Spacechips (c) 2019

Right-First-Time PCB Layout for Spacecraft Avionics

Module 1

1.1 Dielectric material selection for space-grade PCB design
1.2 Right-first-time stack-up design for EMC
1.3 Interplane capacitance and PDN
1.4 Stack-up design using Hyperlynx and importing into Expedition
1.5 Dielectric weave pattern and its impact on high-speed design
1.6 Vias and padstacks

Module 2

2.1 Packaging your design and forward annotation between DxDesigner and Expedition
2.2 Creating/importing PCB footprints (EDX, PDX or ODB++) or where to get land patterns, *e.g.* PartQuest, UltraLibrarian, SnapEDA or parts vendors
2.3 Storing footprints within the Central Library using Library Manager
2.4 Floorplanning and Placement
2.5 Planes - creating power and GND planes within Expedition

Module 3

3.1 Routing, critical length of transmission lines, track geometry and target impedances
3.2 Current-carrying capacity of traces and planes
3.3 Using constraints and Design Re-use
3.4 Right-first-time layout of RF, GHz digital and GSPS analogue/mixed-signal
3.5 Right-first-time layout of DC-DCs and linear/switching regulators
3.6 Right-first-time layout of low-voltage, high-current power distribution

Module 4

4.1 Space-industry specific PCB standards, *e.g.* ESA and IPL
4.2 Should the space industry use silkscreen? For and against!
4.3 Should the space industry use soldermask? What are the alternatives?
4.4 Creating the manufacturing files and selecting a space-qualified PCB fabricator