ENGINEERING CONSULTING

TECHNICAL SALES & SUPPORT

Production Analysis & Learning Services, LLC

PRESENTS

"PCB Layout for CCA Made Easy… Everything You Ever Wanted to Know About Manufacturing but Were Afraid to Ask"

SMTA Surface Mount Technology Association

IPC ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES

• Phone: 949.713 .7229 • Fax: 949.713.7229 •

• For Information Please Call or E-mail •

• palsrvs@palsrvs.com •

“Don’t Even Think About DFM… Cause I Don’t Do It!”

Why DFM…

- *Fewer Design & Prototype Changes*
- *Lower Cost*
- *Better Reliability*
- *Higher Quality*
- *Ensures Success with New / Future Product Introductions*
- *A Richly Rewarding Experience that Motivates Personnel*

Why Should I Care…

- Designers who take the time to learn and understand PCB manufacturing, CCA manufacturing, and Component manufacturing today…will be employed tomorrow.
- Companies who support a PCB Designer in learning PCB manufacturing, CCA manufacturing, and component manufacturing today…will be in business tomorrow.
- Market Analyst reports show that the company's that team up with PCB, CCA, and Component manufacturers today will become the companies of the future by gaining significant market share tomorrow.
- Processes have pushed the envelope on equipment available today. Techniques used by these manufacturing companies may cross over and could become the solution the other was looking for.

Ok fine…WIIFM?

#10 Greater Knowledge Base & Understanding of Mfg.

#9 Improved Efficiency By Understanding More

#8 More Desirable for a Potential Employer

#7 Less Whining by Your M.E. Or C.M.

#6 Fewer Design Change Requests = Less Work For You!

#5 Less Likely to Become the ME's Scapegoat

#4 Still Employed Tomorrow

#3 Cost Savings Realized in Mfg. Directly Attributed to You

#2 On Time Shipments Directly Attributed to You

#1 Able to do More For Less, more profitable, and all due to your improved knowledge = More Pay

Now that I Care…What is DFM?

DFM Means…

- **Coordinating Product Design With Mfg. Plant Capabilities to Maximize Efficiency at the Lowest Cost**
- **The days of Design Engineers "Throwing it Over The Wall"to Manufacturing are Over!**

Where Do We Start?

But of Course…Where Else…

At the <u>Executive</u> Level

Must lead with a commitment to infuse the corporate culture with DFM's inherent advantages

But Wait…Don't Stop There!

- **Orchestrate personnel from different job functions**
  - ✓ ***Design, manufacturing (please keep in mind that Test is considered Part of Manufacturing throughout this presentation), purchasing, production, management, and distribution.***
    - ❖ ***All must have desire to perform with unified vision***
    - ❖ ***All must have in-depth understanding of the delicate interrelationships among their job functions***
    - ❖ ***All must be capable of understanding the various tradeoffs dictated by DFM and determine at which stages in manufacturing these tradeoffs should happen to benefit the manufacturer and customer***

SMT…A Crash Course In Manufacturing

The 4 M's

- **MAN**
  - ✓ ***Improper Training vs. Proper Training vs. No Training***
- **MACHINE**
  - ✓ ***Not capable***
  - ✓ ***Not maintained***
- **METHOD**
  - ✓ ***Not capable or Improper***
- **MATERIALS**
  - ✓ ***Bad PCB layout***
  - ✓ ***Garbage In = Garbage Out***

M.E. AKA Fire Fighter

- Typical scenario upon arriving to work
  - ✓ Line outside office door is 4 deep
    - ❖ SMT Line 2 is down due to a broken belt on the machine (operator forgot to do PM)
    - ❖ SMT Line 4 is having wetting issues on the PCB's but product has to ship today…can't you "*make it solder*"?
    - ❖ SMT Line 6 is having placement issues and has not been running all morning
    - ❖ SMT line 8 is having solder paste issues for some reason…what do you want us to do?
  - ✓ Oh, and by the way…all of the product for all of these lines has to ship by 2pm today or we will be late on our shipment and have to pay a 10% penalty. When do you think you can have it all fixed?

A typical cause and effect diagram for the SMT process has well over 100 variables

M.E.'s Mission

- Eliminate as many of these variables as possible
  - ✓ Promotes quick and efficient process trouble shooting due to less variables having to be investigated, which aids in on-time shipments
- Do more for less
- Ensure process and manufacturing lines are as flexible as possible
  - ✓ Everything is inter-changeable…even operators

Easiest First

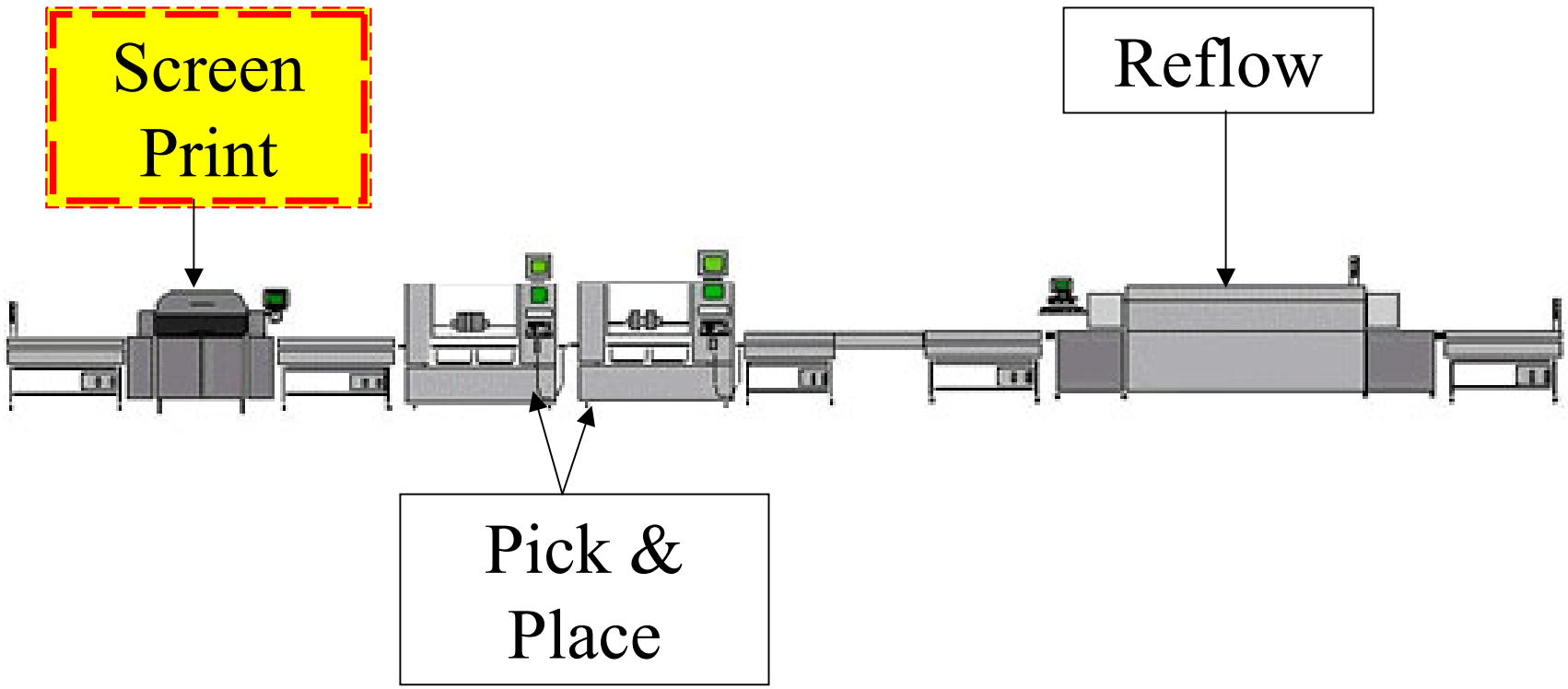
- Machine – TQM (Total Quality Maintenance)
  - ✓ **A good ME will be a stickler about maintenance…Why, because it is one less variable that has to be investigated when trouble shooting process issues**
- Method – Process Validation
  - ✓ **ME will develop a NPI**
- Man – **ARE** *you ready*! **---** AUTOMATE!

  REGULATE! **E**DUCATE!
  - ✓ **Automate as much as possible**
  - ✓ **Where you can't Automate, then Regulate**
  - ✓ **Educate...Educating your work force is the cheapest fastest way to reduce cost and improve quality**

Material…Seems Easy But…

- 80% of MFG. Issues typically falls under Materials
- Designers do it once vs. M.E. does it 10's of thousands of times
- A good PCB layout makes material variable elimination simple
  - ✓ Allows M.E. to focus on the ***"real"*** material related issues
  - ✓ QA gains a better grasp on supplier issues quicker so info can be feedback for improvements
  - ✓ Questionable material is not **"*used anyway*"**
  - ✓ Purchasing and Component Engineering become more aware that a cost savings at the front of the line is not a true cost savings unless the same savings is shown at the back end of the line

SMT MFG LINE



Screen Print Process

- Process Fundamentals
  - ✓ Metal Stencil with matching holes (or apertures) line up with pads on PCB that are to be soldered on
  - ✓ Solder is deposited through the holes in the stencil with a squeegee blade
  - ✓ Thickness of stencil dictates height of deposit (along with several other machine parameters, but deposit height starts here)
  - ✓ The object of this process is to deposit the right amount of solder so as to achieve ZERO defects after reflow

Process Flow

- The PCB is placed on a flat, hard, stable surface
- The stencil is placed over the PCB and the openings on the stencil are lined up with the appropriate pads
- The solder paste is pushed through the holes on the stencil with a squeegee blade

Process Strategy

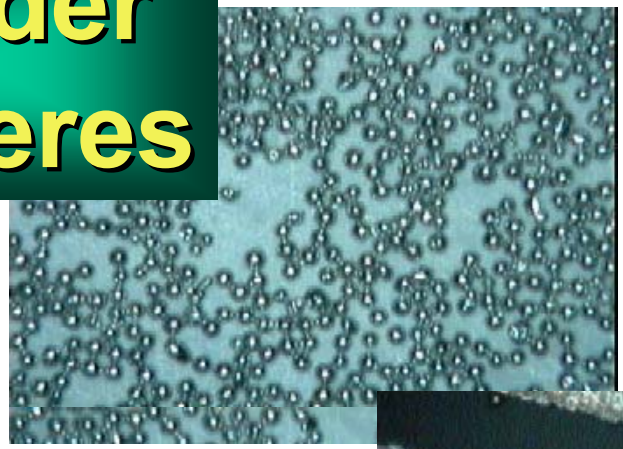
- The trick in this process is to be sure that the PCB is perfectly flat so that the stencil can form a perfect gasket to the PCB

  - ✓ When the stencil does not form a perfect gasket, several different types of solder printing defects can occur which all lead to defects after the reflow process

Manual

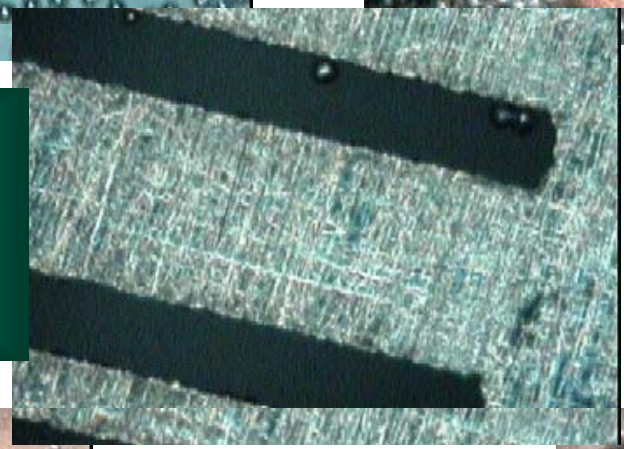
- The PCB is hand placed on a flat tabletop
- The stencil is placed over the PCB and manually lined up to the PCB
- The solder paste is hand printed using a hand held squeegee
- The printed PCB is manually transferred to the next operation
- Stencil can be 20” x 20” Aluminum tubular or cast and smaller. 29” x 29” stencil would be too large to handle and print properly
- Support tooling is typically not used for this approach

