Notice

Notification of Proprietary Information: This document contains proprietary information of TTM and its receipt or possession does not convey any rights to reproduce or disclose its contents, or to manufacture, use, or sell anything it may describe. Reproduction, dissemination, disclosure, or use, in whole or in part, without specific written authorization of TTM is strictly forbidden. All data contained within this document are subject to this restriction.
Objectives of this Presentation

- **Cost Adders:** Factors associated with manufacturing PCBs.
  - Design Guidelines: use green column whenever possible
  - Panel utilization
  - Variable
  - Fixed
  - Other visuals for reference

- **Class 2 vs Class 3:** Cost comparison and options

- **Material:** Cost increased with speed

- **Increased Technology:** How it affects complexity and cost

- **PCB Cost Index:** Progressively more complex stack-ups

- **Microvia Sublaminations (HDI)**
TTM NA Manufacturing Capability: Refer to the technology tables to select the best-cost design when possible

• ‘Standard’
  Current everyday capability with no premium

• ‘Advanced’
  Current everyday capability with a small premium
  – Extra process steps
  – More expensive raw material cost

• ‘Engineering’ = Viasystems maintains experience with and can fabricate on request with a significant premium; contact Viasystems engineering to discuss your requirements

• ‘Development’ = Process development in-process based on customer demand and collaboration, will cost you an arm and a leg
## TTM NA PCB Technology

### Trace & Space

<table>
<thead>
<tr>
<th>External Trace</th>
<th>External Space</th>
<th>Internal Trace</th>
<th>Internal Space</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.004&quot; - 100 µm</td>
<td>0.004&quot; - 100 µm</td>
<td>0.004&quot; - 100 µm</td>
<td>0.004&quot; - 100 µm</td>
</tr>
<tr>
<td>0.003&quot; - 75 µm</td>
<td>0.003&quot; - 75 µm</td>
<td>0.003&quot; - 75 µm</td>
<td>0.003&quot; - 75 µm</td>
</tr>
<tr>
<td>0.0025&quot; - 63 µm</td>
<td>0.0025&quot; - 63 µm</td>
<td>0.0025&quot; - 63 µm</td>
<td>0.0025&quot; - 63 µm</td>
</tr>
</tbody>
</table>

### Drilled Via Size

<table>
<thead>
<tr>
<th>Drill Diameter</th>
<th>Pad Diameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.010&quot; - 254 µm</td>
<td>0.020&quot; - 508 µm</td>
</tr>
<tr>
<td>0.008&quot; - 200 µm</td>
<td>0.016&quot; - 400 µm</td>
</tr>
<tr>
<td>0.006&quot; - 150 µm</td>
<td>0.012&quot; - 300 µm</td>
</tr>
<tr>
<td>0.004&quot; - 100 µm</td>
<td>0.004&quot; - 100 µm</td>
</tr>
</tbody>
</table>

### Aspect Ratio

<table>
<thead>
<tr>
<th>Via Diameter</th>
<th>Pad Diameter</th>
<th>Microvia</th>
</tr>
</thead>
<tbody>
<tr>
<td>150 µm &amp; 125 µm</td>
<td>0.004&quot; - 100 µm</td>
<td>2013</td>
</tr>
<tr>
<td>300 µm &amp; 254 µm</td>
<td>0.008&quot; - 200 µm</td>
<td>2013</td>
</tr>
<tr>
<td>0.6:1</td>
<td>0.8:1</td>
<td>1:1</td>
</tr>
</tbody>
</table>

### Via Diameter

- Standard: 150 µm, 0.004" - 100 µm, 0.006" - 150 µm
- Advanced: 125 µm, 0.003" - 75 µm, 0.005" - 125 µm
- Engineering: 125 µm, 0.003" - 75 µm, 0.005" - 125 µm
- Development: 125 µm, 0.003" - 75 µm, 0.005" - 125 µm

### Pad Diameter

- Standard: 300 µm, 0.010" - 254 µm
- Advanced: 250 µm, 0.008" - 200 µm
- Engineering: 250 µm, 0.008" - 200 µm
- Development: 250 µm, 0.008" - 200 µm

### Microvia

- 2013
- 2013
- 2013
- 2014
### Microvia Stack-up

<table>
<thead>
<tr>
<th></th>
<th>Standard</th>
<th>Advanced</th>
<th>Engineering</th>
<th>Development</th>
</tr>
</thead>
<tbody>
<tr>
<td># Microvia layers</td>
<td>1+1</td>
<td>2+2</td>
<td>6+6</td>
<td>&gt;7+7</td>
</tr>
<tr>
<td>Buried Sub</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Stacked MicroVias</td>
<td>No</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
</tr>
</tbody>
</table>

### Microvia Materials

- **Standard**
  - Std FR4
  - Laser Prepregs

- **Advanced**
  - Low loss Epoxy
  - Polyimide & BT

- **High Temp FR4**
  - Lead Free Assembly
  - HDI
  - Flex & Rigid Flex

- **Layer Count**
  - Up to 24
  - Up to 3.30 mm-0.130"

- **Thickness**
  - Up to 7.62 mm-0.300"

### Attributes

- **Layer Count**
  - Up to 24
  - 26 to 44
  - 46 to 60
  - >60
  - Up to 10.16 mm-0.400"

- **Thickness**
  - >10.16 mm-0.400"

### Laminate Materials

- **High Speed**
  - Low Loss
  - RF & Microwave

- **High Temp FR4**
  - Lead Free Assembly
  - HDI
  - Flex & Rigid Flex

- **High Speed**
  - Low Loss
  - BT & Polyimide

- **Stablcor**

- **2013**

- **2013**

- **2013**

- **2014**

- **LCP**
  - Non-Reinforced

- **Film Based**

- **Non-Reinforced**

- **2013**

- **2013**

- **2013**

- **2014**

- **Film Lased BUM**

- **2013**

- **2013**

- **2013**

- **2014**

- **Film Based**

- **Non-Reinforced**
## TTM NA PCB Technology

### Embedded Passives
- **Standard:** 0.002" - 50 µm Capacitive Cores
- **Advanced:** Buried Resistors FaradFlex™ MC, Dupont™ HK-04
- **Engineering:** High Dk Ceramic Filled
- **Development:** Embedded Active/Passive Devices

### Surface Finishes
- **Standard:** HASL OSP, Immersion Tin
- **Advanced:** Lead free HASL Immersion Silver, Immersion Gold
- **Engineering:** E-Less Gold, Multiple ENEPIG
- **Development:** Neutral Eless Au Ormecon

### Solder Mask
- **Registration Min opening:**
  - Standard: 0.0025"- 64 µm, 0.008"- 200 µm
  - Advanced: 0.002"- 50 µm, 0.006"-150 µm
  - Engineering: 0.001"- 25 µm, 0.006"- 150 µm
  - Development: <0.001"- <25 µm
  - Min opening: 0.004"- 100 µm

### Via Fill/Cap Plate
- **Min Drilled hole Aspect ratio:**
  - Standard: 0.012" - 300 µm, 8:1, Non-CVF & CVF
  - Advanced: 0.010" - 254 µm, 10:1, Non-CVF & CVF
  - Engineering: 0.008" - 200 µm, 6.5:1, Non-CVF
  - Development: 0.008" - 200 µm, Copper CVF
- **Year:**
  - Standard: 2013
  - Advanced: 2013
  - Engineering: 2013
  - Development: 2014

---

**CVF = Conductive Via Fill**

**Non-CVF = Non-Conductive Via Fill**

**TTM Technologies**
Factors that Drive PCB Cost

1) Manufacturing **Panel Utilization** (how many PCBs fit on the master panel)
2) Performance **Class** (IPC 6012B Class 2 vs 3)
3) **Layer Count** (total number of required cores)
4) Design **Complexity**, including minimums/maximums for:
   - Line Widths and Feature Spacing / Impedance tolerance
   - Drilled Hole Size (Aspect Ratios – (PCB Thickness/Diameter)
   - Overall PCB Thickness (Equipment Limitations)/ Aspect Ratios
   - Annular Ring Requirements (Registration Capabilities)
   - Copper Weights (Cost, Availability, and impact on Etching)
5) Type of dielectric **Material**
6) **HDI/High Technology** (via-in-pad, buried capacitance)
7) Process **Yield** (often a **Hidden Cost**)
Panel Utilization is the most important factor affecting PCB cost

- PCB vendors manufacture **panels**, not individual PCBs
- Manufacturing panels come in various sizes. Some are optimized for special process or materials, others are optimized for volume production, and many panel sizes are vendor-specific
- **18” x 24”** manufacturing panels are optimized for “standard” PCB manufacturing equipment and are considered “**universal**” in that all vendors can process them
  - “**Working Area**” (minus edge features - coupons, tooling, etc.) is approx 16”x22”
  - **Other sizes include 21”x24”, 14”x24”, 12”x18”, and 12”x12”**
- Panel utilization must be considered in the very early phases of product design, **NOT AT PCB LAYOUT**, since product attributes often dictate form factor
  - and circuit size
- When calculating units per production panel, include rails/keep-outs required for assembly
Class 3 requires 2.50” min (63.5mm) on 2 sides, dependent on impedance configurations and IST coupon requirements

Note: If board geometry permits, scoring can be used
Manufacturing Panel Utilization- Class 2

Excellent Panel Utilization

- Total usable area 371 in² total
- Circuit area (including assembly rails) 358 in²
- 97% panel utilization

Poor Panel Utilization

- Total usable area 371 in² total
- Circuit area (including assembly rails) 187 in²
- 50% panel utilization

PCB COST = 2X
Manufacturing Panel Utilization – Nesting

No Nesting
Panel Yield: 6 parts

Circuits Nested
Panel Yield: 8 parts
Class 2 vs. Class 3 Cost Adders

IPC6012 Class 2 vs. Class 3 = 10-20% cost adder

- **IPC Class 2** requires cross-sections to be done on a *sampling basis* per customer requirement, if any. Most customers do not request sections.

