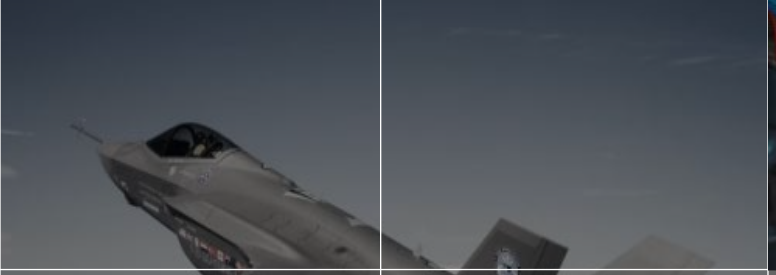
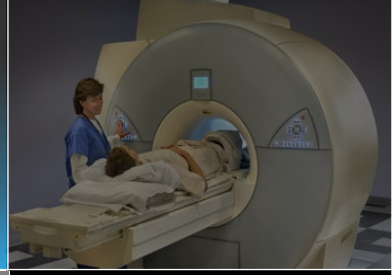
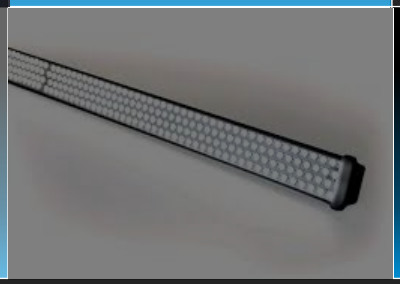
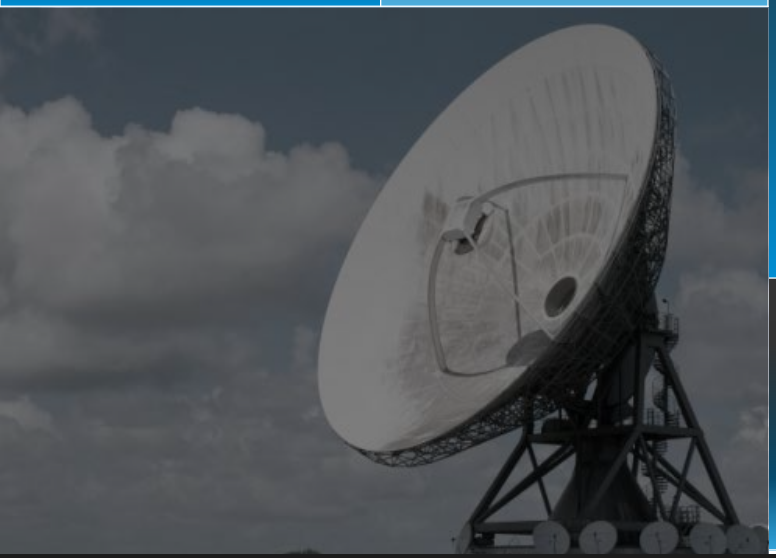




American Standard Circuits

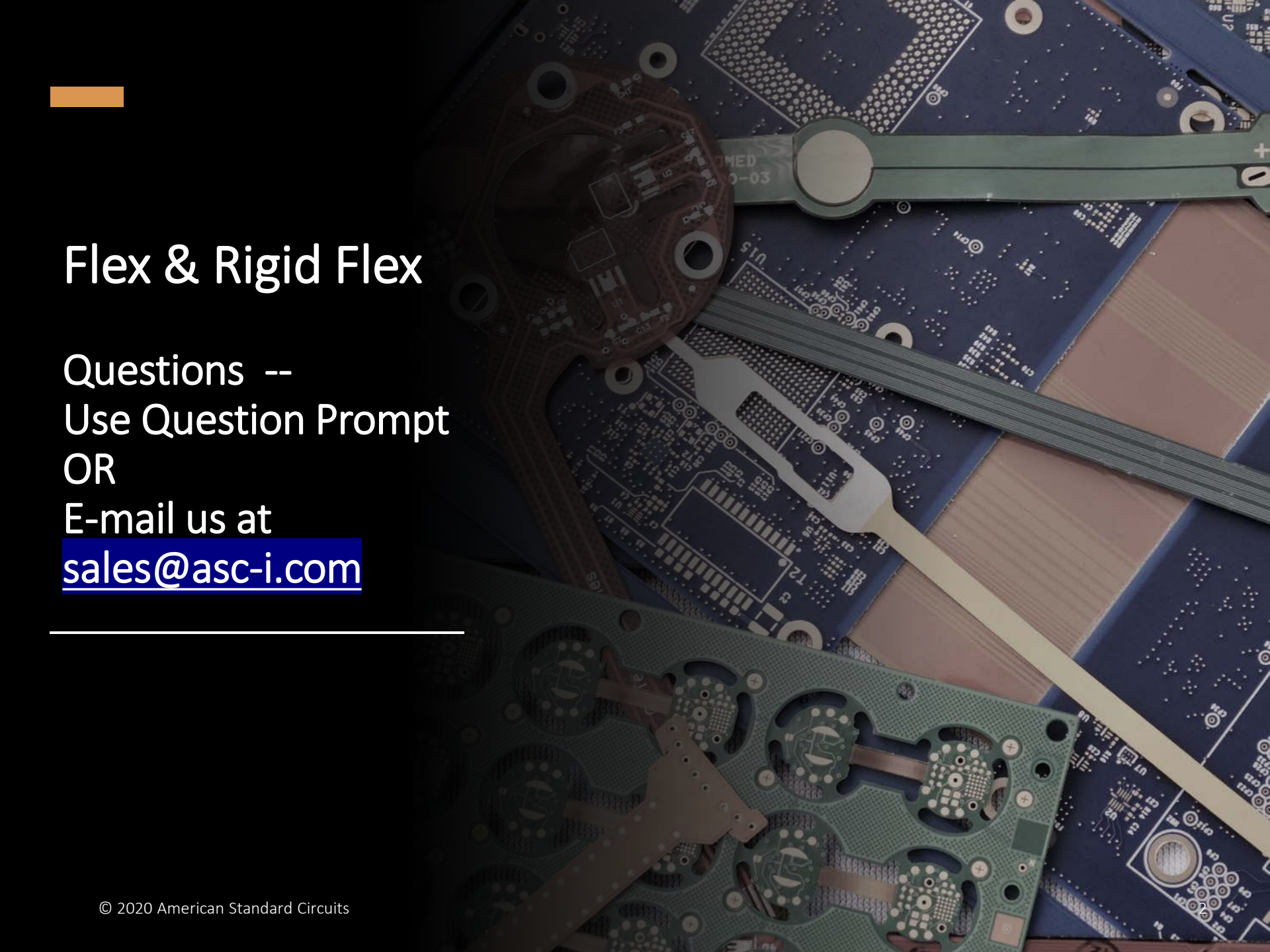
Creative Innovations In Flex, Digital & Microwave Circuits

"Collaborate To Win"

www.asc-i.com

Flex & Rigid Flex

Questions --
Use Question Prompt
OR
E-mail us at
sales@asc-i.com



SPEAKERS

Anaya Vardya

President & CEO @American Standard Circuits

Anaya Vardya has over 35 years in the electronics manufacturing business and is currently the president and CEO of American Standard Circuits, Inc. Anaya has a master's degree in chemical engineering from the University of Cincinnati and a bachelor's of technology from the Indian Institute of Technology.

Email: Anaya@asc-i.com

Phone: (503) 887-1909

Dave Lackey

VP of Business Development @American Standard Circuits

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

Email: Dlackey@asc-i.com

Phone: (815) 403-3415

If you are interested in a customized Webinar presentation for your company's needs, please contact us at sales@asc-i.com

Definitions

- **Flex Circuit:**

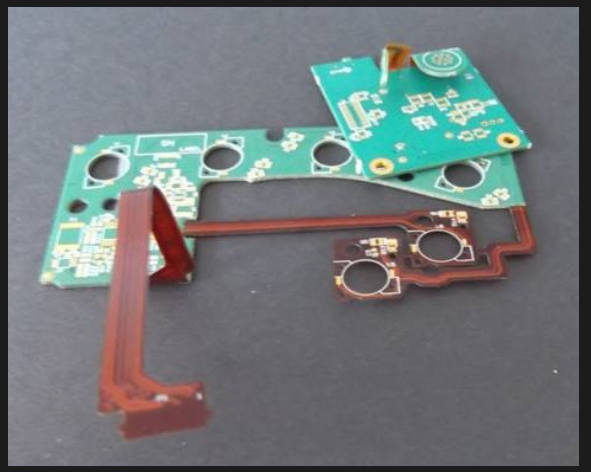
  Made of a flexible polymer film laminated to a thin sheet of copper that is etched to produce a circuit pattern

- **Rigid-Flex Circuit:**

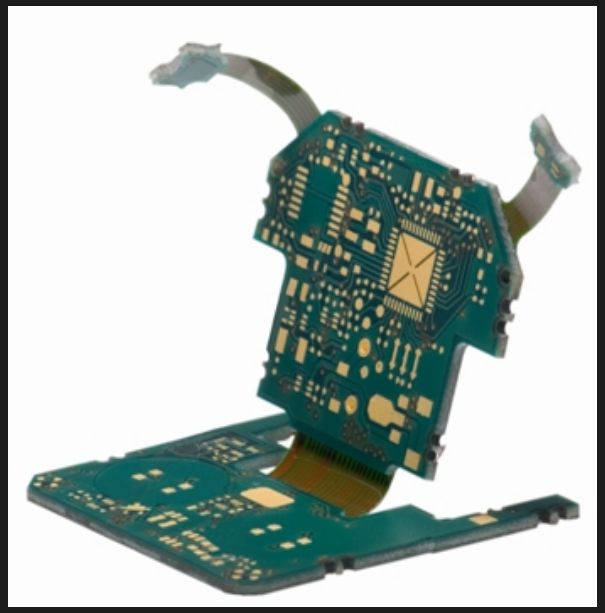
  Combination of a flex circuit with a rigid circuit as one circuit board

Benefits of Flexible **Circuitry**

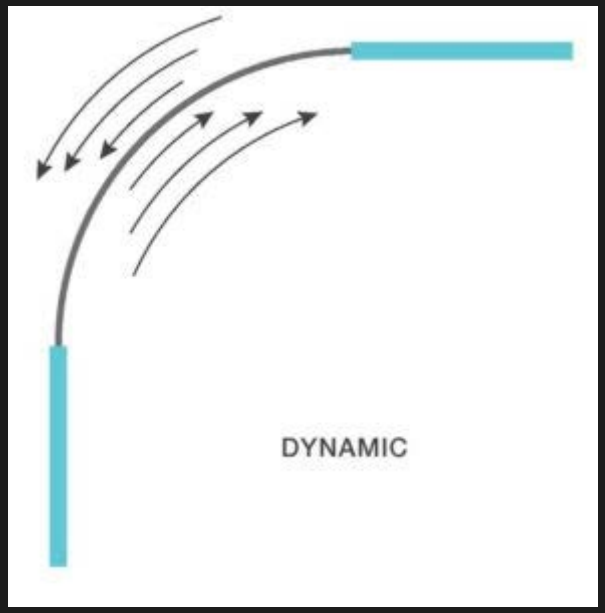
- A solution to a packaging problem
- Replacement for a circuit board and wires
- Aesthetics
- Reduce assembly costs
- Increased Reliability
- Reduce weight and space
- Dynamic flexing
- Thermal management/ High temp applications



Static vs Dynamic



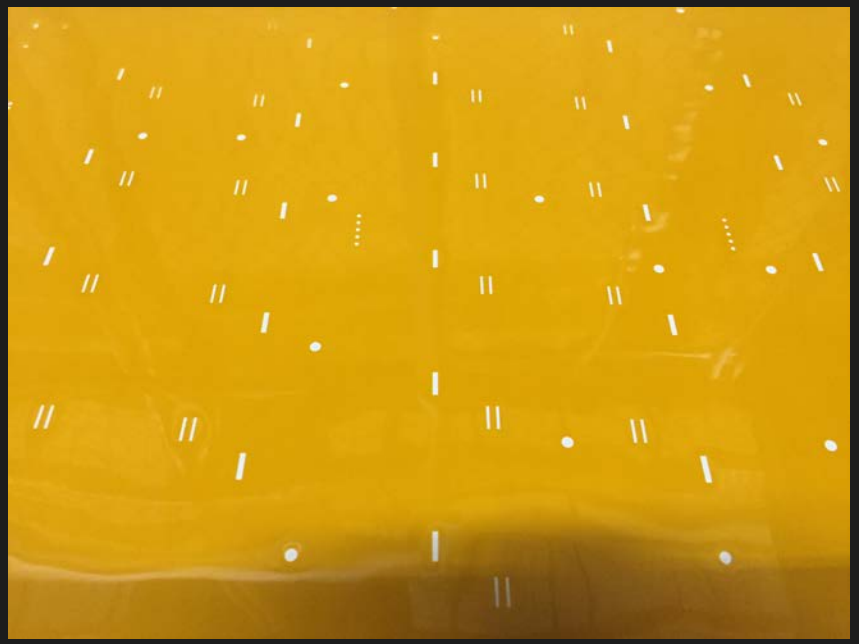
Static Flex (Bend to Install)



Dynamic Flex (Always Moving)

Materials

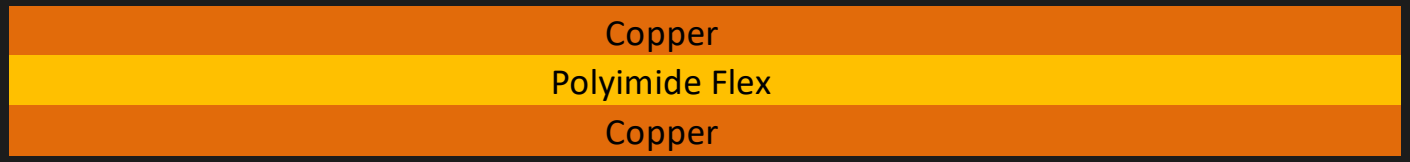
- Base Materials
- Coverlayers
- Stiffeners
- Shielding