Solder Spheres



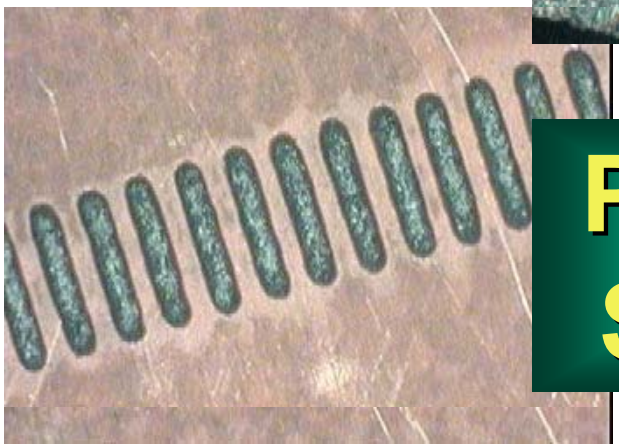
Solder Paste

Stencil



Printed Solder

Printed Solder



Semi-Automated

- The PCB is hand placed inside the screen-printing system
- The stencil is loaded inside the system and manually lined up to the PCB
- The solder paste is automatically printed with squeegee blades mounted inside the machine
- The printed PCB is manually transferred to the next operation
- Stencil size is typically 20” X 20” Aluminum tubular.
  - ✓ *Aluminum cast frame requires a special fixture to mount frame inside machine and typically increases set-up time by 5 to 10 minutes*
- Support tooling is manually placed support pins


SMT

Automated

- The PCB is automatically loaded inside the screen-printing system via a magazine loader and edge conveyor system
- The stencil is partially hand loaded inside the system and is then automatically pulled into the machine
- The machine then automatically lines up the PCB to the Stencil via an upward and downward looking camera, which uses fiducials on the PCB and the Stencil
- A rising table automatically comes up to support the PCB
- The solder paste is automatically printed (and dispensed if option is available) with squeegee blades mounted inside the machine
- The printed PCB is automatically transferred to the next operation via an edge conveyor
- Stencil size can be 29" X 29" or 20" X 20" Aluminum tubular
  - ✓ *Aluminum cast frame requires a special fixture to mount frame inside machine and typically increases set-up time by 5 to 10 minutes.*
- Support tooling is typically manually placed support pins


EKRA

Screen Print Over View

- Key to good screen print process is creating the gasket between stencil and PCB
  - ✓ Number one variable which affects Gasket is Board Support
  - ✓ Ideal fixture for Primary (top) side assembly (no components on Secondary (back) side) is flat stable surface
  - ✓ Ideal fixture for Secondary (back) side (components on Primary (top) side) is vacuumed formed fixture
- Ideal Stencil size is 20” X 20”
  - ✓ Allows flexibility by quick interchanges between various available processes
  - ✓ Max Array sizes are minimized, which are easier to process and manage
  - ✓ Stencil Storage is more manageable as compared to the 29” X 29”
  - ✓ Overall board size exceeds 20” X 20” then use 29” X 29”

DFM Design Guide Lines For…

Screen Print

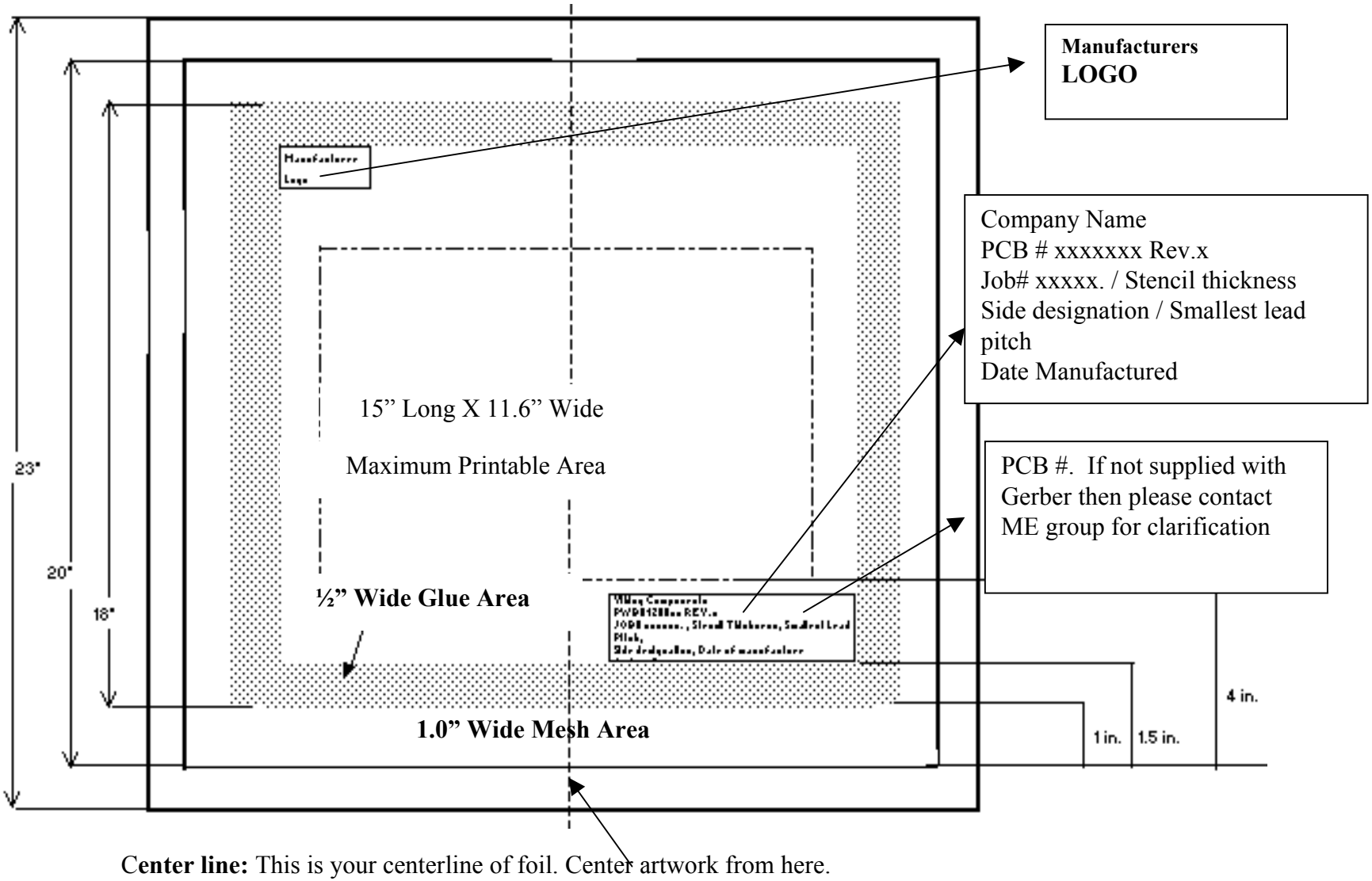
Array Panelization

- Should the PCB layout Engineer specify this…ABSOLUTELY! Don't just let the PCB house control this
  - ✓ A PCB house is only going to optimize the panel size based on their process requirements, and not your CCA houses
- Panelization should be based on…
  - ✓ ***ALL*** machine capability,
  - ✓ Optimized through put on manufacturing line
  - ✓ Optimized use of raw materials for both the CCA & PCB fab house
- PCB layout, CCA Manufacturing, and PCB Fabrication Engineers should all work together to develop a standardized approach

Array Standardization

- 20” X 20” Stencil
  - ✓ Minimum array length is 3" (length of array is X direction)
  - ✓ Minimum array width is 2.5" (width of array is Y direction)
  - ✓ For PCB Thickness = .042" or greater, max length (X) = 14" & max width (Y) = 12“
  - ✓ PCB Thickness = .04199" to .014”, max length (X) = 14" & max width (Y) = 6“
  - ✓ PCB thickness under .014” typically require a carrier for the entire SMT process
  - ✓ Array Width (Y) should never exceed Length (X).
    - ❖ Makes array too unstable for automated process
    - ❖ Weight of components

20”x20” Tubular Frame



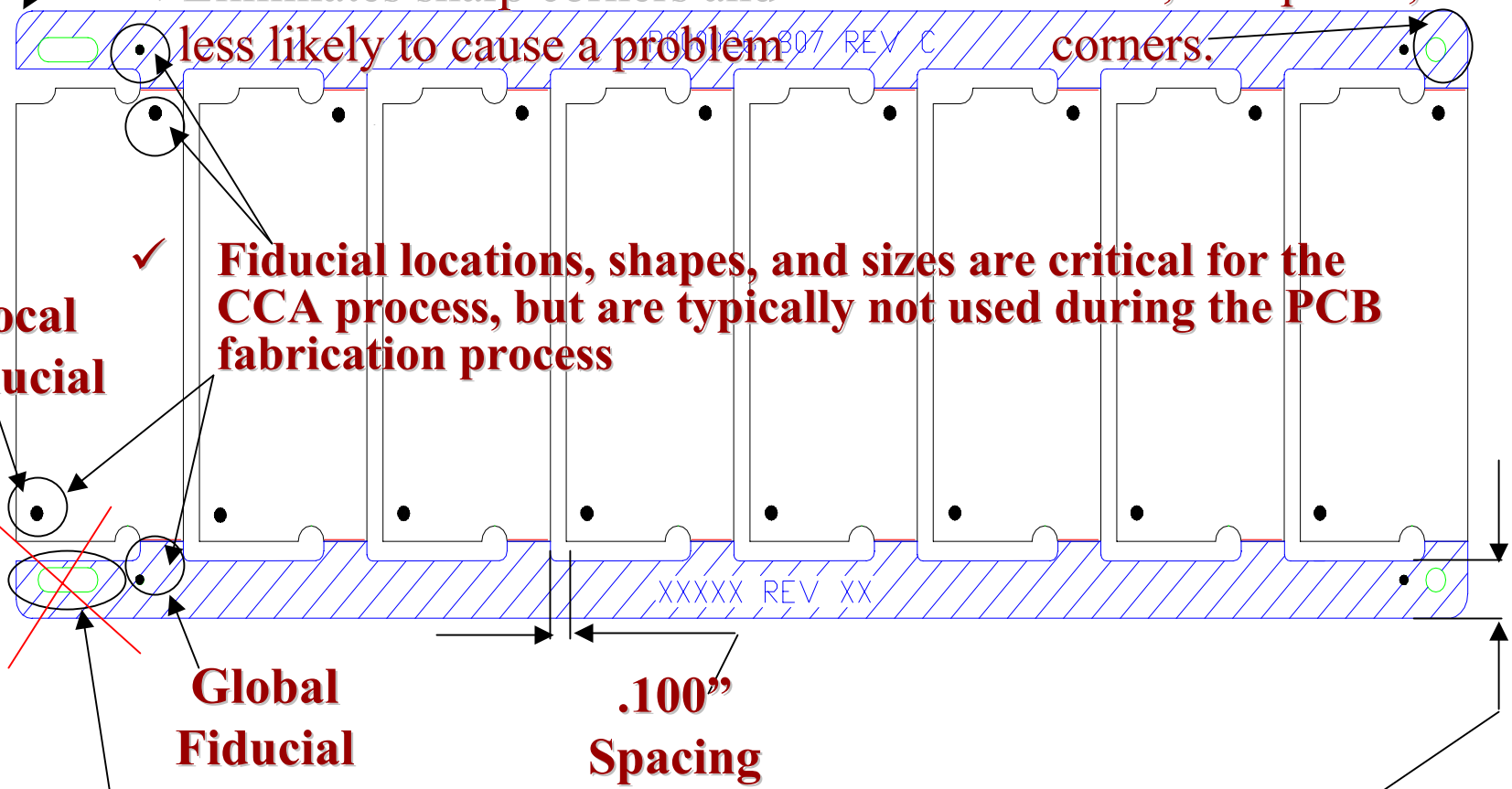
Center line: This is your centerline of foil. Center artwork from here.

