

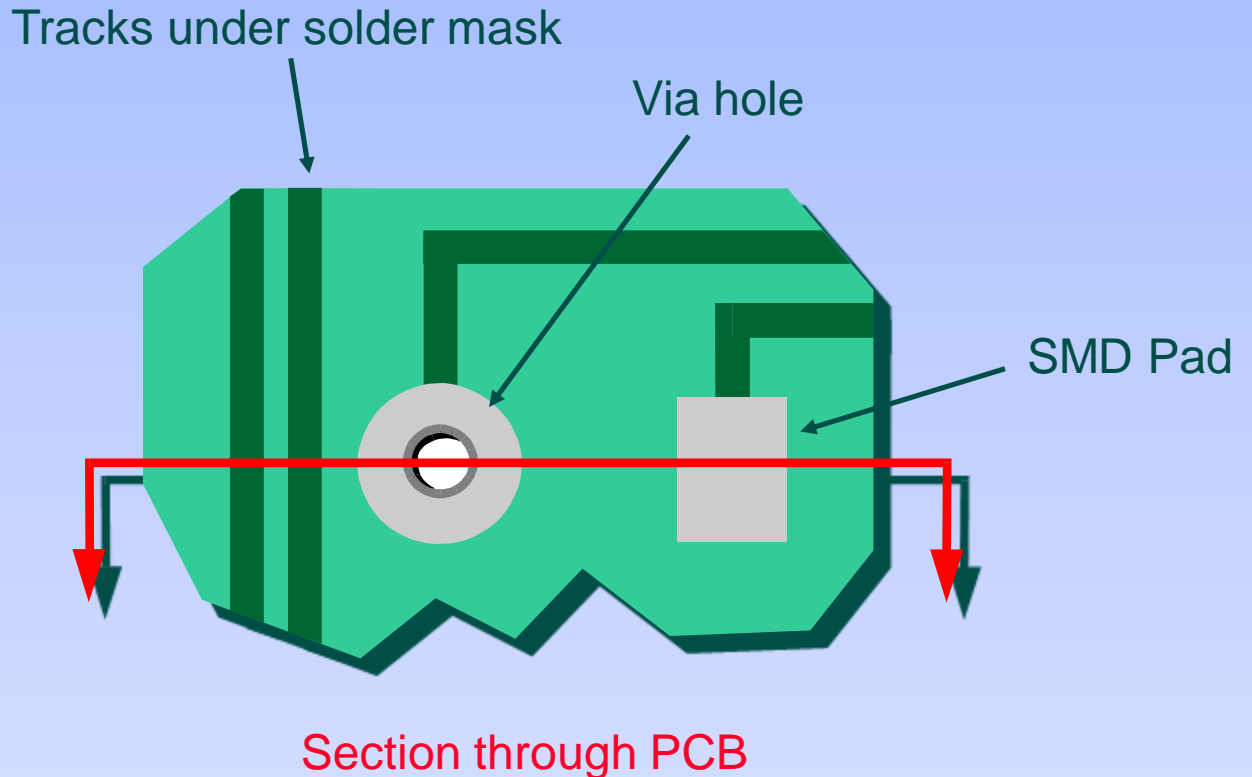
**How is a PCB Made ?**  
**What determines impedance ?**

**[www.fmuser.net](http://www.fmuser.net)**

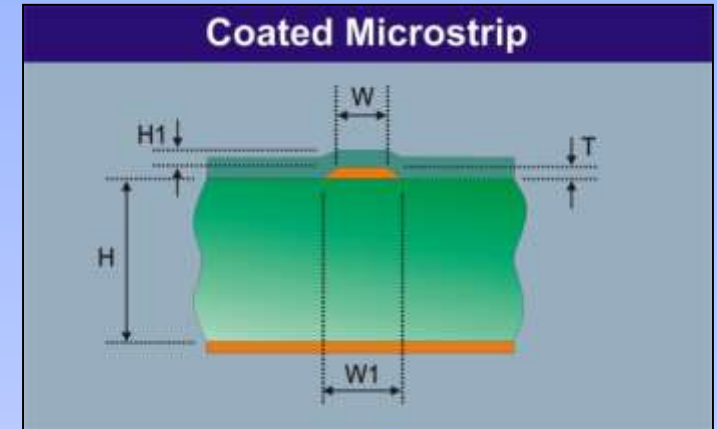
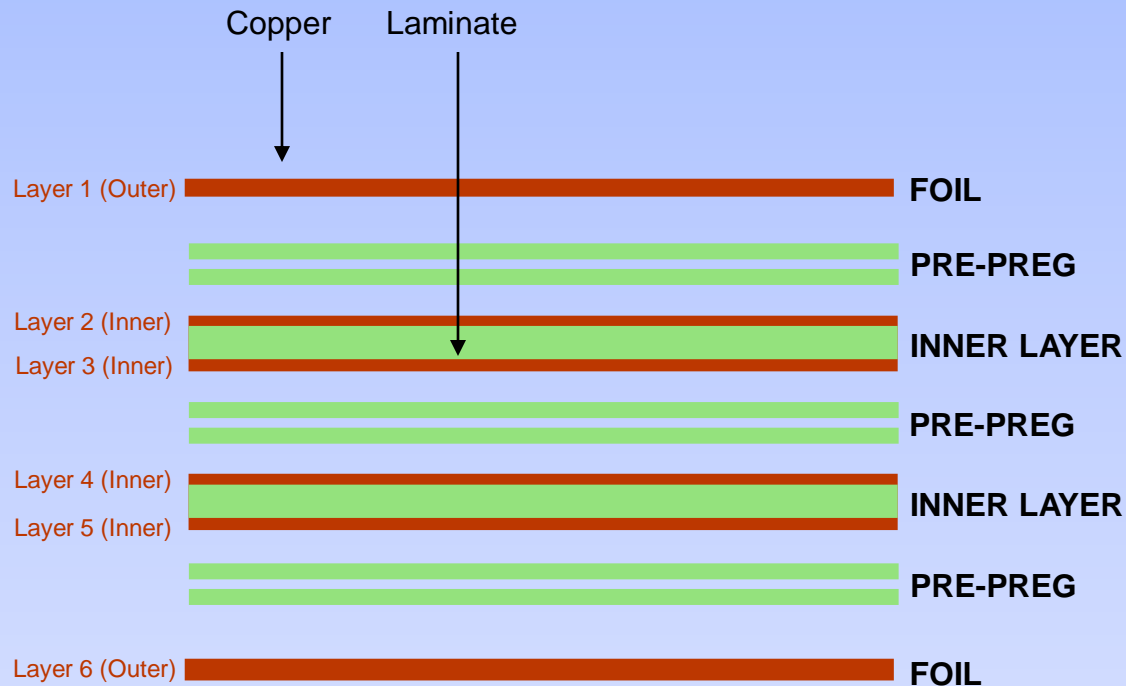
# Manufacturing Processes for a Multi-layer PCB

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.



