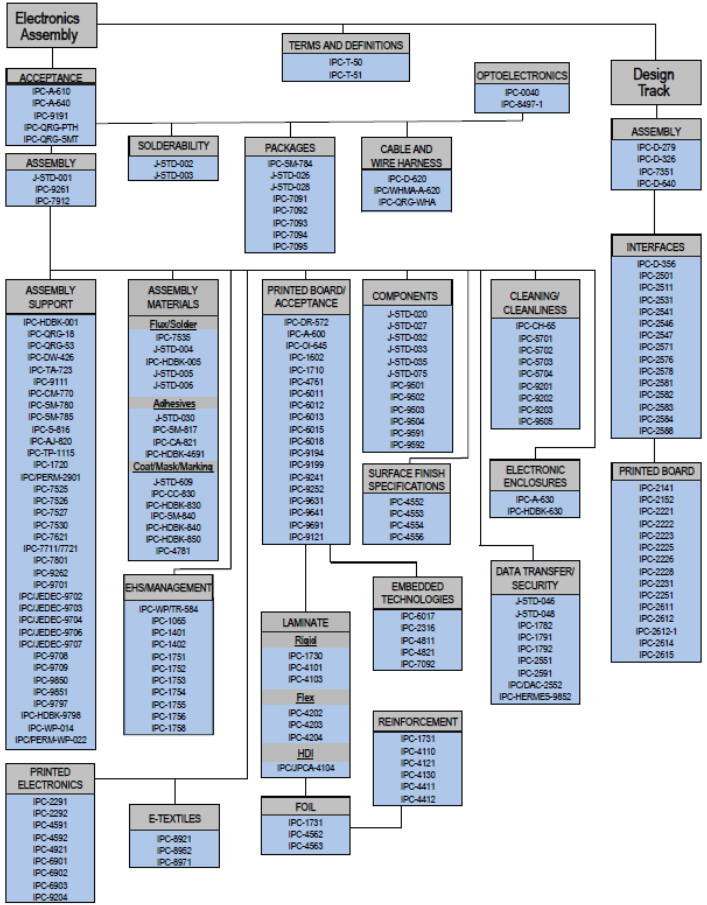
IPC ERFA-møde, torsdag den 21. november 2024

NPD/NPI – Hvad kendetegner et godt design?

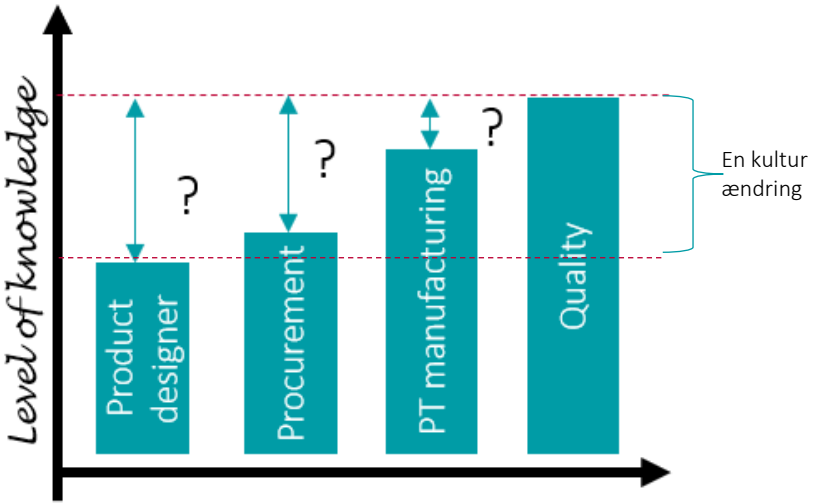
Forståelse og inkorporering af IPC-standarder i produktevaluering

Joseph Vella Ottosen – Kamstrup
Tony Mathiesen – Danfoss Drives

IPC Standards Tree



Niveauet af brug og viden om standarderne adskiller os som individer og som afdelinger. Denne bevidsthed kan gavne os alle, når vi diskuterer forventninger, kvalitet samt produktion…. *med en knivspids* af vilje til at lytte og lære fra hinanden….



Ambition og formål

Når vi designer og udvikler nye produkter

Omkostninger er konge
- Udvikling
- Dele og materialer
- Processer og produktion

Kvalitet er grundlæggende
- Pålidelighed afspejler omdømme
- Levetid forventes

Forsyningskæden
- Indgående logistik
- Udgående logistik

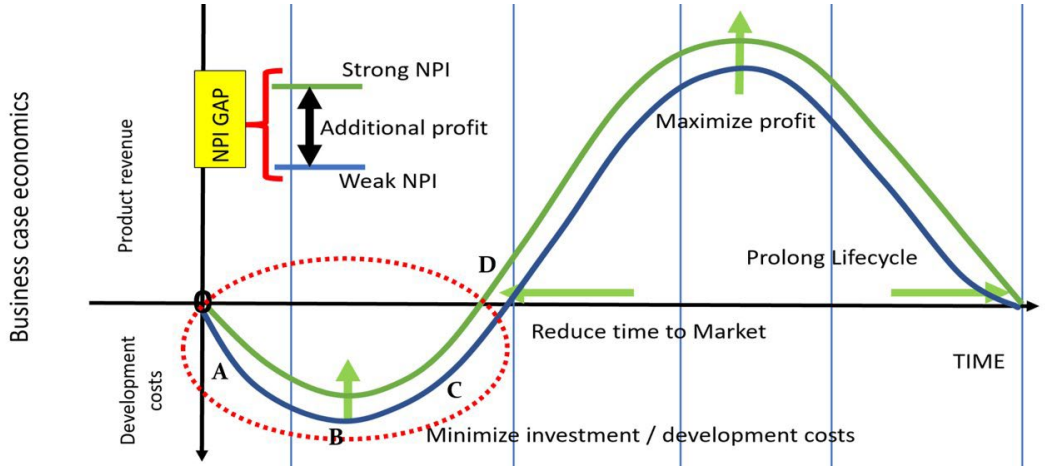
Standardisering
- Design for X
- Design reviews
- Concurrent engineering
- Fremstilling af prototyper
- Evaluering af prototype

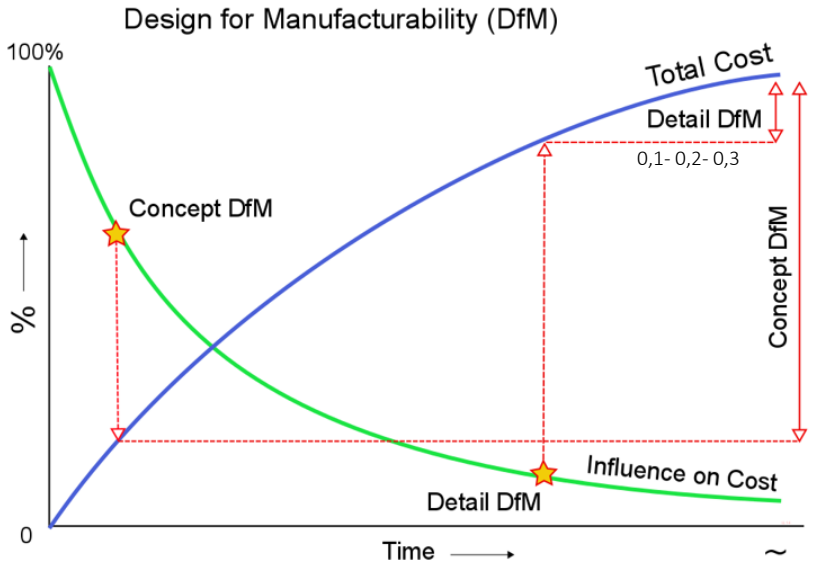
$$$

Mindre er bedst.....

- Udvikling
- Stykliste/indkøb
- Standard dele
- Varianter
- Processer
- Håndtering
- Kontrol

Hvor og hvordan kan IPC-standarder hjælpe?



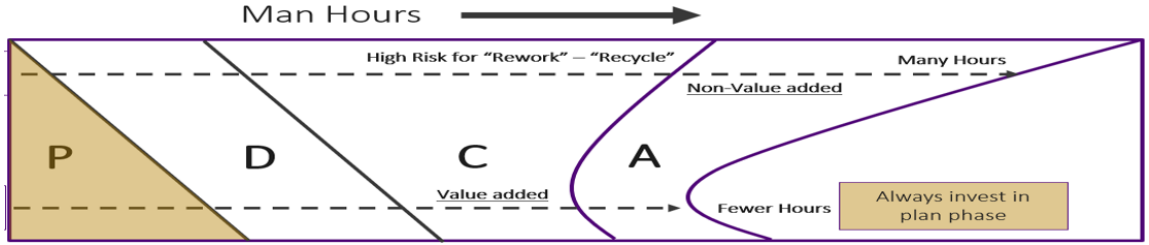


Grafen illustrerer fordelen ved at lave DfM som koncept tidligt i NPI-projekter, og hvordan det vil have en større indvirkning på omkostningsbesparelser, end hvis der foretages en detaljeret DfM-gennemgang i en senere tidsramme i projektet.

IPC-2231A **DFX Guideline**
IPC-2222 **DfM PCB**
IPC-A-630 Acceptability Standard for Manufacture, Inspection and Testing of Electronic Enclosures

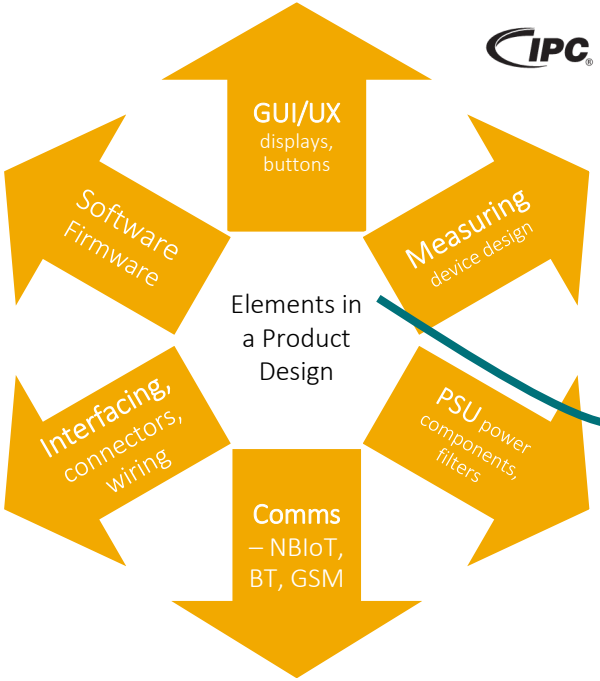
?