- **IPC Class 3** requires cross-sections to be done on 100% of the manufacturing panels. Cost impact is as follows:
  - *Greatly increased lab time* for sectioning and report creation due to lower $C=0$ Sampling Index
  - .10 for Class 3 versus 1.5
  - Coupons must be *removed, documented, and retained* (up to 20 yrs for some customers)
  - *Additional coupons* are required for Class 3 product versus Class 2, due to sectioning and retention requirements. PCB number up and panelization can be affected.

- **IPC Class 3** requirements result in some special process steps that consume production time and resources:
  - All panels are *serialized* to ensure full traceability
  - *Plating times* are 25% longer (1 mil min vs. 0.8 mil min)
  - *Additional in-process micro-sectioning* is completed in order to ensure compliance with Class 3 requirements for copper and annular ring
Aside from the section and reporting requirements, what are the two main design differences between Class 2 and Class 3 PCBs?

1. **Annular Ring** requirements (internal and external)
   - Requires design modification

2. **Plating Thickness** requirements
   - Mainly a process modification for the fabricator

The cost of going to Class 3 is primarily in the **reporting and sectioning** requirements of the spec, as well as the **lost space in panelization** due to addition coupon placement.

If Class 3 product performance without the cost is desired, design with Class 3 annular ring and require extra plating, but specify Class 2 performance.
   - *Eliminates extra coupon requirements and subsequent lost space on the panel.*

High-reliability fabricators usually have one main process flow, which is already set-up for Class 3 performance, for process simplification.
# Class 2 vs. Class 3 Cost Adders

<table>
<thead>
<tr>
<th>Attributes</th>
<th>IPC 6012/Cls 2</th>
<th>IPC 6012/Cls 3</th>
<th>IPC 6012/Cls 3A</th>
<th>MIL P 55110G</th>
<th>MIL-PRF-31032/5</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Annular Ring</strong></td>
<td>90° breakout acceptable</td>
<td>0.002” min, 0.001” min</td>
<td>0.002” min</td>
<td>0.002” min</td>
<td>as specified, default .004” as specified, default .002”</td>
</tr>
<tr>
<td><strong>External</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Internal</strong></td>
<td>90° breakout acceptable</td>
<td>0.002” min, 0.001” min</td>
<td>0.002” min</td>
<td>0.002” min</td>
<td></td>
</tr>
<tr>
<td>Bow &amp; Twist</td>
<td>0.75% max. if SMT used</td>
<td>0.75% max. if SMT used</td>
<td>0.5% max</td>
<td>As specified; otherwise, 1.5% max</td>
<td>As specified; otherwise, 1.5% max</td>
</tr>
<tr>
<td>Dielectric Thickness</td>
<td>If not specified, 0.0035” min .001” 25 μm</td>
<td>If not specified, 0.0035” min .001” 25 μm</td>
<td>If not specified 0.0035” 2 plys min. .001” 25 μm</td>
<td>as specified if not specified 0.0035” min .001” 25 μm</td>
<td>Min dielectric shall be as specified</td>
</tr>
<tr>
<td>Cu plating/Ductility</td>
<td>Elongation 12% min; Tensile strength 36,000 psi min</td>
<td>Elongation 12% min; Tensile strength 36,000 psi min</td>
<td>Elongation 18% min; Tensile strength 40,000 psi min</td>
<td>No Requirement</td>
<td>Elongation 18% min; Tensile strength 36,000 psi min</td>
</tr>
</tbody>
</table>

## Copper Plating

| Through holes               | avg .000787” 20 μm min .00709” 18 μm | avg .000984” 25 μm min .00787” 20 μm | avg .0015” 38μm min .0012” 30μm | .001” avg | .001” avg |
|                            | avg .000787” 20 μm min .00709” 18 μm | avg .000984” 25 μm min .00787” 20 μm | avg .0015” 38μm min .0012” 30μm | .001” avg | .001” avg |
| Blind vias                 | avg .000787” 20 μm min .00709” 18 μm | avg .000984” 25 μm min .00787” 20 μm | avg .0015” 38μm min .0012” 30μm | .001” avg | .001” avg |
| Buried vias                | avg .000787” 20 μm min .00709” 18 μm | avg .000984” 25 μm min .00787” 20 μm | avg .0015” 38μm min .0012” 30μm | .0006” avg | .0006” avg |
| Microvias                  | avg .000472” 12 μm min .00394” 10 μm | avg .000472” 12 μm min .00394” 10 μm | min .0008” 20μm min .00394” 10 μm | .00047” avg | .00047” avg |
| Conductor Width Reduction  | not >20% of min          | not >20% of min          | not >20% of min         | not to exceed 20% | within +/- 20% |
| Etchback                   | when specified .0002” -.003” | when specified .0002”-.003” | when specified .0002”-.0015” when required | .0002”-.002” | .0002”-.002” |
| Negative Etchback          | shall not exceed 0.001”   | shall not exceed 0.0005”  | not acceptable          | not acceptable | not acceptable |
| Cracks, barrel/corner      | None allowed             | None allowed             | None allowed            | None Allowed | None Allowed |
| Crazing                    | shall not span more than 50% of the distance between conductors | shall not span more than 50% of the distance between conductors | no crazing allowed | no crazing allowed | no crazing allowed |
| Delamination/Blistering     | shall not span more than 50% of the distance between conductors | shall not span more than 50% of the distance between conductors | no delamination allowed no blistering allowed | no delamination allowed no blistering allowed | no delamination allowed no blistering allowed |
| Nailheading                | Acceptable               | Acceptable               | Acceptable              | not to exceed 1.5 times cu foil thickness |                 |
Layer Count

- Every two layers (core) added to a multi-layer PCB will add ~20-25% to the unit cost

- Odd layers **do not** save money. For example, a 7 layer PCB costs the same as an 8 layer PCB, due to the standard manufacturing processes involved (**same number of cores**).
  - *Standard cores are supplied to fabricators as 2-sided (Cu foil on both sides)*
  - *Internal layers are processed in pairs (individual cores)*
Fixed Costs – Line Width and Feature Spacing

- Standard is .004”/.004” (0.102mm) L/S on ½ oz material

- .003”/.003” (0.076mm) L/S adds ~ 20-30%

- .0025”/.0025” (0.051mm) L/S adds ~ 50%

- If thinner lines and spacing are required within a confined area (BGA), using them **only where needed** will result in higher yields than using across the PCB.

```plaintext
.003  [-----]  .007

Rout in the center or Pads
.005/.005
```
Impedance Trace Identification Helpful Hint

➢ Use unique ‘D-codes’ (aperture line widths) for each impedance controlled trace so that the traces can be picked out and modified independently of other lines or fills. The D-codes can be as different as 0.0001 mil
  • For example, use a 4.01 mil for an impedance controlled line width, and 4 mil for a non-controlled line width (or different ohm lines), or line-drawn plane fills
Fixed Costs – Impedance Tolerance

- +/-10% tolerance is standard
- +/-7% tolerance adds 20%
- +/-5% tolerance adds 30%

- Paying for the tightened impedance tolerance may be of no benefit if the proposed stackup and material set are not robust
  - Standard TDR coupon per IPC-2221 requirements is used for verification
  - Actual PCB is more likely to have unpredictable variations than the coupon

- Etching tolerances are independent of line widths, and improved tolerances can be obtained by using wider lines
  - A 0.1 mil line width difference on a 3 mil 50 Ohm line equals 1.6%
  - A 0.1 mil line width difference on a 5 mil 50 Ohm line equals 0.9%
Impedance Variation due to Poor Copper Distribution

Thieving squares/dots or solid copper area fill eliminates poor feature distribution

*Maintain 0.200” min keep-out zone from any copper feature
Stackups – Symmetry

- With BGA devices prevalent in today’s designs, **flatness** is critical.
- Symmetry is the most important step that can be taken to ensure that no **warpage** or **residual stresses** are present in the final product.
- The build should be symmetrical about the Z-axis, including copper, prepregs, and cores.
- When possible, use single-ply cores, except for heavy copper designs.
- The best builds will utilize the **same core thickness and prepreg style** throughout.

