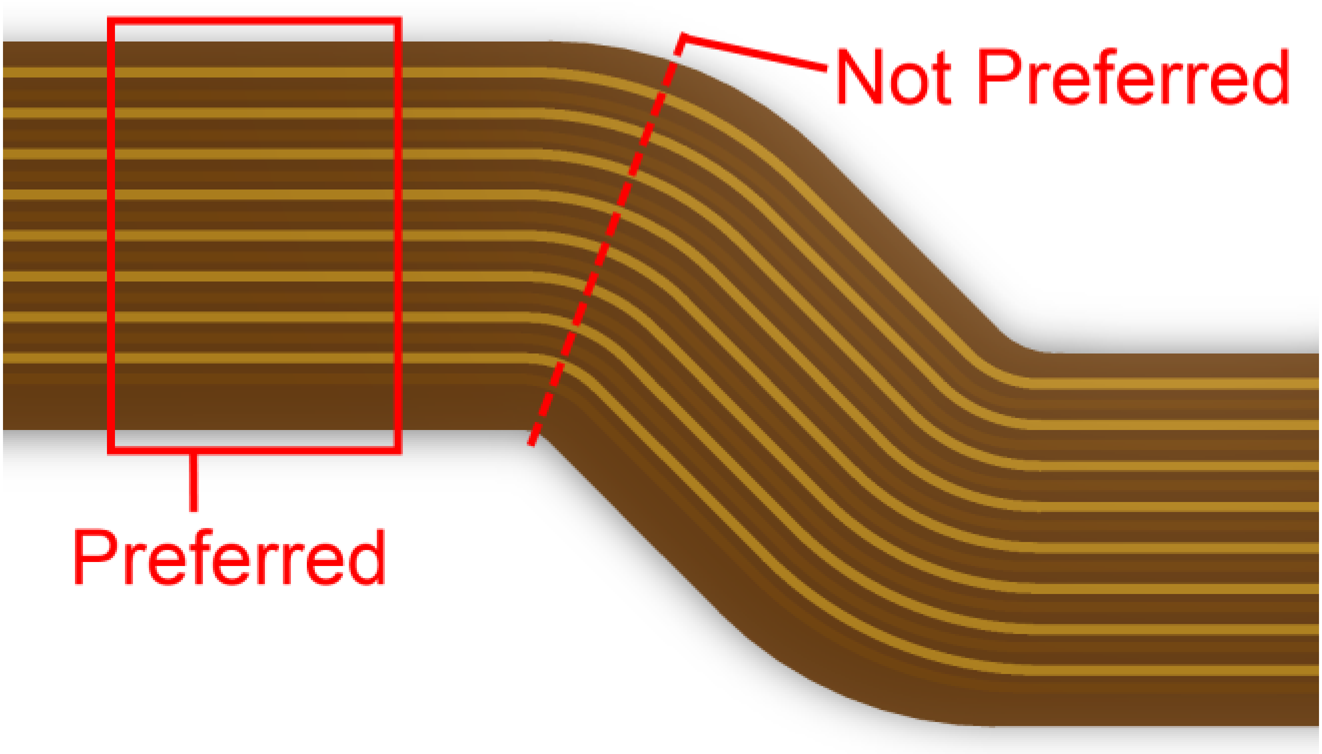


Figure 1: Preferred bend locations.

DO USE CURVED TRACES

Also referring to figure 1 above, 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.

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, it's advisable to taper down from the pads (hint: teardrop the pads and vias in the flex circuit!)

