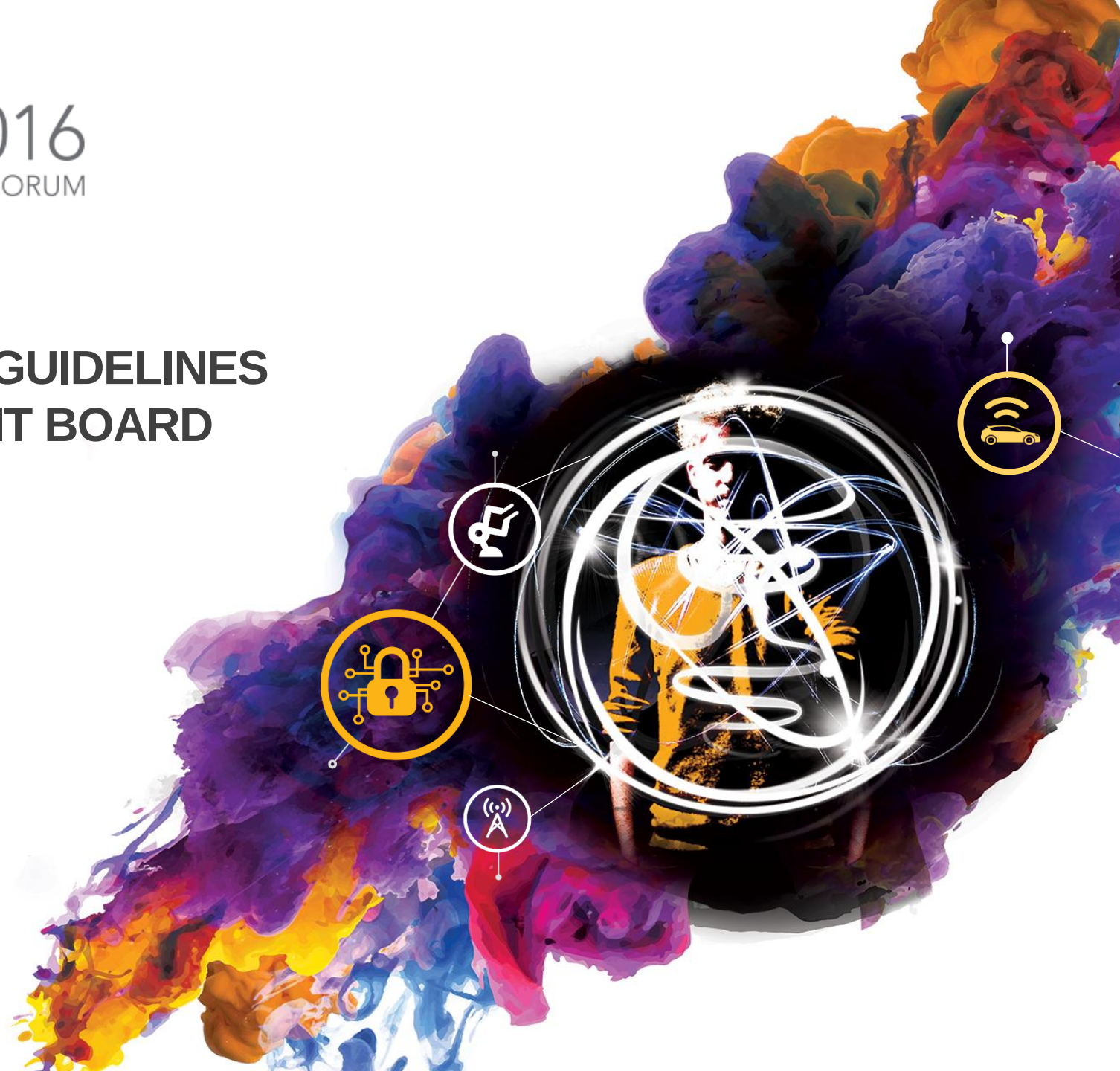
FTF 2016
TECHNOLOGY FORUM

COST DRIVERS AND DESIGN GUIDELINES TO IMPROVE PRINTED CIRCUIT BOARD MANUFACTURABILITY

BOARD SOLUTIONS GROUP

ANDRES MARQUEZ
DFX ENGINEER
FTF-DES-N1992
May 17, 2016



AGENDA

- Introduction
- Importance of Early Engagement
- Design for Manufacturing (DFM)
- PCB Fabrication Cost Adders
- PCB Assembly Recommendations
- Stay Up to Date
- Q & A

INTRODUCTION

Introduction

What this presentation IS about:

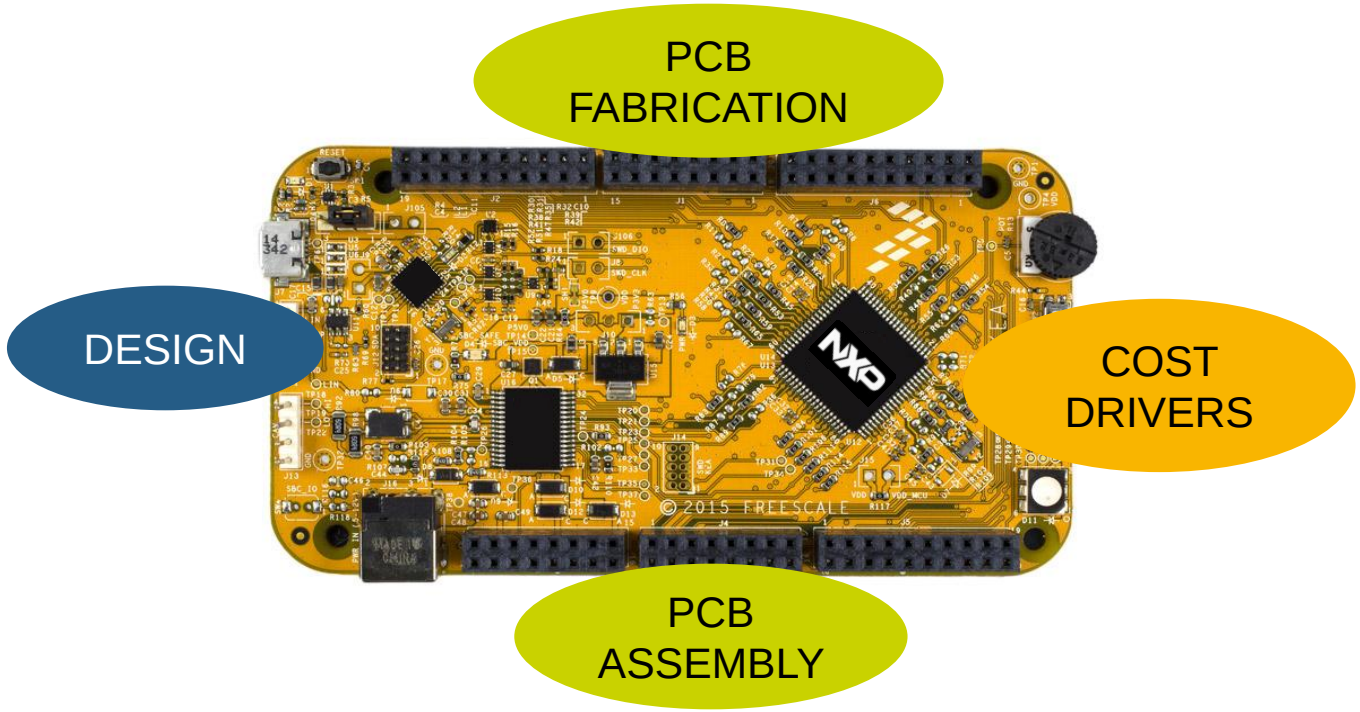
- It aims to talk about printed circuit board's design aspects that drive manufacturing and assembly costs
- It shows guidelines and recommendations for PCB fabrication and assembly

What this presentation IS NOT about:

- It does not show a process to set the cost of PCBs
- It is not project specific

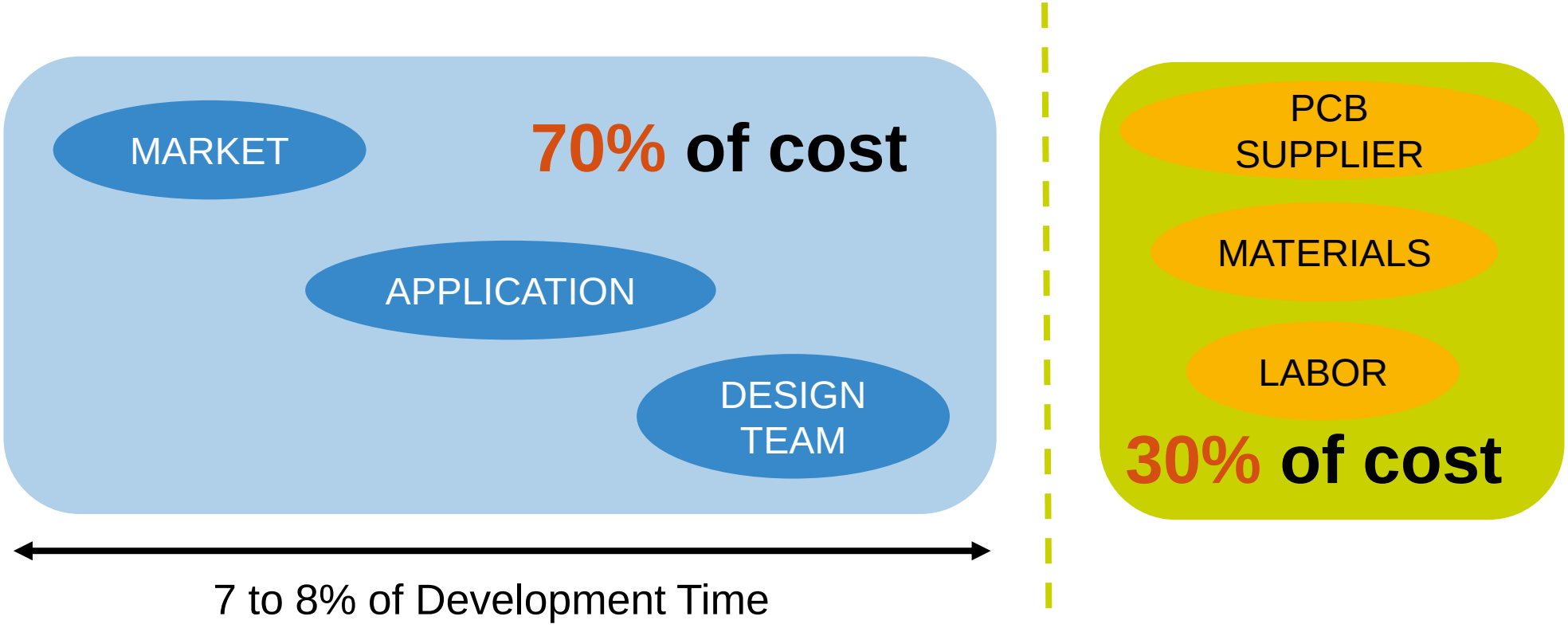
Audience:

- Everyone interested in PCBs



IMPORTANCE OF EARLY ENGAGEMENT

Importance of Early Engagement



PCB Design should have early input from:

a) *Manufacturing Guidelines*

b) *Parties building the board*

Importance of Early Engagement

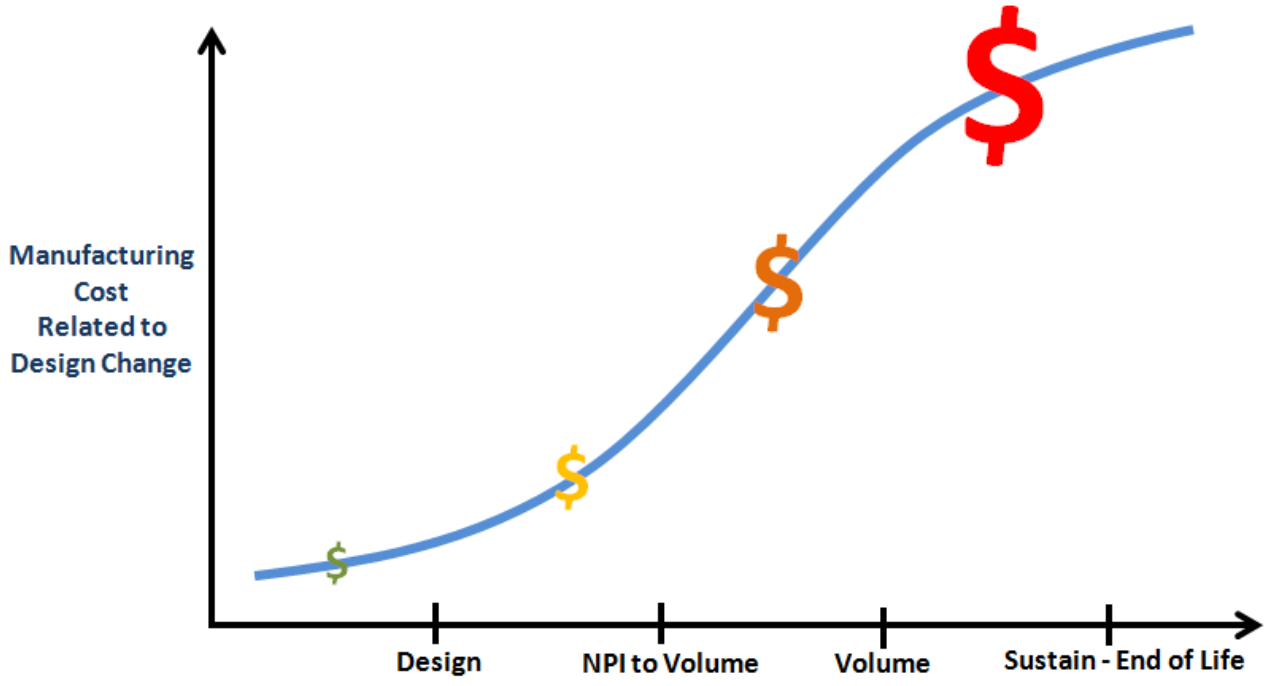


Image reference: Lean NPI With Valor. Mentor Graphics

- Avoid manufacturing cost impact by following manufacturing guidelines
- Early supplier involvement to define manufacturing capabilities and be up to date with new technologies

Early involvement in product life cycle to shift money to the left: *"The earlier the changes in the design, the better."*

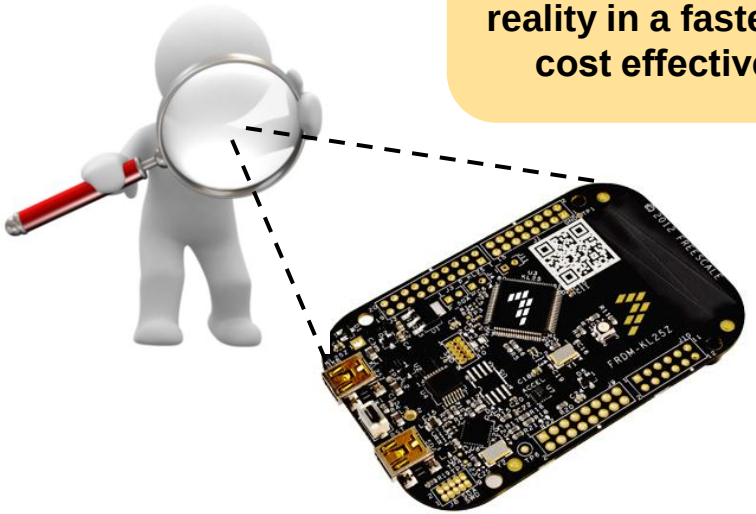
DESIGN FOR MANUFACTURING (DFM)

Design For Manufacturing (DFM)

Awareness of how the way we design is going to affect manufacturing

Several Definitions, all convey in **designing a product in the most efficient manner possible in terms of cost, resources and time**.

“Design Guidelines provide best practices so that boards can be brought to reality in a faster, easier and cost effective manner”



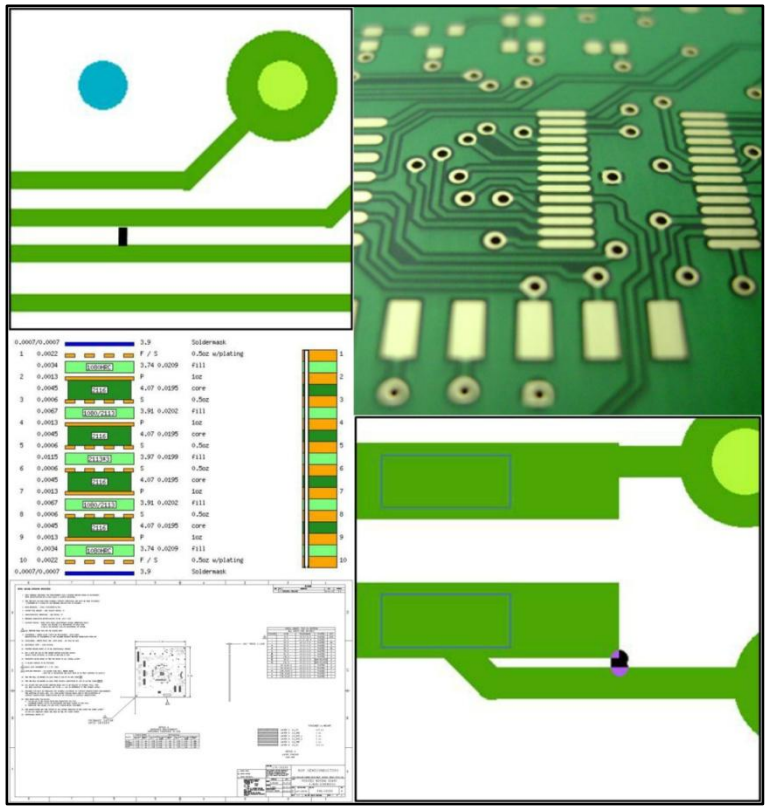
Designer

GAP

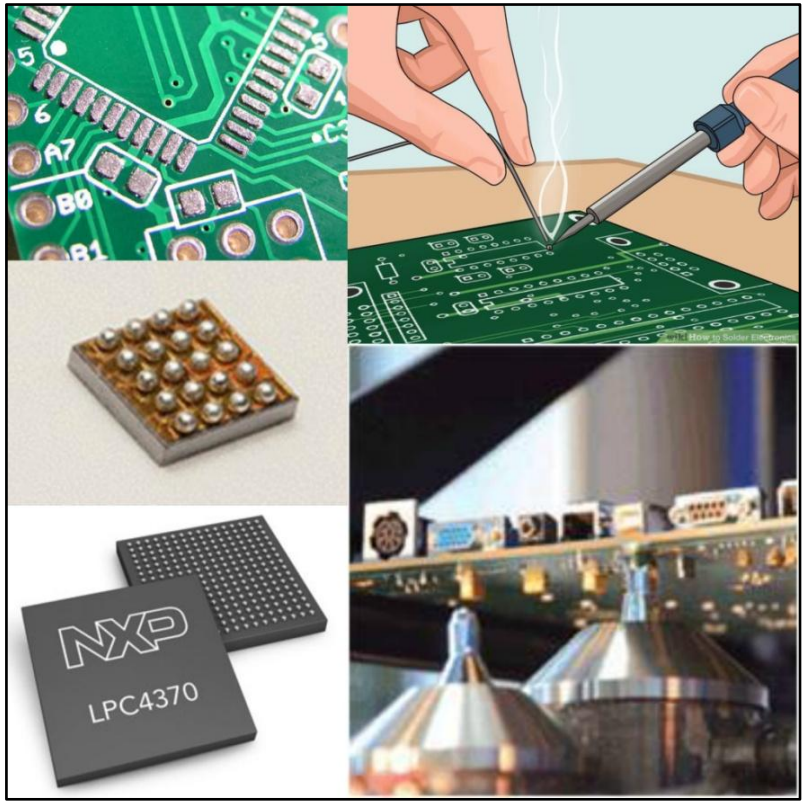
Manufacturer

PCB Fabrication and PCB Assembly

PCB Fabrication



PCB Assembly



PCB FABRICATION COST ADDERS

Printed Circuit Board Fabrication Cost Adders

- Volume and Lead Time
- PCB Fab Shop’s Capabilities
- Panel Utilization
- Stackup – Number of Layers
- Stackup Configuration
- Technology Complexity (HDI)
- Copper Weight vs. Minimum Spacing/Width
- Copper Considerations
- Drilling and Aspect Ratio
- Drill to Copper Distance
- Solder Mask Considerations
- PCB Finish
- Fabrication Drawing

Volume and Lead Time

- Prototypes vs. Production
- Many PCB manufacturers have a minimum cost for an order >> MOQ
- Time is Money. A Quick Turn will cause the PCB Fabshop to change priorities and increase the cost.