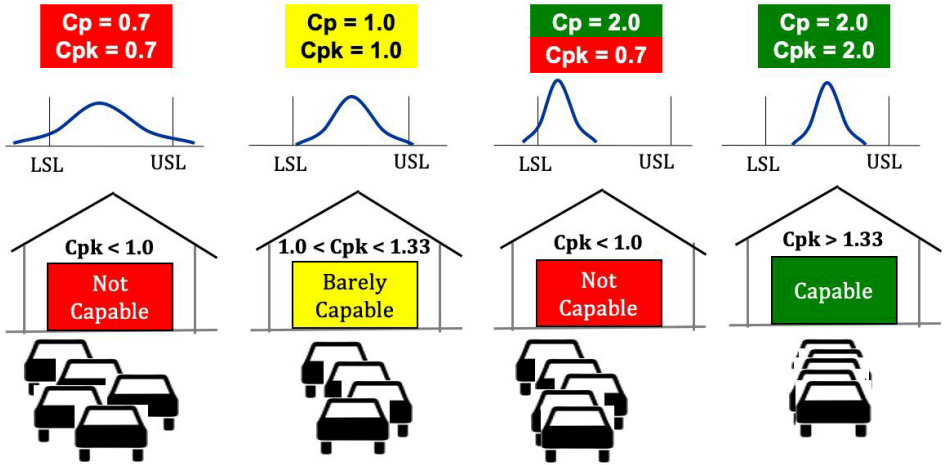
As "one" Supply Chain we need to deliver a product in accordance with what is specified as the finished product. We must look at the product as one concept to have a more influential impact.



Understanding *process capability*

– The Good driver vs. The Unsteady driver.





Produktdesign kan påvirke procesvinduet i et bestemt produktionsområde.

– standarderne kan hjælpe os etablere fælles grundlag mellem produkterne og de forskellige designelementer ift. vores kapabiliteter.

Understanding *process capability*

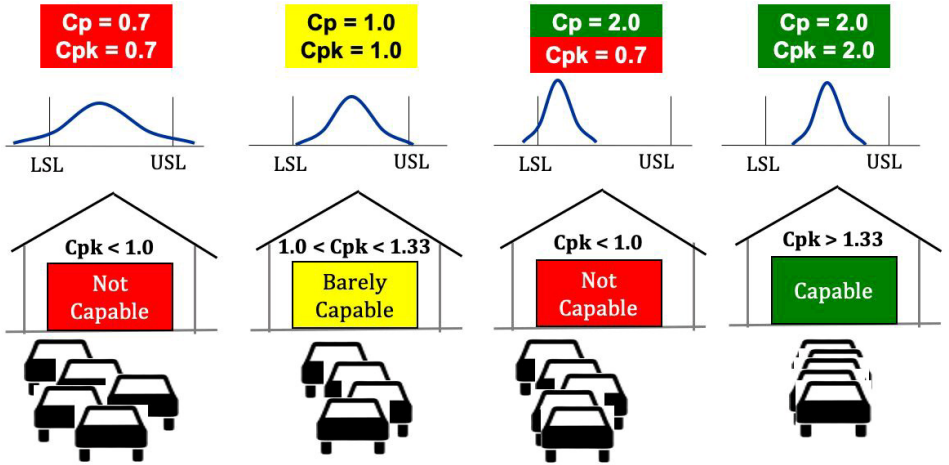
– The Good driver vs. The Unsteady driver.

Standarder fastsætte klare kriterier for, hvad der udgør et acceptabelt produkt.

- IPC-A-610
- IPC-J-STD-001
- IPC-6012…
- IPC-7351
- IPC-2221/22

SPC-diagrammer kan derefter overvåge disse aspekter for at opretholde overensstemmelse med standarden.

…..e.g.
IPC-9191 – Guidelines for implementing SPC
IPC-9701 – Performance test SMT
IPC-TM-650 series



The driver is unsteady. The car often scrapes the walls. You will produce defect parts unless process width is reduced and process is centered.

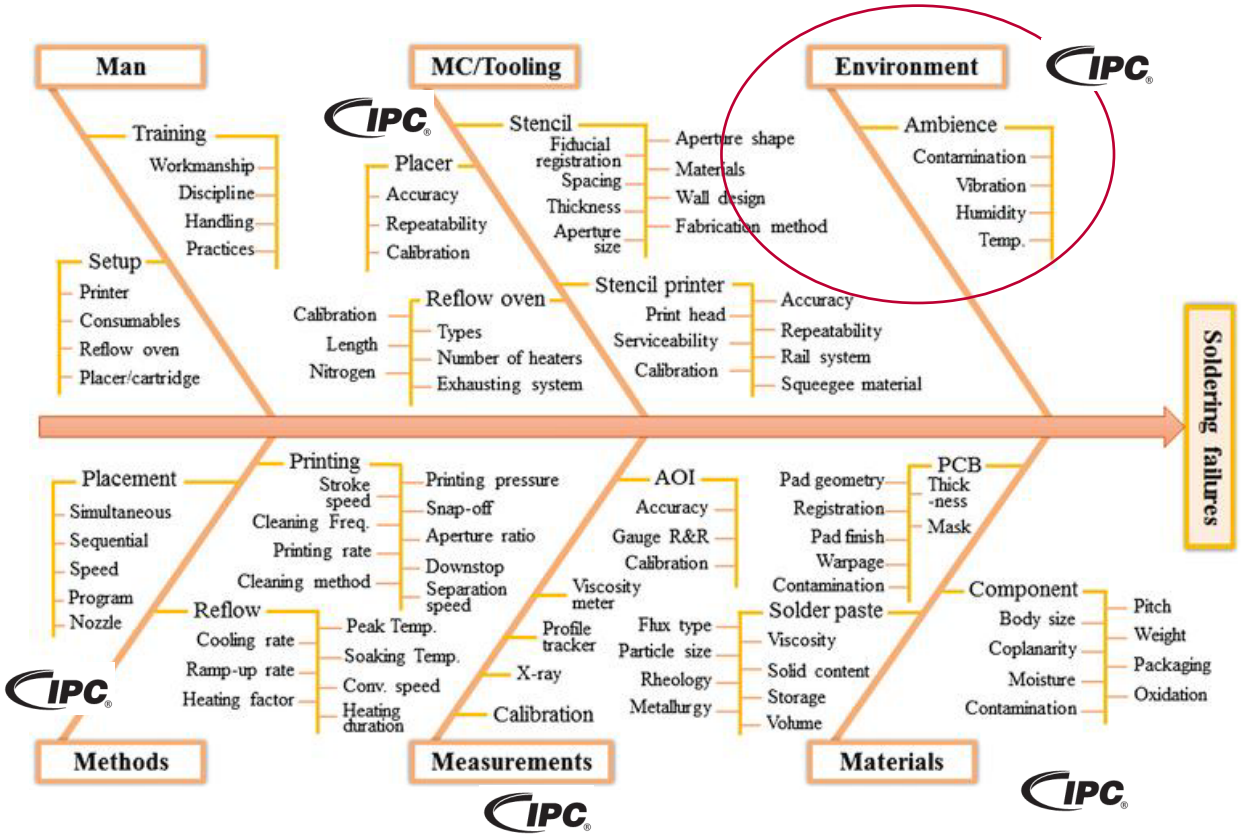
The driver is still unsteady but better than before. He often comes too close to the walls. You are likely to have a defect unless the process width is reduced.

The driver is unable to center the car. But he's consistent – always too close to one side. You are likely to have a defect, unless the process is re-centered.

The driver always parks successfully without scraping the sides. The process is centered, and with a narrow distribution. You are unlikely to have defects even if the process shifts slightly to either side.

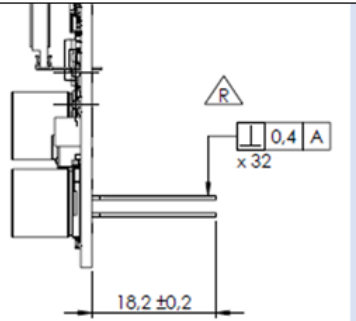
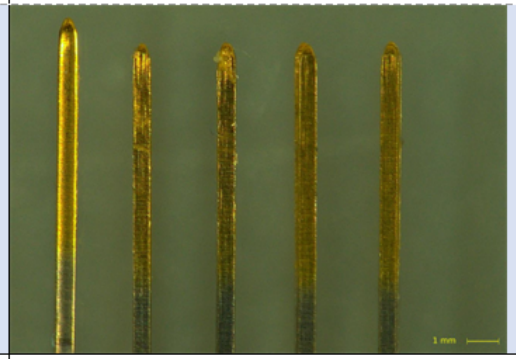
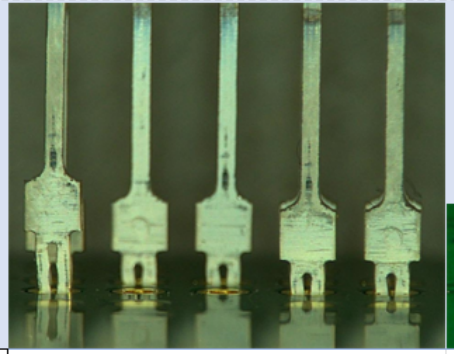
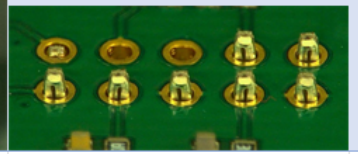
Understanding how to correct *process capability*

– The Fishbone diagram – Cause & Effect – FMEA tools

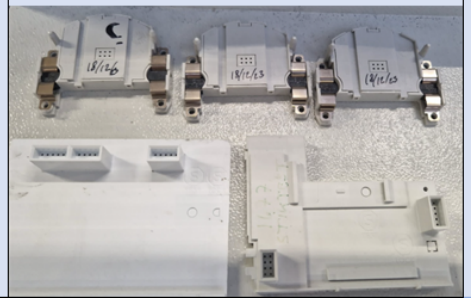
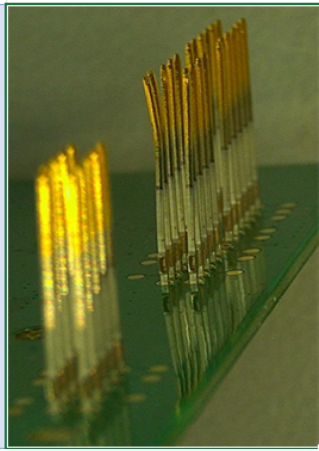


Læringen og erfaringen fra at arbejde med procesevne kan være yderst værdifuld, når man analyserer nye designs, der skal passe ind i det eksisterende produktionssetup.
Disse fakta og tal udgør en væsentlig del af inputkvaliteten til "**Produktionskrav**" baseret på IPC-standarder og metoder.

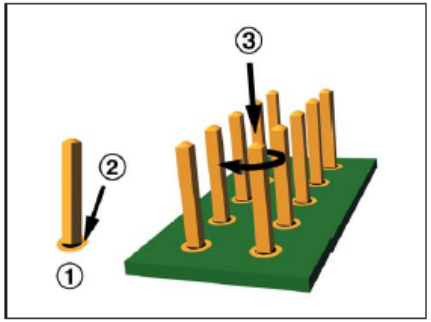
Press fit proces variationer

140.06 140.07	According to the 5538xxxx drawing associated to all corresponding products which require the insertion of press fit pins, the height is specified and needs to be included in the inspection during the process.			
140.06 140.07 140.08	Pin height needs to be checked according to the product requirements. In this failure mode pin height can be influenced by parameters in material or machine alignment.			

