Flex in Focus

Designers Guide to Flex PCBs

Lauren Waslick – Newgrange Design
Matthew Harms – EMA Design Automation
10/29/2024

Newgrange Design

EMA Design Automation

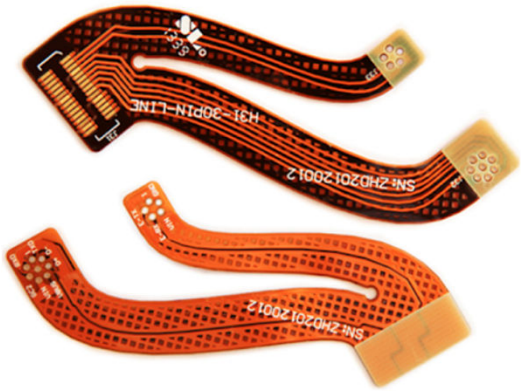
Agenda

- Types of Flex Boards
- Key considerations throughout the flex design process:
  - Mechanical
  - Footprinting
  - Placement
  - Routing
  - Rules
- Early Flex Fabricator Involvement
- Key Takeaways

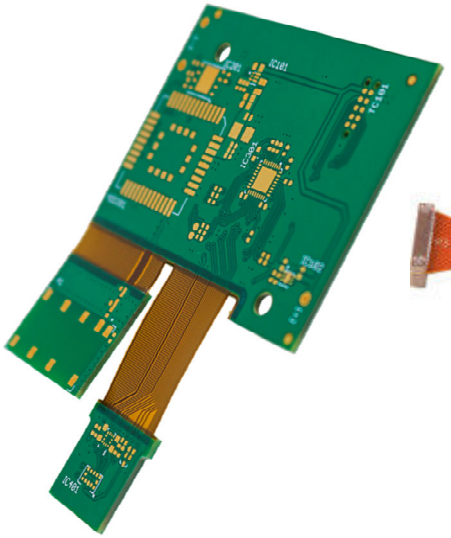
Types of Flex Boards

Flex

Flex with Stiffener



Rigid-Flex



Combination

IMAGES:
https://www.multi-circuit-boards.eu/en/products/printed-circuit-boards/flexible-pcb.html
https://www.gtpcb.com/sale-40891901-automotive-flexible-pcb-board-with-blue-solder-mask-and-white-silkscreen.html

IMAGES:
https://www.epectec.com/flex/industries-market-focus/flex-and-rigid-flex-pcbs-for-itar-applications.html
https://www.glenair.com/flex-circuit-assemblies/index.htm

Should I Use Flex in My Design?

When considering whether or not to utilize a flex or rigid-flex board in your design, ask yourself the following:

Question	Consideration
Does my timeline allow for a longer fabrication?	While some rigid boards can be fabricated in just a few days, most flex or rigid-flex will take 2-3 weeks.
Does my budget allow for a higher cost fabrication?	The custom fabrication process and specialized materials can increase costs up to 5-10X as compared to rigid boards.
What stage of development am I in?	When proving out a design in prototype phase, the cost and time saving of rigid may be worth it, even if you re-spin to be rigid-flex in a future development cycle.
Does flex suit the electrical needs of my design?	When dealing with high power or sensitive signals, flex may limit the ability to meet design requirements. This should be reviewed early in the process.

Key Considerations Throughout the Flex Design Process

- Mechanical
- Footprinting
- Placement
- Routing
- Rules

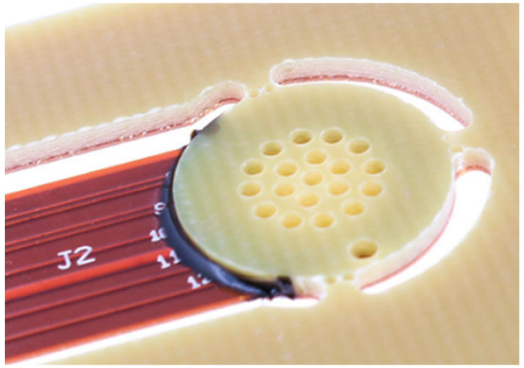
Mechanical

Questions to ask about board mechanicals:

- Is the rigid board size fixed or can it be adjusted?
- Is the flex size/width fixed or can it be adjusted?

