PFC
FLEXIBLE CIRCUITS LIMITED

DESIGN MANUAL 2104

Prepared by PFC Engineering

PFC FLEXIBLE CIRCUITS LIMITED
11 CANADIAN RD. UNIT 7
SCARBOROUGH, ONT.
CANADA, M1R5G1
416 750-8433

WWW.PFCFLEX.COM

INTRODUCTION TO FLEX CIRCUITS

Benefits of Flexible Printed Circuit Board

Flexible printed circuit board technology provides a multitude of new possibilities for different types of product, e.g. **reduced size** and **weight**, **improved packaging** and **interconnect**, **superior mechanical** and **electrical integration**, **increased functionality**.

(1) Reduced Weight and Space

- Considerable weight reduction. Up to 80% lighter than a PCB
- Thickness can be as thin as 0.002”.

(2) Improved Packaging

- Formability of the FPC enables a package size reduction
- FPC allows unique designs which solve interconnection problems

(3) Interconnect Solution

- Reduces the number of levels of interconnection required in an electronic package
- Simplify system design

(4) Dynamic Flexing

- The thinness of the material makes FPC the best solution for flexible applications- up to millions of cycles

(5) Electrical Integration

- Material options for FPC allow tailored designs- electrical characteristics, flexibility
- High-data signal speed greater than 10GHZ

(6) Increased Functionality

- Thermal management for FPC dissipate heat at a better rate than any other dielectric materials while providing the added benefits of greatly improved flexibility

Single-Sided, Double-Sided, Multi-Layer Circuit

▪ Single-Sided FPC

- Single-Sided FPC consist of a single copper conductor layer on a flexible dielectric film

▪ Single-Sided Features

- Very thin construction total thickness 0.002" – 0.006" (50um – 150um)

▪ When to use Single-Sided FPC

- Dynamic flexing application
- Unusual folding and forming application

▪ Double-Sided FPC

- Double-Sided FPC consist of two copper layer with dielectric film, normally connected with a plated through-hole

▪ Double-Sided Features

- Component assembly available on both sides
- Two conductive layer

▪ When to use Double-Sided FPC

- Required when circuit density and layout cannot be routed on a single layer
- Ground and power plane applications

▪ Multi-Layer FPC

- Three or more flexible conductive layers with flexible insulating layers between each one, plated through-holes provide connection between layers

▪ Multi-Layer FPC Features

- Improve rigidity on component areas (e.g. BTB connector)

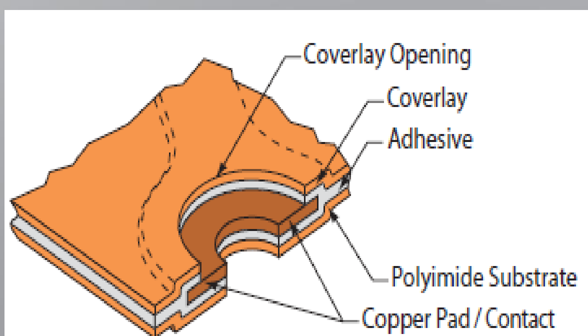
▪ When to use Multi-Layer FPC

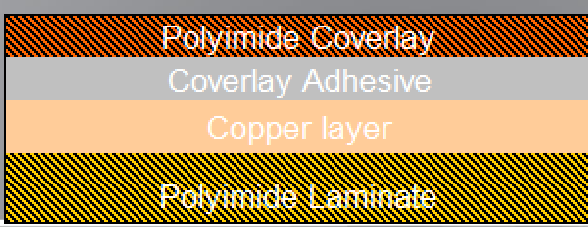
- Used for shielding application
- Multiple I/O signals
- Dense surface mount assembly
- FPC miniaturization

Single-Sided, Double-Sided, Multi-Layer Circuit Constructions

Single-Sided FPC

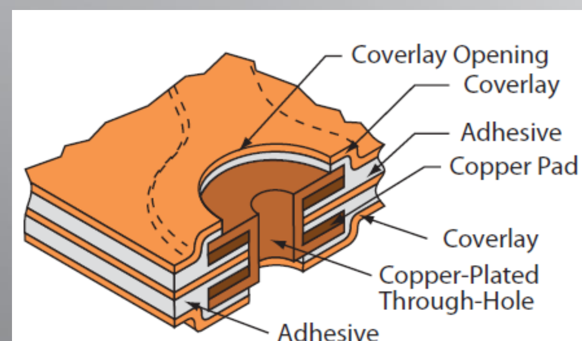
- Cheaper material cost
- Thinnest and most flexible

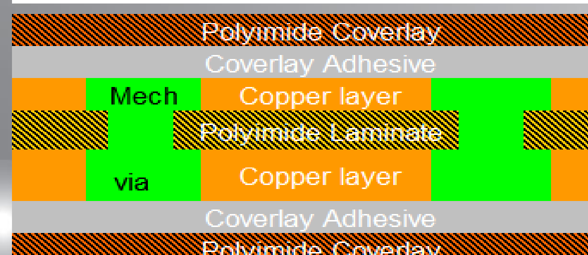




Double-Sided FPC

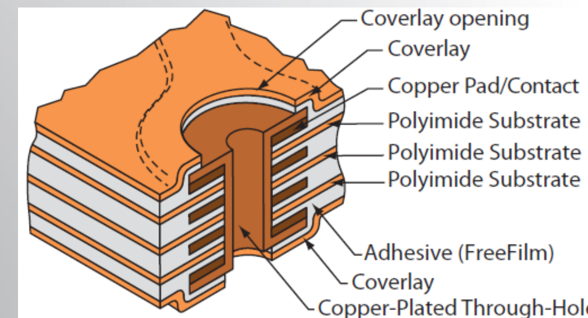
- Improves the layout density and increases I/O count

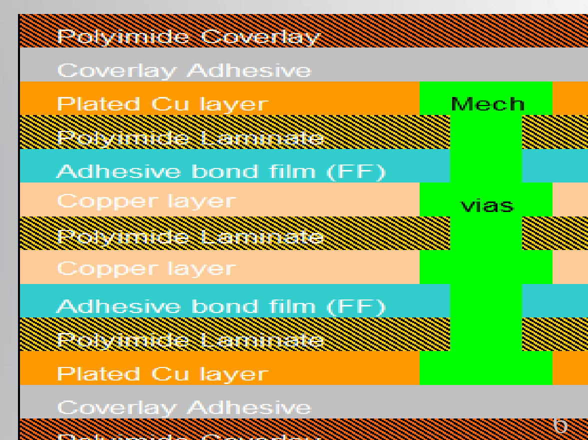




Multi-Layer FPC

- Improved EMI and Impedance applications while maintaining FPC flexural properties





Rigid Flex and Sculptured Flex Circuit Constructions

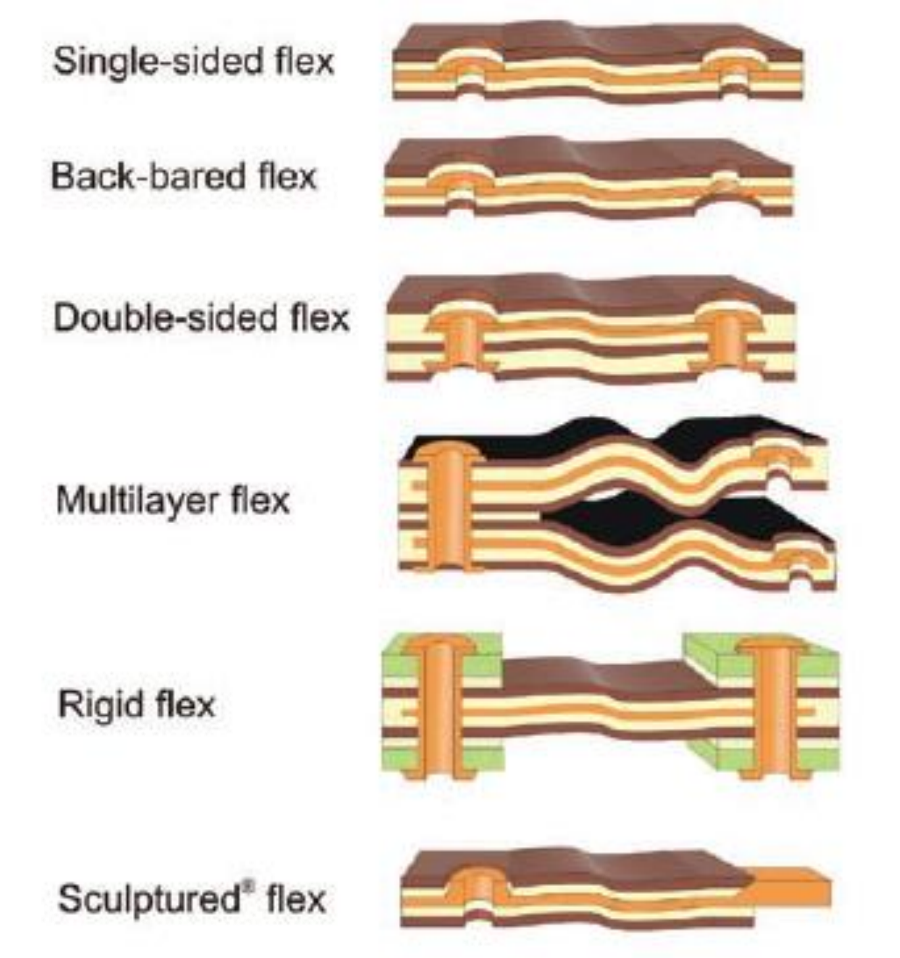
Rigid Flex Circuits

Hybrid constructions consisting of rigid and flexible materials that are laminated together into a single structure.

Sculptured Flex Circuits

Often used to create bare/exposed metal contacts beyond the polyimide. The method involves selective etching of thick copper foils to create exposed plated fingers.

*Patented by Advanced Circuit Technology



HOW IS A FLEX CIRCUIT MANUFACTURED?

Drilling

WHAT IS DRILLING?

Drilling is a process where holes and via holes are generated with drill bits.

The available drill size ranges from 0.0059” to 0.250” in diameter.

Mainly, there are 3 types of drill bits.

1. Slot drill – for radius slot milling
2. Normal drill – for drilling normal round holes
3. Router drill – to make a cutout, for routing FR4

Some of the major quality issues corresponding to the drilling process are summarized in the following table:

Defects	Possible root causes
Drilling burr	Wrong setting, defect materials, drill bit wear off
Drilling dent	Insufficient suction pressure
Shifted hole	Wrong program, shrinkage
Missing hole	Drill bit wear off
Zero offset	Wrong setting

TYPES OF DRILLING

1. CNC Drilling

- Computerized numerical control (CNC) drilling is implemented for mass production
- The fastest machines for drilling varying hole sizes have multiple spindles in turrets with drills of varying diameters already mounted for drilling
- Can be used for drilling holes and routing

2. Laser Drilling

- Drilling hole in material by using a laser beam
- It is cost effective, and has no drill breakage or tool wear
- It has the ability to drill unlimited hole sizes and shapes in a variety of materials including most metals, ceramics, and semiconductors

Imaging

WHAT IS IMAGING?

Imaging is a process whereby circuit patterns are exposed onto photoresist using UV source and photographic artwork.

Imaging is sub-divided into 3 sub-processes

1. Dry film lamination
2. Artwork registration and exposure
3. Develop

Mechanism

UV radiation is applied to crosslink photopolymer chains. The cross-linked polymer, which is the exposed area, has high resistance to alkaline solution, whereas the unexposed area will react with alkaline solution and become soluble in water.

