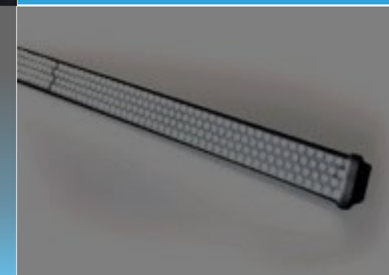
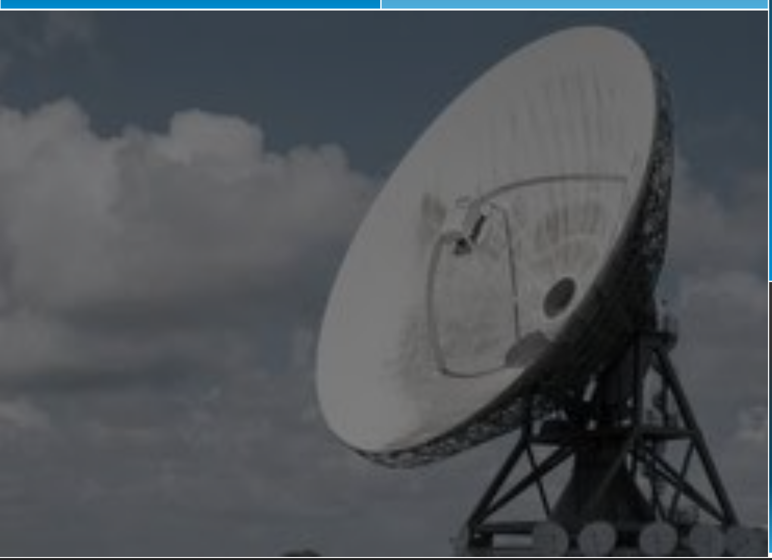


American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits



"Collaborate To Win"

www.asc-i.com

Flex Circuits 101 Guidelines to a successful first-pass build

SPEAKER

David Lackey

VP of Business Development @American Standard Circuits

Been involved with manufacturing PCBs since 1980 working in various shops, most of which had military certifications and utilized higher technology. Having worked in most manufacturing departments throughout the years, developed a strong engineering background and is knowledgeable in most of the industry technologies.

Email: dlackey@asc-i.com

Phone: (815) 403-3415

Presentation Outline

- Flex Material Selections and definitions
- Fabrication Notes
- Data Requirements (Suggestions)
- Cost Factors
- Array Layout
- Benefits of Flexible Circuits

Work With Your PCB Fabricator During The Design Phase

Flex Specific Specifications
Materials

- IPC Slash Callouts (Materials):
  - 4204/11 – Adhesiveless
  - 4204A/1 – Adhesive Coated Dielectric Substrate
  - 4203/1 – Adhesive Coated Coverlayer/Bondply
  - 4203A/18 – Acrylic Adhesive
  - Flex core thickness (This is different for adhesive/adhesivless)
  - Coverlayer/Bondply/Acyrlic Adhesive thickness

General Callouts
  - Which layers are flex vs rigid (Rigid Flex)
  - If multiple layers of flex are together are they bonded together or belted
  - Stiffener types (Polyimide Coverlayer or rigid unclad FR4, polyimide or other)
  - Strain Relief required or not and where is it to be applied
  - Plating flex button plate or entire surface

Fabrication Notes

SAMPLE NOTES FOR A 6 LAYER RIGID-FLEX PCB

1. RIGID-FLEX TO BE FABRICATED USING IPC-6013 CLASS 3 STANDARDS
2. THIS FLEX CIRCUIT TO CONTAIN 4 LAYERS IN RIGID SECTIONS AND 2 LAYERS IN FLEXIBLE SECTION.
3. MATERIALS:
    a. THE RIGID MATERIAL SHALL BE EPOXY GLASS LAMINATE PER IPC-4101/24/26/99/101/126
    b. THE FLEX MATERIAL SHALL BE ADHESIVELESS FLEX COPPER CLAD LAMINATE PER IPC-4204/11
4. COVERLAYER TO BE .001" POLYIMIDE WITH .001' ADHESIVE PER IPC 4203/1
5. COPPER WEIGHT TO BE 1/2 OZ ON ALL LAYERS, PLUS PLATING ON LAYERS 1 AND 6 WITH A MINIMUM OF 1 MIL PLATING IN THE HOLE
6. THIS FLEX CIRCUIT SHOULD BE OF MULTIPLE BEND TYPE.
7. MINIMUM BEND RADII TO BE 6X THICKNESS OF FLEX CIRCUIT.
8. MINIMUM LINE WIDTH TO BE 3 MILS AND MINIMUM LINE SPACE TO BE 5 MILS
9. FINAL FINISH TO BE ENIG, PER IPC-4552
10. SOLDERMASK BOTH SIDES GREEN PER IPC-SM-840
11. OVERALL THICKNESS IN FLEX AREA SHALL BE NO MORE THAN 8 MILS
12. NO SILKSCREEN OR INK ON EITHER SIDE
13. ALL BOARD DIMENSIONS SPECIFIED BY DWG (DIMENSIONS IN GERBERS FOR REFERENCE ONLY).
14. NO X-OUTS ALLOWED IN PANEL

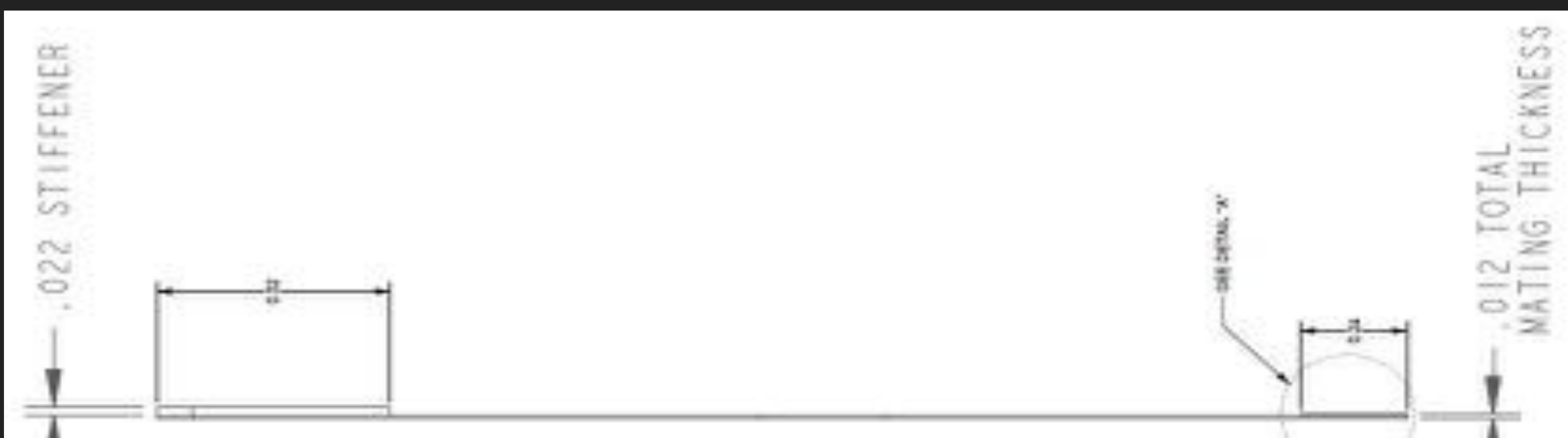
Fabrication Notes

4. MATERIAL:

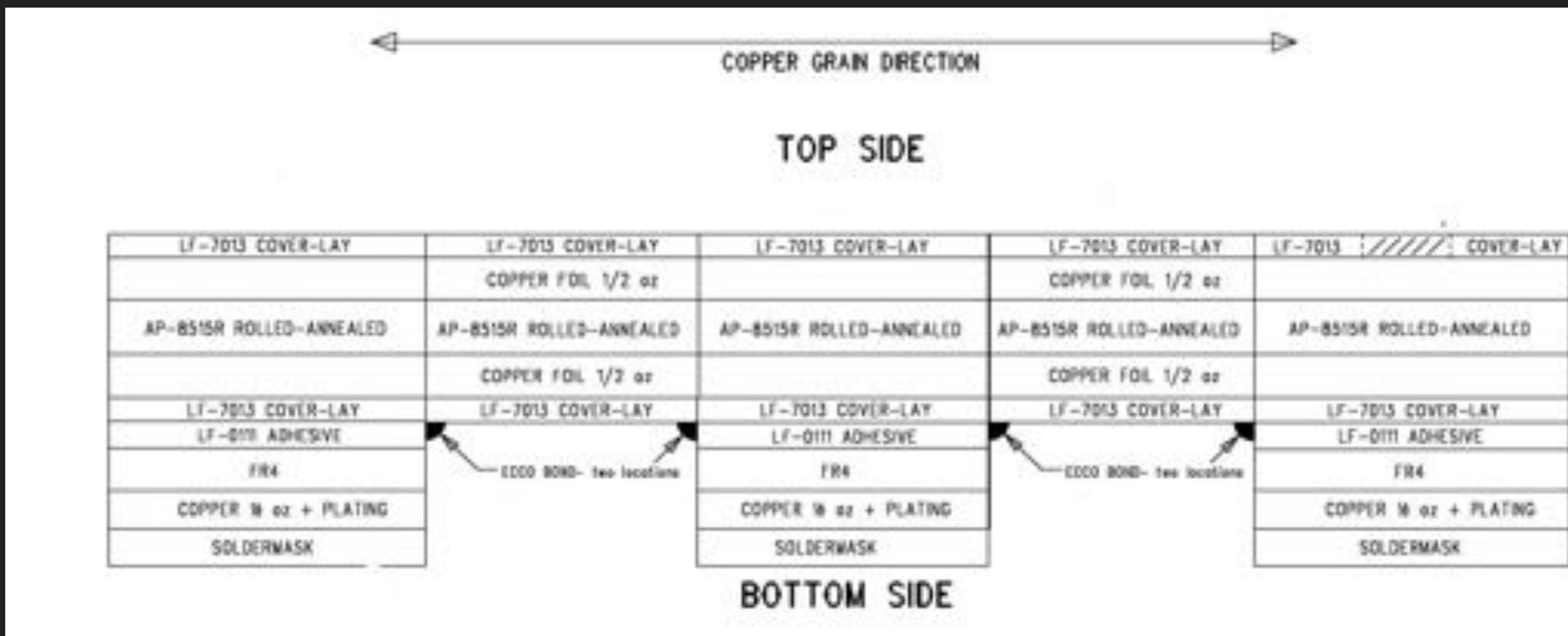
4.A. .008 +/- .002 THICK COPPER CLAD POLYIMIDE FILM PER IPC-4204/11 FOR FLEX CABLE LAYER.

4.B. APPLY POLYIMIDE STIFFENER AS SHOWN, PER IPC-4203/1. STIFFENER THICKNESS SHALL BE SELECTED TO PROVIDE TOTAL THICKNESS DEFINED IN NOTE 5. SIFFENER TO BE BONDED TO SECONDARY SIDE OF FLEX CIRCUIT.

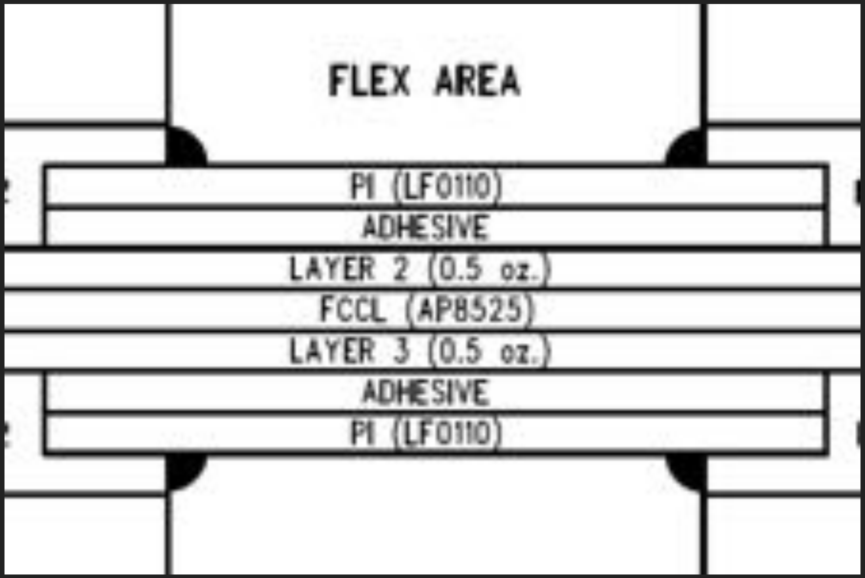
4.C. APPLY .030 (NOMINAL) THICK FR4 STIFFENER AS SHOWN, PER IPC-4101/26. STIFFENER TO BE BONDED TO SECONDARY SIDE OF FLEX CIRCUIT.



