Is Your Spacecraft Straight or Bendy?

Flexible and flex-rigid PCBs are increasingly being used in orbit to reduce mass, size, price, assembly and test, as well as improving reliability by removing the need for physical connectors to interface with high-bandwidth sensors. Direct and indirect financial costs savings are possible by avoiding the need for expensive HDI connector pairs and wire harnesses, and then having to assemble and test these respectively. Bendy circuits can tolerate environmental factors such as extreme temperatures, radiation and vibration, and are 3D in nature conveniently connecting different spacecraft sub-systems. In this post, I summarise their use for space applications, best design practices and the formal IPC and ESCC standards which need to be followed for space-grade fabrication.

Typically for flexible PCBs, adhesives bond copper foil to polyimide (PI) or polyester (PET) films which is more bendy than traditional epoxy resin. The use of PI as a core-substrate material is preferred as its very heat resistant which makes it suitable for multiple reflow cycles and stable to expansion and contraction due to temperature fluctuations. The flexed PCB construction concludes with an outer layer, *e.g.* a PET coverlay, which insulates the outer-surface conductors, provides mechanical protection and endurance, shielding from corrosion and damage, in the same way solder mask does on a rigid board. As a flexible circuit can endure specific mechanical stresses, its conductors also have unique requirements for elasticity and toughness compared to traditional laminated copper foil. An example of a typical, single-layer flex circuit cutaway is illustrated in Figure 1.

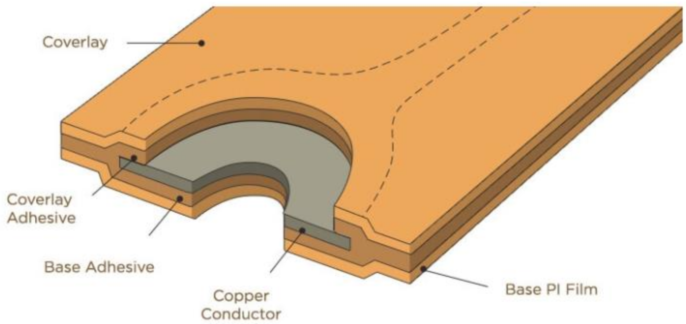

Figure 1 : Single-Layer, Flexible PCB Construction.

The standard design practice is to minimise the thickness of the flexible circuit as much as possible while meeting the electrical and performance requirements of the design, and not incurring unnecessary material costs.

Bend radius is a measure of how much you can bend a flexible circuit board without causing damage or shortening its lifespan. The smaller its value, the more bendy it will be! Flexed circuits are usually classified as either static (stable) or dynamic, with the former bent once (Flex-to-Install) during initial assembly to the desired radius of curvature and bend angle, while the latter is moved repeatedly throughout its operation and lifetime.

The thicker the overall flexible circuit, the less it will bend without damage and there will be a neutral bending axis throughout the flexed board, which defines a curve along which there is no longitudinal tensile or compressive stress. Thinner traces can withstand greater compressive stress than tensile stress, so thinner traces can be placed inside the neutral bending axis. The shift in the neutral bending axis will depend on the bending radius and a good rule-of-thumb for the minimum bend radius for a static, flexed PCB is:

$$Minimum\ Bend\ Radius = 6 * Layer\ Count * Flexible\ PCB\ Thickness$$

If the stack-up and trace thickness are chosen properly, the above rule will ensure the neutral bending axis will not shift appreciably from the centreline of the PCB. As the layer count increases, this ensures you comply with the IPC 2223C standards on the bending ratio (bend radius divided by thickness) listed below for flexible PCBs:

Layer	Stable/Static	Dynamic
Single	10:1	100:1
Double	10:1	150:1
Multi	20:1	Not Recommended

The minimum bending radius can be calculated by multiplying the finished, flexible PCB thickness by the ratio of the bending radii defined in IPC-2223. For example, if the assumed thickness of a multi-layer, static (stable) bendy PCB is 100 µm, the minimum bending radius is (20:1) * 100 µm = 2 mm.

