Continuing..

**PRINTED WIRING BOARD TECHNOLOGIES**
Image Transfer

Video highlights-PTH process sequence- complete steps
PTH Electroless Copper Plating

Video highlights-PTH process sequence- complete steps
Panel Copper Electroplating

Video highlights-PTH process sequence- complete steps
Video highlights-PTH process sequence- complete steps
Dry Film Stripping

Video highlights-PTH process sequence- complete steps
Post operations

1. Quality Inspection (at all stages) ☒
2. Solder Masking ✓
3. Legend printing ✓
4. Scoring and Routing ✓
5. Edge chamfering
6. Bare board Testing _ elec. test / shunts / opens
7. Packaging/Shipping
PWB Microsectioning
PWB Microsection Polishing
Solder Masking On Bare Copper - SMOBC

Steps for SMOBC

Plug all PTH holes...tenting or Screen ink
Chemically strip TIN metal till the underneath Cu is exposed
Treat the exposed copper to convert to Black/Brown oxide
Apply Solder Mask
UV/Thermal cure

The Color of the Solder mask is generally GREEN because White **Legend Ink** is more clearly visible on a green background
[other colors are also used]

---

**Legend Printing**

White Legend Inks on green background

**Edge Chamfer**
Routing

Routing is a cutting operation used to create channels as well as for creating special cut-outs in the board introduced by the design requirement.

Illustration of Cut-outs in PWB
Bare Board Testing Fixture-Bed of Nails  Flying Probe Tester
Board Finishes

Solder Finish  - Electroplated Tin
Gold Finish    - Ni-Gold for planarity
               - ENIG

The Board is now ready for assembly
Multi Layer-Types

1. Laminated Multi layer Structures

Made by stacking separately made layers and Pressing them into to a mono block in a press

Called Conventional MLBs

2. High Density Multi layer Structures

Made by sequentially adding layer by layer onto a core substrate
Construction of Laminated Multi layer Structures

Key Raw Materials:
1. Thin Core laminates 0.5-1.50 mm
2. Prepreg material 0.5-1.40 mm
3. Treated Copper foil 10-35 um

Manufacturing route options:

1. Copper foils bonded with prepreg
2. Rigid laminates bonded with prepreg
Option -1

Stack bonding two DS boards using Prepreg

Routes for 4-layer MLB Construction
Option - 2

Stack bonding a DS board and adding copper foil for final two layers
Major Steps in 4-layer **Construction**

1. Process a Double sided board - on 0.80 mm core
2. Lay 0.40 mm thick prepreg on either side
3. Lay 35 microns copper on top of prepreg on either side
5. Cool to RT
6. Drill the required via holes
7. Plate through the holes
8. Pattern top and bottom
9. Post finish

Legend Print
Interconnect Hole Formation

- **Mechanical Drilling**
- **Laser Drilling**
  - YAG/CO$_2$ Laser
    - Excimer (UV) laser or laser ablation
- **Photochemical via formation**
  - Liquid photo-definable dielectric layer
  - Dry film photo-definable dielectric layer
- **Etching**: Wet etching and dry plasma
Interconnect Hole Formation

Laser drilling & SBU
- build-up layer
- laminate RCC core
- UV-YAG laser ablation
- pattern-plating Cu
- etch tracks and pads

Photovia formation
- inner layer
- non-reinforced resin dielectric base core
- create window by etching
- plasma etching of dielectric
- final pattern formation after plating

Plasma etch and microvia
- copper overhang
- copper etch down to remove overhang

Figure Source: Dr Gerard Edwards, University of Bolton, UK
Mechanical Drilling Limitations

• Technical aspects
  – minimum hole dia ~0.15mm
  – Registration
  – Debris and smear generation

• Economical aspects
  – Sequential process
  – High investment of multi-spindle machines
  – Drill bits, entry/exit foil material
  – De-burring and de-smearing

• The maximum rpm obtainable with conventional spindles has prevented the smallest diameter drill bits from operating at their most efficient cutting speeds.