- Where are the bend lines for the flex?
  - This requires knowing a little about the final assembly but can impact placement and routing.

- Is the size/shape of the stiffener clear?
  - Ideally, it touches at least two edges of the flex.
  - It should exist in all flex areas that have components installed.

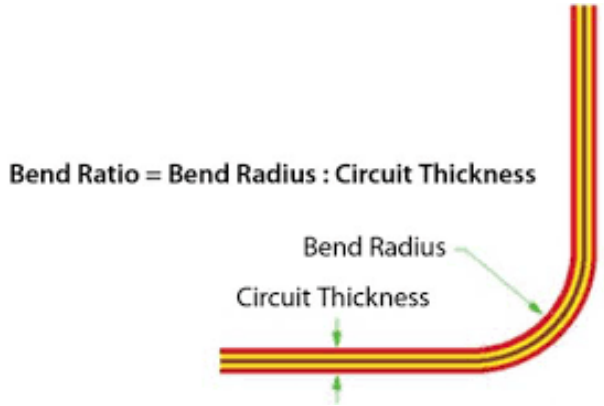


IMAGES:
https://www.epectec.com/flex/flex-circuit-stiffeners.html

Mechanical

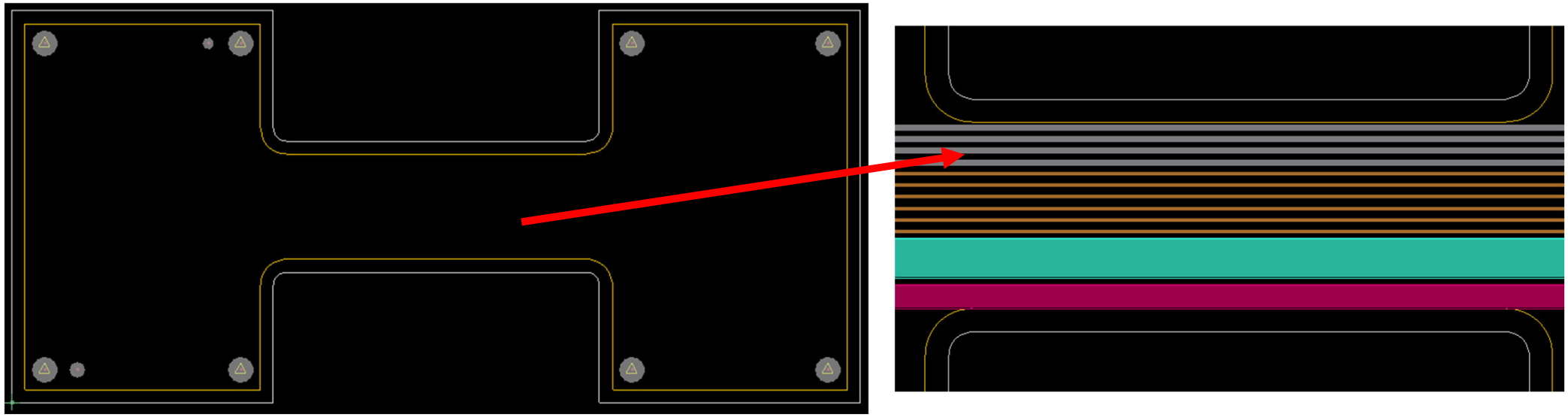
Questions to ask about board mechanicals:

- How is board going to be installed?
  - Dynamic flex: bending multiple times
  - Static flex: bending once and being locked into place
- What is the required bend radius?
  - Consult IPC-2223 for bend radius standards
  - ~6X flex width for 1-2 layer designs
  - ~12X flex width or more for 3+ layer designs