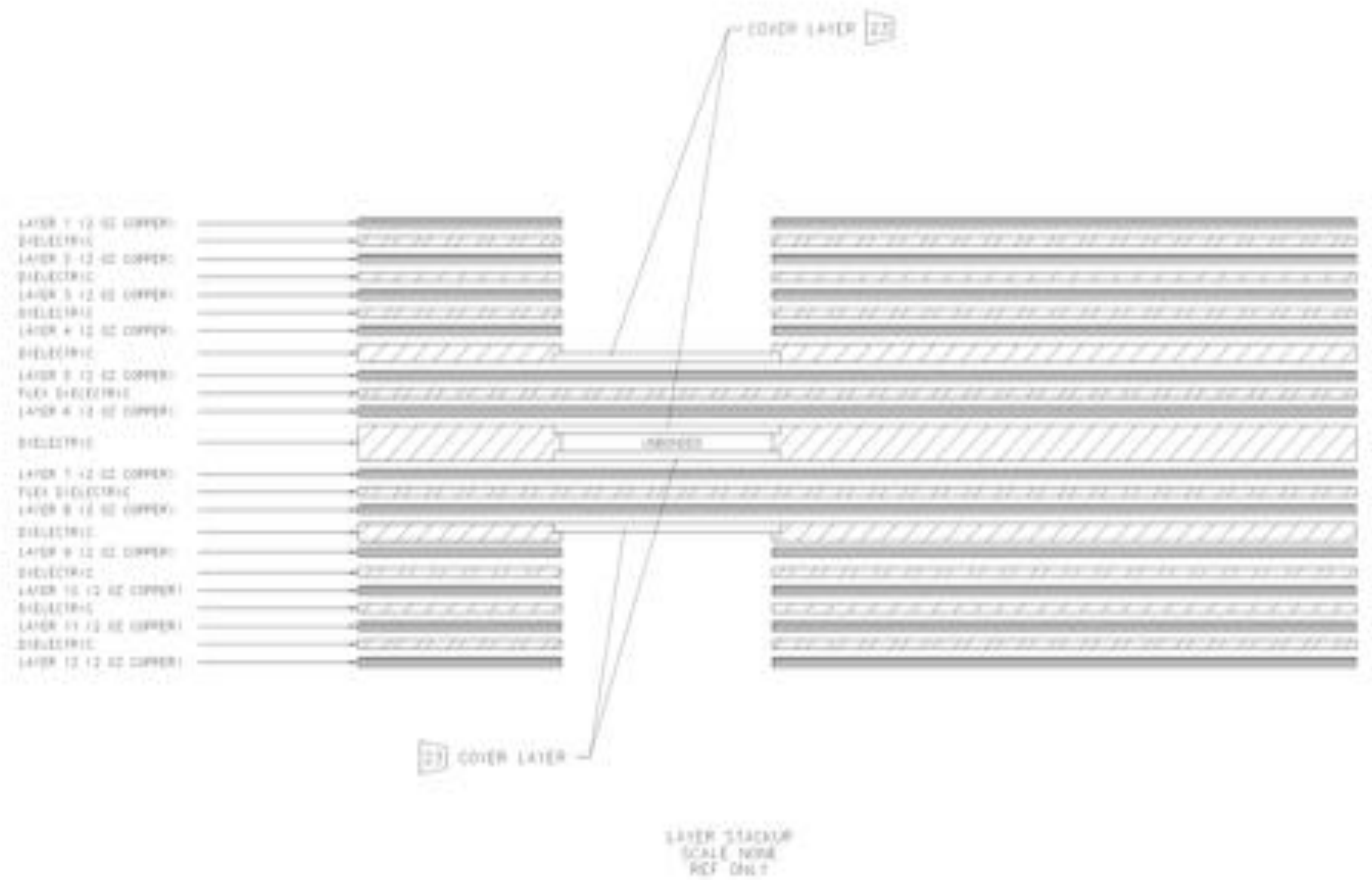
Fabrication Notes



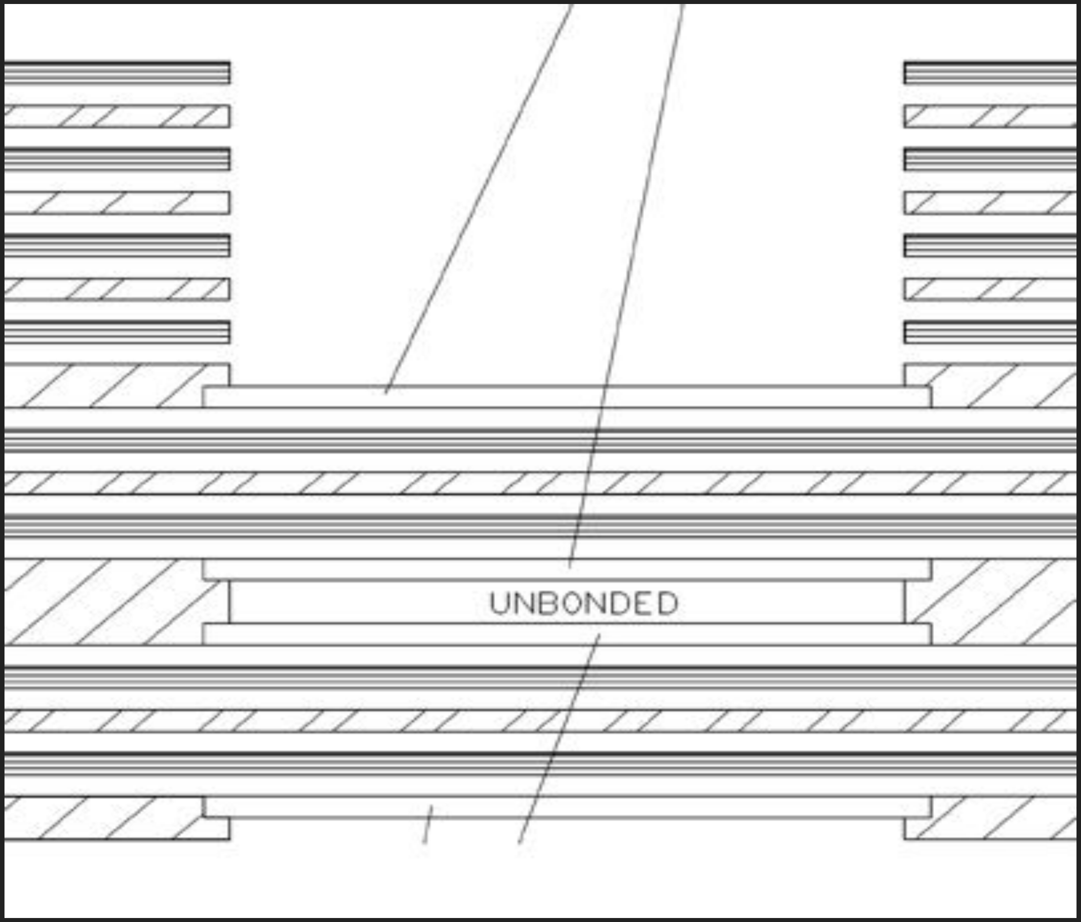
Flex Area Material Callout & Strain Relief



Fabrication Notes



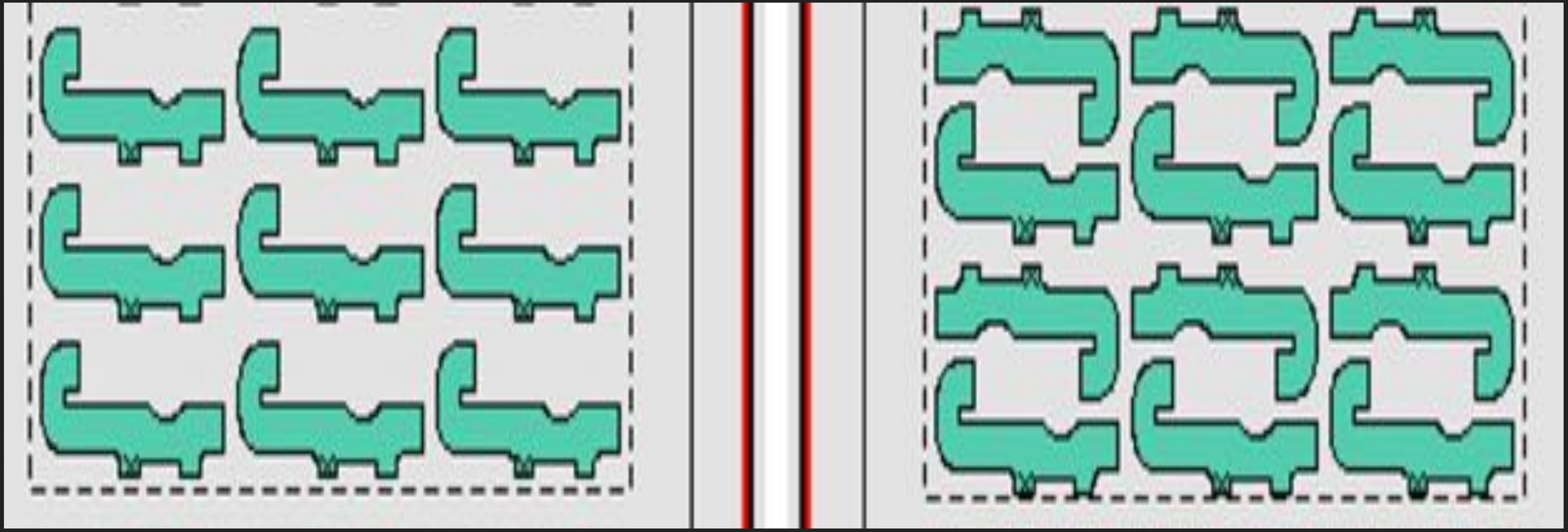
Fabrication Notes



CAD Data Flex
Common Problems

- Array Requirements
  - Either not provided or not enough information provided regarding array layout.
  - Nesting for better panel utilization
  - Scoring callout on a Rigid-Flex Design
  - Plated holes closer than 60 mils at the rigid to flex interface
  - Plated copper surface does not call out for button plate

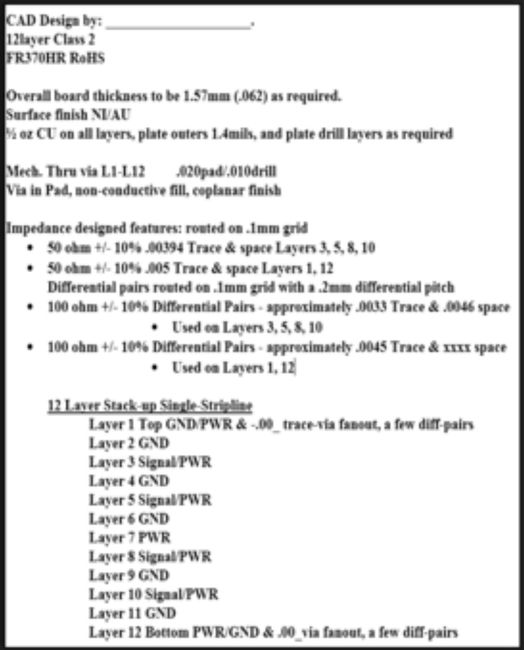
CAD Data Flex
Nested Array (more pcs per panel)



How is Impedance Calculated?

There are many calculators in todays market, its encouraged to think the impedance and routing solvability through, creating your stackup request.

It's *highly encouraged* to work with your fabricator from the beginning. Submit your stackup request - let them provide the actual build stackup.


CAD Design by: ____________________.
12layer Class 2
FR370HR RoHS

Overall board thickness to be 1.57mm (.062) as required.
Surface finish NI/AU
½ oz CU on all layers, plate outers 1.4mils, and plate drill layers as required

Mech. Thru via L1-L12 .020pad/.010drill
Via in Pad, non-conductive fill, coplanar finish

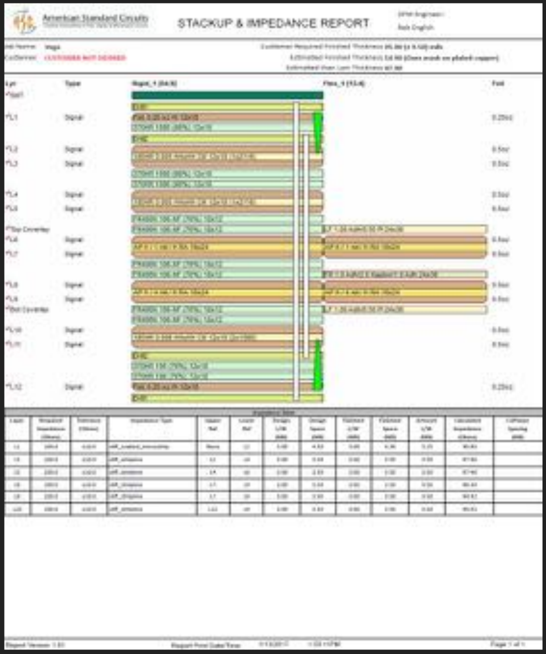
Impedance designed features: routed on .1mm grid

- 50 ohm +/- 10% .00394 Trace & space Layers 3, 5, 8, 10
- 50 ohm +/- 10% .005 Trace & space Layers 1, 12
  Differential pairs routed on .1mm grid with a .2mm differential pitch
- 100 ohm +/- 10% Differential Pairs - approximately .0033 Trace & .0046 space
  - Used on Layers 3, 5, 8, 10
- 100 ohm +/- 10% Differential Pairs - approximately .0045 Trace & xxxx space
  - Used on Layers 1, 12

12 Layer Stack-up Single-Stripline
Layer 1 Top GND/PWR & -.00_ trace-via fanout, a few diff-pairs
Layer 2 GND
Layer 3 Signal/PWR
Layer 4 GND
Layer 5 Signal/PWR
Layer 6 GND
Layer 7 PWR
Layer 8 Signal/PWR
Layer 9 GND
Layer 10 Signal/PWR
Layer 11 GND
Layer 12 Bottom PWR/GND & .00_via fanout, a few diff-pairs

Stackup Request


Collaboration



Fabricators Stackup

Definitions

- **Flex Circuit:**

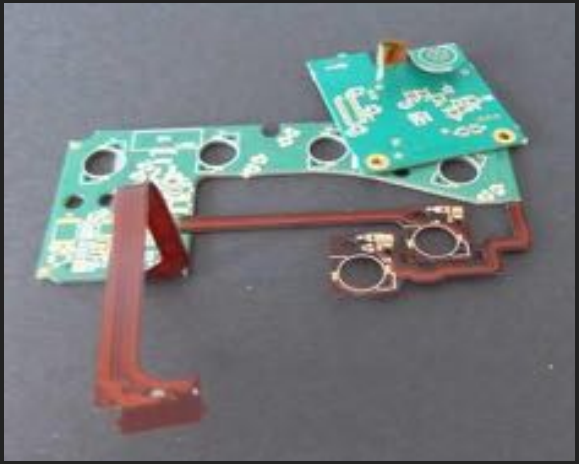
  Made of a flexible polymer film laminated to a thin sheet of copper that is etched to produce a circuit pattern

- **Rigid-Flex Circuit:**

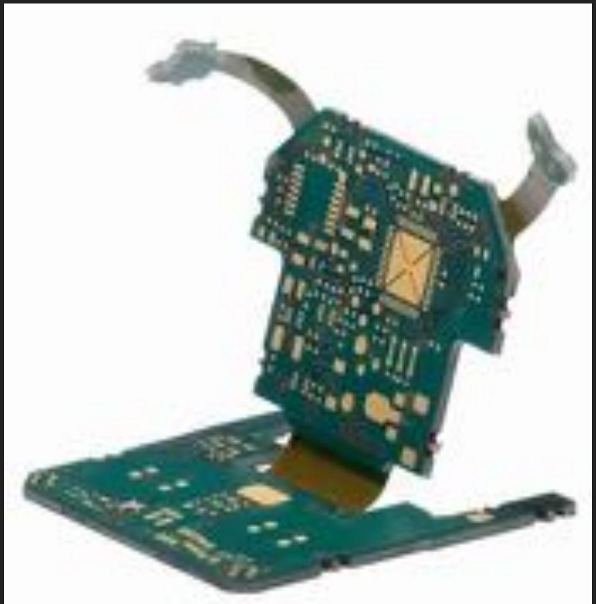
  Combination of a flex circuit with a rigid circuit as one circuit board