PROCESS FLOW OF IMAGING

1. Dry Film Lamination

Laminated a layer of dry film on top of the copper

2. Art Work Registration

Visually align artwork using Carlson pin on top of the panel and adhesive attached to it

3. Exposure

UV-radiation is used to create a circuit pattern on top of the copper surface

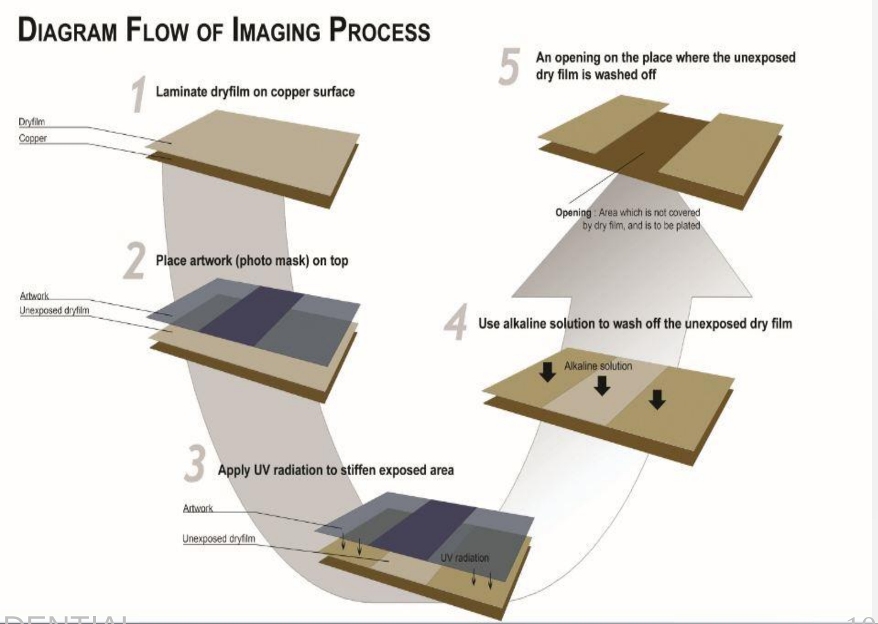
4. Develop

To clean away the unexposed dry film on the panel using alkaline solution

Material Explanation

Dry Film: light sensitive polymer which allow pattern to be formed on it

Artwork: piece of transparency with design of circuit pattern on it



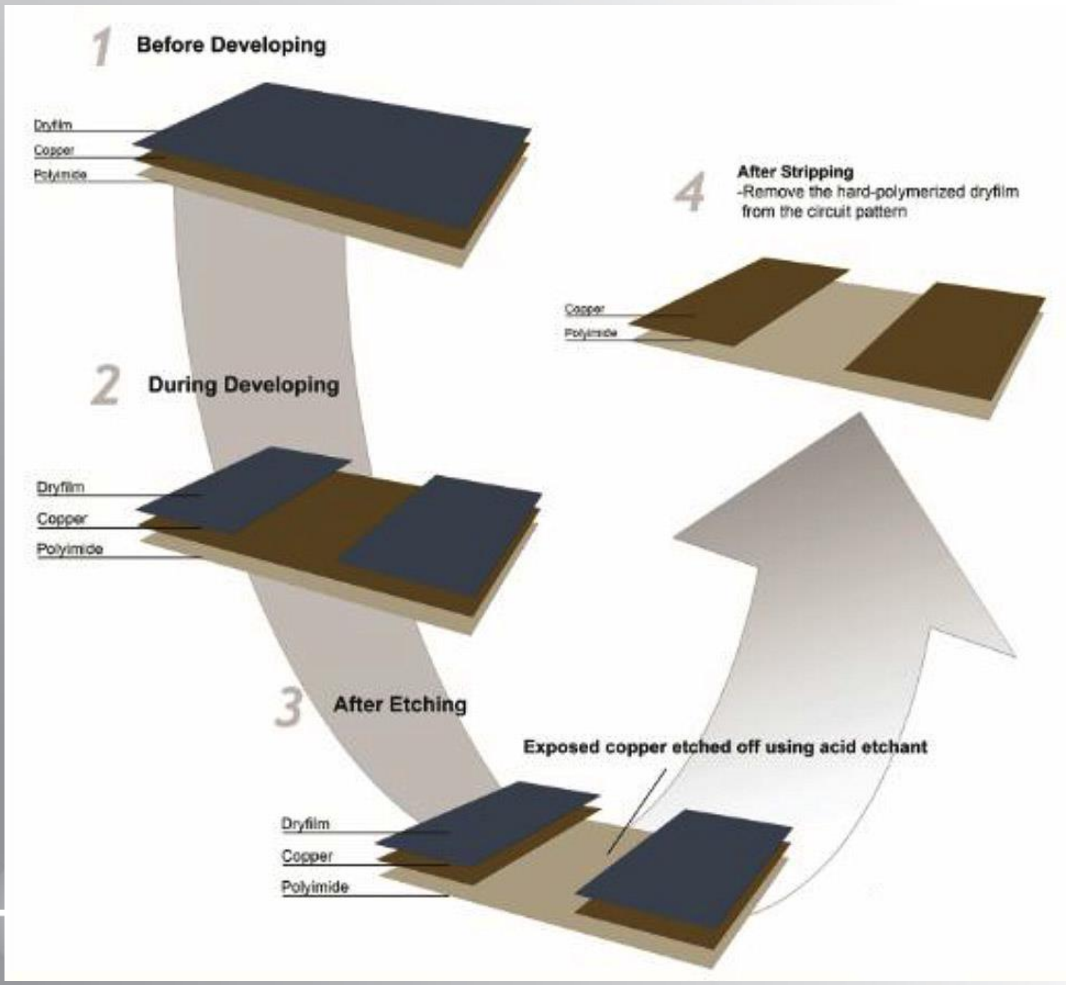
Etching and Stripping

ETCHING

To remove the unwanted copper

STRIPPING

To remove the hard polymerized resist from the remaining copper resist



Electrolytic Plating

WHAT IS ELECTROLYTIC PLATING?

Deposition process of metallic layers using electrical current.

Types of Plating

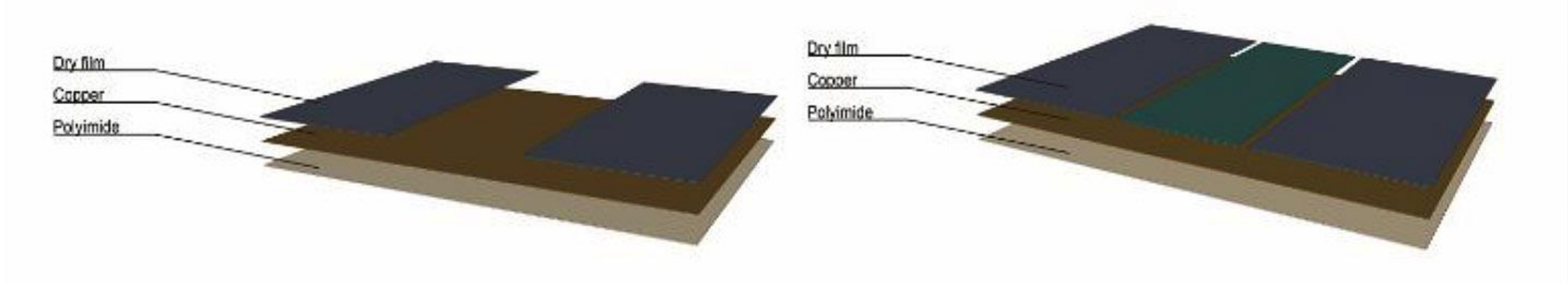
1. Panel Plating
Plate the surface and via holes
2. Pad Plating
Selectively plate via pad and via hole
3. Pattern Plating
Selectively plate circuit and via hole wall

Plating Processes

1. Acid Clean
2. Micro-Etch
3. Acid Dip
4. Copper Plating

AFTER DEVELOPING

AFTER COPPER PLATING



Lay-Up Coverlay

WHAT IS LAY UP?

Lay Up is simply the stacking of materials on top of another.

Materials for Lay Up

1. Sheet adhesive: to provide layer to layer adhesion
2. Coverlay: as a protective shield for the copper surface
3. Stiffener: to stiffen the panel
4. FR4: Frame-retardant 4, a type of stiffener

Lay Up Method for Coverlay

1. Sticky rollers are used to clean surface for lay up
2. Registration with the help of jigs
3. Alignment of pattern or holes of panel with the pattern or holes on lay up material
4. Heat the lay up material on the dummy area by using soldering iron

Important

Misalignment between layers will cause open or short circuit for the final product

PFC FLEXIBLE CIRCUITS LIMITED

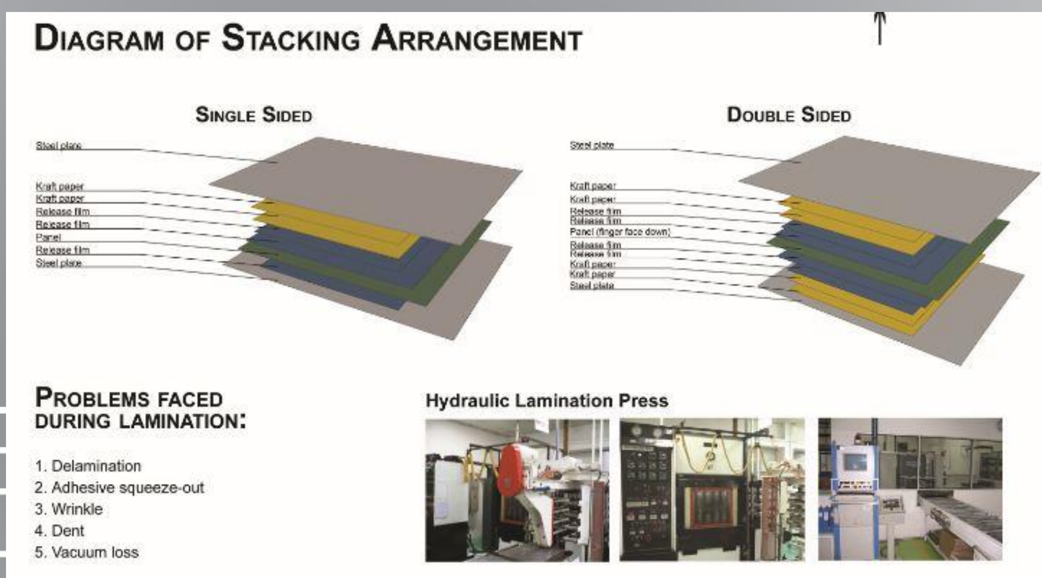
Lamination

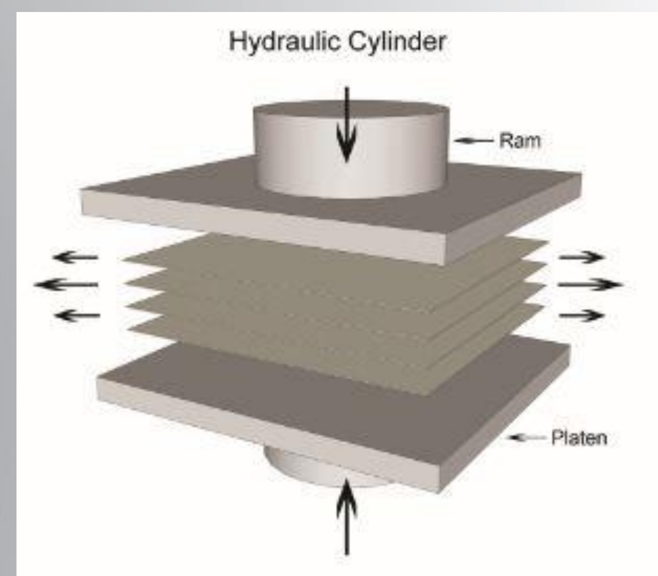
WHAT IS LAMINATION?

A process whereby two or more layers of material are compressively bind together by using heat and pressure.

How it Works

It uses hydraulic cylinders to apply pressure to press platens.
Platens are heated by hot oil or electrical elements.
The operating parameters in this process are lamination temperature, pressure and pressing time.