Array Standardization

✓ A radius should always be used on rail corners

❖Eliminates sharp corners and less likely to cause a problem

✓ Standard tooling hole size is .125” +.002“/-.001", non-plated, 4 corners.



✓ **Fiducial locations, shapes, and sizes are critical for the CCA process, but are typically not used during the PCB fabrication process**

Local Fiducial

Global Fiducial

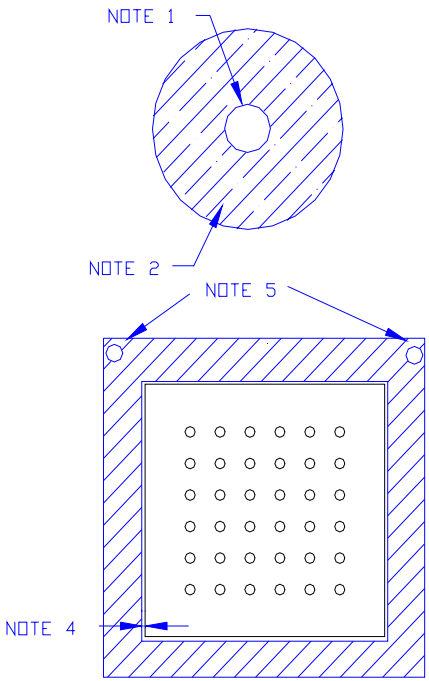
.100” Spacing

✓Do not use slotted tooling holes!

✓ Standard rail width is typically .400”

Array Standardization

- ✓ Standard Global Fiducials should be .05" (round) plated, and placed on all 4 corners of the array rail with 2X Solder Mask clearance
  - ❖ The Primary global fiducials should always mirror the secondary global fiducials



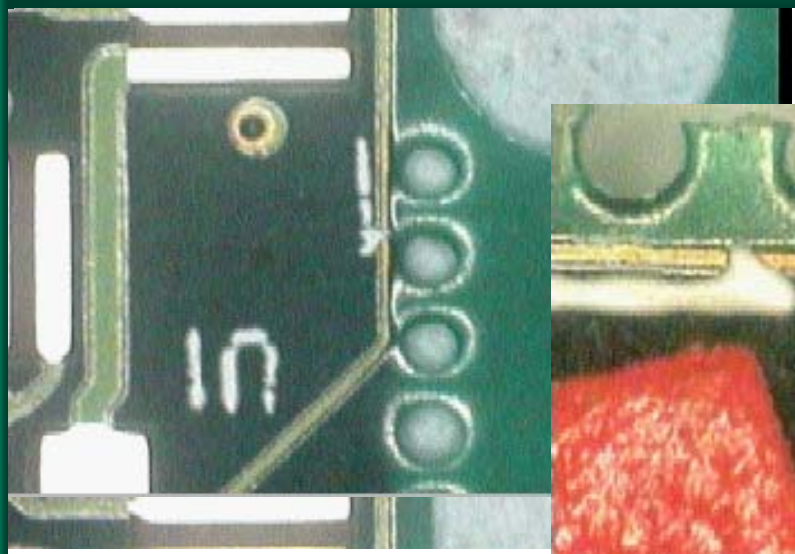
(8) BGA BALLS' PITCH AND DIA. TO BE CONSISTANT UNLESS OTHERWISE SPECIFIELD.
(7) WHENEVER POSSIBLE, DO NOT MIRROR TOP & BOTTOM LAND PATTERNS FOR X-RAY & REWORK PURPORSES.
(6) OBJECT/COMPONENT FREE AREA FOR REWORK .20" FROM COMPONENT EDGE PREFERRED, .05 MIN ACCEPTABLE.
(5) LOCAL FIDUCIALS TO BE ON ADJACENT CORNERS AS NEAR AS POSSIBLE TO COMPONENT AS SHOWN. QTY 2 REQUIRED.
LOCAL FIDUCIALS REQUIRED ON ALL COMPONENT LAND PARTTERN.
(4) SILK SCREEN BOX INSIDE LINE SHOULD BE 0.01 LARGER THAN COMPONENT OUTSIDE LINE.
(3) FIDUCIAL PLATING TIN LEAD PREFERRED, GOLD & SILVER IMMERSION ACCEPTABLE. FLATNESS .005.
(2) OBJECT FREE AREA: AS LARGE AS POSSIBLE, MINIMUM 2X FIDUCIAL DIAMETER.
(1) PCB FIDUCIAL DIA 0.50" REQUIRED. LOCAL FIDUCIAL DIA 0.025 PREFERRED, 0.20" TO 0.50" ACCEPTABLE
NOTES:

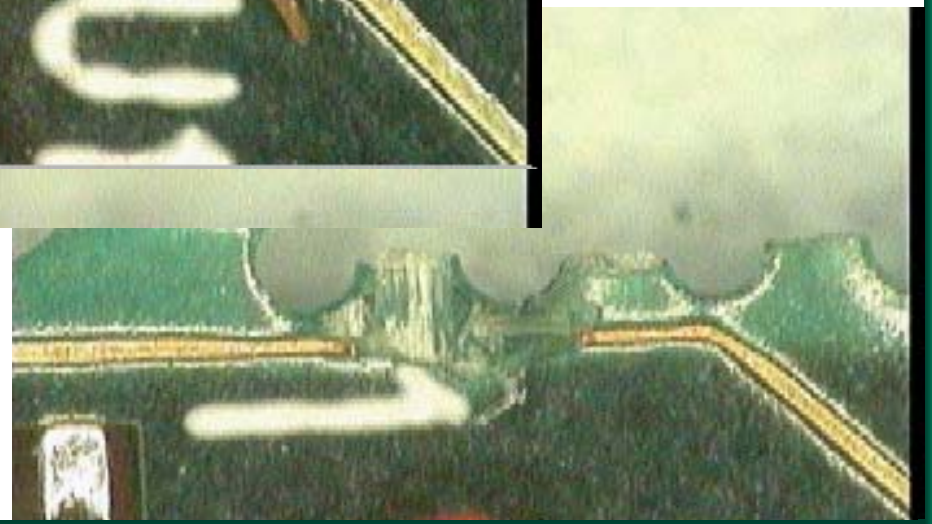
- ✓ Local Fiducials – Glad you asked! I'll take it!

Array Standardization

- ✓ Board edge Clearances
  - ❖ .020”
- ✓ Scoring
  - ❖ Web must be sufficient to allow for the board to be processed through entire CCA process
  - ❖ Web thickness is also dependant on the over all board thickness
  - ❖ A good web to go with is .011” +/- .003” with a 55 to 60^{o} angle
  - ❖ THE DEPTH OF THE SCORE BETWEEN TOP AND BOTTOM MUST BE WITHIN 0.005" +/-0.001" OF EACH OTHER

Reasons For…





Clearing Board Edges

SolderMask

- Extremely critical for screen print process
  - ✓ Biggest mistake seen by board layout engineers…
    - ❖ COMPENSATING DIMENSIONAL CHARACTERISTICS FOR PCB PROCESS
      - All dimensions should be final dimensions
      - Let PCB fab house compensate for you
  - ✓ SolderMask height must be below or even with pad height
  - ✓ Non-Solder Mask defined pads are the best
  - ✓ Solder Mask defined pads used when pad shape is not defined by design
  - ✓ Tent all Via's whenever possible
  - ✓ Clear all solder mask away (as well as traces) from all break-away areas
  - ✓ Use a light Green Solder Mask and stick with the same color solder mask

Silkscreen Images

- IS IT NECESSARY? HECK NO!
  - ✓ Folks have gotten lazy
- Use a Transparency
- It will Force the CCA house to use the Assembly drawing
  - ✓ Less mistakes will be made!

Board Thickness

- PCB material determines Tg
  - ✓ Tg = glass transition phase
    - ❖ Temperature the material begins to change state
- 4101-23 is typical FR4 Tg = 140°C
- 4101-24 is high Tg FR4 Tg = 170°C
- Thin PCB's must have high Tg material
- .045" and < thick PCB's use the 4101-24
  - ✓ Specify 4101-24 for everything if possible
- No Generic Call Outs on FAB Drawing!
  - ✓ Clearly specify material type with 4101 call out