Kamstrup standard

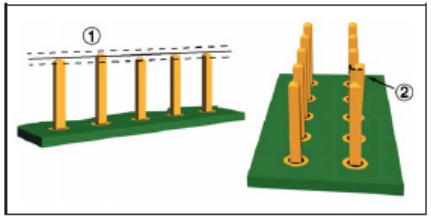
PFMEA item nr	Recommended Actions	Illustration	Illustration2	Illustration3	Illustration4
140.01 140.02 140.04	The corresponding plastic parts used for product compliance for pin straightness are not registered as controlling tools. No Instructions are written and there is no management renewal system established to update these tools when necessary or after a time based wear and tear period. It is also feasable to insert bent pins into the the plastic parts, as can be seen in the pictures on the far right. Would this be within specified criteria?				

4.3.2 Konnektorpins – Pres



Figur 4-31
1. Ingen synlige skader
2. Land
3. Ingen synlig vridning

Ønskelig – I
- Pin termine
- Pin højden

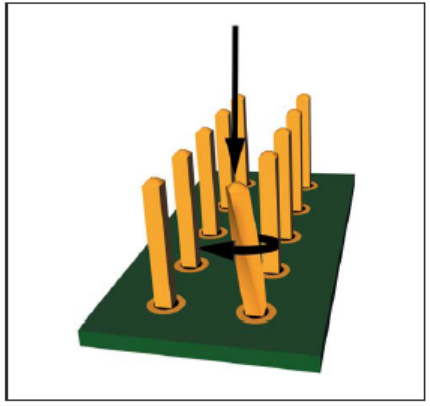


Figur 4-32
1. Højdetolerance på pin terminering
2. Mindre end 50 % af pin termineringens tykkelse

Acceptabel
- Pin termine terminering

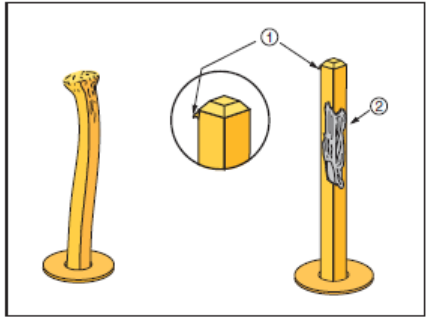
Bemærk: N pinkonnekto Konnektorpi en god elek

9.10 Press Fit Pins



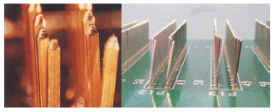
Figur 9-39

Defekt – Klasse
- Beskadiget pin
  - Vredet.
  - Champignon
  - Bøjet mere e
  - Blotlagt basis
  - Grat.



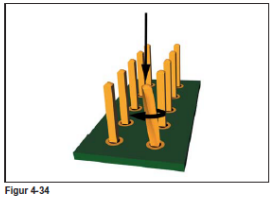
Figur 9-40
1. Grat
2. Manglende plettering

4.3.2 Konnektorpins – Press Fit Pins (fortsat)

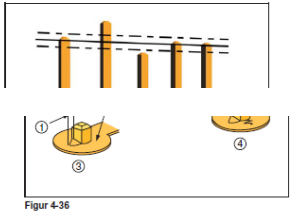


Figur 4-33

Defekt – Klasse 1,2,3
- Pin terminering er bøjet ud fra rækken – er bøjet mere end 50 % af pin termineringens tykkelse, se Figur 4-33.
- Pin terminering er synligt vredet, se Figur 4-34.
- Pin termineringens højde er uden for den specificerede tolerance, se figur 4-35.



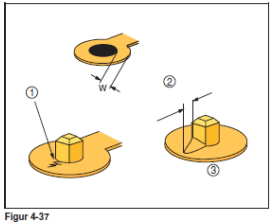
Figur 4-34



Figur 4-36
1. Land løftet 75 % eller mindre af annularringens bredde
2. Land med lederbane
3. Land ikke brudt
4. Land uden lederbane er løftet og brudt, men har stadig sikker befæstelse (ikke funktionelt loddeland)

Acceptabel – Klasse 2
- Ingen visuelle tegn på løftet land på montagesiden.

Acceptabel – Klasse 3
- Ingen løftede eller revnede annularringe.



Figur 4-37
1. Land brudt

Defekt – Klasse 1,2
- Enhver løftet annularring, hvor mere end 75 % af bredden (W) er løftet, på gennembrydningssiden.

Defekt – Klasse 2
- Ethvert løftet land på monteringssiden.

Defekt – Klasse 3
- Enhver løftet eller brudt annularring med press fit pins.

Bemærk: For yderligere information, se 10.3.2.

IPC-9797 – Press-Fit Standard

IPC-A-610

The link between Product Development and Operations

Team consist of:

- Design for manufacturing specialists
- Manufacturing process experts

Iterative Design Reviews: Conduct design reviews at key stages of the project, from early concept development to final handover to operations, ensuring a continuous and iterative process.

Collaborative Optimization: Work closely with designers to challenge and refine the design or manufacturing process when necessary.

High level design inputs

- Avoid unnecessary processes if following general design guidelines.
- Step up in panel to better utilize SMT placement process.
- Split between top & bottom side component count.
- Product family component portfolio. Avoid unnecessary large component setups and mixed shape size.

Early involvement is key in concept phase, as mechanical board contrains and product build up is not fixed yet.

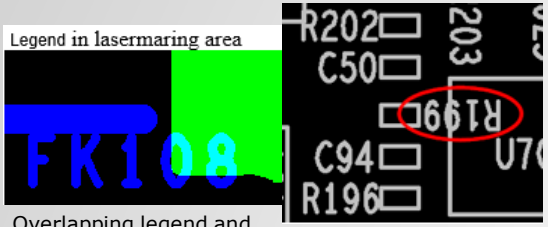
Remove edge PCB

PPL – Prefered Parts List

Internal Design Guidelines

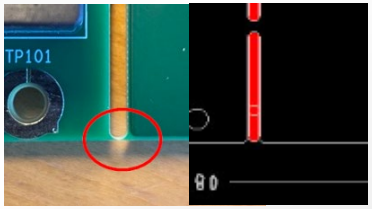
Data for automatic program generation for auto-THT insertion and robot assembly

Review findings

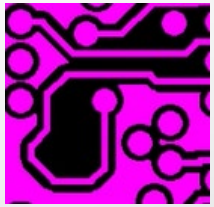


Overlapping legend and orientation for better readability

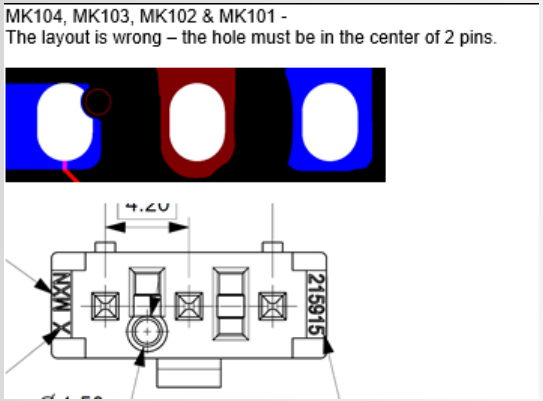
Component too loose in tape, pickup problems
Change design of tape pocket



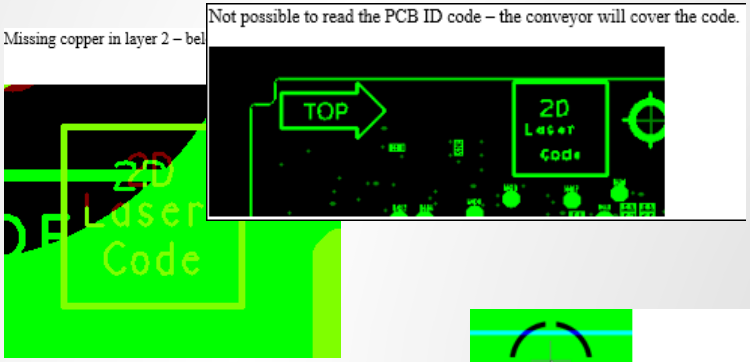
Cannot be removed in depaneling process. Difficult to see from design files.



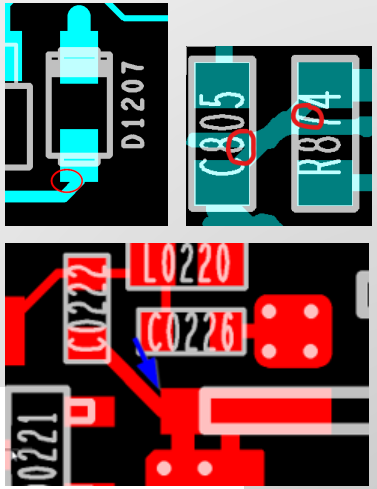
Odd trace route



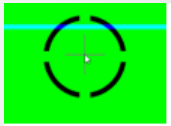
Library data error



Add copper in layer 2 for better contrast / readability



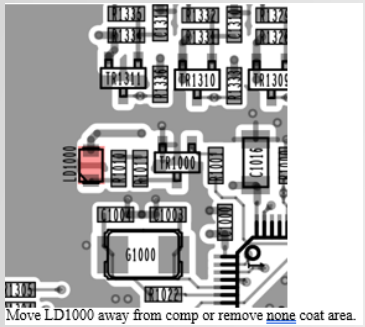
Acid traps



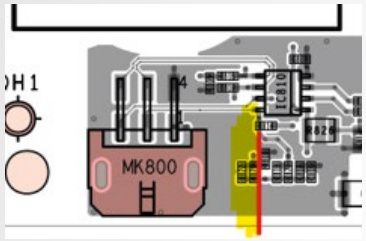
Thermal reliefs :
It is recommended to have spokes longer than wide. E.g. 0.5 wide 0.6 mm long
See PCB layout guideline for lead free tin wave soldering id: 00719858

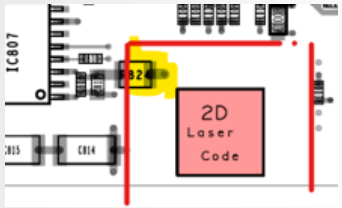
Conformal Coating

For connectors and other non-coating areas keep minimum distance if components should be coated.

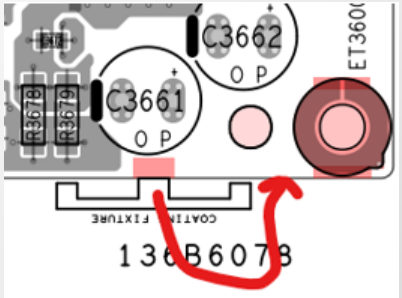


Move LD1000 away from comp or remove none coat area.

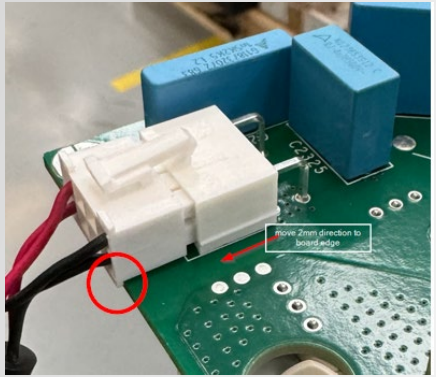




Placement of fixture clips

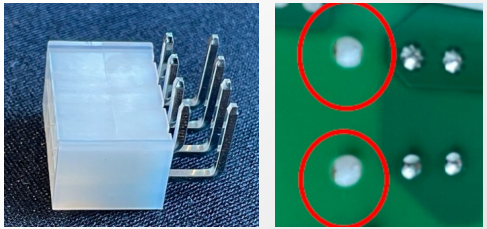


Observations found during early NPI phase, mechanical assembly



Observation found: Connector could not snap in

Solution: Move connector 2mm towards edge



Component has no snap-in pins
(needed to hold the component in position during wave soldering, alternative tool is required to keep in position)

Solution: Added snap-in pins

DPAK stencil design optimization

Too much Solder Paste can be the Reason for ”floating” components.