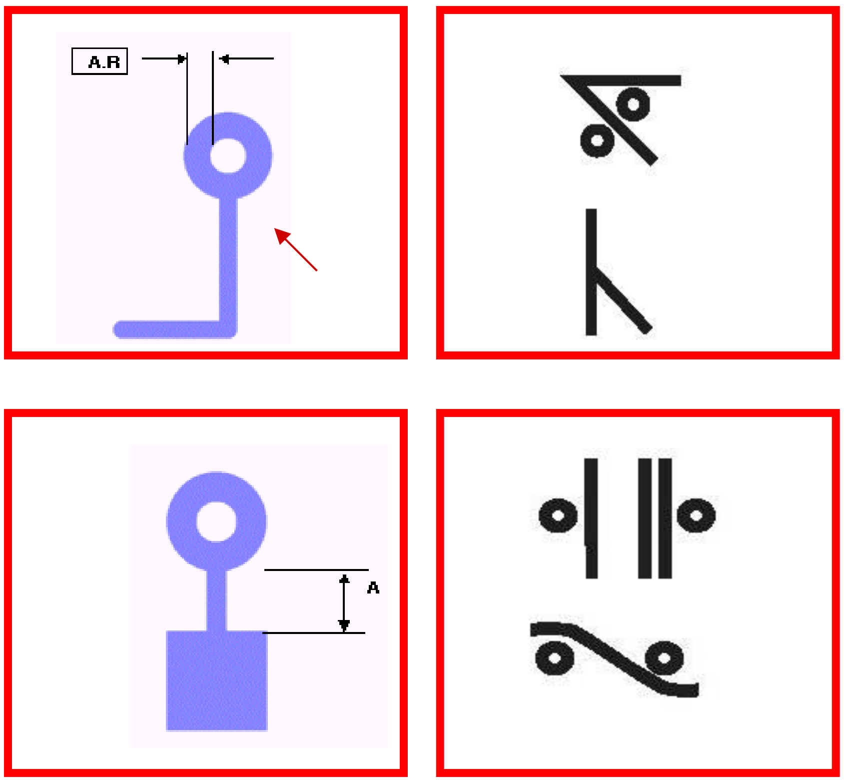


FLEX CIRCUIT LAYOUT

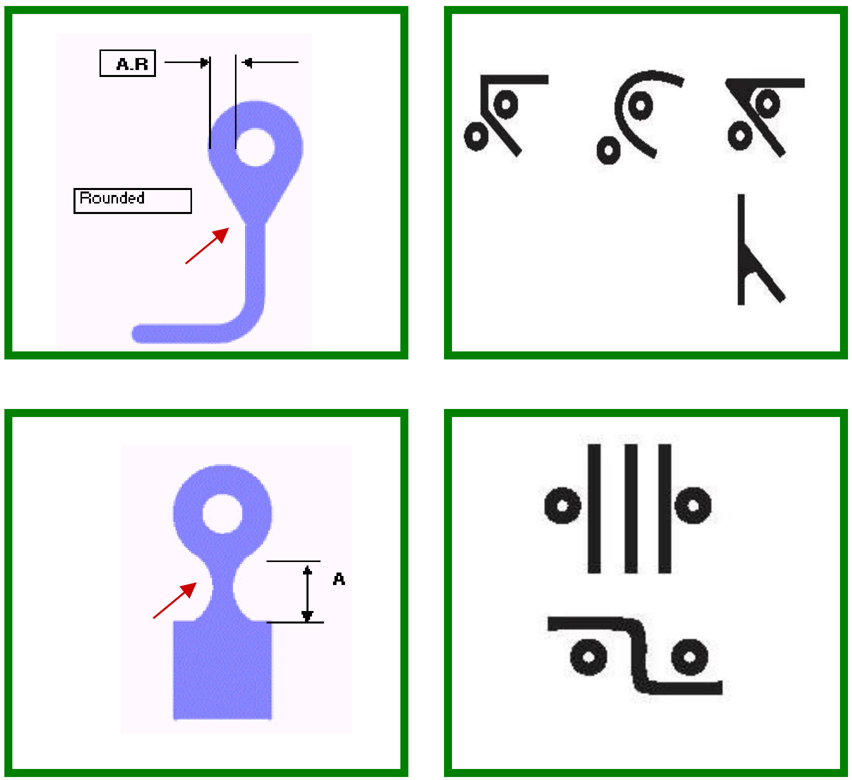
Flex Design Guidelines

CONDUCTOR ANNULAR RING DESIGN AND FILLETING – All pads, on both through-hole and surface mounted pads should be filleted to reduce stress points. This helps eliminate breaking during flexing.

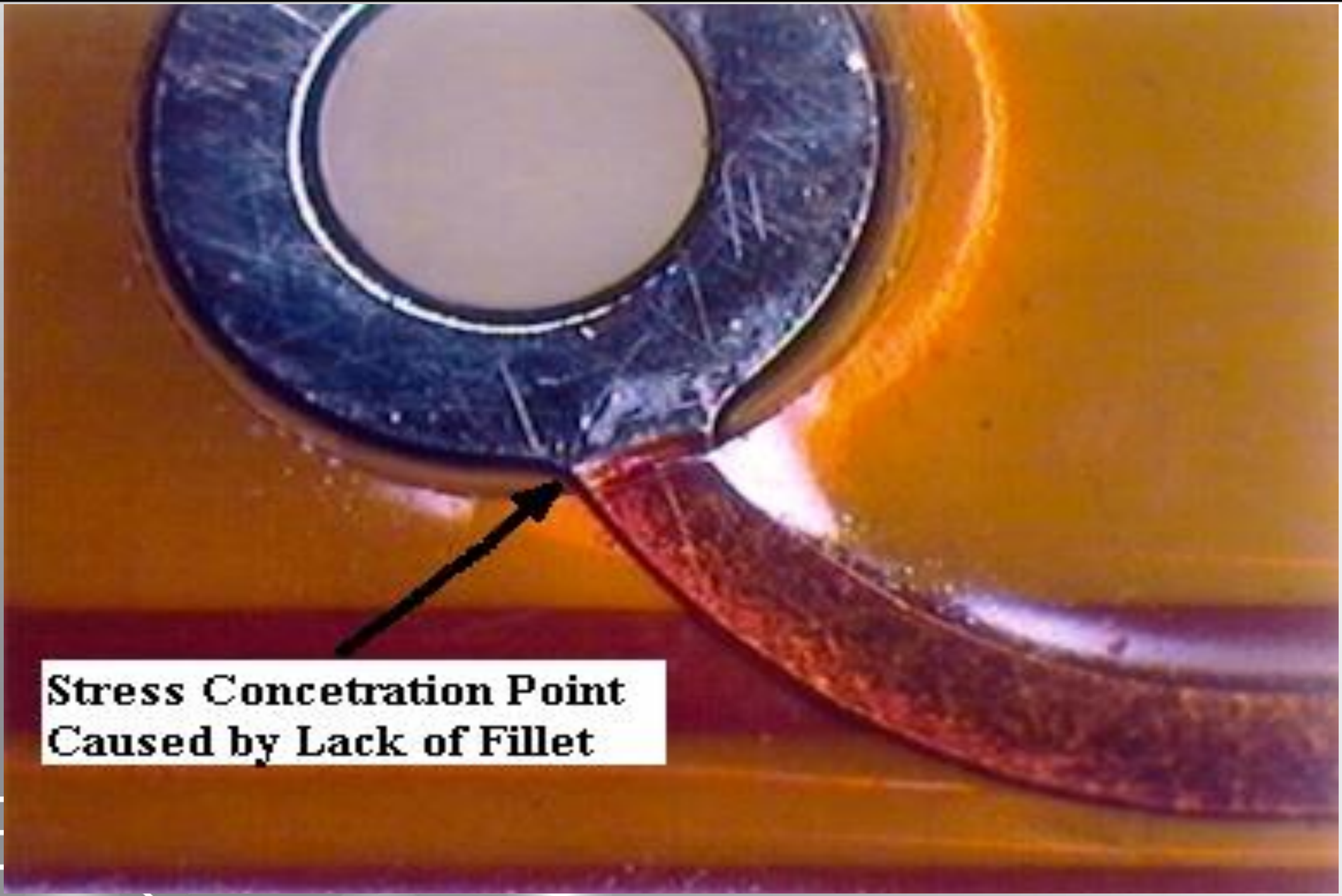
Not Preferred



Preferred



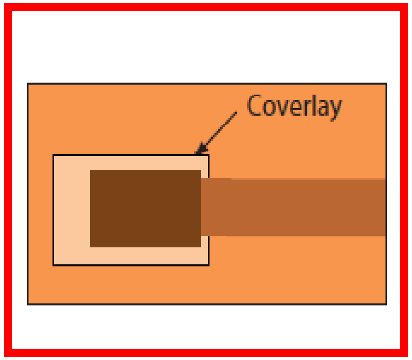
DESIGN GUIDELINES FILLETING OF TERMINATION LANDS

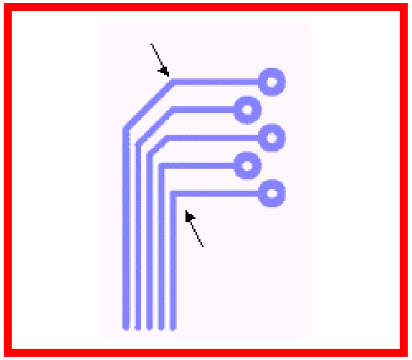


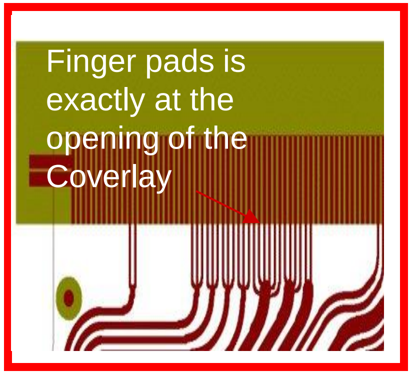
Flex Design Guidelines

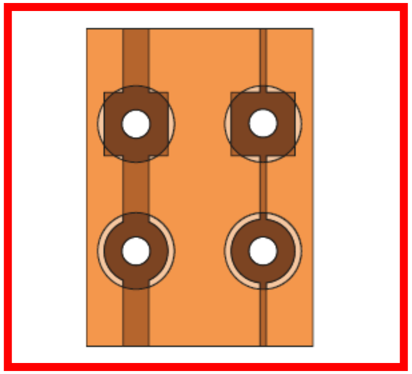
TIE-DOWNS AND RADIUSED TRACES – Tie-downs are captured by the coverlay to anchor the copper to prevent separation between the copper and the base material. Radiused traces help to reduce breaking during folding and bending.

Not Preferred

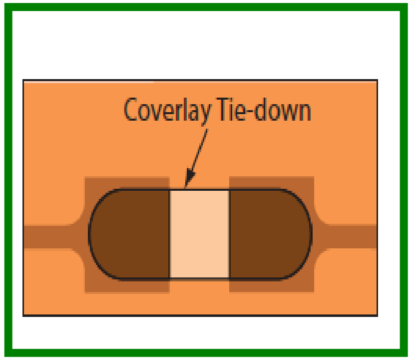


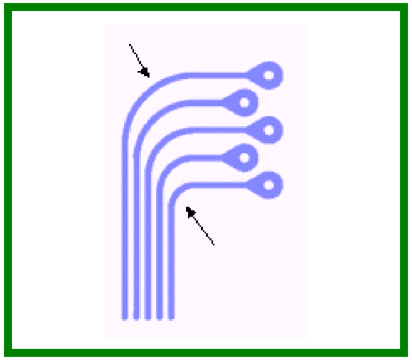


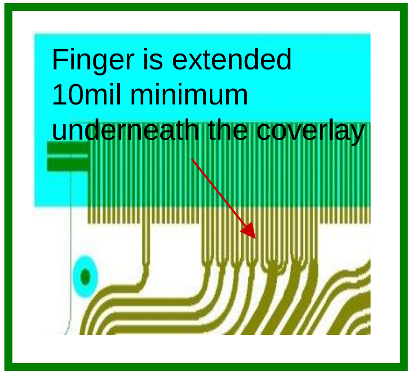


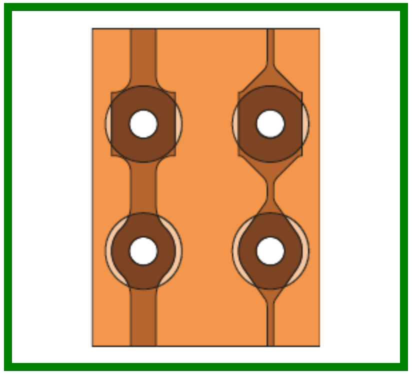


Preferred







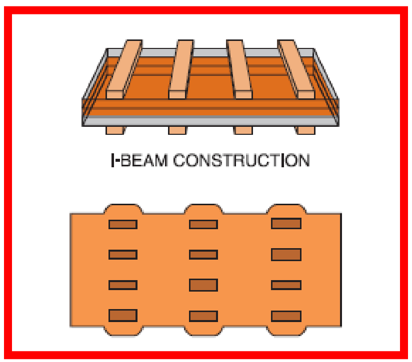


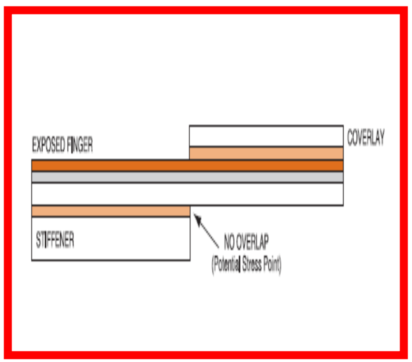
Flex Design Guidelines

I-BEAM DESIGN AND STIFFENERS – I-Beam constructions occur when the inductors on both layers lie directly on top of each other, increasing the stiffness of the circuit through fold areas. A better alternative is stagger conductors, alternating their location to retain the maximum flexibility of the circuit.

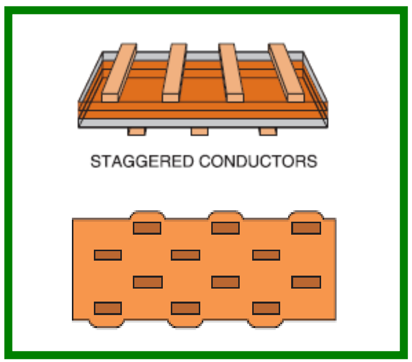
STIFFENERS DESIGN REFERENCE – Stiffener and coverlay termination line should overlap a minimum of 12mil to avoid stress points. Eliminating the stress points reduce the chance of traces breaking.