IMAGES:
https://www.minco.com/wp-content/uploads/Minco_BasicsofFlexDesign.pdf

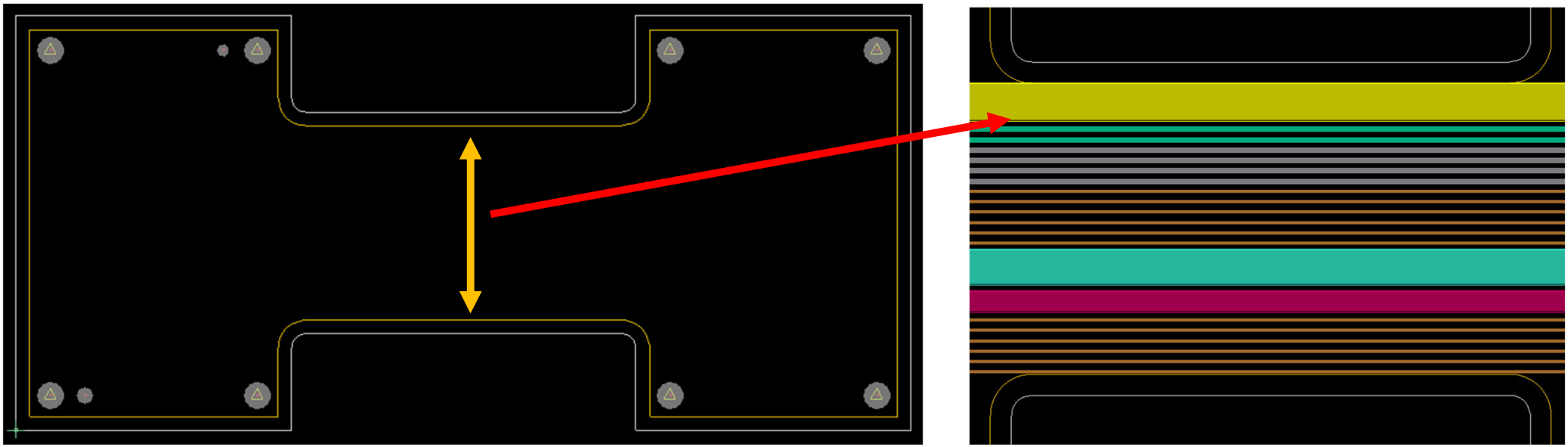
Real-world example



- Design Update! More circuitry is added on both rigid sections, causing new signals and more power needed across the flex…. What should you do?

Real-world example

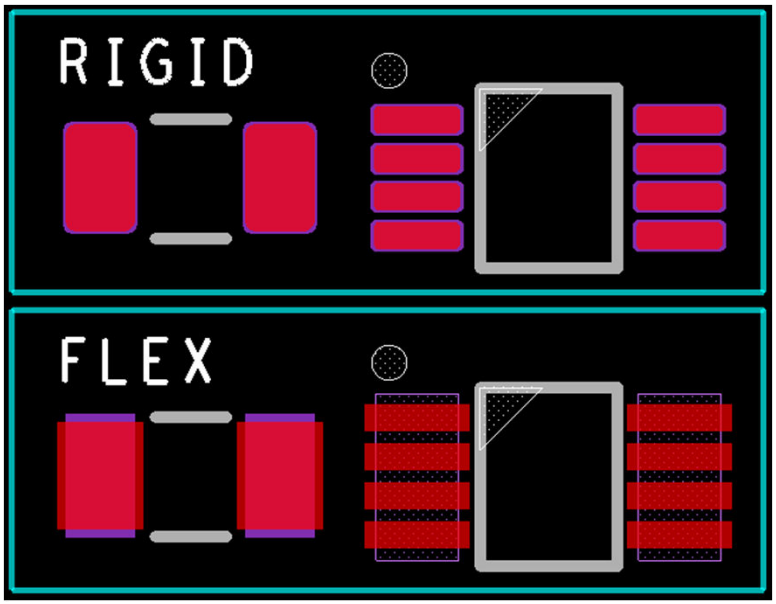
- The bend radius was critical, so adding flex layers might have jeopardized the mechanical stability….
- Instead, we increased the width of the flex.