For a dynamic flexible circuit, copper can harden under repeated bending, eventually becoming brittle and prone to fracture. If your final application involves repeated creasing or movement of the flex circuit, you may need to consider higher-grade, rolled annealed foils which are able to stretch more before fatigue or cracking occurs.

In addition to material selection and the total number of layers, if you want to reduce overall thickness and increase flexibility, there are coverlays available which do not require an adhesive. Reducing the amount of copper on planes through cross-hatching can also increase bendiness!

Siemens' (previously Mentor Graphics) Expedition flow allows a flexible PCB to be created within the same design environment as a traditional rigid board, *i.e.* multiple board outlines and stack-ups, flex-specific layer types, *e.g.* coverlay, adhesive and stiffener to support mounted components, as well as bend areas and flex-aware DRC and fabrication output.

For this article, I will describe a static flex connecting a rigid PCB containing an Earth-Observation image sensor to a smaller rigid board containing a high-density connector which mates to the main payload processor. The flex is being used to transport high-bandwidth differential pairs, some LVCMOS control signals and several, low-voltage, low-current power rails. No parts are to be mounted onto the flex, bendy pad-stacks are not required and the ten-layer stacks for the two rigid boards are identical and fabricated using the same material.

The example was developed within the Siemens' Expedition PCB environment and comprises three separate board outlines drawn in red. Each of the two rigid portions were assigned the same ten-layer master stack, while the flex's sub-stack comprises the inner six layers of the main stack as illustrated below. The latter preserves layer assignment and dielectric constant, avoiding changes to trace geometry to maintain characteristic impedance while also containing EM fields. Layers 2 and 5 of the flex's sub-stack are used for routing the sensor's sub-LVDS outputs as well as some LVCMOS control signals, while Layer 3 contains the low-current, power rails.

	Visible	Color	Pour Draw Style	Layer Name	Type	Usage
1				Soldermask_Top	Dielectric	Solder Mask
2	☑		Solid	GND1	Metal	Solid Plane
3				DIELECTRIC_1	Dielectric	Substrate
4	☑		Solid	Signal1	Metal	Signal
5				DIELECTRIC_2	Dielectric	Substrate
6	☑		None	GND2	Metal	Solid Plane
7				DIELECTRIC_3	Dielectric	Substrate
8	☑		None	Signal2	Metal	Signal
9				DIELECTRIC_4	Dielectric	Substrate
10	☑		Solid	VCC	Metal	Solid Plane
11				DIELECTRIC_5	Dielectric	Substrate
12	☑		Solid	GND3	Metal	Solid Plane
13				DIELECTRIC_6	Dielectric	Substrate
14	☑		None	Signal3	Metal	Signal
15				DIELECTRIC_7	Dielectric	Substrate
16	☑		None	GND4	Metal	Solid Plane
17				DIELECTRIC_8	Dielectric	Substrate
18	☑		Solid	Signal4	Metal	Signal
19				DIELECTRIC_9	Dielectric	Substrate
20	☑		Solid	GND5	Metal	Solid Plane
21				Soldermask_Bottom	Dielectric	Solder Mask

	Visible	Color	Pour Draw Style	Layer Name	Type	Usage
1				Soldermask_Top	Dielectric	Solder Mask
2	☑		None	GND2	Metal	Solid Plane
3				DIELECTRIC_3	Dielectric	Substrate
4	☑		None	Signal2	Metal	Signal
5				DIELECTRIC_4	Dielectric	Substrate
6	☑		Solid	VCC	Metal	Solid Plane
7				DIELECTRIC_5	Dielectric	Substrate
8	☑		Solid	GND3	Metal	Solid Plane
9				DIELECTRIC_6	Dielectric	Substrate
10	☑		None	Signal3	Metal	Signal
11				DIELECTRIC_7	Dielectric	Substrate
12	☑		None	GND4	Metal	Solid Plane
13				Soldermask_Bottom	Dielectric	Solder Mask

Figure 2 : Master Rigid Stack and Flex Sub-Stack, table view.