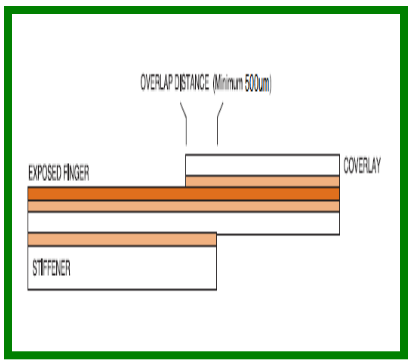
Not Preferred





Preferred





CREATING A RELIABLE FLEX CIRCUIT

Bending

Design for Reliability

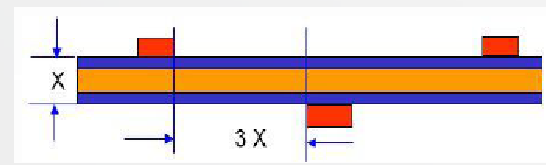
▪ Reduce overall thickness in the flex area

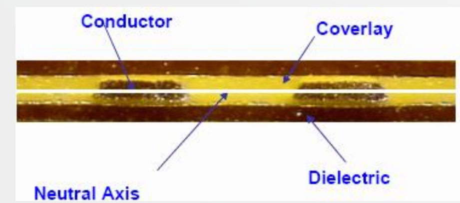
• Reduce the base copper weight (and the corresponding adhesive thicknesses) or reduce the dielectric thickness.

• Use adhesiveless base materials. Adhesiveless materials will usually reduce the starting thickness of each substrate by 12-25um (0.0005"- 0.0010") when compared to adhesive based substrates.

• Eliminate copper plating on the conductors in the flexing area (dynamic region) by utilizing selective (pads plating/button plating-only) allowing the circuit to have increased flexibility.

Polyimide Coverlay		12UM_CVL	12
Coverlay Adhesive		15UM_CVL_ADH	15
Copper layer	L1	12UM_ED	12
Polyimide Laminate		20UM_SS_FCCL	20





▪ Make the circuit robust to withstand flexing

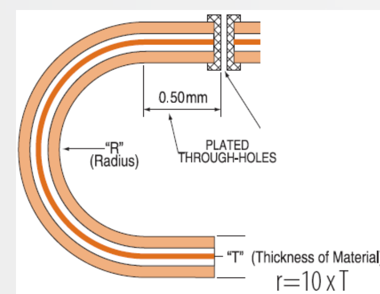
• Conductors should be staggered from layer to layer and not stacked on top of each other to increase flexibility.

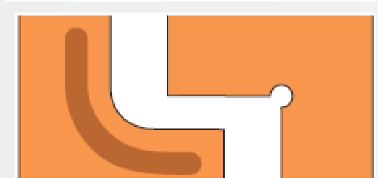
• Conductor thickness and width should remain constant in bending areas.

• Balance the conductor weights and material thicknesses on each side of the neutral bend axis.

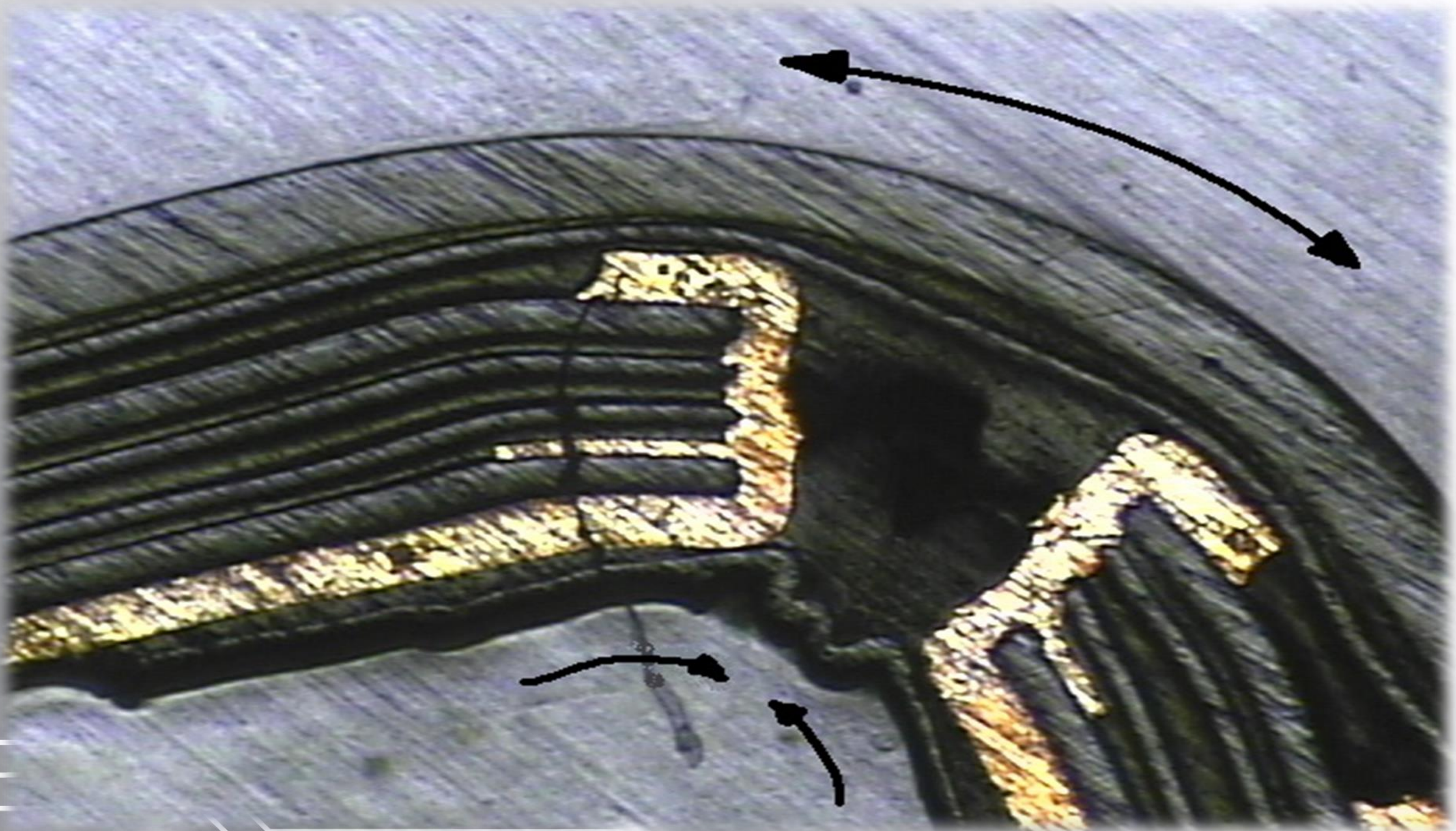
• Bend radius of a flex should be approximately 10 times the material thickness and at least 500um (0.020") away from the plated through hole.

• Incorporate tear stops or reliefs for slits in the circuit. The end of the slit represents a vulnerable point for a tear to start and to propagate.





EFFECT OF BEND ON PLATED THROUGH HOLES



PROPERTIES AFFECTING LOCATION OF NEUTRAL BEND AXIS

- The neutral bend axis will shift towards layers with heavy copper conductors or copper planes
- The neutral bend axis will shift towards layers with thick polyimide dielectric (>.003”).
- Heavy copper layers or thick polyimide layers on opposite sides of the neutral bend axis tend to cancel each other
- A “balanced construction” will tend to keep the neutral axis in the center of the stack

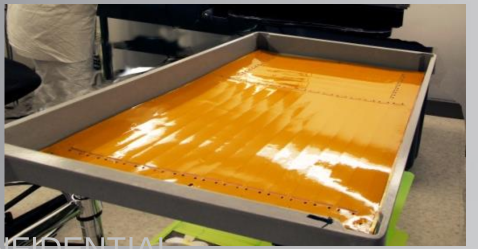
FLEX CIRCUIT MECHANICAL CONSIDERATIONS

- IPC-2223 has a reasonable section on bend radius calculations
- Grain direction in the copper is important for tight bend radii
- The stiffness of a flex is primarily driven by the copper weight but kapton thickness is also critical.
- A .008 (200 u) thick flex will be about 40% more flexible than a .010 (250 u) thick flex

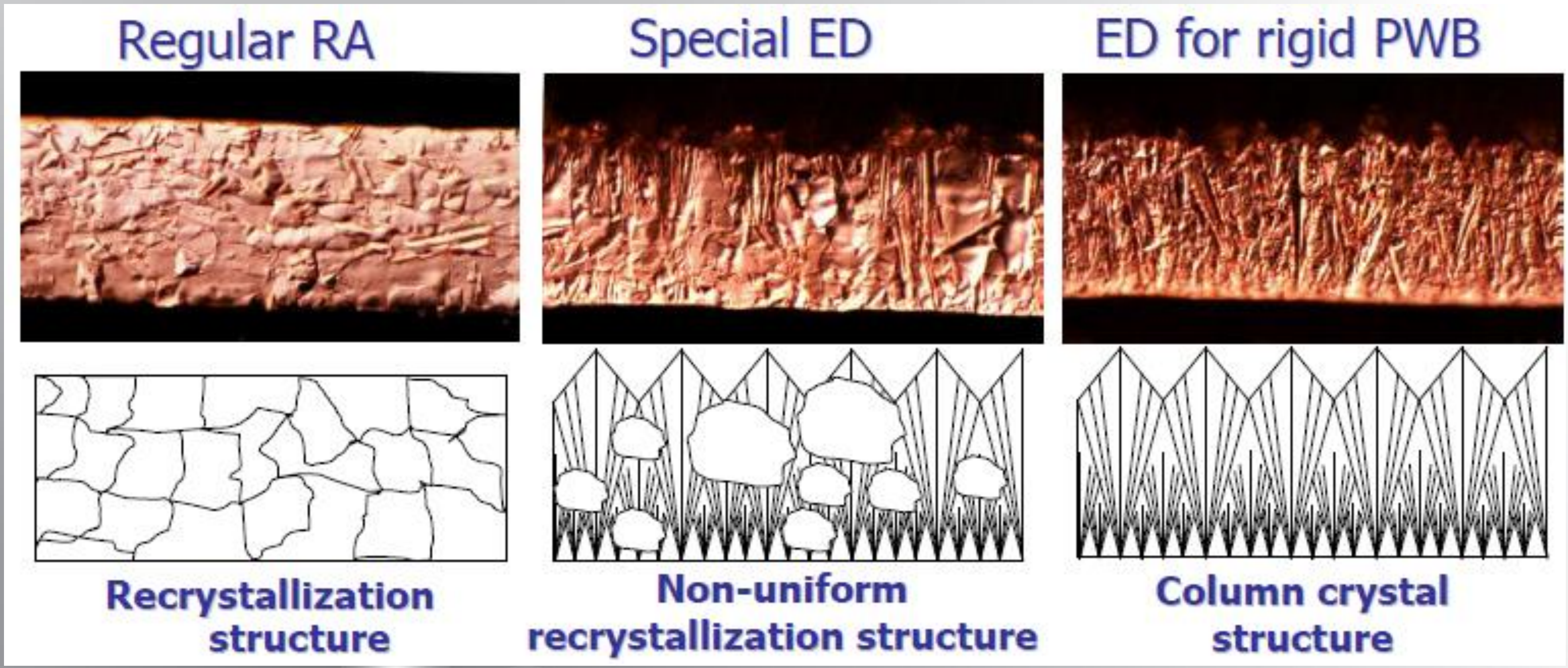
FLEX CIRCUIT MATERIALS

MATERIALS

- Most flex circuits are made using polyimide, adhesive, and copper
- Metal is bonded to the polyimide and then etched to create the features
- The most common metal is copper but just about any metal can be bonded onto polyimide
- Nickel alloys can be bonded and designed as *heaters*

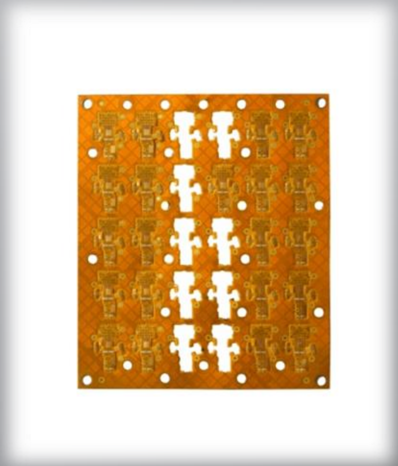


Advance Materials: Copper Foil Trends

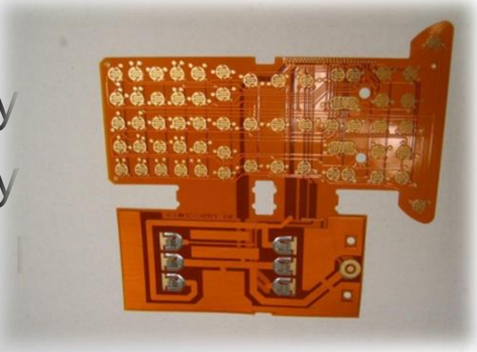


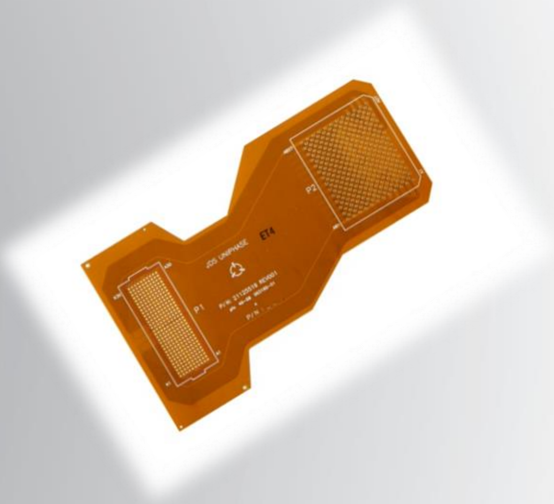
*Comparison of crystal structure – cross-sectional observation of cu. foil crystals after annealed