Benefits of Flexible **Circuitry**

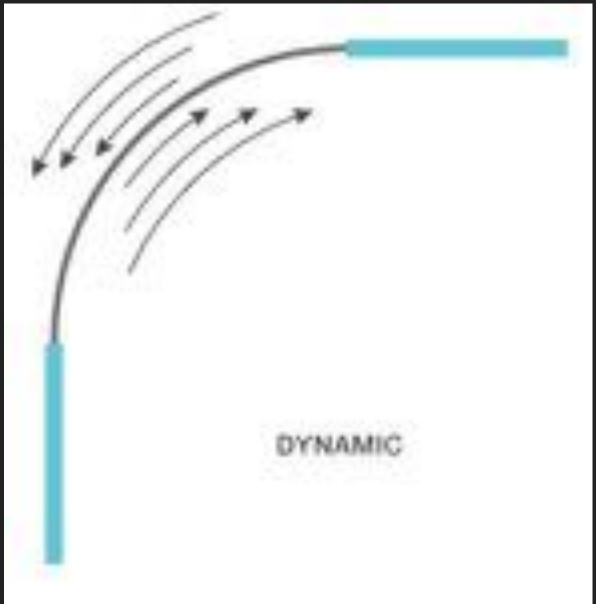
- A solution to a packaging problem
- Replacement for a circuit board and wires
- Aesthetics
- Reduce assembly costs
- Increased Reliability
- Reduce weight and space
- Dynamic flexing
- Thermal management/ High temp applications



Static vs Dynamic



Static Flex (Bend to Install)



Dynamic Flex (Always Moving)

Materials

- Base Materials
- Coverlayers
- Stiffeners
- Shielding

Base Materials

Flex Core with Adhesive

Copper
Adhesive
Polyimide Flex
Adhesive
Copper

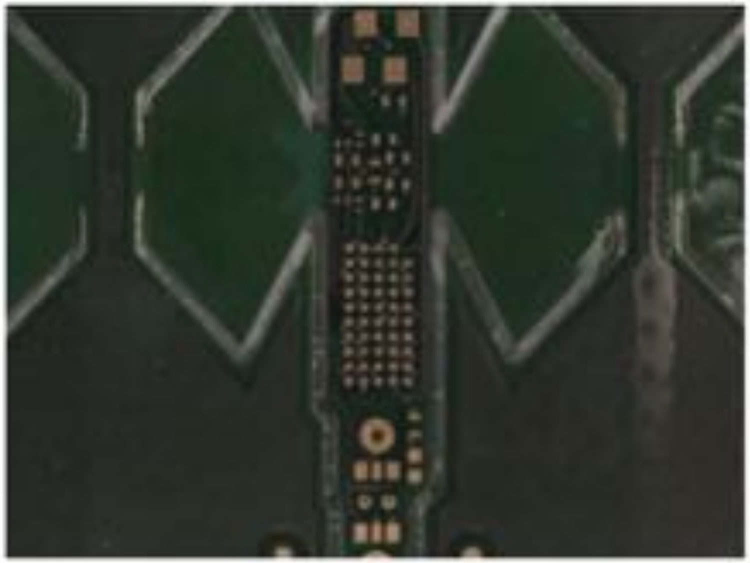
Adhesiveless Flex Core

Copper
Polyimide Flex
Copper

Copper is either RA (Rolled Annealed) or ED (Electrodeposited)

ED usually less than ½ Oz. copper weights

Coverlayer Options



Soldermask Coverlayer

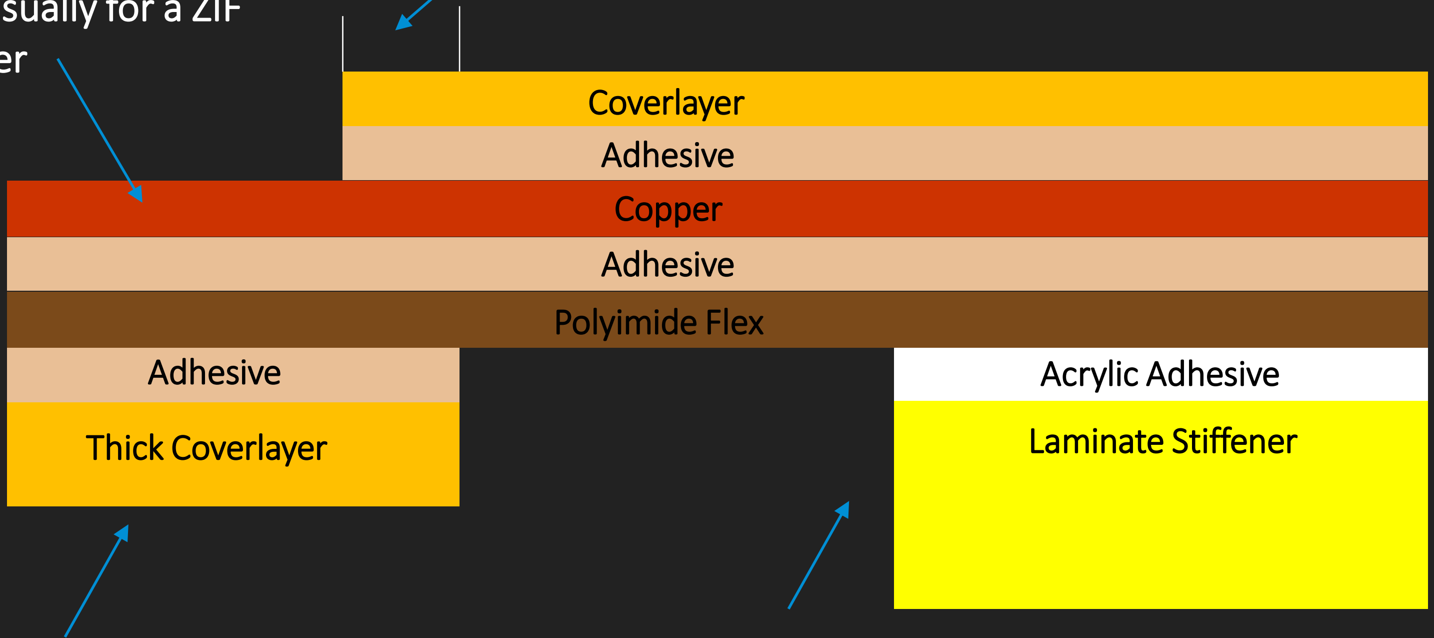
Polyimide Coverlayer

- Use Flexible Soldermasks
- Polyimide Coverlayer – will need to specify poyimide film thickness and adhesive thickness

Stiffener

Coverlayer and
Stiffener Overlap

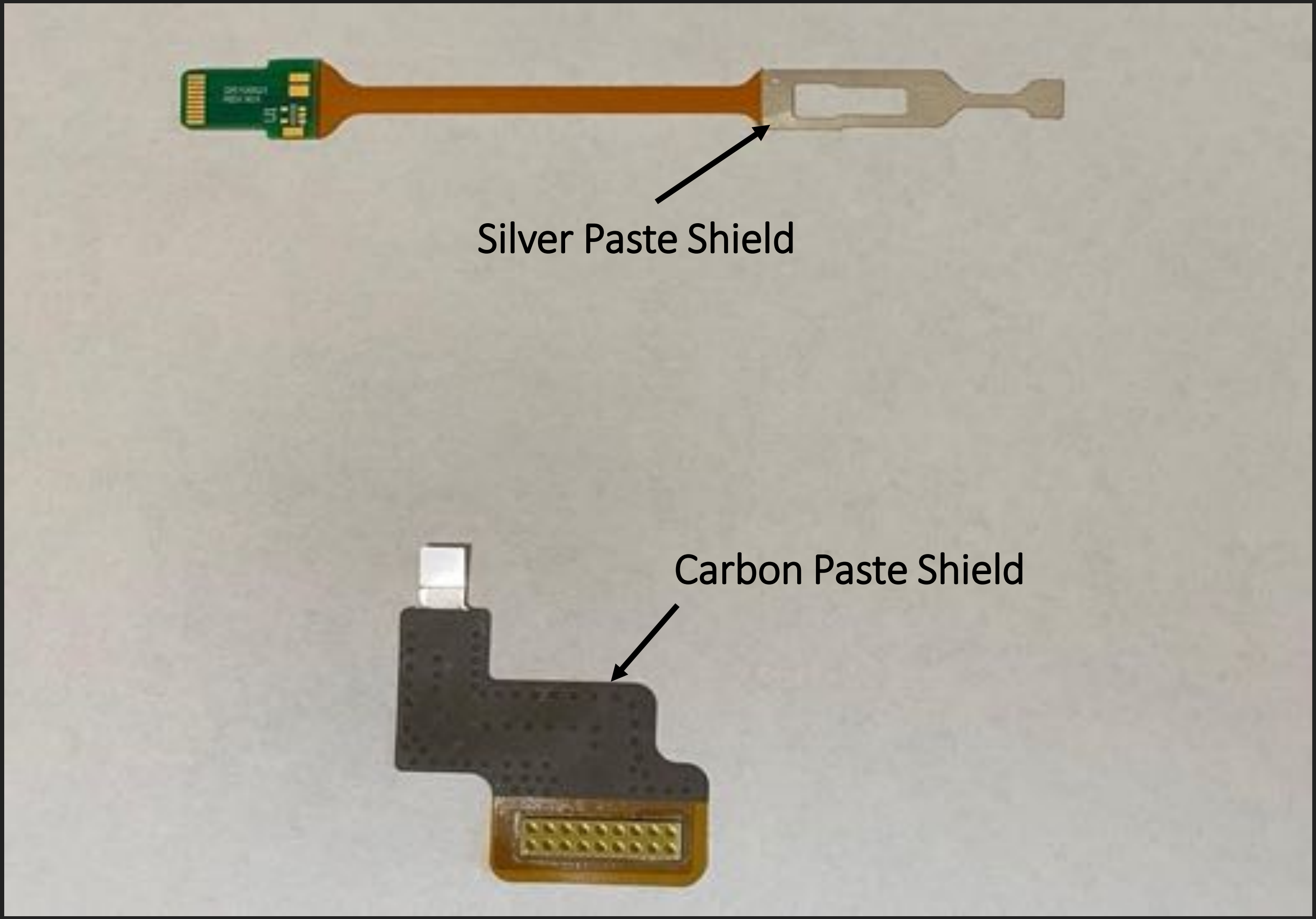
Exposed contact
fingers usually for a ZIF
connecter



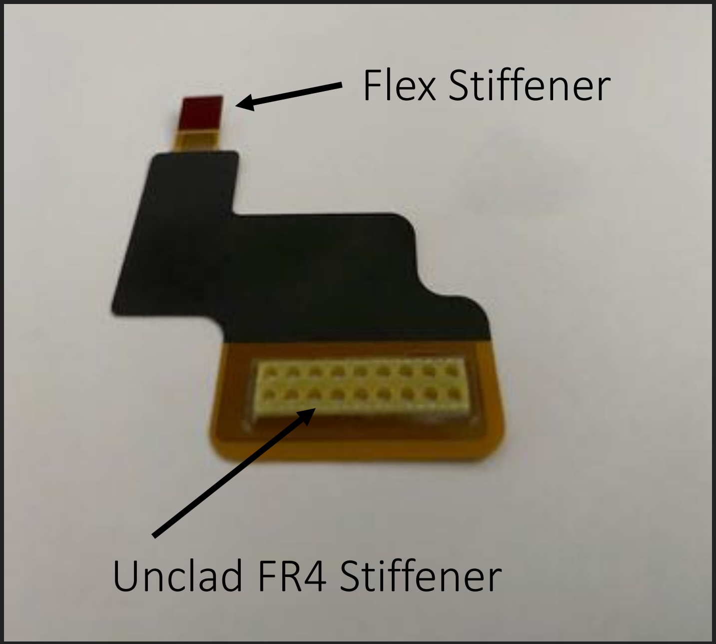
Flex stiffener used
primarily for connector
insertion usually
coverlayer material

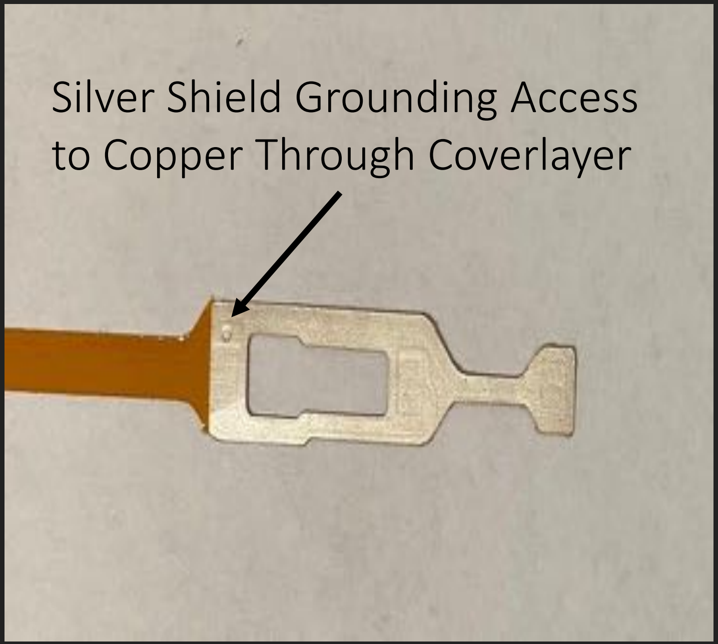
Laminate stiffener
usually for upper
component assembly
usually unclad FR-4 or
Polyimide laminate