---

<table>
<thead>
<tr>
<th>Layer</th>
<th>Thickness</th>
<th>Rigid Stack</th>
<th>Description</th>
<th>Layer</th>
<th>Thickness</th>
<th>Rigid Stack</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layer - 1</td>
<td>0.0020, 0.0005</td>
<td>0.0012</td>
<td>Tajiyo 4000-MP</td>
<td>Layer - 1</td>
<td>0.0020, 0.0005</td>
<td>0.0012</td>
<td>Tajiyo 4000-MP</td>
</tr>
<tr>
<td>Layer - 2</td>
<td>0.0061</td>
<td>0.0005</td>
<td>1oz P/G</td>
<td>Layer - 2</td>
<td>0.0061</td>
<td>0.0005</td>
<td>1oz P/G</td>
</tr>
<tr>
<td>Layer - 3</td>
<td>0.0005</td>
<td>0.0036</td>
<td>1/2oz Sig (Std PR)</td>
<td>Layer - 3</td>
<td>0.0005</td>
<td>0.0036</td>
<td>1/2oz Sig (Std PR)</td>
</tr>
<tr>
<td>Layer - 4</td>
<td>0.0012</td>
<td>0.0020</td>
<td>1/2oz Sig (Std PR)</td>
<td>Layer - 4</td>
<td>0.0012</td>
<td>0.0020</td>
<td>1/2oz Sig (Std PR)</td>
</tr>
<tr>
<td>Layer - 5</td>
<td>0.0012</td>
<td>0.0036</td>
<td>1oz P/G</td>
<td>Layer - 5</td>
<td>0.0012</td>
<td>0.0036</td>
<td>1oz P/G</td>
</tr>
<tr>
<td>Layer - 6</td>
<td>0.0006</td>
<td>0.0050</td>
<td>1/2oz Sig (Std PR)</td>
<td>Layer - 6</td>
<td>0.0006</td>
<td>0.0050</td>
<td>1/2oz Sig (Std PR)</td>
</tr>
<tr>
<td>Layer - 7</td>
<td>0.0012</td>
<td>0.0005</td>
<td>1/2oz Sig (Std PR)</td>
<td>Layer - 7</td>
<td>0.0012</td>
<td>0.0005</td>
<td>1/2oz Sig (Std PR)</td>
</tr>
<tr>
<td>Layer - 8</td>
<td>0.0020, 0.0005</td>
<td>0.0001</td>
<td>Tajiyo 4000-MP</td>
<td>Layer - 8</td>
<td>0.0020, 0.0005</td>
<td>0.0001</td>
<td>Tajiyo 4000-MP</td>
</tr>
</tbody>
</table>

**PREFERRED**

**NOT PREFERRED**
Variable Costs – Mechanical Drilling

- **The largest variable cost** in PCB manufacturing

- Drilling holes consumes drill bits at the rate of one bit for every 1000 holes drilled. (0.006” drills ~500-700 hits)

- Some fabricators (TTM included) will often include a certain amount of drill hits **per manufacturing panel** in the **base cost** (for TTM, up to 25K hits on a manufacturing panel are included)

- Additional costs will amount to 1-2% for every extra 10K holes

- Drill hole **size is not an adder until** size is **less than 8 mils** (0.2mm) in NA
  - 15-20% cost adder for 6 mil (0.15mm) holes
  - **Aspect Ratio (depth/diameter) must be considered**
Estimators look for the smallest drill size on the drill table within the fabrication drawing, which is always the smallest via size in the design.

General rule of thumb is that drill sizes in a drill table are “finished” hole sizes, but this is not entirely correct with respect to vias.

Hole sizes less than 18 mils (0.46mm) are almost always interconnecting vias, and are usually interpreted in the drill table as drill bit size, not finished hole size.

For maximum fabricator flexibility, tolerances on vias should be stated as “+.000”- (hole size)” or “no min”.

<table>
<thead>
<tr>
<th>SIZE</th>
<th>QTY</th>
<th>SYM</th>
<th>PLATED</th>
<th>TOL</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.008</td>
<td>2724</td>
<td>+E</td>
<td>YES</td>
<td>+0.002/-0.008</td>
</tr>
<tr>
<td>0.012</td>
<td>1317</td>
<td>×</td>
<td>YES</td>
<td>+0.003/-0.012</td>
</tr>
<tr>
<td>0.025</td>
<td>172</td>
<td>+</td>
<td>YES</td>
<td>+/-0.002</td>
</tr>
<tr>
<td>0.032</td>
<td>4</td>
<td>+M</td>
<td>YES</td>
<td>+/-0.003</td>
</tr>
<tr>
<td>0.035</td>
<td>14</td>
<td>+A</td>
<td>YES</td>
<td>+/-0.003</td>
</tr>
<tr>
<td>0.037</td>
<td>2</td>
<td>+E</td>
<td>NO</td>
<td>+/-0.003</td>
</tr>
<tr>
<td>0.041</td>
<td>28</td>
<td>+P</td>
<td>YES</td>
<td>+/-0.003</td>
</tr>
<tr>
<td>0.052</td>
<td>8</td>
<td>+I</td>
<td>YES</td>
<td>+/-0.003</td>
</tr>
<tr>
<td>0.06</td>
<td>2</td>
<td>+H</td>
<td>YES</td>
<td>+/-0.003</td>
</tr>
<tr>
<td>0.062</td>
<td>2</td>
<td>×</td>
<td>YES</td>
<td>+/-0.003</td>
</tr>
<tr>
<td>0.07087</td>
<td>5</td>
<td>+N</td>
<td>YES</td>
<td>+/-0.003</td>
</tr>
</tbody>
</table>
Variable Costs – Mechanical Drilling

- How about the **Number of Drill Sizes** on a particular design?

- This is a challenge for many fabricators, as having too many drill sizes results in more drill bits required in each “kit” for manufacturing, particularly if each drill size is used for a small number of holes, and can lead to a cost adder, if excessive.

- Finished hole sizes are drilled oversized by 4-6 mil (0.1-0.15mm) by fabricators in order to account for subsequent plating requirements (copper + final finish).

- Hole diameters that are close will generally be drilled with the same size drill bit, as long as finished diameter is within specified tolerance.
Example of excessive number of drill sizes

44 “unique” drill bit sizes
  - Combine sizes into one that will fit

93% of holes are defined by the first four bit sizes

70% of tool sizes are used for <10 holes on the manufacturing panel (1-up part/panel)
Fixed Cost Adders – Board Thickness and Via Aspect Ratio

Board Thickness Adders
- Any thickness up to 93 mils is considered standard
  - Thin cores cost more
- 94-120 mils adds ~10-20%
- 121-250 mils adds ~25-35%
- Design review is required for anything over 250 mils (6.35mm) (equipment limitations) or under 20 mils (0.5mm) (handling issues)

Aspect Ratio (thickness : min drill size) Adders
- Aspect ratios of 10:1 or less are considered standard
- >10:1-11.99:1 adds ~10-15%
- 12:1-12.99:1 adds ~20-25%
- 13:1-14.99:1 adds ~30-35%
Board Thickness and Via Aspect Ratio

Aspect ratio is the ratio of overall thickness to drill size, but is not a universal indicator of capability. Related to how easily the holes walls can be plated with copper.