DESIGN GUIDE
COPPER WEIGHTS



Inches	Metric
.0002	5 micron ED only
.00035	9 micron ED only
.00047	12 micron ED or
.0007	18 micron – RA
.0014	35 micron – RA
.0028	70 micron – RA
.0042	105 micron
.0056	140 micron
.0098	248 micron
.0168	427 micron
.0224	569 micron





MATERIAL POLYIMIDE

- Polyimide thicknesses range from .0005 (12.5 u) to .012 (300 u) for copper clad products
- Coverlay thicknesses range from .0005 (12.5 u) to .005 (125 u)
- Polyimide coverlays come in four main versions
  - Acrylic
  - Epoxy
  - Phenolic Butryal
  - Polyimide
- In North America acrylic and polyimide are most common
- Polyimide version is good for high temperature (170 C continuous) or extreme chemical environments

DESIGN GUIDE
ADHESIVES

- Adhesives range from .0005 (12.5 u) to .004 (100 u)
- In any flex circuit build, the adhesive systems are the weakest link
  - Adhesive Tg's (glass transition temperature) are lower than the Tg of the copper or polyimide
  - The CTE's (Coefficient of Thermal Expansion) are higher
- Adhesives are harder to drill as the adhesive tends to wraps around the drill bits
- Drilling a flex is much more difficult than drilling a rigid board

Flexible solder masks come in two basic types:

- Screen Floodable or spray on much like conventional rigid board masks
- Dry film like Dupont's *Vacrel*
  - only good for tin-lead finish
- **Advantage:** cost and the ability to resolve small openings
- **Disadvantages:**
  - will crack in extreme bends
  - not as good an insulator as polyimide
  - maximum copper weight that can be covered is 35 micron

Advance Materials: Technology Trend

Black Coverlay

- Introduce by Dupont for Apple flex requirement for flex IP design

Conductive Adhesive

- Requirement for conducting metal stiffener to the flex to improve electrical discharge of device
- Mostly Japanese materials (Tatsuta, Asahi)

Transparent Coverlay

- Increasing trend of transparent devices with see-thru requirement

White Coverlay

- Application for LED flex for increasing reflection firing

LCP (Liquid Crystal Polymer)

- High Frequency application >25GHZ (Similar to TK material from Dupont)

Types of Stiffeners (Other than standard FR4)

- Metal, Aluminum, Ceramic

SHIELDING FLEX

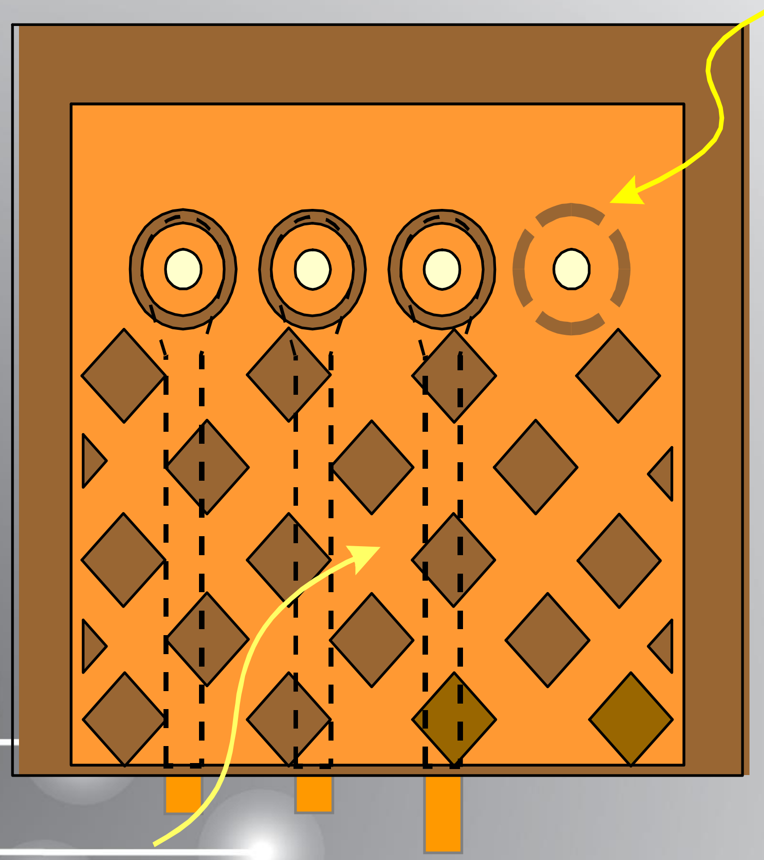
Shields are designed and used for EMI and ESD considerations as well as impedance requirements

- PFC recommends 3 types of shielding
  - Copper clad
  - Copper cross hatch
  - Silver paste/epoxy
- Copper has the best shielding characteristics
- Silver allows more flexibility and at a slightly lower cost

Cross Hatching of Ground Layer

The method improves flexibility but can impact electromagnetic compatibility

Cross hatching of ground plane

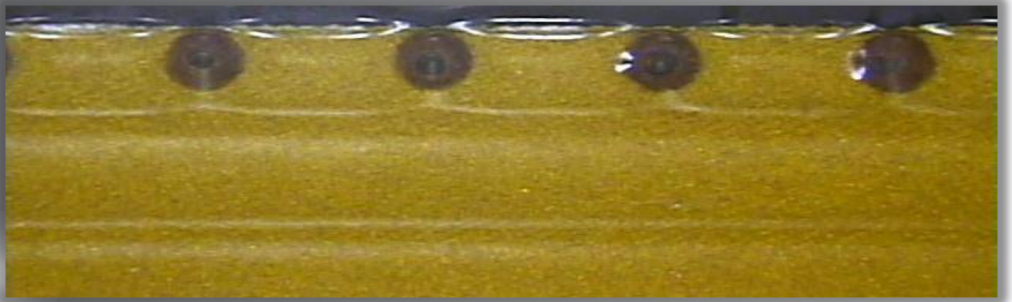


Ground connection with thermal relief

SHIELDING
SILVER SHIELD

Openings are made in the coverlay on top of the copper and Silver is screened in the pattern desired

- The silver goes down the coverlay openings and makes electrical contact to the appropriate copper feature below
- A coverlay needs to be laminated on top of the silver in most areas for proper performance
- Silver shields should be at least .025 (625 u) away from the outline

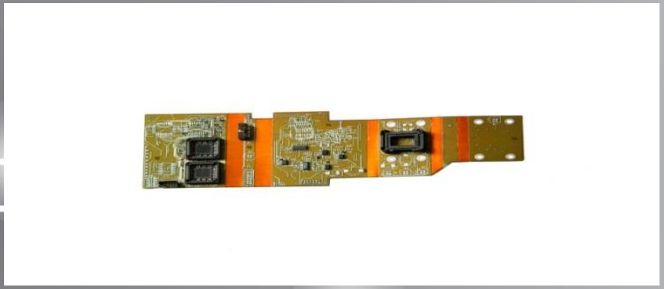


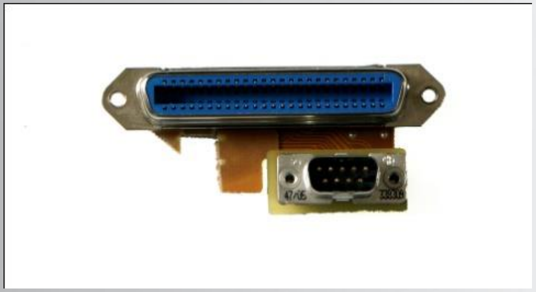
MATERIALS
THERMAL-HEAT DISSIPATION

Various materials are available to attach to a flex circuit to dissipate heat. Some which we have used with success are:

1. 3m heat transfer tapes
2. Thermacore graphite

It is best to contact the factory for design help.





FLEX CIRCUIT IMPEDANCE CONTROL

PFC FLEXIBLE CIRCUITS LIMITED

- Dk (Dielectric Constant) of flex materials ranges from 3.0 to 3.4 depending on the speed of the circuit. New materials have a Dk of 2.6
- Typical impedances are 50-75 ohm end to end and/or 100-110 ohm differential
  - Circuits can be strip line or surface micro strip
  - Proper choice of materials allows +/- 6% on impedance
- Flex material thicknesses are better controlled than rigid board thicknesses. Dupont AP 8525 which has a .002 thick kapton core will range from .00197 to .0022 in thickness

Controlled Impedance: What do I need to know?

THE BASICS:

Electrical Impedance: A measure of opposition to time-varying electric current in an electric circuit.

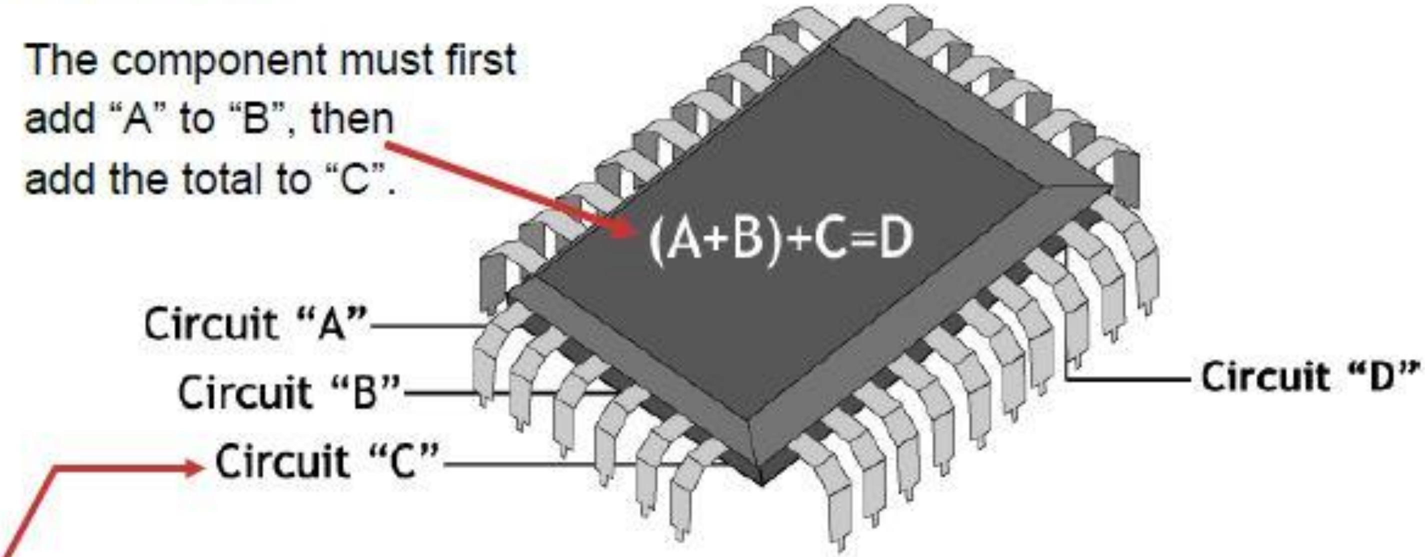
Simplified: To slow down an electrical circuit.

Why: As flex design and components become more complicated, smaller and faster, it becomes necessary to slow certain circuits down, allowing specific functions of components to perform before others.

The increase in processor clock speed and component switching speed on modern flex circuit means that the interconnecting paths (traces) can no longer be regarded as simple conductors.

A Simple Design Example:

Electrical Impedance: A measure of opposition to time-varying electric current in an electric circuit.



The Problem: "A", "B" and "C" signals all reach the component at the same time.

The Solution: Apply Impedance to Circuit "C" to slow the signal enough for the component to first calculate ("A"+B").