Shielding



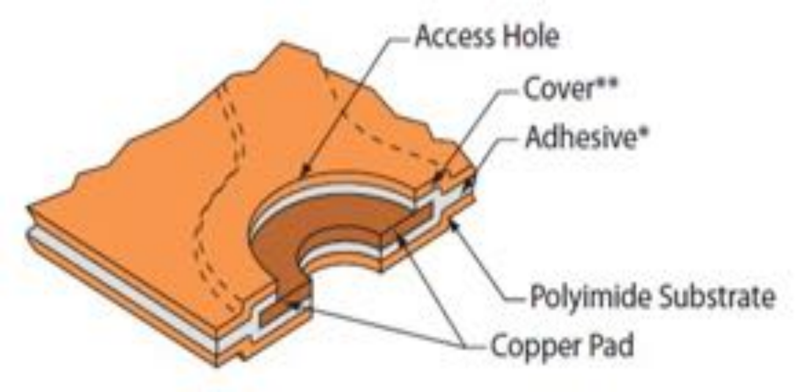
Shielding & Stiffeners



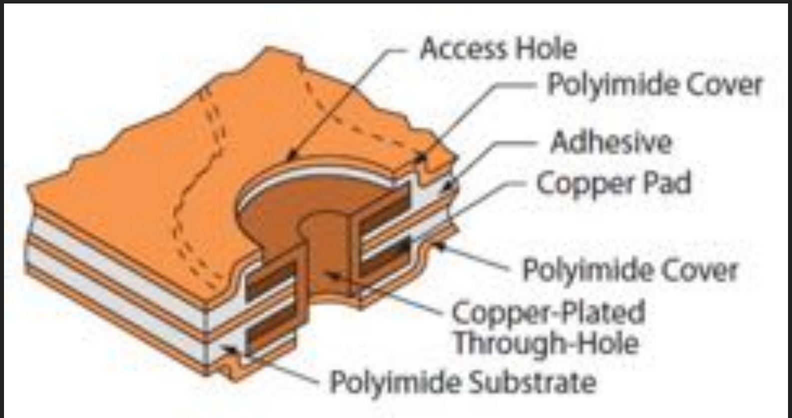


Types of Flex Circuits

- Single-layer (Type 1)

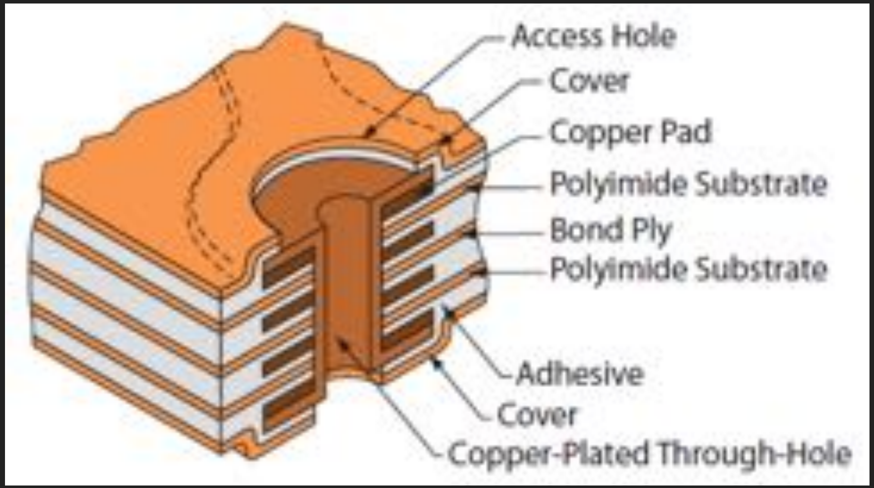


- Double-Layer (Type 2)

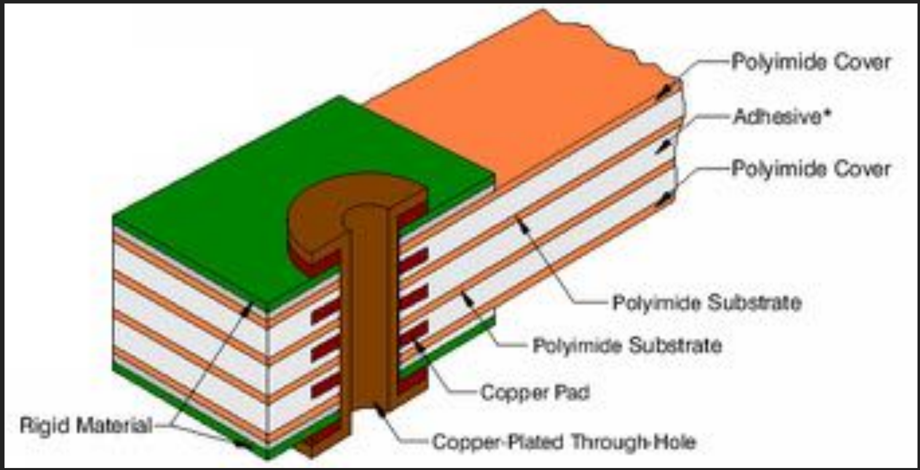


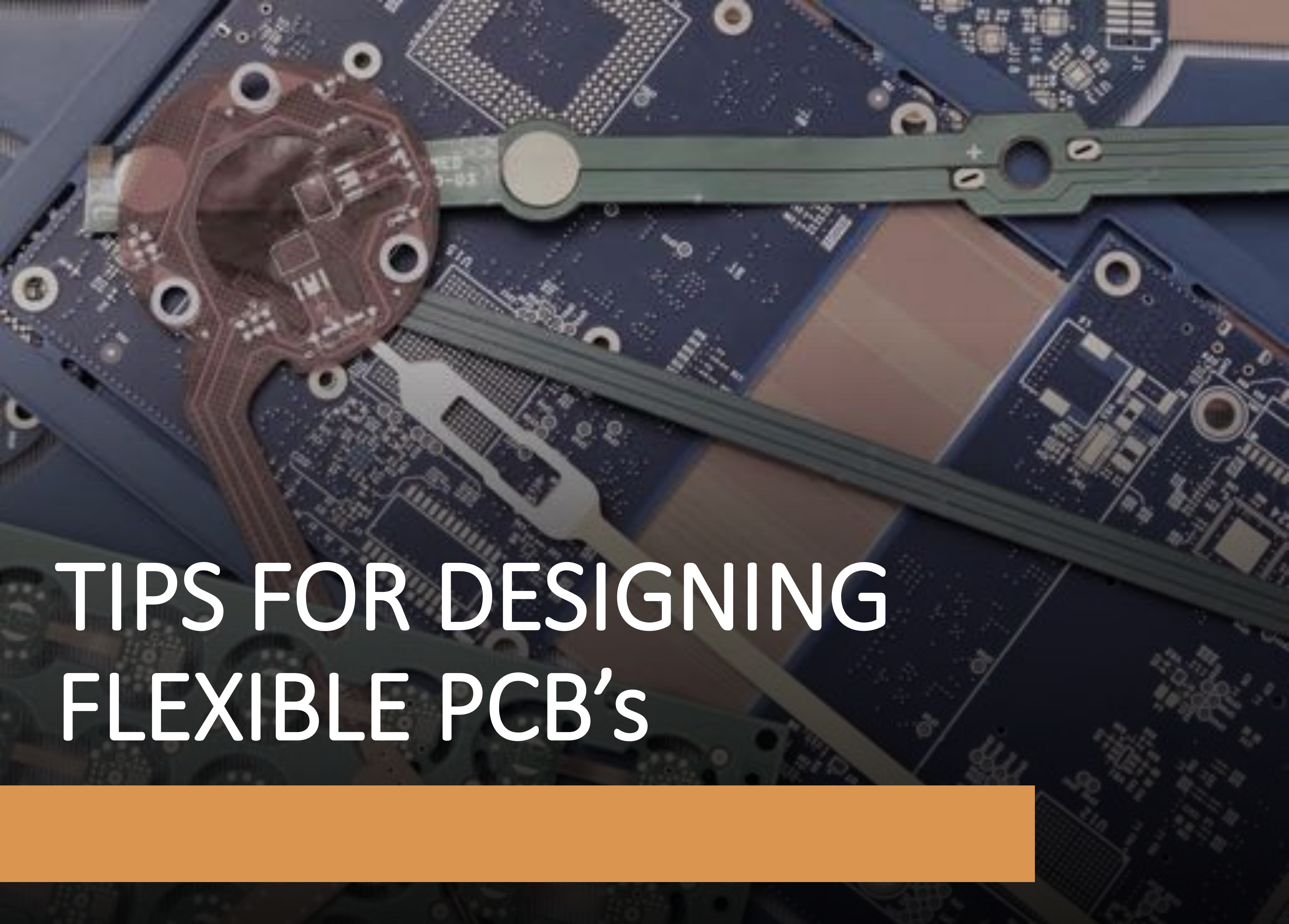
Per IPC-2223

Multi Layer (Type 3)



Rigid Flex (Type 4)



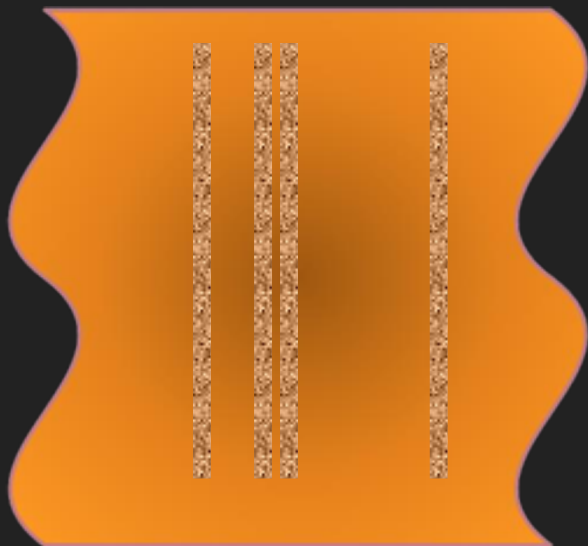


TIPS FOR DESIGNING FLEXIBLE PCB’s

Routing Options For Flex

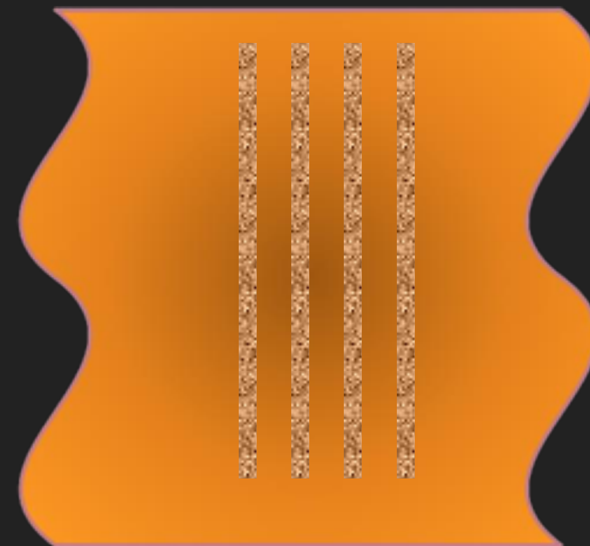
Balanced Circuit Pattern

Poor Design



Conductors routed randomly

Robust Design



Conductors evenly staggered

The examples above show a covercoated circuit pattern in a Type 1, 2 or Type 3 & 4 inner layer applications.

The "poor design" has a major potential flaw:

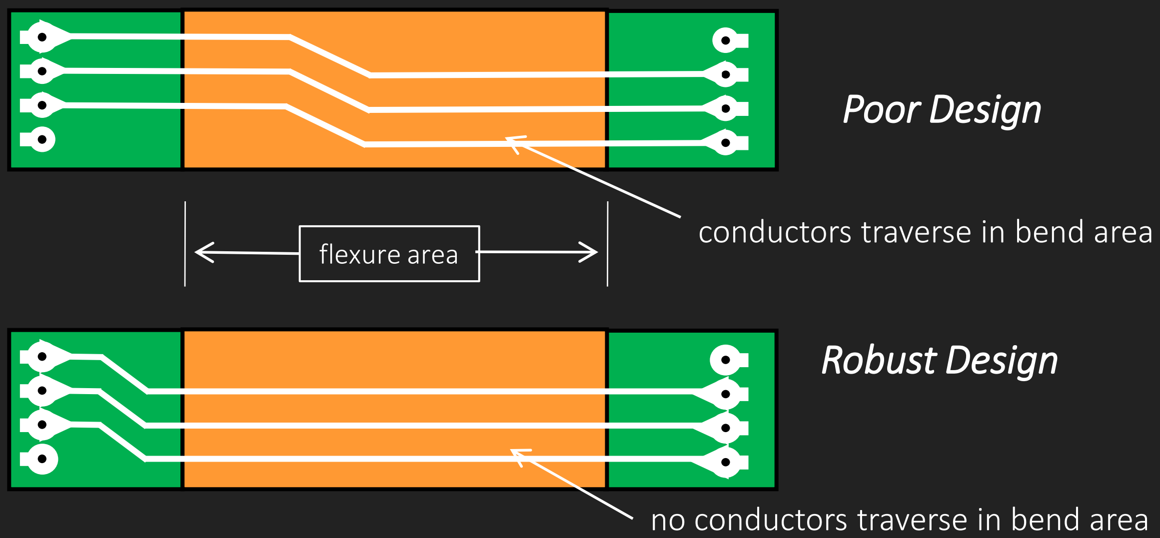
1) If the circuit is flexed perpendicular to the conductors repeatedly, stress will occur in the same location causing the isolated conductor to crack prematurely

The "robust design" has the following advantages:

1) Since the conductors are routed evenly, an isolated stress condition cannot develop and therefore no premature failure will occur