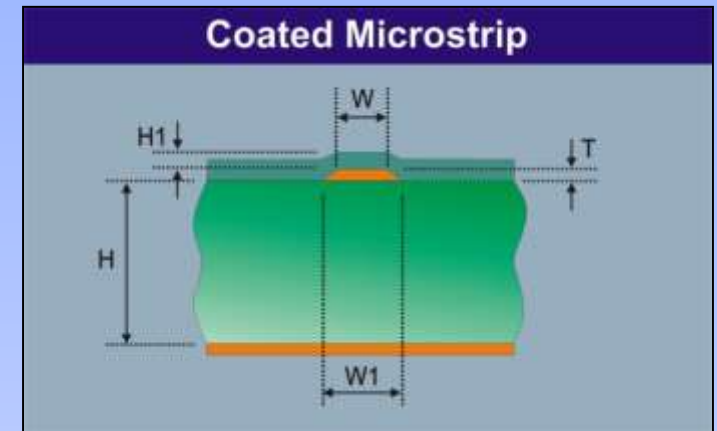
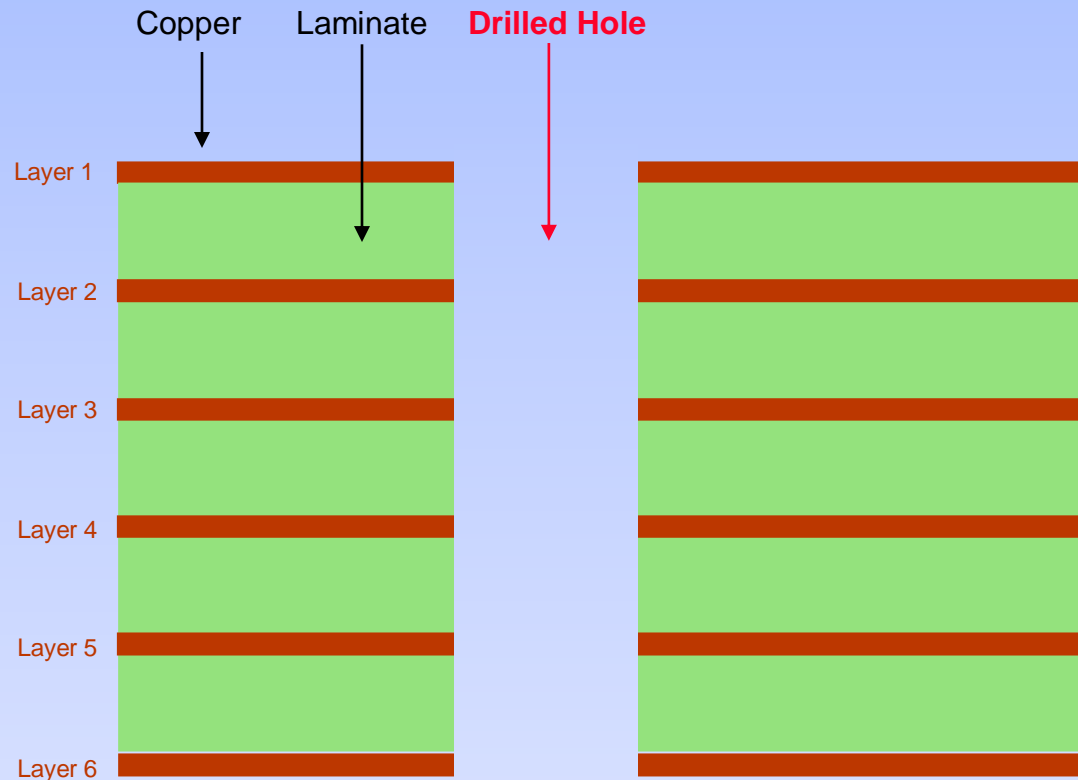
## Typical Layer Construction - 6 Layer PCB



### 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

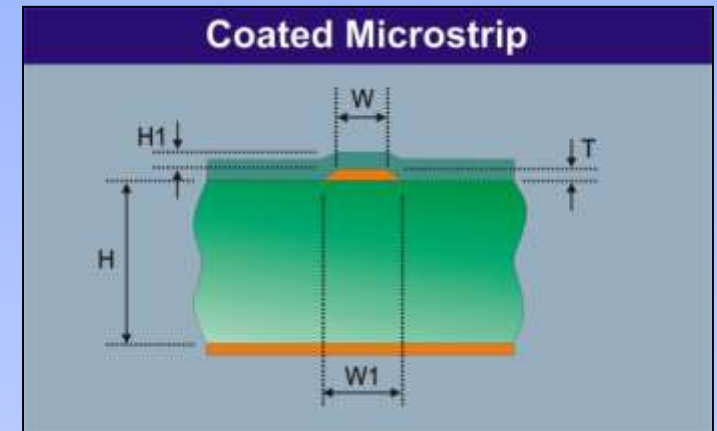
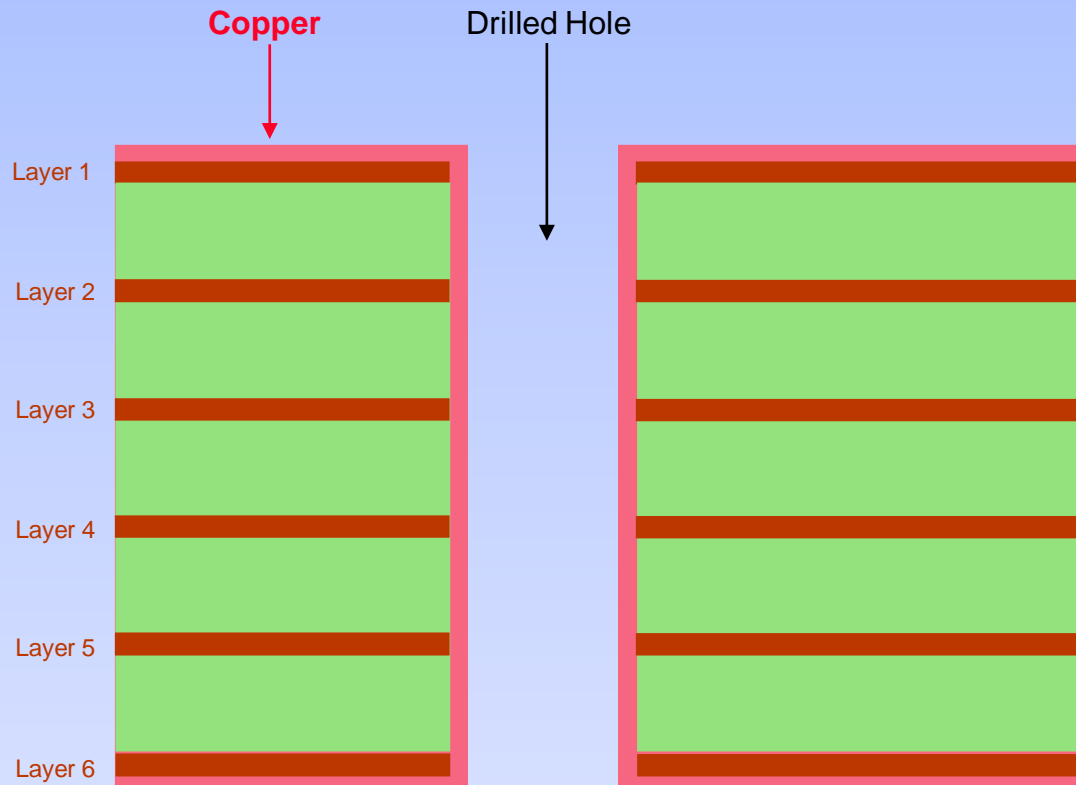