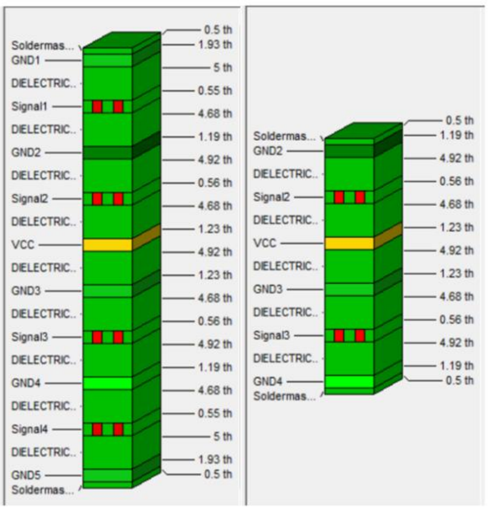

Figure 3 : Master Rigid Stack and Flex Sub-Stack, visual view.

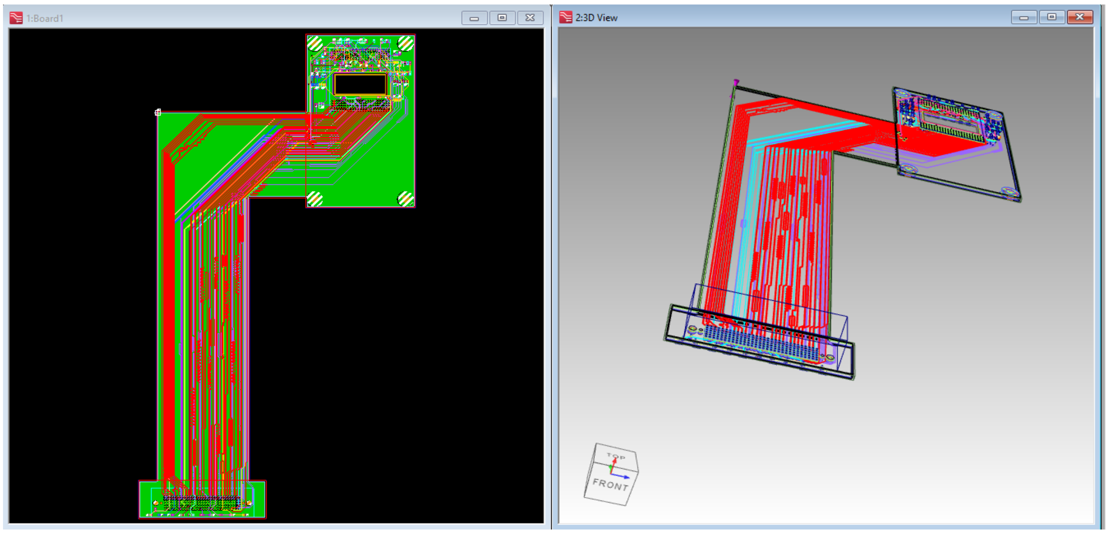

Figure 4 : Example of Earth-Observation sensor flex-rigid PCB.

For this example, a single route border is used as shown by the grey inner border in the 2D view and none of the three board outlines overlap. Routing uses specific trace widths and separation to maintain characteristic impedance, while Hyperlynx confirmed the trace-to-trace clearance was sufficient to avoid crosstalk between neighbouring, parallel tracks.

Flexible PCBs have unique failure modes such as plated through holes and vias over the bend area that can fracture resulting in an open circuit, overlapping traces (I-Beam Effect) which impact flexibility, as well as holes too close to the flex-rigid interface causing adhesive seepage stressing the plated through hole leading to barrel cracking.