- The component will ride on a wave – because liquid solder will try to create a sphere shape – so it can “tilt/float” around

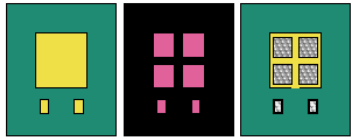


Figure 2.19: Modifications for D-PAK components

With the base pads of D-PAK components it is often necessary to reduce the outside dimensions of this pad and then window the resulting aperture to offer the solder paste volume required by the component without causing unnecessary component float or paste starvation.

Cooperation between Product Development and Supply Chain to test different sized “windowpane” in stencil design.

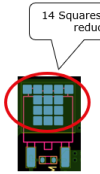
IPC-7525 Stencil Design Guideline

Stencil Design (3001907)

- PCB 134B6060 iss 4 (PC68770) – 127µ.

Part used –
- 612L0451
- 612L0452

Both from STM



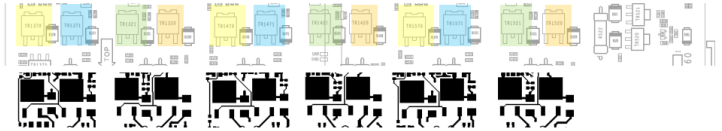
NO CHANGE

TR1370; 1470, 1570
- Solder Paste Area: **1,2x1,2x14= 20,2 mm2**
- Component Cooling Area: 5,0x5,4= ~27,0 mm2
- => Area Reduction ~25%
- Stencil thickness 125µ => 3,375mm3

TR1371; 1471, 1571
- Solder Paste Area: **1,0x1,0x14=14,0 mm2**
- Component Cooling Area: 5,0x5,4= ~27,0 mm2
- => Area Reduction ~48%

TR1321; 1421, 1521
- Solder Paste Area: **0,9x0,9x14=11,34 mm2**
- Component Cooling Area: 5,0x5,4= ~27,0 mm2
- => Area Reduction ~58%

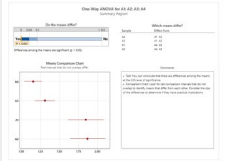
TR1320; 1420, 1520
- Solder Paste Area: **0,8x0,8x14=8,96 mm2**
- Component Cooling Area: 5,0x5,4= ~27,0 mm2
- => Area Reduction ~67%



Result - Offset

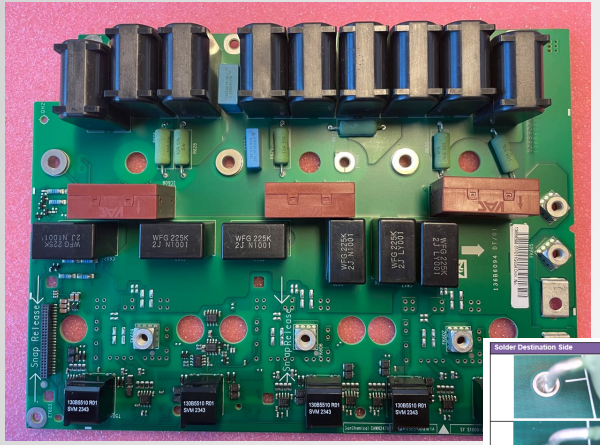
Voids

Offset - Minitab Results

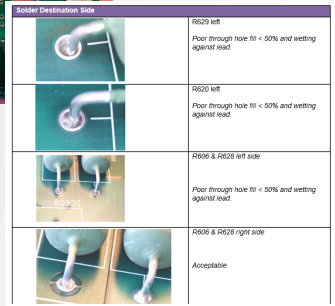


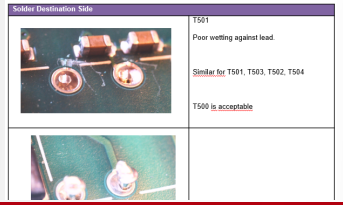
Voids - Minitab Results

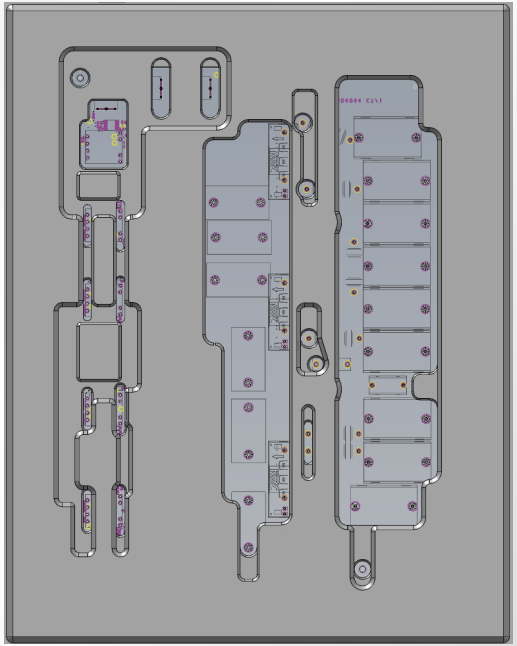
Solder evaluation during first engineering run