### Impedance Considerations

- 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

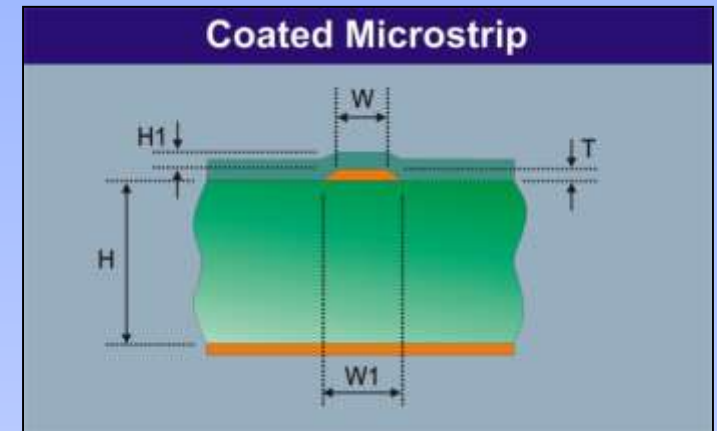
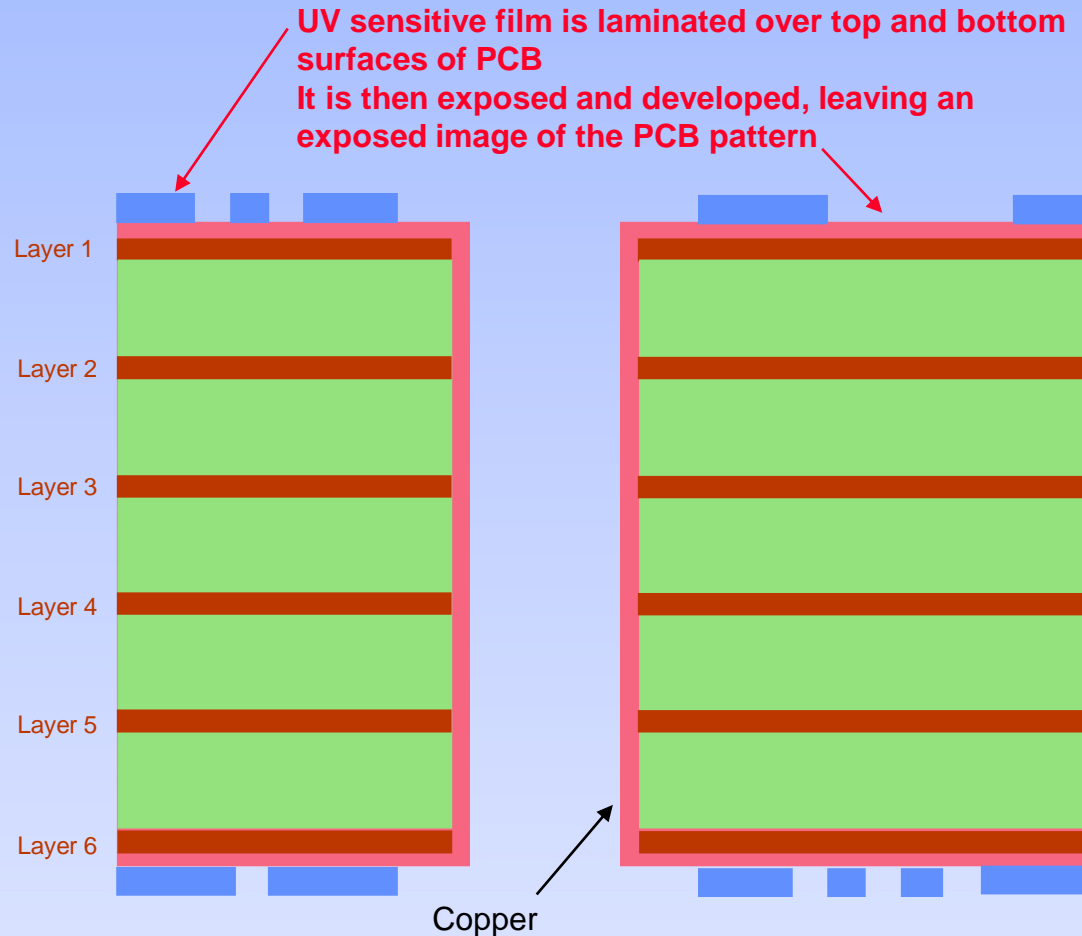
## Addition of Copper to all Exposed Surfaces



### 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

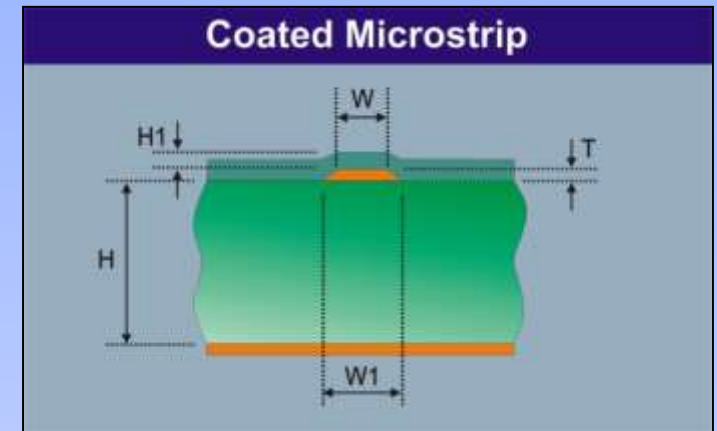
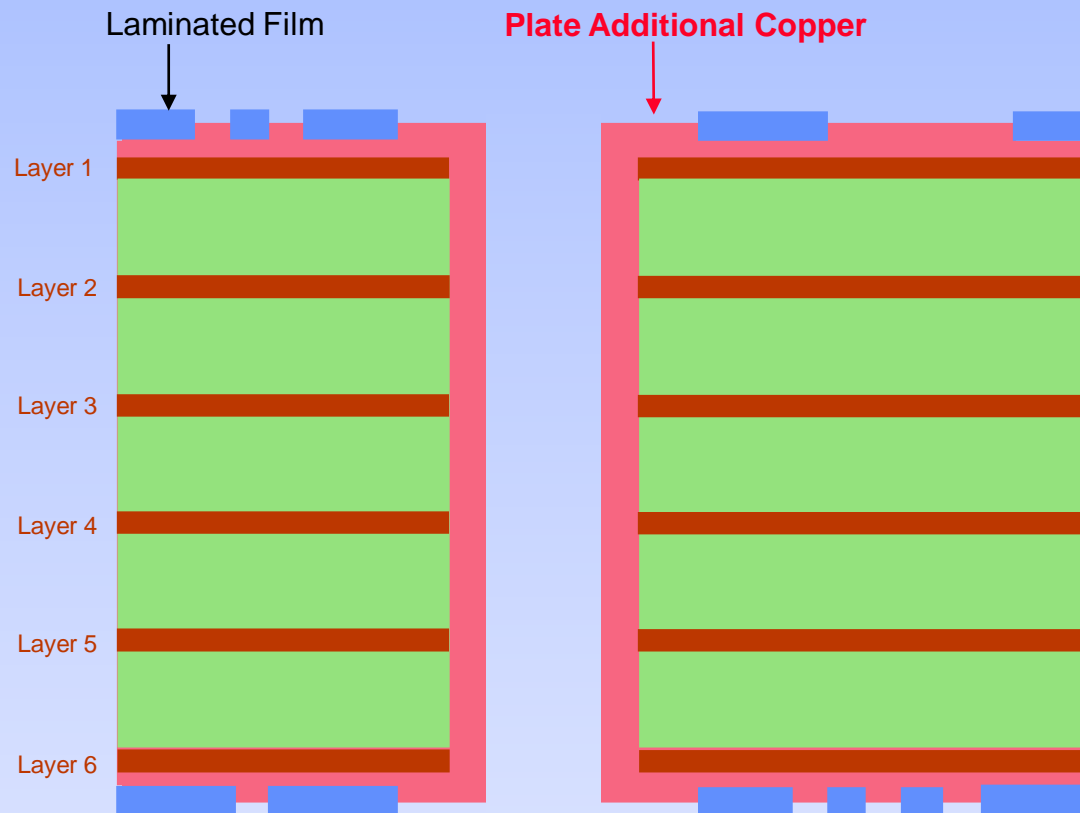


