Hands-On PCB Engineering

Lecture 9: Fabrication, Assembly, DFx
PCB Stackups [Simple]

- Two types of materials:
  1. Laminate (dielectric, insulator)
  2. Foil (copper, conductor)

- Laminate is usually based on fiberglass epoxies
PCB Stackups [Advanced]

- Three types of materials:
  1. Core (dielectric, insulator, copper on both sides)
  2. Prepreg (dielectric, insulator, “glue”)
  3. Foil (copper, conductor)

- Need more layers?
  ○ Add more cores
  ○ Inherently symmetric
Fabrication

- How do we actually draw our board?
- Start with a flat sheet of copper
- End with a bunch of traces
- All about the double negatives
  - We’re not going to draw all the traces we want to have
  - We’re going to remove all the ones we do not want
Fabrication

- First we cover everything with a resist.
- This material is specifically designed to resist UV light.
Fabrication

- Next, we add the **negative** of our board
- This is the opposite of what you route
  - Fills in all the gaps instead!
  - Black is “film”
  - White is air/nothing
Fabrication

- We add the **film** directly on top of the **resist**
- Then we shine UV light onto the board (**develop**)

- Resist + UV = ?
- Resist + Film + UV = ?
Fabrication

- We add the **film** directly on top of the **resist**
- Then we shine UV light onto the board (**develop**)

- Resist + UV = **No Change**
- Resist + Film + UV = **Goodbye!**
Fabrication

- Now we *etch* away all the remaining copper
- Note the same *resist* also stops this chemical
- Two for one!
Fabrication

- Final step!
- **Strip** the last of the **resist**
- Can’t resist everything...
Fabrication

- Repeat the above process as many times as needed
- Sandwich all the layers together
  - Using prepreg as glue
- Add heat+pressure, George Foreman Grill it
Vias

- Now we add our vias!
- Lots of different types, depending on how fancy you want to be
- 99% of all vias are PTH:
  - Non-plated through hole (NPTH) is just a hole
  - Plated through hole (PTH) is a hole that gets connected with copper
Vias

- Quiz!
- Which via is NPTH, and which is PTH?
Soldermask/Silkscreen

- Add **soldermask** to prevent parts from shorting
  - “The green stuff”
- Add **silkscreen** to label/help with bringup
Finish

- Most popular surface finishes: ENIG and HASL

<table>
<thead>
<tr>
<th>Surface Finish Options</th>
<th>HASL (SnPb)</th>
<th>Lead-Free HASL</th>
<th>Electroless Nickel Immersion Gold (ENIG)</th>
<th>Immersion Silver (IAg)</th>
<th>Organic Solderable Coatings (OSP)</th>
<th>Immersion Tin (ISn)</th>
</tr>
</thead>
<tbody>
<tr>
<td>RoHS Compliant?</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Flat Surface?</td>
<td></td>
<td>✓</td>
<td></td>
<td></td>
<td></td>
<td>✓</td>
</tr>
<tr>
<td>Aluminum Wire Bond?</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>✓</td>
</tr>
<tr>
<td>Solderability (Wettability)</td>
<td>Best</td>
<td>Good</td>
<td>Good</td>
<td>Better</td>
<td>Good</td>
<td>Good</td>
</tr>
<tr>
<td>Shelf Life</td>
<td>&gt;1 Year</td>
<td>&gt;1 Year</td>
<td>&gt;1 Year</td>
<td>6-11 Months</td>
<td>6-11 Months</td>
<td>6 Months</td>
</tr>
<tr>
<td>Costs</td>
<td>$</td>
<td>$</td>
<td>$$$$</td>
<td>$§</td>
<td>$</td>
<td>$§</td>
</tr>
<tr>
<td>Popularity</td>
<td>High</td>
<td>Med</td>
<td>High</td>
<td>Medium</td>
<td>Low</td>
<td>Low</td>
</tr>
</tbody>
</table>

- I have never heard of the other ones, TBH
- If you can afford it, just get ENIG
Fabrication – Summary