VOLUME AS A COST ADDER

Quantity	150	500	5000
Cost	$7.36	$2.43	$0.62

LEAD TIME AS A COST ADDER - QTY 1000

Lead Time	10 Days	15 Days	20 Days
Cost ea.	$3.40	$3.00	$2.74

PCB Fab Shop’s Capabilities

- Understand the PCB Fab Shop’s capabilities

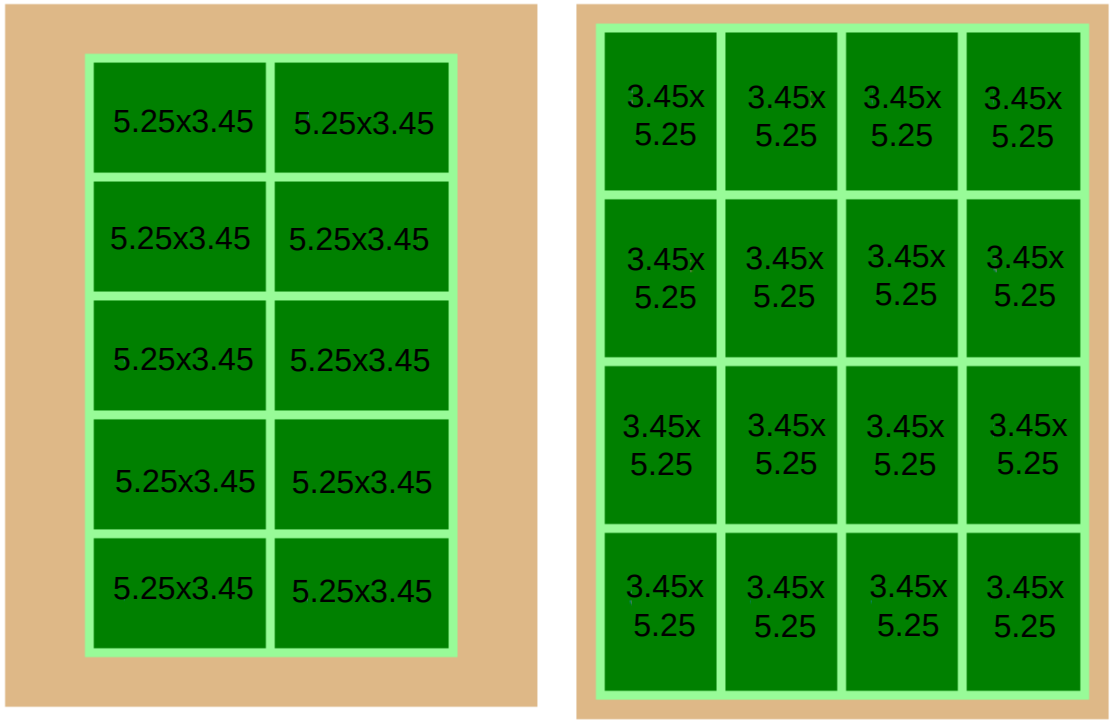
CONVENTIONAL ADVANCED LEADING EDGE

Advanced categories build up. Each parameter contributes independently for the yield of the PCB.

Category	Conventional	Advanced	Leading Edge
Line/Spacing	0.004"/0.004"	0.003"/0.003"	0.0025"/0.0025"
Mechanical Drill	0.010"	0.008"	0.006"
Aspect Ratio	8:1 ~ 10:1	10:1 ~ 15:1	15:1 ~18:1
Solder Mask Registration	0.003"	0.002"	0.001"
DHS to Copper	0.008"	0.007"	0.006"
Copper to Edge	0.02"	0.015"	0.01"

Panel Utilization

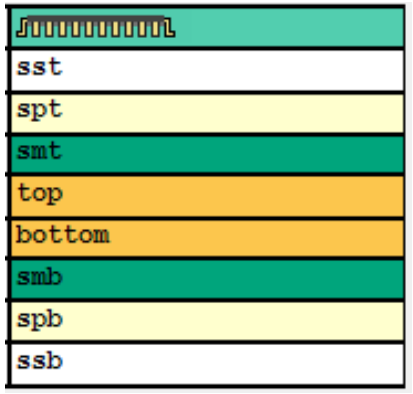
- Know the types of panels your Fab shop uses:
  - The most common measurements are 18”x24” and 21”x24”
- How many parts will fit? Goal: as much parts as possible in a Panel
  - Rectangular boards can usually fit better in a panel than circular boards
  - Consider part rotation
- PCB fab shops have to charge for material not used

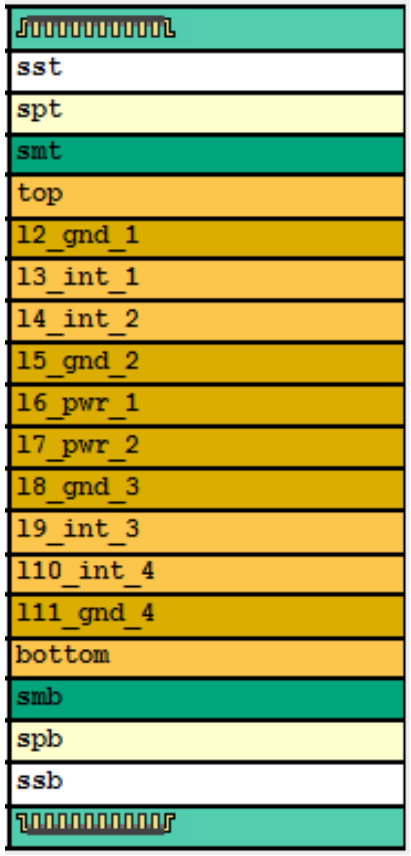


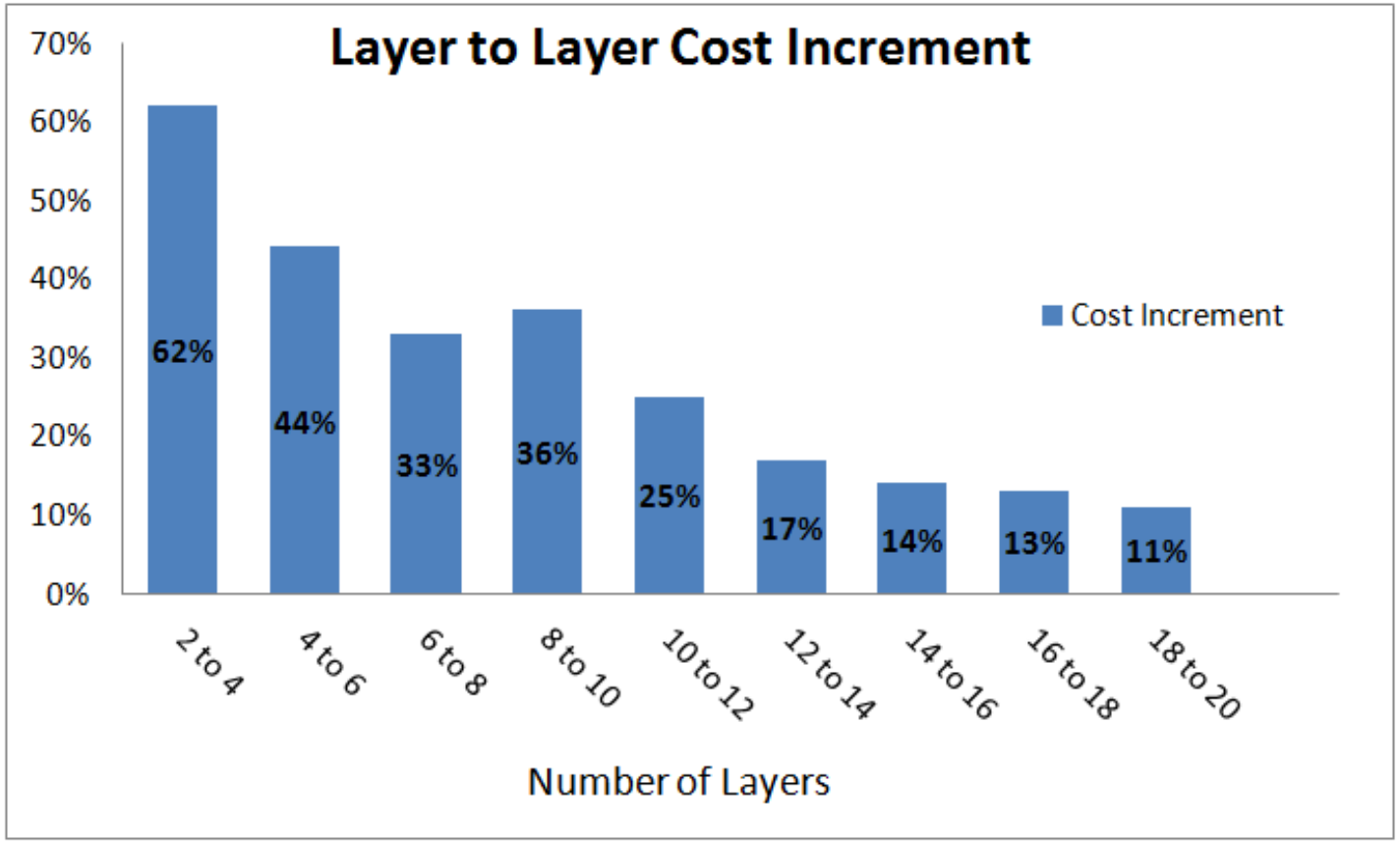
Same panel 18”x24” different arrangement.
41.9% (10 boards) vs. 67.1% (16 boards) utilization

Stackup – Number of Layers

- Number of Layers affects cost by increments







Stackup Configuration

- Material Selection $$$ – Application Driven, mostly signal integrity
- Balanced Stackup:
  - Symmetrical stack up to avoid warpage
  - Thieving pads