## Impedance Considerations

- Does not effect impedance

# Electro-plating Process 1

## Additional Copper to all Exposed Surfaces

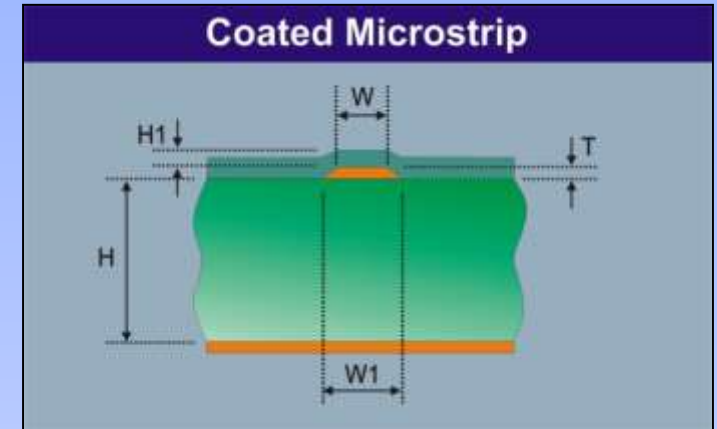
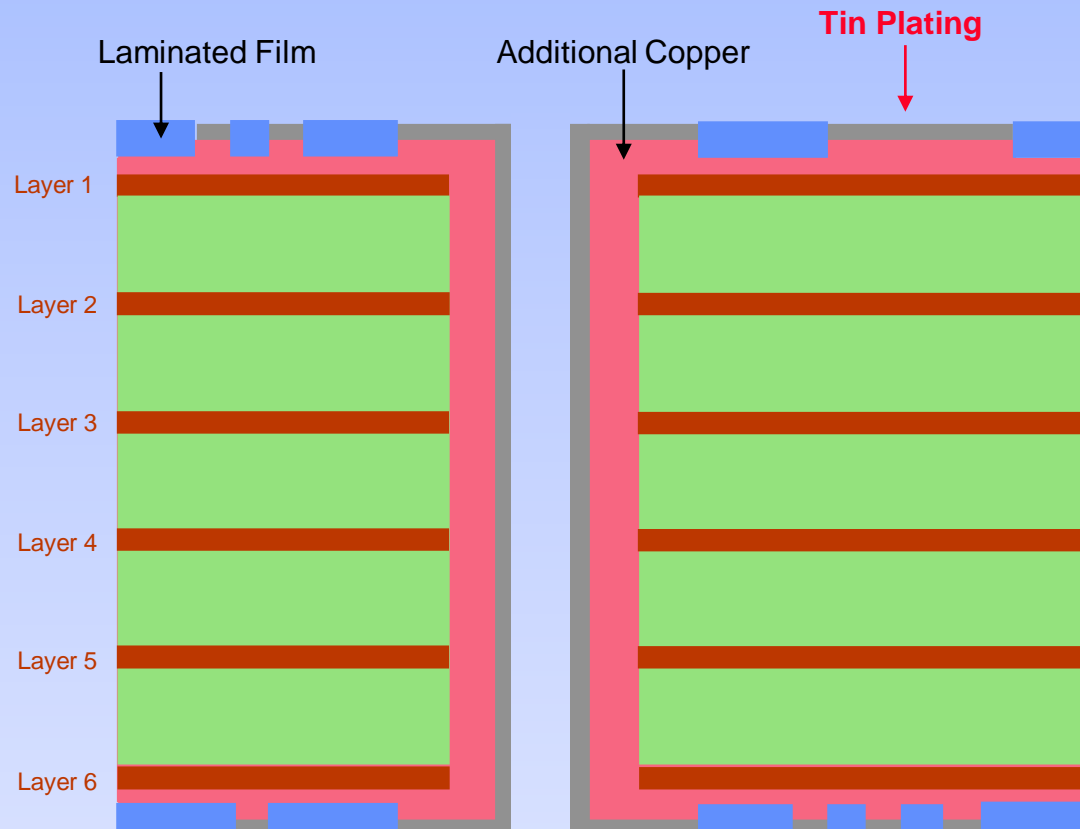


### Impedance Considerations

- 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

## Electro-plating Process 2

### Add Tin over Exposed Copper Areas



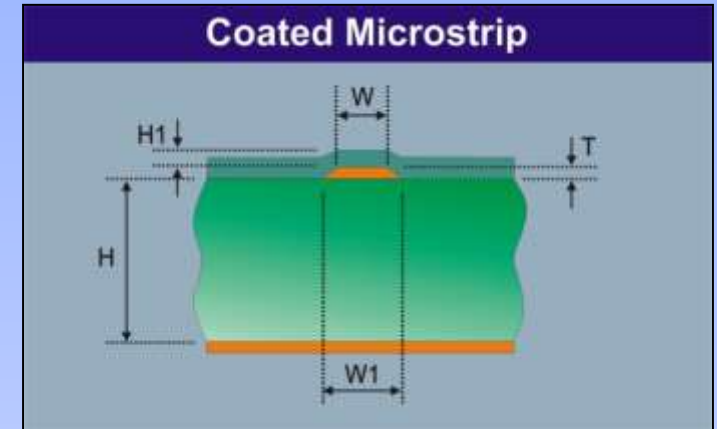
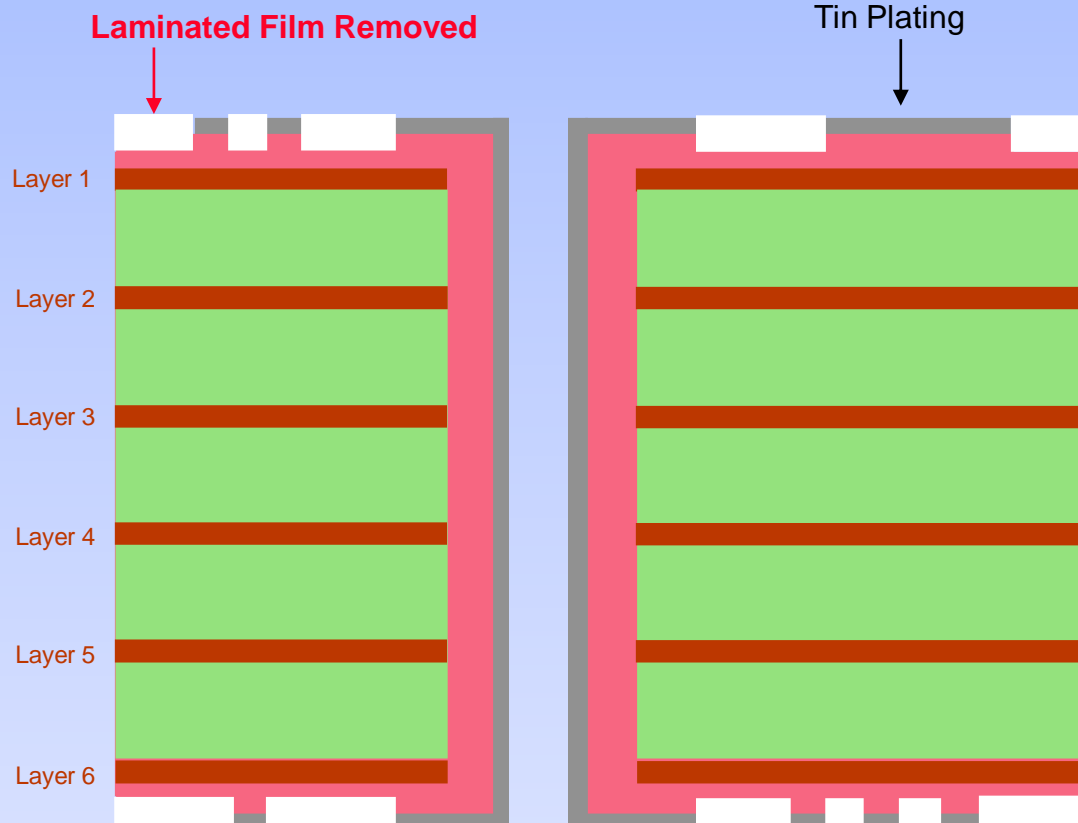
### Impedance Considerations