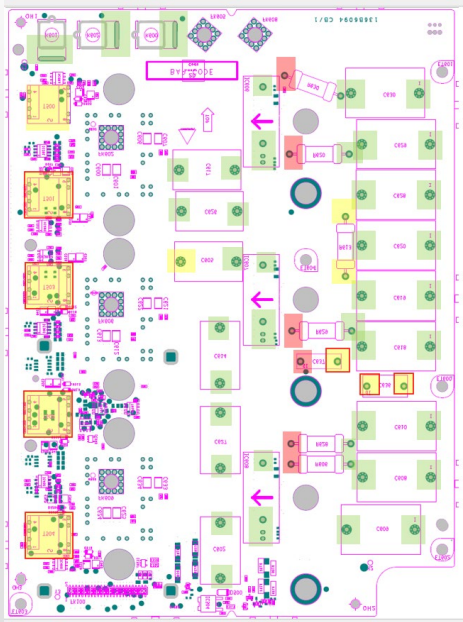


6-layers with 100µm cobber
Pressfit connectors and SMT on bottom side



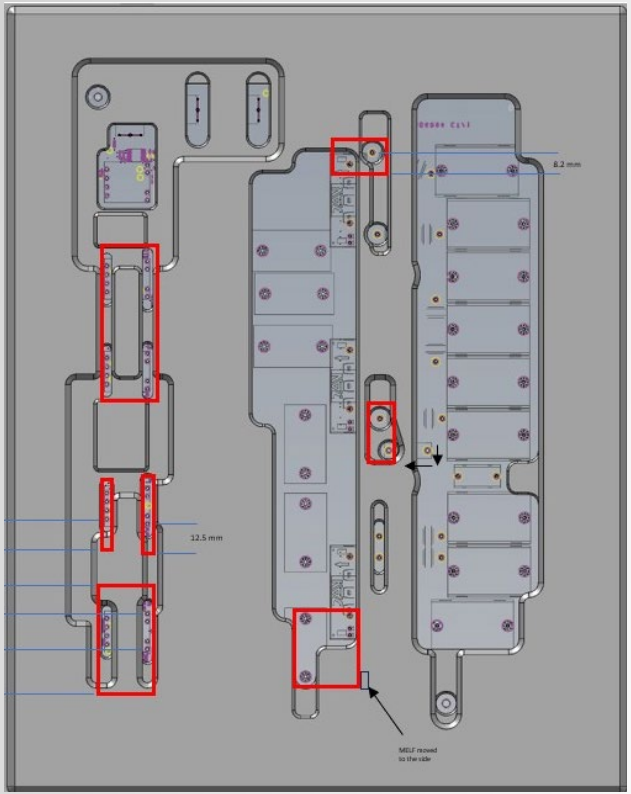


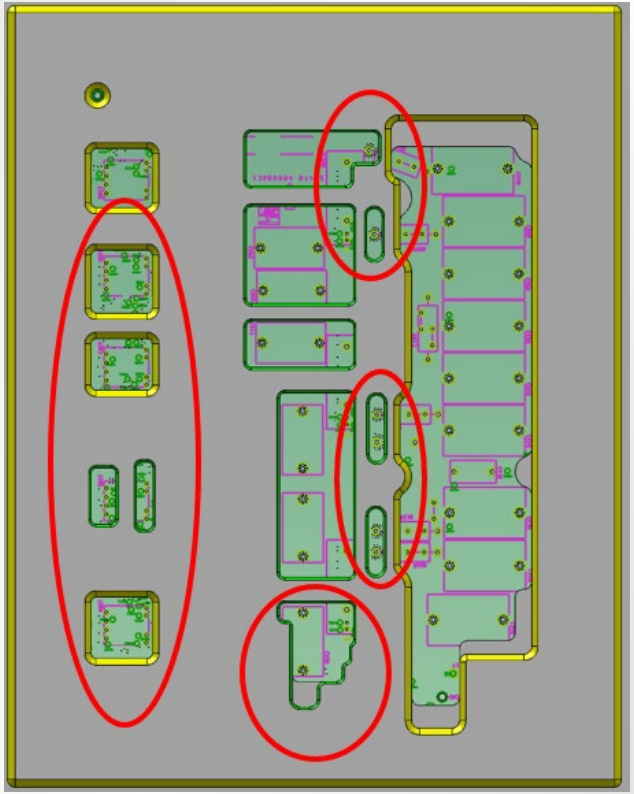




IPC-A-610 & IPC-J-STD-001

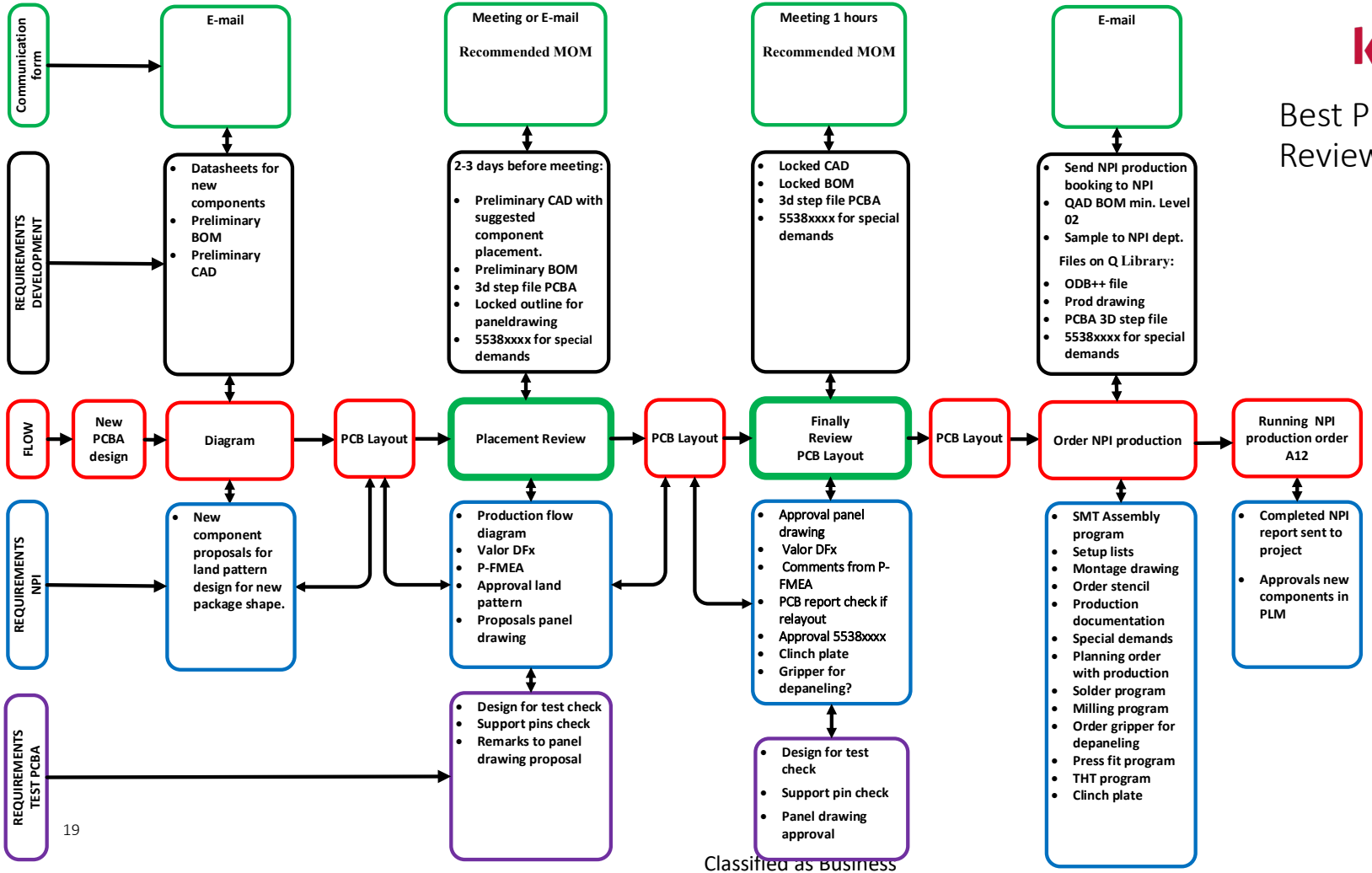
Solder evaluation





In cooperation with designers and process experts several changes to both design and solder frame. To achieve good solder quality.

Best Practice Design Review PCBA



ValorMSS Process Preparation

PCB CAD DATA

BOM DATA

”One point” data handling

DFM / DFA analysis

Data Preparation / Process Equipment.

DfM rapport

Ekstern VPL Database
Adgang til mere end
35 mio. komponent
shapes

Output til proces:
SPI AOI SMT THT 3D-AOI, stencil...

kamstrup

PCB number & revision 5536XXXX_YY	55362183_A1
Date of report	08-11-2023
Prepared by	CLEN
Reposnsible CAD designer	BRH
Report Type	Final DFM review
Report Status	**Approved**

kamstrup

Conclusion

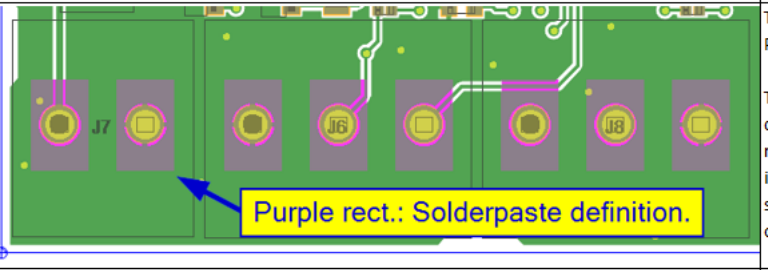
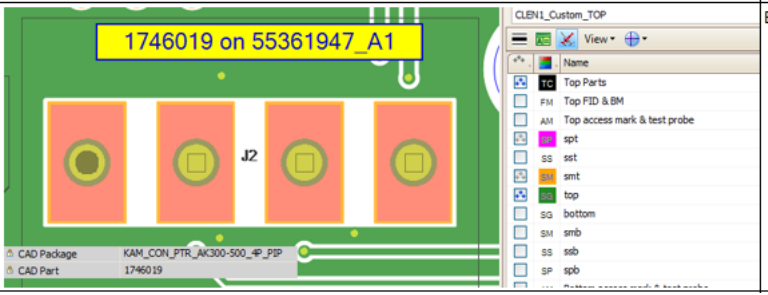
DFM issues category count					DFM issues fixed				
A	B	C	D	E	A	B	C	D	E
11	5	2	1	0	11	4	1	1	0

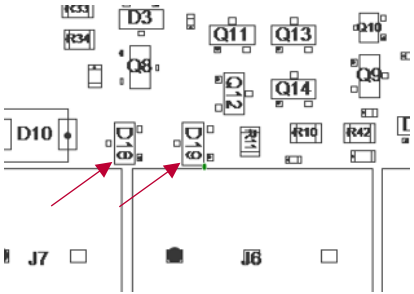
DFM issues rejected					DFM issues N/A				
A	B	C	D	E	A	B	C	D	E
0	1	1	0	0	0	0	0	0	0

Count	"fixed"
19	17
"rejected"	"N/A"
2	0

HOME
Link to Valor Tab

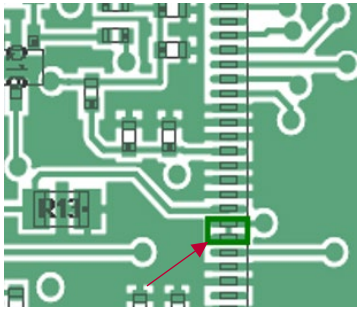
Template rev.: A29

Category	Criteria	Comments	Severity	Agreed Actions	Status
		The J6, J7 and J8 connectors are defined as Pin In Paste connectors. To ensure proper flow of the liquid solder during reflow soldering, we normally have a rectangular soldermask and copper opening in the same size and shape as the solderpaste definition shown in the image on the left.	D		Fixed
		Example of how it is done on 55361947_A1...			

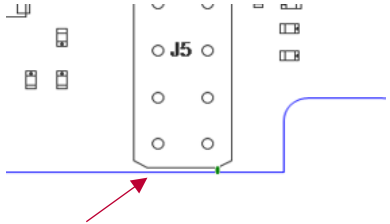


D18 + D19: SOT23 is located very close to the J7 connector. Measured distance is 250 microns which is extremely close to the minimum allow distance.

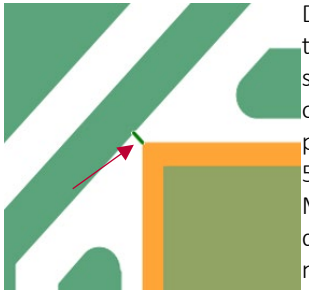
It is recommended to increase the distance to 400 microns.



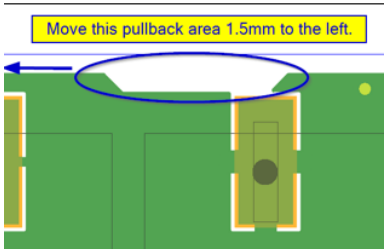
U1: TQFP, gullwing: It is not allowed to connect the two pads in this way



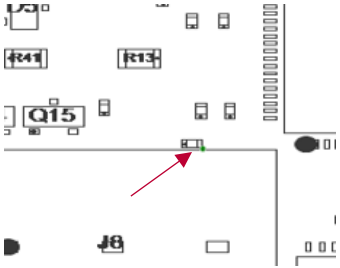
J5: Double row header. Too close to the PCB edge. Measured distance is 250 microns which is below the minimum required distance of 1000 microns.



Distance between a trace and the soldermask opening of one of the Q15 pads is measured to 58.6 microns. Min. required distance is 75 microns.



Move the pullback area 1.5mm to the left, to get clear of the SMD pad of C28.