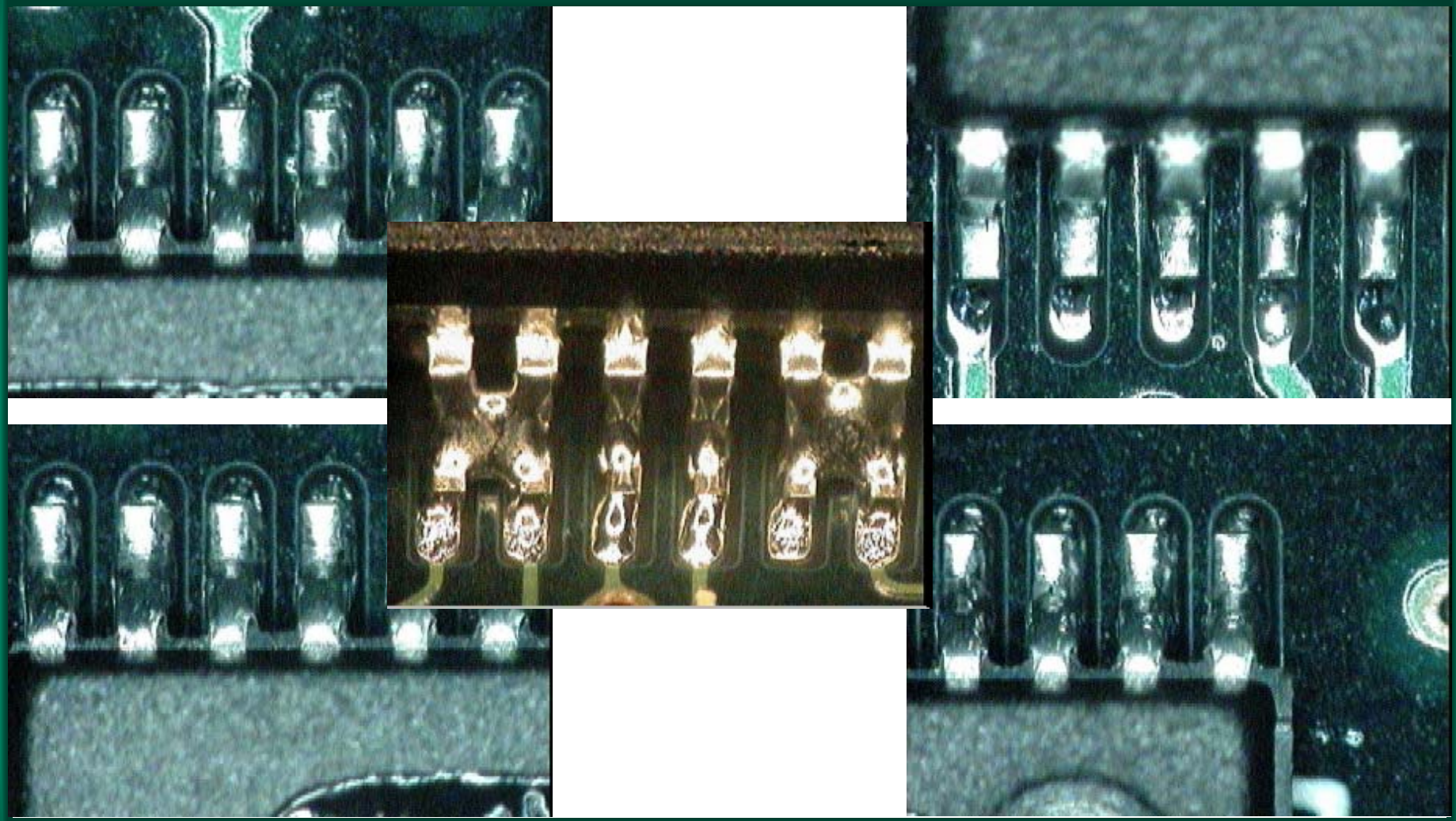
Copper Distribution

- Critical In Reducing Warpage
  - ✓ Especially thin multi-layer substrates
- Copper must be distributed evenly
  - ✓ 4 Layer PCB
    - ❖ Layer 1 & 4 matched
    - ❖ Layers 2 & 3 matched
  - ✓ 6 Layer PCB
    - ❖ Layer 1 & 6 matched
    - ❖ Layers 2 & 5 matched
    - ❖ Layers 3 & 4 matched

Land Creation

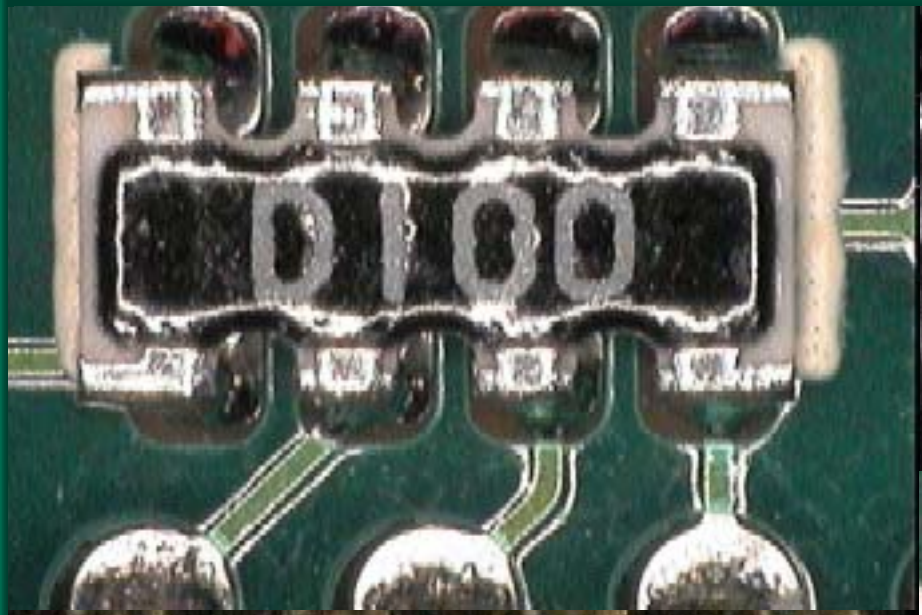
- Standardize!
- Work with M.E. and Optimize!
  - ✓ Use SM782 As Starting Point Only!
- Validate by Testing!
- Lead Pitch Must Match Part Spec.
  - ✓ Metric = Metric & Inches = Inches
- Key things to Remember
  - ✓ Every solder joint must have 3 sides
  - ✓ Larger surface area = more reliable
  - ✓ Too large/long of pads results in bridging
  - ✓ Think SYMETRICAL
  - ✓ Ideal Pin 1 / Polarity Mark = Longer Lead
    - ❖ Every 10^{th} pin for multi-leaded devices.

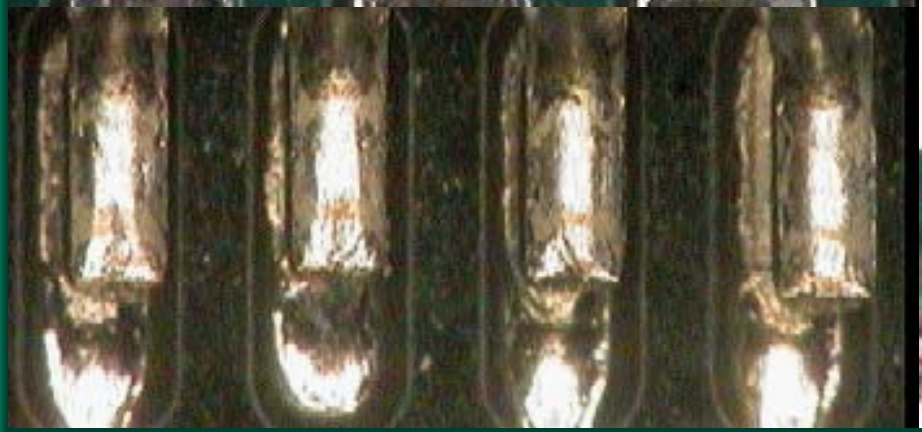
Reasons Lead Pitch

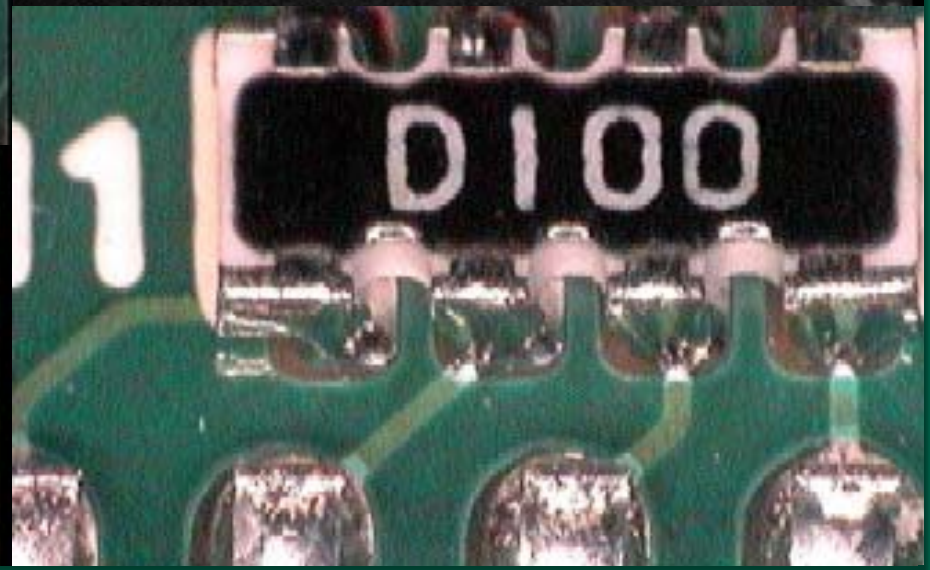


Must Match Part Spec.

Reasons Lead Pitch



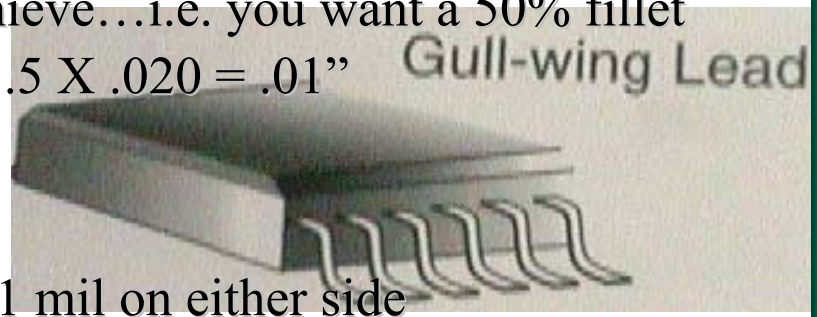




Must Match Part Spec.

Land Creation

- Gull Wing Device
  - ✓ Needs No toe fillet
    - ❖ Pads do not need to extend in front of toe
    - ❖ Remember - 3 sides to a solder joint…so give it a toe fillet
    - ❖ How much?
      - ▪ Take the Thickness of the lead and multiple by the size of the fillet you would like to achieve…i.e. you want a 50% fillet and the lead is .020 thick. .5 X .020 = .01”

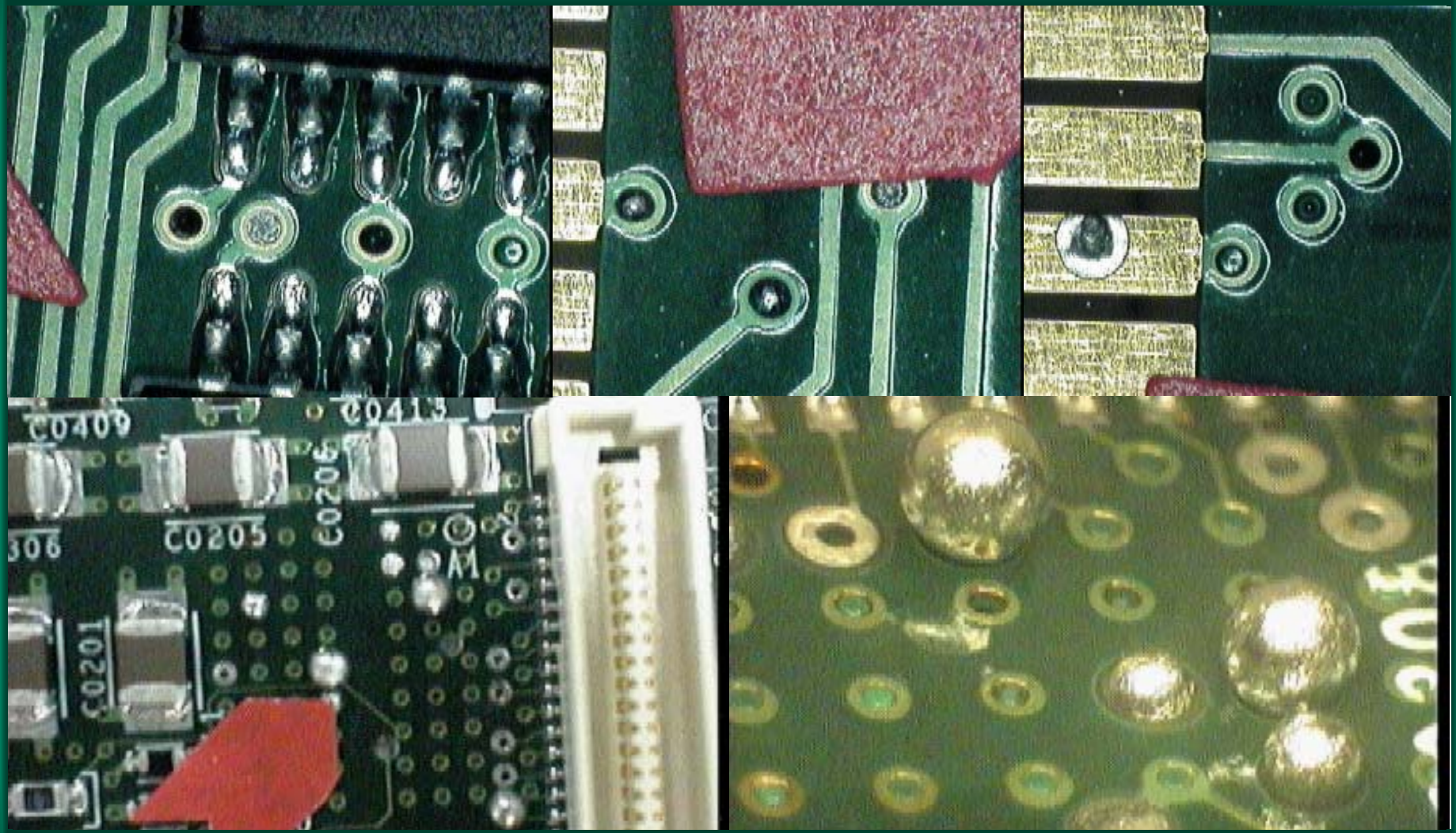


  - ✓ Side fillet
    - ❖ Pitch of the device limits you
    - ❖ You will always need at least 1 mil on either side
  - ✓ 50% Heel fillet Needed (Class 2)
    - ❖ Pad extends underneath device
    - ❖ Usually does not affect real estate (as much!)
    - ❖ Extend pad length pass the heel of the device at least .20 to .40”
    - ❖ Can get away with a .015” extension if necessary.
    - ❖ To much room can lead to shifting of the part from side to side
    - ❖ Just the right amount allows the device to self-center