- Does not effect impedance



## Electro-plating Process 3

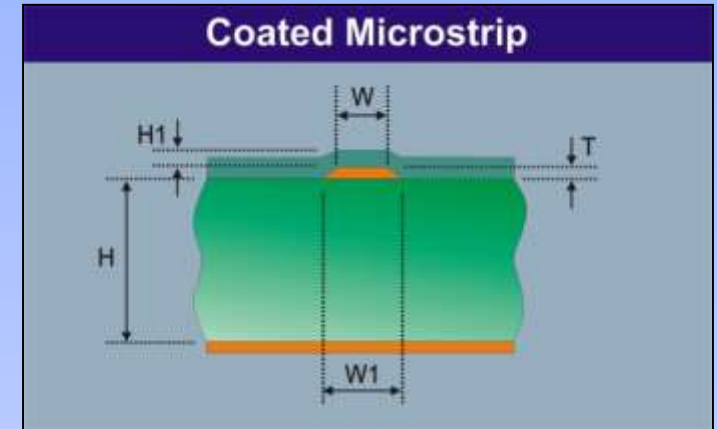
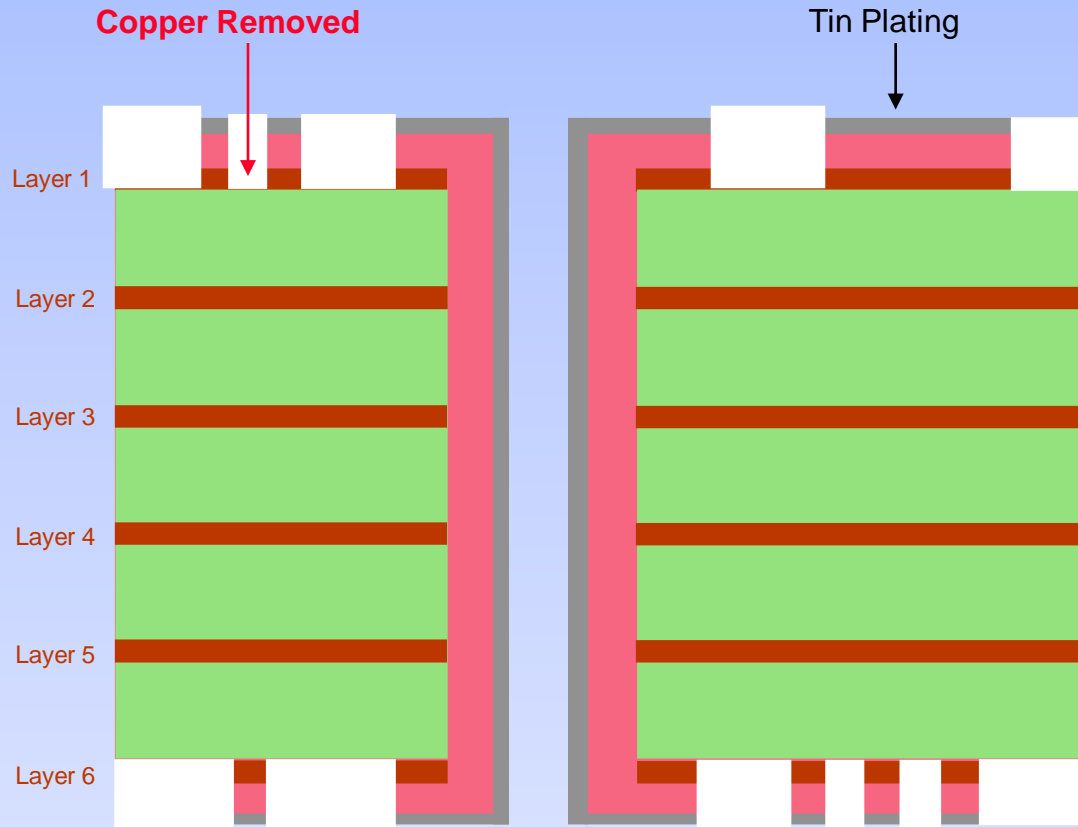
### Remove Laminated Film



### Impedance Considerations

- Does not effect impedance

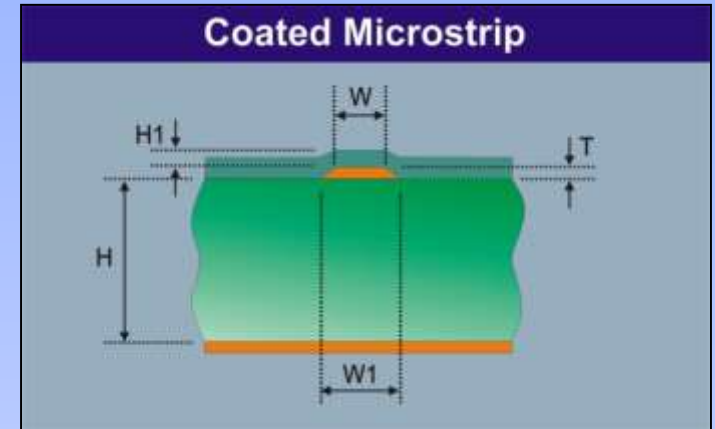
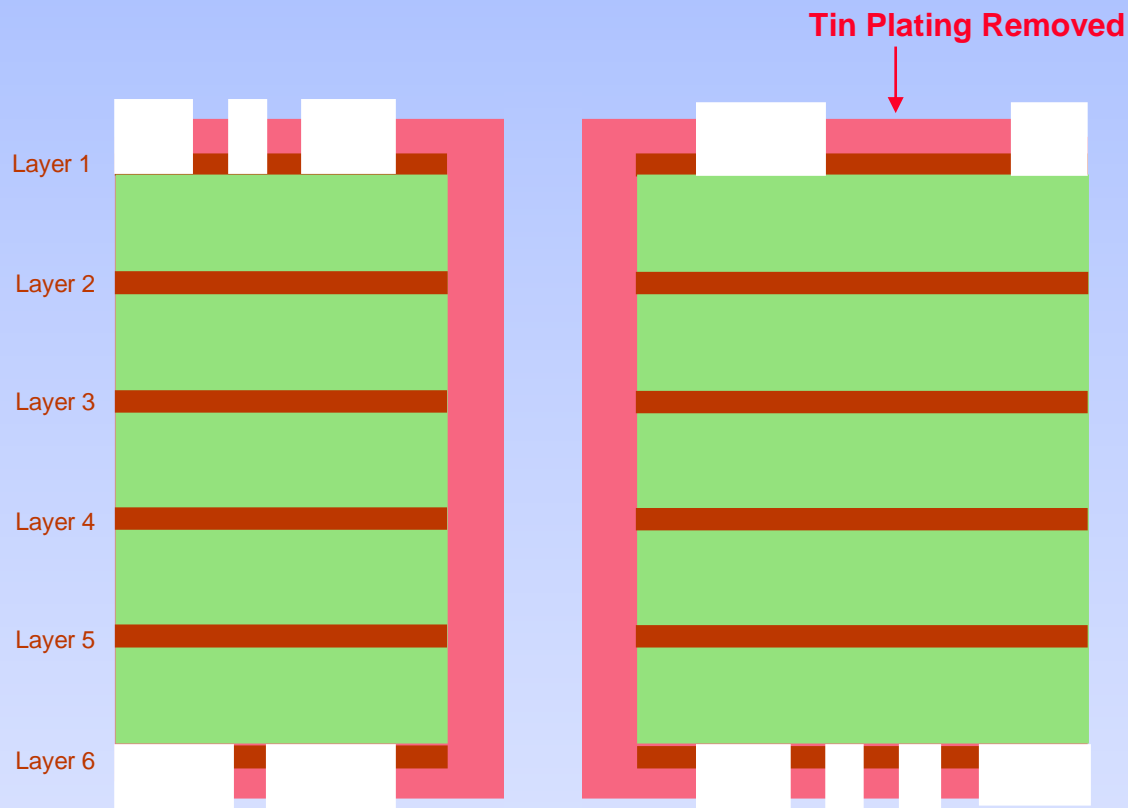
## Etch Process - Remove Exposed Copper