Base Materials

Flex Core with Adhesive

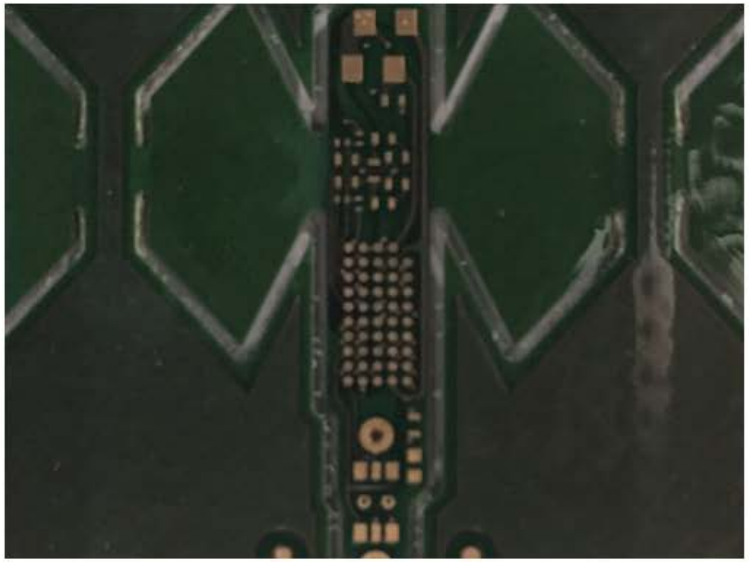
Adhesiveless Flex Core



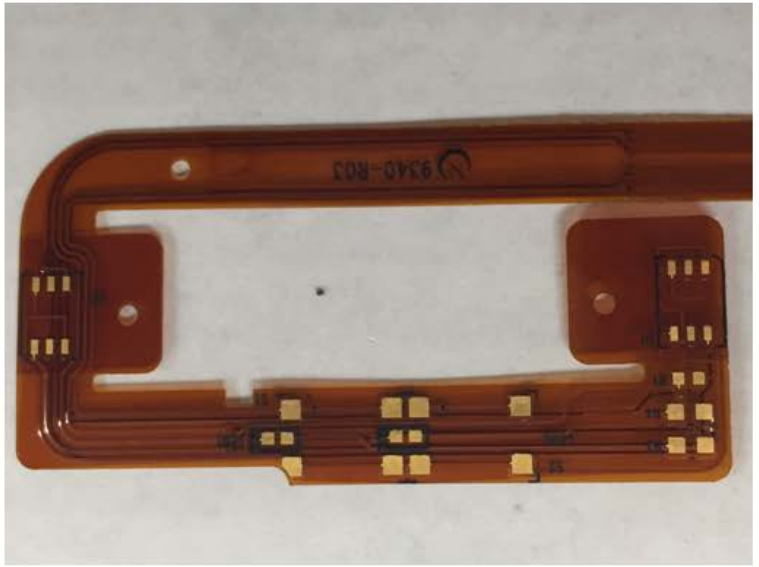
Copper is either RA (Rolled Annealed) or ED (Electrodeposited)

ED usually less than ½ Oz. copper weights

Coverlayer Options



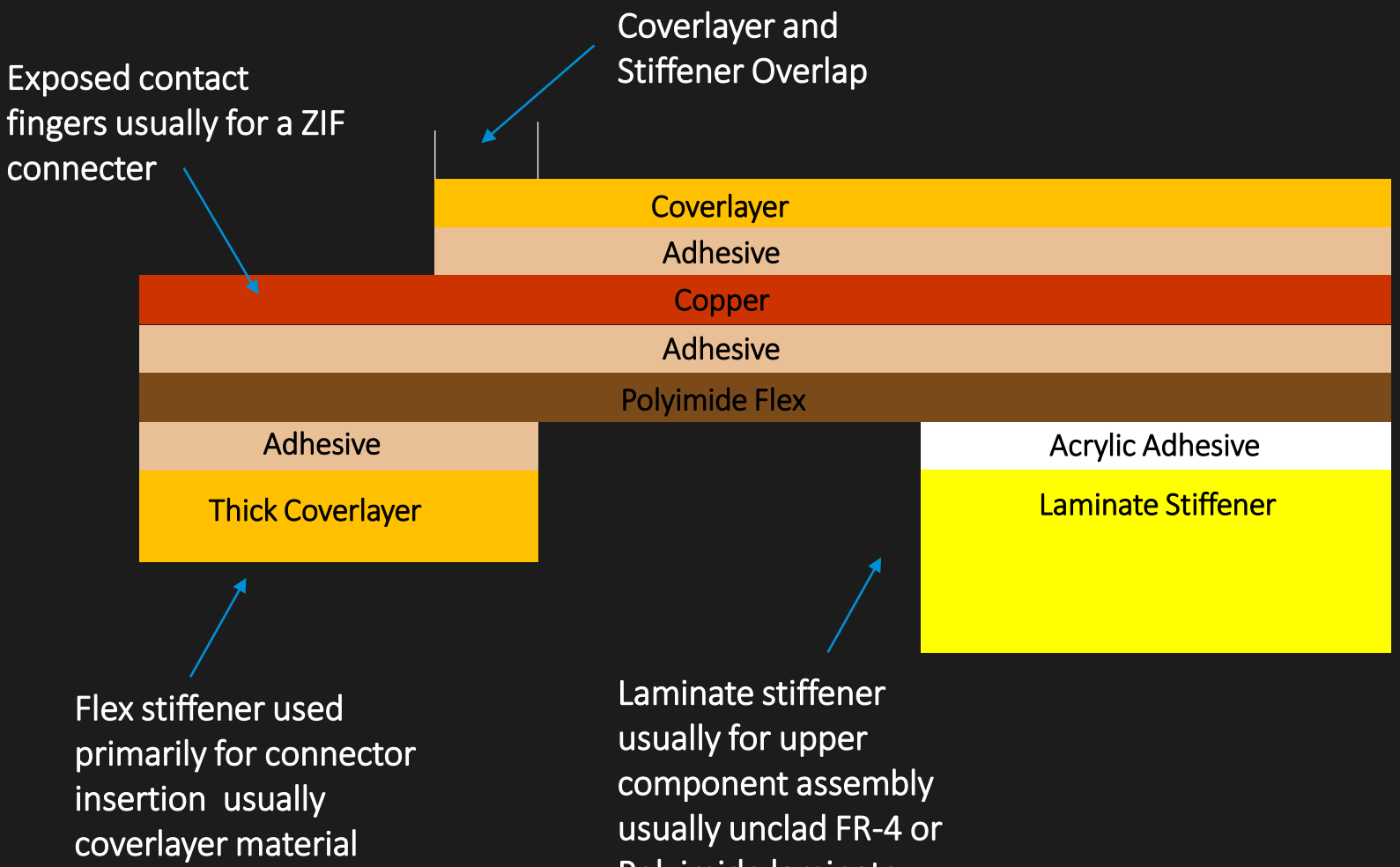
Soldermask Coverlayer



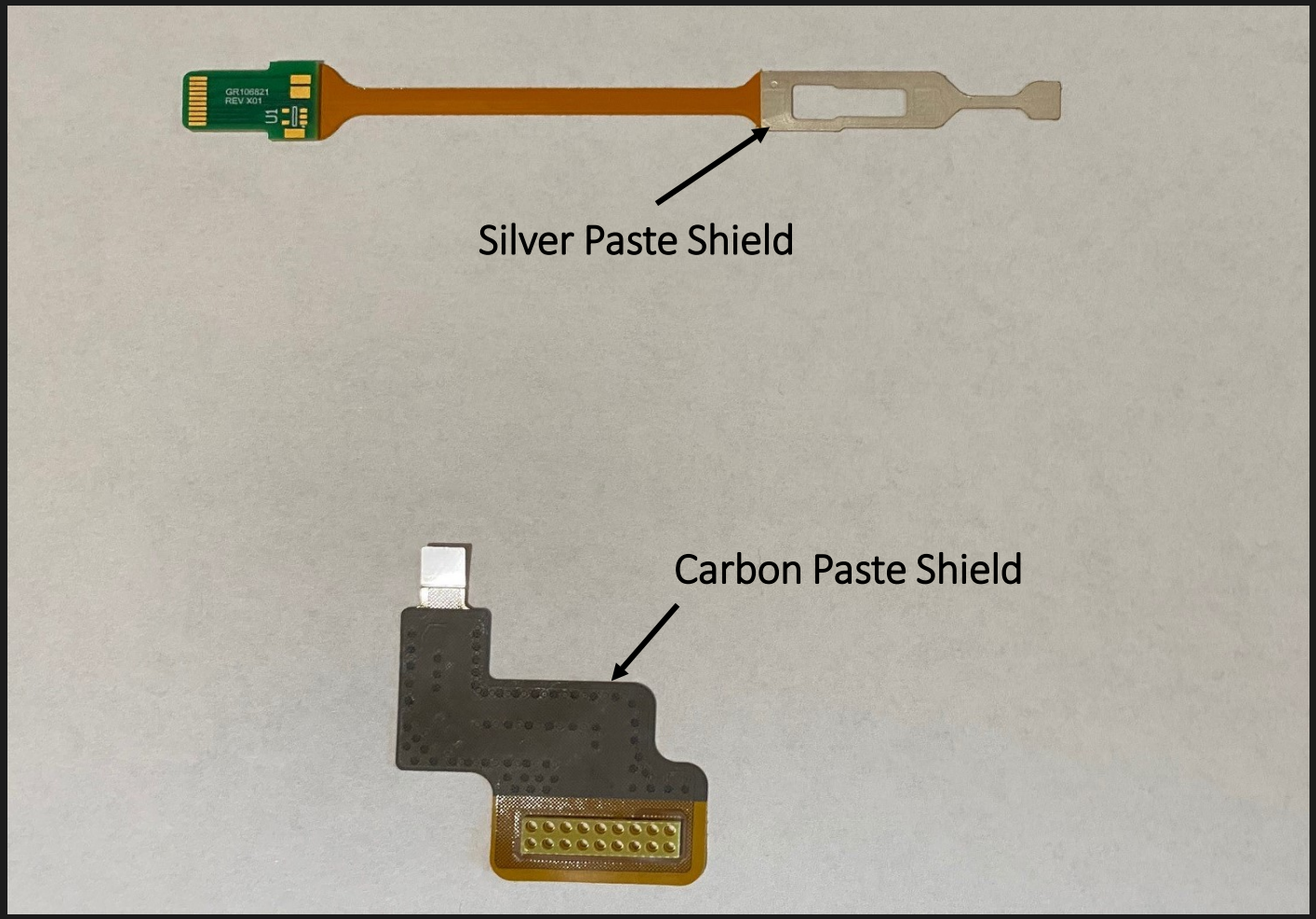
Polyimide Coverlayer

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

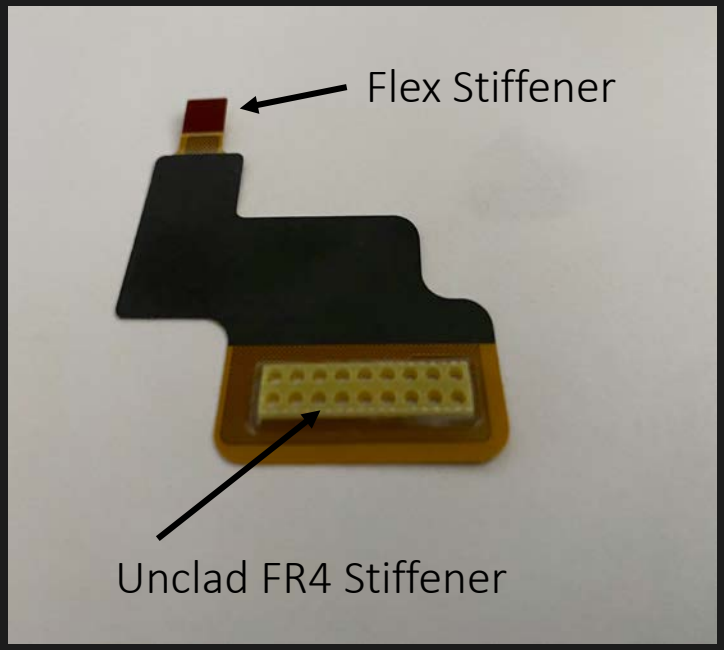
Stiffener

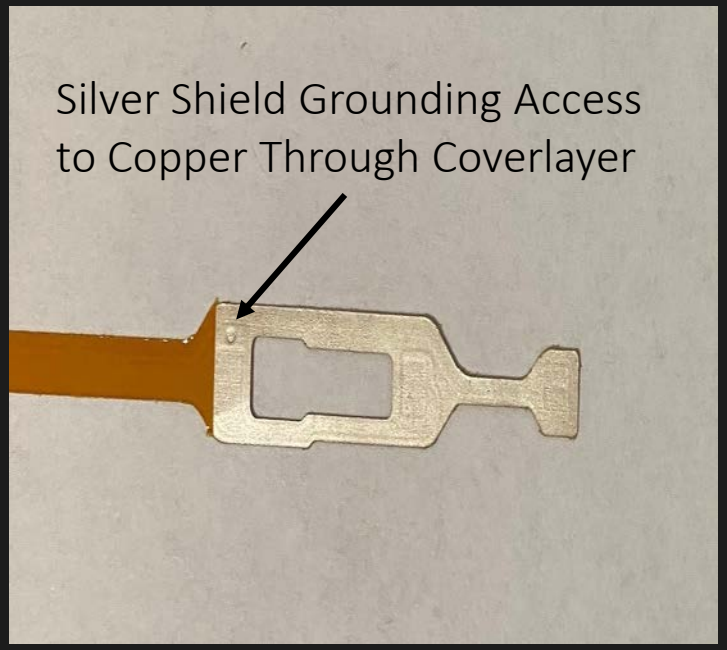


Shielding



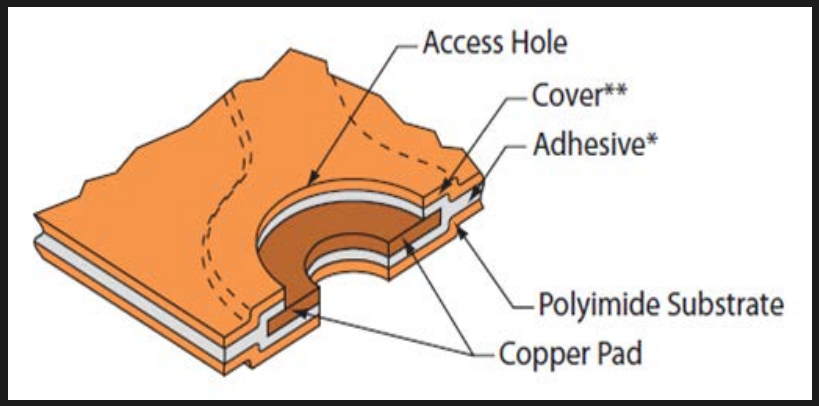
Shielding & Stiffeners



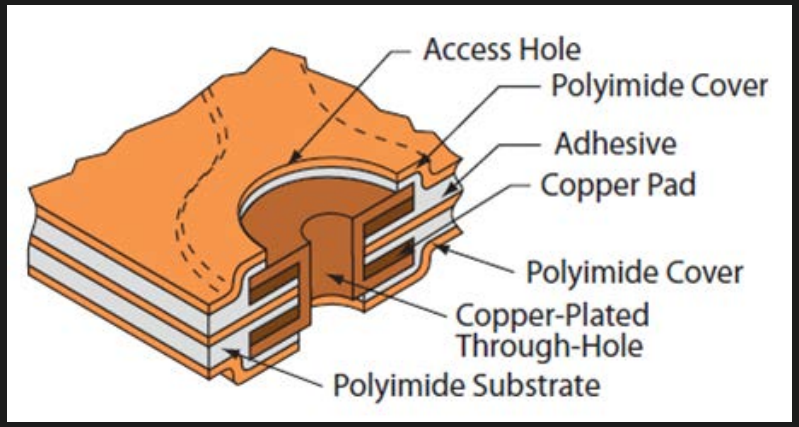


Types of Flex Circuits

- Single-layer (Type 1)