Via Size

- TENT THEM!
- TOO BIG, THEN FORGET IT!
  - 13.5mil hole stacked 3 high for drilling
  - 12mil hole stacked 2 high for drilling
  - 10mil hole drilled 1 at time
    - Small drill bits wander
    - Affects cost

Reasons for…



Complete Via Tenting

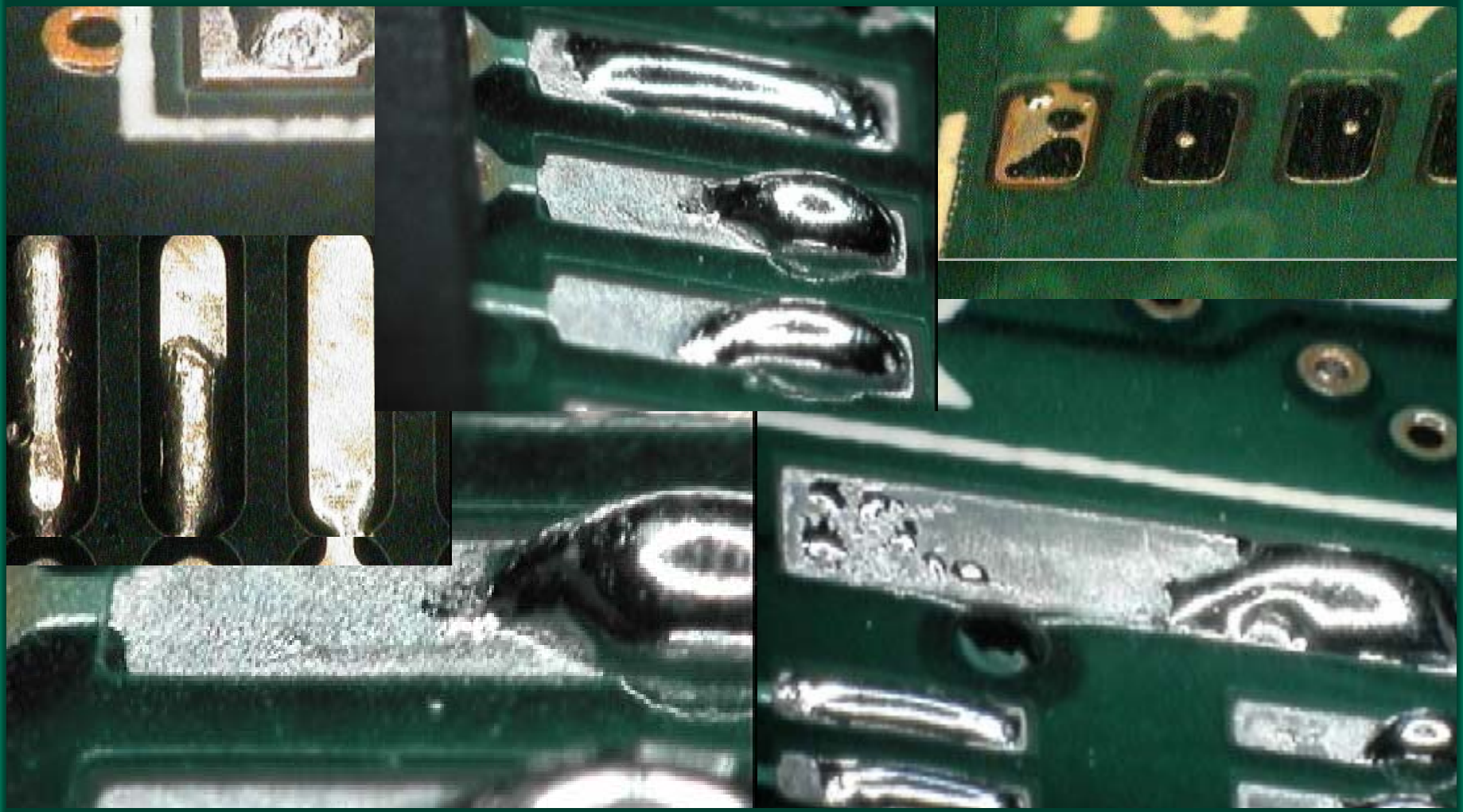
More Reasons for…

Tenting Via's

HASL Surface Finish

- HASL
  - ✓ Board Houses Sub-Out Their HASL Process
  - ✓ Most HASL Shops Lack Process Controls
    - ❖ If it didn't solder the first time (probably due to dirty copper) they send it back through
    - ❖ Each thermal excursion affects PCB MTBF rate
    - ❖ HASL Shops Do Not Properly Maintain Solder Bath
      - ▪ Copper level is too high, which causes Dewetting
      - ▪ Dewetting? No turning back…It can't be fixed!
  - ✓ HASL Hides "*Real*" Issues
    - ❖ Solderability testing absolute must!

Reasons For…



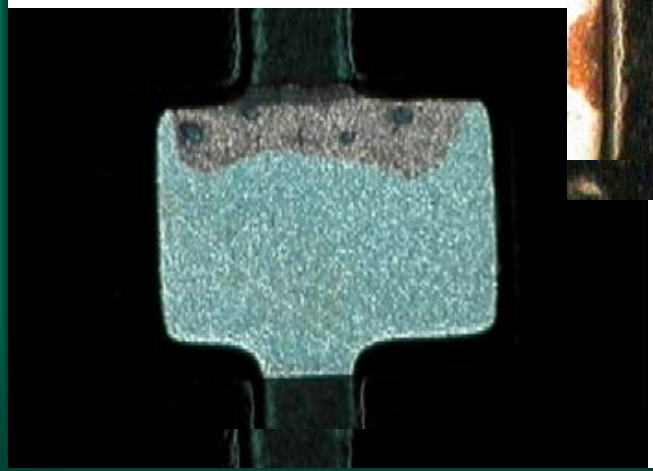
Solderability Testing

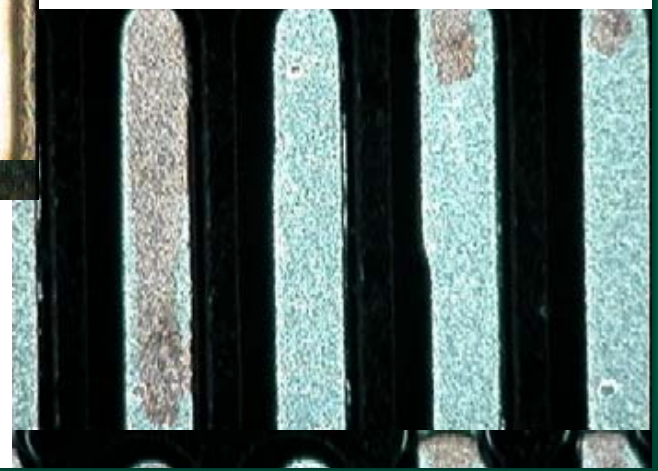
Silver Surface Finish

- Silver Immersion -- Directly Over Copper
  - ✓ Silver Molecular Structure Related to Solder
  - ✓ Silver Amalgamates into the solder
- FLAT, FLAT, FLAT!
- Great Shelf Life
- Neutral PH Bath at Low Temperatures
- Silver Migration…NO!
  - No one can prove this
  - Silver too thin of coating
- Silver Won't Stick to Dirty Copper
  - Great Contrast Allows Naked Eye Detection
  - Silver finish is actually REWORKABLE
    - Silver can be removed, copper cleaned, and re-coated

Silver Surface Finish

- Silver Won't Stick to Dirty Copper
  - Great Contrast Allows Naked Eye Detection
  - Silver finish is actually REWORKABLE
    - Silver can be removed, copper cleaned, and re-coated

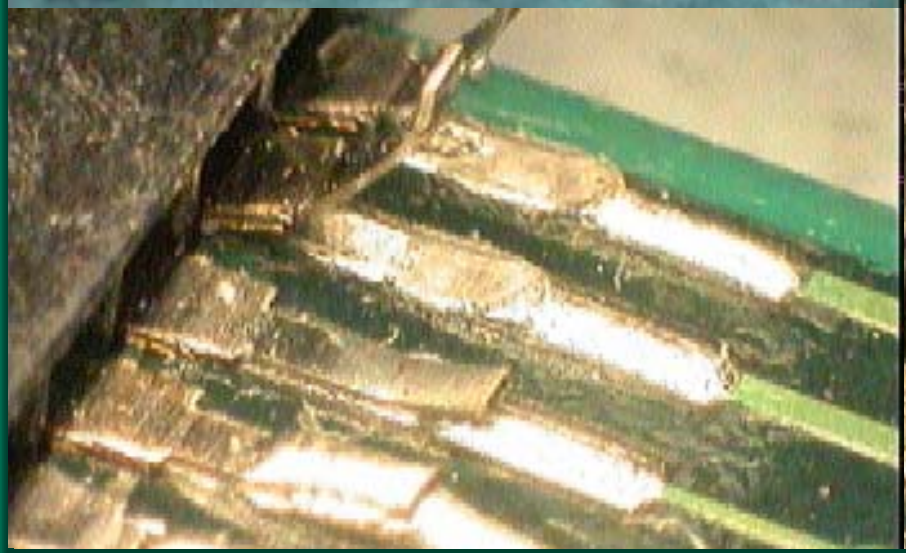




Tin Surface Finish

- Tin Immersion, Here's my thoughts on it…
  - ✓ IT SUCKS!
  - ✓ JUST SAY NO!
  - ✓ DON'T USE IT!
  - ✓ Nothing but soldering issues
  - ✓ You Will Need Two Things
    - ❖ LOTS and LOTS of HIGHLY ACTIVE FLUX
      - ▪ Typical Activator is FLORIDE OR CHLORIDE
        - • Both Are EXTREMELY CORROSIVE
    - ❖ LOTS and LOTS of HEAT
  - ✓ Poor Shelf Life
  - ✓ Multiple handling issues
  - ✓ Hazardous Waste Generated by Bath is Costly