### 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.

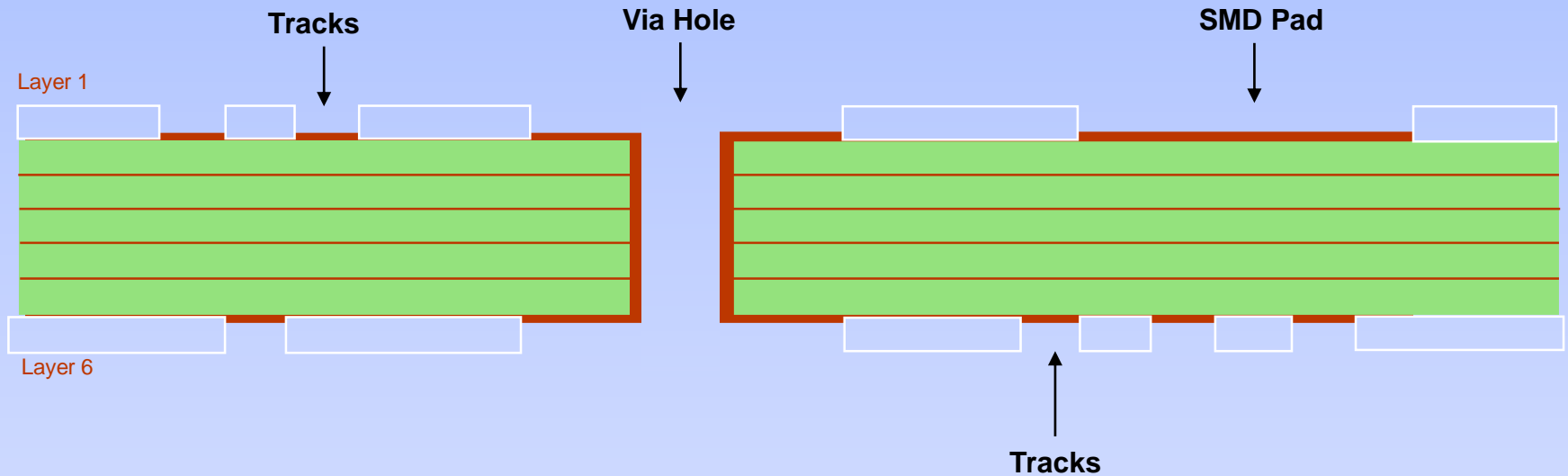
## Tin Strip - Remove Tin Plating



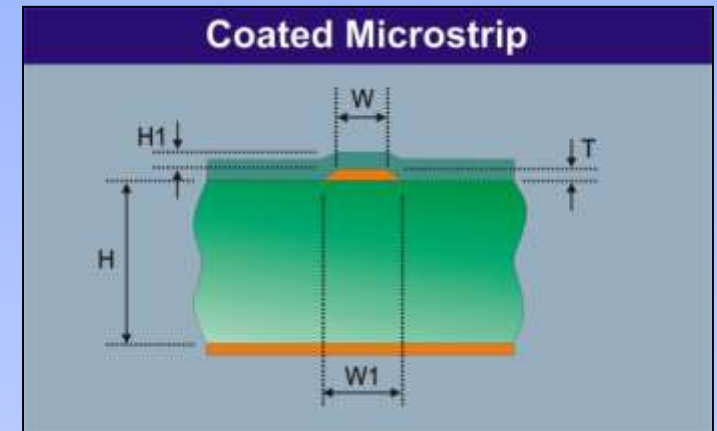
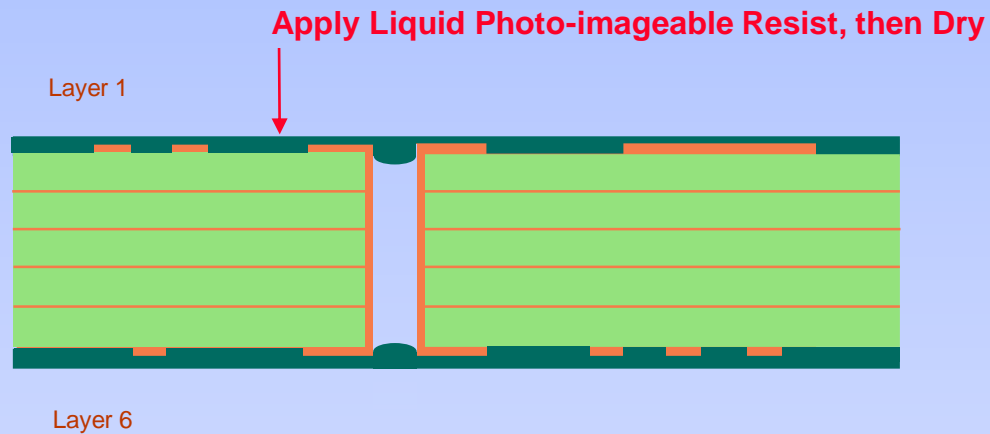
### Impedance Considerations