Results and Impedance Relationships:

- Impedance (Zo) is measured in Ohms (Ω), and should not be confused with resistance. You cannot measure impedance with an ohmmeter.
- Target impedance is usually between 25Ω, and 125Ω.
- Typical result of a 75Ω controlled impedance on a 0.007” trace is a signal slowdown of approximately 166ps/in

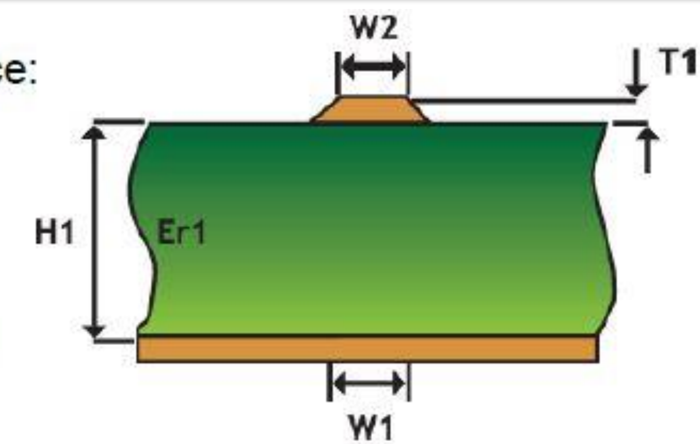
Design Factors affecting Impedance

- Conductor width increase, impedance decreases
- Copper thickness (weight) increase, impedance decreases
- Laminate thickness increase, impedance increases
- Dielectric constant increase, impedance decreases
- Inductance increase, impedance increases
- Capacitance increase, impedance decreases

3 Basic Factors to Impedance:

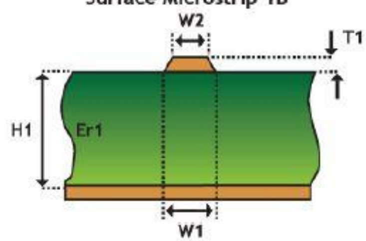
Physical characteristics of impedance trace:

- Height of Trace (T1)
- Width of Trace at Top (W2)
- Width of Trace at Bottom (W1)
- Distance to other Copper Features (H1)
- Dielectric Constant of PCB Material (Er1)

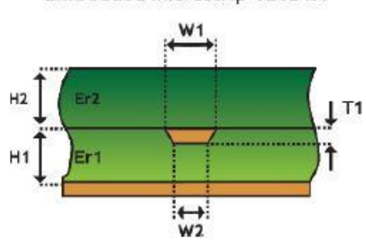


Simple Impedance Models:

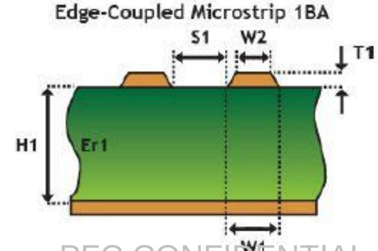
Surface Microstrip 1B



Embedded Microstrip 1E1B1A

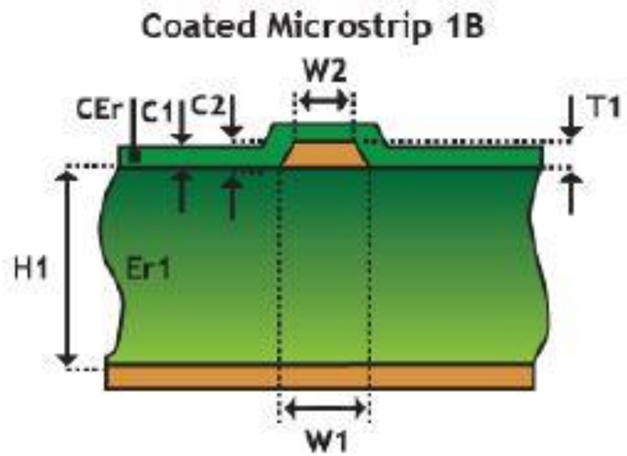


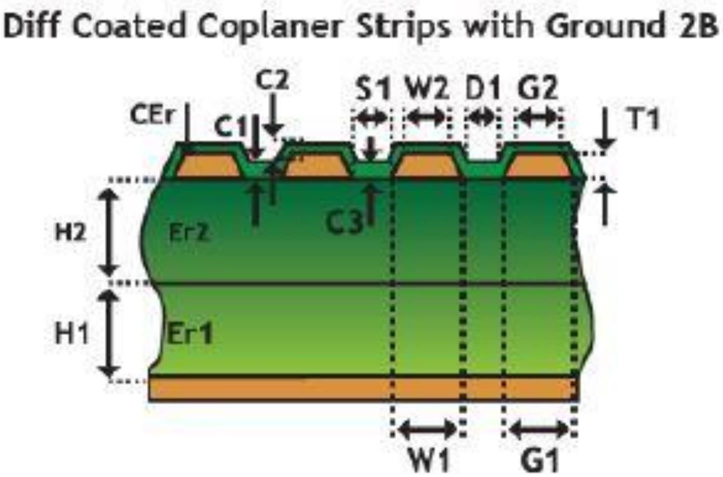
Edge-Coupled Microstrip 1BA



Complicated Impedance Models:

Some PCB's may have multiple Impedance requirements!
More Impedance requirements mean more impedance coupons, which can decrease the amount of usable panel space for PCB's.





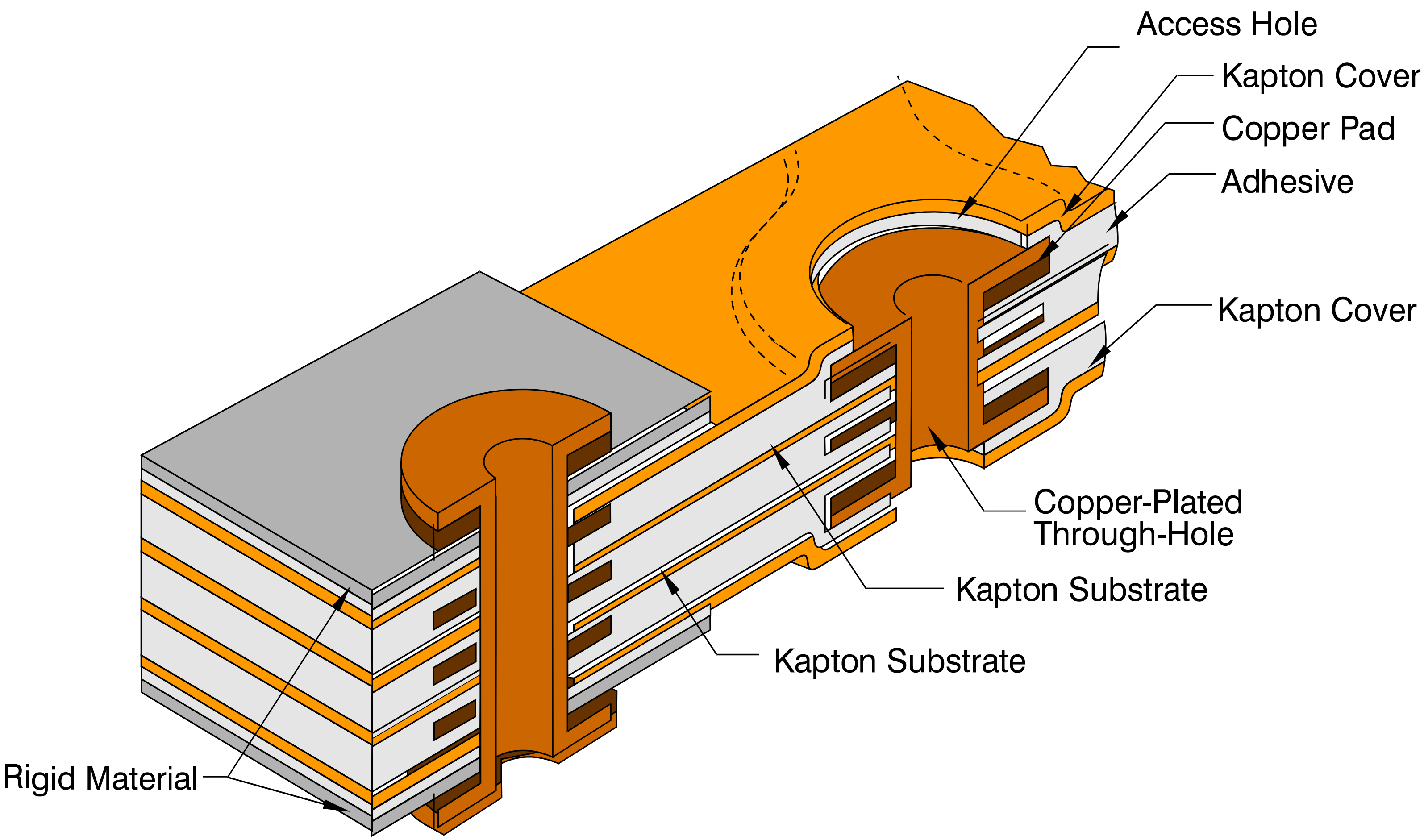
Testing/Modeling:

- Impedance Modeling Software: Polar

*Failed Impedance coupons do not mean failed boards!

RIGID FLEX

RIGID FLEX
IPC 6013, TYPE 4



RIGID FLEX OVERVIEW

A rigid flex consists of:

- layer(s)of flex circuits laminated to/between rigid material (fr-4/bt/gi)
- and electrically connected together through vias

It is preferred to have:

- An equal number of layers of rigid on either side of the flex-balanced design
- Although many rigid flex designs are unbalanced

Rigid flex can be made with:

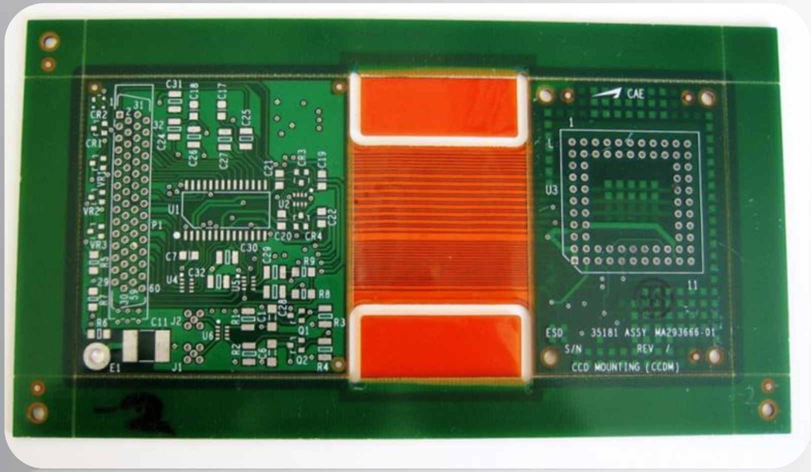
- all the flex layers extending through the rigid areas
- Or, with the covercoat section ending slightly inside (1.27-1.9mm). This is called a ***Bikini Flex***

RIGID FLEX PANELIZATION

Scoring is the common approach to separating unused rigid sections

Stencil registration is not as easy as a rigid board

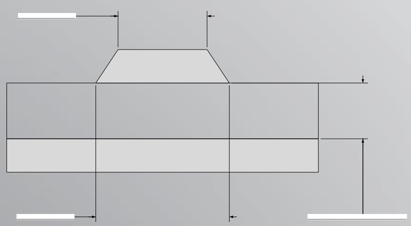
- Scoring is only an option with a bikini build (with limitations)
- You cannot score through kapton
- Normal break-off tab rules apply



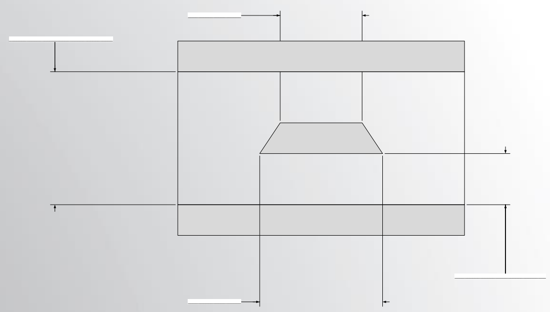
RIGID FLEX
CONTROLLED IMPEDANCE

dk of flex materials ranges from 3.0 to 3.4 depending on the speed of the circuit. New materials have a dk of 2.6 50 ohm end to end and > 100 ohm differential pair are common in either a strip line or micro strip configuration

MICROSTRIP



OFFSET STRIPLINE



- +/-10% typical tolerance
- +/- 5- 6 % optional
- 75 ohm end to end is common
- 130 ohm differential pair occasionally required
- Proper choice of materials allows +/- 5% to 6% impedance tolerance
- Flex material thicknesses are better controlled than rigid board thicknesses. Dupont ap8525 which has a .002 thick kapton core will range from .00197 to .0022 in thickness

RIGID FLEX
CORE MATERIAL

The most common core material used in the flex portion of a rigid flex is an adhesiveless copper clad

There are three main types of adhesiveless materials:

- Cast – copper is fused into the kapton under heat and pressure
- Sputtered – a nickel or chromium tie coat is sputtered onto the polyimide then a thin layer of copper is sputtered on top
- Polyimide is poured onto the copper as a liquid