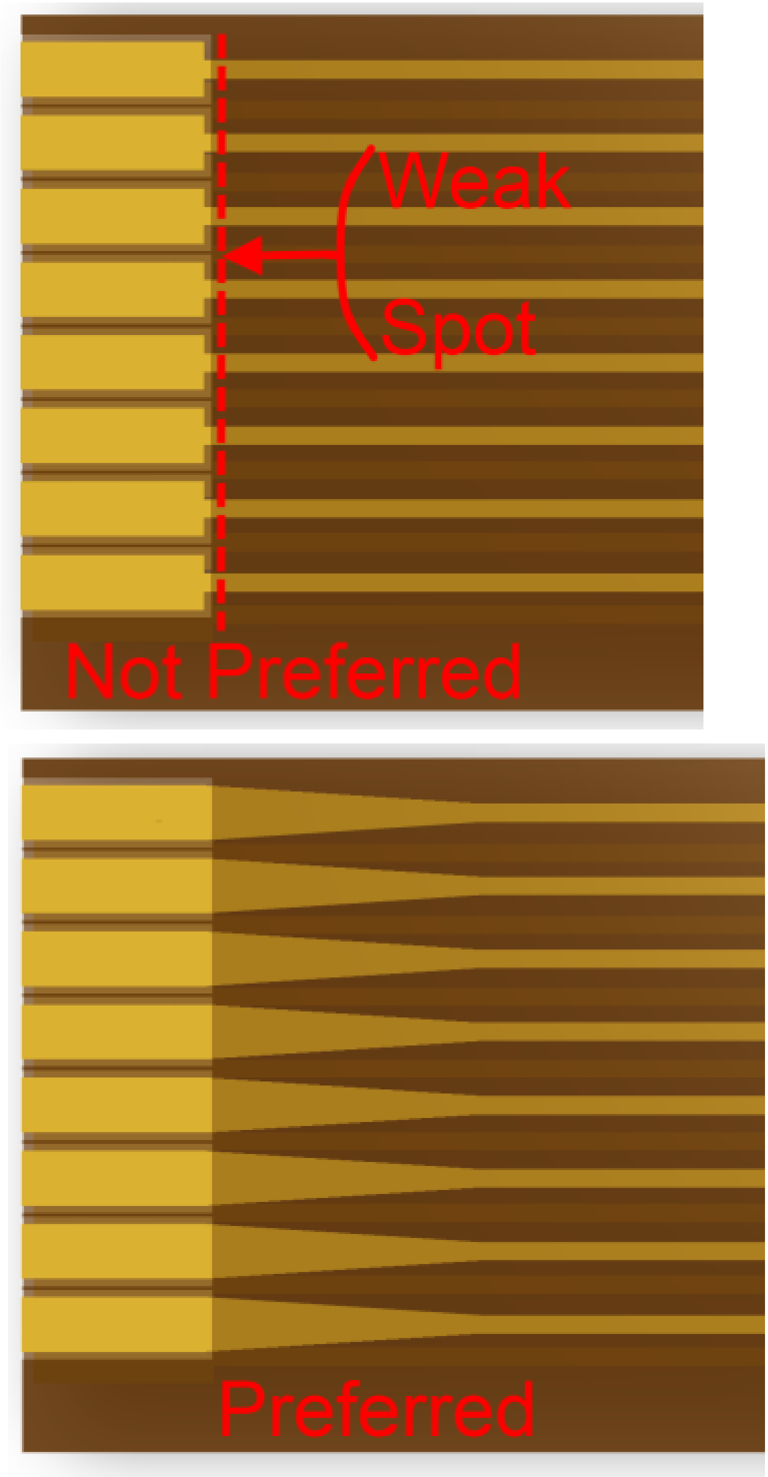


Figure 2: Trace width change and pad entries can cause weak spots.

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.

While thinking about this one, it also occurred to me that 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 I found it fast enough to build the hatch below with cut-and-past.