To maximise the bendiness of a flexible circuit, ideally all traces should route in a straight line as angles may result in an increased risk of tearing. Turns should be radiused or mitred, and tracks entering the rigid portion of the PCB should run straight for at least 1 mm before changing direction, because it's within these transition areas where the board encounters the highest mechanical strain.

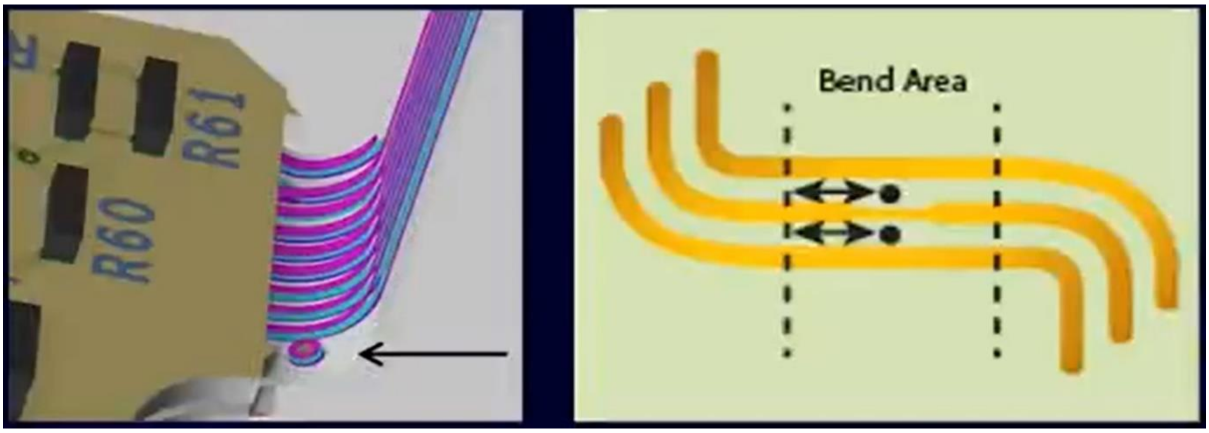

Figure 5 : Plated Through Hole Over the Bend Area.

Running traces over each other in the same direction distributes tensions between the copper layers unevenly. Staggering the traces reduces or eliminates the problem, reducing the total flexible layer thickness allowing for better bending as illustrated below:

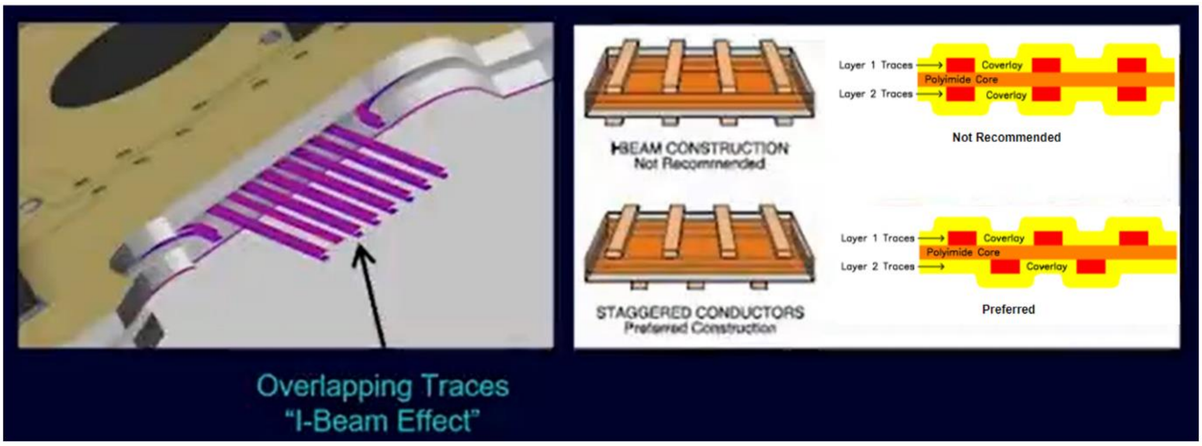

Figure 6 : Overlapping Traces Within a Flex.