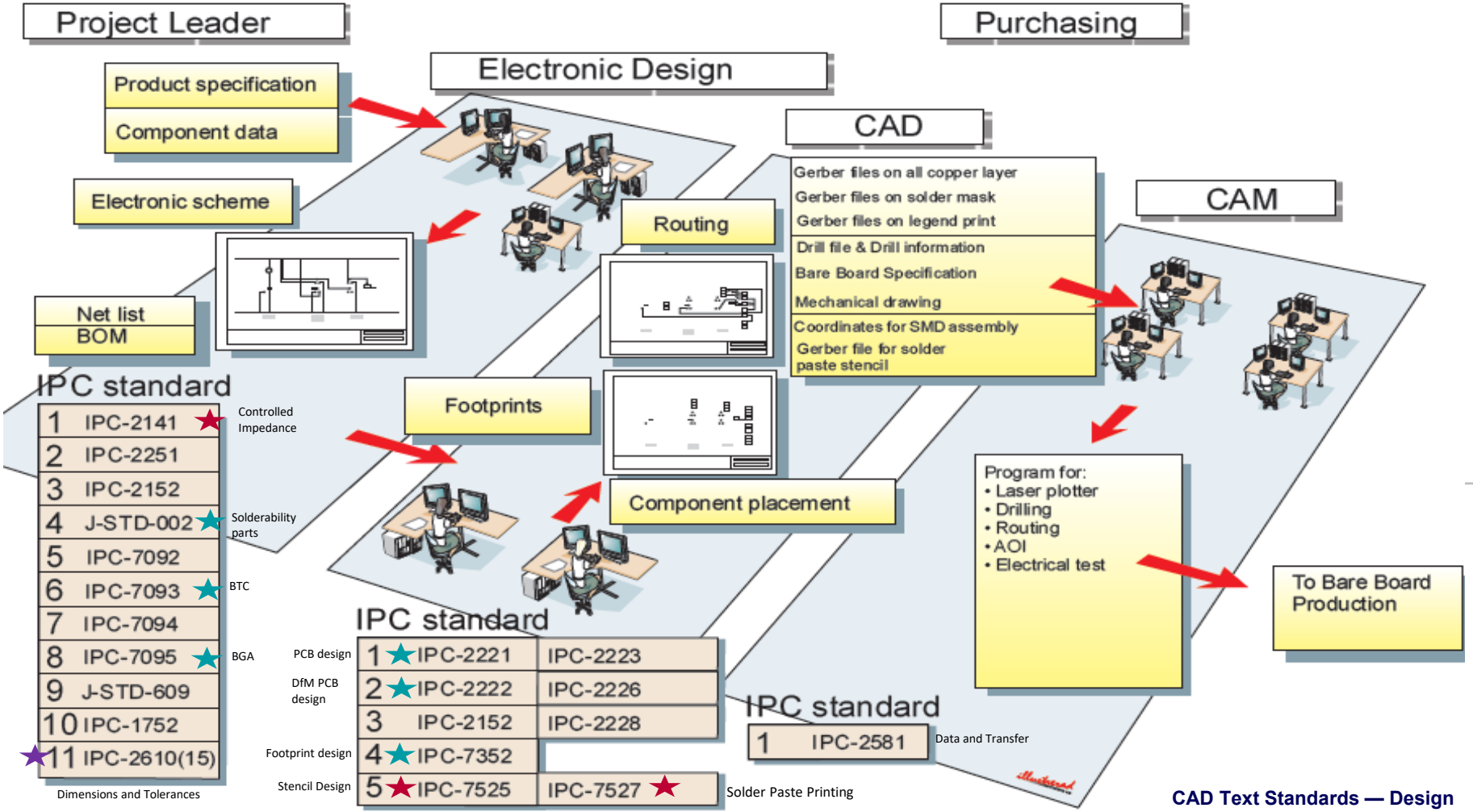


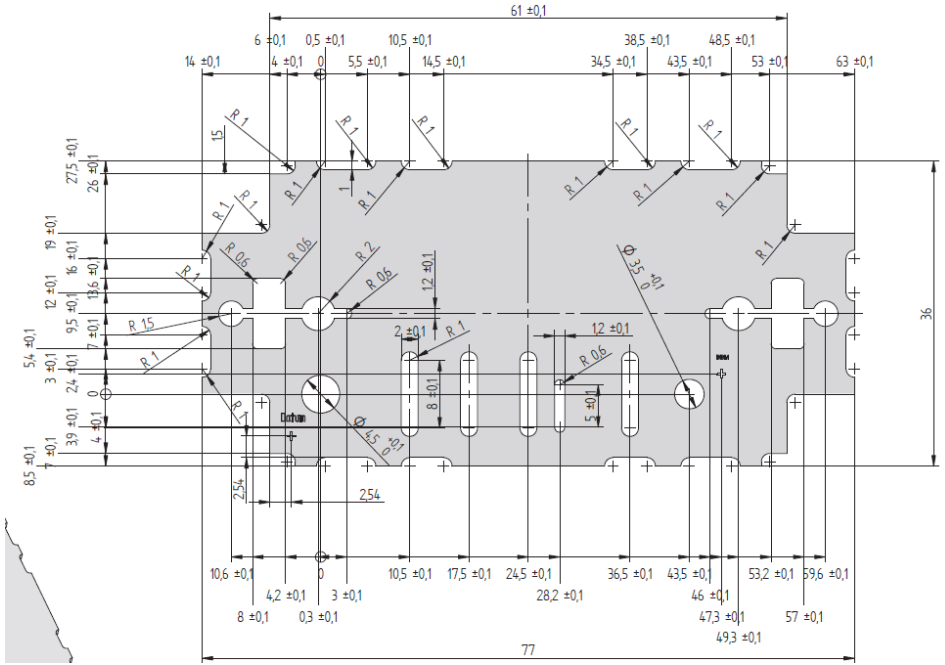
R70 must be moved further upwards by at least 150 microns to restore a safe distance to the J8 connector.

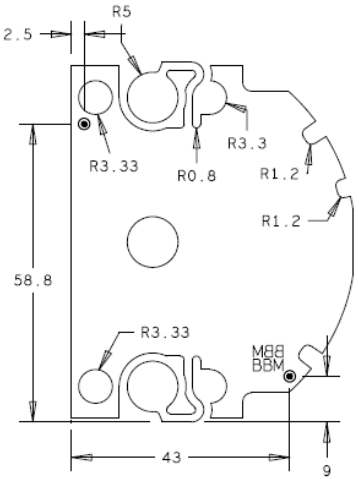
DfM reporting

(5535)	Design		Review	Review	Report	Reported					Solved					
Design Name	Revision	INIT	Date	Type	Link	A	B	C	D	E	A2	B2	C2	D2	E2	Cost Saved
2169	A1	FLAN	18-12-2023	Final	DFM review	1	2	1	0	0	1	2	0	0	0	50000
2188	A1	FLAN	02-02-2024	Placement	DFM review	1	0	2	3	0	1	0	2	3	0	57000
2188	A1	FLAN	07-02-2024	Final	DFM review	1	6	11	0	0	1	6	10	0	0	85000
2169	B1	FLAN	19-03-2024	Final	DFM	0	5	1	1	1	0	5	1	1	0	33000
2159	B1	FLAN	18-04-2024	Final	DFM	0	1	6	0	0	0	1	3	0	0	21000
2188	B1	CLEN	30-04-2024	Placement	DFM review	0	2	3	0	0						0
2179	A1	FLAN	08-05-2024	Final	DFM review	3	6	11	0	1	3	3	8	0	1	81000
2093	C1	FLAN	22-05-2024	Final	DFM review	0	1	3	0	0	0	1	2	0	0	19000
2115	B1	CLEN	19-06-2024	Placement	DFM review	0	4	7	1	0						0
2205	A1	CLEN	26-06-2024	Final	DFM review	0	0	10	0	0	0	0	10	0	0	20000
2065	B1	CLEN	26-06-2024	Placement	DFM review	1	5	7	0	1	1	1	7	0	0	64000
2169	C1	FLAN	27-06-2024	Final	DFM review	2	3	4	1	1	2	3	3	1	1	72000
2201	A1	FLAN	19-06-2024	Final	DFM review	0	1	5	0	0	0	0	5	0	0	10000
2209	A1	CLEN	10-07-2024	Final	DFM review	1	1	24	3	0	1	1	24	3	0	101000
2204	A1	CLEN	10-06-2024	Final	DFM review	0	2	0	1	0	0	2	0	0	0	15000
1951	C1	CLEN	07-05-2024	Additional	DFM review	0	0	0	0	0	0	0	0	0	0	0
2157	B1	CLEN	09-01-2024	Placement	DFM review	0	3	2	0	0	0	1	2	0	0	19000
2188	B2	CLEN	15-08-2024	Final	DFM review	0	1	0	0	0	0	1	0	0	0	15000
2202	A1	FLAN	15-08-2024	Final	DFM review	0	3	5	0	1	0	2	5	0	1	25000
2212	A1	FLAN	29-08-2024	Placement	DFM review	0	6	3	0	0	0	6	3	0	0	36000
2212	A1	FLAN	04-09-2024	Final	DFM review	0	2	2	0	1	0	2	2	0	0	19000
2157	B3	FLAN	18-09-2024	Final	DFM review	2	2	5	2	0	2	2	5	0	0	60000
2123	D2	FLAN	19-09-2024	Final	DFM review	2	2	5	1	0	2	2	5	1	0	61000
2212	A1	FLAN	30-09-2024	Additional	N/A	0	0	0	0	0	0	0	0	0	0	0
2211	A1	FLAN	07-10-2024	Placement												0
2208	B1	CLEN	24-09-2024	Placement	DFM review	0	0	21	6	0	0	0	12	0	0	24000
2209	B1	CLEN	25-10-2024	Final	DFM review	0	0	10	0	0	0	0	10	0	0	20000
2211	A1	FLAN	15-10-2024	Final	DFM review	0	1	8	0	0	0	1	5	0	0	25000
2165	B1	FLAN	04-10-2024	Final	N/A	0	0	1	0	0	0	0	1	0	0	2000

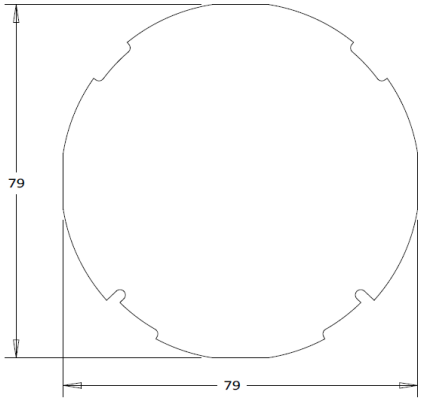
Year	Sum of Cost Saved
2020	1.496.000
2021	1.840.000
2022	1.696.000
2023	2.292.000
2024	930.000
Grand Total	8.254.000


Project Leader
Purchasing
Electronic Design
Product specification
Component data
Electronic scheme
Net list
BOM
Routing
CAD
Gerber files on all copper layer
Gerber files on solder mask
Gerber files on legend print
Drill file & Drill information
Bare Board Specification
Mechanical drawing
Coordinates for SMD assembly
Gerber file for solder paste stencil
CAM
Footprints
Component placement
Program for:
• Laser plotter
• Drilling
• Routing
• AOI
• Electrical test
To Bare Board Production
IPC standard
1 IPC-2141 Controlled Impedance
2 IPC-2251
3 IPC-2152
4 J-STD-002 Solderability parts
5 IPC-7092
6 IPC-7093 BTC
7 IPC-7094
8 IPC-7095 BGA
9 J-STD-609
10 IPC-1752
11 IPC-2610(15) Dimensions and Tolerances
IPC standard
PCB design 1 IPC-2221 IPC-2223
DfM PCB design 2 IPC-2222 IPC-2226
3 IPC-2152 IPC-2228
Footprint design 4 IPC-7352
Stencil Design 5 IPC-7525 IPC-7527 Solder Paste Printing
IPC standard
1 IPC-2581 Data and Transfer