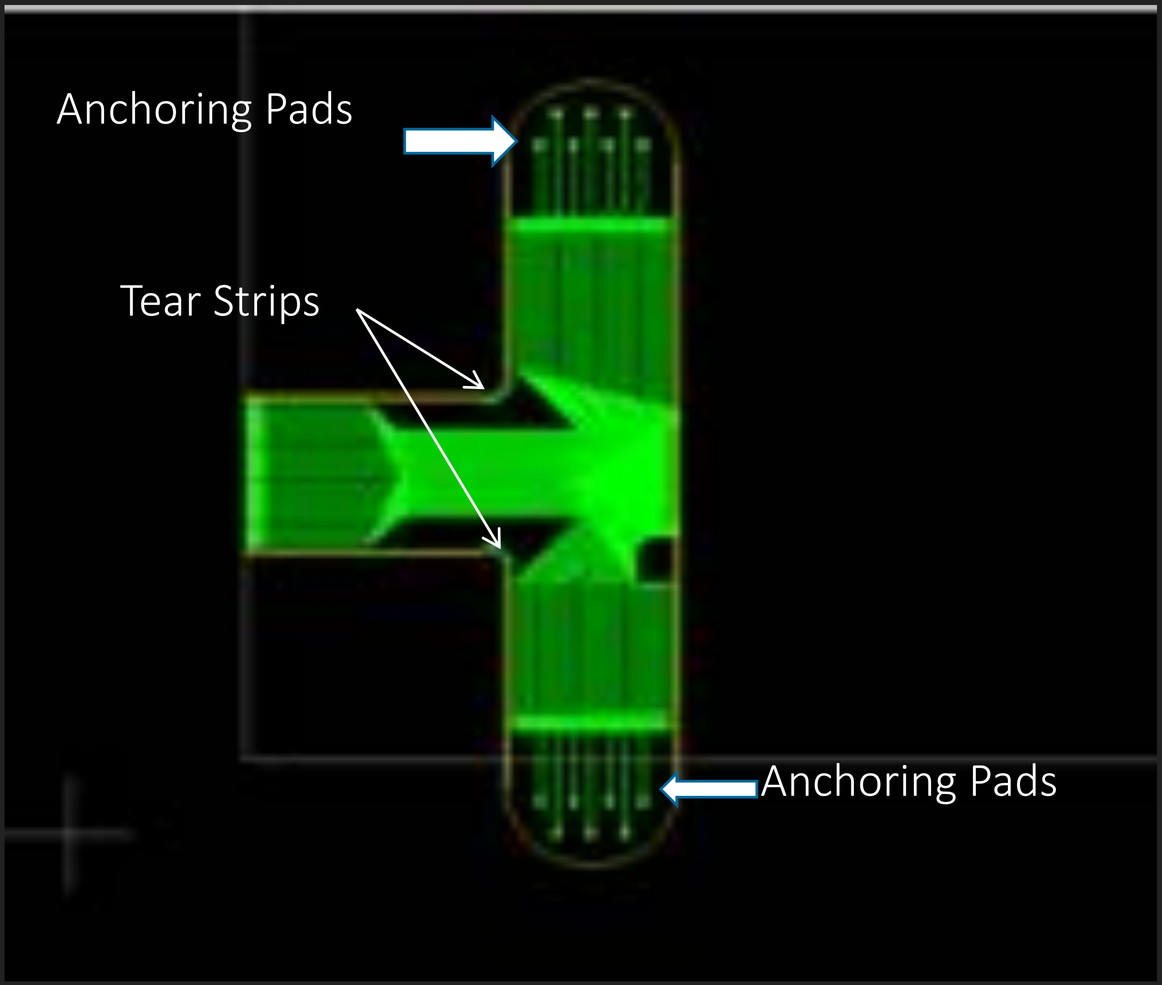
****Note: the robust design is absolutely critical in dynamic flexing applications ****

Dynamic Flexing Applications



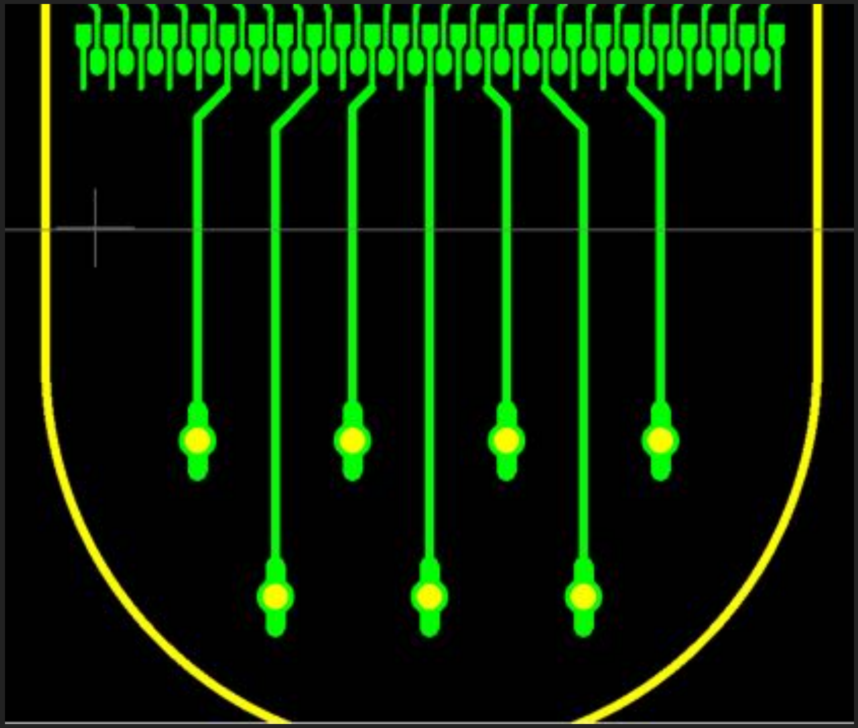
General:
All of the design guidelines apply for dynamic flexing applications. Any imperfections in artwork , materials, etched anomalies, edge roughness, etc. will cause a premature failure.

Flex Design Enhancements

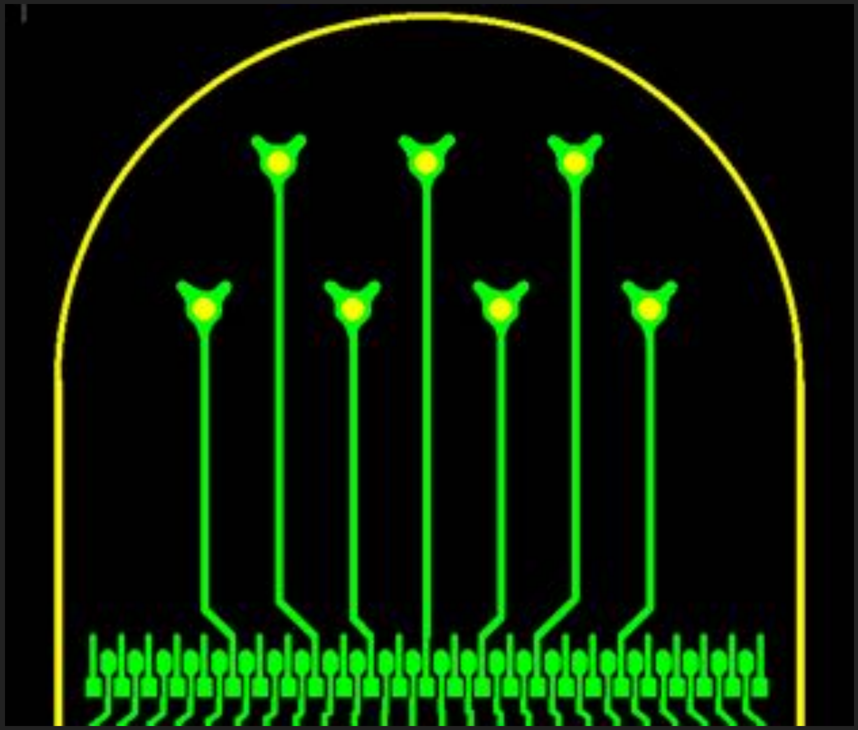


Flex Design Enhancements

Anchored Pads (Tie Downs)

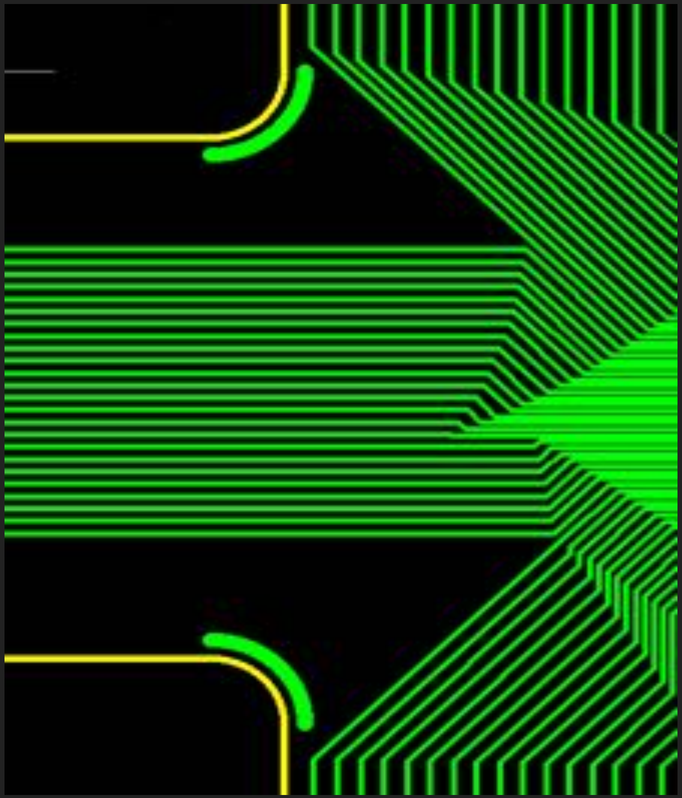


Anchored Pads (Rabbit Ears)