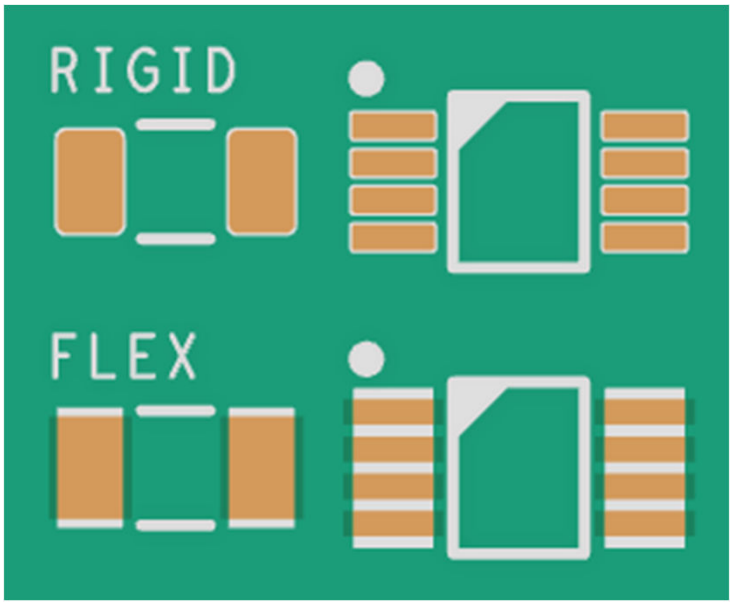


Footprinting - SMT

Surface Mount footprints placed on **FLEX** regions:

- SMD pads
  - Anchor pads by having the coverlay over the pads on at least two sides
- Coverlay sliver
  - 10 mil sliver required with coverlay as opposed to 3-4 mil sliver with soldermask
  - Gang relief is more common on flex

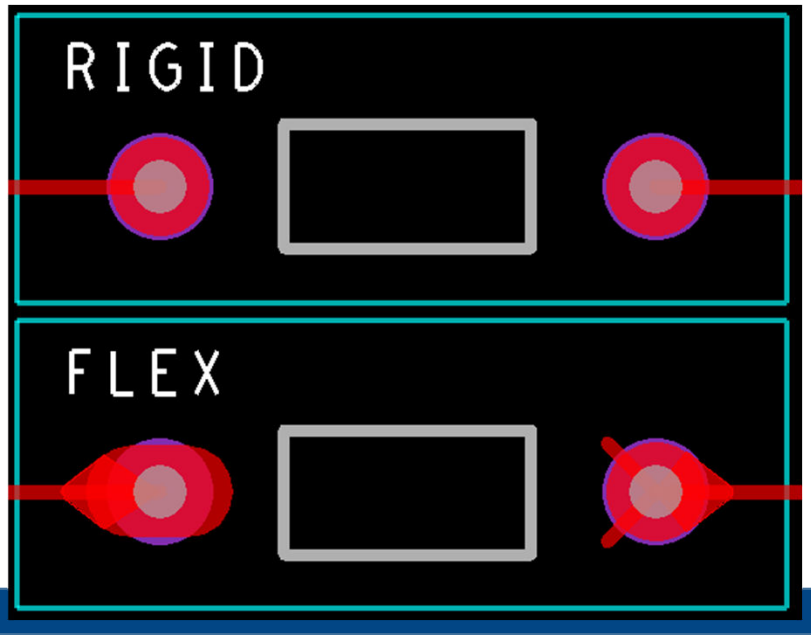


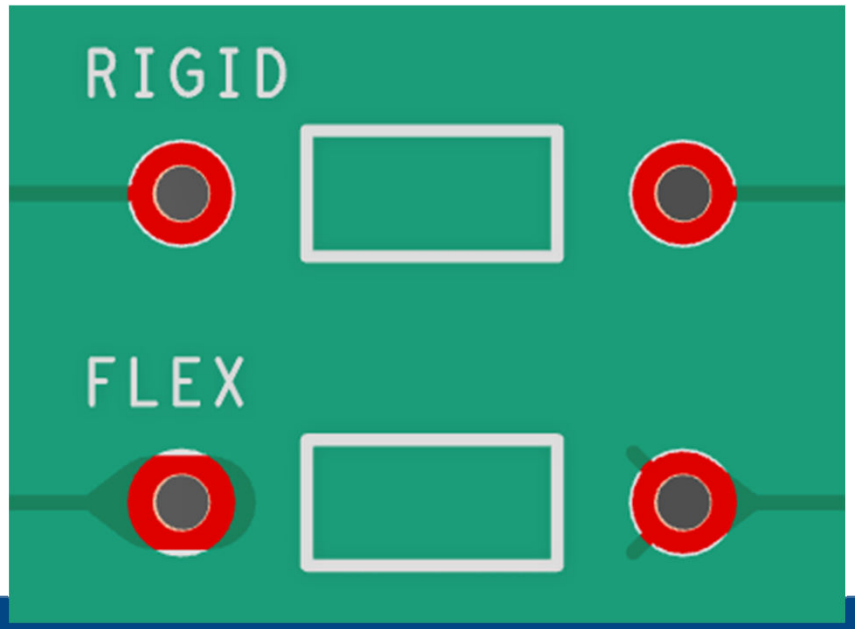


Footprinting - TH

Through Hole footprints placed on **FLEX** regions:

- TH pads
  - Min 30 mil annular ring oversize (15 on each side)
  - Anchor pads with traces and/or teardrops

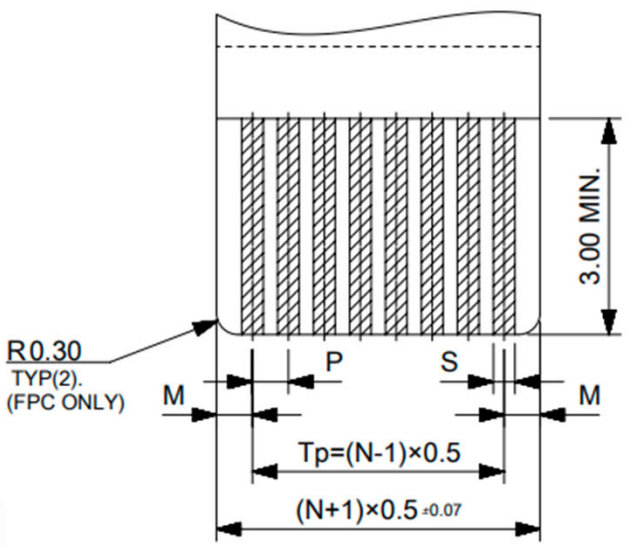


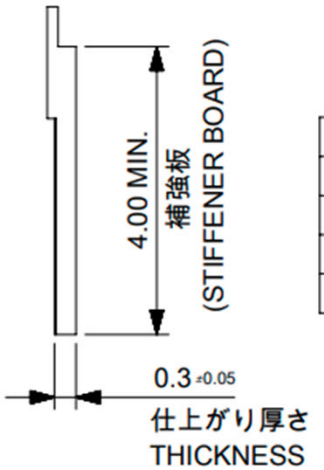


Footprinting – ZIF Connector

ZIF Connectors on **FLEX** regions:

- Coverlay should be mostly removed, but should anchor the end of the pads
- Stiffener thickness is critical for installation
- Copy copper, coverlay & stiffener exactly from datasheet
- Example:
  - Molex 781272210





APPLICABLE FPC/FFC
RECOMMENDED DIMENSION

S	0.30 $^{+0.05}_{-0.02}$	0.35 $^{+0.05}_{-0}$
P	0.50±0.05	0.50±0.05
M	0.50±0.08	0.50±0.10
Tp	Tp±0.05	Tp±0.05
	FFC	FPC

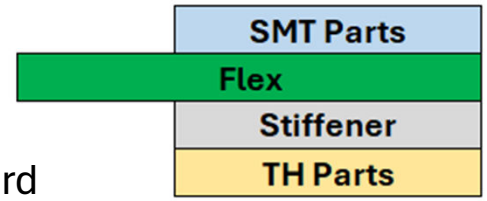
IMAGES:
https://www.molex.com/content/dam/molex/molex-dot-com/products/automated/en-us/salesdrawingpdf/781/78127/781272210_sd.pdf?inline