**Internal Layers**
- Copper Clad Laminate (Core)
- Imaging Resist
- UV Print
- Develop Uncured Resist
- DES – (Develop Etch Strip)
- Oxide
- Unused Cu
- Strip Protective Resist
- Laminate (Multilayer)
- Drill

**External Layers**
- Electroless Plate (Hole Wall)
- Panel Plate
- Imaging Resist
- UV Print
- Develop Resist
- Circuitry Sn plate
- SES – (Strip Etch Strip)

**Soldermask**
- Coat Soldermask
- UV Print
- Develop Uncured SM
- Surface Finish
- Silkscreen (legend)
- Fabrication (Rout & Score)
- Test & Inspect
- Package & Ship
- Etch Cu
- Strip Protective Sn
Assembly

- Now we have a fabricated board!
- What to do with it?
- How do we get to here?
Pads + Holes

- SMD/SMT components sit flush to the board
  - Pads - Use exposed copper for the connection
- THT components use leads that insert into holes
Manual Assembly

- What most of you will probably be doing!
- Pick it up, put it on the board, solder it on
- Repeat until complete!
Automated Assembly

- Manual assembly is not an option for industry
- Extensive use of “Pick-and-Place” machines
- Put each component where it needs to go
Design For X

- Many methodologies exist when designing PCBAs
- Design for:
  - Test
  - Manufacturing
  - Inspection
  - Variability
- Comes down to what you want to prioritize
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Manufacturing</td>
<td>Person who pays for the design</td>
<td>Person who designs the PCBA</td>
<td>Can't use advanced design techniques</td>
</tr>
<tr>
<td>Test</td>
<td>Person who tests / &quot;brings-up&quot; the design</td>
<td>Person who designs the PCBA</td>
<td>Need to fit extra components in</td>
</tr>
<tr>
<td>Inspection</td>
<td>Quality control team</td>
<td>Person who designs the PCBA</td>
<td>Can't optimize design as much</td>
</tr>
<tr>
<td>Variability</td>
<td>Design engineers</td>
<td>Person who designs the PCBA</td>
<td>Need to fit extra components in</td>
</tr>
<tr>
<td>Environment</td>
<td>Person who designs the PCBA</td>
<td>Design engineers</td>
<td>Limited subset of materials to use</td>
</tr>
</tbody>
</table>
Design For Manufacturing

- Designing things that can be fabricated and assembled
- Sometimes simple:
  - Don’t put components on top of each other
  - Don’t have two different nets 1 nm apart
- Sometimes complex:
  - Tall and short components should be farther apart
  - A via’s width has to be at least 1/10th the board thickness
Design For Manufacturing

- Often an iterative process!
- Lots of back and forth with the manufacturer
- Most vendors have software that can do DFM checks
- Sometimes given as a DRC File
  - Design Rule Checklist
  - Sets the limits in your PCB ECAD tool
  - Don’t need to keep uploading and waiting
Tempo Automation Overview

#spinaboard | tempoautomation.com
**Turnkey PCBA Manufacturing**

**SMART SOURCING**
- Real-time API for faster sourcing feedback
- Pre-purchasing on BOM Components
- Network of pre-qualified/audited PCB vendors
- Matrix-based pricing arrangements
- DFM/DFA checks
- Stackup creation

**IN-HOUSE**
- Factory in Downtown San Francisco
- Purpose-built from ground up to optimize speed and quality
- Automated polymerics (underfill/staking)
- 4x Total Lines
- 2x Flight Production Lines
- Selective wave solder for Through Hole parts