RIGID FLEX
CORE MATERIAL

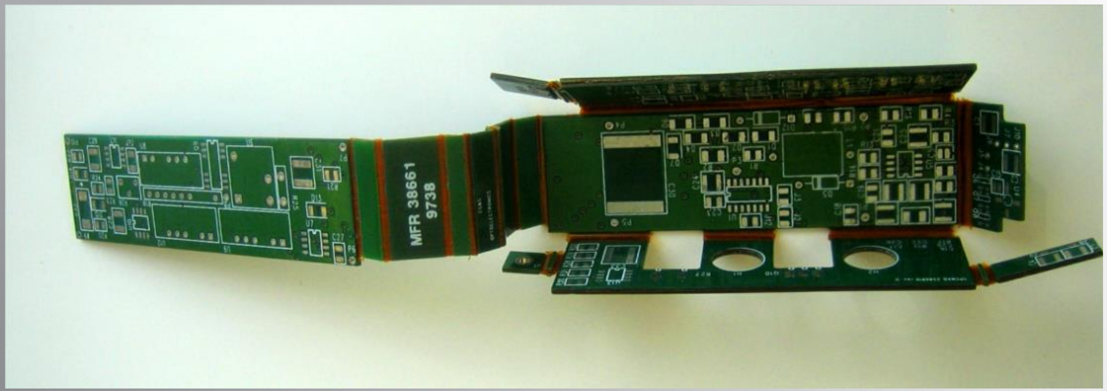
PFC recommends Cast material from Dupont or Nippon as the best material (highest peel strengths, best dimensional stability) but it is also the most expensive

The cast material has the best thermal match to the FR-4 materials:

- Materials with chrome tie coats are the next best
- Nickel tie coat products do not have good peel strength
- Recently a new generation of adhesiveless LCP materials are being offered but there are concerns in high layer count applications

RIGID FLEX COVERCOATS

- In a rigid flex application cover coats are bonded over the copper layers. These are essentially the solder mask layers
- A cover coat consists of a layer of polyimide (kapton) with a layer of adhesive (like a pre-preg) to bond the cover layer onto the copper



RIGID FLEX ADHESIVES

The two most common adhesives are modified acrylics or all Polyimide

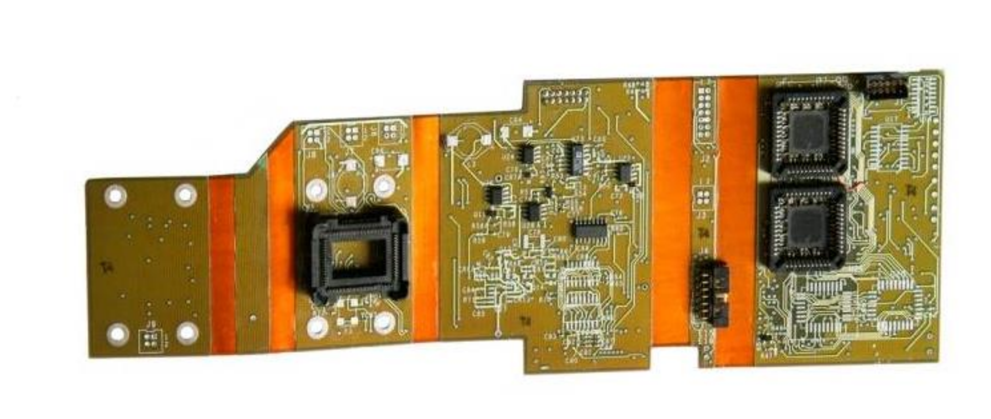
- most prevalent in North America

- The pre-preg used to bond the flex to the rigid is known as a low flow or no flow. (Limited Vendors)

  - It comes in the standard builds such as 1080 and 2116
  - A low flow is used so it does not flow onto the flex rigid interface

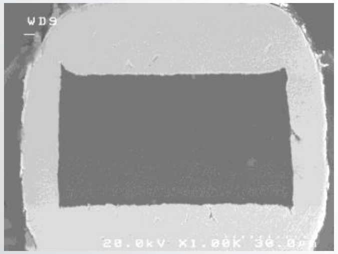
RIGID FLEX
CTE'S

- CTE FOR COPPER – 18 ppm/ deg. C
- CTE FOR KAPTON – 100 ppm/ deg. C
- CTE FOR ACRYLIC ADHESIVE –(<tg) -
  100 ppm/ deg. C (x,y,z)
- CTE FOR ACRYLIC ADHESIVE – (>tg) – 400 pp
- m/ deg. C (x,y,z)



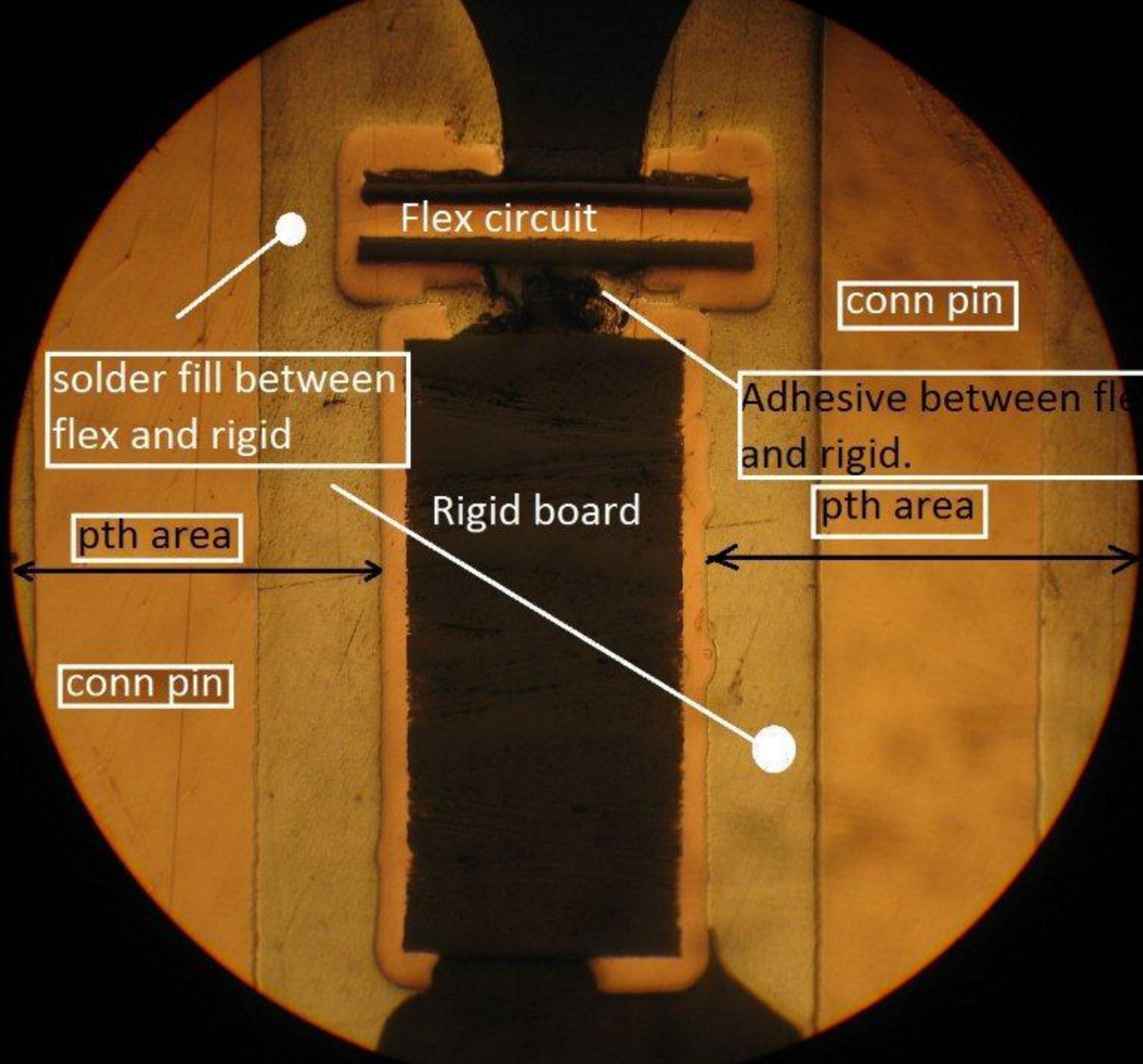
RIGID FLEX DESIGN RULES

- All vias should be at least 1.0 mm away from the flex/rigid interface.
  - 1.27 mm is the Norm



- Routing tolerance - .178 mm is normal - .0508 mm is achievable

- Bending of the flex at the flex/rigid interface is not allowed. Often a bead of epoxy strain relief is placed at this interface to force a radius- enabling a graduated bend


Flex circuit
conn pin
solder fill between
flex and rigid
Adhesive between fle
and rigid.
Rigid board
pth area
pth area
conn pin

- Cleaning
  - PFC recommends plasma treatment of the hole walls as a desmear operation or an etch back depending on the build.
- Assembly
  - It is mandatory to bake rigid flex prior to assembly to remove moisture. Depending on the design, this can be from 4-12 hours @110c. This can be done at the vendor then heat seal the parts with desiccants

FLEX CIRCUIT TOOLING

TOOLING SYSTEMS

Manufacturing a flex circuit involves tooling: artwork, drilling, die cutting, routing, stiffeners etc.

Given the instability of the material do not expect registration to be as easy as on a rigid pwb.

TOOLING SYSTEMS
OUTLINE DIES

- Steel rule dies - +/- .010 (.25 u).
  - Life 10000 hits (cost $700).
  - Minimum radii .030 (.762 u).
- Chemically etched dies +/- .003 (.0762 u).
  - Life 40000 hits (cost $1200).
  - Minimum radii .020 (.500 u).
- Male/female dies +/- .002 (.050 u).
  - Life 100,000 hits (can be re-sharpened) (cost $5000)
  - Minimum radii .010 (.250 u).

TOOLING SYSTEMS
OPTICAL SYSTEMS

- Optical punching
  - allows the punching of tooling holes for die cutting registered from artwork features for precision cutting of a zif feature for example.
- Optical drilling/routing enables programming to "best fit" drilling vias in a high layer count flex or rigid flex.
  - Today's advanced systems allow you to do this on a whole panel or any portion required
  - Optical drilling also allows precise depth drilling for buried and blind vias.
- Optical routing can easily achieve +/- .002 (.050 u) tolerances.

FLEX CIRCUIT FINISHING

DESIGN GUIDELINES
COPPER SURFACE FINSHES

- Hot Air Solder Level
- Electrolytic Plating's:
  - Hard Gold over Nickel (used for contacts)
  - Soft Gold over Nickel (used for bonding gold wire to the gold layer
- Electroless Platings:
  - Electroless Ni, Immersion gold
  - Tin: Flat, solder-able finish
  - Silver: Flat, solder-able finish
  - OSP: Organic Solder-able Preservative

DESIGN GUIDELINES
INTERCONNECTION

In most cases flex circuits are attached to a rigid board via a connector

As densities continue to increase and costs become a consideration/issue, other systems have appeared.

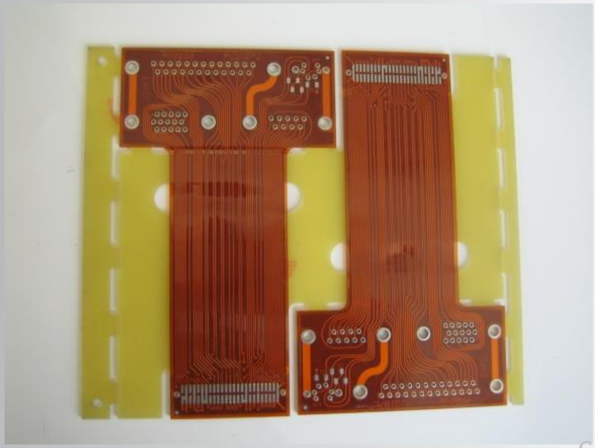
- Hot bar
- Conductive inks
- Direct solder through vias