- Examples (All 10:1 Aspect Ratios):
  - 120 mil (3mm) thick PCB with 12 mil (0.3mm) drill - STANDARD
  - 80 mil (2mm) thick PCB with 8 mil (0.2mm) drill - ADVANCED
  - 60 (1.5mm) thick PCB with 6 mil (0.15mm) drill - ENGINEERING

- Always understand the context of a fabricator’s maximum aspect ratio capability listing

<table>
<thead>
<tr>
<th>Aspect Ratio</th>
<th>Standard</th>
<th>Advanced</th>
<th>Engineering</th>
<th>Development</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.006&quot; drill</td>
<td>6.5:1</td>
<td>10:1</td>
<td>14:1</td>
<td></td>
</tr>
<tr>
<td>0.008&quot; drill</td>
<td>8:1</td>
<td>10:1</td>
<td>12:1</td>
<td>16:1</td>
</tr>
<tr>
<td>0.010&quot; drill</td>
<td>10:1</td>
<td>12:1</td>
<td>16:1</td>
<td>18:1</td>
</tr>
<tr>
<td>0.012&quot; drill</td>
<td>10:1</td>
<td>14:1</td>
<td>18:1</td>
<td>20:1</td>
</tr>
<tr>
<td>0.0135&quot; drill</td>
<td>10:1</td>
<td>16:1</td>
<td>20:1</td>
<td>24:1</td>
</tr>
</tbody>
</table>
Fixed Cost Adders – Surface Finish

Surface Finish Adders

- **OSP**, Immersion **Silver**, **ENIG**, and **SnPb HASL** are **NOT** cost adders
- **ENEPIG** adds 10% per manufacturing panel
- **Lead-Free HASL** adds 5% per manufacturing panel (subcontract is higher)
- **Hard Gold** is priced per in\(^2\) area at the current market price of gold
- Selective Finishes (**combinations**) are priced on an individual basis
Miscellaneous Adders

- Scoring – No Cost Adder
- Milling – 5-10% (at +/-5 mil tolerance)
- Edge Plating – 5-10% (plus additional costs associated with reduced #up on the manufacturing panel, if affected)
- Back Drilling – 15-25%
- Castellations – 10-15% depending on size and density
- Cavities – Avoid if Possible! – price is design specific
Glossy or Matte Soldermask – No Cost Adder

Different Color (non-green) Soldermask – 5-10%, depending on lot size.
- Use Caution with Black or White Soldermask (cost, capability, and yield issues)
- The cost adder for different soldermask is due to having to break down the set-up for one color in order to use another (time consuming, storage issues, etc.)

Soldermask is a Continuous Process, and the application is independent of the PCB design, except for 2 oz copper and greater (double coat)

However, the Silkscreen process is a Batch Process, and is set-up for each particular work order / part number. Only Printed nomenclature is a cost adder, an option for greater legibility when text is smaller than standard (~.035” high)

Customers that use different soldermask colors to identify RoHS or lead-free products should consider using a different silkscreen color instead.
Internal annular ring measurements DO NOT include through hole plating.

Figure 9-3  Internal Annular Ring
Internal annular ring is measured from Drilled edge to etched edge of foil.
External Annular Ring

External annular ring measurements DO include through hole plating.

Figure 9-2  External Annular Ring

Min. Annular Ring

Plated-Through Hole

Land
External Annular Ring
## PCB Design Guidelines

### Mechanical Drilled Blind, Buried, and Through Holes

<table>
<thead>
<tr>
<th>Drill</th>
<th>Pad</th>
<th>Anti-Pad</th>
<th>PCB Thickness</th>
<th>Aspect Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Drill &amp; Pad Diameter</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.006&quot;</td>
<td>0.016&quot;</td>
<td>0.026&quot;</td>
<td>up to 0.039&quot;</td>
<td>6.5:1</td>
</tr>
<tr>
<td>0.008&quot;</td>
<td>0.018&quot;</td>
<td>0.028&quot;</td>
<td>up to 0.062&quot;</td>
<td>7.75:1</td>
</tr>
<tr>
<td><strong>IPC Class 2</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.010&quot;</td>
<td>0.020&quot;</td>
<td>0.030&quot;</td>
<td>up to 0.100&quot;</td>
<td>10:01</td>
</tr>
<tr>
<td>0.012&quot;</td>
<td>0.022&quot;</td>
<td>0.032&quot;</td>
<td>up to 0.120&quot;</td>
<td>10:01</td>
</tr>
<tr>
<td>0.0135&quot;</td>
<td>0.024&quot;</td>
<td>0.034&quot;</td>
<td>up to 0.135&quot;</td>
<td>10:01</td>
</tr>
<tr>
<td><strong>1/2 oz copper</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.012&quot;</td>
<td>0.022&quot;</td>
<td>0.032&quot;</td>
<td>up to 0.120&quot;</td>
<td>10:01</td>
</tr>
<tr>
<td>0.0135&quot;</td>
<td>0.028&quot;</td>
<td>0.038&quot;</td>
<td>up to 0.135&quot;</td>
<td>10:01</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Drill</th>
<th>Pad</th>
<th>Anti-Pad</th>
<th>PCB Thickness</th>
<th>Aspect Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Drill &amp; Pad Diameter</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.008&quot;</td>
<td>0.023&quot;</td>
<td>0.033&quot;</td>
<td>up to 0.062&quot;</td>
<td>7.75:1</td>
</tr>
<tr>
<td><strong>IPC Class 3</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.010&quot;</td>
<td>0.025&quot;</td>
<td>0.035&quot;</td>
<td>up to 0.100&quot;</td>
<td>10:01</td>
</tr>
<tr>
<td>0.012&quot;</td>
<td>0.027&quot;</td>
<td>0.037&quot;</td>
<td>up to 0.120&quot;</td>
<td>10:01</td>
</tr>
<tr>
<td>0.0135&quot;</td>
<td>0.028&quot;</td>
<td>0.038&quot;</td>
<td>up to 0.135&quot;</td>
<td>10:01</td>
</tr>
</tbody>
</table>
# PCB Design Guidelines

## Mechanical Drilled Blind, Buried, and Through Holes

<table>
<thead>
<tr>
<th>Drill &amp; Pad Diameter</th>
<th>8 Layers or less</th>
<th>&gt;8Layers</th>
<th>IPC Class 2</th>
<th>Pad dia over drill</th>
<th>Pad dia over drill</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/4 oz copper</td>
<td>.010&quot;</td>
<td>.010&quot;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3/8 oz copper</td>
<td>.010&quot;</td>
<td>.010&quot;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1/2 oz copper</td>
<td>.010&quot;</td>
<td>.010&quot;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1 oz copper</td>
<td>.012&quot;</td>
<td>.012&quot;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2 oz copper</td>
<td>.014&quot;</td>
<td>.014&quot;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3 oz copper</td>
<td>.016&quot;</td>
<td>.016&quot;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4 oz copper</td>
<td>.018&quot;</td>
<td>.018&quot;</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Drill &amp; Pad Diameter</th>
<th>8 Layers or less</th>
<th>&gt;8Layers</th>
<th>IPC Class 3</th>
<th>Pad dia over drill</th>
<th>Pad dia over drill</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/4 oz copper</td>
<td>.013&quot;</td>
<td>.015&quot;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3/8 oz copper</td>
<td>.013&quot;</td>
<td>.015&quot;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1/2 oz copper</td>
<td>.013&quot;</td>
<td>.015&quot;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1 oz copper</td>
<td>.015&quot;</td>
<td>.017&quot;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2 oz copper</td>
<td>.016&quot;</td>
<td>.018&quot;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3 oz copper</td>
<td>.019&quot;</td>
<td>.021&quot;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4 oz copper</td>
<td>.022&quot;</td>
<td>.024&quot;</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Microvia Annular Ring Guidelines

IPC 6016 Qualification & Performance Specification for HDI PCBs

Microvia Design Guidelines

Standard
Pad Diameter = 0.006” + Laser drill diameter

Advanced
Pad Diameter = 0.005” + Laser drill diameter

3.4.3 Annular Ring (External)

3.4.3.1 Capture Land to Microvia Capture lands shall have tangency at a minimum. Breakout is not allowed, unless the design and procurement documentation specify (i.e., landless microvia). See Figure 3-2.

Figure 3-2 Capture Land Registration

3.4.3.2 Plated-Through Holes External annular ring for plated-through holes shall be in accordance with the applicable sectional performance specification (i.e., IPC-6012, IPC-6013, etc.).
### Laser Drilled Blind & Buried Microvias

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Standard</th>
<th>Advanced</th>
<th>Engineering</th>
<th>DpMV™</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pad diameter</td>
<td>0.012&quot;</td>
<td>0.010&quot;</td>
<td>0.0086&quot;</td>
<td>0.012&quot;</td>
</tr>
<tr>
<td>Laser drill diameter</td>
<td>0.006&quot;</td>
<td>0.006&quot;</td>
<td>0.005&quot;</td>
<td>0.006&quot;</td>
</tr>
<tr>
<td>Dielectric thickness</td>
<td>0.0025&quot; - 0.003&quot;</td>
<td>0.0025&quot; - 0.004&quot;</td>
<td>0.0025 - 0.004&quot;</td>
<td>0.006&quot;</td>
</tr>
<tr>
<td>Aspect Ratio</td>
<td>0.4:1 - 0.5:1</td>
<td>0.5:1 - 0.8:1</td>
<td>0.5:1 - 0.8:1</td>
<td>one-to-one</td>
</tr>
<tr>
<td>Anti Pad</td>
<td>0.005&quot;/side</td>
<td>0.004&quot;/side</td>
<td>0.004&quot;/side</td>
<td>0.004&quot;/side</td>
</tr>
<tr>
<td>Dog Bone Offset</td>
<td>0.005&quot; space Pad-Pad</td>
<td>Pads Tangent</td>
<td>0.001&quot; Pad overlap</td>
<td>0.005&quot;Space</td>
</tr>
</tbody>
</table>
## PCB Design Guidelines: As Cu thickness increases, so do lines/spaces