**Fabrication (PCB)**
- Network of pre-qualified/audited PCB vendors
- Matrix-based pricing arrangements

**Components**
- Real-time API for faster sourcing feedback
- Pre-purchasing on BOM

**DFMx**
- DFM/DFA checks
- Stackup creation

**Assembly**
Factory in San Francisco?
Tempo is good at building...

SWEET SPOT

<table>
<thead>
<tr>
<th>Fab Complexity</th>
<th>Moderate-to-high fab complexity: 4-24 layers, fine pitch BGAs, HDI designs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Assy Complexity</td>
<td>High placement count: 1000+ unique SMT placement count</td>
</tr>
<tr>
<td>Volume</td>
<td>Low volume production &amp; prototype: Qty 1 to 1000 for each run</td>
</tr>
<tr>
<td>Leadtime</td>
<td>Need for speed: Needed it yesterday</td>
</tr>
</tbody>
</table>

**EXAMPLE 1: FIGHTER JET FLIGHT COMPUTER**

<table>
<thead>
<tr>
<th>Fab Complexity</th>
<th>- 18L, 10” x 4”, 87mil final thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>- IPC Class 2</td>
</tr>
<tr>
<td></td>
<td>- 500 vias in pads</td>
</tr>
<tr>
<td></td>
<td>- Mixed copper weights</td>
</tr>
<tr>
<td></td>
<td>- Controlled impedances</td>
</tr>
<tr>
<td></td>
<td>- Fine pitch BGA</td>
</tr>
<tr>
<td>Assy Complexity</td>
<td>- 1200 SMT placement per board</td>
</tr>
<tr>
<td></td>
<td>- 11 TH placement per board</td>
</tr>
<tr>
<td>Volume</td>
<td>10-20 each run</td>
</tr>
<tr>
<td>Leadtime</td>
<td>10 days</td>
</tr>
</tbody>
</table>

**EXAMPLE 2: SURGICAL-ASSISTED ROBOT**

<table>
<thead>
<tr>
<th>Fab Complexity</th>
<th>- 14L, 13” x 9.5”, 93mil final thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>- IPC Class 2</td>
</tr>
<tr>
<td></td>
<td>- 130 vias in pads</td>
</tr>
<tr>
<td></td>
<td>- 3mil width, 4mil spacing</td>
</tr>
<tr>
<td></td>
<td>- Controlled impedances</td>
</tr>
<tr>
<td>Assy Complexity</td>
<td>- 1112 SMT placement per board</td>
</tr>
<tr>
<td></td>
<td>- 10 TH placement per board</td>
</tr>
<tr>
<td>Volume</td>
<td>10 each run</td>
</tr>
<tr>
<td>Leadtime</td>
<td>8 days</td>
</tr>
</tbody>
</table>
Software-driven factory

CAD software file → Exported to image file → Reviewed by humans → Software-enabled labor-dependent production = Fast, Seamless, Transparent, Reliable, High quality
Traditional contract manufacturers take too long

**Quoting**
- Proto: 2-3 DAYS
- Complex: 10-15 DAYS

**Production**
- Proto: 10-12 DAYS
- Complex: 15-20 DAYS

**Debugging & rework**
- Proto: 3-5 DAYS
- Complex: 5-15 DAYS

- Slow
- Opaque
- Arduous
- Low quality
The key to the Tempo process – Automated Data Intake

Automated Feature Extraction & Preservation of Design Intent

- Hosted on ITAR Compliant AWS OpenGov Cloud Server -

<table>
<thead>
<tr>
<th>SMT placement count</th>
<th>315</th>
</tr>
</thead>
<tbody>
<tr>
<td>Through hole placement count</td>
<td>0</td>
</tr>
<tr>
<td>Leadless placement count</td>
<td>9</td>
</tr>
</tbody>
</table>

Board specs:
- Board Height (in): 4.72
- Board Width (in): 2.79
- Layer Count: 4
- Minimum Finished Hole Size (in): 0.01
- Minimum Trace Width (mil): 5
- Minimum Trace to Trace Spacing (mil): 5.7
- Buried Vias: No
- Blind Vias: No
- SMT Placement Sides: single
- Number of Via-in-pad: 1
Real-time BOM Feedback
Package Matching
Upload a design today
Go to:
http://secure.tempoautomation.com/rfq

#spinaboard | tempoautomation.com
Questions?

Thanks for listening!