DESIGN GUIDELINES
RIGIDIZERS/STIFFENERS

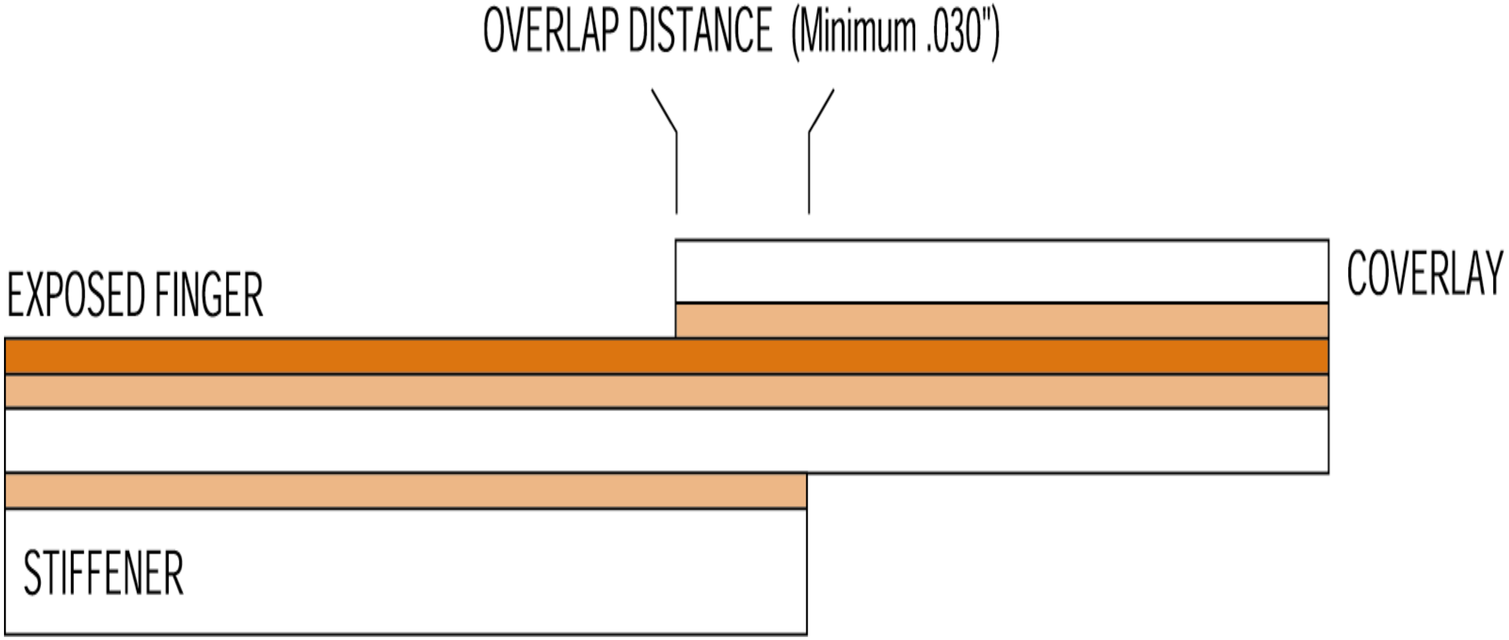
- FR4/G10 Stiffeners:
  - Bonded with PSA or thermal-set adhesive
  - Adds rigidity to connector area or used as carrier panel

- Polyimide Stiffeners:
  - Bonded with PSA or thermal-set adhesive
  - Used to add thickness, or add strength in high wear areas

- Other materials
  - Stainless steel
  - Additional Kapton



STIFFENER AND COVERLAY TERMINATION POINTS SHOULD OVERLAP A MINIMUM OF .030" TO AVOID STRESS POINTS.



FLEX CIRCUIT ASSEMBLY

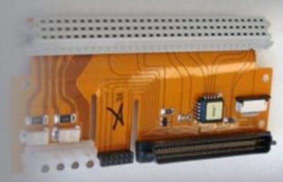
DESIGN GUIDELINES
ASSEMBLY

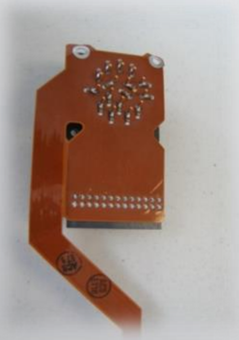
Assembly of flex circuits is not much different than assembly of rigid boards

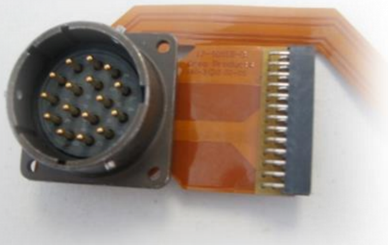
- Care has to be taken to make sure the parts are dry before wave or reflow per IPC-hdbk-001
- Most flex circuits do not have a lot of mass, so the reflow profile needs to be set accordingly
- Flex circuits can often be die cut *after* assembly
- Flex circuits can also be tabbed in and de-panelized much like rigid boards
- Flex circuits cannot be scored
- Flex circuits can be assembled on different levels

DESIGN GUIDELINES
ASSEMBLY-HAND SOLDERING

Typical hand solder temperatures are:
550-580 F (288-305 C) for leaded solder
600-620 F (316-327 C) for lead free solder







The Art of Assembly

Screen Printing

Types of Solder Paste

- Leaded No Clean
- Leaded Water Soluble
- Lead-free Water Soluble
- Lead-free No Clean

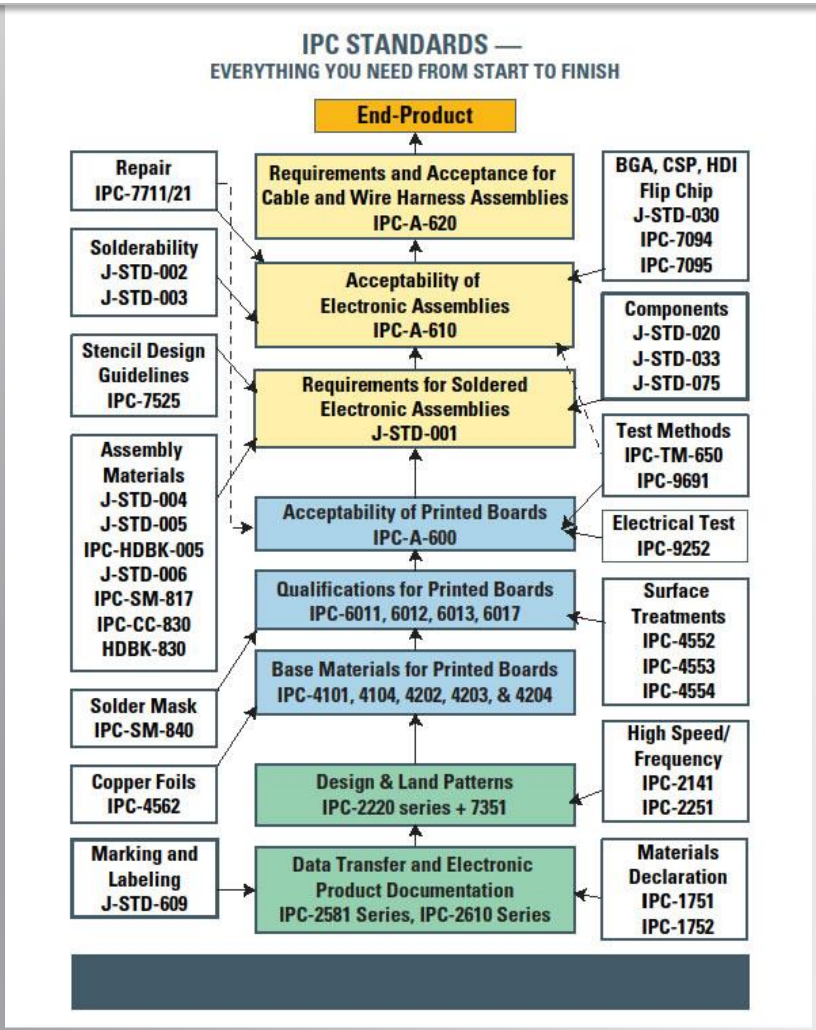
⇒ **Pick & Place** ⇒

Reflow Oven

- Lead-free Temp 245°C +5/-0

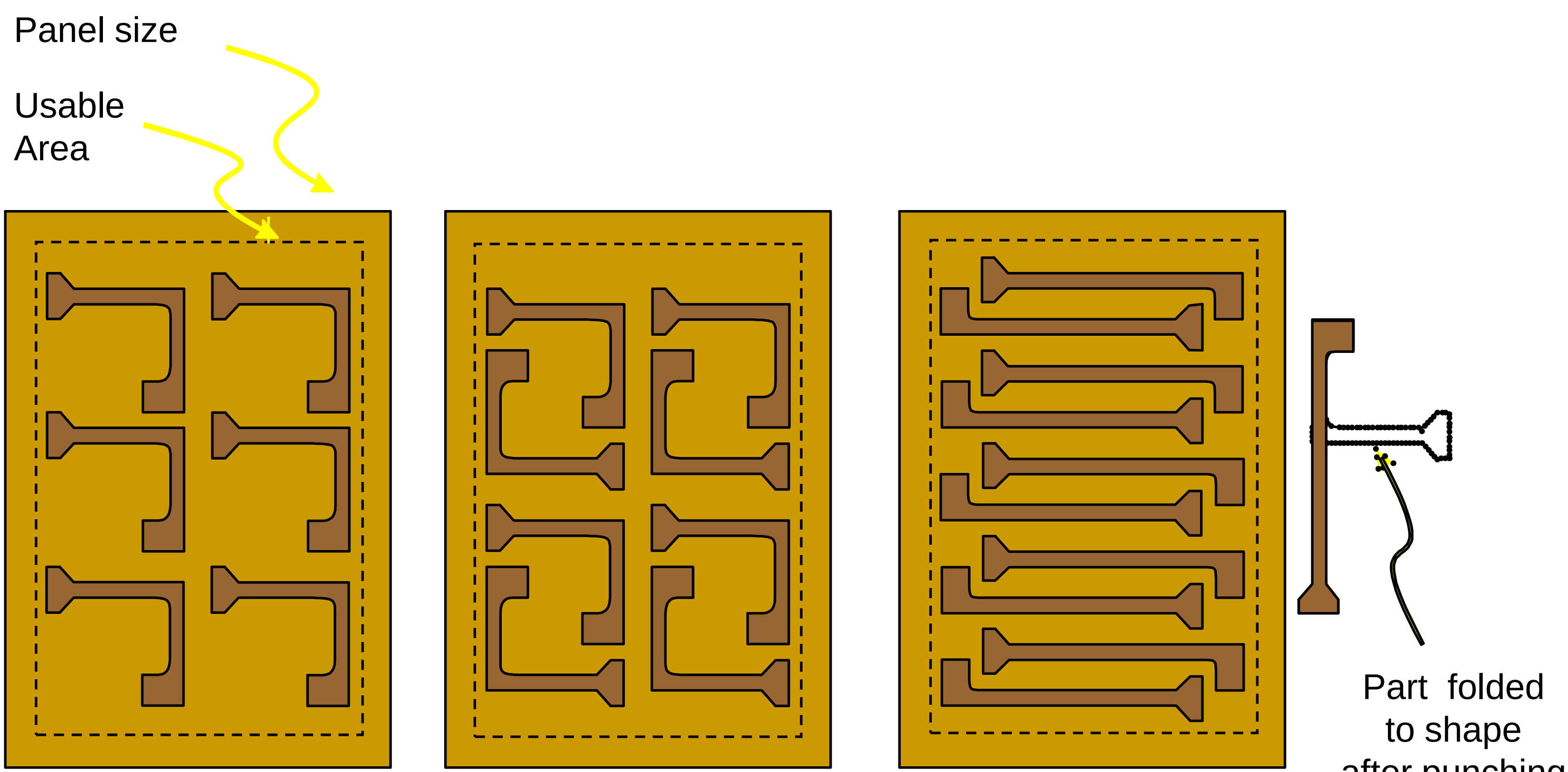
- The Art of Assembly can be summarized into the above diagram.
- SMT is abbreviation for Surface Mount Technology.
- The technology enables assembly within standard capabilities :
    - Fine pitch connector of 0.4mm (16mils)
    - Passive component 0201 (0603)
    - BGA (Ball Grid Array) (0.4mm (16mils)
    - QFP (Quad flat package) / IC / Fine pitch (20 mils)
    - Top & Bottom (double-sided) SMT mounting

Standard and Specification for Flexible Circuits



COST SAVING OPTIONS AND LEAD TIME REDUCTION

DESIGN GUIDELINES
PANELIZATION



No Nesting
Panel Yield 6 parts

Circuits Nested
Panel Yield 8 parts

Optimized Nesting
Panel Yield 10 parts

PFC FLEXIBLE CIRCUITS LIMITED

Design Considerations

Contact manufacturer in the early stages of design

1. Number of copper layers
   - Significant increase in price going from 2 copper layers to 3
2. Trace Width:
   - Keep trace and space width as wide as possible to avoid increased pricing for yield (.005”)
3. Surface Finish
   - Avoid using more than one finish
4. Circuits per panel: Nesting density
5. Feature to Feature Tolerance determines tooling and processing cost

PFC FLEXIBLE CIRCUITS LIMITED

A flex circuit will almost always cost more than a comparable rigid PCB due to the processing issues associated with the very thin materials used in flex circuit manufacturing.

- Flex circuit base materials are much more expensive than base materials for rigid PCBs
- Flex materials are very fragile and dimensionally unstable
  - Lower etching yields
  - Layer to layer registration problems
  - Materials stretch and shrink with temp and humidity and during processing

FLEX CIRCUIT COST DRIVERS

- Choosing Between Rigid-Flex Construction and Rigidized Multi-Layer Construction
  - Are there surface mount components on 1 or both sides of the rigid areas?
  - Will the circuit be cycled often between temperature extremes?

MECHANICAL LAYOUT STEPS

- Determine termination method for each termination area
  - Mating connectors (through hole and/or SMT)
  - ZIF connectors (supported finger pattern on flex)
  - Unsupported finger pattern (etched or brazed)
  - Card Edge
  - Pins
- Determine connector style
  - Straight
  - Right angle

PFC FLEXIBLE CIRCUITS LIMITED
11 CANADIAN RD. UNIT 7
SCARBOROUGH, ONT.
CANADA, M1R5G1
416 750-8433

WWW.PFCFLEX.COM