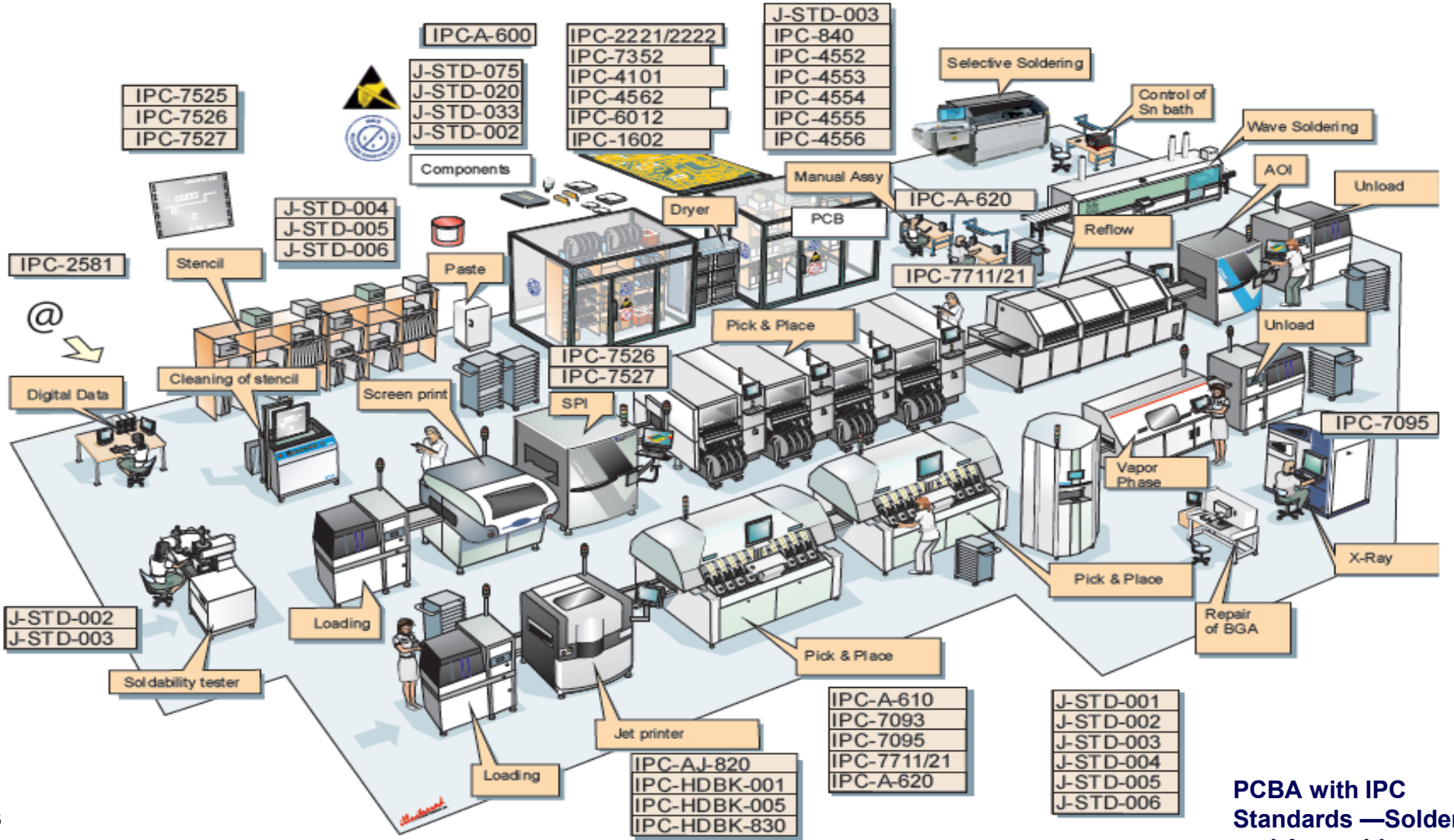


Film Name
BREAKOUT AND MARKINGS

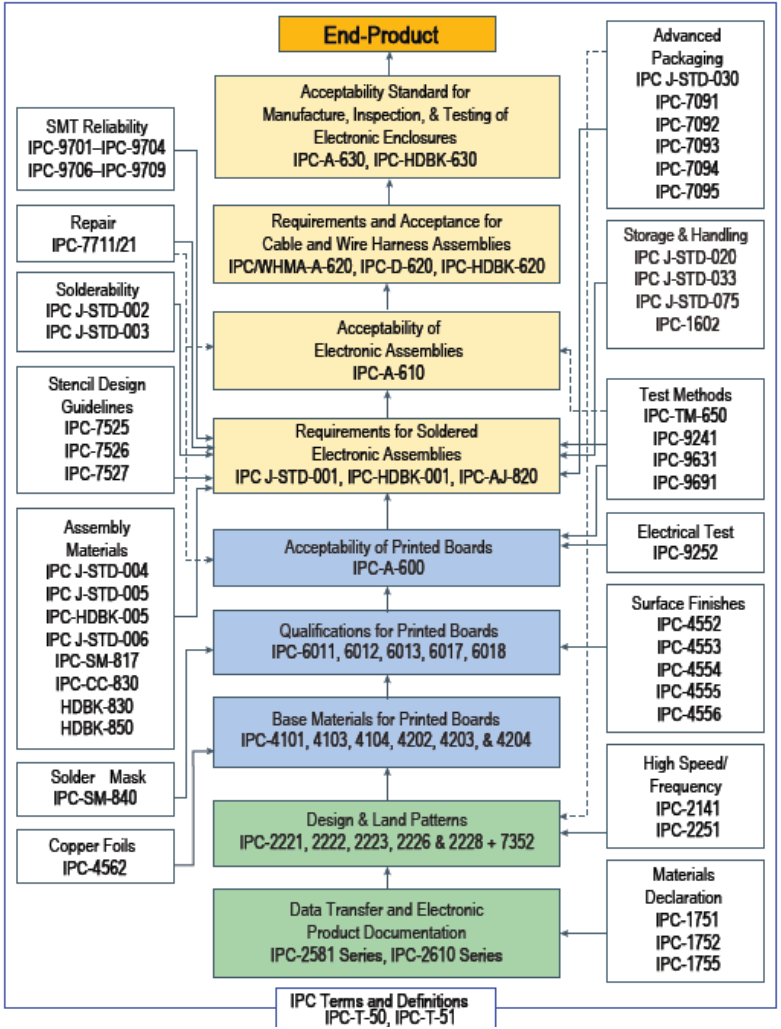


Film Name
BOARD DIMENSIONS

IPC-2615
Printed Board Dimensions and Tolerances



PCBA with IPC Standards —Soldering and Assembly



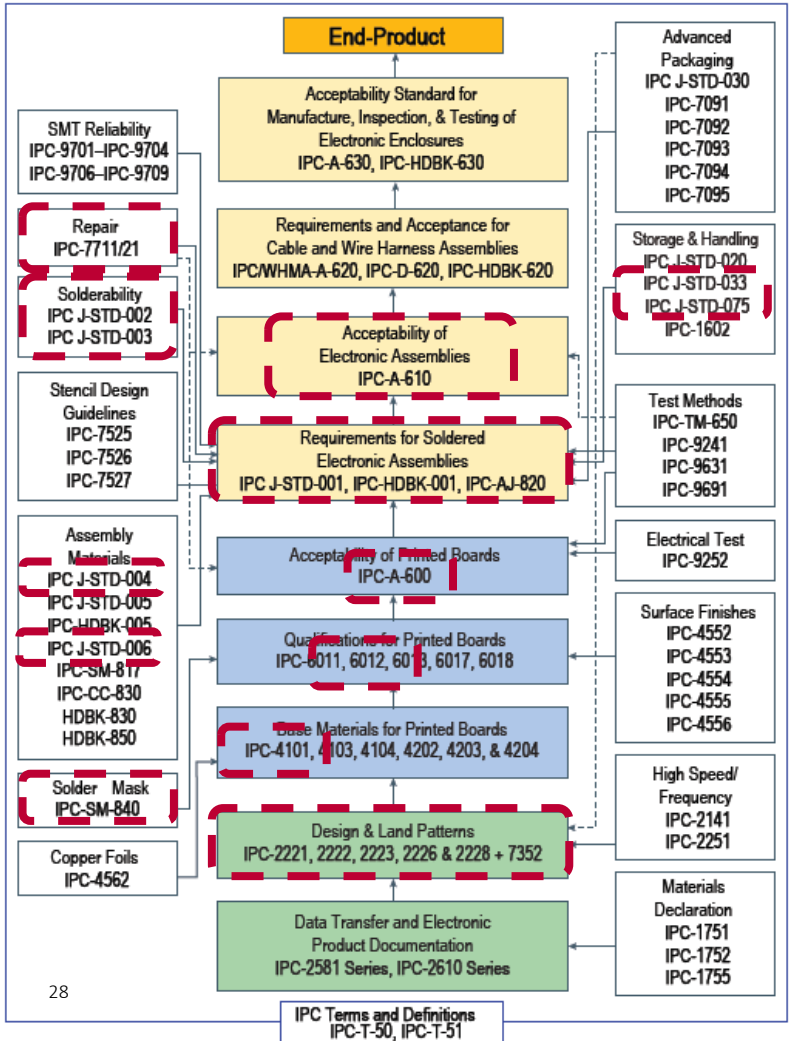
kamstrup

?

Requirements for Soldered Electronic Assemblies
IPC J-STD-001, IPC-HDBK-001, IPC-AJ-820

?

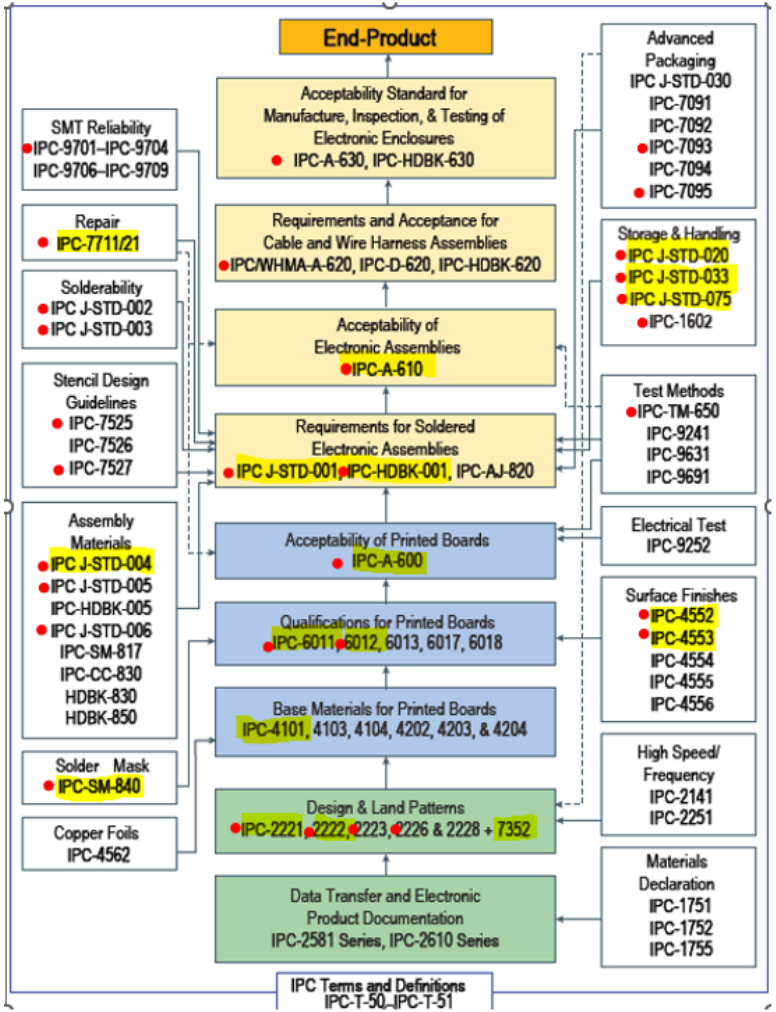
Design & Land Patterns
IPC-2221, 2222, 2223, 2226 & 2228 + 7352



IPC Standards……. for rework

IPC-A-610 – Workmanship standard for electronic assemblies
IPC J-STD-001 – Requirements for Soldered Electronic assemblies
IPC-7711/21 - Rework, modification & repair of electronic assemblies
IPC J-STD-033 - Control of moisture-sensitive components
IPC J-STD-075 – Control of non-semiconductor components PSL+MSL
IPC SM-840 – Soldermask spec and performance
IPC 4101 – Specification for Base Materials for PCB's
IPC 6012 – Qualification and performance for PCB's
IPC-A-600 – Acceptability of PCB's
IPC J-STD-002 – Solderability on component pin/connection
IPC J-STD-003 – Solderability on print
IPC J-STD-004 – Soldering Flux Requirements
IPC J-STD-006 - Solder requirements for solder wire
IPC 2220 series – Design of prints and solder islands/holes

Any more??