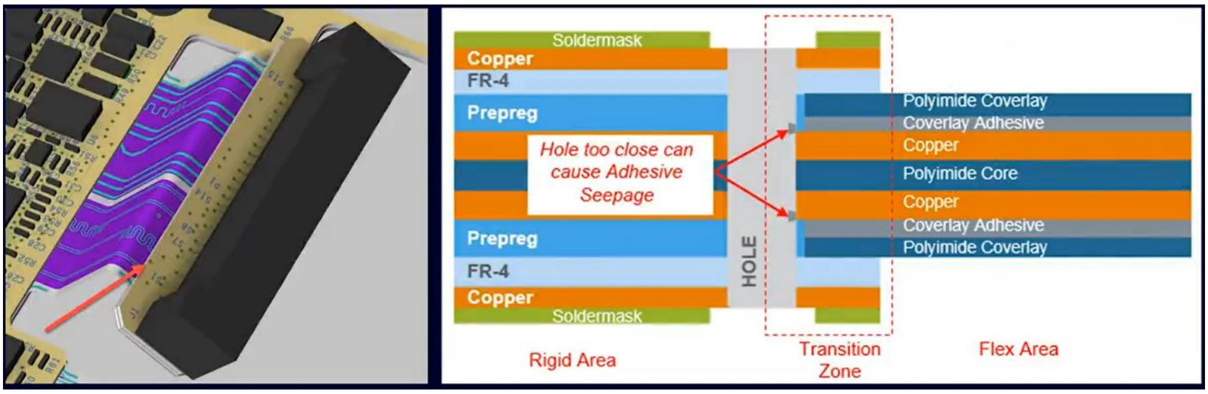

Figure 7 : Holes too close to flex-rigid Interface.

Valor NPI can be used to capture and enforce the above Design for Manufacturing (DFM) checks, including drill-to-copper clearances and minimum annular rings, before sending the completed layout to your fabricator.

Heat, moisture, chemicals, shock and vibration all need to be modelled with accurate material properties to determine the product's reliability and minimum allowed bending radius. There is a limit to the amount of strain flexible PCBs can be subjected to: when they are bent out of shape, the internal bend experiences compressive forces while the external bend experiences tensile forces. Knowing these limits improves functionality, performance and the reliability of your space electronics!

It's important to discuss your design with your fabricator before beginning layout to understand their design rules, what they can and cannot build reliably, and the compliance of materials to support your electrical and functional needs.

In addition to IPC documents, specifically IPC-2223 and IPC-6013 C/D, agencies such as ESA, NASA and ISRO have their own formal standards for space-grade boards. ESA's, ECSS-Q-ST-70-12C, **Design Rules for PCBs**, and ECSS-Q-ST-70-60C, **Qualification and Procurement of PCBs**, are extensively used by the global space industry. Spacechips teaches a course on **Right-First-Time, Space-Grade PCB Design, Layout, Manufacture & Assembly**, during which we show you how to design a reliable, flexible PCB for space applications.

Until next month, the first person to tell me the difference between a Flex-to-Install and a One-Time Crease bendy PCB will win a Courses for Rocket Scientists World Tour tee-shirt. Congratulations to Marta from Austria, the first to answer the riddle from my previous post.

Dr. Rajan Bedi is the CEO and founder of Spacechips, which designs and builds a range of advanced, AI-enabled, L to K-band, ultra high-throughput on-board processors, transponders, SDRs and Edge-based OBCs for telecommunication, Earth-Observation, navigation, 5G, SIGINT, internet and M2M/IoT satellites. The company also offers Space-Electronics Design-Consultancy, Avionics Testing, Technical-Marketing, Business-Intelligence and Training Services. (www.spacechips.co.uk).