Reasons for Avoiding



Tin Surface Finish

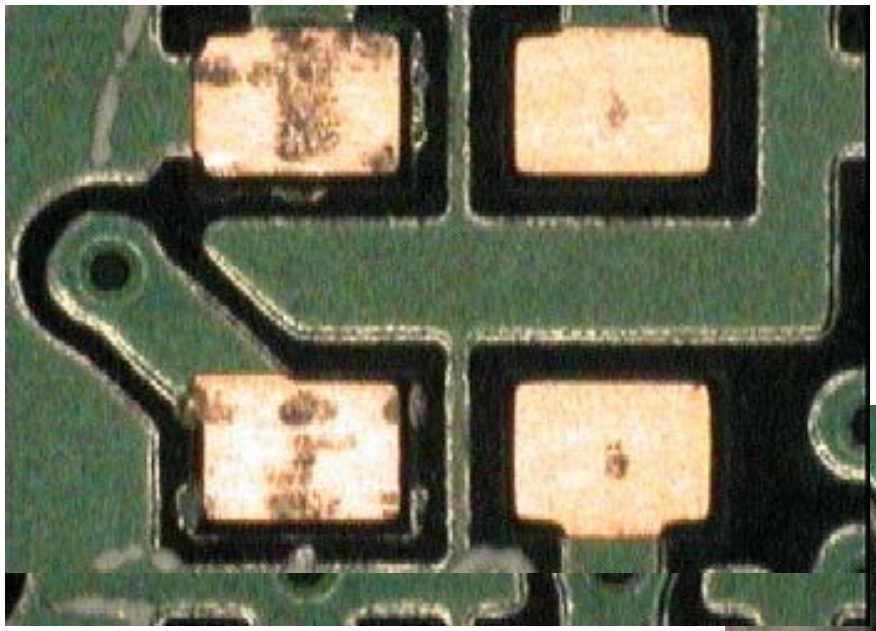
Gold Surface Finish

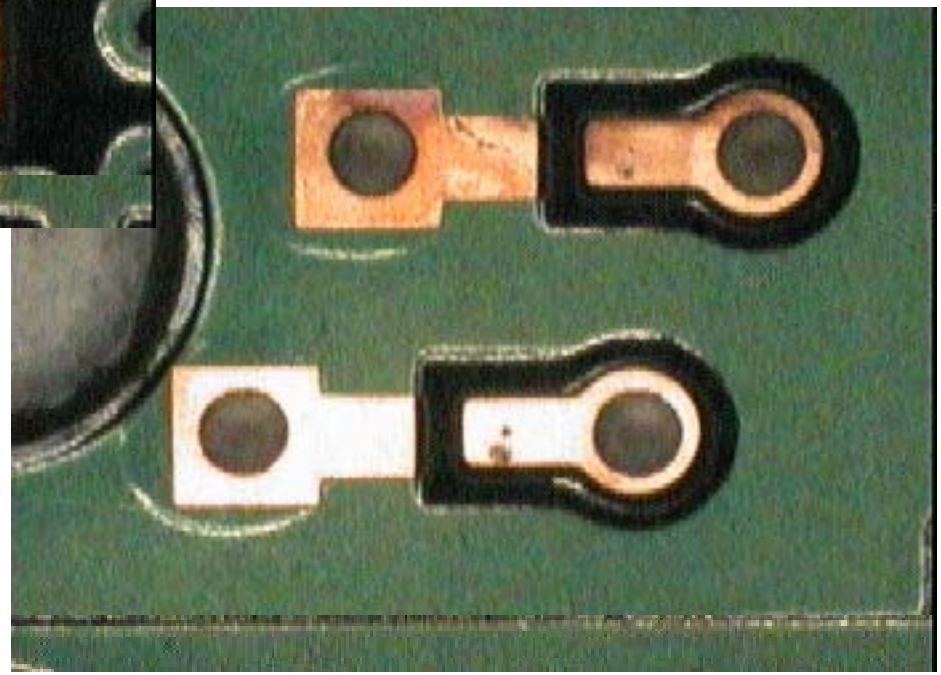
- Immersion Gold over Electrolysis-Nickel
  - Can't get silver…then it will have to do
  - Gold is porous does not seal nickel well
    - Nickel will oxidize over time
    - Becomes impervious to solder
    - Typical Max Shelf Life = 3 months
  - Major issues with black pad
  - Sensitive to handling.

OSP Surface Finish

- OSP, Here's my thoughts on it…
  - ✓ IT SUCKS!
  - ✓ JUST SAY NO!
  - ✓ DON'T USE IT!
  - ✓ Nothing but soldering issues!
- Copper
  - ✓ Whatever happen to just bare copper?
  - ✓ Maybe an ideal approach for a Proto-Type builds

OSP Surface Finish

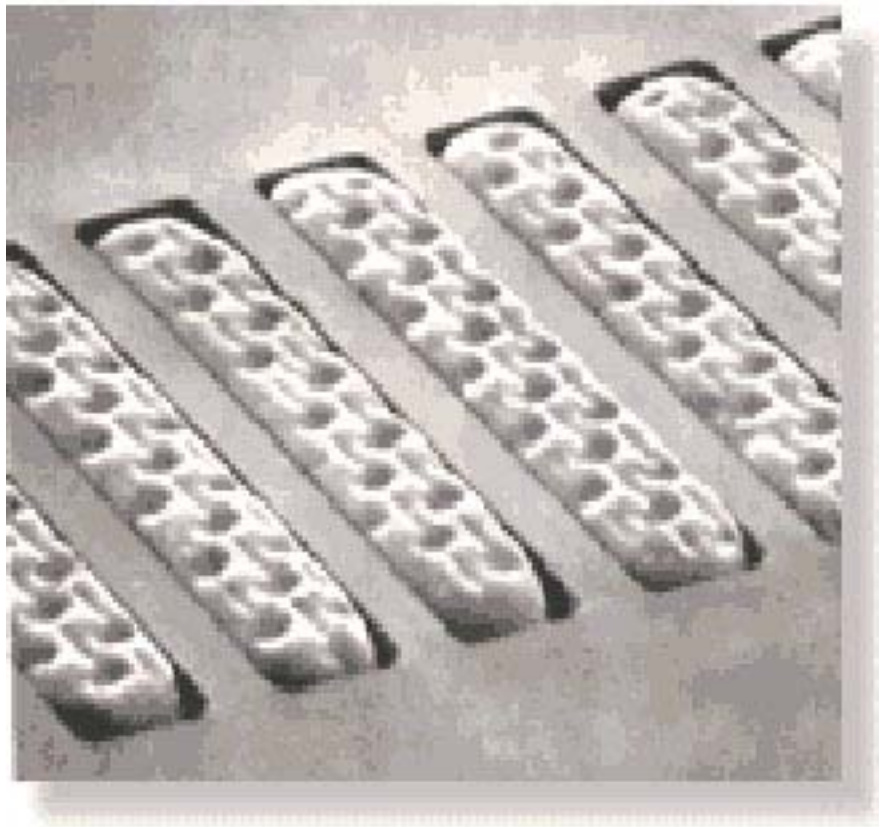




PPT Surface Finish

- PPT or "Precision Pad Technology"
  - ✓ Patented Solid Solder Deposit Process
  - ✓ Solder Coating Over Copper with "*mesh*" impression
  - ✓ CCA does gross print with tacky flux
    - ❖ Phenomenally better End Results
    - ❖ 100% Yields Every Single Time
    - ❖ PPT Passed HALT & HASS Testing
      - ▪ Pad & Laminate Tore while solder joint remained in tact
  - ✓ Ideal for:
    - ❖ Parts with a pitch of .015" or less
    - ❖ uBGA's with solder spheres .015" or less
    - ❖ 0201 devices
  - ✓ Cost:
    - ❖ 5-cents/Sq. Inch for Single-Sided PCB
    - ❖ 10-cents/Sq. Inch for Double-Sided PCB
    - ❖ Potential Cost Center for a PCB House
      - ▪ Typical Charge for HASL = 1 cent/Sq. Inch
      - ▪ Catch 22 - If you don't ask, they don't offer! So Ask!

PPT Surface Finish



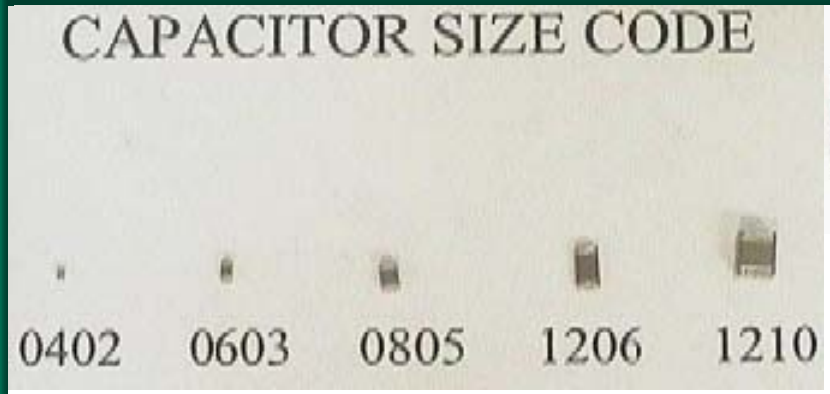
Part Orientation

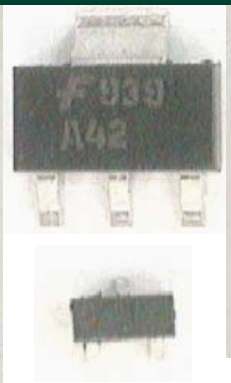
- Apertures Direction VS Solder is Critical
- Long Thin Holes Easier to Fill
- Stroke of Squeegee Blade = Device Leads Direction

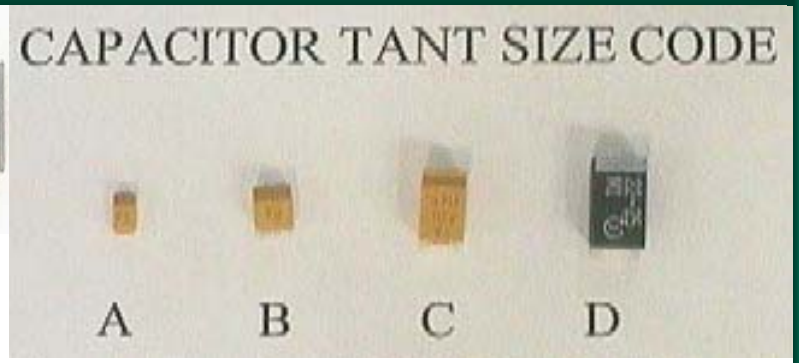
Primary Vs Secondary

- If Possible Make it Single Sided
- Place Odd Shaped on Primary Side
- Symmetrical Layouts Nice
- Don’t Mirror the Parts!