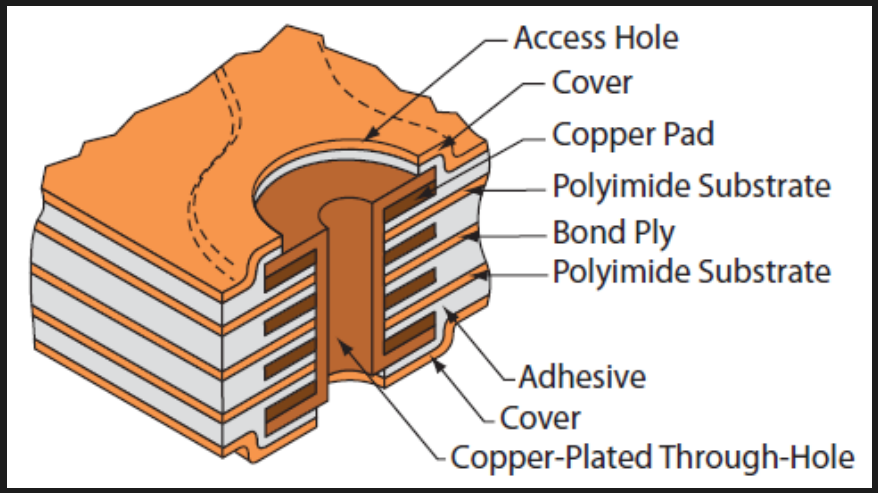


- Double-Layer (Type 2)

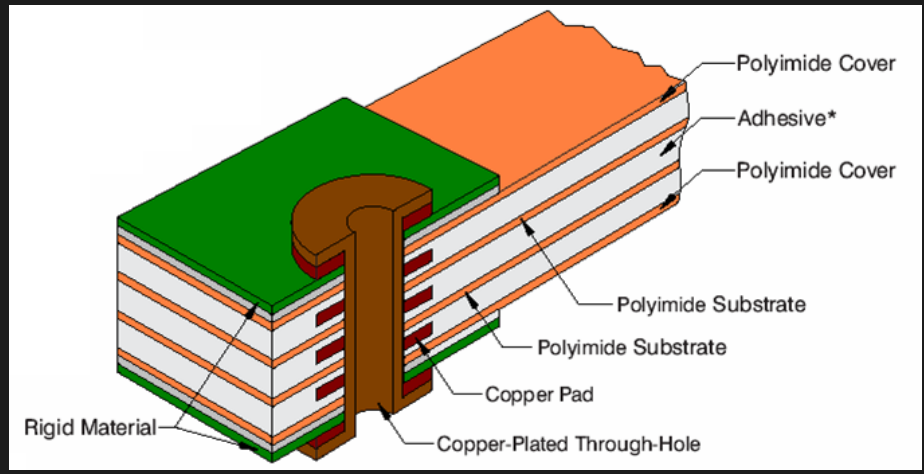


Per IPC-2223

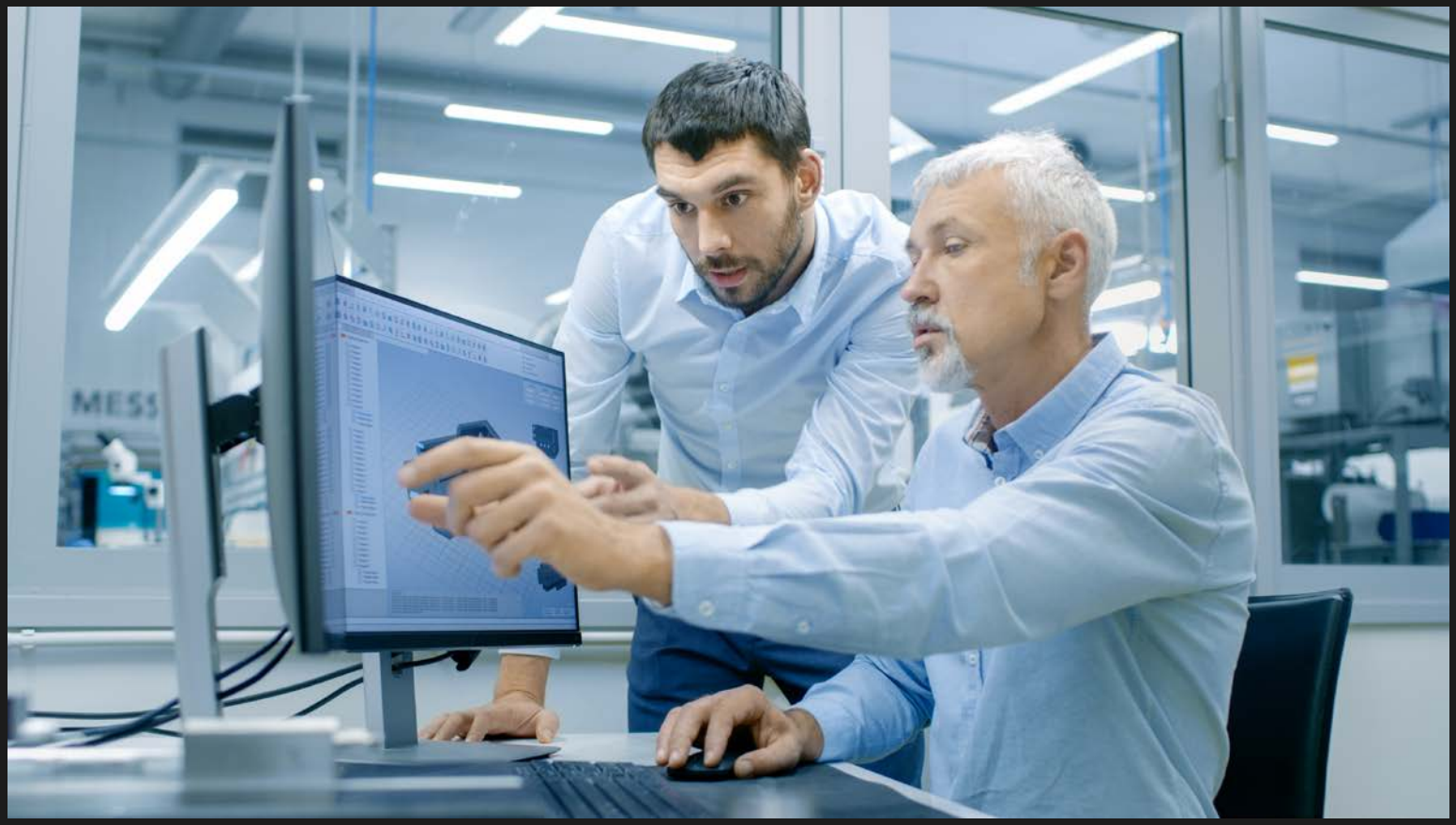
Multi Layer (Type 3)



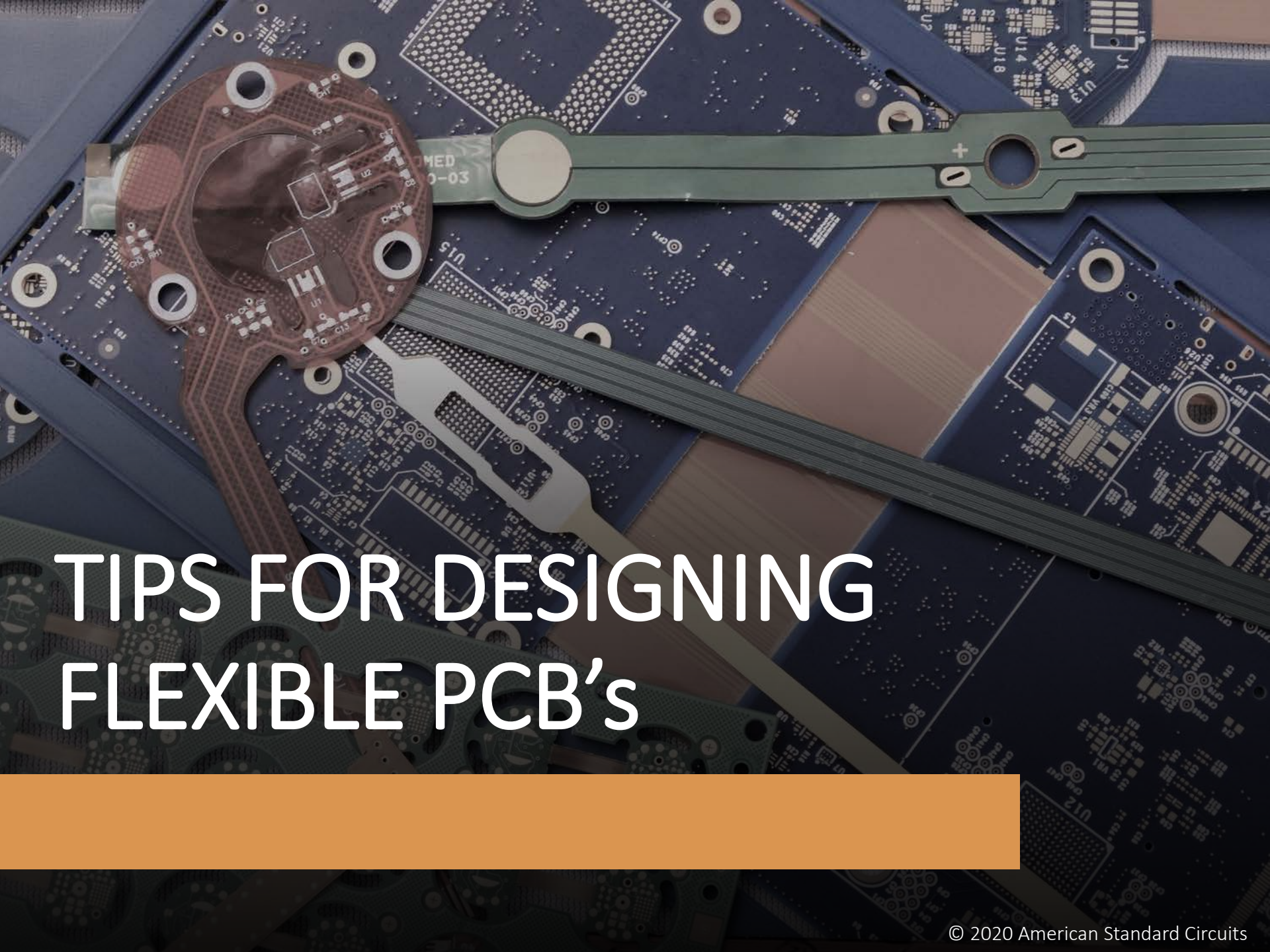
Rigid Flex (Type 4)



Our Most Important Advice



Work With Your PCB Fabricator During The Design Phase

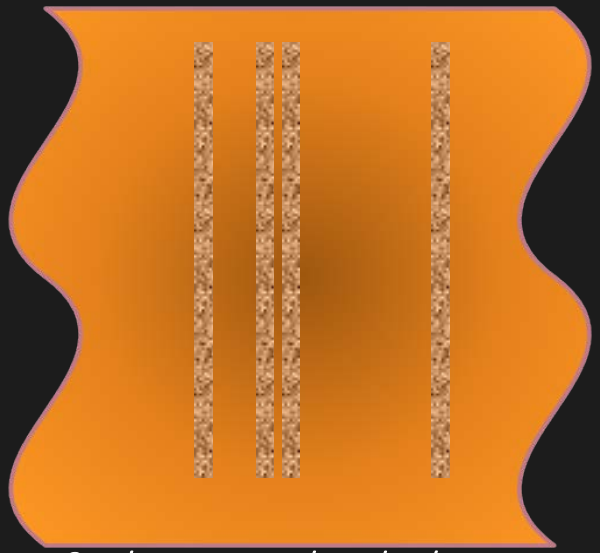


TIPS FOR DESIGNING FLEXIBLE PCB's

Routing Options For Flex

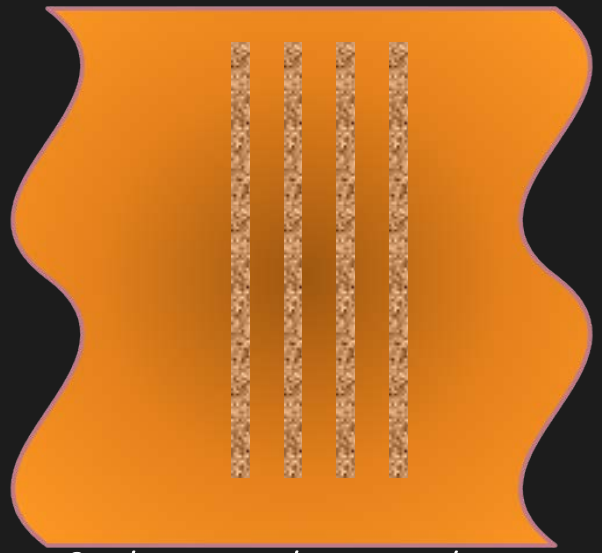
Balanced Circuit Pattern

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:

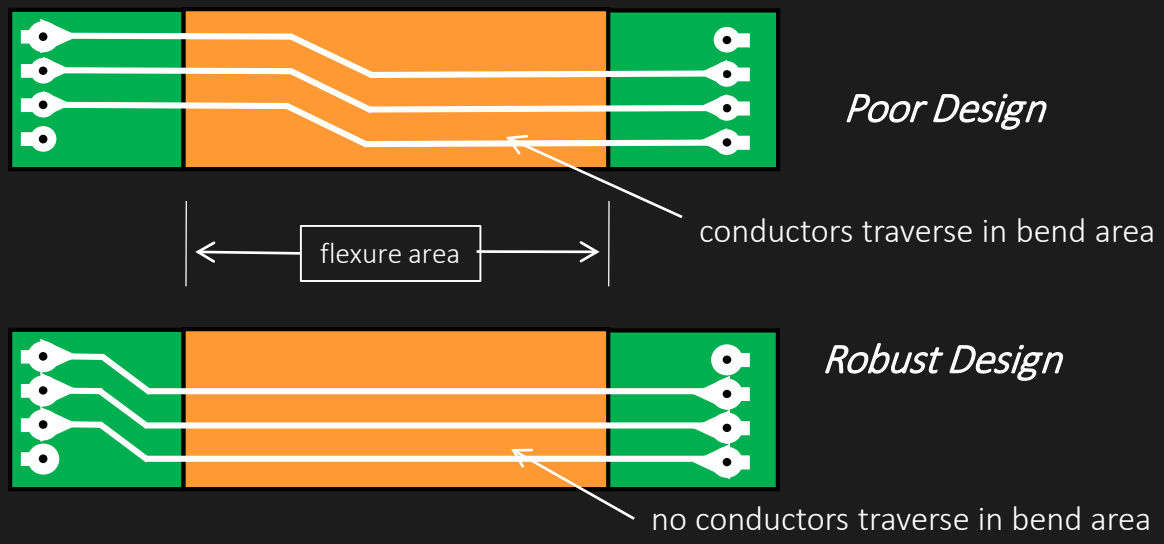
1) If the circuit is flexed perpendicular to the conductors repeatedly, stress will occur in the same location causing the isolated conductor to crack prematurely

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

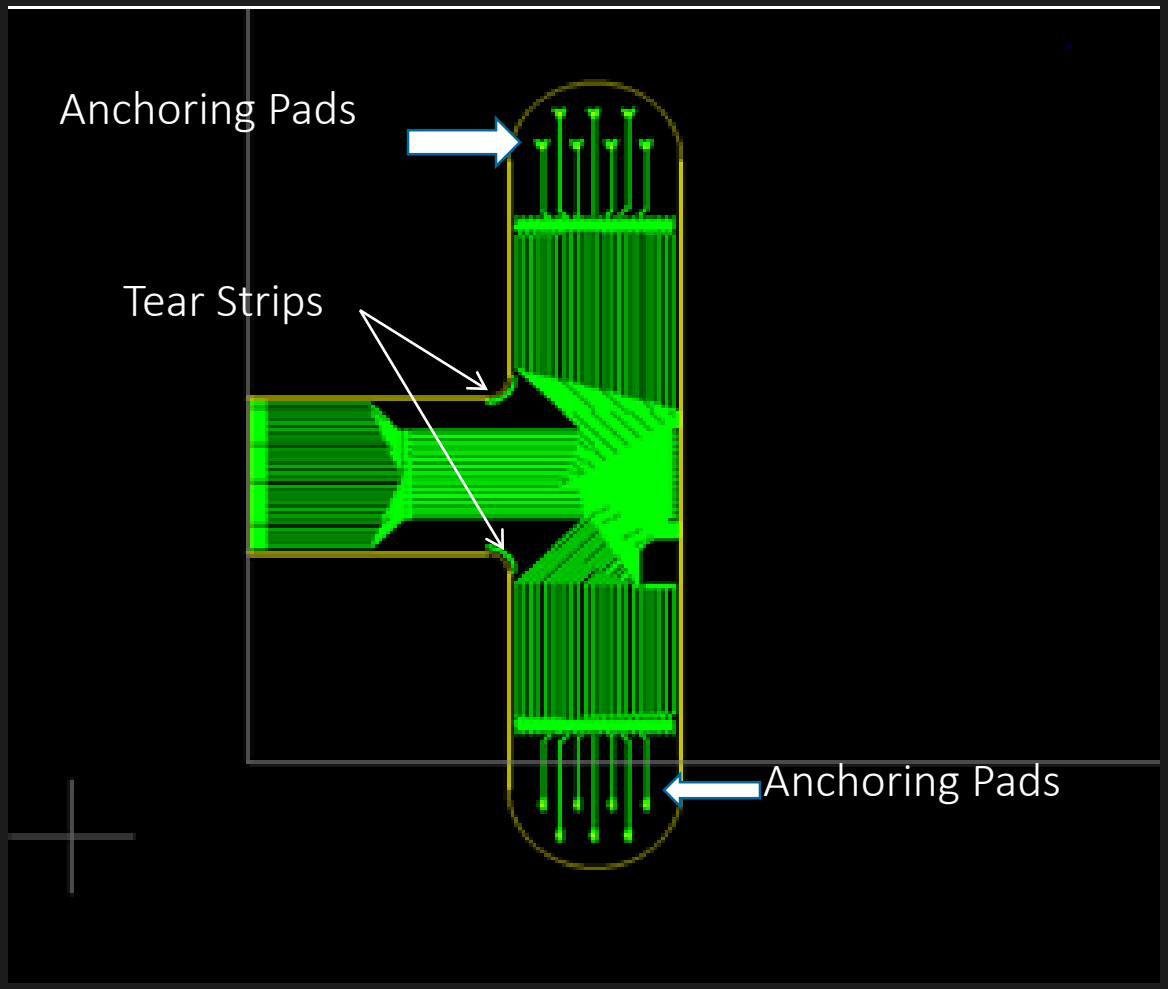
**Note: the robust design is absolutely critical in dynamic flexing applications **

Dynamic Flexing Applications



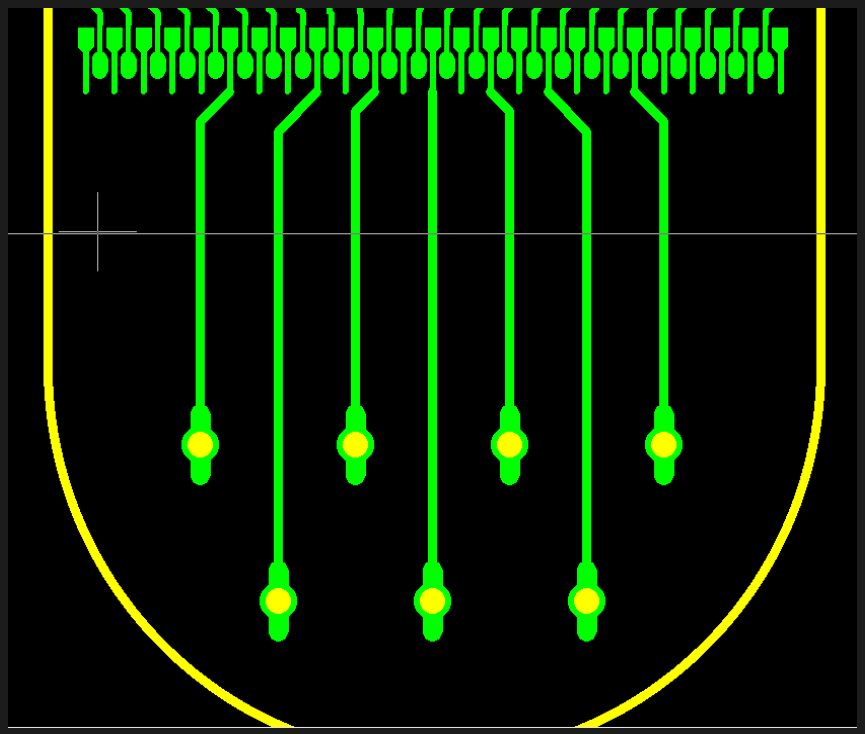
General:
All of the design guidelines apply for dynamic flexing applications. Any imperfections in artwork , materials, etched anomalies, edge roughness, etc. will cause a premature failure.

Flex Design Enhancements

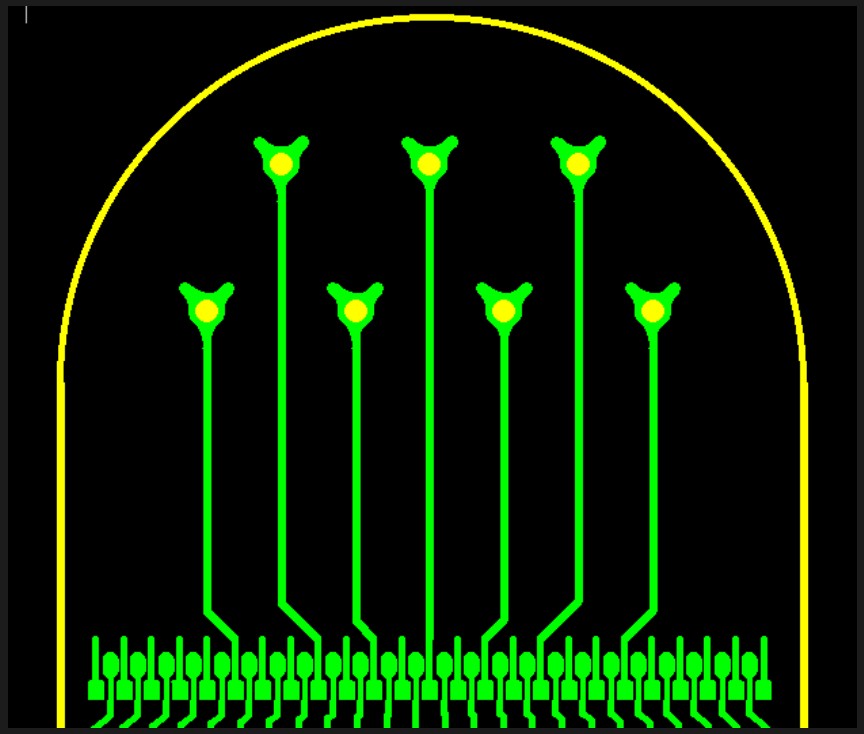


Flex Design Enhancements

Anchored Pads (Tie Downs)

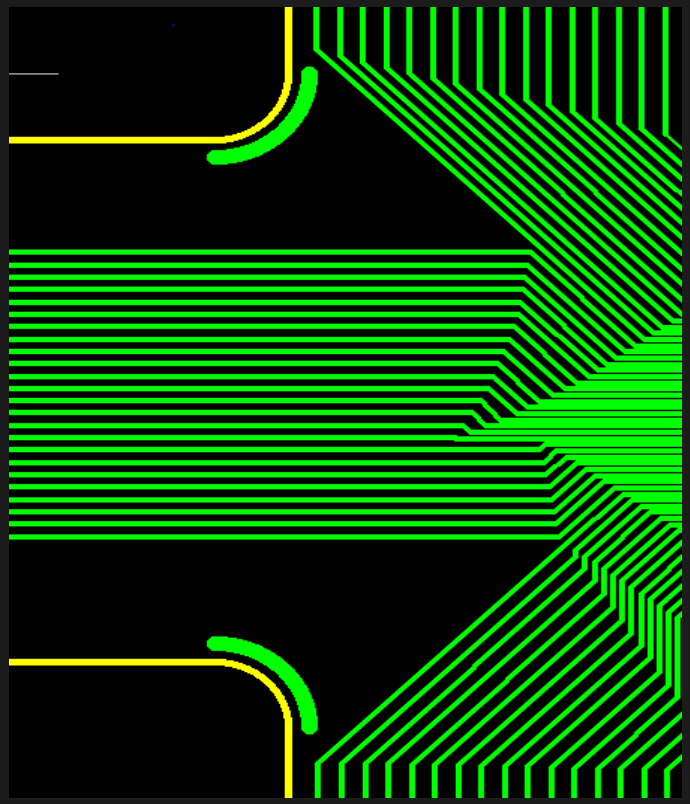


Anchored Pads (Rabbit Ears)