### Internal & External Trace Widths

<table>
<thead>
<tr>
<th>Trace &amp; Space</th>
<th>Standard</th>
<th>Advanced</th>
<th>Engineering</th>
</tr>
</thead>
<tbody>
<tr>
<td>External Trace</td>
<td>.004&quot;</td>
<td>0.003&quot;</td>
<td>0.0025&quot;</td>
</tr>
<tr>
<td>External Space</td>
<td>.004&quot;</td>
<td>0.003&quot;</td>
<td>0.0025&quot;</td>
</tr>
<tr>
<td>Internal Trace</td>
<td>.004&quot;</td>
<td>0.003&quot;</td>
<td>0.0025&quot;</td>
</tr>
<tr>
<td>Internal Space</td>
<td>.004&quot;</td>
<td>0.003&quot;</td>
<td>0.0025&quot;</td>
</tr>
</tbody>
</table>

### Cu Weight

<table>
<thead>
<tr>
<th>Plated Layer starting</th>
<th>Trace &amp; Space</th>
<th>Trace &amp; Space</th>
<th>Trace &amp; Space</th>
<th>Finished Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>External 1/4 oz</td>
<td>0.003&quot;</td>
<td>0.0025&quot;</td>
<td>average 0.0016&quot;</td>
<td></td>
</tr>
<tr>
<td>External 3/8 oz</td>
<td>0.003&quot;</td>
<td>0.00275&quot;</td>
<td>average 0.0018&quot;</td>
<td></td>
</tr>
<tr>
<td>External 1/2 oz</td>
<td>0.004&quot;</td>
<td>0.0035&quot;</td>
<td>average 0.0021&quot;</td>
<td></td>
</tr>
<tr>
<td>External 1 oz</td>
<td>0.005&quot;</td>
<td>0.0045&quot;</td>
<td>average 0.0026&quot;</td>
<td></td>
</tr>
<tr>
<td>External 2 oz</td>
<td>0.008&quot;</td>
<td>0.0075&quot;</td>
<td>average 0.0038&quot;</td>
<td></td>
</tr>
<tr>
<td>External 3 oz</td>
<td>0.011&quot;</td>
<td>0.0105&quot;</td>
<td>average 0.005&quot;</td>
<td></td>
</tr>
<tr>
<td>External 4 oz</td>
<td>0.014&quot;</td>
<td>0.0135&quot;</td>
<td>average 0.0062</td>
<td></td>
</tr>
<tr>
<td>External 5 oz</td>
<td>0.017&quot;</td>
<td>0.0165&quot;</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Internal 1/4 oz</td>
<td>0.003&quot;</td>
<td>0.0025&quot;</td>
<td>0.004&quot;</td>
<td></td>
</tr>
<tr>
<td>Internal 3/8 oz</td>
<td>0.003&quot;</td>
<td>0.00275&quot;</td>
<td>0.006&quot;</td>
<td></td>
</tr>
<tr>
<td>Internal 1/2 oz</td>
<td>0.004&quot;</td>
<td>0.0035&quot;</td>
<td>0.006&quot;</td>
<td></td>
</tr>
<tr>
<td>Internal 1 oz</td>
<td>0.005&quot;</td>
<td>0.004&quot;</td>
<td>0.0012&quot;</td>
<td></td>
</tr>
<tr>
<td>Internal 2 oz</td>
<td>0.007&quot;</td>
<td>0.006&quot;</td>
<td>0.0026&quot;</td>
<td></td>
</tr>
<tr>
<td>Internal 3 oz</td>
<td>0.009&quot;</td>
<td>0.008&quot;</td>
<td>0.004&quot;</td>
<td></td>
</tr>
<tr>
<td>Internal 4 oz</td>
<td>0.011&quot;</td>
<td>0.010&quot;</td>
<td>0.0054&quot;</td>
<td></td>
</tr>
<tr>
<td>Internal 5 oz</td>
<td>0.013&quot;</td>
<td>0.012&quot;</td>
<td>0.0068&quot;</td>
<td></td>
</tr>
</tbody>
</table>
## PCB Design Guidelines

<table>
<thead>
<tr>
<th>Feature</th>
<th>Standard</th>
<th>Advanced</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drill-to-Copper</td>
<td>0.008&quot;</td>
<td>0.0065&quot;</td>
</tr>
<tr>
<td>Anti-Pad</td>
<td>0.010&quot; over Pad dia</td>
<td>0.008&quot; over Pad dia</td>
</tr>
<tr>
<td>Thermal Relief</td>
<td></td>
<td></td>
</tr>
<tr>
<td>non Plated Layer</td>
<td>0.006&quot;/side</td>
<td>0.004&quot;/side</td>
</tr>
<tr>
<td>Plated Layer</td>
<td>.008&quot;/side</td>
<td>.006&quot;/side</td>
</tr>
<tr>
<td>Circuit to PCB edge</td>
<td>0.025&quot;</td>
<td>0.010&quot;</td>
</tr>
<tr>
<td>Plane to PCB edge</td>
<td>0.025&quot;</td>
<td>0.010&quot;</td>
</tr>
<tr>
<td>Routing Tolerance</td>
<td>+/- 0.005&quot;</td>
<td>+/- 0.002&quot;</td>
</tr>
<tr>
<td>Edge of hole barrel to PCB Edge</td>
<td>0.025&quot;</td>
<td>0.020&quot;</td>
</tr>
<tr>
<td>Solder Mask Dam</td>
<td>0.004&quot; min</td>
<td>0.003&quot; min</td>
</tr>
<tr>
<td>Filled Vias</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Conductive</td>
<td>0.012&quot; drill min</td>
<td>0.010&quot; drill min</td>
</tr>
<tr>
<td>non-conductive</td>
<td>0.010&quot; drill min</td>
<td>0.008&quot; drill min</td>
</tr>
</tbody>
</table>

Via-in-Pad filled & plated over require wrap around plating
wrap around plating reduces trace & space capabilities on plated layers
**Copper** - The conductive layer of the PCB

- Thickness is often specified in terms of 'weight' (oz/ft\(^2\))
  - 1 oz copper weight = 1.35 mil
  - 1.35 mil = thickness of 1ft\(^2\) x 1ft\(^2\) copper square that weighs 1 oz

- Different copper weights are used depending on whether the conductor or plane will be used to carry a high current
  - Most common thickness is ½ oz (= 0.68 mil thickness)
  - Other common thicknesses include 1 oz, and 2 oz
  - Thickness above 2 oz will require design review and factory approval
    - Adds 15-20% (Design constraints)

- External layers of a PCB are always plated, and so the final thickness of copper will include the plating thickness
PCB Materials

Base Laminate Materials are comprised of three components:

- Copper Foil
- Woven Glass Fabrics
- Resin
PCB Construction

The printed circuit board stack-up is the combination of various material types, cores, and prepreg, including copper and dielectric thicknesses.

- **Core** = Fully cured dielectric between 2 copper foil layers
- **Prepreg** = partially cured dielectric used to bond cores and/or foil in the stack-up
- **Copper Foil** – The copper used on outer/inner (subs) layers of the PCB or on the core layers
Prepreg Glass Styles

106

1080

2113

2116

7628

Note: Due to material availability and general preferences, glass styles and resin contents used in Asia are often different to those used in North America.
Material Selection – High Performance

What material is best for a high-performance design?

- **Signal Integrity** performance (signal speed and signal loss) is often the driving factor in going to a higher cost material set.

- Other factors include:
  - *Thermal* performance
  - *Assembly* and *Rework*
  - Long term *Reliability*

- Going to a ‘higher speed’ material provides much improved signal loss (*attenuation*), but trace lengths also play a major role in determining loss characteristics.
  - Normally, once speeds of **2.5Gb/s** (1.25GHz) are employed, higher speed materials should be considered.
  - Signal loss on **shorter traces** is less, and standard high-Tg FR4 can therefore be used at speeds as high as **10Gb/s** as long as the trace lengths are short enough (a good rule of thumb is 3” or less).
### Laminate Cost Factors Based on Resin System

<table>
<thead>
<tr>
<th>Material Group/ Resin System</th>
<th>Vendor Specific</th>
<th>Cost Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>High Tg FR4 (Baseline sqft. Price)</td>
<td>Isola 370 HR</td>
<td>1</td>
</tr>
<tr>
<td>High Tg/Reliability-Filled</td>
<td>Nelco 4000-29, Isola 370HR</td>
<td>1</td>
</tr>
<tr>
<td>High Speed/ Mid Low Loss</td>
<td>Rogers Theta</td>
<td>1.1</td>
</tr>
<tr>
<td>High Speed/ Mid Low Loss</td>
<td>Nelco 4000-13 EP, Isola FR408</td>
<td>1.2</td>
</tr>
<tr>
<td>High Speed/ Low Loss</td>
<td>Nelco 4000-13EP SI</td>
<td>3X</td>
</tr>
<tr>
<td>High Speed/ Low Loss</td>
<td>Panasonic Megtron 6</td>
<td>4X</td>
</tr>
<tr>
<td>Polyimide</td>
<td>Nelco N7000-2, Isola P96, Arlon 35N</td>
<td>3X</td>
</tr>
<tr>
<td>High Frequency</td>
<td>Rogers 4350B, RO4003</td>
<td>5X</td>
</tr>
<tr>
<td>BT Packaging Substrate</td>
<td>Nelco N5000-32</td>
<td>3X</td>
</tr>
</tbody>
</table>

Note: Based on square foot pricing
Material Selection – High Speed Options

- Pure package with desired high-speed material
  - Although this will provide superior performance, it will also be the most expensive option, and should be used when all PCB layers require improved signal performance.