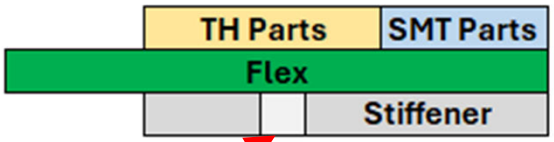
Placement

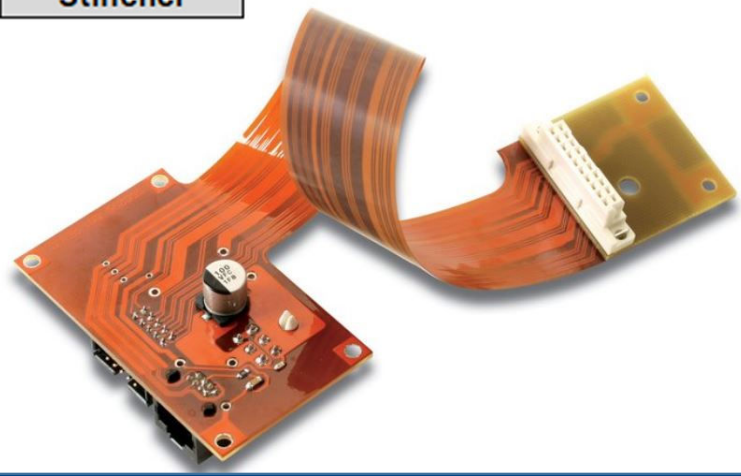
Placement in stiffener regions:

- SMT Parts
  - Stiffener should be on the **opposite side** of the board as the components.
- TH Parts
  - Stiffener should be on the **same side** as the components to allow bottom side soldering
- Mix SMT/TH on same side
  - Stiffener should have holes **larger than** the exposed pad of a TH component to allow bottom side soldering on an area with SMT/TH components

- Absolutely no parts should be placed on or near a bend line.





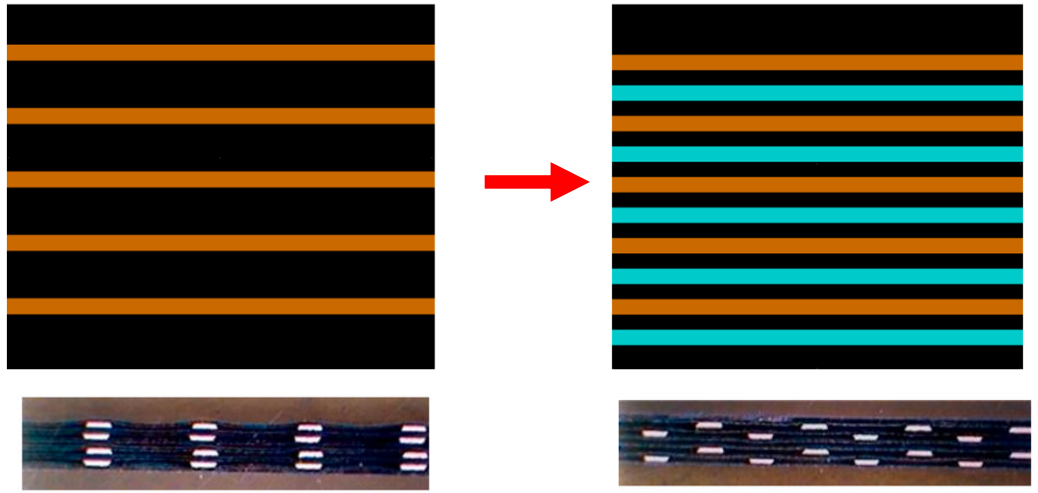


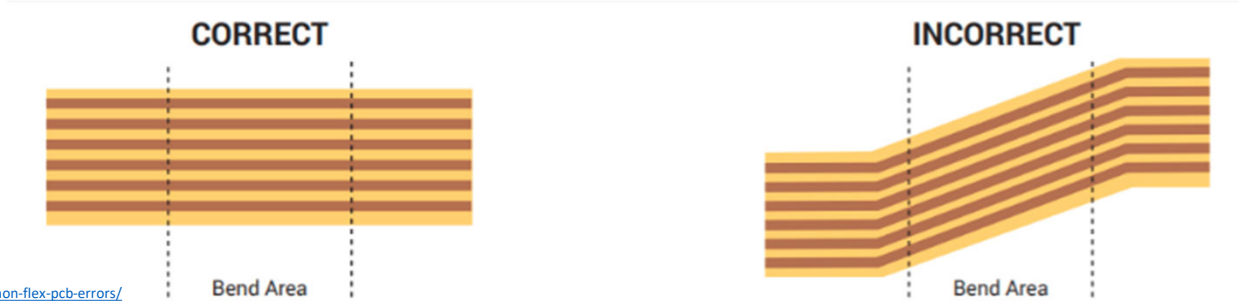
IMAGES:
https://www.greencircuits.com/flex-and-rigid-flex-circuit-designs/