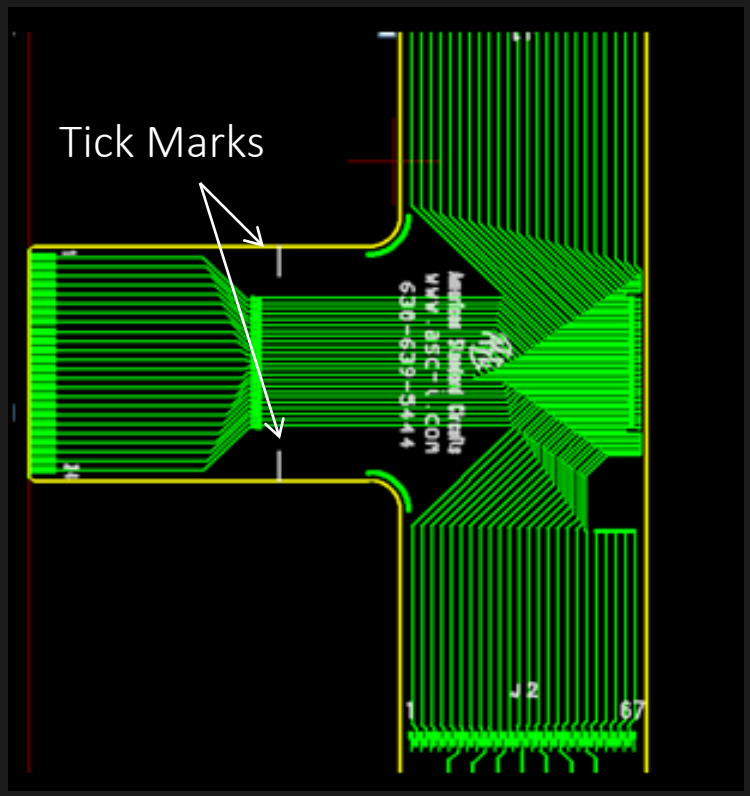


Flex Design Enhancements

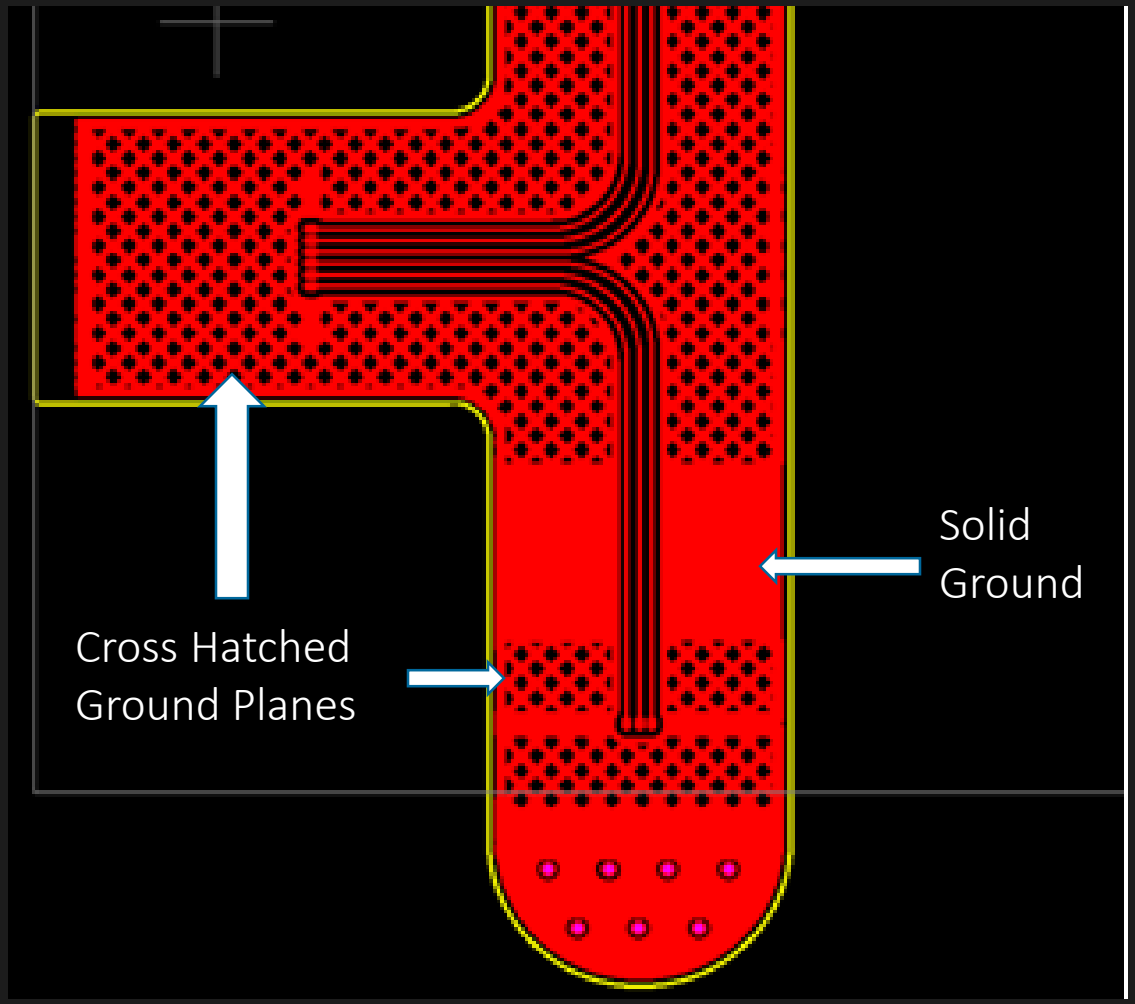
Tear Strips On Corners



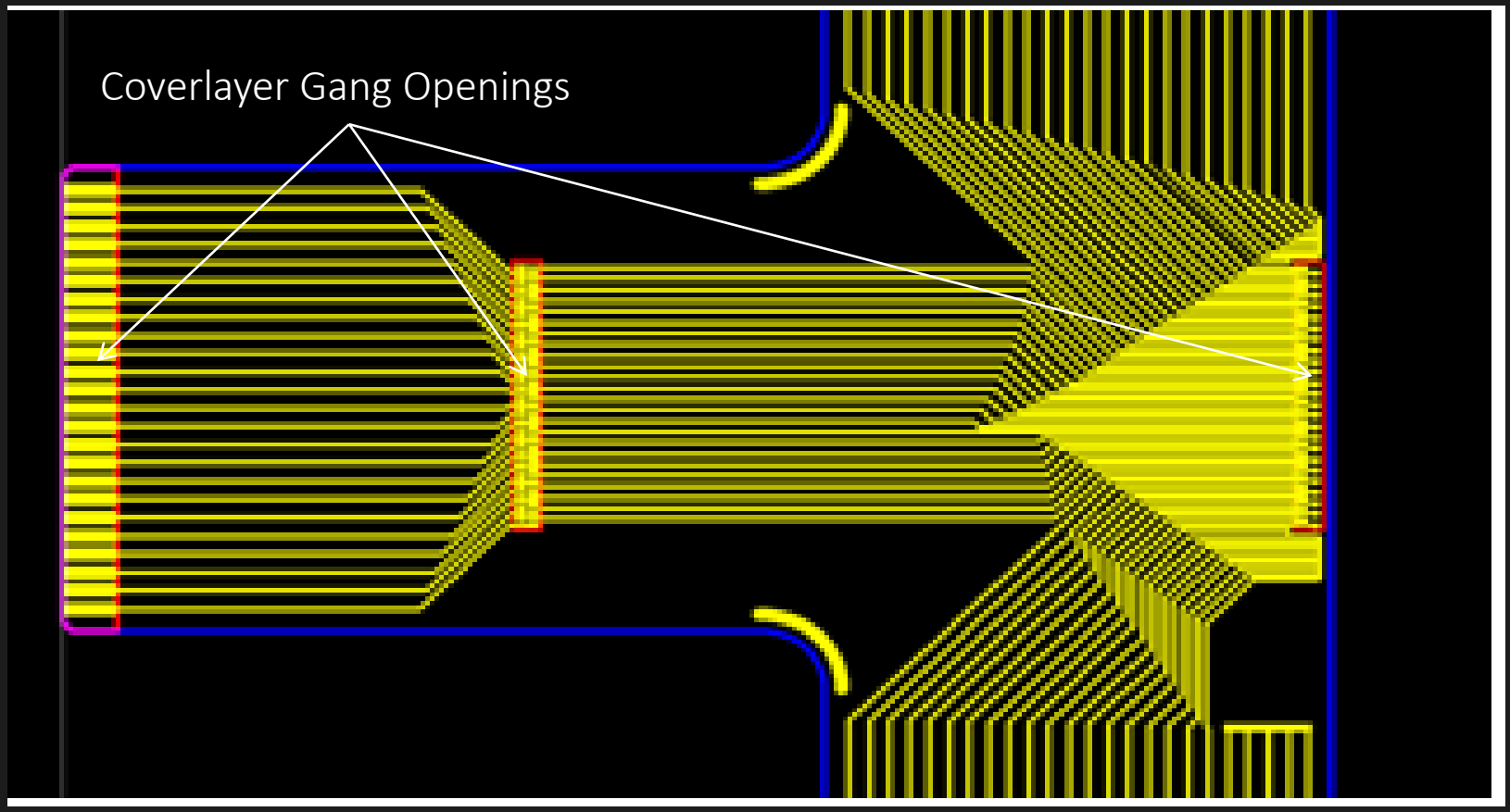
Tick Marks To Show Bend Areas



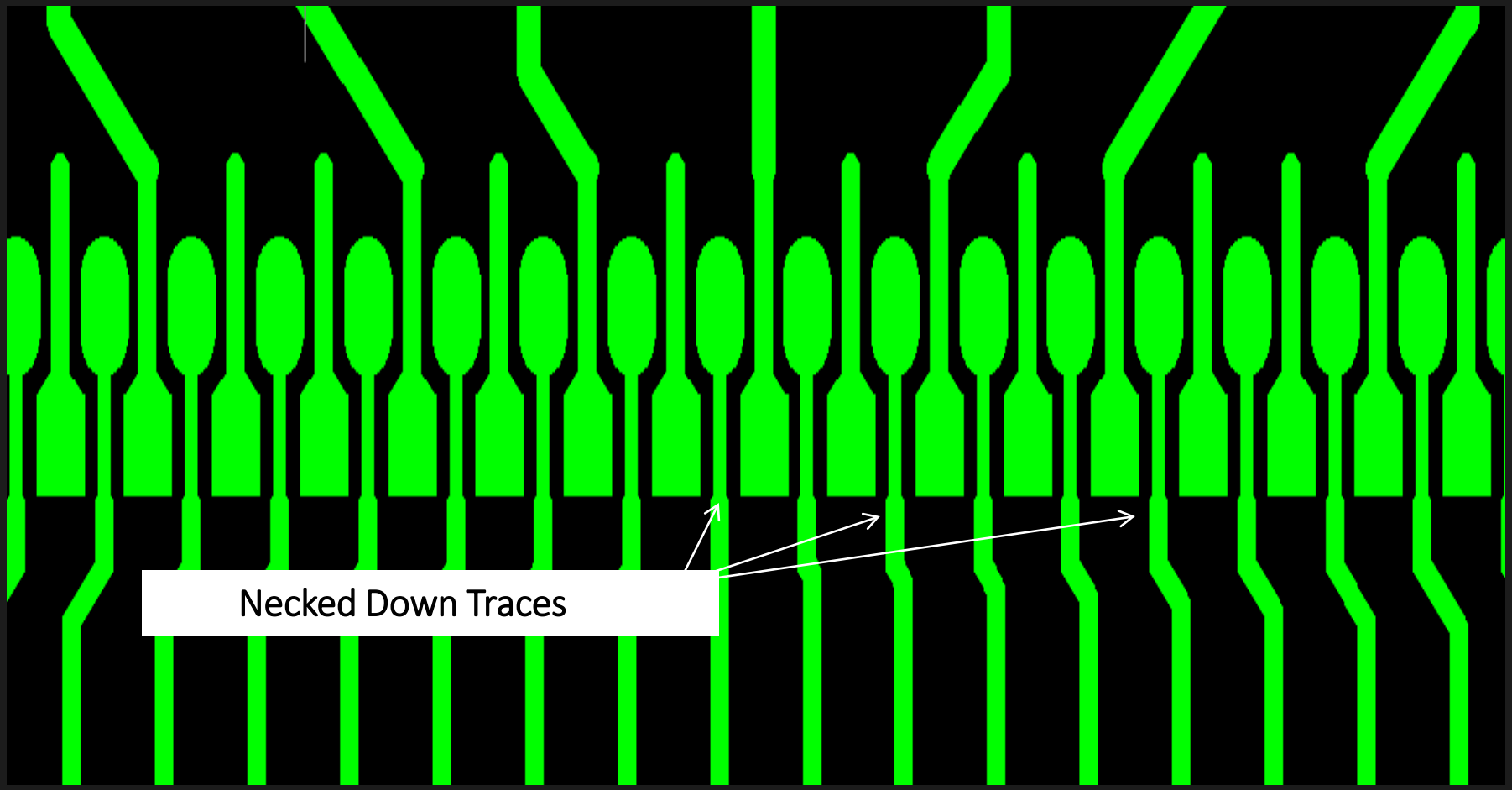
Flex Design Enhancements



Flex Design Enhancements



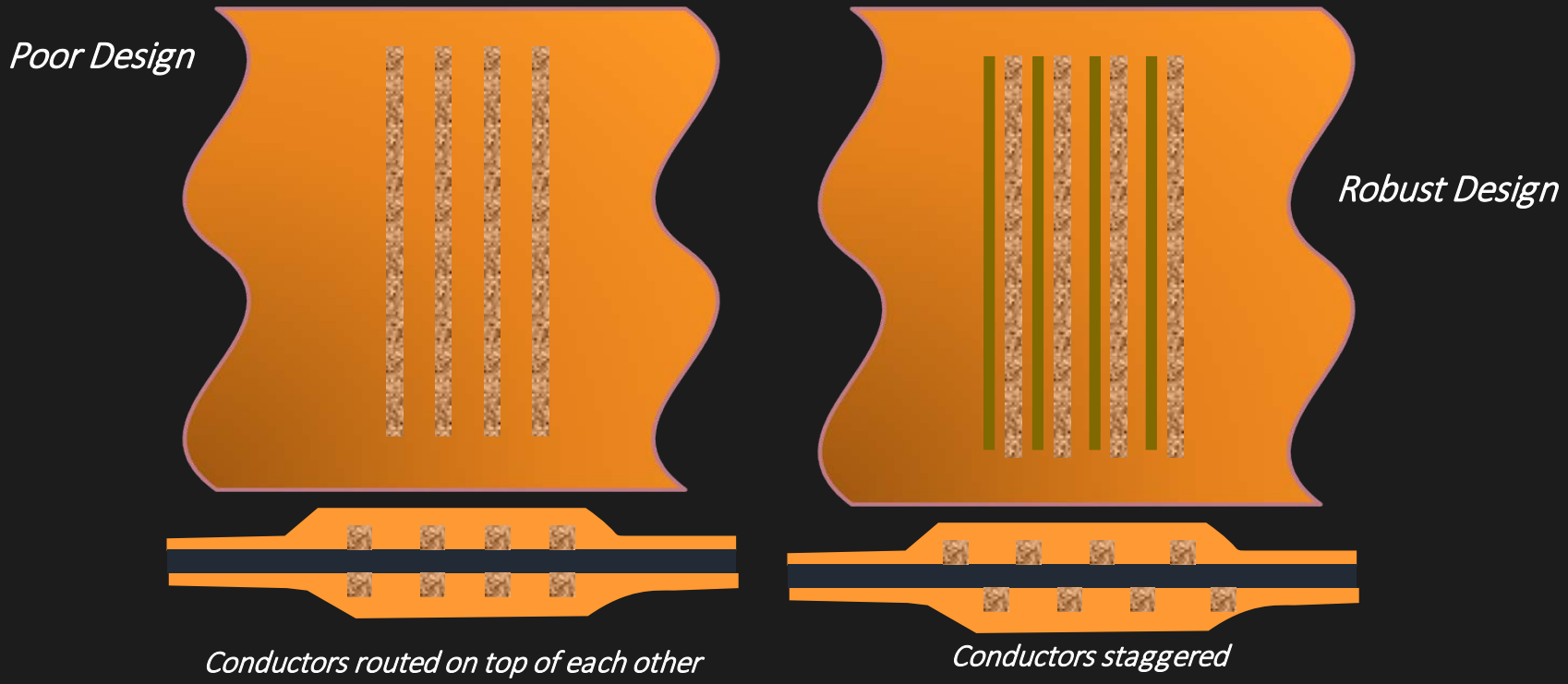
Flex Design Enhancements



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 should be slightly larger than pad 3-5 mils depending on adhesive thickness. 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:

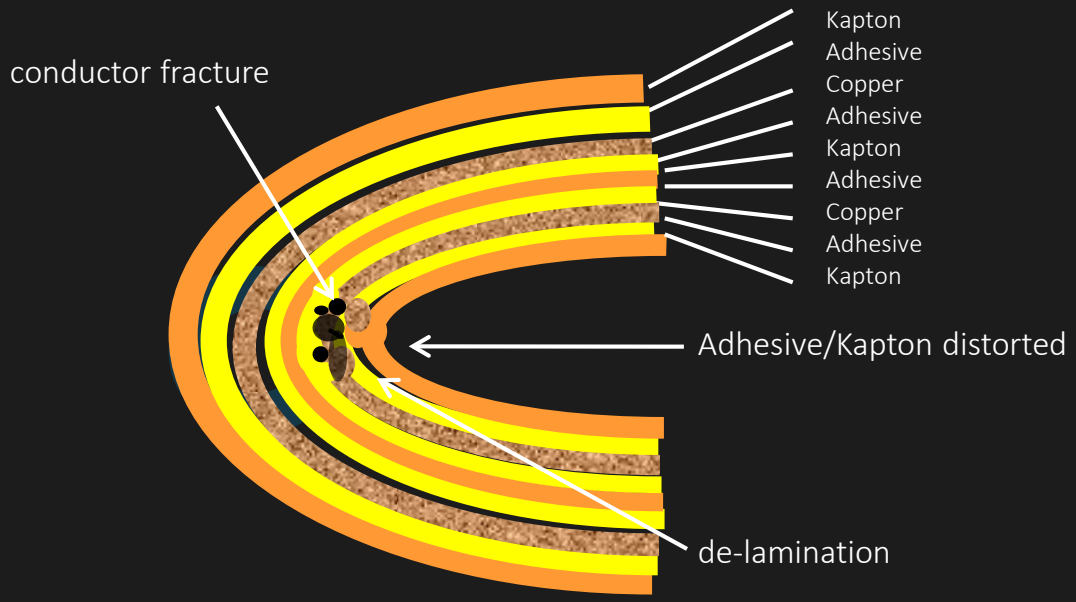
- If the circuit is flexed perpendicular to the conductors repeatedly, stresses force the copper to bend inward against the other conductor causing the conductor to crack.