Design - DfM

IPC 2220 Series

IPC-2221 + 2222

IPC-7351/2

PCB Requirements

IPC 6010 series + IPC-A-600

IPC-6012

IPC-SM-840

IPC-1601

IPC-4552

IPC-4553

J-STD-003

PCB Acceptance

W.I.P

kamstrup

Assembly Requirements & Acceptance

IPC-J-STD-001 Series

J-STD-002

J-STD-003

J-STD-004

J-STD-005

J-STD-006

J-STD-020

J-STD-033

J-STD-075

IPC-A-610

IPC-7711/21

IPC/WHMA-A-620

Reference	Description
IPC-J-STD-003	Solderability Tests for Printed Board
IPC-4552	Specification for Electroless Nickel/Immersion Gold (ENIG) Plating for Printed Circuit Boards
IPC-4553	Specification for Immersion Silver Plating for Printed Circuit Boards
IPC-4761	Design Guide for Protection of Printed Board Via Structures
IPC-6010 series	Specifications
IPC-6012	Qualification and Performance Specification for Rigid Printed Boards – (class 2 + 3)
IPC-9252	Guidelines and Requirements for Electrical Testing of Unpopulated Printed Boards
IPC-A-600	Acceptability of Printed Boards
IPC-SM-840	Qualification and Performance of Permanent Polymer Coating (Solder Mask) for Printed Boards
IPC-TM-650	Test Methods Manual - **2.4.22 - 2.3.25 - 2.6.27 - 2.4.1 - 2.5.7** -
IPC-1601	Printed board handling and storage guidelines

kamstrup

PCB requirements and acceptance -

denne viden er værdifuld som et incitament til at opnå god producerbarhed i DfM-processen

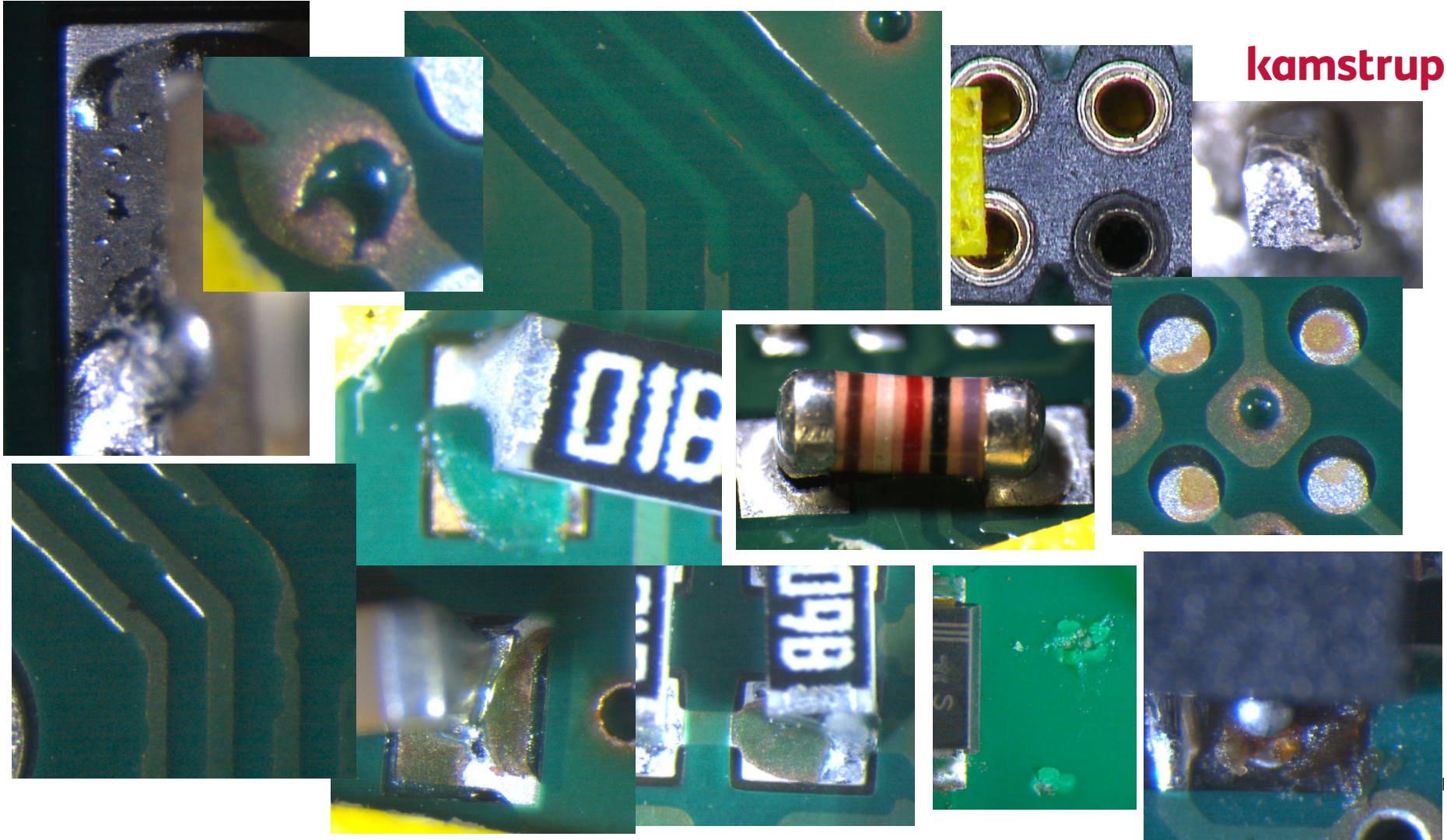
Additional suggestions?:

IPC-TM-650, Method 2.6.25: *CAF Resistance Test (Conductive anodic filament)*

IPC-9691 *User Guide for CAF testing*

IPC-5704 *Cleanliness of Unpopulated PCB (3 pages – 50% given)*

IPC-TM-650	Name	Purpose
2.3.25	Detection and Measurement of Ionizable Surface Contaminants by Resistivity of Solvent Extract (ROSE)	determine the total ionic content extractable from on, and absorbed within, the surface of printed wiring boards
2.4.22	Bow and Twist - Laminate	PCB flatness test
2.4.1	Soldering Resistance of Laminate Materials	determine the resistance of laminate materials (both unclad and etched surfaces) to the thermal abuse of a solder dip
2.5.7	Dielectric Withstanding Voltage, PCB	determine whether insulating materials and/or conductor spacings are adequate.
2.6.27	Thermal Stress, Convection Reflow Assembly Simulation	simulate those effects that are the result of soldering thermal excursions.


kamstrup

Capabilities vs Needs

- A concept DfM is required
- Concurrent engineering
  - Customer expectations - quality – lifetime
  - Design density vs producibility
  - Package types
  - Procurement
  - Manufacturing
  - Handling
  - Installation & Service

PCB supplier capabilities need to match your needs… *comfortably* – no need to choose a supplier where your needs are on the edge of what is capable.
IPC standards can create the common ground of **understanding** the expectations of the customer vs limitations of the supplier.

There is a fine line between assumptions and expectations – but when the decision is based on cost….
"*The bitterness of poor quality remains long after the sweetness of low price is forgotten*." **Benjamin franklin**

DFM

- ☐ Før prototype
  - ☐ Placement
    - ☐ PCB test krav
    - ☐ Slutsamling krav
  - ☐ Valor
- ☐ DFM Review
  - ☐ Produktoverblik
    - ☐ Erfaringer implementeret?
  - ☐ Nye komponenter?
  - ☐ Nye processor?
  - ☐ Process gennemgang
    - ☐ SMT
    - ☐ THT
    - ☐ Pressfit
    - ☐ Fræs
    - ☐ Test
    - ☐ Andet
  - ☐ Mekaniske krav (5538…)
    - ☐ Krav fra tidligere?
    - ☐ Generiske krav?
  - ☐ Vandproduktion / Målersamling
    - ☐ Specielle hensyn
  - ☐

Overleveringer

- ☐ Processer klar til 0-serie?
  - ☐ Route + tider
  - ☐ Alle processer klarmeldt?
  - ☐ Testfikstur offline/inline?
- ☐ Fiksturaflevering
- ☐ Testdatabase

Hvordan forbedrer vi DFx processen?

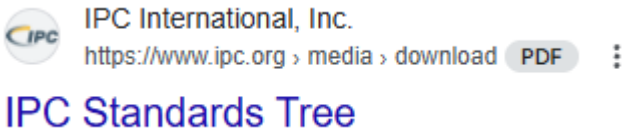
- Vigtigt med feedback fra PT og Test - *understøtter læringsprocessen*
  - Procesgrænser/vinduer – SPC data
- Evaluering og godkendelse af komponent valg i udviklings processen
  - Ift. IPC, maskinmæssige begrænsninger, kapabiliteter…
- Forbedre checkliste og check grænser løbende
  - Ift. proces data, QA, afvigelser
- Bedre regulering af footprint designs på tværs af udviklings divisioner.
- Tidligere involvering i Design fasen fra NPI team og PT team
- Bedre kvalitetssikring / konformitet i forhold til IPC regler, metoder of kvalifikationer

Standard	Name	Standard	Name
IPC-2221	**Generic Standard on Printed Board Design**	**IPC-7351**	**Generic Requirements for Surface Mount Design and Land Pattern Standard**
IPC-2222	**Sectional Design Standard for Rigid Organic Printed Boards**	**IPC-7352**	**Generic Guideline for Land Pattern Design (+THT)**
IPC-6012	Qualification and Performance Specification for Rigid Printed Boards		
IPC-A-600	Acceptability of Printed Boards		
IPC-A-610	Acceptability of Electronic Assemblies		
IPC-J-STD-001	Requirements for Soldered Electrical and Electronic Assemblies	IPC-1782	Standard for Manufacturing and Supply Chain Traceability of Electronic Products
IPC-7095	Design and Assembly Process Implementation for BGAs		
IPC-7093	Design and Assembly Process Implementation for Bottom Termination Components (BTCs)*	IPC-1791	Trusted Electronic Designer, Manufacturer, and Assembler Requirements
IPC-9701	Performance Test Methods and Qualification Requirements for Surface Mount Solder Attachments	IPC-1401	Corporate Social Responsibility and Sustainability Protocols for Electronic Manufacturing Industry
IPC-9252	Requirements for Electrical Testing of Unpopulated Printed Boards	IPC-1402	Standard for Green Cleaners Used in Electronics Manufacturing
IPC-TM-650	Test Methods Manual	IPC-1752	Materials Declaration Management
IPC-CC-830	Qualification and Performance of Electrical Insulating Compound for Printed Wiring Assemblies	IPC-1755	Conflict Minerals Data Exchange Standard
IPC-2615	Printed Board Dimensions and Tolerances		
IPC-9797	Press-Fit Standard for Automotive Requirements and Other High-Reliability Applications		

Tips

IPC Document Revision Table
https://www.ipc.org/ipc-document-revision-table

https://go.ipc.org/en-us/ipc-checklist



Design for Manufacturing

DfM er vigtigt tidligt i NPI-projekter, hvor det vil have en større indvikning på cost. Jo senere i NPI faserne, jo mindre er den potentielle cost besparelse.

- Hvordan kan vi understøtte og forbedre DfM bedre?
- Hvilke IPC-standarder kan understøtte DfM ved NPI?
- Er der andre standarder at overveje?