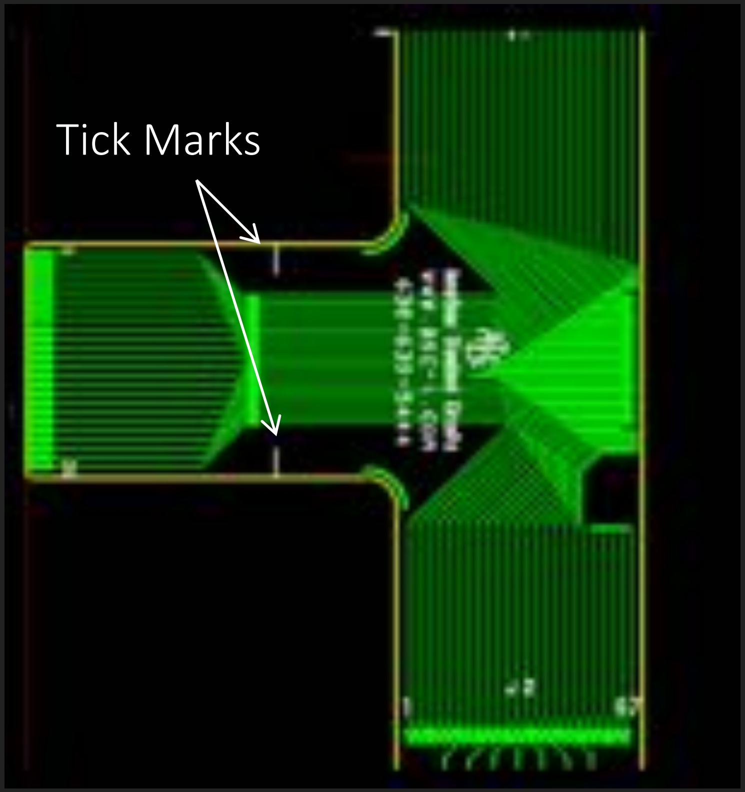


Flex Design Enhancements

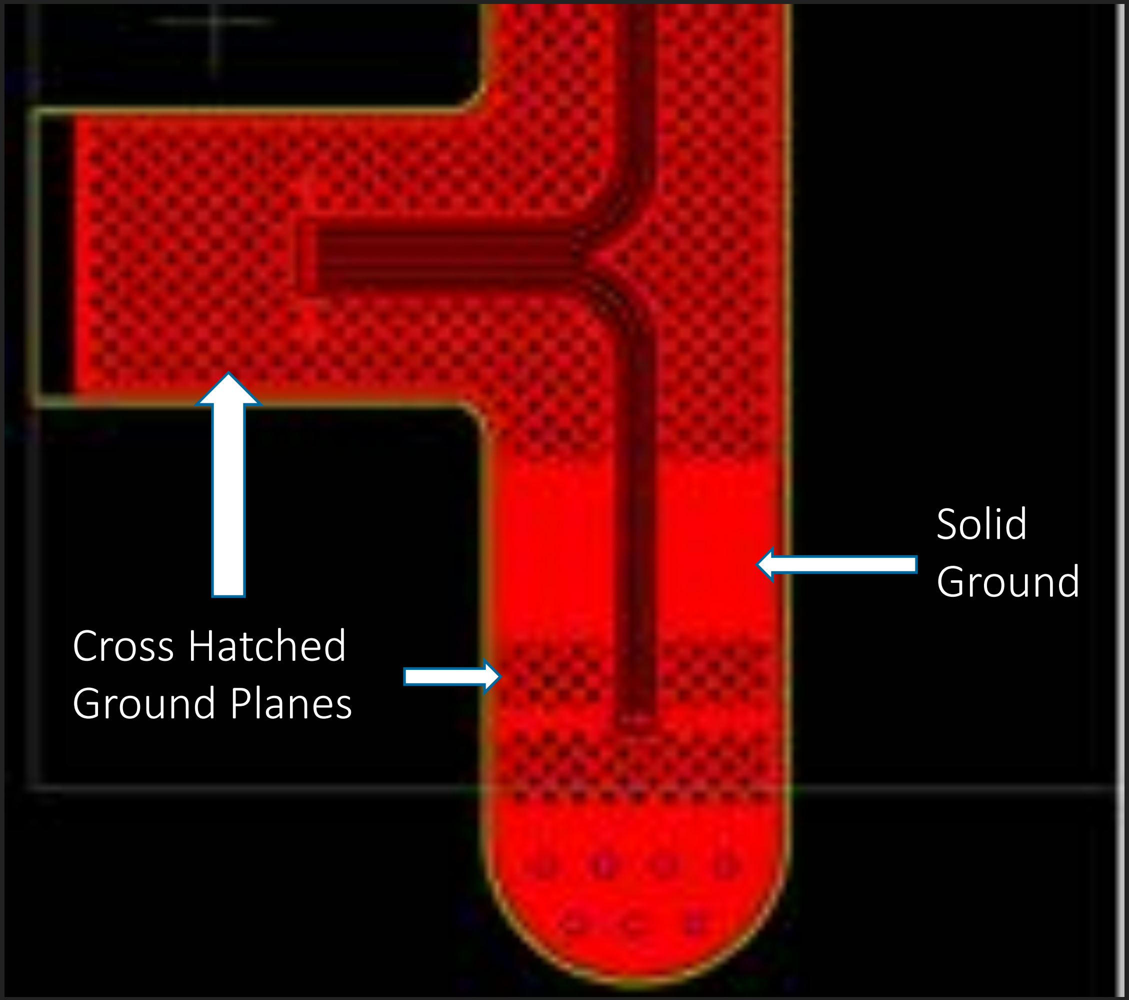
Tear Strips On Corners



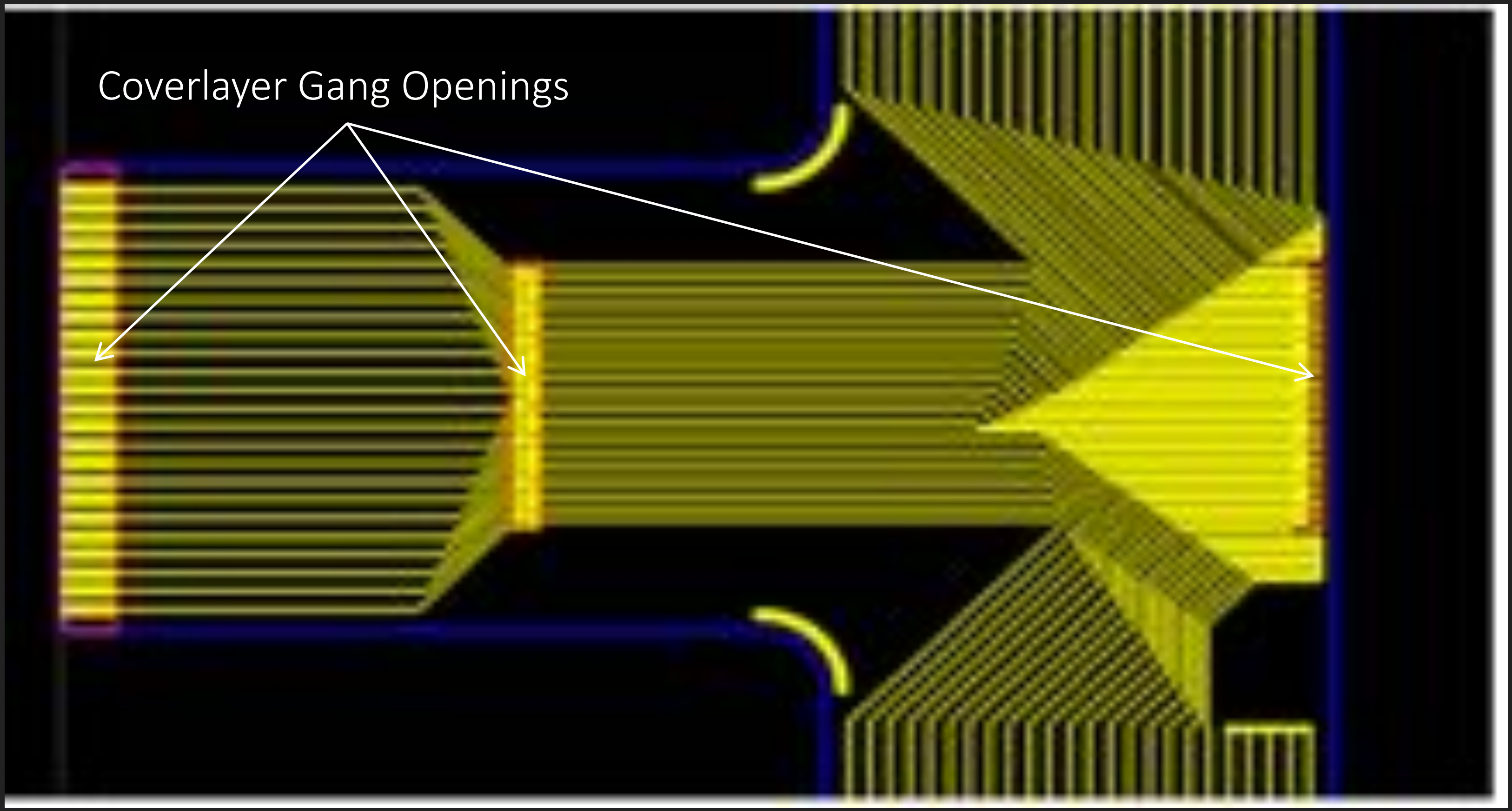
Tick Marks To Show Bend Areas



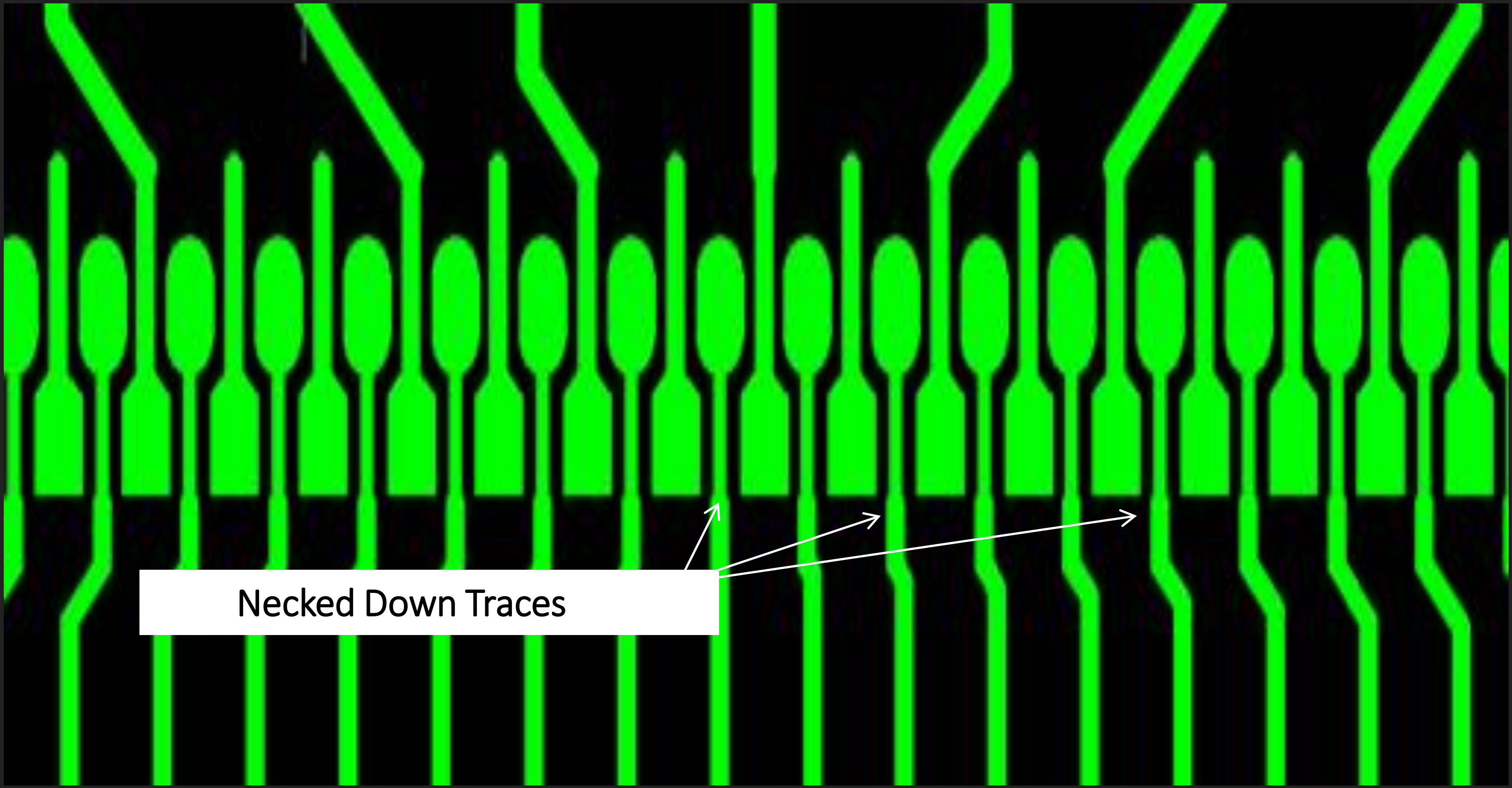
Flex Design Enhancements



Flex Design Enhancements



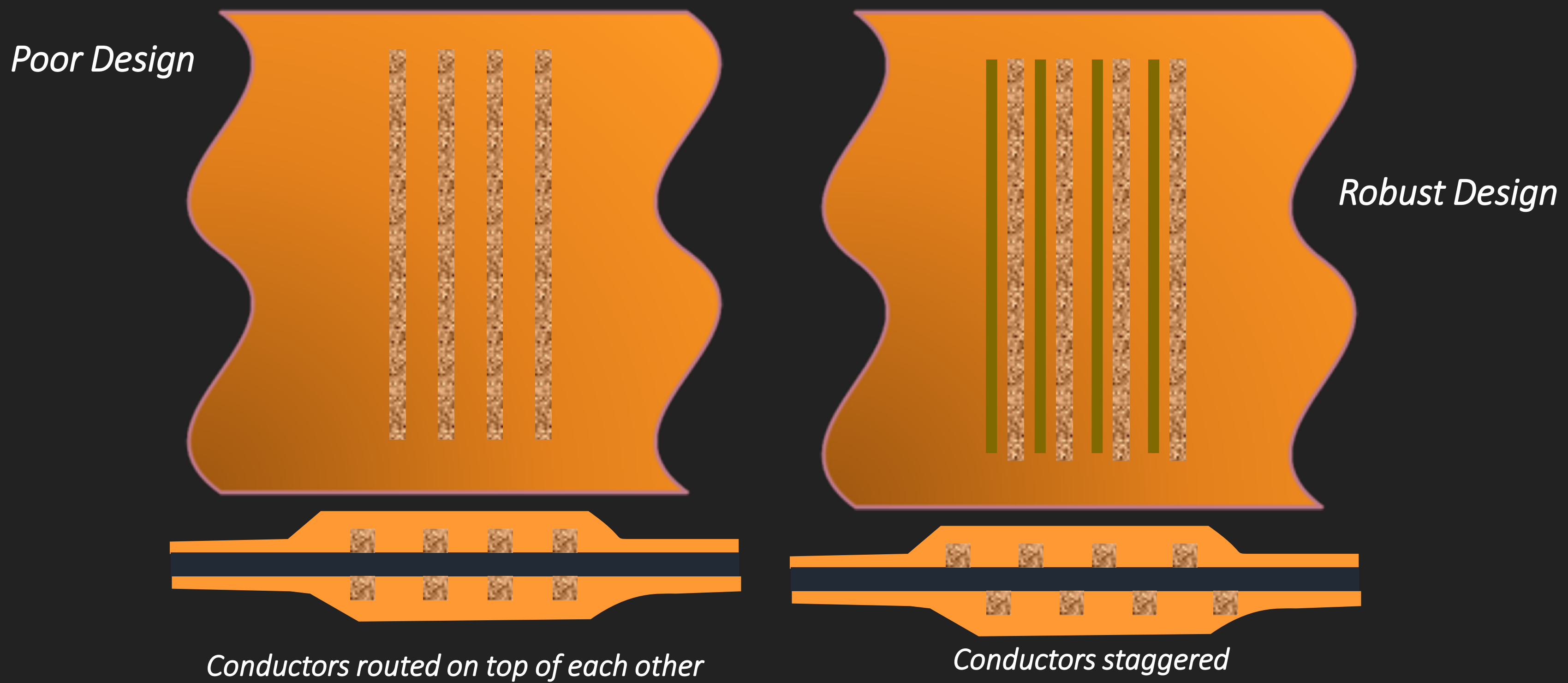
Flex Design Enhancements



Coverlayer Opening

Flex Circuit Type	Coverlayer Opening
Single metal layer flex circuit with land hold down features	Coverlayer opening can be roughly equal to pad diameter
Single metal layer flex without land hold down features or filleted lands	Openings in coverlayer should be 0.010” less than pad diameter
Double sided flex PCB’s and multi-layer flex with plated through holes and filleted lands	Coverlayer opening can be equal to or slightly larger than pad. This minimizes squeeze-out.
Non-component plated through hole vias	No opening unless needed for electrical test purposes

I-Beaming



The "poor design" has a major potential flaw:

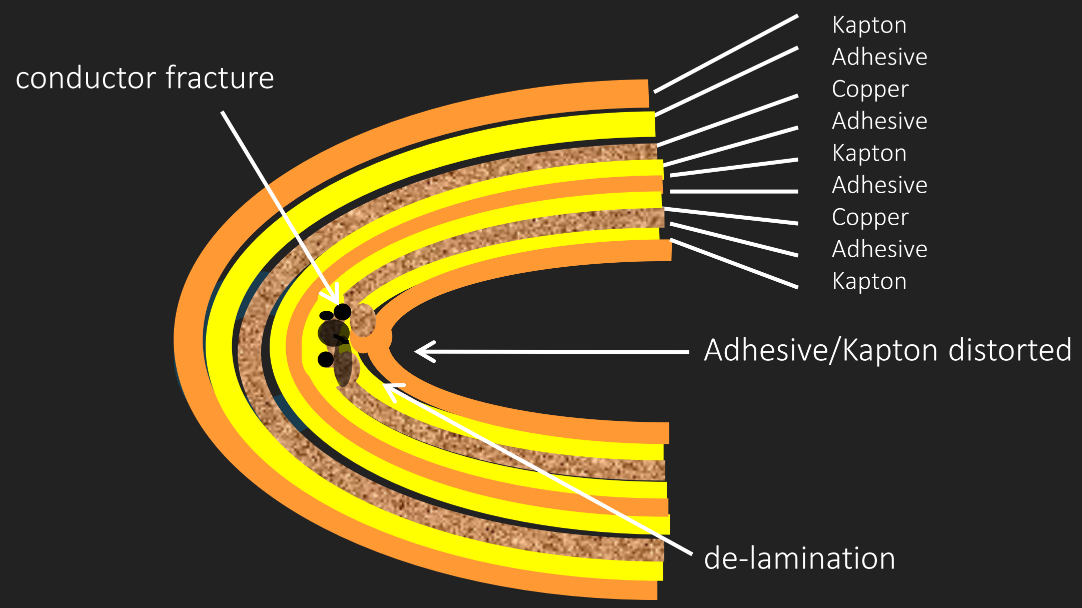
- If the circuit is flexed perpendicular to the conductors repeatedly, stresses force the copper to bend inward against the other conductor causing the conductor to crack.