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 **

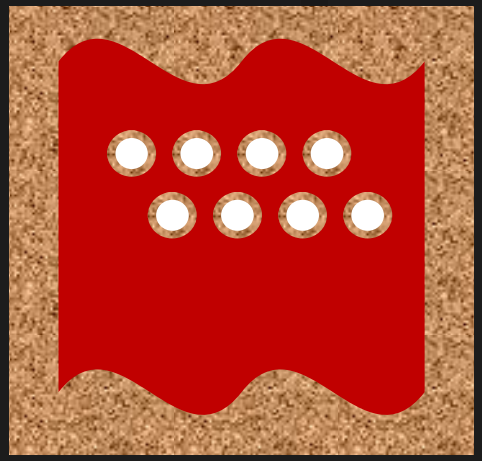
Minimum Bend Radius



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:

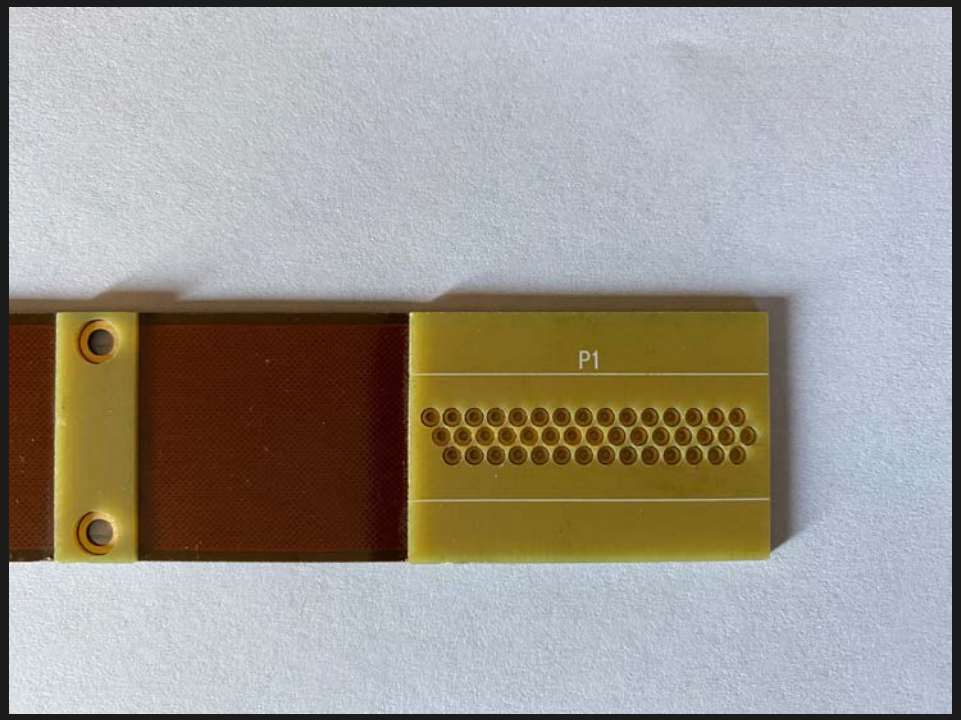
This process is used on Type 2 circuits where maximum flexibility is desired.

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

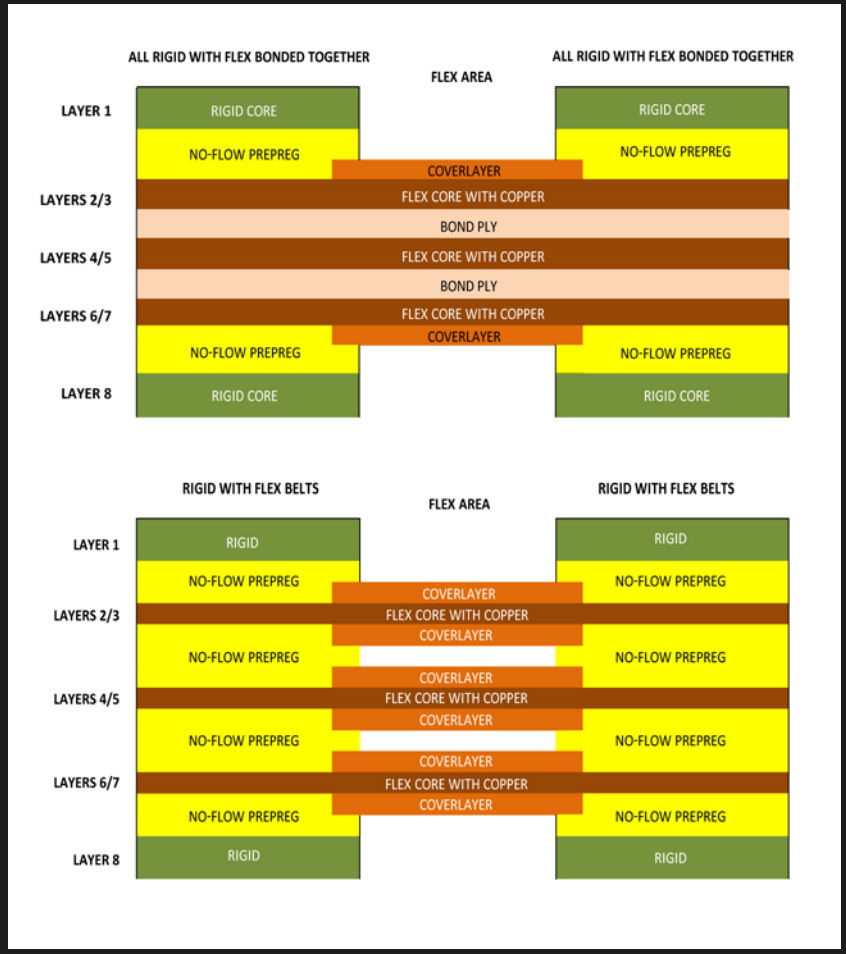
Rigid Flex/Stiffeners

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

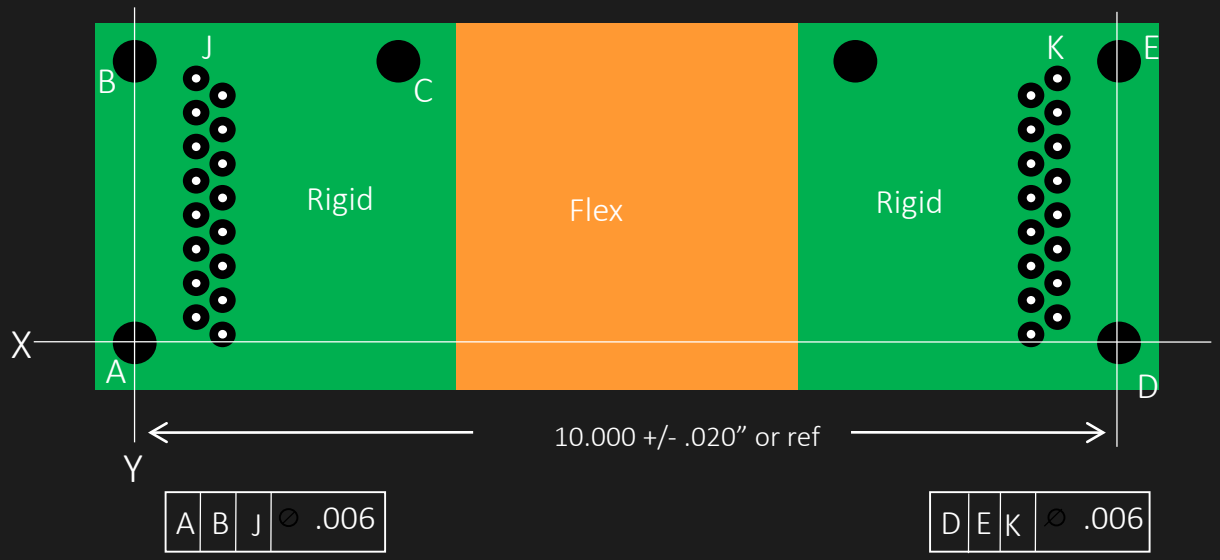


Rigid Flex

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

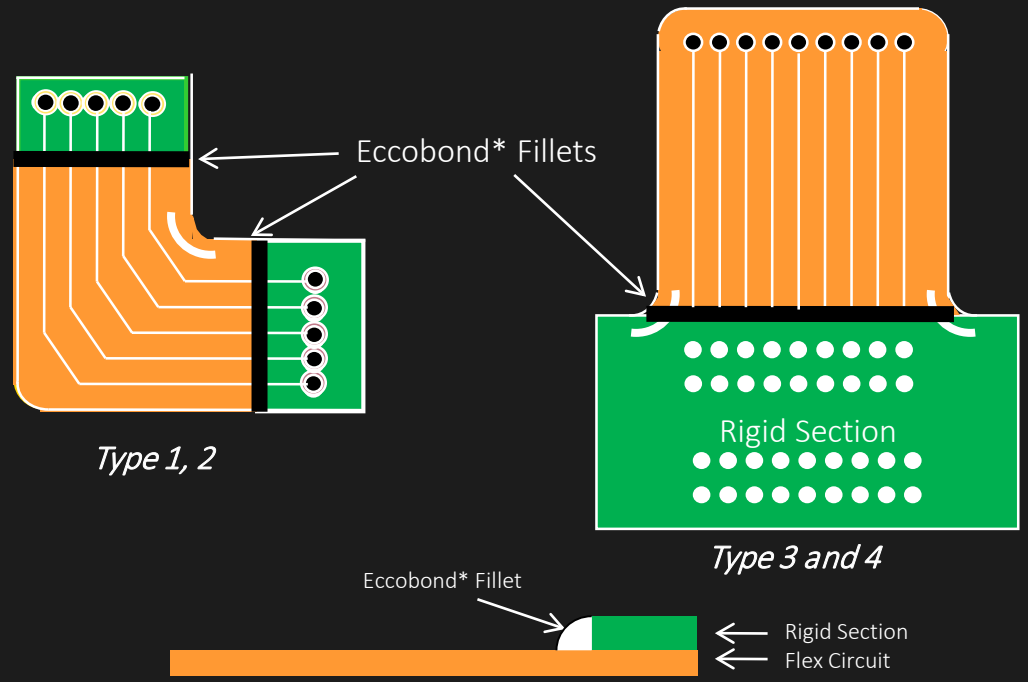


Rigid Flex Hole to Hole Tolerances



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

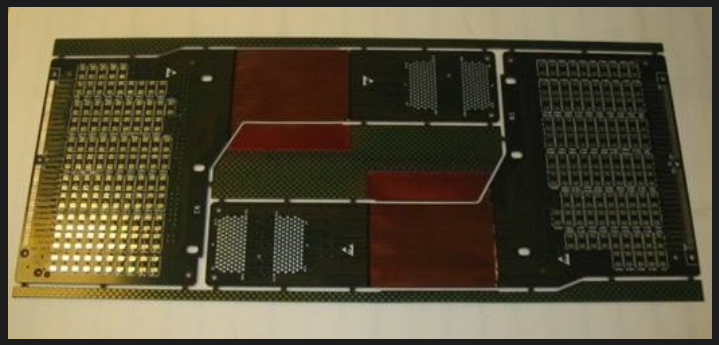


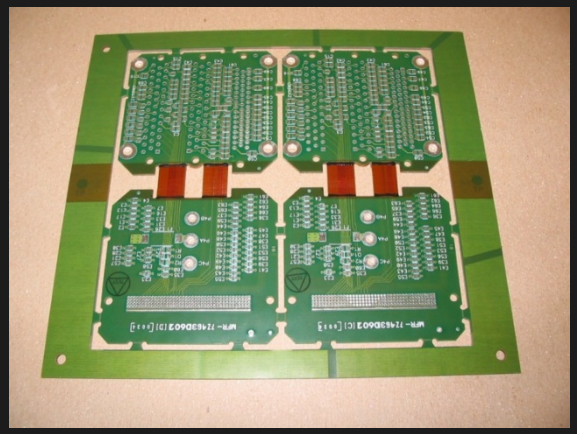
Eccobond* Fillets:

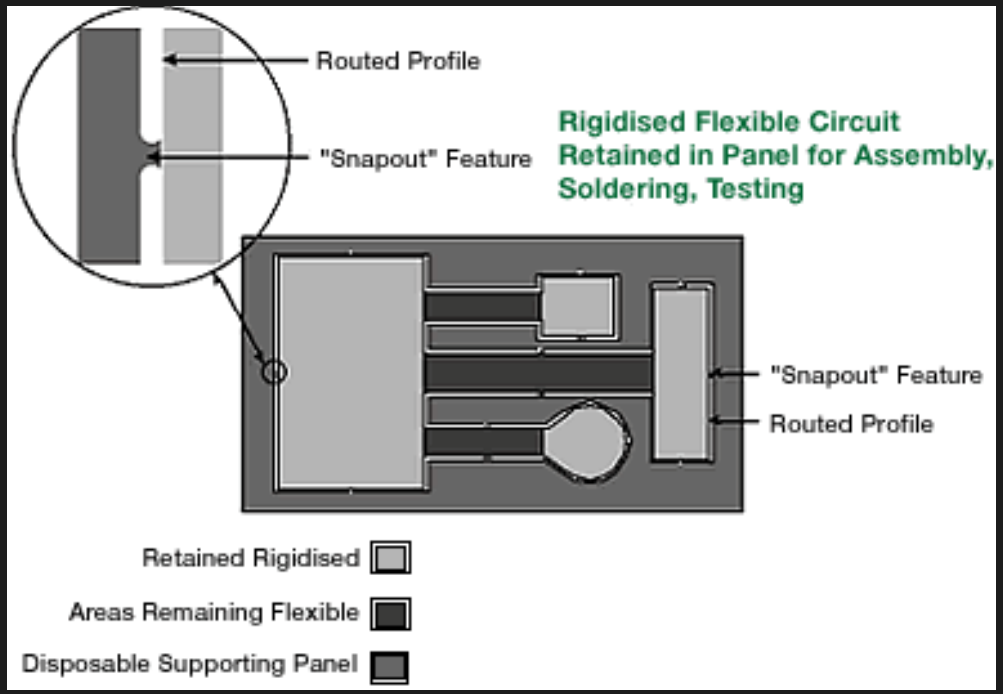
The purpose of the fillet is to prevent conductors from being cracked when the flex circuit is flexed during installation. Basically it prevents the flex circuit from being bent at this transition area. Also, In rigid/flex applications, there may be “prepreg squeeze out” at the rigid edge from the lamination 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.