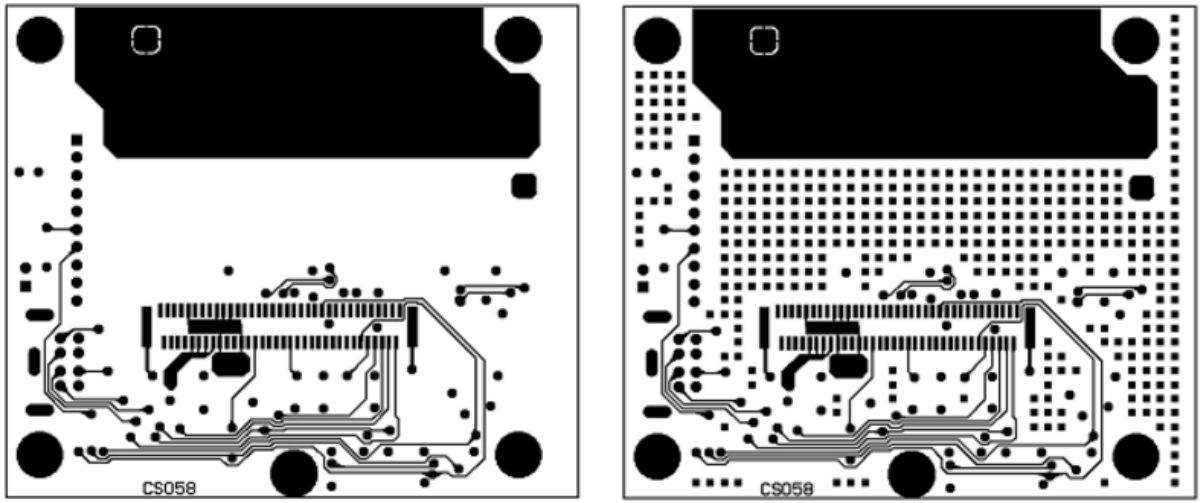
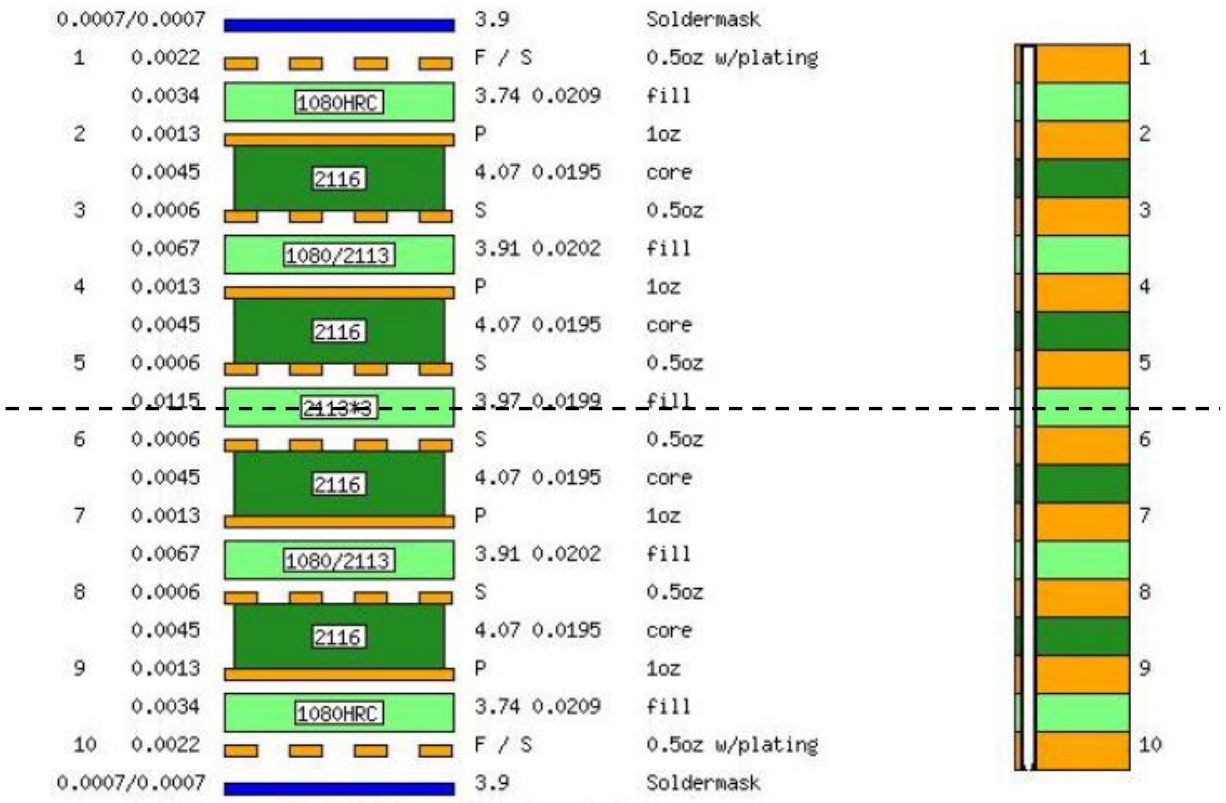


Image by electronics.stackexchange.com



Technology Complexity (HDI)

- Microvias usually < 0.006” and laser drilled

Type III HDI Structure

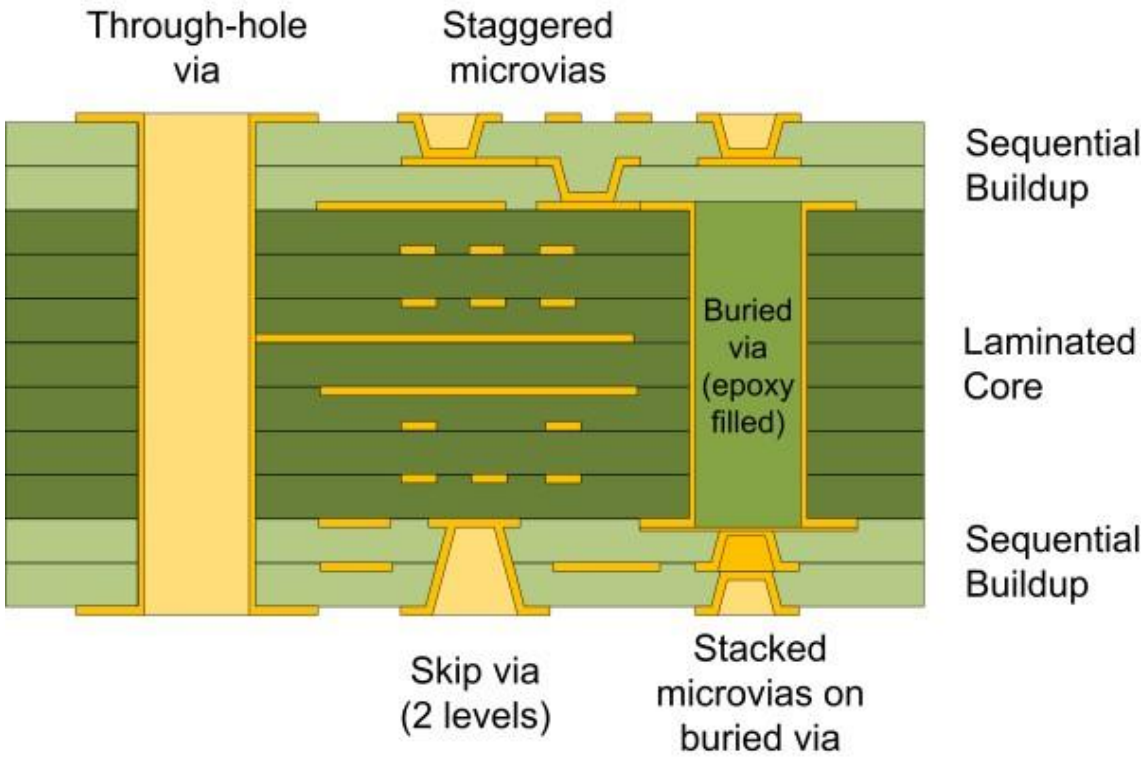


Image by www.datarespons.com - High Density Interconnect

Cost based on Extra Processes

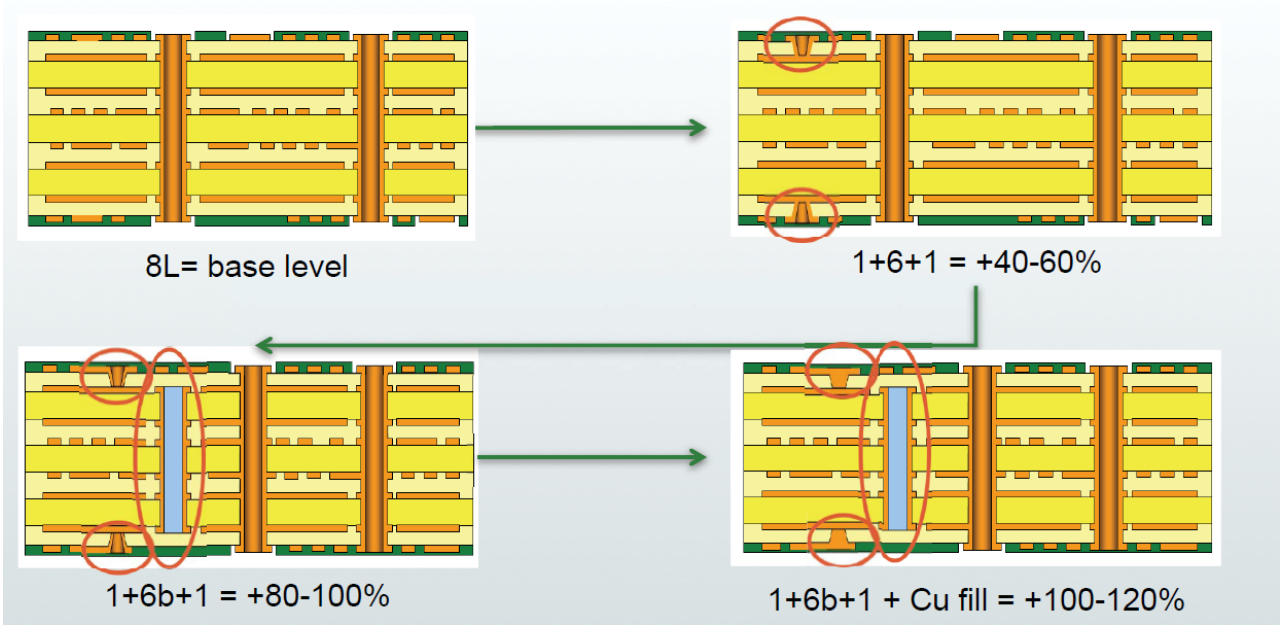


Image by NCAB – Cost Drivers in PCB Production

Copper Weight vs. Minimum Spacing/Width

- The thicker the copper the more difficult to etch
- Etch compensation defines minimum spacing/width