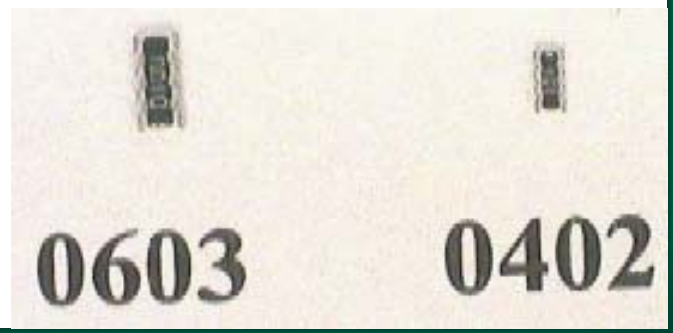
Part Size VS Real-Estate



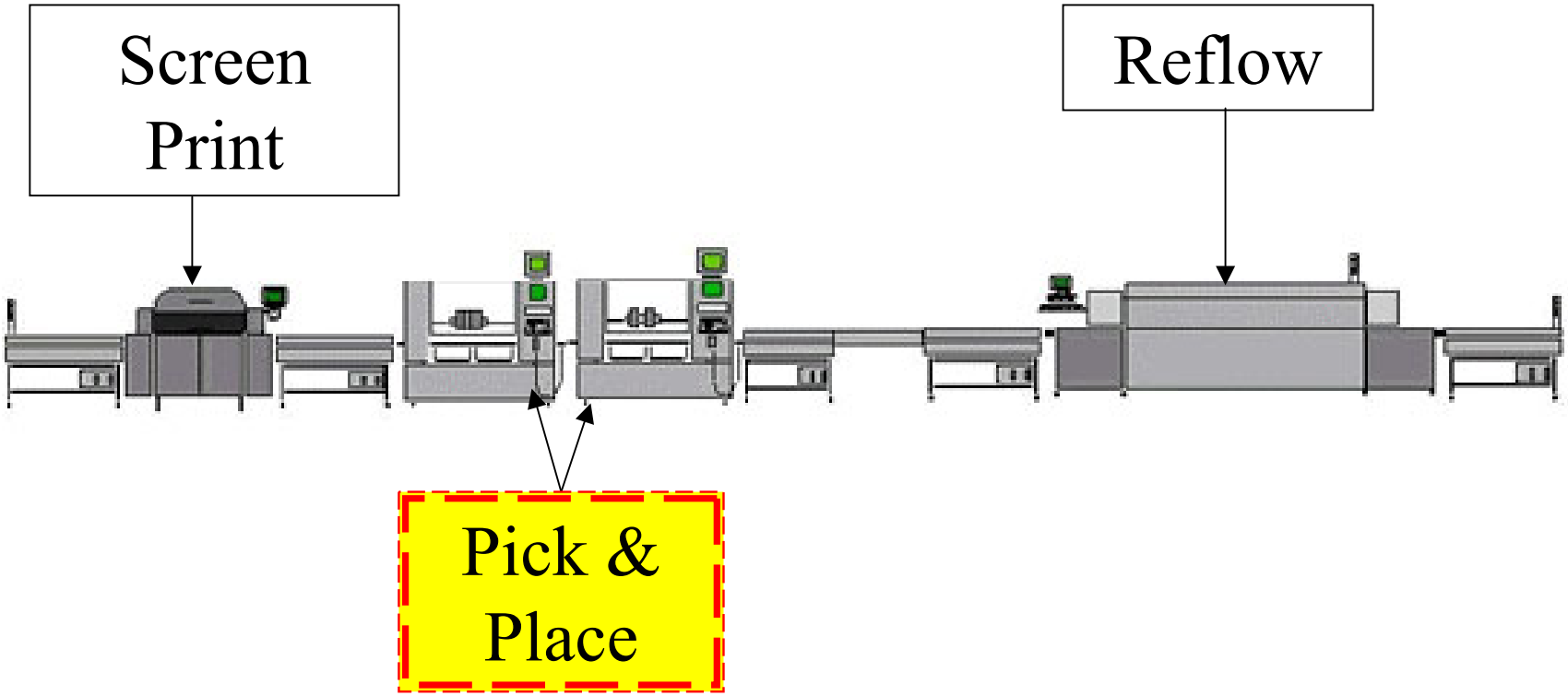




- Don't go crazy on miniaturization!
- Don't use an 0402 when there's room for an 0805…and 0201's – GOOD LUCK!
- Space the parts and/or use appropriately sized parts based on the available support tooling
- Rule of thumb is simple… use the largest part you can for the available PCB real estate & support tooling



SMT MFG LINE



Pick & Place Process

- Process Fundamentals
  - ✓ PCB array is loaded into the machine
  - ✓ Camera looks at global & Local Fiducials
  - ✓ Machines Calculates 0,0 array location based on fiducials
  - ✓ System begins picking, aligning, and placing parts
  - ✓ Object of process is to pick and place components so as to achieve ZERO defects after reflow

Process Flow

- PCB is placed on flat, hard, stable surface
- PCB is aligned to 0,0 array reference point
- Components are picked, aligned, and placed

Process Strategy

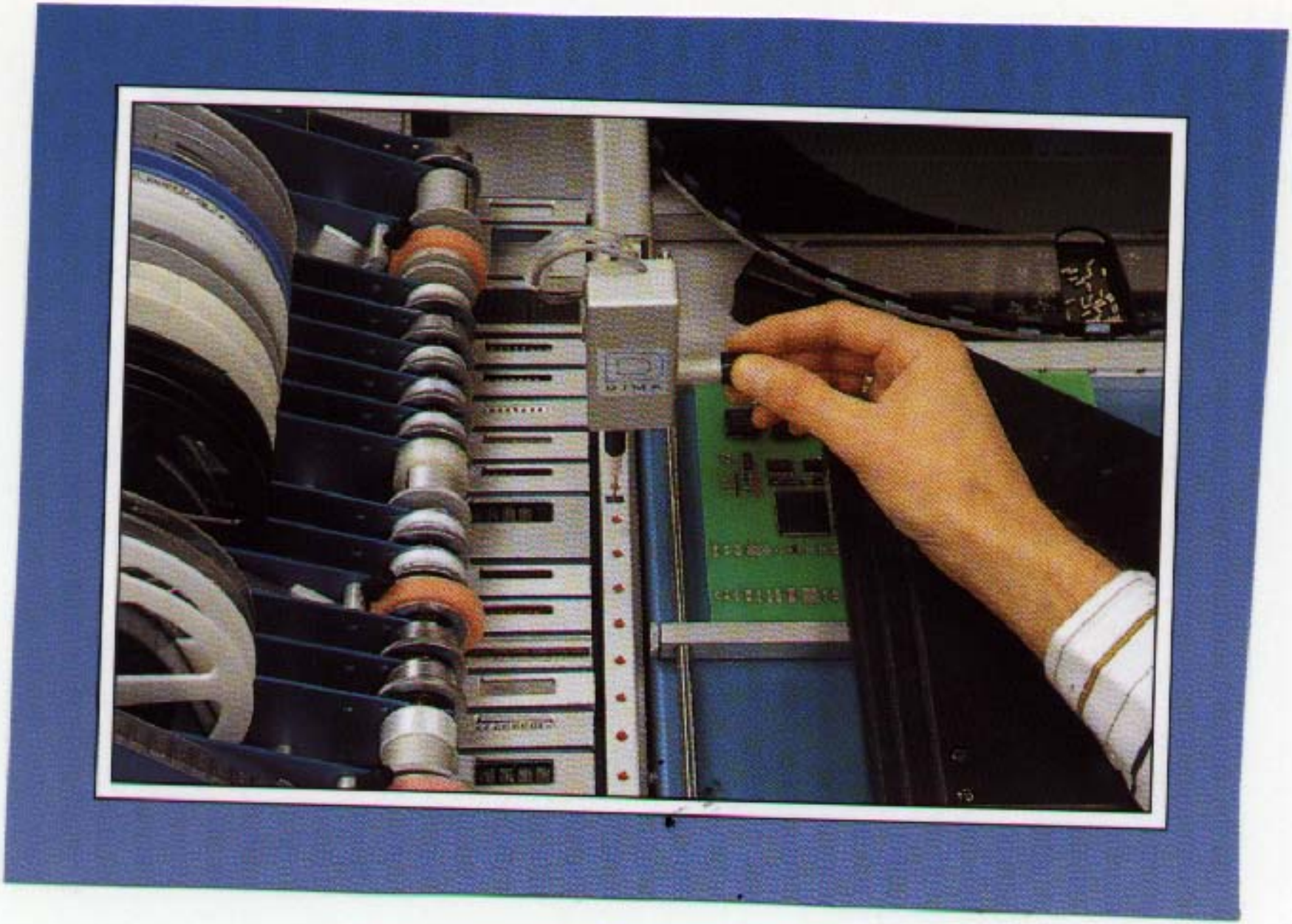
- Generate Placement Program from Gerber
- Machine Set-Up Performed by Capable Operator
- Machine Placement Accuracy Capable

Manual

- The PCB is hand placed on a flat tabletop
- Components are laid out on the flat tabletop
- Components are hand picked with tweezers and hand placed onto the appropriate lands
- CCA is manually transferred to the next operation
- Support tooling is typically not used for this approach

Semi-Automated

- The PCB is hand placed inside pick & place system
- Components are picked and placed either by the machines automated placement head, or manually operated placement head
- The CCA is manually transferred to the next operation.
- Support tooling is manually placed support pins
  - The tip of the support pin diameter is typically 1/16”
  - Parts too close will not allow the proper placement of pins



Automated

- Automated loading Via Edge Conveyor
- Automated Rising Support Table
  - ✓ Supports PCB during placement process
  - ✓ Tooling for this typically manually placed
- Pick & Placement
  - ✓ Overhead Gantry Moves to Pick and Place Location
    - ❖ Overhead Gantry Typically Used as line balancer
  - ✓ Turret - Board Moves to the head
    - ❖ Turret Typically Seen on Chip Shooters
- Alignment
  - ✓ Laser - Scans side of the part and compares
  - ✓ Upward Looking Camera looks upward at part
- Feeders manually placed into position
  - ✓ Reels – Ideal due to location of feeders
  - ✓ Stick(or Tube)- Issues with feeders & packaging
    - ❖ Best to have parts tape and reeled
  - ✓ Tray - Not always desirable
    - ❖ Travel Distance


YAMAHA

Pick & Place Over View

- The PCB must be perfectly flat and as stable as possible
  - ✓ Diving Board
- Feeders Must be Mechanically Stable
  - ✓ Pick Position
- Head/Picking System Must Pull a Good Vacuum
- Alignment Must Be Within at Least +/- .004”

DFM Design Guide Lines For…

Pick & Place

Array Panelization

- Guess what…these requirements are identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for P&P
- Standardizing on Overall Board Size Minimizes Required Set-Up Time for P&P
- Be as Flexible as Possible
  - ✓ So You Can Interchange Tooling

SolderMask

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for P&P

Silkscreen Images

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for P&P

Board Thickness

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for P&P
- Remember Stable Flat Surface!
  - ✓ Warped boards won't lie flat…no matter what!
  - ✓ Boards that don't lay flat Become Diving Boards During P&P

Copper Distribution

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for P&P

Pad Geometry

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for P&P
- Standardize! Standardize! Standardize!
  - ✓ Don't Forget Machine Placement Accuracy
    - ✓ Parts Placed +/-.003" with Placement Accuracy of +/.004"…Just Won't Work!