Routing – Traces & Pours

Routing tips for traces and pours:

- Alternate routing paths when there is more than 1 layer
- Traces should be equally spaced
- Traces crossing bend lines should be perpendicular to bend line



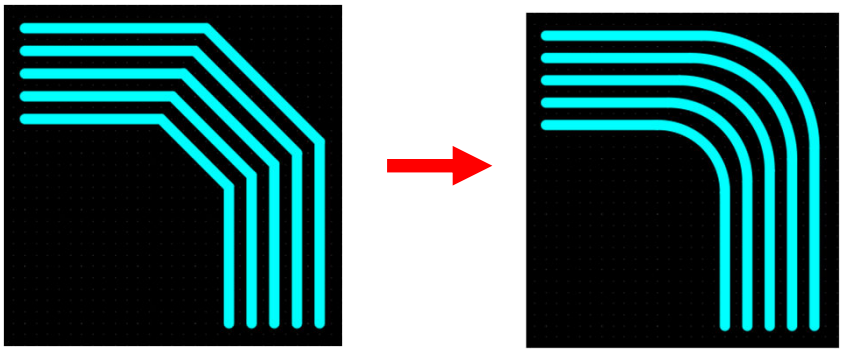


IMAGES:
https://www.protoexpress.com/blog/avoiding-common-flex-pcb-errors/

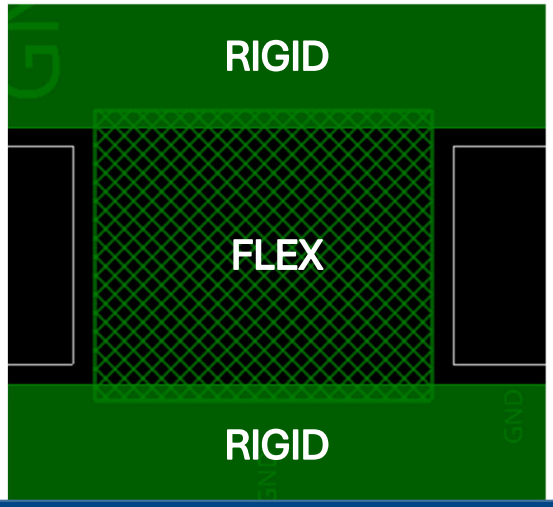
Routing – Traces & Pours

Routing tips for traces and pours:

- Curved traces everywhere, especially along curves of the flex region.



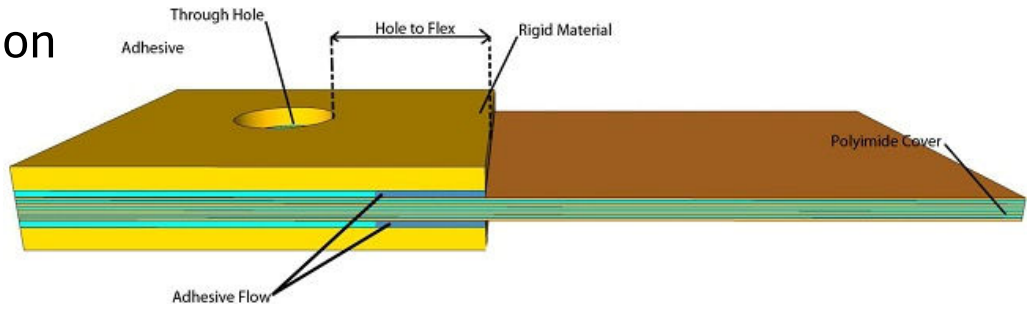
- Use hatched GND planes for the length of the flex to help with flexibility.
  - Should be on 45 degree angle to have more even coverage of signals as a return plane.
  - This is optional, based on flex requirements.