The "robust design" has the following advantages:

- Since the conductors are not routed directly on top of each other, there is a place for the copper to displace and therefore is more resilient when repeatedly flexed.

**Note: the robust design is absolutely critical in dynamic flexing applications **

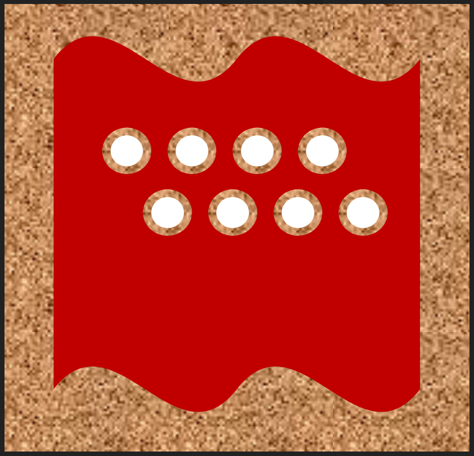
Minimum Bend Radius



Flex Circuit Type	Minimum Bend Radii
Single Sided	3-6x Circuit Thickness
Double Sided	6-10x Circuit Thickness
Multilayer Flex	10-15x Circuit Thickness (or more)
Dynamic Application (only SS recommended)	20-40x Circuit Thickness (increase in radius normally increases life)

Pad Only Plating

result after copper plating



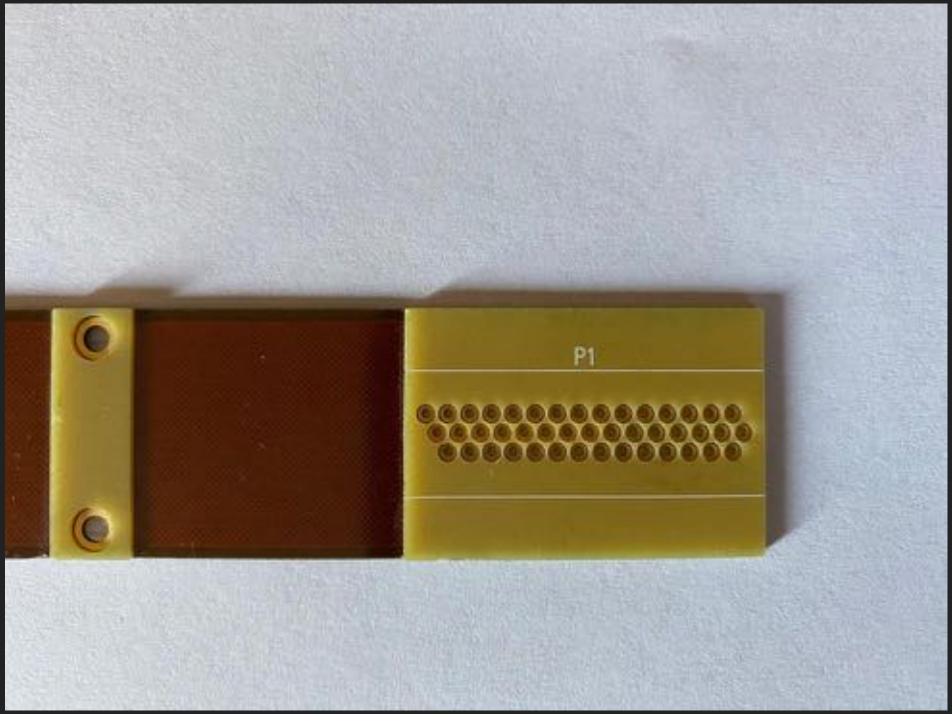
photoresist image for electrolytic copper plating the holes

General:
This process is used on Type 2 circuits where maximum flexibility is desired.
A full thickness of electrodeposited copper will reduce flexure life. Therefore in those cases, it is desirable to limit the amount of electrodeposited copper on the conductor surfaces.

RIGID FLEX

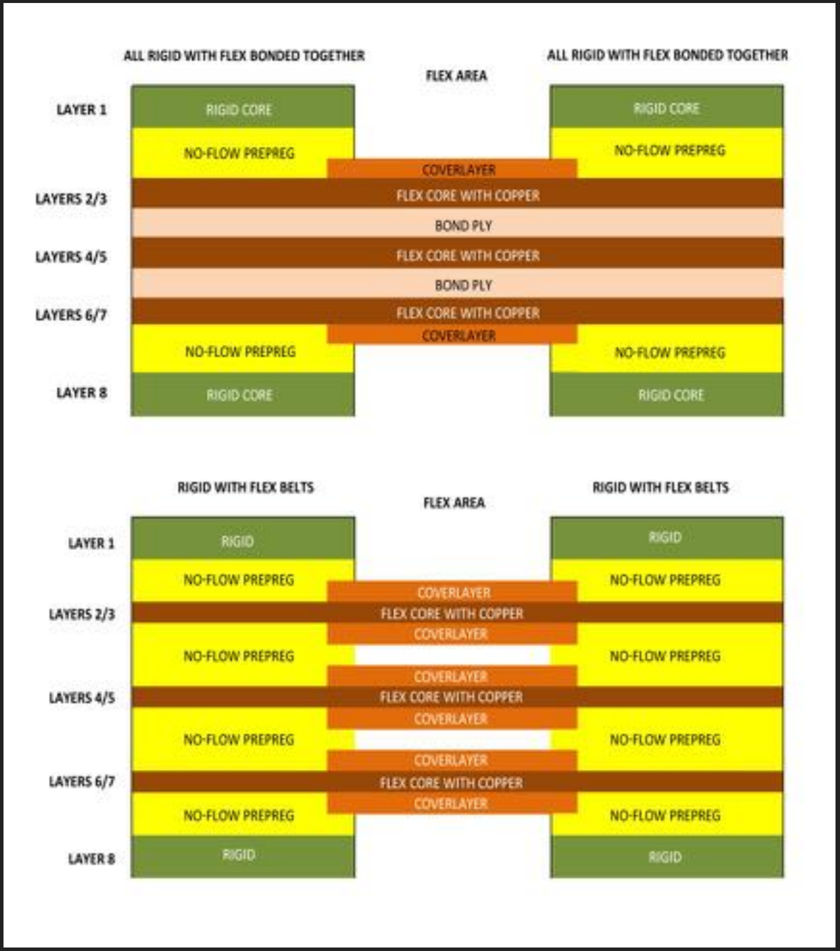
Rigid Flex/Stiffeners

This is an example of a flex PCB with stiffeners. This is a more cost effective option when allowable than designing a rigid flex.

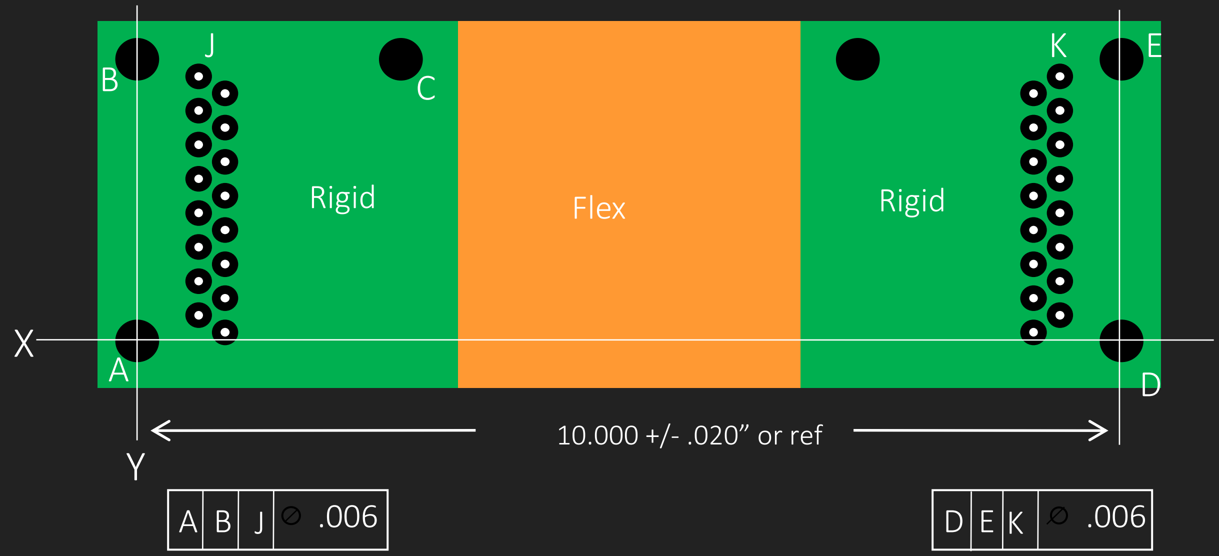


Rigid Flex

- Adhesiveless Materials
- Bikini Cut Coverlayer
  - Overlap of partial coverlayer to be minimum 0.050”
- PTH should be .100” from edge of flex / rigid interface

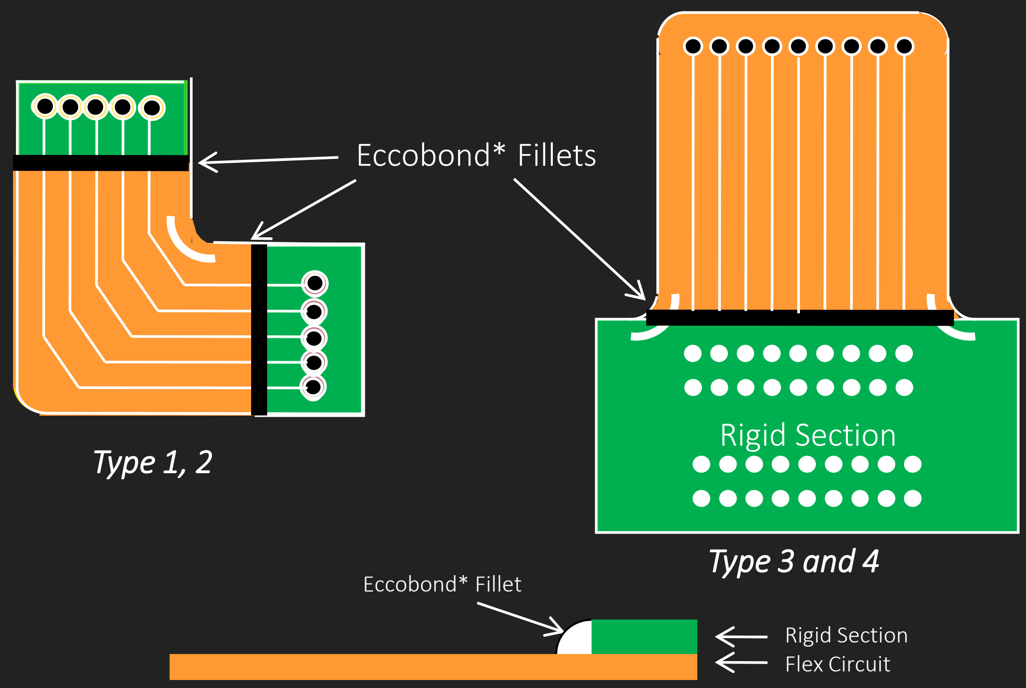


Rigid Flex Hole to Hole Tolerances



The example above shows a Type 4 Rigid/Flex hole locations with datums. Hole locations within a rigid section can be held to typical rigid true hole positions (.005-.010"). However, when applied across the flexible section (rigid to rigid) these tolerances cannot be maintained due to the flexible material may shrink or have slight distortions when in a un-restrained condition. Therefore a preferred practice is dimension datum holes across the across rigid sections with "loose" or "reference" dimensions while maintaining typical true hole position tolerances within a rigid section.

Eccobond* Fillets

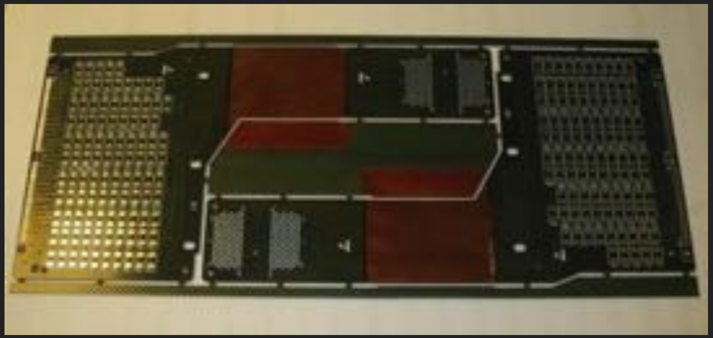


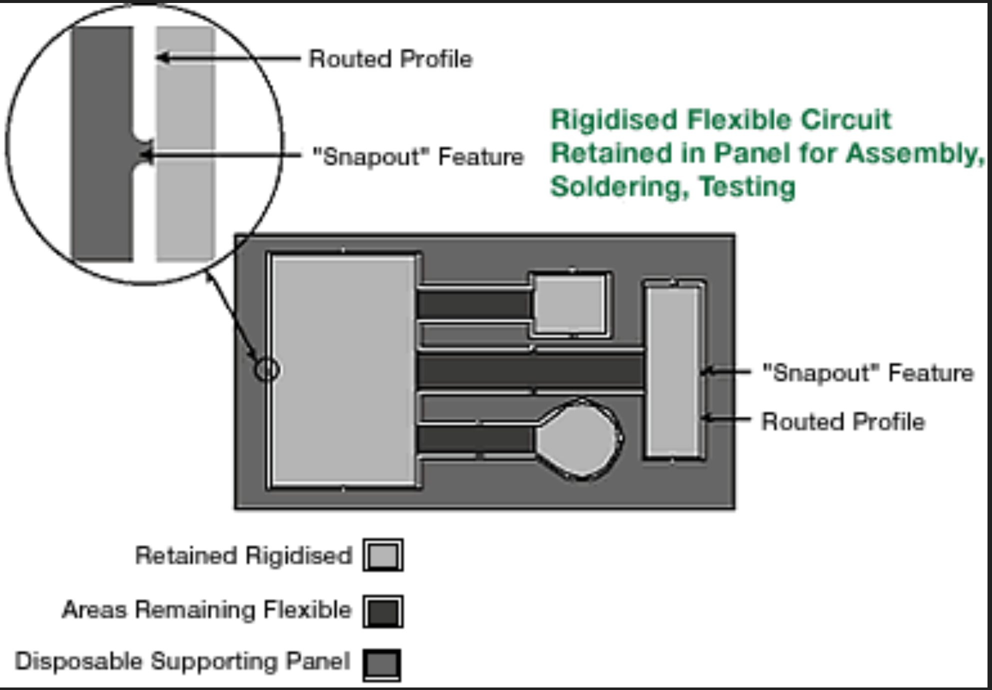
Eccobond* Fillets:

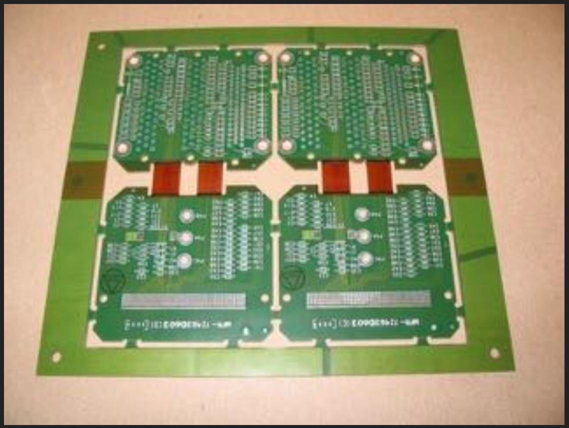
The purpose of the fillet is to prevent conductors from being cracked when the flex circuit is flexed during installation. Basically it prevents the flex circuit from being bent at this transition area. Also, In rigid/flex applications, there may be "prepreg squeeze out" at the rigid edge from the lamination process that may contain sharp edges that can pierce the flex circuit and cause conductor breakage. The eccobond* material will encapsulate those sharp edges and eliminate this issue.