Internal Layers	Cu Weight	Conventional
Min Line	0.5	0.004
Min Separation	0.5	0.004
Min Line	1	0.005
Min Separation	1	0.005
Min Line	2	0.007
Min Separation	2	0.007
External Layers	**Cu Weight**	**Conventional**
Min Line	0.5	0.004
Min Separation	0.5	0.004
Min Line	1	0.004
Min Separation	1	0.004
Min Line	2	0.006
Min Separation	2	0.006

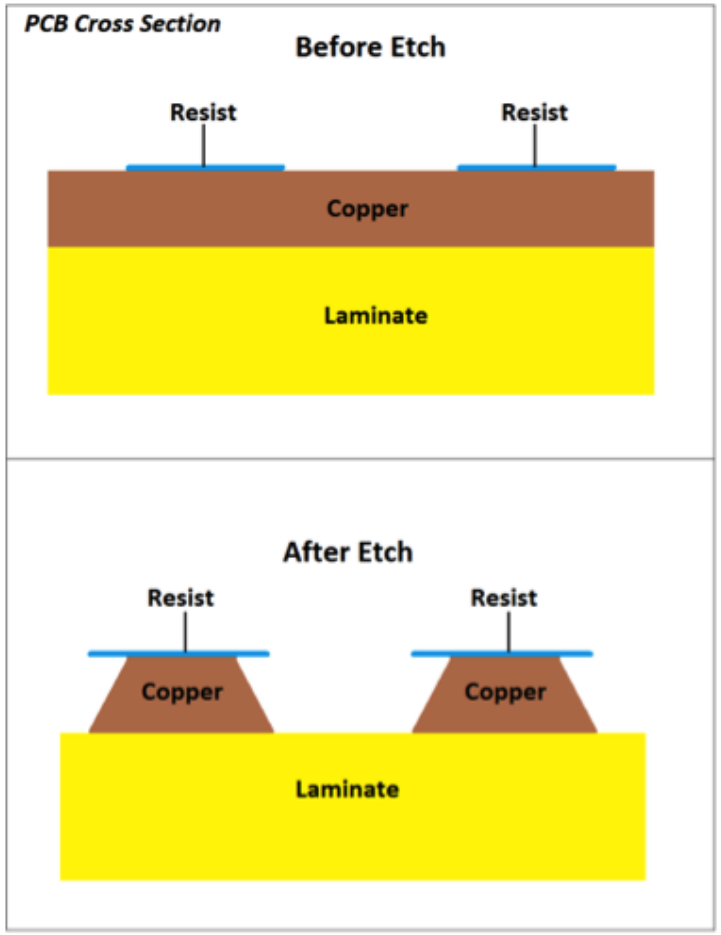
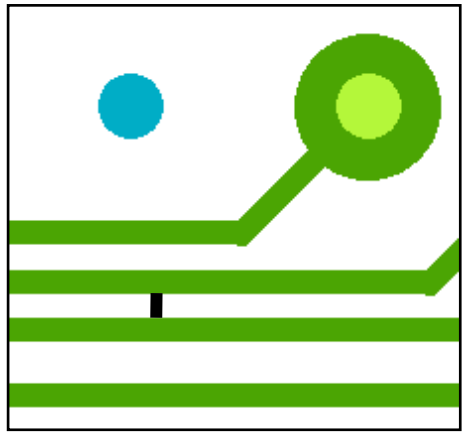


Image by www.pcbuniverse.com

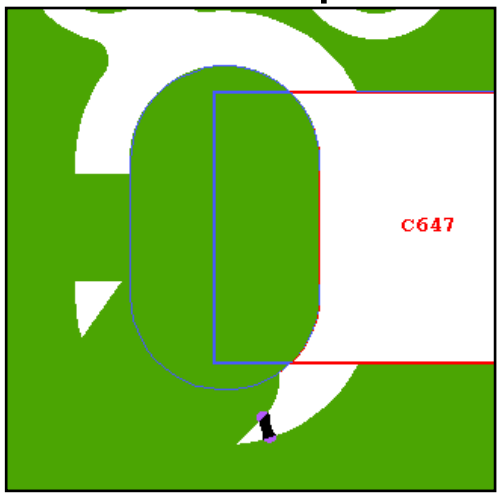
Copper Considerations

Minimum Spacing/Width



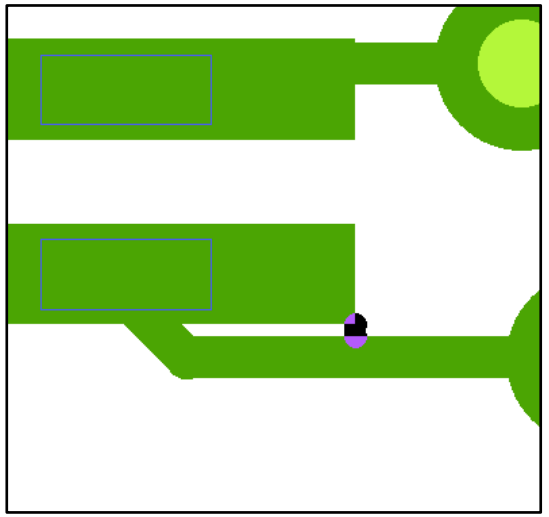
Spacing/Width of 0.003”/0.003”

Acid Traps

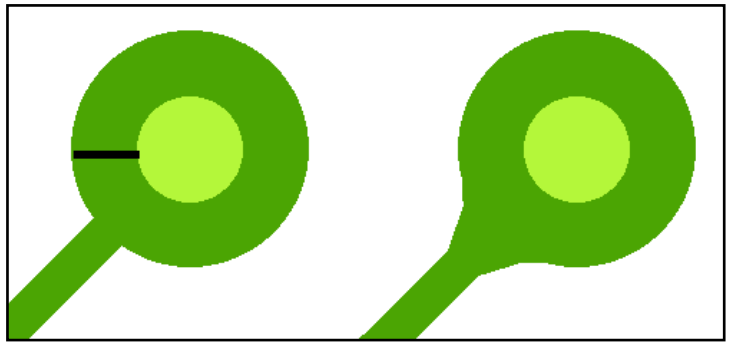


Category	Conventional	Advanced	Leading Edge
Line/Spacing	0.004"/0.004"	0.003"/0.003"	0.0025"/0.0025"
Copper to Edge	0.02"	0.015"	0.01"

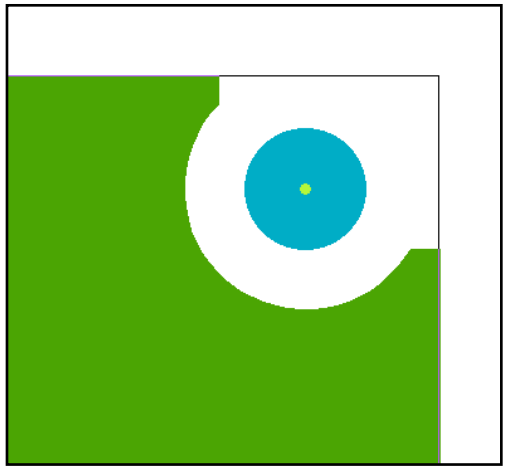
Same Net Spacing



Annular Ring/Teardrops



Copper to Edge



Drilling and Aspect Ratio

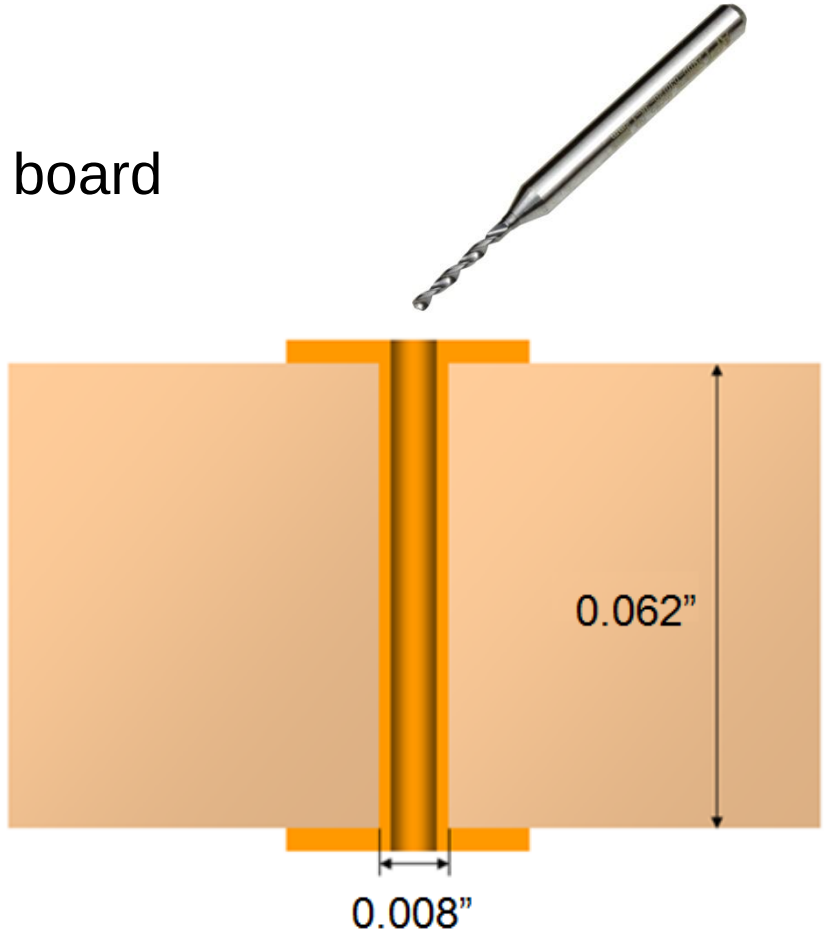
- Affected by quantity, minimum drill and thickness of the board
- Drilling small holes has an impact in cost: < 0.008”

- The Aspect Ratio is defined as the board thickness divided by the drill size
- Higher aspect ratios are more difficult to produce

$$Maximum\ Aspect\ Ratio = \frac{Board\ Thickness}{Smallest\ Drilled\ Hole}$$

$$Maximum\ Aspect\ Ratio = \frac{0.062"}{0.008"} = 7.75:1$$

Category	Conventional	Advanced	Leading Edge
Aspect Ratio	8:1 ~ 10:1	10:1 ~ 15:1	15:1 ~18:1
Mechanical Drill	0.010"	0.008"	0.006"