- Hybrid Builds
  - Often, high-speed signals can fit on just a few of the layers within the build.
  - Utilizing the high speed material on either one or both external layers (with standard FR4 on the remaining layers) may provide a desirable performance/cost benefit.
  - Hybrid builds should use a core construction (cores on external dielectrics) in order to maximize dielectric thickness consistency.

Check with factory for material compatibility.
When using a mixed material build, consider the effects of warpage due to the mixing of materials, and take steps to eliminate it.

PWB cost may increase, but the build will be more robust.
Yield will impact cost as PCB technologies are increased. Some of the major factors impacting yield are as follows:

Aggressive PCB design geometries: *
- Line Widths and Spacing Less than 4 mils (0.1mm)
- Drill Diameters Less than 10 mils (0.25mm)
- Pad Sizes Less than drill dia. + 10 mils (0.25mm) *
- Aspect Ratio greater than 10:1
- Drilled Hole to Copper Spacing Less than 8 mils (0.2mm)
- Anti-Pads Less than drilled dia. + 18 mils (0.46mm) *

Complexity of the manufacturing process (Added processes)
- HDI is a key contributor to higher complexity/ cost.
HDI (High Density Interconnect)

- The decision to go to high-density interconnect (HDI) technology can add **cost** to the PCB

- In many cases (0.5mm BGA, 0.4mm BGA, via-in-pad, reduced PCB size), the use of HDI is **unavoidable**
HDI – Technology Adds Cost (and Time)

Technology and Density Cost Adders include:

- **Blind Vias** *(Laser)*: 10-15% per Layer (second side is 5%)
- **Buried Vias** *(Laser)*: 40-50% (plus any sub filling costs if stacked)
- **Blind Vias** *(Mechanical)*: 50-100% (plus any sub filling costs if stacked)
- **Through Hole Fill** *(non-conductive)*: 20% per Panel
- **Subassembly Hole Fill** *(non-conductive)*: 15% per Layer
- **Microvia Copper Fill**: 20% per Layer
- **Buried Capacitance**: ~5% per Std BC Core (more for high performance matl’s)
- **Buried Resistance**: ~75% per Resistive Layer
HDI Multipliers

- Adders for Hole Fill, Laser Microvias, and/or Mechanical Blind/Buried holes are for the **process itself** and are not dependent on the total number of holes
  - *Once you add one blind/buried/filled via, you can use as needed to take advantage of the increased density*

- When Via Fill is considered, maintain a single hole size for optimized filling
  - *Copper filling **multiple microvias** on the same layer is as cost effective as filling just one microvia*
Laser drilled microvias with internal buried vias are preferred over mechanical blind vias for ease of manufacture, especially with respect to external layer wrap plate processing.

When considering blind vias, the following rules of thumb will help make the design more robust:

- Design the build so that the blind vias are symmetrical about the center axis and extend to the middle.
  - Example: 12L PCB with blinds L1-6 & L7-12, instead of L1-5 & 8-12.
- If possible, consider back drilling on long blind vias.
  - Example: 12L PCB with blinds L1-5 (consider a straight build with back drills)
  - Also less expensive and less time consuming than multiple lams.
- If only one set of blind vias is required, the build should be mirrored on the opposite side using a dummy subassembly and heavier copper to mimic the extra Cu plating on the blind via subassembly.
On subassemblies where the build is unbalanced, use **dummy subassemblies** and **copper weight changes** to force the build into a more symmetric state.

### PREFERRED

<table>
<thead>
<tr>
<th>Layer</th>
<th>Thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layer - 1</td>
<td>0.0020</td>
</tr>
<tr>
<td></td>
<td>0.0005</td>
</tr>
<tr>
<td>Layer - 2</td>
<td>0.0005</td>
</tr>
<tr>
<td></td>
<td>0.0060</td>
</tr>
<tr>
<td>Layer - 3</td>
<td>0.0020</td>
</tr>
<tr>
<td></td>
<td>0.0055</td>
</tr>
<tr>
<td>Layer - 4</td>
<td>0.0006</td>
</tr>
<tr>
<td></td>
<td>0.0060</td>
</tr>
<tr>
<td>Layer - 5</td>
<td>0.0005</td>
</tr>
<tr>
<td></td>
<td>0.0054</td>
</tr>
<tr>
<td>Layer - 6</td>
<td>0.0027</td>
</tr>
<tr>
<td></td>
<td>0.0060</td>
</tr>
<tr>
<td>Layer - 7</td>
<td>0.0006</td>
</tr>
<tr>
<td></td>
<td>0.0099</td>
</tr>
<tr>
<td>Layer - 8</td>
<td>0.0020</td>
</tr>
<tr>
<td></td>
<td>0.0005</td>
</tr>
</tbody>
</table>

**Primary Stack**

- Layer - 1: Taiyo 4000-MP 1/2oz P/G (Std Plt) 370H 1/2oz Sig 1/2oz P/G (Std Plt)
- Layer - 2: 1/2oz Sig 70H
- Layer - 3: 1/2oz Sig 1/2oz P/G (Std Plt) 370H
- Layer - 4: 1/2oz Sig 370H 1/2oz Sig
- Layer - 5: 370H 1/2oz Sig
- Layer - 6: 0.0060 (1-1652) 2oz P/G 370H 1/2oz Sig
- Layer - 7: 370H 1/2oz Sig
- Layer - 8: 1/2oz P/G (Std Plt) Taiyo 4000-MP

### NOT PREFERRED

<table>
<thead>
<tr>
<th>Layer</th>
<th>Thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layer - 1</td>
<td>0.0020</td>
</tr>
<tr>
<td></td>
<td>0.0005</td>
</tr>
<tr>
<td>Layer - 2</td>
<td>0.0006</td>
</tr>
<tr>
<td></td>
<td>0.0060</td>
</tr>
<tr>
<td>Layer - 3</td>
<td>0.0020</td>
</tr>
<tr>
<td></td>
<td>0.0055</td>
</tr>
<tr>
<td>Layer - 4</td>
<td>0.0006</td>
</tr>
<tr>
<td></td>
<td>0.0060</td>
</tr>
<tr>
<td>Layer - 5</td>
<td>0.0005</td>
</tr>
<tr>
<td></td>
<td>0.0054</td>
</tr>
<tr>
<td>Layer - 6</td>
<td>0.0027</td>
</tr>
<tr>
<td></td>
<td>0.0060</td>
</tr>
<tr>
<td>Layer - 7</td>
<td>0.0006</td>
</tr>
<tr>
<td></td>
<td>0.0099</td>
</tr>
<tr>
<td>Layer - 8</td>
<td>0.0020</td>
</tr>
<tr>
<td></td>
<td>0.0005</td>
</tr>
</tbody>
</table>

**Primary Stack**

- Layer - 1: Taiyo 4000-MP 1/2oz P/G (Std Plt) 370H 1/2oz Sig 1/2oz P/G (Std Plt)
- Layer - 2: 1/2oz Sig 70H
- Layer - 3: 1/2oz Sig 1/2oz P/G (Std Plt) 370H
- Layer - 4: 1/2oz Sig 370H 1/2oz Sig
- Layer - 5: 370H 1/2oz Sig
- Layer - 6: 0.0060 (1-1652) 2oz P/G 370H 1/2oz Sig
- Layer - 7: 370H 1/2oz Sig
- Layer - 8: 1/2oz P/G (Std Plt) Taiyo 4000-MP
The following examples use a 12 layer printed circuit to illustrate how different via configurations impact manufacturing complexity. We start with a standard through hole design and then add blind and buried vias, Microvias, and Via-In-Pad technologies.
Major Process Operations

Printed circuit fabrication requires many detailed processing steps. However, for the purpose of developing a basic cost index for engineering purposes, one way to look at the fabrication process is to keep track of the number of major operations:

- Inner layer processing: Photo image and etch
- Pattern plated layer processing: Photo image, copper plate and etch
- Via-In-Pad Wrap plating process (Copper panel plate + Image and barrel plate)
- Blind Via Wrap plating process (Copper panel plate)
- SMV plating process (Photo image and copper button plate)
- High pressure lamination
- Mechanical drilling
- Laser drilling
Via Structure: Basic 12 layer PCB with through hole via (Standard through hole PCB)

Note: to add or subtract layer pairs (i.e. 2 layers) the index will only change by 1 since each inner layer process produces two layers.