- 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**



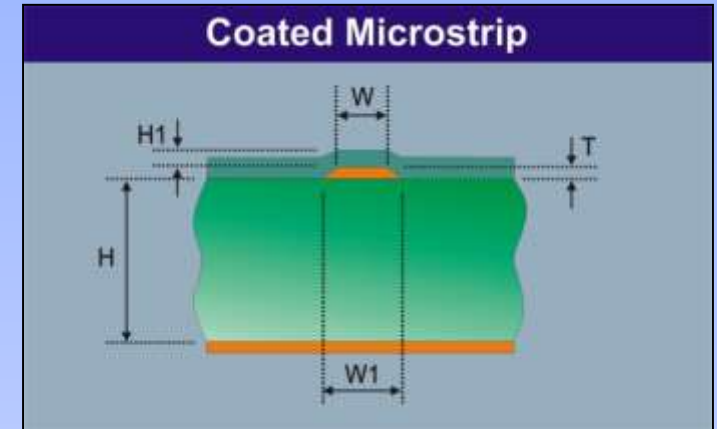
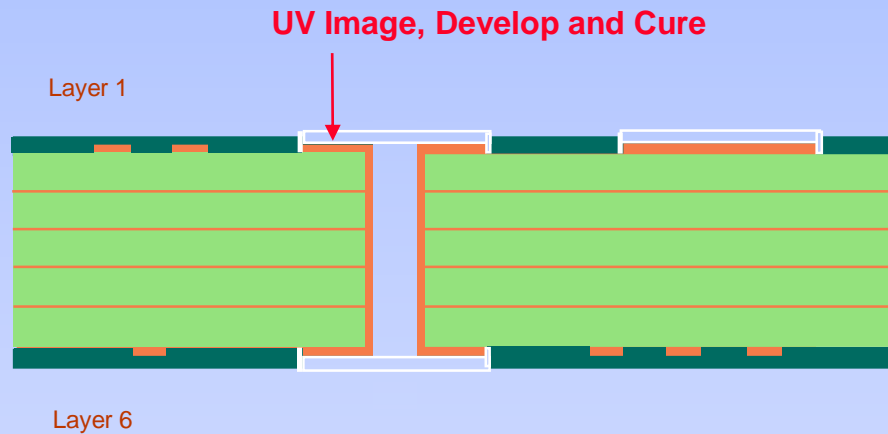
## Solder Mask Application - Curtain Coated Method



### 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

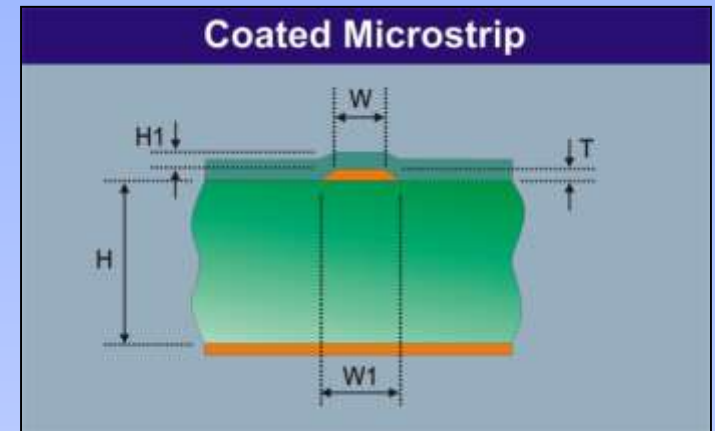
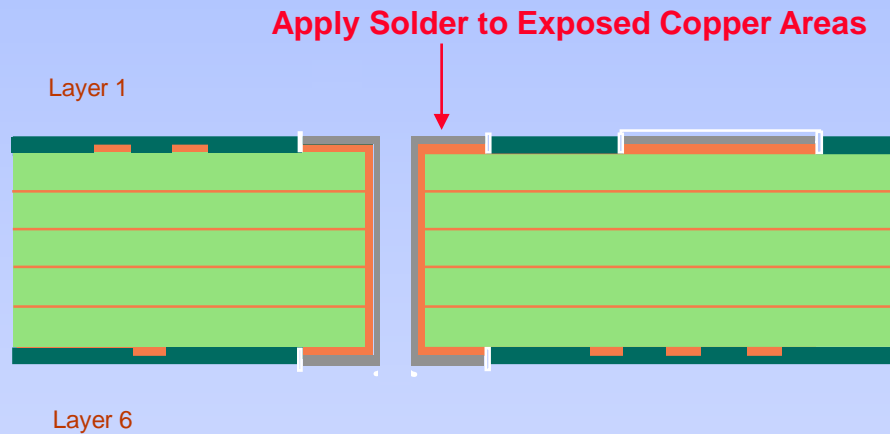
## Solder Mask Application Image, Develop and Cure



### Impedance Considerations

- Does not effect impedance

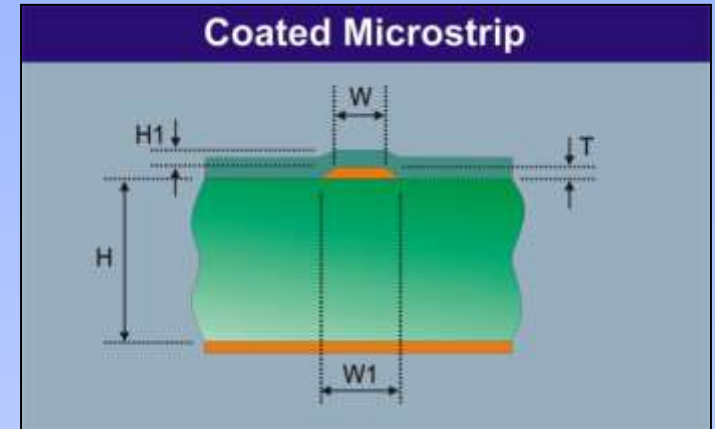
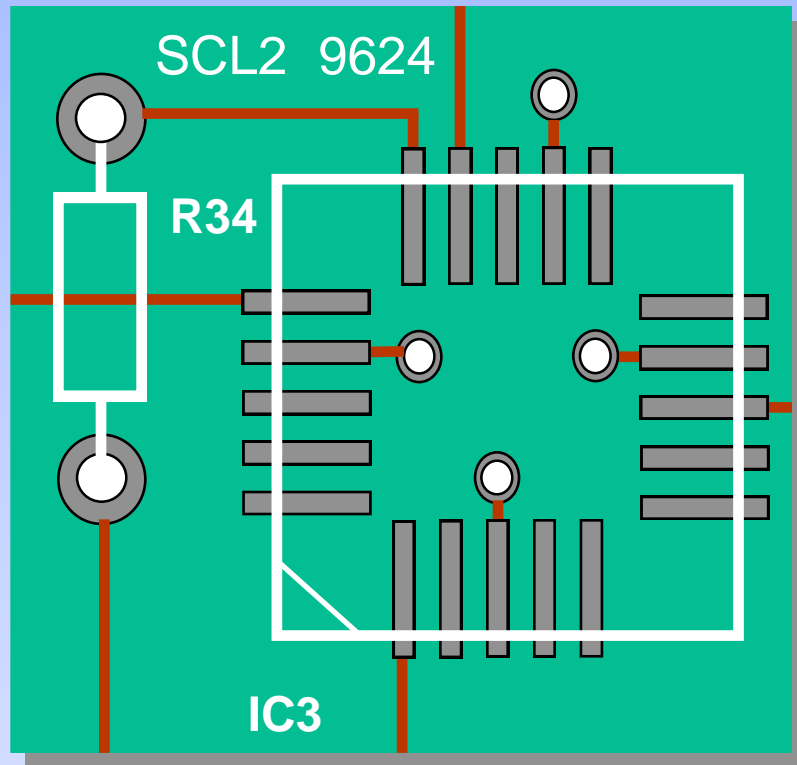
# Surface Finish Process