Routing Practices

- Break off tabs
- Utilizing break off tabs, the rigid flex can be supplied in a panel array to ease in handling at assembly.
- Array size can contribute to the cost of the card.

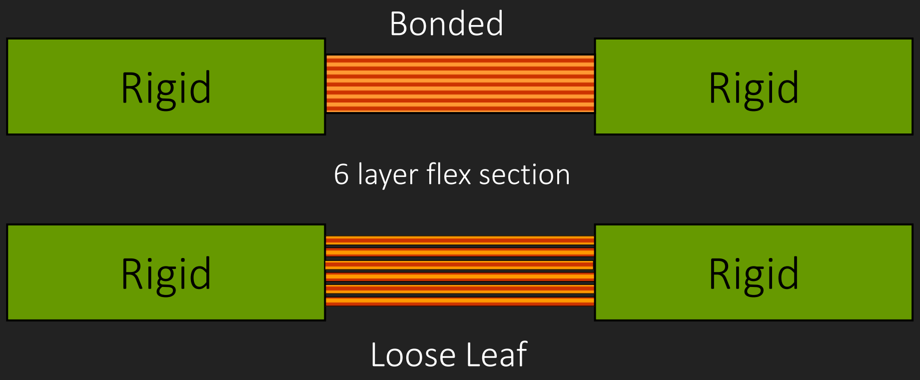






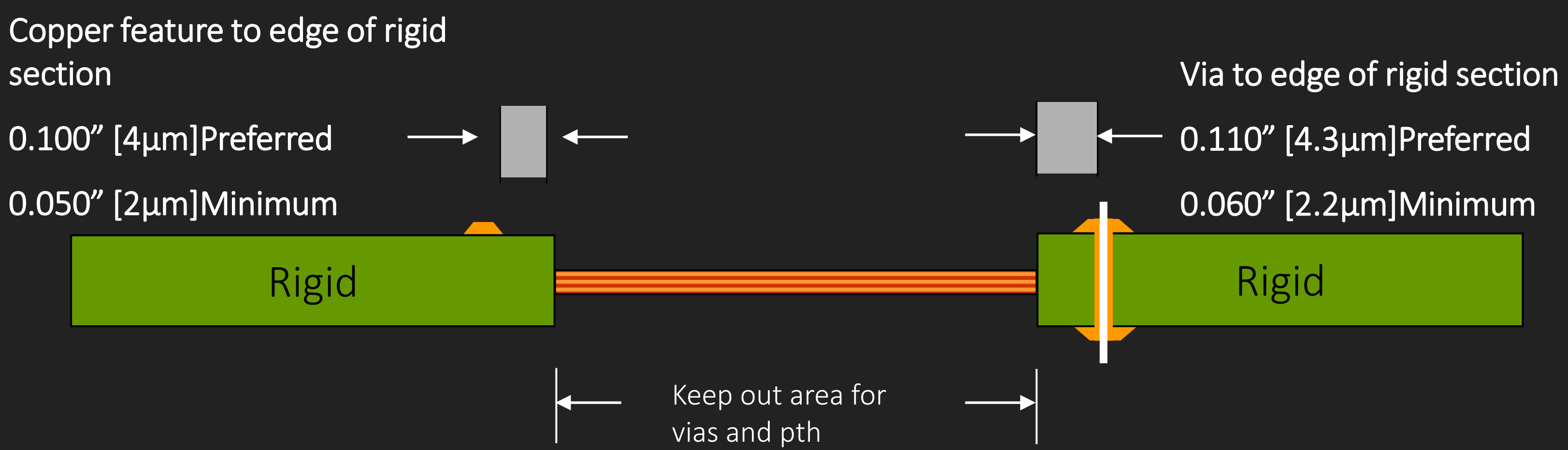
High Layer Flex in Rigid Flex

- As the flex section layer count increases, the ability to bend the flex decreases.
- Utilizing single sided flex sections increases the flexibility.



Routing/Layout Practices

- Keep Out Areas– Minimize cost.



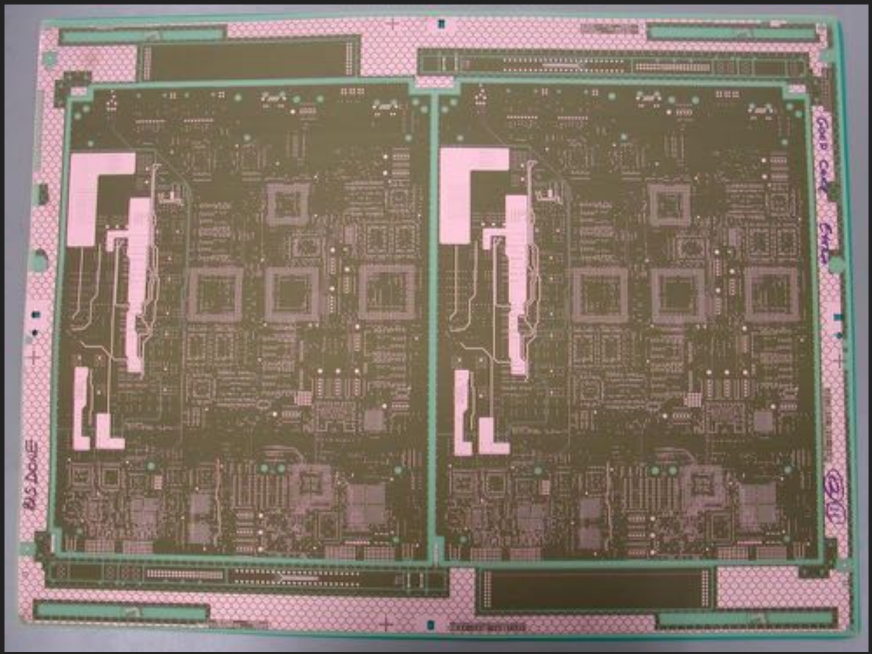
Some Design Mistakes To Avoid

- Placing vias where the flex bends or at edge of stiffeners
  - Can cause via cracking
- Failing to add teardrops at the pad to trace interface.
  - Can lead to trace breakage
- Having sharp angles when routing traces, especially in the bend region
  - Can lead to stress riser and fracture the traces
- Creasing, folding or bending flex circuits beyond it's stress point.
- Failing to anchor unsupported pads
  - Can lead to pad lifting during assembly
- Making the hole size of the stiffener too small. Should be +.020" over finished hole size

COST CONSIDERATIONS

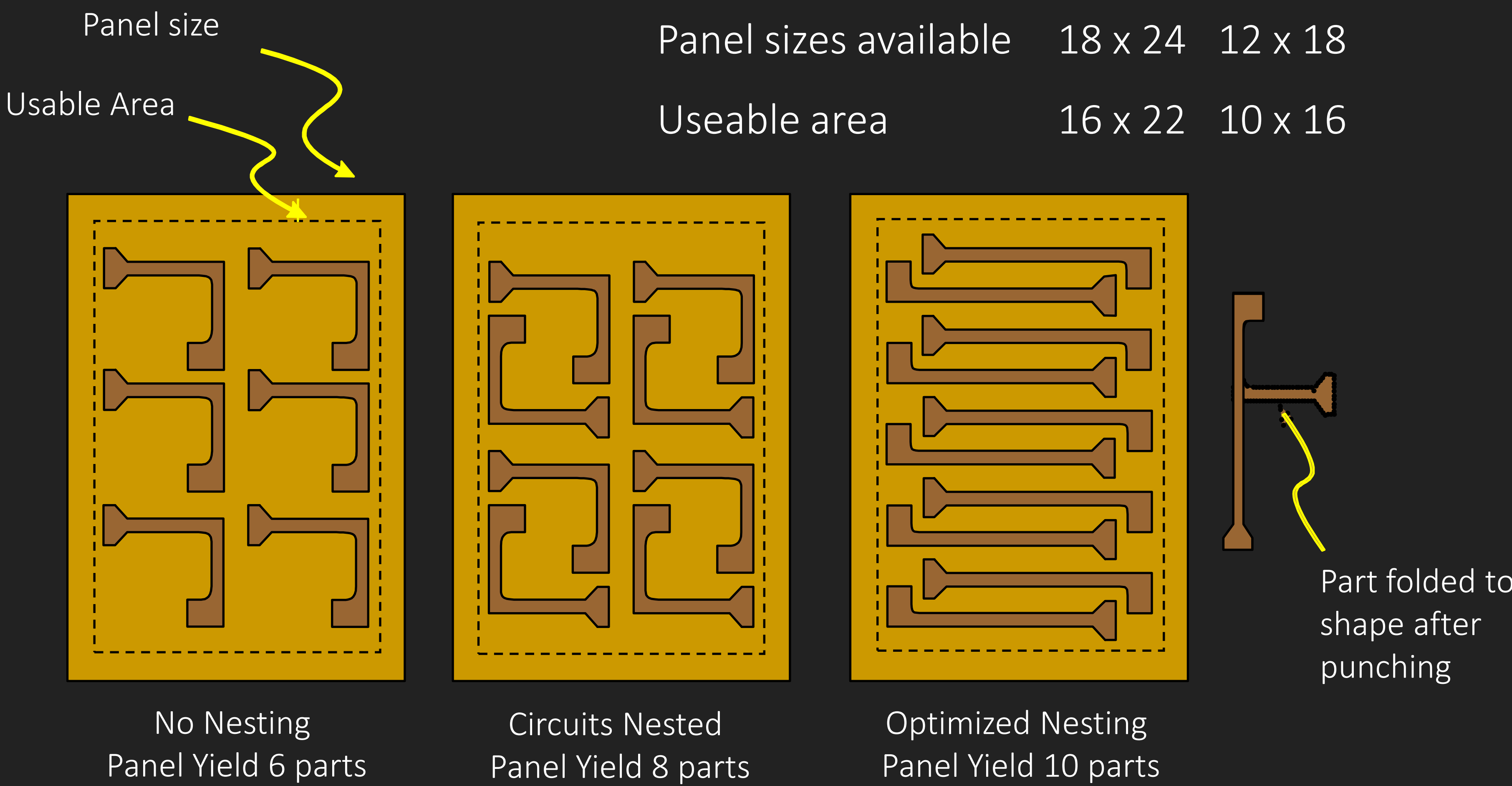
Panelization

- Panelization
  - PWB's are manufactured on standard panel sizes.
  - Cost is a factor of the number of individual cards on a production panel.
  - Impedance/Mil coupons, if required, are placed in the production panel. May effect panelization yield, ie. $$.
  - Industry standard panel size is 18" x 24".
  - General rule:
    - 1.000" boarder
    - 0.25" – 0.5" spacing
    - Spacing is very dependent on design complexity and density. More spacing may be required to improve dimensional stability in Rigid Flex applications.



Nesting Of Circuits Improves Panel Yield

Panel sizes available	18 x 24	12 x 18
Useable area	16 x 22	10 x 16



Material Selection

Thickness of flex material core

- 1 and 2 mil thick are the most common and economical

Copper Weights

- ½ ounce and 1 ounce are most common

Coverlayer Thickness Typically

- 1 mil with 1 mil of adhesive

Cost Drivers

Low Cost Factors (<10%)

- Complex routing/Scoring
- Edge Routing
- >0.093” thick PWB’s
- <0.030” thick PWB’s
- Via Plug (button print)
- Strain Relief
- Adhesive Vs Ahesiveless Mat’l.

High Cost Factors (>25%)

- Advanced Technologies
- Buried Vias
- Layer Count
- Material Utilization
- Selective Plating
- Buried Access (ZIF connectors)
- Dual Surface finish
- Line Width and Space (<.004/.004)

Medium Cost Factors (10%-25%)

- Aspect ratio > 10:1
- Drill hole count (>30k)
- Non-FR4 materials
- Drilled holes <0.012”
- Stiffeners (Rigidizers)
- < 0.005” Line/Space
- Button Plating
- Controlled Impedance
- Annular ring (Pad< Drill + 12)

Additional Considerations

- **Cost Trade-offs**
  - Use a smaller line width/space before adding layers
  - Investigate how boards will fit into a production panel to ensure that maximum material utilization occurs.
  - Consider reliability/issues:
    - Adhesive materials are lower cost that Adhesiveless materials. The increase in acrylic resin in a via stack reduce reliability due to an excessive CTE-Z

THANK YOU!

American Standard Circuits, Inc.
475 Industrial Dr.
West Chicago, IL 60185
PH: 630-639-5444

sales@asc-i.com
www.asc-i.com