How is a PCB Made?
What determines impedance?
Manufacturing Processes for a Multi-layer PCB

The following presentation covers the main processes during the production of a multi-layer PCB.

The diagrams which follow represent a section through a 6 layer PCB, as indicated in red.
**Typical Layer Construction - 6 Layer PCB**

- Layer 1 (Outer)
- Layer 2 (Inner)
- Layer 3 (Inner)
- Layer 4 (Inner)
- Layer 5 (Inner)
- Layer 6 (Outer)

Copper  Laminate

**Impedance Considerations**

- Layer build / stackup is one of the most important aspects of controlled impedance
- Many combinations of material thickness can be used.
- PCB Fabricators manufacturing techniques vary
Drilling of Bonded Panel

Impedance Considerations

• Press temperature and pressure have an effect on the flatness and hence impedance. This should be checked prior to drilling

• Drilling itself does not effect impedance
Electroless Copper Process
Addition of Copper to all Exposed Surfaces

Impedance Considerations

- Electroless copper effects copper thickness on outer layers (T)
- Sometimes other solutions are used containing carbon etc.
Laminating and Imaging of External Layers

UV sensitive film is laminated over top and bottom surfaces of PCB. It is then exposed and developed, leaving an exposed image of the PCB pattern.

Copper

- Does not effect impedance
Electro-plating Process 1
Additional Copper to all Exposed Surfaces

- Electro-plating increases the copper thickness on outer layers (T).
- This will always be variations in the amount of copper added.
- This finished copper thickness should be used in structure calculations.

Impedance Considerations
Electro-plating Process 2
Add Tin over Exposed Copper Areas

**Impedance Considerations**
- Does not effect impedance
Electro-plating Process 3
Remove Laminated Film

Layer 1
Layer 2
Layer 3
Layer 4
Layer 5
Layer 6

Laminated Film Removed
Tin Plating

Coated Microstrip

Impedance Considerations

• Does not effect impedance
Etch Process - Remove Exposed Copper

- The etch process produces an ‘etch back’ or undercut of the tracks. This can be specified by the W / W1 parameters.
- This means that tracks will end up approximately 0.025 mm (0.001”) thinner than the original design.

Impedance Considerations

- The etch process produces an ‘etch back’ or undercut of the tracks. This can be specified by the W / W1 parameters.
- This means that tracks will end up approximately 0.025 mm (0.001”) thinner than the original design.
Tin Strip - Remove Tin Plating

Impedance Considerations

- The Removal of Tin will slightly reduce the copper thickness (T) on the outer layers
PCB is now complete except for surface finishes and panel routing
Solder Mask Application
- Curtain Coated Method

Apply Liquid Photo-imageable Resist, then Dry

Impedance Considerations

- Some PCB Fabricators chose to check the impedance before the solder mask is added
- Structures can be checked in Normal and Coated mode
- Thickness of solder mask should be specified using H1
Solder Mask Application
Image, Develop and Cure

UV Image, Develop and Cure

Layer 1

Layer 6

Impedance Considerations
- Does not effect impedance
Surface Finish Process

- Apply Solder to Exposed Copper Areas

Impedance Considerations

- Surface Finish (Tin / Lead / Gold / Silver) is usually only added to pads.
- If board has no solder mask, the thickness of finish should be added to T.
Component Notation

- R34
- IC3

Impedance Considerations
- Does not effect impedance
Routing (includes second stage drilling)

Impedance Considerations

- Controlled Impedance coupons are routed from the panel
- Good controls are necessary to ensure that coupons can be matched to manufacturing panels
Process finished PCB and coupon for testing

**Impedance Considerations**

- Controlled Impedance coupons are routed from the panel.
- Controls are necessary to ensure that coupons can be matched to manufacturing panels; this should be performed on trial panels prior to production ramp up.
Why as a designer do you need to discuss your design with your PCB fabricator?

PCB manufacture is a process, it uses materials which are not “Ideal”

FR4 for example is a glass resin mix made of two substances with differing electrical properties.

PCB Manufacturers need to make small adjustments to designs to maximise yields.

- Glass Er 6
- Resin Er 3 (FR4)
- Resin Er < 3 (High performance laminates)
Why as a designer do you need to discuss your design with your PCB fabricator?

Process varies from one fabricator to another.

Press pressures temperatures may vary

Pre preg and Core may vary from one Supplier to another.
Process finished PCB and coupon for testing

**Impedance Considerations**

- Controlled Impedance coupons are routed from the panel.
- Controls are necessary to ensure that coupons can be matched to manufacturing panels; this should be performed on trial panels prior to production ramp up.