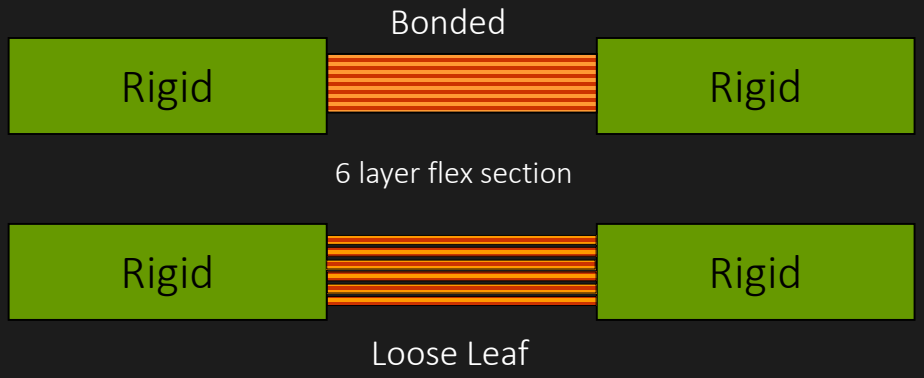


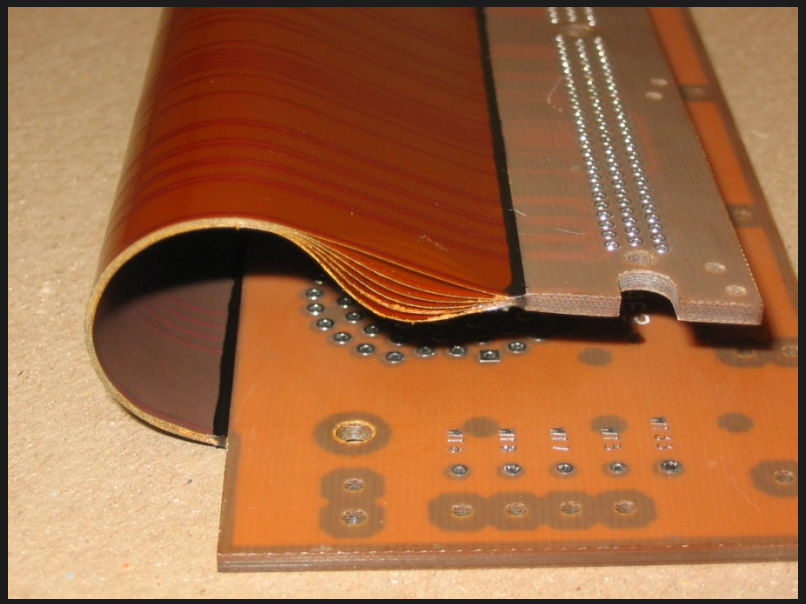




High Layer Flex in Rigid Flex

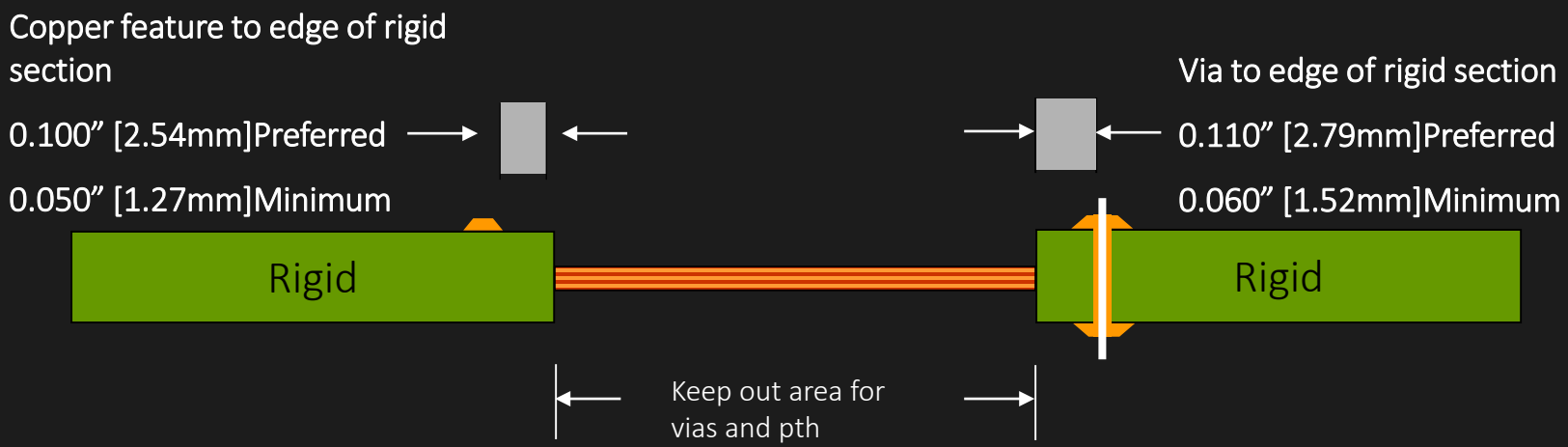
- As the flex section layer count increases, the ability to bend the flex decreases.
- Utilizing single sided flex sections increases the flexibility.





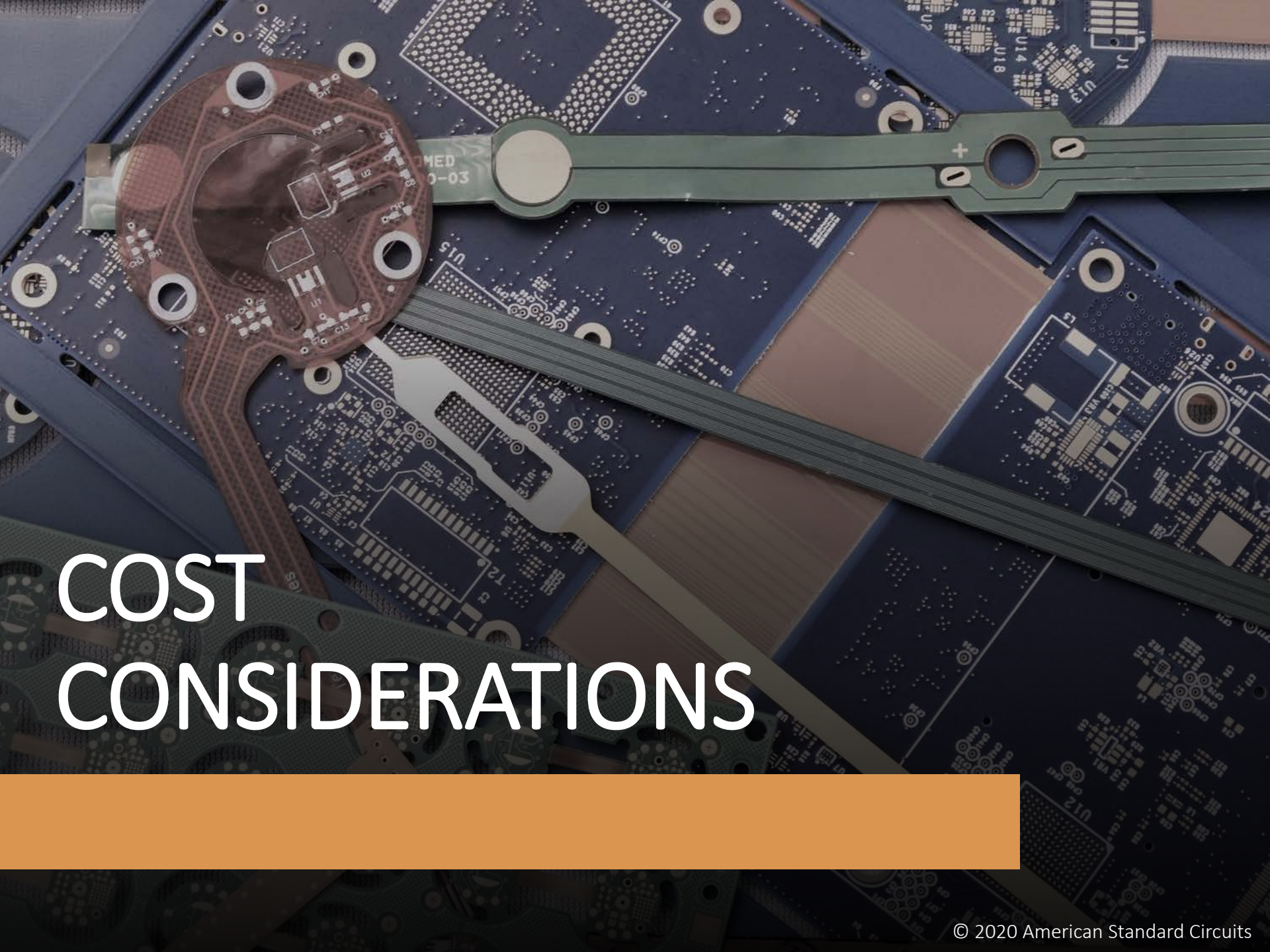
Routing/Layout Practices

- Keep Out Areas– Minimize cost.



Some Design Mistakes To Avoid

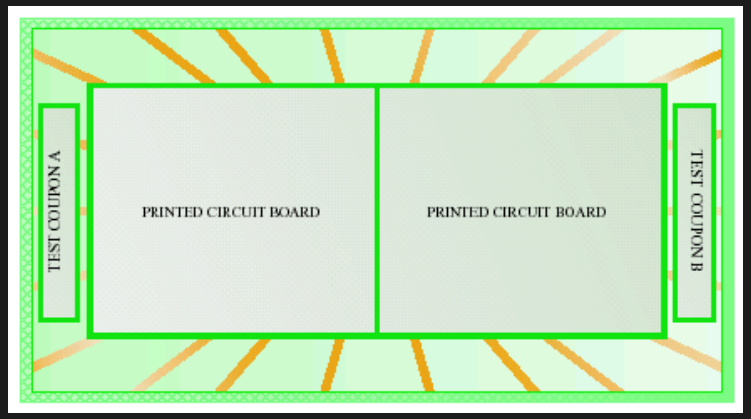
- Placing vias where the flex bends or at edge of stiffeners
  - Can cause via cracking
- Failing to add teardrops at the pad to trace interface.
  - Can lead to trace breakage
- Having sharp angles when routing traces, especially in the bend region
  - Can lead to stress riser and fracture the traces
- 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

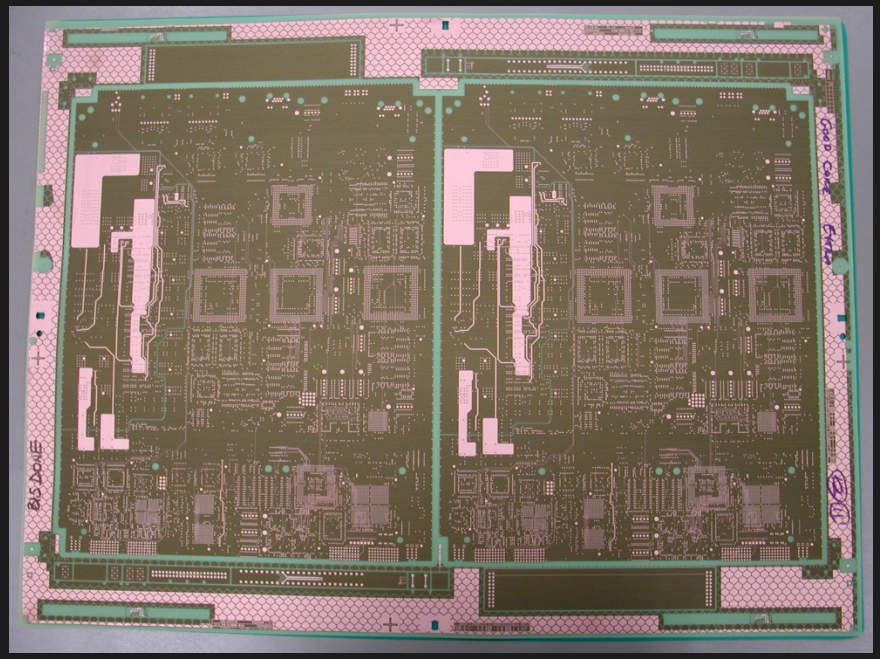


COST CONSIDERATIONS

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" border
    - 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.