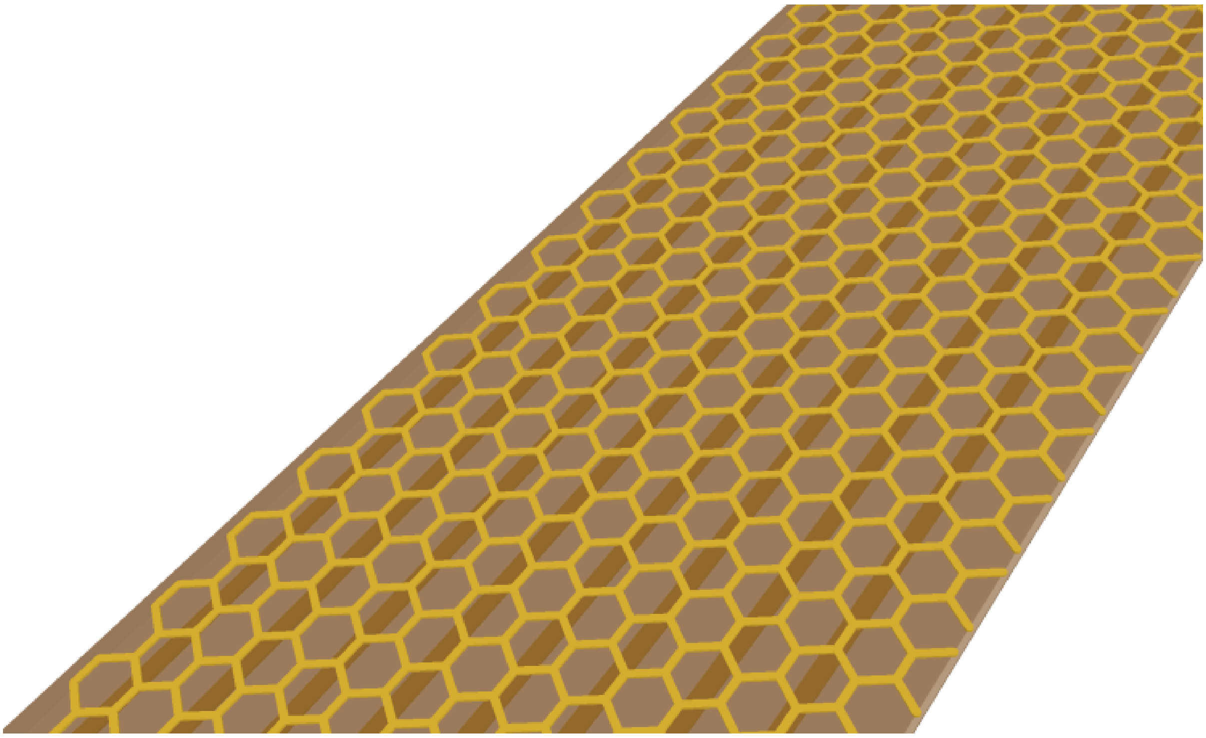


Figure 3: Using hexagonal hatched polygons can spread the tension biases evenly among three angles.

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 (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. Surface mount pads and non-plated-through pads are referred to as unsupported, and need additional measures to prevent detachment.

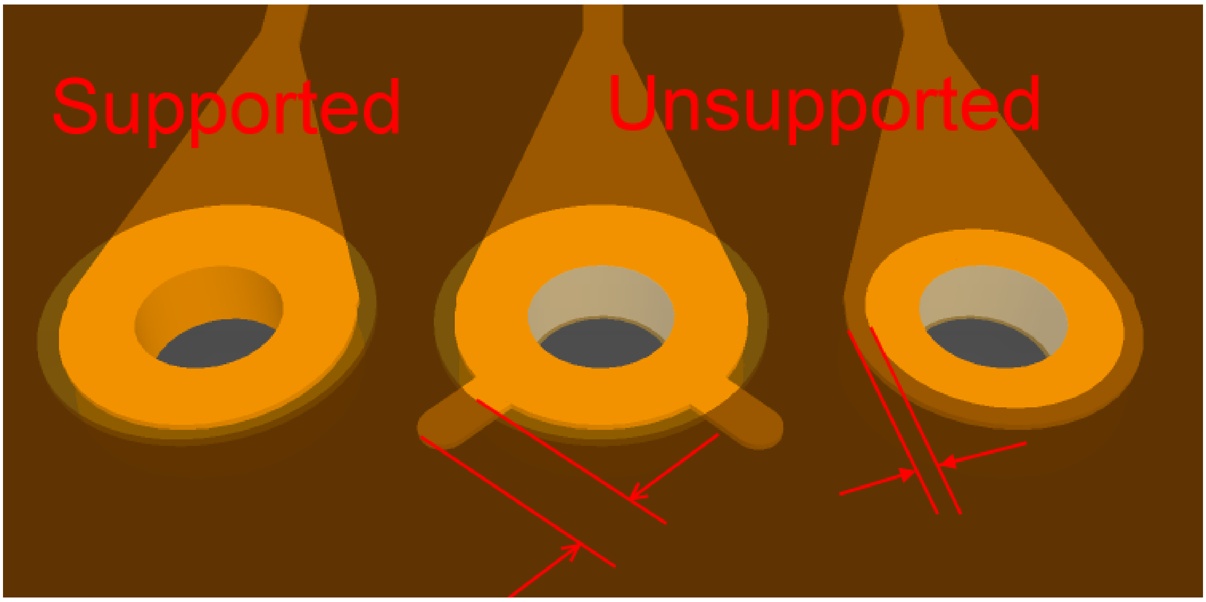


Figure 4: Supporting through-hole pads in flex with plating, anchoring stubs, and reduced coverlay access openings.