Via Size

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for P&P

Surface Finish

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for P&P

Part Orientation

- Direction Part Faces VS Direction it Faces when Placed
  - ✓ Remember movement is time and money
  - ✓ No Rotation is Ideal
- Parts in Reel Typically Facing Ideal Direction

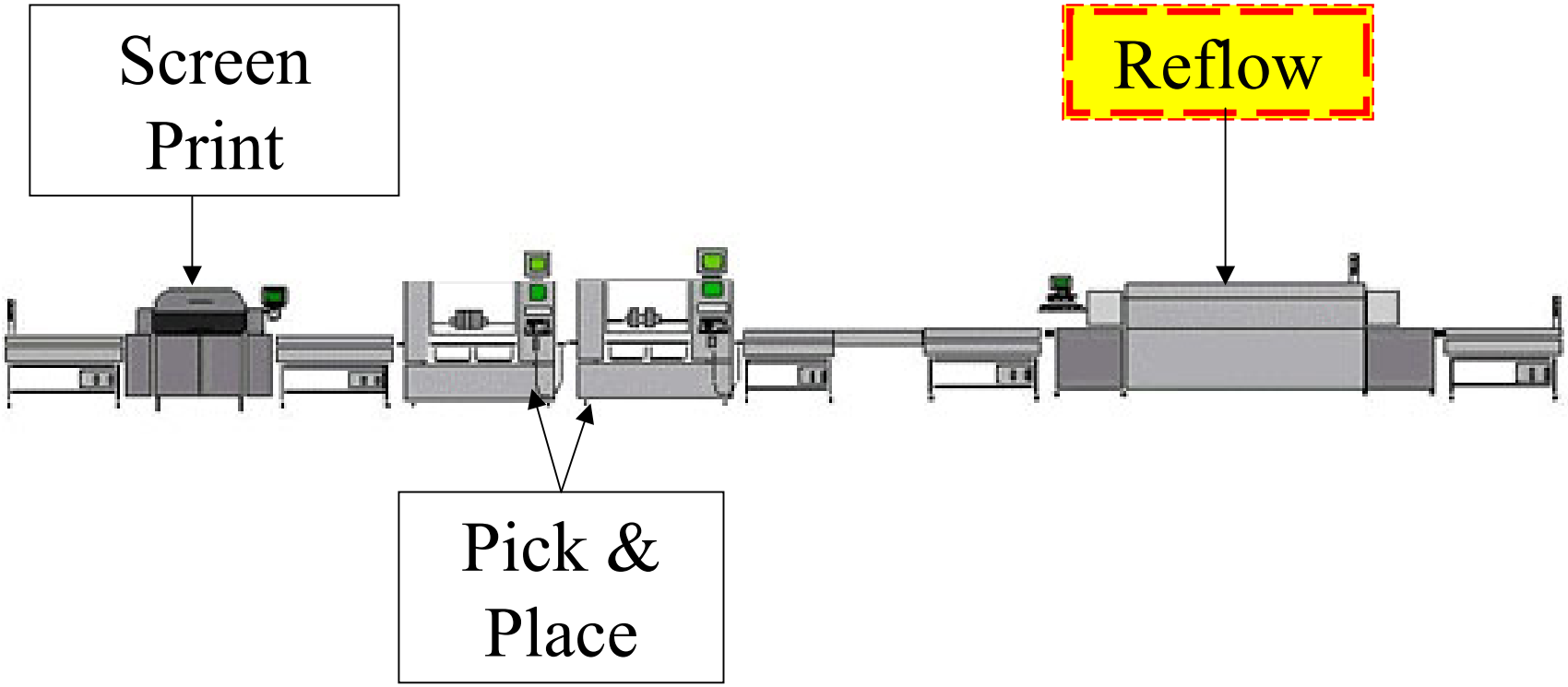
Primary VS Secondary

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for P&P
- Smallest Parts Always Placed First

Part Size VS Real-Estate

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for P&P
- The smaller the part the slower
- Rule of thumb is simple…use the largest part you can for the available PCB real estate & support tooling

SMT MFG LINE



Oven Process

- Process Fundamentals
  - ✓ Based on time VS temperature
  - ✓ Dependant on:
    - ❖ How Oven Performs
    - ❖ M.E. understands thermo-dynamics, metallurgy and raw materials
  - ✓ Solder not properly activated by flux will not allow solder to wet to the pad
    - ❖ Appears Rough with Scales
  - ✓ Solder held in liquidous for too long or too short of time creates large lead/tin intermetallics that become brittle
  - ✓ Solder not cooled fast enough create large granular structured solder joint that will fail prematurely

Process Flow

- PCB Placed On Flat Mesh Belt or Pin Chain conveyor
- Mesh Belt/Pin Chain Speed Set To Achieve Optimum Time in Oven
- Product Travels Through Oven Entering Each Heating Stage at Appropriate Time
- Heats Assembly Enough for Solder to Reflow
- Product Continues Through Cooling Section Where Cool Air is Blown Over the Assembly

Process Strategy

- Oven Profile Based on Total Mass Density, Solder, Flux
- 90 Second Pre-Heat (2.5^{0}C/min max ramp rate)
- 90 Second Soak Temperature (soak temperature depends flux)
  - ✓ Soak Process Used to Equalize Temperature
  - ✓ Minimizes Thermal Shock and CTE Issues
- 45 - 60 Second Reflow
  - ✓ Temperature Quickly Spiked 30^{0}C Above Eutectic Point
- Quickly Cooled
  - ✓ The Faster the Better
  - ✓ Creates Tighter Grain Structure
- 4-minute Oven profiles Ideal!
  - ✓ Take Heated Length of Oven & Divide By 4 = belt speed
  - ✓ Longer Ovens Allow For Better Equalized Temperature

Oven Methods

- Hot Air Convection
  - Blowers force air over heating coils producing hot air
  - Hot Air Continues to Flow Over CCA's
- IR
  - Infra-Red Heating
  - No Air
  - Sensitive to Colors.
  - Works Well With Flex Circuitry
- Convection-IR combo
  - Most Common
  - Ideal for CCA on Standard Rigid PCB's
- Vapor Phase
  - Uses a Flourinert fluid that boils at a specific temperature
  - Works Well on Substrates That Absorb a lot of Heat

Oven Methods

- Laser Soldering / Selective Soldering
  - Ideal for Selective Soldering of Sensitive Devices
  - Expensive, but Worth it if Volume Supports it
- Hot Bar
  - Hot Bar Comes Down On Area Requiring Soldering
  - Typically Used with Flex Circuitry
- Air Pressure Pulsing in Vacuum Chamber
  - Vacuum-Sealed, Nitrogen Atmosphere Chamber
  - Pressure in Chamber Varied Causing Solder to Pulse
  - Referred to as a Flux-Less Soldering System
  - Ideal for Hermetically Sealed Devices


SMR-800

Oven Over View

- Belt Speed
  - ✓ 4 min. profile is ideal
  - ✓ Take heated length of the oven & divide by 4 for belt speed
  - ✓ Adjust Belt Speed When Density of CCA Does Allow Proper Reflow By Adjusting Zone Settings Only
  - ✓ Belt must be stable and not vibrate.
- Delta T
  - ✓ Temperature Difference Across Belt
  - ✓ Ideally should be within 4°C
    - ❖ If Too Great it Causes Uneven Heating / Reflow
    - ❖ Faulty Oven Seals Affect Delta T and Causes Heat Loss
- Heating System
  - ✓ Quick Response Time to Changes Within the Chamber
    - ❖ Faster Response Times Allow Better Heat Control For Load Changes
  - ✓ Proper Oven Maintenance Ensures Faster Response Times

DFM Design Guide Lines For…

Oven / Reflow

Array Panelization

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for Reflow
- Standardizing on Overall Board Size Minimizes the Number of Reflow Profiles Required
- Be as Flexible as Possible
  - ✓ So You Can Interchange Profiles

SolderMask

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for Reflow

Silkscreen Images

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for Reflow

Board Thickness

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for Reflow
- Remember Stable Flat Surface!
  - ✓ Especially While Solder is in Liquidous State

Copper Distribution

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for Reflow

Pad Geometry

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for Reflow
- Standardize! Standardize! Standardize!
  - ✓ Don't Forget Surface Tension!
    - ❖ Proper Pad Design Promotes Self-Alignment!

Via Size

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for Reflow

Surface Finish

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for Reflow
  - ✓ Nothing like Solder to Solder!

Part Orientation

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for Reflow

Primary VS Secondary

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for Reflow
- Evenly Distribute Heat Sinking Parts
  - ✓ Minimizes Uneven Heating

Part Size VS Real-Estate

- Guess what…these requirements are Identical to the Screen Print Requirements
  - ✓ What's OK for Screen Print is OK for Reflow
- Component Size Variation Doesn't Support Soak
- Rule of thumb is simple…use the largest part you can for the available PCB real estate & support tooling

Company Profile

Production Analysis & Learning Services is a full turnkey Design, Manufacturing, Quality, and Maintenance Engineering Consulting Service with problem solving capabilities unique to the Electronic & Microelectronics Assembly Industry.

Company Mission

Production Analysis & Learning Services' Mission is to Provide Problem Solving Capabilities Unique to the Electronic & Microelectronics Assembly Industries with the Highest Possible Level of Design, Manufacturing, Quality, Marketing, Sales, and Technical Engineering Support.

Production Analysis & Learning Services, LLC

•Turnkey Engineering Service Provider for the Electronic & Microelectronics Assembly Industries • Electronic, Mechanical, & PCB Design Engineering • Manufacturing & Industrial Engineering Support • Quality Management & ISO Support • IPC Certification & Manufacturing Training Support • Manufacturing Equipment Maintenance Service Provider • Marketing & Web Site Development & Support • Database Development & Support • Networking Solutions & Analysis

Manufacturer Representative For:

Amtech Solder Products, Inc.

Advanced Metals Technology Inc. is the world leader in manufacturing electronic grade solder powders • Solder Creams • Bar Solder • Core Wire • Flux Righter • BGA Spheres • SMT Stencil Wipes • Bench top Hand Cleaner • Soft Wipes • Powder and Flux Stay Fresh Packaging

Envirosense Inc.

Providing Environmentally Safe Chemicals for the Cleaning of Precision and Electronic Assemblies • Neutralizer and De-fluxing Cleaner • Surface Tension Reducer and Foam Suppresser • Tin-Lead Solder and Metal Protection Additive • Tsunami Class III Cleaning Machine

GC Aero, Inc.

Unique Flexible Circuit Applications • Single Sided • Double Sided • Multi-Layer • Rigid-Flex • Test • Laser • Quick-Turn & Assembly

iFiber Optix, Inc.

Delivering tomorrows cable solutions today • Custom Designs • Fiber Optic Connectors • Fiber Optic Adaptors • Optical Transceivers• Cable Assemblies • Fiber Optic Splitters • Attenuators • Copper Assemblies • Distribution Enclosures Total • Coax Assemblies • 100% Tested • Cisco Approved Supplier • Private Labeling • 24 Hour Turn

Lewis and Clark

Matching Buyers and Sellers of Pre-owned Equipment • Asset management of excess systems • Plant Liquidations • Reconditioning • Field Service • Training and Assistance in Leasing and Financing • Complete examination • Performance testing • Analysis of repairs required • Total reconditioning

Mask Technology, Inc.

Service Company for the PCB, Assembly, & Microelectronics Industries • Solid Solder Deposit (SSD) - Macro Planar Deposits for Rigid or Flex Substrates • SSD Reflow Equipment - Reflow System using PPT™ Technology • Solder Spheres - Solder Spheres for BGA & Chip Scale Packages • Wafer Bumping - to .005" sphere size .008" Pitch

Production Analysis & Learning Services is capable of providing the following engineering support & services:

Electronic & Mechanical Engineering Design

- Adherence to MIL-STD, IPC, ISO, JDEC, & IEEE Specification
- Project Engineering Support
- Turn Key Design to Market Capability

Manufacturing & Industrial Engineering Support

- Supply Chain Management Support
- Lean Manufacturing Implementation and Support
- Manufacturing and Production Analysis
- Reflow Process Development
- uBGA process & rework development
- Work Instruction Development & Training
- Hazardous Materials Control & Training

Quality Engineering Support

- Cost of Quality Analysis
- ISO Certification Assistance Work Instruction Development
- ISO Training Program
- SPC Implementation and Training

Certification & Training Support

- Certified IPC (610, 600, & J-STD-001)
- Training and Testing ESD Training Program

Marketing

- Packaging Design
- Web Development Support