Routing – Vias & Pads

Routing tips for vias and pads:

- No holes/vias within 50 mil of rigid-flex transition
- Teardrops/fillets on pads, vias and traces for anchoring
- Vias should be fully tented to protect them from peeling
- No vias in bending area
  - Avoid vias in non-stiffener flex area as much as possible



IMAGES:
https://www.protoexpress.com/blog/air-gap-construction-method-rigid-flex-pcbs/

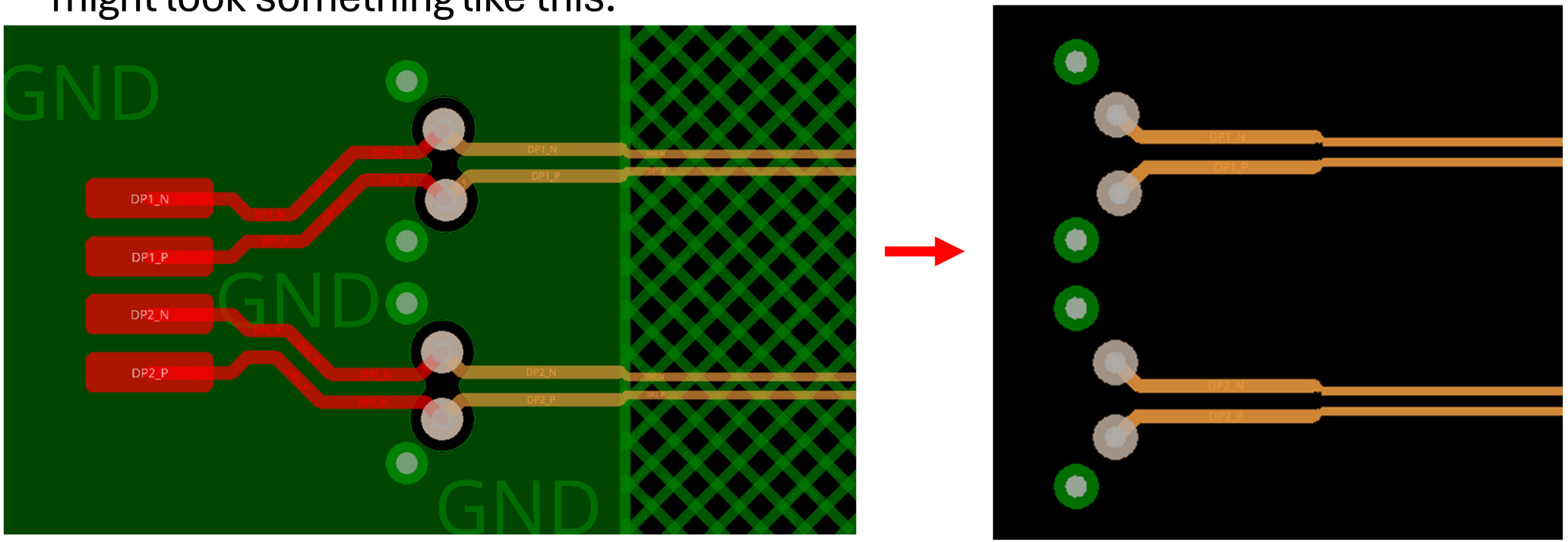
Rules

Work with the flex fab house to determine following items:

- Board edge clearance: can often be lower on flex regions when a laser router is being used for the board edge
- Min trace/space: will be driven by copper weight and fab house capabilities
- Controlled impedance should be requested from a fab house
  - Flex materials and hatched planes will impact impedance calculations
  - Standard calculators may not be accurate
  - Trace/space will change as you cross the rigid-flex boundary

Routing Example – Controlled Impedance

- When controlled impedance is routed over a rigid flex transition, it might look something like this:



Early Flex Fabricator Involvement

Engage flex fabricator early in process to help drive design decisions across multiple stakeholders.

Make sure all necessary departments are involved to ensure the most collaboration.

This allows you to find creative solutions that save time and money.

Electrical

- Material options/requirements
- Copper thickness
- Minimum trace/space
- Controlled impedance calculations

Mechanical

- Board thickness
- Bend radius

Management

- Lead time
- Fabrication cost
- Design cost

Main Takeaways

- Determine appropriate board “type” for needs of the design
- Understand differences in best design practices for flex and rigid design
- Check in often with all involved departments/collaborators
- Identify and communicate with fabricator early in the process

Thank you!

CONTACT:

Lauren Waslick – Newgrange Design

lauren@newgrangedesign.com