Example: For 14 layers the index would be 9, and for 10 layers the index would be 7
The via-in-pad process requires the filled vias to be plated separately, and also requires a fill and planarize step.

Via Structure: Through hole via-in-pad (VIP)

(PCB with 0.8mm pitch BGA)
Via Structure: Through hole and laser microvia L1-2

(PCB with 0.65mm pitch BGA or less – low pin count)
12 Layer PCB

Via Structure: Through hole and microvia L1-2, L12-11
(PCB with 0.65mm pitch BGA or less – low pin count)
12 Layer PCB

Via Structure: Through hole and blind via L1-6
(Increased density PCB with larger BGAs, typically 1.0mm or greater)
Via Structure: Through hole and blind via L1-6 with via-in-pad (two wrap cycles)
(Increased density PCB with 0.8mm pitch BGAs)
12 Layer PCB

Via Structure: Through hole, blind via L1-6 and L7-12
(Increased density PCB with 0.8mm pitch BGAs on both sides)
12 Layer PCB

Via Structure: Through hole and microvia-in-pad L1-2, L12-11 with MV copper fill

(PCB with **0.5mm** BGA or less – low pin count – via in pad)
12 Layer PCB

Via Structure: Through hole and microvia L1-2, L12-11, with buried via L2-11
(Most popular HDI construction – increased density, 0.5mm BGA, medium pin count)
Via Structure: Through hole and microvia L1-2, L12-11, stacked on an internal filled buried via L2-11 with copper over-plate

(Increased density PCB with 0.65mm BGA or less – low pin count)
Via Structure: Through hole and microvia L1-2, L12-11, stacked on an internal filled buried via L2-11 with copper over-plate plus copper filled microvias
(Increased density PCB with 0.5mm BGA or less – low pin count & via in pad)
12 Layer PCB

Via Structure: Through hole and microvia L1-2, L2-3, L12-11, L11-10 (staggered)

(PCB with 0.5mm BGA or less, medium pin count)
12 Layer PCB

Via Structure: Thru hole and microvia-in-pad L1-2, L2-3, L12-11, L11-10 (SMV stacked structure) (PCB with 0.4mm BGA, medium pin count, via in pad)
Via Structure: Through hole and SMV L1-2, L2-3, L12-11, L11-10, with L3-10 buried via (PCB with 0.4mm BGA, high pin count, via in pad – costly)
12 Layer PCB

Via Structure: Through hole and SMV L1-2, L2-3, L12-11, L11-10, with L2-11 buried via
(PCB with 0.4mm BGA, high pin count, via in pad – preferred)
12 Layer PCB

Via Structure: Thru hole & SMV L1-2, L2-3, L3-4, L12-11, L11-10, L10-9, & bur via L3-10
(PCB with 0.4mm BGA or less, very high pin count, via in pad – preferred)
12 Layer PCB

Via Structure: Through hole and SMV L1-2, L2-3, L3-4, L12-11, L11-10, L10-9, all via-in-pad plus a buried via L3-10 with fill and copper over-plate
(Extreme!)
Overview

Microvia Sub-lamination Interface Techniques
TTM Enabling Solutions

SMV®  DpMV™  DpSMV®  DpBV™  *ThermalVia™

FLAT-WRAP™  NextGen-SMV®  *HDI-Link™  *Sub-Link™
Microvia: Sub-Lamination Interface

**Basic design rules for microvia build-up layers:**

- Build-up dielectric layers must be balanced on either side of the sub-lamination.

- Build-up dielectric layers are generally 0.0025” to 0.003” thick.

- The recommended total number of lamination cycles that any one part of the structure should experience is 3 and 4 for advanced structures.

- Microvias stacked on buried mechanical vias should be avoided due to wrap plating requirements and excessive stress on thicker substrates.

- Solid copper mechanically drilled vias can be used on thin sub-lamination cores in place of wrap plating.
Microvia Sub-Lamination Interface: Stacked On Sub-Via

- This approach requires via fill and wrap plating
- CTE issues can be a problem on thick boards

Not Recommended on all designs: Contact engineering
Microvia Sub-Lamination Interface: Offset Via

0.5 mm Pitch or >

Preferred Construction
Microvia Sub-Lamination Interface: Offset Via Stacked

0.4 mm Pitch or <

Preferred Construction
Microvia Sub-Lamination Interface: Sub-Lam Microvia

Preferred Construction with additional low cost Microvia to layer 4
Exotic Process Operations

- Buried resistor technology (2 imaging & etching operations)
  - Process Cost index 2 per core
- Buried capacitance
  - 2 mil ZBC 2000 (Thin core processing & registration issues)
    - Process index 1 per core
  - Dupont HK-4, Farad Flex (Thin core processing & registration issues)
    - Process index 2 per core
  - 3M Cply (Thin core processing, sequential lamination & two imaging operations are required when both sides of the dielectric are unsupported by copper in the design)
    - Process index 3 per core
- Cavity constructions i.e. recessed regions of the PCB. This process requires many additional tooling and process operations
- Buried resistor layers, this requires rather expensive material and special etching chemistry
Rules of Thumb for Cost

- Investigate how PCBs will fit into a production panel to ensure that maximum material utilization occurs.
- Use a *smaller line width/space before* adding layers.
- Use *smaller holes before* adding layers.
- When using tighter impedance tolerances, ensure that the build doesn’t cancel out these gains (glass styles, line widths).
- Add *laser blind vias* (one or two sides) before buried vias.
- *Laser and buried vias* allow for *better yields* than mechanical blind vias.
- Add *staggered laser vias before* stacked vias.
- Consider *hybrid* builds for a performance/cost benefit.

**Always work with your fabricator on cost mitigation strategies.**
Summary and Conclusion

- **Cost is driven by Engineering and Design!**

- For optimum results, PCB cost control really needs to be addressed in the **early phases** of product development.

- **Panel Utilization** is critical to producing PCB’s at the lowest cost.

- Keep it **simple**! Complex PCB designs can add many costly manufacturing operations as well as longer lead times.

- Manufacturing **yield** should be considered in the design process and **concurrent engineering** with vendors can improve yields.

- IPC Class 3, 3A, and IST **requirements** can have a large impact on PCB cost due to large numbers of coupons, documentation, lab time, as well as, panel utilization.
Objectives:

• Introduce DFX for Printed Circuit Board Design
  – Design for Fabrication
  – Design for Assembly
  – Design for Test
  – Design for Quality
• Initial considerations for best cost with highest reliability
• Pre-planning
  – Reduce prototype spins by planning ahead
  – Reduce prototype to mass production modification delays and TQs
  – Keep Purchasing Department happy
What Your Company Can’t Afford

Development

Wall of Confusion

Manufacturing
Many PCB-related Design Inputs are Outside the Board

- Circuit Board Attributes, Layer count, Thickness, Dimensions, Holes, Lines/spaces, Blind/buried Vias
- Component pad size/pitch/density drive design complexity and technology
  - Smallest feature sets price of whole PCB
  - Design the rest of the features as large as possible
- Via in pad solder mask or fill protection required during soldering
  - Cu-cap plated filled vias are 30% more expensive
- PCB Technical difficulty – which fab site fits?
- Fabricator Design Guidelines for Site / technology
- Fab Panel Utilization
- Assembly Panel (Arrays) Utilization
- Assembly Requirements
  - Rails for conveyance
  - Stability of array
  - Over-Size limited by SMTA platforms, such as solder paste printers
- Material
- Stack-up
- DFX (Design for Fabrication, Assembly, & Test); Follow the KISS principle
- Migration path prototype (local) to mass production (Asia)
Design Team:

• Electrical & Mechanical Design Engineers
• Test Engineers
• Quality Engineers
• Supply Chain Management & Supply Chain
• Contract Manufacturers or Company Manufacturing
Assembly Requires Design Teamwork

How will this go down the line?

How many up on an array for SMT?
  Rails?
  Panel utilization killer?

What do we do about the overhanging parts during assembly?

Can a vision system read a broken edge?