Referring to figure 4, the second option is good for adhesive coverlay and the third for adhesiveless. Coverlays attached with adhesive will exhibit “squeeze out” of the adhesive, so the pad land and the access opening must be large enough to allow for this while providing a good solder fillet.

SMT component pads are among the most vulnerable, especially as the flex circuit may bend under the component’s rigid pin and solder fillet. Figures 5 and 6 show 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 is compared in figure 6, with the bottom SMD footprint used for flex mounted components. This obviously reduces the density of flex circuit component mounting, but by nature flex circuits cannot be very dense compared with rigid.

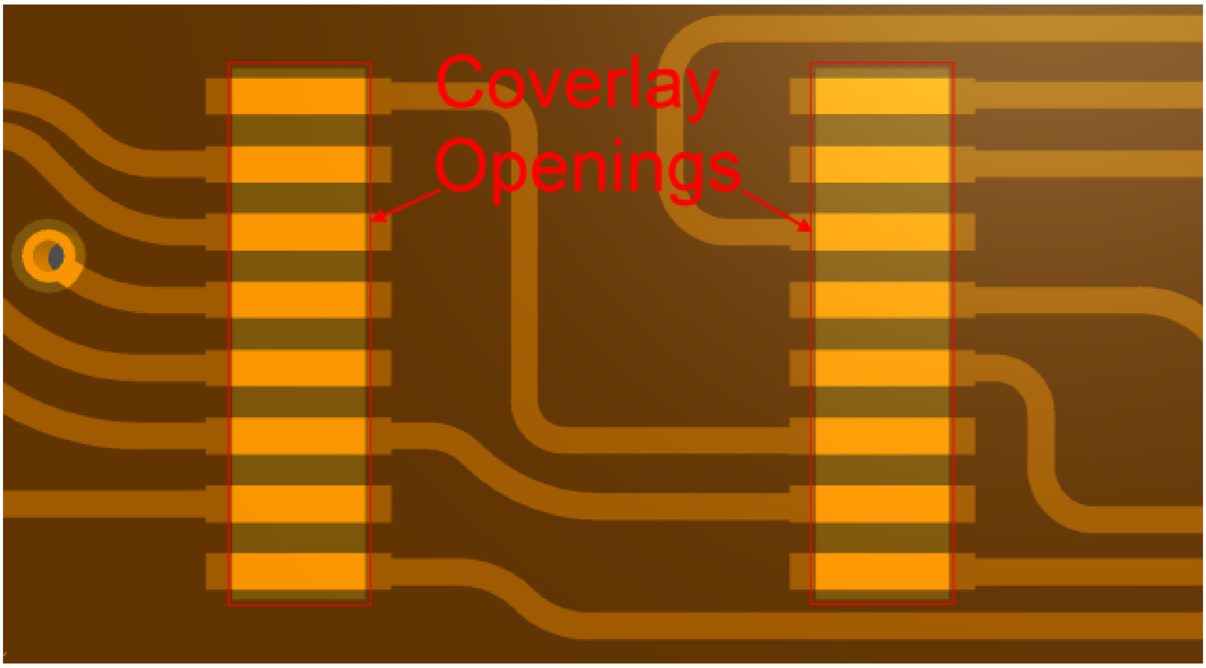


Figure 5: Coverlay openings for an SOW package showing anchoring at each end of each pad.