For fabrication flexibility
Via Size (+0.000”/-Via Size)

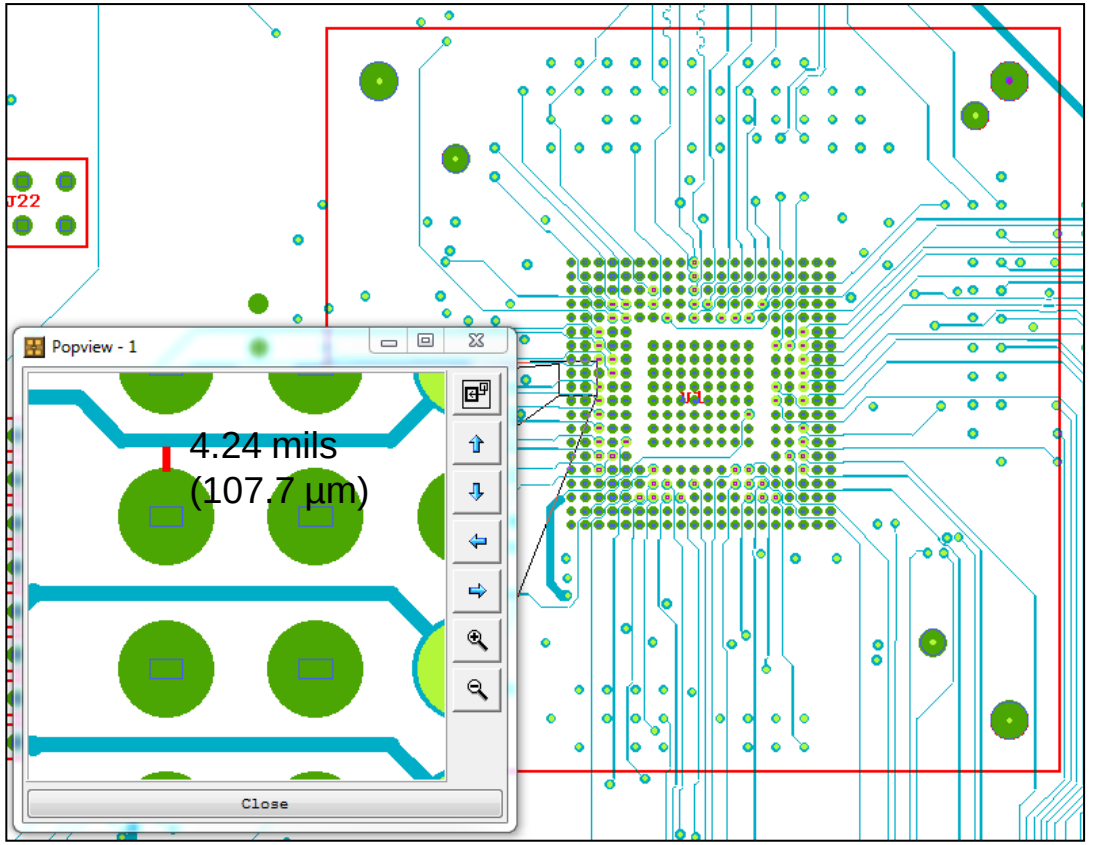
Drill to Copper Distance

- Often overlooked issue
- Imaging tolerance, layer to layer registration and Drill tolerance come into play
- Center FHS in its tolerances + 4 mils (100 µm) plating to obtain approximate Drilled Hole Size

$$DHS = FHS + \frac{ABS(POS\ TOLERANCE + NEG\ TOLERANCE)}{2} + 4$$

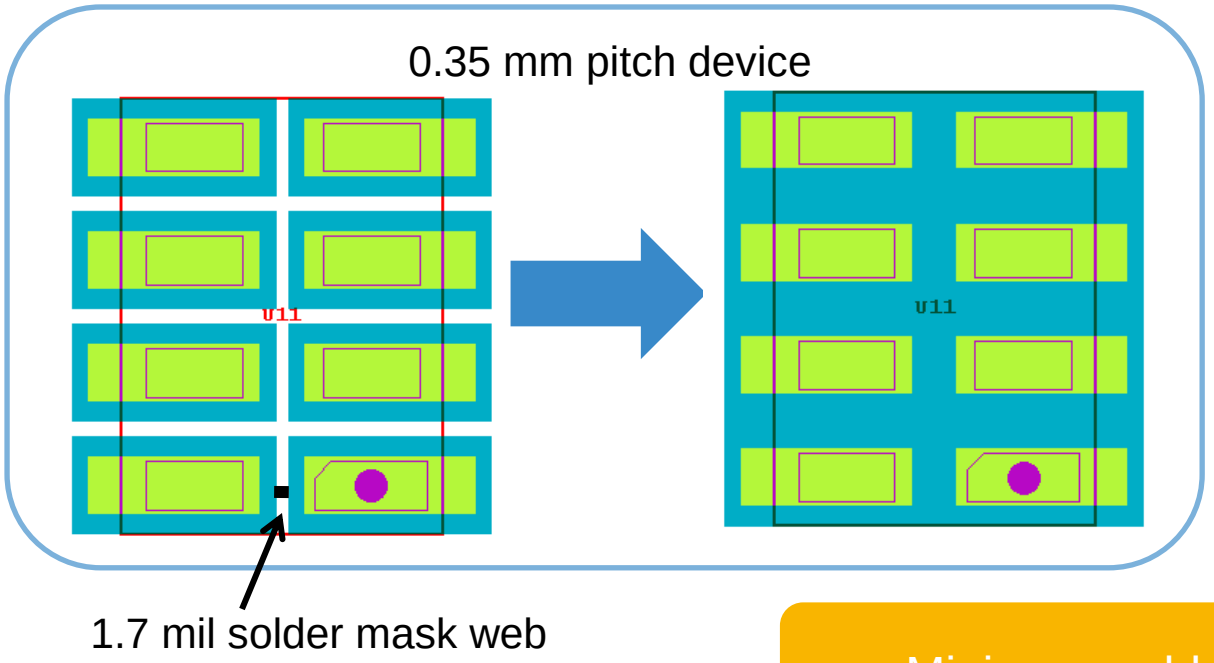
- *POS and NEG Tolerances with signs included. Tolerances specified in fabrication drawing.*

$$DHS = 14 + \frac{ABS(4 + 0)}{2} + 4 = 20$$

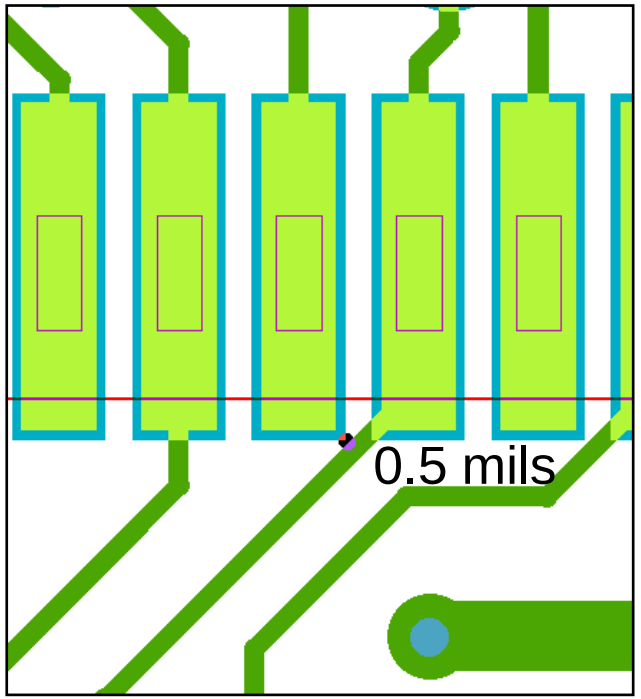


Solder Mask Considerations

- Color may affect the process
- Types of solder mask LDI vs. LPI
- When pitch is small remove solder mask between pads



Coverage Issue



Extra Costs for SM

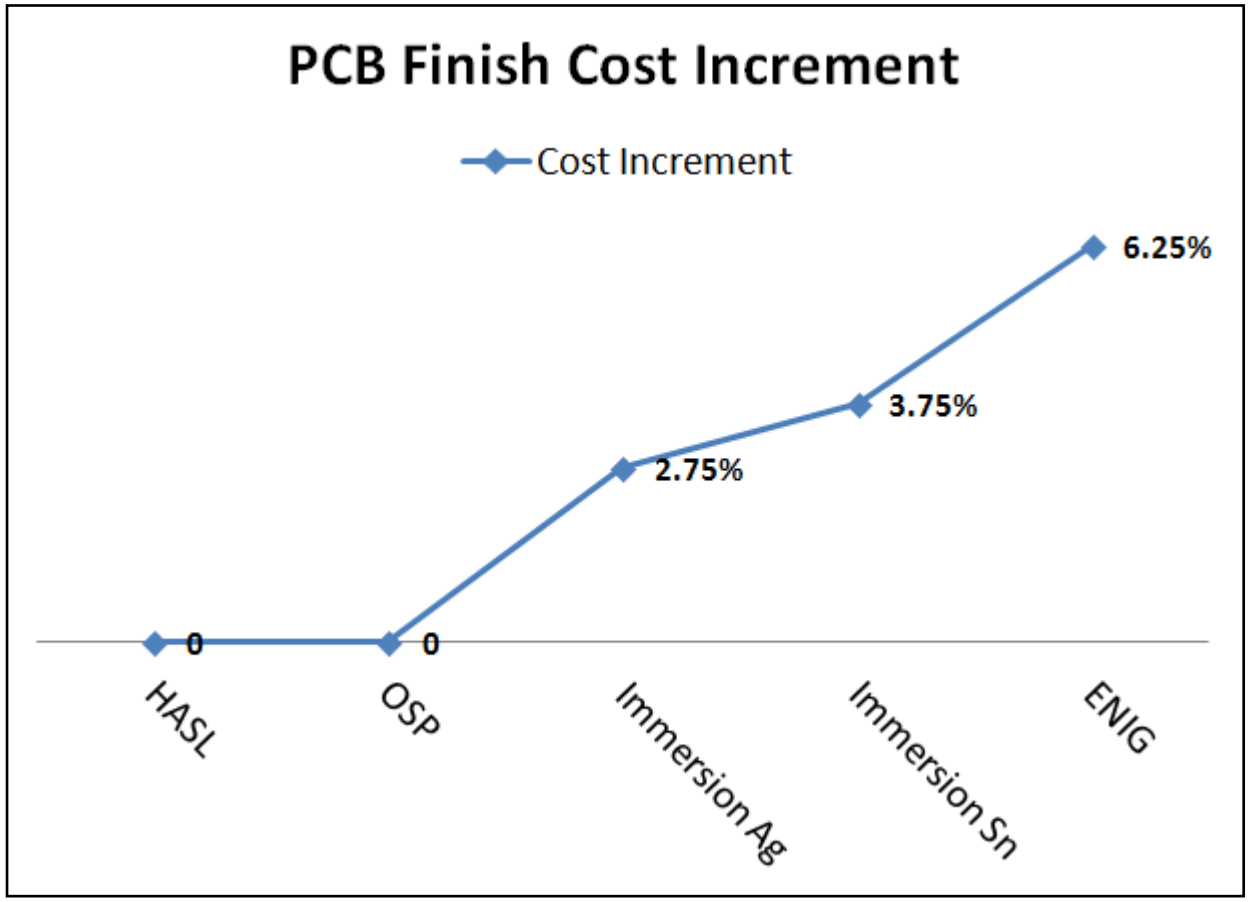
Capping
Plugging

Category	Conventional	Advanced	Leading Edge
Solder Mask Registration	0.003"	0.002"	0.001"