Shock to the overhanging part solder joints during operation?
Dimensions: Fab Panel Utilization Biggest Cost Factor

- The cost of a production panel is stable, with additional units adding slightly additional cost – get as many up on a production panel as possible.
- Plan the PCB size to include rails for over-hanging components and/or clearance for assembly.
- Utilize TTM standard panel sizes (inch):
  - 18x24  20x24  21x24  16x20  16x18  16x21  16x24
5” X 7” Panelization Example
Use an 18” x 24” or 21” x 24” production panel? (consider working panel tooling)
Reduce space between parts by 0.10” and add 3 more units – a 14% unit increase/panel
Panelization Example yields 6 PCBs on 18”x24”

A 2 x 1 array, approx. 11” x 8” would fit 8 per production panel by reducing to approx. 5.2” side
Standard TTM Automotive Materials

- Isola 370HR is TTM standard for High Tg FR-4
- Isola 370HRS is CAF-improved, TTM-required for High Voltage applications
- Immediate Cost Reduction Options
  - Use EM-827 for:
    - 0.5 oz, 1 oz. Cu designs
    - Where no High Voltage is required
      - Already used on 1033983-00-B ADAS, NCAP, Camera Control Module
- Additional Materials Under Consideration
  - Isola 180 with Baotek glass for High V
  - ShengYi Autolad for stacked microvia designs (lower CTE)
Stack-ups for China Should Be Confirmed Before Routing

• Select materials most used by the production site – not reference design stack-ups
  – Higher volume pricing
  – Better availability
  – Site more familiar with material characteristics

• Glass weaves are different
  – Engineers don’t like approving different stack-ups
  – Controlled impedance adjustments

• Lamination equipment presses differently

• Up-front approval reduces production approval

• Potential cost savings with efficient stacks
# Utilize Asia DFM Guidelines for Mass Production

**TTM-Via Asia DFM Checklist**  
**Last Update:** 30-April 2015  
**Version 1.1**

## Characteristic: DFM Standard (minimums)

<table>
<thead>
<tr>
<th>Inner Layer Etch Capabilities</th>
<th>Guangzhou Minimum</th>
<th>Huiyang Minimum</th>
<th>Zhongshan Minimum</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/2 oz. trace/space</td>
<td>4 mil/4 mil</td>
<td>3 mil/3.5 mil</td>
<td>4 mil/4 mil</td>
</tr>
<tr>
<td>1 oz. trace/space</td>
<td>4 mil/4 mil</td>
<td>3 mil/3.5 mil</td>
<td>4 mil/4 mil</td>
</tr>
<tr>
<td>2 oz. trace/space</td>
<td>7 mil/7 mil</td>
<td>6 mil / 6 mil</td>
<td>5 mil/6 mil</td>
</tr>
<tr>
<td>3 oz. trace/space</td>
<td>9 mil/9 mil</td>
<td>8 mil / 8 mil</td>
<td>7 mil/7.5 mil</td>
</tr>
<tr>
<td>4 oz. trace/space</td>
<td>9 mil / 9 mil</td>
<td>9 mil/11 mil</td>
<td>8 mil/8 mil</td>
</tr>
<tr>
<td>5 oz. trace/space</td>
<td>11 mil / 11 mil</td>
<td>11.5 mil/11.4 mil</td>
<td>9 mil/12 mil</td>
</tr>
<tr>
<td>6 oz. trace/space</td>
<td>13 mil / 13 mil</td>
<td>14 mil/15.6 mil</td>
<td>no</td>
</tr>
<tr>
<td>12 oz. trace/space</td>
<td>28 mil / 28 mil</td>
<td>26 mil / 26 mil</td>
<td></td>
</tr>
</tbody>
</table>

## Outer Layer Etch Capabilities

<table>
<thead>
<tr>
<th>Drilled Via Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outer Layer Thickness after Plating</td>
</tr>
<tr>
<td>1.3 - 1.5 mil</td>
</tr>
<tr>
<td>1.8 - 2.0 mil</td>
</tr>
<tr>
<td>2.5 mil / 5 mil GZ</td>
</tr>
<tr>
<td>3.0 - 3.2 mil</td>
</tr>
<tr>
<td>3.8 - 4.4 mil</td>
</tr>
<tr>
<td>5.4 - 5.8 mil</td>
</tr>
<tr>
<td>Drill/Pad Diameter [ for Tangency ]</td>
</tr>
<tr>
<td>(Class 3: IL + .002 / OL + .004)</td>
</tr>
<tr>
<td>10 mil / 20 mil</td>
</tr>
<tr>
<td>12 mil/22 mil</td>
</tr>
</tbody>
</table>

## Mechanical Hole Edge to Copper

<table>
<thead>
<tr>
<th>Mechanical Hole Edge to Copper</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hole Edge to Conductor</td>
</tr>
<tr>
<td>8.5 mil</td>
</tr>
<tr>
<td>Hole Edge to Cu on Anti Pad</td>
</tr>
<tr>
<td>10. mil</td>
</tr>
</tbody>
</table>

## Microvia Edge to Adjacent Copper

<table>
<thead>
<tr>
<th>Conductor</th>
<th>Cu to board edge</th>
</tr>
</thead>
<tbody>
<tr>
<td>9 mil</td>
<td>8 mil</td>
</tr>
<tr>
<td>15 mil</td>
<td>10 mil</td>
</tr>
</tbody>
</table>

## Microvia

<table>
<thead>
<tr>
<th>Microvia</th>
</tr>
</thead>
<tbody>
<tr>
<td>Microvia Diameter</td>
</tr>
<tr>
<td>Pad Diameter</td>
</tr>
<tr>
<td>Aspect Ratio</td>
</tr>
</tbody>
</table>

## SMD to SMD Spacing

<table>
<thead>
<tr>
<th>Solder Mask</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum dam width</td>
</tr>
<tr>
<td>Minimum s/m-defined pad</td>
</tr>
</tbody>
</table>

## Non-Conductive Via Fill / Cap Plate

<table>
<thead>
<tr>
<th>Non-Conductive Via Fill / Cap Plate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aspect Ratio</td>
</tr>
<tr>
<td>Flatwrap</td>
</tr>
<tr>
<td>Copper Fill</td>
</tr>
<tr>
<td>LDI Mask</td>
</tr>
</tbody>
</table>

## Plated Hole Nomen

<table>
<thead>
<tr>
<th>Plated Hole Nomen</th>
</tr>
</thead>
<tbody>
<tr>
<td>DHW to DHW [ CAF ]</td>
</tr>
</tbody>
</table>

## Dimensional Tolerances

<table>
<thead>
<tr>
<th>Dimensional Tolerances</th>
</tr>
</thead>
<tbody>
<tr>
<td>Score off set</td>
</tr>
<tr>
<td>True position</td>
</tr>
<tr>
<td>Routed Profile &amp; Cut out</td>
</tr>
<tr>
<td>Largert non plated drill diameter</td>
</tr>
<tr>
<td>Edge connector Bevel Depth</td>
</tr>
<tr>
<td>Cu Distribution: Thieving</td>
</tr>
<tr>
<td>Common Voidsed areas IL &amp; OL</td>
</tr>
</tbody>
</table>

---

# Table of Contents

- [Characteristic: DFM Standard (minimums)](#characteristic-dfm-standard-minimums)
- [Inner Layer Etch Capabilities](#inner-layer-etch-capabilities)
- [Outer Layer Etch Capabilities](#outer-layer-etch-capabilities)
- [Drilled Via Size](#drilled-via-size)
- [Aspect Ratio](#aspect-ratio)
- [Mechanical Hole Edge to Copper](#mechanical-hole-edge-to-copper)
- [Microvia Edge to Adjacent Copper](#microvia-edge-to-adjacent-copper)
- [Conductor | Cu | to board edge](#conductor-cu-to-board-edge)
- [Microvia](#microvia)
- [SMD to SMD Spacing](#smd-to-smd-spacing)
- [Non-Conductive Via Fill / Cap Plate](#non-conductive-via-fill-cap-plate)
- [Plated Hole Nomen](#plated-hole-nomen)
- [Dimensional Tolerances](#dimensional-tolerances)
- [Cu Distribution: Thieving](#cu-distribution-thieving)

---

**Operator**

<table>
<thead>
<tr>
<th>Name</th>
<th>Date</th>
</tr>
</thead>
</table>

**CAM**

<table>
<thead>
<tr>
<th>Name</th>
<th>Date</th>
</tr>
</thead>
</table>

---

**CAM technologies**

---

**TTM-Via Asia DFM Checklist**

**Last Update:** 30-April 2015

**Version 1.1**

---

**TTM-Via Asia DFM Checklist**

**Last Update:** 30-April 2015

**Version 1.1**
Pre-Design Communication Reduces Technical Queries!
Procedures for Prototype Transfers to China

• Global Supply Chain to cc TTM Engineering on RFQs:
  – Quick-turn N. America PCB Prototypes
  – Procurement by a CM: state if panelization is required for fast QTA
  – Review site-proposed stacks for mass production
  – Review proto arrays for mass production – change if needed when going to Asia

• Asia production orders TBD, but plan accordingly
Conclusion: Working Together Initially Saves $$$$
Thank you!

Julie Ellis, FAE
julie.ellis@ttm.com
714-473-1867