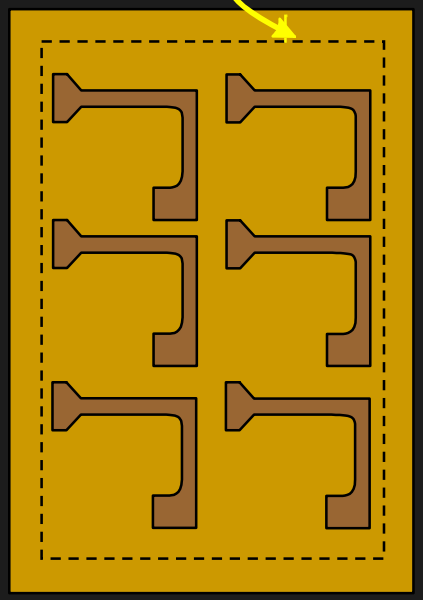


Nesting Of Circuits Improves Panel Yield

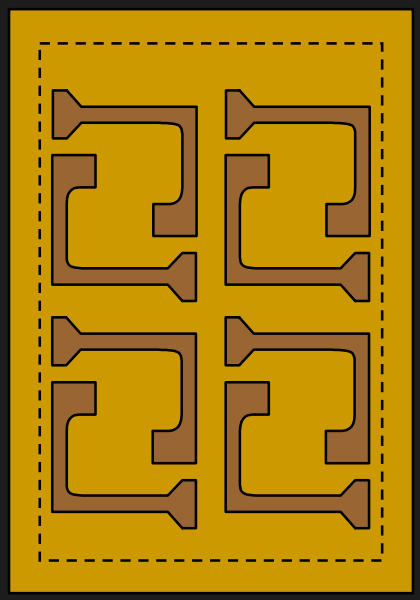
Panel size

Usable Area

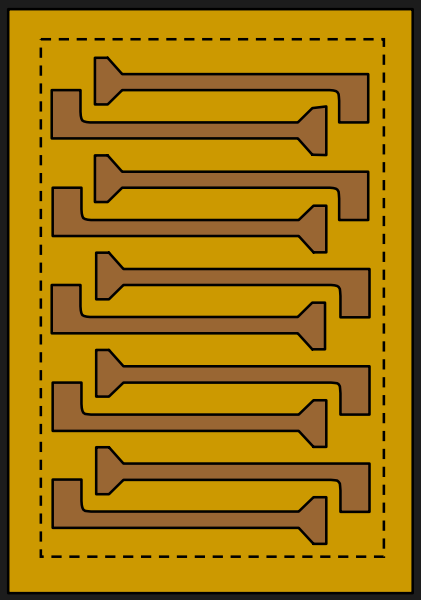
Panel sizes available	18 x 24	12 x 18
Useable area	16 x 22	10 x 16



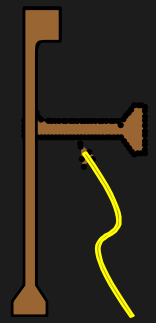
No Nesting
Panel Yield 6 parts



Circuits Nested
Panel Yield 8 parts



Optimized Nesting
Panel Yield 10 parts



Part folded to shape after punching

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)

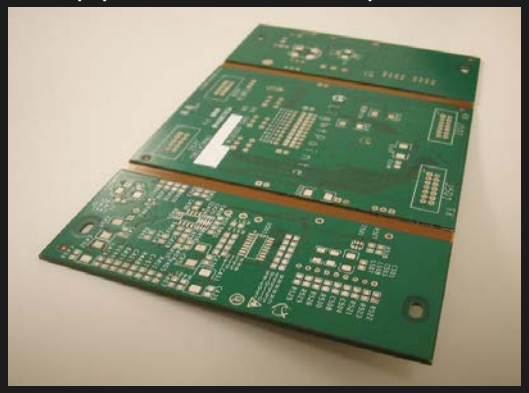
Additional **Considerations**

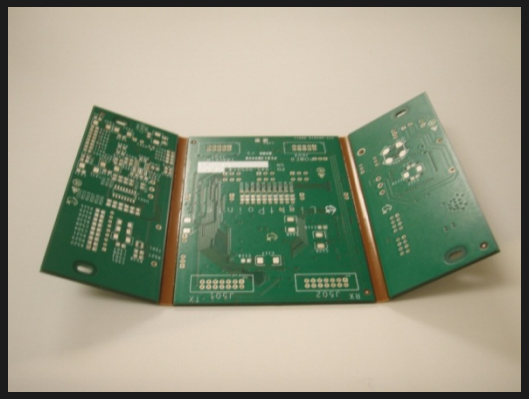
- **Cost Trade-offs**
  - Use a smaller line width/space before adding layers
  - Investigate how boards will fit into a production panel to ensure that maximum material utilization occurs.
  - Consider reliability/issues:
    - Adhesive materials are lower cost that Adhesiveless materials. The increase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

CASE STUDIES

Case Study Telecommunications Rigid-Flex 4 Layer PCB

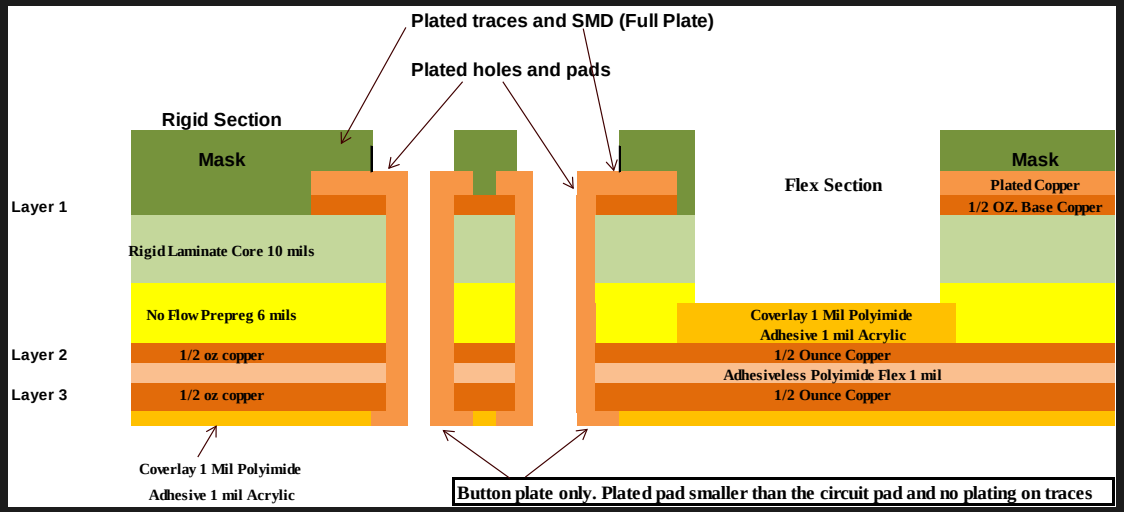
- PROBLEM:
  - Customer had 50% failure rate from existing supplier
  - Copper in Flex area was cracking due to flex area of PCB being bent several times
- SOLUTION:
  - Redesigned stack-up
  - Converted customer to adhesiveless kapton material
  - Decreased Flex Circuit thickness from 11.8 mils to 8.4 mils a 29% decrease
  - The extra thickness was adding rigidity to flex area and causing copper to crack once circuit was bent
  - Shipped over 300 parts and have had no failures





Case Study Medical Application
Rigid-Flex 3 Layer PCB

- PROBLEM:
  - 3 layer Rigid-Flex design using flex as an outerlayer
  - Required button plating to maintain flexibility in the flex region
  - Required laser cutting of the coverlayer for SMT pads
  - Tight registration required smaller manufacturing panel for processing
- SOLUTION:
  - Converted to a 4 layer rigid flex
  - Eliminated the need for button plating the flex layers
  - Eliminated the need for laser cut coverlayer
- RESULTS:
  - Material costs were higher BUT redesign resulted in a 20% savings



RESOURCES

For information on upcoming Webinars, please go to

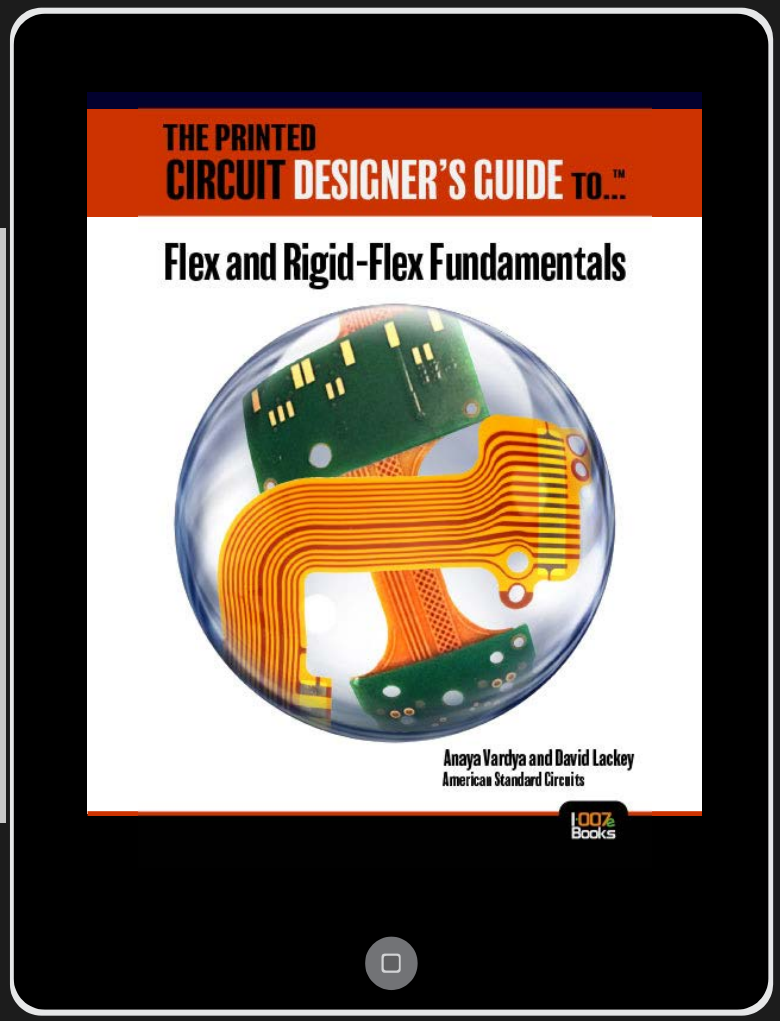
https://www.asc-i.com/webinars

Next week's Webinar Wednesday topic is Surface Finishes.

FLEX / RIGID-FLEX **E-BOOK**

"Provides designers of all experience levels with valuable and important real-world information that will help to assure first-pass success when designing flex and rigid-flex boards for today's evolving market."

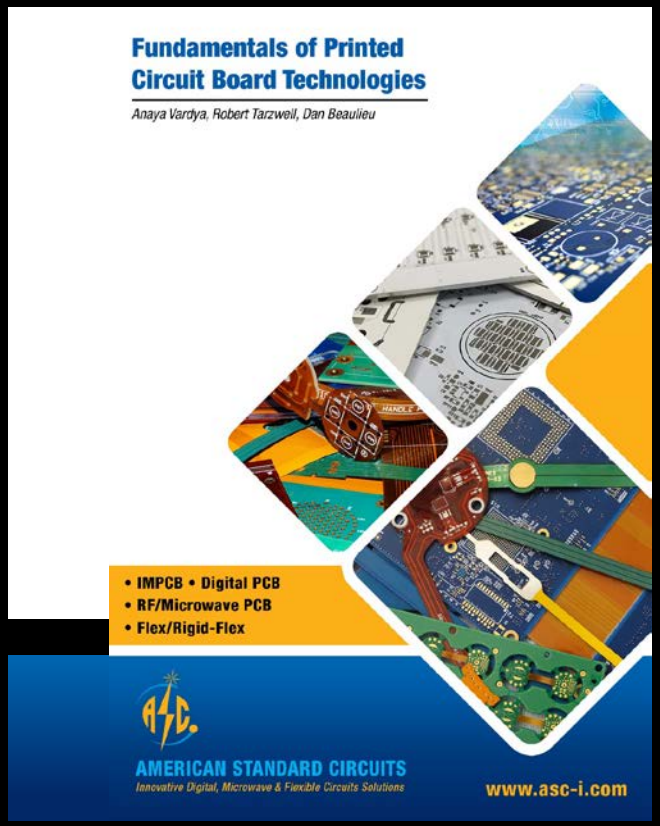
Click here for a free download.



Fundamentals of PWB Technologies
E-BOOK

"The book covers the fundamentals of manufacturing printed circuit boards, starting with simple single-sided product through extremely complicated advanced technology multilayer and hybrid boards. The book explains very complicated technology like HDI, flex/rigid flex, RF microwave and thermal management in a way that is easy to understand and comprehend."

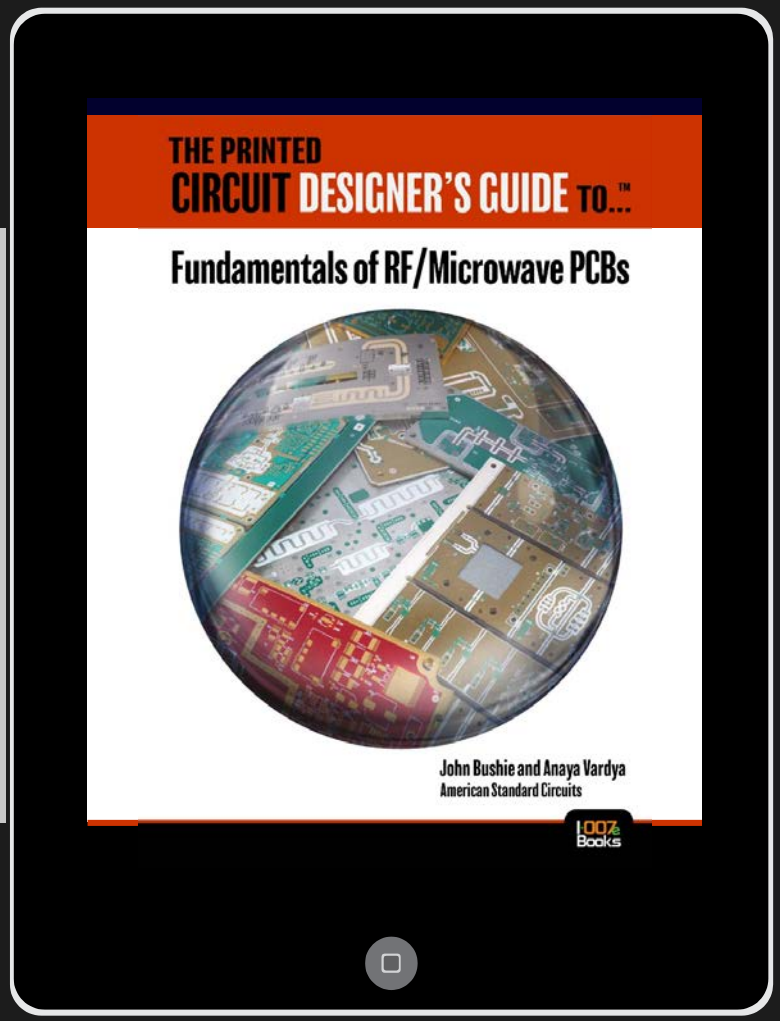
Click here for a free download.



RF / MICROWAVE **E-BOOK**

"This micro eBook provides information needed to understand the unique challenges of RF PCBs. The authors answer two main questions: what is the correct material to use for a particular project, and what can be done at the design stage to make a product more manufacturable?."

Click here for a free download.



American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

With a wide range of capabilities and a solid reputation, ASC is a total solutions provider for today's electronics industry.

- ***Digital PCB***
- ***RF / Microwave PCB***
- ***Flex / Rigid-Flex***
- ***IMPCB***
- ***Prototype to Volume Production***
- ***Domestic & Offshore Production***

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