Minimum solder mask webs are usually 3 mils.

PCB Finish

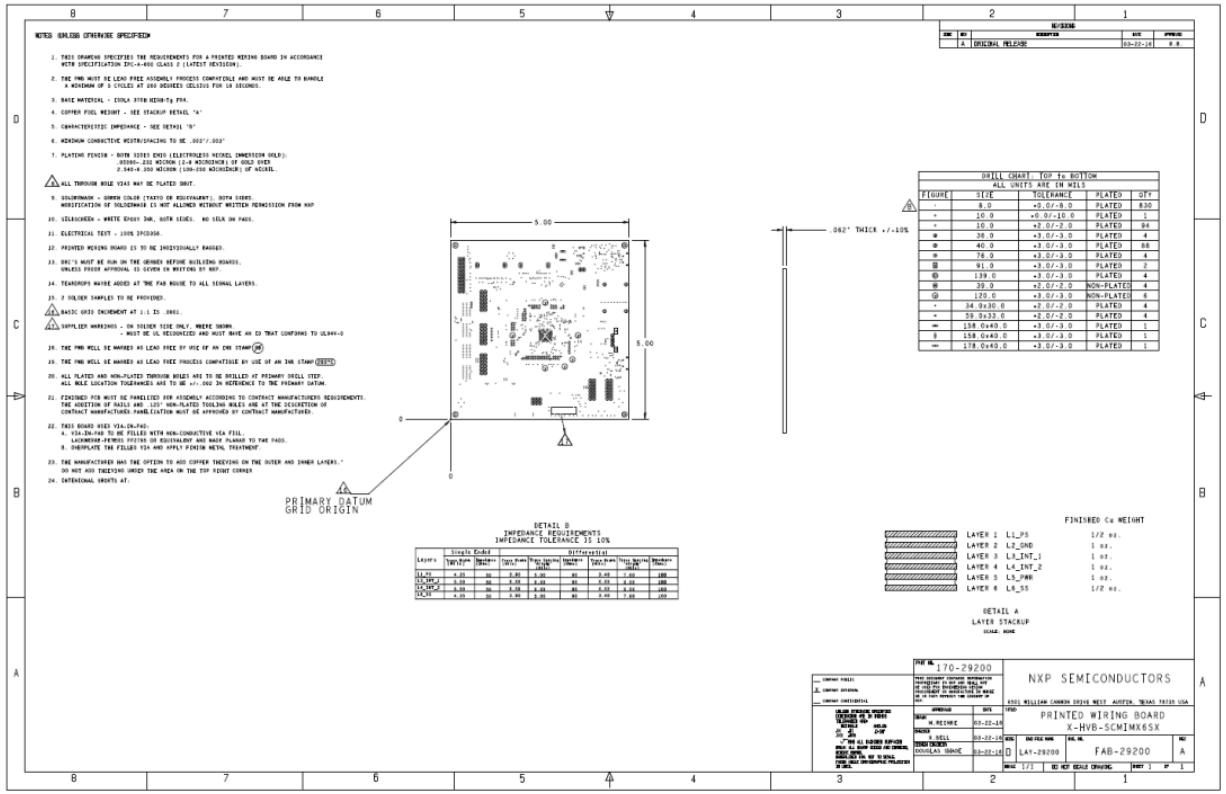
- Hot Air Solder Leveling (SnPb HASL)
- Organic Solderability Preservative (OSP)
- Immersion Silver (Imm Ag)
- Immersion Tin (Imm Sn)
- Lead Free HASL (LF HASL)
- ENIG – Electroless Nickel Immersion Gold
- Electrolytic Hard Gold
- Electroless Nickel Electroless
- Palladium Immersion Gold (ENEPIG)



Fabrication Drawing

Should contain necessary information to avoid delays or confusion

- Some important information to be specified:
  - Dimensions of the outline of the board with cutouts or notches
  - Thickness of the board along with stackup
  - Base material
  - IPC class type
  - Copper thickness
  - Drill map with quantity, tolerances and their respective symbols
  - Surface finish
  - Solder mask color
  - Whether it is allowed for fab shop to add thieving pads for copper balancing, NFP removal and Tear drops for ease of manufacturing
  - List of intentional shorts to avoid confusion
  - Impedance chart
  - Revision and company information



Avoid discrepancies between fabrication drawing and CAD data

PCB ASSEMBLY RECOMMENDATIONS

Printed Circuit Board Assembly Recommendations

- Assembly Cost
- Product Technology Definition
- Package Sizes
- Component Spacing & Footprints
- Stencil Design Rules
- Wave Soldering
- General Recommendations For Assembly

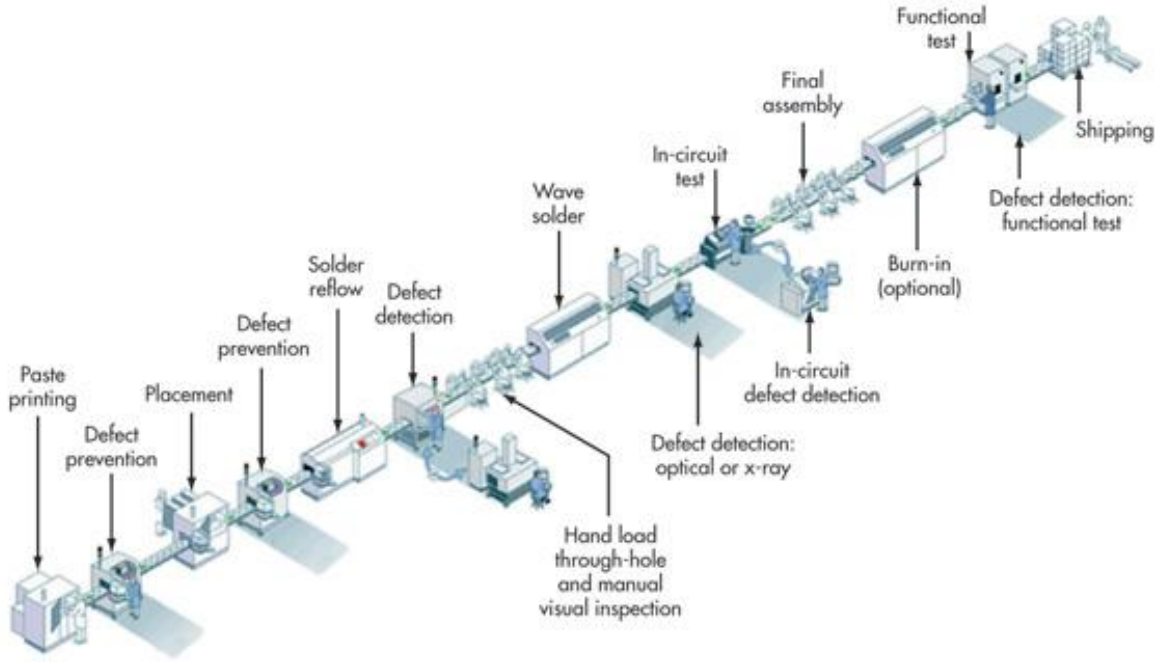
Assembly Cost

The assembly cost from the process point of view will depend upon:

- Number of processes
  - SMT Top and Bottom
  - PTH Top and Bottom
  - Press fit
  - TEST

Process time

- SMT placement time (number of different components, rotation, etc.)
- Board size (nozzle travel)
- Hand placed components, manual soldering, etc.



Product Technology Definition

- Intention of the board
- Application
- Volume
- Prototype vs. Production

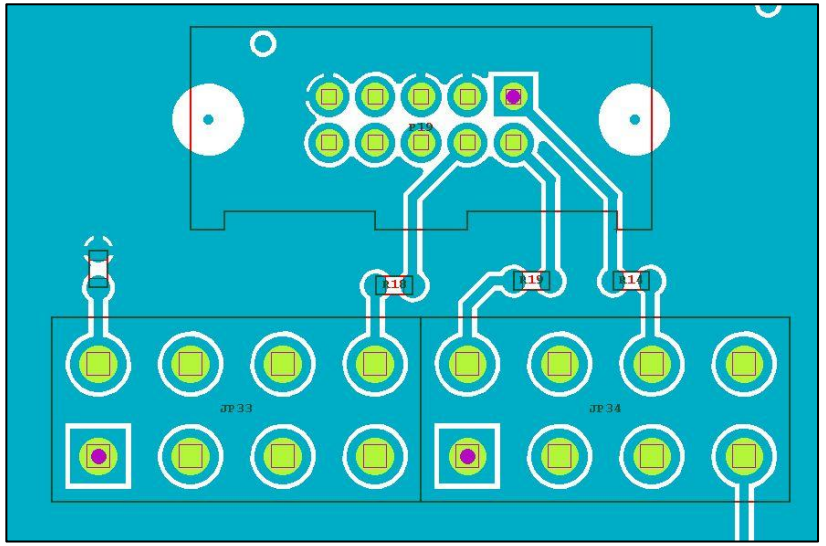
Package Sizes

- Component manufactures have new packages with the same features
- Keep your BOMs up to date, use new parts
- Smaller is not better in all cases, define and standardize the technology you will use. Package drives PCB technology.