## 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

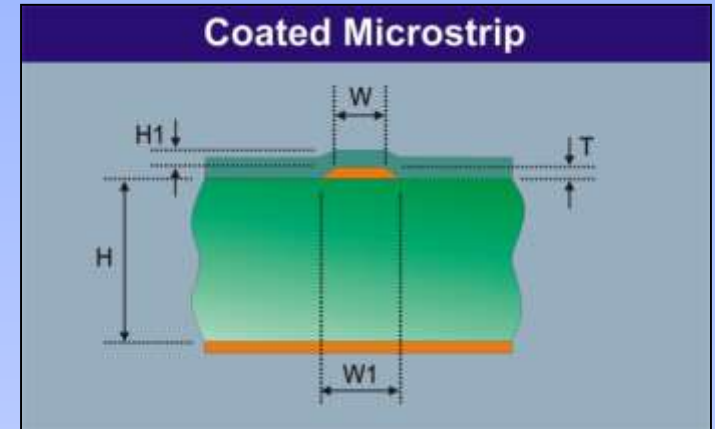
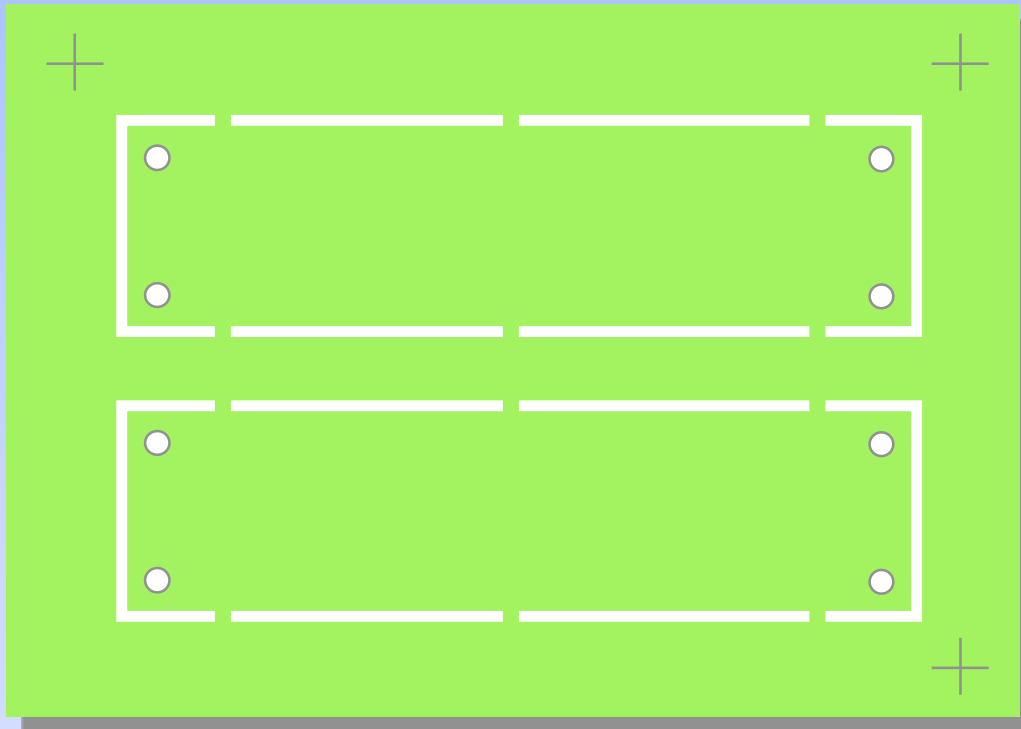


### Impedance Considerations

- Does not effect impedance



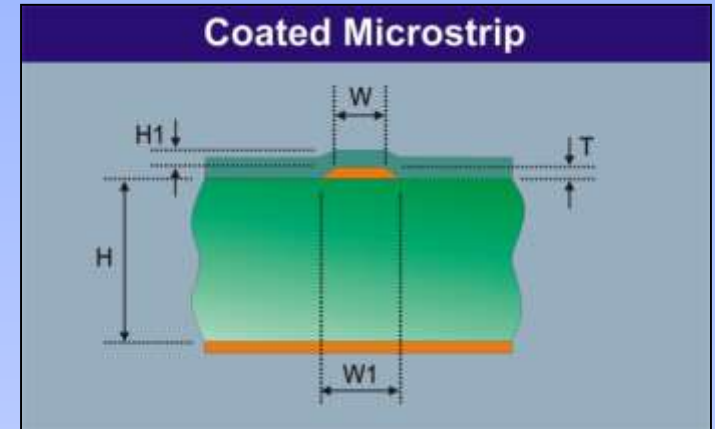
## Routing (includes second stage drilling)



### 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

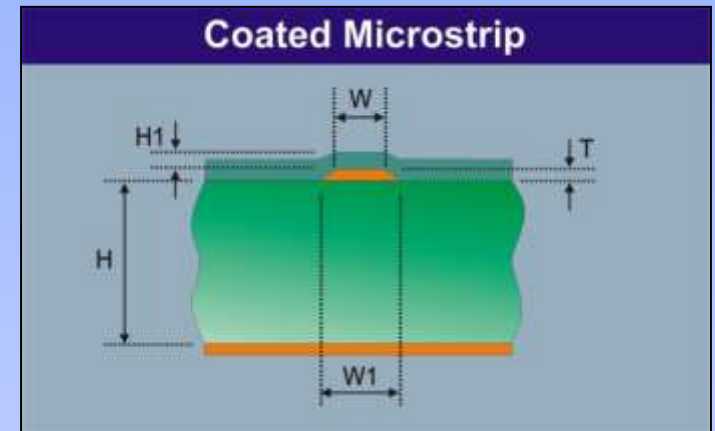
- Controlled Impedance coupons are routed from the panel
- Controls are necessary to ensure that coupons can be matched to manufacturing panels this should be performed on trial panels prior to production ramp up.

**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**



### Impedance Considerations

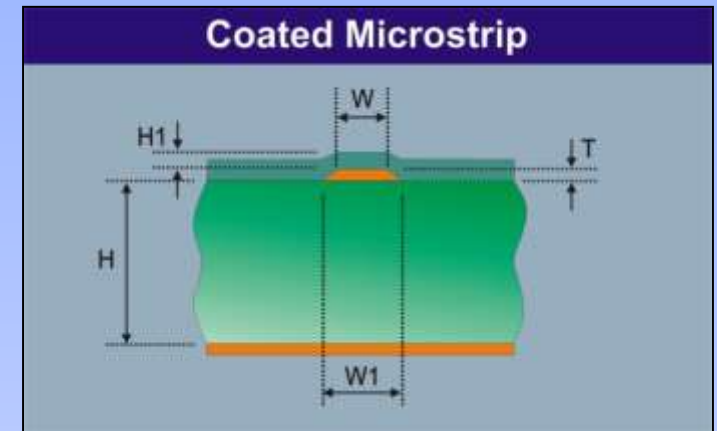
- Glass Er 6
- Resin Er 3 (FR4)
- Resin Er < 3 (High performance laminates)

**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**

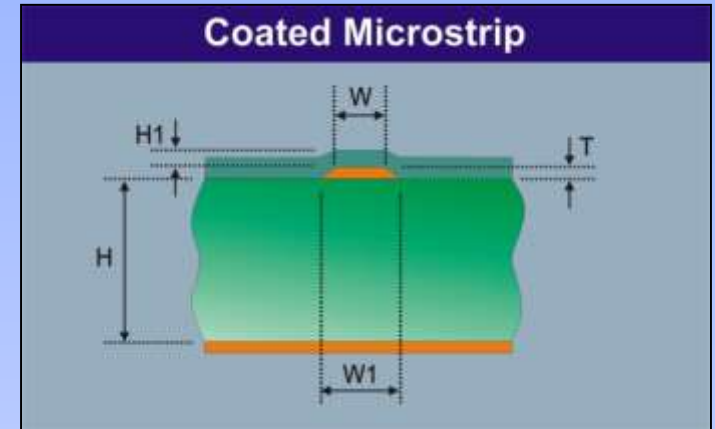
**Pre preg and Core may vary from one Supplier to another.**



### Impedance Considerations

- Supplier variations

## 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.