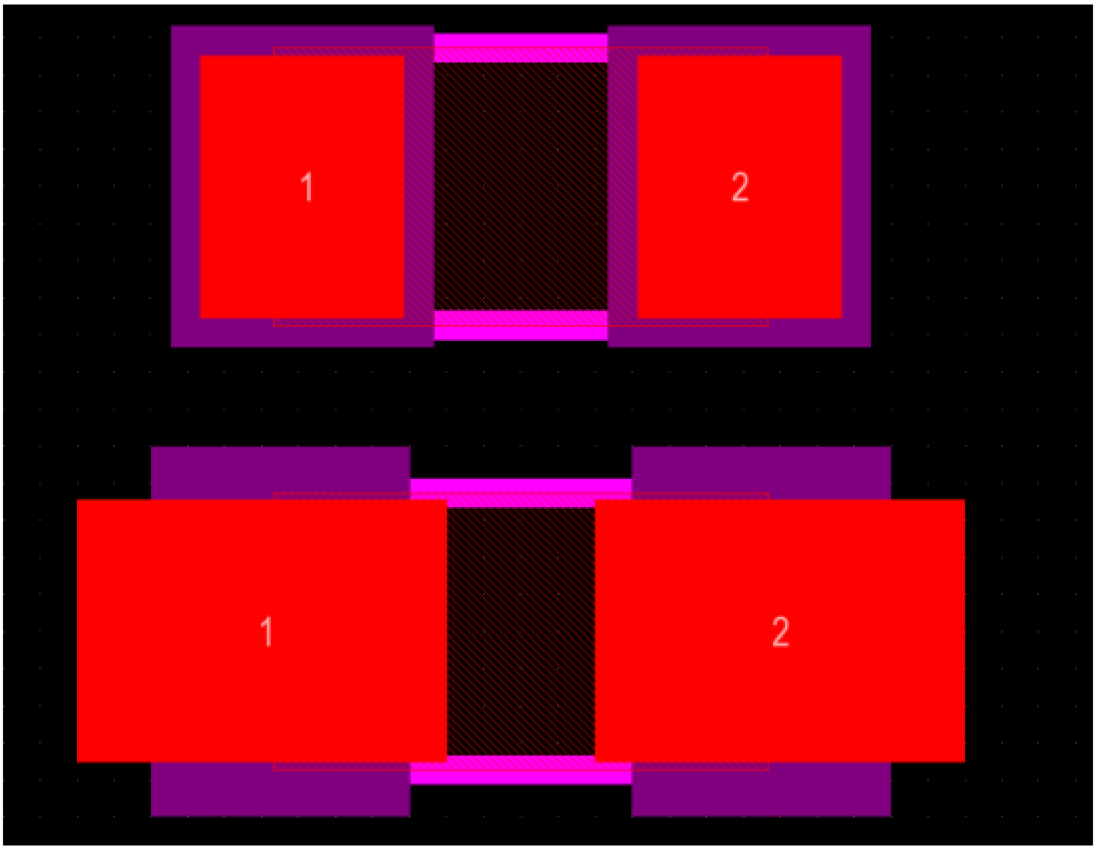


Figure 6: Adjusting the pad sizes and “mask” opening for coverlay. The top land pattern is for a nominal 0603 size chip component, whereas the bottom is modified for coverlay anchoring.

DOUBLE-SIDED FLEX

For dynamic double-sided flex circuits, try to avoid laying traces over each other on the same direction (figure 7). Rather stagger them so the tensions are more evenly distributed between copper layers (figure 8).

Figure 7: Adjacent-layer copper traces are not recommended.

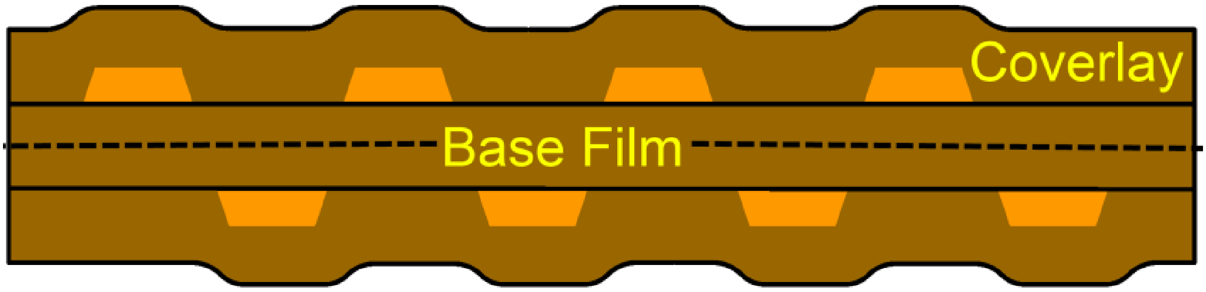


Figure 8: Staggered adjacent-layer traces are preferred.

This is by no means a complete set, but You should now have a few good tips on how to design flex circuits to give the best yield and highest reliability for the product - but be aware of the tradeoffs between cost, performance and reliability.