Component Spacing and Footprints

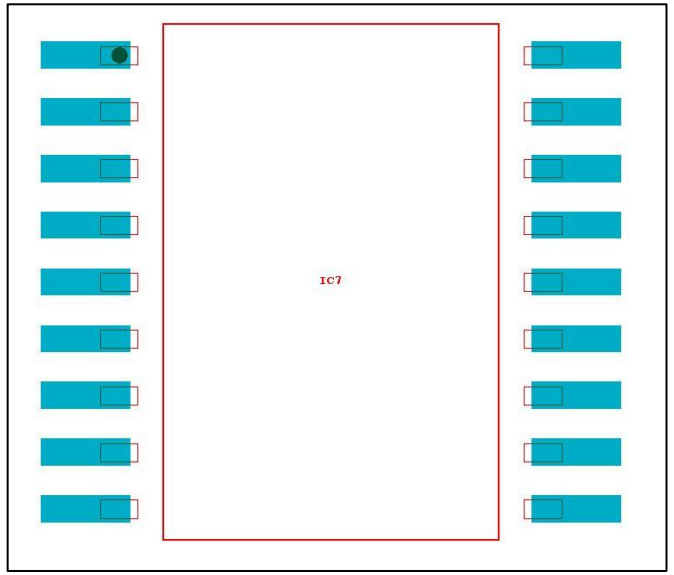
Why is it important to maintain an adequate component to component distance?

- For better machine programming (Placement tolerance)
- To compensate for component and PCB dimensional tolerances
- To facilitate component verification (Visual inspection)
- To facilitate rework process
- To facilitate cleaning process



Footprint design to achieve reliable solder joints

- Follow IPC7351 as a reference



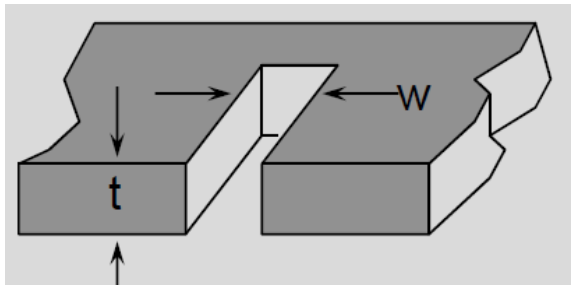
Stencil Design Rules (Paste Layer)

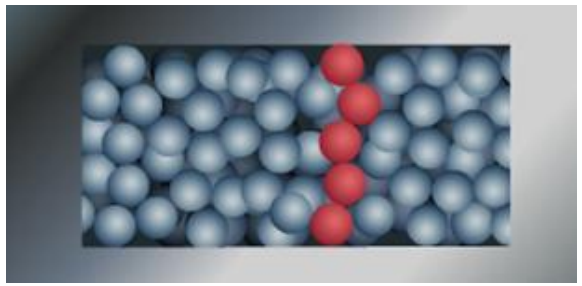
There are two rules used in Stencil Design

Minimum Apertures Guidelines

$$1)\ Aspect\ Ratio = \frac{Aperture\ Width}{Stencil\ Thickness} > 1.5$$

E.g. 16 mils pitch QFP: Aspect Ratio is 9/5= **1.8**



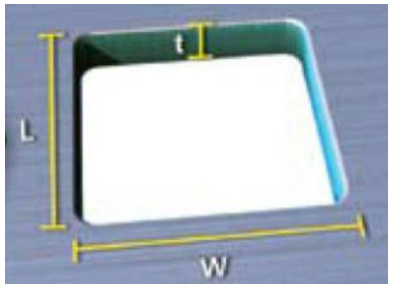


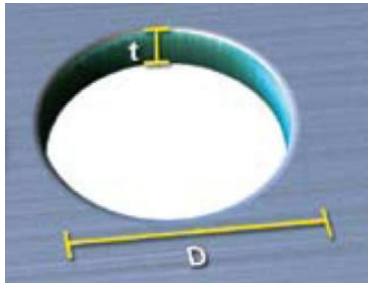
$$2)\ Aperture\ Width >= 5\ particles$$

Area Ratio

$$Area\ Ratio = \frac{Opening\ Area}{Walls\ Area} >= 0.6$$

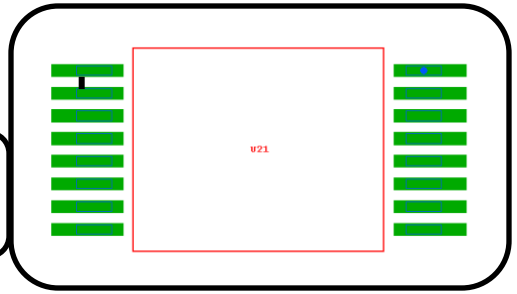
$$AR\ Rect = \frac{LxW}{2(LxT) + 2(WxT)} \qquad AR\ Circle = \frac{D}{4T}$$





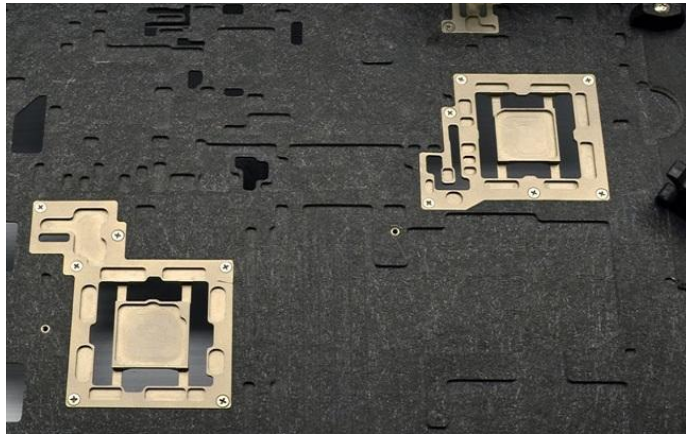
E.g. CSP with 10 mil round aperture.
Area Ratio = 10/(4x4)= **0.625**

Maintain Stencil Webs >= 9 mils

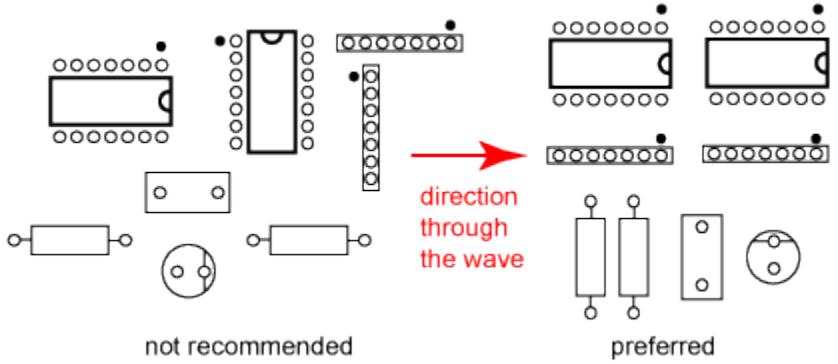


Wave Soldering

- Keep enough distance from SMT to PTH components' pins on the other side in order to avoid high cost wave solder pallets.
- Consider alternate ways: paste in hole, selective wave solders.
- Take care of component orientation to avoid shorts and retouch!



Wave Solder Pallet with Titanium Inserts

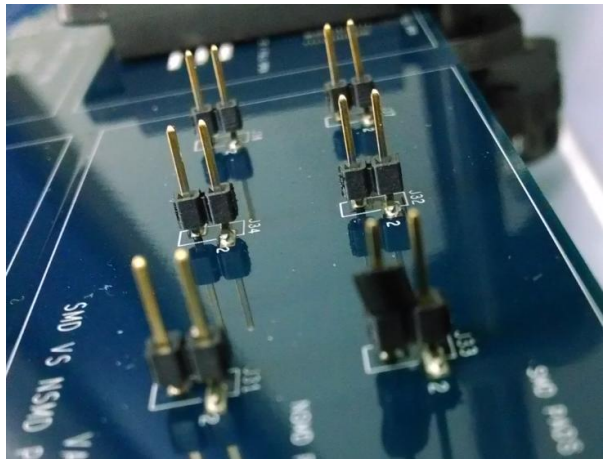


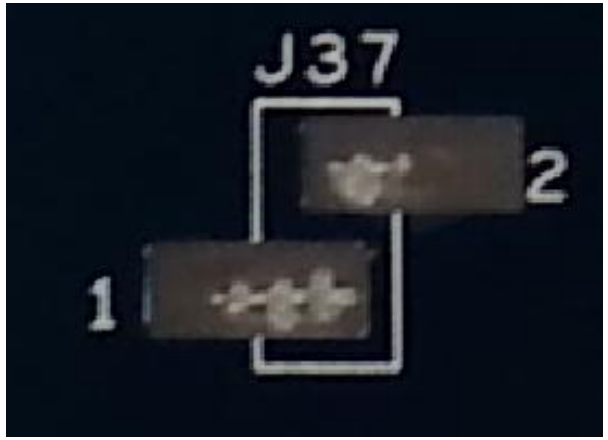
Component's Alignment for Wave Solder

Selective Soldering Process

General Recommendations for Assembly

- Define Type of IPC Class so that CM knows what to accept or not
- Use at least 3 fiducials, spaced by 8 inches
- If having a few SMT comps on bottom move all to top to avoid having duplicated stencil NREs if board is big
- Avoid having lots of PTH comps on both sides so that all of them are processed at the same process on top, otherwise they will have to be hand soldered
- Avoid using two pin SMT headers as they tend to detach along with pads. Recommend to increase the size of pad encroached in case they are used.





STAY UP TO DATE

Stay Up To Date…

- App Notes from NXP: www.nxp.com
- Websites from Fab Shops
- PCB Design Magazine: http://www.iconnect007.com/magazines/pcb-design-magazine/
- SMTA Subscription
- Everything PCB: http://www.everythingpcb.com/
- Follow up on Conferences: such as PCB West, Apex, SMTA…
- Forums such as Sparkfun Electronics

Q & A

Contact: Andres.Marquez@nxp.com

SECURE CONNECTIONS
FOR A SMARTER WORLD

ATTRIBUTION STATEMENT